Development of a Finite Element Model for the study of the impact behaviour of sandwich panels

Karthik Ram Ramakrishnan [a,b], Sandra Guérard [a], Krishna Shankar [b], Philippe Viot[a]

a. *Arts et Métiers ParisTech, I2M, UMR 5295, F-33400 Talence, France*
b. *School of Engineering and Information Technology, UNSW, Canberra, Australia 2600*

Abstract :

Sandwich structures based on Fibre reinforced polymer (FRP) skins bonded to either side of a low density core material, such as polymer foams are finding increasing use in aerospace and marine industries. It is necessary to study the impact response of sandwich structures in order to ensure the reliability and safety of these structures. A finite element (FE) analysis provides the capability to model the impact event, including the contribution of the individual elements of the sandwich panel to the energy absorption. The purpose of this paper is to develop a numerical model of sandwich panels with Kevlar FRP facesheets and core of PMI foam, subjected to low velocity impact loading. An approach for modelling sandwich structures in the commercial finite element code LS-DYNA is presented in the paper. The force and energy absorption characteristics of the sandwich structure are analysed.

1 Introduction

The study of the behaviour of sandwich structures to impact loading is usually accomplished by experimental testing. However, experiments are time consuming, expensive and require ancillaries which can be avoided by using numerical methods, which can help in predicting the energy absorption and peak loads for a given combinations of materials and geometry by considering competing mechanisms [1]. A finite element (FE) analysis provides the capability to model the impact event, including the contribution of the individual elements of the sandwich panel to the energy absorption. Numerical models for the low velocity impact analysis of sandwich plates have been developed using different finite element software by several authors [1, 2]. In this paper, the impact loading response of a sandwich plate with Kevlar fibre reinforced epoxy facesheets and a PMI foam core was examined using the commercially available Finite Element software LS-Dyna.

2 Development of the Finite element model

The facesheets made of three layers woven Kevlar/epoxy composite laminate, is modelled using 4-node shell elements with the Belytschko-Tsay element formulation and 3 integration points corresponding to the 3 layers in the layup. A tetrahedral solid element with 1 point integration is used for the core. The two facesheets are represented by 10000 shell elements each while the core is made of 128843 tetrahedral elements. The impactor is modelled with 3000 solid elements. The support fixture is modelled using shell elements and a fixed boundary condition is applied to the aluminium plate in the bottom while a preload, corresponding to the torque applied on the bolts during the experiments, is applied to the top clamping plate. An initial velocity of $3.1m/s$ is defined to the hemispherical impactor. The material law used for the Kevlar facesheets was LAMINATED COMPOSITE FABRIC material model available in the LS-Dyna material model library (MAT 58). The model is based on the theory of continuum damage mechanics approach developed by Matzenmiller, Lubliner and Taylor – called MLT model [3]. The input parameters for the material model are obtained from characterisation tests conducted by Denneulin [5]. A Crushable Foam model represented by LS-Dyna Material MAT 63 is used to model the behaviour of the core as recommended by Croop and Lobo [4]. The crushable foam material model requires the stress-strain response for the foam obtained from uniaxial compression tests.

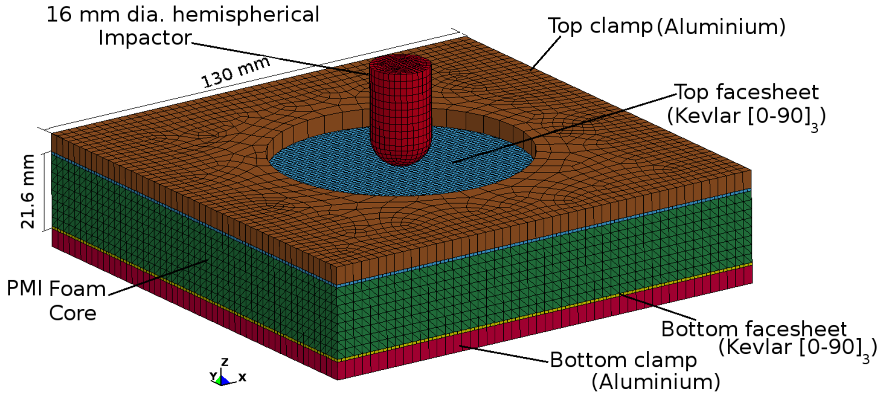

FIGURE 1 – Geometry and Mesh of Sandwich plate

3 Results of the LS-Dyna model

Figure 2 shows the time-history of the kinetic energy of the impactor as well as the Contact force and Displacement of the sandwich plate. The initial kinetic energy is 8.2 J and as the impactor comes into contact with the plate, some of the energy is transferred to the plate. At the end of the impact, not all of the initial kinetic energy is returned to the impactor as some of it is absorbed in plastic deformation and failure of the sandwich plate. It can be seen from the force history that after a linear increase as the impactor contacts the specimen, a sudden drop in the load is observed, which indicates a loss in stiffness due to damage in the facesheet. A Force-Displacement curve is also shown in the figure (d).

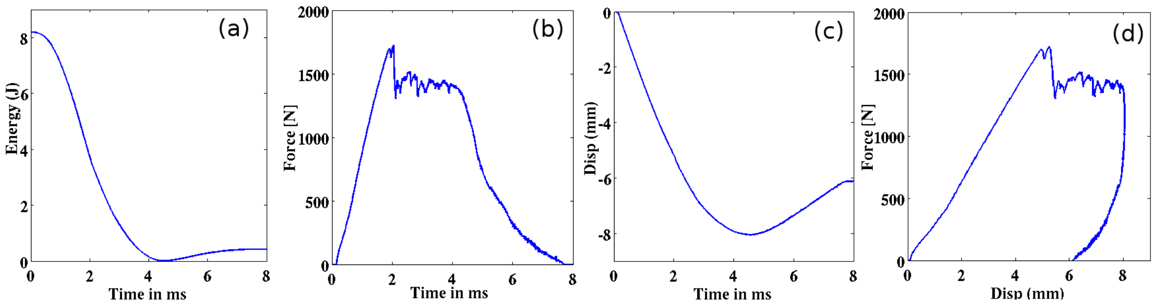

FIGURE 2 – Kinetic energy, Force, Displacement history of low velocity impact of sandwich plate

4 Conclusion

A finite element model developed in LS-Dyna to study the impact behaviour of sandwich structures was presented in this paper. A Kevlar-PMI foam sandwich plate was modelled with a combination of shell elements (facesheets) and solid elements (core). In the future, the numerical models presented here will be validated by comparing it with experimental results. The development of an accurate numerical model will help the designer to maximise energy absorption in sandwich structures.

Références

[1] A. Rajaneesh, I. Sridhar, S. Rajendran, Composite Structures **94**(5), 1745 (2012)

[2] I. Ivanez, C. Santiuste, E. Barbero, S. Sanchez-Saez, Composite Structures **93**(9), 2392 (2011)

[3] *LS-Dyna Keyword User's Manual*, Livermore Software Technology Corporation (LSTC) (2012)

[4] B. Croop, H. Lobo, *Selecting material models for the simulation of foams in LS-Dyna*, in *7th European LS-Dyna Conference* (2009)

[5] S. Denneulin, *Etude du comportement de materiaux composites sous sollicitations dynamiques - Application a un casque aeronatique*, in *PhD Thesis, Arts et Métiers ParisTech* (2011)