Chapter

19

Receiver Gages — Go Gages and Functional Gages

James D. Meadows
Institute for Engineering & Design, Inc.
Hendersonville, Tennessee

James D. Meadows, president of the Institute for Engineering and Design, Inc., has instructed more than 20,000 professionals in Geometric Dimensioning and Tolerancing and related topics over the last 30 years. He is the author of two current hardcover textbooks, a workbook and a 14-hour, 12-tape applications-based video training program on GD&T per the ASME Y14.5M-1994 standard on dimensioning and tolerancing. He is an ASME Y14.5.2 Certified Senior Level Geometric Dimensioning and Tolerancing Professional. Mr. Meadows is a member of eight ANSI/ASME and ISO standards committees and serves as the chairman of the committee on Functional Gaging and Fixturing of Geometric Tolerances. He is a journeyman tool and die maker and a graduate of Wayne State University.

19.1 Introduction

Receiver gaging is one of the most effective ways to determine the functionality of workpiece features.

There are two members of the receiver or attribute gage family: Functional Gages and GO gages. Functional and GO gages both determine feature compliance with a fixed size boundary; hence they are considered attribute gages.

Functional Gages inspect compliance with a constant functional boundary commonly associated with a worst mating condition. This boundary is known as a maximum material condition (MMC) concept virtual condition boundary. Functional Gages are made to the MMC concept virtual condition boundary of the features they inspect, then toleranced so they represent a situation worse than the features will face in assembly conditions.

GO gages are used to determine compliance with the maximum material condition boundary of perfect form required by several American National Standards (ANSI B4.4, ASME Y14.5, and ASME Y14.5.1).

This type of measurement is a physical representation of the theoretical principles of geometric tolerancing of workpieces. It shows the datum feature simulation and virtual condition boundaries as pins and holes that are cylindrical, diamond-shaped, widths, and even oddly configured. It demonstrates that planar features are represented by planar rails and that datum features and controlled features are represented in gages and fixtures by the shape of the MMC or virtual condition they generate. It allows a theoretical boundary to take on a physical form that a person can actually hold in their hands, and, by doing so, is capable of making a difficult geometric concept easy to understand.

Functional and GO gaging are time-tested tools of 3-dimensional (3-D) measurement that determine whether or not workpiece features will actually fit into assemblies. They do this without the use of computers or software. They are reliable and low tech. If used in a well-balanced measurement plan in conjunction with other measurement tools, they can provide the confidence needed to accept produced parts on the basis that they will perform their intended function.

Gaging of this variety is sometimes viewed as inappropriate because it produces no variables data (specifically how a feature has departed from perfect geometric size, form, orientation or location) and is therefore incapable of assisting in the statistical process control of manufacturing methods. However, many measurement techniques that do produce variables data are not representative of worst case assembly conditions and collect very little 3-D data concerning worst case feature high point interference possibilities. The type of data collected by functional and GO gaging is considered attribute (good vs. bad) information.

Both variables data and attribute data have their place in a well-balanced measurement procedure. Unfortunately, many measurement professionals are led to believe that only one of the two types of measurement information is to be used. Therefore, they lose the benefits of the type they do not choose.

19.2 Gaging Fundamentals

In a perfect circumstance, fixed limit gages accept all features that conform to their tolerance specification and reject all features that do not conform to their tolerance specification. The GO gage and the Functional Gage should each completely receive the feature it is inspecting.

GO plug gages should enter holes over the full length of the hole when applied by hand without using extreme force. A GO cylindrical ring gage should pass over the entire length of a shaft when applied by hand. This inspects not only a violation of the maximum material condition size limit, but also the envelope of perfect form at maximum material condition that American National Standards require. The rule in ANSI is that size limits control the surface form of rigid features.

The international rule is not the same. In the International Organization for Standardization (ISO), size is independent of form. Therefore, according to the ISO policy, unless otherwise specified, size inspection does not require a full form GO gage. Simple cross-sectional inspection procedures are all that are necessary to verify size requirements.

In ANSI-approved documents, NOGO gages are designed to inspect violations of the least material condition (LMC) limit of size. The LMC limit of feature size is inspected with a NOGO gage (or a simulation of this gage). The NOGO gage is a cross-sectional checking device, treating a cylinder as though it was a stack of coins. Each coin in the stack represents a circular cross-section of the surface. Each cross-section must not measure less than the least material condition. Since the requirement is that the gage “not go” over the workpiece, the NOGO gage should not be able to pass into or over the workpiece feature being inspected at any orientation or location.

A Functional Gage pin must be able to fully engage the hole it is inspecting over the entire depth of the hole without extreme force being applied. A Functional Gage hole, which is a full form ring gage, should be able to receive the shaft being gaged over the full length of the shaft without extreme force being applied. If planar datum features are being simulated by the gage, the datum features on the workpiece must contact the datum feature simulators on the gage with the required contact specified by ASME Y14.5M-1994 and ASME Y14.5.1M-1994. If restraint is to be used to inspect the workpiece features while on the datum features, it must be specified in notes or other documents relating to the feature measurement requirements. If no restraint is to be used, or restraint insufficient to alter the measurement readings, no note is required. However, a Free State Inspection symbol may be used inside feature control frames to clarify that the part is not to be distorted by restraining forces during the inspection procedure.

19.3 Gage Tolerancing Policies

Gages must be toleranced. There are three gage tolerancing policies commonly practiced throughout the world. These policies are known as: Optimistic Tolerancing, Tolerant Tolerancing, and Absolute Tolerancing (also called the Pessimistic Tolerancing approach).

Optimistic Tolerancing is not an ANSI-recommended practice for gages. It assures that all parts within specifications will be accepted by the gage. Most of the technically out-of-tolerance parts being inspected by the gage will be rejected, but a small percentage of technically out-of-tolerance parts will be accepted. This policy is accomplished by tolerancing the gages from their appropriate MMC or MMC concept virtual condition boundary so that gage pins can only shrink and gage holes can only grow from these boundaries. This method subtracts material from the gage so that gagemaker’s tolerances, wear allowances, form tolerances and measurement uncertainties all reside outside the workpiece limits of size and geometric control.

Tolerant Tolerancing is also not an ANSI-recommended practice for gages. It assures that most parts within specification will be accepted by the gage. Most of the parts outside the specification will be rejected by the gage. A small percentage of parts outside the specifications may be accepted by the gages or a small percentage of parts that are within the specifications may be rejected by the gages. This policy may either add or subtract material from the gage MMC boundary or MMC concept virtual condition boundary since the tolerance is both plus and minus around these boundaries. This means that some of the gagemaker’s tolerances, the wear allowances, the form tolerances and the measurement uncertainties reside both within and outside of the workpiece limits of size and geometric control.

Absolute Tolerancing is recommended. This type of gage tolerancing means that gage pins are toleranced only on the plus side of their MMC concept virtual condition boundary (only allowing them to grow) and that gage holes are toleranced only on the minus side of their MMC concept virtual condition boundary (only allowing them to shrink). This has the effect of rejecting all parts not within tolerance and accepting all parts that are within tolerance except those borderline parts that fall within the range of the gage tolerance. Part features that are produced within the range of the gage tolerance are rejected as though they were not in compliance with their geometric tolerance, even though technically they are within the design specification limits. This is the price we must pay if we choose to accept no parts that have violated their tolerance.

Absolute Tolerancing is the ANSI-recommended practice of applying gage tolerances so that the gages will reject all workpiece features that reside outside of their specifications. This is to assure complete random interchangeability of mating parts in an assembly inspected by these gages. Gagemaker’s tolerances, wear allowances, form tolerances and measurement uncertainties of the gage are all within the workpiece limits of size and geometric control. These gage tolerances add material to the gage. The gages are dimensioned at the MMC limit or MMC concept virtual condition limit of the feature being gaged, then toleranced so that gage pins can only get larger and gage holes can only get smaller. This policy is based on the gaging premise that all parts not within tolerance will be rejected, most parts that are within tolerance will be accepted, and a small percentage of in-tolerance parts that are considered near the borderline between good and bad will be rejected as though they had violated their tolerance requirements.

The ANSI-recommended amount of tolerance is 5% of the tolerance on the feature being gaged plus an optional 5% of the tolerance allowed for wear allowance. This recommendation is only a place from which to begin the decision as to what tolerance will be assigned to the gage. Using the Absolute Tolerancing method, the actual amount of tolerance chosen will depend on the number of parts the gage will accept and the number of parts one is willing to reject with the gage. It is a balance between the cost of the gage and the cost of the rejection of good parts by the gage. The smaller the gage tolerance, the more expensive the gage and the quicker the gage will wear beyond acceptable limits and begin to accept bad parts. On the other hand, the larger the gage tolerance, the less expensive the gage. However, the gage will run the risk of being produced at a size that will reject a larger number of produced parts that are within tolerance but near the borderline.

19.4 Examples of Gages

The following examples show a variety of workpieces and the gages to verify their conformance with common geometric tolerances. The gages may be toleranced using maximum material condition, least material condition, or regardless of feature size concepts. Each has advantages and disadvantages of cost and part acceptance.

19.4.1 Position Using Partial and Planar Datum Features

In Fig. 19-1 the workpiece is a simple rectangular part with two holes. The datum reference frame is constructed from three planar surfaces, two of which are partial datum features of limited specified length.

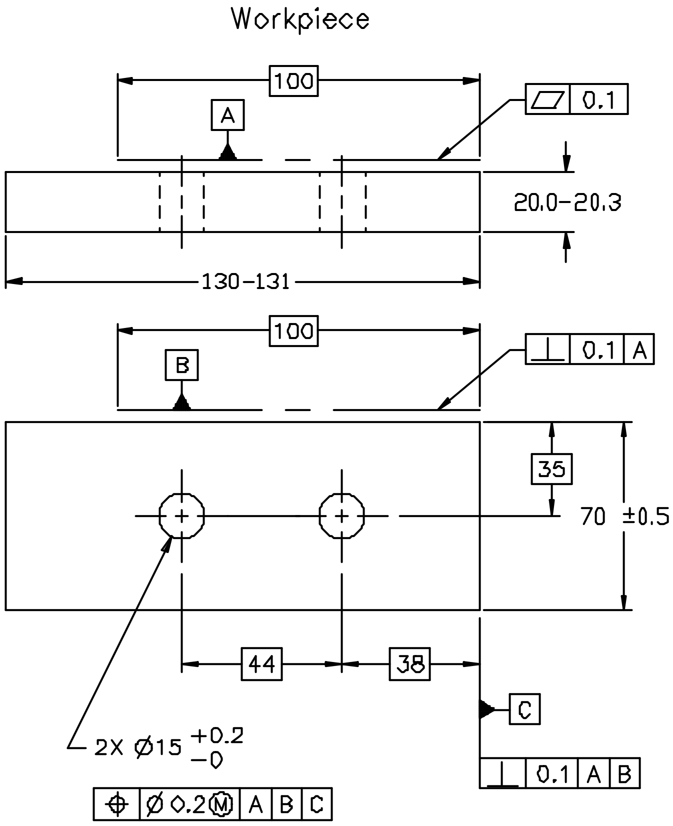


Figure 19-1 Position using partial and planar datum features

This is similar to using two datum target areas. The two partial datum features and the tertiary datum feature are first controlled and interrelated. The primary datum feature is given a flatness control. The secondary datum feature is given a perpendicularity control to only the primary datum plane formed by the three highest points within the primary datum feature. This controls both the orientation of the secondary datum feature and also its flatness. The tertiary datum feature is given a perpendicularity control to both the primary and secondary datums. Again, the perpendicularity control both forms and orients the tertiary datum feature. These three geometric characteristics of flatness, perpendicularity to one datum and then perpendicularity to two datums are used to give progressively more powerful geometric controls to the datum features. This not only gives them a needed interrelationship, but also implies a sequence of events for the reader of the drawing. These controls will also make the tolerancing of the gage easier, since the controls given to the gage elements will simply mimic the controls given to the part and use 5%-10% of the tolerance of the feature it represents.

The fourth and last geometric control shown is to position the two holes in the pattern to one another and to the three datum planes given by the three highest points of the primary datum feature, the two highest points of the secondary datum feature with respect to the primary datum plane, and the one highest point of the tertiary datum feature with respect to the primary datum plane and the secondary datum plane. Fig. 19-2 shows the gage for Fig. 19-1. The gage has, in order of consideration:

- A primary datum feature that is flat to within 10% of the flatness tolerance given to the primary datum feature on the workpiece,
- A secondary datum feature that is perpendicular to the primary datum plane to within 10% of the tolerance given to the secondary datum feature on the workpiece and,
- A tertiary datum feature that is perpendicular to the primary datum plane and the secondary datum plane to within 10% of the tolerance given to the tertiary datum feature on the workpiece.

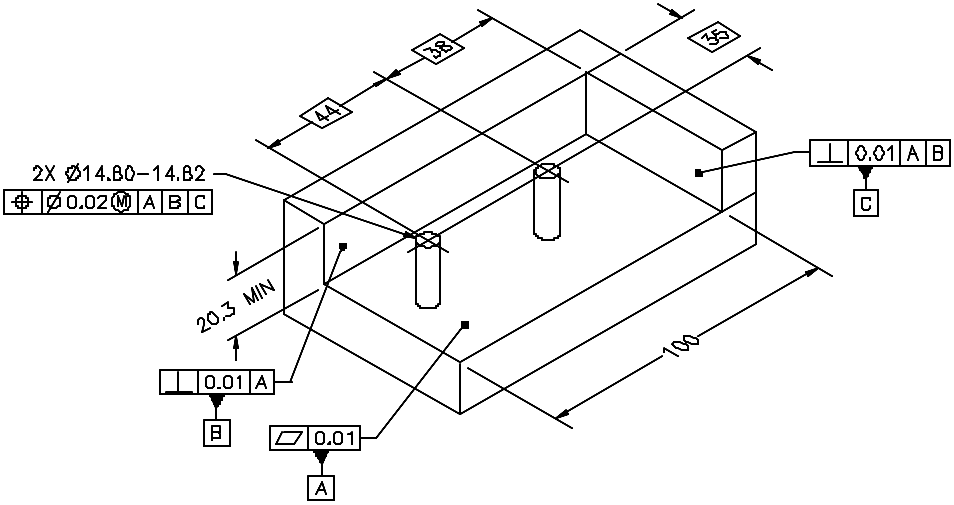
Each datum feature simulator on the gage has enough surface area to entirely cover the datum feature from the workpiece it represents. It must try to hit the highest points of contact on the datum feature to properly construct the datum plane and unless it has enough surface area, it runs the risk of missing the appropriate high points and improperly establishing the datums. Too much surface area and the gage runs a similar risk of establishing nonfunctional and therefore inappropriate datums.

The gage also has two gage pins. Ideally, these gage pins will be at least as long as the holes they are gaging are deep. If these were simply GO gages meant to gage the maximum material condition of the holes, they would not be mounted on a plate, would have no relationship to the datum reference frame, and would be made at the maximum material condition of the holes. But these are Functional Gage pins meant to gage the positional requirement of the holes, so they are mounted and related to the datums and dimensioned to be at the virtual condition of the holes they are to inspect.

The size of the gage pins are dimensioned to begin at the virtual condition of the holes being gaged and go up in size tolerance by 10% of the size tolerance given to those holes. The gage pins also have a positional control based on 10% of the tolerance given to the holes they are gaging. If the workpiece is capable of being applied to the gage (as shown in the illustration), while maintaining its appropriate contact on the datum feature simulators, it is judged to be in compliance with the positional requirement. The size limits of the holes must be inspected separately.

One of the important requirements of workpieces to be gaged is that they are sufficiently defined to allow the gage designer/gagemaker to simply follow from control to control using 5%-10% of the tolerances that the workpiece shows. Unless the workpiece is complete in its definition, the gage designer cannot use it as a guide in the complete geometric definition of the gage. If necessary, the gage designer may add notes or even a procedural sheet to explain the proper use of the gage. As with all inspection, unless otherwise specified, the gage is to be used at 20 degrees Centigrade or 68 degrees Fahrenheit.

Gage



Workpiece Applied to Gage

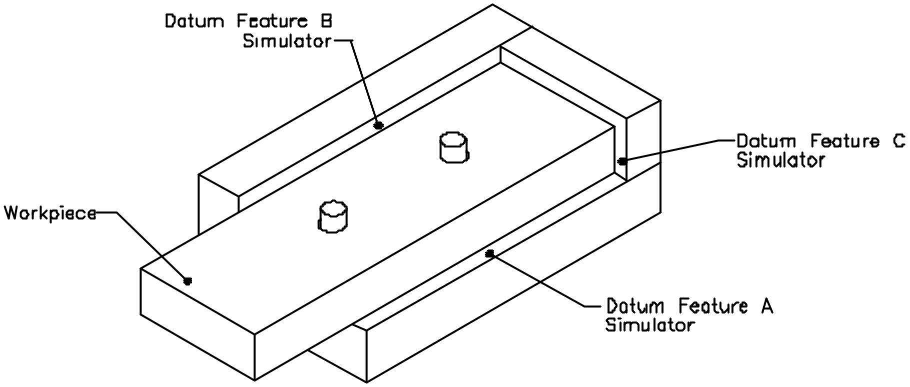


Figure 19-2 Gage for verifying two-hole pattern in Fig. 19-1

19.4.2 Position Using Datum Features of Size at MMC

Fig. 19-3 shows a workpiece that uses a planar primary datum feature, a secondary datum feature of size and a tertiary datum feature of size. By the time one gets to the tertiary datum feature of size, all spatial degrees of workpiece freedom have been eliminated by the primary and secondary datum features except angular orientation (what is commonly referred to as pattern rotation). The workpiece has been sufficiently defined to discuss the construction of the gage to inspect the position of the four-hole pattern. As is the case with many such workpieces, if the workpiece fits the gage used for the four-hole pattern's positional control, that gage will also inspect the position of the slot and the center hole's perpendicularity since they are represented on the gage as datum features for the four holes and they are represented at their virtual condition.

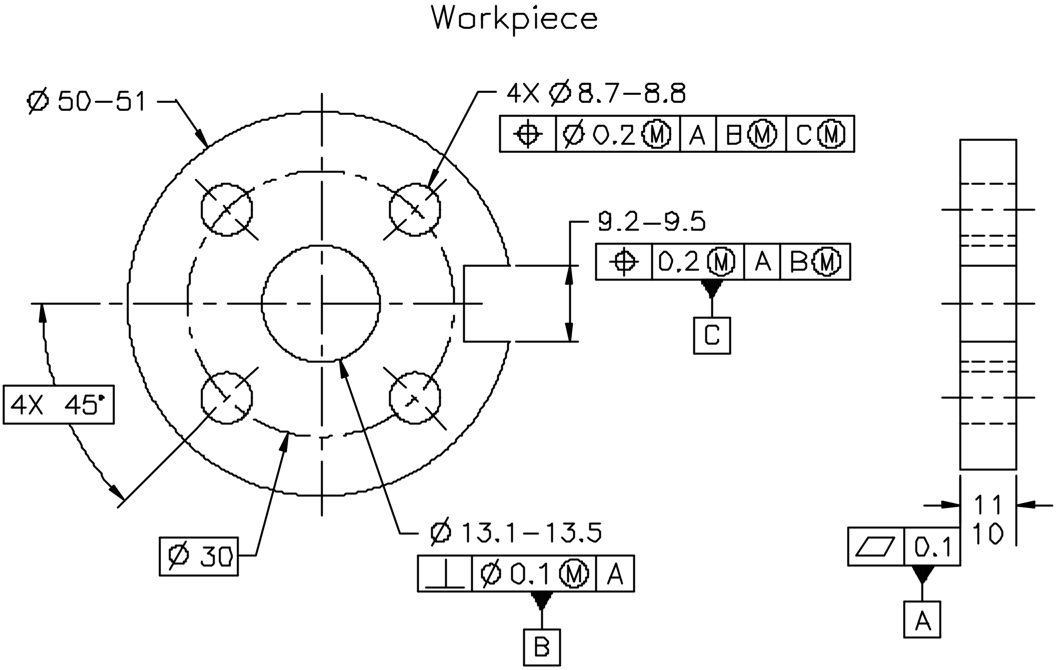


Figure 19-3 Position using datum features of size at MMC

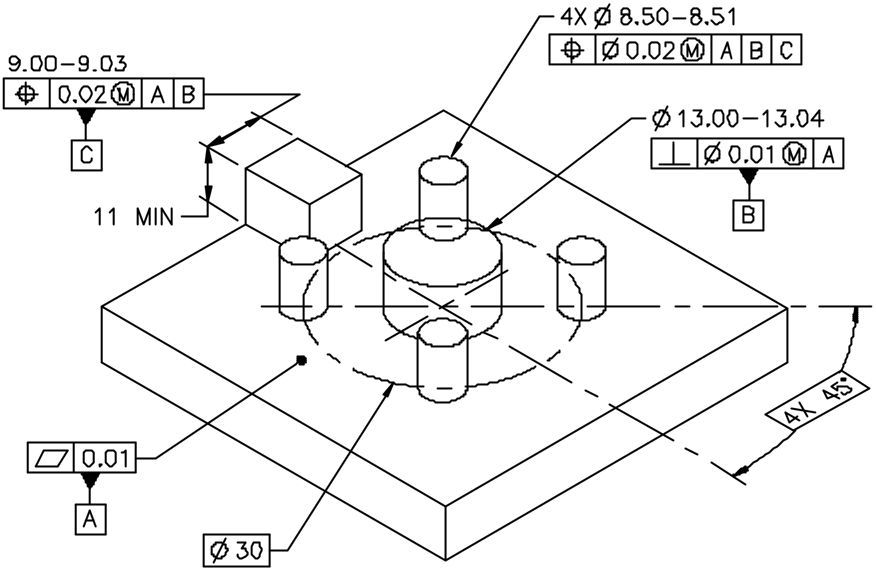
A separate gage to inspect them individually would be considered redundant by most inspectors, since they would be represented at exactly the same size, orientation, and alignment as they are on the gage for the four-hole pattern. Again, as with Fig. 19-1, Fig. 19-3 has used a progressive geometric definition to make the workpiece complete enough to be both produced and inspected (at least for most of the purposes of this discussion).

1. The primary datum feature is controlled for 3-D form (flatness).
2. The secondary datum feature of size is controlled perpendicular to the primary datum plane.
3. The tertiary datum feature of size is controlled for position to the primary datum plane and the secondary datum axis.
4. The hole pattern is then controlled to the primary datum plane (for perpendicularity), the secondary datum axis (for location), and the tertiary datum centerplane (for angular orientation).

The maximum material condition concept has been used everywhere it is allowed for ease of manufacture and increased geometric tolerance while preserving functionality. The use of the MMC symbol after the geometric tolerances and also after the datum features of size will make it easy to represent them with gage pins at their appropriate constant boundary size (their virtual condition size). As in Fig. 19-1, each size tolerance and geometric tolerance has been mimicked by the gage that uses the same geometric characteristics and 10% of the tolerance on the workpiece. This geometric tolerance allows the gage pins to be only larger than the virtual condition boundary of the hole being represented so as to not accept a workpiece that exceeds its allowed tolerances.

This tolerancing of the gage pins to only get larger than the worst case boundary (and in the case of gage holes to only get smaller than the worst case boundary) being inspected will make the gages reject a small percentage of technically good parts that are near the borderline between good and bad. This way the gage doesn't accept a bad part. One must remember that this absolute tolerancing method is preferred by ANSI-approved documents, but is not the preferred practice in the ISO-approved documents on gaging.

The gage in Fig. 19-4 does not show the use of the maximum material condition symbol after the datum features of size. This will reduce the allowed inaccuracies in the gage, increase the chance of producing a more accurate gage and will accept more of the produced workpieces. Use of the regardless of feature size (RFS) concept after datum features of size on the gage design may increase the cost of the gage, but should more than make up for this additional cost by the gage's acceptance of a greater number of per-



Workpiece Applied to Gage

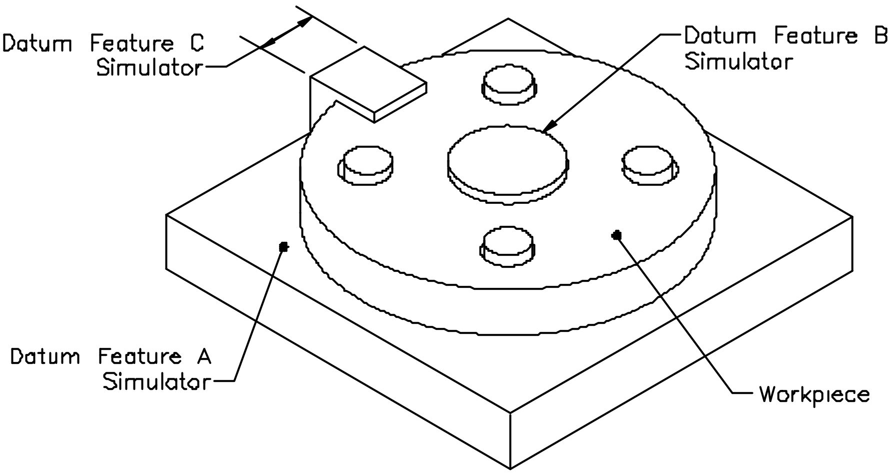


Figure 19-4 Gage for verifying four-hole pattern in Fig. 19-3

drawing technically good parts that are inspected by the gage. Even though the gage may use the regardless of feature size concept, it is commonly understood that receiver type gages, as discussed here, are most often used to inspect workpiece features and represent workpiece datum features that use the maximum material condition concept.

19.4.3 Position and Profile Using a Simultaneous Gaging Requirement

In Fig. 19-5, a simultaneous gaging requirement exists between a four-hole pattern and a profile control because both use exactly the same datum reference frame in exactly the same way. Both use a primary planar datum feature (A) and a secondary datum feature pattern of size (B) at maximum material condition. This creates a situation wherein, unless specified as a SEPARATE REQUIREMENT, the two geometric controls (position of the four-hole pattern and profile of the outside of the workpiece in the front view) must be inspected by the same gage. This is a more restrictive requirement than if both controls were allowed to use their own separate gage.

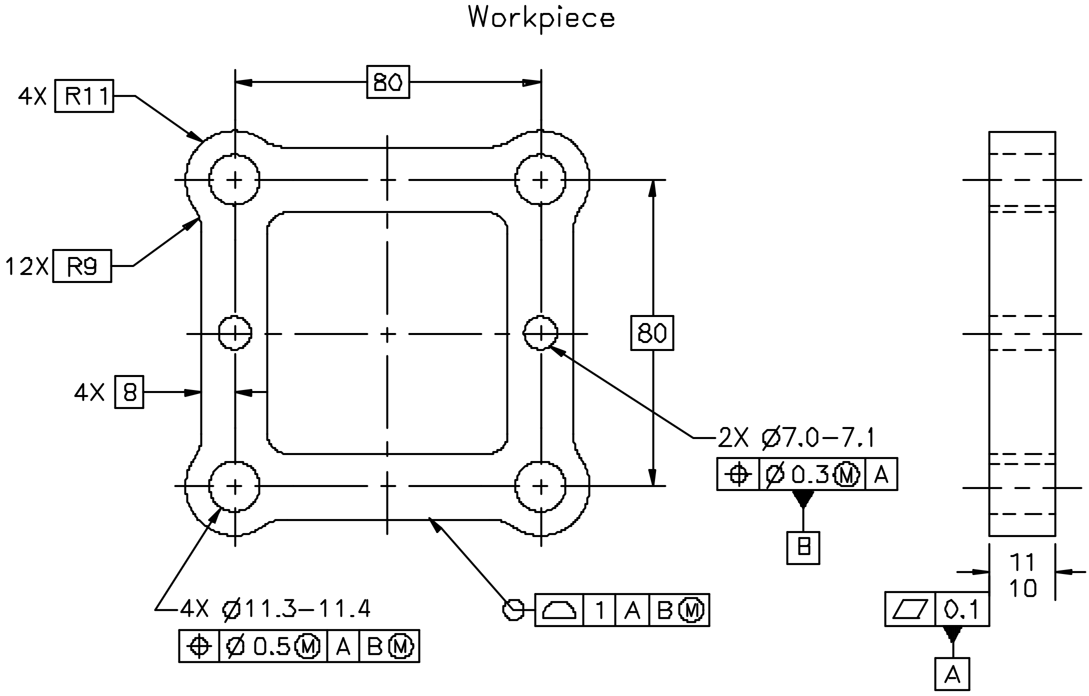
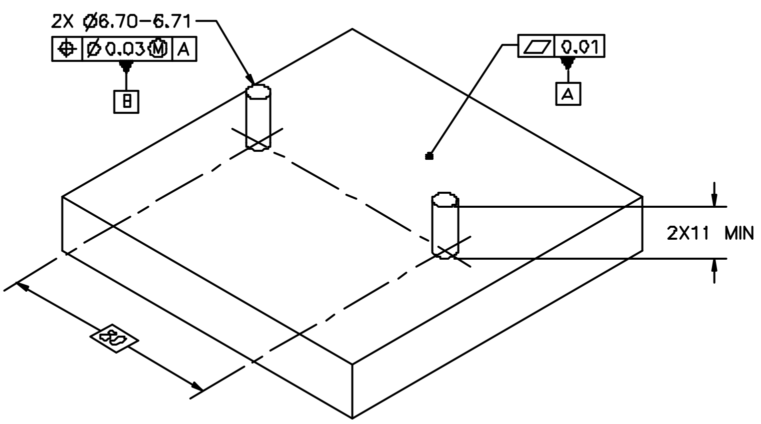


Figure 19-5 Position and profile using a simultaneous gaging requirement

For example, in a separate gaging requirement, the four-hole pattern could rock on datum A. This creates a different angle to be accepted than the rocked orientation on datum A used to accept the profile. Or as the datum pattern B holes grew from their virtual condition boundary toward their least material condition, the four-hole pattern as a group could shift to the left and the profile could shift to the right and be accepted. But in a simultaneous gaging requirement this would not be acceptable. Both the four holes and the profile would have to be accepted by one gage in one rocked orientation, with the four holes and the profile shifted in the same direction (if rock and shift were to occur).

Since Fig. 19-5 contains profile that is a geometric tolerance that cannot be referenced at maximum material condition, one may want to use a fixture to simulate only the datum features. See Fig. 19-6. If this is done, the gage/fixture will be capable of gaging the hole-to-hole requirement between the two holes in datum pattern B as well as their relationship to the primary datum plane A. It is also capable of stabilizing the workpiece to use a variables data collector such as a computerized coordinate measurement machine to measure the position of the four holes and the profile of the workpiece. The workpiece is stabilized in one orientation to measure the four holes and the profile controls. If the four holes and the profile meet their geometric tolerances when measured in that orientation, they may be considered as having met the SIMULTANEOUS REQUIREMENT condition of their inspection.

It is possible to create a complete gage that will not only represent the datum features, but also the four holes at their virtual condition (MMC concept) boundary and the worst case mating condition of the profile's outer boundary. Although the gage as shown in Fig. 19-7 for Fig. 19-5 will not gage the profile's inner boundary (which, if important, can be represented or inspected in other ways), the gage is


Gage

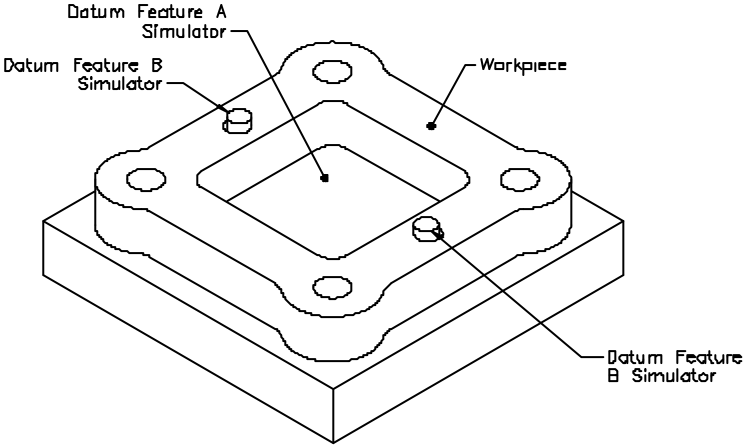
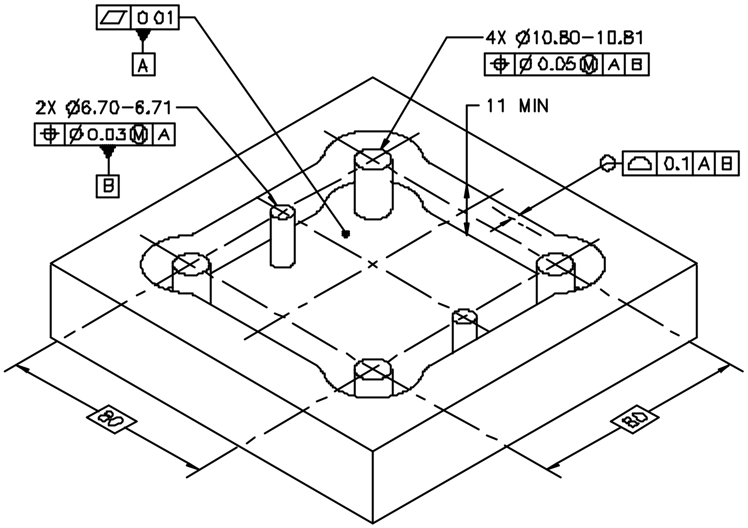

Workpiece Applied to Gage

Figure 19-6 Gage for simulating datum features in Fig. 19-5

Gage



Note: The nominal profile for the gage is the maximum part profile tolerance boundary. The profile tolerance on the gage is unilaterally in. The gage simultaneously verifies the hole locations and profile outer boundary. It does not verify the profile inner boundary.

Workpiece Applied to Gage

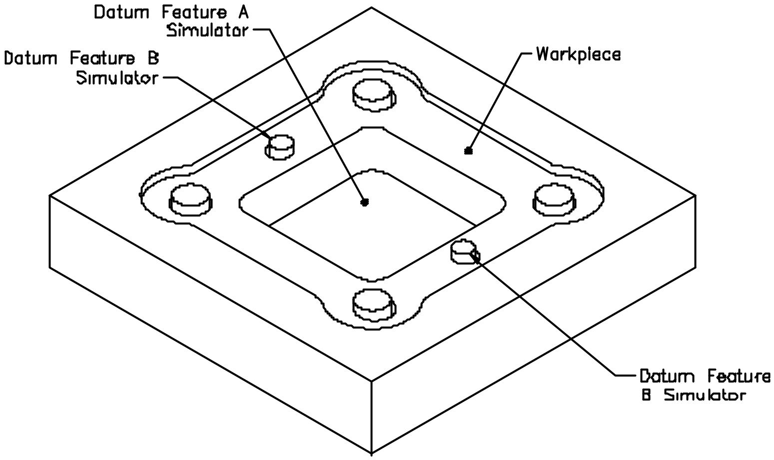


Figure 19-7 Gage for verifying four-hole pattern and profile outer boundary in Fig. 19-5

capable of inspecting the positional tolerance of the four holes, the outer boundary of the profile control, and the interrelationship between the four-hole pattern and the profile under the simultaneous requirement rule. The gage simulation for the profile has a nominal size that is the maximum part profile tolerance boundary. The profile tolerance on the gage (shown as 10% of the profile tolerance on the workpiece) is unilateral inside (as with all gage holes), allowing the gage tolerance to accept no profile that exceeds the outer boundary of the workpiece's profile tolerance.

19.4.4 Position Using Centerplane Datums

Fig. 19-8 shows a simultaneous gaging requirement for a four-hole pattern and a larger center hole. Each uses exactly the same datums in the same order of precedence with the same material condition symbols after the datum features. This creates the simultaneous gaging requirement. This is a very sequential geometric product definition.

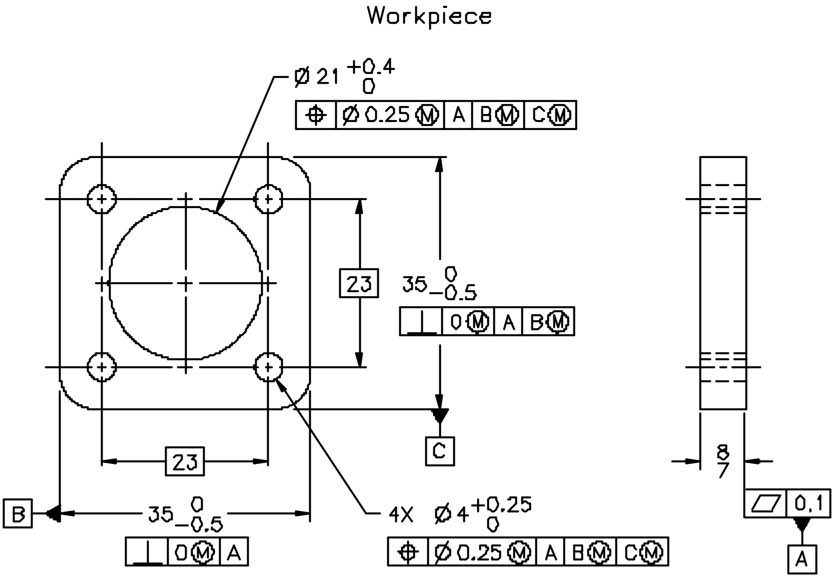
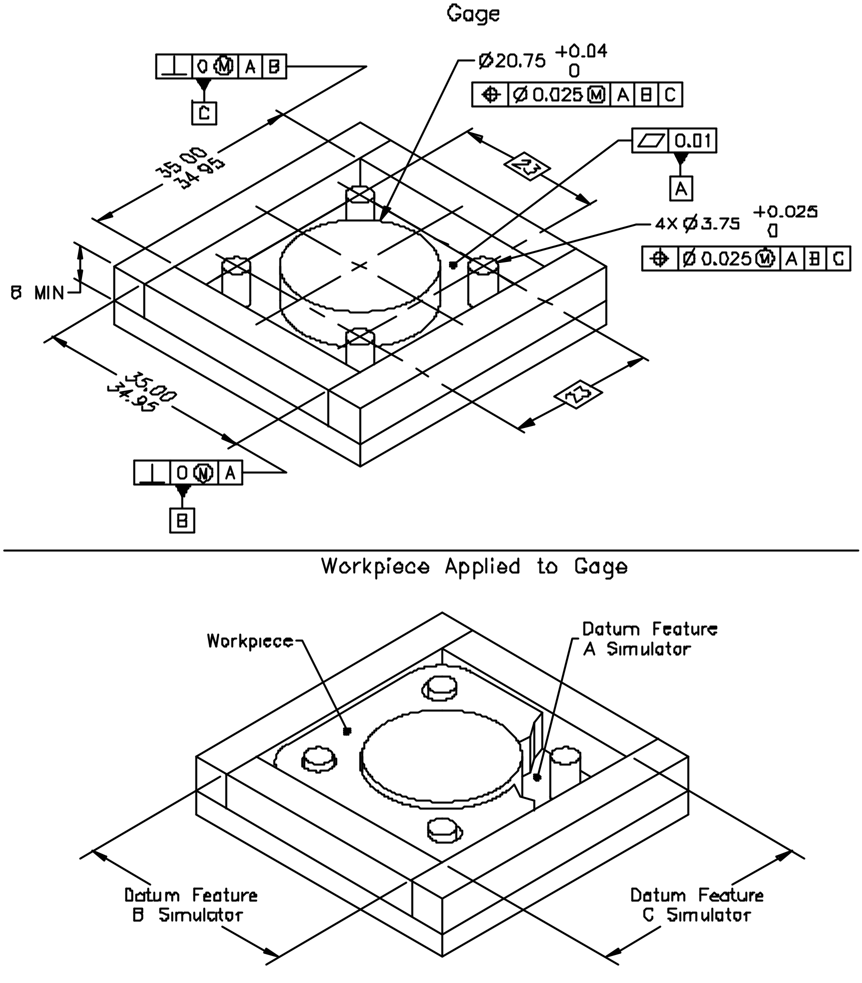


Figure 19-8 Position using centerplane datums

To understand the requirements, one might first look at the configurations and ignore the feature control frames. All four holes are shown centered to the hole in the middle and to the outside of the workpiece. The four holes are dimensioned 23 mm from each other, but since they are depicted centered to the center hole, we must assume each of the four holes is desired to be 11.5 mm from the center hole and from the middle of the workpiece. The hole in the center is exactly that; a hole we desire to be in the middle of the workpiece. The part is then geometrically toleranced in four steps. Step 1, the primary datum feature is identified and given a flatness tolerance. Step 2, the secondary datum feature is identified as one of the 35-mm widths creating a centerplane datum, and the datum feature that generates that centerplane is given a perpendicularity control back to the primary datum plane. Step 3, the tertiary datum feature is identified as the other 35-mm width creating a third datum plane which is also a centerplane datum. The datum feature that generates that centerplane is given a perpendicularity control back to the primary datum plane and the secondary datum centerplane. Step 4 is the simultaneous positional requirement of all five holes to each other and to the primary, secondary, and tertiary datum features. All geometric tolerances of perpendicularity and position are referenced at maximum material condition and use their datum features of size at maximum material condition. This makes it easy to represent each at a constant gage element size, either their MMC or their virtual condition, as applicable. Since in the case of the datum features of size a zero tolerance at MMC has been used, the MMC and the virtual condition are the same. Any gage that simulates these datum features will be able to gage their compliance with their given geometric tolerances and the geometric tolerances of the holes measured from them. The same Functional Gage will also be able to verify compliance with the 35-mm MMC size.

As shown in Fig. 19-9, step 1 on the gage shown represents datum feature A and gives it a flatness tolerance of 10% of the flatness tolerance on the workpiece. Step 2 on the gage represents datum feature B at a size of 35 mm plus zero and minus 10% of its size tolerance. It is then given exactly the same feature control frame the workpiece has on its datum feature B (10% of zero is still zero). Step 3 on the gage represents datum feature C at a size of 35 mm plus zero and minus 10% of its size tolerance. It is then given the same feature control frame the workpiece has on its datum feature C except it references its datum feature of size B at regardless of feature size. As explained in previous examples, this has the effect of increasing the cost of the gage by decreasing the allowed gage tolerance. However, it has a better chance of producing a gage that will accept more of the produced parts that are within their geometric tolerances. Step 4 on the gage represents all five controlled holes with gage pins. The gage pins begin at the virtual condition of the hole they represent and are toleranced for size with minus zero and plus 10% of the size tolerance of the hole. Then the gage pins are given a position tolerance of 10% of the position tolerance of the hole it represents to the datums simulated in steps 1-3.

Again, the datum features of size on the gage are referenced at regardless of feature size, even though the features they simulate are referenced at MMC. Keep in mind this is a personal choice. Gage datum


Gage

Workpiece Applied to Gage

Figure 19-9 Gage for verifying four-hole pattern in Fig. 19-8

feature of size simulations may be referenced at MMC. This will make the gage tolerance larger, and potentially decreases the cost of the gage. It also runs the risk of the gage being made at a size, orientation, and location that rejects more of the technically in-tolerance workpieces it gages.

In these examples, a zero tolerance at MMC was used on the controlled datum features of size and therefore a zero tolerance at MMC was used on the gage simulation of the controlled datum features of size. For the purposes of gage tolerancing, one may consider that a workpiece using a geometric tolerance at MMC has a total tolerance that includes the size tolerance and the geometric tolerance. If one adds the size tolerance and the tolerance from the feature control frame on the feature being considered, a true sense of the total tolerance on the feature can be understood. When distributing tolerance on the gage, the tolerance distribution may be that 5%-10% of the total tolerance on the feature being gaged can be used in the size limits of its gaging element, and a zero tolerance at MMC used in its feature control frame. The effect on the gage of this method of tolerance distribution is usually a more cost-effective gage without the possibility that the gage will accept more or less of the parts that it inspects.

19.4.5 Multiple Datum Structures

In Fig. 19-10, the positional controls shown use zero at MMC for their geometric tolerances. This makes it easy to illustrate that the only tolerance available for the gage designer to take 5%-10% of is the difference between the MMC and the LMC of the controlled features. In each case, both for the center hole that becomes datum feature D and for the four holes that eventually are positioned to A, D at MMC, and B, a total of 2 mm is used as the size tolerance. This means that when the gage is produced, the gaging elements (pins) that are used to simulate these holes will use a percentage of the 2 mm as the total tolerance on the gage pin sizes and their orientation and location geometric tolerances. This tolerance can be split between the gage pin size and its geometric tolerance or simply used as size tolerance while the geometric tolerance uses zero at MMC, or zero at LMC.

Fig.19-10 is sequentially toleranced, with a flatness control given to the primary planar datum feature, a perpendicularity tolerance given to the secondary planar datum feature back to the primary datum, and a perpendicularity tolerance given to the tertiary datum feature back to the primary and secondary datums.

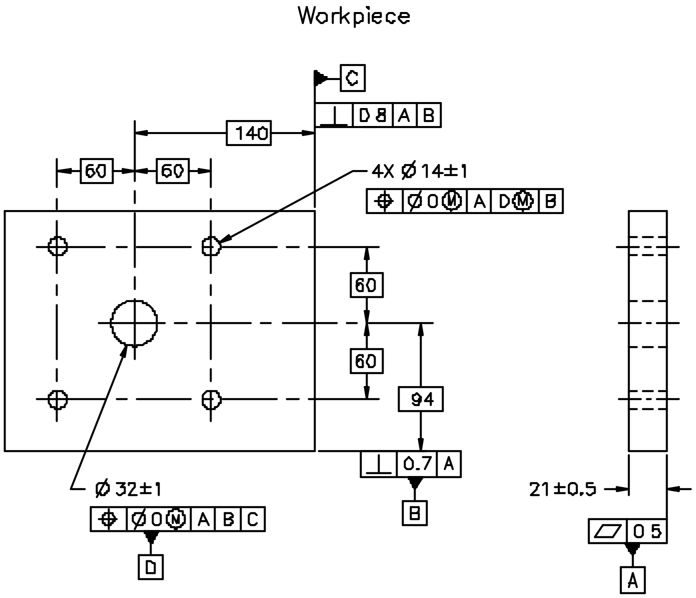
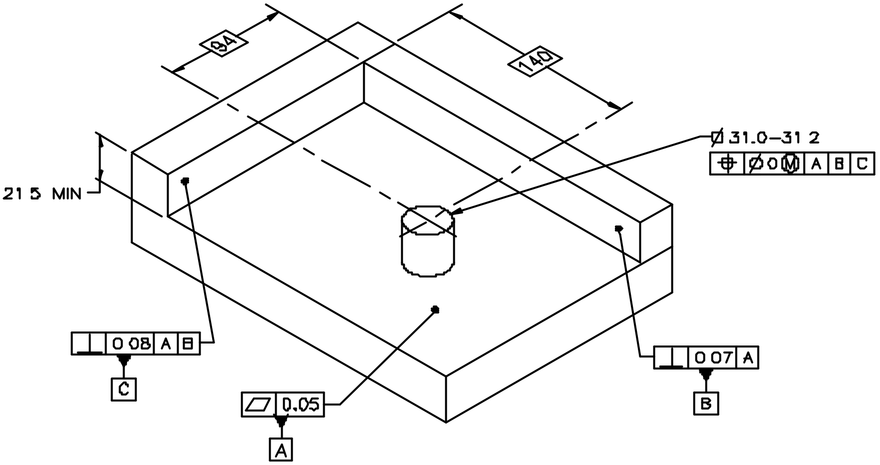


Figure 19-10 Multiple datum structures

This completes the first datum reference frame from which the center hole is positioned. The center hole is then made a datum feature (D) from which the outer four holes may be positioned for location on the X and Y axes while using datum A for perpendicularity and datum B for angular orientation.

Each geometric control is considered separately verifiable. If gaged, each positional control will be considered a different gage. Since each positional control uses a zero at MMC positional tolerance, the gages that inspect position will also be able to verify compliance with the MMC size envelope. The first gage verifies the position of the center hole. It consists of three planar datum feature simulators, each using exactly the same geometric control as the feature it represents. The only difference is that (as illustrated) a geometric tolerance of 10% of the feature it simulates has been used. The center hole being gaged is represented by a gage pin at the desired basic angle and distance from the datums (as depicted in Fig. 19-11). The gage pin is dimensioned at the virtual condition size of the hole it is gaging and is allowed to grow by 10% (0.2) of the tolerance on the hole. The gage pin is then given a positional tolerance of zero at MMC to the datum features used on the gage.

Gage to Verify Datum Feature D



Workpiece Applied to Gage

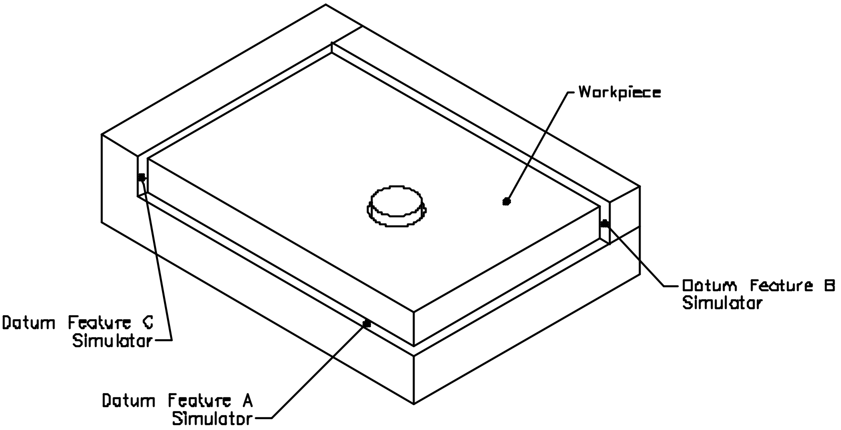
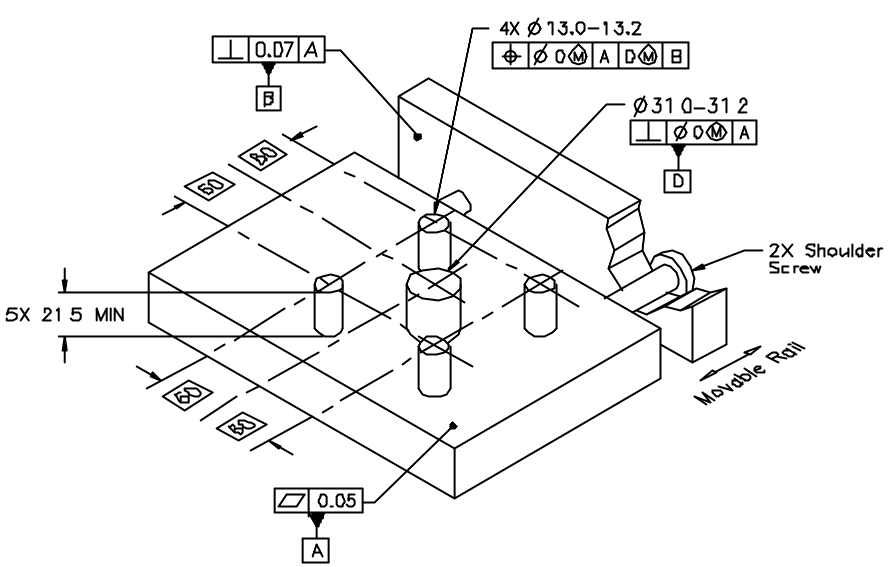


Figure 19-11 Gage for verifying datum feature D in Fig. 19-10

The last gage for Fig. 19-10 in Fig. 19-12 is used to inspect the position of the four-hole pattern. It begins with a datum feature simulator for datum A and uses a flatness tolerance of 10% of the datum feature it simulates. It also has a datum feature simulator for datum feature B (which is used as a tertiary datum feature to construct a fourth datum plane). This is used to control the pattern rotation (angular orientation) of the four holes and will be a movable wall on two shoulder screws. For the part being gaged to pass the gaging procedure, it will have to make contact with a minimum of two points of high point contact on the datum feature B simulator. This is to assure that the four-hole pattern has met the desired angular relationship to datum plane B and datum feature B. If, for example, only one point was contacted by the part on the datum feature simulator B, it would not assure us that the hole pattern's orientation had

Gage



Workpiece Applied to Gage

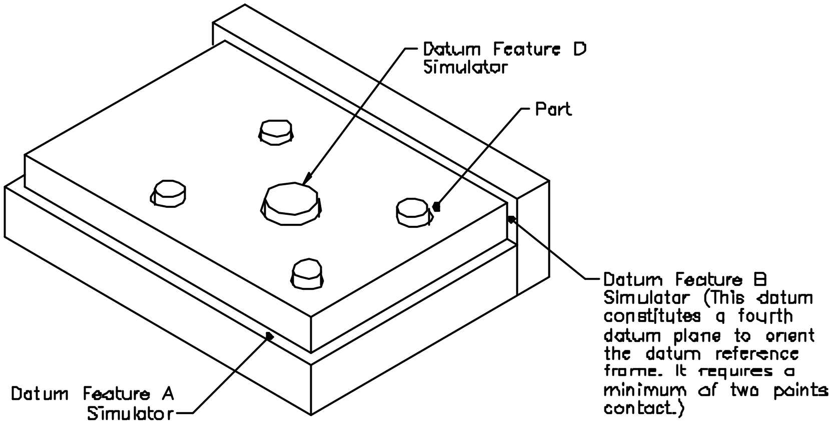


Figure 19-12 Gage for verifying four-hole pattern in Fig. 19-10

been properly maintained to the real surface from which datum B is constructed on the workpiece being gaged. The datum feature simulator for B is given a perpendicularity tolerance back to datum A. The perpendicularity tolerance is 10% of the tolerance on the datum feature it is simulating. Datum feature D is also represented. Again, D is simulated by a gage pin sized to begin at the hole's virtual condition and then the gage pin is allowed to grow by 10% of the tolerance given to the D hole being represented. The gage pin D is then given a perpendicularity requirement of zero at MMC back to the primary datum. A positional tolerance is not needed for gage pin D as long as enough surface area exists for datum feature A to be properly contacted.

The four holes being gaged are then represented with four gage pins of (as required of all gage elements) sufficient height to entirely gage the holes. These gage pins are represented at the virtual condition diameter of the holes they simulate and are allowed a size tolerance of 10% of the tolerance on the size of the holes. This tolerance is all in the plus direction on the gage pin size. The gage pins are then positioned to the datum feature simulators previously described, A primary, D at MMC or RFS secondary, and B tertiary (tertiary datum feature/fourth datum plane used to orient the two planes that cross at the axis of datum D).

19.4.6 Secondary and Tertiary Datum Features of Size

In Fig. 19-13, the position of two holes is established by datums A, B, and C (see gage in Fig. 19-14). Once this has been done, the two holes are used as secondary and tertiary datum features (see gage in Fig. 19-15) from

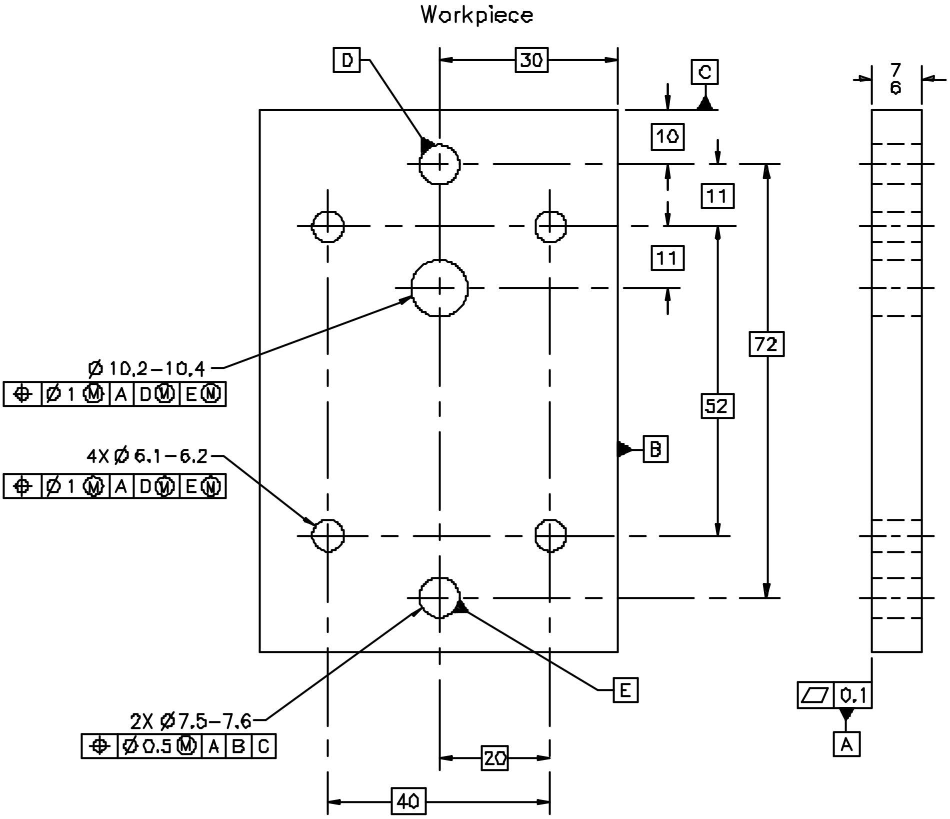
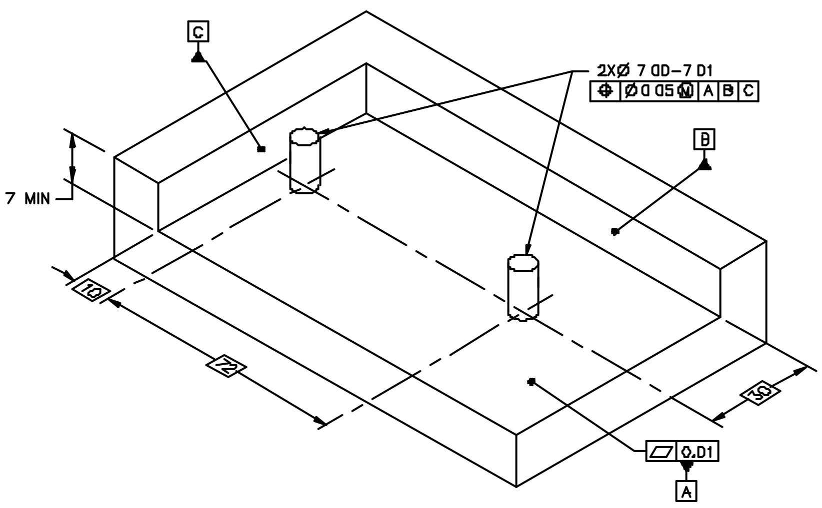


Figure 19-13 Secondary and tertiary datum features of size

Gage



Workpiece Applied to Gage

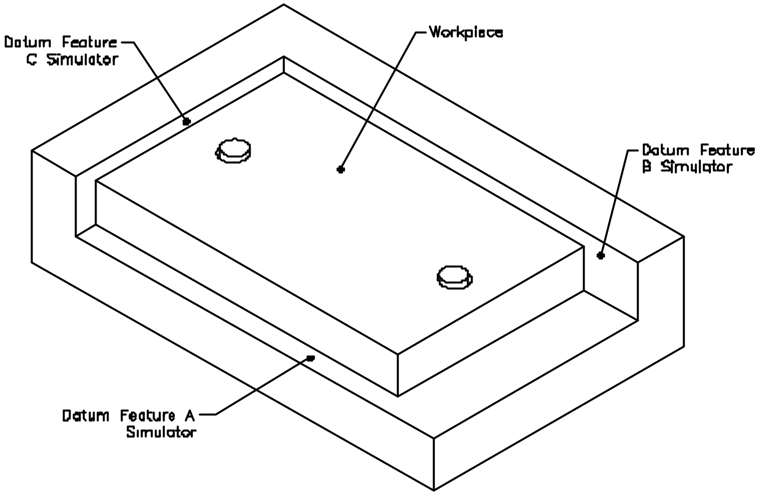


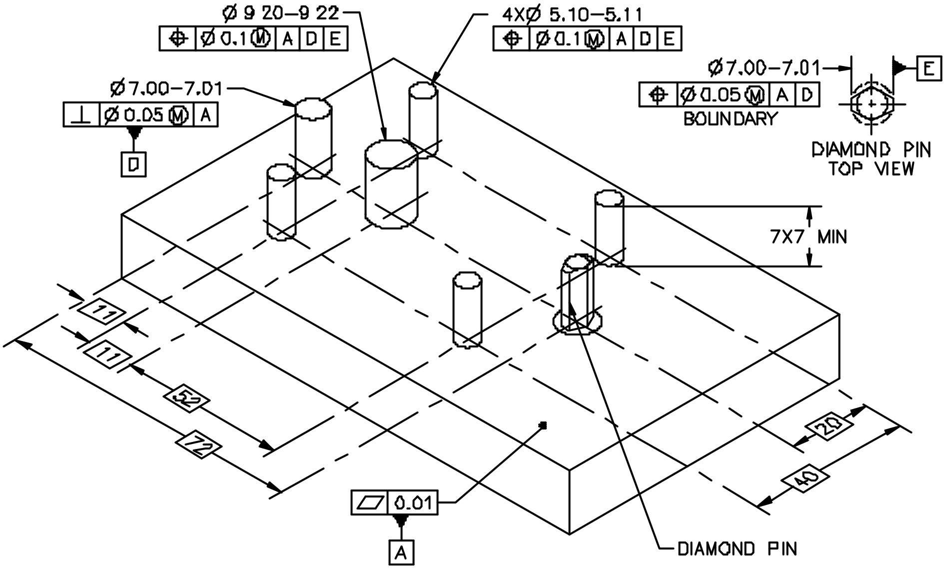
Figure 19-14 Gage for verifying datum features D and E in Fig. 19-13

which to measure the four 6.1-6.2 holes and the one 10.2-10.4 hole. Since datum feature of size D is used as secondary, it establishes the location of the five holes in both the X and the Y directions. Datum feature of size E is used as an angular orientation datum only. This means that the datum feature simulator on the gage for D is a cylindrical pin made at the virtual condition of the hole it represents (sometimes referred to as a four-way locator). Datum feature E, however, is represented by a width only (sometimes referred to as a two-way locator). Datum feature E is like a cylinder made at the virtual condition of the hole it simulates, but is cut away in the direction that locates it from datum feature D. This is to prevent it from acting as a location datum but rather as only a pattern rotation datum.

This use of datum feature simulators in Fig. 19-15 is common. Datum feature simulator E is a tertiary datum feature of size and is represented as an angular orientation datum (a two way locator) with a diamond shaped (or cut-down cylindrical) pin. However, it is not representative of other types of datum feature simulation. Datum features are normally represented by datum feature simulators that have the same shape as they do; for example planar datum features represented by planar simulators, cylindrical datum features represented by cylindrical simulators, and slot/tab/width datum features represented by datum feature simulators of the same configuration.

If datum features D and E had been used as a compound datum (D-E) with both D and E referenced at MMC, D would not have taken precedence over E. Hence, being equal, both would have been used to

Gage



Workpiece Applied to Gage

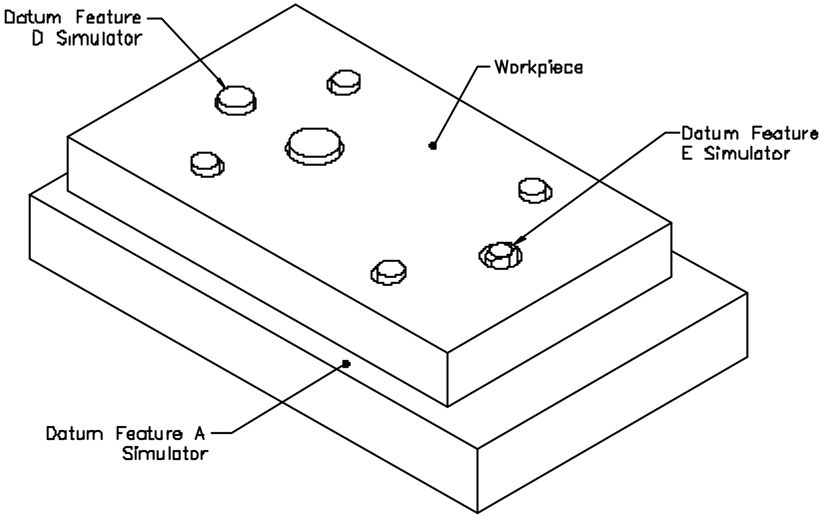


Figure 19-15 Gage for verifying five holes in Fig. 19-13

orient and locate the five holes referred to them as though they were a pattern datum consisting of the two holes. In this circumstance, the gage (as shown in Fig. 19-15) would have represented both D and E with cylindrical pins made at the virtual condition of the holes they represent. Both D and E would be considered four-way locators.

19.5 Push Pin vs. Fixed Pin Gaging

Although the examples used in this section use fixed pin gages, some thought should go toward the use of push pin gages. With push pin gages, the workpiece is first oriented and located on the gage's datum feature simulators. Then the gage pins are pushed through holes in the gage and into the holes on the workpiece. This allows the user of the gage to be certain the appropriate type of contact exists between the gage's datum feature simulators and the datum features on the workpiece being gaged. Push pin gages also provide a better view of which features in a pattern under test are within tolerance and which are out of tolerance. The holes that receive their gage pins are obviously within their geometric tolerance and the holes that are not able to receive their gage pins have violated their geometric tolerance. This information should be helpful to improve the manufacture of subsequent parts.

It must be considered that with a push pin – type gage design, gage tolerances are used in a manner that allow the gage pin to easily enter and exit the gage hole with a minimum of airspace. Gage holes that are to receive push pin gage elements should be given geometric tolerances that use a projected tolerance zone that is a minimum height of the maximum depth of the hole being gaged (since the gage hole gives orientation to the gage pin and is likely to exaggerate the orientation error of the gage hole over the height of the gage pin). The gage hole should be treated as though it is a gage pin when calculating its virtual condition. The projected geometric tolerance zone diameter is added to the maximum material condition of the gage push pin diameter to determine the virtual condition of the gage pin when pushed into the gage hole. In Absolute Tolerancing, this gage pin virtual condition boundary may be no smaller than the virtual condition of the hole on the workpiece being gaged.

19.6 Conclusion

Receiver gaging provides a level of functional reliability unsurpassed by other measurement methods. Instead of verifying compliance with a theoretical tolerance zone, it transfers that tolerance to the controlled feature's surfaces and creates an understandable physical boundary. This boundary acts as a confinement for the surfaces of the part. It assures one that if the boundary is not violated, the part features will fit into assemblies. ASME Y14.5M-1994 (the Dimensioning and Tolerancing standard) and the ASME Y14.5.1M-1994 (the standard on Mathematical Principles of Dimensioning and Tolerancing) both state that occasionally a conflict occurs between tolerance zone verification and boundary verification. They also state that in these instances, the boundary method is used for final acceptance or rejection.

19.7 References

1. American National Standards Committee B4. 1981. ANSI B4.4M-1981, Inspection of Workpieces. New York, New York: The American Society of Mechanical Engineers.
2. Meadows, James D. 1995. *Geometric Dimensioning and Tolerancing.* New York, New York: Marcel Dekker.
3. Meadows, James D. 1998. *Measurement of Geometric Tolerances in Manufacturing.* New York, New York: Marcel Dekker.
4. Meadows, James D. 1997. *Geometric Dimensioning and Tolerancing Workbook and Answerbook*. New York, New York: Marcel Dekker.
5. The American Society of Mechanical Engineers. 1995. *ASME Y14.5M-1994, Dimensioning and Tolerancing.* New York, New York: The American Society of Mechanical Engineers.