FINITE ELEMENT MODELING OF A
CONCRETE STRUCTURE UNDER BLAST LOADING

A Thesis
Presented in Partial Fulfillment of the Requirements
for the Degree Master of Science
by
Susan Michelle Cheney, B.S.
The Ohio State University
1985

Approved by

Copyright © 1985
by Susan Michelle Cheney
All rights reserved

Department of Civil Engineering
# Table of Contents

List of Figures .................................................  iv
List of Tables ...................................................  vi
Acknowledgements ...............................................  vii
Introduction .....................................................  1
Blast Test Description .........................................  5

## The Chen and Chen Model

- General .......................................................  13
- Yield and Failure Criteria .................................  14
- Loading Function ............................................  17
- Incremental Stress-Strain Relationships ...............  18
- Implementation ..............................................  21

## Model Description

- Model Geometry ...............................................  24
- Boundary Conditions ........................................  28
- Material Models .............................................  29
- Loading .......................................................  32

## Analysis of Results

- Abaqus Results ..............................................  38
- Model One .....................................................  39
- Model Two .....................................................  65
- Model Three ..................................................  69
- Comparison of Models One - Three .......................  83

## Conclusions

- General .......................................................  85
- Problems with Abaqus .....................................  86
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Chen and Chen Model</td>
<td>88</td>
</tr>
<tr>
<td>Problems with the Models</td>
<td>88</td>
</tr>
<tr>
<td>Appendix A</td>
<td>90</td>
</tr>
<tr>
<td>Appendix B</td>
<td>98</td>
</tr>
<tr>
<td>Appendix C</td>
<td>106</td>
</tr>
<tr>
<td>References</td>
<td>132</td>
</tr>
</tbody>
</table>
List of Figures

Fig. 1 Test Specimen ........................................ 6
Fig. 2 Typical Instrumentation Layout ....................... 8
Fig. 3 Gauge P21 Pressure Reading .......................... 10
Fig. 4 Yield and Failure Surface for Concrete ............. 16
Fig. 5 Model One ............................................. 25
Fig. 6 Model Three ........................................... 27
Fig. 7 Gauge P64 Pressure Reading .......................... 33
Fig. 8 Percent Pressure-Time History Curve ............... 34
Fig. 9 Blast Pressure Distribution ........................... 35
Fig. 10 Blast Test Results .................................... 40
Fig. 11 Blast Test Results .................................... 41
Fig. 12 Blast Test Results .................................... 42
Fig. 13 Blast Test Results .................................... 43
Fig. 14 Blast Test Results .................................... 44
Fig. 15 Blast Test Results .................................... 45
Fig. 16 Abaqus Results ....................................... 47
Fig. 17 Abaqus Results ....................................... 48
Fig. 18 Abaqus Results ....................................... 50
Fig. 19 Abaqus Results ....................................... 51
Fig. 20 Abaqus Results ....................................... 52
Fig. 21 Abaqus Results ....................................... 53
Fig. 22 Abaqus Results ....................................... 54
Fig. 23 Abaqus Results ....................................... 55
Fig. 24 Abaqus Results ....................................... 56
Fig. 25 Abaqus Results ....................................... 57
<table>
<thead>
<tr>
<th>Fig. 26</th>
<th>Abaqus Results</th>
<th>58</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig. 27</td>
<td>Abaqus Results</td>
<td>59</td>
</tr>
<tr>
<td>Fig. 28</td>
<td>Abaqus Results</td>
<td>60</td>
</tr>
<tr>
<td>Fig. 29</td>
<td>Abaqus Results</td>
<td>61</td>
</tr>
<tr>
<td>Fig. 30</td>
<td>Abaqus Results</td>
<td>62</td>
</tr>
<tr>
<td>Fig. 31</td>
<td>Abaqus Results</td>
<td>63</td>
</tr>
<tr>
<td>Fig. 32</td>
<td>Abaqus Results</td>
<td>64</td>
</tr>
<tr>
<td>Fig. 33</td>
<td>Abaqus Results</td>
<td>66</td>
</tr>
<tr>
<td>Fig. 34</td>
<td>Abaqus Results</td>
<td>67</td>
</tr>
<tr>
<td>Fig. 35</td>
<td>Abaqus Results</td>
<td>70</td>
</tr>
<tr>
<td>Fig. 36</td>
<td>Abaqus Results</td>
<td>71</td>
</tr>
<tr>
<td>Fig. 37</td>
<td>Abaqus Results</td>
<td>73</td>
</tr>
<tr>
<td>Fig. 38</td>
<td>Abaqus Results</td>
<td>74</td>
</tr>
<tr>
<td>Fig. 39</td>
<td>Abaqus Results</td>
<td>75</td>
</tr>
<tr>
<td>Fig. 40</td>
<td>Abaqus Results</td>
<td>76</td>
</tr>
<tr>
<td>Fig. 41</td>
<td>Abaqus Results</td>
<td>77</td>
</tr>
<tr>
<td>Fig. 42</td>
<td>Abaqus Results</td>
<td>78</td>
</tr>
<tr>
<td>Fig. 43</td>
<td>Abaqus Results</td>
<td>79</td>
</tr>
<tr>
<td>Fig. 44</td>
<td>Abaqus Results</td>
<td>80</td>
</tr>
<tr>
<td>Fig. 45</td>
<td>Abaqus Results</td>
<td>81</td>
</tr>
<tr>
<td>Fig. 46</td>
<td>Abaqus Results</td>
<td>82</td>
</tr>
</tbody>
</table>
List of Tables

Table 1  Test Parameters ............................................ 6
Table 2  Initial Velocities ......................................... 34
Table 3  Scratch Gauge Readings ................................. 39
Acknowledgements

I wish to thank my advisor, Dr. Wolfe, for his patience in dealing with me and my project, and for all of his help. Thanks also go to the other members of my committee, Dr. Sandhu, who has advised and encouraged me since my senior year, and Dr. Sierakowski, who has also been a great help. I would also like to thank the Owens-Corning Fiberglas Technical Center, and in particular Barry Burnham, for their generous help in the printing of this thesis.
CHAPTER ONE
-------
INTRODUCTION

The purpose of this investigation was to attempt to recreate the dynamic responses of concrete walls under blast loading. The dynamic responses to be recreated were taken from a series of blast tests provided to the author, briefly described in Ref [1] and in Chapter Two. Geometric, kinematic and material data were taken from the blast tests. Averaged pressure readings from the blast tests were used to determine an appropriate loading for the structure.

The finite element computer code Abaqus, which was installed on a CDC Cyber 370, was used to model the responses. Abaqus is described in detail in Ref [2] and briefly reviewed in Ref [3]. In particular, the concrete material model which Abaqus uses, the Chen and Chen model, was tested, and the overall suitability of the Abaqus program for this type of problem was determined.
There were several reasons for undertaking this investigation. Understanding the dynamic response of structures is important not only to the armed forces, who anticipate blast loading, but also to the civilian civil engineering community, who might anticipate more wind and earthquake loadings rather than blast loading. Also, the program Abaqus has not been tested on this type of problem.

The next chapter of this thesis describes the blast tests. It describes the physical setup of the tests, the dimensions of the structure, the explosive, and the means used to restrain the structure. Next, the test parameters and instrumentation were detailed. The results of the blast tests are examined, and the responses of the concrete structures are discussed. Lastly, the quality and types of data obtained are reviewed.

Chapter Three describes the Chen and Chen concrete material model used in Abaqus. This chapter first discusses the assumptions used in the program and in the model, and then goes on to describe yield and failure criterion. The terms of the failure criterion are expanded upon, and a loading function is defined. Then the incremental stress-strain relationships are derived and placed into matrix form. Lastly, the way in which Abaqus uses this model, how the model works in the program, and the various options available in the program are described.

The fourth chapter of this thesis describes what went
Precisely what was modeled is discussed, and each of the three models used is discussed with respect to the element type used, the number of elements and nodes and the material model used. The method used to generate the meshes is described, and the boundary conditions used are given. Next the material properties of the three models and the methods used to obtain them are detailed. Specifically, the Young's modulus, Poisson's ratio, and the ultimate and yield strengths used are given. The Abaqus default strength ratios are also detailed. Next, the process for determining the loading on the wall is explained, from the interpretation of the pressure gauge data to the calculation of initial velocities.

Chapter Five covers the analysis of the results which Abaqus produced. First the results of the blast tests are analyzed, and the effects of permanent deflections are removed. Next, the two versions of each of the three models are compared against each other and against the blast test results for maximum deflections and period of vibration. The number of steps and amount of time completed are discussed, as well as the amount of cpu time the model required. Next, the three models are compared against each other in terms of cpu time, maximum deflection, and period. Lastly, the effect of changing the boundary conditions is discussed.

In the sixth chapter, the conclusions are drawn. The
suitability of Abaqus for this type of problem is discussed, along with the value of the Chen and Chen concrete material model. Suggestions for possible improvements to the program are made.

There are three appendices. The first appendix discusses a method of calculating smeared elastic constants in concrete. The next appendix presents some of the results of the blast tests, and the last appendix presents some of the Abaqus results that were not included in Chapter Five.
CHAPTER TWO

BLAST TEST DESCRIPTION

A series of blast load tests was conducted on concrete walls. The tests are summarized in Table 1. A more complete description of the test results is given in Appendix B and in Ref [1]. The tests were designed to identify the effect on the dynamic response of the wall of the percentage of vertical steel in the concrete, and to determine the relative value of the use of stirrups over the use of dowels in preventing spalling.

The actual thickness of the wall was approximately one foot and the weight of the explosive charge was half that in a thousand pound bomb. The explosive was packed in steel pipe and placed on the ground surface 1.5 meters away from the center of one wall of the concrete boxes. The actual dimensions of the concrete boxes tested are shown in Fig. 1. In order to minimize rigid body motion, each box was bolted to a large L-shaped concrete reaction block.
Fig. 1 Test Specimen

<table>
<thead>
<tr>
<th>Test</th>
<th>Percent Steel</th>
<th>Shear Reinforcement</th>
</tr>
</thead>
<tbody>
<tr>
<td>SWT-1</td>
<td>1.00%</td>
<td>Stirrups</td>
</tr>
<tr>
<td>SWT-2</td>
<td>0.50%</td>
<td>Stirrups</td>
</tr>
<tr>
<td>SWT-3</td>
<td>0.50%</td>
<td>Dowels</td>
</tr>
<tr>
<td>SWT-4</td>
<td>0.25%</td>
<td>Dowels</td>
</tr>
<tr>
<td>SWT-5</td>
<td>2.00%</td>
<td>Stirrups</td>
</tr>
<tr>
<td>SWT-6</td>
<td>0.25%</td>
<td>Stirrups</td>
</tr>
</tbody>
</table>

Table 1 Test Parameters
prior to being tested. The reaction block had a sand berm behind it to keep it in place.

Six tests were performed, with the percentage of vertical steel varying from a high of 2% down to 0.25%. The shear reinforcement consisted of either stirrups or dowel rods. Percentages of vertical steel and shear reinforcement used in each test are shown in Table 1. The percentage of horizontal steel in the concrete was equal to 0.25% for all tests, except for the walls reinforced by dowels, in which the percentage of horizontal steel was 0.13%. The design strength of the concrete was 5000 psi, and 36 ksi steel was used as reinforcement.

Each box was instrumented in order to measure the pressure exerted on the wall by the blast, the strain in the reinforcing steel, the deflection at the midpoint of the wall, and the acceleration of various points on the structure. A typical instrumentation layout is shown in Fig. 2. Also, high-speed photography was used to record the velocities of fragments from the cased charge and the concrete spall fragments.

The test results showed that the walls with lesser percentages of steel generally experienced greater deflections. The differences in the magnitude of the deflections between the different percentages of steel were not very large however. None of the walls was breached, and all of the walls experienced considerable spalling on
Fig. 2  Typical Instrumentation Layout
the lower part of the wall. Spalling was induced by the blast induced stress wave and not by the structural response of the wall. The spalling normally extended to approximately halfway up the inside face of the wall. In structures with stirrups, the stirrups acted to contain the spalled concrete. Concrete not contained by the stirrups flew off the wall into the interior of the structure. In structures with dowel rods, more concrete flew off the wall. The results of the tests showed that the percentage of steel could be safely lowered without jeopardizing the strength of the wall, but also that stirrups are preferable to dowels in preventing spalling of the concrete of the interior face of the wall.

The instrumentation on the boxes yielded a large amount of data, which varied considerably in both quantity and quality from test to test. The pressure gauges closest to the center of the blast were frequently destroyed either by the blast or by fragments from the steel pipe encasing the explosive. Consequently little good data were obtained on the blast pressures at the bottom center of the wall. A typical pressure reading from gauge P21, is shown in Fig. 3, where the sudden rise in pressure at 0.75 msec followed by a constant steady high signal indicates the destruction of the gauge. The data were increasingly better the further away from the bottom center of the wall. However, at every gauge location, there was considerable
Fig. 3  Gauge P21 Pressure Reading
variation from test to test in the pressures recorded. The data from the pressure gauges were also used to calculate the impulse-time history on the wall.

Accelerometer responses were used to obtain acceleration-time histories. In addition these records were integrated to obtain plots of velocity and displacement-time histories. Because the displacement-time curves did not show the decaying sinusoidal curve that would be expected from an impulsive forcing function followed by free vibration, but rather tended to continuously increase up to levels that were clearly not accurate, it was felt that the displacement curves obtained by integrating the acceleration-time curves were not particularly accurate. This is probably due to permanent rotations in the wall.

The problem with this method of determining displacements from accelerations is that the integration process greatly magnifies any small inaccuracy in the data. Therefore these data were used only to give an estimate of the maximum deflection. The data from the relative displacement gauge was considered to be more accurate because it measured the displacement directly without integration, and because it gave the kind of decaying sinusoidal wave that was expected.

The displacements did not quite follow the expected pattern of lower steel percentages producing higher displacements. Generally this was true, but not for all
tests. Also, walls reinforced with dowels produced larger displacements than walls reinforced with stirrups. All of the walls showed some amount of permanent deflection as a result of the blast as measured by the relative displacement gauge. For walls reinforced with stirrups the permanent deflection was usually on the order of a half an inch or less. Walls reinforced with dowels tended to have two to three times as much permanent deflection. The individual blasts typically moved the concrete box, reaction block, and sand berm back approximately a half an inch.
CHAPTER THREE

THE CHEN AND CHEN MODEL

3.1 General

The finite element code Abaqus, briefly described in Ref [3], was chosen to model the blast tests described in Chapter Two. Abaqus contains a concrete material model which allows the high compressive but low tensile strength of concrete to be modeled. Also Abaqus has special rebar elements which allow the individual steel rebars and their positions in the concrete to be modeled.

Abaqus uses the Chen and Chen stress surface to model the behavior of the concrete in the inelastic range. The concrete is assumed to be a continuous, isotropic, and linearly elastic-plastic strain-hardening material. The nonlinear effects caused by the plasticity of compression concrete and cracking of the tension concrete are considered in the formulation. In the program, the steel is assumed to have compatible displacements but the
constitutive calculations are done separately from the concrete and are governed by its own material definition. Rebar-concrete interaction is modeled by the "tension-stiffening" concept, which is described later.

3.2 Yield and Failure Criterion

The Chen and Chen model assumes the failure criterion is a function of $I_1$, the first invariant of the stress state tensor, $\sigma$, and $J_2$, the second invariant of the deviatoric stress tensor, $S$. The failure criterion takes the following form:

$$ f(\sigma) = K J_2 I_1 I + A I_1 = r \quad (3.1) $$

The yield surface takes the same form as the failure surface, with $f$ given by:

$$ f(\sigma) = K J_2 I_1 I + A I_1 = r \quad (3.2) $$

in which $A_0$, $r_0$, $A_1$, and $r_1$ are material constants which can be determined from simple tests. $K$ was set equal to three because, according to Chen and Chen, it fit
experimental data reasonably well. This results in a parabolic function in the compression region and a hyperbolic function in the tension-compression region. The yield and failure surfaces are shown in Fig. 4 for the case of two non-zero principle stresses. The material constants $A$, $r$, $A_0$, and $r_0$ are functions of $f'_u$, $f'_t$, $f'_c$, $f_u$, $f_t$, and $f_c$. Herein $f'_u$, $f'_t$, and $f'_c$ are the ultimate compressive, uniaxial tensile, and equal biaxial compressive strength of concrete under uniaxial compressive loading, uniaxial tensile loading, and equal biaxial compressive loading, respectively, while $f_u$, $f_t$, and $f_c$ are the initial discontinuous strength of concrete under uniaxial compressive loading, uniaxial tensile loading, and equal biaxial loading, respectively. For $K$ equal to three the material constants $A$, $r$, $A_0$, and $r_0$ can be expressed as follows. For the compression zone:

\[
A_u/f'_u = (f'_u - f'_c)/(2f'_u - f'_c); \quad r_u/f'_u = (3f'_u - 2f'_c)/3(f'_u - f'_c); \\
A_c/f'_c = (f'_c - f'_u)/(2f'_c - f'_u);
\]

\[
r_c/f'_c = (1/3)(2f'_c - 3f'_u - f'_u + 2f'_c)/(2f'_c - f'_u) \quad (3.3)
\]
Fig. 4 Yield and Failure Surface for Concrete
and for the tension-compression zone:

\[
A / f' = (1 - f') / 2; \\
u_{ct} t
\]

\[
2 \quad 2 \\
r / f' = f' / 6; \\
u_{ct} t
\]

\[
A / f' = (\bar{f} - f) / 2; \\
o_{ct} t
\]

\[
2 \quad 2 \\
r / f' = f / 6 \\
o_{ct} t
\]

in which \( \bar{f} \), \( \bar{f}' \), \( \bar{f} \), \( \bar{f} \), and \( \bar{f} \) are the

\( bc \quad c \quad t \quad c \quad t \quad bc \)

nondimensionalized quantities of \( f' \), \( f' \), \( f \), \( f \), \( f \)

\( bc \quad t \quad c \quad t \quad bc \)

with respect to \( f' \).

3.3 Loading Function

The loading surface for this model translates along the \( \sigma = \sigma = \sigma \) axis and expands isotropically simultaneously. This means that the loading function is a combination of kinematic hardening and isotropic hardening models. The loading function takes the form

\[
f(\sigma) = (K J / 3 - K I / 36 + I / 12 + \beta I / 3)(1 - a I / 3) = r \]
where \( a \) and \( \beta \) are given as

\[
a = (A - A_u)/(r_u - r_u);
\]

\[
\beta = A_u r - A_u r_u
\]

\[
(3.6)
\]

In Eq. 3.5 the term \((\beta/3)I\) reflects the concrete property of a much weaker tensile strength than compressive strength. The value of \( r \) determines the rate of isotropic hardening and \((a/3)I\) reflects the kinematic translation of the loading surfaces. For the special case when \( a \) and \( \beta \) are zero, Eq. 3.5 reduces to an extended von Mises criterion.

3.4 Incremental Stress-Strain Relationships

The incremental stress-strain relationships are based on the normality condition. For a stable material, normality requires that:

\[
P \frac{d\varepsilon^{ij}}{d\sigma^{ij}} = G(\delta f/\delta \sigma^{ij})df
\]

\[
(3.7)
\]

in the Chen-Chen model:
\[
G = \sqrt{\frac{p}{p}} \frac{p}{p}
\]
\[
\frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma}
\]
\[
ij \quad ij \quad ij \quad ij
\]

For simplicity denote
\[
1/H = df/\sqrt{\frac{p}{p}} \frac{p}{p}
\]
\[
\frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma}
\]
\[
rs \quad sr
\]

and use the relation
\[
df = \frac{\partial f}{\partial \sigma} d\sigma
\]
\[
\frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma}
\]
\[
\frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma}
\]
\[
mn \quad mn
\]

Then Eq. 3.7 may be rewritten as
\[
d\varepsilon = \frac{\partial f}{\partial \sigma} d\sigma /H \sqrt{\frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma} \frac{\partial f}{\partial \sigma}}
\]
\[
ij \quad ij \quad mn \quad mn \quad rs \quad sr
\]

Assuming that the total incremental strain \(d\varepsilon\) is composed of two parts, namely an elastic and a plastic component such that:
\[
d\varepsilon = d\varepsilon + d\varepsilon
\]
\[
ij \quad ij \quad ij
\]

where
\[
d\varepsilon = H d\sigma
\]
\[
ij \quad ijk \quad kl
\]

Combining Eq. 3.11 and Eq. 3.12
\[
\frac{1}{w} = \left[ 2(1-\nu)J - (1-2\nu)S - 2(1+\nu)\rho + 2(1+\nu)\rho^2 \right]^{\frac{1}{2}}
\]

\[
\phi = \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right]^{\frac{1}{2}}
\]

\[
\phi = \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right] \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right]^{\frac{1}{2}}
\]

\[
\phi = \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right] \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right]^{\frac{1}{2}}
\]

\[
\phi = \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right] \left[ (1-\nu)S - \rho S + (1+\nu)\rho \right]^{\frac{1}{2}}
\]
\[
\begin{align*}
\phi &= \frac{[(1-\nu)S - \rho S + (1+\nu)\rho]}{(1-\nu) r} \\
\phi &= \frac{[(1-\nu) r]}{3} 
\end{align*}
\]

(3.16)

in which \(\nu\) is Poisson's ratio, \(E\) is Young's modulus, \(S\) is the deviatoric stress tensor, and \(\rho\) is defined as:

\[
\rho = \frac{n I (a + \beta r)}{3} 
\]

(3.17)

where \(n\) is zero in the compression zone and \(-1/3\) in the tension zone.

3.5 Implementation

In the program, when the failure surface is reached the response is dependent upon which part of the surface is reached. If it reaches the triaxial compression zone, then the concrete is assumed to crush and immediately loses all strength. If the failure surface is reached in the tension-compression zone, then the concrete is assumed to crack in a plane perpendicular to the largest principle strain. When this happens the concrete is assumed to lose all strength across the crack, but this assumption may be modified by the program options of tension stiffening and shear retention, which will be discussed later. Also, the concrete is assumed to recover its stiffness if the crack...
closes during subsequent loading, although the original
direction of the crack is recorded so that it can reopen.
Since the Chen and Chen failure criterion is based on
reaching a particular stress and not on strain, the theory
tends to overpredict the ductility of concrete in the
tension zone. For this reason an additional failure
criterion based on total mechanical strain has been added
to the program. The criterion is that cracking occurs when
the largest principal strain reaches a value either given
by the user or calculated by the program to be $2f_{tE}$.

In plain concrete, cracking leads to an immediate loss
of stiffness, and stress across the crack drops to zero.
In reinforced concrete, due to the interaction of the
rebars in the concrete, the remaining concrete continues to
provide some stiffness. The tension stiffening option in
Abaqus can be used to enter an unloading curve after
cracking. The default is to have the stress across the
crack drop to zero. The user gives the fraction of
remaining stress as a function of strain above cracking.
The development of a crack will lead to the shear stresses
parallel to the crack being set to zero. However in a thin
crack because of the unevenness of the surface there may be
some interlocking of the two crack surfaces. This effect
can be simulated by the program's shear retention option.
Thus the shear stiffness does not immediately drop to zero
upon the development of a crack but instead drops according
to a user defined curve until a specified strain is reached.
4.1 Model Geometry

In attempting to model the geometry of the concrete box, it was decided to model just half of the box since the box and the effects of the blast are half-symmetric. Also, only the front wall of the box was modeled to cut down on the size of the problem. Three basic models were developed.

Model One used the concrete material and rebar elements to model both the vertical and horizontal steel reinforcement in both the inside and outside faces. The shear reinforcement was not modeled. This model uses the eight node brick element. The first model was two elements (three nodes) deep through the thickness, 12 elements (13 nodes) along the length and 23 elements (24 nodes) high as shown in Fig. 5. There was a total of 552 elements and 936 nodes. In Model One, for 0.5% vertical steel
Fig. 5 Model One
reinforcement, #3 size rebar elements were defined running horizontally and vertically through the center of each element.

Model Two was a simple elastic single material model which contained a larger number of elements than the first model. As with the first model, the second model uses the eight node brick element. Model Two was three elements (four nodes) deep through the thickness of the wall, 12 elements (13 nodes) along the length, and 23 elements (24 nodes) high. There was a total of 828 elements and 1248 nodes. This mesh is similar to the one shown in Fig. 5, except that the second model was three elements deep through the thickness instead of two. The first two models used full integration.

Model Three was a simple elastic single material model which used the twenty node brick element with reduced integration. This model was one element deep through the thickness of the wall, six elements along the length, and 15 elements high for a total of 90 elements and 738 nodes. This mesh is shown in Fig. 6.

The nodal and element data for the first two meshes were generated separately in an available short Fortran routine. The nodal and element data for the third model were generated separately in a different Fortran routine and were input directly into the Abaqus data file. This procedure was found to be easier than using the generation
Fig. 6 Model Three
4.2 Boundary Conditions

The three models were considered to be fixed along the back face of the wall where the roof and the floor joined the wall. This meant that the back of the wall was fixed for a distance of 12.5 inches from both the top and bottom of the wall. The size of the bottom and top elements was increased so that these boundary conditions could be exactly met. The larger size of the elements in the top and bottom layers is noticeable in Fig. 5 and Fig. 6. The larger element size and hence decreased accuracy of results was deemed acceptable in this area because very little displacement would occur there. Since the three models were considered to be fixed along the bottom and top to the concrete reaction block, the energy used in translating the box and reaction block was not accounted for. Because the wall was assumed to be half-symmetric, and only half of the wall was modeled, the wall was fixed in the X direction on the plane of symmetry.

The above boundary conditions do not completely simulate the actual tests. The assumption of fixity at the top of the wall is not quite accurate. In the experiments, the roof of the box whipped up and down, allowing the top of the wall to rotate a small amount. To reduce this
discrepancy between the tests and the computer simulation a second series of computer models was developed in which, the boundary conditions at the top of the wall were relaxed. The top row of elements was allowed to rotate about an axis parallel to the X-axis located 6.25 inches (the center of the area where the wall and the roof meet) below the top of the wall in the back. This axis was not allowed to translate in the Y direction, but was allowed to translate in the X direction. This assumes that the displacement of the roof is small, which is consistent with test measurements obtained by the accelerometers placed on the inside roof of the boxes. Thus there were two versions of each of the three models, one fixed at the top of the wall, and one allowed to rotate at the top of the wall.

4.3 Material Models
---------------------

In the first model, which contained the concrete material model and the rebar elements, properties for both the concrete and the steel had to be provided to the program. Because Abaqus considers the behavior of the concrete to be linear in the elastic range, only Young's modulus and Poisson's ratio had to be supplied for this part. Young's modulus for $f' = 5000$ psi concrete was $6\times10^6$ psi, and Poisson's ratio was taken as 0.2.
For the plasticity modeling, uniaxial compressive yield stress, \( f' \), was taken as 3000 psi and the uniaxial compressive ultimate strength was taken as 5500 psi. The values for the yield and ultimate strength of the concrete were taken from the results of strength tests performed on concrete samples taken from the batches from which the boxes were poured. The results of these tests can be seen in Appendix B. The ratio of biaxial to uniaxial ultimate compressive strength was assumed to be 1.16, which is the Abaqus code default value. The ratio of uniaxial tensile strength to uniaxial compressive strength was assumed to be 0.09, also the Abaqus code default value.

The steel in the first model was considered to be a homogeneous isotropic, elastic-perfectly plastic material. For elasticity, Young’s modulus was taken as 29x10^6 psi, and Poisson’s ratio was taken as 0.3. For plasticity, the yield stress was taken to be 36 ksi.

In the second and third models, which were the plain elastic, single material models, the only material constants which had to be supplied to the program were Young’s modulus and Poisson’s ratio. Young’s modulus for the composite material was calculated by using the rule of mixtures to smear together the properties of the concrete and steel. Recalling from the previous paragraph that Young’s modulus for concrete and steel was 3.84x10^6 psi and 29x10^6 psi, respectively. The smeared Young’s modulus was
calculated to be $3.97 \times 10$ psi for 0.5% vertical steel reinforcement. The composite concrete-steel material was considered to be isotropic, mainly for the sake of simplicity. This was not a particularly good assumption, since the percentage of steel was not the same in all three directions.

Poisson's ratio for the composite material was likewise calculated from the Poisson's ratios of steel and concrete according to a formula based on energy methods. This procedure is described in Appendix A. Poisson's ratio was calculated from the Poisson's ratio of concrete, 0.2, and the Poisson's ratio of steel, 0.3. The smeared Poisson's ratio was calculated as 0.2005. The formula used to calculate Poisson's ratio produced a higher value of Poisson's ratio than would have been obtained by using the rule of mixtures.

A similar procedure for calculating the equivalent Young's modulus for a smeared steel-concrete material is also described in Appendix A. However the equivalent Young's modulus produced from this formula was almost the same as that from the rule of mixtures. Because the rule of mixtures is simpler to use and resulted in similar estimates of Young's modulus, the rule of mixtures was used to calculate the equivalent Young's modulus.
4.4 Loading

Loading information was taken from the data provided by the pressure gauges in the six wall tests. A sample pressure-time curve is shown in Fig. 7. Since the signal was rather noisy, only a few representative points were taken from the curves.

The pressure-time history of the blast was modeled as a double triangle, as shown in Fig. 8. Taken from the pressure records were the amplitude of the pressure peak, the time of arrival of the blast wave, the time of the pressure peak, the intermediate time, which is the time where the slope of the pressure-time history curve is discontinuous after the peak, and the end time, which is the time where the blast pressure becomes zero.

The pressure-time curve used in the analysis was obtained by averaging the data recorded for all six tests at each pressure gauge. However, the pressure gauges closest to the center of the blast (P91, P61, P21) were combined to produce one value considered typical of the three gauges. Using the values averaged at each pressure gauge, three graphs were drawn showing the peak pressure distribution along the three lines of gauges. From these plots, data were taken to obtain the equivalent pressure curves shown in Fig. 9. Equipressure lines are labeled A-H. These equipressure lines were used to break the front face nodes up into equipressure node groups. The average
Fig. 7  Gauge P64 Pressure Reading
Fig. 8 Percent Pressure-Time History Curve

<table>
<thead>
<tr>
<th>Zone</th>
<th>Velocity (in/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1079.604</td>
</tr>
<tr>
<td>2</td>
<td>819.948</td>
</tr>
<tr>
<td>3</td>
<td>546.636</td>
</tr>
<tr>
<td>4</td>
<td>341.640</td>
</tr>
<tr>
<td>5</td>
<td>204.996</td>
</tr>
<tr>
<td>6</td>
<td>102.492</td>
</tr>
<tr>
<td>7</td>
<td>40.992</td>
</tr>
</tbody>
</table>

Table 2. Initial Velocities
Fig. 9  Blast Pressure Distribution
pressure values used as the loads on the node groups are labeled as L1-L7.

A similar procedure of data averaging was followed for the four times of interest. The average arrival time was taken as 0.001 sec, the average peak time was taken as 0.00108 sec, the intermediate time was taken as 0.00143 sec, and the end time was taken as 0.00325 sec, as shown in Fig. 8.

The load and pressure time histories were input into Abaqus by treating the blast as an impulsive loading. The blast could also have been treated as a dynamic loading instead of an impulsive loading. However, this was found to be impractical to implement due to the very small time step needed to adequately define the loading curve. With such a small time step, it would have been necessary to restart the program several times, requiring vast amounts of computer time.

However, when the blast was treated as an impulsive loading, initial velocities needed to be calculated. The equation used to calculate the initial velocity $v$ was

$$\frac{\int F(t) \, dt}{m} = v \quad (4.1)$$

where $\int F(t) \, dt$ is the integral with respect to time of the pressure-time history curve times the area of an element that the pressure acts on. This is calculated as
the area under the percent pressure-time history curve times the peak pressure and the area the pressure acts on. The mass, \( m \), is

\[
m = \rho AT
\]

(4.2)

where \( \rho \) is the density of reinforced concrete, 150 lb/in \(^3\), \( A \) is the area mentioned above, and \( T \) is the thickness of the wall, 12.5 in. Substituting Eq. 4.2 into Eq. 4.1 gives

\[
\frac{\int P(t) A \, dt}{\rho A T} = \frac{\int P(t) \, dt}{\rho T} = v
\]

(4.3)

so that velocity is not dependent upon the area the blast acts upon. Since there are seven equipressure zones, there are seven equal velocity zones corresponding to the equipressure zones. The initial velocity calculated for each zone is shown in Table 2. Initial velocities were input only for the portion of the wall between the roof and the floor. No initial velocities were input for the top and bottom 12.5 inches of the wall. The same initial velocity was input for each node through the thickness of the wall. A suggested time step of 2 msec was chosen for ten steps for a total analysis time of 20 msec. However, to increase program efficiency, an automatic time stepping scheme was used, so that the size of the time step was not known a priori.
5.1 Analysis of Blast Test Results

In order to analyze the results Abaqus produced, first the results of the blast tests must be studied. The blast test results from the pipe gauge are summarized in Table 3 and presented in more detail in Appendix B. From Table 3 it is apparent that in each of the tests the walls suffered permanent deflections, which Abaqus could not model. Thus to compare the results Abaqus produced with the blast test results, it was necessary to remove the effects of the permanent deflection.

Removing the effects of the permanent deflection was done by subtracting the permanent deflection out of the blast test deflections. The method used was to assume that the deflection increased linearly from zero to its maximum value at the same time the peak in the total deflection was observed. From the time of the peak deflection, the
permanent deflection was considered to be constant at its maximum value. The blast test results with the permanent deflections subtracted out are shown in Fig. 10 through Fig. 15.

<table>
<thead>
<tr>
<th>percent steel</th>
<th>Max Defl (in)</th>
<th>Residual Defl (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SWT #1</td>
<td>1.00</td>
<td>0.63</td>
</tr>
<tr>
<td>SWT #2</td>
<td>0.50</td>
<td>0.95</td>
</tr>
<tr>
<td>SWT #3</td>
<td>0.50</td>
<td>2.90</td>
</tr>
<tr>
<td>SWT #4</td>
<td>0.25</td>
<td>2.60</td>
</tr>
<tr>
<td>SWT #5</td>
<td>2.00</td>
<td>0.99</td>
</tr>
<tr>
<td>SWT #6</td>
<td>0.50</td>
<td>1.65</td>
</tr>
</tbody>
</table>

Table 3. Scratch Gauge Readings

5.2 Model One

The results Abaqus produced did not accurately model the blast test results. The first model to be tested, Model One, which utilized the concrete material and rebar elements, produced the poorest results of the three models, as will be seen later. Abaqus completed fifteen time steps of approximately 0.5 msec each for a total analysis time of 9.313 msec before ending due to machine limits on system
Fig. 10 Blast Test Results
Fig. 11 Blast Test Results
Fig. 12  Blast Test Results
Fig. 13 Blast Test Results
SWT-5 2% STIRRUPS

Fig. 14 Blast Test Results
Fig. 15 Blast Test Results
resource units used. Nearly 11,284 sec of cpu time on the CDC Cyber 370 was used before the program execution was halted.

Deflections at the back center of the wall are plotted against time in Fig. 16 and Fig. 17. The marked points are the Abaqus results. Also shown for comparison on the graphs are the blast test deflections for 2%, 1%, 0.5%, and 0.25% vertical steel reinforcement with stirrups for shear reinforcement.

Both Fig. 16 and Fig. 17 show that the model was much too stiff. The results Abaqus produced for both versions of the model underpredicted both the maximum displacement and the period of vibration of the wall. The maximum displacements of the center point of the wall predicted by both versions of Model One were less than one half of the value of the maximum displacement in the blast test for 0.5% vertical steel reinforcement. The difference between the maximum displacements of the two versions small, 0.02 in, or about a seven percent increase in the maximum displacement for the relaxed boundary condition version.

The period predicted by the first model with fixed boundary conditions was approximately one fifth the period in the blast test, and that predicted by the first model with relaxed boundary conditions was approximately one fourth the period in the blast tests. Relaxing the boundary conditions had a greater effect on the period of
MODEL ONE - FIXED BC

Fig. 16 Abaqus Results
MODEL ONE - RELAXED BC

Fig. 17 Abaqus Results
vibration than on the maximum displacement, increasing the period by one msec or nearly twenty percent. This is somewhat to be expected since relaxing the boundary conditions in effect increases the free height of the wall. Likewise, the version with relaxed boundary conditions shows more effects of secondary modes of vibration than the version with fixed boundary conditions.

The deflected shapes of the two versions of the first model were similar for the first 4.5 to 5 msec. From that time onwards the deflected shapes increasingly do not resemble each other, as seen in Fig. 18 through Fig. 32. These figures show the displaced shape of the back vertical centerline of the wall at specific times. The marked points are the nodes. Only the free height of the wall, that between the floor and the roof, is shown. The version of Model One with relaxed boundary conditions is easily distinguished since its displacement is not zero at the juncture of the wall and the roof (z = 77.5 in).

Damping in the blast test walls was stronger than in the Abaqus model. However the damping coefficient was not considered to be of primary interest in the present study so the Abaqus default damping coefficient of -0.05 was used, which was just enough to remove high frequency "noise" from the solution.
TIME = 0.002 SEC

- MODEL ONE - FIXED BC
- MODEL ONE - RELAXED BC

Fig. 18 Abaqus Results
TIME = 0.0025 SEC

Fig. 19  Abaqus Results
TIME = 0.0030 SEC

Fig. 20 Abaqus Results
TIME = 0.0035 SEC

- MODEL ONE - FIXED BC
- MODEL ONE - RELAXED BC

Fig. 21 Abaqus Results
TIME = 0.0040 SEC

Fig. 22  Abaqus Results
Fig. 23 Abaqus Results
Fig. 24 Abaqus Results
Fig. 25 Abaqus Results
TIME=0.0060 SEC

Fig. 26 Abaqus Results

58
TIME = 0.0065 SEC

Fig. 27 Abaqus Results
TIME=0.007125 SEC

- MODEL ONE - FIXED BC
- MODEL ONE - RELAXED BC

Fig. 28 Abaqus Results
Fig. 29 Abaqus Results
TIME = 0.008125 SEC

Fig. 30 Abaqus Results
TIME = 0.008531 SEC

- MODEL ONE - FIXED BC
- MODEL ONE - RELAXED BC

Fig. 31 Abaqus Results
Fig. 32 Abaqus Results
5.3 Model Two

Model Two, the elastic, eight node brick model, produced better results than the concrete model, but the results were still poor when compared with the blast test results. Abaqus completed seven time steps of approximately one msec each for a total analysis time of 6.5 msec for the version of the model with fixed boundary conditions and seven msec for the version with relaxed boundary conditions before the program execution halted due to machine limits on system resource units used. About the same amount of cpu time on the Cyber was used in the second model as in the first, 12430 sec, before the program execution halted.

Deflections at the back center of the wall are plotted against time in Fig. 33 and Fig. 34. The marked points are the Abaqus results. Also shown for comparison on the graphs are the blast test deflections of the back centerpoint of the wall for 2%, 1%, 0.5%, and 0.25% vertical steel reinforcement with stirrups for shear reinforcement.

Both Fig. 33 and Fig. 34 show that, as was the case for Model One, the model was much too stiff. The results Abaqus produced for both versions of Model Two underpredicted both the maximum displacement and period of vibration of the wall. The maximum displacements of the back centerpoint of the wall predicted by both versions of
MODEL TWO - FIXED BC

Fig. 33 Abaqus Results
MODEL TWO - RELAXED BC

Fig. 34 Abaqus Results
the second model were approximately one half of the value of the maximum displacement in the blast test for 0.5% vertical steel reinforcement. The difference between the maximum displacement of the back centerpoint of the two versions was larger than in the last model, 0.032 in, or about an eleven percent increase in deflections in the version with relaxed boundary conditions.

The period predicted by the second model with fixed boundary conditions was approximately thirty percent of the period in the blast test, and that predicted by the second model with relaxed boundary conditions was approximately one third the period in the blast tests. As with the first model, relaxing the boundary conditions had a greater effect on the period of vibration than on the maximum displacement, increasing the period by one msec or nearly eighteen percent. Also, the version with relaxed boundary conditions shows more effects of secondary modes of vibration than the version of the second model with fixed boundary conditions.

The deflected shapes of the two versions of the second model tended to approximate each other until 5.5 to six msec had passed, and from that time onwards the deflected shapes increasingly do not resemble each other, as seen in Fig. 20 to Fig. 27. These figures show the deflected shape of the back centerline of the wall at specific times. The marked points are the nodes.
5.4 Model Three

Model Three, the elastic twenty node brick model produced results that better modeled the tests than Model One but were slightly worse than Model Two. Again, the results were still poor when compared against the blast test results. Abaqus completed 38 time steps of approximately 0.5 msec each for a total analysis time of 21 msec. Both versions of the third model completed program execution without using up the allowable system resource units. Only 6433 sec of cpu time on the Cyber was used, much less than in the other two models.

Deflections at the back centerpoint of the wall are plotted against time in Fig. 35 and Fig. 36. The marked points are the Abaqus results. Also plotted for comparison on the graphs are the blast test deflections of the centerpoint of the wall for 2%, 1%, 0.5%, and 0.25% vertical reinforcement with stirrups for shear reinforcement. Only ten msec of the total analysis time of 21 msec are shown on both figures. This was done to facilitate comparison with the other models.

Both Fig. 35 and Fig. 36 show that Model Three, like the other two models, was much too stiff. The results Abaqus produced for both versions of the third model underpredicted both the maximum displacement and period of vibration of the wall. The maximum displacements of the back centerpoint of the wall predicted by both versions of
MODEL THREE - FIXED BC

Fig. 35 Abaqus Results
MODEL THREE - RELAXED BC

Fig. 36 Abaqus Results
the third model were approximately one half of the value of
the maximum displacement in the blast test for 0.5%
vertical steel reinforcement. The difference between the
maximum displacement of the back centerpoint of the two
versions was quite small, 0.0077 in, only an increase of
only two percent in deflections in the version of Model
Three with relaxed boundary conditions. This increase is
much smaller than in the previous two models.

The period predicted by the third model with fixed
boundary conditions was approximately one fourth the period
in the blast test, and that predicted by the third model
with relaxed boundary conditions was approximately thirty
percent of the period in the blast tests. As with the
previous two models, relaxing the boundary conditions had a
greater effect on the period of vibration than on the
maximum displacement, increasing the period by 0.8 msec or
nearly sixteen percent. The increase in the period was
less than in the other two models. Also, the version of
Model Three with relaxed boundary conditions showed more
effects of secondary modes of vibration than the version of
the third model with fixed boundary conditions.

The deflected shapes of the two versions of the third
model were similar to each other until nearly 2.7 msec had
passed, and from that time onwards the deflected shapes
increasingly do not resemble each other, as may be seen in
Fig. 37 through Fig. 46. These figures show the displaced
TIME = 0.001575 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 37 Abaqus Results
TIME = 0.001959 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 38 Abaqus Results
TIME = 0.002342 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 39 Abaqus Results
TIME=0.002726 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 40 Abaqus Results
TIME = 0.003157 SEC

---

**MODEL THREE - FIXED BC**

**MODEL THREE - RELAXED BC**

---

**Fig. 41 Abaqus Results**
Fig. 42 Abaqus Results
Fig. 43 Abaqus Results
TIME = 0.005120 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 44 Abaqus Results
TIME=0.005869 SEC

- MODEL THREE - FIXED BC

- MODEL THREE - RELAXED BC

Fig. 45 Abaqus Results
TIME = 0.006711 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC

Fig. 46 Abaqus Results
shape of the back centerline of the wall at specific times. The marked points are the nodes. The figures also show the effects of secondary modes of vibration on the displaced shape of the wall. Only some of the deflected shapes for the first ten msec are shown. The third model produced results at many more time steps than these figures show. More complete results for the third model are presented in Appendix C.

5.5 Comparison of Models One - Three

All three of the models grossly underpredicted both the maximum displacement at the centerpoint of the wall and the period of vibration. Among them, Model Two produced the highest maximum displacements and the longest periods. Model One produced the lowest maximum displacements and the shortest periods. In each of the three models, relaxing the boundary conditions improved the results, most especially in lengthening the period. Model Three improved the least of the three models with the relaxation of the boundary conditions at the top of the wall, and Model Two improved the most. Also, in each of the three models, the version of the model with relaxed boundary conditions always showed more of the effects of secondary modes of vibration. However, the maximum displacements generally changed more with different models than with different
versions of the same model. This was not true for the period of vibration, which was affected more by the version of the model than by the model itself.

Comparing the three models shows that the third model was the most efficient of the three, since it completed more than twice as many steps and twice as much time as the first model, and more than five times as many steps and three times as much time as the second model. Also, the third model accomplished this on half as much cpu time as the other two models. Of the three models, Model Two used up the most cpu time while completing the fewest steps and the least time. Between different versions of the same model, there was always less than 10 cpu sec time difference, and for that reason only one time was given for each model. Different versions of the same model always ran for the same number of steps, and usually covered the same amount of time. Only the two versions of the second model ran for different lengths of time. Thus it was the model rather than the version of the model that made the important differences between cpu time, steps and time completed.
CHAPTER SIX

CONCLUSIONS

6.1 General

It has been shown that the dynamic response of the concrete walls predicted using the Abaqus finite element computer code poorly approximated observed behavior. Specifically, the Abaqus results did not match the blast test results in either the period of vibration or the maximum displacement of the wall. The results Abaqus produced for the maximum displacements were closer to the blast test results than the results Abaqus produced for the period of vibration, but even the maximum displacements were approximately half of what was expected. The poor agreement between predicted and observed behavior could be due to either problems in the Abaqus code itself or in problems with the specific model studied.
6.2 Problems with Abaqus

In general, the Abaqus code did not prove to be very well suited to this type of problem. Abaqus is not easy to use for large problems. The nodal point generation routine in Abaqus is very simple, only generating nodes in a straight line between two defined points. The generation routine could easily be made more sophisticated and easy to use. For a large mesh, generating the nodal data requires a large number of statements in which mistakes are difficult to spot until the preprocessor program is run. The preprocessor program itself can take a long time to run due to the fact that the preprocessor requires almost all of the core storage available to execute even for small problems. Thus debugging the nodal point data for a large problem can become quite time consuming. For this reason it is suggested that in the future, for large problems, a Fortran routine which will generate the nodal data in the proper format be written. Such programs are simple and easily written, and the data may be easily checked for errors before being entered into the Abaqus input file.

The element generation routine also suffered from the same problems as the nodal point generation routine. Similarly, a Fortran routine was used to generate the element data in this investigation. However in the most
recent version of Abaqus, a more sophisticated element generation routine is included that might make element generation easier. However it is suggested that as with the nodal point data, a separate Fortran routine is the quickest way to generate element data.

Entry of the remainder of the data is straightforward, though the User Information Manual is not always very clear. In particular, the instructions for restarting programs and for using more than one analysis step are quite confusing. The graphics capabilities are quite nice and are rather easy to use. The user is given many options on the picture or graph drawn, the viewpoint, and the labels used. The default options produce a nice picture, and changes from the default options are easily made.

Another problem encountered in using Abaqus is the overly large amount of output the program produces for large problems. Some of the problems stem from inefficient page usage, since Abaqus only uses 6.5 inches of an 11 inch page. Also, while much of the unwanted analysis results can be suppressed, the preprocessor output cannot be suppressed, and for large problems the preprocessor output was truly voluminous. For the models of this investigation, the preprocessor output was well over a hundred pages even for the smallest model. The default options on the analysis output tend to result in more information than is useful, especially in the element
output, even for rather small problems.

A more serious problem with Abaqus was the limits on the size of problem which could be analyzed. Model Two was about as large a problem as the system could handle. There was no virtual memory system to handle very large problems, although there are plans to install one eventually. The Abaqus results tended to improve with increasing mesh size, however, the models could not be made very much larger due to the limited core memory available.

6.3 The Chen and Chen Model

This problem did not appear to provide a very good test of the concrete failure model used in Abaqus. The results of Model One, which used the Chen and Chen model were not very different from the two elastic models. This would seem to indicate that the response of the wall was mostly in the elastic range, and that the plasticity model was not really used, which would seem to indicate that the failure of the concrete in the blast tests was not due to the structural response.

6.4 Problems with the Models

The model itself could be improved in several ways. First of all, the mesh might have been made larger, since
increasing the mesh size seemed to improve results. However, as previously discussed, this is not possible without changes in the program. Also, more of the structure could have been modeled. Specifically, the roof as well as the front wall of the concrete box could have been included in the analysis. This might have improved estimates of the period of vibration of the wall. Again, because of limitations on the size of the problem, the mesh would necessarily have to be made rather coarse, which would tend to negate the possible benefits of modeling both the roof and the wall together.
APPENDIX A

Conservation of Strain Energy
Conservation of Strain Energy

(Capt. Paul Rosengren)

15 Aug 83

\[ w = \frac{1}{2} e + G E \sum_{i,j} E_{ij} \]  \hfill (A.1)

where \( w \) = strain energy per unit volume

\( E \) = Young's modulus

\( \nu \) = Poisson's ratio

\( \lambda \) = Lamé's constant = \( E / (1+\nu)(1-2\nu) \)

\( G \) = shear modulus = \( E/2(1+\nu) \)

\( e \) = first invariant of strain = \( E = E_{kk} + E_{11} + E_{22} + E_{33} \)

\( E \) = strain matrix

\( V \) = volume

assuming a linear isotropic material:

\[ w V = w V + w V \]  \hfill (A.2)

\[ T T \quad C C \quad S S \]

Total energy = energy in concrete + energy in steel

expanding Eq. A.2 using Eq. A.1:

\[ w V = (1/2\lambda e + G E \sum_{i,j} E_{ij}) V + (1/2\lambda e + G E \sum_{i,j} E_{ij}) V \]  \hfill (A.3)

\[ T T \quad c \quad c \quad i j \quad i j \quad c \quad s \quad s \quad i j \quad i j \quad s \]
Now define an equivalent $\lambda$ and $G$ such that:

\begin{align*}
2
(1/2\lambda V + G E) V = W V \\
\text{eq eq ij ij} & T T \\
\end{align*}

and $V = V + V$ \hspace{1cm} (A.5)

This gives the result that:

\begin{align*}
2
1/2(\lambda V + \lambda V)e + (G V + G V)E E = \\
\text{cc ss cc ss ij ij} \\
2
1/2\lambda V e + G V E E \\
\text{eq T eq T ij ij} \\
\end{align*}

Equating terms:

\begin{align*}
1/2(\lambda V + \lambda V) &= 1/2\lambda V; \\
\text{cc ss eq T} \\
G V + G V &= G V \\
\text{cc ss eq T} \\
\end{align*}

This leads to the definition of $\lambda$ and $G$:

\begin{align*}
\lambda V &= \frac{\lambda V}{cc ss} E V \\
\lambda &= \frac{\lambda V}{cc ss} + \frac{\lambda V}{cc ss} \\
\text{eq eq} & \frac{E V}{cc ss} \\
\text{eq eq} & \frac{V V}{T T} \\
(1+\nu)(1-2\nu) & \text{eq eq} \\
\end{align*}

(A.8)
\[ G = \frac{G_v}{V} + \frac{G_c}{V} = \frac{E}{2(1 + \nu)} \]  

(A.9)

Since the material constants for steel and concrete are known, equivalent material constants can be found using volume percentages. Since Young's modulus and Poisson's ratio are commonly used material constants in computer models, it is convenient to solve for these. Eq. A.8 and A.9 represent a linear mixture using volume percentages of Lamé's constant and the shear modulus. Combining Eq. A.8 and A.9 yields:

\[ \frac{\lambda}{G} = \frac{2\nu}{1 - 2\nu} \]  

(A.10)

and the resulting expressions for \( \nu \) and \( E \):

\[ \nu = \frac{\lambda}{2(\lambda + G)} \]  

(A.11)

\[ E = 2G \left(1 + \nu \right) \]
useful expressions for the Lame's constant and the shear modulus are:

\[
E \nu = \frac{E \nu}{(1 + \nu)(1 - 2\nu)}
\]

\[
G = \frac{E}{2(1 + \nu)} \tag{A.12}
\]

Eq.s A.11 are non-linear expressions for equivalent Young's modulus and Poisson's ratio based on a linear mixture of Lame's constant and the shear modulus. Selecting typical values for the material constants and examining the resulting expressions gives some interesting observations.

Example Problem

\begin{align*}
E &= 3000000 \text{ psi} \\
\nu &= 0.2 \\
\text{concrete:} \\
\lambda &= \frac{3000000(0.2)}{(1 + 0.2)(1 - 2(0.2))} = 833333 \text{ psi} \\
G &= \frac{3000000}{2(1 + 0.2)} = 1250000 \text{ psi}
\end{align*}

\[94\]
steel:

\[
\lambda = \frac{29000000(0.3)}{(1 + 0.3)(1 - 2(0.3))} = 16730800 \text{ psi}
\]

\[
G = \frac{29000000}{2(1 + 0.3)} = 11153800 \text{ psi}
\]

<table>
<thead>
<tr>
<th>( V/V )</th>
<th>( V/V )</th>
<th>( \lambda )</th>
<th>( G )</th>
<th>( \nu )</th>
<th>( E )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.000</td>
<td>0.000</td>
<td>833333</td>
<td>1250000</td>
<td>0.2000</td>
<td>3000000</td>
</tr>
<tr>
<td>0.995</td>
<td>0.005</td>
<td>912820</td>
<td>1299519</td>
<td>0.2063</td>
<td>3135225</td>
</tr>
<tr>
<td>0.990</td>
<td>0.010</td>
<td>992307</td>
<td>1349038</td>
<td>0.2119</td>
<td>3269825</td>
</tr>
<tr>
<td>0.980</td>
<td>0.020</td>
<td>1151282</td>
<td>1448076</td>
<td>0.2215</td>
<td>3537520</td>
</tr>
<tr>
<td>0.970</td>
<td>0.030</td>
<td>1310257</td>
<td>1547114</td>
<td>0.2293</td>
<td>3803662</td>
</tr>
<tr>
<td>0.960</td>
<td>0.040</td>
<td>1469231</td>
<td>1646152</td>
<td>0.2358</td>
<td>4068638</td>
</tr>
<tr>
<td>0.950</td>
<td>0.050</td>
<td>1628206</td>
<td>1745190</td>
<td>0.2413</td>
<td>4332715</td>
</tr>
<tr>
<td>0.750</td>
<td>0.250</td>
<td>4807670</td>
<td>3725950</td>
<td>0.2817</td>
<td>9551025</td>
</tr>
<tr>
<td>0.000</td>
<td>1.000</td>
<td>16730800</td>
<td>11153800</td>
<td>0.3000</td>
<td>29000000</td>
</tr>
</tbody>
</table>

Linear Combination

Comparison to a linear combination of Young's modulus and Poisson's ratio directly by volume ratio.

\[
\frac{\nu}{V} = \frac{\nu}{V}
\]

\[
E = E + E
\]

95
\[ \nu = \nu - \frac{\nu}{V} \quad \text{and} \quad \nu = \nu - \frac{\nu}{T} \quad \text{(A.13)} \]

using the same constants as in the previous example problem:

\[
\begin{align*}
E &= 3000000 \text{ psi} & E &= 29000000 \text{ psi} \\
\nu &= 0.20 & \nu &= 0.30
\end{align*}
\]

<table>
<thead>
<tr>
<th>( V/V )</th>
<th>( V/V )</th>
<th>( \nu )</th>
<th>( E )</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>T</td>
<td>s</td>
<td>T</td>
</tr>
<tr>
<td>1.000</td>
<td>0.000</td>
<td>0.2000</td>
<td>3000000</td>
</tr>
<tr>
<td>0.995</td>
<td>0.005</td>
<td>0.2005</td>
<td>3130000</td>
</tr>
<tr>
<td>0.990</td>
<td>0.010</td>
<td>0.2010</td>
<td>3260000</td>
</tr>
<tr>
<td>0.980</td>
<td>0.020</td>
<td>0.2020</td>
<td>3520000</td>
</tr>
<tr>
<td>0.970</td>
<td>0.030</td>
<td>0.2030</td>
<td>3780000</td>
</tr>
<tr>
<td>0.960</td>
<td>0.040</td>
<td>0.2040</td>
<td>4040000</td>
</tr>
<tr>
<td>0.950</td>
<td>0.050</td>
<td>0.2050</td>
<td>4300000</td>
</tr>
<tr>
<td>0.750</td>
<td>0.250</td>
<td>0.2250</td>
<td>9500000</td>
</tr>
<tr>
<td>0.000</td>
<td>1.000</td>
<td>0.3000</td>
<td>29000000</td>
</tr>
</tbody>
</table>

96
LINEAR COMBINATION

ENERGY METHOD

EQUIVALENT YOUNG'S MODULUS x10³

STEEL VOLUME FRACTION

EQUIVALENT POISSON'S RATIO

97
APPENDIX B

----------

Selected Blast Test Results
### Results of Compressive and Tensile Strength Tests

<table>
<thead>
<tr>
<th>Structure</th>
<th>Location</th>
<th>Comp. Strength (psi)</th>
<th>Tensile Strength (psi)</th>
<th>Days Cured</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Floor</td>
<td>5270</td>
<td>736</td>
<td>113</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5350</td>
<td>731</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>5380</td>
<td>762</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Wall</td>
<td>5090</td>
<td>727</td>
<td>105</td>
</tr>
<tr>
<td></td>
<td>Roof</td>
<td>5290</td>
<td>740</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>5320</td>
<td>742</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>Floor</td>
<td>4950</td>
<td>632</td>
<td>117</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4320</td>
<td>574</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4390</td>
<td>678</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Wall</td>
<td>5450</td>
<td>687</td>
<td>105</td>
</tr>
<tr>
<td></td>
<td>Roof</td>
<td>5300</td>
<td>621</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>5300</td>
<td>709</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>Floor</td>
<td>5520</td>
<td>729</td>
<td>112</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5690</td>
<td>724</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Wall</td>
<td>6150</td>
<td>797</td>
<td>106</td>
</tr>
<tr>
<td></td>
<td>Roof</td>
<td>6190</td>
<td>806</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>6580</td>
<td>630</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>6630</td>
<td>754</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>6610</td>
<td>728</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>6660</td>
<td>674</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>6670</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>D</td>
<td>Floor</td>
<td>4400</td>
<td>694</td>
<td>110</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4540</td>
<td>744</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4540</td>
<td>699</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Roof</td>
<td>5870</td>
<td>619</td>
<td>104</td>
</tr>
<tr>
<td></td>
<td>Wall</td>
<td>5230</td>
<td>729</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>5300</td>
<td>664</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4770</td>
<td>328</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4930</td>
<td>240</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4970</td>
<td>484</td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX C

Abaqus Results - Model Three
TIME=0.001384 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.001767 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.002150 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME=0.002534 SEC

• MODEL THREE - FIXED BC

• MODEL THREE - RELAXED BC
TIME = 0.002917 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.003996 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.004745 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.005494 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.006243 SEC

- MODEL THREE - FIXED BC

- MODEL THREE - RELAXED BC

DEFLECTION (IN.) AT X=0
TIME = 0.007647 SEC

* MODEL THREE - FIXED BC

* MODEL THREE - RELAXED BC
TIME=0.008116 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME=0.008701 SEC

* MODEL THREE - FIXED BC
* MODEL THREE - RELAXED BC
TIME = 0.009286 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.009871 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.010603 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.01133 SEC

• MODEL THREE - FIXED BC

• MODEL THREE - RELAXED BC
TIME = 0.01207 SEC

* MODEL THREE - FIXED BC

* MODEL THREE - RELAXED BC

DEFLECTION (IN.) AT X = 0
TIME = 0.01280 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.01371 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.01463 SEC

- MODEL THREE - FIXED BC

- MODEL THREE - RELAXED BC
TIME = 0.01554 SEC

- MODEL THREE - FIXED BC

- MODEL THREE - RELAXED BC
TIME = 0.01783 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
TIME = 0.01897 SEC

- MODEL THREE - FIXED BC
- MODEL THREE - RELAXED BC
REFERENCES

