TUBE FIXTURE DESIGN AND TOLERANCE

Thomas R. Clark
Clark Fixture Technologies
410 N. Dunbridge Rd.
Bowling Green, OH 43402

TABLE OF CONTENTS

INTRODUCTION

Clark Fixture Technologies has prepared this catalog as an aid to explain tube fixture design, as well as define common concepts related to bent tube design and fixturing. Following the first four sections on design considerations is a section showing some of the standardized fixture details that are utilized here at Clark Fixture Technologies. We hope this catalog will be of use to you.

This publication has been released as a starting point and is not intended to be the final word. If there are areas you disagree with, or aspects you think should be included, please send me your written arguments. My hope is that this will stimulate discussion resulting in a move toward standardization of concepts, methods, and fixture functions.

Although some viewpoints in this report may seem biased toward Clark Fixture Technologies' methods, this work is not intended as a sales proposal, and is not written from a proprietary position. It is presented because we want to do our best to provide you with the important information we have.

If you know someone who may benefit from, or has an interest in this information, please write to me at 410 N. Dunbridge Road, Bowling Green, Ohio 43402. I will make this catalog available to them. I would request that you do not copy this report, as I wish to maintain its integrity and be able to update it.

Sincerely,

THOMAS R. CLARK

Updated August 2001

TUBE GEOMETRY

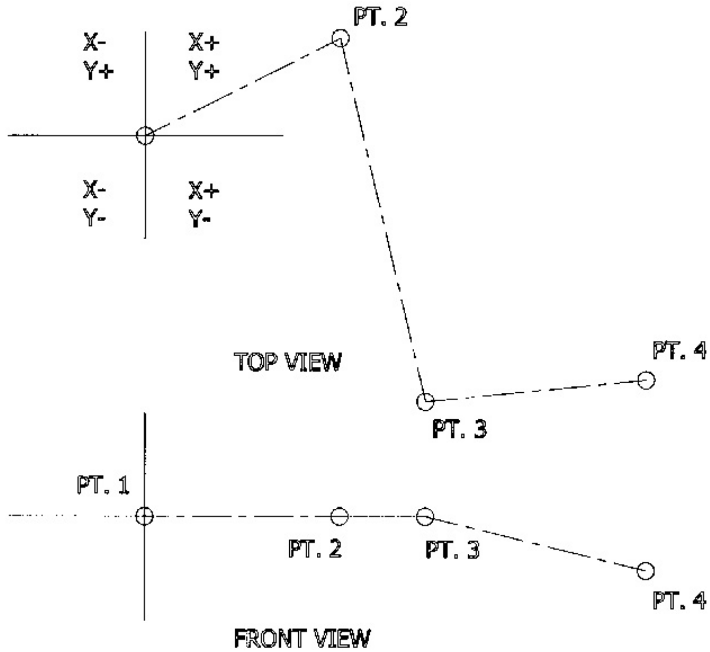
There are four basic methods of defining a tube shape: Cartesian Coordinate System (XYZ), Bend Data, Mechanical Drafting, and Military Specifications. The following is an explanation and discussion of each method.

Coordinate System (X Y Z)

The first and most popular method defining tube shape is the XYZ method, which utilizes the Cartesian coordinate system (see Fig. 1). Three datum planes are established which are mutually perpendicular, or normal, to each other: *the X-plane, Y-plane, and Z-plane*. The intersection of these planes creates an origin. "Control" points are assigned at a given, predetermined distance from each of these planes. These distances fix the point in three-dimensional space. These points are connected in sequence by straight lines, which will become the centerline of the tube. After creating the polyline centerline, radii need to be placed so that a segment of a circle with a given diameter lies on the plane described by two intersecting lines and is tangent to both those lines. These straight lines and arcs become the theoretically perfect centerline of the tube so there will only be one shape that will satisfy these constraints. This centerline may be thought of as the datum. At this stage, a circle of a known diameter is created on an end point, normal to the corresponding vector. The circle is then swept along the centerline. The swept surface flows around and along the centerline, so that any point on the surface of the cross-section of the tube is an equal distance from the centerline. This equal distance will be constant through the straights as well as through the radii. This process finishes the tube body.

A method to insure proper relationship between the datum planes as the centerline is rotated, is called the "right-hand rule" (see Fig. 2). The right hand rule is a way to conceptualize how this relationship works. Using your right hand, place your index finger straight ahead in the X-axis, your middle finger in the Y-axis, and your thumb will be the Z-axis. The same relationship is maintained no matter which way you rotate your hand. This is important so that you do not create a mirror image by flipping the relationship between axis planes. Positive values move away from your hand and negative values move from the fingertips toward the palm of the hand.

XYZ DIMENSIONING SCHEME



Pt. No.	X	Y	Z	Rad.
1	0	0	0	0
2	+36	+18	0	10
3	+52	-49	0	15
4	+93	-45	-10	0

Fig. 1

NOTE:

1. These are centerline values, not tube surface values.
2. The relationship between these values will always remain the same.
3. The values themselves may change if the X-0, Y-0, Z-0 reference changes.
4. This table describes the complete centerline model.
5. This example assumes Pt. 1 is X-0, Y-0, and Z-0.

THE RIGHT HAND RULE

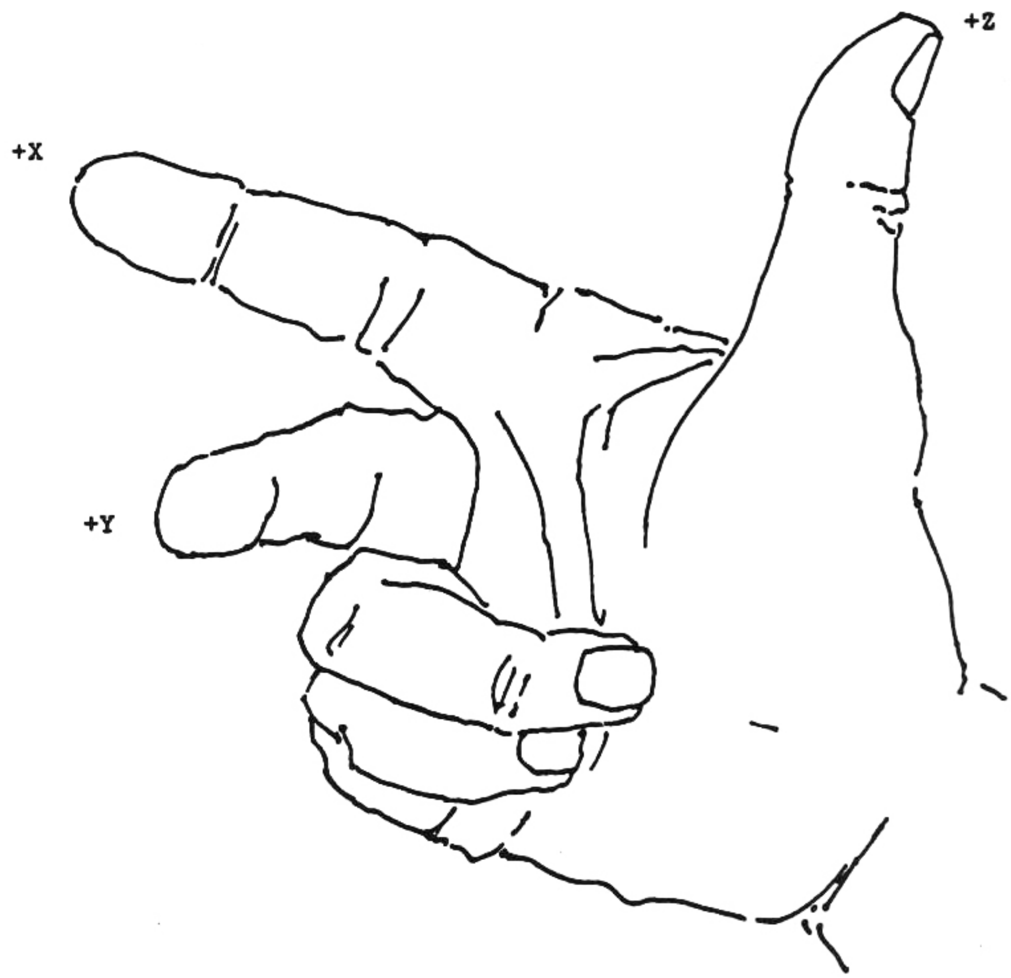


Fig. 2

The preceding is the general anatomy of the XYZ concept used in the tube industry. The tube is constructed of a straight section, an arc segment, and another straight section and so on until it ends with a final straight. The tube is not a smooth flowing continuous curve, but a series of straight and arc segments. This methodology of XYZ points can be expanded to locate and describe a variety of other features such as clamps, brackets, or other attachments.

The advantages are that this coordinate system easily positions the tube in 3D relationship to other features around it, which the tube may need to avoid or be attached to. This system is used for automotive and aerospace design as well as many other industries. Several datums can be established; one for each component and subassembly, and a global datum in which all the subassemblies correspond. By this method, the engineer can quickly determine the relationship of any component to any other component. It is the most common method of design, found on almost all CAD/CAM systems, and is fairly easily understood. Due to the built-in datum source, this method lends itself to Coordinate Measuring Machine (CMM) inspection and the general CMM concepts used for measuring blocks or spheres.

The limitations of the XYZ coordinate system for tubing fall into two general areas. The first is this system describes a three-dimensional model on a two dimensional plane. If the model is rotated any degree out from any of the views, the entire XYZ table is voided and will need to be recalculated. While the tube shape did not change in relationship to itself, its relationship to those planes changed. Therefore, the values describing tubes in terms of XYZ points, relates to whether you are looking at the whole picture or only at the tube.

The second general area of limitation is that parts cannot be bent in XYZ. XYZ data must always be converted to what is generally called "Bend Data".

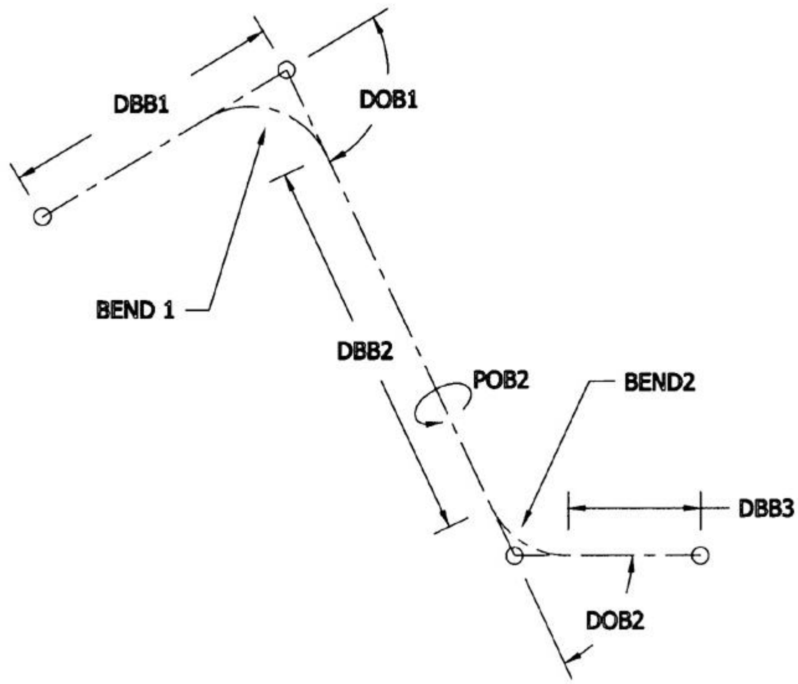
In terms of fixturing and quality assurance, the fixture builder gets caught in the middle. Based on CAD systems, as well as common layout practices, we find it easier and more convenient to describe the model in XYZ terms. As products become more and more sophisticated and tubing becomes increasingly complex, it is helpful to the bender to analyze his or her part in as many, or as close to true views or flat bends as possible. It is difficult to "read" a bend that is going straight up in the air. Tube deviations are easier to see when the tube is flat or in true view.

BEND DATA

"Bend Data" is the next most common method of defining tube shapes (see Fig. 3). Bend data is made up of three elements. First is the "distance between bends", (*D.B.B. or L=Length or "Y"*). This is the straight distance between two bend radii tangencies or the distance from the first or last point to the closest tangency. The second feature of Bend Data is the "degree of bend", (*D.O.B. or A=Angle or "C"*), which is simply the number of degrees in the bend. Most of the practical problems in bending occur here. Spring back, elongation, collapse, tooling marks, tearing, etc., occur in the degree of bend. The third feature is the "plane of bend", (*P.O.B. or R=Rotation or "B"*), sometimes called "phase angle" or "twist". This is defined in degrees and is the rotational relationship between one bend plane and the next bend plane.

Bend planes can be defined as the plane created by the two intersecting centerline vectors, one incoming and one outgoing, with the bend somewhere in between. It takes three points to describe a plane: first, a point somewhere on the first straight leg, preferably the farthest point from the bend in question; second, the intersection point where the two lines meet; and third, a point on the second straight leg, again, preferably the farthest point away from the bend. When a bend is held with its bend plane normal to the angle of sight, it is viewed in a "true view". (When a curve defining a bend is laying flat in the x/y plane [0,0], it is said to be in true view.) In order to put the next bend in a "true view" position, the tube is rotated about the common centerline of the straight segment, which is simple enough. Then, there are some questions to be asked here: Is the rotation clockwise or counter clockwise? Are you looking from the last bend to the new bend or from the new bend to the last one? In terms of design and inspection, it is reassuring to have some fixed absolute. However, in terms of building the part, it is not practical to relate all bends back to the first.

BEND DATA DIMENSIONING SCHEME



Bend #	P.O.B.	D.O.B.	D.B.B.	RAD
	0	-	17mm	0
1	0	95°	50mm	10mm
2	5°	80°	23mm	15mm

NOTE:

1. These values will be the same regardless of part orientation.
2. This table describes the complete centerline model.

The advantages of Bend Data are similar in nature to the weaknesses of the XYZ coordinate system. The part geometry remains the same regardless of orientation. No matter what rotation you find the tube in, you can measure it in terms of Bend Data. Bend Data also lends itself more readily to the construction of a tube.

The disadvantages of Bend Data are generally the opposite of the XYZ system. With Bend Data there is no method of determining the tube's relative position to other close objects. Theoretically, the only absolute considered with Bend Data is the starting point, and occasionally, the ending point.

These two mathematical systems complement each other. The system of absolute XYZ values describing relative positioning is essential in design, and the progressive nature of Bend Data building one feature on top of the previous is ideal for building real parts. There are many software programs that convert these systems back and forth, and while the math is somewhat complex, it is not outside the realm of most shops even if they don't have the software.

The only common breakdown between the two systems comes when parts are inspected. Most inspection is required to be performed in terms of what is on the drawing, and naturally so. The part, however, exists in real space and has no connection to the XYZ values. The section on inspection deals more completely with these problems and possible solutions, but it is helpful here to have a general understanding of the difficulties. The first concern is to correctly orientate the part in space and secure it so it can be checked through Coordinate Measuring Machine inspection. This can involve physical positioning or "best fit" mathematics. The second concern is how to use the theoretical XYZ values and through tolerancing, accept or reject real parts. Utilizing an envelope or jacket best does this best. This will be discussed in the section on Tube Tolerance.

MECHANICAL DRAFTING

This method defines the tube in what might be thought of as a more traditional mechanical drafting. It defines geometry from outer surfaces as well as outstanding features such as bracket edges or hole centers. It will often include degree of bend angles and reference from bend tangents.

Three problems arise, all in some measure related to the view limitations. The first concern is you are not working with the centerline, but the outside surface. This does not allow for material diameter tolerances. Second, the dimensions may come from points or features that are only relevant in the specific drawing view. If the view is rotated, the point may become non-existent. An example is a given distance from a bend tangent to the backside of the bend. Depending on the view, the distance may increase or decrease. The third problem is that frequently, information from one view will conflict with information in another view.

From the construction side, there are no readily usable transfers from this dimensioning method to Bend Data. In building fixtures, we have a similar problem of trying to interpret the data into a usable form. This takes more time, which translates into increased expense.

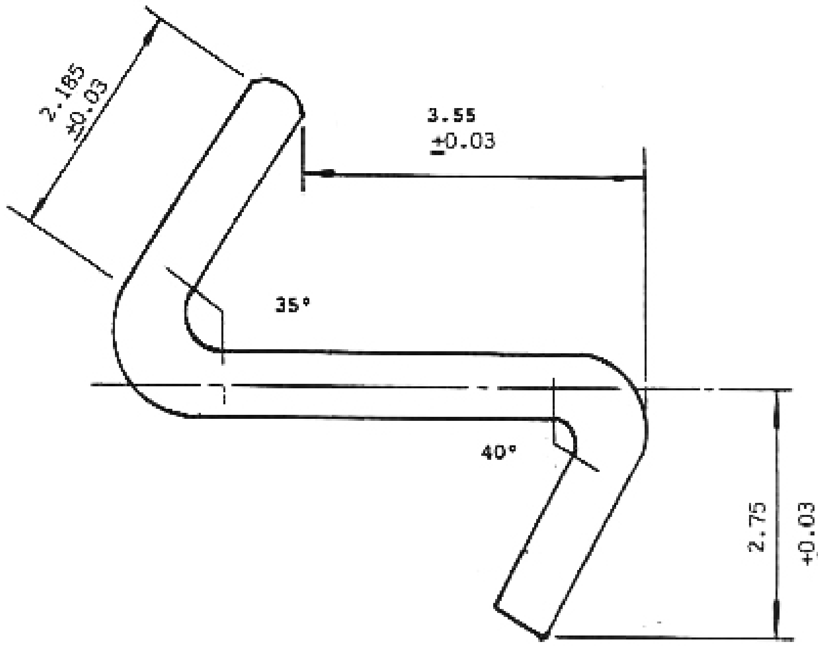


Fig. 4

NOTE:

1. Leader lines reference an end feature valid only in this particular view and will be altered or moved in another rotation.

2. The possibility is high for conflicting values, especially in complex parts, with more than one view.

3. The tolerancing scheme battles against itself and adds the potential for stacking tolerance, but implies a ±0.030 envelope tolerance.

MILITARY SPECIFICATIONS

A copy of these specifications can be found in Appendix D. This method is very functional as it defines the tube in terms of bending information.

TUBE TOLERANCE

There are a variety of ways and interpretations of treating tube tolerances. There are many good books and classes on Geometric Dimensioning and Tolerance that may be referred to for more information. It is beyond the scope and purpose of this document to reiterate that topic. What is discussed are some general considerations of how tolerances relate to tubes and try to establish some general conceptual treatments that concern tubes and tube attachments in terms of designing appropriate fixtures. In the section on fixture details you'll find specific solutions to specific problems. My hope is to clarify the problems so appropriate solutions can be executed.

Datum Points

A "Datum" as defined in Webster's New International Dictionary is "a point assumed or used as a basis of reckoning". In other words, it is a point, plane, line, axis, or feature, which is considered to be a constant from which other features, points, or planes may be consistently measured.

Whether implied or directly stated, almost all tube drawings have datum points. When drawings use geometric dimensioning and tolerancing, the datum points are called out specifically, and the viewer is aware that indicated features merit special consideration. Drawings utilizing only XYZ values have an implied datum point, or more directly here, a "0,0,0" point where all dimensions originate from. The XYZ points can be thought of as fixed or datum references.

Note: When checking a part, one of the two primary methods is to verify that the theoretical centerline of two straight sections will intersect within an area of tolerance around the fixed XYZ point. The other primary method is by verifying that the actual exterior surface falls within the calculated "envelope tolerance" as described below.

There are also datum points that are not called out, but are "assumed" because "we have always done it that way," regardless of what the drawing states. (*i.e., "Bracket holes are always datum points because that is where the tube mounts"; both ends are datum because "that's where the tube meets something else larger than itself -- unless there is a rubber hose on one end -- then only the component would have both ends datum and the full assembly would datum the tube end and tolerance the hose end"; "the tube end that we start bending from is datum because that's where we start")*. All of these assumptions have some validity, but ideally, should be stated explicitly on the drawing. The drawing should determine the tube as it actually is, and not according to tradition.

Envelope Tolerance

Envelope Tolerance, (E.T.), is a concept somewhat exclusive to tubing. This is a theoretical space between maximum material condition of the surface for a perfectly shaped part, plus an imaginary surface a fixed distance from the tube surface, which completely surrounds the tube and avoids interference with other components. It assumes the theoretical centerline is a datum.

While this imaginary surface is not required to be centered around the tube, it is generally thought of as an imaginary tube that has a diameter of the maximum material condition of the tube, plus two times the allowed tolerance, (*for instance, a 12mm tube with a tolerance of ±2mm would have an envelope tolerance of 16mm. See Fig. 5*). The tolerance envelope is driven from the centerline geometry, thus obtaining a theoretically perfect space, which the tube must not breach. The envelope tolerance is calculated as MMC + 2X allowed tolerance, because the theory is the tolerance of + X allows the centerline to move in both a positive and a negative X amount; thus, 1X for positive plus 1X for negative plus the maximum diameter of the tube. This is the zone the designer intended the tube to not intrude and should prevent any interference.

This is the general concept of envelope tolerance. We believe the best method to assure your parts do not intrude through this zone is to construct a hard replica of this zone in which you can lay your tube and verify its correctness.

An envelope tolerance zone adjustment may be a "datum target." This is usually a point on or near the surface, which would reduce or eliminate tolerance at that point. This can be accomplished through the use of pins, (*Ref. Datum Target Pins in Detail Section*). Also, envelope tolerance may change over the length of the tube. One area may call for greater restriction to the point of having no tolerance at all, while other segments may allow very broad zones.

This concept is at the heart of tube tolerancing and should be well understood. Whatever method of tolerance call outs are given, they are best interpreted through envelope tolerance.

ENVELOPE TOLERANCE

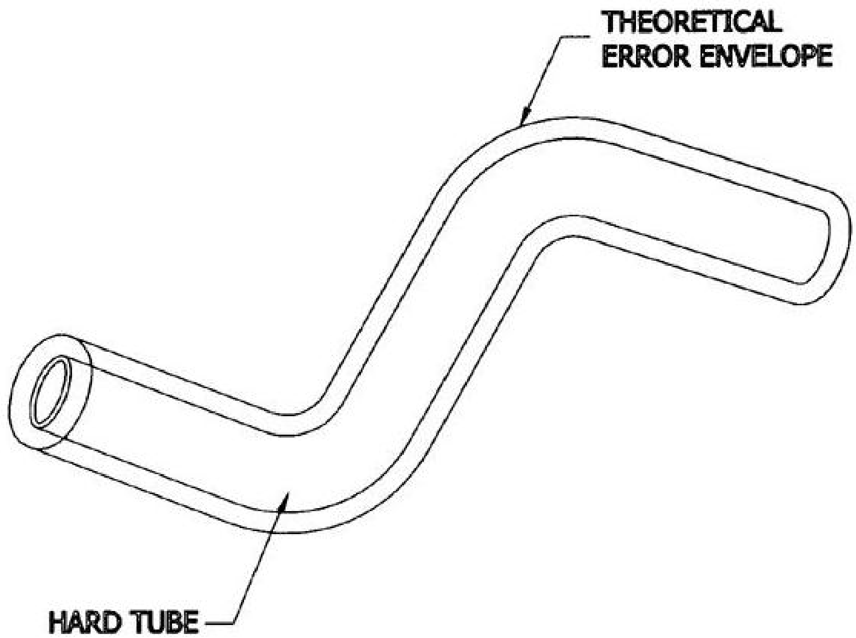


Fig. 5

NOTE:

1. Tolerance is not *required* to be consistent or regular about the tube surface. However, the envelope tolerance maintains its shape.

It is important to discuss that a tube can be built which will satisfy the envelope tolerance, but will not be traceable back to the XYZ control points by analysis of the centerline tube vectors. This means that if you build a tube with an envelope tolerance and your customer is checking your parts using vector intersections, they may reject most of your parts. The envelope approach will allow a variety of different tube shapes to fit within the tolerance zone. This does not invalidate the envelope tolerance, but is an example of how the envelope tolerance satisfies the design intent for clearances, allowing manufacturers full use of tolerances to produce economical parts, which may still fail CMM inspection.

Because the vector intersection method uses the straight sections to determine the centerline vector angles and projects that to another line to form the intersection point, its tolerance is very dependent on not only the length of the straight, but the degree of bend. To understand the differences, let's review how the two concepts are created and briefly discuss two verification methods.

The XYZ control points are the points in space that have been determined by a designer to express the best, or at the least, a good path for a tube to take. Although there are other considerations, the main concern is a direct path that will take the tube from a starting location to an ending location and avoid whatever is in the way or could be in the way in the future. Because a designer knows that tubes cannot be produced perfectly, they specify an amount of tolerance that will allow the part to function as intended.

To create the "error envelope" you begin with the perfect centerline, or datum, and add the maximum tube O.D., plus the allowable tolerance. This is the theoretical path, which is larger than the maximum tube diameter and if the tube remains inside, will avoid all obstructions and meet the designer's requirements.

When verifying a part, there are two schools of thought. One approach is to measure a tube and determine whether or not any part of it breaches the theoretical envelope tolerance. This can be done mathematically utilizing tube readers and computers, or it can be done through the use of fixtures. The second method is to measure the tube, with a tube reader or other coordinate measuring machine (CMM), calculate a centerline, project the centerline to the intersection points and compare the deviation at the intersection point. A tube that passes the envelope test may not pass the intersection point test.

This difference is a new problem, which has been exposed by the greater use of computers. Previously only in the rarest of situations were centerlines calculated, projected to intersection points, the whole math model oriented to the drawing XYZ orientation, and then compared in three-dimensional space. With the use of computers, this is commonplace today, but is not always accessible at the time and place the tube needs to be checked.

The first method reaches for the outside of the tube in a more pragmatic approach and the second method calculates a centerline and compares theoretical points in a theoretical orientation; one on the outside and one on the inside.

While there is no obvious resolution to the problem, there are a few issues to clarify. First of all, it is a fact that the majority of tube drawings utilize a Cartesian coordination system and the control points are specifically called out. This gives them a priority position and suggests that the

final product should make use of them as the standard, and that the tolerance zone should be a sphere, (*or a cube*), around the XYZ point. This can only practically be measured and analyzed by a computer. Many even suggest the repeatability of a tube reader is not reliable enough to provide confidence. As we have seen earlier, that XYZ values, while having great merit for design, are difficult for real world construction.

Second, in the XYZ intersection method, the envelope tolerance, or the actual position of the tube, will increase and decrease depending on the distance of the straight and the degree of bend. (*If the straight is short, a more radical vector angle is accepted and if the bend is deeper, the intersection point is projected farther from the tube*). This is not the intent of the designer and would be very difficult for the tube builder to calculate.

The envelope tolerance method works well in the "real world" with either measuring equipment or a fixture. It is not an improper argument to state that it is the XYZ points that are stated on the drawing and not an error envelope, and that the tube must be verified to the XYZ points.

Until standards are more correct or a change is made to the drawings, what is important is that all parties connected with the tube construction and verification understands these issues.

LINEAR TOLERANCE

Linear Tolerance (L.T.) is simply the allowable deviation at either one or both tube ends according to the drawing. This tolerance is really the envelope tolerance as it applies to the ends of the tube, but it is helpful to consider it separately due to the following practical results.

There are two reasons for treating linear tolerance separately from the envelope. First, because of datum schemes or the methods by which the tube mounts to some larger object, the tolerance is almost always different and sometimes non-existent at the ends. The second reason is that in fixtures, the practical description of end or linear tolerance is traditionally the use of a sliding pin with some fashion of a scribed hi-low band. It seems the greatest reason for this is tradition, yet there are some advantages over a fixed block at the end. The most notable is that it allows you to determine your overall length of the tube to which you can trim to proper length as a secondary operation. This practice is common in larger tubes such as exhaust tubes.

TOLERANCE CALL OUTS

I.
x.x = ±0.020 inch
x.xx = ±0.010 inch
x.xxx = ±0.005 inch

II.
1-100 = ±2mm
101-300 = ±3mm
301-800 = ±6mm

III. Feature Tolerance ±0.030

IV. Bend Tolerance ±1

V.

⌖	Ø 0.060	Ⓜ

Listed above are five common tolerance call outs, one of which can be found somewhere on most drawings, often in the title block or in the notes. Sometimes there is a call out in the title block and a conflicting statement in the notes. If this is the case, the call outs pointing to the part are usually the first priority, notes are second, and general title block notations are third. Also look for disclaimers such as "unless otherwise noted."

As you review the drawing, you may find other call outs or dimensional tolerances. If these exist, they will usually pertain to special features such as brackets or end locations. These will take precedence over general tolerance notes. The suggestions and comments outlined below regarding these five call outs assume that there is no other conflicting information.

Item I above suggests that for single decimal numbers there is a plus and a minus 0.020 inch envelope; for two decimal numbers, a ±0.010 inch envelope; and for three decimal numbers, a ±0.005 inch envelope. If there are no other call outs, it is safe to assume these values hold true also for linear tolerance. All X, Y, Z data is shown at three decimal places or greater. Usually, the tube centerline information will not vary and will probably be either one or two decimal numbers. Three decimal numbers may be used on bracket details, bead sizes, or other various controlled features. There are usually fewer questions and less confusion with this system than others, and it allows the designers greater control.

Item II above suggests that when an element of the tube body is under 100mm there will be a ±2mm tolerance; if greater than 100mm, there will be a ±3mm tolerance, and dimensions greater than 300mm; there will be a ±6mm tolerance. We have experienced three interpretations of this. One suggests that if any point on the tube, in any axis, (*but limited to axis movements*), passes the steps, then the tube envelope will increase to the next step. For instance, if point 8 is

greater than 300mm in the Y axis, only then would the envelope increase to ±6mm for the balance of the tube. (*Linear tolerance will probably be called out elsewhere)*.

The second theory suggests that if a point passes the step value in any axis, the tolerance there may be larger, but only in that axis and only as long as the points remain over the step value in that axis. For instance, if point 8 is over 300mm in the Y-axis, then the tolerance envelope will stretch so that the X and Z values remain at ±2mm. If at point 10 the Y-axis returns under 100mm and the X-axis is greater than that, then the cavity will be stretched in another fashion. This interpretation is more easily constructed than it is verified, especially if you do not have a Coordinate Measuring Machine. What is more important is that, it doesn't follow the intent of the tolerance table. The table suggests that the longer the part is, the more difficult it is to bend. In the above interpretation, you could easily have a 12-foot tube with 30 bends that is held to the first tolerance step of ±2mm in the Y and Z-axes. I do not believe that is the intent.

The third interpretation is to follow the centerline and after 100mm of tubing, there is a sudden change in cavity tolerance from ±2mm to ±3mm. Likewise at 301mm, the cavity will change to ±6mm. I believe this allows for the greatest manufacturing tolerance and is a very legitimate argument given this type of table.

Item III on the previous page is generally interpreted as envelope tolerance and linear tolerance, as well as everything else being ±0.030. This is satisfactory if it is the designer's intent.

Item IV on the previous page is looking at the way the part is bent, which is acceptable; but in practical situations this type of tolerancing will always lead to the "stacking" of tolerances. If a tube has 8 bends and each one makes full use of the one degree, and if the straight lengths are very long, you can end up with a exceptionally large envelope at the finished end, and still satisfy the 1 degree requirement.

Item V on the previous page is a geometric tolerance callout, which means there is a total allowable deviation of 0.060 inch. This symbol is usually pointing to the surface of the tube, but conceptually, it is more helpful to think of it controlling the centerline. It means the total positional movement of the centerline can be no more than 0.060 inch, (*±0.030 from a nominal position*).

INTRODUCTION TO FIXTURE DESIGN

In this section we examine a variety of considerations that may impact the fixture design. We will also look at a variety of specific symbols and typical notes plus discuss their meanings.

My hope is that at the end of this section, you will have a systematic process for designing fixtures and will have an appreciation for the implications of different approaches. This will be helpful in making meaningful choices of fixture details found in the later sections.

METHODOLOGY OF FIXTURE DESIGN

"Understand the Drawing"

As stated before, **fixture design is an exercise in interpretation and application of tolerancing.** The drawing symbols, notes, and dimensions are the final word regarding the tube configuration. The drawing is the best single tool at your disposal to help you build a part that fits your customer’s needs. It is also the foundation on which to stand if your customer rejects your parts. In theory, if you build to their documentation and design, and the part does not function well, the responsibility rests with them and not with you. You will have fulfilled your obligation in producing the tube or assembly. Because we are striving to be excellent suppliers, the design criterion needs to involve the drawing as well as other considerations.

It is helpful to consider various aspects such as your customers' requirements, their assembly methods, what inspection methods the tube may be required to pass, and your manufacturing capabilities. You may also need clarification of the drawing intent, and revisions may be in order. It is important to remember that the drawing may be your only recourse if a problem arises. It is also essential to know whether your interpretation of the drawing fits the function of the part, and whether your process enhances or detracts from that function. The fixture can be designed to allow only parts that fit these requirements to pass the fixture constraints and expose parts that don't pass. The tube or assembly fixture should be designed in a manner that is consistent with the drawing intent and that suits your manufacturing process. This may require a set of fixtures to follow the part through various phases of completion. A more tightly toleranced fixture may be required at the first stages of a complex assembly, and a fully toleranced fixture as a final check after welding and construction.

After gathering all available information, you must insure that the design intent matches the drawing symbols to determine what type of fixture or features will be required. Some of these interpretations of symbols should be discussed with your client. Actual fixture design becomes an exercise in applying tolerances. The final fixture should allow manufacturing full use of all allowable tolerances but restrict manufacturing in every area called for by the drawing. There should be no need to arbitrarily limit manufacturing to reduced tolerances simply because the users "don't have time" or are untrained to use full tolerance. The fixtures can be designed to clearly expose good and bad parts. It should not be left to the subjective decision of an operator. By observing the following guidelines you will possess the ability to give operators tools that make their jobs easier and more precise.

STEP ONE – LOCATE AND UNDERSTAND THE DATUM SCHEME

The first step is to locate all datums and determine their order of importance. If Geometric Tolerancing is used, this is fairly easy. First, look for the primary datum [-A-], then secondary [-B-], etc. In simple terms, these symbols mean that the part must contact these datum planes but not pass through them. At the point where the tube touches the datum plane, there is no tolerance. A good practice here is to highlight datum points. Second, you'll need to consider datums' relationships to each other. Is [-B-] parallel (∥) to [-A-]; is [-D-] perpendicular (⊥) to [-B-]; is [-F-] concentric (O) to [-C-]? In the section containing fixture details, you will find a variety of solutions to various datum schemes.

These symbols in the tubing industry will usually be called out on the tube end or on appendages such as brackets. There may also be an overall positional call out which can be thought of in terms of an envelope tolerance surrounding the tube body. Here is an example of a call out. (See Fig. 6 on the next page also).

⌖	A	0.060	Ⓜ

In terms of tubing, this means that the tube centerline, (*although it may be pointing to the surface)*, cannot be out of true position by more than a total of 0.060, considering the tube in a maximum material condition. Another way to write this would be an envelope tolerance of ±0.030 (0.030 for one side of the envelope and 0.030 for the other = 0.060).

The second type of datum to look for would be target datums (C1) (see Fig. 6 on the next page). These are points at which there is no tolerance and should be taken seriously, as they may indicate some area of interference in the final assembly. (*Tubing is generally part of some larger assembly*). Placing a round pin next to the tube in the proper orientation, usually perpendicular to the tube centerline, can easily control these features. The intersection of the round pin to the round tube creates a point.

If geometric tolerancing is not given, you should decide whether the zero plane must be considered datum or only a reference for the dimensions. There is at least the inference of zero-zero being a datum point. It is often helpful at this point to know if and how your customer will be inspecting the parts. If there is a datum point, you need to determine whether it is in one, two, or three axes, and then pick an appropriate control detail. (*Be careful not to stack up tolerances of attachments to the tube. Valves, brackets, clamps, etc., may have their own tolerance*). It is helpful in terms of manufacturing to begin bending at this end, if possible.

Another possibility is to have tolerance at both ends. This is the norm in rubber hoses and is not uncommon in metal tubing. Whether or not to use the implied datum in this type of dimensioning is often best determined by your customer.

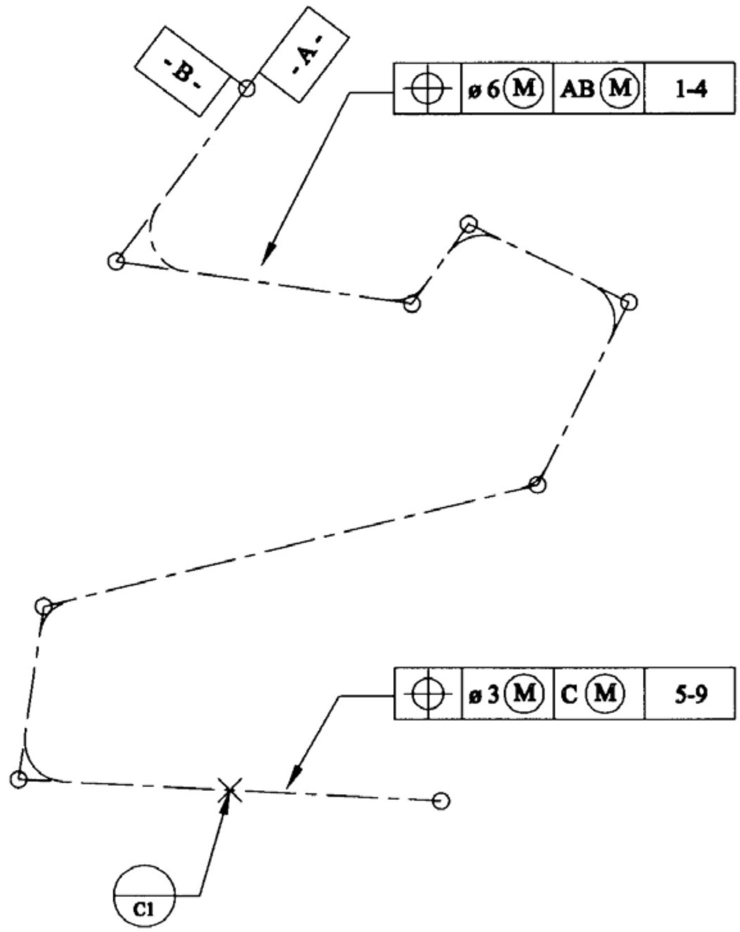


Fig. 6

NOTES:

1. Lock down Pt. 1.
2. From Pt. 1 - Pt. 4 use envelope tolerance of $\pm$3mm.
3. From Pt. 5 - Pt. 9 use E.T. of $\pm$1.5mm.
4. Hold Pt. 9

STEP TWO - LOCATE "SPECIAL FEATURES"

The next area of concern is the treatment of special features and attachments such as brackets. The avoidance of stacking tolerances is the primary focus here. Many features have their own independent tolerancing schemes and are often referenced to some point on the tube body. The tolerances are allowed to move with the tube in a positional tolerance, and also have their own tolerances. The simplest approach is to add both tolerances together and pocket that area, restricting the maximum allowable movement. The problem with this approach is that, while it is functional and often acceptable, it is not really correct. The feature tolerance should be treated separately. For example, if your tube fits the fixture in the maximum "plus" position and a bracket is attached to the tube in its maximum allowable negative position, it will be incorrect by the positional tolerance allowed in the tube body. The correct way to resolve this problem is to construct a floating detail that correctly measures feature size independent of envelope tolerance. An example of this conflict is a pin that has free movement within the tube position envelope of $\pm$0.030 inch but has a diameter of only 0.002 inch below the bracket hole. This method will insure the bracket hole is the allowable size, and at the same time make provision for positional variance. This type of detail is standardized only in its simplest form because of the generally unique nature of it. These types of features often come at great expense in order to correctly achieve the drawing intent.

Two other solutions should be discussed. One mentioned earlier is to simply determine the maximum allowable position by adding feature tolerance and positional tolerance, then limiting movement to that size. This is appropriate where the feature is non-critical or it has been predetermined that the feature is correct and you are looking only for attachment position. This is usually the least expensive solution.

Another approach is to treat the feature as a tertiary datum and provide a reduced positional tolerance for that area. This approach may be appropriate for some bracket treatments. The concern is that you may be limiting functional use of more important tolerancing by restricting this area. For example, if you have a primary datum at one end and then restrict the bracket position further up the tube, you may have limited your positional tolerance between the end of the tube and the bracket. This limitation is appropriate when called for in the drawing, but care should be taken to insure you are not restricting manufacturing more than necessary.

STEP THREE - ORIENTATION

After understanding the datum scheme, you should decide which orientation is the best for the part. All fixtures, both component and final for a given part assembly, should be in the same orientation.

Some orientation considerations include:

1. Have as many bends as close to true view as possible. This allows the operator the best opportunity to correct any deviations.
2. Gravity will play a part in how the tube fits the fixture.
3. It is also helpful if the datum points are accessible and there are not many "apparatus" requirements to satisfy the datum.
4. Account for overall size. A horizontal fixture is easier to use than a vertical one.

5. All fixtures (bend fixture, weld fixture assembly, or final audit fixture) should have the same orientation or rotation, when possible.

Other features such as nuts, beads, sleeves, etc., may be considered as everything from being a datum to being non-existent, depending on the stage of manufacturing this particular fixture will be used. Such features are often treated with clearance pockets and scribed lines showing their location with or without tolerance.

STEP FOUR - END DETAILS AND GAUGE STYLES

The next step is to determine which gauge style best meets your process requirements, as well as which end and supplemental details should be specified to satisfy your process, and to insure drawing compliance. Both of these are discussed in the next section.

GAUGE FUNCTION

"In-Process Gauging"

Once the drawing has been reviewed, datum schemes & tolerances are understood, and all attachments & their relationships to the tube are identified, it is important to consider where your particular part should be inspected in your company's manufacturing process.

This decision has much to do with cost (*including time as a function of money*). There is a good argument for saying that after every operation there should be controls to verify the operation's correctness. This will eliminate value adding to poor quality parts. The other extreme says, "as long as the part does not go out the door incorrect, we're satisfied." Some operations have been observed where there are no checks other than the production forming tools. The belief is that if the *process* is controlled, the results can then be assumed.

The gauge designer needs to be aware of the company's processes. Maintaining this awareness, however, is a dynamic task. Processes change over time. The theoretical goal is total process control. However, until there is total control of everything from incoming material, tooling wear, machine deviations, and personnel issues, some type of part inspection is called for.

Unless parts are simple and do not require much assembly, welding, piercing, trimming, or other secondary processes, the argument for multiple fixtures is as follows: The prime consideration is to limit value being added to parts that are out of specification. If the parts are incorrectly made at the first process, all the labor, material, and time spent on preceding processes will be wasted. This not only costs money, but also often causes delivery schedule problems. Many of our customers purchase a "no frills" fixture with a limited tolerance envelope to be used right at the bender. The few hundred dollars spent here saves thousands of dollars being wasted and helps meet delivery schedules, as well as preventing embarrassment and poor customer confidence.

In-process fixturing also helps in controlling the production process and exposing the weaknesses. This may be done to isolate the variables in a problem part or to better understand a process.

Some of the difficulty in justifying the money spent on gauges is that few accounting processes are not set up to measure money that has been saved. Final profit numbers are measured, but there is usually no value recorded for smoother operations. Quality and delivery targets are then easier to obtain. There are also on-time delivery reports, but nothing that captures the reduced frustration levels.

In-process gauging is an excellent way to limit unnecessary costs being added to inferior parts, and to isolate variables in order to control your process.

OPERATOR ERROR

Another consideration is *who* will be using a particular fixture. The repeatability and reliability of the fixture has much to do with who will be using the gauge as well as how it will be

used. To insure parts are being loaded in a consistent manner, vee-blocks, clamps, gauge instructions, or electronic sensors may be used.

The vee-block also captures the part to insure that it does not move. This design is the same as the letter "V". A v-shaped cavity is created to hold the part steadily. Clamps do two things. First, they insure the part is being loaded and held with the same pressure and that the pressure is in the same location relative to the fixture, not necessarily to the part. The second function is that the part will be held in the clamped position while the part is being "read" in the gauge. The operator cannot knowingly, or unknowingly, rotate or shift the tube's position in the fixture as he looks at one end and then the other. This is very important for repeatability. Clamps are often used with other features to insure consistent loading position from operator to operator. Written gauge instructions attached to the base serve as a handy reference for the operator. Gauge instructions can be as simple as "load here first" to complete instructional diagrams. The complexity and use of instructions depends on operator training, skill, and corporate protocol. Electronic sensors help eliminate operator error by more precisely determining the placement of the part being checked.

MATERIAL

The last issue to address is material. Material considerations include the number of parts to be inspected, the inspection process, expected fixture accuracy, size of the part, (*weight as well as physical size*), type of use it will endure, style of fixture, delivery time of the fixture, and of course, cost.

Aside from machinable ceramics and other exotic materials, steels will provide the longest wearing, closest tolerance fixtures. However, the price will increase from hundreds of dollars to thousands and the weight for even small assemblies can require a rolling cart or forklift to move them. In addition, even steel fixtures are subject to wear on high volume parts or parts with abrasive details such as powder metal blocks.

At this time, we do not believe there is a "perfect" material that satisfies price, machinability, wearability, weight, and the ability to hold close tolerances. Some might say that there is not even a "good" material. We believe that with the use of a variety of materials, there are acceptable compromises to meet all situations. We are always searching for better material solutions and believe that there will eventually be a material developed that will better satisfy the various demands of our industry.

At this time we recommend one of three materials as the parent or base material (*see Appendix A for material qualities*): aluminum, especially where high temperature is a factor; a plastic urethane foamed fixture board for general fixturing, (*this material when used with pockets of other materials will satisfy 80%-90% of all automotive and aerospace tube fixturing*), and laminated wood when tolerances are greater than a few thousandths and fixture life is not as long. To get the most out of wood, it is important to use as few single blocks as possible. Wood construction should consist of cutting or removing the part out of a single block rather than building with a cluster of small pieces. The reason is obvious in terms of longevity and accuracy. Uses for the laminated wood plank, like the fixture plank, can be greatly increased by incorporating it with other materials. Where there are high wear areas, such as brackets or any metal corners, blocks of Delrin, UHMW, or steel can be added and the part cavity machined

through them. For fixtures exposed to high temperatures, such as welding, hi-temperature phenolic blocks can be added. Sometimes these may even be added to wood fixtures to obtain excellent results at a substantially reduced cost, in addition to quicker delivery.

To reduce the total cost and the weight of large fixtures, we would recommend in most cases, an aluminum honeycomb base instead of an aluminum or steel plate, even over blocked wood construction. Honeycomb will provide great strength at a reduced weight and in most cases, will also save money.

The fixture material deserves to be looked at closely. With a little care and thought, the fixture may be made much more serviceable for a much longer period of time and at a lower cost.

A SUMMARY OF FIXTURE DESIGN CONSIDERATIONS:

I. Obtain all information pertaining to the part to be checked.
II. Establish gauging schedule - what gauge will be needed where.
III. Analyze tolerances - watch for stacking tolerance.
IV. Determine cavity diameter/tolerance.
V. Review special features.
VI. Pick appropriate details, materials, and gauge style.
VII. Review the gauging schedule to see if it meets your processes.

Remember, fixture design is an exercise in applying tolerance.

GAUGE STYLES

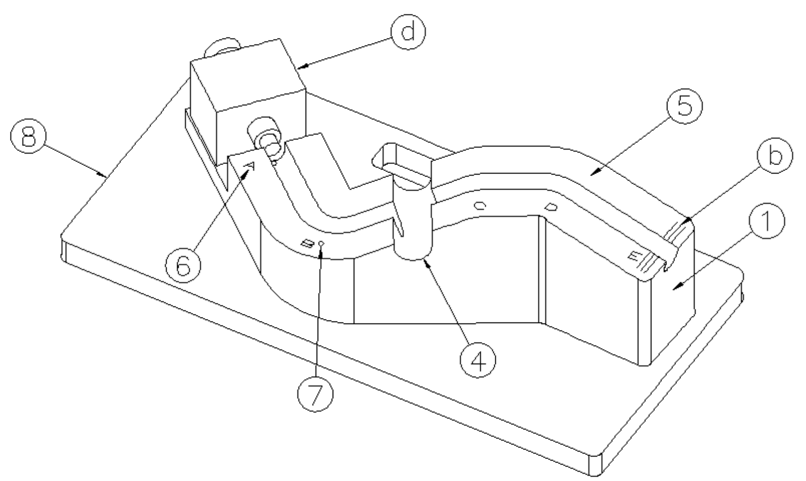
"Full-Contour Illustration"

A full-contour gauge is a gauge that represents the theoretical tolerance envelope with a hard feature so as to reject the tubes or assemblies that do not meet the requirement. Not only does it include only the straights, bends, only one side or the other, but the *entire* envelope.

Full contour fixturing is one of the best ways to insure that inspected parts meet the drawing requirements. If the parts fall (without being forced) into the gauge and do not exceed the envelope top surface, then they have fulfilled the requirement of shape.

Many options are available to gauge how much tolerance is used at a specific point on the fixture and to capture other features that may be added. Care must be taken to not build in more than the drawing would allow. There may also be reasons to allow less in certain areas.

The function of a full contour fixture is to insure correct parts are being produced and to eliminate the operator's need to make judgment calls on the accuracy of the part. It is by nature fail-safe, or "poke-yoke".



Product Features:	Dimensional Data Required:
1. Material (Fixture Plank, Jelutong, Mahogany, Ren470)	* Data Sheet
2. Part Rotation (to Print, Dynamic)	* Cavity Diameter (Include Tolerance of End Feature)
3. Full Contour, Full Diameter Height = (1/1) ratio or Variable	* CMM Data
4. Finger Slots for easy part removal	**Optional Features**
5. Controlled <u>upper surface</u> 1.0" minimum width	a. Alignment Bowls or Tooling Ball
6. All data points labeled on fixture itself	b. Green Zones at specified locations
7. 0.125 " drill holes at each intersection point	c. Datum Options
8. Base Material – ¾" Birch Plywood, Fixture Plank,	d. Endpoint and other Tolerancing Solutions
Jelutong, Mahogany, Aluminum, Honeycomb	e. Inserts – Metal – Delrin – Ren470
	f. CMM Report
	g. Drawing

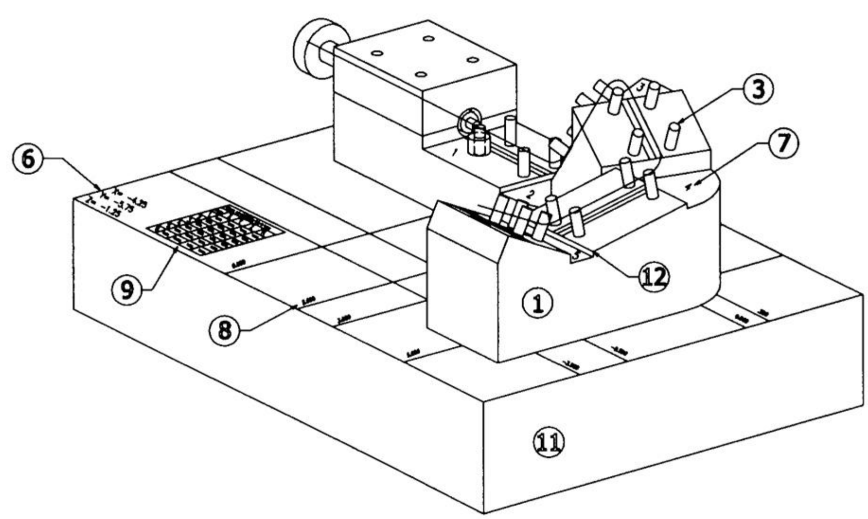
BENDER SET UP

A bender set-up gauge is a gauge that is usually built with nominal tolerance, and is oriented to bend data. It will have a flat bottom and one sidewall with the opposite side open. The side block will be the length of each straight, tangent to tangent.

This is similar to the old traditional style of gauge and is valuable in setting up a dedicated bender. The machinists building the bender are highly skilled and have more time than a production worker. They are constantly working toward producing a perfect part. Since the gauge is set at nominal, this will allow them to do this.

This bender set-up gauge should be thought of as bender tooling and not production tooling. When the bender is shipped, the gauge should be set aside and used only to reset the bender. The groups of small blocks do not hold up as well in a production environment. If used in production when a bender needs to be retooled or adjusted for wear, the fixture is in doubtful condition to perform its best function.

For this style of guage to also be used in production, it should be designed with the pins positioned at full tolerance. Bushings that slide over the pine and are removable, are made to remove the tolerance both in the Z-axis and side to side. This will allow the dedicated bender builder the flexibility needed to construct the equipment.



Product Features:

1. Material (Fixture Plank, Mahogany)
2. Part Rotation (to Print, 90°, Dynamic)
3. Steel Dowels Gauge Full Height to Width ratio
4. Steel bushings easily added for "zero" gauging
5. Green Tolerance Zones Endpoint locations
6. Square Corner with labeling for verification "start" position
7. Coordinate point locations identified with number/letter
8. X and Y scribe lines on the base for key points
9. Tube Part X, Y, Z, R data table labeled on base
10. 0.125" drill holes at each intersection point
11. 2.00" Jelutong Base
12. Scribe lines for centerline nominal tube and tangencies

Dimensional Data Required:

* X, Y, Z, R Data Points
* Cavity Diameter (Including Tolerance of End Features)
* Linear (Length) Positive Tolerance
* Linear (Length) Negative Tolerance

Optional Features:

a. Endpoint Datum Options
b. Endpoint and Other Tolerancing Solutions
c. Mechanical SPC
d. Electronic SPC
e. Saddle Blocks instead of Dowels
f. Radius can be taken out

ZERO-TOLERANCE

If you use a zero-tolerance gauge for production, you run the risk of shipping incorrect parts, and scrapping or reworking good parts.

In Figure 7 below, the cross section of a perfect part in a zero tolerance fixture is shown. It is rare to consistently produce perfect parts. For purposes of discussion, we will confine the deviation to four directions: up, down, left, and right. The problems are compounded when you consider several bends over the length of the tube. If the deviation is either up or to the left (see Fig. 7) a significant amount, a good operator on a good day will catch it. The operator will need to decide if it has deviated the allowable amount or more. This may not be difficult in this example, especially if they have a feeler gauge time to evaluate the part. Next, compound this by several bends, and one can realize it becomes a much more difficult job. In the process of measuring the part, the operator has to ascertain whether the part orientation to the gauge changes as it is reviewed from one end to the other. This is an example of where a clamp would be useful.

The truth is that the process above usually does not happen. The operator lays the tube in the gauge and quickly checks for any gross variation. If any are found, they may hold the part to the fixture (not deforming the part, but rocking it) in a different area and see if that particular section of the tube fits better. The operator then may or may not observe the results to the rest of the tube. This is not to question the operator's ability, but if this type of fixture is used, their job has been made more difficult, and they have been given a task without the appropriate tools to accomplish it.

The other problem with this type of zero-tolerance fixture is if the part moves an allowable deviation on the right side or down, it will force the rest of the tube out of position. Depending on the degree of bend and the amount of straight tube, it will likely show a good part to be incorrect. In most situations, a better approach is to use the full contour gauge as shown on page 30.

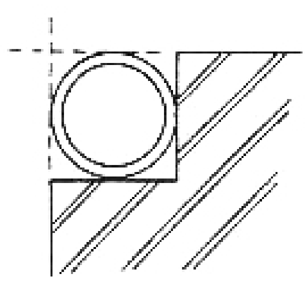


Fig. A
Perfect Part
In Zero Tolerance Fixture

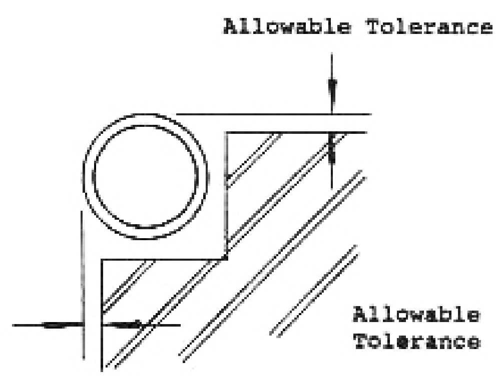


Fig. B
Good Part
In Zero Tolerance Fixture

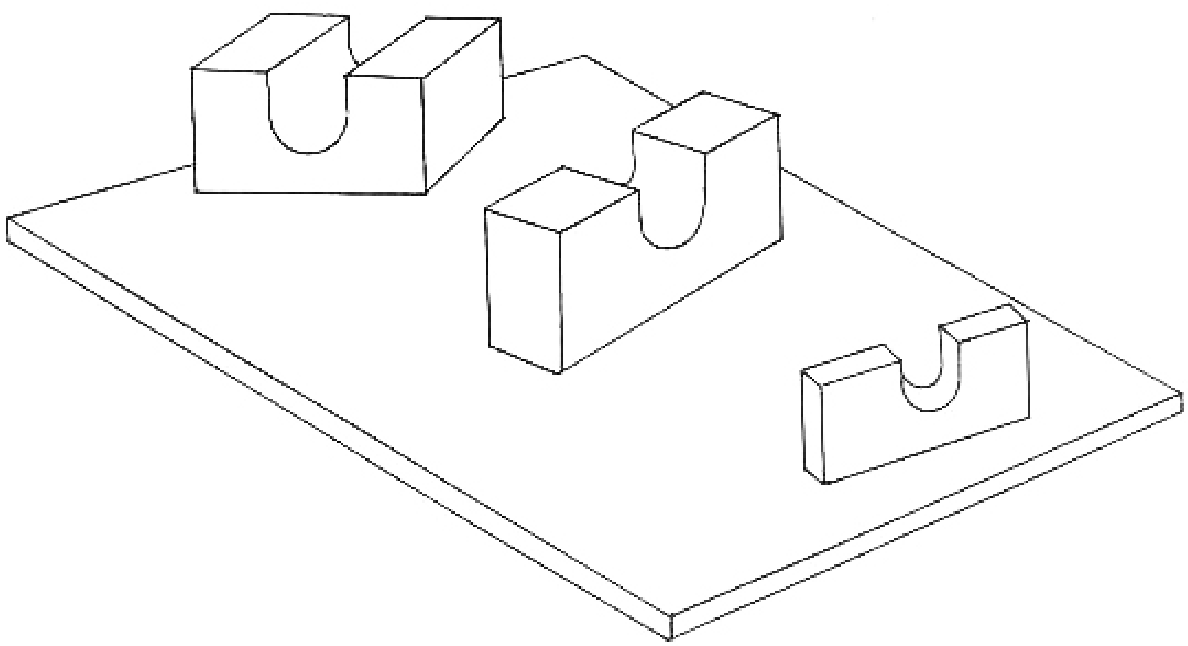
Fig. 7

SADDLE or GATE STYLE

Another common style of gauge utilizes a saddle or gate through which the straight portions of the tube must pass. These gauges usually have full tolerance depending on their function. The gates generally are 1/2" to 1" wide and are oriented on stands that hold them in the correct vector position. Because of the many tall, narrow stands, these fixtures are generally built of steel, which makes them heavy and expensive, or rigid plastics, which are less expensive but not as durable.

By controlling only a small segment of a straight and no portion of the bend, this only insures that the tube fits the general shape desired. This type gauge is not effective in verifying a tube to a close tolerance. The distance between straights, as well as the degree of bend, may vary and still be accepted by the fixture. In some industries or certain applications, this type of fixture may be acceptable. Automotive and aerospace parts that are robotically welded require closer tolerances than can be insured by saddle gauges.

We recommend this style fixture when *only* the end points are critical. If the tube has been previously verified more closely, this gauge may serve as a final check for the tube.



VARIABLE DATA CONTROL

"The Concept"

Variable data collection can be as valuable to tube bending as in any industry. You can watch for tool wear, changes in material specs, operator variables, etc. This information can be obtained at a reasonable cost on a production fixture. The basic process follows classic statistical process control concepts. Because there are many good books on the subject, the interest here is not in what to do with the data, but in how to collect it.

Tube bending has a number of variables, each of which may have a bearing on the other, and can compound any problems. The solution lies in understanding your process and the placement of the data collection device. Given your process and the particular job you want to check, will the results and errors be different if a particular part fails or begins to fail? For example, if the bending tool wears, will the straights be longer, or if the material comes in harder, will the bends not be as deep? Variable data collection will help in tracing problems once a deviation is seen.

Another approach suggests that if you collect data from only one point, you will expose process changes. While this makes the decision of placement simpler, it may or may not be helpful in isolating a problem.

Whichever method is used, it will be important to design a fixture that has good repeatability and has some datum or zero reference. The collected information is more reliable when the degree of repeatability is high. **Unless the part is loaded or positioned in the fixture in a consistent manner, you will not know if the collected data is a result of a process change or simply a difference in the loading of the part into the fixture.**

A datum point, zero reference, or net surface is important because what you are trying to measure is the deviation from one feature to another. Another method is to use two collection devices, allowing deviation in both areas, and calculating the difference between them. Without the relationship from some fixed feature to the measured feature, your information is suspect i.e., obtaining a raw number without relationship to other features on the tube is of little value.

With the instruments that are currently available, it is relatively inexpensive to collect data from rotational relationships as well as linear measurements. Rotational relations would be bend plane to bend plane, the "phase angle" or angular relationship between the two tubes in a hose assembly or the angular position of an end block or bracket position. Examples of linear measurements would be end lengths and bracket position. If you lock the leg of a bend, you could use a linear device to measure the deviation in the degree of bend, or to measure the position of any feature along the body of the tube.

It is still difficult to directly measure the length of a particular straight on the body of the tube without some calculations because there are no "hard" features to get a measurement from. If this were desired, a good approach would be to use a Coordinate Measuring Machine or tube reader.

The output from these devices can be in a variety of formats: Dial Indicator, LED, or hard copy printout with date-time coding, which can be downloaded directly to a computer for full SPC analysis.

FIXTURE DETAILS

Introduction

Fixture details are the features that will make the fixture come alive. These are the features that transform the drawing restraints and parameters into a functional tool that is easily operated and understood by the person who will have to use it. The fixture will be used to grade the process as well as the operator. Its clarity and ease of use will have much to do with the ability to produce correct parts at a minimum frustration level. The following examples are by no means the only solutions. (Liberty should be taken to mix, match, and utilize whatever options you need to conform the fixture to the drawing parameters or to your particular function.)

Following are several common options with a description of their purposes and functions, and some occasional comments.

Pin Materials

For a high volume part, steel or possibly tool steel is an obvious choice. In most applications where steel is used, it is a good practice to harden it or, in some way prevent it from rusting, or use stainless steel.

Delrin, UHMW (Ultra High Molecular Weight Polyethylene) REN 460, 470, 472, or other stable plastics can be used when there is concern over the possibility of marring the tube surface.

Aluminum is not recommended for most pin applications due to its wear characteristics. If a metal pin is called for, the few extra dollars spent for a hardened or stainless steel pin will be well worth it.

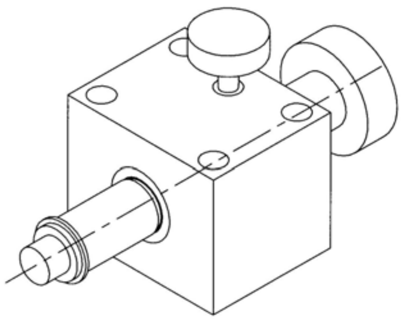
Electronics

While there is some connection between variable data collection and electronics, it is valuable to note some advantages of electronic devices in a Go/No-Go concept.

Lights or buzzers can be used to indicate good or bad parts. You may want incorrect parts to be clamped until a supervisor acknowledges them. Bracket planes can be established and will not show a correct part until three small switches are made. Switches can be placed so an end or a block needs to be "datumed" and an indicator light is turned on before the balance of the tube can be inspected. Also, you may want to insure that a particular number of parts are inspected. A switch can be placed in the fixture that will indicate how many parts have been checked.

Devices such as these can be added to the fixture to make it more useful for the operator and more consistent for production.

INSIDE DIAMETER CHECK PIN



The Inside Diameter Pin is used to check the position of an open end tube or tube assembly by referencing the tube I.D. The tube is laid into the fixture and the pin is slid into the tube end until the pin face touches the tube end.

Linear tolerance can then be read from the band or line on the pin body as referenced to some fixed detail such as the face of the pin block.

OPTIONS:

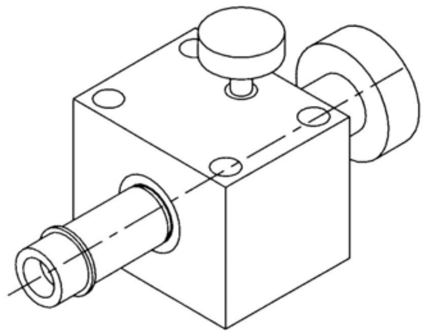
A. **INCLUDE POSITIONAL TOLERANCE** – In this case, the pin diameter "X" would be the smallest tube I.D. minus the allowed positional tolerance, (*Tube I.D. = 8mm -(±2mm) = 4mm Dia*.). The "Y" dimension, which is the I.D check length, needs to be of an appropriate length to engage the part. This is the most common use and fits well with the envelope tolerance concept.

B. **NO POSITIONAL TOLERANCE** – In this case, the "X" diameter would be the MMC with no tolerance. Often times 0.002-0.004 inches are added to more easily fit the tube. Obviously a 1.25 inch opening will not fit a 1.25 inch tube. The "Y" dimension may be determined by the drawing. There may be a note to hold some diameter for a particular distance from the tube end back. This distance could become the "Y" dimension. This is a means of eliminating envelope tolerance at the tube end.

C. **TO ELIMINATE TUBE LINEAR TOLERANCE** – The pin linear tolerance zone can only be a line. A more positive method is to build the pin with a large boss, or handle, that is constructed so that when the pin face is at the datum position, the large diameter handle will bump against some hard feature that will stop the pin travel.

D. **TO CHECK FOR AN END DETAIL ORIENTATION SUCH AS A FLANGE** – One method is to use a large diameter faced pin. In this case, the small pin diameter will pass into the flange hole, (*2 pins may be required to check flange rotation; either both with tolerance, or one with tolerance and one datum, or both datum*), and the large face of the pin will touch off on the flange face. The operator will then check for consistent contact between the pin face and the flange and will check the linear tolerance band. This method may be used with feeler gauges to obtain a more precise reading of the flange angle.

OUTSIDE DIAMETER PIN



The Outside Diameter Pin is used to check the position an open end tube or tube assembly by referencing the tube O.D.

The tube is laid into the fixture and the pin is slid over the tube end until the pin face touches the tube end. There should be at least one window, opened past the pin face, to provide a visual reference that the pin is fully engaged.

Linear tolerance can then be read from the band or line on the pin body as referenced to some fixed detail such as the bushing in which the pin slides.

OPTIONS:

A. **INCLUDE POSITIONAL TOLERANCE –** In this case, the pin diameter "X" would be the largest tube O.D. plus the allowable positional tolerance. (*Tube O.D. or MMC 8mm + (+2mm) = 12mm Dia.*). The "Y" dimension, which is the O.D check length, needs to be of an appropriate length to engage the part.

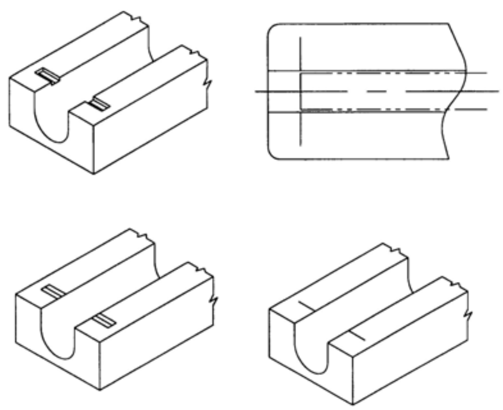
There is a good argument for not using a positional toleranced pin when using a full-contour fixture. If the walls of the fixture limit tube movement by the allowable positional amount, a full positional pin at the tube end is not only redundant, but it is also an unneeded concern for the operator. The end length can be established with a green zone.

B. **NO POSITIONAL TOLERANCE** – In this case, the "X" diameter would be the MMC with no additional tolerance. Often 0.002 to 0.004 inches are added to more easily fit the tube. Obviously, a 1.25 opening will not fit a 1.25 tube. The "Y" dimension may be determined by the drawing. There may be a note to hold some diameter for a particular distance from the tube end back. This distance could become the Y dimension. This is a means of eliminating envelope tolerance at the tube end.

C. **TO ELIMINATE TUBE LINEAR TOLERANCE** – The pin linear tolerance zone can only be a line. A more positive method is to build the pin with a large boss, or handle, that is constructed so that when the pin face is at the datum position, the large diameter handle will bump against some hard feature that will stop the pin travel.

D. **TO CHECK FOR AN END DETAIL ORIENTATION SUCH AS A FLANGE** – One method is to use a large diameter faced pin. In this case, the small pin diameter will pass into the flange hole, (*2 pins may be required to check flange rotation; either both with tolerance, or one with tolerance and one datum, or both datum*), and the large face of the pin will touch off on the flange face. The operator will then check for consistent contact between the pin face and the flange and will check the linear tolerance band. This method may be used with feeler gauges to obtain a more precise reading of the flange angle.

E. GREEN TOLERANCE ZONES



The Green Tolerance zone is used to locate the tube end.

The tube is laid in the fixture and viewed to see if the tube end falls within the green band or zone.

This is the quickest, easiest, and least expensive method to establish tube lengths. It is suitable for many more applications than is presently used, but does require minimal judgment by the operator.

OPTIONS:

BLACK LINE ONLY – NO TOLERANCE BAND

This option is acceptable in several places on non-critical parts or as a reference for parts that will later be trimmed. It assumes zero tolerance parts. A tube cannot be easily positioned accurately to a single line for a datum reference.

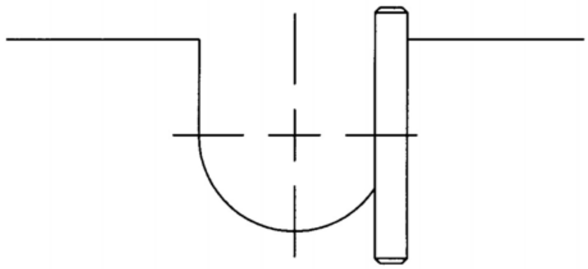
GREEN TOLERANCE

This is a tolerance zone with either a milled shelf or scribed lines, plus a black line at the nominal position. This demonstrates the idea that if the tube end falls outside the green band, it is unacceptable.

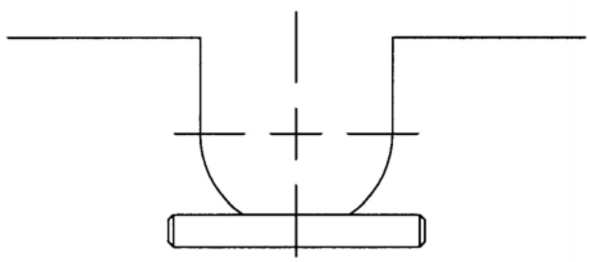
DATUM TARGET PINS

This is a pin option that will remove the envelope tolerance in a specific area in order to reflect the datum target call out on the print.

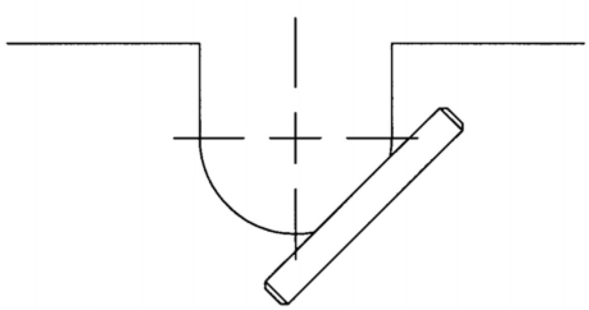
Most target datums fall on the surface of the tube. These datums are points, not surfaces or lines. By installing a pin in the proper orientation, you can locate the target point. The intersection created by the tangency of the tube diameter and the pin diameter create the datum target.



Datum Pin
Perpendicular To Cavity

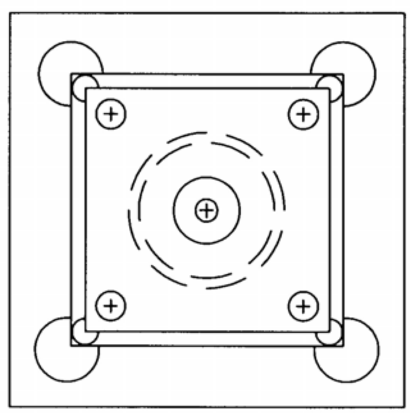


Datum Pin
Parallel To Cavity



Datum Pin
45° To Cavity

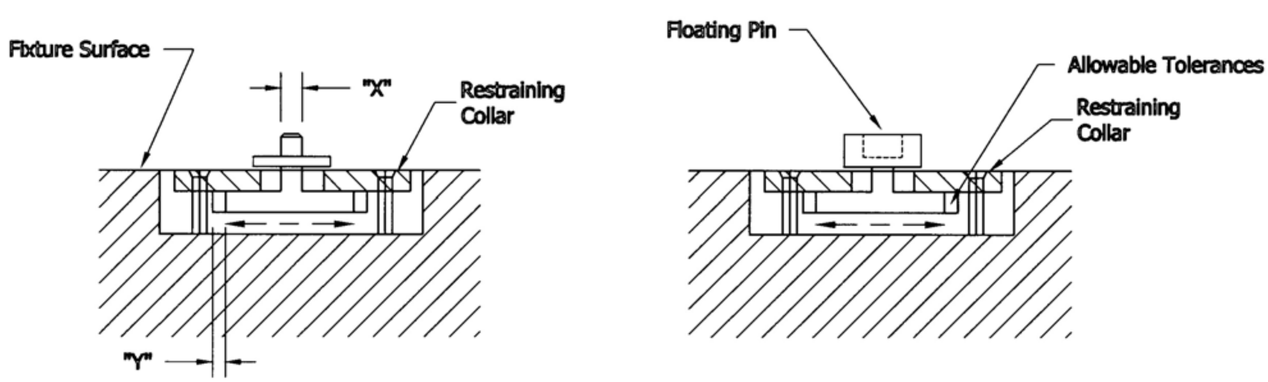
FLOATING, OR ORBITAL PINS



Top View

A floating pin is useful to check the size of a bracket or clamp, yet maintain positional tolerance. It is also useful when the tube inside or outside diameter size needs to be checked but in a way that does not interfere with the positional tolerance.

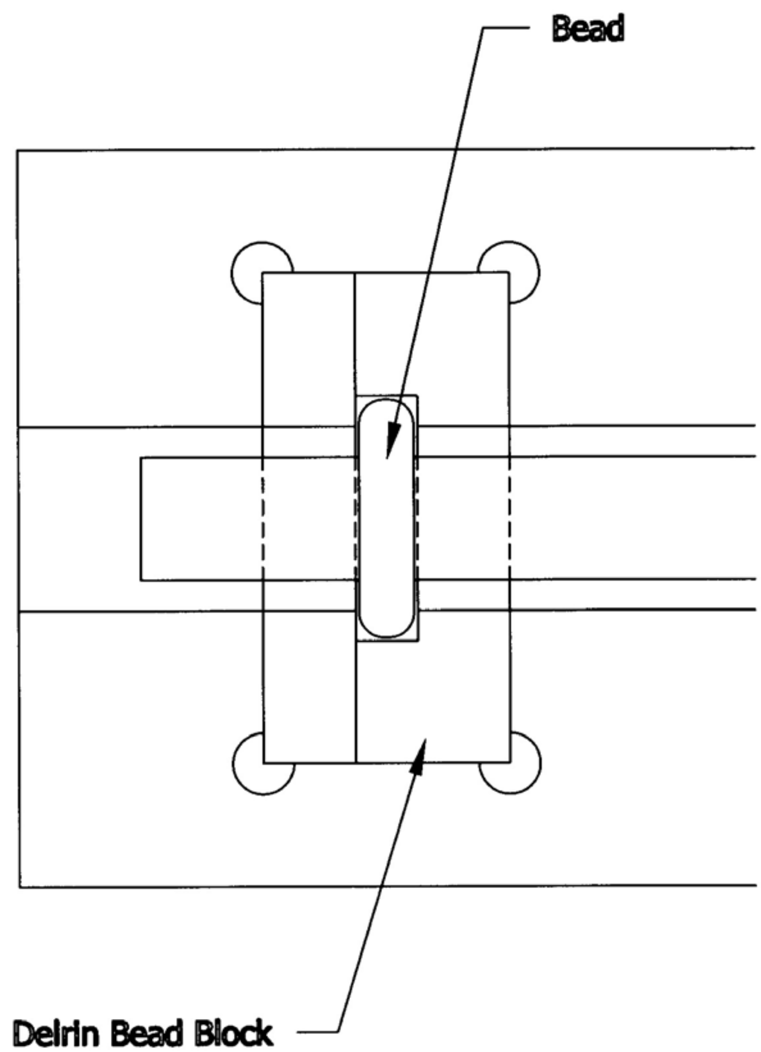
In this way, stacking of tolerances may be avoided. The pin diameter "X" is cut so that it fits the bracket hole or the end of the part. The allowable movement or float of the pin is restricted to the positional tolerance "Y".



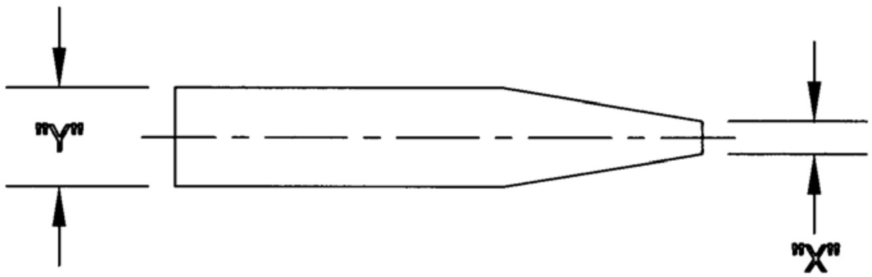
Side View

BEAD LOCATION BLOCK

One side or the other of a tube bead is often a datum that usually needs to be closely captured. This location block is precisely cut to capture that point. These details are generally constructed from a material with good wear properties such as Delrin.



TAPERED PIN



This pin has a gradual bevel at its end with the small diameter "X" being significantly less than the minimum diameter of the tube. The pin diameter "Y" is significantly larger than the maximum diameter of the tube.

The purpose of this feature is to center any type of hole. It can be used on brackets, flange blocks, etc., or even tube ends.

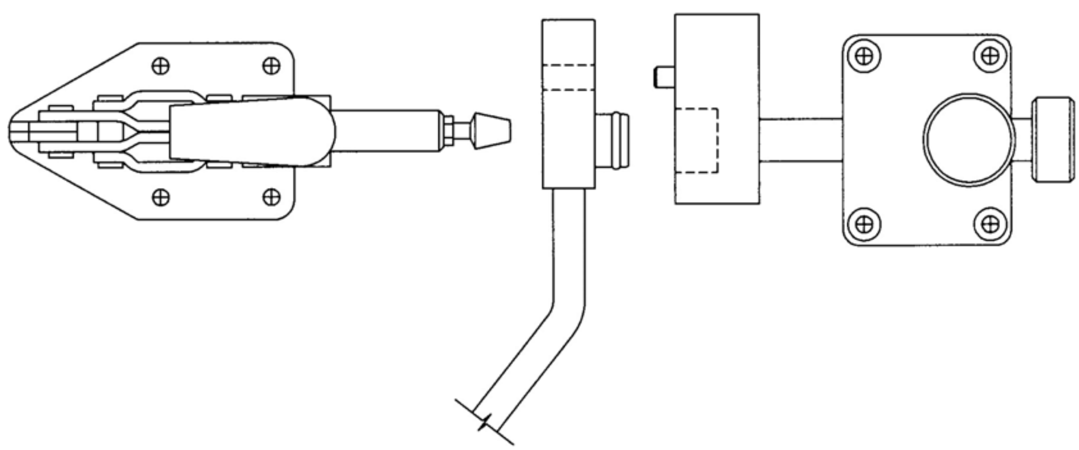
It is not useful for linear tolerances because as the diameter of the tube increases, the pin will slide further into the tube.

FLANGE OR CONDENSER BLOCK

This detail is designed to capture a flange block and verify that the tube and flange orientation to the mounting hole(s) is correct. Flange blocks usually have one face as being datum. There is generally no need to verify the entire flange block if it has been previously inspected. However it is sometimes helpful to check the profile to insure the proper block was installed.

These requirements can be resolved in a number of ways. If the block face is datum, that surface should be established with a wearable surface. There is sometimes an advantage in using a leaf type spring or a clamp to ensure positive contact.

If the flange block has a nipple, a receiving pocket (with or without tolerance) can be provided to check shape and orientation. The hole(s) can be treated with a tapered pin since you are not trying to size the hole, but only ensure that it is in the correct location.



VERIFICATION

This section discusses several concepts and general practices used with some common equipment. It is beyond the scope of this catalog to be a training manual for verification of fixtures.

Verification has the implication of communication. Like all communication, it is valuable to understand the same definition of terms so that the desired results can be expected.

The purpose of fixture verification is to insure that the fixture conforms to the design criteria. The design criteria should be similar to, but not always exactly the same, as the part drawing criteria. Oftentimes, for a variety of reasons, the design criteria may alter the drawing parameters. As we have seen, the manufacturing process may reduce tolerance or leave off features that will not be attached at the time the fixture will be used. Likewise, a final assembly fixture may not require all features if the tubes have been adequately checked previously. On the other hand, if you are inspecting the tube only once and are looking for a final audit fixture, that fixture should closely resemble the drawing. It is too simplistic to say that the fixture should be a hard replica of the drawing and contain all the information on the drawing. The point is that the fixture verification should see the drawing through the eyes of the fixture design criteria.

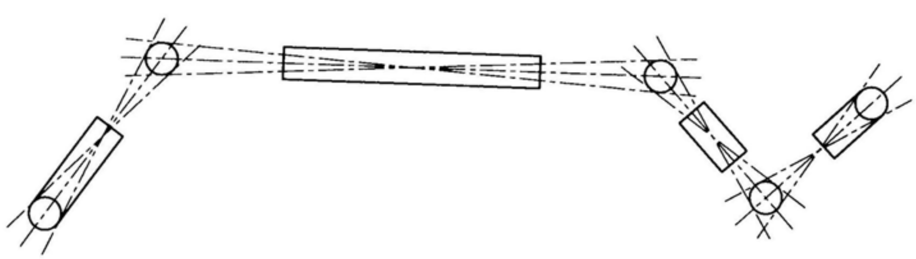
BASIC UNDERLYING CONCEPTS

There are a few general concepts that should be addressed and some understanding reached so that everyone in your organization, suppliers, and customers, can treat these issues in a similar fashion. Some of these have great implications regarding the correctness of a part. Some will have only infrequent implications, and for your particular application, may be of only theoretical implication.

INTERSECTION POINTS VERSUS CYLINDER ENVELOPE

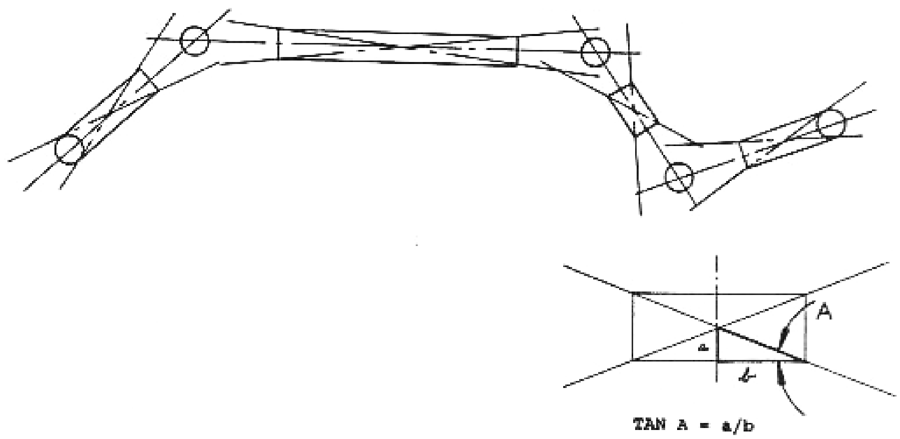
As we have seen, there is a theoretical envelope tolerance surrounding the tube. When measuring this envelope, there are at least two basic approaches that are currently being used.

The first takes a strict approach to the XYZ control points. This method inspects the fixture walls, calculates a centerline, extends two centerlines, and determines the location of the intersection point. By comparing the deviations of this location to the nominal point position, a determination is made regarding the quality of the shape (see Fig. 8)



Centerline deviation limited by Tolerance sphere around the XYZ point

The second method measures the tube or fixture surface, calculates the centerline, places the centerline vector into a cylinder, which is the allowable tolerance diameter, and determines whether or not the vector breaches the tolerance cylinder (see Fig. 9)



Centerline deviation limited by Tolerance Cylinder of the straight section

Fig. 9

<u>NOTE</u>: If the straight is reduced by 50%, the Vector angle will double.

On the surface, the two concepts may seem similar. Indeed, in some shaped tubes, this may yield the same results. However, if you push them both to their extremes and compare them, they will produce very different results. Usually, the second method will allow a greater degree of variance. Because of this, it is very important to determine which method you will use and by which method your customer will evaluate you.

The reason for the first method is that the XYZ values are the final and only control feature acceptable. There is no derived or calculated position related to them that has any bearing on the tube shape. XYZ values are on the drawing and that is what should be adhered to.

The reason for the second method is that the XYZ positions direct the way to the real tube shape. The XYZ points are often not even located on the tube, but are projections in space. The justification is that the tube vector angle is critical and if that angle is controlled within the allowable tolerance, all the features the tube should stay away from are avoided.

The first method will hold much closer tolerances. The difference is a function of how long the straight is and how deep the bend is (the deeper the bend, the farther from the tube the XYZ point is).

If the first method is used, there are other considerations with less implication. If you are controlling the XYZ point within some given tolerance, the shape the allowable movement of the points creates should be thought out. Should that shape be a cube since the point can move x amount in the XY or Z-axis, which are all perpendicular to each other? Or should the shape be a sphere since the point is allowed to shift in any direction a given amount from its fixed position?

In this document, we are not recommending one approach over the other. If you do not utilize a Coordinate Measuring Machine (CMM) or similar computerized equipment to measure your parts, you are limited to probing as accurately as possible some rotation on the fixture and comparing your measurements to the drawing. Checking an actual tube for correctness without a fixture or CMM analysis is more easily done by checking bend data, (degree of bend, etc.), and not XYZ projections.

Considering the implications, it is helpful to look at the equipment and procedures to measure tubes and tube fixtures in two areas. One process involves the use of computers to analyze information it has received about the part. The other process does not, and is limited to directly stating measured information.

When a computer is used, usually in the form of a CMM or a tube reader, the general concept is to touch off at a minimum of four locations on each straight, (the more points checked, the better, because of the ability to average surface deviations or operator errors). From these four points, a cylinder is mathematically created and the center of the cylinder is found and projected to the previous centerline. This repeated process creates the basic tube polyline. This model can then be compared with the drawing model and deviations noted. A great advantage here is that through a "best fit" program, you have the option to average the deviation by positioning the two mathematical models over each other in a way to show the least possible error. This can be done in a variety of weighted parameters, from being a totally free body, to completely restraining one or more features, such as datums.

A CMM can also analyze the part in terms of bend data. It can calculate the centerline radius, the degree of bend, etc. This tremendous flexibility also carries a responsibility to understand the limits of the constraints you may have on a particular job, as well as the implications of the measurements and the concepts behind the source code. Without this, good parts can be made to look bad and bad parts may appear acceptable. It is common to experience a tube being measured five separate times with five very different results.

When using a granite stone and a height gauge or some expanded version of this, (such as a height gauge that can slide in three axes with or without an LCD readout), you do not have the means to practically analyze your results. Without the means to analyze the measured results, you are limited in what you can do. Sometimes this can be a benefit, however, it is difficult in terms of correcting a deviation. With tubes and tube fixtures, it is not always helpful to deal with one element of the part at a time. The location of point 8 has a great deal to do with the positions of point 7 and point 9; the positions of point 2 and point 3 can also affect it. This is where a "best fit" program is helpful since it can deal with the entire model at once.

When you are using this type of measuring system, it is helpful to specify a square corner on the external surface of the fixture to position the fixture in the proper axis. It is also useful to

have steel pins with tapered points turned to the diameter of the cavity, or just under. Two of these pins can be placed in the adjoining cavities and the points on the pins brought together. This will provide a close approximation to the XYZ point. When measuring this point, it is good to have probes with sharp points and some that have angles so that all positions on the fixture can be reached.

It should be noted that due to the variables with this method, even with a qualified operator, is to some degree is only an approximation. Some of the variables include how well the pins fit the cavity, how sharp the points are, how closely they can be matched, and how closely the probes can be positioned. It would be good to analyze these variables to determine the accuracy and repeatability of your system and determine if that level of accuracy is acceptable.

Tube and fixture verification is not an easy quick matter. It should be performed with care, understanding the limits of the system used. If the results are not repeatable, a better method, such as CMM, should be determined and implemented.

APPENDIX A

HARDNESS CONVERSION TABLE

Material	Hardness	Abrasion	Specific Gravity	Pounds per Cubic Foot	Coefficient of Linear Expansion per Unit Length per Degree F.	Coefficient of Static Friction
Sugar Pine			0.35	25		Wood on Metal 0.20 – 0.60
Cherry			0.47	35		
Mahogany	Janka 800#		0.46	40		
Urethane (Modulan) Fixture Plank	Shore 65-70	0.045			0.000027	
Aluminum	B60 Rockwell		2.70	168.50	0.00001244	Aluminum on Aluminum 1.35
Mild Steel 121/15	B67/80			490	0.00000633	0.80
Stainless	B80	84				
Tool Steel 271/319	C-28/34	100	17.69	512		
Teflon						0.04
Jelutong	Janka 390#	10	0.46	29		
UHMW *	R64			58.75	0.000072	0.12
Delrin						0.14
Nylon	R120	50		70.75	0.00004	0.35

*UHMW = Ultra High Molecular Weight Polyethylene Plastic

These values, due to the nature of deviations in the material, may not be exact in every situation and should be looked at for comparison.

APPENDIX B

SYMBOL FOR:	ANSI Y14.5	ISO
STRAIGHTNESS	⏤	⏤
FLATNESS	⏥	⏥
CIRCULARITY	○	○
CYLINDRICITY	⌭	⌭
PROFILE OF A LINE	⌒	⌒
PROFILE OF A SURFACE	⌓	⌓
ALL AROUND-PROFILE		NONE
ANGULARITY	∠	∠
PERPENDICULARITY	⊥	⊥
PARALLELISM	//	//
POSITION	⌖	⌖
CONCENTRICITY/COAXIALITY	◎	◎
SYMMETRY	NONE	⌯
CIRCULAR RUNOUT	↗	↗
TOTAL RUNOUT	⌰	⌰
AT MAXIMUM MATERIAL CONDITION	M	M
AT LEAST MATERIAL CONDITION	L	NONE
REGARDLESS OF FEATURE SIZE	S	NONE
PROJECTED TOLERANCE ZONES	P	P
DIAMETER	∅	∅
BASIC DIMENSION	$\boxed{50}$	$\boxed{50}$
REFERENCE DIMENSION	(50)	(50)
DATUM FEATURE	$\boxed{\text{-A-}}$	OR $\boxed{A}$
DATUM TARGET	∅6 / A1	∅6 / A1
TARGET POINT	×	×

APPENDIX C

GLOSSARY OF TERMS

It should be understood that these definitions are of common usage in the tube bending industry and if taken out of that context, may need to be refined or expanded.

BEND DATA: This is a numeric system used to define tube shapes. The three components are the degree of bend, plane of bend, and the distance between bends.

BEST FIT: A mathematical method used to calculate or position a real tube centerline to the closest possible placement on the nominal centerline, which is done by comparing individual elements and the whole model. This method can be restrained to include datums or weighted values.

CMM: Coordinate Measuring Machine.

DATUM: A datum is a fixed point or plane with no tolerances. It is a starting or reference location from which other points can be measured.

DATUM TARGET: A datum target is a point or area on a part that defines a datum plane. An "X" marks them.

ENVELOPE TOLERANCE: A theoretical space between maximum material condition of the surface of a perfectly shaped part and an imaginary surface a fixed distance from the tube surface which completely surrounds the tube.

LINEAR TOLERANCE: The allowable deviation located at the ends of the tube.

L.R.A: Tube geometry abbreviation for Length, Rotation, and Angle.

MMC: The maximum material condition. The largest, allowable size the tube material can be and still be acceptable.

NET SURFACE: A surface with no allowable tolerance.

POKE YOKE: A manufacturing tool that forces the operator or process to complete correct parts. In fixturing, a "go-no go" or "fool proofing" fixture may be said to be Poke Yoke.

REPEATABILITY: The idea that a fixture can produce consistent results regardless of operator. Repeatability is necessary when obtaining variable data. It is more important in terms of datums and is less important in Poke Yoke or "fail safe" fixturing.

SPC: Statistical Process Control is a numerical method of charts or graphics that describe process changes for the purpose of quality control.

TRUE VIEW: The position of a bend that is normal or perpendicular to the line of sight. When the curve defining a bend is "flat", it is said to be in "true view".

VECTOR OR VECTOR ANGLE: A Vector is a line composed of a direction and a length

APPENDIX D

"MILITARY SPECIFICATION"

MIL-D-9898C

MS33583	Tube End, Double Flare, Standard Dimensions for.
MS33584	Tubing End, Standard Dimensions for Flared.
MS33611	Tube End Radii.
MS33660	Tubing End, Hose Connection, Standard Dimensions for.

* (Copies of specifications and standards required by manufacturers in connection with specific acquisition functions should be obtained from the contracting activity or as directed by the contracting officer.)

2.2 Other publications. The following document(s) form a part of this specification to the extent specified herein. The issues of the documents which are indicated as DoD adopted shall be the issue listed in the current DoDISS and the supplement thereto, if applicable.

AMERICAN SOCIETY FOR TESTING AND MATERIALS (ASTM)

ASTM D 3951-82 Standard Practice for Commercial Packaging.

Application for copies of ASTM publications should be addressed to the American Society for Testing and Materials, 1916 Race Street, Philadelphia, Pennsylvania 19103.)

2.1.3 Order of precedence. In the event of a conflict between the text of this specification and the references cited herein, the text of this specification shall take precedence.

3. REQUIREMENTS

3.1 Engineering drawings of tube bend data shall be design activity documents and shall comply with the following.

3.1.1 Drawing requirements. Symbols, references, designations, code, and abbreviations shall be in accordance with DOD-STD-100.

3.1.2 Tube bend drawings shall indicate color code requirements as specified by MIL-STD-1247.

3.2 Drawing sizes. Tube bend data shall be furnished on a vertical “A” size accordance with figure 1.

3.3 Drawing format. Drawing format and required informational entries shall be in accordance with figure 1. For some special tubes having wye branches, brackets, or other details which cannot be shown in the space limitations of figure 1, a detail drawing of any suitable standard size

may be provided and the bend data shall be shown on the detail drawing in the format of figure 1.

3.4 Drawing numbers. Drawings shall be assigned Government design activity drawing numbers, or assigned contractor design activity drawing numbers.

3.5 Drawing materials. Materials used for the preparation of tube bend drawings shall be in accordance with DOD-STD-100.

3.6 Duplicates which become drawing media. Materials in this category shall conform to the requirements for "duplicate originals" specified in MIL-D-5480.

3.7 Delineation on tube bend drawings. Unless otherwise specified in procurement documents or orders, tube bend drawings, including those for vendor items, shall contain the following minimum engineering data as applicable.

3.7.1 The data furnished under this specification shall be suitable for use on bending machines utilizing the principle of "draw bending," e.g., the tube to be bent is clamped against a bending form which rotates and draws the tube through a pressure die and over a mandrel.

3.7.2.1 The "Bend Nobler" column shall specify the number of the bend that is to be performed. Bends shall be numbered consecutively from the "B End" of the tube as shown in figure 2.

3.7.2.3 The bend radius "F (inches to center line)" column shall specify the radius of bend to be performed, measured from the center line of the tube, a shown in figure 2.

3.7.3.4 The turn angle "E (degrees)" measured from the plane of the first bend, CCW viewed from B
end, shall specify the angle through which the tube is rotated from the zero reference plane established by bend number one to the place of the bend to be performed. The "tube turn" shall be expressed as counter-clockwise rotation when looking from the "B end" toward the "A end," using a 360° dial. An equivalent dial setting for specified tube rotation will depend on the type of dial used (figure 3) and must be determined by the tube bending machine operator when converting from one dial system to another.

3.7.2.5 The bend angle "G (degrees)" column shall specify the angle through which the tube is to be
bent as shown in figure 4. The bend angle is the finished angle and does not include spring-back. Each entry shall be CW(clockwise) unless noted as CCW(counter-clockwise).

3.7.3 The tube size (inches) shall specify the nominal outside diameter, wall thickness, and length necessary to make the finished tube assembly.

3.7.4 The "D" length is the length of the tube before bending which includes the necessary allowance for beading or flaring the ends. This length does not, however, include any allowances past the "A end" for clamping the tuber-which will be required when the last bend is very close to the "A end." Also, allowance may be required past the "B end" to complete a close starting bend. These additional lengths will be shown in the notes.

3.7.5 The "type of end" column shall be used to designate the end fittings. Any variation from the standard shall be noted and if necessary, illustrated in the "note" block.

3.7.6 Special treatment, marking, protection and testing of tubes shall be noted in the "note" block.

3.7.7 Reference to specifications and standards for requirements materials, processes, treatments, test methods, and procedures; the applicable type, grade, class, or condition shall be shown.

3.7.8 Delineate directly or by reference sufficient engineering requirements and characteristics for the item such as material, dimensions, tolerances, form, and finish necessary to enable the Government or contractor to procure or reproduce tile item or obtain an adequate interchangeable substitute.

3.7.9 Each tube bend drawing shall show engineering date for only one tube or tube assembly. If the number of tube bends exceed. the available spaces, additional drawings shall be used as indicated as a set (sheet l of 2; sheet 2 of 2; etc.). The note "continued on next sheet" shall appear as the last entry on all sheets except the last.

3.8 Item identification and part number. Each tube. or tube, subassembly shall be identified by one part number in accordance with DOD-STH-100. If a substitute material for a tube is authorized, a special note on the drawings shall describe how the tube would be marked to identify the material from which the tube has been made.

3.8.1 Identification on drawings. In accordance with 3.8 items shall be identified by showing the identifying part number.

3.8.2 The design activity shall indicate on the drawing the part number to be marked and the method of marking of the finished tube. Identification marking requirements shall be in accordance with MIL-STD-130.

3.8.3 Identification of materials, processes, and protective treatment. materials, processes, and protective treatment necessary to meet the design requirements for the tube shall be identified on the drawing by reference to applicable specifications, or standards. The applicable type, grade, class, condition, etc., shall be indicated. The applicable amendment symbol of the specification or standard shall not be shown. Additional reference to other equivalent specifications is permitted.

3.8.4 Other identification. When materials, processes, and protective treatment are used which cannot be identified adequately in accordance with 3.8.3, the drawing shall provide additional information for complete identification including the following:

a. Trade names or commercial designation.

b. Name address and Federal Supply Code for Manufacturers (FSCH) of the producer of the material, process, or protective treatment.

c. Physical and mechanical properties in sufficient detail to disclose strength and safety

characteristics where required by the design.

3.9 Care and maintenance of original drawings and referenced documents.
The design activity shall care for and maintain original engineering drawings and referenced documents in such manner to assure satisfactory reproducibles in accordance with MIL-D-5480.

4. QUALITY~ASSURANCE PROVISIONS

4.1 Responsibility for inspection. Unless otherwise specified in the contract or purchase order, the contractor is responsible for the performance of all inspection requirements as specified herein. Except as otherwise specified in the contract or order, the contractor may use his own of any other facilities suitable for the performance of the inspection requirements specified herein, unless disapproved by the Government. The Government reserves the right to perform any of the inspections set forth in the specification where such inspections are deemed necessary to assure supplies and services conform to prescribed requirements.

4.2 Examination. The contractor shall review and check the drawings and referenced documents for completeness, technical and engineering accuracy, sufficiency of information for accomplishment of the functions specified in the contract requirements, legibility reproducibility, and for conformance to all applicable requirements of the governing documents. These inspections. shall be accomplished prior to submission of the data for Government acceptance. Evidence or records of performance of the required inspection shall be made available to the Government inspector, upon request, at the time the data are presented for acceptance.

4.3 Inspection conditions. Sample items or packs and the inspection of the preservation, packaging packing and marking for shipment and storage shall be in accordance with the. requirements of section 5, or the documents specified these in.

5. PACKAGING

5.1 Original and duplicate original drawings shall be prepared for delivery in accordance with DOD-D-l000.

5.2 Microfilm shall be prepared for delivery in accordance with MIL-M-9868.

5.3 Reproducibles and nonreproducibles shall be prepared for delivery in accordance with MIL-D-5480.

5.4 Preservation. The levels of preservation shall be A and Commercial.

5.4.1 Level A. (Use data in MIL-D-5480).

5.4.2 Commercial. Commercial preservation shall be in accordance with ASTM D-3951.

5.5 The levels of packing are A, B and Commercial.

5.5.1 Level A. (Use data in MIL-D-5480).

5.5.2 Level B. (Use data in MIL-D-5480).
5.5.3 Commercial. Commercial packing shall be in accordance with ASTM D3951.

5.6 Marking of shipments. Interior packages and exterior shipping containers shall be marked in accordance with HIL-STD-129. Each interior package shall be marked in such manner that tile markings will not become damaged when the packages are opened. The nomenclature for interior packaged and exterior shipping containers shall be as follows:

Drawing for (name or item) Specification
____________________________(as applicable)
Name of manufacture________x________________________________
Name of contractor__________x________________________________
(If different from design activity)

Contract or Order No.________x________________________________

x = Information to be entered by the design activity

6. NOTES

6.1 Intended use. Tube bend drawings covered by this specification may be used by the Military activities for design, procurement manufacture, testing, evaluation, production, production and reviewing inspection, overhaul, shipping, storage, identification of stock, ordering and storage of replacement parts, inspection of items at overhaul, general maintenance of equipment, and wherever engineering drawings are needed.

6.2 Ordering data.

6.2.1 Acquisition requirements. Acquisition documents should specify the following:

a. Title number, and date of this specification.

b. Level of drawings to be provided (see 1.2). when drawings are required, include the applicable code identification and drawing numbers.

c. Kinds of materials for originals if they are to be provided (sec 3.5).

d. Levels of preservation and packing (see 5.4 and 5.5).

e. Quantity and type of reproducible, non-reproducible, or microfilm copies of drawings to be provided.

f. Any applicable instructions and procedures pertaining to control arid approval of drawings and revisions thereto.

g. Delivery dates and destination for drawing and other documents.

6.3 Submittal of tube bend drawings, standards, and related documents. In addition to the requirements of the contract or order for levels of drawings to be submitted, it shall be the responsibility of the prime contractor to include the requirements of this specification in contracts or orders with sub-contractors and to insure compliance therewith.

6.3.1 Contractor design activity documents referenced in tube bend data drawings. All contractor design activity documents referenced on drawings shall be furnished as part of the set of drawings and shall show the appropriate FSCM in accordance with DOD-STD-100. Duplication of a prior submission of these documents to the military procuring activity is not required.

6.4 The margins of this specification are marked with an asterisk to indicate where changes (additions, modifications, corrections, deletions) from the previous issue were made. This was done as a convenience only and the Government assumes no liability whatsoever for any Inaccuracies in these notations. Bidders and contractors are cautioned to evaluate the requirements of this document based on the entire content irrespective of and relationship to the last previous issue.

Custodian:
Air Force - 99

Review activity:
Air Force - 16

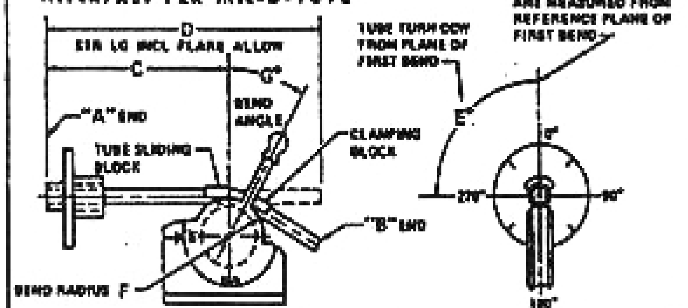
User activity:
Navy - AS

Preparing activity:
Air Force - 82

(Project DRPR-F262)

APPLICATION		REVISIONS			
NEXT ASSY	USED ON	LTR	DESCRIPTION	DATE	APPROVED

INTERPRET PER MIL-D-9898



TUBE SIZE		
OD	WALL	D LENGTH

HEAT TREAT

FINISH

COLOR CODE

PART IDENT.

BEND NO.	C DISTANCE	F BEND RADIUS	E TURN ANGLE	G BEND ANGLE

LIMIT WRINKLES, FLATNESS PER MS33611

END	FITTING	TYPE
A		
B		

TYPE OF END

1 FLARE MS33584
2 DBL FLARE MS33583
3 BEAD MS33660
4 SQUARE
5 NOTED

PROTECT ENDS PER MIL-C-5501

NOTES

1. ADD_____INCHES TO D LENGTH AT "A" END FOR CLAMPING, ETC WHEN REQUIRED
2. ADD_____INCHES TO D LENGTH AT "B" END FOR FIRST BEND WHEN REQUIRED
3. PRESSURE TEST TO______PSI PER______

DASH NO.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES
TOLERANCES ON
FRACTIONS DECIMALS ANGLES
± .XX ± ±
.XXX ±

MATERIAL AND MATERIAL SPEC.

DR

CHK

ENG

A.F. AUTHENTICATION

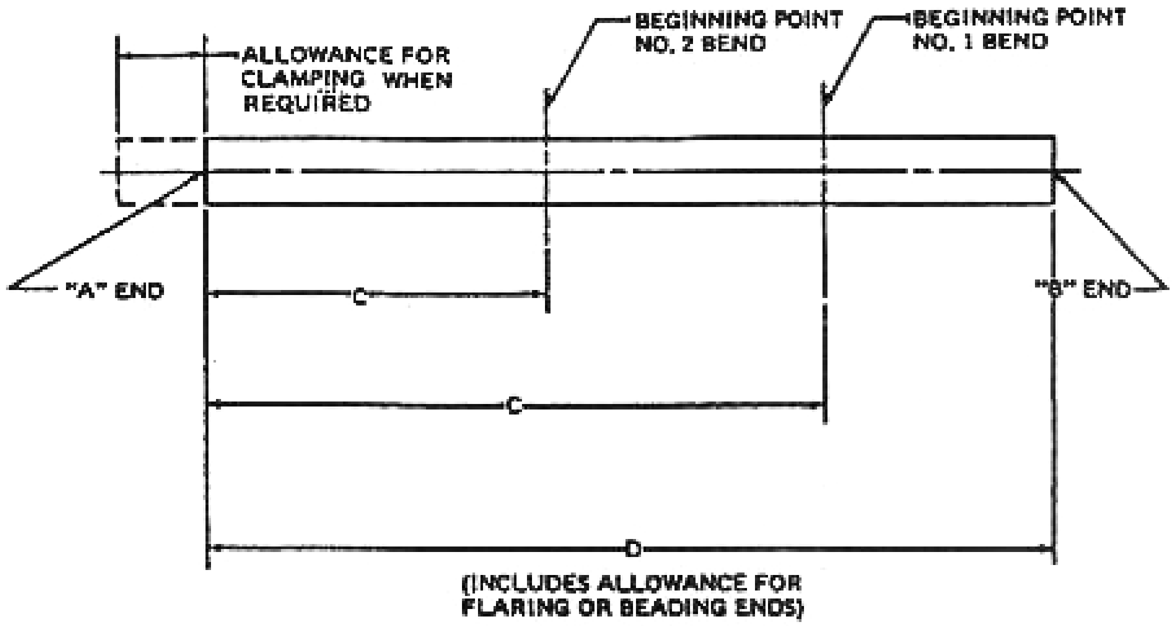
A.F. RELEASE

U.S. AIR FORCE

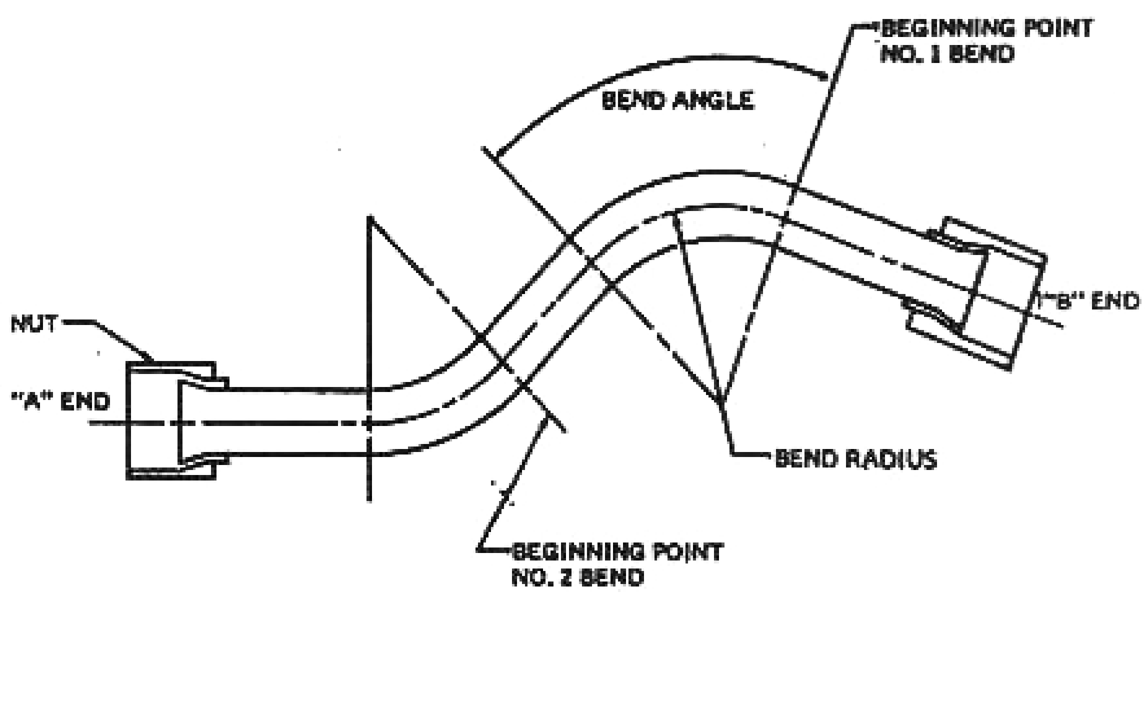
TITLE

SIZE	FSCM	DRAWING NO.
A		

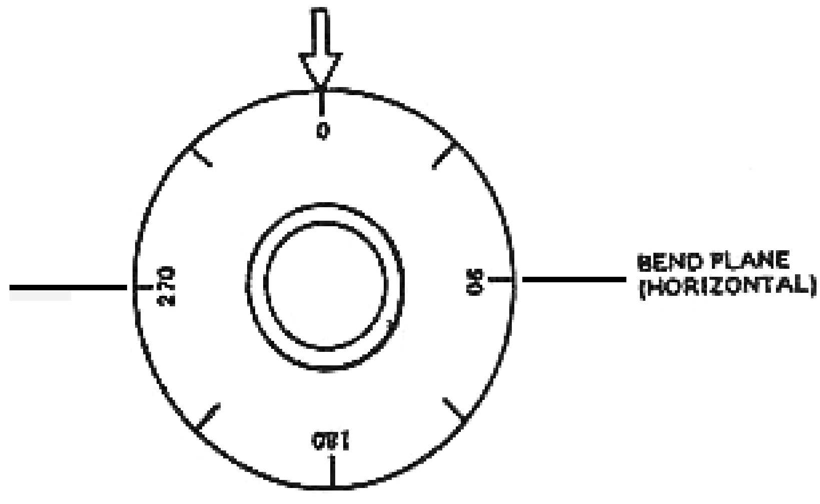
SCALE	REF SPEC MIL-D-9898	SHEET OF



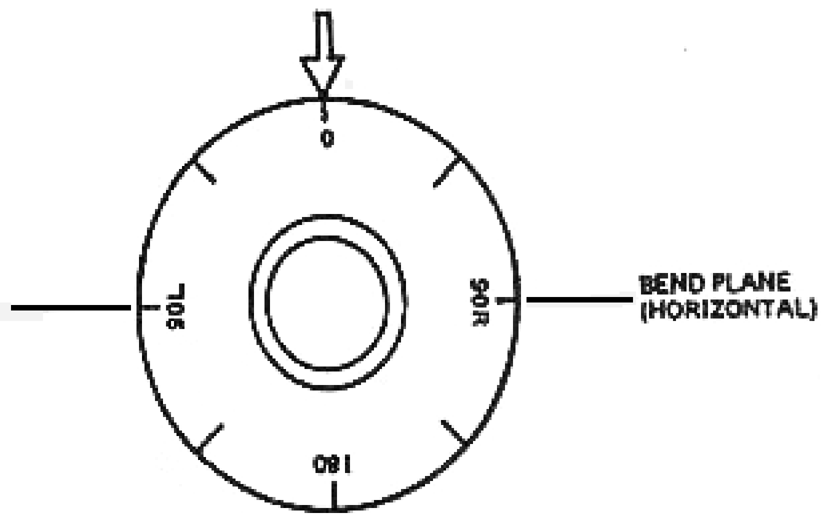
ALLOWANCE FOR
CLAMPING WHEN
REQUIRED
BEGINNING POINT
NO. 2 BEND
BEGINNING POINT
NO. 1 BEND
"A" END
"B" END
C
C
D
(INCLUDES ALLOWANCE FOR
FLARING OR BEADING ENDS)



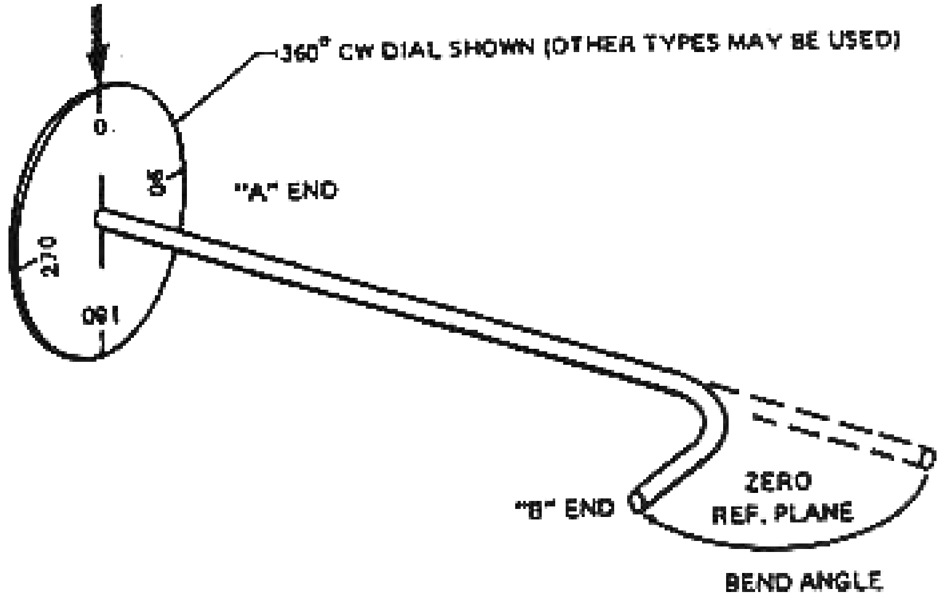
BEGINNING POINT
NO. 1 BEND
BEND ANGLE
NUT
"A" END
"B" END
BEND RADIUS
BEGINNING POINT
NO. 2 BEND

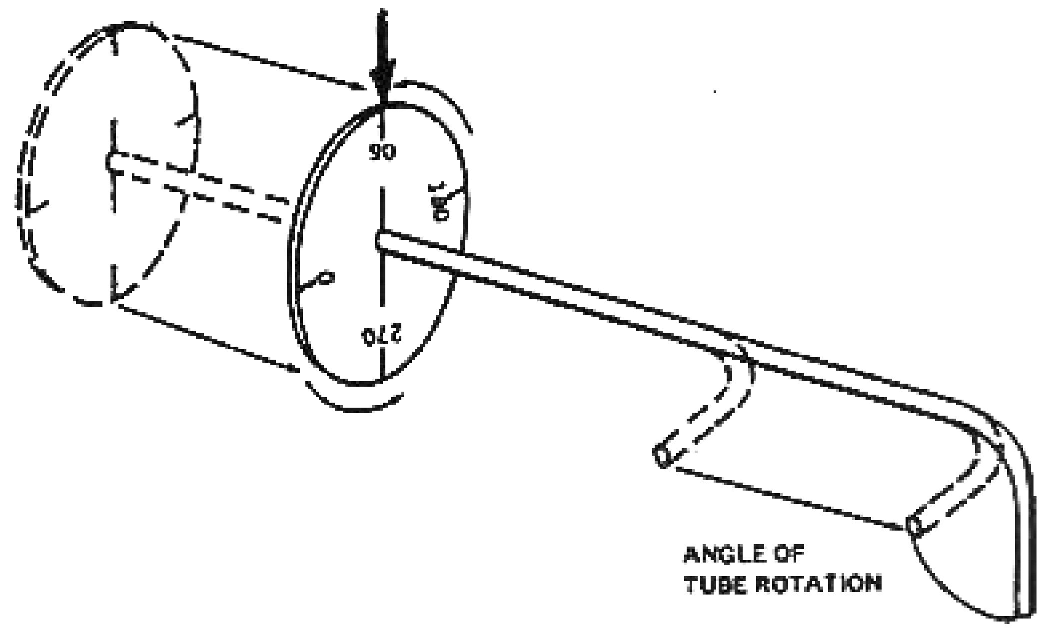


360° DIAL



180° R OR L DIAL


360° CW DIAL SHOWN (OTHER TYPES MAY BE USED)
0
90
270
180
"A" END
"B" END
ZERO
REF. PLANE
BEND ANGLE


90
180
0
270
ANGLE OF
TUBE ROTATION

APPENDIX “E”

PIN TYPE	USE	INFORMATION REQUIRED
I.D. Pin	Check linear tube I.D. position tube end	Linear tolerance minimum tube I.D. without positional tolerance
O.D. Pin	Check linear tube O.D. position tube end	Linear tolerance tube diameter
Green Zones	Check linear tolerance	Linear tolerance
Datum Target Pins	Removes the tolerance to a given area	Need to know the spot of limited tolerance
Floating Pins	To check the bracket/clamp without stacking tolerance	What tolerance, which feature
Bead Location Block	Datum point(s) tube at bead location	Which end is datum, and which side of the bead
Orbital Pin	To size the tube I.D. or O.D., and maintain the positional tolerance	What tolerance I.D. or O.D. to capture
Tapered Pin	To center a hole or tube end	Which feature
Non-Rotating Flange Pin	To verify the orientation of a mounting block	I.D. or O.D. tolerance
Flange or Condenser Pin	To verify the mounting block and verify orientation	Which surface is the datum surface, plus pin tolerance

CLOSING

In the current environment, delivery times are being reduced, contracts are determined by a fraction of a cent, and high quality is assumed as a prerequisite for entering the game. The customer is also looking for that "extra" that its suppliers can provide. We have some catching up to do in terms of our standards and analysis procedures.

Closer tolerances and the ability of computers to analyze geometry have called into question old procedures.

This is not a time for us to kick the sand and hope the old easy days will return. The future lies ahead and there is tremendous opportunity for those willing to change. There is also opportunity, in some measure, to direct an industry and help take it into the future. We live in a time of rapid change.

While it seems too much to ask that industry standards can be agreed upon and implemented, it seems good to start to clarify the questions and look at the implications.

If you have any thoughts or concerns, please let me know by writing me.

Thomas R. Clark
410 Dunbridge Road
Bowling Green, OH 43402