INTRODUCTION

One engineering activity necessary for the continuous operation of manufacturing plants is dealing with in-plant or field problems. These problems occur in spite of designer's efforts, after equipment is put into service. Commonly called "troubleshooting", this activity has a need for quick accurate analysis. There is seldom time for "prototypes" or extended analysis as results are often required within days, if not hours. This paper deals with three troubleshooting cases where the use of Finite Elements played a major role.

The Du Pont Engineering Department's workhorse structures program, MSC NASTRAN, was used in all three of the problems. Although these case histories may not be unique as far as analysis is concerned, it is hoped that the applications and modeling strategy will prove interesting and useful. All three examples involve diagnosing and correcting problems with existing equipment. One involves pressure pulsation in a fluid system; one deals with dynamic loading from an explosion; and one covers buckling of a tank support structure.

PRESSURE PULSATION IN A 175-LB STEAM LINE

A. Problem Description

A large 175-lb steam distribution system provided steam for process heat in several manufacturing areas and power for three reciprocating steam engines. After one plant area was shut down and mothballed, the 175-lb steam distribution system was blanked off at the supply points for that area. Violent vibration was then noticed in the pipe bridge carrying the steam lines and other process piping. It occurred at random times and with varying intensities. An analysis was undertaken to explain the phenomenon and recommend ways of reducing the vibration.

B. Analysis

1. Measurements and Diagnosis

Vibration measurements made at various points on the piping bridge showed higher amplitudes on the 175-lb steam lines than neighboring pipes. The measurements also showed large frequency components at harmonics of the steam engine running speeds. This, combined with the occurrence of the problem after blanking off of the 175-lb steam system, suggested that the problem was originating in that system.

Measurement of piping natural frequencies gave no indication that mechanical resonance was responsible; however, the correlation between vibration frequency and harmonics of steam engine speeds suggested the possibility of an acoustical resonance. Measurements of dynamic pressure in the steam header showed large pulsation occurring simultaneously with the vibration, and at the same frequencies. Pressure pulsation was monitored while steam engines changed speed and sharp variations in amplitude were observed, apparently signifying natural frequencies. It was concluded that an acoustical resonance had been created by blanking off the mothballed area supply points and was being excited by relatively small pressure pulsations from the steam engines at certain operating speeds. The random nature of the vibration was postulated to be due to the engines running at various speeds, depending on process conditions.

Although reopening steam supply lines to the mothballed area would probably detune the resonance and reduce the vibration, it would also result in unacceptable energy losses. The engine speeds were determined by process requirements; hence substantial speed changes were not practical. A system modification was needed to eliminate the problem for the full range of machine speeds without removing the blind flanges.

2. Modeling with NASTRAN

The first task was to produce a computer model of the steam system that matched the measured data. Pressure pulsation in a fas filled pipe is governed by the same wave equation as a longitudinal vibration of a rod:

$$\frac{\partial^2 S}{\partial X^2} = \frac{1}{C^2} \frac{\partial^2 S}{\partial t^2}$$

where: S = displacement along the pipe

t = time

C = speed of sound in the medium in question

Solving the problem of longitudinal vibration of an elastic solid rod is straightforward using NASTRAN. Analysis of pressure pulsation in a pipe may be accomplished by using an "equivalent" rod whose speed of sound is the same as that of the gas.

The speed of sound in an elastic solid is

$$C = (E/\rho)^{1/2}$$

where: E = Young's modulus

ρ = mass density

For an ideal gas

$$C = (KgRT)^{1/2}$$

Since Pv = RT for an ideal gas and $1/v = g\rho$:

$$C = (KP/\rho)^{1/2}$$

where: K = the specific heat ratio for the gas

P = pressure in the gas

v = specific volume

R = ideal gas constant

ρ = Density = $\frac{1}{vg}$

g = Acceleration of gravity

Equating the speed of sound in the "equivalent" rod to that in the gas gives

$$C = (E'/\rho)^{1/2} = (KP/\rho)^{1/2}$$

$$\text{or } E' = KP$$

where: E' = Young's modulus for the material in the "equivalent" rod

Thus, we can analyze fluid oscillation in a one dimensional system with rod or bar elements and using PK for E and the gas density for the material density. Axial stress in the rod represents pressure in the fluid and motion in the rod represents motion in the fluid. This provides us with an analysis tool subject to the same basic assumptions as found in the classic organ pipe solution.

To test the method against theory, and to determine the degree of approximation introduced by the finite elements, several organ pipes were analyzed. Figure 1 shows percentage error in a model introduced by the elements when compared to closed form solutions of natural frequencies in an open-ended organ pipe. The percent error is plotted versus the ratio of rod element length to wave length of the frequencies of interest. The error is seen to be less than 2% when the element length is shorter than 10% of the wave length.

Since the required boundary conditions of equality of velocity and force are automatically satisfied at finite element junctions, we can use simple rods to model very complex systems of pipes. The models are essentially one dimensional in that they consist of a series of straight lines held together by MPC's. The lines can be conveniently chosen parallel to a reference coordinate axis, regardless of the pipe's actual orientation. An example is shown in Figures 2 and 3. Coupling the fluid system to a system of beams representing the piping is straightforward.

Figure 1

ERROR IN ORGAN PIPE FREQUENCIES

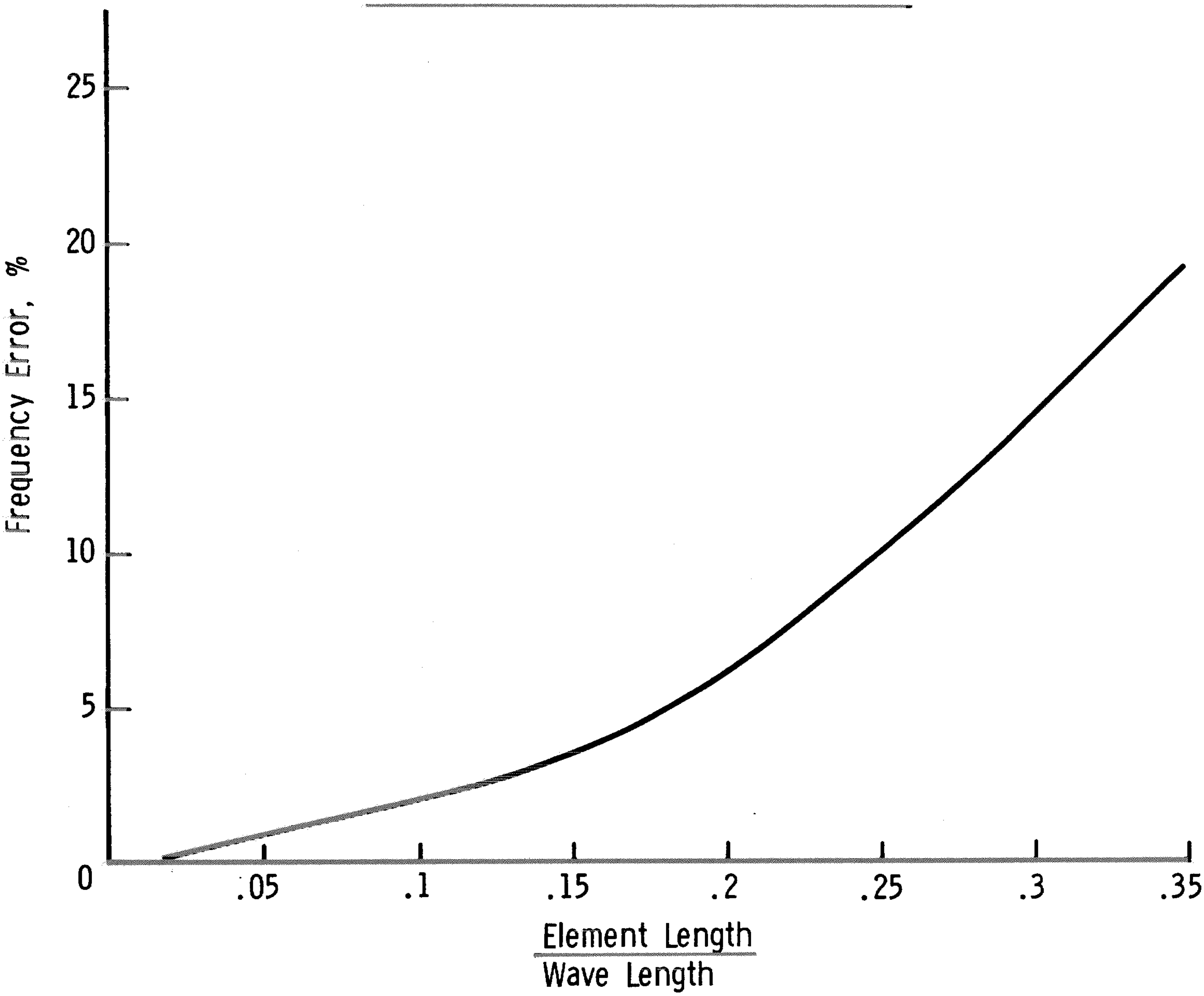


Figure 2

SCHEMATIC OF EXAMPLE PIPING SYSTEM

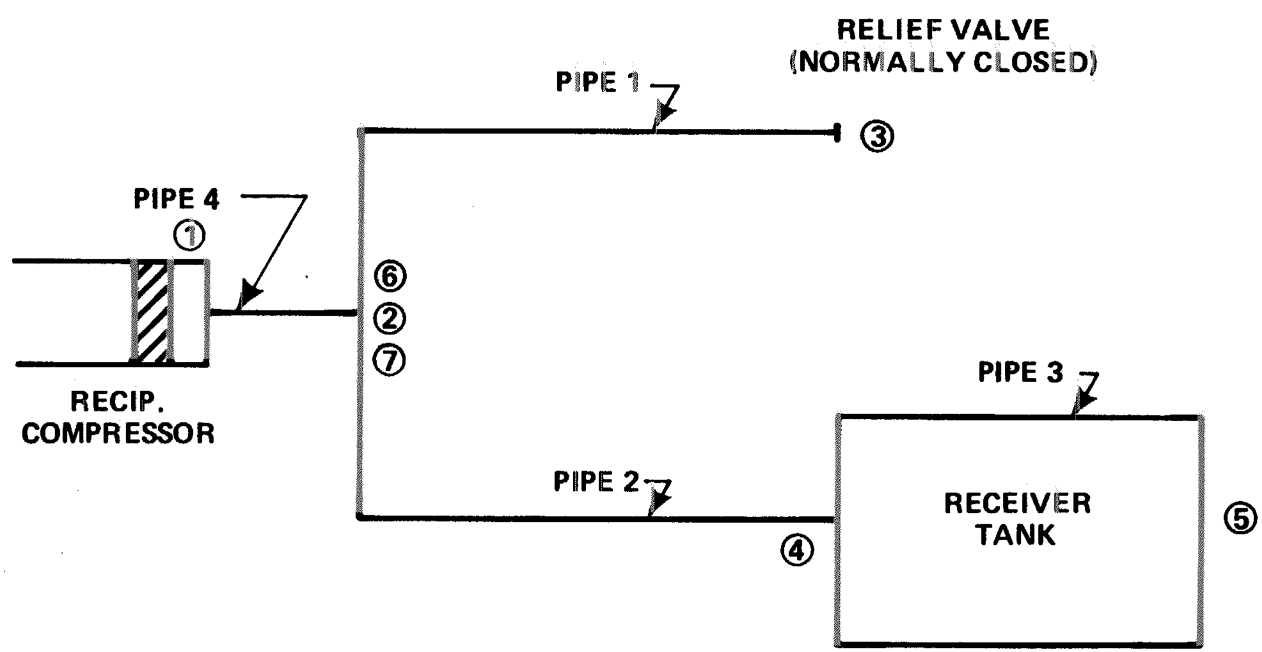
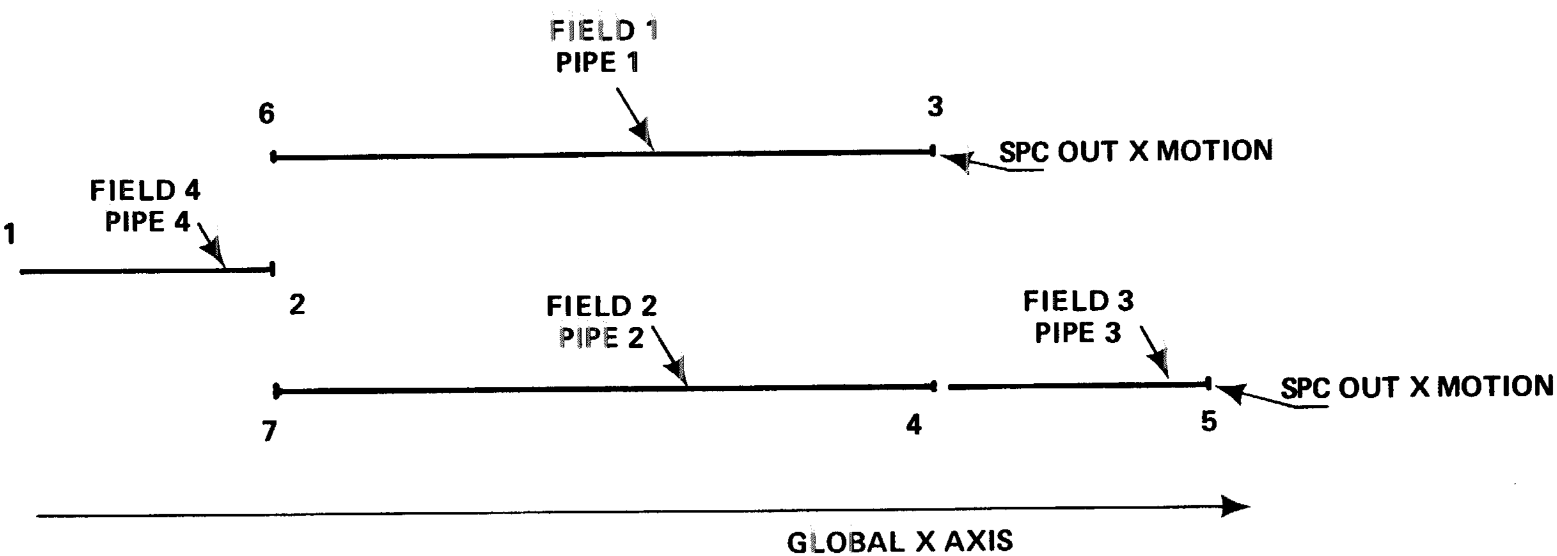


Figure 3

NASTRAN MODEL OF EXAMPLE PIPING SYSTEM



POINTS 2 & 6 & 7 CONNECTED BY MPC
SPC OUT ALL DOF EXCEPT 'X' DIRECTION

A model for the steam distribution system was produced with rod elements whose lengths were chosen using Figure 1 and whose material properties were derived from steam tables. Junctions in the system involving several pipes were formed using MPC equations. No effort was made to directly couple the fluid and steel systems in this case. Natural frequencies and mode shapes were calculated for the system and compared with measurements. Frequency response calculations were made using constant amplitude excitation at the steam engines. This provided some insight as to which modes would be most readily excited by the engines and should be expected in the measurement. Pressures at given points were shown by plotting stresses in the equivalent model. Peaks in the calculated pressure response compared well with measured data.

Unbalance forces acting on the piping segments with blind flanges due to fluctuating pressures were produced using scaler degrees of freedom. These scaler points were set equal to the difference between displacements at opposite ends of a pipe element and multiplied by appropriate constants.

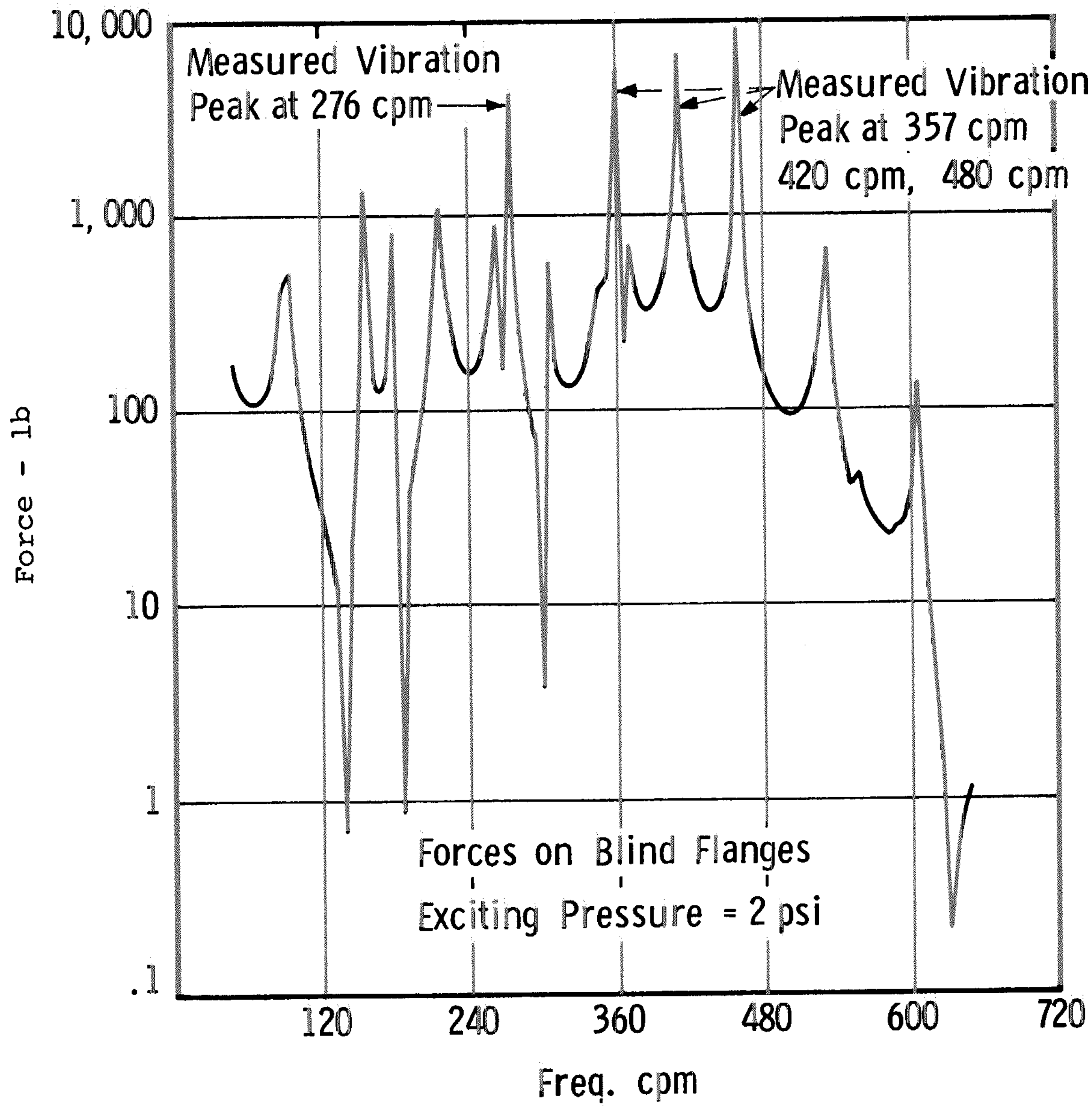
Figure 4 is a frequency response plot showing unbalanced force acting on a blind flange at a point of high vibration. Relative amplitudes were not well matched since the excitation for the calculation was arbitrarily constant in amplitude, while the actual excitation, especially at the higher frequencies, was due to harmonics of engine speed with unknown and probably varying amplitudes. Also, losses due to damping were ignored.

C. Conclusions and Recommendations

One approach toward solving this type of problem is to install orifices at locations of high fluid motion to reduce pulsation response. A study of the mode shapes of the stronger modes excited in the response calculation and present in the measurements showed a single location where a small bore

Figure 4

STEAMLINE FREQUENCY - RESPONSE PLOT



orifice could be installed that would affect all of the modes without severely reducing the distribution system efficiency. The orifice was modeled as a very short element having a small diameter. No damping was added to the system. Frequency response calculations with the orifice in the model showed a 20-fold decrease in forces on the blind flanges in the system and a shift in the natural frequencies. The orifice was physically added to the system and the vibration problem disappeared.

ANATOMY OF A TANK FAILURE

A. Problem Description

A holding tank used as a collecting point for waste liquid (mostly water and organics) had five pipes attached to it bringing incoming liquid from sump pumps in several manufacturing areas. In addition, there were two outlet lines taking away material to a waste processing facility. The tank had a slightly conical top, and sat on a concrete pad anchored only by its weight.

One afternoon, without warning, the top was blown off the tank and flew 157 ft., severely damaging overhead heating and ventilating ducting on its way. Piping sections from the top were found on a roof 300 ft. from the tank. Figure 5 shows the tank and ducting after the incident. Figure 6 shows scattered wreckage.

Witnesses to the incident reported no flames or smoke and no evidence of heat was found in the wreckage. Subsequent investigation showed the vent line for the tank to be blocked, but whether the blockage was caused by the incident or was there prior to it was unknown. The number of sump pumps running at the time of the incident was also unknown; but a stream of liquid was observed flowing from a damaged inlet pipe just after the failure.

Any one of the sump pumps had pressure capability to burst the tank, and since the vent was possibly blocked, it appeared that the failure could have occurred due to static over-pressure of air trapped between the tank top and the rising liquid surface. The alternative possibility was an explosion of an unknown gas from an unknown ignition source in the tank. The design modification to prevent a recurrence, of course, depended on the cause. A finite element analysis was one of the channels of inquiry followed.

Figure 5

DAMAGED WASTE HOLDING TANK

Figure 6

DEBRIS FROM WASTE TANK FAILURE

B. NASTRAN Model

Two mathematical analyses were made of the tank: one a geometric nonlinear calculation to determine the stress distribution under static overpressure, the other a transient with a rapidly rising pressure approximating an explosion. The stresses corresponding to both cases were contour plotted separately and compared with the apparent stress distribution that caused the failure. Figures 7, 8, and 9 show stresses due to static and dynamic loading. The dynamic stress distribution showed maximum stresses occurring around the top edge of the tank, which matched the failure pattern. The maximum static stresses occurred around the bottom edge of the tank, far from the failure point. This provided evidence that the cause of the failure was a very rapid overpressure. Further analysis by explosion experts determined the possibility of hydrogen being present in the tank, and identified a potential ignition source. The system was redesigned to prevent this type of problem from recurring.

Figure 7

MAXIMUM SHEAR IN TOP AND BOTTOM SEAMS UNDER DYNAMIC LOAD

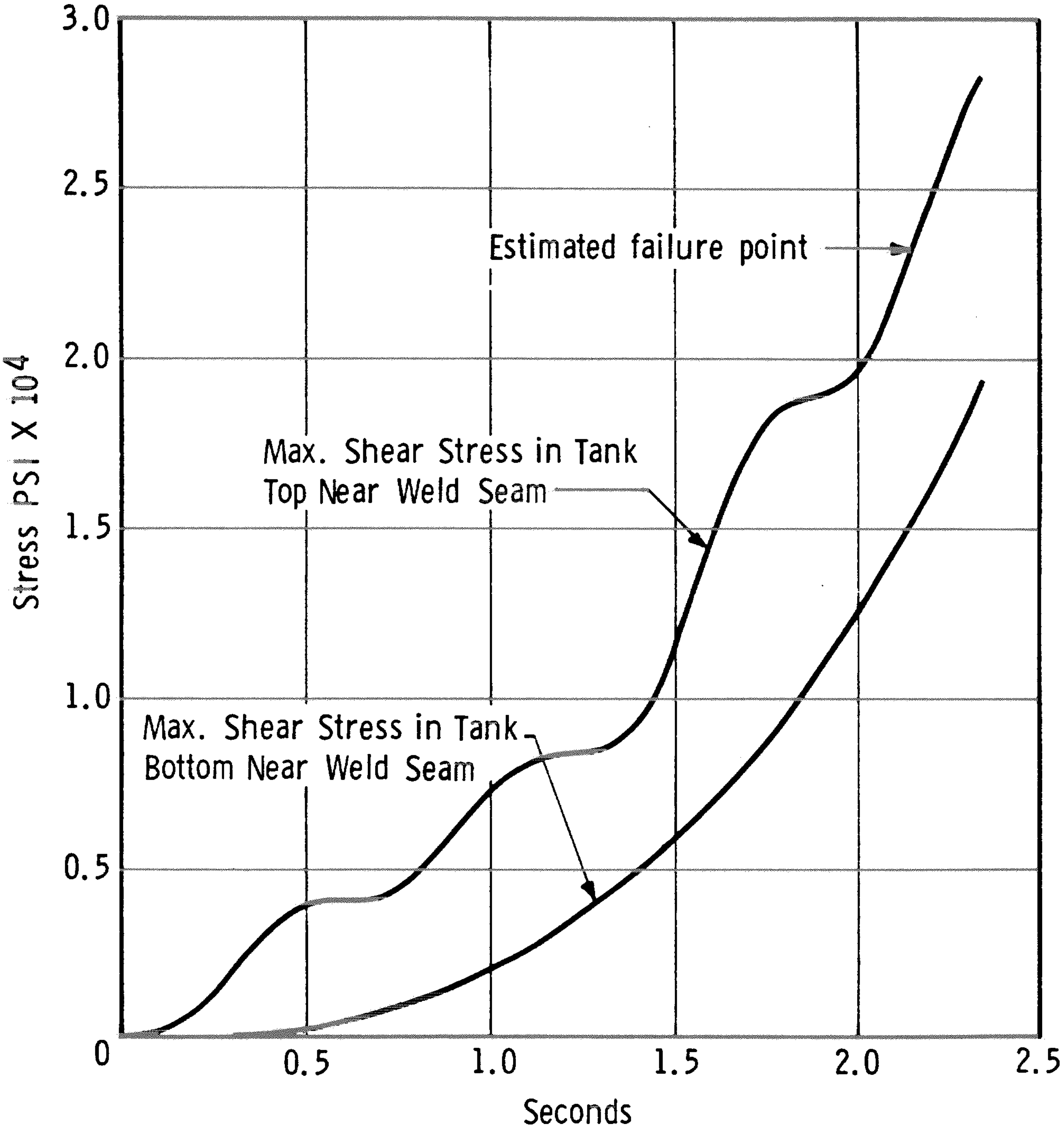
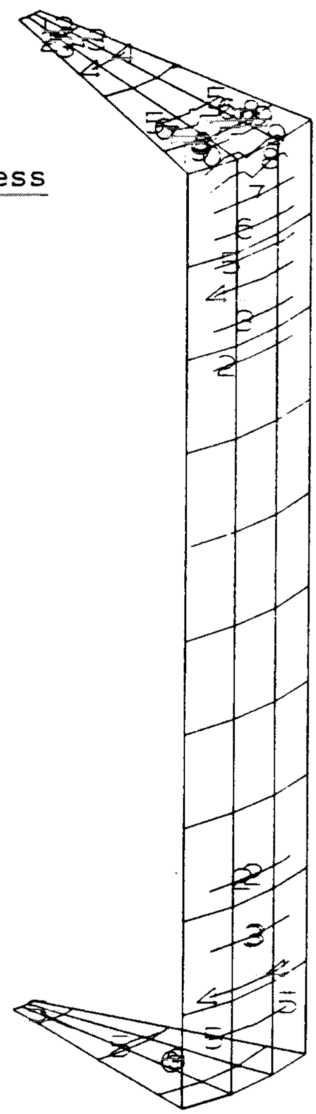


Figure 8

WASTE HOLDING TANK DYNAMIC LOAD CONDITION

Symbol	Max. Shear Stress
1	325.3
2	2351.3
3	4377.2
4	6403.1
5	8429.0
6	10455.0
7	12480.9
8	14506.8
9	16532.7
10	18558.7

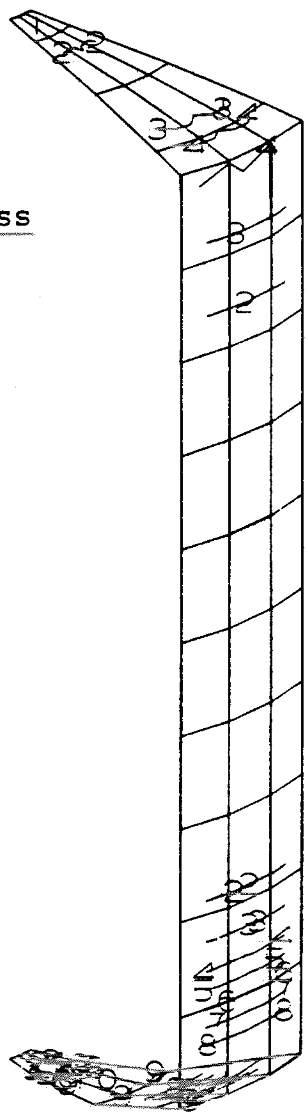


Onset of Yield - .018 Secs.
after ignition
Press. = 38.5 psig

Figure 9

STATIC LOAD CONDITION

Symbol	Max. Shear Stress
1	42.6
2	2442.0
3	4841.4
4	7240.8
5	9640.2
6	12039.6
7	14438.9
8	16838.3
9	19237.7
10	21637.1



Max. def. = 1.46''

Static Pressure At 20 psig

STABILITY OF ACID TANK SUPPORT STRUCTURE

A. Problem Description

A plant had purchased a 7500-gal, glass-lined tank for storage of sulfuric acid. The tank was mounted approximately 20 ft. above the ground on a steel structure designed by an outside consulting firm, as shown in Figure 10. During its first filling, when the level reached approximately 1700 gal., the supporting structure collapsed, dropping the tank onto the cross beams in the support structure, severely yielding them, and throwing its cast iron legs as far as 20 ft. away. Figures 11 and 12 show the collapsed structure. The glass service piping was sheared off, considerable damage to the glass tank lining occurred, and there was a significant acid spill.

B. Analysis

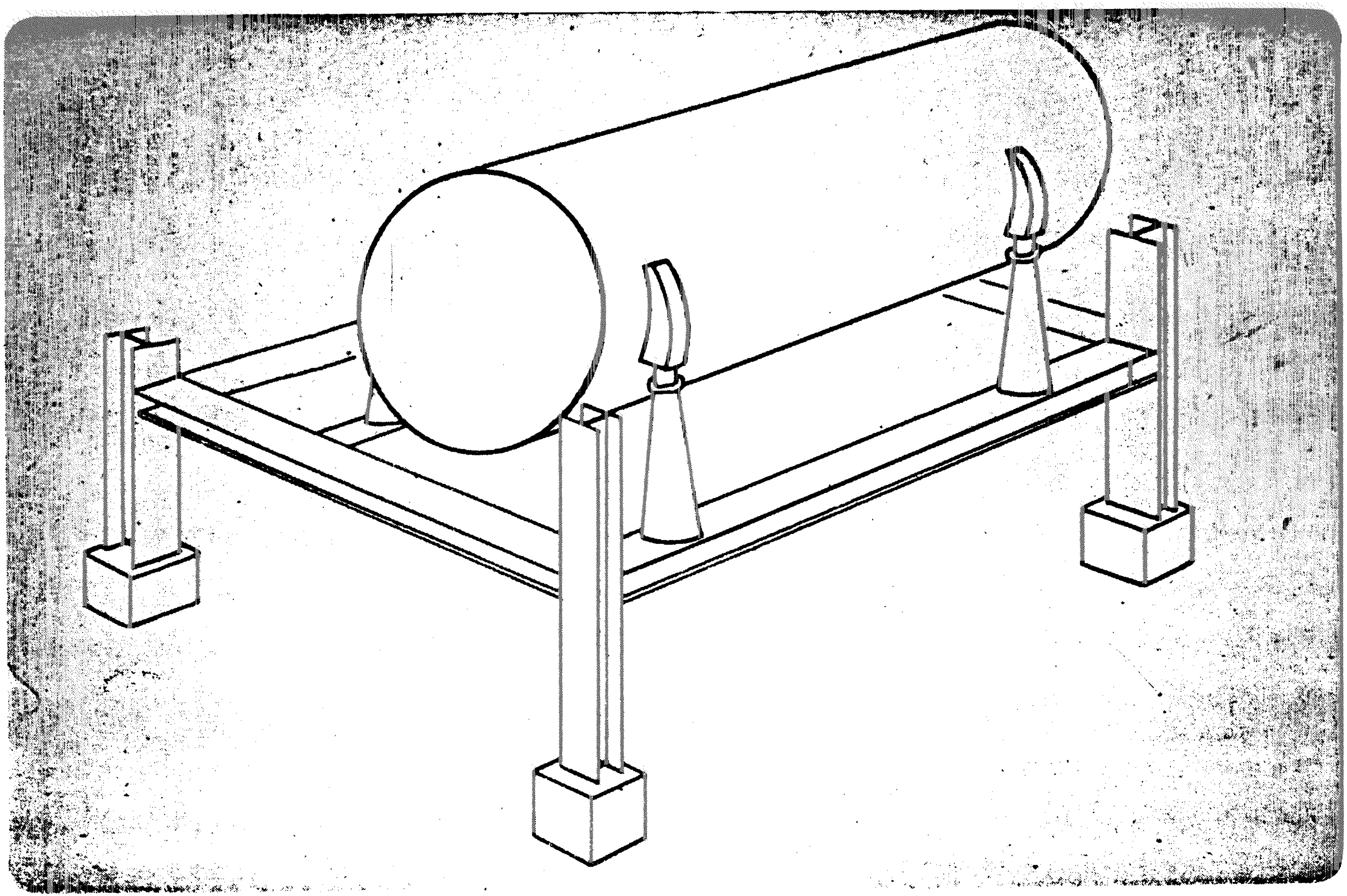
1. Diagnosis

The wreckage was examined very carefully looking for structural flaws that might have caused such a premature failure. The structure steel was tested and tolerances of the beams used in the support structure were checked. No irregularities were found. Due to the mode of the failure and lack of warning, we suspected a stability problem. A fairly crude hand analysis described on Figure 13 was applied to the main beams supporting the tank. It suggested the possibility of a torsional instability in the main su-port beams. It was decided to perform a more general elastic stability analysis of the structure using NASTRAN.

2. Modeling with NASTRAN

A buckling analysis was done using NASTRAN's Rigid Format 5. Since it is standard practice to install beams using "clip angles", section warping was considered possible. Supporting steel was modeled using beam elements having elastic

Figure 10



SCHEMATIC OF ACID TANK SUPPORT STRUCTURE

Figure 11

DAMAGED ACID STORAGE TANK

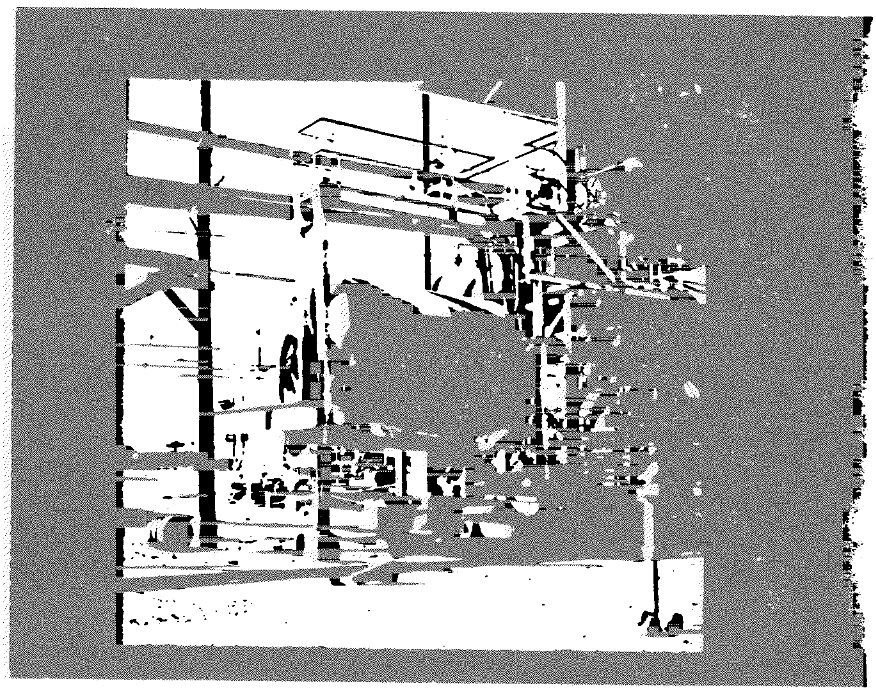


Figure 12

RESULTS FROM ACID SPILL

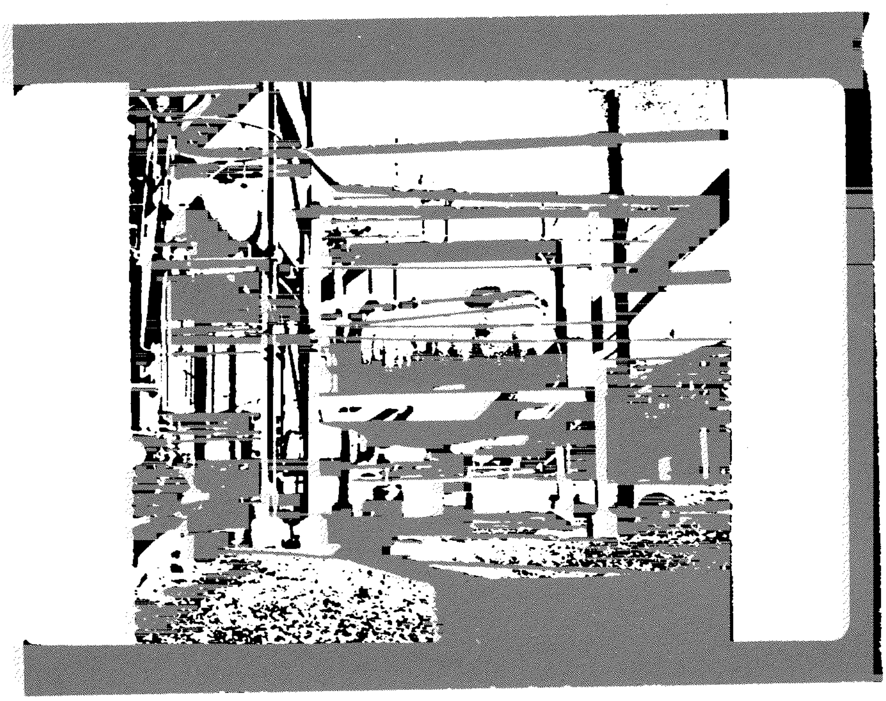
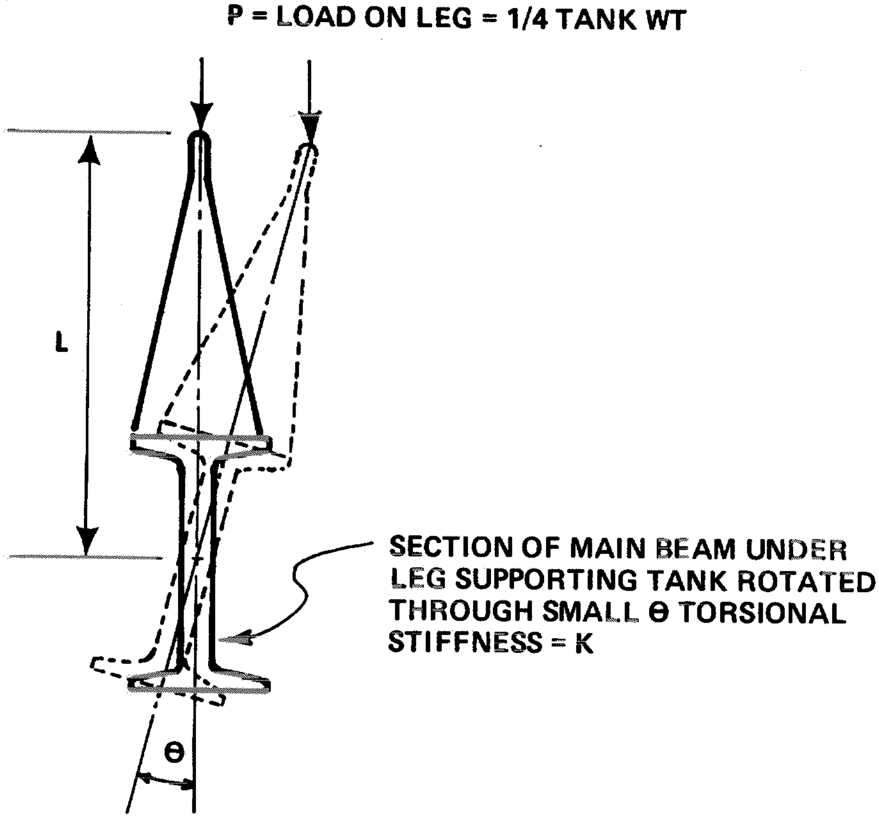


Figure 13



OVERTURNING MOMENT = LθP

RESTORING MOMENT = Kθ

FOR STABILITY K > LP

> warping capability. A point of elastic instability having the same failure mode as experienced was predicted within 15% of the actual failure load. Interestingly enough, if bar elements or if beam elements without elastic warping capability were used, the structure appeared to be safe. Using the buckling model, it was easy to recommend a design fix. "Clip angle" connections were replaced by framed-in welding connections and stiffer framed-in cross members were added to the structure.

Finite element programs, particularly when available with interactive graphic front ends and graphic output, are becoming more and more useful. They are finding application not only with design problems, but with field problems that require rapid engineering response, and analysis that can change direction as needed. The broad capability of MSC NASTRAN has made it an ideal tool to this end.