Material characterization and parametric studies of explosion-proof vessel

Jagadeep Thota
University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation
https://digitalscholarship.unlv.edu/rtds/2008

This Thesis is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Retrospective Theses & Dissertations by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
MATERIAL CHARACTERIZATION AND PARAMETRIC STUDIES OF
EXPLOSION-PROOF VESSEL

by

Jagadeep Thota

Bachelor of Engineering in Mechanical Engineering
Bangalore Institute of Technology, Bangalore University, India
2001

A thesis submitted in partial fulfillment
of the requirements for the

Master of Science Degree in Mechanical Engineering
Department of Mechanical Engineering
Howard R. Hughes College of Engineering

Graduate College
University of Nevada, Las Vegas
August 2006
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.
The Thesis prepared by

JAGADEEP THOTA

Entitled

MATERIAL CHARACTERIZATION AND PARAMETRIC STUDIES OF
EXPLOSION-PROOF VESSEL

is approved in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

Examination Committee Chair

Dean of the Graduate College

Examination Committee Member

Examination Committee Member

Graduate College Faculty Representative
ABSTRACT

Material Characterization and Parametric Studies of Explosion-Proof Vessel

by

Jagadeep Thota

Dr. Mohamed B. Trabia, Examination Committee Chair
Professor and Chairperson of Mechanical Engineering
University of Nevada, Las Vegas

And

Dr. Brendan J. O'Toole, Examination Committee Chair
Associate Professor of Mechanical Engineering
University of Nevada, Las Vegas

Light-weight explosion-proof vessels can be an important tool for the temporary storage and transportation of explosive materials. Design, analysis and testing of these types of vessels present many technical challenges. Several composite vessel designs have been developed by the Russian Nuclear Federal Center (RFNC-VNIIEF). This thesis characterizes the materials used in the manufacture of the explosion-proof vessel. The objective of the research also involves the determination of an efficient analysis procedure for the explosion-proof vessel that provides a combination of accuracy, computational speed and modeling simplicity in carrying out a parametric analysis. The explosion-proof vessel is finally optimized to reduce the peak strains produced during simulation. The results obtained in this thesis are compared with the experimental and computational results provided by RFNC.
TABLE OF CONTENTS

ABSTRACT ............................................................................................................................ iii

TABLE OF CONTENTS........................................................................................................ iv

LIST OF FIGURES.............................................................................................................. viii

LIST OF TABLES.................................................................................................................. xii

ACKNOWLEDGEMENTS................................................................................................... xiv

CHAPTER 1  INTRODUCTION .............................................................................................. 1
1.1 Explosion or Blast ............................................................................................................. 1
1.2 Blast Wave ....................................................................................................................... 2
1.3 Confined Explosion ........................................................................................................... 5
   1.3.1 Shock Pressure ............................................................................................................. 5
   1.3.2 Gas Pressure ................................................................................................................. 6
1.4 Conwep (Conventional Weapons – An Air Blast Function) ........................................... 7
1.5 Blast Scaling ..................................................................................................................... 8
1.6 Materials Used in Explosion-Proof Vessels ................................................................... 9
1.7 An Overview of Recent Research Activities in Internal Blast Loading of Explosion-
   Proof Vessels ..................................................................................................................... 10
1.8 Objective ............................................................................................................................. 18

CHAPTER 2 MATERIAL CHARACTERIZATION .......................................................... 19
2.1 Material Testing System (MTS) ...................................................................................... 22
   2.1.1 Load Frame ................................................................................................................. 23
   2.1.2 Computer and Software ............................................................................................ 24
   2.1.3 Load Unit Control Panel (Pod) ................................................................................. 25
   2.1.4 Hydraulic Grip Supply ............................................................................................... 26
   2.1.5 Strain Gage Conditioning System ............................................................................ 26
   2.1.6 MTS Control Unit ...................................................................................................... 27
   2.1.7 Hydraulic Service Manifold ..................................................................................... 27
   2.1.8 Hydraulic Pump ......................................................................................................... 27
2.2 Strain Gages ....................................................................................................................... 27
2.3 Displacement Measuring Devices .................................................................................. 28
2.4 Material Characterization of Anti-Fragment Shield (Steel Wire Mesh) ....................... 28
   2.4.1 Samples Provided by RFNC ..................................................................................... 29
   2.4.2 Test Specimen ............................................................................................................. 29
LIST OF FIGURES

Figure 1. 1: Blast wave or blast pressure profile [9] .................................................................4
Figure 1. 2: A typical shock and gas loading during confined explosion [7] .........................7
Figure 1. 3: Romero's [15] explosion-proof vessel model in MESA-2D .............................12
Figure 1. 4: Predicting structural responses of Romero's [15] model using DYNA-3D 12
Figure 1. 5: Stevens's [19] spherical explosion-proof vessel .................................................14
Figure 1. 6: AT595 Explosion-Proof Vessel [1] .................................................................16
Figure 1. 7: DRAKON-R/2D code predicting the pressure distribution in the AT595 vessel [1] .................................................................17
Figure 2. 1: AT595 explosion-proof vessel .................................................................19
Figure 2. 2: Material Testing System (MTS) .................................................................22
Figure 2. 3: Scheme showing the MTS components ...........................................................23
Figure 2. 4: Scheme showing the load frame of the MTS ....................................................24
Figure 2. 5: Scheme of the POD and the functions of the controls ......................................25
Figure 2. 6: Grips and load platens used for compression tests ..........................................26
Figure 2. 7: Schematic view of the 90° tee rosette strain gage ...........................................28
Figure 2. 8: Anti-fragment shield (steel wire mesh) material provided by RFNC ..........29
Figure 2. 9: Different types of SWM specimens .............................................................30
Figure 2. 10: 30 Stacked SWM samples so as to form 2.54 cm thick specimen ..............30
Figure 2. 11: Scheme showing the application of load on the SWM specimens .............31
Figure 2. 12: Strain rate plot for the SWM specimens .....................................................32
Figure 2. 13: Determination of SWM material properties from the stress-strain plot ....33
Figure 2. 14: Typical engineering stress-strain curve for 2.5 cm SWM specimen 35
Figure 2. 15: Typical engineering stress-strain curve for 5.1 cm SWM specimen ........35
Figure 2. 16: Typical engineering stress-strain curve for 7.4 cm SWM specimen ..........36
Figure 2. 17: Comparison of stress-strain curves for different types of SWM specimen .36
Figure 2. 18: Crushed SWM specimen .............................................................................37
Figure 2. 19: Polymer-foam panel provided by RFNC .....................................................38
Figure 2. 20: Two different types of polymer-foam specimens ........................................39
Figure 2. 21: Scheme showing the application of load on the polymer-foam specimen 40
Figure 2. 22: Strain rate for the polymer-foam specimen ...............................................41
Figure 2. 23: Material properties obtained from the stress-strain plot for polymer-foam 42
Figure 2. 24: Typical engineering stress-strain curve for 2.5 cm polymer-foam specimen from panel 1 .................................................................43
Figure 2. 25: Typical engineering stress-strain curve for 5.1 cm polymer-foam specimen from panel 1 .................................................................45
Figure 2. 26: Typical engineering stress-strain curve for 2.5 cm polymer-foam specimen from panel 2 .................................................................46
Figure 2. 27: Typical engineering stress-strain curve for 5.1 cm polymer-foam specimen from panel 2 .................................................................47
Figure 3.12: FE Model 2 of the AT595 explosion-proof vessel .........................................97
Figure 3.13: Detailed view of the end-cap portion of Model 2 ..........................................97
Figure 3.14: Meshed cap portion of the vessel ....................................................................99
Figure 3.15: End-cap meshed with solid elements ..............................................................99
Figure 3.16: FE Model 3 of the AT595 explosion-proof vessel .......................................100
Figure 3.17: FE Model 4 of the AT595 explosion-proof vessel .......................................102
Figure 3.18: FE Model 5 of the AT595 explosion-proof vessel .......................................104
Figure 3.19: Solid band of elements depicting the gusset plate in Model 5 ......................105
Figure 3.20: Stress-strain curve for a plastic-kinematic material ..................................107
Figure 3.21: Element having orthotropic material property [8] .......................................108
Figure 3.22: Model 4 with the symmetry boundary conditions .......................................111
Figure 3.23: Model 4 with the pressure loading condition ................................................115
Figure 3.24: Comparison of peak circumferential strains at different locations .............124
Figure 3.25: The circumferential strain contour in Model 1 .............................................126
Figure 3.26: Circumferential strain of Model 1 and DRAKON at the central region .........127
Figure 3.27: The circumferential strain contour in Model 2 .............................................128
Figure 3.28: Circumferential strain of Model 2 and DRAKON at the central region .........129
Figure 3.29: The circumferential strain contour in Model 3 .............................................130
Figure 3.30: Circumferential strain of Model 3 and DRAKON at the central region .........131
Figure 3.31: The circumferential strain contour in Model 4 .............................................132
Figure 3.32: Circumferential strain of Model 4 and DRAKON at the central region .........133
Figure 3.33: The circumferential strain contour in Model 4 .............................................134
Figure 3.34: Circumferential strain of Model 4 and DRAKON at the central region .........135
Figure 4.1: FE model, Model 5, of the AT595 explosion-proof vessel ..............................138
Figure 4.2: Comparison of peak circumferential strain at different locations on the cylindrical portion of the vessel .................................................................139
Figure 4.3: Element in the central region from which the results are outputted ..............139
Figure 4.4: AT595 explosion-proof vessel depicting the inert explosive casing .............142
Figure 4.5: Fixed portion of the vessel obtained from the LS-DYNA file of Model 4 ......144
Figure 4.6: Variable portion of the vessel generated from the MATLAB optimization code .................................................................145
Figure 4.7: Increment/Decrement of the angle of the cap due to the variable $X_1$ .........146
Figure 4.8: The first node created for the -33° basalt-plastic layer from the optimization code ................................................................................................................147
Figure 4.9: The first row of nodes created in the cylindrical portion of the vessel .........148
Figure 4.10: The first row nodes created in the cap portion of the vessel .......................149
Figure 4.11: Angle of the segment formed by each element in the cap portion .............149
Figure 4.12: The rest of the nodes created for the cylindrical portion of the vessel ......151
Figure 4.13: The rest of the nodes created for the cap portion of the vessel .................152
Figure 4.14: Order of nodes in an element .......................................................................153
Figure 4.15: The first row of elements created ..............................................................154
Figure 4.16: Rest of the elements created to obtain the -33° basalt-plastic layer ..........155
Figure 4.17: The +33° basalt-plastic layer created from the MATLAB optimization code ................................................................................................................156
Figure 4.18: The 90° basalt-plastic layer created from the MATLAB optimization code ................................................................................................................156
Figure 4.19: The gusset and throttle plates obtained from the MATLAB database file 161
Figure 4.20: Flowchart representing the optimization process ........................................164
Figure 4.21: Plot depicting the reduction of peak circumferential strain after optimization
........................................................................................................................................168
Figure 4.22: Comparing peak strain contours for Case 4 at the initial guess and final
result ...................................................................................................................................169
Figure 5.1: Comparison of stress-strain plots for different SWM specimens ..........172
Figure 5.2: Typical engineering stress-strain plot for the ploymer-foam material......173
Figure 5.3: Typical stress-strain plot for stainless steel ..................................................175
Figure 5.4: AT595 explosion-proof vessel .................................................................177
Figure 5.5: FE model, Model 4 of the AT595 explosion-proof vessel .................179
Figure 5.6: Plot depicting the reduction of peak circumferential strain after optimization
........................................................................................................................................182
LIST OF TABLES

| Table 2.1: | Summary of tests done and properties computed | ............................................21 |
| Table 2.2: | Load frame specifications of the MTS | .............................................................23 |
| Table 2.3: | Strain rates used during the compression test of SWM specimen | ...........................................33 |
| Table 2.4: | Results of SWM specimens | .............................................................34 |
| Table 2.5: | Densities of the polymer-foam panels provided by RFNC | ...........................................38 |
| Table 2.6: | Results of 2.5 cm polymer-foam specimens from panel 1 | ...........................................42 |
| Table 2.7: | Results of 5.1 cm polymer-foam specimens from panel 1 | ...........................................44 |
| Table 2.8: | Results of 2.5 cm polymer-foam specimens from panel 2 | ...........................................45 |
| Table 2.9: | Results of 5.1 cm polymer-foam specimens from panel 2 | ...........................................46 |
| Table 2.10: | Results of stainless steel specimens from plate 1 | ...........................................55 |
| Table 2.11: | Results of stainless steel specimens from plate 2 | ...........................................56 |
| Table 2.12: | Types of basalt-plastic panels provided by RFNC | ...........................................58 |
| Table 2.13: | Results of transverse specimens from panel 1 | ...........................................64 |
| Table 2.14: | Results of transverse specimens from panel 6 | ...........................................65 |
| Table 2.15: | Results of axial specimens from panel 7 | ...........................................67 |
| Table 2.16: | Results of ±45° specimens from panel 4 | ...........................................70 |
| Table 2.17: | Summary of results for polymer-foam specimens | ...........................................72 |
| Table 2.18: | Summary of stainless steel results | .............................................................74 |
| Table 2.19: | Different types of steel materials used in AT595 explosion-proof vessel | ...........................................74 |
| Table 2.20: | Summary of basalt-plastic results | .............................................................75 |
| Table 2.21: | Basalt-plastic values provided by RFNC | ...........................................75 |
| Table 3.1: | Basic dimensions of the AT595 explosion-proof vessel | ...........................................78 |
| Table 3.2: | The function of the basic parts of AT595 explosion-proof vessel | ...........................................79 |
| Table 3.3: | The basic units used in LS-DYNA | .............................................................80 |
| Table 3.4: | The derived units of LS-DYNA | .............................................................81 |
| Table 3.5: | Basalt-plastic material properties from RFNC | ...........................................82 |
| Table 3.6: | Polymer-foam material properties from RFNC | ...........................................82 |
| Table 3.7: | Steel material properties from RFNC | .............................................................84 |
| Table 3.8: | Geometric characteristic of composite layer in the cylindrical portion | ...........................................85 |
| Table 3.9: | Geometric characteristic of the composite material in the cap portion | ...........................................85 |
| Table 3.10: | 90° basalt-plastic material properties | .............................................................86 |
| Table 3.11: | ±33° basalt-plastic material properties | .............................................................86 |
| Table 3.12: | Material properties of the composite shell layer in Model 1 and Model 2 | ...........................................92 |
| Table 3.13: | Characteristic of the individual material layers in Model 3 | ...........................................101 |
| Table 3.14: | Symmetry boundary conditions along the three planes | ...........................................112 |
| Table 3.15: | Characteristics of the contact cards used | .............................................................118 |
| Table 3.16: | Peak strain results of the FE models and RFNC | ...........................................123 |
| Table 3.17: | Error measure of the FE models with respect to experimental results | ...........................................125 |
| Table 4.1: | Coordinates of the first node | .............................................................147 |
Table 4.2: Coordinates of the first row of nodes in the cylindrical portion ..........148
Table 4.3: Coordinates of the first row of nodes in the cap portion ..................150
Table 4.4: Coordinates of the reminder nodes on the cylindrical portion ..........151
Table 4.5: Coordinates of the reminder nodes in the cap portion ....................152
Table 4.6: Identification of nodes surrounding an element for the first row ........153
Table 4.7: Identification of nodes surrounding an element for rest of the layer ....154
Table 4.8: Coordinates of the first node ......................................................156
Table 4.9: Coordinates of the first row of nodes in the cylindrical portion ........157
Table 4.10: Coordinates of the first row of nodes in the cap portion ...............157
Table 4.11: Coordinates for the remainder of the cylindrical portion ..............158
Table 4.12: Coordinates of the remainder of cap portion .............................158
Table 4.13: Coordinates of the first node ......................................................159
Table 4.14: Coordinates of the first row of nodes .........................................159
Table 4.15: Coordinates of the reminder nodes in the layer ............................159
Table 4.16: Results of peak strain obtained by varying just the variable $X_3$ ......165
Table 4.17: Initial guesses on variables $X_1$ and $X_2$ .................................166
Table 4.18: Results obtained after optimization of the initial guesses ..............167
Table 5.1: Summary of tests conducted and properties computed ...................171
Table 5.2: Summary of polymer-foam results ..............................................174
Table 5.3: Summary of results for stainless steel .........................................176
Table 5.4: Summary of results for basalt-plastic .........................................176
Table 5.5: Parts to be modeled and their features .........................................178
Table 5.6: Results of the FE analysis of explosion-proof vessel models ............179
Table 5.7: Initial guesses on variables $X_1$ and $X_2$ ....................................180
Table 5.8: Results obtained after optimization of the initial guesses ...............181

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
ACKNOWLEDGEMENTS

I express my sincere gratitude to my thesis advisors Dr. Mohamed B. Trabia and Dr. Brendan J. O’Toole for their invaluable guidance and suggestions throughout the entire course of this research project.

I would like to thank Dr. Ajit K Roy, Dr. Samaan G. Ladkany and Dr. Georg F. Mauer for their time in reviewing the prospectus, participation of defense, and counseling of the thesis as the committee members.

I would also like to express my heartiest gratitude to my colleagues, friends and family for their unrelenting support and motivation throughout this research activity.
CHAPTER 1

INTRODUCTION

Light-weight explosion-proof vessels can be an important tool for the temporary storage of explosive materials. They could be also used in emergency situations for containment of explosives in public places or for planned detonation of explosive materials. Bomb disposal units can use these vessels to transport small explosive devices away from populated areas for safe detonation in remote locations. Detonating big explosive experiments in large-scale containment vessels is becoming a popular alternative to open-air activities. Many test facilities across the country require that large debris and particulate matter be captured. Testing in the containment vessel offers several operational and safety advantages, as well as modest environmental enhancements, compared to open-air testing. These requirements drive the need for designing light-weight explosion-proof vessels that can be easily transported.

1.1 Explosion or Blast

An explosion is described as a physical or a chemical change in the material with the creation of blast wave and a powerful sound. Explosions occur under sudden change of stored potential energy into mechanical work [1]. Buri et al. [2] further added that explosion generally produces vast destruction because of associated shock and blast
waves. An explosion is a rapid release of energy in atmosphere over an extremely short
time and is distinguished by the formation of shock wave subsequently resulting into a
blast wave. The blast wave propagates in media and interacts with structures producing
large deformation.

1.2 Blast Wave

Sudden release of energy in atmosphere at a rate higher than the speed of sound
results in the formation of a system of blast wave. The strength of blast waves depends on
the amount and the rapidity of the energy release. The blast wave attenuates in strength as
it propagates outwards from its point of formation and produces large deformation.

Blast waves are fully described by the time of arrival, peak-over pressure, positive
phase duration, positive phase impulse and negative phase impulse. The air shock
produced due to an explosion causes a wave of almost instantaneous increase in pressure
above the atmospheric pressure, this is known as over pressure. Over pressure is the
difference between the peak pressure of the blast wave and the ambient pressure. Positive
duration is the time in which the over pressure decays to the ambient pressure. The
positive phase impulse is the measure of the energy carried by the wave as shown in
Figure 1.1. The negative phase is formed by the inertia of the media and has a longer
duration than positive phase. For low yield conventional explosions, the damaging effects
of negative phase are negligible [2,3].

In order to study the effects of the blast wave on a structure, it is necessary to know
the properties of the blast, such as density, wind velocity, shock front velocity, peak
pressure and dynamic pressure. Blast pressures are applied as time varying blast profiles of atmosphere overpressure loading on a model.

The entire time history of the blast overpressure at a point can be calculated using the Friedlander's Equation, which is given as [4],

\[
P(T) = P_o + P_s \left( 1 - \frac{T - T_a}{T_d} \right) e^{-\alpha \left( \frac{T - T_a}{T_d} \right)}
\]

(1.1)

where,

- \( P_o \) = Ambient pressure
- \( P_s \) = Peak pressure
- \( T_a \) = Arrival Time
- \( T_d \) = Positive duration
- \( \alpha \) = Decay coefficient
- \( T \) = Time measured from the instant the shock front arrives
The modified Friedlander equation is given as [5],

\[ P(T) = P_o \left( 1 - \frac{T}{T_a} \right) e^{-c \left( \frac{T}{T_a} \right)} \]  

(1.2)

It has been found that the effect of a blast wave on a structure is greatly influenced by the ratio of the positive duration of the air shock and the natural period of vibration of the element of the structure under consideration. If this ratio is less than 0.1, then the specific impulse, the time integral of pressure, is the dominant characteristic of the load, and the loaded area of the structure acquires a velocity in an extremely small displacement. If this ratio is greater than about six, it is the peak overpressure that dominates the structural response [6].
1.3 Confined Explosion

Confined and contained explosions that occur within structures normally develop complicated pressure-time histories on the inside surfaces. Such loading cannot be predicted exactly, but approximations and model relationships exist to define blast loads with a good confidence. The loading from a high-explosive detonation within a confined vessel consists of two almost distinct phases. The first phase is the reflected blast loading which typically consists of an initial high-pressure, short-duration, reflected wave plus several later reflected pulses. The second is called the gas loading phase [7].

1.3.1 Shock Pressure

Incident and reflected shocks inside structures consist of the initial high-pressure, short-duration reflected wave, plus several later reflected shocks, which are a result of reverberation of the initial shock within the structure. These later pulses are usually attenuated in amplitude because of an irreversible thermodynamic process. These are complicated in waveforms because of the involved reflection process within the structure. The simplest case of blast wave reflection is that of normal reflection of a plane shock wave from a plane, rigid surface. In this case, the incident wave moves at velocity $U$ through still air at ambient conditions. The conditions immediately behind the shock front are those for the free-air shock wave. When the incident shock wave strikes the plane, rigid surface, it is reflected and moves away from the surface with a velocity $U_r$ into the flow field and compressed region associated with the incident wave. In the reflection process, the incident particle velocity $U_s$ is arrested ($U_s = 0$ at the reflecting surface), and the pressure, density, and temperature of the reflected wave are all increased above the
values in the incident wave. The overpressure at the wall surface is termed the normally reflected overpressure and is designated $P_r$.

Following the initial internal blast loading, the shock waves reflected inward will usually strengthen as they implode toward the center of the structure, and then attenuate as they move through the air and re-reflect to load the structure again. The second shocks will usually be somewhat less in strength than the initial pulse, and after several such reflections, the shock wave phase of the loading will be over [7].

1.3.2 Gas Pressure

When an explosion from a high-explosive source occurs within a structure, the blast wave reflects from the inner surfaces of the structure, implodes toward the center, and rereflects one or more times. The amplitude of the re-reflected waves usually decays with each reflection, and eventually the pressure settles to what is termed the gas pressure-loading realm. When considering poorly vented or unvented chambers, the gas load duration can be much longer than the response time of the structure, appearing nearly static over the time to maximum response. Under this condition, the gas load is often referred to as a quasi-static load. When considering vented chambers, the gas pressure drops quickly in time as a function of room volume, vent area, mass of vent panels, and energy release of the explosion, and depending on the response time of structural elements under consideration, may not be considered quasi-static.

The gas load starts at time zero and overlaps the shock load phase without adding to the shock load, as illustrated in Figure 1.2, where the shock phase and the gas phase are idealized as such as that shown and which should be used in design. They intersect at the load time pair $(P_r, T_r)$ to form the bilinear load history, such as that shown and that should
be used in design. Since the shock and gas loading are parts of the entire load history, although they are calculated separately, they should not be considered separately in design or analysis. Various procedures are available for predicting the peak gas pressure in a structure (e.g., ConWep, Hyde 1993) [7].

Figure 1.2: A typical shock and gas loading during confined explosion [7]

1.4 ConWep (Conventional Weapons — An Air Blast Function)

ConWep blast function is used to apply simple blast loading rather than to explicitly simulate the shock wave from the high explosive. The LOAD_BLAST [8] boundary condition in LS-DYNA is based on an implementation by Randers-Pehrson and Bannister in 1997 of the empirical load blast functions implemented in the ConWep code. Kingery and Bulmash in 1984 wrote the ConWep code. The Blast functions can be used for two cases, the free air detonation of a spherical charge and surface detonation of a hemispherical charge. While the surface detonation approaches the conditions of a mine blast, anti-vehicular mines are most commonly buried anywhere from 5 to 20 cm
(sometimes more if a road is resurfaced for example) below the surface of the soil. The depth of burial, among other things, has a significant effect on the energy directed on the target by funneling the force of the blast upwards. Other variables such as soil, moisture content and soil type have an equally important effect on the mine but none of these effects are included in the ConWep blast model. The only variable available is the mass of the explosive [9]. ConWep model accounts for the angle of incidence of the blast wave, but does not account for the shadowing or confinement effect. In reality when front of blast pressure hits an object, it bounds back generating secondary pressure; however, ConWep does not however account for the secondary pressure.

1.5 Blast Scaling

Two different weight TNT explosives will generate the same overpressure, but they will do so at a different distance from the explosive center. For a target to experience the same overpressure with a smaller explosive, the target will need to be much closer to the bomb than with a more massive explosive. This is the basic idea behind explosive scaling put forth by Neff [10]. Since the same overpressure will be generated by different weight explosives, the weight of the bomb can be combined with distance from the explosive to create a scaled distance parameter. The scaled distance \( Z \) used for this purpose is defined as follows,

\[
Z = \frac{R}{W^{3/2}}
\]

(1.2)

where \( R \) is the radius from the center of the explosion given in meters and \( W \) is the equivalent weight of TNT given in kilograms. This method of blast scaling was proposed by Sachs in 1944 [11]
As well as the above scaling, it is possible to scale between different explosives to see where an equivalent impact will be delivered. Hopkinson formulated this method in 1915. For example, if a given overpressure is felt at radius $R_1$ for an explosive with TNT equivalent mass $W_1$, a second explosive with equivalent mass $W_2$ will generate the same overpressure at radius $R_2$ as given by the following equation [10],

$$\frac{R_1}{R_2} = \left[\frac{W_1}{W_2}\right]^{\frac{1}{3}}$$

(Coggin et al. [11] studied the aforementioned methods of blast scaling and inferred that Sachs scaling produced excellent scaled results compared to Hopkinson scaling. They also proposed that Hopkinson scaling is a special case of Sachs scaling.

1.6 Materials used in Explosion-Proof Vessels

The resistance of vessels to blast loading can be improved by selecting appropriate materials for the vessels. Composite materials are the most common materials used for the explosion-proof vessels because of their specific strength and stiffness, their tolerance to damage due to their inherent ability to resist catastrophic failure and formation of small light weight debris instead of large secondary projectiles that might occur in the metallic case [12]. Some of the materials which have been used in the explosion-proof vessels are Woven roven S-2 glass reinforced polyester composites, Hard steel alloy (RHA), Aluminum alloy 5083, Titanium alloy Ti-6Al-4V, Basalt-Plastic, etc [1,12].

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
1.7 An Overview of Recent Research Activities in Internal Blast Loading of Explosion-Proof Vessels

Baker [13] surveyed research approaches to the dynamic response of a pressure vessel to severe loading conditions such as external blast effects, internal blast effects, effects of nuclear reactor core melt accidents, and missile impact effects. He highlighted deficiencies of various approaches used to predict complex loads and dynamic responses.

Oswald et al. [14] designed an explosion containment vessel using a simplified dynamic analysis procedure based on single degree-of-freedom (DOF) and three DOF "equivalent" systems. These simplified systems were used to calculate the elastic deflections of complex vessel structural components at critical locations caused by blast pressures from an internal explosion. A static finite element analysis was used to determine the shape function of the most complex vessel structural components and to relate the peak stress in the component to the peak deflection determined in the simplified dynamic analysis.

Martineau et al. [15] investigated the response of a closed 2:1 elliptical ended stainless steel cylindrical vessel with asymmetric internal blast loading by experiments, computations and analysis. The high explosive charge was located on the vessel's axis of symmetry but was axially offset from its geometric center. Two-dimensional, Eulerian finite difference calculations from a hydrodynamic code, MESA-2D, was used to study the blast phenomenon and pressure loading on the vessel walls. The calculated two-dimensional loading was applied to a three-dimensional explicit finite element model, DYNA-3D, to predict the structural response of the vessel. DYNA-3D is a non-linear,
explicit Lagrangian finite element analysis tool with excellent structural transient analysis capabilities [16].

In 1997 Los Alamos National Laboratory was designing a large containment vessel to elastically withstand a 50 kg internal high explosive detonation. Romero et al. [16] fabricated a one-tenth scaled model of the containment vessel and used it to obtain experimental results of both pressure loading and strain response. They further developed the work of Martineau et al. [15]. Romero’s FE model comprised of 124,474 elements and 150,891 nodes. The numerical predictions of the pressure loading and strain response were obtained from a two-dimensional Eulerian hydrodynamic code, MESA-2D, and an explicit, non-linear finite element code, DYNA-3D. The approximate simulation time was 27 hours and the machine used to do the computation was a single 195 MHz R10000 processor. The initial pressure peaks predicted analytically compared reasonably well, with the average deviation being 20% from the experimental results. However the reverberated pressure peaks and duration did not correlate as well. This was attributed to the two-dimensional versus three-dimensional differences between the analytical and the physical model. The predicted strain histories show good correlation and are within 20% to the experimental data in both magnitude and frequency content [16].
Whenhui et al. [17] studied the structural dynamic behavior of cylindrical explosive vessels in response to internal blast loading produced by a concentrated explosive charges. Special attention was given to the formation and development of dynamic strain growth and the factors that affect it when the vessels are subjected to dynamic loading. This work included theoretical treatment of the strain growth and strain growth factors. Duffey et al. [18] further pointed out that the superposition and interaction of different
modes of response with similar frequencies is the mechanism of strain growth. Whenhui et al. [17] concluded that the geometry of the vessel plays a dominant factor in controlling the degree of strain growth. The following assumptions were made in their analysis,

- Symmetry can be ensured during response by locating the charges at the geometrical centre of the chambers.
- Detonation of the charges is transient.
- The connection of structural elements is continuous and perfect.
- Plastic deformation is not permitted during the operation of the chambers.

Stevens et al. [19] performed a series of hydrodynamic and structural analyses of a spherical confinement vessel to understand the dynamic response of the vessel when subjected to internal blast loadings. The transient pressures acting on the inner surfaces of the vessel were computed using an Eulerian hydrodynamics code MESA-2D, and CTH: which simulated the burn of the high explosive and the internal gas dynamics. The vessel’s structural response to these pressures was then analyzed using an explicit finite element structural dynamics code, PARADYN. PARADYN is a massively parallel version of DYNA-3D. PARADYN was chosen because of DYNA-3D’s reputation and history for handling very large models by effectively using the available computational tools.
Mahmadi et al. [20] modeled the airblast wave for a C4-explosion by using an explicit finite element code LS-DYNA3D. The hydrocode LS-DYNA is used to simulate large deformation material responses and dynamic processes and to solve the continuum equations for nonlinear responses of materials and structures. In the detonating modeling case, the fluid mesh undergoes large deformations. As the LS-DYNA3D scheme is explicit, the mesh can become distorted. Then, the rezoning of Arbitrary Lagrangian-Eulerian (ALE) method is used to create a new undistorted mesh for the fluid domain.
They modeled the fluid by an Eulerian mesh based upon the Eulerian multi-material method.

ALE [21] formulation can be thought of as algorithms that perform automatic rezoning. Users perform manual rezoning by stopping the calculation when the mesh is distorted, smoothing the mesh or remapping the solution from the distorted mesh to the smooth mesh. An ALE formulation consists of a time step followed by a remap or advection step. The advection step performs an incremental rezone, where incremental refers to the fact that the positions of the nodes are moved only a small fraction of the characteristic lengths of the surrounding elements. The accuracy of an ALE calculation is often superior to the accuracy of a manually rezoned calculation. An ALE formulation contains the Eulerain formulation as a subset. Eulerain codes can have more than one material in each element, but most ALE implementations are simplified ALE formulations, which permit only a single material in each element. The primary advantage of a simplified formulation is its reduced cost per time step. The overall flow of an ALE time step is,

- Perform a Lagrangian time step.
- Perform an advection step.
- Decide which nodes to move.
- Move the boundary nodes.
- Move the interior nodes.
- Calculate the transport of the element-centered variables.
- Calculate the momentum transport and update the velocity.
The Russian Federal Nuclear Center (RFNC-VNIIEF) has designed and developed several composite explosion-proof vessels. Figure 1.6 shows the AT595 explosion-proof vessel [1], built by RFNC, which mainly consists of a central cylindrical portion with two hemispherical end caps on either end. This AT595 explosion-proof vessel is taken as the base model for all the finite element models in this thesis.

RFNC determined the mechanical properties for several of the materials used in the AT595 explosion-proof vessel including basalt-plastic, polymer-foam, stainless steel and steel wire mesh material [22,23]. They also conducted experimental and numerical simulations of internal blast loading in open and closed vessels. They used a two-dimensional axisymmetric analysis code known as DRAKON-R/2D [1,22] to model the two-dimensional problems of non-stationary flows of continuous medium including the effects of shock waves. DRAKON-R/2D program divides the simulation into two stages: first stage includes modeling of the gas-dynamics associated with the explosion while the second stage calculates the structural response of the vessel. The code used to perform
the first stage in known as B-71 code whereas the DRAKON code is used for the second stage.

Figure 1.7: DRAKON-R/2D code predicting the pressure distribution in the AT595 vessel [1]

Matta [9] created various finite element (FE) models of the AT595 explosion-proof vessel starting from a simple shell model and moving towards a more geometrically accurate solid model. He conducted computational simulations on these FE models and predicted their structural responses by using the LS-DYNA code. He determined the effectiveness of the material models within LS-DYNA and also predicted the maximum deformations in the composite and steel layers. Finally he compared his results with the computational results obtained from RFNC.

Surveyed literature shows that the design, analysis, and testing of these types of containment vessels present many technical challenges. A standardized method of analysis and design is required to ensure confidence in the performance level of a particular design. Current ASME pressure vessel codes do not account for dynamic blast loading.
1.8 Objective

The main objectives of this thesis are:

- To characterize the materials used in AT595 explosion-proof vessel by conducting static tests. These materials include steel wire mesh, polymer-foam, stainless steel and basalt-plastic composite. The results obtained are compared with those provided by RFNC.
- To determine an efficient analysis procedure and create a FE explosion-proof vessel model that provides a combination of accuracy and computational speed for carrying out parametric analysis or design optimization.
- To optimize the FE model of the explosion-proof vessel in order to reduce the peak strains produced during simulation.
The AT595 explosion-proof vessel, which was designed by RFNC, is shown in Figure 2.1. The vessel is made up of filament wound basalt fiber/epoxy composite with an inner steel liner [1]. The basalt-plastic composite was fabricated by winding Rb9-1250-4S basalt fiber rovings, consisting of 9 mm diameter fibers in a 3πT-10 epoxy binder. Polymer-foam has been placed on either end of the vessel. Anti-fragment shield, which is made up of layers of steel wire mesh material, is located below the cylindrical steel liner. The vessel is stiffened by two throttle plates located at either end of the anti-fragment shield. Throttle plates are radially supported by the stainless steel gusset plates.

Figure 2. 1: AT595 explosion-proof vessel
Material characteristics of the AT595 explosion-proof vessel components should be known to define the material models in the computational analysis code. RFNC conducted experiments on the aforementioned materials. The material characteristics obtained from these experiments were used in the DRAKON-R/2D code to simulate the effect of internal blast loading in the AT595 explosion-proof vessel. Samples of basalt-plastic, polymer-foam, stainless steel, and steel wire mesh materials were given to University of Nevada, Las Vegas (UNLV) for conducting independent material characterization experiments. Static tests were done on the material samples given, and the behavior of these materials were studied and the mechanical properties determined. Compression tests were carried out on polymer-foam and steel wire mesh specimens whereas on basalt-plastic and stainless steel specimens’ tension tests were done. Table 2.1 lists the summary of static tests conducted on these materials and the properties computed.
Table 2. 1: Summary of tests done and properties computed

<table>
<thead>
<tr>
<th>Material</th>
<th>Test</th>
<th>Objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>Anti-Fragment shield</td>
<td>Compression</td>
<td>Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>(Steel wire mesh)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Polymer-Foam</td>
<td>Compression</td>
<td>Collapse Strength</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>Tension</td>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yield Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Ultimate Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fracture Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>Basalt-Plastic</td>
<td>Tension</td>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shear Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fracture Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Engineering Stress-Strain curve</td>
</tr>
</tbody>
</table>
2.1 Material Testing System (MTS)

All the static experiments were conducted on the MTS Axial/Torsional Material Test System shown in Figure 2.2. Figure 2.3 shows a scheme of the MTS layout with its primary components.
Components of the MTS Testing System

Figure 2.3: Scheme showing the MTS components

2.1.1 Load Frame

The load frame specifications of the MTS are listed below in Table 2.2. Figure 2.4 portrays a scheme of the load frame components in the MTS.

Table 2.2: Load frame specifications of the MTS

<table>
<thead>
<tr>
<th>Specification</th>
<th>Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Working Dimensions</td>
<td>635 mm wide by 1463 mm high</td>
</tr>
<tr>
<td>Axial Load Transducer Limit</td>
<td>250 KN</td>
</tr>
<tr>
<td>Torsional Load Transducer Limit</td>
<td>2200 N-m</td>
</tr>
<tr>
<td>Linear Actuator Limit</td>
<td>± 76 mm, measured with an LVDT</td>
</tr>
<tr>
<td>Rotary Actuator Limit</td>
<td>100° static rotation (± 50°), measured with ADT 90° dynamic rotation (± 45°)</td>
</tr>
</tbody>
</table>
2.1.2 Computer and Software

The controlling computer is a PC system running on Windows 2000 operating system. TestStar is the main program for operating the MTS, it provides menus for activating the hydraulics to move the actuators and grip specimen. It also displays information such as displacement, angle, force, torque, strains, etc. while conducting tests on the MTS. Even test limits, such as the load limit, can be defined using the TestStar software.

Figure 2.4: Scheme showing the load frame of the MTS
2.1.3 Load Unit Control Panel (POD)

The load unit control panel or the POD is the main interface with the system for manually controlling the MTS. Figure 2.5 shows a scheme of the POD and the briefly explains the control buttons on the it.

![Scheme of the POD and the functions of the controls](image)

- **Controllers**
  - Axial
  - Torsional

- **Sensors**
  - Load Cell
  - Length

- **Current Mode**
  - Length A SG
  - Angle A SG

- **LUCP Mode**
  - Force A Pod
  - Torque A Pod

- **Auto Zero/Control Mode**
  - Move cursor up and down in the display.
  - Left button Auto zeros the selected sensor in the display or selects a new control mode under LUCP.

- **Test Control**
  - Stop
  - Hold
  - Run/Resume

- **Interlocks**
  - Must be reset if lit.

- **Power Control for Main Pump**
  - Always go from Low to High and from High to Low.

- **Power Control for Service Manifold**
  - Always go from Low to High and from High to Low.

- **Actuator Positioning Control**
  - Clockwise moves the actuator down.

- **Rotary Actuator Control**
  - Clockwise rotates actuator to the right.

**Emergency Stop**

Display (2 windows) which are toggled on/off with the 'Next Pane' button.

The current control mode is displayed under current mode for each actuator. The next selected control mode is displayed under LUCP mode.

Display Controls: Middle buttons move cursor up and down in the display. Left button Auto zeros the selected sensor in the display or selects a new control mode under LUCP.

Test Controls: (Also available on Computer screen.

Interlocks: Must be reset if lit.

Power Control for Main Pump: Always go from Low to High and from High to Low.

Power Control for Service Manifold: Always go from Low to High and from High to Low.

Linear Actuator Control: Clockwise moves the actuator down.

Rotary Actuator Control: Clockwise rotates actuator to the right.

Figure 2.5: Scheme of the POD and the functions of the controls
2.1.4 Hydraulic Grip Supply

The hydraulic grip supply is used to apply gripping pressure to the specimen during a test. Gripping pressure is controlled by a combination of air and hydraulic fluid. The maximum gripping pressure of this supply unit is 45 MPa.

Figure 2.6: Grips and load platens used for compression tests

2.1.5 Strain Gage Conditioning System

The strain gage conditioning system is used to interpret resistance change across the strain gage and convert it into micro-strain. The conditioner system on the MTS can
monitor up to eight separate strain gages and report continuous strain readings to the MTS control program for data processing.

2.1.6 MTS Control Unit

This box houses all the electronic signal conditioners and controllers for the hydraulic components.

2.1.7 Hydraulic Service Manifold

This unit regulates the hydraulic supply to the actuators and grip supply unit.

2.1.8 Hydraulic Pump

This is the pump and reservoir for the hydraulic system.

2.2 Strain Gages

Strain gages were used on the tension test specimens of basalt-plastic and stainless steel to compute the poisson's ratio. The 90° tee rosette strain gage is used for obtaining the axial and transverse strains in the specimen during a test. From these obtained strains the poisson's ratio is determined. The 90° tee rosette strain gages used in the static experiments have a gage factor of 2.02 and can measure strain up to a resistance of 120 Ω.
2.3 Displacement Measuring Devices

For all the tests, displacement or change in dimensions of the specimen was measured using the linear variable differential transformer (LVDT) present in the MTS machine. The strain values in the Engineering Stress-Strain curves were formulated from the LVDT displacement values. For tension tests, strain gages were also used but to only determine the poisson’s ratio and elastic modulus.

2.4 Material Characterization of Anti-Fragment Shield (Steel Wire Mesh)

Anti-fragment shield is located in the cylindrical portions of the vessel below the steel liner and between the throttle plates. Anti-fragment shield is used to absorb and distribute the shock loads from the shrapnel of the explosive material in the AT595 explosion-proof vessel.
2.4.1 Samples Provided by RFNC

RFNC provided UNLV with 130 square SWM sheets. The sheets were approximately 33 cm in length and having a thickness of 0.85 mm. The mesh spacing was approximately 2.8 mm. Figure 2.8 depicts the SWM sheets.

Figure 2. 8: Anti-fragment shield (steel wire mesh) material provided by RFNC

2.4.2 Test Specimen

Tests were done on 2.5 cm, 5.1 cm and 7.4 cm square SWM samples respectively. Figure 2.9 shows the different sizes of the test specimen. Thirty SWM samples were stacked together, as shown in Figure 2.10, to obtain a thickness of 2.5 cm for all the tests. Paper-cutter was used to cut the 33 cm SWM sheets to the required specimen size. Paper-cutter is chosen because it was cost effective, easy to handle and importantly resulted in a good end product, i.e. specimen with the required dimensions.
Figure 2.9: Different types of SWM specimens

Figure 2.10: 30 Stacked SWM samples so as to form 2.54 cm thick specimen
2.4.3 Testing of Steel Wire Mesh

Compression tests were conducted on the SWM specimens using the MTS machine. Load was applied along the thickness direction of the specimen, on the square surface area as shown in Figure 2.11. The upper platen was fixed by the upper grips of the MTS and is not allowed to move in any direction. The SWM specimen was placed on the lower platen. The lower platen was held by the lower grips of the MTS and was free to move in the longitudinal direction. Compressive load was applied on the specimen by the movement of the lower platen.

![Scheme showing the application of load on the SWM specimens](image)

**Figure 2.11:** Scheme showing the application of load on the SWM specimens

Due to the limitation of the fixtures on the MTS, the maximum load was limited to 88 KN and also the distance between the platens was limited to 0.03 mm, i.e. the
compression test was aborted automatically once the distance between the platens came below 0.03 mm or if the load reached 88 KN. The strain values were measured using the LVDT present in the MTS and the laser equipment. The maximum strain value was taken from the LVDT data. The tests were not carried out with a constant strain rate; it was varied with respect to the distance between the load platens. Table 2.3 shows the different strain rates being used. The strain rate was varied in such a way so as to obtain more data points initially and at the end of the experiment, but at the same time not to make the experiment slow, the strain rate was increased in the middle portion of the test. Figure 2.12 shows the plot of strain versus time.

![Plot of strain versus time](image)

**Figure 2.12: Strain rate plot for the SWM specimens**

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 2. 3: Strain rates used during the compression test of SWM specimen

<table>
<thead>
<tr>
<th>Distance Between the Platens</th>
<th>Strain Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 25.40 mm to 12.70 mm</td>
<td>0.0250 mm/mm/second</td>
</tr>
<tr>
<td>From 12.70 mm to 06.35 mm</td>
<td>0.0500 mm/mm/second</td>
</tr>
<tr>
<td>From 06.35 mm to 02.54 mm</td>
<td>0.0010 mm/mm/second</td>
</tr>
<tr>
<td>From 02.54 mm to 00.03 mm</td>
<td>0.0005 mm/mm/second</td>
</tr>
</tbody>
</table>

2.4.4 Results and Comments

For all the tests, the experiment terminated as the load limit of 88 KN had been reached. Figure 2.13 shows a typical Engineering Stress-Strain curve obtained for the SWM specimen and the determination of maximum stress/strain value from the plot. Table 2.4 represents the results obtained for the SWM specimens. The maximum strain reported was measured with the help of the LVDT present in the MTS machine.

![Figure 2.13: Determination of SWM material properties from the stress-strain plot](image.png)
Table 2.4: Results of SWM specimens

<table>
<thead>
<tr>
<th>Specimen No.</th>
<th>Specimen Type</th>
<th>Maximum Stress MPa</th>
<th>Maximum Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.5 cm</td>
<td>137.87</td>
<td>0.7794</td>
</tr>
<tr>
<td>2</td>
<td>2.5 cm</td>
<td>137.15</td>
<td>0.7728</td>
</tr>
<tr>
<td>3</td>
<td>2.5 cm</td>
<td>138.00</td>
<td>0.7750</td>
</tr>
<tr>
<td>4</td>
<td>5.1 cm</td>
<td>34.76</td>
<td>0.6225</td>
</tr>
<tr>
<td>5</td>
<td>7.4 cm</td>
<td>16.39</td>
<td>0.5141</td>
</tr>
<tr>
<td>6</td>
<td>7.4 cm</td>
<td>16.38</td>
<td>0.5129</td>
</tr>
</tbody>
</table>

The maximum load applied on 2.5 cm and 7.4 cm specimens was same whereas the cross-section of the 2.5 cm specimen is less compared to the 7.4 cm. Hence the 2.5 cm specimen should be crushed to a larger extent when compared to the 7.4 cm specimen. From Table 2.4 it is seen that the 2.5 cm specimen is crushed to about 78% whereas the 7.4 cm specimen is crushed to a smaller extent of 51%. Small amount of slipping was noticed between the stacked SWM samples during the experiment. The Engineering Stress-Strain curves for all the samples are plotted as shown below,
Figure 2.14: Typical engineering stress-strain curve for 2.5 cm SWM specimen

Figure 2.15: Typical engineering stress-strain curve for 5.1 cm SWM specimen
Figure 2.16: Typical engineering stress-strain curve for 7.4 cm SWM specimen

Figure 2.17: Comparison of stress-strain curves for different types of SWM specimen
From Figure 2.17 it can be seen that all the three curves follow the same path, hence proving that the results obtained from compression test of SWM specimens to be consistent. Figure 2.18 shows the crushed SWM specimen after conducting the compression test.

![Crushed SWM specimen](image)

Figure 2.18: Crushed SWM specimen

2.5 Material Characterization of Polymer-Foam

Polymer-Foam is located at either ends of the AT595 explosion-proof vessel. The primary function of polymer-foam is to absorb the energy released by the blast wave.

2.5.1 Samples Provided by RFNC

RFNC provided UNLV with eleven square polymer-foam panels. The panels were approximately 30.5 cm in length and width, and the thickness of each panel was 5 cm. Even though all the panels were of same dimensions and material, the densities varied between 170 kg/m³ to 220 kg/m³. Table 2.5 lists the densities of all the panels provided by RFNC. Tests were conducted on polymer-foam specimens from panel 1 and panel 2.
Table 2. 5: Densities of the polymer-foam panels provided by RFNC

<table>
<thead>
<tr>
<th>Panel</th>
<th>Mass (kg)</th>
<th>Density (kg/m$^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.9936</td>
<td>222.17</td>
</tr>
<tr>
<td>2</td>
<td>0.8073</td>
<td>180.51</td>
</tr>
<tr>
<td>3</td>
<td>0.8515</td>
<td>190.40</td>
</tr>
<tr>
<td>4</td>
<td>0.8981</td>
<td>200.81</td>
</tr>
<tr>
<td>5</td>
<td>0.7922</td>
<td>177.14</td>
</tr>
<tr>
<td>6</td>
<td>0.8016</td>
<td>179.24</td>
</tr>
<tr>
<td>7</td>
<td>0.8963</td>
<td>200.42</td>
</tr>
<tr>
<td>8</td>
<td>0.9182</td>
<td>205.31</td>
</tr>
<tr>
<td>9</td>
<td>0.9002</td>
<td>201.29</td>
</tr>
<tr>
<td>10</td>
<td>0.8675</td>
<td>193.98</td>
</tr>
<tr>
<td>11</td>
<td>0.7940</td>
<td>177.54</td>
</tr>
</tbody>
</table>
2.5.2 Test Specimen

Tests were done on 2.5 cm and 5.1 cm square polymer-foam specimens. A thickness of 2.5 cm was maintained for all the specimens. Figure 2.20 shows the different sizes of the test specimen. The cutting tool used to cut the required polymer-foam specimens was band saw. Initially, coring tool was used to cut the polymer-foam specimens, but it was observed that after cutting a polymer-foam specimen, the surface of the specimen adjoining the coring tool surface melts and sticks on to the tool, making it impossible to remove the specimen without damaging the tool or the specimen. On the other hand, band saw proved to be a more appropriate choice for cutting the polymer-foam specimens.

![Figure 2.20: Two different types of polymer-foam specimens](image)

2.5.3 Testing of Polymer-Foam

Compression tests were conducted on the polymer-foam specimens using the MTS machine. Load was applied along the thickness direction of the specimen, on the square surface area as shown in Figure 2.21. The upper platen was fixed by the upper grips of the MTS and was not allowed to move in any direction. The polymer-foam specimen was
placed on the lower platen. The lower platen was held by the lower grips of the MTS and was free to move in the longitudinal direction. Compressive load was applied on the specimen by the movement of the lower platen.

![Scheme showing the application of load on the polymer-foam specimen](image)

**Figure 2.21:** Scheme showing the application of load on the polymer-foam specimen

Due to the limitation of the fixtures on the MTS, the maximum load was limited to 88 KN and also the minimum distance between the platens was limited to 0.03 mm, i.e. the compression test was aborted automatically once the distance between the platens came below 0.03 mm or if the load reached 88 KN. The strain values were measured using the LVDT present in the MTS and the laser equipment. Laser equipment was used to measure the modulus of the specimen whereas LVDT predicted the maximum strain. The strain rate used for crushing the specimens was 2.0E-3 mm/mm/second.
2.5.4 Results and Comments

For all the tests, the experiment terminated as the load limit of 88 KN had been reached. Figure 2.23 shows a typical Engineering Stress-Strain curve for the polymer-foam specimen and the material properties obtained from these plots.
Figure 2.23: Material properties obtained from the stress-strain plot for polymer-foam

Table 2.6: Results of 2.5 cm polymer-foam specimens from panel 1

<table>
<thead>
<tr>
<th>Specimen No.</th>
<th>Density Kg/m³</th>
<th>Crush Strength MPa</th>
<th>Elastic Modulus MPa</th>
<th>Maximum Stress MPa</th>
<th>Maximum Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>231.16</td>
<td>5.84</td>
<td>No Data</td>
<td>137.92</td>
<td>0.8474</td>
</tr>
<tr>
<td>1.3</td>
<td>237.38</td>
<td>5.84</td>
<td>No Data</td>
<td>138.01</td>
<td>0.8479</td>
</tr>
<tr>
<td>1.4</td>
<td>227.56</td>
<td>5.91</td>
<td>No Data</td>
<td>138.02</td>
<td>0.8524</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The results for 2.5 cm polymer foam specimens from panel 1 are tabulated in the Table 2.6. Figure 2.24 shows the Engineering Stress-Strain plot for the 2.5 cm specimen from panel 1. From the Table 2.6 and Figure 2.24 it can be concluded that the results are very consistent. The thicknesses of the 2.5 cm polymer-foam specimens were greater than the required 2.5 cm. MTS was programmed to start obtaining the test data when the distance between the platens were 2.5 cm. Hence from the plot in Figure 2.24 it can be seen that the initial test data could not be obtained due to the thickness of the specimen being greater than 2.5 cm. Therefore it was not possible to determine the elastic modulus of these specimens.

![Figure 2.24: Typical engineering stress-strain curve for 2.5 cm polymer-foam specimen from panel 1](image-url)
Table 2.7: Results of 5.1 cm polymer-foam specimens from panel 1

<table>
<thead>
<tr>
<th>Specimen No.</th>
<th>Density Kg/m³</th>
<th>Crush Strength MPa</th>
<th>Elastic Modulus MPa</th>
<th>Maximum Stress MPa</th>
<th>Maximum Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>211.30</td>
<td>4.34</td>
<td>91.90</td>
<td>34.51</td>
<td>0.8064</td>
</tr>
<tr>
<td>1.3</td>
<td>214.02</td>
<td>4.76</td>
<td>120.61</td>
<td>34.48</td>
<td>0.7974</td>
</tr>
<tr>
<td>1.4</td>
<td>239.35</td>
<td>5.21</td>
<td>119.94</td>
<td>34.45</td>
<td>0.7814</td>
</tr>
</tbody>
</table>

Table 2.7 depicts the results obtained from 5.1 cm polymer-foam specimens from panel 1. Figure 2.25, gives the Engineering Stress-Strain curve for the 5.1 cm specimens from panel 1. From the results it is observed that the maximum strain values of the specimens are very consistent but there is a slight variation in the crush strength of the specimens, and the elastic modulus of specimen 1.2 compared to the other specimen, which can be attributed to the significant difference in densities. The elastic modulus of specimen 1.2 is less when compared to the other specimens.
Figure 2.25: Typical engineering stress-strain curve for 5.1 cm polymer-foam specimen from panel 1

Table 2.8: Results of 2.5 cm polymer-foam specimens from panel 2

<table>
<thead>
<tr>
<th>Specimen No.</th>
<th>Density Kg/m$^3$</th>
<th>Crush Strength MPa</th>
<th>Elastic Modulus MPa</th>
<th>Ultimate Strength MPa</th>
<th>Ultimate Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>186.45</td>
<td>4.12</td>
<td>132.97</td>
<td>138.21</td>
<td>0.8903</td>
</tr>
<tr>
<td>2.2</td>
<td>189.03</td>
<td>4.05</td>
<td>134.55</td>
<td>138.21</td>
<td>0.8905</td>
</tr>
<tr>
<td>2.3</td>
<td>191.81</td>
<td>4.00</td>
<td>108.48</td>
<td>137.90</td>
<td>0.8898</td>
</tr>
</tbody>
</table>

The results of 2.5 cm specimens from panel 2 are listed in the Table 2.8. The typical Engineering Stress-Strain curve for 2.5 cm polymer-foam specimens from panel 2 is as shown in Figure 2.26. All the results for this type of polymer-foam specimen are very consistent.
Table 2.9 provides the results for 5.1 cm specimens from panel 2. Figure 2.27 shows the typical Engineering Stress-Strain curve for the 5.1 cm specimens from panel 2. Again all the results are very consistent. Figure 2.28 shows the crushed polymer-foam specimen.

Table 2.9: Results of 5.1 cm polymer-foam specimens from panel 2

<table>
<thead>
<tr>
<th>Specimen No.</th>
<th>Density Kg/m³</th>
<th>Crush Strength MPa</th>
<th>Elastic Modulus MPa</th>
<th>Maximum Stress MPa</th>
<th>Maximum Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.2</td>
<td>181.76</td>
<td>3.07</td>
<td>91.50</td>
<td>34.52</td>
<td>0.8417</td>
</tr>
<tr>
<td>2.4</td>
<td>185.98</td>
<td>3.71</td>
<td>94.81</td>
<td>34.52</td>
<td>0.8348</td>
</tr>
<tr>
<td>2.5</td>
<td>185.55</td>
<td>3.50</td>
<td>95.63</td>
<td>34.48</td>
<td>0.8371</td>
</tr>
</tbody>
</table>
Figure 2.27: Typical engineering stress-strain curve for 5.1 cm polymer-foam specimen from panel 2

Figure 2.28: Crushed polymer-foam specimens
2.6 Material Characterization of Stainless Steel

The inner pressurizing layer, throttle plates, gusset plates and the layer covering the polymer-foam material in the AT595 explosion-proof vessel is made of stainless steel. Stainless steel material is used so as to add rigidity to the vessel and also to attenuate the shock from the explosion material placed inside the vessel.

2.6.1 Samples Provided by RFNC

RFNC provided UNLV with four square stainless steel plates. The plates were approximately 20 cm in length and width, and having an approximate thickness of 0.41 cm. Figure 2.29 shows the stainless steel plates provided to UNLV by RFNC. Two of the four plates were utilized to make specimens for static tension tests.

![Stainless steel plate provided by RFNC](image)

Figure 2. 29: Stainless steel plate provided by RFNC
2.6.2 Test Specimen

ASTM E-8 standard [23] was used in designing the stainless steel specimen. From the E-8 standard, sheet type specimen was chosen because the thickness of the plates, 0.41 cm, falls within the required specification of this type of specimen. Figure 2.30 shows the ASTM E-8 standard for sheet type specimen.

Specifications as given in ASTM E–8 standard for Sheet type (1.27 cm wide) specimen
(Dimensions are in cm)

- \( G = \text{Gage length} = 5.08 \pm 0.005 \)
- \( W = \text{Width} = 1.27 \pm 0.010 \)
- \( T = \text{Thickness} = \text{Max. of} 1.91 \)
- \( R = \text{Radius of fillet} = \text{Min. of} 1.27 \)
- \( L = \text{Overall length} = 20.32 \)
- \( A = \text{Length of reduced section} = \text{Min. of} 5.72 \)
- \( B = \text{Length of grip section} = 5.08 \)
- \( C = \text{Width of grip section} = 1.91 \)
  (can be \( \frac{2}{3} \)rd of the length of the grip)

Figure 2.30: ASTM E-8 standard for sheet type specimen
From the design parameters given in ASTM E-8 standard for sheet type specimen, the stainless steel specimens were designed to the required dimensions. Figure 2.31 shows the final dimensions of the stainless steel specimens.

![Diagram of stainless steel specimen dimensions](image)

**Figure 2.31: Scheme of the designed stainless steel specimen**

A water jet machine tool was used to cut the stainless steel specimens. Water jet machine was preferred as the cutting tool because of its ability to give straight edges and precise finished dimensions without any surface damage. Figure 2.32 depicts the cut stainless steel specimens from the two plates.
The stainless steel specimens were then strain gaged on one side with the 90° tee rosette strain gage to obtain the lateral and longitudinal strains and hence the poisson’s ratio.

2.6.3 Testing

Tension tests were conducted on the stainless steel specimens using the MTS machine. The specimen was held by the grips of the MTS as shown in Figure 2.33. The upper portion of the specimen was fixed such that it cannot move in any direction during the test. Tensile loads were applied by pulling the lower portion of the specimen in the longitudinal direction as shown in Figure 2.33.
The maximum tensile load on the specimen was limited to 88 KN due to the limitation of the fixtures on the MTS, i.e. the tension test was automatically aborted by the MTS machine once the load limit of 88 KN had been reached. The strain was measured in three ways during the experiment. Strain gages were used to measure longitudinal and transverse strains, from which the poisson’s ratio was computed. The strain values for computing the elastic modulus was also taken from the strain gage. LVDT present in the MTS was used to measure the yield, maximum and fracture strains. Laser equipment was also used for measuring strains, but is only used for verifying the strains from the aforementioned devices. The strain rate used for tension testing the
stainless steel specimens is $2.0 \times 10^{-4} \text{ mm/mm/second}$. Figure 2.34 depicts the strain versus time for the stainless steel specimen, the slope of which represents the strain rate used.

![Graph showing strain rate for the stainless steel specimen](image)

Figure 2.34: Strain rate for the stainless steel specimen

2.6.4 Results and Comments

For all the tests the load limit was never achieved as all the specimens failed much before 88 KN. Figure 2.35 shows the typical Engineering Stress-Strain curve obtained for the stainless steel specimen. The path of this curve is similar to a standard steel stress-strain curve and the regions of yield, ultimate and fracture can be easily spotted on the plot. From the plot the material properties of stainless steel is computed.
Table 2.10 below provides the results for stainless steel specimens from plate 1. The typical Engineering Stress-Strain curve for the stainless steel specimens from plate 1 is as shown in Figure 2.36. The results in Table 2.10 are consistent.
Table 2. 10: Results of stainless steel specimens from plate 1

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson’s ratio</th>
<th>Elastic Modulus GPa</th>
<th>Yield Stress MPa</th>
<th>Yield Strain mm/mm</th>
<th>Ultimate Stress MPa</th>
<th>Ultimate Strain mm/mm</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.25</td>
<td>183.60</td>
<td>245.43</td>
<td>0.0040</td>
<td>628.31</td>
<td>0.6725</td>
<td>516.35</td>
<td>0.7305</td>
</tr>
<tr>
<td>2</td>
<td>0.24</td>
<td>190.00</td>
<td>241.37</td>
<td>0.0040</td>
<td>628.95</td>
<td>0.6693</td>
<td>No Data</td>
<td>No Data</td>
</tr>
<tr>
<td>3</td>
<td>0.25</td>
<td>165.75</td>
<td>236.27</td>
<td>0.0035</td>
<td>581.07</td>
<td>0.5989</td>
<td>446.40</td>
<td>0.6564</td>
</tr>
<tr>
<td>4</td>
<td>0.25</td>
<td>174.37</td>
<td>237.22</td>
<td>0.0032</td>
<td>626.00</td>
<td>0.6416</td>
<td>506.85</td>
<td>0.6958</td>
</tr>
</tbody>
</table>

Figure 2. 36: Typical engineering stress-strain curve for stainless steel specimen from plate 1

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 2.11 depicts the results obtained from the stainless steel specimens of plate 2. All the results are very consistent. Figure 2.28 shows the Engineering Stress-Strain curve for the stainless steel specimens from plate 2. The fractured stainless steel specimen is shown in Figure 2.38.

Table 2.11: Results of stainless steel specimens from plate 2

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson’s ratio</th>
<th>Elastic Modulus GPa</th>
<th>Yield Stress MPa</th>
<th>Yield Strain mm/mm</th>
<th>Ultimate Stress MPa</th>
<th>Ultimate Strain mm/mm</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.27</td>
<td>176.07</td>
<td>258.21</td>
<td>0.0036</td>
<td>649.30</td>
<td>0.6717</td>
<td>536.23</td>
<td>0.7286</td>
</tr>
<tr>
<td>2</td>
<td>0.27</td>
<td>181.05</td>
<td>257.21</td>
<td>0.0036</td>
<td>640.67</td>
<td>0.6679</td>
<td>540.99</td>
<td>0.7280</td>
</tr>
<tr>
<td>3</td>
<td>No Data</td>
<td>No Data</td>
<td>256.98</td>
<td>0.0037</td>
<td>640.13</td>
<td>0.6590</td>
<td>545.49</td>
<td>0.7151</td>
</tr>
</tbody>
</table>

Figure 2.37: Typical engineering stress-strain curve for stainless steel specimen from plate 2

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
2.7 Material Characterization of Basalt-Plastic

The outer composite layer of the AT595 explosion-proof vessel is made up of basalt-plastic. Basalt-plastic consists of epoxy matrix reinforced with basalt fibers. Basalt-plastic material is used so as to make the AT595 explosion-proof vessel light weight and at the same time increase its overall strength. Also the basalt fibers are cheap, hence cost effective, when compared to glass and carbon fibers.

2.7.1 Samples Provided by RFNC

RFNC provided UNLV with seven square basalt-plastic panels. The panels were approximately 33 cm in length and width. The thicknesses of the panels were not consistent but varying as shown in Table 2.12, this can be attributed to the ridges present in the panels and also due to the uneven distribution of the fibers throughout an individual panel. Out of the seven panels, five were unidirectional and the other two were \(\pm 45^\circ\) panels. Static tension test was conducted on the specimens of four panels. Table 2.12 lists the type of panels used for conducting tests.
Table 2.12: Types of basalt-plastic panels provided by RFNC

<table>
<thead>
<tr>
<th>Panel</th>
<th>Type</th>
<th>Thickness mm</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Unidirectional</td>
<td>1.5 – 2.3</td>
<td>Tested - Transverse</td>
</tr>
<tr>
<td>2</td>
<td>Unidirectional</td>
<td>5.3 – 6.4</td>
<td>Not Tested</td>
</tr>
<tr>
<td>3</td>
<td>Unidirectional</td>
<td>5.1 – 6.4</td>
<td>Not Tested</td>
</tr>
<tr>
<td>4</td>
<td>±45°</td>
<td>2.8 – 3.0</td>
<td>Tested - ±45°</td>
</tr>
<tr>
<td>5</td>
<td>±45°</td>
<td>2.8 – 3.3</td>
<td>Not Tested</td>
</tr>
<tr>
<td>6</td>
<td>Unidirectional</td>
<td>2.8 – 3.0</td>
<td>Tested - Transverse</td>
</tr>
<tr>
<td>7</td>
<td>Unidirectional</td>
<td>2.8 – 3.8</td>
<td>Tested - Axial</td>
</tr>
</tbody>
</table>

2.7.2 Test Specimen

ASTM D-3039 standard [23] was used in designing the basalt-plastic specimen. From the design parameters given in ASTM D-3039 standard, the basalt-plastic specimens were designed to the required dimensions as shown in Figure 2.39.

![Figure 2.39: Scheme of the designed basalt-plastic specimen](image)

Dimension units are in cm
(T=approximately ranges from 0.15 cm to 0.64 cm)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The basalt-plastic specimens were tested on the MTS machine, whose gripping pressure is high enough to crush the basalt-plastic material, hence end tabs were used to prevent this crushing and at the same time transfer the applied load into the test specimen from the loading device. Fiber-glass was used as the end tab material. Fiber-glass end tabs were bonded onto the basalt-fiber panels by the use of epoxy.

The end tabs were beveled to the required angle at one end with the help of a milling machine tool. The end tabs were first bonded onto the basalt-plastic panels and then the panels were cut for the required test specimens. A water jet machine tool was used for cutting the basalt-plastic specimens. Water jet machine is preferred as the cutting tool because of its ability to give straight edges and precise finished dimensions without any surface damage. Figure 2.40 depicts cutting of the basalt-plastic panels, bonded with end tabs, by the water jet machine.

Figure 2. 40: Basalt-plastic panel being cut by water jet cutting tool
From each panel two different types of specimens were cut, one was 1.27 cm wide basalt-plastic specimen and the other was 2.54 cm wide. The stainless steel specimens were then strain gaged on one side with the 90° tee rosette strain gage, as shown in Figure 2.41, to obtain the lateral and longitudinal stains and hence the poisson’s ratio.

![Figure 2.41: 90° tee rosette strain gage glued onto two different basalt-plastic specimens](image)

2.7.3 Testing

Tension tests were conducted on the basalt-plastic specimens using the MTS machine. The specimen was held by the grips of the MTS at the end tab region as shown in Figure 2.42. The upper portion of the specimen is fixed such that it cannot move in any direction during the test. Tensile loads are applied by pulling the lower portion of the specimen in the longitudinal direction as shown in Figure 2.42.
The maximum tensile load on the specimen was limited to 88 KN due to the limitation of the fixtures on the MTS, i.e. the tension test was aborted automatically by the MTS machine once the load limit of 88 KN had been reached. Similar to stainless steel specimens, the strain values were measured in three ways during the experiment. Strain gages were used to measure longitudinal and transverse strains, from which the poisson’s ratio was computed. The strain values for computing the modulus was also taken from the strain gage. LVDT present in the MTS was used to measure the fracture strain. Laser equipment was also used for measuring strains, but was only used for verifying the strains from the aforementioned devices. A strain rate of $8.0 \times 10^{-5}$
mm/mm/second was used for testing the axial and transverse basalt-plastic specimens whereas for the ±45° basal-plastic specimens the strain rate used was 1.0E-4 mm/mm/second. Figure 2.43 depicts the strain versus time for the basalt-plastic specimen, the slope of which represents the strain rate used.

Figure 2.43: Strain rate for the basalt-plastic specimens
2.7.4 Results and Comments

For all the tests the load limit was never achieved as all the specimens failed much before 88 KN.

2.7.4.1 Transverse Basalt-Plastic Specimen

Panel 1 and panel 6 were tested for transverse or across the fibers basalt-plastic properties. In this case the load was applied perpendicular to the fiber direction. Figure 2.44 shows the typical Engineering Stress-Strain curve obtained for the transverse basalt-plastic specimen and the material properties formulated from the obtained plots.

![Stress-Strain curve for Transverse basalt-plastic specimen](image)

Figure 2.44: Material properties obtained from the stress-strain plot of transverse basalt-plastic specimens
Table 2.13 provides the results for transverse basal-plastic specimens from panel 1. Panel 1 had large thickness variations and voids, due to which the material properties were not consistent as shown in Table 2.13. Figure 2.45 depicts the typical Engineering Stress-Strain curve for the transverse basalt-plastic specimen from panel 1.

Table 2.13: Results of transverse specimens from panel 1

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson’s ratio</th>
<th>Elastic Modulus (GPa)</th>
<th>Fracture Stress (MPa)</th>
<th>Fracture Strain (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-1</td>
<td>0.064</td>
<td>6.68</td>
<td>22.5</td>
<td>0.0035</td>
</tr>
<tr>
<td>1-2</td>
<td>0.047</td>
<td>6.12</td>
<td>28.9</td>
<td>0.0044</td>
</tr>
<tr>
<td>1-5</td>
<td>0.072</td>
<td>8.32</td>
<td>25.6</td>
<td>0.0028</td>
</tr>
<tr>
<td>1-7</td>
<td>0.044</td>
<td>6.65</td>
<td>16.0</td>
<td>0.0024</td>
</tr>
<tr>
<td>1-8</td>
<td>0.066</td>
<td>6.89</td>
<td>27.0</td>
<td>0.0039</td>
</tr>
</tbody>
</table>

Figure 2.45: Typical engineering stress-strain curve for transverse specimen from panel 1
The results of transverse basalt-plastic specimens from panel 6 are tabulated in the Table 2.14. The results of these specimens are very consistent compared to the specimens from panel 1 and can be attributed to the less variation in thickness and more even distribution of fibers. The typical Engineering Stress-Strain curve for the transverse basalt-plastic specimens from panel 6 is shown in Figure 2.46. Figure 2.47 shows the fractured transverse basalt-plastic specimen.

Table 2.14: Results of transverse specimens from panel 6

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson’s ratio</th>
<th>Elastic Modulus GPa</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>6-1</td>
<td>0.055</td>
<td>7.42</td>
<td>23.2</td>
<td>0.0031</td>
</tr>
<tr>
<td>6-2</td>
<td>0.060</td>
<td>7.83</td>
<td>23.0</td>
<td>0.0029</td>
</tr>
<tr>
<td>6-3</td>
<td>0.055</td>
<td>7.70</td>
<td>22.9</td>
<td>0.0030</td>
</tr>
</tbody>
</table>

Figure 2.46: Typical engineering stress-strain curve for transverse specimen from panel 6

65
2.7.4.2 Axial Basalt-Plastic Specimen

Panel 7 is tested for basalt-plastic properties in the axial or along the fibers direction. In this case the tensile load was applied along or parallel to the direction of the basalt fibers. Figure 2.48 shows the typical Engineering Stress-Strain curve obtained for the transverse basalt-plastic specimen and the material properties formulated from the obtained plots.
Figure 2.48: Axial basalt-plastic materials properties obtained from the stress-strain plot

Table 2.15 provides the results of axial basal-plastic specimens. The results are very consistent. Figure 2.49 depicts the typical Engineering Stress-Strain curve for the axial basalt-plastic specimen. Figure 2.50 shows the fractured axial specimens.

Table 2.15: Results of axial specimens from panel 7

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson's ratio</th>
<th>Elastic Modulus GPa</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>7-3</td>
<td>0.29</td>
<td>29.6</td>
<td>650</td>
<td>0.044</td>
</tr>
<tr>
<td>7-5</td>
<td>0.29</td>
<td>28.5</td>
<td>617</td>
<td>0.042</td>
</tr>
<tr>
<td>7-7</td>
<td>0.29</td>
<td>29.6</td>
<td>592</td>
<td>0.044</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 2. 49: Typical engineering stress-strain curve for axial specimen from panel 7

Figure 2. 50: Axial basalt-plastic specimen after the test
2.7.4.3 ±45° Basalt-Plastic Specimen

The specimens from Panel 4 were tested for shear properties of the basalt-plastic materials. In this case the fibers were oriented ±45° to the applied tensile load. The shear stress and shear strain values were computed from the obtained engineering stress-strain values using Equation 2.1 and Equation 2.2. Figure 2.51 shows a typical Engineering Stress-Strain curve and Shear Stress-Strain curve for the ±45° basalt-plastic specimen, and the determination of shear modulus.

\[ \tau = \frac{\sigma}{2} \]  \hspace{1cm} (2.1)

\[ \gamma = \varepsilon_x - \varepsilon_y \]  \hspace{1cm} (2.2)

where,

\[ \tau \] = Shear Stress (GPa)

\[ \sigma \] = Normal/Tensile Stress (MPa)

\[ \gamma \] = Shear Strain (mm/mm)

\[ \varepsilon_x \] = Longitudinal Strain (mm/mm)

\[ \varepsilon_y \] = Lateral/Transverse Strain (mm/mm)
Table 2.18 provides the results of ±45° basal-plastic specimens. From the Table 2.16 it can be stated that the shear modulus is consistent. Figure 2.52 depicts the typical Engineering Stress-Strain curve for the ±45° basalt-plastic specimen. Figure 2.53 shows the fractured ±45° specimens.

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Poisson’s ratio</th>
<th>Shear Modulus GPa</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>4-1</td>
<td>0.62</td>
<td>10.8</td>
<td>130</td>
<td>0.068</td>
</tr>
<tr>
<td>4-2</td>
<td>0.67</td>
<td>13.5</td>
<td>112</td>
<td>0.062</td>
</tr>
<tr>
<td>4-3</td>
<td>0.64</td>
<td>11.8</td>
<td>102</td>
<td>0.054</td>
</tr>
</tbody>
</table>
Figure 2.52: Typical engineering stress-strain curve for ±45° specimen from panel 4

Figure 2.53: Fractured ±45° basalt-plastic specimen
2.8 Conclusion

2.8.1 Anti-Fragment shield

Not much of experimental data was provided for anti-fragment shield from the RFNC. The only information RFNC had reported about anti-fragment shield is that a pressure of 5 MPa is good to crush the shield by 50%. There is no mention of the number of layers or the thickness of the shield. Also they noted that the 'layer' noticed a considerable resistance after 50% compression. Again no specifications were given about the anti-fragment shield 'layer'.

From the testing done in UNLV, the results are observed to be consistent and the curve patterns accordingly. It is difficult to compare these results with RFNC, due to the limited knowledge of the RFNC test procedure and the data available.

2.8.2 Polymer-Foam

All the polymer-foam specimens from both the panels are summarized and tabulated in the Table 2.17. From these results it is observed that all the specimens are crushed to 80% or more of their original thickness, if not for the load limitation due to the fixture the polymer-foam specimens could have been crushed to more than 90%.

Table 2.17: Summary of results for polymer-foam specimens

<table>
<thead>
<tr>
<th>Specimen Type</th>
<th>Density (Kg/m³)</th>
<th>Crush Strength (MPa)</th>
<th>Elastic Modulus (MPa)</th>
<th>Maximum Strain (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg.</td>
<td>S.D. (%)</td>
<td>Avg.</td>
<td>S.D. (%)</td>
</tr>
<tr>
<td>2.5cm-Panel 1</td>
<td>230±2.1</td>
<td>5.86±0.7</td>
<td>No Data</td>
<td>No Data</td>
</tr>
<tr>
<td>5.1 cm-Panel 1</td>
<td>220±6.5</td>
<td>4.77±9.0</td>
<td>110.8±14.8</td>
<td>0.80±1.6</td>
</tr>
<tr>
<td>2.5cm-Panel 2</td>
<td>190±1.4</td>
<td>4.06±1.5</td>
<td>125.3±11.7</td>
<td>0.89±0.4</td>
</tr>
<tr>
<td>5.1cm-Panel 2</td>
<td>180±1.3</td>
<td>3.42±9.4</td>
<td>94.0±2.3</td>
<td>0.84±0.4</td>
</tr>
</tbody>
</table>
The only data of polymer-foam available from RFNC is the density of the foam, which is 200 kg/m$^3$ and the Engineering stress-strain curve, shown in Figure 2.54. From the curve the crush strength can be interpreted as being 5 MPa, which is close to Panel 1 tested in UNLV. The variations of the crush strength for the specimens tested in UNLV can be attributed to the significant differences in the densities of the samples.

![Graph](image)

Figure 2.54: Engineering stress-strain curve for the polymer-foam material provided by RFNC [1]

2.8.3 Stainless Steel

The results from specimens of both the stainless steel plates are summarized in Table 2.18. From the summarized results it is seen that the measured material properties are consistent and repeatable for both the plates.
Table 2.18: Summary of stainless steel results

<table>
<thead>
<tr>
<th>Poisson’s ratio</th>
<th>Elastic Modulus GPa</th>
<th>Yield Stress MPa</th>
<th>Yield Strain mm/mm</th>
<th>Ultimate Stress MPa</th>
<th>Ultimate Strain mm/mm</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25 ±4.0</td>
<td>178 ±4.7</td>
<td>248 ±3.9</td>
<td>0.0036 ±7.9</td>
<td>627 ±3.4</td>
<td>0.65 ±3.1</td>
<td>515 ±7.2</td>
<td>0.71 ±4.2</td>
</tr>
</tbody>
</table>

Table 2.19: Different types of steel materials used in AT595 explosion-proof vessel

<table>
<thead>
<tr>
<th>No.</th>
<th>Used for</th>
<th>Yield Strength MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Thick wall parts of the inner vessel like mouths and port lids</td>
<td>245</td>
</tr>
<tr>
<td>2</td>
<td>Cylindrical part of inner shell and throttle plates</td>
<td>500</td>
</tr>
<tr>
<td>3</td>
<td>Parts with steel material other than the ones mentioned above</td>
<td>800</td>
</tr>
</tbody>
</table>

RFNC reported three types of steel materials, which are listed in Table 2.19. From the three, the first one looks closest to the one provided to UNLV as the modulus is matching. It is made of whole-rolled preforms and has an ultimate strength of 520 MPa.

2.8.4 Basalt-Plastic

All the basalt-plastic composite results are summarized in the Table 2.20. From these summarized results it can be stated that excluding panel 1, the specimens from same panels provide consistent results. The large variation in the material properties of panel 1 is due to the presence of large number of voids, uneven thickness of the panel and also non-uniform distribution of the basalt fibers.
Table 2. 20: Summary of basalt-plastic results

<table>
<thead>
<tr>
<th>Panel</th>
<th>Poisson’s ratio</th>
<th>Modulus GPa</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg.</td>
<td>S.D. %</td>
<td>Avg.</td>
<td>S.D. %</td>
</tr>
<tr>
<td>Panel 1 (Transverse)</td>
<td>0.059 ±2.00</td>
<td>6.93 ±12.0</td>
<td>24 ±21.0</td>
<td>0.34 ±8.1</td>
</tr>
<tr>
<td>Panel 6 (Transverse)</td>
<td>0.057 ±0.29</td>
<td>7.65 ±2.7</td>
<td>23 ±0.7</td>
<td>0.30 ±1.0</td>
</tr>
<tr>
<td>Panel 7 (Axial)</td>
<td>0.290 ±0.32</td>
<td>29.2 ±2.2</td>
<td>620 ±4.7</td>
<td>4.32 ±7.9</td>
</tr>
<tr>
<td>Panel 4 (±45°)</td>
<td>0.640 ±2.00</td>
<td>12.0 ±12.0</td>
<td>115 ±12.0</td>
<td>6.12 ±7.4</td>
</tr>
</tbody>
</table>

Table 2. 21: Basalt-plastic values provided by RFNC

<table>
<thead>
<tr>
<th>Density</th>
<th>2060 kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity for Axial specimen</td>
<td>53.55 GPa</td>
</tr>
<tr>
<td>Modulus of Elasticity for Transverse specimen</td>
<td>15.15 GPa</td>
</tr>
<tr>
<td>Modulus of Rigidity for ±45° specimen</td>
<td>5.9 GPa</td>
</tr>
<tr>
<td>Poisson’s Ratio for Axial specimen</td>
<td>0.29</td>
</tr>
</tbody>
</table>

The elastic modulus of the axial and transverse specimens is less when compared to the values provided by RFNC, listed in Table 2.21. The density of the basalt-plastic material tested in UNLV is 1700 kg/m³ whereas the basalt-plastic material tested in RFNC had a density of 2060 kg/m³. The density of the UNLV panels corresponds to a fiber volume fraction of 30% while the density of the basalt-plastic material tested in RFNC corresponds to 56% fiber volume fraction. Hence the low values of modulus can be attributed to the lower fiber volume fraction of the panels. The poisson’s ratio and failure strain values obtained are reasonable.
The overall results of all the material samples were consistent and comparable with the RFNC values.
CHAPTER 3

FINITE ELEMENT ANALYSIS

The AT595 explosion-proof vessel built by the RFNC is taken as the base model for all the FE models in this thesis. Figure 3.1 shows a quarter section of the AT595 explosion-proof vessel, which mainly consists of a central cylindrical portion with two hemispherical end caps on each side. The length of the cylindrical portion of the vessel is 2.45 m with an outer diameter of 0.984 m. The total length of the vessel is 3.32 m. The diameter of the polymer-foam is 0.44 m and has a length of 0.36 m. The throttle plate has a thickness of 0.02 m and an inner diameter of 0.44 m. The gusset plates are triangular shaped with the length of the sides being 0.15 m and having a thickness of 0.015 m. Table 3.1 summarizes the dimensions of AT595 explosion-proof vessel.

![Figure 3.1: AT595 explosion-proof vessel](image)

Table 3.1 summarizes the dimensions of AT595 explosion-proof vessel.
Table 3.1: Basic dimensions of the AT595 explosion-proof vessel

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overall Length of Vessel</td>
<td>3.320 m</td>
</tr>
<tr>
<td>Length of the Cylindrical Portion of the Vessel</td>
<td>2.450 m</td>
</tr>
<tr>
<td>Outer Diameter of Vessel</td>
<td>0.984 m</td>
</tr>
<tr>
<td>Inner Diameter of Vessel</td>
<td>0.911 m</td>
</tr>
<tr>
<td>Diameter of Polymer-Foam</td>
<td>0.440 m</td>
</tr>
<tr>
<td>Length of Polymer-Foam</td>
<td>0.360 m</td>
</tr>
<tr>
<td>Inner Diameter of Throttle Plate</td>
<td>0.440 m</td>
</tr>
<tr>
<td>Thickness of Throttle Plate</td>
<td>0.020 m</td>
</tr>
<tr>
<td>Thickness of Gusset Plate</td>
<td>0.015 m</td>
</tr>
<tr>
<td>Length of the sides of Gusset Plate</td>
<td>0.150 m</td>
</tr>
</tbody>
</table>

The vessel is made up of a filament wound basalt fiber/epoxy composite with an inner steel liner. Polymer foam has been placed on either end of the vessel to absorb the energy released by the blast. An anti-fragment shield made up of steel wire mesh has been placed inside the cylindrical liner to absorb and distribute the shock loads from shrapnel. Two steel throttle plates are added to increase the rigidity of the structure and attenuate the shock from the explosive materials that are placed inside the container. Radially placed steel gusset plates support the throttle plates to add rigidity to the vessel. Therefore the important and basic parts of the AT595 explosion-proof vessel that need to be incorporated in the FE model of the vessel for conducting explicit internal blast loading analysis are the outer basalt-plastic composite layers, inner steel layer, polymer-foam,
anti-fragment shield, throttle plates and the gusset plates. Table 3.2 summarizes the 
aforementioned basic parts and their functions in the AT595 explosion-proof vessel.

Table 3.2: The function of the basic parts of AT595 explosion-proof vessel

<table>
<thead>
<tr>
<th>Basic Feature</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basalt-Plastic Layer</td>
<td>✓ Increase strength</td>
</tr>
<tr>
<td></td>
<td>✓ Make the vessel light weight</td>
</tr>
<tr>
<td>Steel Lining</td>
<td>✓ Add rigidity to the vessel</td>
</tr>
<tr>
<td>Polymer-Foam</td>
<td>✓ Absorb energy released by the blast</td>
</tr>
<tr>
<td>Anti-Fragment Shield</td>
<td>✓ Absorb the shock loads</td>
</tr>
<tr>
<td></td>
<td>✓ Distribute the shock loads from the shrapnel</td>
</tr>
<tr>
<td>Throttle Plates</td>
<td>✓ Increase rigidity of the vessel</td>
</tr>
<tr>
<td></td>
<td>✓ Attenuate the shock from the explosion</td>
</tr>
<tr>
<td>Gusset Plates</td>
<td>✓ Support the throttle plates</td>
</tr>
<tr>
<td></td>
<td>✓ Adds rigidity to the vessel</td>
</tr>
</tbody>
</table>

In this chapter, five FE models of the AT595 explosion-proof vessel with different 
levels of detail are studied. The effectiveness of various models is measured by 
comparing computational predictions of strains on the outer surface of the vessel to the 
experimental and computational results obtained by RFNC. The objective in conducting 
FE analysis of the AT595 explosion-proof vessel is to,

- To reach general recommendations regarding the creation of FE models for closed 
cylindrical vessels subjected to internal blast loads.
• To check if the CONWEP code, embedded within LS-DYNA, can properly model internal blasts as this will reduce the need to use an Eulerian-Lagrangian code, which is computationally expensive.

• To create a FE explosion-proof vessel model that provides a combination of accuracy and computational speed for carrying out parametric analysis or design optimization.

3.1 System and Software

All the computational analysis was done on a 2.8 Ghz AMD Athlon processor, having a 2 GB RAM. Altair HyperMesh v7.0 was used as the pre-processor to create and mesh the 3D models of AT595 explosion-proof vessel. Explicit FE code LS-DYNA v970 was used to simulate the structural response of the FE models. LS-POST and Altair HyperView v7.0 were used for post-processing the analysis.

3.2 Units

The standard S.I. system of units is used throughout this thesis report. The basic and derived units used in defining the FE model in LS-DYNA are listed in Table 3.3 and Table 3.4 respectively.

<table>
<thead>
<tr>
<th>Basic Parameter</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>Meter (m)</td>
</tr>
<tr>
<td>Mass</td>
<td>Kilogram (kg)</td>
</tr>
<tr>
<td>Time</td>
<td>Second (s)</td>
</tr>
</tbody>
</table>

Table 3.3: The basic units used in LS-DYNA
Table 3.4: The derived units of LS-DYNA

<table>
<thead>
<tr>
<th>Derived Parameter</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Force</td>
<td>N*</td>
</tr>
<tr>
<td>Stress / Pressure / Modulus</td>
<td>GPa**</td>
</tr>
<tr>
<td>Strain</td>
<td>m/m (dimensionless)</td>
</tr>
</tbody>
</table>

*N = Newton = kg.m/s$^2$

**GPa = $10^9$ Pascal = $10^9$ N/m$^2$

3.3 Material Properties Obtained from RFNC

The material properties required to define the material models in LS-DYNA are obtained from RFNC. Material properties are obtained for basalt-plastic, polymer-foam and stainless steel material.

3.3.1 Basalt-Plastic Material Properties

The outer basalt-plastic layer of the AT595 explosion-proof vessel is fabricated by spiral circular winding of ribbon Pb9-1250-4S over the inner steel lining. The ribbon consists 9 mm diameter basalt fiber filaments, which is impregnated with epoxy binder 3πT-10. Circular winding on spherical parts of container was performed in accordance with the law of geodesic profile. The cylindrical section of the composite has alternating layers of 90° annular windings and balanced ±33° layers. The combined thickness of all the 90° annular layers is 16 mm and the overall thickness of all the balanced ±33° layers is 16 mm. Hence the total thickness of the basalt-plastic composite layer in the cylindrical
portion of the vessel is 32 mm. The cap portion of the vessel only has 16 mm of balanced ±33° layers. The basalt-plastic material properties obtained from RFNC is listed in Table 3.5.

Table 3.5: Basalt-plastic material properties from RFNC

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (p)</td>
<td>2060 kg/m³</td>
</tr>
<tr>
<td>Modulus of Elasticity in the Fiber Direction (E₁)</td>
<td>53.55 GPa</td>
</tr>
<tr>
<td>Modulus of Elasticity across the Fiber Direction (E₂)</td>
<td>15.15 GPa</td>
</tr>
<tr>
<td>Modulus of Rigidity (G₁₂)</td>
<td>5.9 GPa</td>
</tr>
<tr>
<td>Poisson’s Ratio (v₁₂)</td>
<td>0.29</td>
</tr>
</tbody>
</table>

3.3.2 Polymer-Foam Material Properties

The material properties of the cylindrical shaped polymer-foam reported to UNLV by RFNC are shown in Table 3.6. RFNC also provided UNLV with an engineering stress-strain curve, as shown in Figure 3.2, for the polymer-foam material.

Table 3.6: Polymer-foam material properties from RFNC

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (p)</td>
<td>200 kg/m³</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>0.108 GPa</td>
</tr>
<tr>
<td>Crush Strength (Sₜ)</td>
<td>5.50 MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio (v)</td>
<td>0.33</td>
</tr>
</tbody>
</table>
In order to define the polymer-foam material model in the FEA code, the engineering stress versus volumetric strain plot is required. Hence from the engineering stress-strain curve given by RFNC the volumetric strain values were derived and used in defining the polymer-foam material model.

3.3.3 Stainless Steel Properties

The inner lining, below the outer basalt-plastic composite layer as shown in Figure 3.1, throughout the AT595 explosion-proof vessel is made of stainless steel. The thickness of this inner steel lining is 4.5 mm. The lining over the polymer-foam is also made of stainless steel material. The material properties of stainless steel used for the inner lining is obtained from RFNC and are listed in Table 3.7. The stainless steel properties listed in the table below are also used for defining the material models of gusset and throttle plates in the FEA code.
Table 3.7: Steel material properties from RFNC

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density ((\rho))</td>
<td>7850 Kg/m³</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>200 GPa</td>
</tr>
<tr>
<td>Plastic Tangent Modulus ((E_t))</td>
<td>0.10 MPa</td>
</tr>
<tr>
<td>Yield Strength ((S_y))</td>
<td>500 MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio ((v))</td>
<td>0.30</td>
</tr>
</tbody>
</table>

3.4 Assumptions in Modeling the Vessel

Some assumptions are made in modeling the AT595 explosion-proof vessel due to difficulty in modeling the material or to simplify the analysis procedure so as to reduce the overall computational time.

3.4.1 Assumption of Anti-Fragment Shield

It can be noted that the anti-fragment shield does not contribute significantly towards the structural performance of the vessel. The steel wire mesh material of the anti-fragment shield is also difficult to model. However, the mass of the anti-fragment shield affects the dynamic response of the explosion-proof vessel. Therefore, the material was not included in the structural analysis but the mass was accounted for by adjusting the density of the inner steel lining in the shield location accordingly [Appendix I].

3.4.2 Assumption in Modeling the Composite Layer

As mentioned earlier, the basalt-plastic layer in the central cylindrical portion of the AT595 explosion-proof vessel is made up alternate layers of 90° annular windings and balanced ±33° layers whereas the cap portion consists of just the balanced ±33° layers. There is an inner stainless steel lining, below the outer basalt-plastic layer, throughout the
vessel. In order to simplify the analysis, the FE model of the explosion-proof vessel was assumed to be created with several homogeneous layers as shown in Table 3.8 and Table 3.9.

Table 3. 8: Geometric characteristic of composite layer in the cylindrical portion

<table>
<thead>
<tr>
<th>Layer No. (Starting from the outermost one)</th>
<th>Orientation / Material Type</th>
<th>Thickness (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>90° Basalt-Plastic</td>
<td>0.016</td>
</tr>
<tr>
<td>2</td>
<td>+33° Basalt-Plastic</td>
<td>0.008</td>
</tr>
<tr>
<td>3</td>
<td>-33° Basalt-Plastic</td>
<td>0.008</td>
</tr>
<tr>
<td>4</td>
<td>Stainless Steel</td>
<td>0.0045</td>
</tr>
</tbody>
</table>

Table 3. 9: Geometric characteristic of the composite material in the cap portion

<table>
<thead>
<tr>
<th>Layer No. (Starting from the outermost one)</th>
<th>Orientation / Material Type</th>
<th>Thickness (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+33° Basalt-Plastic</td>
<td>0.008</td>
</tr>
<tr>
<td>2</td>
<td>-33° Basalt-Plastic</td>
<td>0.008</td>
</tr>
<tr>
<td>3</td>
<td>Stainless Steel</td>
<td>0.0045</td>
</tr>
</tbody>
</table>

A program was written in MATLAB to derive the material properties of 90° and ±33° basalt-plastic layers from the properties of basalt-plastic composite given by RFNC [Appendix II]. Table 3.10 lists the material properties of 90° basalt-plastic layer while Table 3.11 shows the material properties of ±33° basalt-plastic layers.
Table 3.10: 90° basalt-plastic material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (ρ)</td>
<td>2060 kg/m³</td>
</tr>
<tr>
<td>Modulus of Elasticity in the Fiber Direction (E₁)</td>
<td>15.15 GPa</td>
</tr>
<tr>
<td>Modulus of Elasticity across the Fiber Direction (E₂)</td>
<td>53.55 GPa</td>
</tr>
<tr>
<td>Modulus of Rigidity (G₁₂)</td>
<td>5.9 GPa</td>
</tr>
<tr>
<td>Poisson's Ratio (ν₁₂)</td>
<td>0.082</td>
</tr>
</tbody>
</table>

Table 3.11: ±33° basalt-plastic material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (ρ)</td>
<td>2060 kg/m³</td>
</tr>
<tr>
<td>Modulus of Elasticity in the Fiber Direction (E₁)</td>
<td>20.77 GPa</td>
</tr>
<tr>
<td>Modulus of Elasticity across the Fiber Direction (E₂)</td>
<td>14.84 GPa</td>
</tr>
<tr>
<td>Modulus of Rigidity (G₁₂)</td>
<td>9.28 GPa</td>
</tr>
<tr>
<td>Poisson's Ratio (ν₁₂)</td>
<td>0.433</td>
</tr>
</tbody>
</table>

3.4.3 Symmetry Assumption

The AT595 explosion-proof vessel is symmetric along the three global axes x, y and z. Hence one-eighth models of the vessel were used for FEA. This also helps in reducing the computational time required to run the FE model of the vessel.

3.5 Model Run Time

As stated in earlier chapters, RFNC conducted both experimental testing and computational simulation of the AT595 explosion-proof vessel. The manufactured vessel
for experimental testing is as shown Figure 3.3. The vessel was mounted with gages at various locations to measure the strain induced in the vessel due to internal blast loading. The strain data reported to UNLV is limited to 3 ms from the start of the experiment, i.e., the inception of the blast. Also, the RFNC computational models were simulated for 3 ms. Hence, to compare the FE results obtained by UNLV with those of RFNC, all the FE models in this thesis were simulated for 3 ms of run time.

![Figure 3.3: AT595 explosion-proof vessel](image)

3.6 Element Types

Two basic types of elements were used in meshing the five 3D models of the AT595 explosion-proof vessel. The majority of the 3D models were meshed with shell elements while one FE model was completely meshed with solid elements.

3.6.1 Shell Element

The shell elements used in meshing the 3D vessel models are 4-noded with bending capabilities. Both in-plane and normal loads are permitted. The element has twelve
degrees of freedom at each node: translations, accelerations and velocities in the nodal x, y and z directions and rotations about the x, y and z axes. This type of element is used in explicit dynamic analysis [21].

The node numbering is done in the anticlockwise direction for this type of element as shown in Figure 3.3. With this type of node numbering the pressure loads act towards the element, i.e., positive pressure acts in the negative z direction with respect to the Figure 3.4. The Belytschko-Lin-Tsay shell type of element formulation is used for this shell element. This is the default shell element formulation used in LS-DYNA due to its computational efficiency. The Belytschko-Lin-Tsay shell element [21] is based on a combined co-rotational and velocity strain formulation. The efficiency of the element is obtained from the mathematical simplifications that result from these two kinematical assumptions. The co-rotational portion of the formulation avoids the complexities of nonlinear mechanics by embedding a coordinate system in the element. The choice of

Figure 3.4: Scheme of a shell element [21]

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
velocity strain, or rate deformation, in the formulation facilitates the constitutive
evaluation.

3.6.2 Solid Element

The solid elements used in meshing the 3D vessel models are defined by eight nodes. These elements have nine degrees of freedom at each node: translations, accelerations and velocities in the nodal $x$, $y$ and $z$ directions and rotations about the $x$, $y$ and $z$ axes. This type of element is used in explicit dynamic analysis [21].

![Figure 3.5: Scheme of a solid element [21]](image)

The geometry, node locations, and the coordinate system for this element are as shown in Figure 3.5. Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 3.5. Positive normal pressures act into the
element. By default this element uses reduced (one point) integration for faster element formulation.

3.7 FE Models of AT595 Explosion-Proof Vessel

As mentioned earlier, five FE models of the AT595 explosion-proof vessel were considered in this thesis to reach general recommendations regarding the creation of FE models for closed cylindrical vessels subjected to internal blast loads. Although the five FE models have different levels of details, the basic features of AT595 explosion-proof vessel were incorporated in all of them. All the models were subjected to an internal blast load of 8 kg of TNT. This blast loading is modeled through ConWep function available in LS-DYNA and is explained in detail in section 3.8.2.2.

3.7.1 Model 1

Model 1 is the most simplified version of the AT595 explosion-proof vessel when compared to all the five FE models. While this model is simple, it retains all the basic features of the AT595 explosion-proof vessel as shown in Figure 3.6.

Excluding the polymer-foam component, which is meshed with solid elements, the rest of the FE model comprises of shell elements to reduce the simulation time. The polymer-foam material at the end portion of cap is enclosed in a 0.012 m steel casing. There are a total of five shell gusset plates, each having thickness of 0.015 m, supporting the throttle plate in the one-eight FE model of the explosion-proof vessel.
The thickness of the shell in the cylindrical portion is 36.5 mm whereas the thickness of the shell in the cap portion is 20.5 mm due to the absence of the 90° basalt-plastic layer in the cap portion of the vessel. The inner steel lining in the cylindrical and cap portion of the vessel is incorporated in the shell layer. The combined material properties of basalt-plastic and steel were calculated, using the classical laminated plate theory [24], and assigned to the cylindrical and cap portion respectively. Table 3.12 depicts these combined properties. The density of the cylindrical portion of the vessel, upto 0.68 m from the center, was increased to accommodate the anti-fragment shield. Model 1 comprises of 35,122 elements and 32,385 nodes. The total CPU run time for this model is 9 minutes and 11 seconds.
Table 3.12: Material properties of the composite shell layer in Model 1 and Model 2

<table>
<thead>
<tr>
<th>Material Properties</th>
<th>Cylindrical Portion</th>
<th>Cap Portion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density ( (\rho) ), kg/m(^3)</td>
<td>2773.8</td>
<td>2773.8</td>
</tr>
<tr>
<td>Modulus of Elasticity in the Fiber Direction ( (E_1) ), GPa</td>
<td>24.25</td>
<td>23.80</td>
</tr>
<tr>
<td>Modulus of Elasticity across the Fiber Direction ( (E_2) ), GPa</td>
<td>50.92</td>
<td>46.50</td>
</tr>
<tr>
<td>Modulus of Rigidity ( (G_{12}) ), GPa</td>
<td>9.75</td>
<td>11.65</td>
</tr>
<tr>
<td>Poisson’s Ratio ( (\nu_{12}) )</td>
<td>0.15</td>
<td>0.15</td>
</tr>
</tbody>
</table>

3.6.1.1 Model Creation and Meshing

Model 1 was created and meshed in Altair HyperMesh. Initially temporary nodes were created in HyperMesh. These temporary nodes were joined by lines and arcs option, in the `geom` menu of HyperMesh, to form the skeleton for different parts of the model. The cylindrical portion of the vessel in Model 1 was meshed by using the drag option in the `2D` menu of HyperMesh. In the drag \( (x) \) direction, the model was meshed with fifty elements and along the arc there were thirty elements present as shown in Figure 3.7.
Next the cap portion of the vessel in Model 2 was meshed with the spin option present in the 2D menu of HyperMesh. The arc drawn in the cap portion was spun over the x axis and meshed with thirty elements, both along the arc and in the spin direction as shown in Figure 3.8. The next part to be drawn was the polymer-foam material. Using the planes and drag option present in the 2D menu of HyperMesh the polymer-foam material was meshed. The number of elements in the drag direction for the polymer-foam material was forty and the number of elements in the other directions was thirty each, Figure 3.9. The steel casing enclosing polymer-foam material has similar number of elements along its lines and arcs as that of the polymer-foam material.
Figure 3.8: The meshed cap portion of the vessel

Figure 3.9: Polymer-foam meshed with solid elements
After the steel casing, the throttle plate was meshed, again making use of the planes option in HyperMesh. The number of elements along the arcs region of the throttle plate was thirty, as against ten elements along the lines. The last feature to be created and meshed in Model 1 was the gusset plate. The gusset plates were meshed similar to the composite shell layer. All the three lines representing the triangular area of each gusset plate were meshed with six elements as seen in Figure 3.11. Duplicate nodes get created at the interface of two or more components. These duplicate nodes were merged into one by the edge option in the tools menu of HyperMesh to obtain the FE model of the AT595 explosion-proof vessel.

Figure 3.10: Meshed shell throttle plate
3.7.2 Model 2

Model 2 is another FE model variation of AT595 explosion-proof vessel as shown in Figure 3.12. The finer end-cap details are completely made of solid elements, while the rest of the FE model comprises of shell elements. The major objective of Model 2 is to investigate the need for a geometrically accurate model of the end-cap portion of the AT595 explosion-proof vessel. Figure 3.13 depicts a detailed view of the end-cap portion of Model 2. If no significant change in the peak strain values is observed in Model 2 when compared to Model 1 then the complex design of the end-cap portion can be replaced with a simpler design. This would not only reduce the simulation time but also the time required to model and mesh the complex end-cap profile.
The detailed end cap portion of the vessel is modeled with solid elements, while the rest of the model comprises shell elements similar to Model 1.
The thickness of the shell layer in the cylindrical and cap portion is similar to Model 1. The combined material properties of basalt-plastic and steel for the shell layers were evaluated and assigned similar to Model 1, and are listed in Table 3.13. The process of adjustment of density in the cylindrical shell layer, to accommodate the anti-fragment shield, is similar to Model 1. This model has a total of 10,479 elements and 12,069 nodes. The total time required to simulate the results for this model was 7 minutes and 24 seconds.

3.7.1.1 Model Creation and Meshing

Model 2 was created and meshed in Altair HyperMesh. Excluding the end-cap portion, the rest of the model was meshed similar to Model 1. For the end-cap portion, it was initially modeled in SolidWorks software. This model was then opened in HyperMesh as an IGES file. Initially, this IGES file was meshed using the planes option in the 2D menu of HyperMesh. Then using the spin option, these 2D elements were spun in the radial direction to obtain the solid elements as depicted in Figure 3.15. Along the radial direction the number of elements specified is eight.
Figure 3. 14: Meshed cap portion of the vessel

Figure 3. 15: End-cap meshed with solid elements
Then the rest of the model was created similar to Model 1. Again duplicate nodes get created at the interface of two or more components. These duplicate nodes were merged into one by the edge option in the tools menu of HyperMesh to obtain the FE model of the AT595 explosion-proof vessel.

3.7.3 Model 3

Model 3 is almost similar to Model 1 in the design aspect but the major difference is the element type used and the interpretation of the gusset plates. Model 3 depicts the most accurate representation of the gusset plates when compared to the AT595 explosion-proof vessel. This model is meshed entirely with solid elements as shown in Figure 3.16.

![Figure 3.16: FE Model 3 of the AT595 explosion-proof vessel](image)

Model 3 is created to study the effect of using solid elements, when compared to shell elements, on the accuracy of the results. The composite stacking sequence is simplified by grouping the 90°, +33° and -33° windings into three separate layers as listed in the section 3.4.2. The number of elements in the thickness direction for each of the basalt-plastic and steel layer is listed in Table 3.13. During analysis of Model 3, each layer was
defined with an individual material model unlike Model 1 and Model 2. Hence, this model can predict the separation of composite layers, which was impossible in the previous two models. A CONTACT_SURFACE_TO_SURFACE card was created in LS-DYNA to define the contact between the various material layers. By using this card it is assumed that the different material layers are bonded together. This card best describes the contact option for solid elements. There is a 0.0045 m thick steel lining housing the polymer-foam material for this model. The density of the inner steel lining, upto the throttle plate from the center, was increased to accommodate for the anti-fragment shield. Model 3 was finely meshed and comprises of 329,287 elements and 408,772 nodes. The total CPU run time for this model was 10 hours 13 minutes and 33 seconds.

<table>
<thead>
<tr>
<th>Layer No. (Starting from the outermost one)</th>
<th>Orientation / Material Type</th>
<th>Number of Elements along thickness direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>90° Basalt-Plastic</td>
<td>4</td>
</tr>
<tr>
<td>2</td>
<td>+33° Basalt-Plastic</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>-33° Basalt-Plastic</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>Steel</td>
<td>2</td>
</tr>
</tbody>
</table>

3.7.3.1 Model Creation and Meshing

The modeling and meshing of this model was achieved in HyperMesh. All the parts were initially meshed with 2D elements from the planes option of HyperMesh present in
the 2D menu. The cylindrical portion of the individual composite and steel layers were dragged along the longitudinal direction while the cap portions were spun in the radial direction. The polymer-foam material was also meshed by dragging the 2D elements in the longitudinal direction. The model has five solid gusset plates evenly distributed in the vessel contacted with the solid throttle plate. The solid mesh of throttle plate was generated by spinning the 2D elements in the radial direction. The steel casing enclosing the polymer-foam material was also created by the spin option in the 2D menu of HyperMesh. For all the parts the meshing was very fine.

3.7.4 Model 4

Model 4 can be called as the shell version of Model 3. The entire model, excluding the polymer-foam material, is constructed of shell elements as represented in Figure 3.17. Model 4 is created so as to keep all the features of Model 3 but at the same time reduce the overall simulation time.

![Figure 3.17: FE Model 4 of the AT595 explosion-proof vessel](image)
The shell gusset and throttle plates, along with the solid polymer-foam material and the steel lining enclosing it, were modeled similar to Models 1 and 2. The composite and the inner steel shell layers were created similar to Model 3 and are assumed as in section 3.4.2. CONTACT_AUTOMATIC_SURFACE_TO_SURFACE card present in LS-DYNA defines the contact between the shell material layers in the FE model. Similar to Model 3, the anti-fragment shield was accommodated in the inner steel layer for Model 4. This FE model comprises of 38,685 elements and 41,725 nodes. The total CPU run time required in simulating the results for this model was 18 minutes and 29 seconds.

3.7.4.1 Model Creation and Meshing

Model 4 was created and meshed similar to Model 1. The individual material layers were created and meshed similar to the shell layer of Model 1, except for the 90° basalt-plastic layer where only the cylindrical portion was created and meshed. Each layer is created in the thickness mid-plane. The polymer-foam material was meshed similar to Model 1. The steel casing enclosing polymer-foam material has similar number of elements along its lines and arcs as that of the polymer-foam material. The rest of the model, such as the throttle plate and gusset plate was meshed identical to Model 1. Wherever duplicate nodes were created in the model they were merged into one by the edge option in the tools menu of HyperMesh to obtain the FE model of the AT595 explosion-proof vessel.

3.7.5 Model 5

Model 5 is similar to Model 4 in all respects except the modeling of the gusset plate. A solid band of same material replaces the individual gusset plates as shown in Figure 3.18.
Figure 3.18: FE Model 5 of the AT 595 explosion-proof vessel

This was done so as to improve the stiffness of the vessel and also to make the vessel more rigid. The density of the solid band of gusset plate was made same as the equivalent density of the individual gusset plates in the aforementioned models. Model 5 comprises of 39,360 elements and 42,356 nodes. Time taken for the CPU to run the simulation for this model was 18 minutes and 38 seconds.

3.7.5.1 Model Creation and Meshing

The modeling and meshing of Model 5 was similar to Model 4, except for the gusset plate. Similar to the polymer-foam material the gusset plate was created by initially meshing through the planes option in the 2D menu of HyperMesh. All the three lines representing the triangular area were meshed with six elements. Then using the spin option these elements were spun over the x axis or in the radial direction to obtain a band or ring of solid elements as seen in Figure 3.19. The number of elements in the radial direction is thirty. Duplicate nodes get created at the interface of two or more
components. These duplicate nodes were merged into one by the edge option in the tools menu of HyperMesh to obtain the FE model of the AT595 explosion-proof vessel.

Figure 3.19: Solid band of elements depicting the gusset plate in Model 5

3.8 LS-DYNA Input Cards

The aforementioned five FE models are converted into files having text format so as to be processed by LS-DYNA. In these files the FE model and its definitions are divided into headings known as input cards. Each card represents some aspect of the FE model. The LS-DYNA input cards used to define the four FE models are explained in this section.

3.8.1 Material Models

In the FE models three types of material models are defined. The type and properties of these materials can be defined in the MAT cards available in LS-DYNA. Material type MAT3 is used to define the steel properties for the inner lining and enclosure over the
polymer-foam material for Model 3 and Model 4. MAT3 also characterizes the gusset and throttle plate properties for all the five models. MAT2 is used to represent the material model for the combined composite in Model 1 and Model 2, and the individual basalt-plastic layers in Model 3 and Model 4. The polymer-foam material in all the five FE models is defined by the MAT63 material model present in LS-DYNA.

3.8.1.1 MAT3

MAT3 card is named as *MAT_PLASTIC_KINEMATIC in the LS-DYNA input file. This material model essentially behaves like a bilinear elastic-plastic material and is used to model isotropic and kinematic hardening plasticity materials. This material model covers for the stress strain curve in the elastic region (until yield stress) and also in the plastic region (beyond yield stress). The stress-strain curve is assumed to be linear within each of these regions and hence made up of two straight lines. Such a simplified stress strain curve is shown in Figure 3.20 below. The slope of the stress-strain curve (from origin to the yield point) is defined as the Elastic Modulus of the material. While the slope of the stress-strain curve (beyond yield point) is defined as the Tangent Modulus for this material model. To determine the linear portion of the curve in the plastic region, a point that lies intermediate to the points corresponding to the ultimate stress and failure stress values on the stress-strain curve is selected so as to achieve a reasonable value for the Tangent Modulus.
Figure 3.20: Stress-strain curve for a plastic-kinematic material

This material model can be used for beam, shell and solid elements, and is cost effective. MAT3 card is defined in the LS-DYNA input file as shown below,

*MAT_PLASTIC_KINEMATIC
$HETNAME MATS 11mat_steel
$--- MID RO E PR SIGY ETAN BETA
11 7850.02.0000E+11 0.350000000 100000.0
$--- SRC SRP FS VP
$--- MID RO E PR SIGY ETAN BETA

where,

MID Material identification number
RO Density (kg/m³)
E Modulus of Elasticity (N/m²)
PR Poisson’s Ratio

107
3.8.1.2 MAT2

MAT2 card is named as *MAT_ORTOTROPIC_ELASTIC in the LS-DYNA input file. This material model is used to define the elastic-orthotropic behavior of the composite layers. Since the basalt-plastic material is an orthotropic material, it is required to define the properties in three mutually perpendicular directions shown in Figure 3.21. The thickness of the orthotropic material is very small and hence plane-stress condition is assumed for these materials. Therefore the properties in the thickness direction (c) are assumed to be greater than or equal to those along (a) and across (b) the fibers for the composite.

![Figure 3.21: Element having orthotropic material property [8]](image)

This card can be used for models with solid or shell elements that are orthotropic in nature. Below is a sample of MAT2 card used in defining the 90° basalt-plastic layer for Model 4,
*MAT_ORTHOTROPIC_ELASTIC

<table>
<thead>
<tr>
<th>HISNAME MATS</th>
<th>5mat_b90</th>
</tr>
</thead>
<tbody>
<tr>
<td>$</td>
<td>$</td>
</tr>
<tr>
<td>$</td>
<td>$</td>
</tr>
<tr>
<td>$</td>
<td>$</td>
</tr>
<tr>
<td>$</td>
<td>$</td>
</tr>
<tr>
<td>$</td>
<td>$</td>
</tr>
</tbody>
</table>

where,

MID | Material identification number

RO | Density (kg/m³)

EA, EB & EC | Modulus of Elasticity in a, b and c direction (N/m²)

PRBA, PRCA & PRCB | Poisson’s Ratio for ba, ca and cb

GAB, GBC & GCA | Shear Modulus for ab, bc and ca (N/m²)

AOPT | Material axes option [-]

3.8.1.3 MAT63

MAT63 card is named as *MAT_CRUSHABLE_FOAM in the LS-DYNA input file. This card is used to define any basic foam material. Compression properties of the foam material are required to activate this card. There is a need to define the engineering stress versus volumetric strain curve in this card. Hence from the engineering stress-strain curve given by RFNC the volumetric strain values are derived and used in defining the polymer-foam material model. This card is defined in the LS-DYNA input file as shown below,
where,

MID Material identification number

RO Density (kg/m³)

E Modulus of Elasticity (N/m²)

PR Poisson’s Ratio

LCID Load curve id, defining yield stress versus volumetric strain

3.8.2 Boundary Conditions

Two sets of boundary conditions are defined for all the five FE models. First set deals with the constraints applied on the symmetry nodes and the second set comprises of the load definition.

3.8.2.1 Symmetry Constraints

The nodes along the symmetry planes are constrained as shown in Figure 3.22 for Model 4. The LS-DYNA card used to define constraints is *BOUNDARY_SPC_NODE. This card has the option of constraining a specified node or a set of nodes along the six degrees of freedom (three translational along the three coordinate axes x, y and z, and three rotational about these axes). Below is a sample of this card defined in the LS-DYNA input file,
where,

NID  Node identification number
CID  Coordinate system id
DOFX, DOFY, DOFZ  Translational constraint along the x, y and z axes
DOFRX, DOFRY, DOFRZ  Rotational constraint about the x, y and z axes

Figure 3. 22: Model 4 with the symmetry boundary conditions
Table 3.14: Symmetry boundary conditions along the three planes

<table>
<thead>
<tr>
<th>Boundary conditions for nodes in symmetry plane</th>
<th>DOFX</th>
<th>DOFY</th>
<th>DOFZ</th>
<th>DOFRX</th>
<th>DOFRY</th>
<th>DOFRZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Y Plane</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Y-Z Plane</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>X-Z Plane</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

3.8.2.2 Load Definition

The five FE models are subjected to internal blast loads. ConWep blast function embedded in the LS-DYNA code is utilized in defining the pressure profile of the internal blast load. Three sets of cards are used to define the ConWep blast function on the specified area or elements. First the location of the high explosive and its mass is to be specified. The LS-DYNA card used for this purpose is *LOAD_BLAST. This card defines an airblast function for the application of pressure loads due to ConWep. The *LOAD_BLAST card is defined in all the models as shown below.

*LOAD_BLAST

```
$ WGT XBO YBO ZBO TBO IUNIT ISURF
$--------1---------2---------3---------4---------5---------6---------7
  8    0    0    0    0    2     2
$ CFH CFL CFT CFP
$--------1---------2---------3---------4---------5---------6---------7
```

where,

- **WGT**
  - Equivalent mass of TNT

- **XBO, YBO, ZBO**
  - x, y & z-coordinate of point of explosion

- **TBO**
  - Time-zero of explosion

- **IUNIT**
  - Unit conversation flag

- **ISURF**
  - Type of blast (surface or air blast)
The mass of the explosive is 8 kg of TNT. The type of units used in defining the explosive and the blast profile, is specified in the IUNIT column. The ISURF column specifies if the explosion is either surface blast (then specify ‘1’ in the column) or air blast (then specify ‘2’ in the column). The *LOAD_BLAST is used in conjunction with the *LOAD_SEGMENT (or *LOAD_SEGMENT_SET or *LOAD_SHELL) card in LS-DYNA to define the blast pressure profile. The *LOAD_SEGMENT card applies the distributed pressure load on the elements specified in the card. The *LOAD_SEGMENT card is defined as shown below for all the models,

```
*LOAD_SEGMENT
$\text{HNAME LOADCOLS}$ 2LoadSegment_2
$\text{HCOLOR LOADCOLS}$ 2 1
$\text{LCID SF AT N1 N2 N3 N4}$
$\text{--------------------------}$
-2 -1.0 0.0 9953 9322 9233 9234
-2 -1.0 0.0 9952 9321 9322 9953

where,

- \text{LCID} \quad \text{Load curve id}
- \text{SF} \quad \text{Load curve scale factor}
- \text{AT} \quad \text{Arrival or birth time of pressure}
- \text{N1, N2, N3, N4} \quad \text{Node numbers of the element on which the (blast) pressure acts}
```

By incorporating “-2” under the LCID column of the *LOAD_SEGMENT card, LS-DYNA voluntarily invokes the ConWep blast pressure profile on the specified segment or elements during the simulation. Load curve multipliers may be used in the SF column to increase or decrease the pressure profile but the time values are not scaled. The
activation time, AT, is the time during the simulation that the ConWep blast pressure should begin to act. If the element on which the blast load is supposed to act is triangular, then the node number N3 is repeated for N4. Hence the *LOAD_SEGMENT card can be used for quadrilateral, as well as triangular elements.

A minimum of two load curves are defined, even if unreferenced, in the model when the ConWep blast function is utilized. The *DEFINE_CURVE present in LS-DYNA is made use off in defining the load curves. The format of this card is depicted below,

```
*DEFINE_CURVE
$HNAME CURVES 2curve2
$HNCOLOR CURVES 2 1
$HNCURVE 1 3 curve2
$ LCID SIDR SFA SFO OFFA OFFO DATTYP
$--------1--------2--------3--------4--------5--------6--------7
 2 0 1.0 1.0 0.0 0.0 0
$ A1 0.0 0.0
$--------1--------2--------3--------4--------5--------6--------7
 0.0 1.0
 .
 7.0 7.0
 8.0 8.0
```

where,

- **LCID**: Load curve id
- **SIDR**: Stress initialization by dynamic relaxation
- **SFA, SFO**: Scale factor for abscissa & ordinate values of the curve
- **OFFA, OFFO**: Offset for abscissa & ordinate values of the curve
- **A1**: Abscissa (x) values of the curve
- **O1**: Ordinate (y) values of the curve

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
When the scale factors (SFA & SFO) and offset values (OFFA & OFFO) are defined in the *DEFINE_CURVE card, then the new abscissa and ordinate values are given as,

Abscissa value = SFA*(Defined value + OFFA)

Ordinate value = SFO * (Defined value + OFFO)

The above set of cards is used in defining the loading condition of all the five models.

Figure 3.23 shows the pressure acting on the elements of the FE Model 4.

![Figure 3.23: Model 4 with the pressure loading condition](image)

3.8.3 Contact Algorithms

All the models comprises of more than one component, hence contact definitions are needed between the different parts either by merging the common nodes between two or more components, or by defining the contact cards in LS-DYNA.
Model 1 comprises of only one shell layer to define the basalt-plastic composite material. Also there is no steel lining enclosing the polymer-foam material, hence no contact cards are defined for this model. The common nodes of different components are merged to obtain the contact definitions. Similarly Model 2 also does not contain any contact cards as even this model has only one shell layer to define the basalt-plastic composite material. Even though the polymer-foam material is enclosed within a steel liner for this model, since both these components are modeled with solid elements the common nodes of these components are merged to obtain the contact definitions.

Model 3 is a pure solid element based FE model. The basalt-plastic composite and steel layers are differentiated as individual components and hence a contact card namely *CONTACT_SURFACE_TO_SURFACE is defined in LS-DYNA for this model. This contact card is only used to define contact definitions between the individual material layers. For the rest of the model the common nodes are merged to obtain the contact definitions. The format of *CONTACT_SURFACE_TO_SURFACE card is as shown below.

```
*CONTACT_SURFACE_TO_SURFACE
$ElHNAME GROUPS 1steeltobasalt
$ElHCOLOUR GROUPS 1 15
$     SSID  MSID  SSTYP  HSTYP  SEOXID  MBOXID  SPR  HPR
$-------1---------2---------3---------4---------5---------6---------7---------8
$     6 2 3 3
$     FS  FD  DC  VC  VDC  PENCHK  BT  DT
$-------1---------2---------3---------4---------5---------6---------7---------8
$     SF  SFM  SST  HST  SFST  SFRT  F5F  VSF
$-------1---------2---------3---------4---------5---------6---------7---------8
```
where,

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSID</td>
<td>Slave segment id</td>
</tr>
<tr>
<td>MSID</td>
<td>Master segment id</td>
</tr>
<tr>
<td>SSTYP</td>
<td>Slave segment type</td>
</tr>
<tr>
<td>MSTYP</td>
<td>Master segment type</td>
</tr>
</tbody>
</table>

The *CONTACT_SURFACE_TO_SURFACE* contact card is used as the interface elements are solid and also this is the simplest card available to define contacts for implicit analysis. It comprises of three mandatory rows. The first row is used to define the slave and master segments of the contact. The second row is used if there is a need to define the coefficient of friction values between the interfaces. If any scale factors are to be utilized then the third row of the card is applied. In the case of Model 3 only the first row is utilized. While defining the contacts between two components, master component is taken as the one, which experiences the blast pressure first and the slave component is the one that gets affected due to the master component. In the case of contact definition between the inner steel lining and the -33° basalt-plastic layer over it, the inner steel lining is taken as the master as the blast pressure first hits this layer and the -33° basalt-plastic layer is the slave because this layer is affected only due to the movement of the inner steel lining. Table 3.15 gives the list of contact interfaces and the master and slave components for these contacts.
Table 3. 15: Characteristics of the contact cards used

<table>
<thead>
<tr>
<th>Contact Interface Between</th>
<th>Master</th>
<th>Slave</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel lining &amp; -33° basalt-plastic</td>
<td>Steel lining</td>
<td>-33° basalt-plastic</td>
</tr>
<tr>
<td>-33° basalt-plastic &amp; +33° basalt-plastic</td>
<td>-33° basalt-plastic</td>
<td>+33° basalt-plastic</td>
</tr>
<tr>
<td>+33° basalt-plastic &amp; 90° basalt-plastic</td>
<td>+33° basalt-plastic</td>
<td>90° basalt-plastic</td>
</tr>
<tr>
<td>Anti-fragment shield &amp; -33° basalt-plastic</td>
<td>Ant-fragment shield</td>
<td>-33° basalt-plastic</td>
</tr>
<tr>
<td>Steel lining &amp; Polymer-foam&lt;sup&gt;§&lt;/sup&gt;</td>
<td>Steel lining</td>
<td>Polymer-foam</td>
</tr>
</tbody>
</table>

<sup>§</sup> This contact definition is only for Model 4.

For Model 4 and Model 5, which are mainly meshed with shell elements, the contact card *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE* is utilized similar to Model 3. This contact card is specifically used since the material layers are made of shell elements and these shell elements are created along the mid plane of the material i.e., there are gaps between the material layers. This contact card does not neglect these gaps but assumes material to be present. The format of this card is similar to *CONTACT_SURFACE_TO_SURFACE* card. Table 3.15 can also be used for this card for defining the master and slave components. There is an extra interface definition for Model 4, between the polymer-foam material and the steel lining enclosing it.

3.8.4 Property Definitions

These cards give the overall property of the component, such as if the component comprises off solid or shell elements, the material type, nodes forming the element, etc. The *NODE* card is used to define the x, y and z coordinates of the nodes present in the FE model. The format of this card is as shown below,
where,

NID           Node id
X, Y, Z      The global coordinates of the specified node

For LS-DYNA to know if the specified element is shell or solid, there is a need to define the *ELEMENT_SHELL or *ELEMENT_SOLID card. The former card is used if the element type is shell and the later for solid elements. The nodes associated with each element are defined in these cards. The format of these cards are as shown below,

*ELEMENT_SHELL

$ EID  PID  N1  N2  N3  N4
$---1----2----3----4----5----6----7----8
  1  2 257 260 262 256
  2 2 256 262 263 255
  .
  13 2 238 268 229 230
  14 2 268 266 228 229

*ELEMENT_SOLID

$ EID  PID  N1  N2  N3  N4  N5  N6  N7  N8
$---1----2----3----4----5----6----7----8----9----10
  65925 9 69616 69631 69632 69617 70337 70352 70353 70338
  65924 9 69615 69630 69631 69616 70336 70351 70352 70337
  .
  65913 9 69604 69619 69620 69605 70325 70340 70341 70326
  65912 9 69603 69618 69619 69604 70324 70339 70340 70325

119
where,

EID  Element id

PID  Part id

N1, ..., N8  Node id's comprising of an element

To define the sectional properties of the shell and solid elements, *SECTION_SHELL and *SECTION_SOLID cards are respectively used. The general *SECTION_SHELL card is as shown below,

*SECTION_SHELL
$HEXNAME PROPS 1shell_b90
$SECID ELFORM SHRF NIP PROPT QR/IRID ICOMP SETYP
$---1---2---3---4---5---6---7---8
1 0 1.0 2 0.0
$T1 T2 T3 T4 NLOC AREA
$---1---2---3---4---5---6---7---8
0.016 0.016 0.016 0.016

where,

SECID  Section id

ELFORM  Element formulation options

SHRF  Shear correction factor

NIP  Number of through thickness integration points

T1, T2, T3, T4  Shell thickness at nodes N1, N2, N3 and N4

For all the *SECTION_SHELL cards the default Belytschko-Lin-Tsay shell element formulation present in LS-DYNA is utilized. The number of integration points in the thickness direction of any shell layer is taken as two. For the *SECTION_SOLID card the constant stress solid element formulation type is used, which is again the default
parameter in LS-DYNA for this card. The general format of the *SECTION_SOLID card is as shown below,

```
*SECTION_SOLID
$HNAME PROPS 7solid_foam
$SECID ELFOM AET
$----1----2----3
 7 1
```

The final card in this section is the *PART card. This card is used to define the information of a particular component or part, i.e., the material information and the section properties. The general format of this card present in all the four FE models is as given below,

```
*PART
$HNAME COMPS 2b90
$HCOLOR COMPS 2 6
$ PID SECID MID EOSID HGID GRAV ADPOPT THID
$----1----2----3----4----5----6----7----8
  2  1  9
```

where,

<table>
<thead>
<tr>
<th>PID</th>
<th>Part id</th>
</tr>
</thead>
<tbody>
<tr>
<td>SECID</td>
<td>Section id</td>
</tr>
<tr>
<td>MID</td>
<td>Material id</td>
</tr>
</tbody>
</table>

3.8.5 Control Cards

Control cards are optional cards in an LS-DYNA input file and can be used to change the defaults, activate solution options such as mass scaling, adaptive meshing, and an implicit solution. A control card defines the properties such as termination time, time step controls, warpage angle for shell, hourglass effect, etc.

Two types of control cards are used for the five FE models. First is the *CONTROL_TERMINATION card, which specifies the termination of the analysis after
a given time. The termination time for all the four FE models is taken as 3 ms. The second card is *CONTROL_TIMESTEP; this card sets the structural time step size. The format of these two cards are listed below,

\begin{verbatim}
*CONTROL_TERMINATION
$ ENDTIM ENDCYC DTMIN ENDENG ENDMAS
$-----------------1-----------------2-----------------3-----------------4-----------------5
  0.003

*CONTROL_TIMESTEP
$ DTINIT TSSFAC ISDO TSLINT DT2MS LCTM ERODE MSIST
$-----------------1-----------------2-----------------3-----------------4-----------------5-----------------6-----------------7-----------------8
  0.0    0.67
\end{verbatim}

where,

ENDTIM  Termination time
DTINIT  Initial time step size
TSSFAC  Scale factor for computed time step

For all the FE models, by giving a zero value under the DTINIT column, we allow LS-DYNA to determine the initial time step size. The default scale factor in LS-DYNA is 0.9, but in case of high explosive loading, this default value is reduced to 0.67.

3.8.6 Database Cards

The database definitions are optional but are necessary to obtain the output files comprising the results information. *DATABASE_BINARY_D3PLOT card is used to obtain the complete output states of the analyses. The time interval between the outputs, DT/CYCL, is taken as 3E-5. This card also contains the plotting information of the three dimensional geometry of the model. Also *DATABASE_EXTENT_BINARY card is used to obtain and write the strain results and plots in the D3PLOT file.
3.9 Results

RFNC-VNIIF conducted experimental testing and computational simulation on the AT595 explosion-proof vessel; these results [1, 22, 23] are compared with the FEA results obtained from the five models. The RFNC-VNIIF used a two-dimensional axisymmetric explicit code known as DRAKON to do their computational simulations. The DRAKON uses Eulerian gas-dynamic code to determine the pressure loading on the inside of the vessel. The strain at the central portion and the peak strains throughout the cylindrical portion of the vessel are compared. RFNC-VNIIF predicted a peak strain of 2.36% at 0.432 ms computationally and experimentally they obtained a peak strain of 2% at 0.55 ms. Both these peak strains occurred at the center of the cylindrical region of the vessel. The peak strains of the five FE models are tabulated in the Table 3.16 along with the RFNC results for comparison. Similar to the RFNC results, the peak strain for the FE models also occurred in the central region of the cylindrical portion of vessel.

Table 3.16: Peak strain results of the FE models and RFNC

<table>
<thead>
<tr>
<th>Model</th>
<th>Peak strain (%)</th>
<th>Time (ms)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>2.00</td>
<td>0.55</td>
</tr>
<tr>
<td>DRAKON</td>
<td>2.36</td>
<td>0.43</td>
</tr>
<tr>
<td>Model 1</td>
<td>2.13</td>
<td>0.51</td>
</tr>
<tr>
<td>Model 2</td>
<td>2.09</td>
<td>0.48</td>
</tr>
<tr>
<td>Model 3</td>
<td>1.58</td>
<td>0.42</td>
</tr>
<tr>
<td>Model 4</td>
<td>2.22</td>
<td>0.51</td>
</tr>
<tr>
<td>Model 5</td>
<td>2.27</td>
<td>0.54</td>
</tr>
</tbody>
</table>
Model 5 reported the highest peak strain among the FE models but less than what DRAKON predicted. The least peak strain was obtained from Model 3, the complete solid model. This can be attributed to the fact that the shell models are more elastic in nature when compared to the rigid solid Model 3. The peak strains of the five FE models at eleven different locations on the cylindrical portion, from the center towards the cap region, is predicted and plotted with the RFNC results as shown in Figure 3.24. The plot also depicts the average throttle plate location.

![Graph showing peak circumferential strains comparison](image)

**Figure 3.24:** Comparison of peak circumferential strains at different locations

Average error is defined for each model relative to the experimental results by comparing the area under the curves shown in Figure 3.24. Using Simpson's 3/8 rule [Appendix III], the area under each curve for the four FE models is calculated. The error is defined using the following equation,
\[ \text{Error} = \left[ \frac{I_m}{I_e} - 1 \right] \times 100\% \]  

where,

\[ I_m = \text{area under the FEA model curve} \]

\[ I_e = \text{area under the experimental curve} \]

Table 3.17: Error measure of the FE models with respect to experimental results

<table>
<thead>
<tr>
<th>Model</th>
<th>Area under the curve</th>
<th>Error (%)</th>
<th>CPU Time (min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>0.0079</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Model 1</td>
<td>0.0093</td>
<td>17.8</td>
<td>9.2</td>
</tr>
<tr>
<td>Model 2</td>
<td>0.0090</td>
<td>14.4</td>
<td>7.4</td>
</tr>
<tr>
<td>Model 3</td>
<td>0.0063</td>
<td>19.7</td>
<td>613</td>
</tr>
<tr>
<td>Model 4</td>
<td>0.0093</td>
<td>17.3</td>
<td>18.5</td>
</tr>
<tr>
<td>Model 5</td>
<td>0.0091</td>
<td>15.4</td>
<td>18.6</td>
</tr>
</tbody>
</table>

Table 3.17 summarizes the modeling error and efficiency results. Efficiency is defined as the CPU time required to complete the analysis. From the above results it is clear that the best model in terms of error is Model 2. The efficiency of this model is also best of the lot. Model 3 is the worst of the lot in terms of both efficiency and error. The error in Model 5 is close to Model 2 and the efficiency is also good. When compared to Model 3, Model 5 is lot simpler to model.

The error for all the five FE models is within 20% of the experimental results. The detailed results of each individual model is as given below,
3.9.1 Model 1

Model 1 generally produces higher strains than the experimental results, except near the gusset plate location. This behavior may be explained by the method of representing the gusset plates as a solid band or ring of elements that increases the stiffness at this location. FEA of Model 1 predicts a peak strain of 2.13% at 0.51 ms in the central region of the vessel. As the composite layer is modeled as a single shell layer, the separation between the layers cannot be predicted in this model. Figure 3.25 shows the maximum circumferential strain contours on the cylindrical portion of the explosion-proof vessel.

Figure 3.25: The circumferential strain contour in Model 1

A plot of circumferential strain versus time is plotted for the model at the central location and compared with the DRAKON result as shown in Figure 3.26. The initial peak of the Model 1 curve is almost same as DRAKON curve but in the later stages there
is a considerable shift in the phase of the curve. This can be attributed to two reasons, one
due to the elastic nature of the shell elements and two since all the material layers are
incorporated as a single shell layer.

![Graph showing circumferential strain of Model 1 and DRAKON at the central region]

Figure 3.26: Circumferential strain of Model 1 and DRAKON at the central region

3.9.2 Model 2

From the FEA of Model 2 we get a peak strain of 2.09% at 0.48 ms in the central
region of the vessel. The circumferential peak strain values at the center of the vessel and
before the throttle plate do not coincide with the experimental results. Except for these
regions, results closely follow the experimental ones. Similar to Model 1, this model
cannot predict separation between the composite layers. Figure 3.27 shows the maximum
circumferential strain contours on the cylindrical portion of the explosion-proof vessel.
A plot of circumferential strain versus time is plotted for the model at the central location and compared with the DRAKON result as shown in Figure 3.28. The Model 2 plot is similar to Model 1 due to the same reason that the elements are elastic in nature and also all the material layers are incorporated as a single shell layer. From this it can be postulated that the fine end-cap detail does not affect the peak strain in the central region of the vessel. Hence the tedious process of creating the end-cap detail can be abandoned.
3.9.3 Model 3

A peak circumferential strain of 1.58\% at 0.42 ms is obtained from the FEA of this model. While an extremely large number of elements are used to create this model, it has the most deviation from experimental results in general. Also the time required to run this model is huge compared to the other three FE models. Due to the individual modeling of the material layers, this model can predict the separation of the composite layers. Figure 3.29 shows the maximum circumferential strain contours on the cylindrical portion of the explosion-proof vessel.
A plot of circumferential strain versus time is plotted for the model at the central location and compared with the DRAKON result as shown in Figure 3.30. Although the prediction of peak strain is poor with this model, the rest of the curve follows the DRAKON result with a minor phase shift. The number of peaks obtained is also similar to the DRAKON result. This can be attributed to the more rigid nature of the solid elements. This model has a very high computational cost not only due to its high run time, but also due to the time consuming creation of this fine model. As shown earlier the error evaluated is also very high for this model. Hence this model is not found appropriate for the iterative optimization process.
3.9.4 Model 4

Model 4 predicts a peak circumferential strain of 2.22% at 0.51 ms at the central region of the vessel. Comparing the peak strain results with the results of RFNC-VNIIF as shown in Figure 3.24, we notice that the peak strain curve of Model 4 at the central portion of the vessel is close to the experimental peak strain, but after sometime it starts deviating from the RFNC results. Similar to Model 3, this model takes into account the discrete representation of the steel and composite layers, hence the separation between layers can be predicted. Figure 3.31 shows the maximum circumferential strain contours on the cylindrical portion of the explosion-proof vessel.
A plot of circumferential strain versus time is plotted for the model at the central location and compared with the DRAKON result as shown in Figure 3.32. This model predicts the peak strain on the central region similar to the DRAKON. But after the initial peak there is major phase shift in the circumferential strain of this curve when compared to the DRAKON curve. This can again be attributed to the elastic nature of the shell layers. Also the type of contact used between the material layers for this model affects the circumferential strain curve. The contact used only acts as the layers are placed one above the other but not firmly bonded together. As the primary objective of this study is to reduce the peak strain present in the cylindrical portion of the vessel by an iterative optimization process, the contact card does not posses much of a problem.
3.9.5 Model 5

Model 5 predicts a peak circumferential strain of 2.27% at 0.54 ms at the central region of the vessel. The peak strain curve of this model initially follows the peak strain curve for Model 4. At the throttle plate the peak strain values for Model 5 deviate more towards the experimental result, provided by RFNC, than the Model 4 values. Similar to Model 3 and Model 4, this model takes into account the discrete representation of the steel and composite layers, hence the separation between layers can be predicted. Figure 3.33 shows the maximum circumferential strain contours on the cylindrical portion of the explosion-proof vessel.
A plot of circumferential strain versus time is plotted for the model at the central location and compared with the DRAKON result as shown in Figure 3.34. This curve behaves similar to the one predicted by Model 4, but with a slightly less amount of noise at the peak strain value. This is due to the solid band of gusset plate, which makes the vessel more rigid. From the earlier results it was seen that the error for this model was very close to Model 2 and also the time required to simulate this model is good for an iterative optimization process. Hence Model 5 is chosen for further study to carry out the optimization process.
3.10 Conclusion

There is a need for proposing finite element approaches that can be useful in modeling cylindrical composite vessels subjected to internal blast loads. This paper presents five possible models. All models use a simplified ConWep code to calculate the internal blast pressure instead of the more time-consuming Eulerian-Lagrangian codes. From the results of all the five FE models it is seen that the error is within 20% when compared to the experimental results done by RFNC. The following is the summary of the results obtained in this chapter,

- Results show that aspect ratio and element size used in shell elements of Model 1, Model 2, Model 4 and Model 5 is reasonable.
• Results however show that Model 3, which uses brick elements, is the least accurate even though it has the largest number of elements. This may indicate that a denser mesh is needed in this case, which can be disadvantageous as Model 4 that uses shell elements is sufficiently accurate.

• Various representations of the end cap region produce no significant difference in strains in the cylindrical portion of the vessel.

• ConWep function is successful in modeling the internal blast loads, thus reducing the need for more computationally expensive codes.

Results show that Model 5 can adequately describe behavior in the cylindrical portion of the vessel. Model 5 provides the best combination of accuracy, computational efficiency and modeling simplicity. It can be used as a base model for carrying out an iterative optimization of the AT595 explosion-proof vessel.
CHAPTER 4

OPTIMIZATION

Optimization can be defined as the procedure for achieving the most desirable design of any product. Optimization is predominantly iterative and hence a series of operation are performed sequentially to obtain the optimal result. In the past, optimization of a product was done manually. This made the optimization process very tedious and time consuming, and hence the field was very limited. But with the advancement of technology and advent of computers, there is more scope available for optimization. With regard to explosion-proof vessels, the field of optimization is fairly new and the research available is minimal, as was noticeable from the literature survey listed in Chapter 1. Optimization can be an important tool for the explosion-proof vessels to minimize the overall mass of the vessel, which can be an essential aspect for the mobility of the vessel. The structural integrity of the vessel can be improved with the help of optimization technique. This can lead to increase in the ability of the vessels to withstand higher explosive loading. The objective of this chapter is,

- To propose and validate an optimization technique for the explosion-proof vessels.
- To reduce the peak strain produced in the AT595 explosion-proof vessel due to internal blast loads.
Model 5, shown in Figure 4.1, is taken as the base FE model for conducting the optimization study. As detailed in Chapter 3, Model 5 is a FE version of the AT595 explosion-proof vessel and is most suitable for the iterative optimization procedure due to its combination of accuracy, computational efficiency and modeling simplicity.

![Diagram of FE model](image)

Figure 4.1 FE model, Model 5, of the AT595 explosion-proof vessel

4.1 Definition of the Optimization Problem

The primary step in optimization is to identify the objective of the problem and to define the parameters/variables that can alter this objective to an optimum level.

4.1.1 Objective Function

The objective of this optimization study is to reduce the peak strains produced in the cylindrical portion of the AT595 explosion-proof vessel due to the internal blast load. From experimental and computational results, as shown in the plot in Figure 4.2, it is observed that the maximum circumferential strain consistently occurs at the center of the vessel. An element in the central region of the vessel, on the outer layer, Figure 4.3, is
identified to output the circumferential strain values and the objective function of the optimization code is to reduce the peak circumferential strain at this element.

Figure 4.2: Comparison of peak circumferential strain at different locations on the cylindrical portion of the vessel

Figure 4.3: Element in the central region from which the results are outputted
4.1.2 Design Variables/Parameters

In order to achieve the aforementioned objective function, three design variables are identified for Model 5. They are,

- \( X_1 \): the thickness of the \(-33^\circ\) basalt-plastic composite layer throughout the vessel. This design variable is related to two conditions, first being that the thickness of the \(+33^\circ\) basalt-plastic layer should be equal to the \(-33^\circ\) basalt-plastic layer. The second is that the overall thickness of the basalt-plastic composite is always equal to 0.032 m as represented below,

\[
T_{90} = 0.032 - 2X_1 \tag{4.1}
\]

where,

\[
T_{90} = \text{Thickness of the 90°basalt-plastic layer (m)}
\]

- \( X_2 \): the second design variable is the angle of orientation of the fibers in the \(33^\circ\) basalt-plastic layer. This parameter is bound by the condition that the angle of orientation of the fibers in the \(-33^\circ\) basalt-plastic layer is numerically equal to 33 basalt-plastic layer but with the negative sign. The said condition is represented in Equation 4.2.

\[
A_{-33} = -X_2 \tag{4.2}
\]

where,

\[
A_{-33} = \text{Angle of orientation of the fibers in the -33° basalt-plastic layer (°)}
\]

- \( X_3 \): the position of the gusset and throttle plate in the cylindrical portion of the vessel, is the third design variable. This variable is dimensionless and represents the number of elements the throttle plate has moved along the axial direction. The length of each element in the axial direction is 0.02445 m. A positive value of \( X_3 \)
indicates a movement of the plates towards the cap portion of the vessel while a negative number implies a movement towards the center of the vessel. This variable is subjected to the condition that the gusset and throttle plates remain within the cylindrical portion of the vessel.

4.1.3 Constraints for the Design Variables

Limits are set for the design variables, known as constraints, which help the optimization code to concentrate the guesses of the variables within a specified region and hence reach the optimum objective function value quicker. There are a total of six constraints, two for each variable, and are represented by the three equations listed below,

\[ 0.003m < X_1 < 0.012m \]
\[ 10^\circ < X_2 < 80^\circ \]
\[ -10 \leq X_3 \leq 10 \]

The movement of the plates, variable \( X_3 \), is restricted to maximum of ten elements on either side of the actual location due to the presence of the inert explosive casing, shown in Figure 4.4, inside the vessel which houses the high explosive (HE) charge.
Penalty terms are introduced for each limit of the design variable such that if any of the constraints are violated i.e., if any of the guesses of the design variables fall outside the above set limits, then the objective function is not computed [Appendix IV]. The modified objective function due to the presence of the penalty terms is as given in Equation 4.6.

\[ F = Af + \sum_{i=1}^{m} \Omega_i \]  

If \( g_i \leq 0 \) \( A = 0 \)

\[ \Omega_i = Rg_i^2 + B \]

If \( g_i > 0 \) \( A = 1 \)

\[ \Omega_i = 0 \]
The variable $f$ is the maximum circumferential strain values at the center of the vessel. The constraint functions $g_i$ correspond to the upper and lower limits in Equation 4.3, Equation 4.4 and Equation 4.5 respectively. $R$ and $B$ are penalty parameters, whose values are $10^{12}$ and 600,000 respectively. The variable $A$ is introduced in the formulation to avoid calculating the objective function when a constraint is violated, as the problem is computationally demanding.

4.2 Organization of the FE code

As outlined in Chapter 3, the FE model created in LS-DYNA is in the form of a text file. The information pertinent to the model is divided into cards in this text file. Changes in the FE model are possible by changing the information in these cards. For the optimization study of the AT595 explosion-proof vessel, the LS-DYNA file of Model 4 is bifurcated into the fixed code and the variable code.

4.2.1 Fixed Code

This code comprises of all the features of the FE model that remain constant irrespective of the change in values of the design variables. The FE code related to polymer-foam model, cap portion of the inner steel lining, shown in Figure 4.5, and some of the LS-DYNA cards such as the material cards, part cards, database cards, etc are part of this code. This code is obtained from the LS-DYNA file created for Model 4.
4.2.2 Variable code

This portion of the FE code combines all parts that depend on the design variables, such as the basalt composite layers, cylindrical portion of the inner steel lining and, gusset and throttle plates, depicted in Figure 4.6. This code also includes some of the LS-DYNA cards accompanying these parts such as the boundary conditions, contact definition, and the CONWEP blast loading. A program is written in MATLAB to create this code. Sections 4.2.2.1 to 4.2.2.5 defines this MATLAB program and explains the creation of various LS-DYNA cards in the order the optimization program is written.
4.2.2.1 Creation of -33° Basalt-Plastic Layer

In the MATLAB optimization code, for creating the composite layers, the original location of the -33° basalt-plastic layer is taken as the base from which the new layers are created. Hence the initial thickness variable $X_i$ is reduced to Equation 4.7, where 0.008 m is the original thickness of the -33° basalt-plastic layer. As the shell layer is created along the thickness mid-plane, the thickness variable is divided by two in Equation 4.7.

$$X_1 = \left\lfloor \frac{X_i - 0.008}{2} \right\rfloor$$  \hspace{1cm} (4.7)
For the cap portion of the vessel, due to the change in the thickness variable $X_i$, there is an increase or decrease in the increment/decrement of the angle, $d\theta$, of the cap as shown in Figure 4.7. This angle increment/decrement is given by the Equation 4.8 [Appendix V].

$$d\theta = \sin^{-1}\left(\frac{d}{R_i}\right) - \sin^{-1}\left(\frac{d}{R_i + X_i}\right)$$ (4.8)

where,

- $d$ = The constant longitudinal distance of the cap portion of the vessel (m)

- $R_i$ = Radius of the -33° basalt-plastic layer (degrees)

For the rest of the sections, the elements or nodes along the longitudinal direction or length of the cylinder are called as rows and the circumferential or radial direction is
interpreted as columns. For all the layers built through the MATLAB optimization code, initially the nodes are created. The coordinates of the first node created is listed in the Table 4.1 and represented in the Figure 4.8. The radial distance between center of the vessel and original -33° basalt-plastic shell layer, $r_i$, is 0.464 m. This node is taken as the base for the creation of rest of the nodes.

Table 4.1: Coordinates of the first node

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>0</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>$r_i + X_i$</td>
</tr>
</tbody>
</table>

Figure 4.8: The first node created for the -33° basalt-plastic layer from the optimization code
The first step in the creation of the -33° basalt-plastic layer through the MATLAB optimization code is to create the nodes. Initially the first row of nodes, as shown in Figure 4.9, in the cylindrical portion of the vessel is created. The coordinates of these nodes are obtained by the equations listed in Table 4.2. The distance between each node in the row direction is 0.02445 m and is represented in the X-coordinate, while \( n \) represents the node number.

Table 4.2: Coordinates of the first row of nodes in the cylindrical portion

<table>
<thead>
<tr>
<th></th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-coordinate</td>
<td>( xc(n) = xc(n-1) + 0.02445 )</td>
</tr>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = 0 )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = zc(n-1) )</td>
</tr>
</tbody>
</table>

Figure 4.9: The first row of nodes created in the cylindrical portion of the vessel
Figure 4.10: The first row nodes created in the cap portion of the vessel

The next step is the creation of the first row of nodes in the cap portion, shown in Figure 4.10. From Figure 4.11, $\theta^*$ is the angle of the segments formed in the cap portion due to each element and is given by the Equation 4.9.

Figure 4.11: Angle of the segment formed by each element in the cap portion
\[
\theta^* = \left[ \frac{(\theta_i - d\theta) \pi}{180} \right] \frac{N}{N}
\]

where,

\[\theta_i\] = Angle subtending the cap radius in the row direction (degrees)

\[d\theta\] = Angle increment/decrement due to the variable \(X_i\) (degrees)

\[N\] = Number of elements the cap portion is divided into.

The coordinates of these nodes are obtained from the equations represented in Table 4.3. The variable \(m\) is the increment in the angle.

<table>
<thead>
<tr>
<th align="left">X-coordinate</th>
<th align="left">(xc(n) = xc(n-1) + [(R_i + X_i) \sin(m\theta^*)])</th>
</tr>
</thead>
<tbody>
<tr>
<td align="left">Y-coordinate</td>
<td align="left">(yc(n) = 0)</td>
</tr>
<tr>
<td align="left">Z-coordinate</td>
<td align="left">(zc(n) = (R_i + X_i) \cos(m\theta^*) -</td>
</tr>
</tbody>
</table>
Figure 4.12: The rest of the nodes created for the cylindrical portion of the vessel

Then the remaining nodes, shown in Figure 4.12, in the cylindrical region of the vessel are created. The coordinates of these nodes are given by the equations represented in Table 4.4.

Table 4.4: Coordinates of the reminder nodes on the cylindrical portion

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(p) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = yc(1) - ((r_j + X_j) \times \sin[(j-1) \times (\pi / 2)] / k] )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = ((r_j + X_j) \times \cos[(j-1) \times (\pi / 2)] / k] )</td>
</tr>
</tbody>
</table>

where,

\[ p = 1 \text{ to number of columns or number of elements in the longitudinal direction} \]
\( j \) = Row number

\( k \) = Number of rows or number of elements in the circumferential direction

Finally the rest of the nodes in the cap portion, depicted in Figure 4.13 are created and the equations for the coordinates of these nodes are listed in Table 4.5.

**Table 4. 5: Coordinates of the reminder nodes in the cap portion**

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(p) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = zc(p) \cdot \sin[-(j-1) \cdot (\pi/2)/k] )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = zc(p) \cdot \cos[-(j-1) \cdot (\pi/2)/k] )</td>
</tr>
</tbody>
</table>
The last step in the creation of the -33° basalt-plastic layer is the generation of elements. The nodes are labeled as shown in Figure 4.14 in order to create an element. Initially the first row of elements in the cylindrical and cap portion of the vessel, shown in Figure 4.15 is created. The nodes surrounding the element are identified by the equations listed in Table 4.6.

![Figure 4.14: Order of nodes in an element](image)

Table 4.6: Identification of nodes surrounding an element for the first row

<table>
<thead>
<tr>
<th>Node 1</th>
<th>( N1(q) = T + q - 1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 2</td>
<td>( N2(q) = N1(q) + 1 )</td>
</tr>
<tr>
<td>Node 3</td>
<td>( N3(q) = N2(q) + g )</td>
</tr>
<tr>
<td>Node 4</td>
<td>( N4(q) = N3(q) - 1 )</td>
</tr>
</tbody>
</table>

where,

- \( T \) = Node id
- \( q \) = Element number
- \( g \) = Total number of nodes in the longitudinal direction for the vessel layer
The remaining elements, depicted in Figure 4.16, are generated and the equations required to identify the nodes surrounding an element is given in Table 4.7.

Table 4.7: Identification of nodes surrounding an element for rest of the layer

<table>
<thead>
<tr>
<th>Node</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 1</td>
<td>( N1(q) = T + u + j )</td>
</tr>
<tr>
<td>Node 2</td>
<td>( N2(q) = N1(q) + 1 )</td>
</tr>
<tr>
<td>Node 3</td>
<td>( N3(q) = N2(q) + g )</td>
</tr>
<tr>
<td>Node 4</td>
<td>( N4(q) = N3(q) - 1 )</td>
</tr>
</tbody>
</table>

where,

\[ u \quad = \quad \text{Node increment for the element (} = c \times j) \]

\[ c \quad = \quad \text{Total number of elements in the longitudinal direction for the vessel layer} \]
4.2.2.2 Creation of +33° Basalt-Plastic Layer

As mentioned earlier, the original -33° basalt-plastic layer is the taken as the base from which all the other composite layers are created. Hence Equation 4.10 is used to identify the location of the new +33° basalt-plastic layer. The angle increment/decrement, $d\theta$, of the overall angle, $\theta$, subtending the arc of the cap portion of the vessel in the row direction due to the thickness variable $X_1$ is given by the Equation 4.11. $R_2$ is the radius of the cap in the longitudinal or row direction.

$$X_1^* = [2X_1' + 0.008] + X_1'$$  \hspace{0.5cm} (4.10)

$$d\theta = \sin^{-1} \left( \frac{d}{R_2} \right) - \sin^{-1} \left( \frac{d}{(R_2 + X_1^*)} \right)$$  \hspace{0.5cm} (4.11)

Similar to the -33° basalt-plastic layer, for the creation of +33° basalt-plastic layer initially the nodes are created and then the elements. The pattern of creation nodes and
elements for this layer are also akin to -33° basalt-plastic layer. Figure 4.17 shows the meshed +33° basalt-plastic layer created from the MATLAB optimization code. The coordinates of the first node for this layer is listed in Table 4.8 and is taken as the base for the creation of rest of the nodes.

![Figure 4.17: The +33° basalt-plastic layer created from the MATLAB optimization code](image)

Table 4.8: Coordinates of the first node

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>0</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( r_i + X_i^* )</td>
</tr>
</tbody>
</table>

Then the first row of nodes in the cylindrical portion of the vessel is created. The coordinates of these nodes are obtained by the equations listed in Table 4.9. This is followed by the generation of first row of nodes in the cap portion, whose coordinates are obtained from the equations listed in Table 4.10. The rest of the nodes in the cylindrical
and cap portion of the $+33^\circ$ basalt-plastic layer are established one after the other and the
equations representing the coordinates of these nodes are listed in Table 4.11 and Table
4.12 respectively.

Table 4.9: Coordinates of the first row of nodes in the cylindrical portion

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>$x_c(n) = x_c(n-1) + 0.02445$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>$y_c(n) = 0$</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>$z_c(n) = z_c(n-1)$</td>
</tr>
</tbody>
</table>

\[
\theta' = \frac{(\theta_2 - d\theta) \cdot \pi}{180} \frac{1}{N} \tag{4.12}
\]

where,

$\theta_2$ = Angle subtending the cap radius in the row direction (degrees)

Table 4.10: Coordinates of the first row of nodes in the cap portion

<table>
<thead>
<tr>
<th>$x$</th>
<th>$x_c(n) = x_c(n-1) + [(R_2 + X'_1) \cdot \sin(m\theta')]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y$</td>
<td>$y_c(n) = 0$</td>
</tr>
<tr>
<td>$z$</td>
<td>$z_c(n) = (R_2 + X'_1) \cdot \cos(m\theta') -</td>
</tr>
<tr>
<td></td>
<td>$z_c(n) = (R_2 + X'_1) \cdot \cos(m\theta') +</td>
</tr>
</tbody>
</table>
Table 4.11: Coordinates for the remainder of the cylindrical portion

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(p) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = yc(1) - \left{ (r_1 + X_1^*) \sin[(j-1)\pi/2]/k \right} )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = \left{ (r_1 + X_1^*) \cos[(j-1)\pi/2]/k \right} )</td>
</tr>
</tbody>
</table>

Table 4.12: Coordinates of the remainder of cap portion

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(p) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = zc(p) \sin(\pi/2)/k )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = zc(p) \cos(\pi/2)/k )</td>
</tr>
</tbody>
</table>

After the creation of nodes, elements are created, initially the first row followed by the rest of the elements for the layer. The equations used for this purpose are akin to the ones used for -33° basalt-plastic layer and are listed in Table 4.6 and Table 4.7.

4.2.2.3 Creation of the 90° Basalt-Plastic Layer:

For the 90° basalt-plastic layer, it is only required to create the cylindrical portion. The procedure for creation of this portion is similar to the previous two composite layers. The Equation 4.13 identifies the new location of this layer, due to the thickness variable \( X_i \).

\[
X_i^s = \left[ \frac{L + X_i^* - X_1^*}{2} \right] + X_i^*
\]

(4.13)

where,

\[
L = T_e - [4(X_i^* + 0.004)] and,
\]

158
\( T_c \) = Thickness of the total basalt-plastic composite (m)

The coordinates of the first node or the base node for this layer is given in the Table 4.13. Then the nodes in the first row are created, whose coordinates are represented by the equations depicted in the Table 4.14, followed by the generation of rest of the nodes in this layer, Table 4.15 lists the equations needed to create these nodes.

**Table 4. 13: Coordinates of the first node**

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>0</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( r_1 + X_1^5 )</td>
</tr>
</tbody>
</table>

**Table 4. 14: Coordinates of the first row of nodes**

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(n-1) + 0.02445 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = 0 )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = zc(n-1) )</td>
</tr>
</tbody>
</table>

**Table 4. 15: Coordinates of the remainder nodes in the layer**

<table>
<thead>
<tr>
<th>X-coordinate</th>
<th>( xc(n) = xc(p) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y-coordinate</td>
<td>( yc(n) = yc(1) - (r_1 + X_1^5) \cdot \sin[(j-1) \cdot (\pi/2)/k] )</td>
</tr>
<tr>
<td>Z-coordinate</td>
<td>( zc(n) = (r_1 + X_1^5) \cdot \cos[(j-1) \cdot (\pi/2)/k] )</td>
</tr>
</tbody>
</table>
After the creation of nodes, elements are created, again initially the first row followed by the rest of the elements for the layer. The equations used for this purpose are listed in Table 4.6 and Table 4.7. Figure 4.18 pictures the meshed 90° basalt-plastic layer generated from the MATLAB optimization code.

![Figure 4.18: The 90° basalt-plastic layer created from the MATLAB optimization code](image)

4.2.2.4 Movement of Gusset and Throttle Plates

The gusset and throttle plates are not created all over again but rather restored from the Model 4 LS-DYNA file, due to the fact that the geometry of the plates are not affected with the guesses of design variables but only the location. A program is written in MATLAB [Appendix VI], which reads the coordinates of the nodes and elements pertaining to these plates and stores it in a database MATLAB file. In the optimization code, these nodes and elements are read from the database file and the coordinates germane to the longitudinal direction (X-coordinates in the present case) are incremented.
with the design parameter $X_3$ [Appendix VII]. This helps in shifting the gusset and throttle plates to and fro from its original location in the longitudinal direction. This concept not only saves programming time, for the generation of complex solid elements of the gusset plates, but also the computational time. Also the optimization program is made a lot simple.

![Gusset plate and Throttle plate diagram](image)

Figure 4.19: The gusset and throttle plates obtained from the MATLAB database file

4.2.2.5 Creation of Rest of the LS-DYNA Cards

Due to the change in the location of the gusset and throttle plates for each optimization iteration, the blast pressure does not act on the same elements of the inner steel layer throughout the optimization process but differs. Hence in the optimization
code, a program is incorporated such that the *LOAD_SEGMENT card is created and the elements on which the blast pressure acts is selected relative to the movement of the gusset and throttle plates [Appendix VIII]. Similar to the creation of elements, the first row of elements on the inner steel layer, from the center to the new location of the throttle plate, are chosen for the application of blast pressure. Then the first row of elements from the end of gusset plate to the end of the cylindrical portion are recognized. The between elements of the steel layer on which the gusset plate rests is ignored. The rest of the elements in the inner steel layer are selected for the application of the blast pressure in the identical manner.

Owing to the creation of *LOAD_SEGMENT card, there is a need for generating the inner steel layer in the cylindrical portion of the vessel through the MATLAB optimization code. When the layers are created through HyperMesh, the numbering of the nodes are random and not in order. In selecting the elements for the application of blast pressure through the MATLAB program requires the numbering of nodes and elements to be in an orderly fashion or else the program becomes very complicated and long to write and implement. Hence a program is written, which is fused in the optimization code, for the creation of the inner steel layer in the cylindrical portion of the vessel. The procedure for the generation of this layer is akin to the creation of the 90° basalt-plastic layer.

The constraints and the sectional properties for the generated composite and steel layers are produced by creating the *BOUNDARY_SPC_NODE and *SECTION_SHELL card respectively through MATLAB.

The final LS-DYNA cards created through the optimization code are the *MAT_ORTOTROPIC_ELASTIC cards for the ±33° basalt-plastic layers. Due to the
change in the angle of orientation of the fibers in these two layers, $X_2$, the material properties are computed and assigned to the respective cards [Appendix IX].

4.3 Optimization Process

The flowchart in Figure 4.20 shows the optimization process followed. The objective function along with the design variables is defined in MATLAB. With the initial guess of the variables, the dynamic code is prepared in the MATLAB. This code is incorporated into the static code. Then by coupling LS-DYNA through MATLAB, the combined static and dynamic code is run. The needed results are then extracted from the LS-DYNA output file and the objective function is evaluated. Based on the objective function value the optimization algorithm creates a new set of guesses for the design variables. The optimization algorithm is a fuzzy simplex code developed by Trabia et al [25]. The initial simplex is created according to Spendley et al [26] by generating $n+1$ equally spaced points according to the Equation 4.13.

$$X_i = X_0 + \delta_1 U_i + \sum_{j=1, j \neq i}^{n} \delta_2 U_j \quad (4.14)$$

where,

$$\delta_1 = \frac{\sqrt{n+1} - n - 1}{n \sqrt{2}} \alpha \quad (4.15)$$

$$\delta_2 = \frac{\sqrt{n+1} - 1}{n \sqrt{2}} \alpha \quad (4.16)$$

The variable $\alpha$ is the simplex size factor and is taken as one for optimizing the explosion-proof vessel. Fuzzy Simplex uses fuzzy controllers to compare the function values at the simplex points and determine the amount of expansion and contraction of
the simplex accordingly. A third fuzzy controller also determines shifts the center of the simplex toward the point with the lowest function value. Extensive testing [25] shows that this algorithm can reach a minimum faster than the standard simplex algorithm [26], which uses fixed steps for expansion and contraction of the simplex. Fuzzy Simplex algorithm is used in this research as it offers flexibility in deciding the best search direction, especially since the problem of optimizing a composite vessel is highly nonlinear. The termination criterion is set based on a difference in the consecutive objective function values. This is known as the error parameter and its value is set to $1E^{-4}$.

![Flowchart representing the optimization process](image)

Figure 4. 20: Flowchart representing the optimization process
4.4 Results and Discussions

The optimization process for the AT595 explosion-proof vessel is initially carried out with just one design variable, $X_3$, to check the effect it has on the optimization problem. The other two variables, $X_1 = 0.008 \text{ m}$ and $X_2 = 33^\circ$, are kept constant for different initial guesses of $X_3$. The results show that the optimization problem is not sensitive to the changes in $X_3$ as listed in Table 4.16. Hence the variable $X_3$ is not included for further optimization processes of the vessel and the location of the gusset and throttle plates are left in their original positions.

Table 4.16: Results of peak strain obtained by varying just the variable $X_3$

<table>
<thead>
<tr>
<th>$X_3$</th>
<th>Peak Strain (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-9</td>
<td>2.17</td>
</tr>
<tr>
<td>-5</td>
<td>2.17</td>
</tr>
<tr>
<td>-2</td>
<td>2.18</td>
</tr>
<tr>
<td>0</td>
<td>2.17</td>
</tr>
<tr>
<td>2</td>
<td>2.17</td>
</tr>
<tr>
<td>5</td>
<td>2.17</td>
</tr>
<tr>
<td>9</td>
<td>2.18</td>
</tr>
</tbody>
</table>

For the rest of the optimization study, to survey the search space several initial guesses of the variables $X_1$ and $X_2$ are used as shown in Table 4.17, ranging from the lower limit of the variables to the upper limit. The results of Table 4.18 are consistent regardless of the initial guess used. From Table VIII the average peak strain value after
optimization is computed to be 1.55\%. RFNC-VNIIF obtained a peak strain value of 2\% in their experimental testing of the AT595 explosion-proof vessel and the simulated value of peak strain from Model 5 is 2.25\%. Percentage reduction in the peak strain, obtained from Equation 4.16, due to the proposed optimization technique when compared to the experimental and computational values is 23\% and 31\% respectively.

\[
R = \left(\frac{\epsilon_{EF} - \epsilon_{O}}{\epsilon_{EF}}\right) \times 100\%
\]  
(4.17)

where,

\[R\] = Percentage reduction of peak strain (\%)

\[\epsilon_{EF}\] = Peak strain obtained experimentally or computationally from FEA (\%)

\[\epsilon_{O}\] = Peak strain obtained after the optimization process (\%)

Table 4.17: Initial guesses on variables \(X_1\) and \(X_2\)

<table>
<thead>
<tr>
<th>Initial Guess</th>
<th>(X_1) (m)</th>
<th>(X_2) (degrees)</th>
<th>Peak Strain (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.003</td>
<td>80</td>
<td>2.55</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.003</td>
<td>10</td>
<td>2.16</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.008</td>
<td>80</td>
<td>2.52</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.008</td>
<td>33</td>
<td>2.26</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.008</td>
<td>10</td>
<td>1.71</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.012</td>
<td>80</td>
<td>2.50</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.012</td>
<td>10</td>
<td>1.63</td>
</tr>
</tbody>
</table>
Table 4. 18: Results obtained after optimization of the initial guesses

<table>
<thead>
<tr>
<th>Final Result</th>
<th>$X_1$ (m)</th>
<th>$X_2$ (degrees)</th>
<th>Peak Strain (%)</th>
<th>Run Time (minutes)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.0113</td>
<td>13.99</td>
<td>1.57</td>
<td>400</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.0113</td>
<td>10.87</td>
<td>1.52</td>
<td>450</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.0114</td>
<td>11.44</td>
<td>1.55</td>
<td>400</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.0113</td>
<td>10.90</td>
<td>1.53</td>
<td>331</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.0113</td>
<td>10.90</td>
<td>1.53</td>
<td>331</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.0102</td>
<td>11.96</td>
<td>1.61</td>
<td>450</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.0102</td>
<td>10.44</td>
<td>1.57</td>
<td>297</td>
</tr>
<tr>
<td>Average</td>
<td>0.0110</td>
<td>11.50</td>
<td>1.55</td>
<td>431</td>
</tr>
</tbody>
</table>

The circumferential strain at the center of the vessel for the FE model before and after optimization is plotted as depicted in Figure 4.21. The amount of reduction in the peak strain after optimization can be gauged from this plot. Also a phase shift is noticed in the curve after optimization, which can be attributed to the increase in rigidity or strength of the vessel due to the changes in the design parameters such as the thickness of the basalt-plastic layers and the angle of fiber orientation for these layers.
The proposed optimization technique not only results in a significant reduction of peak strain, but also the computational time required to finish the optimization process is less. From the Table 4.18 it is observed that the average time taken to finish the optimization is around seven hours, which is less than the simulation time of the solid FE model, Model 3, of the explosion-proof vessel. Figure 4.22 depicts the peak circumferential strain contours for Case 4 at the initial guess and after the optimization process. The peak strain contours after optimization is less when compared to the initial value.
4.5 Conclusion

From the literature survey it is evident that the area involving the optimization studies regarding explosion-proof vessels is very limited. There is a need for designing an universal optimization technique for explosion-proof vessels and this research lays a platform for such a study. This chapter involves in proposing the optimization technique and also validates it by comparing the optimization results with the experimental ones.

The objective of this optimization study is to reduce the peak strain obtained in the AT595 explosion-proof vessel due to internal blast loading. Three design variables are chosen initially and as one of the variables does not contribute to the objective of the study, it is abandoned. From the remaining design variables, a reduction in peak strain of 23% and 31% is obtained when compared to the experimental and computational results respectively. This change can be attributed as significant. Also the time taken to finish the optimization process is around seven hours, which can be stated as fast. Hence the optimization technique proposed in this chapter is validated.
CHAPTER 5

CONCLUSION

This chapter summarizes the work presented in the earlier chapters. The objective of this thesis threefold and is as listed below,

- To characterize the materials used in AT595 explosion-proof vessel by conducting static tests. These materials include steel wire mesh, polymer-foam, stainless steel and basalt-plastic composite. The results obtained are compared with those provided by RFNC.

- To determine an efficient analysis procedure and create a FE explosion-proof vessel model that provides a combination of accuracy and computational speed for carrying out parametric analysis or design optimization.

- To optimize the FE model of the explosion-proof vessel in order to reduce the peak strains produced during simulation.

5.1 Material Characterization

RFNC provided UNLV with some of the materials used in the manufacture AT595 explosion-proof vessel. The materials included anti-fragment shield (SWM), polymer-foam, stainless steel and basalt-plastic composite. Static tests are conducted on these materials and their properties evaluated. Table 5.1 gives the summary of tests conducted properties estimated. All the tests are conducted on the MTS.
Table 5.1: Summary of tests conducted and properties computed

<table>
<thead>
<tr>
<th>Material</th>
<th>Test</th>
<th>Objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>Anti-Fragment shield</td>
<td>Compression</td>
<td>➤ Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>(Steel wire mesh)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Polymer-Foam</td>
<td>Compression</td>
<td>➤ Collapse Strength</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>Tension</td>
<td>➤ Poisson's Ratio</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Yield Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Ultimate Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Fracture Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Engineering Stress-Strain curve</td>
</tr>
<tr>
<td>Basalt-Plastic</td>
<td>Tension</td>
<td>➤ Poisson's Ratio</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Elastic Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Shear Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Fracture Stress and Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td>➤ Engineering Stress-Strain curve</td>
</tr>
</tbody>
</table>

5.1.1 Anti-fragment Shield (SWM)

From the static compression tests conducted on the SWM specimens, the Engineering stress-strain curve behavior is assayed. Tests are conducted for three different types of specimens. Figure 5.1 gives a summary of the Engineering stress-strain curves for these specimens. All the curves follow the same path and the results are consistent. Small
amount of slipping is noticed between the SWM layers in the specimen. No comparable experimental data was available from RFNC. As the anti-fragment shield does not contribute much to the structural performance of the vessel and also due to the difficulty in the modeling of the SWM, this is not included in the FE analysis of the AT595 explosion-proof vessel. But the mass of the anti-fragment shield is adjusted in the inner steel layer due to the affect it has on the dynamic response.

![Graph showing stress-strain plots for different SWM specimens](image)

**Figure 5.1:** Comparison of stress-strain plots for different SWM specimens

5.1.2 Polymer-Foam

Two panels from the eleven provided by RFNC are tested for compressive loading. From each panel two types of specimens are tested. The general Engineering stress-strain
curve obtained for the polymer-foam specimens is shown in Figure 5.2.

Figure 5. 2: Typical engineering stress-strain plot for the polymer-foam material

Table 5.2 gives a summary of the results obtained from the testing of polymer-foam specimens. The results are consistent within the different types of specimens but varying when compared between the two panels. The pattern of the stress-strain curve for all the specimens follow similar to the curve given by RFNC. The crush strength of panel 1 is close to the experimental RFNC values provided. The variation of the results between the two panels and also with the RFNC value can be attributed to the significant differences in the densities of the specimen.
Table 5.2: Summary of polymer-foam results

<table>
<thead>
<tr>
<th>Specimen Type</th>
<th>Density Kg/m³</th>
<th>Crush Strength MPa</th>
<th>Elastic Modulus MPa</th>
<th>Maximum Strain mm/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Avg.</td>
<td>S.D. (%)</td>
<td>Avg.</td>
<td>S.D. (%)</td>
</tr>
<tr>
<td>2.5cm-Panel 1</td>
<td>230</td>
<td>±2.1</td>
<td>5.86</td>
<td>±0.7</td>
</tr>
<tr>
<td>5.1 cm-Panel 1</td>
<td>220</td>
<td>±6.5</td>
<td>4.77</td>
<td>±9.0</td>
</tr>
<tr>
<td>2.5cm-Panel 2</td>
<td>190</td>
<td>±1.4</td>
<td>4.06</td>
<td>±1.5</td>
</tr>
<tr>
<td>5.1cm-Panel 2</td>
<td>180</td>
<td>±1.3</td>
<td>3.42</td>
<td>±9.4</td>
</tr>
</tbody>
</table>

5.1.3 Stainless Steel

The stainless steel specimens are subjected to tension tests. The specimen is designed based on the ASTM E-8 standard for sheet type specimen. The specimens are cut from two plates. The typical engineering stress-strain curve for the stainless steel specimen is shown in Figure 5.3. This figure also portrays the materials properties obtained from the curve.
Figure 5.3: Typical stress-strain plot for stainless steel

The results of the specimens from both the plates are summarized in the Table 5.3. All the stress-strain curves follow the similar pattern and also the results are consistent. The obtained yield strength matches with one of the steel results of RFNC. This matched steel is used for the thick wall parts of the inner vessel like the mouths and port lids.
Table 5.3: Summary of results for stainless steel

<table>
<thead>
<tr>
<th>Poisson’s ratio</th>
<th>Elastic Modulus GPa</th>
<th>Yield</th>
<th>Ultimate</th>
<th>Fracture</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>±4.0</td>
<td>178</td>
<td>±4.7</td>
<td>248</td>
</tr>
</tbody>
</table>

5.1.4 Basalt-Plastic Composite

The basalt-plastic specimens are divided into Transverse (90°) specimens, Axial (0°) specimen and ±45° specimens. Tension tests are carried out on all the specimens. Totally four panels are tested, two for transverse specimens and one each for axial and ±45° specimens. Two panels are used for the transverse specimens since one of the panel had large number of voids, uneven thickness and also non-uniform distribution of the basalt fibers. This produced a large variation in the material properties for that panel, which can be observed from the summarized results in Table 5.4. But the rest of the panels show consistent results within themselves.

Table 5.4: Summary of results for basalt-plastic

<table>
<thead>
<tr>
<th>Panel</th>
<th>Poisson’s ratio</th>
<th>Modulus GPa</th>
<th>Fracture Stress MPa</th>
<th>Fracture Strain %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Panel 1</td>
<td>0.059 ±2.00</td>
<td>6.93 ±12.0</td>
<td>24 ±21.0</td>
<td>0.34 ±8.1</td>
</tr>
<tr>
<td>(Transverse)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Panel 6</td>
<td>0.057 ±0.29</td>
<td>7.65 ±2.7</td>
<td>23 ±0.7</td>
<td>0.30 ±1.0</td>
</tr>
<tr>
<td>(Transverse)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Panel 7</td>
<td>0.290 ±0.32</td>
<td>29.2 ±2.2</td>
<td>620 ±4.7</td>
<td>4.32 ±7.9</td>
</tr>
<tr>
<td>(Axial)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Panel 4</td>
<td>0.640 ±2.00</td>
<td>12.0 ±12.0</td>
<td>115 ±12.0</td>
<td>6.12 ±7.4</td>
</tr>
<tr>
<td>(±45°)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
The modulus properties obtained were far less than the RFNC values. This is attributed to the low fiber volume fraction of the panels provided to UNLV when compared with those tested in RFNC. The poisson's ratio and the strains are reasonable and consistent.

5.2 Finite Element Analysis

The primary objective of this study is to develop a FE explosion-proof vessel model that provides a combination of accuracy, computational speed and simplicity in creation of the model for carrying out parametric analysis or design optimization. The primary parts of the AT595 explosion-vessel, which are taken into consideration for FE modeling, are listed in the Table 5.5 and shown in Figure 5.4.

Figure 5. 4: AT595 explosion-proof vessel
Table 5.5: Parts to be modeled and their features

<table>
<thead>
<tr>
<th>Basic Feature</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basalt-Plastic Layer</td>
<td>✓ Increase strength</td>
</tr>
<tr>
<td></td>
<td>✓ Make the vessel light weight</td>
</tr>
<tr>
<td>Steel Lining</td>
<td>✓ Add rigidity to the vessel</td>
</tr>
<tr>
<td>Polymer-Foam</td>
<td>✓ Absorb energy released by the blast</td>
</tr>
<tr>
<td>Anti-Fragment Shield</td>
<td>✓ Absorb the shock loads</td>
</tr>
<tr>
<td></td>
<td>✓ Distribute the shock loads from the shrapnel</td>
</tr>
<tr>
<td>Throttle Plates</td>
<td>✓ Increase rigidity of the vessel</td>
</tr>
<tr>
<td></td>
<td>✓ Attenuate the shock from the explosion</td>
</tr>
<tr>
<td>Gusset Plates</td>
<td>✓ Support the throttle plates</td>
</tr>
<tr>
<td></td>
<td>✓ Adds rigidity to the vessel</td>
</tr>
</tbody>
</table>

Five FE models of the AT595 explosion-proof vessel having different levels of details are considered. Model 1 is a simple FE model comprising of all the features of the AT595 explosion-proof vessel. Excluding the polymer-foam material, rest of the model comprises of shell material. The cap and cylindrical portion of the vessel is represented as single shell layer. Model 2 is similar to Model 1 in the cylindrical portion, but the end-cap portion is modeled with high detail solid elements. Model 3 is a complete solid FE model and the mesh is very fine. The basalt-plastic composite is modeled as individual layers. The concept of Model 4 is identical to Model 3 but it is predominantly created from shell elements. Model 5 is akin to Model 4 in all aspects except the design of the gusset plate, which is created as a solid band of material. The basalt-plastic composite layers are modeled as individual shell layers. Model 5 is depicted in Figure 5.5.
Figure 5.5: FE model, Model 4 of the AT595 explosion-proof vessel

All the models are checked for percentage error when compared with the experimental results from RFNC and for computational efficiency. The results are summarized in the Table 5.6. From the results it is clear that Model 4 provides the best combination of accuracy and computational efficiency. Hence this model is chosen for carrying out an iterative optimization process.

Table 5.6: Results of the FE analysis of explosion-proof vessel models

<table>
<thead>
<tr>
<th>Model</th>
<th>Peak strain (%)</th>
<th>Time (ms)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>2.00</td>
<td>0.55</td>
</tr>
<tr>
<td>DRAKON</td>
<td>2.36</td>
<td>0.43</td>
</tr>
<tr>
<td>Model 1</td>
<td>2.13</td>
<td>0.51</td>
</tr>
<tr>
<td>Model 2</td>
<td>2.09</td>
<td>0.48</td>
</tr>
<tr>
<td>Model 3</td>
<td>1.58</td>
<td>0.42</td>
</tr>
<tr>
<td>Model 4</td>
<td>2.22</td>
<td>0.51</td>
</tr>
<tr>
<td>Model 5</td>
<td>2.27</td>
<td>0.54</td>
</tr>
</tbody>
</table>
5.3 Optimization Study

The field of optimization of the explosion-proof vessel is fairly new and hence not a lot off research available. The main objective of the study is to create an optimization procedure for the explosion-proof vessel and to validate it. The objective function is chosen as the reduction in the peak strain of the vessel in the cylindrical portion. Three design variable are identified for the accomplishment of the objective. First is the thickness of the -33° basalt-plastic layer, \( X_1 \), second is the angle of orientation of fibers in the +33° basalt-plastic layer, \( X_2 \), and the final variable is the position of the gusset and throttle plates in the cylindrical portion of the vessel, \( X_3 \). Preliminary results showed that the optimization problem is not sensitive to the changes in the variable \( X_3 \), when \( X_1 \) and \( X_2 \) are kept constant. Hence this variable was not included in further optimization process. For different initial guesses of \( X_1 \) and \( X_2 \), shown in Table 5.7, the final results were consistent as seen in Table 5.8.

<table>
<thead>
<tr>
<th>Initial Guess</th>
<th>( X_1 ) (m)</th>
<th>( X_2 ) (degrees)</th>
<th>Peak Strain (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.003</td>
<td>80</td>
<td>2.55</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.003</td>
<td>10</td>
<td>2.16</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.008</td>
<td>80</td>
<td>2.51</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.008</td>
<td>33</td>
<td>2.26</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.008</td>
<td>10</td>
<td>1.71</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.012</td>
<td>80</td>
<td>2.50</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.012</td>
<td>10</td>
<td>1.63</td>
</tr>
</tbody>
</table>
Table 5.8: Results obtained after optimization of the initial guesses

<table>
<thead>
<tr>
<th>Final Result</th>
<th>$X_1$ (m)</th>
<th>$X_2$ (degrees)</th>
<th>Peak Strain (%)</th>
<th>Run Time (minutes)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.0113</td>
<td>13.99</td>
<td>1.57</td>
<td>400</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.0113</td>
<td>10.87</td>
<td>1.52</td>
<td>450</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.0114</td>
<td>11.44</td>
<td>1.55</td>
<td>400</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.0113</td>
<td>10.90</td>
<td>1.53</td>
<td>331</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.0113</td>
<td>10.90</td>
<td>1.53</td>
<td>331</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.0102</td>
<td>11.96</td>
<td>1.61</td>
<td>450</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.0102</td>
<td>10.44</td>
<td>1.57</td>
<td>297</td>
</tr>
<tr>
<td>Average</td>
<td>0.0110</td>
<td>11.50</td>
<td>1.55</td>
<td>431</td>
</tr>
</tbody>
</table>

Percentage reduction in the peak strain due to the proposed optimization technique when compared to the experimental and computational values is 23% and 31% respectively. The proposed optimization technique not only results in a significant reduction of peak strain, but also the computational time required to finish the optimization process is less. From the Table 5.8 it is observed that the average time taken to finish the optimization is around seven hours, which is less than the simulation time of the solid FE model, Model 3, of the explosion-proof vessel. The circumferential strain at the center of the vessel for the FE model after simulation and optimization is plotted as depicted in Figure 5.6. The amount of reduction in the peak strain after optimization can be gauged from this plot. Also a phase shift is noticed in the curve after optimization, which can be attributed to the increase in rigidity or strength of the vessel due to the
changes in the design parameters such as the thickness of the basalt-plastic layers and the angle of fiber orientation for these layers.

Figure 5.6: Plot depicting the reduction of peak circumferential strain after optimization

5.4 Conclusion

The following conclusions can be interpreted from this thesis study,

- The overall results of all the material samples were consistent and comparable with the RFNC values.
- ConWep function is successful in modeling the internal blast loads, thus reducing the need for more computationally expensive codes.
- Various representations of the end cap region produce no significant difference in strains in the cylindrical portion of the vessel.
• The optimization procedure discussed in this thesis has laid a platform for the parametric analysis of the explosion-proof vessels.

• The optimization procedure is validated due to the significant decrease of 23% in the peak circumferential strain for the cylindrical portion of the vessel when compared to experimental results.

5.5 Scope for Future Work

• Need to find a standard test procedure for the SWM material.

• As less than half of the samples provided by RFNC were tested, there is scope for more material testing.

• Find a modeling procedure for the anti-fragment shield.

• To identify suitable composite material models for incorporation within FE analysis codes.

• Including the parameters of the end-cap portion of the vessel may further improve the optimization results.

• More design variables can be added such as the inner steel liner, the inert casing of the explosive, polymer-foam model, etc. to study their effects.

• To conduct failure analysis for the FE model of AT595 explosion-proof vessel.
APPENDIX I

Adjustment of Density in the Inner Steel Lining Due to Anti-Fragment Shield

The adjustment of density in Model 4, due to the anti-fragment shield, is depicted below. The same procedure is followed for the remaining FE models. Table 1 gives the parameters of the anti-fragment shield and the inner steel lining.

Table 1: Characteristics of anti-fragment shield and steel lining

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Anti-Fragment Shield</th>
<th>Inner Steel Lining</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, ( \rho ) (kg/m(^3))</td>
<td>1080</td>
<td>7850</td>
</tr>
<tr>
<td>Outer Diameter, ( d_1 ) (m)</td>
<td>0.911</td>
<td>0.920</td>
</tr>
<tr>
<td>Inner Diameter, ( d_2 ) (m)</td>
<td>0.731</td>
<td>0.911</td>
</tr>
<tr>
<td>Length, ( L ) (m)</td>
<td>0.679</td>
<td>0.679</td>
</tr>
</tbody>
</table>

Calculations for Mass \((m_i)\) of Anti-Fragment Shield:

Area of the anti-fragment shield,

\[
A = \frac{\pi}{4} \left[ d_1^2 - d_2^2 \right] = 0.2321 m
\]  \hspace{1cm} (1)

Volume of anti-fragment shield,

\[
V = A \times L = 0.1576 m^2
\]  \hspace{1cm} (2)

Mass of the anti-fragment shield,

\[
m_i = \rho \times V = 170.208 kg
\]  \hspace{1cm} (3)
Calculation for Mass ($m_2$) of the Inner Steel Lining Over the Shield:

Similar to the calculation of mass of the anti-fragment shield, the mass of the inner steel lining over the anti-fragment shield is evaluated from the parameters given in Table 1 and is given as,

$$m_2 = 68.996\text{kg} \quad (4)$$

The masses of the anti-fragment shield and the inner steel lining obtained are added to give the total mass ($M$) of the adjusted steel lining.

$$M = m_1 + m_2 = 239.204\text{kg} \quad (5)$$

The combined new density is obtained by dividing the total mass, $M$, by the volume of the inner steel encasing the anti-fragment shield.

$$\rho' = \frac{M}{V'} = 27215.4\text{kg/m}^3 \quad (6)$$

where,

$\rho'$ = Adjusted density value in the inner steel lining (kg/m$^3$)

$V'$ = Volume of the inner steel lining enclosing the anti-fragment shield (m$^3$)
APPENDIX II

Program to Calculate the Material Properties for 90° and ±33° Basalt-Plastic Layers

```matlab
1 %To find the Lamina Elastic properties of
2 %90 degree and ±33 degree Basalt composite
3 clc
4 clear
5 x=[90,33];
6 %Inputing the Basalt properties
7 E11=53.55e9; %Longitudinal Modulus
8 E22=15.15e9; %Transverse Modulus
9 G12=5.9e9; %Shear Modulus
10 PR12=.29; %Major Poisson's Ratio
11 PR21=.082; %Minor Poisson's Ratio
12
13 %Inputing the 90 degree fiber angle
14 thetal=(x(1)*pi/180); %Fiber angle
15 %Calculation of Lamina Elastic properties
16 %Modulus in the X-direction
17 Ex1=inv((cos(thetal)^4/E11)+(sin(thetal)^4/E22)+(0.25)*(((1/G12)-(2*PR12/E11))*sin(2*thetal)^2));
18 %Modulus in the Y-direction
19 Ey1=inv((sin(thetal)^4/E11)+(cos(thetal)^4/E22)+(0.25)*(((1/G12)-(2*PR12/E11))*sin(2*thetal)^2));
20 %Shear modulus
22 %Major Poisson's ratio
23 PRxy1=Ex1*((PR12/E11)-(0.25)*((1/E11)+(2*PR12/E11)+(1/E22)-(1/G12))*sin(2*thetal)^2);
24 %Minor Poisson's ratio
25 PRyxl=(Ey1/Ex1)*PRxy1;
26
27 %Printing out the 90 degree basalt-plastic properties
28 thetah=(thetal*(180/pi));
29 E11=Ex1 %Longitudinal Modulus
30 E22=Ey1 %Transverse Modulus
31 G12=Gxy1 %Shear Modulus
32 PR12=PRxy1 %Major Poisson's ratio
33 PR21=PRyxl %Minor Poisson's ratio
34
35 %Inputing the 33 degree fiber angle
36 thetah=(x(2)*pi/180); %Fiber angle
37 %Calculation of Lamina Elastic properties
38 %Modulus in the X-direction
39 Ex2=inv((cos(thetal)^4/E11)+(sin(thetal)^4/E22)+(0.25)*(((1/G12)-(2*PR12/E11))*sin(2*thetal)^2));
40 %Modulus in the Y-direction
41 Ey2=inv((sin(thetal)^4/E11)+(cos(thetal)^4/E22)+(0.25)*(((1/G12)-(2*PR12/E11))*sin(2*thetal)^2));
42 %Shear modulus
44 %Major Poisson's ratio
45 PRxy2=Ex2*((PR12/E11)-(0.25)*((1/E11)+(2*PR12/E11)+(1/E22)-(1/G12))*sin(2*thetal)^2);
46 %Minor Poisson's ratio
47 PRyxl2=(Ey2/Ex2)*PRxy2;
48
49 %Printing out the 33 degree basalt-plastic properties
50 thetah=(thetal2*(180/pi));
51 E11=Ex2 %Longitudinal Modulus
52 E22=Ey2 %Transverse Modulus
53 G12=Gxy2 %Shear Modulus
54 PR12=PRxy2 %Major Poisson's ratio
55 PR21=PRyxl2 %Minor Poisson's ratio
56
57
```

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
APPENDIX III

Calculation of Area under the Curve for Finding the Error between the FE Models

The Figure 1 shows the plot of peak strains, for the four FE models along with the RFNC results, along the length of the cylindrical portion of the explosion-proof vessel. The areas under the curves are computed using the Simpson's 3/8 rule, which is shown in Equation 1.

\[ I = 3(b-a) \left\{ \frac{f(x_0) + 3 \left[ \sum_{i=1,4,7} f(x_i) + 2 \sum_{k=5,6,9} f(x_k) + f(x_n) \right]}{8n} \right\} \]

Figure 1: Comparison of peak circumferential strain at different locations on the cylindrical portion of the vessel

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
where,

\[ I \quad = \quad \text{Area under the curve.} \]

\[ b \quad = \quad \text{The last x-axis point in the curve.} \]

\[ a \quad = \quad \text{The first x-axis point in the curve.} \]

\[ f(x_0), \ldots, f(x_n) \quad = \quad \text{From the first to the last y-axis coordinate values or points.} \]

\[ n \quad = \quad \text{Number of segments of equal width the curve is divided into.} \]

For all the models \( a = 0 \) and \( b = 1.125 \text{ m} \), while the number of segments of equal width the curve is divided into is, \( n = 3 \).
APPENDIX IV

Optimization Code depicting the Penalty Terms

```matlab
function y=f23mesh(x)
% clc
% clear
% x=[1,1,0];
delete('combined.k');
load database throttle;
a(1)=x(1)*0.008;  % First variable-Thickness of B-33 lamina
a(2)=x(2)*33;     % Second variable-Orientation of the fibers in B-33 lamina
a(3)=x(3)*0;      % Third variable-Throttle plate movement by no. of elements

% Constraints
g(1)=a(1)-0.003;
g(2)=0.012-a(1);
g(3)=a(2)-10;
g(4)=80-a(2);
g(5)=a(3)+10;
g(6)=10-a(3);
s2=0;
penalty(1)=0;
penalty(2)=0;
penalty(3)=0;
penalty(4)=0;
penalty(5)=0;
penalty(6)=0;

if(g(1)<0)
    penalty(1)=(-20*g(1)+1)*100000.0;
elseif(g(2)<0)
    penalty(2)=(-20*g(2)+1)*100000.0;
elseif(g(3)<0)
    penalty(3)=(-20*g(3)+1)*100000.0;
elseif(g(4)<0)
    penalty(4)=(-20*g(4)+1)*100000.0;
elseif(g(5)<0)
    penalty(5)=(-20*g(5)+1)*100000.0;
elseif(g(6)<0)
    penalty(6)=(-20*g(6)+1)*100000.0;
else
    xvariables(1)=(a(1)-0.008)/2;
end
```

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
APPENDIX V

Derivation for the Angle Increment/Decrement in the Cap Portion of the Vessel

Due to the design variable $X_i$, there is an increase or decrease in the radial angle subtending the cap for each layer. This angle increment/decrement, $d\theta$, is derived as shown below. Figure 2 shows the original radius, $R_i$, and radial angle, $\theta$, of the -33° basalt-plastic layer. Due to the thickness variable $X_i$, there is an increase or decrease in the cap radius, $R_i'$ and radial angle, $\theta_i$. The position of the new -33° basalt-plastic layer is obtained from the variable $X_i'$, given in Equation 1. The longitudinal distance of the cap portion of the vessel, $d$, is always kept constant at 0.4376 m.

\[
X_i' = \left[ \frac{X_i - 0.008}{2} \right]
\]  

(1)

Figure 2: Angle increment/decrement in the cap portion of the vessel due to $X_i$
\[ R_l \]  =  Radius of the -33° basalt-plastic layer (m)

\[ R_1' \]  =  Radius of the new -33° basalt-plastic layer, due to the variable \( X_i \) (m)

\[ \theta \]  =  Angle subtending the cap portion in the longitudinal direction (°)

\[ \theta_i \]  =  New angle subtending the cap portion in the longitudinal direction (°)

\[ d \]  =  Length of the cap portion in the longitudinal direction (m)

\[ d\theta \]  =  Angle increment/decrement due to the variable \( X_i \) (°)

From Figure 2 we get the following equations,

\[ \cos(90° - \theta) = \frac{d}{R_l} = \sin(\theta) \tag{2} \]

\[ \cos(90° - \theta_1) = \frac{d}{R_l + X_1} = \sin(\theta_1) \tag{3} \]

Hence,

\[ \theta = \sin^{-1}\left(\frac{d}{R_l}\right) \tag{4} \]

\[ \theta_1 = \sin^{-1}\left(\frac{d}{R_l + X_1}\right) \tag{5} \]

\[ d\theta = \theta - \theta_1 \tag{6} \]

\[ \therefore d\theta = \sin^{-1}\left(\frac{d}{R_l}\right) - \sin^{-1}\left(\frac{d}{R_l + X_1}\right) \tag{7} \]

Similar procedure is also followed for the +33° basalt-plastic layer.
APPENDIX VI

Program for Storing the Node and Element Definitions of the Gusset and Throttle Plates in a MATLAB Database File

```matlab
2 clear all;
3 clc;
4 B=textread('static2.k','s','whitespace','
');
5 B(41);
6 B(1312);
7 B(1621);
8 k=0;
9 for i=41:1312
10 b=B(i);
11 nid=b(1:5);
12 nids(1-40)=str2num(nid);
13 n_xcoord=b(6:21);
14 n_xcoords(1-40)=str2num(n_xcoord);
15 n_ycoord=b(22:37);
16 n_ycoords(1-40)=str2num(n_ycoord);
17 n_zcoord=b(38:53);
18 n_zcoords(1-40)=str2num(n_zcoord);
19 k=k+1;
20 end
21 for j=1:k
22 struc_node_database(j)=struct('node',nids(1),'
_xcoord',...
23 n_xcoords(1),'
_ycoord',n_ycoords(1),'
_zcoord',n_zcoords(1));
24 end
25 
26 
27
28 j=0;
29 for i=1317:1616
30 b=B(i);
31 eid=b(1:5);
32 eids(1-1316)=str2num(eid);
33 realconst_set=b(13);
34 realconst_sets(1-1316)=str2num(realconst_set);
35 node1=b(17:21);
36 nodes1((1-1316))=str2num(node1);
37 node2=b(25:29);
38 nodes2((1-1316))=str2num(node2);
39 node3=b(33:37);
40 nodes3((1-1316))=str2num(node3);
41 for f=1:k
42 if struc_node_database(f).node==nodes1(i-1316)
43 %Taking node1 coordinates from the node database
44 x1(i-1316)=struc_node_database(f)._xcoord;
45 end
```

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
45     y1(i-1316)=struc_node_database(f).y_coord;
46     z1(i-1316)=struc_node_database(f).z_coord;
47     end
48     if struc_node_database(f).node==nodes2(i-1316)
49         %Taking node1 coordinates from the node database
50         x2(i-1316)=struc_node_database(f).x_coord;
51         y2(i-1316)=struc_node_database(f).y_coord;
52         z2(i-1316)=struc_node_database(f).z_coord;
53         end
54     if struc_node_database(f).node==nodes2(i-1316)
55         %Taking node1 coordinates from the node database
56         x3(i-1316)=struc_node_database(f).x_coord;
57         y3(i-1316)=struc_node_database(f).y_coord;
58         z3(i-1316)=struc_node_database(f).z_coord;
59     end
60     end
61     j=j+1;
62     end
63     e=0;
64     for i=1621:2430
65         b=B(i);
66         eid=b(1:5);
67         eids((i-1620)+j)=str2num(eid);
68         realconst_set=b(13);
69         realconst_sets((i-1620)+j)=str2num(realconst_set);
70     %Element Coordinates
71         node1=b(17:21);
72         nodes1((i-1620)+j)=str2num(node1);
73         node2=b(25:29);
74         nodes2((i-1620)+j)=str2num(node2);
75         node3=b(33:37);
76         nodes3((i-1620)+j)=str2num(node3);
77     for g=1:k
78         if struc_node_database(g).node==nodes1((i-1620)+j)
79             %Taking node1 coordinates from the node database
80             x1((i-1620)+j)=struc_node_database(g).x_coord;
81             y1((i-1620)+j)=struc_node_database(g).y_coord;
82             z1((i-1620)+j)=struc_node_database(g).z_coord;
83         end
84     if struc_node_database(g).node==nodes2((i-1620)+j)

% Taking node coordinates from the node database
x2((i-1620)+j)=struct_node_database(g).x_coord;
y2((i-1620)+j)=struct_node_database(g).y_coord;
z2((i-1620)+j)=struct_node_database(g).z_coord;
end

if struct_node_database(g).node==nodes2((i-1620)+j)
% Taking node coordinates from the node database
x3((i-1620)+j)=struct_node_database(g).x_coord;
y3((i-1620)+j)=struct_node_database(g).y_coord;
z3((i-1620)+j)=struct_node_database(g).z_coord;
end

end

e=e+1;

for k=1:(j+e)
  struct_line_database1(k)=struct('Element_Id',eids(k), 'Part_No', ..., 'real_const_sets(k), 'Node1', nodes1(k), 'Node2', nodes2(k), 'Node3', nodes3(k), ..., 'x_1', x1(k), 'y_1', y1(k), 'z_1', z1(k), 'x_2', x2(k), 'y_2', y2(k), 'z_2', z2(k), ..., 'x_3', x3(k), 'y_3', y3(k), 'z_3', z3(k));
  struct_line_database1(k);
end

for k=1:(j+e)
  struct_line_database1(k);
end
APPENDIX VII

Optimization Code Required to call and Increment the Nodes and Elements of the Gusset and Throttle Plates According to the Design Variable $X_3$

```matlab
platevar=a(3); % Variable-number indicates how many elements it has

% moved, the sign indicates the direction (-ve means towards the center,
% +ve means towards the spherical portion)
for j=1:1272
    struc_node_database(j).x_coord=struc_node_database(j).x_coord+
    (platevar*0.02445);
end

for j=1:1272
    Node_number(j)=struc_node_database(j).node;
    Xcoordinate(j)=struc_node_database(j).x_coord;
    Ycoordinate(j)=struc_node_database(j).y_coord;
    Zcoordinate(j)=struc_node_database(j).z_coord;
end

% Writing out the moved nodes of the gusset and throttle plates
A(k)=['NODE';
k=k+1;
for i=1:1272
    xs=num2str(Xcoordinate(i));
    ys=num2str(Ycoordinate(i));
    zs=num2str(Zcoordinate(i));
    Nodeval=num2str(Node_number(i));
    A(k)=[Nodeval,'',',xs','',',ys','',',zs'];
k=k+1;
end
```

195

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Optimization Code for the Creation of ConWep Blast Function Card

```matlab
% Creating the blast pressure card, which is acting on the cylindrical
% portion of steel layer
A(k)='"LOAD_SEGMENT"';
k=k+1;
q=1;
T=60000;
lcid=-2;
SF=-1;
AT=0;
ipre=(28+platevar); % The position of the throttle plate from the center
% after getting moved
% Creation of the first row of pressure, from the center to the throttle
% plate
for p=1:ipre;
    N1(q)=T+q-1;
    N2(q)=N1(q)+1;
    N3(q)=N2(q)+50;
    N4(q)=N3(q)-1;
    lcids=num2str(lcid);
    sfs=num2str(SF);
    ats=num2str(AT);
    N1s=num2str(N1(q));
    N2s=num2str(N2(q));
    N3s=num2str(N3(q));
    N4s=num2str(N4(q));
    A(k)=[lcids,'','sfs','','ats','','N1s','','N2s','','N3s','','N4s];
k=k+1;
q=q+1;
end
% Number of elements to be skipped to accommodate the gusset plate
% Creation of first row of pressure, from the gusset plate to the end
% of the cylindrical portion
for p=1:(49-ipre-6);
    N1(q)=T+q-1;
    N2(q)=N1(q)+1;
    N3(q)=N2(q)+50;
    N4(q)=N3(q)-1;
    lcids=num2str(lcid);
    sfs=num2str(SF);
    ats=num2str(AT);
    N1s=num2str(N1(q));
    N2s=num2str(N2(q));
```
% Creation of the rest of the pressure card, from the center to the
% throttle plate
for i=1:29:
    mdn=mdn+u;
    for p=1:ipre;
    N1(q)=T+mdn+u;
    N2(q)=N1(q)+1;
    N3(q)=N2(q)+50;
    N4(q)=N3(q)-1;
    lcids=num2str(lcid);
    sfs=num2str(SF);
    ats=num2str(AT);
    N1s=num2str(N1(q));
    N2s=num2str(N2(q));
    N3s=num2str(N3(q));
    N4s=num2str(N4(q));
    A(k)=[lcids,'','sfs','','ats','','N1s','','N2s','','N3s','','N4s];
    k=k+1;
    q=q+1;
    mdn=mdn+1;
end
mdn=mdn+6; % Number of elements to be skipped to accommodate the
% gusset plate
% Creation of rest of the pressure card, from the gusset plate to
% the end of the cylindrical portion
for p=1:(49-ipre-6):
    N1(q)=T+mdn+u;
    N2(q)=N1(q)+1;
    N3(q)=N2(q)+50;
    N4(q)=N3(q)-1;
    lcids=num2str(lcid);
    sfs=num2str(SF);
    ats=num2str(AT);
    N1s=num2str(N1(q));
    N2s=num2str(N2(q));
    N3s=num2str(N3(q));
end
N4s=num2str(N4(q));
A(k)=[lcids,'','sfs','','ats','','N3s','','N2s','','N1s','','N4s];
k=k+1;
q=q+1;
mdn=mdn+1;
end
\end

% Creation of pressure on the connecting elements between the cylindrical portion and the spherical portion of the steel layer
node1=23;
node3=61499;
SF=1;
for p=1:30;
    N1(q)=node1;
    N2(q)=N1(q)+1;
    N3(q)=node3;
    N4(q)=N3(q)+50;
    lcids=num2str(lclid);
    sfs=num2str(SF);
    ats=num2str(AT);
    N1s=num2str(N1(q));
    N2s=num2str(N2(q));
    N3s=num2str(N3(q));
    N4s=num2str(N4(q));
    A(k)=[lcids,'','sfs','','ats','','N3s','','N2s','','N1s','','N4s];
k=k+1;
    node1=node1+1;
    node3=node3-50;
    q=q+1;
end
Creation of the ±33 Basalt-Plastic Material Cards Through the Optimization Code

% Creation of material cards for B+/-33 lamina
angle=a(2); % Second variable in the optimization code-orientation of
% the fibers in B-33 lamina
% Inputting the Basalt properties
E11=53.5e9; % Longitudinal Modulus
E22=15.15e9; % Transverse Modulus
G12=5.9e9; % Shear Modulus
PR12=.29; % Major Poisson's Ratio
PR21=.082; % Minor Poisson's Ratio

A(k)='$$$$$';

% Inputting the positive fiber angle for B+33 lamina
theta=(a(2)*pi/180); % Fiber angle ranging b/w 10-80
% Calculation of Lamina Elastic properties for positive basalt angle
% Modulus in the X-direction
Ex1=inv(((cos(theta)^4/E11)+(sin(theta)^4/E22)+(0.25)*((1/G12)-...
(2*PR12/E11))*sin(2*theta)^2));
% Modulus in the Y-direction
Ey1=inv(((sin(theta)^4/E11)+(cos(theta)^4/E22)+(0.25)*((1/G12)-...
(2*PR12/E11))*sin(2*theta)^2));
% Shear modulus
Gxy1=10*Ex1;
Gx1=10*Gxy1;

% Major Poisson's ratio
PRxy1=(Ex1/Ey1)-0.29*((1/E11)+(2*PR12/E11)+(1/E22)-...
(1/E22)-(1/G12))*cos(2*theta)^2);

% Minor Poisson's ratio
PRx1=0.1*PRxy1;
PRy1=(Ey1/Ex1)*PRxy1;

% Inputting the negative fiber angle for B-33 layer
theta=-theta; % Fiber angle ranging b/w -10-(-80)
% Calculation of Lamina Elastic properties for negative basalt angle
% Modulus in the X-direction
Ex2=inv(((cos(theta)^4/E11)+(sin(theta)^4/E22)+(0.25)*((1/G12)-...
(2*PR12/E11))*sin(2*theta)^2));
% Modulus in the Y-direction
Ey2=inv(((sin(theta)^4/E11)+(cos(theta)^4/E22)+(0.25)*((1/G12)-...
\[(2*PR12/E11)*\sin(2*\theta)^2)\); 

% Modulus in the Y-direction

\[Ez2=10*Ex1;\]

% Shear modulus

\[Gxy2=10*Gxy2;\]

\[Gxz2=10*Gxy2;\]

% Major Poisson's ratio

\[PRxy2=Ex2*(PR12/E11)-(0.25)*((1/E11)+(2*PR12/E11)+(1/E22)-...\]

\[(1/G12))*\sin(2*\theta)^2);\]

% Minor Poisson's ratio

\[PRzx2=0.1*PRxy2;\]

% Some of the constant terms in the material card

% Part id of +basalt mat model

\[pid1=10;\]

% Part id of -basalt mat model

\[pid2=11;\]

% Density of basalt

\[density=2060.02;\]

% Converting the variables to string format

\[densltys=num2str(density);\]

\[pidls=num2str(pid1);\]

\[pid2s=num2str(pid2);\]

% The properties needed to define the +basalt mat card

\[Exs=num2str(Exl,2);\]

\[Eys=num2str(Eyl,2);\]

\[Ezs=num2str(Ezl,2);\]

\[Gxys=num2str(Gxyl,2);\]

\[Gysz=num2str(Gyzl,2);\]

\[Gzxs=num2str(Gsxl,2);\]

\[PRxys=num2str(PRxyl,2);\]

\[PRyx2=num2str(PRyxl,2);\]

\[PRzx2=num2str(PRzxl,2);\]

% Writing the +basalt material card in the combined.k

% Totally 5 lines are needed to define the card (including card title)

\[A(k)='MAT_OTHROPTROPIC_ELASTIC');\]

\[k=k+1;\]

\[A(k)=[pidls,'','densitys',' ','Exs',' ','Eys',' ','Ezs',' ','PRxys',' ','...\]

\[PRyx2',' ','PRzx2];k=k+1;\]
% Converting the variables to string format
% The properties needed to define the -basalt mat card
Exs = num2str(Ex2,2);
Eys = num2str(Ey2,2);
Ezs = num2str(Ez2,2);
Gxys = num2str(Gxy2,2);
Gyzs = num2str(Gyz2,2);
Gzs = num2str(Gzx2,2);
PRxys = num2str(PRxy2,2);
PRyxs = num2str(PRyx2,2);
PRzxs = num2str(PRzx2,2);

% Writing the -basalt material card in the combined.
% Totally 5 lines are needed to define the card (including card title)
A(k) = {'*HAT_ORTHOTROPIC_ELASTIC'};
k = k+1;
A(k) = {'pid2s', ',', 'density', ',', 'Exs', ',', 'Eys', ',', 'Ezs', ',', 'PRxys', ',', '...
        PRyxs', ',', 'PRzxs'}; k = k+1;
A(k) = {'Gxys', ',', 'Gyzs', ',', 'Gzs', ',', 'aopts'}; k = k+1;
A(k) = [0]; k = k+1;
A(k) = [0]; k = k+1;
A(k) = {'*END'};
APPENDIX X

Program for creation of *SECTION_SHELL Cards for the Basalt-Plastic Layers

1023  % Creating Section_Shell cards
1024  % For B90 lamina
1025  A(k)='*SECTION_SHELL';
1026  k=k+1;
1027  A(k)='1,0,1.0,2,0.0';k=k+1;
1028  SH90=(0.032-(2*actualthickminus33)); % Thickness of B90 lamina
1029  SH90s=num2str(SH90);
1030  A(k)=[SH90s,' ,SH90s,' ,SH90s,' ,SH90s];
1031  k=k+1;
1032  % For B+33 layer
1033  % Thickness of the layers are same for B+33 and B-33 lamina
1034  A(k)='*SECTION_SHELL';
1035  k=k+1;
1036  A(k)='2,0,1.0,2,0.0';k=k+1;
1037  SHplus33=actualthickminus33; % Thickness of B+33 lamina
1038  SHplus33s=num2str(SHplus33);
1039  A(k)=[SHplus33s,' ,SHplus33s,' ,SHplus33s,' ,SHplus33s];
1040  k=k+1;
1041  % For B-33 layer
1042  A(k)='*SECTION_SHELL';
1043  k=k+1;
1044  A(k)='3,0,1.0,2,0.0';k=k+1;
1045  SHminus33=actualthickminus33; % Thickness of B-33 lamina
1046  SHminus33s=num2str(SHminus33);
1047  A(k)=[SHminus33s,' ,SHminus33s,' ,SHminus33s,' ,SHminus33s];
1048  k=k+1;

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
REFERENCE


VITA

Graduate College
University of Nevada, Las Vegas

Jagadeep Thota

Local Address:
969 East Flamingo Road, Apt #122
Las Vegas, NV 89119

Permanent Address:
D4, Samhita Meadows, 1st Cross, Kaggadasapura Main Road
Bangalore, Karnataka, INDIA 560093

Degrees:
Bachelor of Engineering, Mechanical Engineering, 2001
Bangalore Institute of Technology, Bangalore University, INDIA

Honors and Awards:
• Outstanding Academic Achievement, Honors Convocation 2005-2006.
• Outstanding Paper in the SAMPE 2006 Conference.
• Member of Phi Kappa Phi, Honor Society.
• Member of Tau Beta Pi, Engineering Honor Society.

Publications:

Thesis Title: Material Characterization and Parametric Studies of Explosion-Proof Vessel

Thesis Examination Committee:
Chairperson, Dr. Mohamed B. Trabia, Ph. D.
Chairperson, Dr. Brendan J. O'Toole, Ph. D.
Committee Member, Dr. Ajit K. Roy, Ph. D.
Committee Member, Dr. Georg F. Mauer, Ph. D.
Graduate College Faculty Representative, Dr. Samaan G. Ladkany, Ph. D.