

Numerical Modelling of a Triple Core Sandwich Structure under Axial Collapse

Siavash T. Taher*, S. Debnath
 Department of Mechanical Engineering
 Curtin University of Technology
 Miri, Sarawak, Malaysia
 *siavash@curtin.edu.my

Simin Ataollahi
 Mechanical and Materials Engineering Department
 Universiti Kebangsaan Malaysia, 43600 UKM
 Bangi, Selangor, Malaysia
 siminataolahi@gmail.com

Abstract— A finite element numerical investigation was carried out to evaluate the reaction and energy absorbing capability of composite energy absorber blocks positioned under a the floor of helicopter. A finite element explicit dynamic analysis module incorporated ANSYS/LS-DYNA computer software put into practice to simulate crashworthy behaviour and energy absorption characteristics of a new economic crushable composite triple layer sandwich structure. In a previous paper, the authors developed the concept of the triple layer thin wall foam-filled block structure and submitted experimental results of the crash behaviour and crashworthiness characteristics of such structure. The achieved finite element data of axial compression simulation of composite crushable structure are compared with actual experimental results of total absorbed energy, force-displacement diagram and crush region features, showing very good conformity. Numerical data and experimental results showed good correlations in pre-crush and post crush loads, average load and absorbed crush energy with two different types of triggering modifications.

Keywords— Composite sandwich structure; crashworthy; finite element analysis; explicit dynamic; Polyurethane foam; Energy absorption; trigger; ANSYS/LS-DYNA.

I. INTRODUCTION

Collapsible impact energy absorbers made of fibre reinforced composite materials are structural elements used in a broad range of automotive and aerospace applications. The energy absorption capability of an exposed crashworthy element or system is greatly affected by its structural design and material properties. There are a number of potential economic and functional benefits to be derived from the use of fibre reinforced plastic (FRP) composite materials in automobile and aircraft construction. The gains arise through increased strength and durability features to weight reduction and lower fuel consumption [1, 2].

Crash energy management is one of the primary design requirements that the body structure must meet. In particular, researcher's attention has been directed towards the improvement of structural vehicle crashworthiness by using FRP composites in specific vehicle parts as collapsible absorbers of crash energy i.e. as structural members that are able to absorb

large amounts of impact energy, while collapsing progressively in a controlled manner [3, 4].

Nowadays most of the research on the energy absorption of composite materials has been limited to the axial compression of tubular structures and most of the research done has been experimental [5]. With no doubt, the validation of analytical and numerical tools for accurate simulation of structural response to crash impacts is an important aspect of crashworthiness research.

Conventional sandwich panel under high energy impact loading, often collapse in a brittle and somewhat unpredictable manner. Using everting devices to control failure of sandwich panel has been reported previously by Taher et al. [6]. A set of designs are based on the concept of the "tied-core" sandwich, i.e. the use of additional core reinforcements that act to tie the opposing facings of a sandwich together had been tested by Pitarresi et al. [7].

In a previous paper, the authors developed the concept of the "triple-layered" foam-filled block and submitted experimental results of the crash behaviour and crashworthiness characteristics of such structure.

The main objective of this present investigation is dealing with the implementation of the finite element explicit dynamic analysis code module incorporated ANSYS/LS-DYNA computer software to the simulation of the crash behaviour and energy absorption characteristics of a novel triple-cell cost-effective crashworthy composite sandwich structure. The obtained numerical results are compared with actual experimental data.

II. FINITE ELEMENT MODEL

A. Model size

The model comprised two types of finite elements: shell and brick elements. The composite faceplates and internal laminates were modelled with four-node thin shell elements, while the hollow composite shell structure has a uniform thickness of 0.2 mm all over its cross-section. Moreover, eight-node brick elements were employed in

the modelling of the sandwich material foam core. An overall picture of the shape and the dimensions of the test specimens are given in Fig. 1a. Finite element model of this structure is depicted in Fig. 1b.

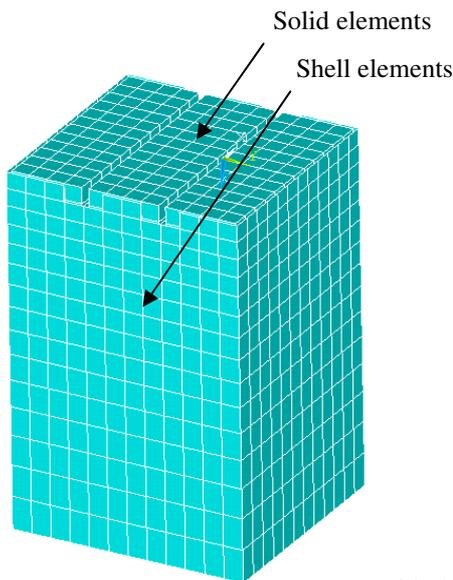
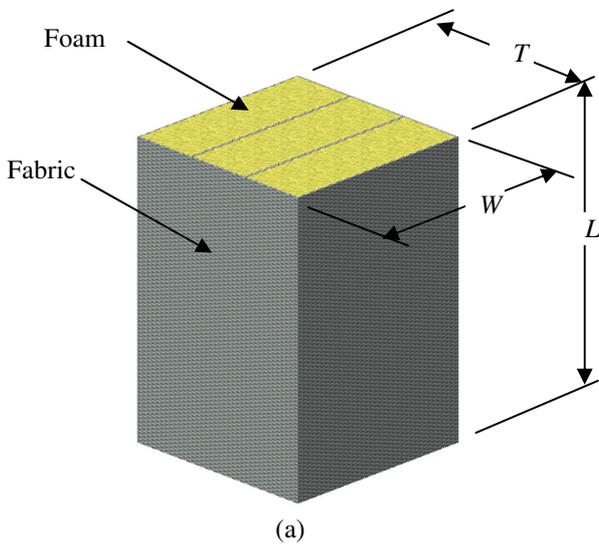


Figure 1. General layout and dimensions of the sandwich energy absorbing tested specimens, (a) general layout and dimensions of the sandwich energy absorbing tested specimens, (b) finite element model of specimen.

B. Trigger Modelling

As reported by Thornton [3], composite structures generally require a collapse trigger mechanism to promote stable, progressive crushing. Otherwise, sudden catastrophic failure can occur. Generally, collapse trigger mechanisms create a region of locally elevated stress at one end of the block from which

failure initiates and then propagates. Two types of collapse trigger mechanism were investigated:

Two straight cut-outs with width and depth 3 and 5 mm respectively to one end of the specimen that trim internal fibre-glass laminates (also called groove trigger or I type trigger that simulated by trimming internal composite shell laminates as shown in Fig. 2a.

Two chamfer cut-out (5x5 mm) to two opposite corners of one end of the specimen perpendicular to intermediate fibreglass reinforcements in the name of bevel trigger or “V” trigger that simulated by two step cut-out on corners of model as depicted in Fig. 2b.

III. FINITE ELEMENT DISCRETIZATION

The specimen model cross-section geometric configuration and the necessary dimensions are shown in Fig. 1a. Both the upper moving and the lower stationary head of the press used in the experimental testing works were modelled using one eight-node brick element for each of them without any further discretization since the material model assigned to these parts was the rigid one. The mesh of finite elements was not uniform along the specimen axis and was featured by a space ratio during meshing. Therefore, sizes of mesh increases from frontal crushing face toward base of block as depicted in Fig. 3. Regarding the specimen length, two types of specimens were tested: short and long blocks with length, L equal to 100 and 150 mm respectively. The width, W of blocks was also variable, taking three distinct average values that were equal to 40, 60 and 80 mm. For specimens two different thickness, T was used approximately equal to 40 and 60 mm. Fibre orientation was $\pm 45^\circ$ to the block axis.

A. Material Property

This section refers to the material types used to model the distinct parts of the specimen, the moving platen and the stationary base. Three different types of material models were used from the library of the ANSYS/LS-DYNA code in the development of finite element analysis described in ANSYS/LS-DYNA user's guide:

- Composite damage model material,
- Crushable foam model,
- Rigid model.

1. For the fabric bidirectional laminated composite material modelling, the Composite damage model material was used, which is based on Material Model 22 in library of the LSDYNA code. Unfortunately Material Model 55 “mat_enhanced_composite_damage” is not available in ANSYS/LS-DYNA which considers more failure options than Material Model 22.

2. Compressive and high strain rate properties of low density foams are widely studied in the published literature [8-11]. For the foam core material modelling, Crushable foam

model was used, where the properties and the corresponding stress-strain curve under compression derived from experimental test.

3. The upper moving and down stationary platens were modelled using the material rigid model. In such a model, where a theoretically deformable body can be considered as a practically rigid due to its much higher mass and stiffness related with those it interacts, this modelling technique may be preferred. Elements, which are rigid, are bypassed in the element processing and no storage is allocated for storing history variables (no stresses, no strains occur in a rigid body); consequently, the rigid material type is very cost efficient. Additionally, meshing of a rigid body with a very simple shape is not necessary as it happens in the current model where the cubic rigid platens are modeled each with a single eight-node brick element. The appropriate properties, may be given as input, are the mass density, Young's modulus and Poisson's ratio (the corresponding values of a mild steel have been given).

B. Contact interfaces modelling

The treatment of sliding and impact along interfaces has always been very important in order to simulate a "physical" performance between interacting structural members being in contact, i.e., keeping geometric boundaries without models independent parts penetrating each other. Moreover, from contact treatment point of view, attention must be paid in special cases, as in the current one described herein, where self-contact may occur in an excessively buckled structural member and/or new material boundaries become apparent due to the applied element erosion/deletion modelling technique described above. The contact interface types required for the appropriate processing of any possible contact interaction among all parts of the model are defined below. Five in total different types of contact interface were used in the modelling of the sandwich block, namely in terms of the ANSYS/LS-DYNA code:

- The automatic general (AG) contact interface type,
- The eroding single surface (ESS) type,
- The tied surface to surface (TDSS) type,
- The Eroding surface to surface (ESTS) type, and
- The tiebreak surface to surface (TSTS) type,

IV. RESULTS AND DISCUSSION

The validation of the finite element model described so far is made by direct comparison to the results and visual observations of the experimental works pertaining to the quasi-static axial compression of the modeled sandwich blocks. The geometry and structure of the specimen, the properties of the construction materials and all other testing details used in the finite element model were exactly the same as the ones of the experimental works. The only difference between the compressive tests and the numerical simulation model, was that the actual crosshead speed during the tests was just 20 mm/min

2nd CUTSE International Conference 2009

instead of the 30 mm/ms that was used in the modelling in order to reduce the calculation time and achieve a reasonable time step that would not give erroneous results in the explicit time integration. Load-displacement diagram including the experiment curve and the corresponding one produced by the numerical simulation and marking the characteristic points corresponding to the sequences of pictures of progressive specimen collapse have been included in Fig. 4.

Finally, the peak load, mean crushing load and load-displacement history calculated by means of the developed finite element model are almost identical to the corresponding values obtained by the experimental works. This feature of the model is of particular importance, since it proves that the simulation model can be effectively used to accurately calculate some of the most significant parameters in the design of sandwich structures.

V. CONCLUSION

The obtained numerical results of axial compression model of composite blocks are compared with actual experimental data of crash energy absorption, load-displacement history and crush zone characteristics, showing very good agreement. Theoretical and experimental results showed good similarities in peak load, average load and energy absorption with and without use of two types collapse trigger mechanism.

REFERENCES

- [1] A. G. Mamalis, M. Robinson, D. E. Manolacos, G. A. Demosthenous, M. B. Ioannidis, and J. Carruthers, "Crashworthy capability of composite material structures," *Compos. Struct.*, vol. 37, pp. 109-134, 1997.
- [2] P. H. Thornton and R. A. Jeryan, "Crash energy management in composite automotive structures," *Int. J. Impact Eng.*, vol. 7, pp. 167-180, 1988.
- [3] P. H. Thornton, "Energy absorption in composite structures," *J. Compos. Mater.*, vol. 13, pp. 247-262, Jan 1979.
- [4] A. G. Mamalis, D. E. Manolacos, M. B. Ioannidis, and D. P. Papapostolou, "Crashworthy characteristics of axially statically compressed thin-walled square CFRP composite tubes: experimental," *Compos. Struct.*, vol. 63, pp. 347-360, 2004.
- [5] C. Bisagni, G. Di Pietro, L. Frascini, and D. Terletti, "Progressive crushing of fiber-reinforced composite structural components of a Formula One racing car," *Compos. Struct.*, vol. 68, pp. 491-503, 2005.
- [6] S. T. Taher, E. Mahdi, A. S. Mokhtar, D. L. Magid, F. R. Ahmadun, and P. R. Arora, "A new composite energy absorbing system for aircraft and helicopter," *Composite Structures*, vol. 75, pp. 14-23, 2006.
- [7] G. Pitarresi, J. J. Carruthers, A. M. Robinson, G. Torre, J. M. Kenny, S. Ingleton, O. Velecela, and M. S. Found, "A comparative evaluation of crashworthy composite sand-

wich structures," *Compos. Struct.*, vol. 78, pp. 34-44, 2007.

- [8] P. R. Marur, "Effective elastic moduli of syntactic foams," *Mater. Lett.*, vol. 59, pp. 1954-1957, 2005.
- [9] N. Gupta, E. Woldesenbet, and P. Mensah, "Compression properties of syntactic foams: effect of cenosphere radius ratio and specimen aspect ratio," *Composites Part A*, vol. 35, pp. 103-111, 2004.

- [10] N. Gupta, Kishore, E. Woldesenbet, and S. Sankaran, "Studies on compressive failure features in syntactic foam material " *J. Mater. Sci.*, vol. 36, September, 2001 2001.

- [11] D. Rittel, "Adiabatic shear failure of a syntactic poly-meric foam," *Mater. Lett.*, vol. 59, pp. 1845-1848, 2005.

