Transactions, **SMiRT-24**
BEXCO, Busan, Korea - August 20-25, 2017
Division IX (include assigned division number from I to X)

MODELLING OF STEEL CONCRETE PANEL SYSTEMS SUBJECTED TO IMPACT LOADS

Samer Gendy[1], Ashraf Ayoub [2]

[1] Ph.D. Candidate, School of Mathematics, Computer Science & Engineering, Department of Civil Engineering; City, University of London, London
[2] Professor, School of Mathematics, Computer Science & Engineering, Department of Civil Engineering; City, University of London, London

ABSTRACT

Steel concrete (SC) panel system is widely used in nuclear containment structures. They are believed to be very efficient in protecting structures against blast and impact loads. The system typically consists of a thick concrete core, sandwiched between two steel plates. The steel plates are regularly connected by shear connectors spaced in both directions.

In this paper, the finite element method is used to simulate the behaviour of SC panels subjected to impact and shock loads. The adopted finite element model is based on a fibre beam element formulation. A new displacement based fibre beam element is developed that considers large displacement using the geometric stiffness matrix to account for the P-Delta effect.

The element also employs an explicit time integration method instead of the traditional implicit method for the solution of the equation of motion. In this method, the accelerations of the nodes are calculated directly without the need to invert the stiffness matrix but only the mass matrix is inverted. No iterations are required and no convergence checks are needed, therefore a large number of inexpensive time steps is required. The Stability of the explicit method is also discussed.

Moreover, fibres added to the concrete mix with different percentages are also considered in the material model along with the strain rate effect of both concrete and steel. The model can predict the displacements and internal forces of the element with adequate precision. A non-composite SC panel and an SC panel with shear connectors, both tested under impact loads, available from the literature were used to be modelled using the new fibre beam element implemented in the multi-purpose finite element program FEAP.

The programme can be extended to model large nuclear containment structures subjected to impact loads, giving the designer an efficient and accurate tool to simulate the behaviour of such important structures under impact and dynamic loading.

INTRODUCTION

Composite steel-concrete (SC) panels consist of a thick concrete core, sandwiched between two steel plates. The steel plates are typically connected by shear connectors (anchors) spaced in both directions as shown in figure (1). The system usually does not contain any horizontal, vertical reinforcement or any stirrups. These panels are believed to be very efficient in protecting structures against blast and impact loads [Anandavalli et al. (2012), Liew and Wang (2011), Vecchio and McQuade (2011) and Sohel and Liew (2014)]. Also Remennikov (2013) found that SC panels are an effective means of protecting structures against extreme impact and blast loading due to their high strength and high ductility characteristics.

Another advantage of this system is the use of the steel plates as formwork during the construction which accelerates the construction process. The steel concrete panel system is widely used especially in nuclear containment structures, offshore structures, liquid containments, blast walls in factories and basements, caissons, tunnels and in shear walls and cores of multi-storey buildings.

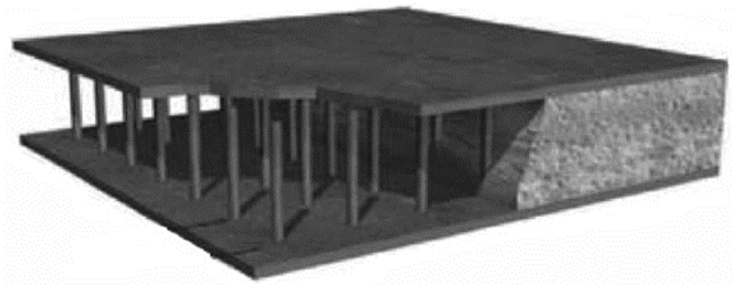

Figure 1. Steel concrete steel panel, figure from Liew and Wang (2011)

The finite element method is regularly used to simulate the behaviour of SC panels when subjected to impact and blast loads. The finite element method allows detailed understanding of the performance of the SC panels under dynamic loading. The models can depict the displacement and internal forces with adequate precision. Most detailed finite element models, however, are computationally expensive. On the other hand, fibre beam models have proven to be able to provide a good balance between simplicity and accuracy. However, the improvement of the software's algorithm of the fibre beam element is still ongoing in order to more accurately consider material and geometric nonlinearity, large deformations, failure analysis, crack propagation and different time integration techniques.

Zhu et al. (2013) used the LS-DYNA commercial finite element software to analyse impact tests on steel plate concrete panels against scaled-aircraft. They found that the outcomes in terms of fracture process, residual velocities, damage parameters, velocity time histories, deformations and damage to aircraft correlated well with the test and analytical results of discrete element method. Anandavalli et al. (2011) used the ANSYS commercial software for the blast analysis of steel-concrete composite panels. They used shell elements to model the steel plates, solid elements to model the concrete core and link elements to model the shear connectors. Kong et al. (2012) examined non-composite SCS panels with axially restrained connections experimentally and numerically under impact loading using LS-DYNA. The concrete core of the panel was simulated using constant stress solid elements while the steel faceplates were simulated using Belytschko-Tsay shell elements. Beam elements were used to model the reinforcing steel and the bolts of the keyed inserts were simplified as square bars. The researchers found that the strain rate effects of the steel and the concrete have major effect on the numerically predicted bending strength and tensile membrane resistance of the panels.

THE FIBRE BEAM ELEMENT

The adopted finite element model in this study is based on a fibre beam element formulation. The fibre beam element is created by the division of the element into several sections. These sections are also divided into what is called fibres that represent concrete and steel. Each of the concrete and steel fibres is modelled using a constitutive model that is able to simulate all the nonlinear behaviour of the material. The concrete and steel material models adopted in this fibre beam element consider the strain rate effect, so that the material parameters of the concrete and steel were adjusted using a dynamic amplification factor. The dynamic amplification factor was investigated for concrete, fibre reinforced concrete and reinforcing steel using the Split Hopkinson pressure bar device [Pająk (2011), Lok and Zhao (2004) and Malvar (1998)]. Furthermore, the section stiffness and section forces are determined by the integration of the fibres for the required section. Using the assumption that plane sections remain plane after deformation, the strain in each fibre can be calculated from the curvature and the section strain.

The present element use a displacement based formulation which is easy to implement and gives accurate results making the model a good compromise between simplicity and accuracy. However, a large number of members division is required to accurately simulate the response of the structure, mainly in plastic zone regions, since the calculation of the curvature is performed using a linear equation which is derived from a cubic shape function. If accepting a small number of division, the inelastic curvature will not be properly presented and the efficiency of the element will be questionable.

An incremental solution is always adopted to solve nonlinear problems. In this technique, the displacement is divided into increments and the problem is solved like a support displacement problem. The displacement control method is a stable method and convergence is easily achieved.

EXPLICIT TIME INTEGRATION METHODS

The explicit dynamic analysis is a mathematical method for integrating the equations of motion through time. The explicit procedure is suitable for short time duration analysis such as impact and shock problems. This method is known to be conditionally stable so a small time increment has to be used to ensure that the solution is stable. Table (1) shows a comparison between the explicit and implicit methods.

Table 1: Comparison between explicit and implicit methods

Explicit method	**Implicit method**
Suitable for short transient problems (dynamic)	Suitable for static and quasi static problems
Use the inversion of the M matrix	Use the inversion of the K matrix
Conditionally stable	Unconditionally stable
No convergence check is required	Convergence check is required
Requires many relatively inexpensive time steps	Requires small number of expensive time steps
Example: central difference method	Example: the Euler backward method

To ensure the stability of the explicit method a small time step is required. The selected time increment (Δt) must be less than the stable time increment (Δ_{tmin}). From Askes et al. (2014), the stable time increment can be estimated by:

$$\Delta t_{min} = \frac{L}{\sqrt{\frac{\lambda + 2\mu}{\rho}}}$$

Where:

$$\lambda \equiv \frac{\nu E}{(1+\nu)(1-2\nu)}$$
$$\mu \equiv \frac{E}{2(1+\nu)}$$

Where λ and μ are lame's parameters, ν is poisons ratio, E is the young's modulus, ρ is the material density and L is the element length. The formulation of the new elements involves that a mass matrix be constructed. A diagonal lumped mass matrix is used to make its inversion possible. All diagonal terms of the lumped mass matrix have to be specified.

EXPLICIT FORMULATION OF THE ELEMENT

The beam element presented herein uses a displacement-based formulation where equilibrium is satisfied in a weighted integral sense. The internal and external forces are summed at each nodal point. Then the nodal accelerations are calculated by multiplying the forces with the inverse of the nodal mass. The solution for an acceleration increment in the global system $\Delta\ddot{U}_{(global)}$ is computed by:

$$\Delta\ddot{U}_{(global)} = [M]^{-1} \times \Delta\hat{F}_{elem(global)}$$

Where M is the lumped mass matrix and $\hat{F}_{elem(global)}$ is the global internal dynamic load. Hence,

$$\hat{F}_{elem(global)} = F_{elem\,(global)} + (M)\,\ddot{U} + (C)\acute{U}$$

In which C is the damping matrix and the velocity Ú and the acceleration Ü are calculated explicitly using the Newmark beta-gamma method while taking $\beta = 0$ and $\gamma = 0.5$.

The damping matrix C can be determined using the Rayleigh damping equation:

$$C = \alpha_m M + \beta_k K$$

The determination of the stiffness matrix K_{elem} and load vector F_{elem} is first calculated in the local system, where rigid body modes are eliminated, then transformed to the global system.

Large Displacement

The new displacement-based fibre beam element is also enhanced to consider the second order effect, which allow the calculation of large displacement that commonly accompany impact problems. The new element is developed within a two-dimensional Lagrangian corotational beam formulation. Both the element internal and external geometric stiffness matrices are considered. The transformation between the local and the global system is made using an enhanced transformation matrix that takes into consideration the current angle of the co-rotating system with respect to the global coordinate system.

The element tangent stiffness matrix is derived from virtual displacements by multiplying the equilibrium equations by specific weighting functions and integrating over the length of the element. The differential equations of equilibrium are therefore satisfied in a weighted integral sense.

The stiffness matrix is updated by adding the internal geometric stiffness matrix term K_g.

$$\boldsymbol{K} = \left(K_g + \int_0^{L0} N_\delta^T K N_\delta dx\right)$$

Where:

$$K_g = P\begin{bmatrix} \frac{2L}{15} & \frac{-L}{30} & 0 \\ \frac{-L}{30} & \frac{2L}{15} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

And the resisting load vector is evaluated as:

$$F_{elem} = \int_0^{L_0} N_\delta^T D_\Sigma^i \, dx$$

And :

$$N_\delta = \begin{bmatrix} \frac{1}{L} & \left(1-\frac{4x}{L}+\frac{3x^2}{L^2}\right)^2 q_1 + \left(1-\frac{4x}{L}+\frac{3x^2}{L^2}\right)\left(-\frac{2x}{L}+\frac{3x^2}{L^2}\right) q_2 & \left(1-\frac{4x}{L}+\frac{3x^2}{L^2}\right)\left(-\frac{2x}{L}+\frac{3x^2}{L^2}\right) q_1 + \left(\frac{2x}{L}+\frac{3x^2}{L^2}\right)^2 q_2 \\ 0 & -\frac{4}{L}+\frac{6x}{L^2} & -\frac{2}{L}+\frac{6x}{L^2} \end{bmatrix}$$

q_1 and q_2 are the rotations in the local system.

And D is the section resisting forces in the local system and consists of P and M, where P is the axial force while M is the bending moment.

The transformation between the local and the global system is made using an enhanced transformation matrix that takes into consideration the current angle of the co-rotating system with respect to the global coordinate system.

$$Q_{global} = T^T Q_{int}$$

$$\boldsymbol{K}_{global} = K_G + T^T \boldsymbol{K}\, T$$

Where:

$$T = \begin{bmatrix} -\frac{\sin\beta}{L} & \frac{\cos\beta}{L} & 1 & \frac{\sin\beta}{L} & -\frac{\cos\beta}{L} & 0 \\ -\frac{\sin\beta}{L} & \frac{\cos\beta}{L} & 0 & \frac{\sin\beta}{L} & -\frac{\cos\beta}{L} & 1 \\ -\cos\beta & -\sin\beta & 0 & \cos\beta & \sin\beta & 0 \end{bmatrix}$$

K_G is the external geometric stiffness matrix, and β is the angle between the element chord and the global system.

VALIDATION OF THE NUMERICAL MODEL AGAINST IMPACT

Two experiments from the literature are used to validate the fibre beam finite element. In those two experiments, instrumented drop weight impact tests were used to examine the dynamic behavior a non-composite and an SC panel with shear connectors.

Modelling of non-composite SC panel under Impact

The behaviour of non-composite SC panels under impact loads is examined using the explicit fibre beam element. The experimental work of Remennikov et al. (2013) is used for the comparison with the new element. A 600 kg drop hammer was used to fall freely on a non-composite SC panels. The panel was axially restrained and the hammer was situated at 3 m height and was allowed to hit the panel several times. Load cell, displacement gauges and a data acquisition system were installed in this experiment. The tested panels had 80 mm concrete core thickness and 3 mm steel plates. The panel had a mild steel faceplates with a yield stress of 271 MPa and a normal weight concrete infill with compressive strength equal to 23 MPa.

Only half of the panel was modelled as a beam element fixed from one side. The panel length was divided into 4 elements and each element was further divided into 5 sections. The sections were also divided into 12 fibres that represent the concrete and 2 fibres that represent the steel plates. The strain rate effect was taken into account by the finite element model.

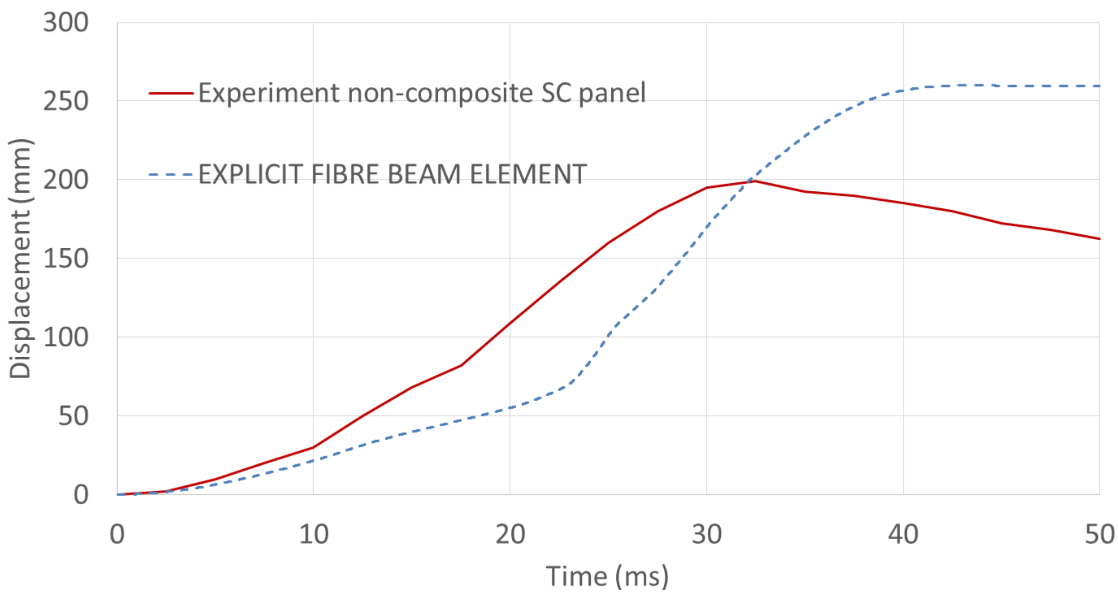

Figure 2. Comparison between the experimental results and the explicit fibre beam element for the non-composite SC panel subjected to impact.

The non-composite SC panel faced severe local collapse in the concrete core due to the lack of confinement. In figure (2), the finite element slightly overestimated the displacement at the final stage of the analysis (i.e. between 35 to 50 ms). However, It is clear that the element can simulate the impact test to a very good extend.

Modelling of SC Panel & Element Stability Discussion

Next, the SC panels tested by Sohel and Liew (2014) are simulated using the explicit fibre beam element. In their work, the researchers tested Eight SC sandwich slabs measuring 1200 × 1200 mm^2 under impact. Panel SLFCS6-80 had a steel plate thickness of 5.96 mm, a Lightweight concrete core with 1% fibre of 80 mm thickness. The concrete density was equal to 1445 kg/m^3, the cylinder strength of the concrete was 28.5 MPa, the elastic modulus of concrete was 14.0 GPa and the yield strength of steel plate was 315.0 MPa.

The SC panel was simply supported with a span of 1000mm. The drop height was chosen as 3m while the projectile mass was about 1246 kg. Only half of the panel was modelled using one fibre beam element. The load displacement curves of the experiment are compared with the fibre beam element. Several time steps were used, the first one was 0.01 sec and the smallest was 0.00001 sec. However, it was found that the solution became stable only when a small Δt is used. By using a time step of 0.001 sec for instance, the output was still numerically unstable. It can also be seen that using a bigger time step has produced a higher load displacement curve although the shape of the load displacement path was maintained. In figure (3), SLFCS6-80 is modelled using the explicit fibre beam element. All the parameters are fixed except the time step.

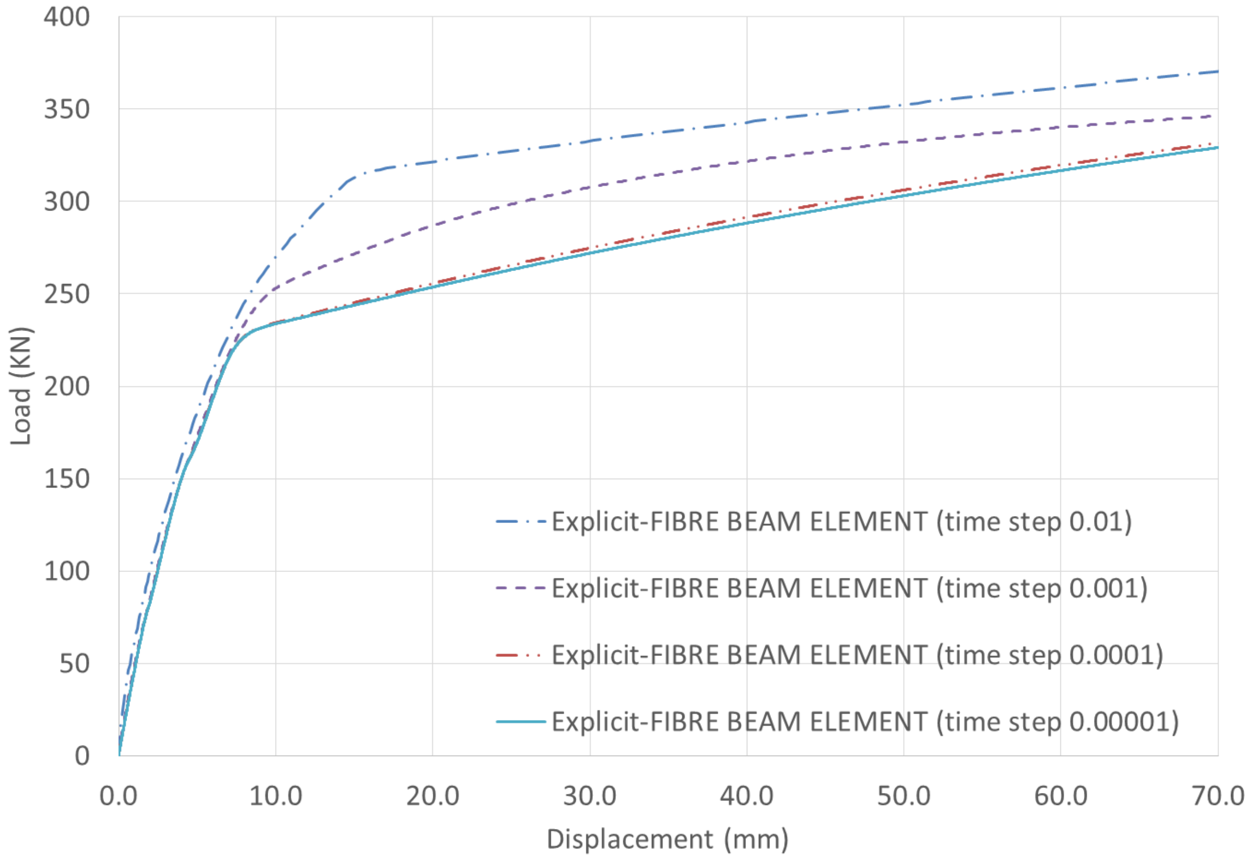

Figure 3. Load-Deflection Comparison of the Explicit Fibre Beam Element for SC panel SLFCS6-80 using different time step values.

Care should be taken when using explicit beam elements to ensure the numerical stability of the solution. Consequently, in figure 4, the stable explicit fibre beam element is compared with the experimental results. Good agreement with the refined model is observed.

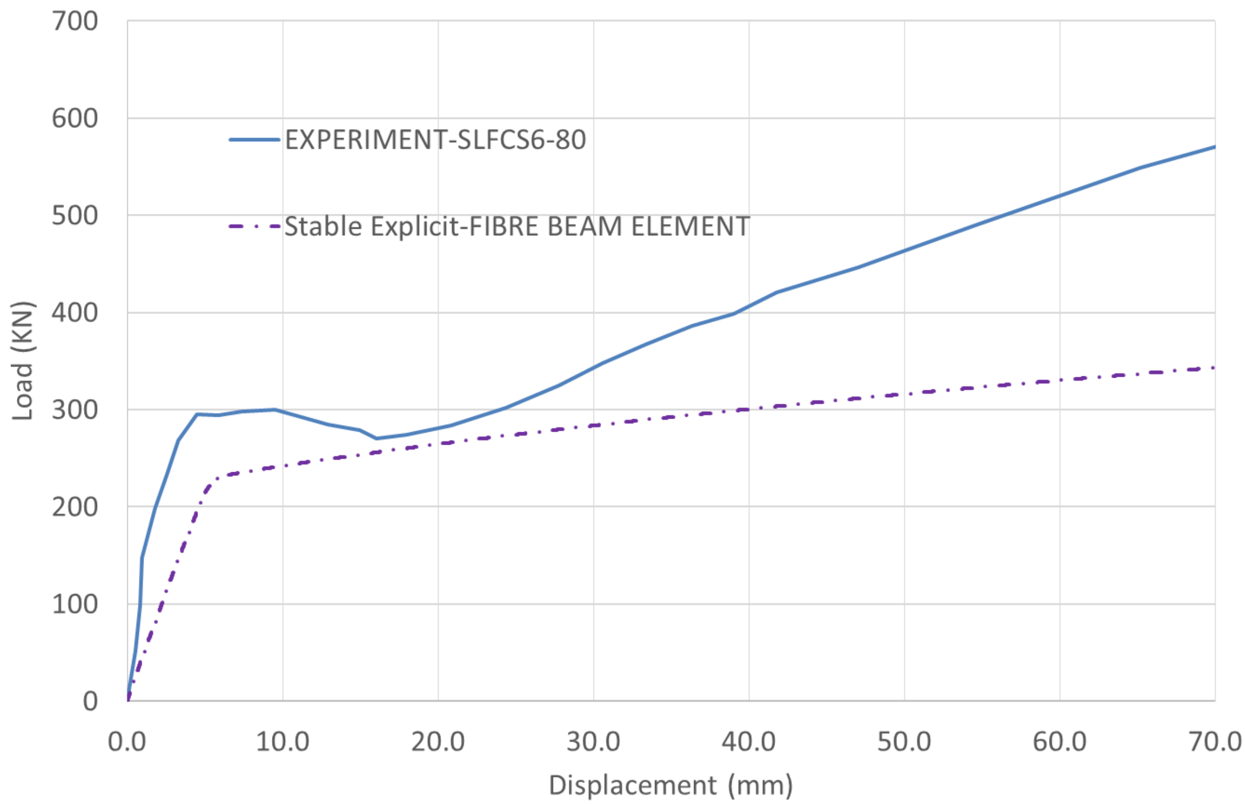

Figure 4. Load-Deflection Comparison between the Experiment Results and the numerically stable Explicit Fibre Beam Element for SC panel SLFCS6-80

CONCLUSION

A new displacement based fibre beam element was presented. The element uses an explicit time integration method and can be utilized to solve dynamic and impact problems. No internal element iterations are required and no convergence checks are needed. The element also considers large displacement and benefits from advanced material models for steel and concrete that account for the strain rate effect. The numerical stability of the element was discussed.

The element can be used to model large nuclear containment structures subjected to impact loads, giving the engineer an effective tool to simulate the nonlinear behaviour of such important structures under impact and dynamic loading.

REFERENCES

Anandavalli, N., Lakshmanan, N., Iyer, N., Samuel Knight, G. and Rajasankar, J. (2011). “A Novel Modelling Technique for Blast Analysis of Steel-Concrete Composite Panels,” *Procedia Engineering*, 14, pp.2429-2437.

Anandavalli, N., Lakshmanan, N., Rajasankar, J. and Prakash, A. (2012). “Numerical Studies on Blast Loaded Steel-Concrete Composite Panels,” *Journal of Civil Engineering and Science*, 1(3), pp.102-108.

Askes, H., Rodríguez-Ferran, A. and Hetherington, J. (2014). “The effects of element shape on the critical time step in explicit time integrators for elasto-dynamics,” *International Journal for Numerical Methods in Engineering*, 101(11), pp.809-824.

Kong, S., Remennikov, A. and Uy, B. (2012). “Numerical simulation of the response of non-composite steel-concrete-steel sandwich panels to impact loading,” *Australian Journal of Structural Engineering*, 12(3).

Liew, J. and Wang, T. (2011). “Novel Steel-Concrete-Steel Sandwich Composite Plates Subject to Impact and Blast Load,” *Advances in Structural Engineering*, 14(4), pp.673-688.

Lok, T. and Zhao, P. (2004). “Impact Response of Steel Fiber-Reinforced Concrete Using a Split Hopkinson Pressure Bar,” *Journal of Materials in Civil Engineering*, 16(1), pp.54-59.

Malvar, L. (1998). “Review of Static and Dynamic Properties of Steel Reinforcing Bars,” *ACI Materials Journal*, 95(5), pp.609-616.

Pająk, M. (2011). “The Influence of the Strain Rate on the Strength of Concrete Taking Into Account the Experimental Techniques,” *Architecture Civil Engineering Environment Journal*, 3/2011, pp.77-86.

Remennikov, A., Kong, S. and Uy, B. (2013). “The response of axially restrained non-composite steel–concrete–steel sandwich panels due to large impact loading,” *Engineering Structures*, 49, pp.806-818.

Sohel, K. and Liew, J. (2014). “Behaviour of steel–concrete–steel sandwich slabs subject to impact load,” *Journal of Constructional Steel Research*, 100, pp.163-175.

Vecchio, F. and McQuade, I. (2011). “Towards improved modeling of steel-concrete composite wall elements,” *Nuclear Engineering and Design*, 241(8), pp.2629-2642.

Zhu, X., Pan, R. and Sun, F. (2013). “FEM Analysis of Impact Tests for Steel Plate Concrete Panels against Scaled-Aircraft Impact,” *Applied Mechanics and Materials*, 477-478, pp.777-783.