Parametric Studies on the FEM Analysis of Foam Core Sandwich Panels with Holes

Ali Vakilazad Sarabi, Masood Mofid
1-Post Graduate Student, Civil Engineering Department, Sharif University of Technology, Tehran, Iran
2- Associate Professor, Civil Engineering Department, Sharif University of Technology, Tehran, Iran
Alivakilazad@gmail.com
Mofid@sharif.edu

Abstract
The aim of the work presented in this research is deal with some of the aspects in the FEM analysis of sandwich panels containing holes which comprised with foam core. In this research, the FEM modeling was produced, analyzed and computed considering laboratory conditions. An extensive parametric study was investigated under different load conditions; different geometrical parameters, such as; dimensions, face thickness, core thickness, size and location of the opening.

Keywords: Sandwich structure, Finite element method analysis, Foam core, Opening.

1. INTRODUCTION

A typical sandwich panel has a three-layer structure. The rigid faces with a relatively high modulus of elasticity are kept apart by the much lighter core which has shear stiffness sufficient to carry most of shear force. The core also acts as a highly effective thermal insulation layer. Variations of this pattern also exist. In multi-layer panels, the faces consist of more than one component and the purpose of the extra layers is usually to improve the performance in fire.

Very light closed-cell polymeric (Divinycell) foam is a favored material for the core of sandwich structures. In classical works (e.g., Gibson and Ashby [1], Shipsha et al. [2], the failure of this material in tension or compression has been described as ductile, failure criteria expressed in terms of stresses have been used. Such failure can exhibit no size effect.

Such ductile response, however, does not take place when high tensile stress concentrations exist, induced for example by notches in laboratory specimens, or various structural holes or accidental damage of real structures. In such a case, the failure of the foam may be brittle. This was first revealed by the notched specimen tests of Zenkert [3] in which the fracture toughness of foam was measured, by the tests of holed panels tests by Fleck and co-workers in Cambridge, and by the finite element studies of foam based on microplane model for foam by Brocca et al. Consequently, the foam, when notched or damaged, must be expected to exhibit size effect, and this is what is confirmed by a recent study (Bazant, Zhou, Novak and Daniel), in which the size effect in Divinycell H100 foam noticed[4].

2. MODELING THE BEHAVIOR OF SANDWICH PANELS

The purpose of the finite element analysis was to predict the behavior of foam core sandwich composites by taking into account both physical and geometrical non-linearity.

The finite element model of an indented sandwich beam was based on the geometry and dimensions of the specimens as used in the indentation tests. The numerical simulations were carried out by using a 3D computational model. The length of the model was reduced to 250 mm. In this way, the number of the elements and the computational time was decreased. The modeling was performed using a finite element computer code. The test specimens were supported by a rigid substrate. Thus, all degrees of freedom were constrained at the lower boundary of the model. The nodes on the vertical axis of symmetry were restrained in horizontal direction.
The indentor was modeled as a rigid body. All rigid body motions were removed, except the translation in the direction perpendicular to the loading. The load was imposed by prescribing vertical displacements to the panel. The contact area between the indentor and the face sheet was computed automatically by the contact algorithm of the FE software.

### 2.1. Material Properties

The face sheets were modeled as a linear-elastic material. For this purpose the *ELASTIC* option in the FE code was used. The input data for this option is given in Table 1. The moduli of the face composite laminates and the foam core were measured using the ASTM methods [5]. The values of the Poisson’s ratio were obtained by the lamina theory or taken from the Divinycell Technical Manual. It should be noted that no damage onset and growth in the face sheet was implemented in the modeling.

<table>
<thead>
<tr>
<th>Table 1 - Mechanical properties of the sandwich constituents, as used in the <em>ELASTIC</em> option of the FE package</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Core(H60)</td>
</tr>
<tr>
<td>Core (H200)</td>
</tr>
<tr>
<td>Face sheet</td>
</tr>
</tbody>
</table>

### 2.2. Models Parameters

As mentioned before, there are some parameters that have important role on the structural behavior of foam core sandwich panels. Dimension parameters are
- Panel dimension(length & width)
- Face sheet thickness
- Foam core thickness
- Thickness ratio: thickness of core/thickness of facesheet ($t_c/t_f$)

#### 2.2.1. Constant parameters

In this research panel is as a square shape and dimension of panel is $250 \times 250 \text{ mm}^2$. The face sheet material is Aluminum. The other parameters are transitive.

#### 2.2.2. Transitive parameters

The definition of effect of transitive parameters is goal of this research. Face sheet thickness, core thickness, core material, opening dimension and opening position are the transitive parameters that predicted in this thesis.

With respect to hole dimension and it’s position, we assumed 10 cases. They are:
- Without hole panel
- Left sided lateral axisymmetric $50 \times 50 \text{ mm}^2$ hole
- Left sided centroid axisymmetric $50 \times 50 \text{ mm}^2$ hole
- Central symmetric $50 \times 50 \text{ mm}^2$ hole
- Right sided centroid axisymmetric $50 \times 50 \text{ mm}^2$ hole
- Right sided lateral axisymmetric $50 \times 50 \text{ mm}^2$ hole
- Left sided axisymmetric $50 \times 50 \text{ mm}^2$ hole
- Central symmetric $100 \times 100 \text{ mm}^2$ hole
- Right sided axisymmetric $100 \times 100 \text{ mm}^2$ hole
- Central symmetric $200 \times 200 \text{ mm}^2$ hole

With respect to other parameters, each of above cases contains 8 types, these are given in table 2
The hardening behavior was defined on basis of the results from foam uniaxial compression test. Characteristic points in terms of nominal stresses and strains were selected along the uniaxial compression curve (figure. 1) and (figure. 2)

![Figure 1. Uniaxial compression response of Divinycell H60 foam [7]](image1)

![Figure 2. Uniaxial compressive and responses of the H200 foam [8]](image2)

The nominal uniaxial stress $\sigma_{nom}$ and nominal strain $\varepsilon_{nom}$ were transformed into true (Cauchy) stress $\varepsilon_{true}$ and volumetric logarithmic plastic strain $\varepsilon_v$ by the equations

$$\sigma_{true} = \sigma_{nom}(1 + \varepsilon_{nom})$$  \hspace{1cm} (1)

$$\varepsilon_{true} = \ln(1 + \varepsilon_{nom})$$  \hspace{1cm} (2)

$$\varepsilon_{pl} = \varepsilon_{true} - \frac{\sigma_y}{E}$$  \hspace{1cm} (3)

$$\varepsilon_v = \ln(1 + \varepsilon_{pl})$$  \hspace{1cm} (4)

where $\sigma_y$ is the uniaxial yield stress ($\sigma_y = 0.80 \text{ MPa}$), $E$ is Young’s modulus ($E = 60 \text{ MPa}$). The nominal stress and strain from uniaxial compression test are defined as follows:

$$\sigma_{nom} = \frac{F}{A}$$  \hspace{1cm} (5)

$$\varepsilon_{nom} = \frac{u}{h}$$  \hspace{1cm} (6)

where $F$ is compressive force, $A$ is the cross-section of the cylindrical foam specimen, $u$ is the axial displacement, $h$ is the height of the undeformed specimen. Since in the tests the sandwich beams undergo large deformations, it is essential to model the geometrical non-linearity. For this purpose the *NLGEOM option in the FE program was used.
2.3. Analysis type

2.3.1. Static Indentation

In this test the elastic–plastic response of sandwich panels with a foam core to local static applied. The study
deals with a 3D configuration, where a sandwich is indented by a rigid cylinder across the whole width of the
specimen. The finite element software is used to model the indentation response of the panels. Both physical
and geometrical nonlinearities are taken into account. The plastic response of the foam core is modeled by
the characteristic options of the FE code. The purpose of the numerical modeling is to develop correct 3D
simulations of the non-linear response in order to further understand the failure modes caused by static
indentation.

The core keeps together the face sheets and resists shear. The primary purpose of sandwich structures is
to produce load-bearing parts with a high bending stiffness-to-weight ratio. Thus, one of major concerns in
the use of sandwich composites is the loss of load caring ability due to a local damage (indentation). Indentation
of sandwich structures can result in considerable crushing of the core and of the face-core
interface without damage of the face sheet. Very often the indentation are barely visible and, therefore, rather
difficult to detect by visual inspections. However, the effect of such sub-surface damages on the shear or
compressive strength may be detrimental for sandwich structures. Generally, the local bending behavior of
sandwich structures can be evaluated from static indentation tests [6].

In order to avoid expensive large scale testing, various analytical models for predicting the indentation
behavior of sandwich composite structures have been developed. The problem of local bending effects in
sandwich panels and beams has been treated by many authors. Very often the analyses are based on the
Winkler foundation model. Other analytical solutions have been obtained by use of the laminate theory
including the transverse shear effects.

An elastic foundation model has been applied to describe the deflection of the loaded face sheet in
sandwich panels. The mechanical behavior of the core has been modeled by continuously distributed. The
effect of localized loads on the overall bending behavior of sandwich composite beams has been considered
by superposing two analyses: the first beam analysis is based on the assumption that the core contribution to
the deformation pattern of a sandwich structure is due to the core transverse shear deformation only; the
second analysis considers the change of the core height due to transverse normal stresses. The superposition
of the two analyses shows that the localized effects on the overall bending behavior become significant when
the ratio of length to depth of the beams is reduced.

The above mentioned analyses are based on the assumption that the core material is linear. However,
when the foams undergo large deformations, the core behavior is non-linear (it crushes) and this leads to
formation of a residual dent after unloading. Thus, it is necessary to introduce this non-linearity in the
analysis of the indentation response of sandwich composite structures. However, consideration of the non-
linear behavior necessitates implementing numeric methods. Therefore, the main aim of this research is to
develop a 3D finite element model characterizing the non-linear static indentation response of foam cored
sandwich composite beams for both loading (indentation). The model will be capable of predicting the
residual dent in the face sheet.

Thus, the model will open opportunities for studying the effect of residual dent on post-indentation
strength of sandwich composite beams. In order to verify the potentialities of the model, the numerical
analysis results are compared to experimental data from indentation tests.
So as to make a three-dimensional analysis, the sandwich beam specimens were indented by using steel
cylinder (30 mm in diameter) across the whole width of the beam cross-section. This relatively small
indentor radius was chosen because it was desirable that the contact between the indentor and the specimen
could be considered as a line load. (figure 3).
The indentation process in foam core sandwich beams strongly depends upon the mechanical properties of the core material.

2.3.2. Inplane test

Finite element modeling of sandwich beam sections with core junctions has been used to evaluate the local stresses and strains at the core junctions. The nonlinear modeling adopted geometrical nonlinearity (large deformations and rotations) as well as nonlinear material behavior (e.g., material plasticity). Three dimensional (3D) models were used in the initial stage of the study. However, the plain strain assumption would artificially increase the overall inplane stiffness of the structure and may lead to transverse stresses that are not present in reality. Further, the 3D modeling showed that the deviations from a plane stress condition were generally small. The material combinations and face and core thicknesses were chosen in accordance with the sandwich configurations used in the experiments. Isoparametric three-dimensional 8 node elements (C3D8R) were used for the finite element mesh of the 3D model. The mesh was refined at the core junction to obtain sufficiently small element edge lengths that ensure convergence of the results. The bond between the face sheets and the core and between the two core materials at the junction was assumed to be perfect and very thin, and hence no additional layers of adhesive were considered in the analysis. Symmetry with respect to the y axis was chosen as the boundary condition for the panel end with the compliant core. The symmetry boundary condition is not used as a geometric representation of the real structure, but it simply corresponds to a zero displacement in the end of x-direction and a constraint of rotation around the z-axis. For the nonlinear model, the displacement was incrementally increased by the FE program until achieved to 3 mm. (figure 4)

![Inplane test (3mm displacement) simulated by FEM code](image)

2.3.3. Out-of-plane compression test

Sandwich structures provide an efficient method to increase bending rigidity without a significant increase in structural weight. These structures, based upon a minimum-gage thickness adequate to carry out-of-plane loads and remain stable under compression without a significant weight penalty. The lack of understanding is partly due to the nonlinear behavior exhibited by composite sandwich structures. Out-of-plane deflections of the order of the facesheet thickness (rather than the sandwich thickness) can result in nonlinear response due to membrane effects. Structural features to accommodate fastening and unsymmetrical damage result in out-of-plane deformations and additional sources of nonlinearities. Thus, linear models cannot adequately predict the structural response let alone damage progression and residual strength. Without adequate models, the ability to predict damage onset due to overloading or growth of existing damage is problematic.

Because of the potentially large out-of-plane deflections compared to the thickness of the facesheet, a nonlinear, large deflection analysis was used to solve the model. Analysis results showed that the total load transfer between the facesheets was small even for the large damage area used in the model. The average loads at the edge of the front and back facesheet were similar, showing little transfer between the facesheets. In the proposed 3D model for simulation out of plane loading the parameters same as inplane test, only the boundary conditions has been changed. Displacements of bottom nodes of a face sheet is completely fixed. The top nodes of another face sheet displaced 5 mm (figure.5). This loading analyzed by displacement controller of FEM software. All papers will take evaluation process in the referee committee.
3. Results

3.1. Indentation bending test

3.1.1. Effect of hole position

In panels with hole, as the displacements increases, in panels with right and left sided centroid holes the maximum principal stress is going to be greater than other holes positions. In these panels in all of the hole positions, reaction force- displacements diagrams are similar together. This situation happened to equivalent plastic strain- displacement diagrams. It shows that reaction force and equivalent plastic strain due to displacement are not dependant on the hole position.

In panels with $10 \times 10 \text{cm}^2$ hole, as the displacements increases, in panels with right and left sided holes the maximum principal stress is going to be greater than centroid hole panel. It seems that the above situations occur because standing of indentor at the edge of these holes.

3.1.2. Effect of hole dimension

The maximum principal stress in all types, initially increases with low rate or constant rate, then increases with high rate. This happens because that the core material has linear behavior at first.

By increasing hole dimensions, the reaction force value is decreased considerably. The opening causes to decrease the loading capability of structure, whatever the hole dimension is greater, the area of section is to be lessen, so the panel shows mellow manner due to loading.

3.1.3. Effect of core material

In similar conditions, maximum principal stress of Divinycell H200 is greater than Divinycell H60. Equivalent plastic strain value (PEEQ) in Divinycell H200 core materials increases linearly with low rates, but in Divinycell H60 core materials, it increases nonlinearly with high rates.

In same conditions, the reaction force value in less displacements for Divinycell H60 is greater than Divinycell H200. By going on the Reaction force value of Divinycell H200 becomes more than Divinycell H60.

3.1.4. Effect of thickness ratio

In thickness ratio of 12.5 the maximum principal stress is greater than others. The analyses predict that when the core depth is small, the shear stress between the core and facing are large. As the core becomes thicker, the stress drops dramatically. This is also the reason why the failures always happen to the cross section where the core is very thin.

The plastic strain value for Divinycell H200 in thickness ratio of 4.17 is greater than others. In sandwich panels the max plastic strain occurs in core, in this case the thickness of core is the least (12.5 mm).

3.2. Inplane test

3.2.1. Effect of hole position

In panels with $5 \times 5 \text{cm}^2$ hole, the increment of maximum principal stress is linear. The maximum principal stress value in lateral holes is less than other cases.

In panels with $10 \times 10 \text{cm}^2$ hole, by increasing displacement value, maximum principal stress in panel with centroid holes is less than others. By displacement increasing, the reaction force value in centroid holes becomes greater than left and right sided holes.
3.2.2. Effect of core material

The maximum principal stress value is nearly same in both kinds of material. It shows that in this test the face sheet has important rule, so the behavior of panel depends on the mechanical properties of facesheet.

3.2.3. Effect of thickness ratio

In thickness ratio of 8.34, value of reaction force is maximum value. In this case the thickness of face and core are the most, so this panel has the most thickness due to other thickness ratios.

3.3. Out-of-plane test

3.3.1. Effect of hole position

The plastic strain value does not change by changing hole position. When the loading area is equivalent and the loading is constant, the maximum value of plastic strain will be constant.

3.3.2. Effect of hole dimension

By increasing hole dimension, reaction force value decreases. By decreasing section area, the loading amount is decreases due to 5 mm displacement

3.3.3. Core Material effect

In Divinycell H200, the maximum principal stress value is greater than Divinycell H60 value, and the reaction force value for Divinycell H200 is considerable greater than Divinycell H60, it is because of mechanical characteristics of Divinycell H200 that these are greater than Divinycell h60.

3.3.4. Thickness ratio effect

In 4.17 and 6.25 thickness ratios, the reaction force value is greater than in 8.34 and 12.5 thickness ratios, total load transfer between the facesheets was small even for the large damage area used in the model, it is considerable that the thickness of core in thickness ratios of 4.17 and 6.25 is equal 12.5 mm and the thickness of core in thickness ratios of 8.34 and 12.5 is equal 25 mm.

4. Conclusions

An analytical method for predicting global and instabilities of a sandwich panel is presented. The sandwich beam is modeled as a 3D inelastic continuum, for this purpose the finite element package was used. The core was modeled as an elastic–plastic material with hardening. The elastic–plastic model was based on a yield criterion which took into account the hydrostatic pressure (mean stress). The input data for the foam elastic–plastic model were obtained from uniaxial compression test in terms of true (Cauchy) stresses and volumetric logarithmic plastic strains. The large deformations induced in the case of localized loading also were taken into account in the modeling. The face sheets were assumed linear-elastic and isotropic. During the testing the beams exhibited pronounced non-linear behavior due to the foam core crushing in the area under the loadings.

The results obtained to given displacements, show that the maximum principal stress for the axisymmetrical deformation mode is always lower that of the symmetrical one. The behavior of the modes is parameterized according to the ratio of core thickness to the face sheet thickness. The results are compared with together, since he present analysis has fewer assumptions than previous analyses, but different combination of geometry and material properties are discussed. The results presented here, reproduce the non-linear indentation behavior of foam core sandwich composite materials behavior accurately for 3D approach. The results that have been presented here are a good prediction of the overall behavior of a sandwich panel in a compressive and bending load environment regardless of the core modulus and thickness ratio. In particular, for thick face sheets and for relatively stiff cores, the present model is found to be more accurate than previous models that assume 2D behavior approach for the sandwich panel and neglect the nonlinear behavior of the foam core.
Fully coupled analysis of inplane and out-of-plane displacements has done to find the condition of the onset of static buckling. Small openings and cut-outs may cover only a part of the width of the sandwich panel. However, small openings reduce the bending strength of a sandwich panel because the area of the face and core is diminished and because the corners of the openings cause stress concentrations that may give rise to an earlier failure than for sandwich panels without openings. Similar stress concentrations are also caused by the corners of large openings. Thus, in addition to the study of the load transfer from a sandwich panel with large openings to adjacent panels, the reduction of the capacity of the sandwich panel with large openings due to the stress concentrations in the corners of the openings has also to be examined.

The multifaceted results of the research capturing plastic strain value, reaction force versus displacement and maximum stress value by using FE models to predict structural response, have provided valuable insight and encouragement to seriously consider foam core sandwich components in structural applications.

11. REFERENCES


7. Victor Iliev Rizov, Non-linear indentation behavior of foam core sandwich composite materials—A 2D approach, Elsevier, February 2005


