4 2-D vertical piping span Stress Analysis in *Tmin*

The original computer code of *Tmin* analyzed horizontal piping configurations. The addition of a 2-D vertical piping span analysis to *Tmin* was completed at the request of my sponsor and is the basis of this thesis. In order to analyze this piping span, the theories discussed in Chapter 2, Section 2-1 is followed for the entire span.

This chapter, details the analysis of a 2-D vertical piping span model with valves included in various spans. Using this model, shear and moment diagrams will be crated, detailed, and explained in Sections 4.1 and 4.2. Stress intensity factors (*SIF*) that are used for valve connections and elbows as required by ASME. These factors are found in Section VIII of the Unfired Boiler and Pressure Vessel code are used in the 2-D vertical piping span are discussed in Section 4.3 [14]. The use of a stress intensity factor increases the moment at a location along the piping span where the valve or elbows are located. Using the shear and moment diagrams created, the differential stress elements, discussed in Chapter 2, will be evaluated at certain sections on the piping span in Section 4.4. By performing the differential stress element analysis, the maximum stress was calculated for each section of the pipe. Finally, in Section 4.5, the stress at each section are equated to the ASME allowed stress and is used to evaluate the maximum pipe-wall thickness using a root-solver.

4.1 Shear and Moment Analysis

Static load analysis of the 2-D vertical piping span was completed first. Static analysis is performed on the 2-D vertical piping span, using pinned-pinned ends of the piping span [7]. Using pinned-pinned ends allowed for analysis of the 2-D vertical piping span to be locked into position at the ends of the piping spans, ensuring zero lateral movement. Moreover, these boundary conditions insure maximum moment estimates internal of the 2-D vertical piping span. The piping components of the 2-D vertical piping span included the weight of the pipe, elbows, internal fluid, and any valves or elbows that may have been included.

A test case showing the 2-D vertical piping span with valves at different locations is seen as the Q diagram seen in Figure 4-1. The purpose of a Q diagram is to simplify the sums of forces and moments that are required to find the reactions at the pinned ends [9]. The test case model dimensions are seen in Table 4-1. The reader is encouraged to use these dimensions and values to verify calculations for further models. The full derivation of the shear and moment analysis on the 2-D vertical piping span can be found in Appendix A.

Table 4-1. Values used for Test-Case Model

W_e	0.5	lbs
w_p	0.5	lbs/ft
L_1	10	feet
L_2	5	feet
L_3	10	feet
F_1	2	lbs
F_2	5	lbs
F_3	2	lbs
Dist-F_1	5	feet
Dist-F_2	5	feet
Dist-F_3	5	feet

In Figure 4-1, the weight of the pipe and fluid (W_p) are added together and are seen as distributed downward arrows. The valves are identified as vectors F_1, F_2 and F_3, also in the downward position. The distances to the valves are identified as *Dist*-F_1, *Dist*-F_2, and *Dist*-F_3. Finally, since the piping span has elbows incorporated, the weights of each elbow (w_e) are seen as additional forces only in the vertical span L_2.

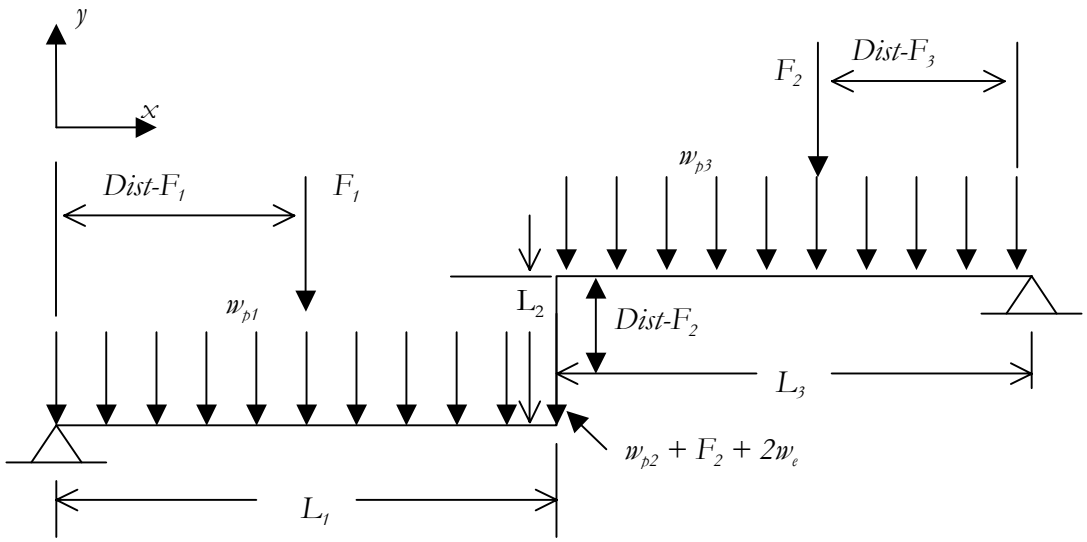

Figure 4-1. 2-D Vertical Pipe-Span. Showing Forces on Piping

The combined weights of the pipe spans, internal fluid, valves and elbows are then shown in an equivalent load diagram, Q_e. Since the weight of the pipe and fluid are seen as distributed forces acting along the entire span of pipe, a Q_e diagram formulates these forces into concentrated loads at the center of each span. A Q_e diagram of the 2-D vertical piping span is seen in Figure 4-2. Using this diagram, the reaction force, R_2, can be found using the sum of moments seen as Equation (4.1).

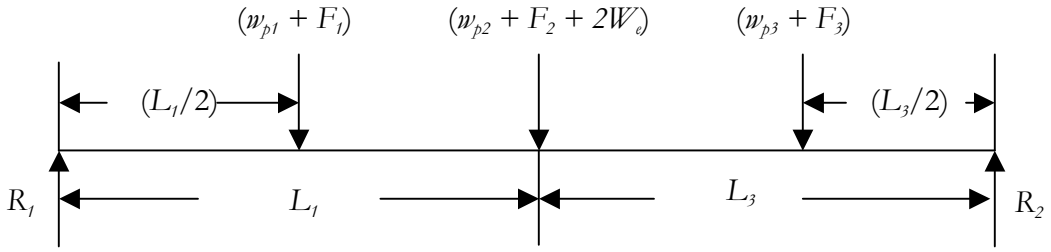

Figure 4-2. Q_e Diagram of 2-D Vertical Piping Span

According to Shigley and Mishke [8], the sum of moments is defined as the force multiplied by its distance for the reference point.

.

$$\sum M_{R_2} = R_2 = \frac{(w_{p1} + F_1)(L_1) + (w_{p2} + F_2 + 2w_e)(L_1) + (w_{p3} + F_3)\left(L_1 + L_3/2\right)}{L_1 + L_3} \tag{4.1}$$

Using R_2, the second reaction force, R_1, can be found using summation of forces in the y-direction. Equation (4.2) yields R_2 as:

$$\sum F_y = R_2 = (w_{p1} + F_1) + (w_{p2} + F_2) + (w_{p3} + F_3) - R_1 \tag{4.2}$$

Once these reaction forces are found, a shear diagram can be created. A shear diagram is used to visually evaluate the piping span for critical areas of load. The moment diagram is used to complement the shear diagram as to where to evaluate the piping stresses, for example, by looking at maximum moment locations. However, the moment diagram is normally the more important of these two diagrams.

4.2 Shear and Moment Diagrams

The fundamentals of shear analysis are seen as the forces that are present at all locations on the piping span being analyzed. When the reaction forces are found (R_1 and R_2), these values are now seen as the change in the end load values. Since the piping has a distributed, downward load, the distributed load will be seen as a decreasing slope on the shear diagram. Figure 4-3 shows a shear (upper figure) and moment diagram (lower figure) of the previous piping configuration. Both were created in *Matlab.* The *Matlab* code was created for use in creating correct *IF-THEN* statements used to determine where the maximum moment would occur in the piping span. In addition, it was used to independently verify the accuracy of *Tmin's* output shear and moment diagrams. As a result of this preliminary code, the addition to the *Tmin* code for the 2-D vertical piping span was made easier.

The shear diagram is used to show the entire set of forces on the member. Note that the shear diagram closes to zero at the end. This indicates proper reaction analysis. As seen in this figure, the shears, V, are largest at the ends. These loads decrease the shear value along the piping span. When the piping has a concentrated load such as a valve, a step function is found in the shear diagram [9].

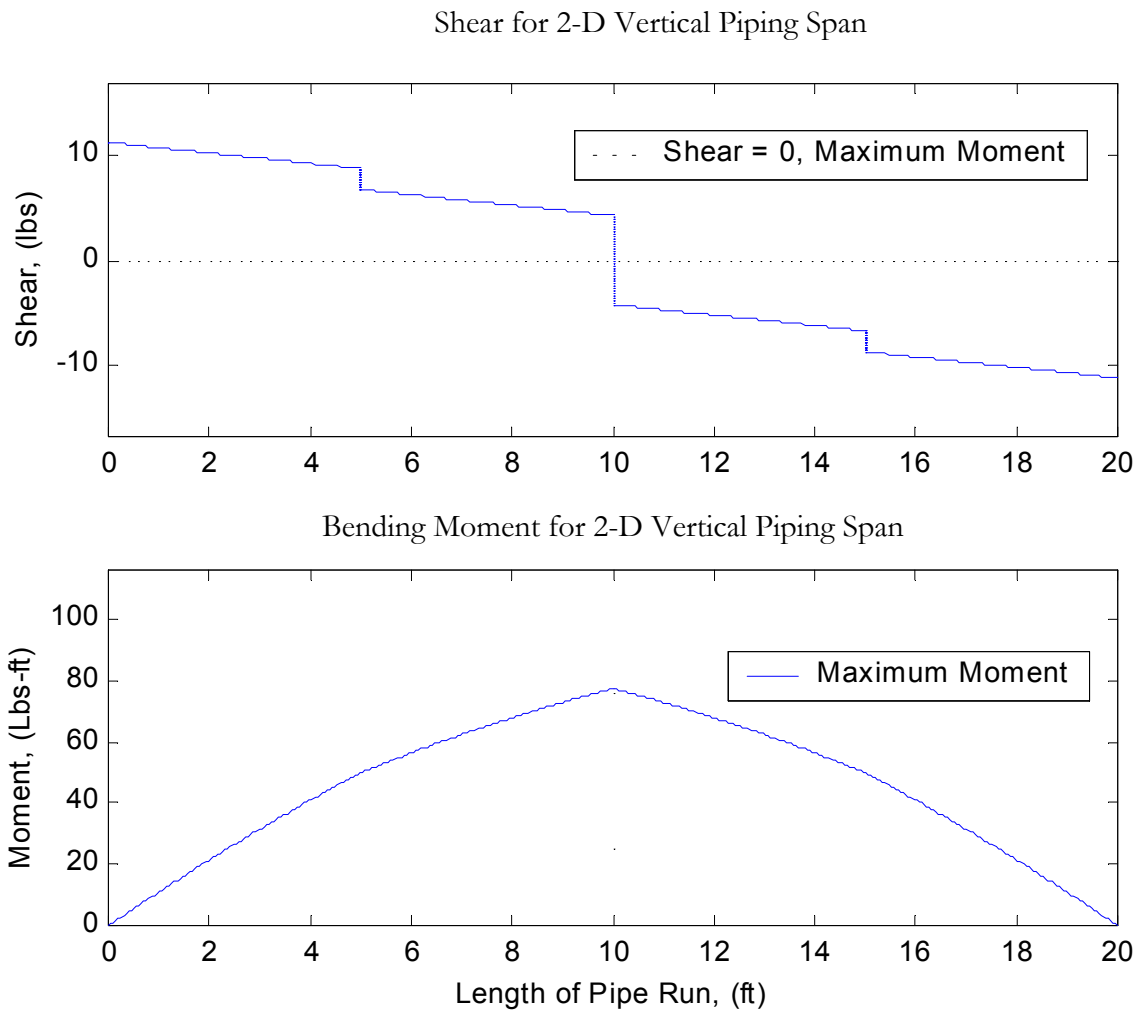

Figure 4-3. Shear and Bending Moment Diagram of a 2-D Vertical Piping Span

Note that in the *Matlab* code the moment diagram was created by using the trapezoids of the shear diagram at each step seen. As a result, when the moment diagram was created the moment will appear to have sharp edges. In addition, this moment diagram will be used as a guideline as to where evaluation of moments will take place. A moment diagram is created by the integration of the shear diagram starting from the left side of the shear diagram as seen by Equation (4.3) [8]. That is, the change in the moment diagram equals the area under the shear diagram.

$$\int_0^{L_1/2} (V_o - Kx)dx = \left(V_o x - K\frac{x^2}{2} \right)_0^{L_1/2} \tag{4.3}$$

Integrating the shear areas between each step function causes the moment diagram to increase as long as the shear diagram remains positive in sign. When the shear diagram becomes zero, in this case at the center of the span, the maximum moment occurs. As shown in Figure 4-3 (lower figure), the moment is zero at the left end and increases until the maximum occurs at the center of the piping span. Continuing to the right of the piping span after the maximum moment occurs, the moment decreases until the right end of the 2-D vertical piping span is complete. This moment reduction is a direct result of the negative shear diagram integration.

Now that the shears and moments have been evaluated for this piping span, differential stress element analysis can be started at various sections along the piping system. The obvious places to evaluate the stress state would be at the maximum shear (end of piping spans) and at the maximum moment locations because of the *SIF* factors that apply at these points. In addition, the location where the step function occurs are also choices for evaluation.

Since the 2-D vertical piping span uses elbows, 5 different elbows are available for choice by the user. With each of these elbows is a stress-intensity factor (*SIF*) is different and is described in the next Section 4.3. If the user chooses a pipe configuration with a valve, the *SIF* connection type of the valve, must be accounted for [28]. In respect to the valves, additional *SIF* values are incorporated into the program for valve connection type [1].

4.3 Stress-Intensity Factors

A stress-intensity factor (*SIF*) is a numerical value titled and determined by the ASME and is found in the B31.3 standards and codes [28]. The *SIF* values for each elbow type are seen in Table 4-2. When an elbow or valve is chosen by the end-user of the program, the corresponding *SIF* value will be multiplied by the moment at that location.

Table 4-2. Stress-Intensity Factors for Elbow Choices

Elbow Description	**Stress-Intensity Factors (*SIF*)** [28]
Standard Long Radius Bend	1.5
Short Radius Bend	1
3D Bend	3
5D Bend	5

Detailed in Chapter 5 are input screens that will enable the user to choose the type of elbow in the 2-D vertical piping span being evaluated. A check in the computer code finds where the elbow starts in comparison to the pipe-span itself [1]. The moment at the end of the elbow connection is multiplied by the *SIF* value corresponding to the elbow chosen. This may force a moment that may have had a low numerical value, in comparison to the maximum moment value, to be increased by the *SIF* factor. This will appear as a spike in the moment diagram as seen in Figure 4-4.

Using the shear and moment diagrams of Figure 4-3, the *SIF* values for a pre-chosen elbow and valve connection type are multiplied by the appropriate moments. Stress-intensity factors can have numerical values of 1 (one) to 2.6 as seen in the Table 4-1 of *SIF* values. This can dramatically change the final *Tmin* pipe-wall thickness output.

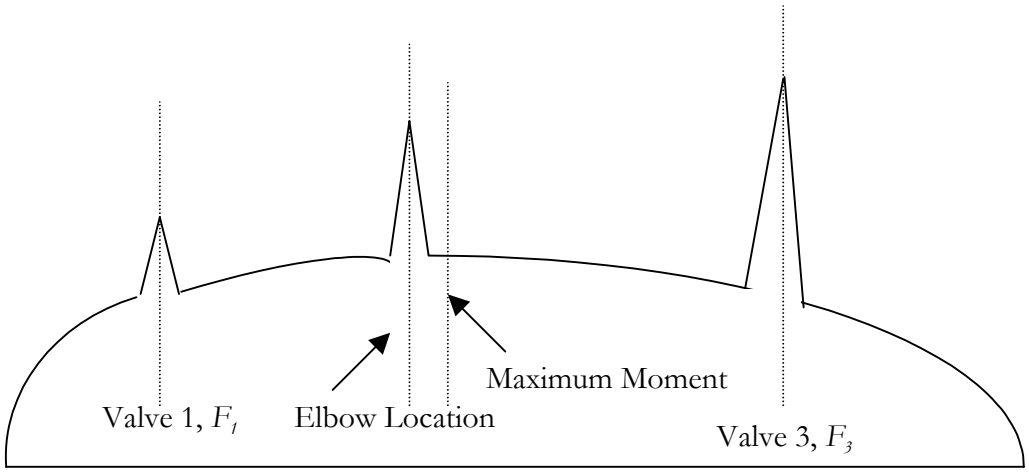

Figure 4-4. Example Moment Diagram with Additions of *SIF* Values Showing Drastic Moment Values

In addition to the *SIF* values for the elbows, the ASME also has *SIF* values for the valve connection types. These valve connection SIF values are obtained in the ASME B31.3 standards and codes [28]. When the user chooses a 2-D vertical piping span for analysis, the program will prompt the user for a valve-connection type. This connection type could be a weld, a threaded, or even a bolt-on flange valve type. Again, what this does is increase the moment according to the connection type due to the *SIF* value. In comparison, when the user chooses a valve connection type, a different set of *SIF* values are used. However, due to the limited knowledge about the length of various valves, the *SIF* value will only be multiplied at the exact center of the valve. These limitations of valve size are due to the wide variety of valve lengths used in the industry today.

The preliminary evaluations of the shear and moment diagrams are now complete. The next section will detail the evaluation of these sections using differential stress elements and Mohr's circle analysis.

4.4 Differential Stress Element Analysis of the 2-D vertical piping span

Using critical sections in Figure 4-3, differential elements are used to determine the maximum stress states at these locations. Reviewing differential stress elements and Mohr's circle analysis from Chapter 2 will be helpful for understanding of this section. Only three areas will be evaluated in this test case because of symmetry of the 2-D vertical piping span: the lower left-end of the piping span, the valve, F_1, location, and the vertical piping span section. The full derivation of the 2-D vertical piping span can be followed in Appendix A, while numerical examples are given in Chapter 6.

Figure 4-5 starts the evaluation of the left end of the piping span. At this location, as seen in Figure 4-3, the shear is at a maximum and thus requires the use of Mohr's circle. To begin, the piping section will be evaluated using 4 differential stress elements located 90 degrees apart from one another. The left end of the piping system is shown with the 4 elements used for evaluation with a shear, V, seen on both sections of the span.

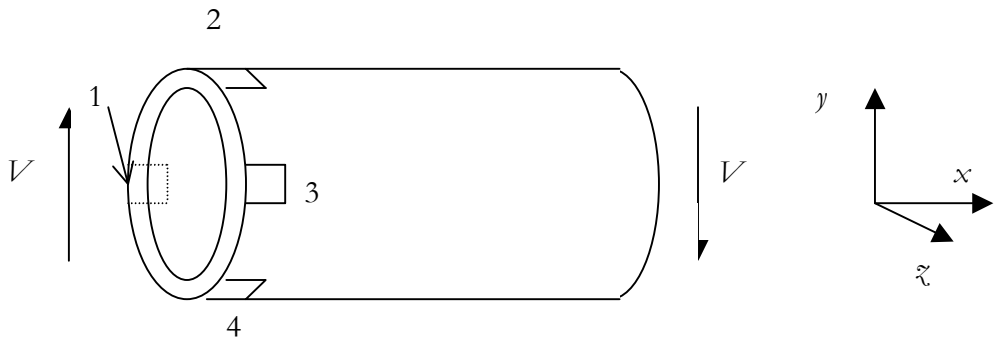

Figure 4-5. Differential Elements Shown on Left-End of Piping Span Section

As seen in this figure, the shears, *V,* passes through elemental points 2 and 4, but are parallel to elements 1 and 3. Elements 2 and 4 have an outside face that cannot sustain shear stress. Thus the shear at 2 and 4 must go to zero. Evaluating elemental points 1 and 3 shear stress are expected. These shear stresses seen are the maximum in the section [8]. The differential stress elements 1 and 3 will be evaluated first, seen as Figure 4-6 (a and b). In this figure, the longitudinal stress will be used to find a maximum stress using Mohr's circle and the Maximum-Shear-Stress Theory (MSST), as detailed in Chapter 2. It can be seen that the shear passes down on the left side of differential stress element 3. At the same time the direction of the shear goes up on the right of the same element; this ensures static vertical equilibrium.

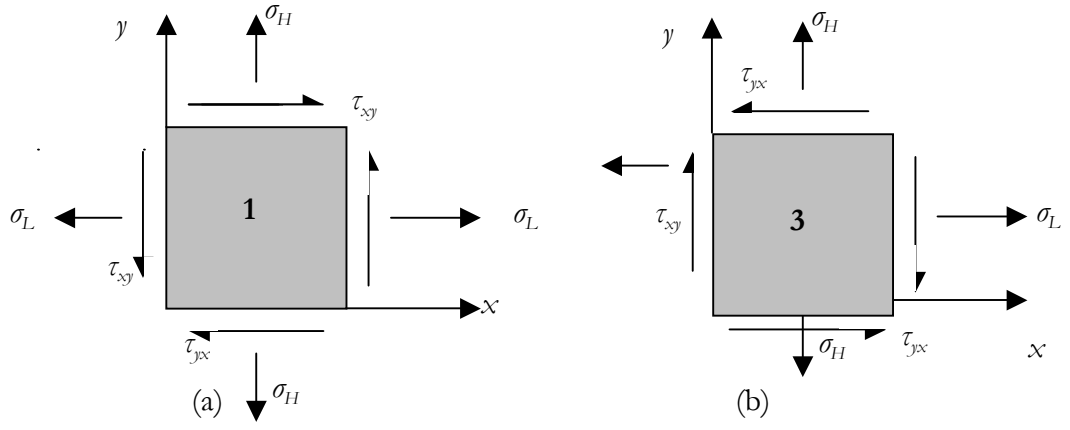

Figure 4-6. Differential Elements 1 (a) and 3 (b) on a Piping Span End

In Figure 4-7 (a and b) differental stress elements 2 and 4 only principal stresses are observed.

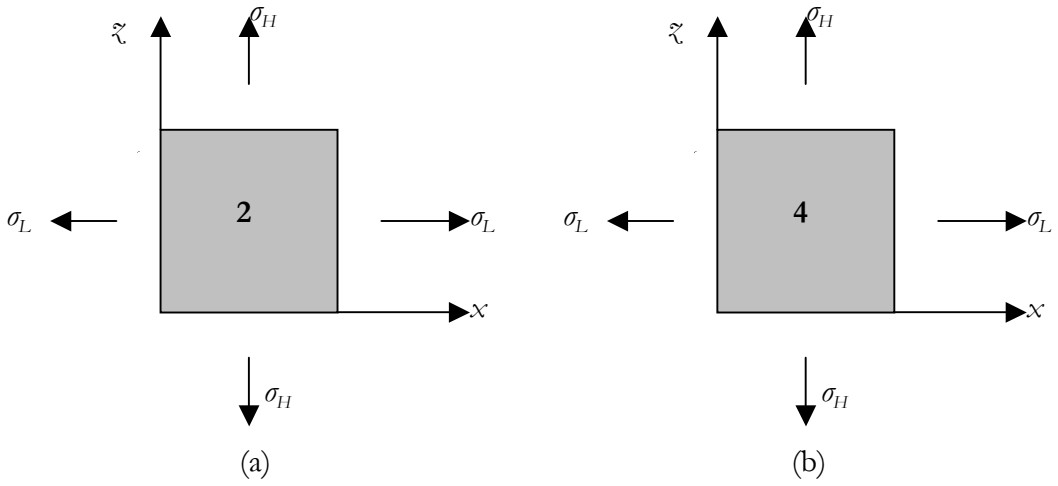

Figure 4-7. Differential Elements 2 (a) and 4 (b) on a Piping Span End

Using the stress states in Figure 4-6 (a and b) using 3-D Mohr's circles, the maximum circle created will define the maximum shear stress. Noting that the stress elements in Figure 4-6 have one added stress to those in Figure 4-7, once must consider those of Figure 4-5 as critical. Figure 4-8 shows the Mohr's circle evaluation of element (a) in Figure 4-5. Noting that the largest principal stress is seen as σ_1, this stress is equated to one-half of the maximum shear, τ_{Max}, and following the Maximum-Shear-Stress Theory results in Equation (4.4) and (4.5) for the evaluation of element (a) in Figure 4-5. Since there are other locations where the piping span must be evaluated, this is but one equation of several which will be used in solving for the pipe-wall thickness.

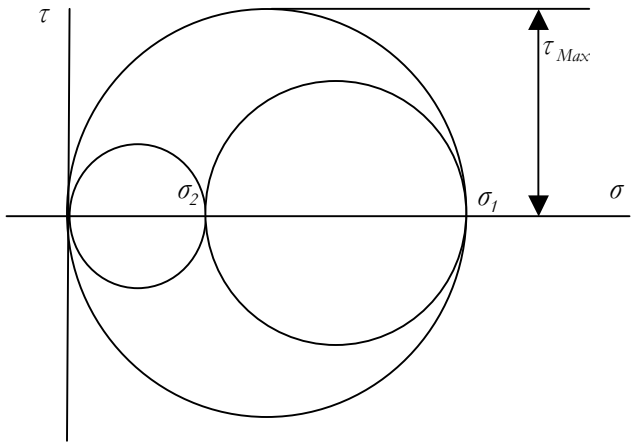

Figure 4-8. Mohr's Circle Analysis of Element (a) in Figure 4-5

$$\sigma'_{MSST} = 2\tau_{Max} = 2\left(\frac{\sigma_1}{2}\right) = \sigma_1 \tag{4.4}$$

$$\sigma_1 = \left[\left(\frac{\sigma_H + \sigma_L}{2}\right) + \sqrt{\left(\frac{\sigma_H - \sigma_L}{2}\right)^2 + (\tau_{xy})^2}\right] \tag{4.5}$$

The next section to be evaluated on the shear and moment diagrams is where the concentrated load occurs. The concentrated loads occur because of the valves at the locations 5 and 15 feet. At these locations a step function is seen. Because there is a moment observed in addition to shear in Figure 4-3, Figure 4-9 shows the evaluated differential stress elements with the inclusion of a moment, *M*.

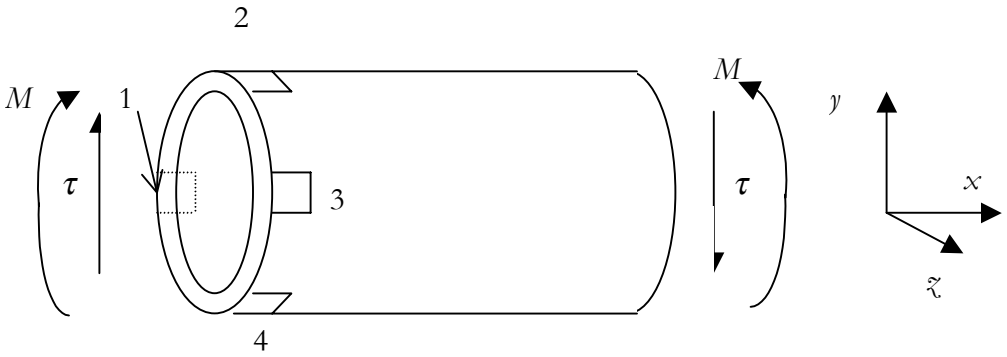

Figure 4-9. Differential Elements Shown on Piping Span Section-Shear and Moment Loads

Elements 2 and 4 experience moment-induced compressive and tensile stresses respectively, with no shear stress. Like in the previous analysis, elements 1 and 3 will have maximum shear. But since shear load is not a maximum in the longitudinal piping span, it will not dominate over the previous stress calculations. Observing differential stress elements 1 and 3, it is seen that they are identical to Figure 4-5, but with a lesser shear stress. The moment-induced stress will not be observed in elements 1 and 3 because they are zero at the neutral axis. However, differential stress elements 2 and 4, seen in Figure 4-10 (a and b), include an additional bending stress due to the moment. In these figures, the additional stress is the resultant of a moment-induced stress, σ_B. Its direction to the differential stress

element is dependent on the moment being positive or negative. When the added moment-induced stress is compressive, the bending stress goes into the differential stress element, while a tensile moment-induced stress is seen as moving away from the differential stress element.

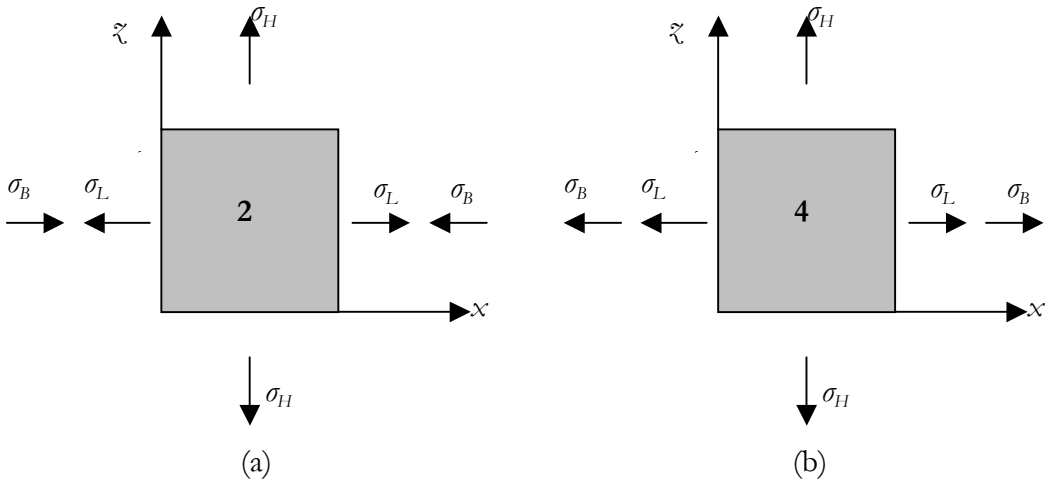

Figure 4-10. Differential Elements 2 (a) and 4 (b) on 2-Axis Vertical piping Span End

Figure 4-11 (a) shows the Mohr's circle analysis that was used for the analysis of differential stress element (a) in Figure 4-8. Two cases are observed. One case has the difference between the hoop, bending and longitudinal stresses being dominant, seen in Figure 4-11 (a), while the other case has the hoop stress the dominant stress (Figure 4-11 b)).

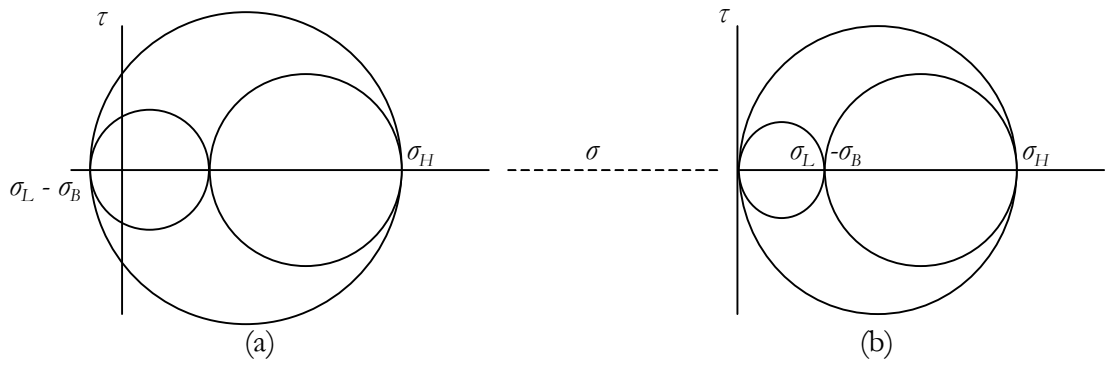

Figure 4-11. Mohr's Circle Analysis of Element (a) in Figure 4-10

Equation (4.6) is the result of the Mohr's circle analysis completed on element (a) in Figure 4-10. Mohr's circle analysis of the differential stress element in Figure 4-12 (b) again results in two more cases observed. From Figure 4-11 (a), this case shows that the hoop stress is the dominant stress observed. While in Figure 4-11 (b), the bending plus the longitudinal stress are the dominant stresses observed. From this analysis, Equation (4.7) was created and will be used for analysis at this section.

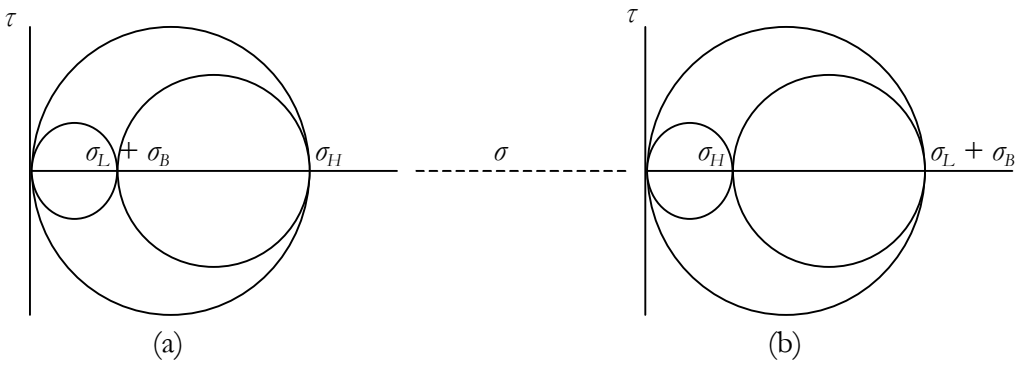

Figure 4-12. Case (a) Shows Hoop Stress Dominant. Case (b) Shows the Bending and Longitudinal Stresses as Dominant

As a result of the 3-D Mohr's circle analysis of the elements in Figure 4-12 (a and b) two independent MSST stress states will be evaluated. The first is for element (a) in Figure 4-10. The second is for element (b) in Figure 4-12. The stress-states are seen as Equations (4.6) and (4.7).

$$\sigma'_{MSST} = \sigma_H \text{ or } \sigma'_{MSST} = (\sigma_H + \sigma_B - \sigma_L) \tag{4.6}$$

$$\sigma'_{MSST} = \sigma_H \text{ or } \sigma'_{MSST} = (\sigma_B + \sigma_L) \tag{4.7}$$

The maximum pipe-wall thickness will be found between these four stresses. This maximum thickness value will be used for final comparison in the entire 2-D vertical piping span. Performing analysis of the differential stress elements on the vertical piping section proved to be different than previous analysis. The difference occurred because of the

addition of another stress term. Because there is an additional normal force on the vertical piping span due to gravity and also due to effects on the longitudinal piping the weight of the vertical pipe. This force may be compressive or tensive.

Since the moment observed in the vertical piping span is constant along its length an additional load diagram was needed. Figure 4-13 shows such a plot. However, the loads need not be entirely compressive as shown here. In order for this case to occur, a short vertical span is observed and at the same time the upper piping span would need to be much longer than the lower piping span. When this happens the vertical span would be in compression from the weight of the upper piping span pushing down on the vertical span. If the loads on the vertical span are in compression, Figure 4-13 will be used for evaluation. However, if the loads in the vertical section experience a tension, a figure showing a tension plot will need to be identified. A tension case will be discussed later in this section. In Figure 4-13 it is seen that at the top of the piping span, the load value is lower than the load value at the bottom of the piping span. Following the rules of distributed loads seen in the shear diagram, the compressive state is seen to increase because of the piping and insulation weight. Next, a concentrated load is observed. This is due to the valve weight.

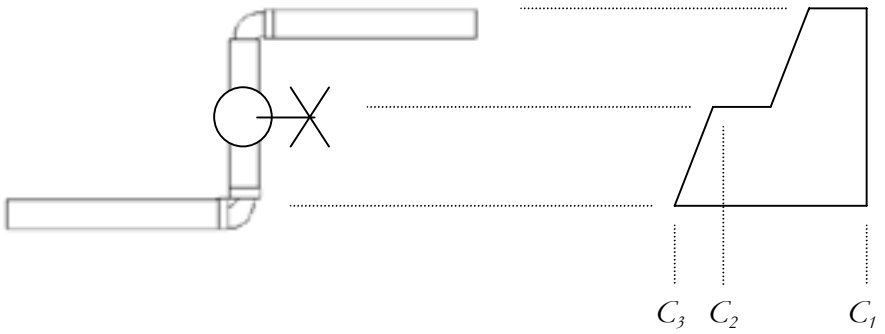

Figure 4-13. a) Vertical Piping Span, b) Axial Load Plot of Vertical Piping Segment

From this figure, three compressive forces are observed: C_1, C_2, and C_3. However, since compressive forces at C_2 and C_3 are always numerically greater than state C_1, only these two will be evaluated seen as Equations (4.8) through (4.10). Because the compressive forces at

the bottom of the piping span are equal to the negative value of the shear value at that location in the left horizontal piping span, a simple substitution was used inside the *Tmin* computer code for these equations. The compressive force C_3 is equated to the shear at the bottom of the vertical piping span, *VBottom* while the compressive force C_2 is at the center of the vertical span. Finally, the compression at the top of the piping span is equal to the shear at the top of the vertical span. In this test case it is seen that the compression at the top and the bottom of the span is necessary to evaluate. In addition, when the user selects a valve to be included in the vertical span, this section will also be evaluated. However, when no valve is present in this section, a calculation will not occur.

$$C_1 = -V_{Top} \tag{4.8}$$

$$C_2 = -Dist\text{-}F_2 * (w_{p2}) - F_2 - V_{Top} \tag{4.9}$$

$$C_3 = -V_b \tag{4.10}$$

Now that the compressive state has been identified, a tension case will be investigated. In the next Figure 4-14, the tension load is the largest at the top of the plot. This is due to a case where the lower piping span is much longer than the upper span. As a result of the longer lower span the weight of the vertical span pulls down on upper span, which creates a pure tension case in the vertical span.

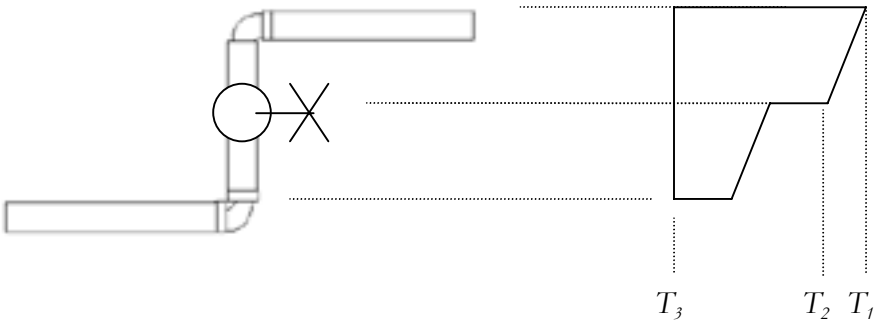

Figure 4-14. a) Vertical Piping Span, b) Axial Tension Plot of Vertical Piping Segment

The resulatant equations that are created for this tension case are seen as Equations (4.8a) through (4.10a).

$$T_1 = -V_{Top} \tag{4.8a}$$

$$T_2 = -Dist - F_2 * (wp2) - F_2 - V_{Middle} \tag{4.9a}$$

$$T_3 = -V_{Bottom} \tag{4.10a}$$

The tension in the vertical section will not be as detrimental to the vertical section as compression would be because the fibers of the piping will be stretched. In comparison to the compression values found, the tension equations will always be less of a risk to the vertical piping span. As a result, Equations (4.8) and (4.10) will be used for all cases when a valve is not present. When a valve is present in the vertical section Equation (4.9) will be used as well.

Once the values were found, the differential stress element analysis could be completed. A vertical piping segment with the 4 differential stress elements used for evaluation is seen in Figure 4-15.

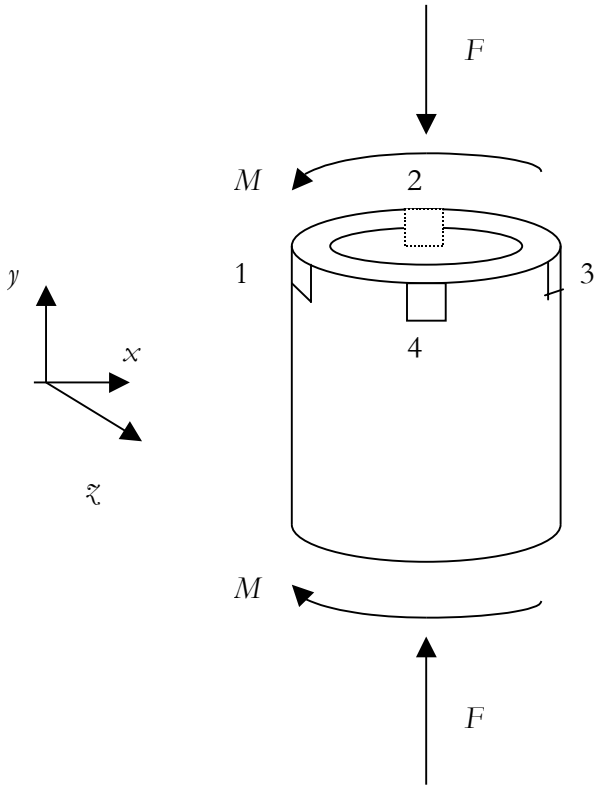

Figure 4-15. Differential Elements with Compressive Forces and Moments on Vertical Segment

As seen in this figure the compressive force pushes into the piping section from both top and bottom. As a result of the compressive force, it is expected that a compressive stress, σ_C, will appear in all differential stress elements. Figure 4-16 (a and b) shows the stress-states that appear on differential stress elements 1 and 3. In the vertical segment location the moment is constant. However, as discussed earlier, the moment induces a bending stress only on elements 1 and 3, while the bending stress in element 2 and 4 are zero.

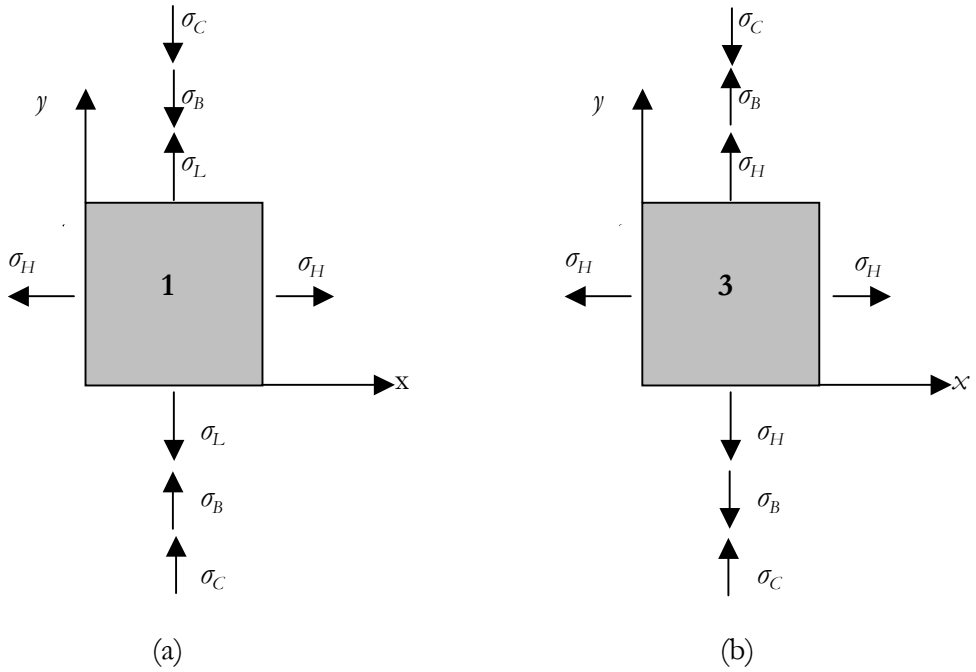

Figure 4-16. Differential Elements 1 (a) and 3 (b) on 2-Axis Vertical piping Span End

Since the moment is zero at the edges of the piping span, as detailed in Section 2.1, the bending stress is zero on elements 2 and 4 seen in Figure 4-17.

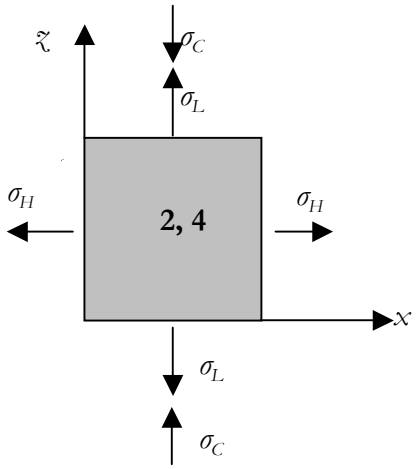

Figure 4-17. Differential Elements 2 and 4 on a Vertical piping Span End

Upon evaluation of all 4 differential stress elements, it was found that elements 1 and 3 would generate the largest 3-D Mohr's circle. Elements 2 and 4 are eliminated from consideration. The 3-D Mohr's circles and MSST for elements 1 and 3 result in the

Equations (4.11) and (4.12). These equations will be used to evaluate pipe-wall thickness in the vertical segment. In addition to these two equations, it was found that the hoop and the summation of the bending and longitudinal stresses might also be a factor. Therefore, when evaluating the vertical piping span, the largest pipe-wall thickness is found from all 4 of these Equations (4.6), (4.7), (4.11), and (4.12). Note that σ_c must be a negative number for Equations (4.11) and (4.12) to function properly. Equation (4.12) is from element 1 in Figure 4-# and Equation (4.12) was obtained from element 3 in Figure 4-16.

$$\sigma'_{MSST} = \sigma_H \text{ or } \sigma'_{MSST} = \sigma_L + \sigma_B - \sigma_C \text{ or } \sigma'_{MSST} = \sigma_L + \sigma_B + \sigma_C \tag{4.11}$$

$$\sigma'_{MSST} = \sigma_H \text{ or } \sigma'_{MSST} = \sigma_L - \sigma_B + \sigma_C \tag{4.12}$$

Using these equations for the analysis of the pipe-wall thickness for the vertical section will result in different pipe-wall thickness values that will be compared for the largest value. Once the evaluation is complete, the pipe-wall thickness values will be compared in an array [32] for the largest value. The array created was used for comparison of all pipe-wall thickness values along the entire piping span.

As stated earlier, because the 2-D vertical piping span test case is symmetric about the vertical section, the same procedures can be followed for the piping sections while moving along to the right of the shear diagram. The procedure of analysis of the Tmin 2-D piping analysis is as follows:

a) Input all data (pipe size, schedule, material, valves present, elbow type, etc.)
b) Numerically compute the shear diagram
c) Determine where the maximum moment occurs using the shear values
d) Find out if valves are in the piping span, if so, apply the stress-intensity factors to the moment at the location of the valve
e) Compute the distance the elbow starts in comparison to the pipe span itself
f) If any moment occurs in the elbow section, apply the stress-intensity factor for the elbow choice

g) Compute the pipe-wall thickness and if necessary pass the values to the root-solver (discussed in the next section). If there are more than one equation for the section being evaluated, get the largest *Tmin* value (critical value) from each equation
h) Compare all *Tmin* values calculated at each section, then display what section is the critical one (discussed in Chapter 5) by coloring the section red
i) Use the critical *Tmin* value for final *Tmin* comparison checks (setup by DuPont previously)

Using the analysis procedure documented in this chapter, the minimum pipe-wall thickness could be found. A detailed analysis of a pre-determined 2-D vertical piping span will be computed numerically in Chapter 6. Since some of the summation of stresses observed involved a pipe-wall thickness, *t*, to an exponent power, this created another problem. In response to this problem, a root solver was incorporated into *Tmin*.

4.5 False-Position Root Solver

Since the stress equations have varying t^N powers, the minimum pipe-wall thickness equations would be tedious to solve for the pipe-wall thickness directly. To show the difficulty of solving directly for the pipe-wall thickness directly in Equations (4.7), (4.11), and (4.12), *Mathematica*, a math solver was used. The resulting solution equations found were too complex and tedious to enter into the program for a direct pipe-wall thickness to solve. In addition, to trouble shoot these equations for errors would be tedious. Results from the program *Mathematica*® solution can be seen in Appendix E. As a consequence of the complex result obtained, a root-solving method was used instead.

The root-solving method used is called the *Regula Falsi*, or False-Position method [33]. In order to use this type of root-solver, the stress equations are equated to an allowed strength. The equations are then rearranged to be equal to zero. Once this was done, an initial guess of $[a_0, b_0]$ was used to create an interval. As seen in Figure 4-18, the plot of

the stress equation, with a crossing interval is shown. In this interval, one end of the interval must result in a negative sign at b_o, and the other a positive sign at a_o. Once this is done, the root solver will begin solving for the actual pipe-wall thickness that will make this equation equal to zero.

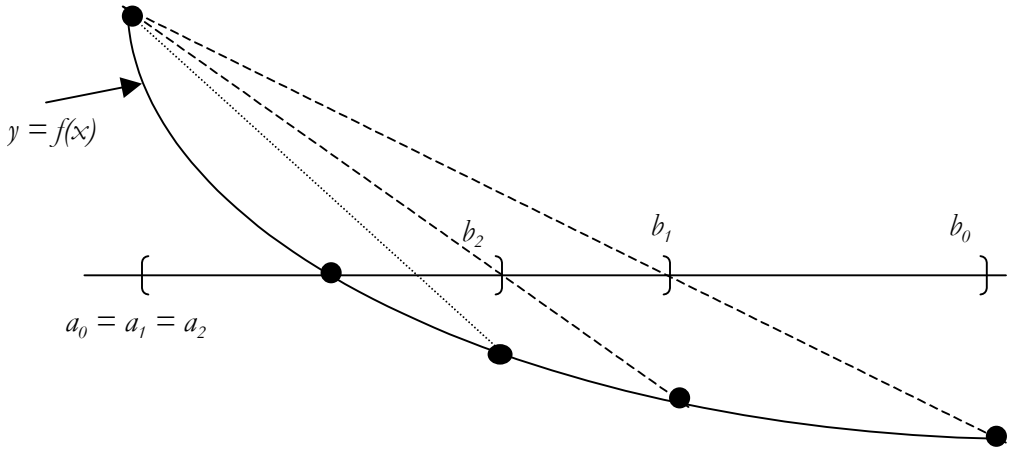

Figure 4-18. Stationary Endpoint of a Stress Equation for the False Position Method

With the interval identified, the root-solver will pass additional guesses into the stress equations. The guesses passed to the solver create a smaller interval repeatedly until a pipe-wall thickness has been found. The process is as follows, an initial guess is given to b_o, the crossing point of the zero axes is then identified as b_1. If b_1 is creates a smaller interval, then another guess, b_2, is passed through the equations. Again, if the interval is smaller than the previous interval, another guess in passed through the equations until the interval of guesses becomes small enough that the root of the equation is found.

Identified in this chapter were the stress equations that were derived from use of differential stress elements that were used for solution to the minimum pipe-wall thickness. In addition, stress-intensity factors that may create a larger minimum thickness have been discussed. Throughout all the discussion differential stress elements have been used for a solution in conjunction with the maximum-shear stress theory. Finally, a root-solving routine has been used to solve for the pipe-wall thickness when the stress equation becomes too difficult to solve for the pipe-wall thickness directly.

In the next Chapter 5, the additions to the *Tmin* program are discussed. These include user-friendly additions, valve-connection screens, the 2-D vertical piping span, as well as an output to a Microsoft Word document.