On the Low-Velocity Impact and Quasi-Static Indentation Studies of Nomex Honeycomb Composite Sandwich Panels

S. Sadeghnejad[1*], Y. Taraz Jamshidi[2], M. Sadighi[1]

[1] Thermoelasticity Center of Excellence, Mechanical Engineering Department, Amirkabir University of Technology Tehran, Iran

[2] Mechanical Engineering Department, Shahid Rajaee Teacher Training University, Tehran, Iran

ABSTRACT: An experimental–numerical methodology for investigation of quasi-static indentation and low velocity impact on sandwich panels with composite skins and Nomex™ honeycomb core is presented. Sandwich panels with glass/epoxy skins and a NomexTM honeycomb core were modeled by a three-dimensional finite element model implemented in ABAQUS/Explicit. The model was validated with experimental tests by comparing numerical and experimental results. The comparison has not only been based on a load–displacement and load time history curves, but has been further exemplified by detailed photographical images throughout the whole loading process and the local behavior of the cells crushing. Results show that fine micromechanical models based on shell elements give good correlation with honeycomb compression tests for Nomex™ honeycombs. Also the reference finite element numerical model demonstrates its capability to accurately reproduce the shape of the local damage of the panel. In other words, the calibrated micromechanical model is obtained to predict both quasi-static and low-velocity impact behavior. The mentioned model could be used for structural optimization with enhanced accuracy in contrast to conventional macro-mechanical models. The calibrated model is used to predict the cell size effect, friction and also geometric scale.

1- Introduction

Sandwich panels are widely used in several industries such as aerospace, automobile, marine, military, and even sports. The major performance index of these structures is their enhanced stiffness to weight ratio. Among these, relatively inexpensive Nomex papers, in form of corrugated cores are good choice to be used as sandwich cores. The honeycomb form serves good shear and compressive strength while low weight. The mentioned sandwiches are designed to be used as secondary structures in civil aircraft structures exposed to both quasi-static and dynamic loading. Manipulating composite skins also improve performance of the structures, both in load bearing capacity and offers damage tolerance capability. Furthermore, using the micro-mechanical approach in honey comb core modeling improves the accuracy of simulation and hence optimization and tailoring the complex structures being obtained.

The composite sandwich structure consists of a lightweight core material and two stiff facings. Foams and honeycombs are two famous classes of cores in composite materials [1-3]. These structures exhibit static properties such as high stiffness-to-weight ratio, high buckling loads and low maintenance cost which are of great importance in marine, transport, civil construction and aeronautics fields [4-6]. Sandwich panels are optimized light structures characterized by good stiffness, strength, and energy absorption properties. Sandwich panels made with thin composite skins and Nomex™ honeycomb core are a particular type, fit for aeronautical components which are exposed to low velocity impacts due to the presence of a ductile material in the skins [7]. Important data about

honeycomb cores in NomexTM can be found in references [8-11], but some uncertainness remains due to technological history. Nomex possesses an extremely high strength to weight ratio. It is also electrically and thermally insulating, chemically stable, self-extinguishing and corrosion as well as shock and fatigue resistant.

Among core materials, honeycomb materials have been widely used in aerospace and submarine applications. Tests in the industry and most publications agree that in low velocity impact and for aeronautical sandwiches with skins that not exceed 2.5 mm, there is equivalence between dynamic tests and static indentations [12-14]. However, static tests are easier to set up and show low dispersion [15, 16]. An experimental and numerical methodology for the investigation of Three Point Bending Test (TPBT) on sandwich panels with Al skins and Nomex™ honeycomb core has been done in previous researches [17]. Yamashita and Gotoh [14] studied the quasi-static compression response of aluminum honeycomb in the thickness direction.

A number of researchers have investigated the response of sandwich structures to localized impact loading and much of this work has been summarized in detailed reviews on the topic [18-20]. The impact behavior of a sandwich panel depends on many factors, not only the mechanical properties of its constituents, skins, and core but also the adhesive capacity of the skin-core interface. Mines et al. [21] investigated the low velocity impact response of glass fiber/vinyl ester sandwich panels based on aluminum honeycomb. Gibson and Ashby [22] provide a complete review of the mechanical behavior of cellular solid cores and also analyze honeycomb cores. In reference [23] developed a theoretical model for determination of the mean crushing strength of the

Corresponding author, E–mail: s.sadeghnejad@aut.ac.ir

foam filled metal hexagonal honeycomb under quasi-static loading. The mentioned authors also studied metal hexagonal honeycomb crushing behavior under quasi-static and low velocity impact loading [24].

Analytical models of beams or plates on elastic foundations have also been developed by several authors [25]. In reference [26] used a displacement-based, plate bending Finite Element (FE) analysis to model impact damage in sandwich plates based on a Nomex honeycomb core. A comparison with low velocity impact and quasi-static indentation data highlighted the model's ability to predict important features of the behavior of the sandwich panels. In reference [1] used the finite element package to predict impact damage in sandwich panels based on both PolyEtherImide (PEI) foam and Nomex honeycomb cores and showed that the model was capable of reproducing the physical behavior of the two types of structure.

Understanding and modeling the crushing phenomenon are the key points of the general problem of impact on sandwich structures with honeycomb core. Core crushing is a complex mechanical phenomenon characterized by the appearance of various folds and failures in the hexagonal structure [21]. This phenomenon is known for its energy-absorbing capacities and has been analyzed since 1963 by reference [27]. Constitutive equivalent models have been developed [28, 29] and have been applied successfully to experiments on large structures subjected to blast loads. Foo et al. [30] analyzed the failure of sandwich panels with a three-dimensional Finite Element Model (FEM) of the honeycomb core subjected to low-velocity impact. Nia et al. [31] determined experimentally that the armor capacity of an aluminum honeycomb panel was small. Yin and Wen [32] analytically and numerically studied the axial crushing of honeycomb structures with various cell specifications.

Besides having a variety of different researches on honeycomb panels and following the previous static and dynamic study of the present authors [7] on the sandwich panels with composite skins and Nomex™ honeycomb core, in the present work, an experimental–numerical methodology for the investigation of quasi-static indentation and low velocity impact on sandwich panels with composite skins and Nomex™ honeycomb core is presented. Sandwich panels with glass/epoxy skins and a NomexTM honeycomb core are modeled by a three-dimensional finite element model implemented in ABAQUS/Explicit. The skins were modeled as laminates applying the Hashin criteria [33] for the prediction of the failure and the honeycomb core was modeled reproducing the NomexTM hexagonal cells. The model was validated with experimental tests by comparing numerical and experimental results. Recently Audibert et al. [34] in 2019 works on new modeling concepts. They used non-linear springs as discrete elements for modeling mechanical behavior of Nomex honey comb core in sandwich panels. The mentioned approach reduce solution time significantly while the results are in rather a good accuracy. Seemann et al. [35] in 2018 works on Nomex core sandwich panels modeling using non-linear but homogenized model. The loading condition in their study is predicting the pull-out strength of sandwich structures numerically. Comparing with benchmarked experiments showed good accordance.

A dedicated focus on this research numerical models is presented. Finally, the results validation and also the

parametric studies will be done to study the effects of different parameter variations. The comparison has not only been based on a load–displacement curve but has been further exemplified by detailed photographical images, throughout the whole loading process, of the local behavior of the cells crushing. Results show that fine micromechanical models based on shell elements give good correlation with honeycomb compression tests for Nomex™ honeycombs. Also the reference FE numerical model demonstrates its capability to accurately reproduce the shape of the local damage of the panel. The major difference between present investigation and previous works are summarized as the micro-mechanical approach of modeling and validating with experimental works. Furthermore, we manipulate the dynamic procedure to obtain the quasi-static response. This approach offers several benefits such as more realistic contact modeling, obtaining validated and calibrated numerical model for both static and dynamic behavior study and also concerning the inertia effects similar to the real test conditions. Finally, the accurate numerical model is obtained.

2- Material

In this work, square sandwich specimens (250mm×250mm and 25.2mm thick) were used. The skins were plain woven laminates of glass-fibers and epoxy resin and 1.2mm in thickness. The core is a NomexTM honeycomb of 22.8mm thick and 59kg/m^3 in density. The cells were hexagonal, with a size of 3.0mm (1/8 in) and a wall thickness of 0.1 to 0.4mm. This variable measured thickness is due to measuring complexity. Fortunately, the simulations are not too sensitive to these values. The equivalent density for core material is determined by means of overall geometry dimensions, measured mass of panel and paper wall thickness. Hence the calculated density of core material for 0.1 and 0.4mm Nomex paper thickness are 500 and 125kg/m^3. For the core properties determination, flat-wise compression tests under square specimens were performed, according to ASTM C 365/C365 M-05 Standard [36]. The load–deflection curves were used to determine the compressive and crush strengths and the compressive modulus of the core (see Fig. 1). Fig. 2 illustrates impact force vs time for the crush test diagram of the sandwich plate with NomexTM honeycomb core and plain woven laminates of glass-fibers skins for the impactor height of 80cm and weight of 2.9kg, as a sample.

In addition, the material properties of the core can be seen in Table 2. In which, σ_y is the core ultimate strength, E core Young's modulus, v core Poisson's ratio and ε_f is the core plastic strain. Face skins have 1.2mm thickness. This value
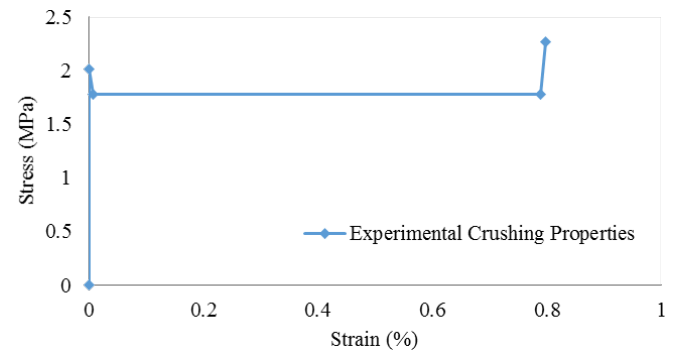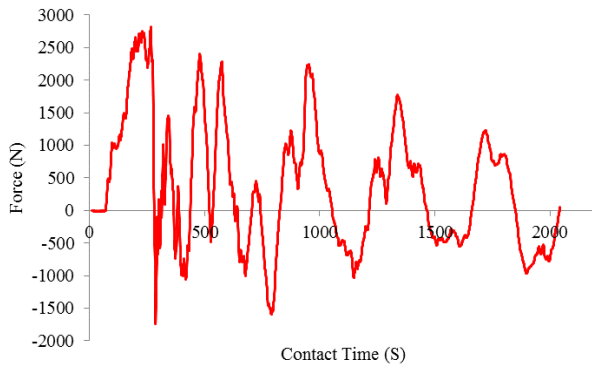

Fig. 1. NomexTM honeycomb crushing properties

Fig. 2. Impact force vs time for the crush test diagram of the sandwich plate with NomexTM honeycomb core

is a little above measured quantity for considering the glue thickness. Core failure is also modeled by shear failure criteria with 0.81 plastic strains. Composite skins material properties are defined in the following manner: it is assumed as a planar orthotropic or lamina elasticity with Young's modulus (E_1 and E_2), shear modulus (G_{12}), Poison's ratio (v_{12}), mass density (ρ) and tensile, compressive and shear ultimate strength values (X, X' and S) presented in Table 1.

Table 1. Face skins material properties

Property	Value	Unit
E_1	29.7	GPa
E_2	29.7	GPa
v_{12}	0.17	-
G_{12}	5.3	GPa
ρ	2200	kg/m³
$X(Y)$	367	MPa
$X'(Y')$	549	MPa
S	97	MPa

Table 2. Core material properties

Property	Value	Unit
σ_y	2	MPa
E	400	MPa
v	0.01	-
ε_f	0.81	-

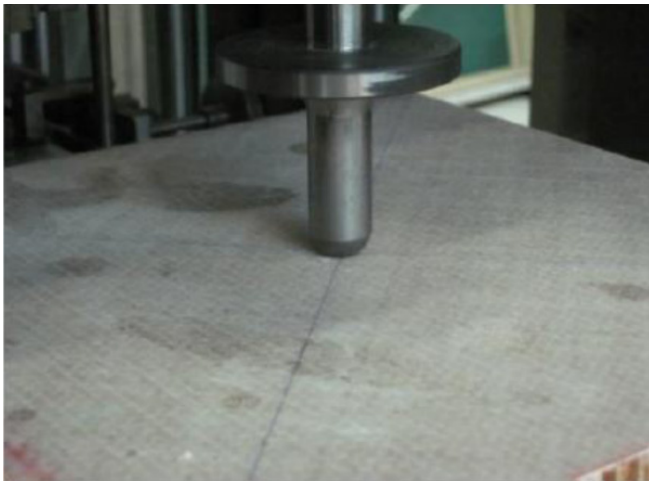## 3- Experimental quasi-static and low velocity impact tests

To measure the static indentation behaviour of the panels, a compression test has been set up by using Zouich machine according to existing standards, ASTM D1621 [37]. Impact tests were performed using an instrumented impact testing system, used in Thermo-Elasticity Center of Excellence at Amirkabir University of Technology (Tehran Polytechnic), which consists of a drop tower equipped with an impactor with variable crosshead, weight arrangement, a high speed data acquisition system, and a force transducer mounted in the impactor as shown in Fig. 3.

The impactor end has an instrumented tip of a hemisphere with diameters of 12.36mm and 16mm and impactor weight was kept constant at 2.9kg for all tests (Fig. 4). The sandwich panels were subjected to impact with increasing drop heights.

Fig. 3. The Amirkabir Thermo-elasticity Center of Excellence drop hammer test system
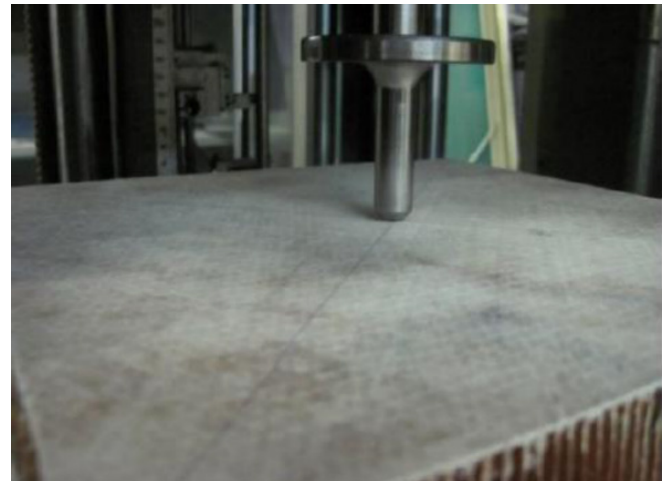
The panel was square clamped condition during the low-velocity impact event. After the completion of the impact tests, the barely visible impact damage size and depth of the permanent indentation was measured precisely. The panels of size 250mm-250mm were tested at the impact height range of 60cm and 80cm and the indentation rate of 5mm/min. Equivalently, the impact energy is about 20 to 25J. In numerical modeling, it is assumed that the initial velocity of the impactor is about 4mps.

4- Numerical Model

Quasi-static indentation and low-velocity impact finite element modeling of the sandwich panel were implemented in the Abaqus/Explicit nonlinear commercial code. According to the nature of the experiments, the boundary conditions of the numerical model are supposed to be clamped. For this reason, two rigid rectangular plates with rectangular openings, at their middle, were manipulated. Inspecting the impactor surface, no permanent plastic deformation was seen. Therefore, this part was modeled as a discrete rigid surface with fine mesh. Fine mesh selection of the impactor, helps output noise reduction and enhances the contact simulation accuracy. Two different rigid hemi-spherical impactors, with diameters 16mm and 12.63mm, are included in the present analysis. The other rigid surfaces, in the present simulation, are specimen holders. These parts were used in upper and lower regions of the panel, to make it clamped. The specimen and the holders opening dimensions are 25×25cm² and 20×20cm² respectively. All the rigid surfaces are modeled with R3D4 elements. A 725gr concentrated mass element attached to the impactor reference point. The component mass plays a very important role in impact behavior and energy absorption phenomenon. The total mass of the impactor in experiments is 2.9kg. Although with the use of two orthogonal planes of symmetry in assembly, only a quarter of the mass (725gr) was defined in the numerical model. For more emphasis, it was evident that only a quarter model needed to be analyzed. Obviously, the symmetry cut regions had the appropriate boundary conditions. Although the exact symmetry conditions were not existed basically in the whole model because of the particular geometric arrangement of the hexagonal honeycomb cells, the full model analysis proves

(a) 16mm impactor

(b) 12.36mm impactor

(c) Specimen cross section

Fig. 4. The impactor end with the instrumented tip of the hemisphere (a) 16mm, (b) 12.36mm, (c) Specimen cross section

that the error due to this assumption was quite small. The only remaining degree of freedom of the impactor was supposed to be an axial translation. So the undesired translation and/or rotation of impactor is not really a concern.

The sandwich panel consists of two major parts: e-glass/epoxy woven laminate skins and Nomex™ hexagonal honeycomb core. To model face composite skin damage and failure, the Hashin criteria was used. Fracture energy in Hashin criteria is used for modeling the damage evolution in composite [33]. The values of these energies are assumed to be different in upper and lower skins. The major reasons are, the upper face was supported by the core in spite of the lower one and elevated strength of upper face glue joint because of compressive stress state. So, the upper and lower fracture energies are selected to be 37.5kJ and 12.5kJ, respectively. To enhance the accuracy of the numerical model and to observe the exact localized deformations and damages, the micromechanical model with a full generation of honeycomb cells is implemented. The sandwich plate including two faces and a core is modeled with linear quadrilateral shell elements with reduced integration (S4R in Abaqus). This element type was chosen for better contact simulation in very complex interaction situation and also to reduce the time and cost of simulation. In addition to careful selection of the element type and mesh density, extensive mesh independency study was performed to ensure the numerical analysis. For instance, in some FE models, the locally refined mesh in the vicinity of the impact and coarser mesh elsewhere are manipulated. To reach a compromise between analysis efficiency (time and hardware requirements) and accuracy, many different meshes were used. To emphasize the procedure of mesh independency analysis for validation purposes, four FE models are generated

for each simulation. These models called coarse, moderate, fine and locally refined respectively with about 50200, 67000, 120000 and 75000 shell element. Furthermore, the element type, simulation period, mass scaling factor, loading rates and maximum time incrementation independency studies are also conducted for validation and verification purposes. Peak load and energy absorption are chosen to be an index for independency analysis. The error related to the mean values from the experiment for coarse, moderate, fine and locally refined meshes are respectively 37, 23, 10, 11 percent. As obviously observed, the locally refined mesh with local refinement in impact region is completely acceptable. On the other hands, using fine mesh is reduced error only 1 percent while increasing more than 50 percent in solution time. Physical and mechanical properties of core material were taken from the manufacturers' documentations. Then they were calibrated and updated to enter the model. These slight variations are used to calibrate the numerical model, because the values from documentations, varied due to manufacturing processes. Finally, the core material used in the analysis is a homogeneous isotropic linear elastic-perfectly plastic model. The above mentioned model was validated by the honeycomb compression virtual test. Additionally, the validation and verification process was done by calibration of the first impact load peak and plateau region. The plateau portion of the impact load time history represents the amount of energy absorption. The whole mesh of the assembly consists of two distinct types of element: S4R and R3D4. The statistical report of the mesh is presented in Table 3. In Fig. 5, the selected portions of the mesh are shown. In these figures, the mesh of face skin, cells, impactor and holder in the quarter model are shown respectively.
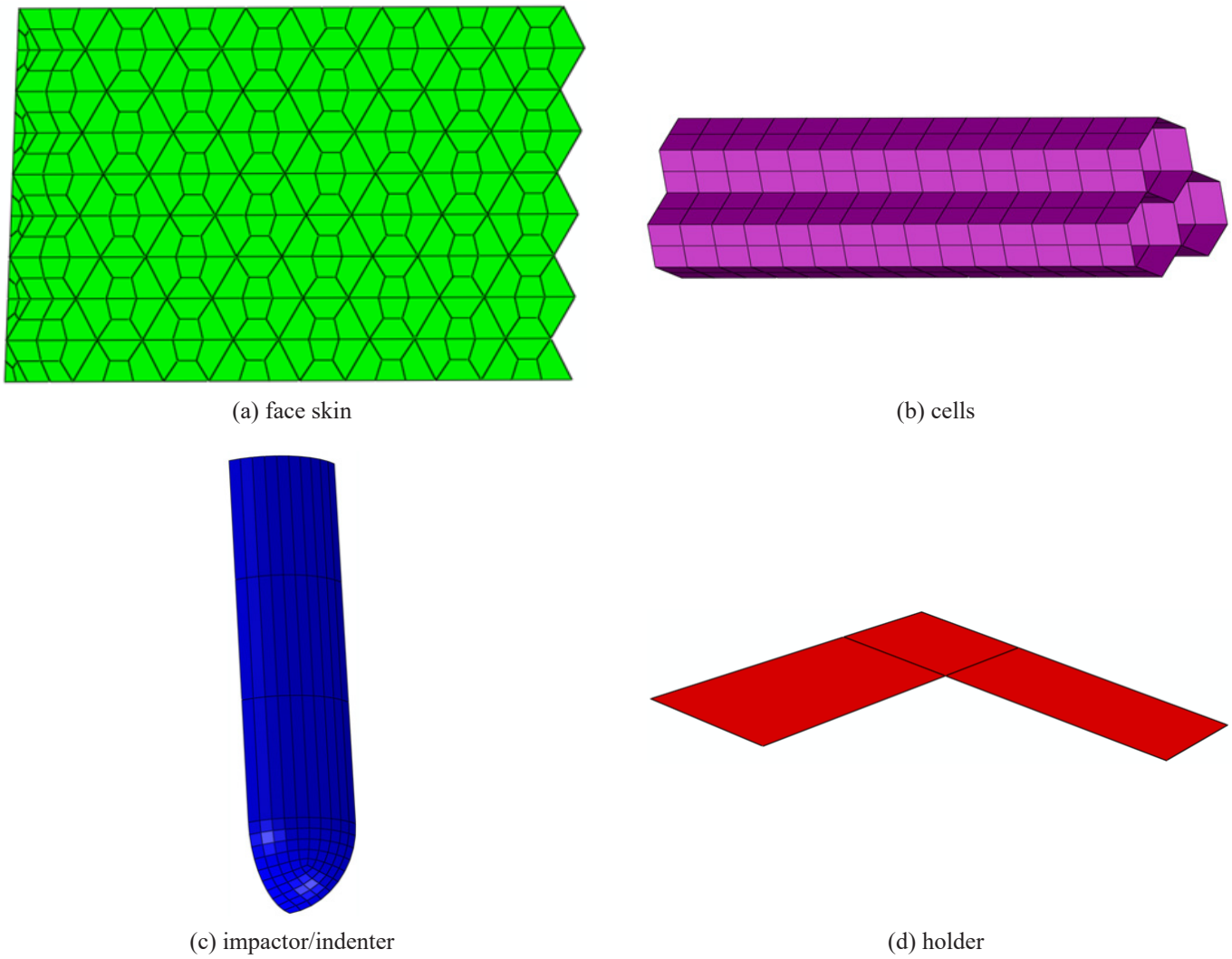
(a) face skin

(b) cells

(c) impactor/indenter

(d) holder

Fig. 5. FE model of the system (a) Face Skin, (b) Cells (c) Impactor/Indenter and (d) Holder

Table 3. Statistical report of the FE model

Element type	No. of Elements	No. of Nodes
S4R	84120	113358

General contact algorithm is used to simulate surface to surface interactions. The interaction properties are assumed to be frictionless for tangential behavior, and hard contact, for the normal one. For better contact establishment and tracking of surfaces in the algorithm, an infinitesimal initial gap was used. This gap also prevents undesired initial over-closure. Fig. 6, shows the selected frames of impact simulation procedures.

Fig. 6. Impact simulation frames

5- Results

Numerical load–displacement FE model curves have been compared with experimental ones, Fig. 7. As described before, the whole experimental impactor/indenter has been numerically modeled, also shows an average experimental curve. This curve has been obtained with a data fitting procedure using the commercial elaboration software MATLAB™. Data have been fitted using a smoothing spline function. This means that the experimental curve is subsequently employed for the comparison with numerical results. The fitting operation is based on Eq. (1) accordingly with [30]:

$$\min(q \sum_j w_j (y_j - \widehat{S}(X_j))^2 + (1-q)\int (\frac{d^2\widehat{S}}{dX^2})^2 dX \qquad (1)$$

In which, (y_j, X_j); $y_1 < y_2 < ... < y_n$, $j \in Z$ is a sequence of observations, modeled by the relation $X_j = S(y_j)$. The smoothing spline estimate $\widehat{S}$ of the function S is defined to be the minimizer (over the class of twice differentiable functions). q is a parameter called smooth factor and belongs to the interval [0, 1] and w_i^{-1}; $j=1,...,n$ are the quantities controlling the extent of smoothing (they represent the weight w_j^2 of each point X_j).

As dissipated in Fig. 7, experimental behavior (described by experimental curves, indenter diameter of 16mm) can be divided into four different zones:

The first one: (1) is the elastic zone with a linear curve that ends reaching a peak of the force. This force value is the maximum of the entire experimental test. The second zone (2) starts after the peak and it ends at a displacement of about 7mm for all tests. In this zone, the various experimental curves show less force variation than the FE curve. The general trend is, however, similar for each curve, exhibiting a sudden drop from the peak value (due to local cells buckling) followed by a slight increase in force. Each of the tests confirms the same overall behavior. In a quasi-static indentation test in fact, after buckling, a folding mechanism appears (both global and local), generating a plateau in the load–displacement curve in the zone (3). In zone (4) the cell walls reach a point at which they cannot accommodate additional folds, and consequently the load–displacement response curve exhibits a sudden increase of the load, densification. Numerical FE reference results are in good accordance with the experimental results in the first region. In particular, load-displacement curves to a fixed displacement are very similar between the FE and the experimental results. This means that the FE model is able to reproduce the load-displacement behavior of a real component with a low error in the linear region. As it can be seen in Fig. 7, all the experiments are quite similar. Furthermore, similarity between experiment results and FE reference is seen just for the zone (1). This is due to the elastic-perfectly plastic model of the honeycomb core. In reality, the honeycomb core cells will crush and they will not carry load any more. This leads to a quick decrease of force that can be seen in Fig. 7. On the other hand, in the numerical model, the crushed and collapsed cells also have the load carrying capacity. These all are the major reasons for the difference between FE reference and experiments. The agreement between numerical and experimental results can also be seen in low velocity impact tests of impactor diameter of 16mm and impact height range of 80cm (Fig. 8).
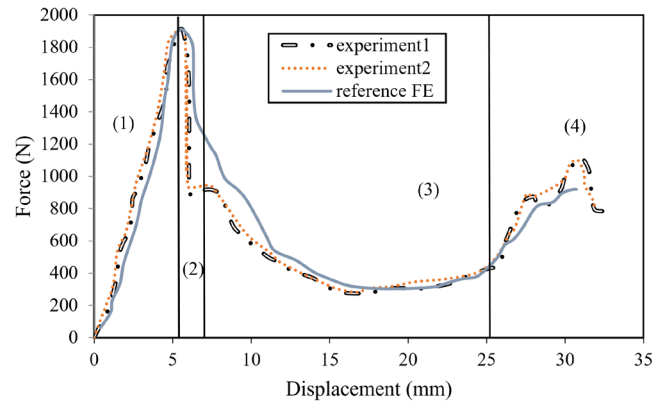

Fig. 7. Comparison between numerical (FE Model) and experimental indentation curves
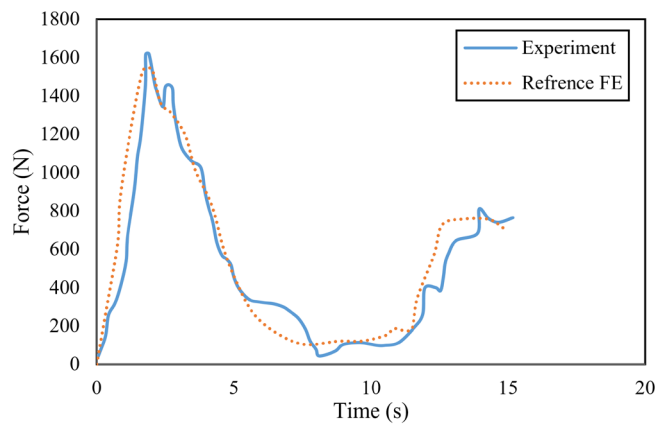

Fig. 8. Impactor velocity time history, impactor diameter of 16mm and impact height range of 80cm

Another interesting comparison refers to the peak force value (first zone) for both indentation and low velocity impact tests. When the peak value is compared the error of the FE model, it is even lower with a value of only 5% for static indentation test of 16mm indenter and 0% for 12.36mm indenter. In low velocity impact tests, the error of the FE model is 3% for 16mm indenter and 2% for 12.36mm indenter. The densification point in Fig. 7 is of further importance. At this point, the displacement load stops decreasing and a sudden increase in its value occurs. These results, apart from the load vs. displacement data, subsequently reported in Fig. 7, are summarized in Table 4.

To the careful selection of the element type and mesh density, extensive mesh independency study was performed to ensure the numerical analysis.

In some FE models, the locally refined mesh in the vicinity of the indentation and coarser mesh elsewhere are manipulated. To reach a compromise between analysis efficiency (time and hardware requirements) and accuracy, many different meshes were used.

Fig. 9 shows the force vs. indentation displacement of 16mm diameter. The results show that there is no meaningful difference between reference FE mesh and two other fine meshes. These two meshes are fine mesh which is a finite element model with very fine and regular mesh and locally refined mesh. So it can be said that the results are mesh independent. Hence for computational efficiency purposes, the rather coarse mesh is chosen as reference FE mesh. A

Table 4. Comparison between numerical and experimental data with different impactor/indenter

(a) quasi-static indentation tests

Indenter		$F_{top}^{\,max}$ (N)	$D_{top}^{\,max\,F}$ (mm)	$F_{bot}^{\,max}$ (N)	$D_{bot}^{\,max\,F}$ (mm)	$F_{Plateau}$ (N)
16mm	Experiment	1500-1600	5-6	800-900	30-31	300-350
	FEM	1450-1500	6-7	850-900	32-33	250-300
	Error (%)	5	18	3	6	18
12.36mm	Experiment	1200-1300	5-6	550-600	30-31	100-150
	FEM	1200-1300	5-6	600-650	31-32	100-170
	Error (%)	0	0	8	3	8

(b) low velocity impact tests

Indenter		$F_{top}^{\,max}$ (N)	$D_{top}^{\,max\,F}$ (mm)	$F_{bot}^{\,max}$ (N)	$D_{bot}^{\,max\,F}$ (mm)	$F_{Plateau}$ (N)
16mm	Experiment	1500-1650	2	820-920	13	320-370
	FEM	1480-1580	2.2	850-930	14	310-350
	Error (%)	3	10	2	7	4
12.36mm	Experiment	1230-1300	2.3	560-610	13.5	110-150
	FEM	1210-1270	2.1	620-670	14.1	120-150
	Error (%)	2	9	10	4	4

mean of the experimental parameters, coming from different tests, has been calculated and the mean values have been compared with values from the numerical analyses. These results are summarized in Table 5 for 16mm and 12.63mm diameter impactors.

Numerical FE reference results are in good accordance with the experimental results. In particular, the total energy absorbed is very similar between the FE and the experimental results. This means that the FE model is able to reproduce the energy absorption behavior of a real component with a low error of less than 2.5%. Another interesting comparison refers to the peak force value (first zone). When the peak value is compared to the error of the FE model it is even lower with a value of only 0.7%. The peak value is significant, not only because it is related at the elastic response of the panel, but more so because it determines the maximum load that a panel can suffer without the occurrence of any irreversible damage. The FE model reproduces the experimental displacement phenomenon during the whole test duration with good agreement.
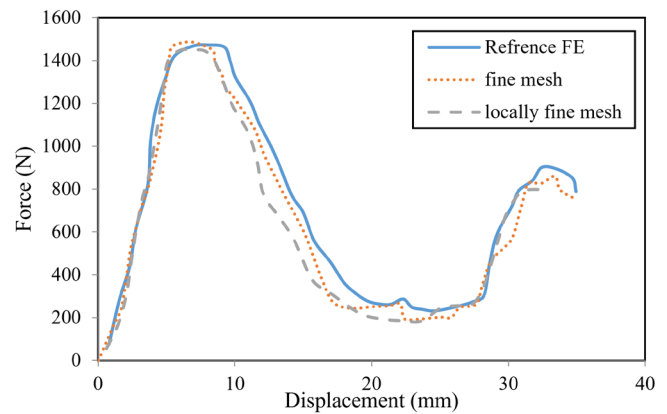

Fig. 9. Mesh density comparison between numerical (FE reference) and experimental curves, indenter diameter of 16mm

Table 5. Comparison between the numerical and experimental test for two impactors in low-velocity impact

Impactor	Test	Peak load (kN)	Time to maximum load (ms)	Total absorbed energy (J)
16mm	Experiment 1	1456	1.75	13.1
	Experiment 2	1461	1.76	14.5
	Experiment 3	1439	1.83	14.2
	Mean	1452	1.78	13.93
	Reference FE	1443	1.8	13.7
	Error (%)	0.6	1.1	1.7
12.63mm	Experiment 1	1282	1.43	11.3
	Experiment 2	1291	1.46	12.8
	Experiment 3	1255	1.48	12.5
	Mean	1276	1.46	12.2
	Reference FE	1267	1.45	11.9
	Error (%)	0.7	0.6	2.5

5- 1- Geometrical scale

As previous underlined, the reference simulation described in the previous section has been carried out by modeling only one quarter of the structure with a symmetry constrain on one side. Thus, symmetry constraints are not entirely accurate. In order to analyze the capability of this reduced model to be representative of the full size panel, a larger model representing the real structure has been analyzed with the same properties of the previously described FE reference model. In Fig. 10, a comparison between these two differently sized models is reported. Indeed, the curves are very similar demonstrating the capability of the quarter model to precisely describe the experimental phenomena.

Fig. 10. Comparison between the numerical load–displacement curves of a full panel model and of a quarter of real panel model with symmetry constrains, indenter diameter of 16mm

There are many advantages using this reduced model (quarter model – FE reference) mainly related to the computational time. Using the full model (full panel) the duration of the analysis is more than 2 weeks with a quad core 3.8 GHz frequency Central Processing Unit (CPU), instead of with the quarter (FE reference) model requires only about 1 week of analysis time. After two weeks of computational analysis, the numerical curve arrives at a displacement depth of only about 35 mm, Fig. 10. Indeed, once the similarity between the two models had been established, the analysis of the half model was stopped.

5- 2- Friction effect

As previous stated, one of the main aims of this work is to investigate the relevant parameters, which influence the behavior of the low velocity impact behavior. Following the validation of the numerical model (FE reference), it turns out to be a very powerful tool for parametric analyses of the effects of different parameters on the low velocity impact. These parameters would otherwise be difficult to modify experimentally. For instance friction between the impactor and the upper skin in the time of impact is an important parameter and its effect has been evaluated numerically. The FE reference analysis does not take the friction between the cylindrical surface of the impactor and the upper skin into consideration; instead, it models the contact as frictionless. The reason for this choice is mainly related to the lack of a reference value for the friction coefficient. However, the contact between the impactor and the skin has a very important role in the simulation. Hence, two different analyses with a static friction value of 0.3 and 0.6 have been carried out to be compared with the frictionless one FE reference analyses.

Numerical results are summarized in Fig. 11.

As it can be seen in the figure, there is no meaningful relation between friction coefficient and the load-time graph for the 16mm diameter impactor in low-velocity impact. On the other hand, this parameter can increase peak load for up to 10% in quasi-static indentation phenomenon. The corresponding graph for 12.63mm diameter impactor is also the same and so neglected. Focusing on the micromechanical behavior, Fig. 12, it can be seen that in absence of friction coefficient, the failure mechanism in FEM modeling is in a good accordance with the experimental one. These image comparisons show a similarity of the experimental and the numerical result, also at a micromechanical level (local indentation and folding). The crushing and failure of honeycomb cells are in accordance with the numerical model. Note that the shown experimental figure is cut in impact region and failure of other cells in the cut plane is inevitable. General contact algorithm is used to simulate surface to surface interactions.

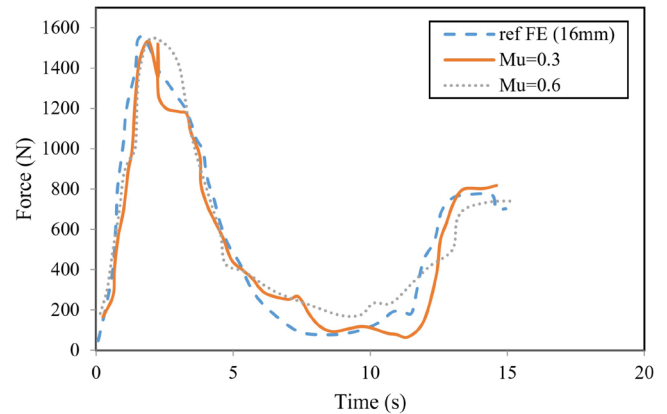The interaction properties are assumed to be frictionless for

Fig. 11. Effect of friction on the numerical load–displacement curve, impactor diameter of 16mm and impact height range of 60cm

tangential behavior and hard contact for the normal one. For better contact establishment and tracking of surfaces in the algorithm, an infinitesimal initial gap was used. The use of the finite element model provides information about the failure modes in the perforation process. Figs. 13 and 14 show the detailed field of plastic strain during the sandwich and honeycomb core perforation.

As it is obviously seen, the mode of failure and perforation is completely similar to the realistic failure mode. The results show the localized perforation and also the importance of skins in strength enhancement. On the other hand, it is seen that Nomex core is majorly playing the role of energy absorbing and also shear resistant in failure. Combining the Nomex core and composite skins offer excellent strength and energy absorption capability and accurate micro-mechanical model is used for tailoring and optimization purposes in applied aspects.

5- 3- Cell size effect

To investigate the cell size behavior with the micromechanical approach it is faced to both modeling and computer resources difficulties. So just two different cell sizes have been studied, i.e. half size and double size which can be seen in Fig. 15. The results are taken from 16mm diameter indenter and again it is emphasized that the results for small indenter are in complete similarity.
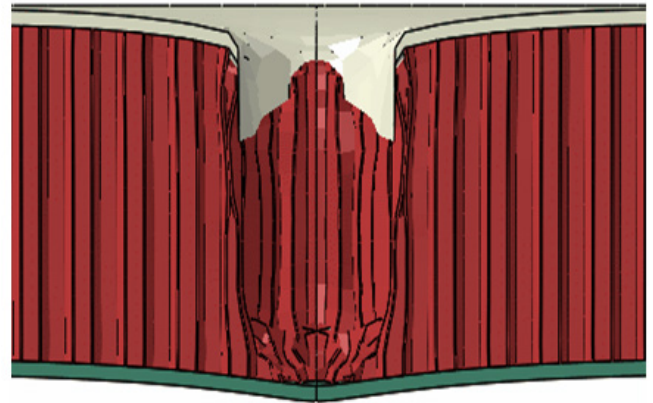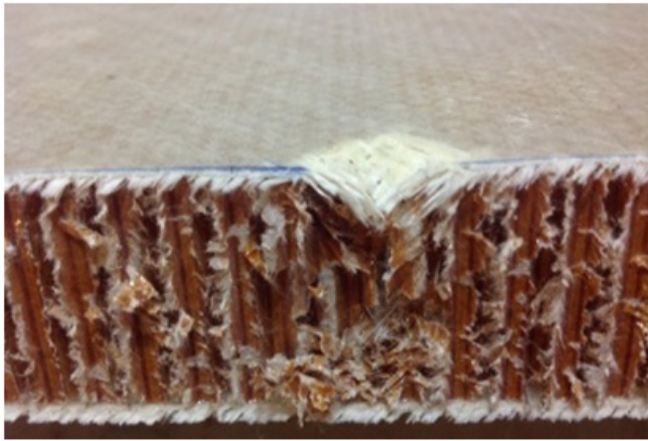
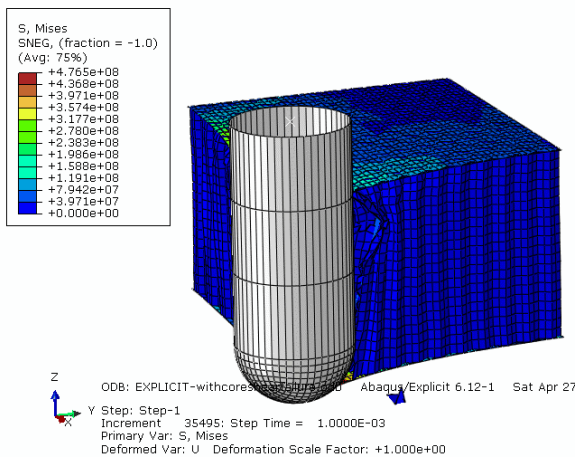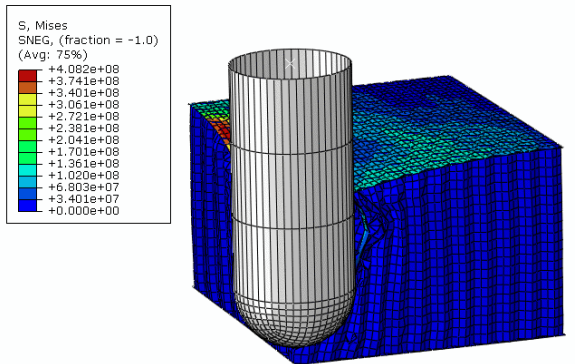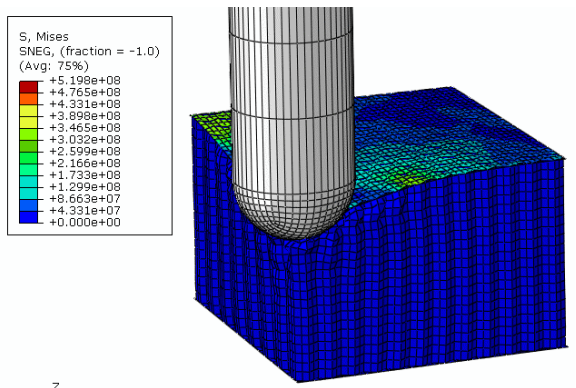**Fig. 12. Detailed comparisons between numerical and experimental damage**

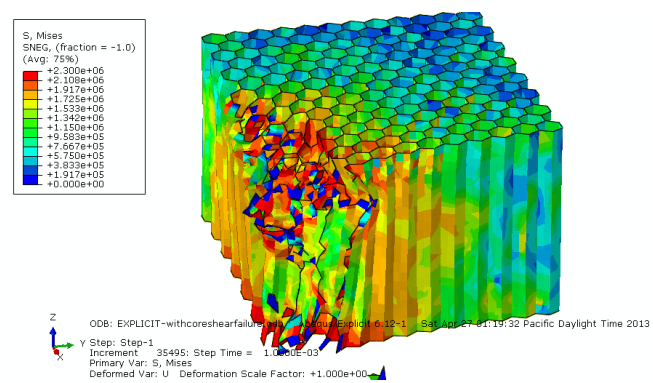**Fig. 13. Detailed fields of von-Mises Stress during the sandwich perforation**
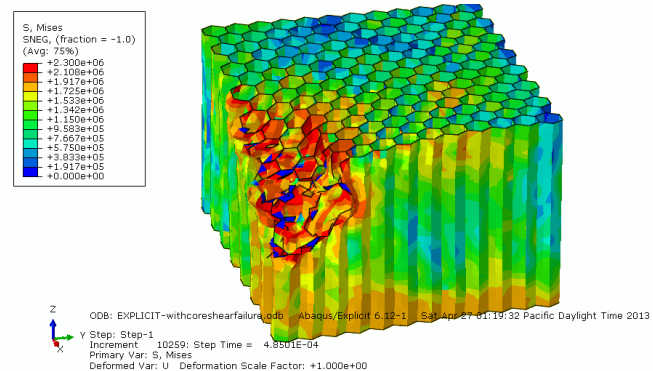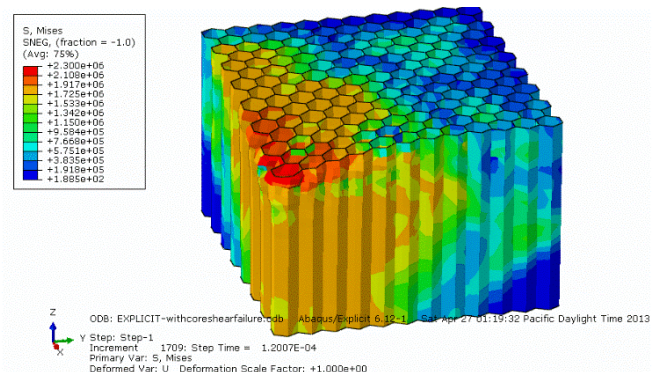

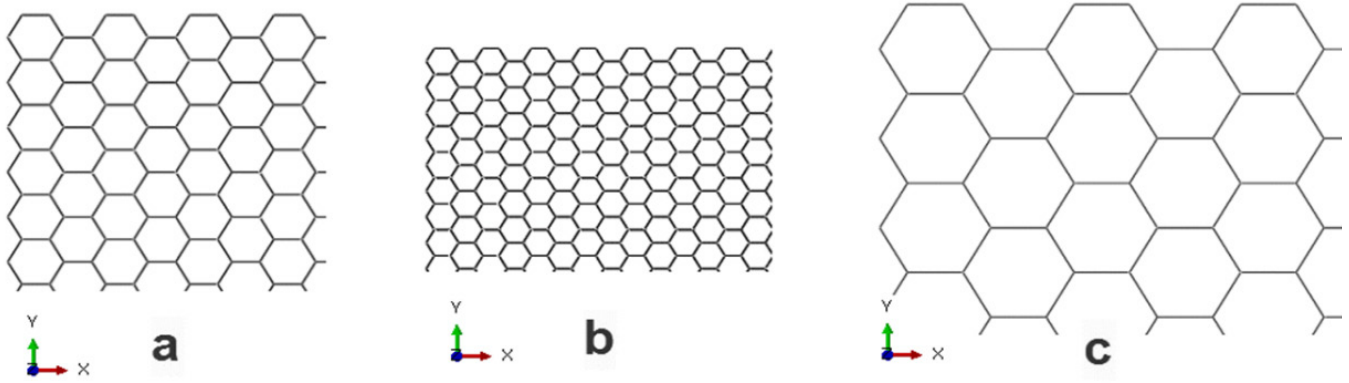**Fig. 14. Detailed fields of plastic strain during the honeycomb core perforation**

Fig. 15. Micromechanical models with (a) real size (b) half-cell size and (c) double cell size

The application of the FE micro model for parametric design studies on honeycomb core materials was studied. Fig. 16 shows the effect of cell size on the crushing behavior.
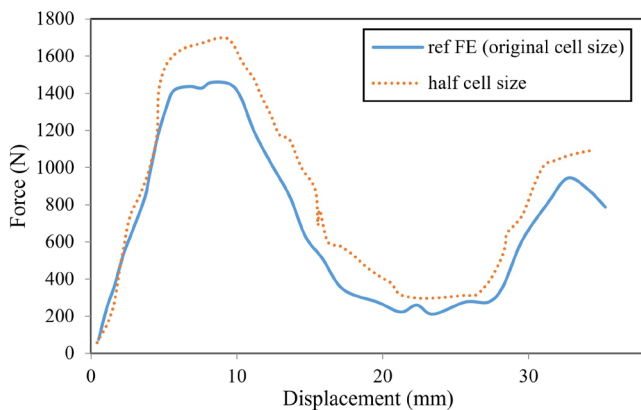

Fig. 16. Effect of cell size on crush response on NomexTM honeycomb in compression, indenter diameter of 16mm

The standard cell size is 3.17mm. The peak and plateau load values for the half-cell size model is about 1.2 times higher than the original model. The model with original cell size but having a double wall thickness as mentioned in the material section, showed the equivalent results with the original model. The model with doubled cell size gave lower (i.e. 0.7) peak and plateau load values which can be related to the lower buckling stability. Designed curves such as Fig. 16 obtained from the core micro models may be used to design honeycomb cores with given crush loads and energy absorption. Although the micromechanical model produces accurate results and gives an insight view about the effect of cell size and the cell wall thickness on the mechanical crushing behavior, it is computationally very expensive.

6- Conclusions

This study focused on modeling of quasi-static indentation and low velocity impact behavior of Nomex™ honeycomb core sandwich composites materials using a detailed micromechanical core model. In the experimental part of this study, a series of tests were conducted, including quasi-static indentation and low velocity impact tests compression with sandwich specimens fabricated with Nomex™ honeycomb and fiber glass/epoxy facings.

The results obtained, demonstrate particular accuracy, not only for the values of the numerical load–displacement curve and the maximum force but also with regards to the detailed comparisons between numerical and experimental damage. The FE reference curve is to some extend similar to the experimental one, showing the same trend and behavior. The reference FE numerical model also demonstrates its capability to accurately reproduce the shape of the local damage of the panel. This approach is therefore reliable to be taken to a higher level (i.e. full scale components). It makes the increased use of virtual tests in extreme cases finally a reliable option in the design and assessment phase. In this study, a reference numerical analysis has previously been carried out without a friction coefficient (with results similar to experimental behavior). The comparison of numerical experiments with numerical results shows that the numerical investigation is in good accordance with the experimental one in the supposed assumption. In the numerical part, it is demonstrated that fine micromechanical models based on shell elements give good correlation with honeycomb compression tests for Nomex™ honeycombs. Since these models are highly meshed dependent and CPU intensive for larger structures, this kind of FE modeling is time saving.

The scaled modeling approach has numerical advantages. This choice has drastically reduced the simulation time with an acceptable approximation in the reproduction of the phenomena. The contact between the impactor and the skin has a very important role in the simulation. There is no meaningful relation between friction coefficient and the load-time graph for the 16mm diameter impactor in low-velocity impact. On the other hand, this parameter can increase peak load for up to 10% in quasi-static indentation phenomenon. Neglecting the friction coefficient, the failure mechanism in FEM modeling is in a good accordance with the experimental one.

The comparison of numerical experiments with numerical results for cell size and thickness effects show that the numerical investigation is in good accordance with the experimental one in the supposed assumption. In the numerical part, it is demonstrated that fine micromechanical models based on shell elements give good correlation with honeycomb compression tests for Nomex™ honeycombs. Since these models are highly meshed dependent and CPU intensive for larger structures, this kind of FE modeling is time saving. The scaled modeling approach has numerical advantages.

References

[1] Levent Aktaya, A.F. Johnsona, B.-H. Kröplinb, Numerical modelling of honeycomb core crush behaviour, *Engineering Fracture Mechanics*, 75(9) (2008) 2616–2630.

[2] S. Sadeghnejad, Y. Taraz Jamshidi, R. Mirzaeifar, M. Sadighi, Modeling, characterization and parametric identification of low velocity impact behavior of time-dependent hyper-viscoelastic sandwich panels, Proceedings of the Institution of Mechanical Engineers, *Part L: Journal of Materials: Design and Applications*, (2017) 1464420716688233.

[3] A. Alavi Nia, M. Kazemi, Analytical and numerical investigations on the penetration of rigid projectiles into the foam core sandwich panels with aluminum face-sheets, Proceedings of the Institution of Mechanical Engineers, *Part G: Journal of Aerospace Engineering*, (2017) 0954410017730090.

[4] S. Sadeghnejad, M. Sadighi, A. Ohadi, Free vibration analysis of composite sandwich plate with viscoelastic core, *Key Engineering Materials* 471-472 (2011) 658-663.

[5] B. Castaniéa, C. Bouveta, Y. Aminandab, J.-J. Barrauc, P. Thevenetd, Modelling of low-energy/low-velocity impact on Nomex honeycomb sandwich structures with metallic skins, *International Journal of Impact Engineering*, 35(7) (2008) 620-634.

[6] Z. Sun, S. Shi, X. Guo, X. Hu, H. Chen, On compressive properties of composite sandwich structures with grid reinforced honeycomb core, *Composites Part B: Engineering*, 94 (2016) 245-252.

[7] Y. Taraz Jamshidi, S. Sadeghnejad, M. sadighi, Static and Dynamic Study of Sandwich Panels with Composite Skins and Nomex™ Honeycomb Core, in: The 22nd *Annual International Conference on Mechanical Engineering-ISME*2014, Shahid Chamran University, Ahvaz, Iran, 2014.

[8] S. Ferri R, Static indentation and low velocity impact tests on sandwich plate, in: The 1997 *ASME international mechanical engineering congress and exposition*, ASME, Dallas, TX, USA, 1997, pp. 485–490

[9] Y. Aminanda, B. Castanie, J.-J. Barrau, P. Thevenet, Experimental analysis and modeling of the crushing of honeycomb cores, *Applied Composite Materials*, 12(3-4) (2005) 213-227.

[10] M.S. Hoo Fatt, K.S. Park, Dynamic models for low-velocity impact damage of composite sandwich panels – Part A: Deformation, *Composite Structures*, 52(3–4) (2001) 335-351.

[11] C. Yuan, M. Li, Z. Zhang, Y. Gu, Experimental investigation on the co-cure processing of honeycomb structure with self-adhesive prepreg, *Applied Composite Materials*, 15(1) (2008) 47-59.

[12] J.E. Williamson, A.L. Paul, Response mechanism in the impact of graphite/epoxy honeycomb sandwich panels, in: *Eighth technical conference of the American society for composites*, Technomic Publishing Company, Cleveland, OH, USA, 1994, pp. 287–297

[13] E.J. Herup, A.N. Palazotto, Low-velocity impact damage initiation in graphite/epoxy/Nomex honeycomb-sandwich plates, *Composites Science and Technology*, 57(12) (1998) 1581-1598.

[14] M. Yamashita, M. Gotoh, Impact behavior of honeycomb structures with various cell specifications—numerical simulation and experiment, *International Journal of Impact Engineering*, 32(1–4) (2005) 618-630.

[15] S. Sadeghnejad, M. sadighi, Theoretical and Experimental Static Behavior of Sandwich Structures with Viscoelastic Core, in: *The Bi-Annual International Conference on Experimental Solid Mechanics and Dynamics* (X-Mech-2014), Tehran, Iran, 2014.

[16] L. Lv, Y. Huang, J. Cui, Y. Qian, F. Ye, Y. Zhao, Bending properties of three-dimensional honeycomb sandwich structure composites: experiment and Finite Element Method simulation, *Textile Research Journal*, (2017) 0040517517703602.

[17] M. Giglio, A. Gilioli, A. Manes, Numerical investigation of a three point bending test on sandwich panels with aluminum skins and Nomex™ honeycomb core, *Computational Materials Science*, 56(0) (2012) 69-78.

[18] G.B. Chai, S. Zhu, A review of low-velocity impact on sandwich structures, Proceedings of the Institution of Mechanical Engineers, *Part L: Journal of Materials Design and Applications*, 225(4) (2011) 207-230.

[19] S. Sadeghnejad, H. Ali, M. Sadighi, Contact Force Estimation of Viscoelastic Sandwich Structures under Low Velocity Impact, Using Artificial Neural Network (ANN), in: The 3rd *International Conference on Composites: Characterization*, Fabrication and Application (CCFA-3), Tehran, Iran, 2012.

[20] C. Kralovec, M. Schagerl, K.-U. Schröder, Elastic body impact on sandwich panels at low and intermediate velocity, Proceedings of the Institution of Mechanical Engineers, *Part G: Journal of aerospace engineering*, 229(2) (2015) 221-231.

[21] R. Mines, C. Worrall, A. Gibson, Low velocity perforation behaviour of polymer composite sandwich panels, *International Journal of Impact Engineering*, 21(10) (1998) 855-879.

[22] L.J. Gibson, M.F. Ashby, *Cellular solids: structure and properties*, Cambridge university press, 1999.

[23] M. Zarei Mahmoudabadi, M. Sadighi, A study on the static and dynamic loading of the foam filled metal hexagonal honeycomb–Theoretical and experimental, *Materials Science and Engineering: A*, 530 (2011) 333-343.

[24] M. Zarei Mahmoudabadi, M. Sadighi, A theoretical and experimental study on metal hexagonal honeycomb crushing under quasi-static and low velocity impact loading, *Materials Science and Engineering: A*, 528(15) (2011) 4958-4966.

[25] S.R. Swanson, J. Kim, Design of sandwich structures

under contact loading, *Composite structures*, 59(3) (2003) 403-413.

[26] A.N. Palazotto, E. Herup, L. Gummadi, Finite element analysis of low-velocity impact on composite sandwich plates, *Composite Structures*, 49(2) (2000) 209-227.

[27] R. Mc Farland, Hexagonal cell structures under post-buckling axial load, *AIAA journal*, 1(6) (1963) 1380-1385.

[28] Z. Xue, J.W. Hutchinson, Constitutive model for quasi-static deformation of metallic sandwich cores, *International Journal for Numerical Methods in Engineering*, 61(13) (2004) 2205-2238.

[29] A.-J. Wang, D. McDowell, Yield surfaces of various periodic metal honeycombs at intermediate relative density, *International Journal of Plasticity*, 21(2) (2005) 285-320.

[30] C. Foo, G. Chai, L. Seah, A model to predict low-velocity impact response and damage in sandwich composites, *Composites Science and Technology*, 68(6) (2008) 1348-1356.

[31] A. Alavi Nia, S. Razavi, G. Majzoobi, Ballistic limit determination of aluminum honeycombs—experimental study, *Materials Science and Engineering: A*, 488(1) (2008) 273-280.

[32] H. Yin, G. Wen, Theoretical prediction and numerical simulation of honeycomb structures with various cell specifications under axial loading, *International Journal of Mechanics and Materials in Design*, 7(4) (2011) 253-263.

[33] Z. Hashin, Failure Criteria for Unidirectional Fiber Composites, *Journal of Applied Mechanics*, 47(2) (1980) 329-334.

[34] C. Audibert, A.-S. Andréani, É. Lainé, J.-C. Grandidier, Discrete modelling of low-velocity impact on Nomex® honeycomb sandwich structures with CFRP skins, *Composite Structures*, 207 (2019) 108-118.

[35] R. Seemann, D. Krause, Numerical modelling of partially potted inserts in honeycomb sandwich panels under pull-out loading, *Composite Structures*, (2018).

[36] A. International, *Standard Test Method for Flatwise Compressive Properties of Sandwich Cores*, in, 2000.

[37] Test Method for Compressive Properties Of Rigid Cellular Plastics, in, *ASTM International*, 2004.