THE NUMERICAL MODEL OF SANDWICH PANELS USED FOR SPECIFYING WRINKLING STRESS

***Jolanta Pozorska*[1]*, Zbigniew Pozorski*[2]**

[1] *Institute of Mathematics, Czestochowa University of Technology*
Częstochowa, Poland
[2] *Institute of Structural Engineering, Poznan University of Technology*
Poznań, Poland
[1] *jolanta.pozorska@im.pcz.pl,* [2] *zbigniew.pozorski@put.poznan.pl*

Abstract. The paper presents a numerical model, which allows one to determine the wrinkling stress in sandwich panels. The wrinkling of the compressed face of sandwich panel can be caused by a bending moment or axial force. The developed model enables one to take into account various boundary conditions and micro-profiling of sandwich faces. Damage initiation and propagation in the interface between the face and the sandwich core is defined, too. The numerical procedures and parameters of the FE model are discussed. The obtained solutions are related to the results of actual experiments.

Keywords: *sandwich panels, local instability, wrinkling stress, numerical simulations*

Introduction

Sandwich panels, due to their specific structure, have many attractive features. The panels are characterized by excellent thermal insulation and relatively high load capacity at low weight. Unfortunately, failure of the sandwich structure can be caused by a number of mechanisms: yielding of the tensioned face, shear of the core, compression of the core, wrinkling of the compressed face and others. The basis of mechanics of sandwich structures and damage mechanisms are presented in [1] and [2].

The phenomenon of wrinkling is the most complex and important because it is the most common reason of sandwich structure failure. Therefore, determination of reliable wrinkling stress is crucial. There are three groups of methods of determining of wrinkling stress: analytical, experimental and numerical. Analytical solutions are based on energy methods [3] or differential equations of equilibrium [4]. An analytical determination of the wrinkling stress is very complex and the resulting solutions do not take into account many important factors. Another way to solve the problem are real experiments. It is currently the primary method also recommended in [5]. Laboratory tests are used both to control the current production, as well as in research on new products [6, 7]. A disadvantage of the experiments is the relatively high cost and the limited ability to observe and record phenomena occurring within the tested object. To overcome these difficulties and to support real experiments, numerical simulations are often used [6, 8]. The experimental, theoretical, and FE characterization of the local buckling in foam-core sandwich beams is presented in [9]. The influence of geometrical and mechanical parameters on the nonlinear behavior of sandwich structures was presented in [10]. The extensive parametric, numerical simulations of sandwich beams under pure bending (4-point bending) are presented in [11]. Especially, the influence of the core depth and core mechanical parameters on wrinkling failure was analyzed. The methods of identification of shear modulus of the core were presented and discussed in [12].

This paper presents the numerical model appropriate for the determination of wrinkling stress in sandwich panels. The FE model is 3D and can take into account micro-profiling of faces. The additional interface layer has been proposed to account for failure between the face and the core. This failure is observed in the final stage of the real experiment.

1. Description of the problem

In order to develop a reliable model, which allows for the determination of wrinkling stress, dozens of numerical analyses were conducted. The analysis evaluated the influence of the model parameters on the obtained results. A static system presented in Figure 1 was considered.

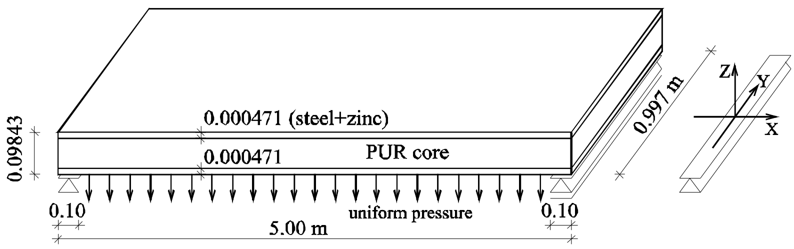

Fig. 1. The static system of a one-span sandwich panel

Sandwich panel with a length of 5.00 m is located on two supports with a width of 10 m. The supports have the freedom of rotation around the y-axis. One of the supports is free to shift along the x-axis. The structure is loaded by pressure perpendicular to the lower face of the sandwich. The loading is directed downwards, i.e. opposite to the z-axis, and causes tension of the lower face ($F2$) and compression of the upper face ($F1$). The total depth of the panel is 98.43 mm. The thickness of each of the faces is $t_{F1} = t_{F2} = 0.471$ mm. The width of the panel is b = 0.997 m. All dimensions are in accordance with the geometrical dimensions of the actual plates, which were subjected to experimental tests.

In order to determine the material parameters of the core and faces, basic experimental tests were performed. Faces, which consist of a steel core with a thickness of 0.414 mm and the zinc layer with a total thickness of 0.057 mm, were subjected to a tensile test. Young's modulus was in the range of 184÷204 GPa. The value 195 GPa was assumed in the model. In order to designate the shear modulus of the core, displacements of the sandwich panel in bending were measured. It was assumed that the core material is isotropic and behaves linearly. Despite these simplifications, it turns out that the designation of the G_C is very difficult [12]. Based on the number of tests, comparing the real and numerical results, the value $G_C = 4220.6$ kPa was designated. The difference between the displacements of the numerical model and actual results in the various tests did not exceed 1%. Because the G_C module is essential for the behavior of the sandwich panel, the other parameters were chosen satisfying the relation $G_C = E_C/2(1+\nu_C)$. On the basis of our own research and observations $\nu_C = 0.02$ was assumed and consequently $E_C = 8610$ kPa.

To develop an appropriate numerical model, the following issues need to be resolved:

a) type of numerical procedure,
b) influence of the initial step in the numerical procedure,
c) necessity of introduction of a interface layer,
d) necessity of introduction of imperfections,
e) type and size of the FE mesh,
f) influence of the constitutive relation of the face material,
g) influence of the numerical parameters improving solution convergence,
h) influence of parameters of a interface,
i) change of the model of the core material.

All of these issues have been studied, and the results are presented and commented on below.

2. The research for reliable numerical model

In the first step the basic 3D model consisting of two steel faces and the PUR core was created. Flat facings were assumed as elastic-plastic material with the yield stress $f_y = 280$ MPa and strengthening to 320 MPa at plastic strain 0.15. Facings were modeled using four node, doubly curved, thin or thick shell, reduced integration, hourglass control and finite membrane strain elements SR4. The core of the panel was modeled using eight node linear brick elements C3D8R.

Since the phenomenon of face wrinkling is associated with the local deformation of the compressed face, a geometrically nonlinear static analysis of the problem was used from the beginning. The possibility of application of Newton's procedure or Riks method was considered. In the analysis of the problem, the Newton procedure proved to be unreliable. There were huge problems with convergence of the solution, and the obtained results in an unpredictable manner depend on the size of the initial step of the procedure. Application of the Riks method made it possible to obtain consistent results, which does not depend on the predetermined initial step.

The influence of the initial step, the introduction of interface and the application of geometrical imperfection on the stress value are presented in Table 1. The value of the initial step refers to the assumed load $q = 1.0$ kN/m^2 (0.1 means 10% of the load q).

The interface layer introduced between compressed face ($F1$) and the core has the thickness equal to 0.5 mm (core thickness was reduced respectively). The interface was modeled using COH3D8 8-node, 3D cohesive elements. The elasticity uncoupled law for cohesive material of the interface was used:

$$\begin{bmatrix} t_n \\ t_s \\ t_t \end{bmatrix} = \begin{bmatrix} K_{nn} & 0 & 0 \\ 0 & K_{ss} & 0 \\ 0 & 0 & K_{tt} \end{bmatrix} \begin{bmatrix} \varepsilon_n \\ \varepsilon_s \\ \varepsilon_t \end{bmatrix}, \tag{1}$$

where t_n is normal traction (stress) and t_s, t_t are shear tractions. Corresponding nominal strains are defined as $\varepsilon_n = \delta_n/T_0$, $\varepsilon_s = \delta_s/T_0$, $\varepsilon_t = \delta_t/T_0$ using separation δ and constitutive thickness of cohesive element T_0. The failure initiation was conditioned by a stress state. The quadratic nominal stress criteria of damage initiation and displacement type with linear softening damage evolution were used. The damage initiation criterion has the form:

$$\left\{\frac{\langle t_n \rangle}{t_n^0}\right\}^2 + \left\{\frac{t_s}{t_s^0}\right\}^2 + \left\{\frac{t_t}{t_t^0}\right\}^2 = 1, \tag{2}$$

where $\langle . \rangle$ is Macaulay bracket with the usual interpretation. The following parameters of the interface were used initially: $K_{nn} = 8610$ kPa, $K_{ss} = K_{tt} = 4220$ kPa, $t_n^0 = 123$ kPa, $t_s^0 = t_t^0 = 112$ kPa.

Geometric imperfections were introduced as a combination of buckling modes of the panel with the adequate multipliers. The problem of buckling of the sandwich structure has been solved independently. The size of introduced imperfections was very small. For the multiplier 0.00005 the initial difference between coordinates of the model nodes is the order of 0.0019 mm.

In the last two columns of Table 1, the extreme stresses occurring in the lower face (σ_{F2}) and upper face (σ_{F1}) are listed. Wrinkling stress determined analytically or calculated on the basis of experiments should be rather compared to the value of σ_{F2} (to the absolute value). The obtained values σ_{F1} express local concentrations in wrinkles of the compressed face.

Table 1

The dependence of the stress on the initial step and the imperfection value

Model	Initial step	Imperfections		σ_{F2} [MPa]	σ_{F1} [MPa]
		No. of modes	Multiplier		
Models without the interface layer					
W1	0.01	–	–	+213.8	–298.2
W2	0.10	–	–	+221.0	–294.3
W1b	0.02	4	0.00005	+158.8	–329.4
W1c	0.10	4	0.00005	+158.8	–329.3
W1d	0.10	4	0.00002	+161.3	–306.5
Models with the interface layer					
Wa1	0.01	–	–	+209.3	–222.9
Wa2	0.10	–	–	+268.7	–272.6
Wa1b	0.02	4	0.00005	+157.4	–307.5
Wa1c	0.10	4	0.00005	+157.3	–307.5
Wa1d	0.10	4	0.00002	+165.1	–307.5

Analyzing the results shown in Table 1, it can be concluded that the initial step size affects only the model without imperfections. For these models, obtained stresses σ_{F2} are very high, inconsistent with the values achieved in the experiments (despite preserved care about the accuracy of the other parameters of the model). In these models, there is also a significant difference between the results obtained for systems with and without the interface. For these reasons, it must be recognized that obtaining correct results is possible only through the assumption of geometric imperfections. The introduction of the interface allows one to differentiate the quality of the connection between the face and the core. In practice it is known that both facings have different adhesion to the core. We analyzed the different types of imperfection (buckling and free-vibration modes), with different values of the multipliers. It turned out that the results closest to the reality are obtained by using buckling modes. The value of the multiplication factor is not so important. Applying a multiplier 0.005 for the model Wa1c we obtain σ_{F2} = 146.7. This means that one hundred times greater multiplier caused the stress reduction 7.3%.

The results shown in Table 1 were obtained for the FEM mesh size 0.03 m. It must be surely recognized that the issue is mesh-dependent. Other dimensions of the mesh were examined: 0.025 and 0.020 m, and the results are shown in Table 2. The difference between the results for the 0.03 and 0.02 m reached 5.5%, while for 0.025 and 0.02 m - only 0.4%.

Very interesting was the analysis of the influence of face material yield strength on the values of wrinkling stress. Instead of declared by the manufacturer, the yield strength 280 MPa, the actual relationship between stress and strain was introduced.

In the tensile test the yield strength was 360 MPa, and the tensile strength reached 436 MPa. In the models W4 and Wa4 the constitutive relation was modified accordingly and the respective wrinkling stress σ_{F2} equal to 164.1 and 166.1 MPa was obtained. It turns out that the change only affects the stress in the compressed face. Change in this stress occurs without significant changes in the tensioned face.

Table 2

The influence of the mesh size on the extreme stress in faces

Model	Mesh size [m]	σ_{F2} [MPa]	σ_{F1} [MPa]
Models without the interface layer			
W1c	0.030	+158.8	–329.3
W3	0.025	+164.3	–292.4
W4	0.020	+163.7	–246.1
Models with the interface layer			
Wa1c	0.030	+157.3	–307.5
Wa3	0.025	+167.1	–295.6
Wa4	0.020	+166.0	–247.8

Fig. 2. Wrinkling of the compressed, upper face ($F1$), sandwich in bending, model Wa4

Potentially, there are a number of numerical tools that can help to obtain a solution in the form of a full course of equilibrium path (with the softening). Already in the interface definition, we can reduce the maximum level of degradation of the finite element. We can introduce damage stabilization by a viscosity coefficient or control the value of displacement at failure. To weaken the requirements for the accuracy of the solution, non-default solution controls can be used. The viscosity coefficient certainly positively influences the solution of the problem. Despite the many possibilities of job control, the equilibrium path after (global) critical point is attained very rarely. Local face debonding and local stress concentrations are realized in the compressed face. Following this, there are problems with obtaining convergence of the solution.

Selection of the interface parameters and change of the model of the core material should still be discussed. Stress results shown in Table 2 are close to real values. The failure mode by wrinkling obtained for the model Wa4 is shown in Figure 2. Selection of each interface parameters must be preceded by numerous experimental studies and parametric analyses.

The presented model uses a simple constitutive relation of the core material. In fact, polyurethane foam is the hyper-elastic material. Fortunately, in the analyzed range of stress and strain, polyurethane foam behaves linearly. The assumed isotropy is also a significant simplification. Please note, however, that it has been consistently assumed both at the stage of experimental tests (to determine the shear module), as well as later in the numerical model. Known to the authors, the latest research shows that the behavior of the foam is more similar to the orthotropic material. Unfortunately, it does not have a complete set of results, which allows one to define the core material in this way.

Conclusions

The presented model was developed for the numerical determination of wrinkling stress. It turned out that in these types of problems, the Riks method is effective. In addition, the introduction of appropriate imperfections is crucial to disturb the symmetry of the numerical problem. The loading, which causes wrinkling, does not depend on the yield strength of the face material. The problem is mesh-dependent, but with a reasonable mesh size, the impact of this effect is negligible

The presented model makes it possible to introduce any face micro-profiling. Insertion of an interface layer enables the definition of failure between the face and core. The choice of interface parameters remains an open question. The introduction of a more advanced model of the core is possible only after the relevant research on material.

References

[1] Davies J. M., Lightweight Sandwich Construction, Blackwell Science Ltd., Oxford 2001.

[2] Zenkert D., An Introduction to Sandwich Construction, Engineering Materials Advisory Services Limited, London 1995.

[3] Allen H.G., Analysis and Design of Structural Sandwich Panels, Pergamon Press Ltd., Oxford 1969.

[4] Stamm K., Witte H., Sandwichkonstruktionen, Berechnung, Fertigung, Ausführung, Springer--Verlag, Wien 1974.

[5] EN 14509 Self-supporting double skin metal faced insulating panels - Factory made products - Specifications.

[6] Steeves C.A., Fleck N.A., Collapse mechanisms of sandwich beams with composite faces and a foam core, loaded in three-point bending. Part II: experimental investigation and numerical modelling, International Journal of Mechanical Science 2004, 46, 585-608.

[7] Pozorski Z., Sól R., Szajda J., Błaszczuk J., Structural behavior of sandwich panels with external deep-profiled and internal soft facing, Engineering Transactions 2013, 61, 3, 173-184.

[8] Pozorski Z., Mechanical models of sandwich panels allowing for local instability, Proc. 8th International Conference on Sandwich Structures, Porto 2008, 1180-1187.

[9] Koissin V., Shipsha A., Skvortsov V., Effect of physical nonlinearity on local buckling in sandwich beams, Journal of Sandwich Structures and Materials 2010, 12, 477-494.

[10] Léotoing L., Drapier S., Vautrin A., Nonlinear interaction of geometrical and material properties in sandwich beam instabilities, Int. Journal of Solid and Structures 2002, 39, 3717-3739.

[11] Jasion P., Magnucki K., Face wrinkling of sandwich beams under pure bending, Journal of Theoretical and Applied Mechanics 2012, 50, 4, 933-941.

[12] Chuda-Kowalska M., Metodyka eksperymentalnych badań trójwarstwowych płyt z cienkimi okładzinami, Wydawnictwo Politechniki Poznańskiej, Poznań 2013.