Engineering Problems in Dimensions and Tolerances

By W. W. WERRING

DIMENSIONAL UNITS

The basic unit in most considerations of dimensions in the United States is the inch. The value of the inch is so important that many companies including the Bell System maintain in their measurement laboratories a standard yard bar calibrated against the standard at the National Bureau of Standards. In spite of this it is an interesting and curious fact that though all have been much concerned over the legal value of the dollar there has been little interest even among engineers in the exact legal value of the inch. Actually there is no single answer to so simple a question as "What is an inch?" In fact, we have changed from a British inch and our own legal meter, to our inch and the International meter and now through action of the American Standards Association we are actually using an inch based on conversion from the International meter which is neither our own legal inch or the British legal inch—and the British are using it too. Table I shows this history of the legal inch in the United States.

It will be seen that under the present status there exists a difference of two parts in a million between the legal inch and the inch used in the dimensional work of industry. This difference is more theoretical than real in small dimensions and industrial use. The bill before Congress, sponsored by the Bureau of Standards is intended to eliminate this as well as any possible ambiguity in the U. S. inch.

DECIMAL DIMENSIONING

In subdividing the inch the modern trend in industry is toward the use of decimals instead of the older common fractions although fractions continue to be used, especially for dimensions of certain materials such as iron pipe, lumber, phenol fiber. In fact even a special decimal system based on using only the tenths and fiftieths of an inch is being considerably discussed by general industry. This system would use a scale on which the smallest division is $\frac{1}{50}''$ or $.020''$ instead of $\frac{1}{64}'' = .0156''$. It is in use by the Ford Motor Company and the values shown in Table II are some of those used in place of common fractions. The decimal equivalents of these common fractions are also shown rounded to 3 decimal places in accordance with American Standard Rules for Rounding off Numerical Values Z25.1-1940.

In the Ford system one and two-digit decimals carry the general tolerance of ± .010″. When greater accuracy is required three-place decimals are used to express a minimum and a maximum value.

The adoption of decimal dimensioning for all drawings prepared at Bell Telephone Laboratories is being actively considered. However, adoption of decimal dimensioning would not of itself result in any changes in our system for establishing tolerance values.

TABLE I

History of United States Dimensional Standards

Year	Action	Resulting Dimensional Relationships
1830–36	Adoption for Customs Service and for distribution to individual states of standards intended to be the English yard based on a certain portion of an 82 inch bar imported in 1813. The portion selected was supposed to be identical with the English yard.	
1856	Official copy of new British Imperial Yard accepted as standard	International Meter = 39.370147 British Inch
1866	Congress declared metric units lawful and established legal equivalents	Legal Meter in U. S. = 39.37 British Inch
1893	Mendenhall Order set up International meter as the fundamental standard	International Meter = 39.37 U. S. Inch
1933	American Standards Association (Representing Industry) adopts 1 inch = 2.54 centimeters	International Meter = 39.370078 U. S. Inch
1937–41	Bill before Congress but held in committee for amendments	International Meter = 39.370078 U. S. Inch

TABLE II

Examples of Ford Decimals Compared to Common Fractions

Ford Decimal	Common Fraction	Decimal of Existing Common Fraction	American Standard Decimal Equivalents (3 Place)
.02	1/64	.015625	.016
.03	1/32	.03125	.031
.05	3/64	.046875	.047
.06	1/16	.0625	.062
.08	5/64	.078125	.078
.3	7/32	.21875	.219
.46	15/32	.46875	.469

Raw Material Sizes

In contrast to this continued trend toward simplification and rationalization of our systems of dimensional units raw material supply is still complicated by a multitude of obsolete systems of gauge sizes in every day use.

Many in industry have probably grown used to the standard gauges in particular fields but though gauge numbers were undoubtedly initiated as a simplified identification the variety of gauges and the variety of names for the same gauge now merely increases confusion. Sheet metals are handled in terms of a number of gauges such as B&S gauge, U. S. standard gauge and BWG gauge; and sheet soft rubber is even designated in decimals of $\frac{1}{64}$ such as $\frac{4.3}{64}''$. It has become good practice to specify sizes by decimal dimension values and not by gauge numbers and holes by actual decimal size rather than by drill numbers. The actual sizes used, however, are determined in many cases by the values corresponding to old gauge numbers long used commercially, though in large running items mills will and do manufacture to any specified decimal size. For some time it has been the practice of material manufacturers and other large industries thus to discontinue the use of gauge numbers though still using the decimal values of gauge sizes.

There is now under way an effort, organized under committee B32 of the American Standards Association, to eliminate the old wire and sheet metal gauge systems entirely and set up a rational series of American standard thicknesses for all metal sheets and preferred diameters for wire, and insure availability in these sizes. The basic conception of a rational series of sizes is that a uniform degree of choice should be presented between successive sizes. Therefore each size should differ from the next by a fixed percentage. The series should therefore be geometric. A variety of geometric series could be used but in order to permit extending the series indefinitely by shifting the decimal point, the particular series based on the root of 10 has been established internationally as the Preferred Numbers Series for standard sizes. The 5 series is one having 5 numbers between 1 and 10 (or between 10 and 100) and is produced by using as the multiplier the fifth root of 10; the 10 series is produced by multiplying by the 10th root of 10; the 20 series by multiplying by the 20th root of 10 etc. The complete Preferred Numbers Series is explained and listed in various forms in American Standard Z17.1-1936.

The subcommittee working on the sheet metal sizes has recently issued a proposed American Standard of preferred thicknesses for all uncoated flat metals thinner than .250″. These thicknesses are all decimals based on the 20 series of preferred numbers rounded in the standard manner to 3 decimal places. The Preferred Numbers and the proposed thicknesses are shown by Table III. It happens that this series closely approximates the Brown and Sharp gauge used in the nonferrous metals which simplifies that portion of the changeover. If this proposed American Standard is generally approved, as now appears most promising, we will be able to choose thicknesses of any metal interchangeably without the restrictions of ancient gauge sizes es-

TABLE III

Decimal Series of Preferred Numbers 10–100			Proposed Preferred American Standard Thicknesses		
5 Series $\sqrt[5]{10} = 1.6$	10 Series $\sqrt[10]{10} = 1.25$	20 Series $\sqrt[20]{10} = 1.12$	Under .010	.010 to .100	.1120 to .250
10	10	10		.010	
		11.2		.011	.112
	12.5	12.5		.012	.125
		14		.014	.140
16	16	16		.016	.160
		18		.018	.180
	20	20		.020	.200
		22.4		.022	.224
25	25	25		.025	
		28		.028	
	31.5	31.5		.032	
		35.5		.036	
40	40	40	.004	.040	
		45		.045	
	50	50	.005	.050	
		56		.056	
63	63	63	.006	.063	
		71	.007	.071	
	80	80	.008	.080	
		90	.009	.090	
100	100	100		.100	

tablished for reasons which were possibly good and sufficient but which certainly have long been forgotten. Meanwhile, another subcommittee is investigating the possibility of applying a similar series to the diameters of wire. Probably diameters to 4 decimal places will be required.

DIMENSIONAL TOLERANCES

Part Tolerances

Regardless of the dimension decided upon in a design it is obvious that it cannot be regularly manufactured to the exact size. Certain manufacturing variations or tolerances must be expected and these introduce a large share of our dimensional problems.

The usual statement on tolerances is that the larger the tolerance allowed the cheaper the part is to manufacture and, therefore, the tolerance specified should be the widest that will permit functioning. However, this is generally true only of overall tolerances which define the manufacturing methods that may be used. It is true in the sense that apparatus is inexpensive to manufacture if it can be so designed that its functioning is largely independent of variations in dimensions. However, such design is not usually achieved and in much apparatus fairly good overall accuracy of dimensions and fit is necessary for uniform functioning. The problem of setting tolerances then becomes one of distributing certain tolerances over various dimensions and different parts. This is a very difficult problem and in the case of any individual tolerance a larger value does not necessarily mean lower apparatus cost and may even mean the reverse.

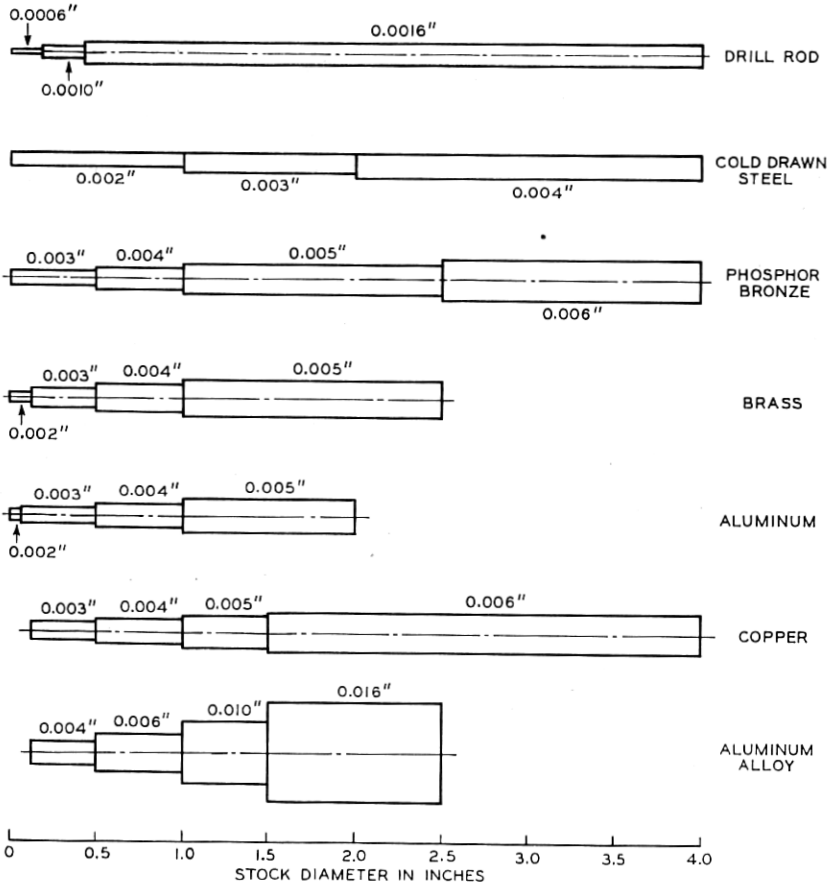

Fig. 1—Total diameter tolerances of commercial round stock

This is easily demonstrated in the case of part tolerances on dimensions which correspond to the dimensions of raw materials. Figure 1 shows the tolerances of commercial grades of round stock. If, for example, engineering requirements dictate the use of a particular material there is no gain in specifying larger tolerances than those to which it is regularly furnished and doing so may require greater accuracy in the mating part. There may even be economy in the use of higher priced material produced to closer tolerances, as for example, drill rod instead of cold drawn steel through economy in the manufacture of associated parts. Similarly manufacture of cantilever springs from sheet stock produced to closer tolerances may reduce the cost of subsequent adjustments. Therefore, when individual part tolerances are involved consideration must always be given to the size tolerances of raw materials.

The same situation exists in the case of tolerances on dimensions produced by a manufacturer's own tools. While close overall limits will require greater overall accuracy of the tools provided and greater frequency of set-ups the most economical distribution of tolerances will be that based upon the normal tolerances that can be expected from various manufacturing operations. Certain degrees of accuracy are inherent in certain types of machines and tools and allowing variations not in proportion to these values serves little if any purpose. Also there are types of combination tools and automatic machines, familiar in mass production practice with which wide tolerances are not an economy because accuracy is required for locating or nesting the part for subsequent operations. Since the distribution of tolerances involves such complex factors of manufacturing method and cost as these, it is desirable for the designing engineer to determine and to indicate unmistakably the effect of tolerances upon functioning and, where interchangeability of individual parts in service is not involved, to allow manufacturing considerations to determine the distribution of tolerances in an assembly.

It is apparent that considerable study of the requirements for functioning of the design, of available materials and the limitations of manufacturing process are required to establish the most economic balance between performance of the apparatus and the required tolerances. Consideration should be given to these tolerance factors in cooperation with manufacturing engineers in an early stage of a design problem so that they may influence the trend of design. This step may avoid the necessity for slow and costly manufacturing developments and delays in starting production. However, completely rigid adherence to the status quo of tolerances is not necessary in long range planning of major design projects. In such cases the trend of progress in materials and manufacture should be determined and anticipated. For example, some cantilever spring design requiring narrow control has been based on sheet material produced to tolerances not commercially available at the time but made so by the time it was needed for production. The extent of progress in this direction is shown by Fig. 2.

Similar progress in manufacturing technique can also be expected. For example, the development of broaching from a comparatively crude operation to the precision method it is today is recent and outstanding.

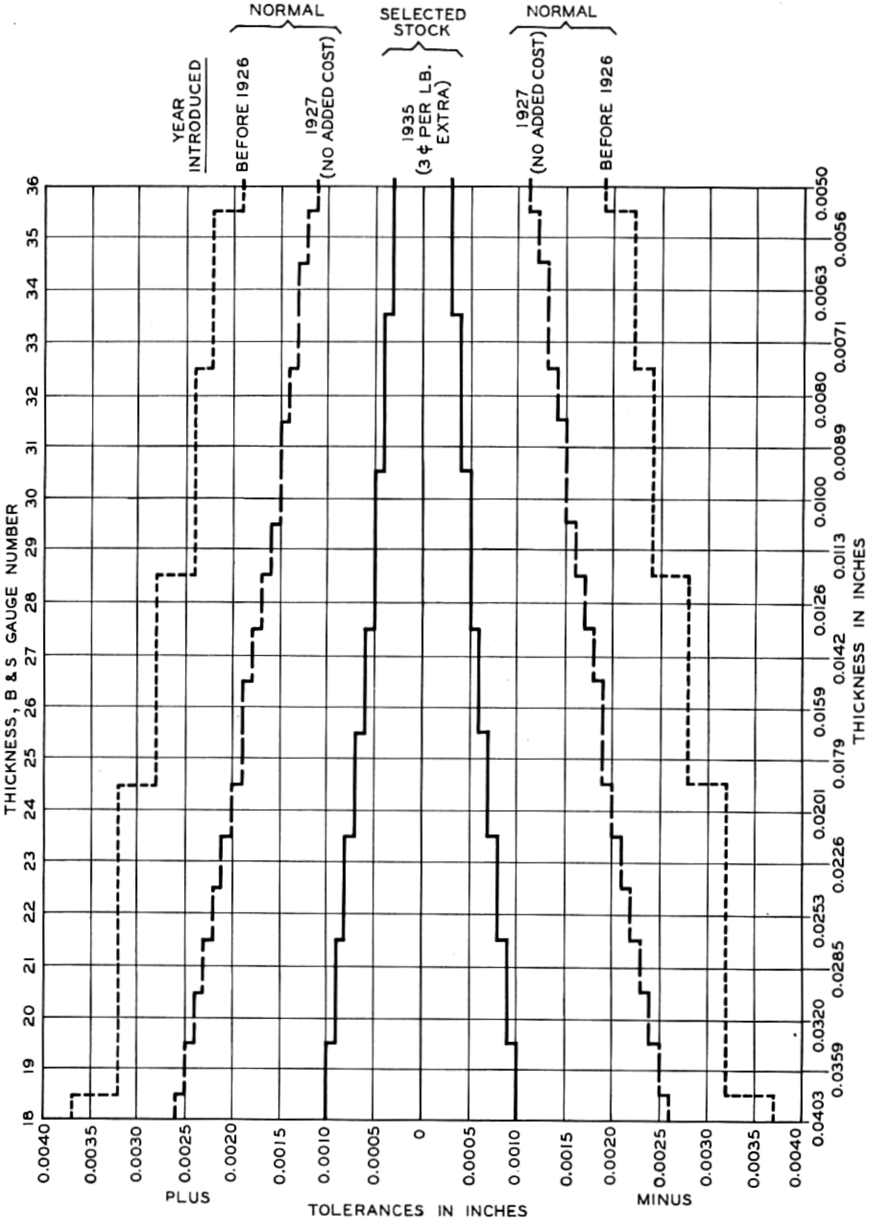

Fig. 2—Improvement in thickness tolerances for brass sheet 1926–1939

Cumulative Assembly Tolerances

Another problem in choice of tolerances is in those cases where a considerable number of parts are additively assembled into a unit as in the case of "spring pileups" used on electrical contacting apparatus such as relays and switches. These consist of considerable numbers of sheet metal springs and insulators alternating and clamped by screws. If the overall tolerance on such an assembly must be taken as the sum of the tolerances of the individual parts various courses of action are presented, the extremes of which are:

1. Very small tolerances must be maintained on the individual parts or
2. Adequate space must be provided in the apparatus for extremely large variations in the assembly.

Small tolerances on the individual parts may be extremely expensive and large space allowances and provisions in associated parts for variations in the assembly may be a serious design handicap.

However, it is recognized that there is obviously small probability that all minimum or all maximum parts will appear in any one assembly. It has been found satisfactory in certain types of such pileups to assume that the maximum dimensional variation that will actually be encountered in an assembly will not be greater than 70% of the sum of the part tolerances. A similar situation exists in many kinds of assemblies or associations of tolerances.

The statistical relationships involved in this problem are indicated by Fig. 3. The curves show the percentage of the cumulative part tolerances within which 99.7% of the assemblies may be expected to be found with different numbers of similar units in the assembly. The solid line is deduced from theoretical relationships. It assumes that the parts are all of one kind, that the parts going to assembly are controlled, of normal distribution and the limits are rationally set to represent the actual conditions. The dotted curves have been deduced from relationships which have been proposed as representing rectangular and triangular distributions of individual part tolerances. The curves may not be truly representative of specific cases because of inconsistent selection of limits or erratic distributions. However, they indicate that the 70% rule on pileups is probably on the safe side in most cases and that closer design of assembly or less restrictive tolerances and cheaper manufacture of piece-parts might be readily possible either (1) by better control, (2) by actual mixing of lots of piece parts or (3) even merely by knowledge of the actual statistical distribution of part dimensions.

The three points indicated in Fig. 3 show the results of a limited experiment in which pileups were assembled from 2083 individual insulators of $\frac{1}{32}''$ phenol fiber taken from factory stock. The establishment of curves by this type of experiment using a sufficiently large and representative sample would be practicable and would permit considerable condensation of design on a sound basis. In this particular case the parts used apparently came from only two different sheets of fiber as indicated by the distribution of thickness of the individual parts shown by Fig. 4.

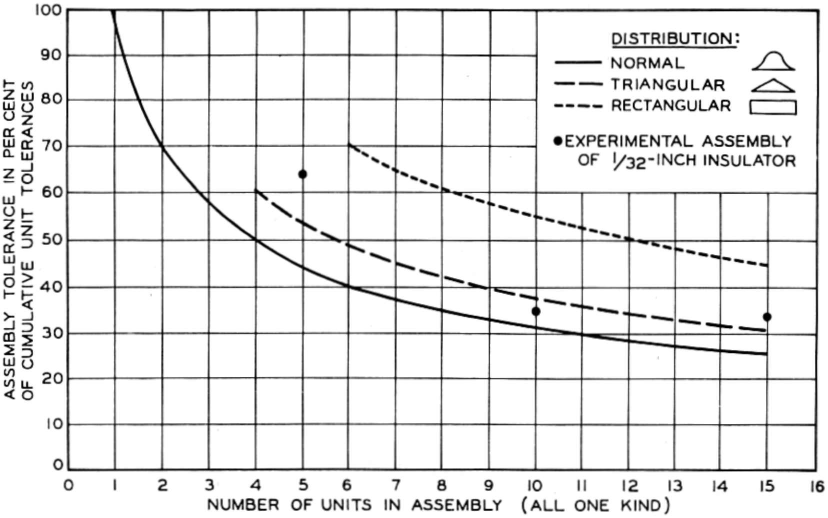

Fig. 3—Statistical relationship of overall tolerance on an assembly and the sum of the individual part tolerances

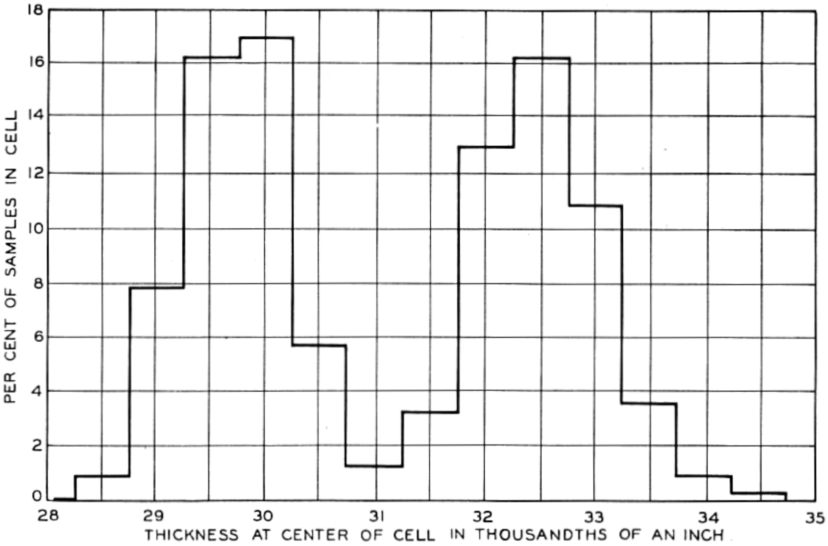

Fig. 4—Distribution of thickness in 2083 pileup insulators

Further statistical analysis of this type of situation is needed together with experimental determination of the distribution of dimensions actually encountered in specific cases.

The distribution of dimensions in a product arises from a variety of causes. One type of cause is the variations such as those in the dimensions and physical properties of raw material which may produce different product dimensions even from a particular tool. A different and more systematic type of cause is the change in the dimensions of tools as a result of wear. The practice followed in establishing tool wear allowances will therefore affect the limits and statistical distribution of part dimensions during the life of the tool. Some designers and some tool makers consider that the specification of a nominal value with plus and minus variations requires a different handling of initial tool dimensions and wear allowances than does the specification only of minimum and maximum limits for a part dimension. Equally good authority maintains that a manufacturer recognizes no difference. Establishment of standard practices in such matters is a needed step in determining the distribution of dimensions to be expected in machined parts. In the present absence of standards or of any consistent attitude on the subject it is necessary for designing and manufacturing engineers to reach an agreement in specific cases where this factor is important.

Such are the factors which determine the tolerances which can be obtained economically or which perhaps will be unavoidably encountered. It is necessary for a designer to keep informed of the interaction of these factors as his design crystallizes and he must also determine the effect of such tolerances upon functioning in order to complete a design which will function properly when assembled in quantity production.

FUNCTIONAL DIMENSIONING

Effect of Tolerances

If apparatus parts are minute or have complicated relative motions it is recognized that manufacturing drawings to the usual scale have serious limitations to their usefulness in the analysis of the effects of combinations of tolerances. In such cases designers frequently make layouts to larger scales or large scale adjustable models to investigate the effect of variations on functioning. Illustrations of this practice are numerous in the experience of most designers of small apparatus.

Even in large parts which are stationary in use the application of tolerances, in effect, establishes several possible positions for each element and may present problems similar to those involving motion. These are not easily recognized because of a curious limitation inherent in small scale drawings. This limitation is probably well known to most engineers but it is worthwhile to analyze it because it is important to be always aware of it.

This limitation is the fact that in drawings the shape of the part and the effect of all nominal dimensions are actually shown graphically whereas, it is possible to indicate tolerances numerically but not graphically. We are therefore apt to visualize the part as it is graphically shown, that is, without tolerances and to think of the numerical tolerances one at a time rather than in combinations as they affect each other and the shape of the part.

If any dimension, significantly affecting the design of a part, is changed the drawing is immediately corrected so that its meaning will be clear and the functioning of the part can be checked. This obviously facilitates design and manufacture. Yet because they cannot be shown directly by regular drawing methods, we have grown accustomed to not being shown the effect of tolerances or changes in tolerances upon the shape of the part. Nevertheless it is obvious that these effects are critical in the functioning of the part or tolerances would not be set. The fact that these critical features of the design are not actually graphically shown and therefore are not easily seen and understood on the drafting board is a serious detriment in working out a design and in all later analysis of it. The full effect of interrelated variations particularly if in three dimensional space may appear only after tools are in process or the first parts produced and this may be rather late for economy.

Originally this difficult analysis of the effect of tolerances upon functioning probably involved only the designer. The manufacturer tried to make the part as nearly as possible to the nominal values shown and variations from them were accidental. Tolerances were looked upon as an indication of the care required and as a means of inspection for acceptance or rejection. With increasingly complex manufacturing tools the permitted tolerances are utilized more and more in the design of tools to allow the greatest possible wear before defective parts are produced and the tools must be replaced. For mass production parts progressive step type tools are used in which a continuous strip of stock advances by various stages from blank sheet to finished part. Tools of this type are extremely expensive and in order to obtain maximum life full use of allowed variations is made in their design. Design of such tools and the gauges required to maintain quality in mass production therefore also requires analysis of the effect of combinations of variables upon the desired part. As the designer has presumably already made this analysis, and incidentally is best qualified to do it, economy and accuracy dictate that his analysis be transmitted to the manufacturing engineer. The problem is to find means by which he can indicate unmistakably on the drawing his analysis of the required functioning of the part and the manner in which he intends the tolerances to apply, in the event that there is any possibility of misunderstanding.

The essence of this problem and some of the possibilities of solution can best be seen by reference to drawings which illustrate the major points.

Figure 5 shows the drawing of a flat plate dimensioned from center lines but without any tolerances whatever. Some minor dimensions not involved in this discussion are omitted in the interest of simplification but the part shown is in every way a normal one. The meaning of the drawing is completely clear and can be interpreted in but one way no matter from what standpoint the analysis is made. The reason for this is obviously that but one value is shown for every dimension.

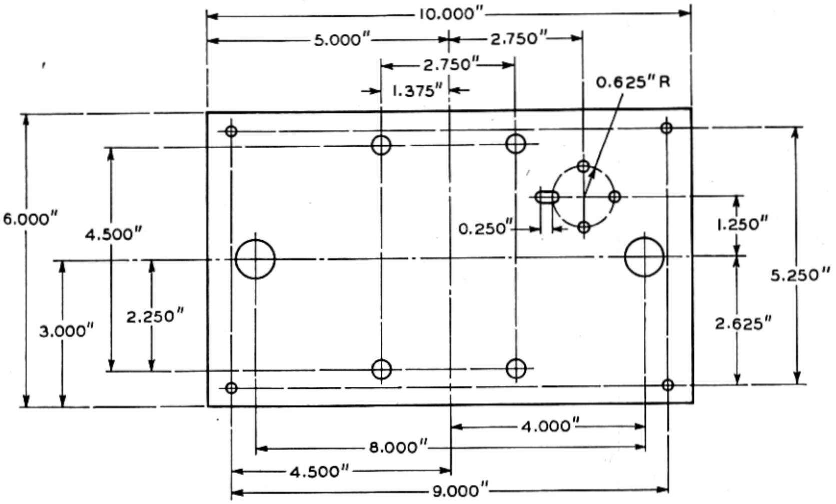

Fig. 5—Flat plate dimensioned without tolerances

Figure 6 shows this same drawing dimensioned in exactly the same way with the exception that tolerances are shown for most of the dimensions. To the uninitiated it might appear to present no more problem than the previous drawing without tolerances because of the tendency to visualize the drawing in terms of the nominal dimensions only.

When the engineer analyzes the effect of the combinations of the various tolerances shown, interesting questions immediately arise. In the first place the combination of holes dimensioned 1.25" ± .002" from the center line appears to be definitely located because on the drawing the center line is shown in a definite position. Yet when the tolerances are considered the center line of this drawing could actually be shown in several different places as, for example:

1. It may be a line through the centers of the two large holes.
2. It may be a line anywhere from 2.992″ to 3.008″ from the outside edges.
3. It may be 2.247″ to 2.253″ from the small holes in the center of the plate.
4. It may be 2.615″ to 2.635″ from the holes numbered 2 and 4.

In brief, the center line which appears so definitely located on the drawing may actually be rather an indefinite location on the part when the various tolerances are considered. While the differences in the possible interpretations are in the order of thousandths of an inch nevertheless this order of magnitude is critical in this part or the indicated tolerances would not have been used. The interpretation of the center line which should be adopted will depend entirely upon the manner in which the part is intended to function and therefore should be indicated by the designing engineer. Obviously, not all designs or all dimensioning will present this difficulty but all should be studied from this viewpoint to determine whether or not they do.

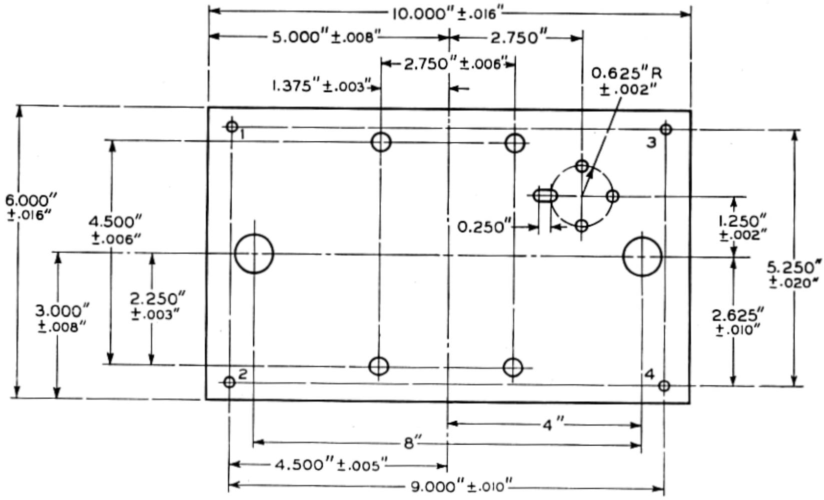

Fig. 6—Flat plate of Fig. 5 with the addition of tolerances

Functional Datum Positions

When the type of uncertainty illustrated exists, it is necessary to indicate clearly the effect of tolerances on functioning by establishing the functional positions to which dimensions should refer. It may be difficult to do this graphically, in which case it is necessary to indicate by notes the particular interpretation which the designer intends. As an example, if the part of Figs. 5 and 6 functions by being located in position by means of the four holes numbered 1, 2, 3 and 4, the intentions of the designer are readily indicated by the following notes:

1. Functional datum line I is midway between the centers of holes 1 and 2 and the centers of holes 3 and 4.
2. Functional datum line II is perpendicular to datum line I at a point midway between the centers of holes 2 and 4.

These notes establish both horizontal and vertical center lines specifically in terms of the center of the one set of dimensions between the holes marked 1, 2, 3 and 4. The term functional datum line is suggested as completely descriptive but other equivalent terms might be used. This information could be indicated on the drawing without the use of notes by the adoption and use of some standard convention or symbol to indicate the particular dimension bisected by the center line.

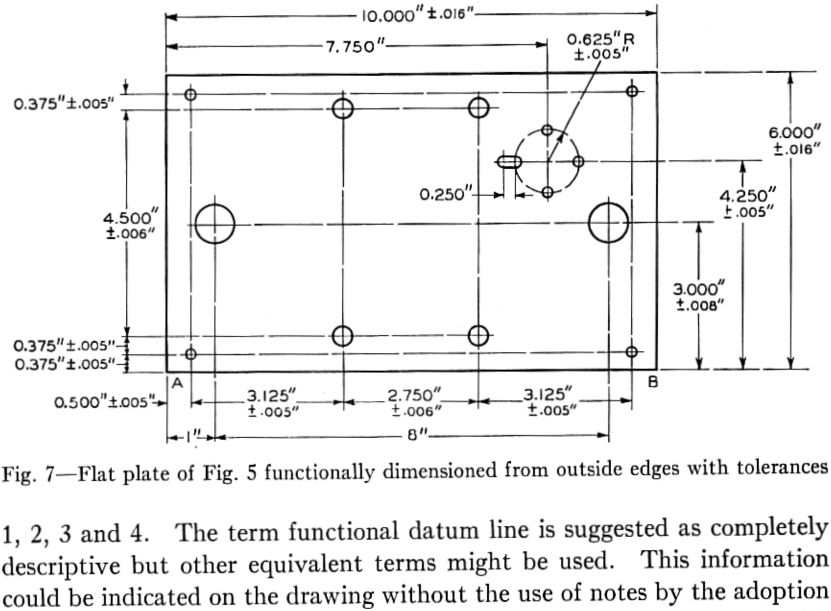

Fig. 7—Flat plate of Fig. 5 functionally dimensioned from outside edges with tolerances

If the functioning of this part were determined by location against the outside edges, this could be readily indicated by dimensioning the part as shown by Fig. 7 and using notes establishing the line A–B as one datum line and the perpendicular to it through A as the other.

In either of these cases the drawing becomes completely definite and subject to only one interpretation. In drawings of this type no change in the method of dimensioning may be required and the problem is solved simply by the addition of suitable notes or symbols indicating the intention of the designer as to functional datum lines.

It is sufficient to establish datum lines in the case of parts which are practically flat pieces with little depth but when a part has substantial depth it will be noted that center lines or other datum lines on a drawing really represent planes in space. In such parts it becomes necessary to establish datum planes rather than lines and three planes at right angles to each other are required.

Figure 8 illustrates such a part which might be an armature such as is used in many pieces of electrical contacting apparatus. In the typical operation of such a part its functioning is determined by the relation of its various dimensions to the position of the pole face and the axis support. In order to indicate this on the drawing it is necessary to establish dimensioning as shown and add to the drawing the notes shown.

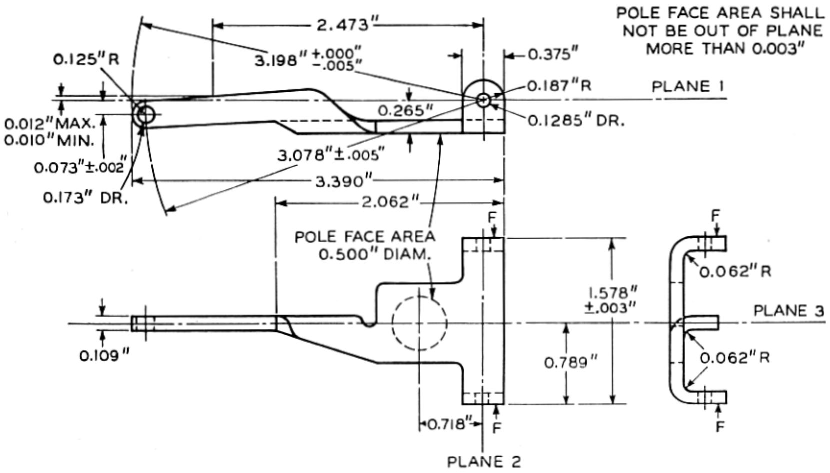

Fig. 8—Functional datum plane dimensioning of magnetic armature type of part

Functional datum plane I passes through the common axis of the two .1285 in. diameter holes and .265 in. above the pole face gauge position.

Functional datum plane II is perpendicular to plane I and passes through the common axis of the two .1285 in. diameter holes.

Functional datum plane III is perpendicular to planes I and II and passes midway between the finished surfaces which are 1.578 ± .003 in. apart.

These notes establish three functional datum planes, the first through the axis at the point of support and a distance .265″ from the pole face area; the second at right angles to the first through the axis of support and the third at right angles to both the first and second and halfway between the finished surfaces 1.578″ apart. With these planes established the application of all the limits and tolerances shown is based on the operating position and analysis of the design is simplified. The drawing and the intentions of the design engineer cannot be misunderstood.

The clear expression of the designer's intentions by datum plane dimensioning will be appreciated by all concerned with the drawing or the resulting part. Inspection of the part is expedited no less than production. The inspector can usually by means of gauge blocks or simple fixtures set the part up on a surface plate as indicated by the drawings datum planes and positions. He can then establish the conformance of the part with the drawing by simple measurements to the indicated horizontal and vertical planes. When production quantities justify special gauges the required design of the gauge is established clearly by the datum planes.

Invariable or Gauge Dimensions

The drawing of Fig. 8 just described illustrates the use of gauge dimensions. The dimensions .265″ and .718″ and the indicated half-inch diameter for the pole face are all gauge dimensions without tolerances and some statement must be made or understanding reached that they are considered invariable and tolerances not permitted. They represent, it might be said, theoretical dimensions, on the drawing, or in practice they represent tools or gauging apparatus made to the highest standards of accuracy. These invariable dimensions are necessary in order to establish a starting point for the dimensioning of the part. It may appear at first that stating that a dimension has no manufacturing tolerance or variation is a hardship upon the manufacturer but this is not really so because the dimensions are not ones which are actually manufactured in the part. They represent usually dimensions built into tools or gauging equipment which are made to a precision greatly superior to that represented by part tolerances.

Invariable dimensions, or better, gauging dimensions or whatever it is proposed to call them are really not a new invention and it is possible to cite easily recognized examples. For instance, the dimension 2.473″ on Fig. 8 is an invariable gauging dimension not associated with the setting up of datum planes but typical of long standing use of invariable dimensions. We all can recall also the use of the term "theoretically correct position" and it is present practice in the case of vacuum tube bases and similar apparatus to designate the location of the contact studs in terms of a gauge having holes located on "true centers." Last but not least a minimum or maximum limit in its application is itself an invariable dimension.

In effect, datum lines or planes established when necessary by use of invariable or gauging dimensions remove the uncertainty as to the designer's intentions and prevent misunderstandings between design, production and inspecting engineers. Admittedly they do not completely solve all problems of dimensions as probably nothing will. They do, however, transfer whatever problems remain from the field of tolerances on finished product to the realm of tool making tolerances and gauging tolerances. The problem of how invariable is "invariable" remains but we are obviously then considering differences of an order of magnitude not usually vitally significant in the functioning of product parts. Theoretically, all "invariable" dimensions should be taken to the best accuracy of good gauging methods which means that any differences of opinion will be reduced at least to one-fifth and probably to one-tenth of the order of magnitude of those where tolerances themselves are involved.

It will be necessary to specially identify gauging dimensions on drawings to distinguish them from ordinary unlimited dimensions and to indicate that they are dimensions for gauges to which only gauge tolerances apply.

Practical Use of Datum Lines and Planes

It is not usual to establish datum lines on all drawings but if their use is necessary in the layout and design of the part they need to be permanently identified. This use of datum lines and planes on drawings, where necessary, may require somewhat greater drafting effort in the actual production of the drawing but their use results in a simplification of design and of the work of those subsequently using the drawings. It reduces the effort expended in analysis of drawings preparatory to the construction of tools and minimizes the possibility of misunderstandings or errors in tools. In products manufactured only intermittently it is particularly valuable as it minimizes the need for understandings and instructions supplementary to the drawings which may be forgotten between production periods or lost through shifts in personnel.

The overall economy in engineering effort and the reduction of the numerous possibilities of error more than compensate for the increase in the actual work of indicating datum positions, lines or planes upon drawings. In addition the choice of design of punches and dies and similar tools by production engineers is better guided by the designer's requirements if functional datum lines are clearly identified. An obvious example is the use of either the inside or outside of a punched and formed part as the starting point. In brief datum plane dimensioning is a more explicit expression on the drawing, of the designers "end point requirements".

When establishing datum planes, it is important to consider them in terms of the actual physical part rather than in terms of the drawing. Lines which appear as definite points on a drawing may not be actually part of the product when it is completed or may be on surfaces shown as a line on the drawing but rough or unfinished in the part. It is difficult to establish any set of rules covering what shall or shall not be done because each drawing and each part must be considered practically as an individual case. That this is so will be amply demonstrated by a serious study of even one part. However, there are obvious generalities which can be established and Fig. 9 shows some of them.

An example of functional datum plane analysis and dimensioning in three dimensions of a complicated part is shown by Fig. 10. This is the die cast frame for a special selector switch. It is the base upon which many interrelated parts and subassemblies are mounted. The proper functioning of the completely assembled switch depends in large measure on proper manufacture of this casting. In effect, the switch is designed around a vertical shaft passing through points P and Q and planes 1 and 2 are, therefore, established through the axis of this shaft. The production planning engineers intend to design the die and withdraw die plugs from such directions that the mounting surfaces will be smooth, flat and without any taper and they intend to use these surfaces as guiding points for their jigs and fixtures. It is for this reason that Plane 1 is established parallel to these mounting surfaces and an indicated distance from them. The other planes are established as shown on this drawing and described by the notes. With this arrangement of planes the designer's analysis in terms of Plane 1 is easily worked out and the reference of Plane 1 to the mounting surfaces permits the production or tool engineer to translate the design of the part into the design of his tools without necessity for further analysis and without the possibility of different interpretations. It will be noted that invariable or gauge dimensions are again used. The complete drawing of this part is very complicated and occupies a drawing practically 4 ft. x 6 ft. The perspective sketch shown and the accompanying notes are incorporated in the drawing as a separate view.

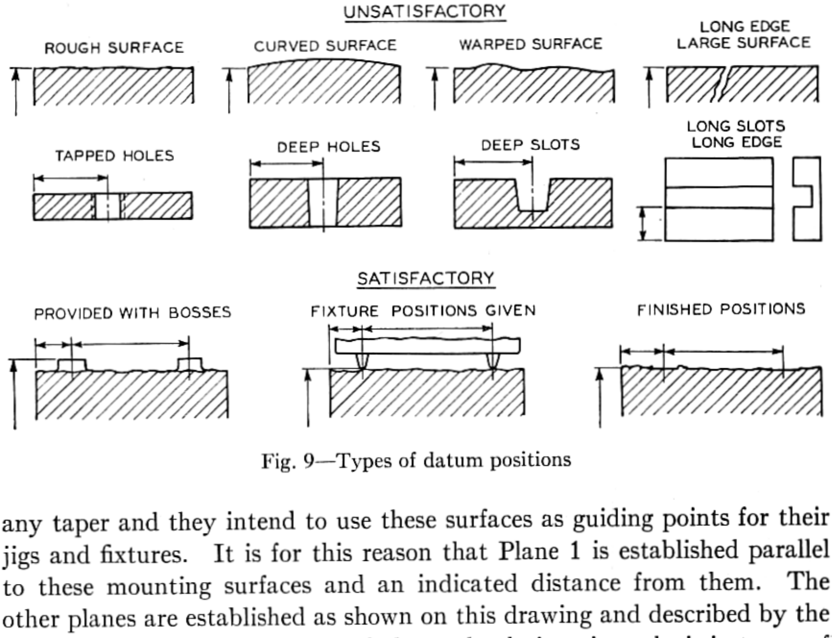

Fig. 9—Types of datum positions

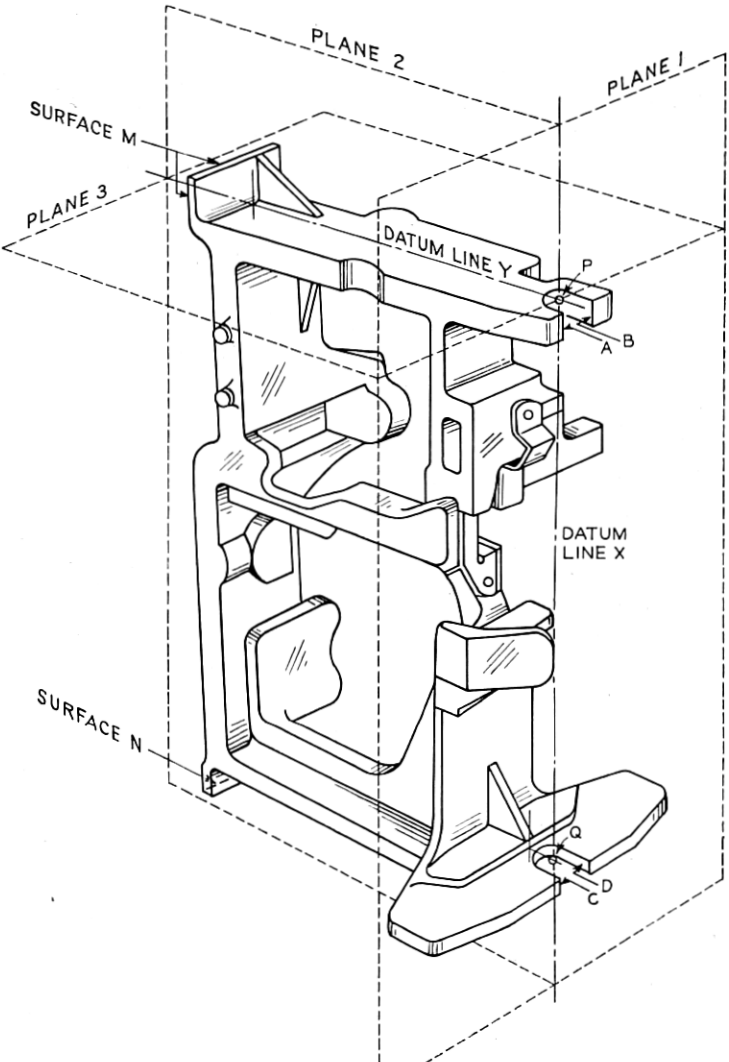

Fig. 10—Functional datum planes of complicated switch frame

Dimensions to datum line "X" or "Y" of the drawing of the frame refer to functional datum planes 1, 2 or 3 described below. Points "P" and "Q" are gauge points used in establishing these datum lines and planes. Points "P" and "Q" shall be half-way between the surfaces "A" and "B" and "C" and "D" respectively and 4.358 in. from the plane of surfaces "M" and "N" on the mounting lugs.

Datum line "X" shall pass through the points "P" and "Q".

Plane 1 shall be parallel to surfaces "M" and "N" and shall include datum line "X".

Plane 2 shall be perpendicular to plane 1 and shall also include datum line "X".

Plane 3 shall be perpendicular to plane 1 and to plane 2 at the point "P".

Datum line "Y" passes through point "P" and is the intersection of planes 2 and 3.

REQUIRED STANDARDIZATION

It is not suggested that the drawings shown and the notes referred to represent a final practice on datum planes. A standard practice in designation of planes and standard terminology and understanding on gauge points and gauge dimensions is required. It will probably be desirable to adopt some symbol or designation for use on drawings to distinguish gauge dimensions which are invariable from ordinary unlimited dimensions to which manufacturing engineers for their own purposes usually add shop tolerances. One thing is certain and that is that datum planes, dimensions and tolerances when established should be primarily in terms of the required functioning of the apparatus. When that is done no one using the drawing in any capacity will have any doubts as to the designer's intention and this results in a great reduction in the discussions and analysis which might otherwise be necessary.

SUMMARY

In summary it may be said that the whole approach to these problems in dimensions and tolerances should be on the basis of functioning. However, good engineering of dimensions and tolerances requires knowledge of what can reasonably be produced and the sources of reasonable tolerance values are:

1. Raw material limits including some knowledge of future trends and developments.
2. The normal accuracy of manufacture, also including anticipation of future improvement.
3. Discussion of trend of design with manufacturing engineers.

Solution of tolerance problems in the final design may involve all of the following steps:

1. Study of the effect of combinations of tolerances on functioning, allowing for statistical effects in accumulations of tolerances.
2. Discussion of this analysis with the production planning engineer because the analysis of tolerance combinations is important in the design of long life tools.
3. Indication of the results of such an analysis by the method of dimensioning drawings.
4. Indication on drawings of functional datum positions, lines or planes established on geometrically correct principles to permanently and unmistakably record the intentions of the designer regarding combinations of variations wherever this is necessary.