Research Paper Engineering

Failure Modelling of Aluminium Plates Under Blast Loading

Agesh Markose Assistant Professor, TKM College Of Engineering,Kollam

ABSTRACT

Protection of the structures against blast loads is an important research topic in defense and domestic sector. Usually plates are used as the structural element for the blast mitigation and hence the analysis of the response of plates are important. These protective plates are used as blast mitigation elements against blast loads. These plates absorb the energy imparted to them. Thus the imparted energy to the structure and the occupants is reduced ensuring their safety. It is essential to predict the failure in order to avoid over design. In this paper, flat plates made of Aluminum are analyzed, with ABAQUS. Comparisons can be drawn relating to the failure initiation loads and locations and extent of damage.

KEYWORDS Blast Loading,Aluminum Plates, Finiteelement Method.

INTRODUCTION

The blast mitigation of structure has become a prominent area of research in the last few decades. Investigations were done in to estimate and simulate the resistance of buildings and structures against blast loads[1]. Plates and shells of varying materials and configurations have been used for the purpose of blast resistance. The explosion initiates a wave front which moves supersonically toward the boundaries [1, 2]. The duration and amplitude of the pulse depends on the explosion, geometry of the target and the obstacles around the target. Usually the blast loading occurs in a matter of milliseconds, with the exponential decay of the pressure pulse. The wave produces plastic deformation and subsequent fracture in target plates.It is seen that the target plates continue to deform plastically due to the inertia effect even after the loading. High frequency strain gauges or digital cameras can be used for capturing the response of the plates. But the experimental validation in blast scenarios is a time consuming and procedure. The alternative is establishing a simulation procedure or capturing the salient steps during the failure. It is not possible to include all the failure modes in a single simulation, due to the complexity and and sensitivity of the blast loading with respect to the various factors. The blast loading process is a highly dynamic and nonlinear process, and several investigations have been done for estimating the blast response. Balden andNurick [1] have tried to numerically simulate the blast response of steel plates. The effect of curvature and fixture on the boundaries were investigated by Kumar et.al[6]. Comprehensive literature on the blast mitigation effects of plates is not available easily because of the complexity of the loading and the various failure modes associated with it. The objective of the present paper is to numerically analyze the blast loading response of Aluminum pates and make a comparison against published experimental data[4].

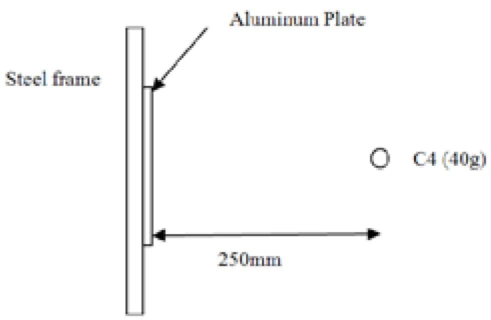


Figure 1: schematic of the set up

METHODOLOGY

In this paper we are trying to numerically simulate the blast interaction of aluminum plate and correlate with available experimental data published by Spranghers et al.[4]. Flat plates have been used widely as a structural element for validation of the numerical procedure with experiment. In the reported experiment [4] an Aluminum plate 400mmx400mmx3mm is clamped and supported on a steel sheet, Figure 1. Here steel sheet is provided with an aperture of 300mmx300mm exactly at the center. The aluminum sheet is mounted at the center, exactly above the aperture. An explosive (40gm C4) is located at standoff distance of 250mm away from the center of the plate. The detonation allows the high pressure gas waves to expand and fall normally on the plates. This induces shock waves [7], and plastic deformation of the aluminum plate through the aperture. The supporting plate undergoes only elastic deformation. In the reported experiment [4], author has measured the deformation of the plates with the help of high speed cameras. This allows for the validation of the numerical model with the experiment. The numerical simulation is done in ABAQUS 6.10.

a. Blast load modeling

An Eulerian approach is usually used for modeling the expanding gases. A coupled Eulerian and Lagrangian (ALE) approach is used by many authors to model the blast and its interaction with surroundings. Such simulations utilize fluid and structure interaction between the expanding gas and the structure (FSI) to arrive at the deflection of the structure [5]. This approach has been widely utilized and is found to be computationally expensive and time consuming. In many engineering problems the blast load generation and its propagation are of negligible interest. The focus is more on the structural response as a result of the applied dynamic pressure on the structure. In such cases it is much faster to use a pure Lagrangian method for finding blast response. The blast load can be implemented by using the blast function CONWEP [3] It helps to reduce the burden of explicitly simulating the progress of the shock wave and its interaction with surroundings. The model is capable of considering the effects of parameters like charge weight, distance of the charge and geometry of the target. The parameters in CONWEP function are based on experimental data and they can be used only by appropriately including the scaling parameters based on the actual explosion. The model does not consider the tunneling and shadowing effects during the explosion. The empirical model CONWEP has been implemented in the ABAQUS 6.10, and is used as such in the simulation.

b. Element Types

The selection of the element plays a crucial role in the accuracy of the results. The usual deformations for simple plate geometry include both in plane and out of plane plastic deformations. The formation of plastic hinges and the initiation of the crack tips follow. Previous authors have already done LS Dyna based simulations for the impact situation described here [4]. In this paper the simulation is attempted with the ABAQUS software which is very widely used for nonlinear deformations. In this paper we are trying to capture the plastic deformation and its shape which marks the onset of failure. The elements for the present analysis should be able to capture both in-plane and out of plane deformations. Due to this reason the solid element is selected. Hexahedral explicit elements are preferred for the simulation. The mesh quality can be assessed with the convergence studies. It is known that the central portion of the plate will undergo maximum deformation as compared with the outer regions. So the maximum mesh density is provided at the central region. Moreover taking advantage of the symmetry only quarter model is simulated. These steps are crucial for saving time, and computational effort. It can be seen that the aluminum plates will undergo the plastic deformation while the supporting plate will only undergo elastic deformation, due to its higher stiffness. For simplicity the clamp is omitted in the simulation. The initial response is plastic followed by an elastic rebound. This is due to the elastic contribution from the surrounding steel plates and partly due to the aluminum plates.

Material modelling under high strain rate is captured with Johnson – Cook model. The model describes the material flow stress as a function of strain, strain rate, and temperature. The dynamic simulation can be carried out with Johnson -Cook model with appropriate constants for Aluminum under consideration. The model parameters adopted for Aluminum are taken from literature. An explicit dynamic analysis with 3D dynamic elements was carried out. The response of the surrounding steel plate is purely elastic and can be modelled with elastic constitutive law. The choice of the explicit scheme or the implicit scheme largely depends on the type of the problem. Implicit schemes are unconditionally stable but still time consuming. If the step size is large it can again introduce errors. The explicit schemes are employed in dynamic cases where the numbers of steps are not excessive.

For optimizing the mesh density ,a quarter plate as shown in Figure 2. was modelled, utilizing the symmetry conditions. 3D solid elements were used to model the plate. The mesh density is maintained maximum at the center where the maximum deformation occurs. The symmetry and fully constrained boundary conditions are sufficient for imposing the boundaries conditions in the experiment. The Quad type mesh, with C3D8R is used which can give sufficiently accurate results under similar simulations. The mesh convergence study was conducted and the final deformation does not seem very sensitive to the mesh density. However the response at .45 sec milliseconds, just after the loading step, showed sensitivity to the mesh density. The mesh density can be increased to increase the solution accuracy as long as the computing resources permit. A certain value of mesh density was selected which is compromise between speed and accuracy. This value of the mesh size is taken for subsequent simulations. The validity of the result is cross checked with the experimental data.

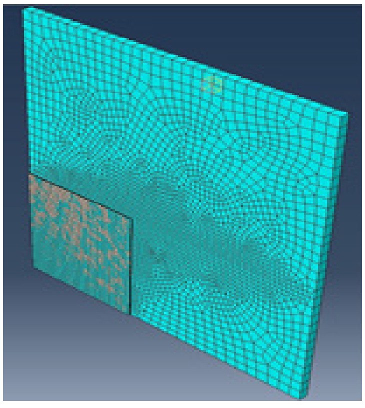


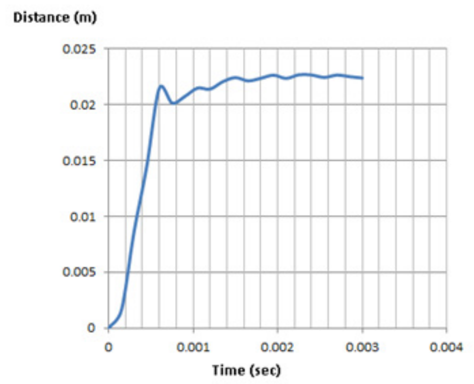
Figure 2. Quarter Plate with varying mesh size

RESULTS

We have carried out impact loading aluminum panel and obtained the plastic deformation and stress distribution contours. For flat plates of about 3mm thickness wehave obtained the plastic deformationresults comparable with the literature as represented Table 1 and Figure 4. The mesh densities were varied to see the effect on the deformation. The numerical results were validated by comparing with data in the published literature. So it is seen that blast response under CONWEP type loading is reasonable. Proper use of the scaling factor can bring down the error further, which can be attempted as future work. The in plane strains (Exx) were also found from the simulations and compared with the published experimental data. The deviations strains were found within 5%. The average error is in displacement is about 10%.

TABLE – 1
Mid-point Displacement

Mid-point deflection(mm)	Experiment	FEM simulation
	20.0	22.3



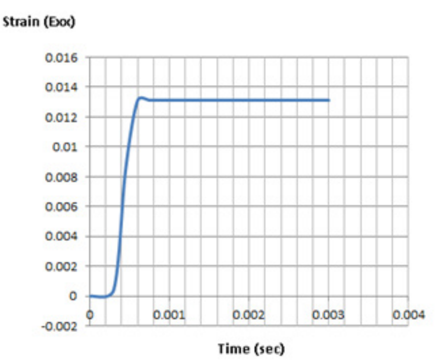


Figure3. Plate deformation

CONCLUSIONS

The paper presents numerical simulation of the blast interaction of aluminum plate, subjected to air blasts from explosives kept in front of the mid-section. The CONWEP blast function with ABAQUS produces reasonable prediction for the given standoff distance. For further estimation of the simulation capabilities the procedure may be repeated by varying the parameters.

REFERENCES

[1] Balden VH, Nurick GN. Numerical simulation of the post-failure motion of steel plates subjected to blast loading. International Journal of Impact Engineering 2005;32:14-34. | [2] Bulson PS. Explosive loading of engineering structures: a history of research and review of recent developments. UK: E&FN SPON; 1997. | [3] Showichen A, 2008, Numerical analysis of vehicle bottom structures subjected to anti-tank mine explosions. PhD Thesis, Cranfield University, UK. | [4]Spranghers K, Vasilakos I, Lecompte D, Sol H, Vantomme J, 2013, Numerical simulation and experimental validation of the dynamic response of aluminum plates under free air explosions, International Journal of Impact Engineering 2013,(54), 83-95 | [5]Børvik T, Hanssen AG, Langseth M, Olovsson l. Response of structures to planar blast loads - a finite element engineering approach. Computers and Structures 2009,(87), 507-20. | [6] Kumar P, LeBlanc J, Stargel DS, Shukla A. Effect of plate curvature on blast response of aluminum panels. International Journal of Impact Engineering 2012;46:74-85 | [7]Yuen SCK, Langdon GS, Nurick GN, Pickering EG, Balden VH, 2012, Response of V-shape plates to localised blast load: Experiments and numerical simulation. International Journal of Impact Engineering, (46), 97-109. | [8] Zukas A, Impact Dynamics,USA: Jihn Wiley and Sons.1982. | [9] Jones N. Structural impact. UK: Cambridge University Press; 1989.