Evaluation of Vehicle Fuel Tank Impact Resistance

Edmund Maputi[1], Dr. G. Satish Babu[2]

[1]Corresponding Author, Graduate Student, Jawaharlal Nehru Technological University, School of Engineering and Technology, Mechanical Department, 500085, Kukatpally, Hyderabad, Andhra Pradesh, India

[2]Professor, Jawaharlal Nehru Technological University, School of Engineering and Technology, Mechanical Department,500085, Kukatpally, Hyderabad, Andhra Pradesh, India

Abstract: *Automotive components are subject to testing and should meet standards set by regulatory authorities according to the type of vehicle. Using ANSYS V14.5 software, an impact simulation, on a fuel tank, is simulated and analyzed. The analysis is based on a recent bus accident that occurred on NH 44 near Palem Village, in Mahaboobnagar District, Andhra Pradesh, India, in which a bus crashed into a road culvert and burst into flames within seconds. It is widely believed that if the fuel tank and floor had not ruptured, the accident would have been less catastrophic. Simulation of the impact test procedure is done and evaluated against the real life event and occurrence. In this paper the impact will be simulated to investigate possible measures that can be taken to avoid any future accidents of this type.*

Keywords: Impact, Analysis, Bus, Fuel Tank, ANSYS

1. Introduction

Recent trends in traffic accidents indicate that fire accidents on buses are on the increase. On the 30th of October 2013, [1], a city to city sleeper bus travelling from (Bangalore to Hyderabad) in India, crashed into a road culvert and burst into flames. A total number of 45 passengers were burnt to death, since fire progression was too rapid for passengers to escape.

According to, [2], the following are some of the facts and conclusions reached at the accident scene:

- "The design of the bus is such that, one main fuel tank of 300 litres is very close to the battery compartment."
- "The sparks from the battery compartment may have ignited the fuel in the main fuel tank".
- "The main fuel tank is located between the front two tyres by design. Another two (2) auxiliary tank of 150 litres each and located, exactly behind the front two tyres, by design".
- "Within few minutes, the main fuel tank (300 Litres) and the two auxiliary fuel tanks (each 150 litres) caught fire". As per eye witness testimony
- "The iron pipe that was on the top of the culvert railing broke and got pierced into the main fuel tank. The other end of the iron pipe peered through the floor of the bus, thus one end of the pipe got immersed in the main fuel tank and the other end opened up into the bus, through the floor. This pipe acted like a capillary and spilled the fuel into the bus".
- "The tanks were makeup of hard bold plastic the floor of the bus was not steel but of plywood material and rubber matting".
- "The entire interior of the bus was made of PVC material, which is highly inflammatory".

Fire accidents are of great concern, all over the world and statistics show that, about 1 percent of Swedish buses, [3], suffer some form of fire incident each year. Also that, during the five-year period of 1999-2003, the U.S. fire departments responded to an estimated average of 2,210 bus or school bus fires per year.

Bus fires have catastrophic consequences in terms of loss of life and property. Often times such accidents occur in areas where the response time of emergency fire services is hampered by distance and heavy traffic in cities. This means that a bus worth thousands of dollars is destroyed beyond repair within minutes. The catastrophic loss of life is a major concern for the public. This research will analyze fuel tank impact resistance and suggest ways to avoid fuel becoming a source of fuel to the fire.

2. Literature Review

Fuel tanks carry a flammable liquid, which if ignited can cause serious damage to a vehicle by the way of a fire and/or explosion, with the capacity to cause damage to adjacent property or human occupants in or around a vehicle. This therefore means that the safety requirement for fuel tanks installed on vehicles must be such that there is no breach of the fuel tank in the event of a crash. The increased demand for more fuel efficient vehicles has resulted in the use of light weight materials in automotive applications. Blow molded plastic tanks are, increasingly, now being used on vehicles of all sizes as opposed to metal tanks. Research has been largely focused on improving material properties so that stronger, low weight tanks can be made. [4], discusses the environmental impact of plastic tanks, while weighing the cost of production, lifecycle and recyclability against steel tanks.

According to, [5], plastic fuel tanks are mostly made of High-density polyethylene (HDPE). Plastic has better formability properties as compared to steel. This allows the tank to be shaped according to available space in the vehicle frame. Plastic tanks have the advantage of less weight as compared to steel tanks, and they do not have seam lines, stress raisers on the component, which are prone to premature failure upon impact. Since steel tanks have seams and join areas, these often fail leading to fuel leak.

Plastic tanks do not transmit heat as much as steel tanks transmit heat. This could lead to expansion of the fuel gas in the tank, thereby causing leaks or in a worst case scenario, an explosion. In the event that a collision occurs the plasticity in plastic tanks allow them not to permanently deform, however steel tanks permanently deform and increase the temperature in the tank.

A variety of fuel tank designs have been engineered by vehicle makers such that an assessment of the safety level of these solutions is necessary ,in order maintain a set level of safety in the design of automobile vehicles. According to **[6]**, the various regulations are set out in the UN standard ECE R.34 Annex 5.SP and in summary they can be expressed as; Experience has shown that, **[7]** a continuously recurring problem, is that of the fuel storage facilities contribution to ignition and the spread of a fire under impact of a collision.

[7], recommends an examination of fuel systems involving polymeric materials in order to eliminate the fuel system, as an ignition source, as an aid to flame spread, or as a source of fuel or fire.

2.1 Explicit Analysis

An impact or crash analysis is a highly nonlinear analysis. According to **[8]**, the 'explicit dynamics' method is highly suitable for high-speed impact problems and highly non-linear problems. The explicit method, used in the analysis system is based on half-step central differences. Most explicit dynamics computer programs adapt a staggered, time-marching procedure with the nodal velocities being computed at the half time steps, $\dot{d}_{n+1/2}$ and the stresses, displacements and accelerations at the whole time step , σ_{n+1}, d_{n+1} and $\ddot{d}_{n+1}$. The time integration algorithms, **[8]** may be expressed as:

$$\ddot{d}_n = \frac{1}{\Delta\tau}\left(\dot{d}_{n+1/2} - \dot{d}_{n-1/2}\right) \qquad (2.4.1)$$

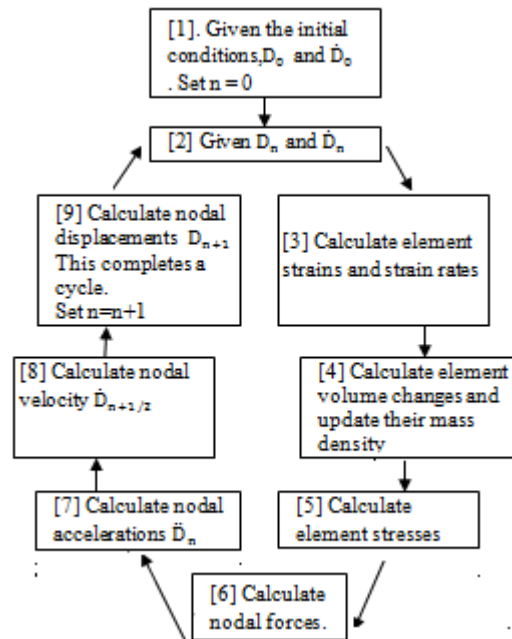$$\dot{d}_{n+1/2} = \frac{1}{\Delta t}(d_{n+1} - d_n) \qquad (2.4.2)$$

Figure 2.1: Explicit Dynamics Solution Procedure

As shown in Figure 2.1, the procedure that is used in the" Explicit Dynamics" analysis system, is explained in the following steps.

Step1- Initial conditions defined

Step 2- In the beginning of a cycle the displacement d and velocity $\dot{d}$ of the last cycle are already known.

Step 3-Using information from step 2, the strain and strain rate for each element is calculated using the following sets of equations;

$$\varepsilon_x = \frac{\partial u_x}{\partial x}, \; \varepsilon_y = \frac{\partial u_y}{\partial y}, \; \varepsilon_z = \frac{\partial u_z}{\partial z} \qquad (2.4.3)$$

$$\gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x}, \gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x}, \gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \qquad (2.4.4)$$

$$A. \quad \{u\} = [N] \qquad (2.4.5)$$

Where [N] is the matrix of shape functions, {d} represents the nodal displacements

Step 4 -The volume change for each element is then calculated, according to the equations of state, and mass density is updated

Step 5- Element stresses are calculated according to the constitutive model, relation between stresses and strains/strain rates

Step 6- The stresses are integrated over the elements, and the external loads are added to form the nodal force, F_n.

Step 7- Nodal accelerations are now calculated according to

$$\ddot{D}_n = \frac{F_n}{m} + \frac{b}{\rho}$$

Where b is the body force,
M is the modal mass
ρ is the mass density

Step 8- Nodal velocities at $t_{n+1/2}$ according to equation 2.4.1

Step 9- Nodal displacements at t_{n+1} are calculated according to equation 2.4.2

2.5 Finite element simulation and geometric modeling

The crash simulation will be carried out in the ANSYS-AUTODYN "explicit dynamics" environment. In this research paper two tests will be carried out on a plastic tank and on a steel tank.

The deformation results will be observed and conclusions drawn against the observed results of the two different materials. According to the accident report **[2],** the bus crashed into a roadside culvert and a pipe, which was part of the culvert, pierced through the middle front of the bus and ruptured the fuel tank and bus floor. This led to a spray of diesel fuel in the passenger compartment. When an ignition occurred, fire progression was too rapid for the escape of passengers. The analysis will focus on the impact of a blunt object on a fuel tank

2.5.1 Simulation and Analysis Steps

1) Launch Workbench, create an Explicit Dynamics analysis, and SAVE as "Tank Impact".
2) Add materials – Steel, Plastic (HDPE).
3) Specify the maximum equivalent plastic strain rate and the tangent modulus.
4) Import geometry.
5) Startup ANSYS Mechanical.
6) Assign materials to the various parts.
7) Delete contact regions.
8) Generate Mesh.
9) Input initial conditions data
   - Velocity = 33m/s
   - Force =30N
   - Supports – On fuel tank.
10) Enter the analysis End time.
11) Turn on Material failure.
12) Highlight the following solutions;
    - Total deformation
    - Equivalent stress
    - Equivalent plastic strain
    - User defined result
13) Solve

3. Test Procedure

According to E/ECE/TRANS/505, Regulation No. 34 page 26 Annex 5.the testing procedure for plastic fuel tanks is as follows;

3.1 Collision Resistance

3.1.1 -The tank must be filled to its capacity with a water-glycol mixture or with another liquid having a low freezing point, which does not change the properties of the tank material, and must then be subjected to a perforation test.

3.1.2- During this test the tank temperature must be (- 40 °C).

3.1.3- A pendulum collision testing fixture must be used for the test. The collision body must be of steel and have the shape of a pyramid with equilateral-triangle faces and a square base, the summit and the edges, being rounded to a radius of 3 mm. The centre of percussion of the pendulum must coincide with the centre of gravity of the pyramid; its distance from the axis of rotation of the pendulum must be 1 m. The total mass of the pendulum must be 15 kg. The energy of the pendulum at the moment of collision must be not less than 30 Nm and as close to that value as possible.

3.1.4- The tests must be made on the points of the tank which are regarded as vulnerable to frontal or rear collisions. The points regarded as vulnerable are those which are most exposed or weakest having regard to the shape of the tank or the way in which it is installed on the vehicle. The points selected by the laboratories must be indicated in the test report.

3.1.5- During the test, the tank must be held in position by the fittings on the side or sides opposite the side of collision. No leak must result from the test.

3.1.6- At the choice of the manufacturer, all the impact tests may be carried out on one tank or each may be carried out on a different tank.

3.2 Test Models

The following figures show two parts of the model setup with the following specifications

Table 3.2

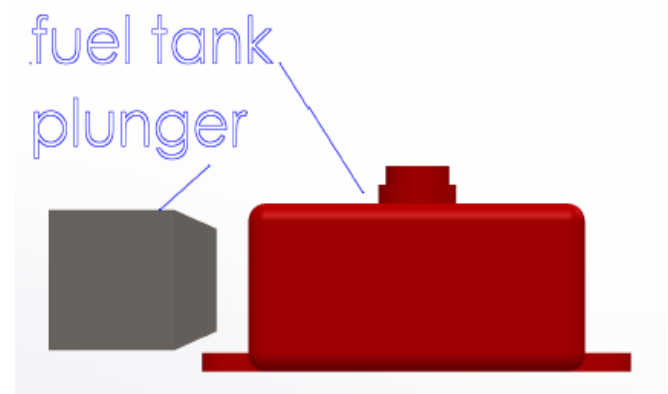| Parameter	Plunger	Fuel Tank
Volume	$1.907 \times 10^{-3} m^3$	$1.7036 \times 10^{-3} m^3$
Mass	$14.967 kg$	$13.373 kg$
Centroid X	$0.2043 m$	$0.2852 7m$
Centroid Y	$0.24672 m$	$0.2352 5m$
Centroid Z	$0.566 m$	$0.2569 9m$

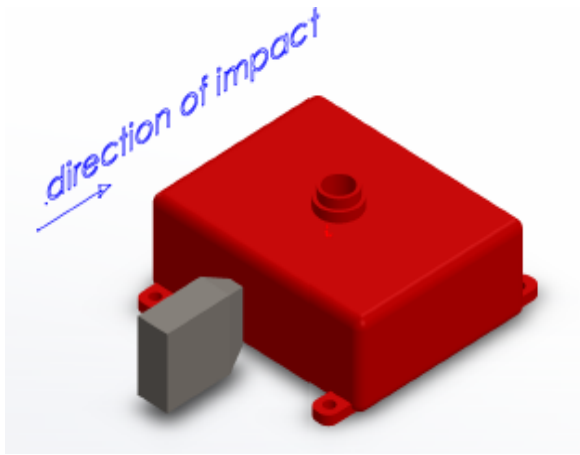

Figure 3.2: Test Model Side View

Figure 3.3: Test Model Isometric View

4. Results

4.1 Plastic Tank

The simulation of the plastic tank showed that the plastic material allowed the tank to deform without significantly deforming the inlet of the tank. In such a case, if the tank is not totally full, the fuel may not spill. However as shown in the diagram the impact causes most damage at the point of impact. According to this test the plastic tank if slightly improved would meet the required standards.
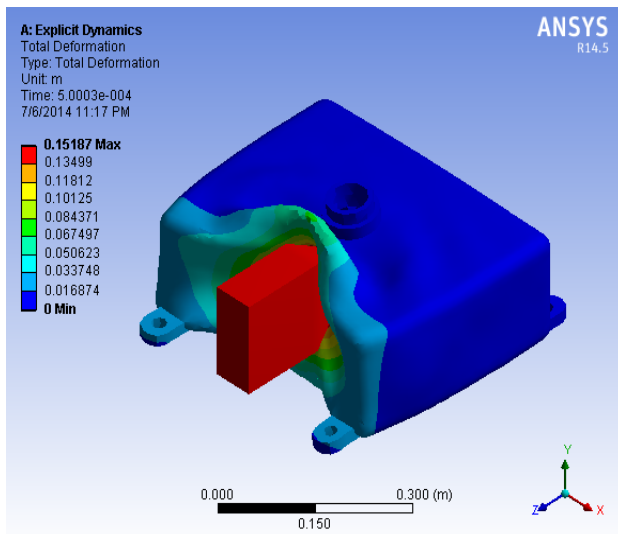

Figure 4.1: Plastic Fuel Tank

4.2 Metal tank

The metal tank, due to the nature of metal, has shown the effect of less plasticity in such kind of impacts. This can be observed as compared to the deformation observed on the plastic tank. The inlet to the tank has been distorted which could lead to the sealing cap falling off. Another side effect of metal tanks is that seam areas and weld points are stress raisers on the fuel tank. This means that in the event of an impact or deformation, these very same areas will be the first to crack open thereby spilling fuel.

These tests show that the standard fuel tank will meet the basic requirements of the standard, which takes into account

the most likely impact limits. However real life events usually result in unpredictable outcomes as reflected the nature of the accident.
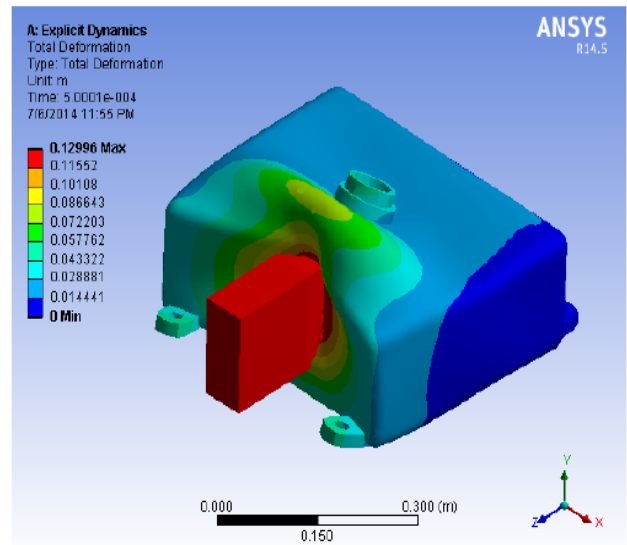

Figure 4.2: Metal Fuel Tank

5. Mahububnagar Accident Scenario

As outlined in the accident report**, [2]** a pipe pierced the fuel tank and caused the spill of fuel into the passenger compartment.

In the actual accident scenario the fuel tank and the bus body where in motion and the pipe on the bridge was stationary. The impact caused the bus body to decelerate rapidly while the fuel tank instantly changed direction of motion as it was ruptured and moved a small distance in the opposite direction.

Two scenarios are presented in the following results, with tank in motion and pipe stationary, and also with pipe in motion and tank stationary. Weight and speed conditions of the bus at the point of impact were not ascertained but assumed to be as follows;

Speed =100km/h = 27.7m/s
Vehicle mass = 16200kg
Force of impact = Mass x Velocity= 450kN

Applying these values as the initial conditions, the figure below shows the deformation observed; Two tank shapes where used in the impact simulation.

Tank material :(HDPE) High Density Polyethylene
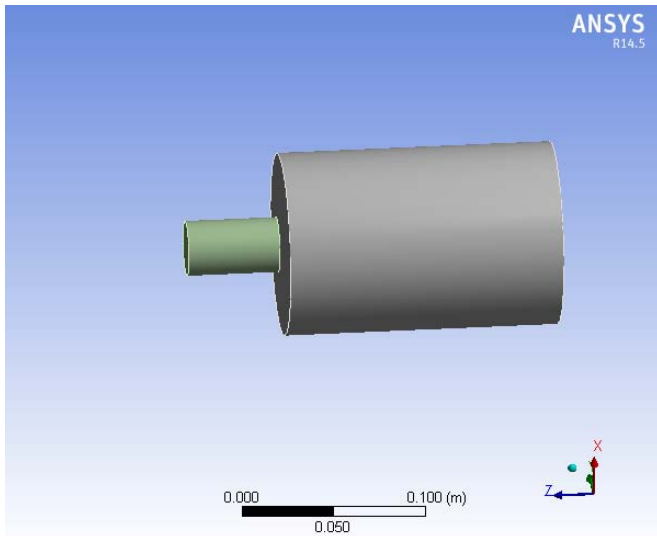Pipe material: Structural steel
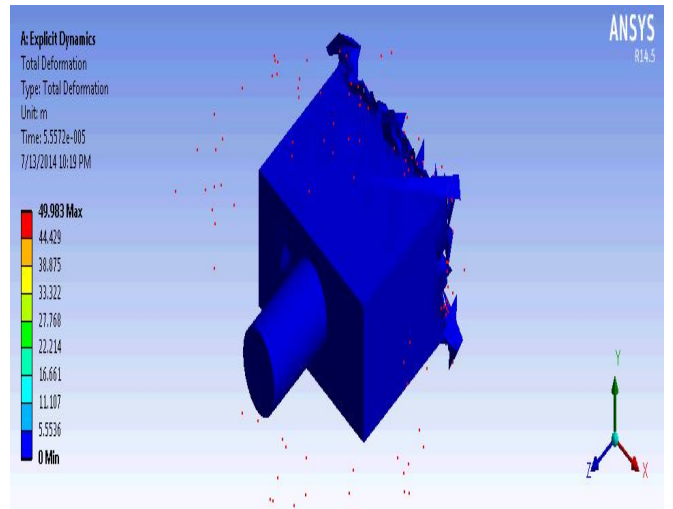
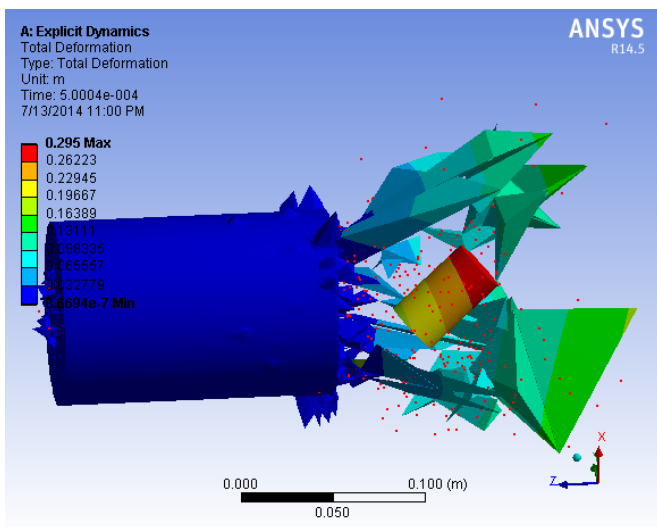**Figure 5.1:** Cylindrical Tank Model

Figure 5.4: Post Impact1
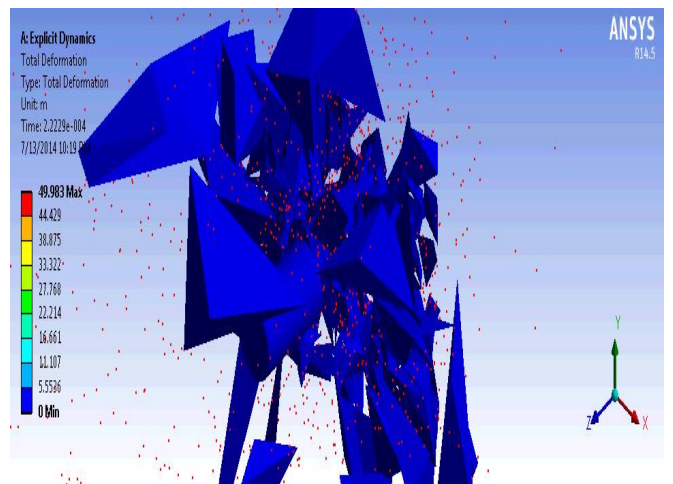

Figure 5.2: After Impact

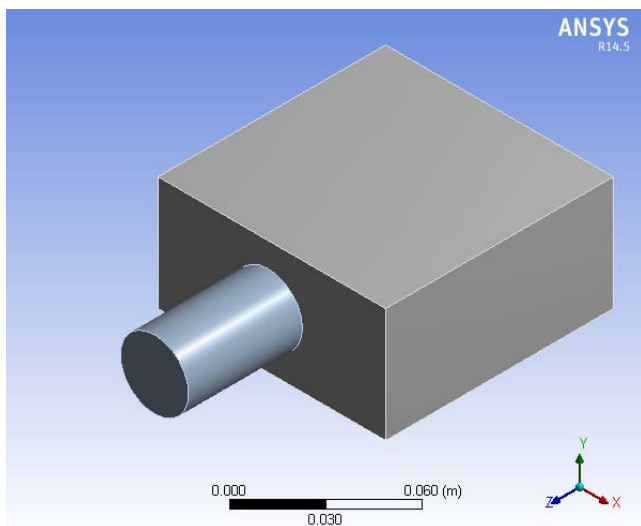**Figure 5.5:** Post Impact 2

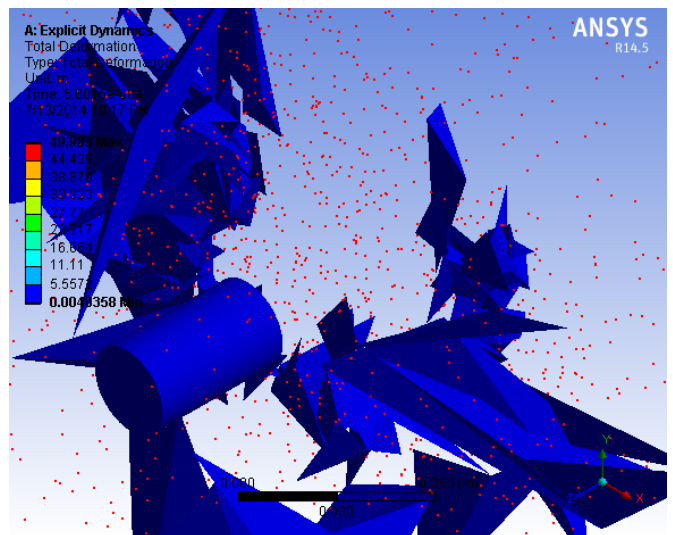**Figure 5.3:** Rectangular Tank Model

Figure 5.6: Post Impact 3

Figure 5.1 shows a cylindrical tank subjected to an impact along its axis. Figure 5.2 shows the aftermath of the impact. The piercing pipe protrudes to the other end while fracturing the whole tank on both ends. The nature of the impact on the exit end shows a splatter of broken pieces which could also resemble the splatter of fuel that can occur in such a scenario.

Figure 5.3 shows a rectangular tank in the same configuration as figure 5.1.intermittently the effect of a crushing force is observed at the back of the tank, as shown in figure 5.4, which is caused by obstruction from other body parts on the vehicle. Figure 5.5 and 5.6 show the different stages of the after impact destruction of the fuel tank. In this analysis there is total destruction of the fuel tank.

6. Discussions and Conclusions

It is clear from the analysis carried out that the test procedure does not imply that if a particular fuel tank passes the test then it is safe from all accidents situations. The bus accident in question was unique in the sense that a metal pipe pierced the bus body and had direct impact with the fuel tank. It is likely that the full weight of the bus was applied on the pipe via the fuel tank. This means that the point of contact was the fuel tank surface. Accident scenarios investigations predicted that the tank was displaced and spilled fuel into the passenger compartment, whose floor was made of chequered plywood.

The analysis reflects that it is not sufficient for manufacturers to install fuel tanks and argue safety based solely on the test standard. Although the fuel tank was not in the immediate frontal impact zone, it was in this case vulnerable to piercing impact because of the peculiar construction of the bus. Rear engine buses as compared to front engine buses are more prone to extensive damage in frontal impacts.

The authors suggest the use of a barrier to protect the tank from such kinds of impact. Manufacturers are usually reluctant to install added materials in the interest of cost such that only the base safety measures are adhered to.

The authors also suggest the removal of fuel tanks from the crash zone of a vehicle. Manufacturers of bus body structures usually place the fuel tanks at extreme ends of the wheel base, so as to accommodate the luggage at the centre. This method is advantageous in terms of lowering the centre of gravity of the whole bus on load. However the fuel tank is now moved to areas near the crash zones of the bus structure. A more balanced decision criterion should be used in favor of fuel tank safety over luggage space

References

[1] www.thehindu.com/news/cities/Hyderabad/45-killed-as-bus-goes-up-in-flames-on-hyderabad-bangalore-highway/article5287797.ece (accessed 4/03/2014)

[2] http://www.cidap.gov.in/Videos/Press/Volvo%20bus%20mishap.pdf (accessed on 09/07/2014)

[3] F Rosen, Improving fire safety of buses and coaches, SP Technical Research Institute of Sweden p11, (accessed from, http://www.saboa.co.za/documents/conference_2013/Improving%20fire%20safety%20of%20buses%20and%20coaches.pdf.)

[4] R.D Stephens, et al, Comparative Life Cycle Assessment of Plastic and Steel Vehicle Fuel tanks, SAE Technical series paper.

[5] P.J. Alvarado, Steel vs. Plastics: The Competition for Light Vehicle Tanks, The Minerals, Metals And Materials Society, 1996, 48 (7) (1996), pp. 22-25.

[6] http://www.sp.se/en/index/services/fueltank/sidor/default.aspx (accessed 3/7/2014).

[7] Fire Safety Aspects of Polymeric Materials, Volume 8-Land Transport Vehicles, National Materials Advisory Board.

[8] H. Huang Lee, Finite Element Simulations with ANSYS Workbench 12, SDC Publications 2010 pp 554-555.

[9] M.A Crisfield, Non-Linear Finite Element Analysis of Solids and Structures, Advanced Topics, Vol 2, Wiley, 1997, pp 451.

Author Profiles

Edmund Shingirayi Maputi is a Graduate student pursuing M.Tech Degree in Engineering Design at Jawaharlal Nehru Technological University Hyderabad, he received a B.Tech. Degree in Industrial and Manufacturing Engineering from Harare Institute of Technology, Zimbabwe in 2011.Prior to university education he was trained as an apprentice Fitter and Turner Machinist by Delta Technical Institute. His research interests include Machine design, Drive-train design and Heavy-duty automotive applications.

Dr G Satish Babu is Professor of mechanical Engineering Department at Jawaharlal Nehru Technological University Hyderabad. He received his B.Tech Degree from JNTU.M.E in Machine Design from Andhra University.PHD from JNTU Kakinada. His research interests include Mechanisms, Machine design and kinematics of Robotic Manipulators