

**User's Manual for  
SafeRoof<sup>®</sup> Version 2.1:**

**A Program for the Analysis  
of  
Storage Tanks  
with  
Frangible Roof Joints**

**Manual Release 2.1  
February, 2005**

Version 2.0 by  
Thunderhead Engineering Consultants, Inc.  
1006 Poyntz Ave.  
Manhattan, KS 66502-5459

Version 1.0 by  
Zhi Lu  
Mechanical Engineering Department  
Kansas State University  
Manhattan, KS 66506

# TABLE OF CONTENTS

<b>INTRODUCTION .....</b>	<b>1</b>
<b>INSTALLING SAFEROOF .....</b>	<b>3</b>
<b>USING SAFEROOF .....</b>	<b>4</b>
<b>FILE MENU.....</b>	<b>4</b>
New .....	4
Open.....	4
Save .....	4
Save as .....	4
Print.....	5
Exit .....	5
<b>DESIGN MENU.....</b>	<b>5</b>
Auto Design.....	5
Appendix A Design .....	7
User Design.....	7
Foundation.....	8
Dead Weight on Shell.....	9
Anchors .....	9
<b>ANALYSIS MENU.....</b>	<b>9</b>
Solution Controls .....	9
Liquid Level .....	11
Combustion Parameters.....	11
Begin Calculation.....	12
API 650 Calculation .....	13
Overturning Stability .....	13
<b>POSTPROCESS MENU .....</b>	<b>14</b>
Access Data .....	14
Plot .....	14
Zoom In .....	16
Zoom Reset.....	16
Reverse Contour .....	16
Displace Scale.....	16
List.....	16
List Font.....	16
<b>HELP MENU .....</b>	<b>16</b>
Index.....	16
Using Help .....	17
About.....	18
<b>EXAMPLE PROBLEM .....</b>	<b>19</b>
<b>STEPS.....</b>	<b>19</b>
<b>THEORY AND IMPLEMENTATION: STATIC ANALYSIS .....</b>	<b>22</b>

## Table of Contents

---

AUTOMATIC DESIGN .....	22
STRUCTURE ANALYSIS.....	22
Shell Elements.....	22
Foundation Elements.....	27
Finite Element Mesh Generation .....	29
Nonlinear Solution Procedure.....	33
COMBUSTION ANALYSIS.....	34
Solution Procedure .....	34
Combustion Wave Geometry.....	36
Burning Velocity.....	40
Venting .....	41
FRANGIBLE JOINT FAILURE.....	42
<b>THEORY AND IMPLEMENTATION FMA-3D ANALYSIS.....</b>	<b>45</b>
<b>VERIFICATION PROBLEM .....</b>	<b>48</b>
<b>REFERENCES .....</b>	<b>54</b>
<b>ACKNOWLEDGMENTS.....</b>	<b>55</b>
<b>AUTHOR'S COMMENTS ON PROGRAM STATUS AND FUTURE DEVELOPMENT .....</b>	<b>ERROR! BOOKMARK NOT DEFINED.</b>
<b>BLANK PAGES FOR USER NOTES .....</b>	<b>ERROR! BOOKMARK NOT DEFINED.</b>
<b>INDEX .....</b>	<b>56</b>



# INTRODUCTION

This manual describes the use of SafeRoof<sup>®</sup>, version 2.0, a program that has been developed to design and analyze storage tanks with frangible roof joints. The program is the result of a research program into frangible joint design, sponsored by the American Petroleum Institute and the Pressure Vessel Research Council.

API 650 (API, 1993) gives design rules for tanks used to store flammable liquids which could be subjected to sudden over-pressurization. Over-pressurization can occur due to the ignition of flammable vapors and can lead to the catastrophic loss of tank integrity. Undesirable modes of failure include loss of the shell-to-bottom joint or failure of the tank shell, either of which could result in loss of containment and spillage of the contents. To reduce this hazard, the frangible roof-to-shell joint is designed to fail first, providing a large venting capability to relieve the pressure. Research was initiated to address concerns about the general applicability of the present frangible joint design rules.

As part of this research, scale model tanks were dynamically tested by igniting an air/methane mixture in the tanks. Experimental observations showed that failure initiated when yielding was reached at the roof-to-shell joint. This was followed by buckling at the joint, tearing of the weld, and failure propagation in the weld and joint. Detachment of the roof relieved pressure in the tank. Because pressure relief does not occur instantaneously, the peak pressure in the tank can be larger than the failure initiation pressure. Further details of the model and testing are documented in the final research report, Swenson, et al., (1994).

SafeRoof incorporates the results of this research. It includes design, analysis, and post-processing modules. In the design module, the user can input tank parameters and SafeRoof will develop a design following API 650 guidelines. This design can either be accepted or modified. The user can then analyze the stresses and displacements in the tank at pressures corresponding to selected tank failure modes. The analysis can be coupled to a combustion/joint failure analysis. The pressures at each failure mode can be used to help evaluate safety of the tank due to overload pressures.

This manual is arranged with instructions on installing and running SafeRoof first. Details of the automatic design, finite element theory, and combustion analysis are given in later sections. The user is reminded that many questions can be answered using the help functions.

Version 2.0 incorporates the capability to perform dynamic, large displacement, elastic-plastic analyses of tank response. This was accomplished by incorporating FMA-3D (FMA, 2004)

## INSTALLING SAFEROOF

SafeRoof runs in Microsoft 9x, NT, 2000, and XP. Using the CD, installation is run automatically. If installation does not start, select Run C:\setup, where C is the name of the drive of the installation disc. The user can now run SafeRoof.

# USING SAFEROOF

SafeRoof runs in Microsoft Windows, so a basic knowledge of Windows usage is assumed. The menus are designed so that the user works from left to right to define, perform, and post-process results.

This section provides a description of the menus and dialog boxes in SafeRoof. Example problems are given in the following section.

SafeRoof is started by either selecting the executable file in the file directory or by creating a program group with SafeRoof as a program item. When SafeRoof is defined as a program item, an icon will be displayed.

## FILE MENU

When SafeRoof is started, the user is shown a blank window with menu selections. It is necessary to select the **FILE MENU** before an analysis can be performed or previous results accessed. After selecting **FILE MENU**, the following choices are available.

### **New**

**New** is used to define the file name of a new analysis. The user can select the directory and file name.

### **Open**

Open can be used to open a previous analysis. When this is done, the user can immediately proceed to the **POSTPROCESS MENU** to examine previously calculated results. Alternately, the user can modify the portions of the design or analysis options and perform new calculations.

### **Save**

**Save** is used to save a design or results. Only active if a design or analysis has been performed.

### **Save as**

This allows the user to save the current design under a new name. However, this does not make copies of the results files. Only active if a design has been performed.

## Print

**Print** sends the current window display to the default printer. When printing a plot of the design or results, using the landscape mode of the printer will likely result in the best fit of the graphic image on the page. Portrait mode is best for listings.

## Exit

Exits the program.

## DESIGN MENU

The **DESIGN MENU** is used to define the tank for which a design is requested. The user supplies sufficient data for SafeRoof to design a tank following API 650 rules. After a design is proposed the user can override the suggestion and define a tank using a different design.

### Auto Design

Selecting **Auto Design** displays the Design Parameters dialog box shown in Figure 1.

Design Parameters	
<b>Basic Geometry</b>	
Tank Diameter (ft):	50.00
Tank Height (ft):	36.00
Roof Slope (in/ft):	0.75
Course Height (in):	96.00
Maximum Liquid Level (ft):	35.00
Specific Gravity of Liquid:	0.95
<b>Position of Top Angle</b>	
<input type="radio"/> In	<input checked="" type="radio"/> Out
<b>Corrosion Allowance</b>	
Shell (in):	0.0625
Bottom (in):	0.0000
<b>Permissible Plate Material</b>	
ASTM A36	Yield Strength (ksi): 36.0
Allowable Product Design Stress (ksi): 23.2	
Allowable Hydrostatic Test Stress (ksi): 24.9	
OK Cancel Help	

Figure 1: Design parameters dialog box

The data supplied in this dialog box is sufficient to develop a tank design following API 650 rules.

First, the user supplies the tank geometry, including: diameter, height, roof slope, and course height. The course height is the

height of the sheet steel used to form the tank sides. In addition, the user defines the direction of the top angle.

The corrosion allowances, maximum design height of the liquid, and specific gravity of the liquid to be stored are also defined. Note that the liquid height defined in this dialog box is used for design. The user can specify a different height for use in the stress analyses.

The material to be used in the design is selected. The allowable stress values for each material are those defined in API 650. If desired, the user can scroll to the bottom of the list and add a user-defined material. If this is done, the user will be prompted for the allowable stresses for the user-defined material.

After the tank and parameters are defined, the user selects **OK**. SafeRoof will then design a tank according to API 650 rules, Figure 2.

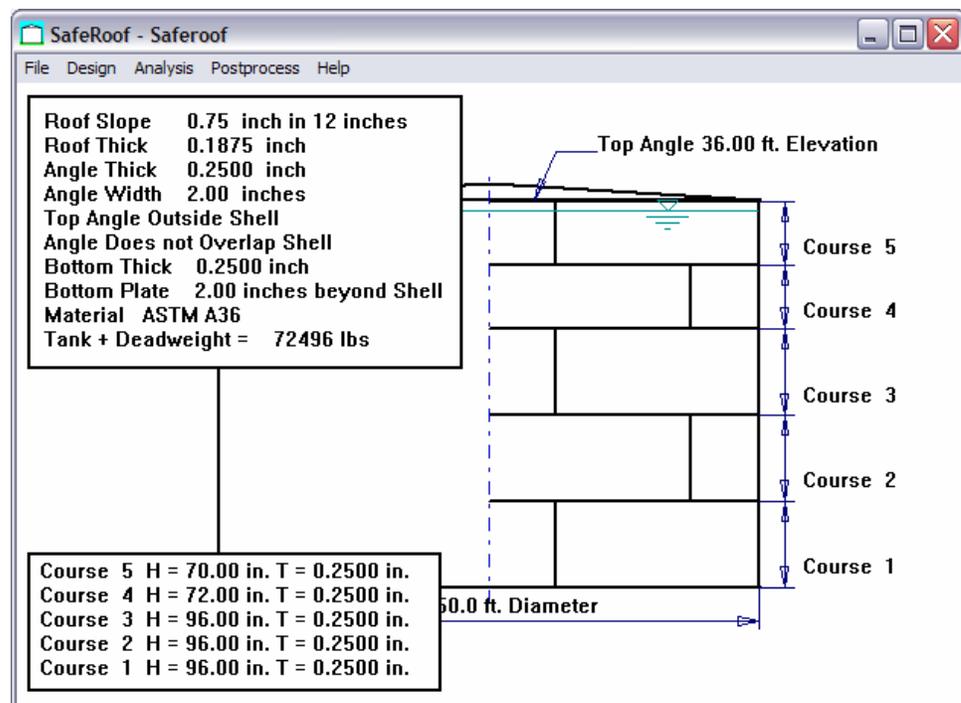


Figure 2: Window showing tank design

## Appendix A Design

Appendix A of API 650 offers an alternate design basis for small tanks. The user can prepare a design using Appendix A rules by selecting the Appendix A Design menu. This displays the dialog box shown in Figure 3.

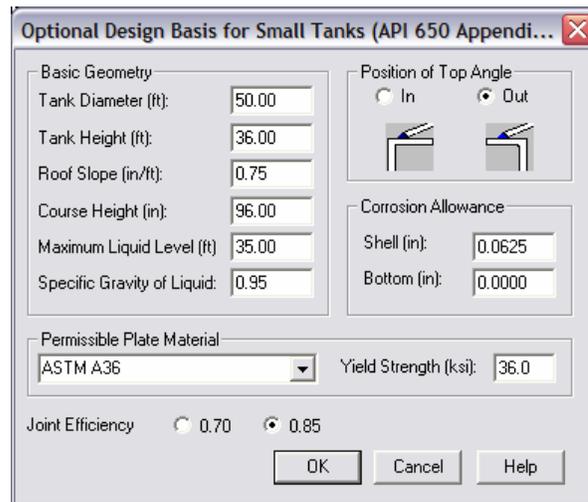


Figure 3: Appendix A design dialog box

Input to this design is similar as that described above.

## User Design

The user can also specify a design directly by selecting **User Design**, which displays the dialog box shown in Figure 4. This allows the user to over-ride the automatic design. The user can specify directly the number of courses, thickness, and height of each course. The user can also specify a top angle that overlaps the shell; this reduces frangibility of the joint, so is not recommended, but is used in some designs.

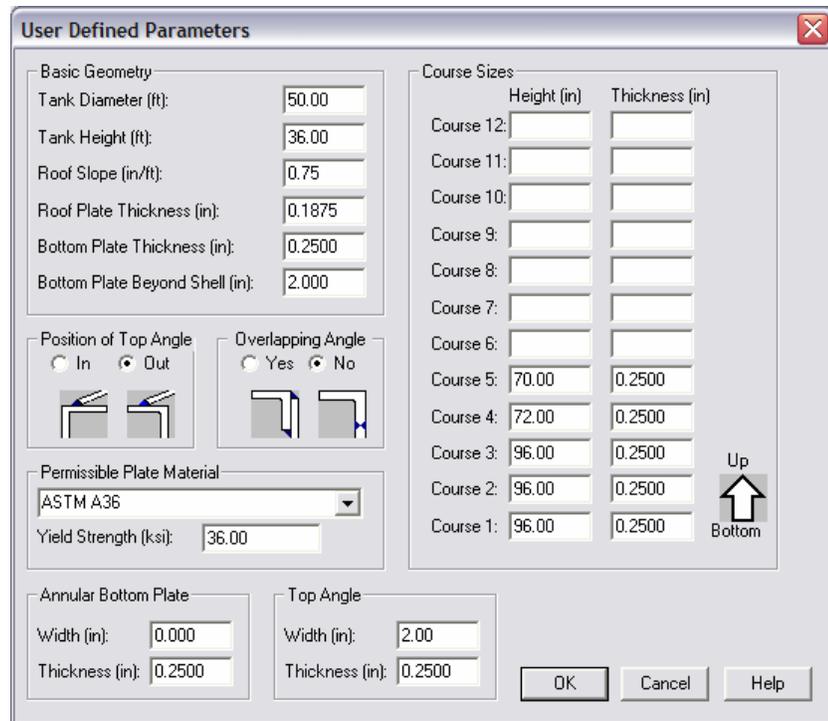


Figure 4: User defined menu

## Foundation

As described in the Theory Section, the model assumes that the tank rests on an elastic foundation. Two stiffnesses and a radius are used to describe the foundation. These can be used to model a tank resting on a sand foundation, with concrete supports under the shell. Inside the specified radius, the first stiffness is used and outside the radius, the second stiffness is used. Selecting **Foundation** displays the foundation dialog box shown in Figure 5.

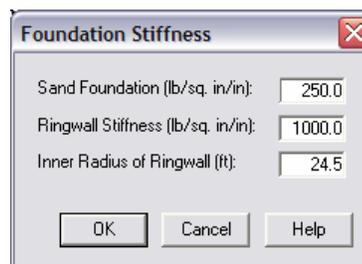


Figure 5: Foundation stiffness dialog box

## Dead Weight on Shell

The user can specify a dead weight on the shell. This weight is assumed to be distributed uniformly around the circumference at the bottom of the shell. This weight will act to reduce uplift of the bottom and is used in the analysis of overturning stability due to wind. When the **Dead Weight on Shell** menu is selected, the dialog box shown in Figure 6 is displayed.



Figure 6: Dead weight dialog box

## Anchors

Anchors can be used to resist the overturning moment. If anchors are included, they are used in the calculation of overturning stability. In addition, the anchor force will be distributed around the circumference of the bottom of the shell in the finite element analysis. The anchors are specified by selecting the **Anchor** menu, which displays the dialog box shown in Figure 7.

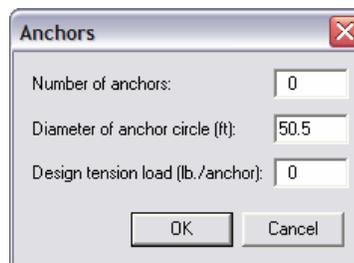


Figure 7: Anchor dialog box

## ANALYSIS MENU

The **ANALYSIS MENU** is used to control and perform a detailed finite element analysis of the tank. The program will also give the failure pressure predicted by API 650.

### Solution Controls

Selecting **Solution Controls** displays the dialog box shown in Figure 8. The user can either select a SafeRoof Linear Analysis,

which performs a large displacement, elastic analysis of the design, or a FMA3D Dynamic Analysis, which performs a dynamic, large displacement, elastic plastic analysis. If the SafeRoof Linear Analysis is selected, the user then selects the pressures for which the solution is desired: zero, bottom uplift, top angle yield, yield at the bottom joint, peak combustion pressure, or user specified. The user also specifies whether to include buckling effects in the roof and floor.

If the dynamic analysis is selected, the user can select either to specify a ramp loading or a combustion analysis, in a combustion calculation will first be performed and then the dynamic combustion pressure loads used in the analysis.

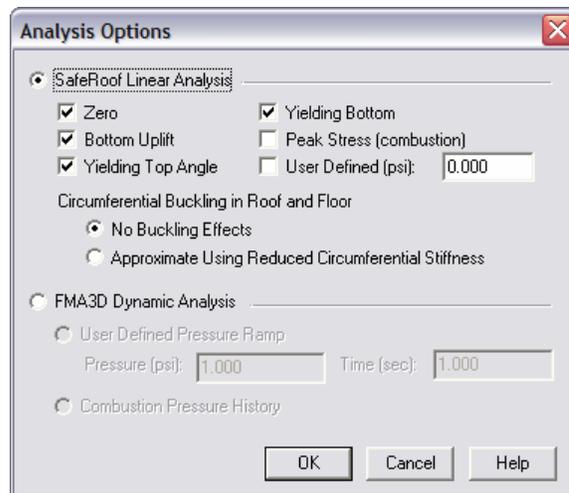


Figure 8: Solution controls dialog box

A zero pressure analysis does not have any internal pressure, but the tank will be loaded by gravity loads on the tank (floor, shell, roof, dead weight, and anchor) and the gravity load of any liquid in the tank. For the FMA3D analysis option, the zero pressure analysis is performed and used as the initial condition for the dynamic analysis.

If the SafeRoof Linear Analysis option is selected the user can specifically specify which pressure solutions they desire (uplift, top angle yield, etc). For the FMA3D analysis option, a transient analysis is performed using the zero pressure state as an initial condition. The transient analysis results are scanned to identify which failure conditions were met and the user can then plot those results.

For both static or dynamic analyses, the conditions are identified as:

- Uplift analysis is defined as the pressure to just cause uplift of the shell.
- Top angle yield is the pressure that causes yielding in the top angle. Again, gravity loads are included.
- Bottom joint yield is the pressure necessary to cause yielding at the bottom joint (including gravity loads).

For only the elastic analysis, the peak pressure analysis is performed using the combustion model. For this case, the pressure to cause yielding at the top angle is first calculated. A combustion analysis is then performed. If the pressure during combustion exceeds the pressure to yield the top angle, the combustion analysis proceeds with a top failure analysis also included. The final result is calculated at the peak pressure, while combustion is occurring, but before venting at the roof lowers the tank pressure.

Finally, for an elastic analysis, the user may specify a pressure for the analysis.

Results will be stored in a data file and available for postprocessing.

## Liquid Level

It is not necessary that the liquid level used for analysis be the same as used for design. In fact, yielding of the bottom joint typically occurs at lower pressures for lower liquid levels. Selecting **Liquid Level** allows the user to change the value used for analysis. The default value is the design value.

## Combustion Parameters

If the user has selected an analysis that requires a coupled combustion calculation, the user can modify the combustion parameters. Selecting Combustion Parameters displays the dialog box shown in Figure 9.

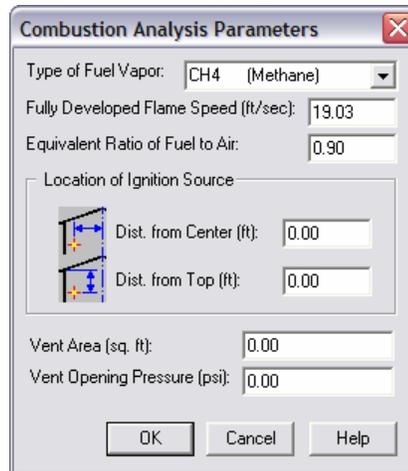


Figure 9: Combustion parameters dialog box

In this dialog box, the user can select different fuels and the location of the ignition. The most rapid pressure rise in the tank will occur if the ignition source is far from the wall, roof, and liquid. If the combustion front contacts any of these surfaces, the volume of gas participating in the combustion is reduced.

### **Begin Calculation**

Selecting **Begin Calculation** starts the calculations.

## API 650 Calculation

Selecting **API 650 Calculation** provides a dialog box (Figure 10) that gives the results predicted using API 650.

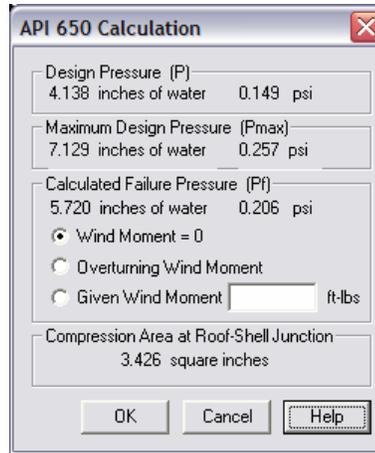


Figure 10: API 650 calculation dialog box

The API 650 results include the API 650 maximum design pressure, the API 650 predicted uplift pressure, the API 650 frangible joint failure pressure and the compression area at the roof-to-shell junction. Since wind moment is used in the design pressure calculation, the user can either use the default of zero, supply a wind moment, or use the moment as defined in the overturning calculation.

## Overturning Stability

The user can evaluate the design for overturning stability. When this is selected, the user is prompted for the wind velocity, operating internal pressure, and the additional overturning moment due to additional exposed area of structures on the tank (Figure 11). The moment due to the tank shell and roof are automatically calculated by the program. After selecting **OK**, the user will be advised whether the present design meets API 650 specifications. If not, the program gives a suggested anchor design. Selecting **ACCEPT** on the suggested anchor dialog box will add the anchors to the design.

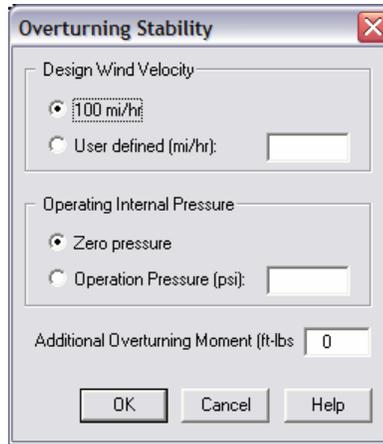


Figure 11: Overturning Stability dialog box

## POSTPROCESS MENU

Postprocessing allows the user to examine previously calculated results, either results for which analyses were just performed or to access the results of previous analyses.

### Access Data

Selecting **Access Data** allows the user to select a data set for postprocessing (Figure 12). Available sets are indicated in black type. As soon as a selection is made, the corresponding pressure and time are displayed.



Figure 12: Access data dialog box

### Plot

After a data set has been selected, the type of plot desired is indicated by selecting **Plot**. In the plot dialog box (Figure 13),

the user can select the plot type: tank design, finite element mesh, pressure time-history (if appropriate), nodal displacements, or element stresses. If element stresses are selected, the user specifies the surface and stress component to be plotted. Finally, the user specifies whether the plot should be sent to the screen (window) or to the printer. Selecting **OK**, generates the plot, as shown in Figure 14.

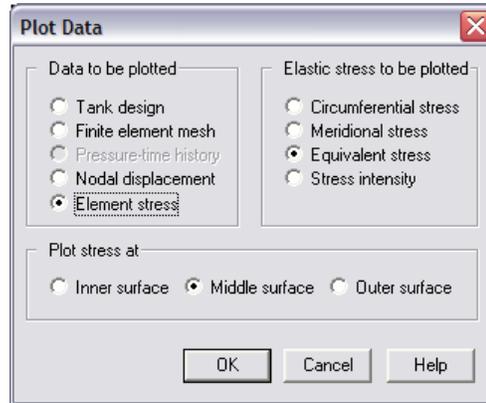


Figure 13: Plot data dialog box

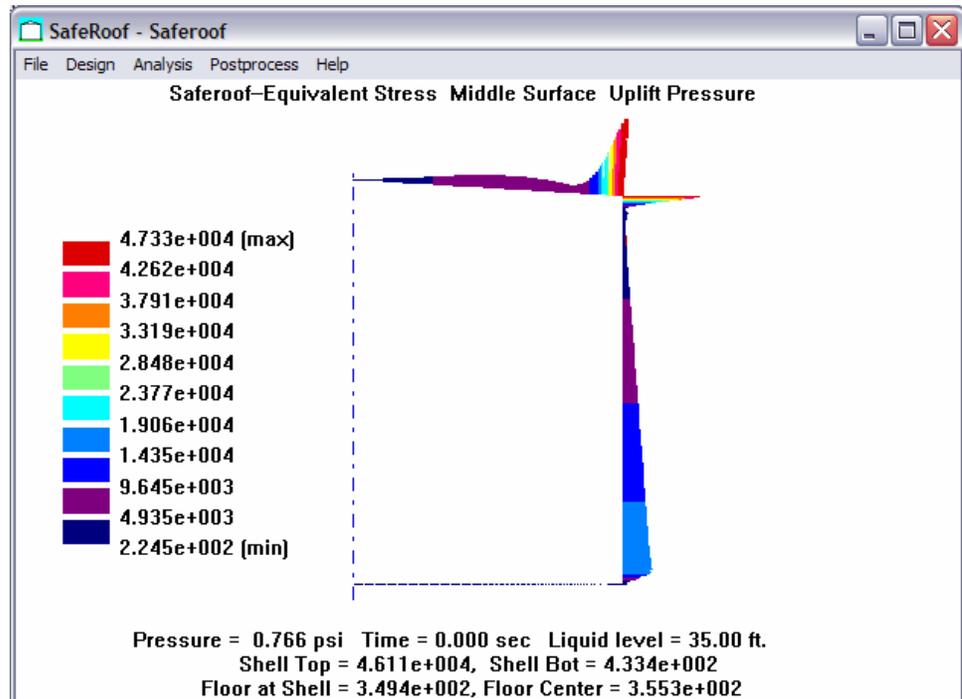


Figure 14: Typical plot

### **Zoom In**

Selecting **Zoom In** allows the user to indicate a region for more detailed display. The user indicates one corner with a mouse click, defining the box by moving the mouse with the mouse button depressed, and then releasing the button.

### **Zoom Reset**

**Zoom Reset** is used after zooming to return to the full display with automatic scaling of displacements.

### **Reverse Contour**

There are times when stresses can be viewed more clearly by reversing the direction of the stress contours. Selecting **Reverse Contour** does this.

### **Displace Scale**

If the user wants to control the magnification used in the displacement plots, selecting **Displace Scale** allows this.

### **List**

The user may list results to the screen, the printer, or to a file by selecting **List**. This is useful for documentation or for saving results to a file that then allows incorporating the results in another document.

### **List Font**

To have a well formatted list, the user can select the font for the list. Selecting a fixed-pitch font (e.g. Courier) aligns columns of the list.

### **Dynamic Display of Transient Analysis**

For a transient analysis, the user can hold down the Cntrl and Shift keys and then select the right or left arrows to move forward or backward in time. This can create a “movie” of the results. It is recommended that the user first set the Displace Scale to an appropriate value so that all images are displayed with the same magnification.

## **HELP MENU**

SafeRoof makes use of the Microsoft Windows help system. The user can obtain further information on most aspects of the program from the help file.

### **Index**

Selecting **Index** shows an index of the help available and allows the user to search for keywords.

### **Using Help**

Explains how to use the help functions.

## **About**

Gives information about the version of SafeRoof.

---

## EXAMPLE PROBLEM

This section provides an example problem. The user can follow this example to help understand the use of the program. It is assumed that the user has installed SafeRoof and is familiar with Windows. In the following discussion, menu selections are capitalized and typed user responses are underlined.

### STEPS

1. Begin running SafeRoof by double-clicking on the SafeRoof icon.
2. Select FILE/NEW and give example1 as the file name.
3. Select DESIGN/AUTO DESIGN. This displays the auto design dialog box.
4. This dialog box allows the user to specify all details of a tank needed for design. We will leave most of the parameters at their default values. By using the tab key or with the mouse, change the tank diameter to 60 feet and the tank height to 40 feet, then select the OK button. A warning box will appear stating that the liquid level for design is suspiciously low (by default 35 feet and the height is now 40 feet). Similar checks are made for several parameters to help ensure a valid design. Select OK to continue.
5. SafeRoof will now display a design that meets API 650 rules for frangible joints. We will continue with this design, however the user could select DESIGN/USER SPECIFIED to change the design.
6. Select ANALYSIS/SOLUTION CONTROLS. This displays the analysis controls dialog box.
7. Using the mouse, select pressures of Zero, Bottom Uplift, Yielding Top Angle, Yielding Bottom. This means SafeRoof will calculate stresses at these four pressures. Since the uplift and yielding pressures are not known, this will require iteration to determine these values. Select OK to continue.
8. Select ANALYSIS/LIQUID LEVEL to specify the liquid level for analysis. By default the analysis level is one foot below the top angle or the user defined design value, whichever is smaller. Note that some free space must be available in the tank if a combustion analysis is to be performed. In this example, change the liquid level to 10 feet and then select OK.

9. Select ANALYSIS/BEGIN CALCULATION to start the calculations. SafeRoof will solve for each of the pressures. Because the problem is nonlinear (large deformation and contact on the bottom), each solution is iterative. In addition there is a larger iteration loop when determining the pressures for yielding and uplift. This is the reason for the iterations displayed during the solution process. When each requested solution is completed, a file with the solution data is written. On a 25 MHz 486 computer, the zero pressure solution takes about 2 minutes, the uplift pressure solution takes 3 minutes, the top yield solution takes 6 minutes, and the bottom yield solution 22 minutes.
10. At the completion of the analysis, select POSTPROCESS/ACCESS DATA to examine results. The data access dialog box allows the user to select from the available data sets for plotting. For this example, we have four data sets available: zero pressure, bottom uplift, yielding top angle, and yielding bottom. Only one set can be accessed at a time, however, as you click on each selection box, the pressure and time corresponding to the data set is displayed. For this example, the bottom uplift pressure is 0.385 psi, the yielding top angle pressure is 0.452 psi, and the yielding bottom pressure is 1.366 psi. Thus, we would expect a small amount of uplift before failure initiation at the top angle, but there is a large margin before yielding at the bottom occurs. For illustration, select the yielding top angle data set, then OK.
11. For each data set, the user can plot the design, displacements, and stresses. To indicate the type of plot desired, select POSTPROCESS/PLOT. This displays the plotdata dialog box. Select Nodal Displacement then OK. This will make a displacement plot.
12. Use POSTPROCESS/ZOOM IN to examine a region more closely.
13. Use POSTPROCESS/DISPLACE SCALE to change the displacement scale.
14. Use POSTPROCESS/ZOOM RESET to view the full displaced mesh.
15. Select POSTPROCESS/PLOT to change the variable being plotted. From the plot data dialog box, select Element Stress, Middle Surface, Equivalent Stress to see the stresses in the tank. This creates a color stress contour display.

16. Select POSTPROCESS/LIST to list selected data to the screen or to a file. To improve the appearance of the list, select a non-proportional font (such as courier) using POSTPROCESS/LIST FONT.

17. Select FILE/EXIT to exit SafeRoof.

# THEORY AND IMPLEMENTATION: STATIC ANALYSIS

This describes the theory and implementation for the SafeRoof static, large deformation, elastic analysis capability.

## AUTOMATIC DESIGN

The automatic design option allows the user to input basic tank parameters, SafeRoof will then automatically design a tank to API 650 rules. Two automatic design options, API 650 Section 3 and Appendix F, are available. The basic design parameters are input by the user of the program. The following assumptions are made for the auto-designed tanks:

1. 3/16 inch thick roof plate supported by rafters,
2. Roof slope 3/4 : 12 inches,
3. Top angle does not overlap top shell,
4. Bottom plate (or annular bottom plate if there is one) projects two inches beyond the shell,
5. Shell thickness calculated by one foot method or Appendix A method.

The tank is assumed to rest on a sand foundation with a default modulus of 250 lb/in<sup>2</sup>/in. A ringwall with a default modulus of 1000 lb/in<sup>2</sup>/in and 6 inches width, measured from the tank shell, is also assumed.

The design can be checked for overturning stability due to a wind velocity of 100 miles per hour (or user specified value) and anchors will be designed if necessary.

## STRUCTURE ANALYSIS

### Shell Elements

The roof, shell, and bottom of the tank are modeled using axisymmetric conical shell elements, as shown in Figure 15. The element (Zienkiewicz, 1991) has constant thickness with three degrees of freedom at each node: translations in the nodal x and y directions and a rotation about the nodal z axis. The element has membrane and bending capabilities, as well as large deflection capability.

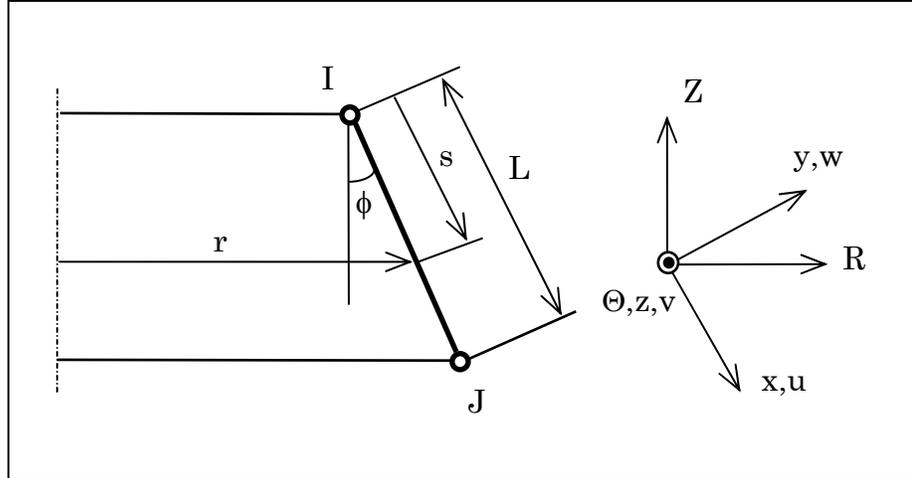


Figure 15: Axisymmetric shell element

The motion,  $\mathbf{u}^n$ , of a general point in the element is:

$$\mathbf{u}^n = \begin{Bmatrix} u \\ w \end{Bmatrix} \quad (1)$$

where  $u$  and  $w$  are the in-plane and lateral displacements, respectively. The circumferential displacement  $v$  is identically zero due to axisymmetry. The strain at a point is a function of the displacement:

$$\begin{aligned} \boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \end{Bmatrix} &= \begin{Bmatrix} \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 \\ \frac{1}{r} (u \sin \phi - w \cos \phi) \end{Bmatrix} + y \begin{Bmatrix} -\frac{\partial^2 w}{\partial x^2} \\ -\frac{\sin \phi}{r} \frac{\partial w}{\partial x} \end{Bmatrix} \\ &= \mathbf{A} \mathbf{u}^n \end{aligned} \quad (2)$$

$\mathbf{A}$  is an operator matrix defined in equation (3) where the subscripts  $c$ ,  $y$ , and  $l$  represent the center surface, bending and large deformation effects, respectively.

$$\begin{aligned} \mathbf{A} &= \mathbf{A}_c + \mathbf{A}_y + \mathbf{A}_l \\ &= \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ \frac{\sin \phi}{r} & -\frac{\cos \phi}{r} \end{bmatrix} + y \begin{bmatrix} 0 & \frac{\partial^2}{\partial x^2} \\ 0 & -\frac{\sin \phi}{r} \frac{\partial}{\partial x} \end{bmatrix} + \begin{bmatrix} 0 & \frac{1}{2} \frac{\partial w}{\partial x} \frac{\partial}{\partial x} \\ 0 & 0 \end{bmatrix} \end{aligned} \quad (3)$$

For a linear material, the stress-strain relation has the form:

$$\begin{aligned} \boldsymbol{\sigma} &= \mathbf{D} \boldsymbol{\varepsilon} \\ &= \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu \\ \nu & 1 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \end{Bmatrix} \end{aligned} \quad (4)$$

Using cubic shape functions for lateral displacement and nodal rotation and linear shape functions for in-plane displacement, the motion of a point  $\mathbf{u}^n$  can be interpolated in terms of nodal displacement  $\mathbf{u}_i$ :

$$\begin{aligned} \mathbf{u}^n &= \mathbf{N} \mathbf{u}_i \\ &= \begin{bmatrix} 0 & N_3^I & 0 & 0 & N_3^J & 0 \\ N_1^I & 0 & N_2^I & N_1^J & 0 & N_2^J \end{bmatrix} \begin{Bmatrix} w_I \\ u_I \\ \beta_I \\ w_J \\ u_J \\ \beta_J \end{Bmatrix} \end{aligned} \quad (5)$$

where  $\beta$  is the rotation at the node and the shape functions are:

$$\begin{aligned} N_1^I &= \frac{1}{2} - \frac{s}{4}(3-s^2) \\ N_1^J &= \frac{1}{2} + \frac{s}{4}(3-s^2) \\ N_2^I &= \frac{L}{8}(1-s^2)(1-s) \\ N_2^J &= -\frac{L}{8}(1-s^2)(1+s) \\ N_3^I &= \frac{1}{2}(1-s) \\ N_3^J &= \frac{1}{2}(1+s) \end{aligned} \quad (6)$$

The nodal displacements in the local coordinate system can be written in terms of the global system using the following transformation:

$$\mathbf{u}_l = \mathbf{T}_r \mathbf{u}_e$$

$$= \begin{bmatrix} \cos \phi & \sin \phi & 0 & 0 & 0 & 0 \\ \sin \phi & -\cos \phi & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos \phi & \sin \phi & 0 \\ 0 & 0 & 0 & \sin \phi & -\cos \phi & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_x^I \\ u_z^I \\ \beta^I \\ u_x^J \\ u_z^J \\ \beta^J \end{Bmatrix} \quad (7)$$

The strain-nodal displacement relation can now be written as:

$$\begin{aligned} \{\varepsilon\} &= \mathbf{A}\mathbf{N}\mathbf{u}_l \\ &= (\mathbf{A}_c + y\mathbf{A}_y + \mathbf{A}_l)\mathbf{N}\mathbf{u}_l \\ &= (\mathbf{B}_c + y\mathbf{B}_y + \mathbf{B}_l)\mathbf{u}_l \end{aligned} \quad (8)$$

Assuming a linear material, the total strain energy is the strain energy density integrated over the volume:

$$U = \int_{\text{vol}} \frac{1}{2} \boldsymbol{\sigma} : \boldsymbol{\varepsilon} \, d(\text{vol}) \quad (9)$$

The work done by an external force is the product of force and displacement:

$$V = \mathbf{F} \cdot \mathbf{u} = \mathbf{u}^T \int_{\text{vol}} \mathbf{N}^T \mathbf{f} \, d(\text{vol}) \quad (10)$$

where  $\mathbf{f}$  is the external loading on the element. By applying Hamilton's principle, the shell has the following governing equation:

$$\delta(U + V) = \delta \mathbf{u}^T \mathbf{K} \mathbf{u} - \delta \mathbf{u}^T \mathbf{F} = 0 \quad (11)$$

where the global stiffness matrix  $\mathbf{K}$  is the integration over the entire volume:

$$\mathbf{K} = \int_{\text{vol}} (\mathbf{B}_c + \mathbf{B}_l + \mathbf{B}_y)^T \mathbf{D} (\mathbf{B}_c + \mathbf{B}_l + \mathbf{B}_y) \, d(\text{vol}) \quad (12)$$

When integrating through the thickness of a conical shell, a factor  $(1 + y \frac{\cos\phi}{r})$  needs to be included. For any constant H, we have:

$$\int_{-t/2}^{t/2} H(1 + y \frac{\cos\phi}{r}) dy = Ht$$

$$\int_{-t/2}^{t/2} H(1 + y \frac{\cos\phi}{r}) y dy = H \frac{t^3 \cos\phi}{12 r}$$

$$\int_{-t/2}^{t/2} H(1 + y \frac{\cos\phi}{r}) y^2 dy = H \frac{t^3}{12} \tag{13}$$

The element stiffness matrix in local coordinates is:

$$\begin{aligned} \mathbf{K}_1 = & \int_{\text{area}} t \mathbf{B}_c^T \mathbf{D} \mathbf{B}_c d(\text{area}) \\ & + \int_{\text{area}} \frac{t^3 \cos\phi}{12 r} (\mathbf{B}_c^T \mathbf{D} \mathbf{B}_y + \mathbf{B}_y^T \mathbf{D} \mathbf{B}_c) d(\text{area}) \\ & + \int_{\text{area}} \frac{t^3 \cos\phi}{12 r} (\mathbf{B}_c^T \mathbf{D} \mathbf{B}_y + \mathbf{B}_y^T \mathbf{D} \mathbf{B}_c) d(\text{area}) \\ & + \int_{\text{area}} \frac{t^3}{12} \mathbf{B}_y^T \mathbf{D} \mathbf{B}_y d(\text{area}) \\ & + \int_{\text{area}} t (\mathbf{B}_c^T \mathbf{D} \mathbf{B}_1 + \mathbf{B}_1^T \mathbf{D} \mathbf{B}_c + \mathbf{B}_1^T \mathbf{D} \mathbf{B}_1) d(\text{area}) \\ & + \int_{\text{area}} \frac{t^3 \cos\phi}{12 r} (\mathbf{B}_y^T \mathbf{D} \mathbf{B}_1 + \mathbf{B}_1^T \mathbf{D} \mathbf{B}_y) d(\text{area}) \end{aligned} \tag{14}$$

The element stiffness matrix in global coordinates will be:

$$\mathbf{K}_e = \mathbf{T}_r^T \mathbf{K}_1 \mathbf{T}_r \tag{15}$$

As is shown in Figure 16, the loads considered are gravity loads,  $\mathbf{F}_1$ , pressure loads,  $\mathbf{F}_2$ , and the secondary effect of pressure load,  $\mathbf{F}_3$ . The reaction force from the elastic foundation will be converted to stiffness and will be discussed later.

$$\mathbf{F} = \mathbf{F}_1 + \mathbf{F}_2 + \mathbf{F}_3 \tag{16}$$

For an element, the gravity load in local coordinates is:

$$\mathbf{f}_{11} = -2\pi t \int_0^L r \mathbf{N}^T \begin{Bmatrix} \rho g \cos \phi \\ -\rho g \sin \phi \end{Bmatrix} dx \quad (17)$$

The element pressure loading vector in local coordinate is:

$$\mathbf{f}_{21} = -2\pi \int_0^L r \mathbf{N}^T \begin{Bmatrix} 0 \\ p_1 - p_2 \end{Bmatrix} dx \quad (18)$$

The normal stress due to pressure converted to strain in the plane of the shell has the form:

$$\boldsymbol{\varepsilon}^{pr} = -\frac{\nu}{2E} (p_1 + p_2) \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} - \frac{\nu y}{2Et} (p_2 - p_1) \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \quad (19)$$

The equivalent load vector is:

$$\mathbf{f}_{31} = \int_{vol} \mathbf{B}^T \mathbf{D} \boldsymbol{\varepsilon}^{pr} d(vol) \quad (20)$$

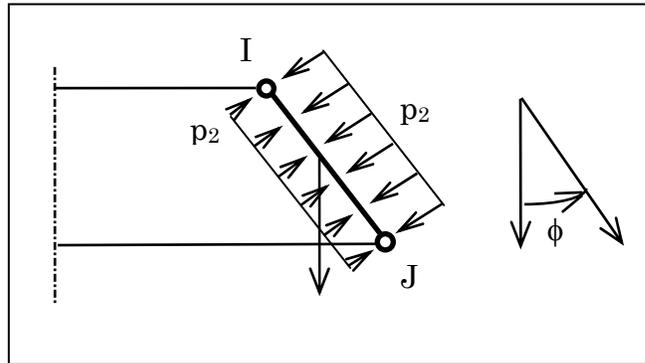


Figure 16: Loading on element

### Foundation Elements

Foundation elements are used to represent the nonlinear reacting forces on the bottom of the tank by the foundation. The foundation of the tank is modeled as an elastic interface without friction.

The total surface traction force in the global coordinate can be written as an integral over the entire surface:

$$\mathbf{F}^e = \int_s \mathbf{N}^T \mathbf{t} dS \quad (21)$$

where  $\mathbf{t}$  is the surface traction in global coordinate. For an element, the surface traction in local coordinates can be obtained by the following transformation:

$$\begin{aligned} \mathbf{t} &= \mathbf{R}^T \mathbf{t}^n \\ &= \begin{bmatrix} \cos\phi & \sin\phi & 0 \\ \sin\phi & -\cos\phi & 0 \\ 0 & 0 & 1 \end{bmatrix} \mathbf{t}^n \end{aligned} \quad (22)$$

where  $\mathbf{R}$  is the coordinate transformation matrix. The local surface traction and local displacement has the relation:

$$\begin{aligned} \mathbf{t}^n &= -\mathbf{T} \mathbf{u}^n \\ &= - \begin{bmatrix} 0 & 0 \\ 0 & k_f \end{bmatrix} \mathbf{u}^n \end{aligned} \quad (23)$$

where

$$k_f = \begin{cases} 0 & \text{when interface open} \\ \text{stiffness of foundation} & \text{when interface closed} \end{cases}$$

Substituting equations (5), (22) and (23) into (21), gives the foundation force vector and foundation stiffness matrix:

$$\begin{aligned} \mathbf{F}^e &= -\mathbf{K}^e \mathbf{u}_1 \\ &= -\int_S \mathbf{N}^T \mathbf{R}^T \mathbf{T} \mathbf{R} \mathbf{N} dS \mathbf{u}_1 \end{aligned} \quad (24)$$

$$\mathbf{K}^e = \int_S \mathbf{N}^T \mathbf{R}^T \mathbf{T} \mathbf{R} \mathbf{N} dS \quad (25)$$

The foundation stiffness matrix is evaluated every iteration and added to the global stiffness matrix.

### Finite Element Mesh Generation

The tanks considered in this program are assumed to be at least 5.0 feet in diameter with two shell courses at least 2.5 feet high. The finite element mesh used for modeling the tank is generated as shown in Figure 17 and 18 and described in Tables 1 and 2.

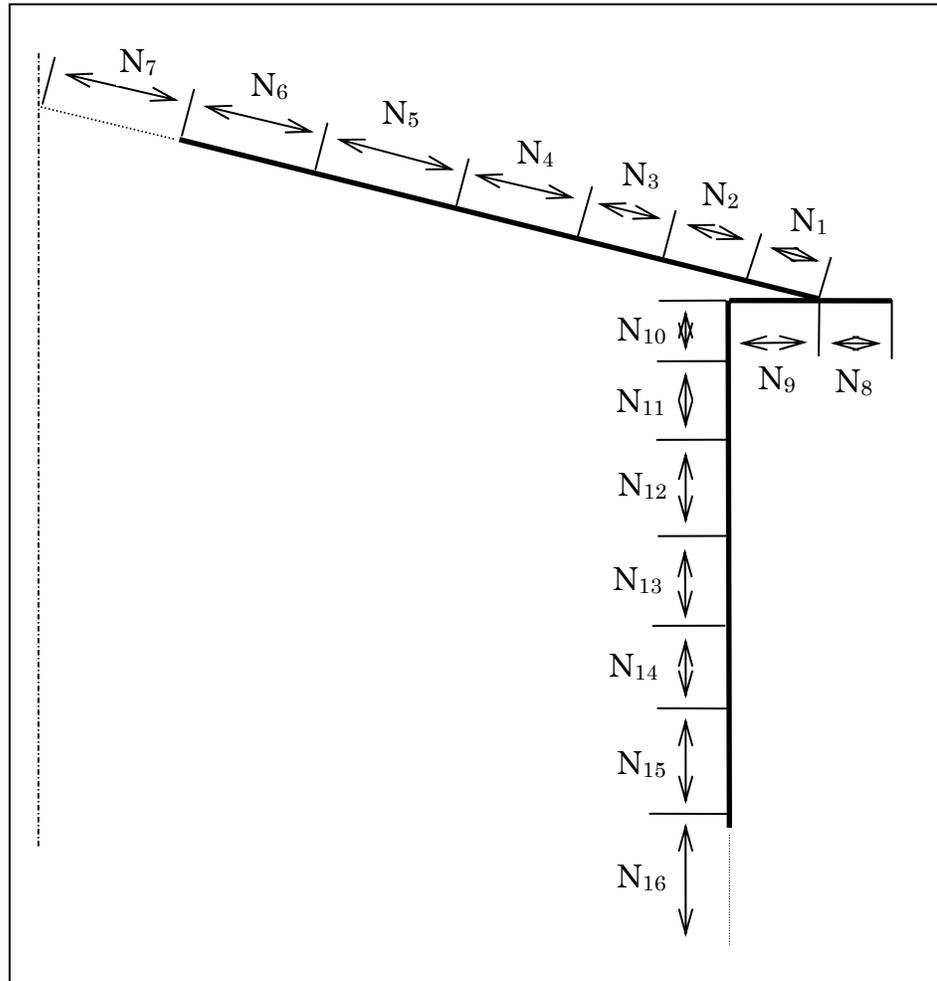


Figure 17: Finite element mesh on upper part of the tank

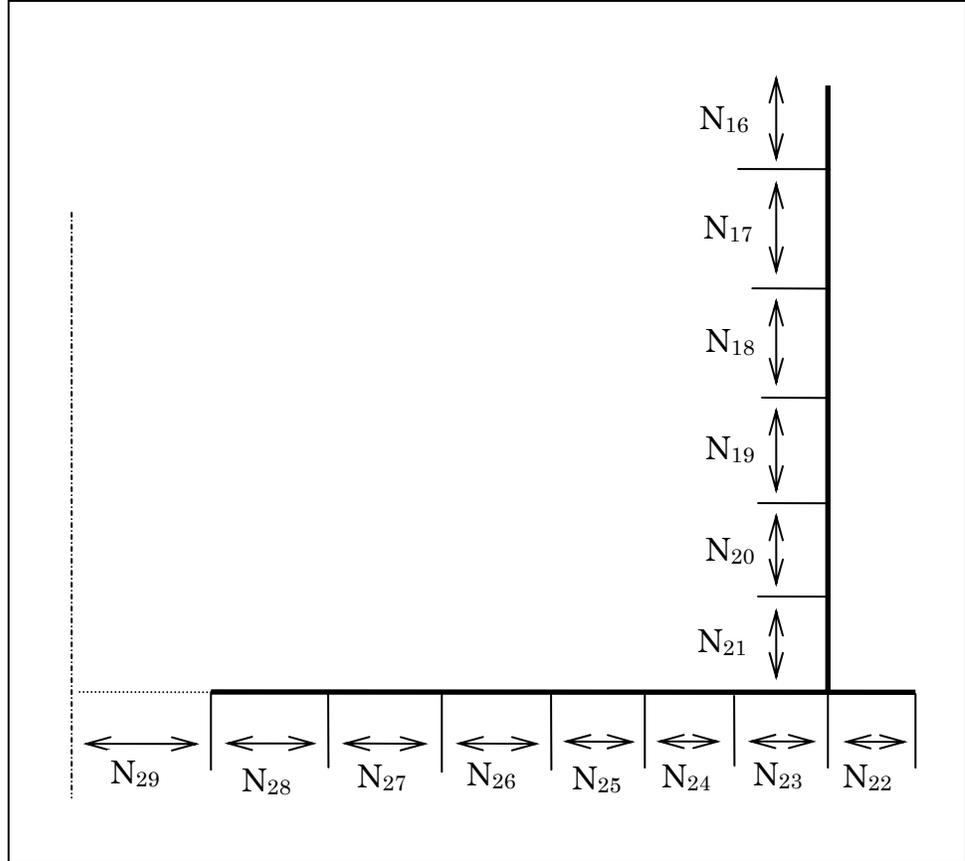


Figure 18: Finite element mesh on lower part of the tank

Region	Number of Elements	Size of Elements	Condition
N <sub>1</sub>	12	0.5	Always
N <sub>2</sub>	12	1.0	Always
N <sub>3</sub>	(R-18)/2 6	Calculated 2.0	24 < R < 33 33 < R
N <sub>4</sub>	0 (R-30)/3 6	- Calculated 3.0	R < 33 33 < R < 52 52 < R
N <sub>5</sub>	0 (R-48)/4 12	- Calculated 4.0	R < 52 52 < R < 102 102 < R
N <sub>6</sub>	0 (R-96)/6 10	- Calculated 6.0	R < 102 102 < R < 168 168 < R
N <sub>7</sub>	0 (R-156)/12	- Calculated	R < 168 168 < R
N <sub>8</sub>	2 2	0.5 1/3 Angle Width	Angle Width > 1 Angle Width < 1
N <sub>9</sub>	Angle Width - 1 1	0.5 1/3 Angle Width	Angle Width > 1 Angle Width < 1
N <sub>10</sub>	2 x Angle Width 1	0.5 2 x Shell Thick	Angle not overlap shell Angle overlap shell
N <sub>11</sub>	12	0.5	Always
N <sub>12</sub>	12	1.0	Always
N <sub>13</sub>	6	2.0	Always
N <sub>14</sub>	6 (Top course height-30)/3+1	4.0 Calculated	Top course height > 48 102 < R < 168
N <sub>15</sub>	0 (Top course height-48)/2+1	- Calculated	Top course height < 48 Top course height > 48
N <sub>16</sub>	10	Calculated	All intermediate courses

Table 1: Finite element mesh on upper part of tank

<b>Region</b>	<b>Number of Elements</b>	<b>Size of Elements</b>	<b>Condition</b>
N <sub>16</sub>	10	Calculated	All intermediate courses
N <sub>17</sub>	0 (Bottom course height-48)/2+1	- Calculated	Bottom course < 48 Bottom course > 48
N <sub>18</sub>	6 (Bottom course height-48)/3+1	4.0 Calculated	Bottom course < 48 30 < Bottom course < 48
N <sub>19</sub>	6	2.0	Always
N <sub>20</sub>	12	1.0	Always
N <sub>21</sub>	12	0.5	Always
N <sub>22</sub>	2	Calculated	Always
N <sub>23</sub>	12	0.5	Always
N <sub>24</sub>	12	1.0	Always
N <sub>25</sub>	(R-18)/2 6	Calculated 2.0	24 < R < 33 33 < R
N <sub>26</sub>	0 (R-30)/3 6	- Calculated 3.0	R < 33 33 < R < 52 52 < R
N <sub>27</sub>	0 (R-48)/4 12	- Calculated 4.0	R < 52 52 < R < 102 102 < R
N <sub>28</sub>	0 (R-96)/6 10	- Calculated 6.0	R < 102 102 < R < 168 168 < R
N <sub>29</sub>	0 (R-156)/12	- Calculated	R < 168 168 < R

Table 2: Finite element mesh on lower part of tank

## Nonlinear Solution Procedure

At the start of the structural analysis, the finite element model is meshed and a linear stiffness matrix formed. For the specified pressure, a global loading vector is obtained and saved. After each iteration, part of the newly obtained displacement will be added to the previous one as the next estimate of the solution. The nonlinear part of the stiffness matrix (due to the large deflection and elastic foundation) are evaluated using the new solution and then used for the next iteration. If available, a previous solution is used as the first guess. The algorithm can be written as:

$$\begin{aligned}
 &\text{do}\{ \\
 &\quad \mathbf{U}'_n = (\mathbf{K}_0 + \mathbf{K}_1^{n-1})^{-1} \mathbf{F} \\
 &\quad \mathbf{U}_n = (1 - \alpha)\mathbf{U}_{n-1} + \alpha\mathbf{U}'_n \\
 &\quad \mathbf{K}_1^n = \mathbf{K}_1^n(\mathbf{U}_n) \\
 &\text{while}(\|\mathbf{U}_n - \mathbf{U}_{n-1}\|_\infty \leq 0.001\|\mathbf{U}_n\|_\infty) \tag{26}
 \end{aligned}$$

where:

- $\mathbf{U}'_n$  = new calculated displacement vector
- $\mathbf{U}_n$  = displacement vector after n-th iteration
- $\mathbf{K}_1^n$  = nonlinear portion of stiffness after n-th iteration
- $\mathbf{K}_0$  = linear portion of stiffness, constant for a given mesh
- $\mathbf{F}$  = global loading vector, constant for a given pressure
- $\alpha$  = relaxation coefficient, changes between 0.05 to 0.5

If the yielding pressure calculation is required for the top joint, a trial pressure will be used first. After the converged displacement has been calculated, the mean equivalent stress at the entire horizontal part of the top angle is calculated and compared to the yielding stress of the material. A new trial pressure is found by Newton's method for the next pressure step until the calculated stress is within  $\pm 0.5\%$  of the yielding stress.

The calculation of bottom yield pressure is similar to that of top yield. The equivalent stress is checked for both shell and bottom elements near the joint and the maximum element stress is compared to the yielding stress.

Due to the nonlinearity of the elastic foundation at the bottom, Newton's method does not give a reliable estimate of the bottom uplift pressure while doing the large iteration loop to determine that pressure. An assumed upper and lower bound of pressure is tried first and a new trial pressure is found by reducing the searching region by half. The calculation is completed when the vertical displacement of the bottom joint node is less than 1% of that of the node at the center of the tank bottom or the pressure between the upper and lower bounds is less than 0.02 psi.

To find a peak pressure in the tank during combustion, the top yield pressure is first calculated. Then a combustion analysis is performed with venting beginning when the top yield pressure is reached. The combustion calculation finishes when the pressure in the tank reduces to zero due to venting. The peak pressure during the combustion process is used for the peak stress calculation.

After each set of calculations is done, the displacement, finite element mesh, and other analysis parameters are written to a solution file along for use during post-processing.

## **COMBUSTION ANALYSIS**

### **Solution Procedure**

The pressure rise in the vapor space inside the tank is calculated by evaluating combustion in small increments of time. The flame front is assumed to grow in a spherical fashion originating from a point source ignition in the tank. The volume of reactants burned in a given time increment is used to determine the pressure after each time increment. To evaluate this pressure, the speed at which the flame front moves is multiplied by a time increment, giving a distance. From this distance, an incremental volume is calculated. This volume is then burned at constant volume combustion and allowed to expand until the pressure inside the tank is uniform.

To calculate the pressure rise during a time increment, an adiabatic flame temperature is calculated for the flame front, assuming constant volume combustion and chemical equilibrium. The pressure inside the volume swept out by the flame front, during a single time increment, increases due to the burning of the reactants at constant volume. The volume swept by the flame front is then allowed to expand, causing the

reactants and products to compress, with no mixing, until the pressure inside the tank is uniform. After the pressure inside the tank is balanced, time is incremented and the process repeated. When the internal pressure reaches the roof-to-shell joint yielding pressure, a venting calculation starts and part of the combustion products are vented, reducing the pressure. The peak pressure found during the combustion calculation and is used for the structural calculation. The combustion calculation continues until the pressure in the tank is reduced to atmospheric pressure.

In these calculations, the tank is assumed to be adiabatic and the tank rigid. The analysis also neglects radiation heat transfer between the flame front and wall of the tank.

## Combustion Wave Geometry

This analysis assumes that the combustion wave will burn as a radial deflagration, having a smooth spherical shaped flame front from a point source ignition.

The flame front is assumed to grow in a spherical fashion originating from a point source ignition in the tank. As the flame front burns across the reactants, the products formed will occupy the volume inside the flame front, and are not allowed to mix with the reactants. If the ignition source is not in contact with the tank or the fluid being stored and the time duration from the ignition is small, the volume is a sphere. Otherwise the fireball will be constrained by the tank or liquid in the tank. Numerical integration is necessary to calculate the volume of the fireball with sufficient accuracy.

Consider a tank of radius  $R$  with height of vapor space  $H$  as shown in Figure 19.

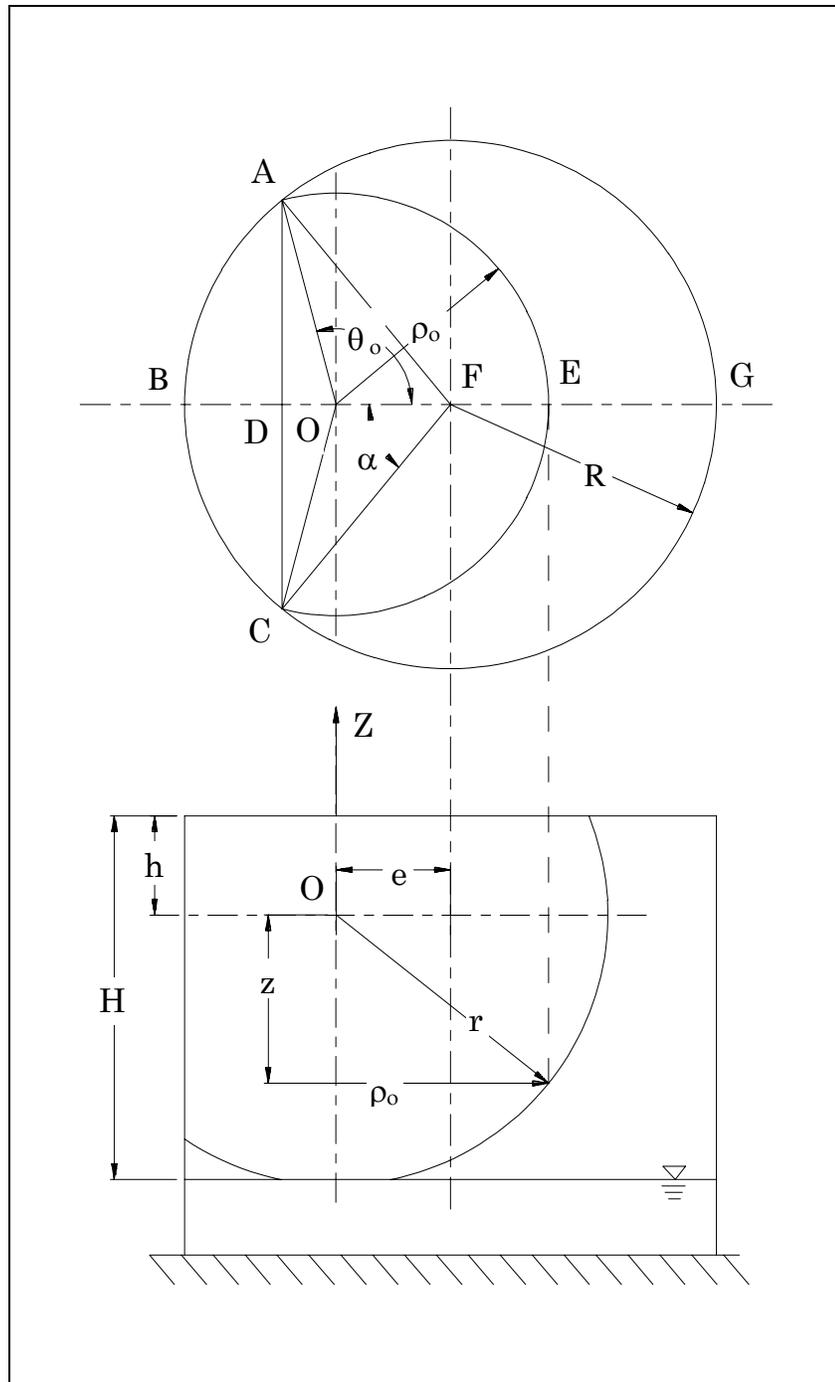


Figure 19: Geometry of fireball at position z

The ignition source is located at point  $O$ , which is distance  $e$  away from center line of the tank shell and distance  $h$  below the roof of the tank. At a given instance, the radius of the fireball is  $r$ . Using a cylindrical coordinate system with the origin at the ignition source and the  $z$  axis up, the cross section area of the

fireball S at height z can be determined analytically. The total volume of the fireball V can be obtained using the following integration:

$$V = \int_{\max(h-H, -r)}^{\min(h,r)} S dz \tag{27}$$

When the ignition source is on the centerline of the tank, the value of e equals zero and the cross section at any z location will be a circle with radius :

$$\rho_o = \sqrt{r^2 - z^2} \tag{28}$$

When e is not zero, the calculation of cross sectional area can be performed for three cases. When the fireball and the tank shell do not touch each other for the given z, the cross sectional area is that of a circle with radius r, as given in equation 28. Otherwise, the cross sectional area is the common part of two circles. For a fireball as shown in Figure 19, the total cross section area S at height z is:

$$\begin{aligned} S = & (r^2 - z^2) \cos^{-1} \left( \frac{R^2 - e^2 - r^2 + z^2}{2e} \right) \\ & + R^2 \sin^{-1} \left[ \frac{1}{R} \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \right] \\ & - \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \sqrt{R^2 - r^2 + z^2 + \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \\ & + \frac{R^2 - e^2 - r^2 + z^2}{2e} \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \end{aligned} \tag{29}$$

Equation (29) is valid for  $\alpha < \pi/2$ , that is, cross section with z coordinate satisfies the following relations:

$$(R - e)^2 < r^2 - z^2 < R^2 + e^2 \tag{30}$$

In case of  $\alpha$  is bigger than  $\pi/2$ , the geometry of the cross section is shown in Figure 20. In this case, the total area of the cross section will be:

$$\begin{aligned}
 S = & (r^2 - z^2) \cos^{-1} \left( \frac{r^2 - z^2 + e^2 - R^2}{2e\sqrt{r^2 - z^2}} \right) \\
 & + \frac{R^2 - e^2 + z^2 - r^2}{2e} \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \\
 & + \pi R^2 - R^2 \sin^{-1} \left[ \frac{1}{R} \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \right] \\
 & + \sqrt{r^2 - z^2 - \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}} \sqrt{R^2 - r^2 + z^2 + \frac{(r^2 - z^2 + e^2 - R^2)^2}{4e^2}}
 \end{aligned}
 \tag{31}$$

The calculated area S is then integrated with respect to z using Runge-Kutta-Gill method. The integration step length is set to 1/150 of total vapor space height, which will give an error equal or less than  $4.2 \times 10^{-10} r^5$ .

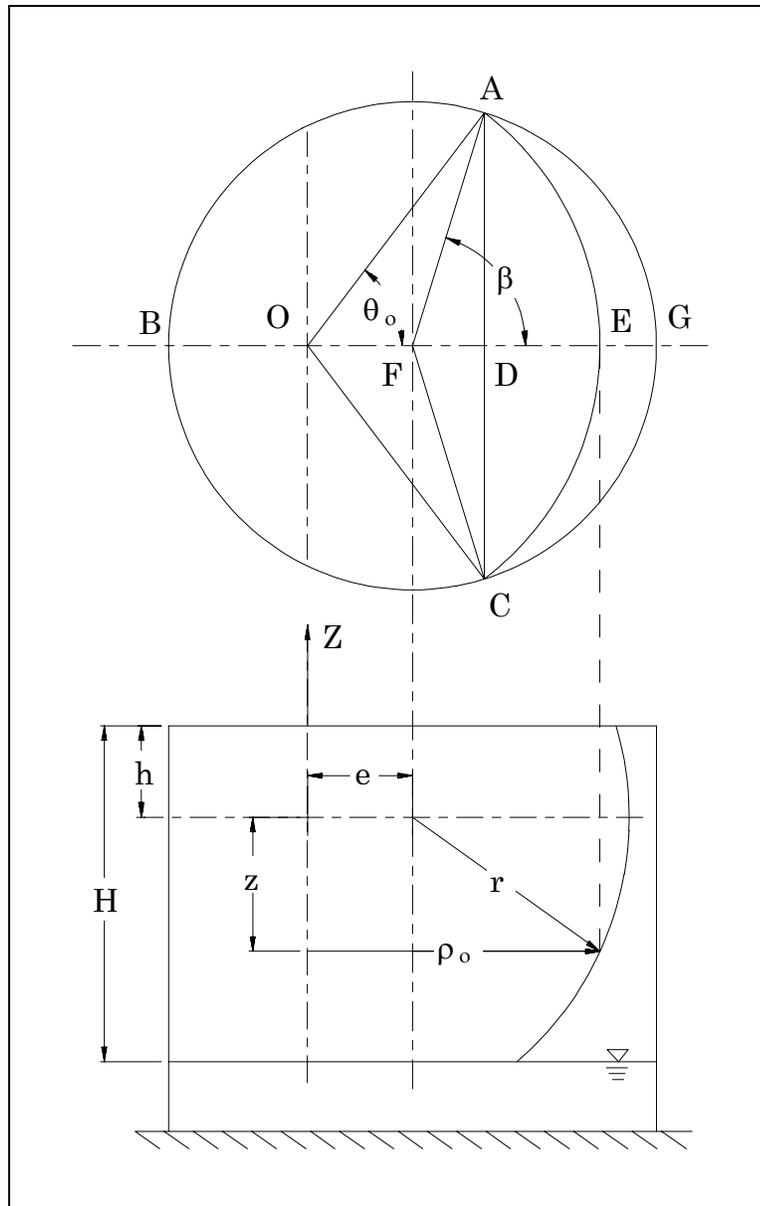


Figure 20: Geometry of the Fireball at Position  $z$   
 when  $R^2 + e^2 < r^2 - z^2 < (R + e)^2$

### Burning Velocity

The combustion process is assumed to be a deflagration, where the burning velocity starts at the laminar velocity and accelerates as a function of the temperature, pressure, and level of turbulence inside the tank. The speed of the flame front as observed from a stationary coordinate has the following form:

$$S = U_t \left[ 1 - e^{-0.2 \left( \frac{T_k}{\tau_s} \right)^{0.75}} \right]^{0.5} \left( \frac{P_u}{P_o} \right)^{-0.04} \left( \frac{T_u}{T_o} \right)^2 (-2.614 + 7.288\Phi - 3.614\Phi^2) \quad (32)$$

Where:

- S = flame speed (m/s),
- $P_u$  = reactant pressure (Pa),
- $P_o$  = initial pressure (Pa),
- $T_u$  = reactant temperature (K),
- $T_o$  = initial temperature (K),
- $U_t$  = fully developed turbulent burning flame speed (m/s),
- $T_k$  = elapsed time from ignition (s),
- $\tau_s$  = integral time scale (s),
- $\Phi$  = equivalence ratio, defined as the actual mass air-fuel ratio divided by the stoichiometric air-fuel ratio.

The above expression for the burning velocity allows for variation of the fuel type, equivalence ratio, temperature, pressure, and changes in turbulence levels of the reactants.

## Venting

The venting calculation starts when the roof-to-shell joint reaches yielding (failure initiation). A venting mass flow rate is calculated and the total mass that has left the tank during the time step is used to determine the pressure drop due to venting for that given time step. The mass flow rate of the combustion products is calculated assuming constant pressure and temperature inside the tank for the time increment. The pressure and temperature are then recalculated based on the amount of mass that is left inside the tank.

For venting during frangible joint failure, a simplified analysis assuming adiabatic incompressible open-channel flow through a sluice gate is made. For a known roof-to-shell joint opening area the volume flow rate through the opening is given by the following equation:

$$Q = C_d A \sqrt{\frac{2P}{\rho}} \quad (33)$$

where

- $C_d$  = coefficient of contraction, between 0.6 to 0.9
- P = pressure difference

$\rho$  = density of the combustion products to be vented  
 $A$  = opening area of the roof-to-shell joint failure

With the time increment small, the change in pressure due to venting over the time increment is sufficiently low for the temperature to be assumed to remain constant for venting during each time increment. After the mass flow rate is calculated for each time step, a new pressure is found using the ideal gas law.

## FRANGIBLE JOINT FAILURE

Frangible joint failure is a very complicated dynamic process. The fracture of the joint vents the combustion products out the opening, reducing the pressure distribution on the tank roof and shell with time. The geometry of the tank during joint failure cannot be described by an axisymmetric shell model or simple formulas. To estimate the peak pressure in the tank, a simplified frangible joint failure model is used.

After failure initiation, the roof of the tank is assumed to rotate as a flat plate about a point on the roof-to-shell joint with rotation angle  $\phi$ . The vent opening is assumed only over part of the circumference defined by the opening angle  $\theta$ , as shown in Figure 21. The moments acted on the roof are:

$$M_w + M_i + M_r + M_f + M_m - M_p = 0 \quad (34)$$

where:

$M_p = \pi R^3 P \cos\phi$  = moment due to pressure the roof,

$M_w = \rho g t_h \pi R^3 \cos\phi$  = moment due to weight of the roof,

$M_r = \frac{7}{8} \pi \rho_{\text{air}} \dot{\phi}^2 c_D R^5$  = moment due to air resistance,

$M_i = \frac{5}{4} \pi \rho t_h R^4 \ddot{\phi}$  = moment due to inertia force,

$M_f = 2 F R^2 \cos\phi (\pi - \theta - \sin\theta)$  = moment due to edge force,

$M_m = 2 m_j R \sin\theta$  = moment due to edge moment,

$P$  = pressure difference between two sides of the roof,

$t_h$  = thickness of the roof,

$g$  = acceleration of gravity,

$\rho$  = density of steel,

$\rho_{\text{air}}$  = density of air,

$C_d$  = drag coefficient (1.143 for a flat disk),

$F$  = downwards force acted on the roof, obtained at roof-to-shell joint yield pressure,  
 $m_j$  = edge moment, obtained at roof-to-shell joint yield pressure.

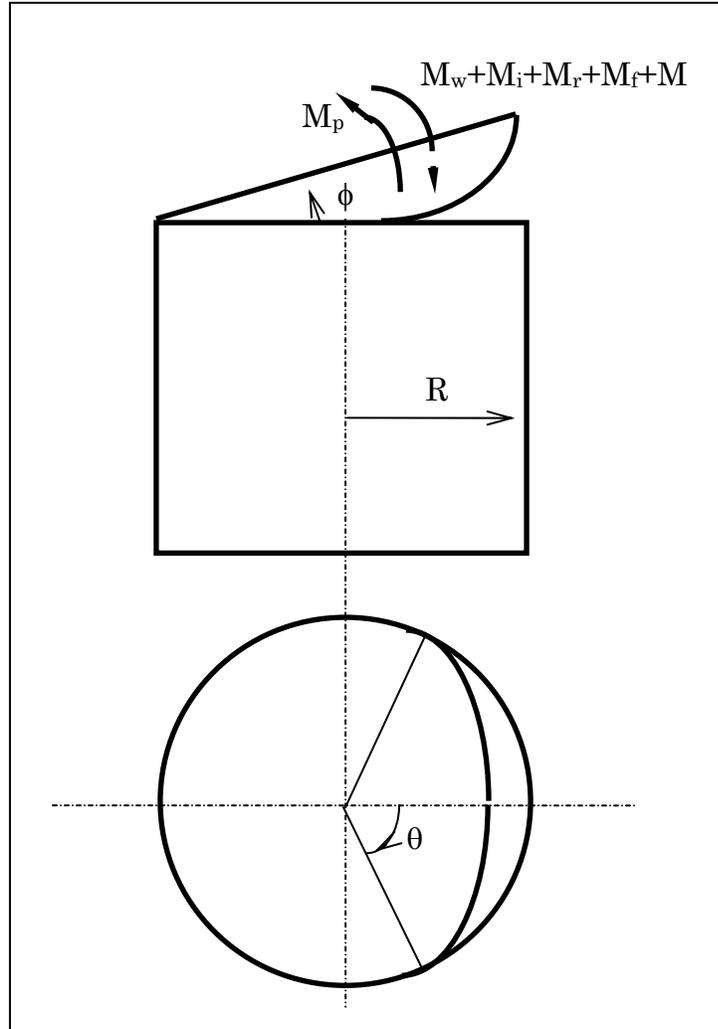


Figure 21: Forces acting of roof during venting

When the roof has rotated an angle  $\phi$  while the opening has an angle  $\theta_{max}$  at a given instant, the maximum opening height is:

$$y_{max} = 2R \sin \phi \tag{35}$$

Assume the opening width as the function of  $\theta$  is:

$$y = \frac{1}{2} y_{max} \left[ 1 + \cos \left( \frac{\theta \pi}{\theta_{max}} \right) \right] \tag{36}$$

The total opening area can be calculated as:

$$A_{\text{open}} = R y_{\text{max}} \theta_{\text{max}} \quad (37)$$

It is reasonable to say that when  $\phi = 0$ ,  $\theta_{\text{max}} = 0$ , and if the roof rotates  $\phi = \pi/2$  then the roof will fly off and  $\theta_{\text{max}} = \pi$ . Also assuming that when  $\phi$  equals or is larger than  $\pi/4$ , the opening area will reach its maximum possible value, (the cross section area of the tank), we have:

$$\theta_{\text{max}} = \pi \sin \phi \quad (38)$$

The opening area can then be written as a function of  $\phi$  only and calculated in each step of combustion analysis.

$$A_{\text{open}} = 2\pi R^2 \sin^2 \phi \quad (39)$$

# THEORY AND IMPLEMENTATION FMA-3D ANALYSIS

This describes the dynamic, large displacement, elastic-plastic analysis capability in SafeRoof. This was accomplished using the FMA-3D code (Key, 2004). The following description of FMA-3D is taken from the Theoretical Manual.

“The program is designed to compute the time-dependent displacements, velocities, accelerations, and stresses within elastic or inelastic, three-dimensional bodies composed of arbitrary shapes and materials. The program is a candidate for use in applications either where large energies and forces are present, where stress wave propagation is occurring, or where large displacements and strains dominate. Applications characterized by small forces and infinitesimal strains are not precluded, however. Applications for which the programs of this class have seen use fall into the areas of spent nuclear fuel shipping cask impact studies, explosive and high-rate metal forming, structures subjected to internal or external blast, and buried structure survivability studies.

An accident is a typical situation requiring the analysis of a structure's transient dynamic response. Virtually every industry is faced with the problem of accidents, either in terms of the plants and facilities it operates or the products it designs. Accident analyses focus on the worst case, the unexpected, and the disabling situation. It is not the normal operating loads which are considered, but rather the extreme loads to which a structure is to be subjected. The structure may no longer be useful for its original purpose, but it should not become part of a chain of events leading to damage beyond that from the original stimulus. In these cases, it is the large deformation, the sudden dynamic excursions, and the inelastic material behavior that need to be modeled.”

FMA-3D is distributed freely under the GNU license. To meet GNU license requirements and still incorporate FMA-3D with

SafeRoof, the approach taken was to modify SafeRoof so that the user can select either a static analysis or a dynamic analysis. If a static analysis is requested, then the internal SafeRoof solution is used. If a dynamic analysis is requested, an input file for FMA-3D is written and the FMA-3D calculation automatically started. At the end of the FMA-3D analysis, the results are read from the FMA-3D output files and plotted in SafeRoof. By taking this approach, it was not necessary to make changes to FMA-3D, while, to the user, the FMA-3D analysis appears seamlessly integrated.

FMA-3D is implemented only as a 3D program, with no 2D axisymmetric elements. The element used for the analysis was a 4-node plate element (P4EL), Figure 22. The 3D model represented a 5 degree slice of the tank, with appropriate symmetry boundary conditions so that the response was axisymmetric. All material properties were assigned from the SafeRoof input data. The internal load of product in the tank is represented by a linearly increasing pressure with depth. This does not include any mass effects due to the product, which would only serve to further slow uplift and bottom joint failure and thus is conservative.

The FMA-3D model uses contact elements under the tank floor to represent the deformation and uplift. At the roof, rafter support is also modeled using contact elements.

The FMA-3D calculation is actually performed in two stages. In the first calculation, all static loads including gravity and product loads are applied to the model. A dynamic relaxation run is then performed, where damping is included in the analysis and the dynamic analysis is continued until static equilibrium is reached. This gives a static solution in which the tank has settled on its foundation and the roof is being supported by rafters. The static equilibrium state is then used as the initial condition for the transient analysis in which the internal pressure loads are added.

The user has the option of specifying a linear ramp pressure loading on the tank or using the SafeRoof combustion capability to calculate the pressure load. The linear ramp can be used to specify such a slow ramp that the tank response is essentially static.

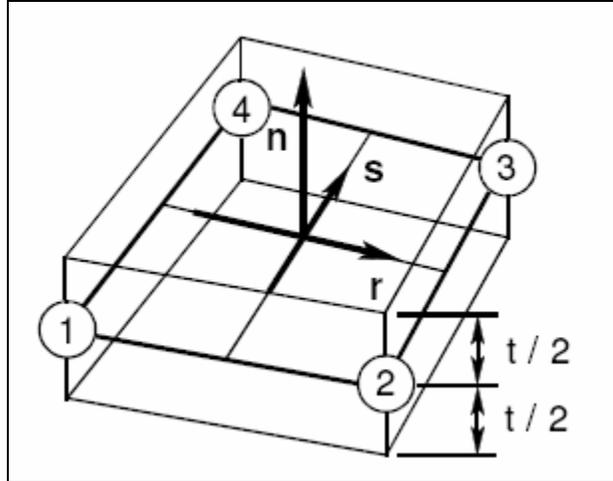


Figure 22: 4-node plate element (FMA, 2004)

During post-processing, displacements at the floor and elements at the top and bottom joints are monitored to identify the onset of yielding and uplift. The time steps at which yielding occurred are then available for plotting.

## VERIFICATION PROBLEM

In this section, an example/verification calculation is presented. The design used in the calculation is based on an actual tank design. The SafeRoof solution is compared to both analytic and ANSYS solutions.

The example tank conforms to the API 650 standard for frangible roof joint design. The design parameters are given in Table 3.

Diameter of the tank:	26.000	ft.
Height of the tank:	36.000	ft.
Roof slope:	0.750	inch in 12 inches
Thickness of roof plate:	0.1875	in.
Top angle width:	2.000	in.
Top angle thickness:	0.1875	in.
Top angle at outside of tank		
Angle not overlap top shell		
Number of courses:	4	
Course	Height (in.)	Thickness (in.)
1	108.000	0.2500
2	108.000	0.2500
3	108.000	0.2500
4	106.000	0.1875
Thickness of bottom plate:	0.2500	in.
Bottom plate beyond shell:	2.00	in.
No annular bottom plate		
Inner radius of ringwall:	12.500	ft.
Stiffness of ringwall:	1000.0	psi/in.
Stiffness of sand foundation:	250.0	psi/in.
Material of the tank plate:	ASTM A36	
Minimum yielding strength:	36.00	ksi

Table 3: Parameters for verification problem

The SafeRoof finite element model has 240 nodes, 239 shell elements and 61 interface elements. The same mesh was used for the ANSYS 5.0 finite element model. In ANSYS, the shell was modeled using the Shell 51 element and the tank foundation was modeled using the Interface 12 element. Calculations were performed at 0.1 psi and 1.0 psi pressure, respectively. The ANSYS solution used the large deformation option. The linear closed form solution was also calculated for

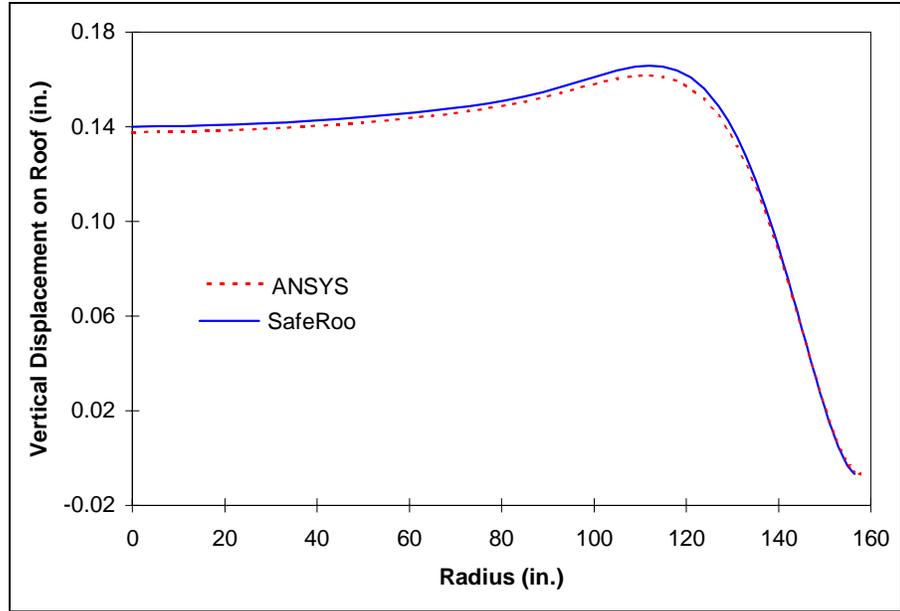
the stresses near the top joint, with an annular ring representing the angle. Since the linear closed form solution does not include the gravity loading, its internal pressure was reduced by 0.052 psi, which is the value required to overcome the weight of the roof. Because of the nonlinear support at the bottom, no analytic solution is available. The calculated displacements and stresses are plotted in Figures 22 through 25.

The verification calculations indicate that SafeRoof, ANSYS and the linear closed form solution all give similar results at the roof-to-shell joint when the internal pressure is low (Figure 23 and Figure 24). In this case, the deflections are small, so that the nonlinear results are close to that of a linear analysis.

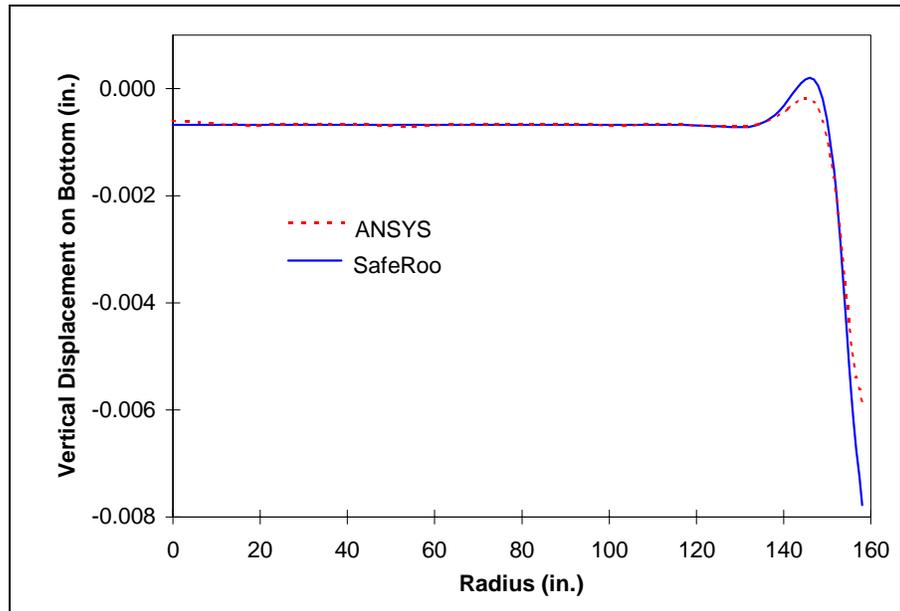
Under higher pressure, the nonlinear finite element solutions give smaller stresses than that of linear analysis, as shown in Figure 25 and Figure 26. SafeRoof gives somewhat larger displacements than ANSYS, hence larger stresses result. Near the roof-to-shell joint, the differences are not significant. At the shell-to-bottom joint, SafeRoof gives larger uplift of the bottom plate and larger stresses than given by ANSYS.

SafeRoof and ANSYS use somewhat different large deflection formulations. The ANSYS finite element model is stiffer than that of SafeRoof. The differences are smaller at the roof-to-shell joint than at the shell-to-bottom joint. Both programs give similar yield pressure at the roof-to-shell joint. The larger bottom deformation predicted by SafeRoof will give a more conservative (but still reasonable) bottom failure prediction than ANSYS. The relative strengths of the roof-to-shell joint and shell-to-bottom joint calculated by SafeRoof will also be a little higher than that given by ANSYS.

Additional verification for version 2.0 of SafeRoof has resulted from comparison with the new FMA3D implementation, using a slow ramp to approximate a static condition. The FMA3D results compared well with the linear SafeRoof calculations.

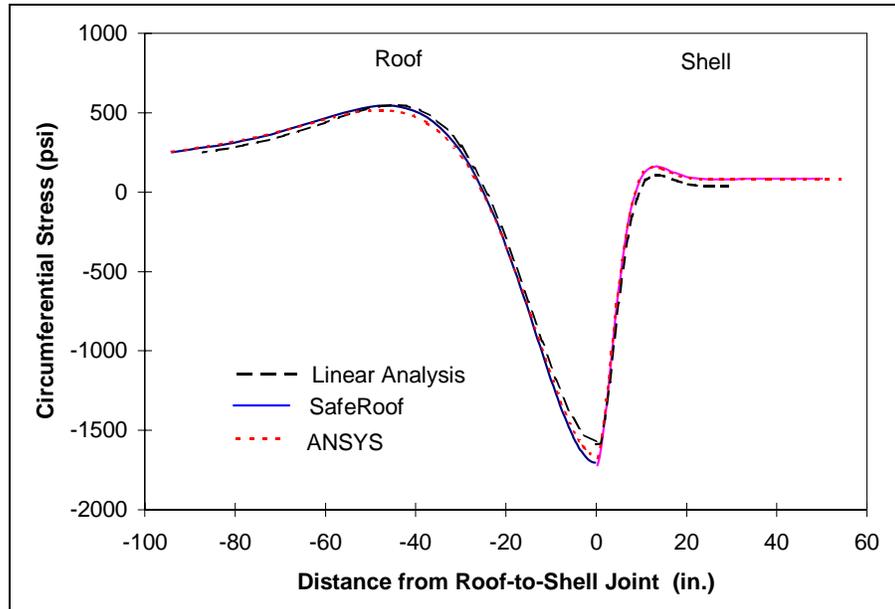


a. Roof deformation

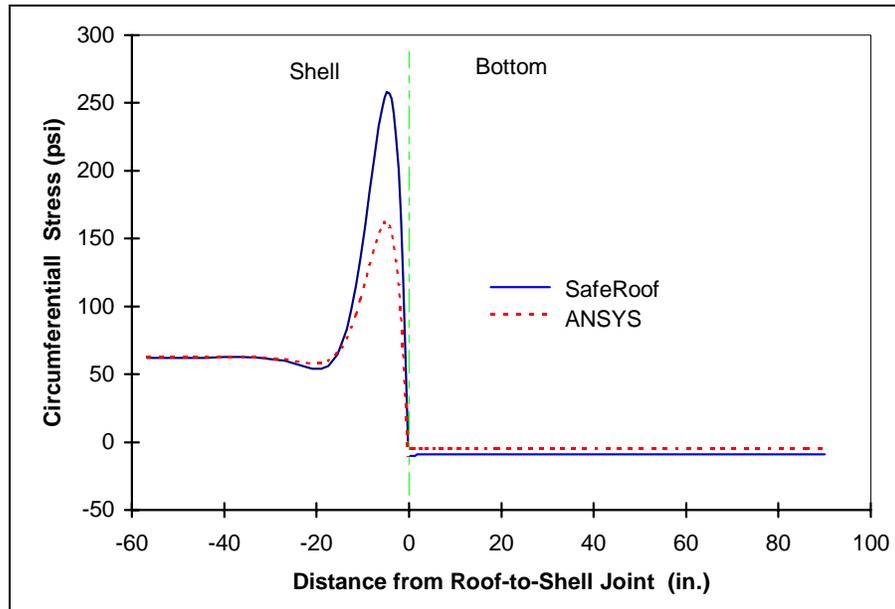


b. Bottom deformation

Figure 23: Deformation of tank at 0.1 psi pressure

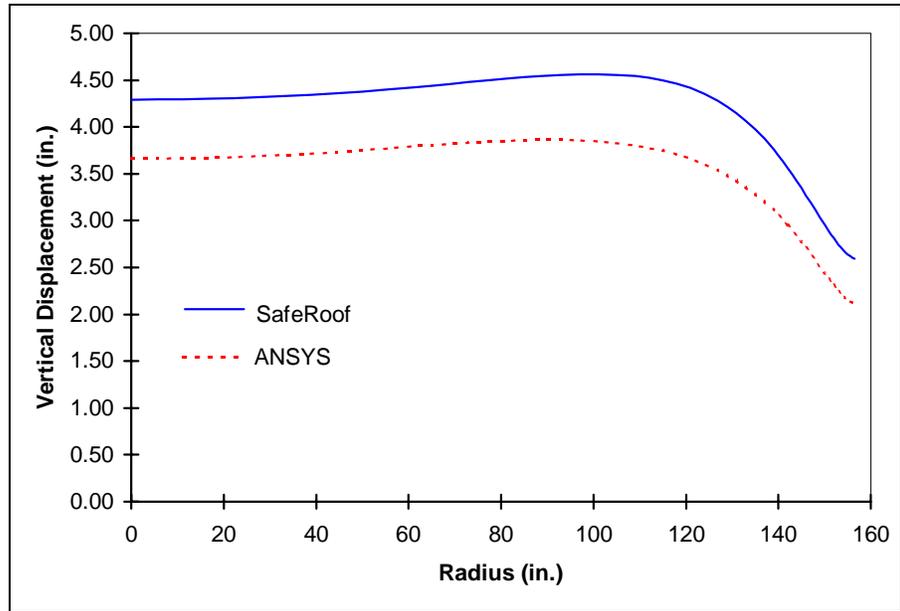


a. Roof-to-shell joint

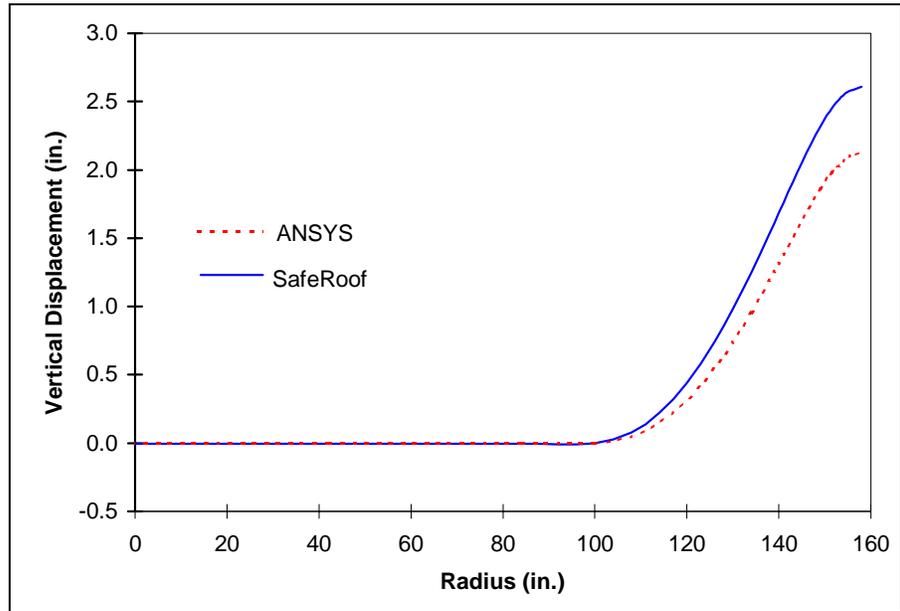


b. Shell-to-bottom joint

Figure 24: Circumferential stresses at 0.1 psi

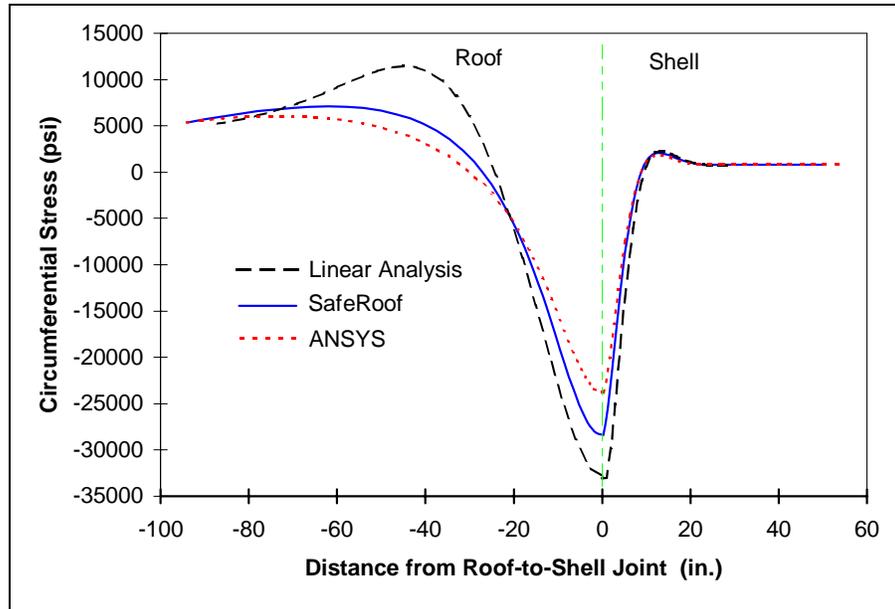


a. Roof deformation

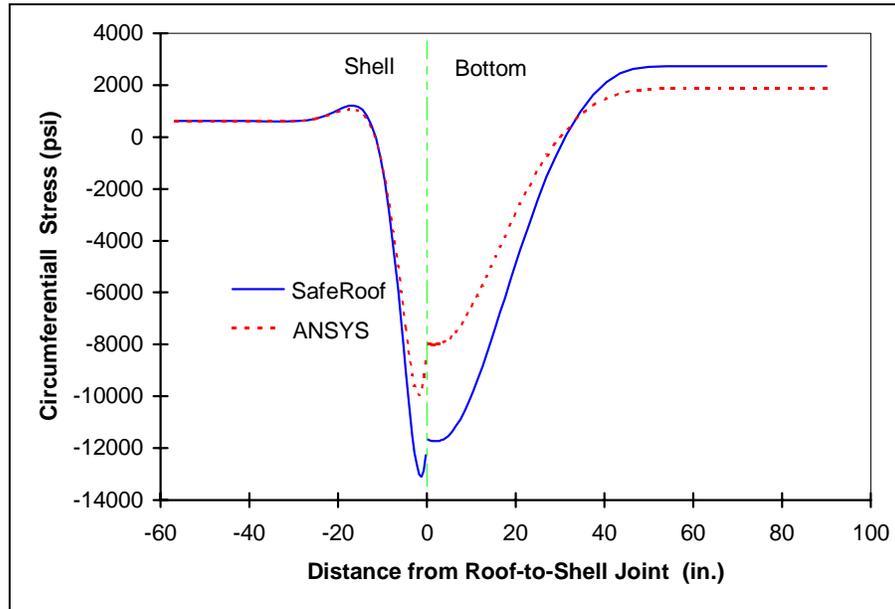


b. Bottom deformation

Figure 25: Deformation of tank at 1.0 psi



a. Roof-to-shell joint



b. Shell-to-bottom joint

Figure 26: Circumferential stresses in tank at 1.0 psi

## REFERENCES

- American Petroleum Institute, 1993, API Standard 650: Welded Steel Tanks for Oil Storage," 9th ed., 1220 L. Street, Northwest, Washington, DC.
- FMA, "FMA-3D User's Guide," Code and Documentation by Samuel Key, FMA Development, LLC, 1851 Tramway Terrace Loop NE, Albuquerque, New Mexico, 87122, (505) 856-1588, Version 20.00, August, 2003.
- Swenson, D., Fenton, D., Lu, Z., Ghori, A., and Baalman, J., 1996, "Evaluation of Design Criteria for Storage Tanks with Frangible Roof Joints," WRC Bulletin 410, Welding Research Council, 345 East 47<sup>th</sup> Street, NY, NY, 10017.
- Zienkiewicz, O. C., and Taylor, R. L., 1991, *The Finite Element Method*, 4th ed., McGraw-Hill Book Company, New York, NY.

## ACKNOWLEDGMENTS

### **SafeRoof Version 1.0**

The original development was supported by the Pressure Vessel Research Council (PVRC), Subcommittee on Dynamic Analysis and Testing, and the American Petroleum Institute (API). We especially thank Jerry Bitner, Martin Prager, Guido Karcher, and Richard Basile for useful discussions. We sincerely appreciate the support of Texaco, El Dorado Plant, in allowing us to examine actual tanks and in constructing the scale model tanks. Texaco contributed about 1/2 of the costs of the tanks to the project, with actual construction performed by Ace Construction. Mark Devries, an undergraduate honors student, and Asif Ghori, graduate student, helped in the testing of the tanks. The Agriculture Department at Kansas State provided the test site. The Kinesiology Departments at Kansas State and the University of Kansas provided and ran the high-speed cameras.

### **SafeRoof Version 2.0**

The development of Version 2.0 was supported by the American Petroleum Institute, with development done at Thunderhead Engineering Consultants, Inc.

We also thank Sam Key for providing the FMA-3D code, which he has developed based on his lifetime experience in large deformation analysis and distributes through the free GNU license. He graciously responded to our questions on the use of FMA-3D.

# INDEX

- About, 19
- Access Data, 15
- Analysis Menu, 11
  - API 650 calculation, 13
  - begin calculation, 13
  - combustion parameters, 13
  - liquid level, 13
  - solution controls, 11
- API 650, 1
- API 650 Calculation, 13
- Auto Design, 5
- Begin Calculation, 13
- Combustion
  - burning velocity, 43
  - solution procedure, 36
  - venting, 43
- Combustion Parameters, 13
- Design Menu, 5
  - auto design, 5
  - foundation, 8
  - user design, 7
- Displace Scale, 18
- Example Problems
  - dynamic, 22
  - static, 20
- Exit, 5
- File Menu, 4
  - exit, 5
  - new, 4
  - open, 4
  - save, 4
  - save as, 4
- Finite Element
  - loads, 27
  - mesh generation, 31
  - nodal displacements, 25
  - shape functions, 25
  - shell elements, 23
  - solution procedure, 35
  - stiffness matrix, 27
  - strain, 24
- Foundation, 8
- Foundation Elements, 28
- Frangible Joint Failure, 44
- Help Menu, 18
  - about, 19
  - index, 18
  - using help, 18
- Index, 18
- Liquid Level, 13
- List, 18
- List Font, 18
- Mesh Generation, 31
- New, 4
- Open, 4
- Plot, 16
- Postprocess Menu, 15
  - access data, 15
  - displace scale, 18
  - list, 18
  - list font, 18
  - plot, 16
  - reverse contour, 18
  - zoom in, 18
  - zoom reset, 18
- Reverse Contour, 18
- Save, 4
- Save as, 4
- Shell Elements, 23
- Solution Controls, 11
- Solution Procedure, 35
- Structure Analysis, 23
- User Design, 7
- Using Help, 18
- Zoom In, 18
- Zoom Reset, 18