ELSEVIER

CrossMark

Procedia Computer Science

Volume 80, 2016, Pages 1202–1211

ICCS 2016. The International Conference on Computational Science

Procedia Computer Science

Using Computational Fluid Dynamics (CFD) for Blast Wave Propagation under Structure

Arif S. M. Sohaimi[*], M. S. Risby[†], Saiddi A. F. M. Ishak, Khalis S., M. N. Norazman, Ariffin. I., M. A. Yusof

Protection and Survivability Research Unit (PROTECT), Faculty of Engineering, Universiti Pertahanan Nasional Malaysia, Sungai Besi Camp, 57000 Kuala Lumpur, Malaysia.

arifmsohaimi@gmail.com, risby@upnm.edu.my

Abstract

In recent years, improvised explosive devices has been an aspect of crusades by terrorist or movements around the world. The blast wave propagation of an explosive detonation can cause disastrous damage on the buildings, vehicles and also injuries to vehicle occupants. Full scale blast tests are expensive and time consuming but by using computational based numerical simulations can virtually predict these wave propagations and minimize the need of experimental testing. Computational fluid dynamics (CFD) is a common tool to do an analysis of free-field blast wave and against structure. This paper presents two different blast analyses; free field air blast and blast loading towards a structure using ANSYS FLUENT software. A high explosive of 1 kg blast peak overpressure data from an experiment has been patched at the specific domain of the symmetry plane. The computed results were found to be in agreement with theoretical and additional experimental data. The verified free field air blast model was expanded to study the blast loading response towards a structure. It was found that developed CFD can be further used to predict the blast wave propagation subjected to the vehicle structures or buildings.

Keywords: Blast wave, Computational fluid dynamics, Boundary condition, ANSYS FLUENT

1 Introduction

Explosion is a massive release of energy. It is can be classified as physical, nuclear or chemical events. In physical explosions, solid explosives such as high explosives on the blast effects are commonly studied. Explosives such as (TNT) and ANFO when exploded would generate blast waves that can result in extensive damages to the buildings and environments (Ngo *et al.*, 2007). Blast waves can be defined as pressure-based waves propagating in the vicinity of the explosive charge and shock waves serve as the stress-based wave in the protective structure and in turn developed as an outcome of

---

[*] Masterminded EasyChair and created the first stable version of this document

[†] Created the first draft of this document

Selection and peer-review under responsibility of the Scientific Programme Committee of ICCS 2016

doi:10.1016/j.procs.2016.05.463

the interaction of structure and blast wave (Grujicic *et al.*, 2013). The interactions of blast wave with an object ignites with an explosion followed by fast chemical reactions. Blast waves caused the air to be rejected by the explosion. Destruction of an object that is exposed to the blast wave entirely depends on the complicated interactions of blast wave with the structure. When a blast appears near to the military vehicles, the explosion wave formed by the detonating explosive impacts skeptically to the combating capacity of the vehicle and passengers *(*Yang *et al.*, 2013). Blast explosions induced by accidents or intentional attacks have resulted in severe damages to buildings and extreme casualties all over the world (Lee, 2009).

Nevertheless, conducting a field blast test has its own significant disadvantages such as it requires expensive sensors and the demand for remote test facilities. Experimental setup to generate blast waves needs the usage real explosives. So, they may be risky and costly. Hence, numerical simulations can be a good alternative in the investigation of blast events. Recent research and advancement work related to numerical simulation is vital to overcome these considerable issues, which is structural response and propagation of blast waves using finite element methods (Fairlie, 1998). Additionally, computational fluid dynamics (CFD) approach is seldom used but can be utilized for simulating blast waves (Alpman *et al.*, 2007).

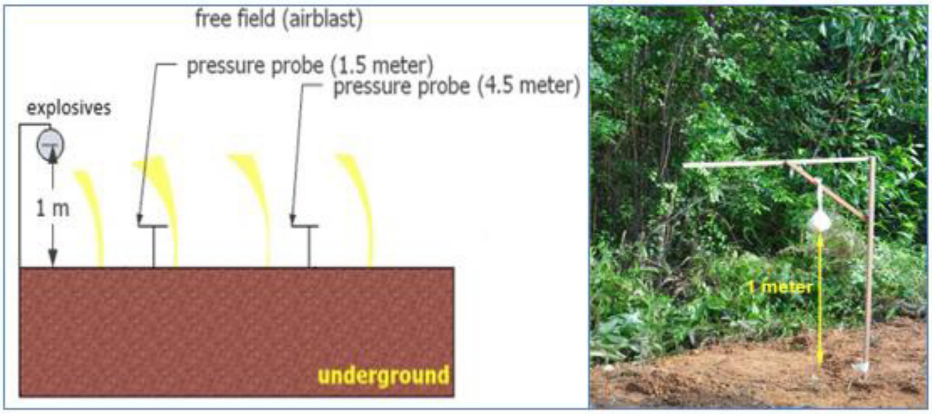
In recent years there has been considerable interest in evaluating the use of Computational Fluid Dynamics (CFD) in the blast wave related studies (Hansen *et al.*, 2010 and Tulach *et al.*, 2015). CFD has wide application prospects because it is able to create big domains as required and present varied post-process functions such as pathlines, contour, vector, report, animation and plot. Anyway, there a few studies found on blast wave propagation, mostly related to validating the applicability of Computational Fluid Dynamics (CFD) in simulating blast events. The challenges of using CFD to simulate blasts are that it does not possess a specific blast detonation algorithm or subroutine as typically available in the finite element based solvers. The present work describes method for modeling and predicting free field air-blast and the blast loading under structure using ANSYS FLUENT. For this simulation work, an initial "guess" for the solution of the blast wave flow field where certain functions or “patch” value is provided for selected flow variables in selected CFD boundary mesh zones. The initial value was taken from a free field air blast experimentation using high explosives. The validated CFD simulation model is then extended to investigate the blast wave response upon impact towards a structural object.

2 Experimental Work

A free field air blast test was conducted using 1 kg plastic explosive (PE4) at 1m standoff as shown in Figure 1. The transmitted wave or blast peak-overpressure from the detonation was measured using pressure probes at two different position from the explosive position namely, 1.5 and 4 meters. The pressure probe implements the following Eq. 1 (Sidik & Risby, 2013):

$$Pressure = \left(\frac{C_{fs}}{V_{ex}}\right) \times \left(\frac{V_{meas}}{CF}\right) \qquad \text{(Eq. 1)}$$

C_{fx} = Full scale capacity (kPa)
V_{ex} = Excitation voltage
V_{meas} = Measured voltage
CF = Calibration factor (mV/V)



(a) (b)

Figure 1: (a) Schematic of free field air blast test setup; (b) Actual test setup using 1 kg PE4.

The maximum blast peak-overpressure can also be computed based from the shock wave parameters for an explosion occurring in the unlimited atmosphere of an ideal gas as shown in Eq. 1.1 (Brode, 1959) and Eq.1.2 (Cranz, 1926; Hopkinson, 1915):

$$Q'^3 = \frac{Q*w}{P_0} = \frac{4\pi}{P_0}\int_0^{R_\Phi} Q\left(E + \frac{u^2}{2}\right)R^2 dR - \frac{4\pi R^3 \Phi}{3(k-1)} \qquad \text{(Eq. 1.1)}$$

Where,

$Q *_w$ = Explosion energy
R_Φ = Radius of the shock-wave front

$$\bar{R} = \frac{R}{\sqrt[3]{W}}\ [m/kg^{1/3}] \qquad \text{(Eq. 1.2)}$$

Where,

$\bar{R}$ = reduced distance $[m/kg^{1/3}]$
R = charge center distance [m]
W = charge weight [kg]

Naumenko & Petrovskyi (1956) have also established similar formula on the basis of theory of model similarity whereas the coefficients were derived from several experimental results (Eq. 1.3).

$$\Delta P_\Phi = \frac{0.76}{\bar{R}} + \frac{2.55}{\bar{R}^2} + \frac{6.5}{\bar{R}^3}\left[\frac{kp}{cm^2}\right], \qquad 1 \le \bar{R} \le 15 \qquad \text{(Eq. 1.3)}$$

3 Numerical Simulation

In general, ANSYS FLUENT do not possess any specific explosive algorithm or subroutine for blast problems compared to establish finite element solver such LSDYNA 3D and ABAQUS. For inlet condition (blast source), the initial blast pressure needs to be defined using profile or patch (Cong, 2014). Within this study, an initial blast pressure (from experimental data) was used as the initial input value for blast source using a patch method. Several points in the computational domain was set for correlation purpose with the experimental setup of the pressure probes. The pressure vs time history of these points was computed by ANSYS FLUENT, which later can be compared with experiment data.

3.1 Free field air-blast model

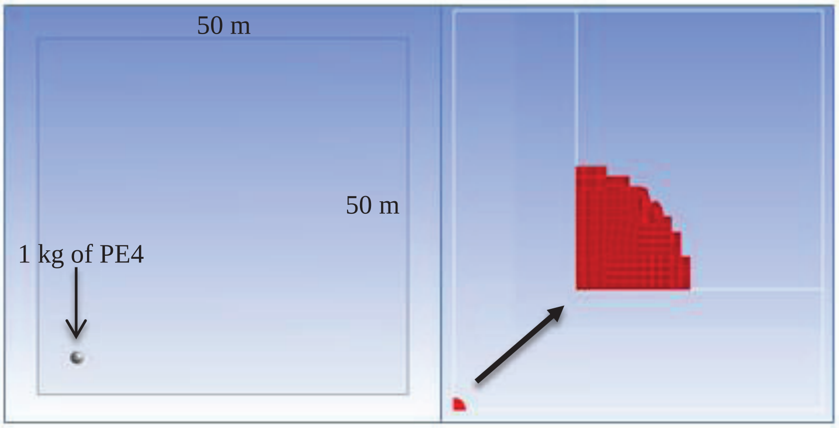
The overall test domain created was modelled as a rectangular box with the dimension of 50 m x 50 m. Blast input boundary was modelled as an arc with radius of 1.5m, which corresponds to the standoff distance from the first pressure probe as shown in Figure 1(a). The two monitored points corresponding to the two pressure probe are shown in Figure 2. Meshing process, which is integrated in ANSYS Workbench 15.0, was used to generate mesh for the domain. The model was developed using 15,575 nodes and 15,333 elements for the computational domain. Body sizing setup for the blast source was set to ‘fine’ in order to obtain a high accuracy in the computed results. Density-based and transient solver was selected for the supersonic compressible flow and also due to the time-dependent nature of explosions. The inviscid flow function was used as the shock wave capturing method and typically used in shock wave simulation in shock tunnel (Ofengeim & Drikakis, 1997). Inviscid flow gave a results in solving complicated flow physics. ANSYS FLUENT software solves the Euler equations. The mass, momentum and energy conservation equations are shown in Eq. 2.1, 2.2 and 2.3 (ANSYS Inc, 2009):

$$\frac{\partial \rho}{\partial t} = \nabla \cdot (\rho \vec{v}) = S_m \qquad \text{(Eq. 2.1)}$$

$$\nabla \cdot \vec{v} = \frac{\partial v_\chi}{\partial \chi} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r} \qquad \text{(Eq. 2.2)}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot \left(\vec{v}(\rho E + p)\right) = -\nabla \cdot \left(\sum_j h_j J_j\right) + S_h \qquad \text{(Eq. 2.3)}$$

For the initial value or “Patch Method” to execute the CFD solution process, the measured blast peak overpressure from 1.5 m standoff distance will be used as the startup value.



(a) (b)

Figure 3: (a) Model of geometry in the ANSYS FLUENT; (b) Patched method

3.2 Blast Loading under Structure

In order to investigate the blast wave response towards a structural object, a capsule model with dimension of 2.4 m x 1.4 m x 2 m was developed (as shown in Figure 4). The computational domain was created with the dimension of 10 m x 15 m. Blast input boundaries was modelled as a half spherical with a diameter of 1.5m, which corresponds to the standoff of the first pressure probe as shown in Figure 5.

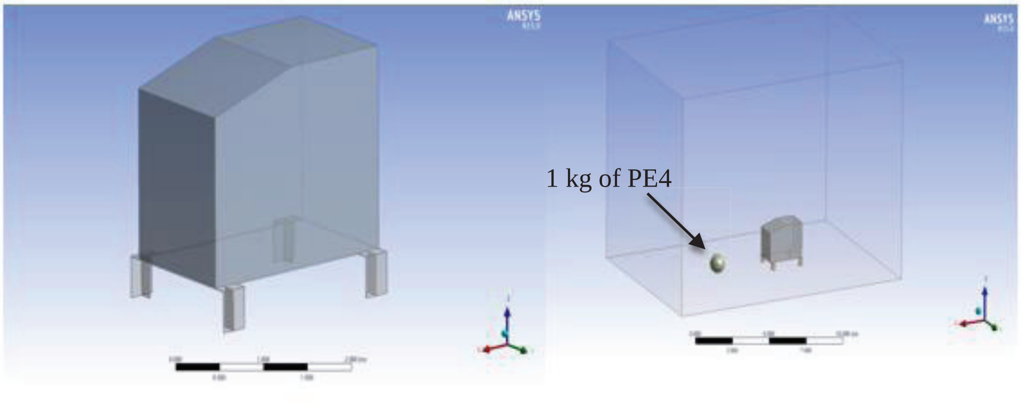


Figure 4: Geometry of capsule

Parallel processing mode was selected in order to reduce the computational time and double precision mode was selected because of the high pressure simulation. Density-based and transient solver was selected for the supersonic compressible flow (time-dependent). Roe-FDS with a Green-Gauss node based was preferred for convective flux types and spatial discretization scheme, which is more accurate and recommended for most cases by ANSYS FLUENT. Patch method updates the flow-field data based on the input and allows to patch different values of flow variables into different cells. In this case, pressure variable was used in patch panel to initiate the blast wave based on pressure input.

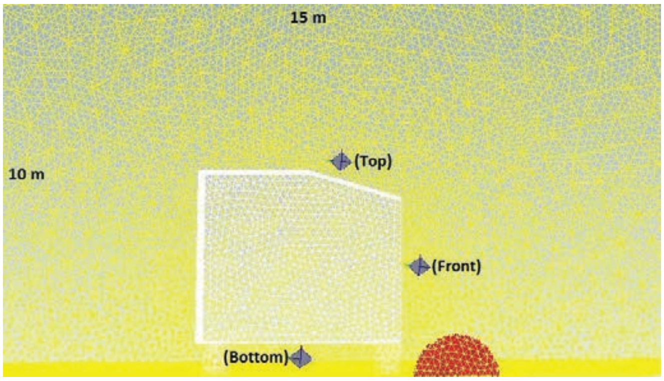


Figure 5: Boundary condition for blast loading on capsule

4 Results

4.1 Free Air Blast

Result from Table 1 and Figure 6 shows that theoretical blast over-peak pressure value is in agreement with the measured experimental data at 1.5 and 4.5 meter standoff distance. At 4.5 m distance, the over-pressure largely differs and can be assumed the weakening of shock waves flow was due to the condensation of humidity (Doerffer, 2000), which is predominant in tropic countries. The CFD simulation result at 4.5 meter using the initial patch value of 413 kPa also shows a slightly 57% error compared to the theoretical but a huge difference compared to the experimental value. This can be assumed that due to CFD simulation was modelled in an ideal condition and no humidity parameter was induced into the computed model.

Table 1: Results of peak overpressure in free air blast from various methods

Standoff Distance	1.5 m	4.5 m
Theoretical (Naumenko & Petrovskyi, 1956)	432 kPa	41.88 kPa
Experiment	413 kPa	13.2 kPa
CFD Simulation	413 kPa (initiate as Patch Value)	65.73 kPa

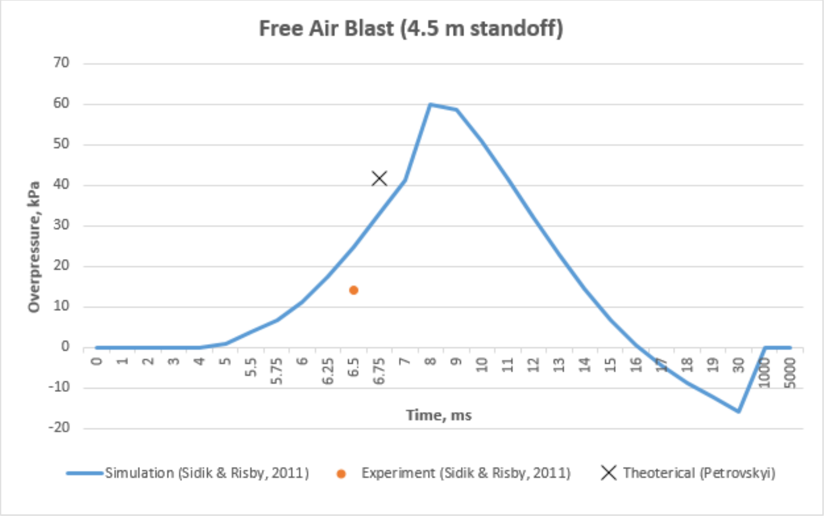


Figure 6: Free air blast results

Figure 7 shows the propagation of blast wave and interaction in computational domain. Blast phenomenon can be seen from Figure 7, starting from 5 ms where it can be shown that the blast wave expanded uniformly and at high speed.

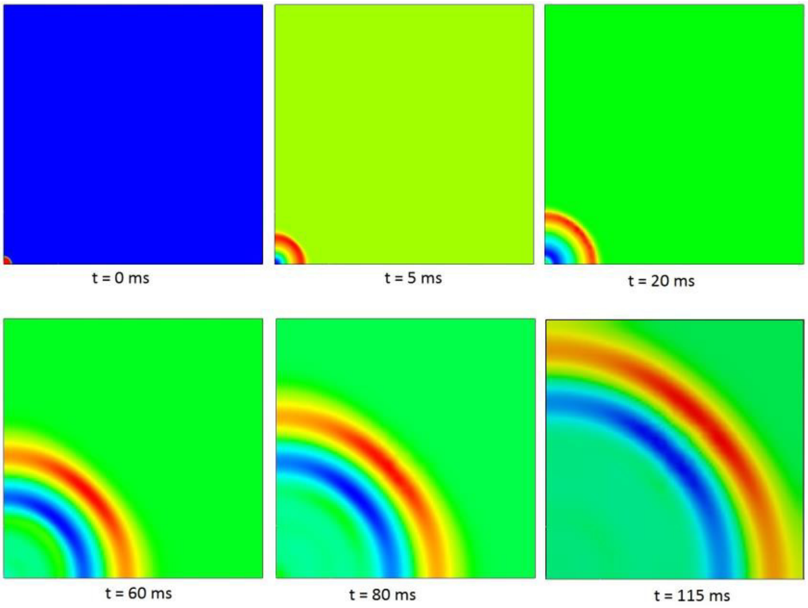


Figure 7: Pressure contour of free air-blast

4.2 Blast Loading response toward a structure

Figure 8 shows the pressure contour and propagation of blast wave and interaction with the capsule model on a symmetry plane. Blast wave phenomenon started from 1 ms onwards. The blast wave start to interact with the structure at t = 1 ms. When the blast wave hits the capsule wall, its start to reflect the blast wave and decrease the pressure of the blast. The diffraction of blast wave can be seen from t = 1.5 ms until t = 9.5 ms. Pressure contour subjected to capsule model was shown in Figure 9. Blast wave start to interact with capsule wall at t = 0.5 ms and start to propagate around the capsule wall. Figure 10 shows the blast loading under structure pressure-time monitored point data with three different locations; front, bottom and top of the capsule. The highest pressure been recorded from ‘front’ monitored point (Figure 12) is 63.76 kPa at 2.21 ms due to the position near from the blast source. For the ‘bottom’ monitored point (Figure 12) the highest pressure data been recorded is 33.87 kPa at 3.98 ms and lastly for ‘top’ monitored point (Figure 11) is 6.61 kPa at 7.15 ms. ‘Top’ pressure lower than ‘bottom’ and ‘front’ because the blast wave start to decrease the pressure due to the diffraction and reflected wave against the capsule wall. These results are helpful to better understand blast wave propagation and effects against the structure wall.

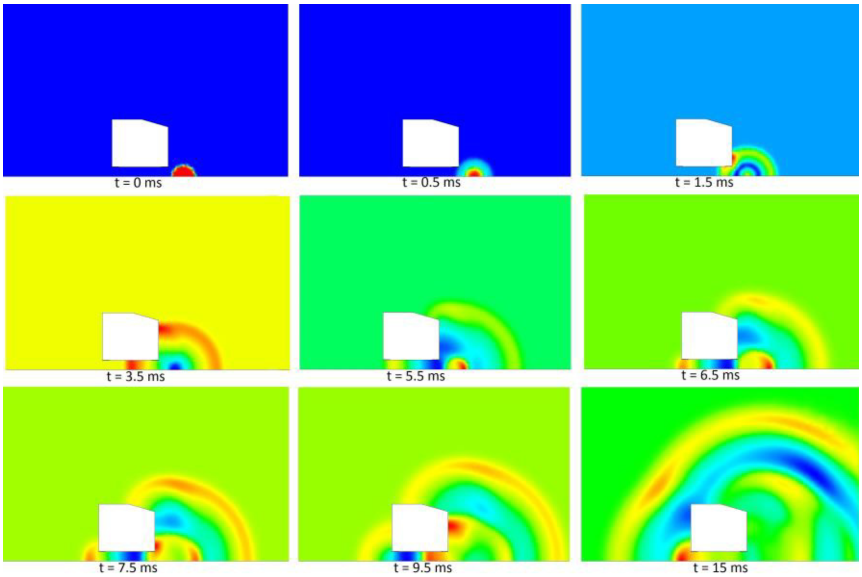


Figure 9: Pressure contour of blast loading under capsule

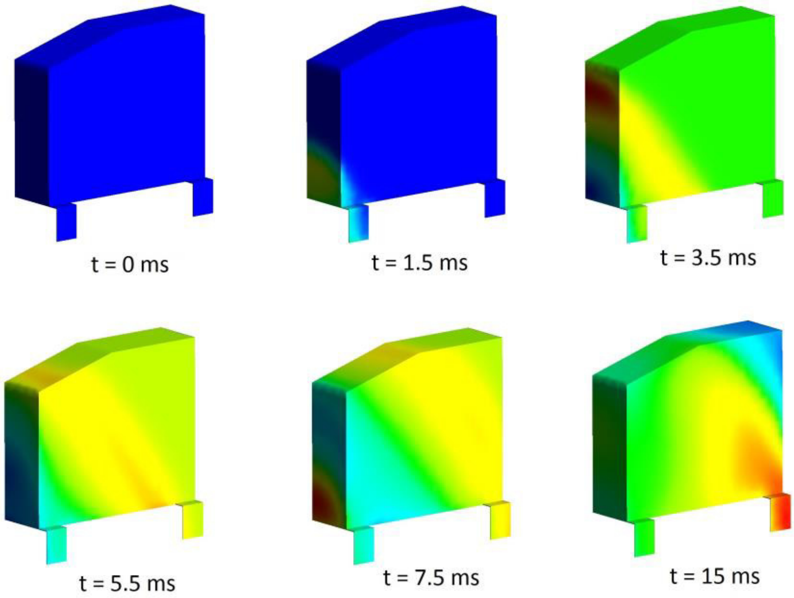


Figure 10: Pressure contour of the capsule

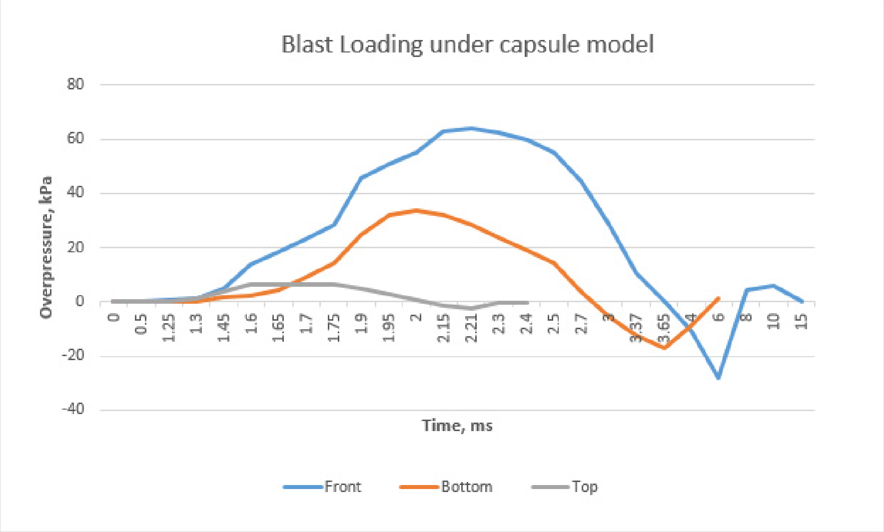


Figure 11: Blast loading under structure pressure-time monitored point data; Front, bottom and top

5 Conclusion

Predicting blast wave propagation in a free field condition and its interaction on structures such as a vehicle is important to understand the severity of its tenacious nature. CFD has advantages in accuracy, saves time and is widely used since it can visualize the wave patterns from the explosive detonation. To verify the capability of CFD in simulating blast wave, this research performed CFD simulations of free field blast and blast loading subjected to capsule model. Based on the results on free field air blast simulation using CFD, it is slightly different data on experimental result but moderately similar with the theoretical data. Factors that affected the accuracy of these results are based on the mesh size and solver settings. One of the major limitations of CFD when simulating blast waves, is CFD does not provide explosive materials to be set up for initial blast pressure. With this setting, ANSYS FLUENT will be more efficient and become one of tools to predict blast wave propagation.

6 Acknowledgement

The authors wish to acknowledge the research grant provided by Long Term Research Grant Scheme (LRGS/B-U/2013/UPNM/DEFENCE&SECURITY-P3) from the Malaysian Ministry of Education that lead to the realization of this work.

References

Alpman, E., Long, L. N., Chen, C.-C., & Linzell, D. G. (2007). Prediction of Blast Loads on Deformable Steel Plate Using Eular Equations. In *18th AIAA Computational Fluid Dynamics Conference*.

ANSYS Inc. (2009). ANSYS FLUENT 12.0 User's Guide.

Brode, H. L. (1959). Blast Wave from a Spherical Charge. *Physics of Fluids*, *2*(2), 217–229.

Cong, Y. (2014). *Use of Efficient Computational Fluid Dynamics (CFD) Techniques to Predict Blast Loads on Structures*.

Cranz, C. (1926). LIehrbuch der Ballistik. *Springer-Verlag Berlin Heidelberg*.

Doerffer, P., Szumowski, A., & Yu, S. (2000). The Effect of Air Humidity on Shock Wave Induced Incipient Separation. *J. of Thermal Science*, *9*(1), 45–50.

G. E. Fairlie. (1998). *The Numerical Simulation of High Explosives using AUTODYN-2D & 3D*. Explo 98', Institute of Explosive Engineers, 4th Biannual Symposium, England.

Grujicic, M., Snipes, J., & Chandrasekharan, N. (2013). Computational analysis of fluid-structure interaction based blast-mitigation effects. *Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications*, *227*(2), 124–142.

Hansen, O. R., Hinze, P., Engel, D., & Davis, S. (2010). Using computational fluid dynamics (CFD) for blast wave predictions. *Journal of Loss Prevention in the Process Industries*, *23*(6), 885–906.

Hopkinson, B. (1915). *British Ordnance Board Minutes 13565*.

Lee, K. (2009). Local response of W-shaped steel columns under blast loading, *31*(1), 25–38.

Naumenko, I. A., & Petrovskyi, I. G. (1956). *The Shock Wave of a Nuclear Explosion*.

Ngo, T., Mendis, P., Gupta, A., & Ramsay, J. (2007). Blast Loading and Blast Effects on Structures – An Overview. *eJSE International*, (Special Issue : Loading on Structure), 76–91.

Ofengeim, D. K., & Drikakis, D. (1997). Simulation of blast wave propagation over a cylinder. *Shock Waves*, *7*, 305–317.

Sidik, U., & Risby, M. S. (2013). *Experimental and Numerical Investigations of Free Field and Underground Wave Propagations Subjected to Blast Loading*.

Tulach, A., Mynarz, M., & Kozubková, M. (2015). CFD simulation of vented explosion and turbulent flame propagation. In *EPJ Web of Conferences* (Vol. 1, pp. 1–6).

Yang, Y., Liou, W. W., Sheng, J., Gorsich, D., & Arepally, S. (2013). Shock wave impact simulation of a vehicle occupant using fluid/structure/dynamics interactions. *International Journal of Impact Engineering*, *52*, 11–22.