Engineering Design I: Methods and Skills
Topic Readings

Jun Hong Park and Steven H. Collins

December 5, 2015

Chapter 11

Engineering Drawings

An engineering drawing is a representation of an engineered product used to clearly define all features and requirements. Good engineering drawings must be clear, organized and thorough. In this reading, we will learn where engineering drawings are used, what their basic elements are, and how to generate a drawing with geometric dimensioning and tolerancing.

11.1 How are Engineering drawings used?

Documenting a Product's Life: Engineered products are often designed and revised by multiple people over long periods of time. A CAD model can be very useful for visually understanding the product, but it is not very good at documenting changes and highlighting important features. With engineering drawings, you can denote important features and document revisions and changes made in the revision.

Manufacturing: Engineering drawings are still the primary tool machinists and technicians use to understand the geometry of a product and its important design features. Depending on how you dimension and tolerance your drawing, the manufacturing method can vary significantly, effecting the final cost of production.

Quality Inspection: No part will ever be a perfect match to the CAD model. Parts will have variation in geometries and dimensions based on material and manufacturing methods. Proper engineering drawings give inspectors ranges of acceptable dimensions and geometries and any other important information.

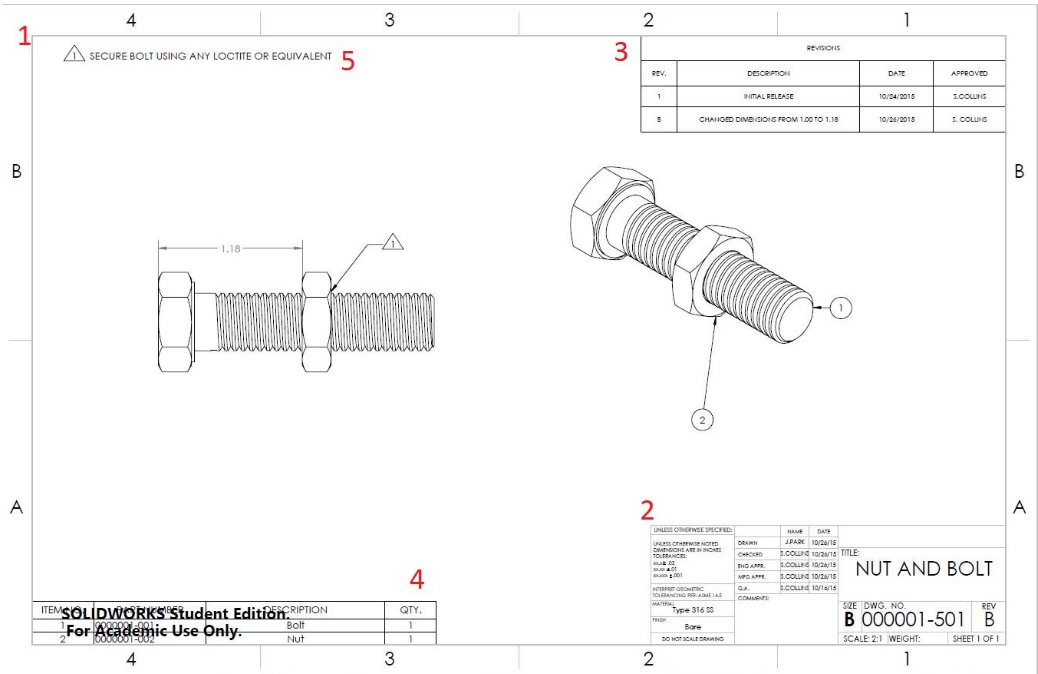


Figure 11.1: Primary elements of an engineering drawing. See Section 11.2.1 for an explanation of each.

11.2 Elements of a Drawing

Most engineering companies will have a unique template, but there are some engineering standards set by the International Organization for Standardization (ISO). These standards are adopted by most if not all engineering firms, and cover most important aspects of engineering drawings. Some symbols, footnotes and aspects of the layout style can vary, but in this reading we will go over the most widely used conventions and common elements.

11.2.1 Basic Layouts & Notes

The primary elements of an engineering drawing (Figure 11.1) are:

1. *Grid System:* Drawings are designed to communicate an idea, and a convenient way to locate specific parts of the drawing is to have a grid. For example, in this drawing, the dimension of 1.18” will be in grid B-4

2. *Title Block:* Title block holds general information like the part number, description, author and approvers. It also contains general information that

can be important when reading and interpreting the document, such as general tolerances, units, and scales. Often legal disclaimers are included here.

3. *Revision Block:* The revision block contains number of this revision and a brief description of changes made for said revision. Some revision blocks also include the location of the change in the drawing by specifying page number and grid location. It is important to note every revision.

4. *Bill of Materials:* The bill of materials (BOM) contains all parts included in an assembly. In a part drawing, the BOM will not be present. As seen on the example above, there are bubbles (section A2) with numbers in them. These numbers correspond to BOM lines. The BOM should at least have part numbers, descriptions and quantities used in the assembly. For complex assemblies, you can find BOM's with multiple sub-assemblies.

5. *Notes:* Notes are added to drawings to provide additional information that cannot be represented with pictures. In this case, the drawing has a flag note, which refers to a specific location of the drawing. A general note (note without symbols) applies to the entire drawing. Notes can specify material, instructions for technicians and inspectors, finish, or assembly methods.

11.2.2 Drawing Views

Drawings are two-dimensional, and therefore need multiple views in order to display all features of a three-dimensional model. We will go over some commonly used views. Note that a drawing will have as many views as necessary to show the entire geometry.

Front, Top and Side Views

Front, Top and Side views are orthographic projections of an object onto three planes, typically the three principal planes (the x-y plane, y-z plane and the x-z plane). The top view should appear above front view and the right view should appear to the right of the front view (Figure 11.2). When laying the views out on the drawing, make sure the views line up with each other.

Isometric View

An isometric view is a projection of the three-dimensional object onto a plane that is at equal angles to the principal coordinate system of the part. Isometric views are helpful for showing the general shape of the object, and provide a good visual reference. Isometric views should not be used for dimensioning, because it can be difficult to identify which features are being referenced.

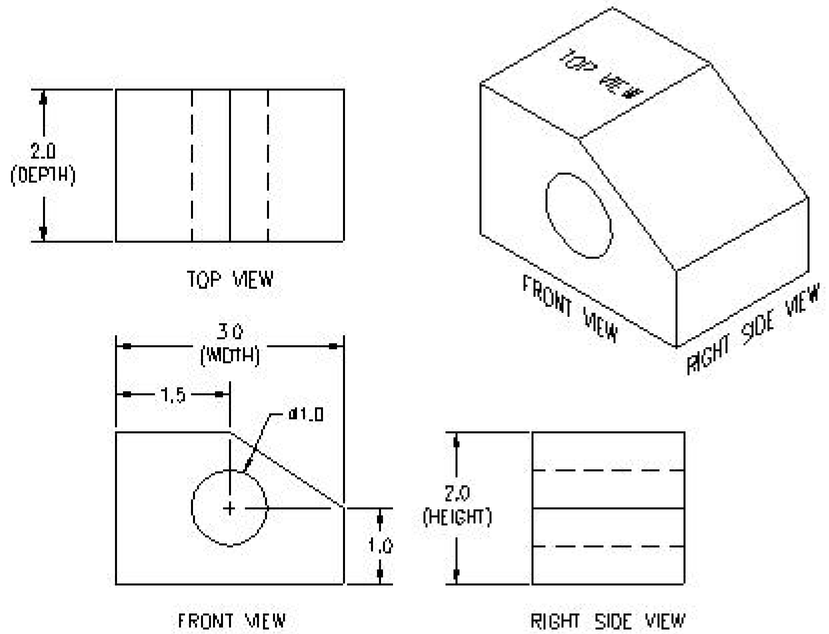


Figure 11.2: The most common views in an engineering drawing are front, top and side orthographic views and an isometric view.

Cross-Sectional View

Cross-Sectional Views are used to display internal features that are difficult to see in other views. They are also a useful way to identify and dimension internal features. Cross-sectional views have a cut line on another view, and will have a projection of the object if it were to be cut along the plane perpendicular to the referenced view, and intersecting the cut line (Figure 11.3.*A*).

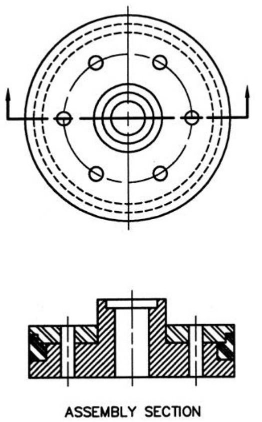
Auxiliary View

Auxiliary views are used to show the true shape of a feature that is not parallel to any principal plane. Auxiliary views are parallel to the primary plane of the feature of interest, and sometimes do not show the entire object (Figure 11.3.*B*).

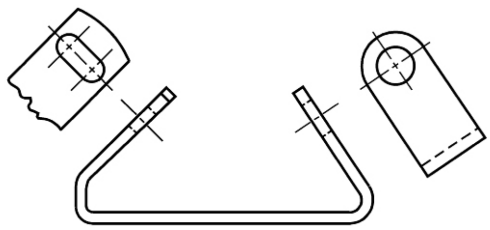
Exploded View

Exploded views show the individual parts of an assembly slightly separated from each other, usually following a consistent pattern. In the example provided, all components are moved only along the primary axis of rotation of the assembly (Figure 11.3.*C*). Exploded views can be helpful in understanding how things fit together.

A. Cross-Sectional View



B. Auxiliary Views



C. Exploded View

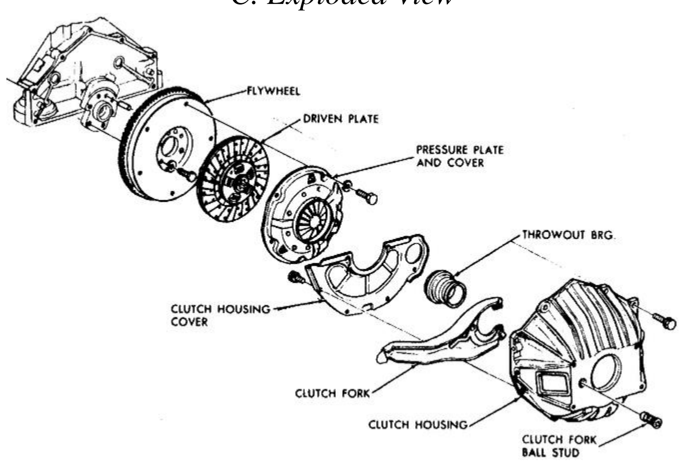


Figure 11.3: Examples of specialty views.

Line		Description	General Application	
A		Continuous thick	A1 A2	Visible outlines. Visible edges.
B		Continuous thin (straight or curved)	B1 B2 B3 B4 B5 B6 B7	Imaginary lines of intersection. Dimension lines. Projection lines. Leader lines. Hatching lines. Outlines of revolved sections in place. Short centre lines
C		Continuous thin free hand	C1	Limits of partial or interrupted views and sections, If the limit is not a chain thin.
D		Continuous thin (straight) with zigzags	D1	Long break line
E		Dashed thick	E1 E2	Hidden outlines. Hidden edges.
F		Dashed thin	F1 F2	Hidden outlines. Hidden edges.
G		Chain thin	G1 G2 G3	Center lines. Lines of symmetry. Trajectories
H		Chain thin, thick at ends and changes of direction	H1	Cutting planes.
J		Chain thick	J1	Indication of lines or surfaces to which a special requirement applies
K		Chain thin double dashed	K1 K1 K3 K4 K5	Outlines of adjacent parts. Alternative or extreme position of movable parts. Centroidal lines. Initial outlines prior to forming Parts situated in front of the cutting plane

Figure 11.4: Line types used in engineering drawings and their meaning.

11.2.3 Lines Used in Engineering Drawings

Line weight, dash and other properties are used to indicate the meaning of a line in engineering drawings. The most common line type is ‘continuous thick’, used to depict visible part edges (corners or places where a curve becomes tangent to the line of view). Another common line type is ‘dashed thick’, used to depict hidden edges (part edges behind material from the current view). Dimensioning lines are typically ‘continuous thin’. Additional line types are explained in Figure 11.4.

11.3 GD&T Basics

Geometric Dimensioning and Tolerancing (GD&T) refers to a set of symbols commonly used in engineering drawings to define allowable deviations in geometry. The language of GD&T consists of dimensions, tolerances, symbols, definitions, and conventions that can be used to precisely communicate functional requirements for the location, orientation, size, and form of each feature of a design. These symbols communicate design design intent and requirements to manufacturers and quality inspectors. The current standard is ASME Y14.5-2009.

11.3.1 Dimensions

By dimensioning a feature, the author of the drawing is implying the importance of the accuracy and the acceptable tolerance. In the title block, default tolerances are often specified based on number of decimal points, e.g.: "Unless specified otherwise, x.xxx $\pm$ 0.005, x.xx $\pm$ 0.01, x.x $\pm$ 0.1". If a different tolerance is required, $\pm$ x.xxx should be added to the dimension. All specified dimensions imply importance, and will be assessed during quality inspection. When designing a part, think carefully about which dimensions are most important and how much error is acceptable. If required tolerances are tight, try to make design changes that improve robustness against manufacturing errors and therefore increase the acceptable error and reduce part cost. After working through this process iteratively, be sure to note the acceptable tolerances in the engineering drawing of the part.

Reference dimensions are used to specify overall dimension (good for picking starting material) or dimensions driven by other features (useful for helping the reader to understand the intent of the non-reference dimensions). Reference dimensions are identified by putting parentheses around the dimension value, such as: "(3.450)". Reference dimensions are considered secondary, and will not be checked by quality inspectors.

11.3.2 Datums

Datum symbols define the locations which will be used as a reference for other drawing dimensions and constraints. All GD&T symbols except for the set defining form (Straightness, Flatness, Circularity, Cylindricity) can or must use datums. A datum can be plane, axis or point that is theoretically exact. Because datums are theoretically exact it is good to use datums on surfaces, axes or points that can be realistically measured. Keep in mind that no real surface or edge can be exact. Datums are listed in alphabetical order by importance, so when manufacturing a part datum A will be set first, then datum B and so on.

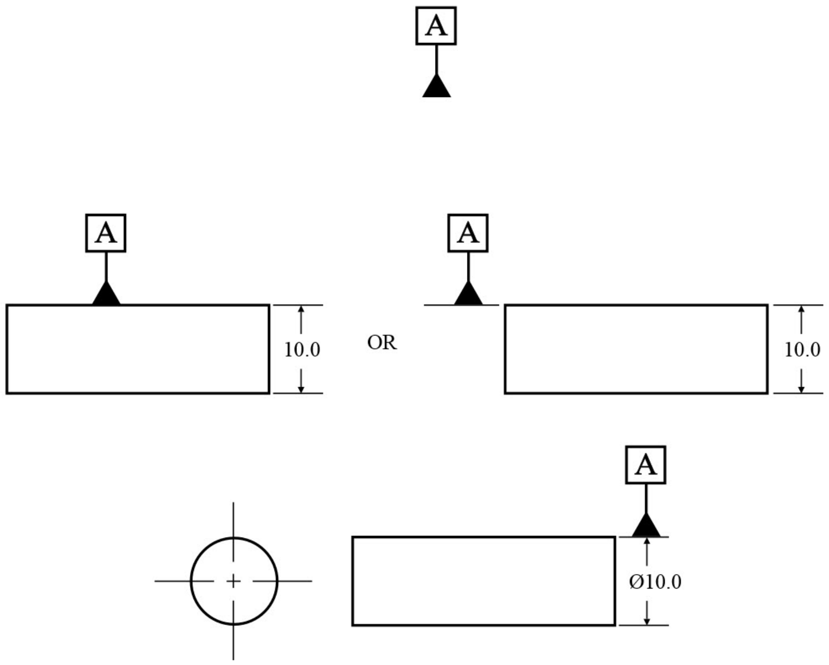


Figure 11.5: Examples of datums. *Top:* The datum symbol. *Middle:* Two ways of defining a datum surface. *Bottom:* Cylindrical axis datums are defined by placing the datum symbol on the leader of the diameter dimension.

11.3.3 Symbols

GD&T symbols are used when dimensions alone cannot describe the required geometry or to simplify the drawing. Generally, the GD&T symbols are shown directly on the related dimension or pointing to the feature being referenced. Convention is to display a box with the symbol inside, then a box with the tolerance, and then datums it references, if applicable. Because there are many possible symbols, we will go over only the most commonly used ones. More information can be found at: `http://www.gdandtbasics.com/` or in reference texts such as "GeoTol Pocket Guide" by Neumann & Neumann.

11.3.4 Form Symbols

Symbols that control form indicate acceptable tolerances for the shape of a part. The indicated feature generally needs to be within two imaginary offset features. The two most commonly used symbols for form are flatness and cylindricity. Form does not reference any datum.

Flatness

By adding the flatness symbol, the engineer is requiring a surface to be flat within specified tolerance. This means that the irregularity of the surface has to be within two parallel surfaces that are separated by the specified tolerance. The largest difference (highest peak and lowest valley) must be within the tolerance or the part will be rejected. Usually when setting a surface as a datum, the surface will also have a flatness requirement.

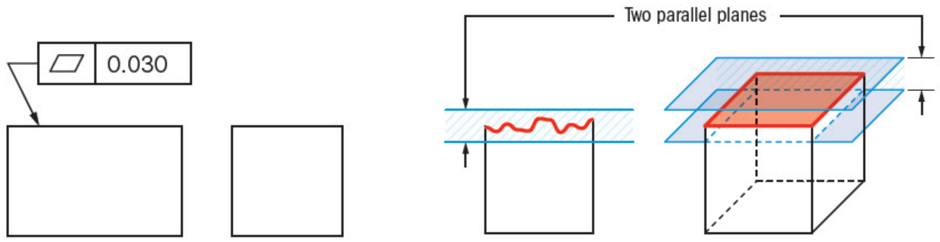


Figure 11.6: The flatness symbol and its use and meaning.

Cylindricity

By adding the cylindricity symbol, the engineer is requiring a surface to be cylindrical. This means that the irregularity of the surface has to be within two concentric cylinders that are separated by the specified tolerance. The largest difference must be within the tolerance.

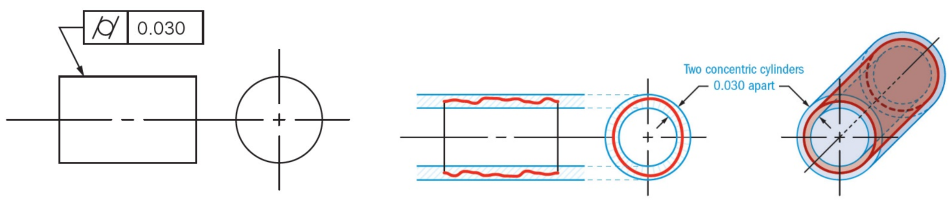


Figure 11.7: The cylindricity symbol and its use and meaning.

11.3.5 Profile Symbols

Profile requirements are used for splines and curved surfaces to which flat or cylindrical form requirements cannot be applied. Due to the difficulty of checking curved surfaces, datums can be useful in defining the ideal profile, but datums are optional. There are two symbols in this category, line profile and surface profile, of which surface profile is more commonly used.

Surface Profile

As with form, the surface profile symbol indicates that a surface must be true to the model within a tolerance. If no datum is present, the method of checking is similar to form. Most of the time, a surface profile callout will reference datums in order to create the ideal profile to check against.

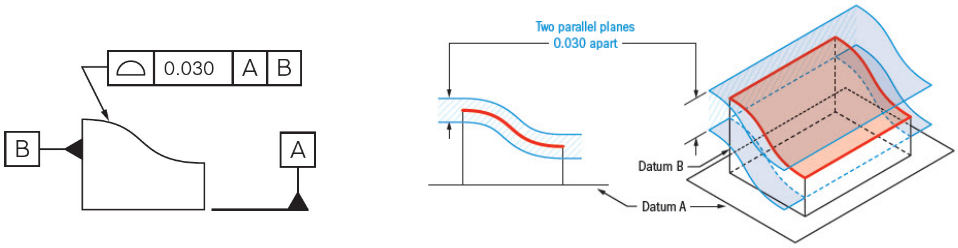


Figure 11.8: The surface profile symbol and its use and meaning.

11.3.6 Orientation Symbols

The two most commonly used orientation symbols are perpendicularity and parallelism. Both require a surface datum. As with other symbols, orientation requirements use an ideal plane and two parallel tolerance planes.

Perpendicularity

Perpendicularity is used very often, and just like other symbols, the tolerance zones are 2 parallel planes that are equidistance away from the ideal parallel plane. Perpendicularity can also be applied to an axis, and often used for hole alignments. For axis perpendicularity, a tolerance cylinder is drawn around the ideal axis that is perpendicular to the datum surface.

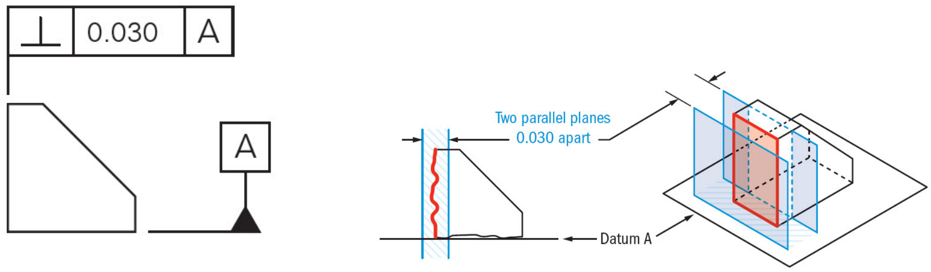


Figure 11.9: The perpendicularity symbol and its use and meaning.

Parallelism

Parallelism is very intuitive, in a sense that that the tolerance surfaces will be parallel to the datum surface. Sometime, an inspector gets a large set of coordinates of the datum surface and the called out surface, and individually check the parallelism, to get even more accurate data, but if the datum surface is out of tolerance, this method will not work.

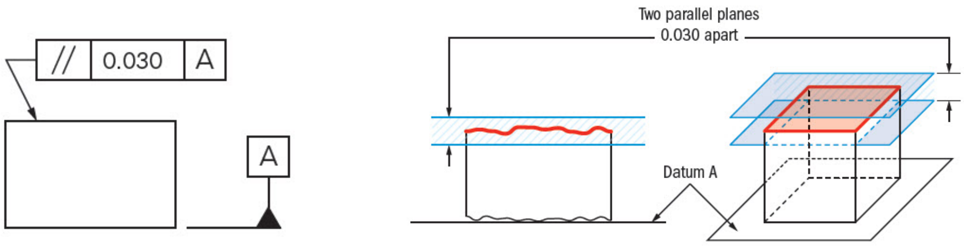


Figure 11.10: The parallelism symbol and its use and meaning.

11.3.7 Location Symbols

Location requirements are important, but can be complicated to implement because they require more datums and dimensions to be defined and incorporated.

Position (True Position)

True position is one of the most useful GD&T symbol but also the most complicated. Unlike other symbols, True Position uses relative distance from the “true position” to actual position. True position is commonly used for finding centers of holes, and typically requires two datums. Because the tolerance is set by the radius around the true position, the equation shown below must be used to see if the true position is met.

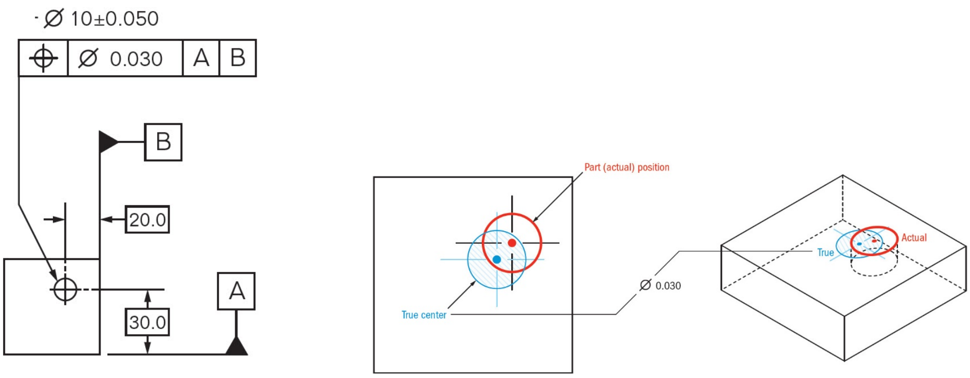


Figure 11.11: The position symbol. The top diameter callout defines the diameter and diameter tolerance for the hole. The boxed elements on the dimension leader are, from left to right, the position symbol, the hole axis position tolerance (the diameter of the circle defining the tolerable region), and the datums of the first and second reference edges. Note that the dimensions defining hole location are also boxed, as they are part of the position requirement.

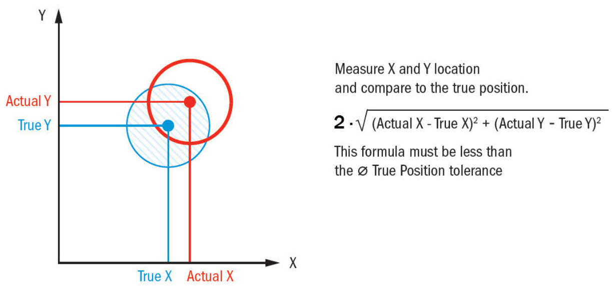


Figure 11.12: Evaluating the position tolerance.

Concentricity

Concentricity is checked by making a tolerance cylinder around the ideal axis that is set by the datum axis. The datum axis needs to be set by a cylindrical feature with an axis. The tolerance appearing to the right of the concentricity symbol defines the diameter of the cylindrical tolerance zone in which the feature axis can be located.

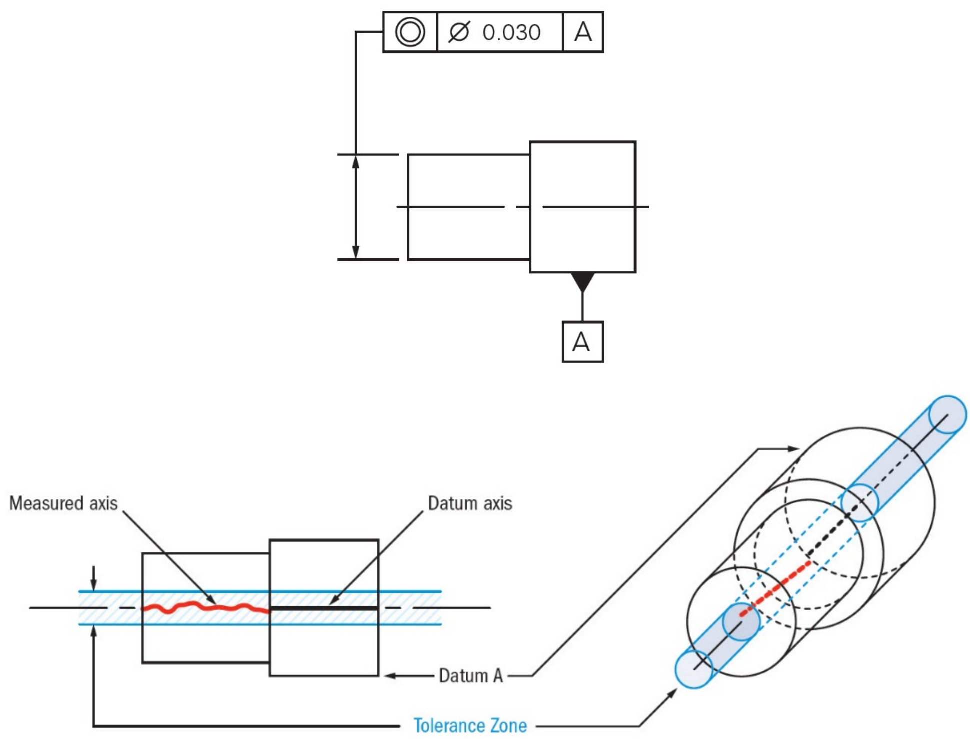


Figure 11.13: The concentricity symbol and its use and meaning.

11.3.8 Runout Symbols

Runout refers to the variation in the radius of a cylindrical feature as it is rotated about a datum axis. This is similar to cylindricity, with the difference being that the datum axis is not the same as the feature axis. There are two types of runout, circular and total, of which total runout is the more common and more restrictive.

Total Runout

Total runout is useful when the entire rotating surface is critical to the part's function. The tolerance zone is in the shape of a tube with the wall thickness of the tube specified to the right of the total runout symbol. Usually, inspection will be performed by running an indicator on a surface while rotating the part within a v-block. For total runout, the indicator will be slowly moved axially, so as to sample a large portion of the outer surface of the feature.

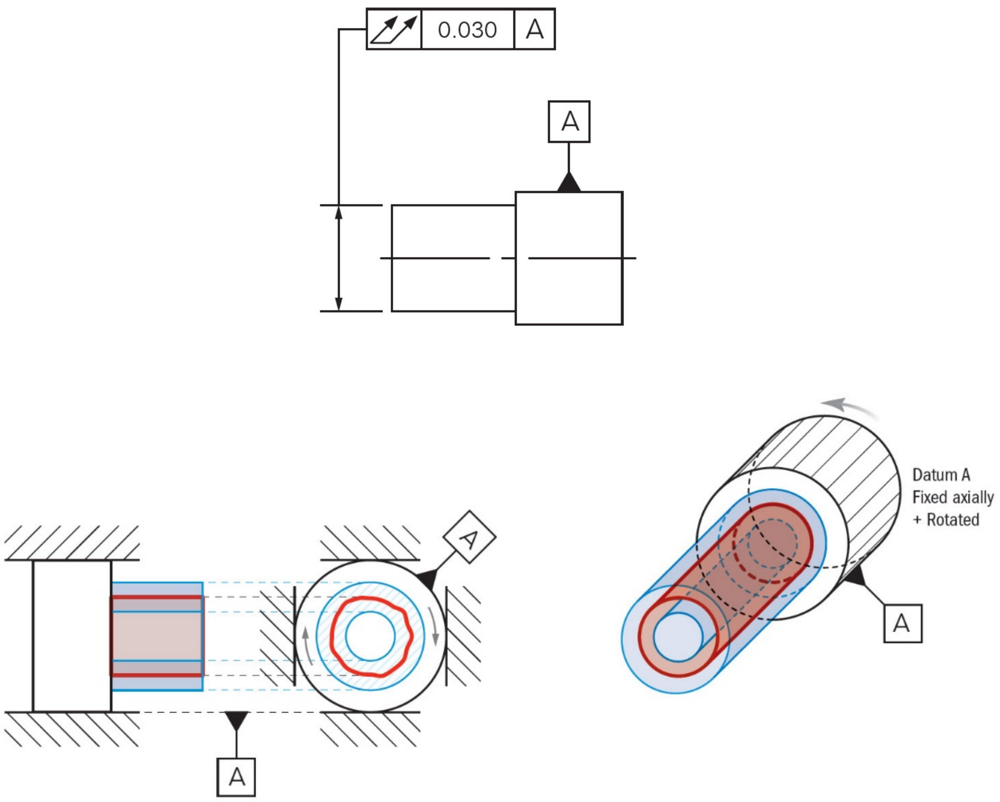


Figure 11.14: The total runout symbol and its use and meaning.

11.4 Acknowledgments

Thanks to Jun Park for help in drafting this chapter.

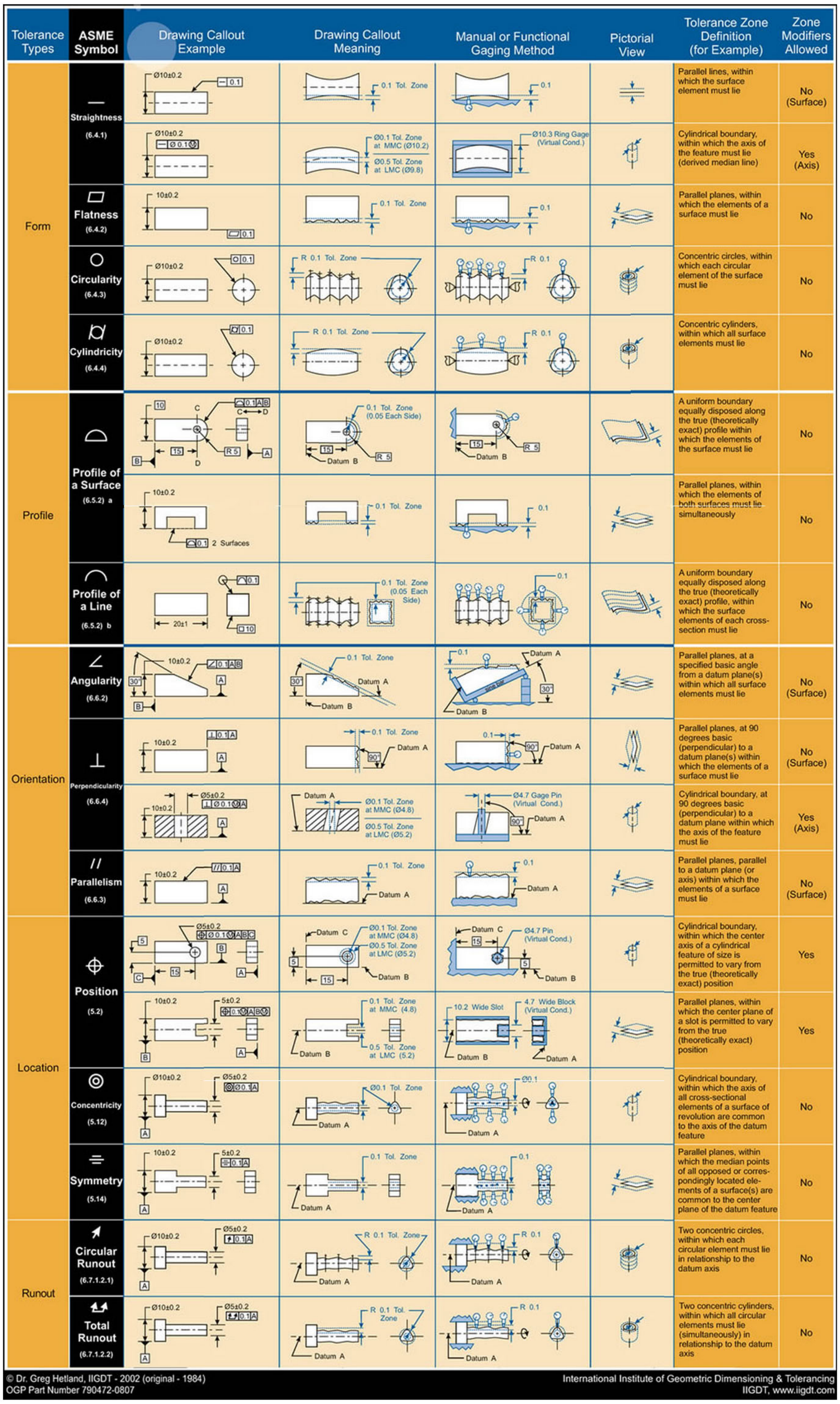


Figure 11.15: Table of common GD&T symbols with abbreviated meaning and use. From http://opensourceecology.org/w/images/0/03/Gdt_diagram.jpg)