COMPOSITE PLATES MODELING STRATEGIES FOR THE ESTIMATION OF TRANSVERSE STRESSES

D. Pastorino[a], A. Blázquez[a*], J.A. Reinoso[b], F. París[a]

[a]*Grupo de Elasticidad y Resistencia de Materiales, ETS de Ingeniería, Universidad de Sevilla*
[b] *Institut für Statik und Dynamik, Leibniz Universität Hannover, Germany*
**abg@etsi.us.es*

Keywords: Laminated plates, transverse stresses, FEM.

Abstract
The development of mechanical models that incorporate accurate estimations of the transverse normal and shear stress components are of notable importance in many engineering applications, and therefore it has attracted the interest of the research community along the last decades. This is the case of debonding and delamination of composite laminates, in which these mechanical actions play a crucial role in the initiation and propagation of these damage mechanisms. In this contribution, the accuracy levels of some numerical techniques that are used to compute these stress components are assessed. Geometrically linear and non-linear benchmark problems are solved using such techniques, whose results are compared between them.

1. Introduction

Plate and shell structures are widely used in many engineering branches: aircrafts, ships, bridges, arquitecture, industrial containers, etc. The mathematical model used to solve these structures involves two well-known hypotheses attributed to Love and Kirchhoff, [1]:

(1) Kinematic hypothesis: straight lines normal to the undeformed middle surface remain straight and normal (thin shells) to the deformed middle surface, and do not change length.

(2) Dynamic hypothesis: the normal stress component acting on surfaces parallel to the middle surface may be neglected in comparison with the other stress components.

Through hypotheses, transverse shear strains are constant through the shell thickness, transverse shear stresses being constant along each ply for a laminate. Nevertheless, there are cases which revealed that transverse stresses are directly involved in the failure of the structure. This is the case of debonding and delamination in FRP (Fibre-Reinforced Plastic) due to their small transverse strength.

Nowadays, most of the thin-walled structures are analyzed by FEM using shell elements that implement the aforementioned hypotheses. The vast majority of the commercial FEM programs provide displacements, forces in the elements and the in-plane stresses and strains through the thickness. Regarding the out-of-plane stresses, first, tangential transverse stresses are usually omitted or, at best, are assumed constant along the thickness. Second, normal transverse stress component is always assumed null or computed along the post-processing stage of the analysis by means of enforcing equilibrium conditions. Therefore, if one were interested in computing all of the stress tensor components, i.e. to dismiss the shell model hypotheses, the most immediate method is to use solid elements.

Trying to get a higher level of efficiency from the computational and accuracy points of view, in this communication, a set of alternative modeling techniques that could be used to compute out-of-plane stress components within the numerical analysis has been studied, the possibilities being detailed in section 2. For this purpose, geometrically linear and non-linear problems have been solved, results being compared between them. The commercial software ABAQUS® version 6.12, [2], has been used for the computations here presented.

2. Estimation of transverse stresses: modeling techniques

It is well known that using solid elements a satisfactory accuracy in the computation of the full stress tensor can be obtained. Based on the intrinsic characteristics of standard solid elements, several aspects should be taken into account: (1) the element aspect ratio (maximum length versus minimum one, less than 4 being recommended), (2) the element distortion (angle between faces, between 45 and 135 degree being usually recommended for bricks elements), and (3) the element size. Whereas the first and the second aspects can be more or less controlled using a powerful meshing tools or properly mesh rules, the third one is the main factor responsible for the computational cost, because it is directly related with the number of degrees of freedom of the problem.

2.1. Modeling with solid elements. Global-local strategies

Persisting in the use of solid elements, one solution could be to use a global-local strategy. Thus, solid elements are only used around the area we are interested in, shell elements being used for the rest of the model. There are two main global-local strategies called submodeling and shell-to-solid coupling, see [2]. Both methods are based on a mesh refinement of the zone of interest, but the nature of each of them is significantly different.

On the one hand, the Submodeling technique is based on solving the global and local model sequentially. First the global model is solved and, subsequently these results are used to define the displacements along the interface for the local model. Therefore, global conditions affect the local model but results from the local model do not have any influence on the global one. On the other hand, the Shell-to-Solid Coupling technique solve a unique model coupling degrees of freedom of the fine and the rough meshes using proper interface conditions. Thus, global and local results interact each other along the full process of loading.

In any case, the local model size must be: (1) large enough to include the stress concentration to be analyzed while the disturbances introduced by the boundary conditions from the global model are avoided, (2) and small enough to be attractive from the computational point of view, see [3].

Based on the previous arguments, the selection of the size and, in some cases, the area susceptible for a local analysis require some knowledge of the stress field, which we ought obtain from more efficient tools than a global-local methodology. It seems reasonable that these tools were based on shell models. Two ways have been considered: (1) to obtain expressions of the out-of-plane stress components from the shell forces and moments, (2) to develop numerical tools including these components.

2.2. Modeling with conventional shell elements.

Conventional shell elements use the hypothesis of the mathematical shell model; only the reference surface is discretized, the resulting FE meshes being then two-dimensional. Within these models, the translational and the rotational degrees of freedom are defined at the nodes. ABAQUS® conventional shell element outputs are section forces and moments, stresses: σ_{11}, σ_{22}, σ_{12} (1 and 2 being the in-plane directions); strains: ε_{11}, ε_{22}, ε_{12}; curvatures: κ_{11}, κ_{22}, κ_{12}; and strains: ε_{13}, ε_{23}, ε_{33} only for some specifics types of elements.

Estimation of transverse stresses requires the shell theory to be enhanced. In [4], instead of determining transverse shear stresses from the shell constitutive relations, the use of the differential equilibrium equations (assuming membrane stresses to remain those obtained from the shell theory) is proposed. Expressions for σ_{xz}, σ_{yz} and σ_{zz} were obtained, although some simplified assumptions were carried out. Analyzing these fields the adequate size for the local model could be estimated.

2.3. Shell-solid-shell model

Trying to have advantages from the bending behavior of shell elements and the capacity of solid element to consider all the stress components, a combination of shell and solid elements is proposed in this section.

A very thin (0.2 mm in thickness) layer of solid elements are placed on the height (along the thickness) where the full stress tensor is desired to be known, and shell elements are used to model the material above and below this layer. In this way, a complete numerical prediction of the stresses can be obtained where the solid element layer is located.

However, this modeling alternative also presents some drawbacks that should be highlighted. First, it is necessary to couple the degrees of freedom of the nodes belonging to the solid elements (with 3 degrees of freedom) with the corresponding nodes of the shell elements (with 5 or 6 degrees of freedom). Furthermore, the choice of the proper master nodes (driver) and slave nodes (driven) cannot be careless; shell degrees of freedom must be the master ones. Second, the aspect ratio of the solid element would be very poor (greater than 10 or even more) for a competitive mesh from the computational cost point of view. This last difficulty could be avoided substituting solid elements by continuum shell elements (commented in Section 2.4.2). Nevertheless, problems associated to coupling between degrees of freedom persist, and it does not seem justified the use of the shell-continuum shell coupling instead of a full continuum shell mesh.

2.4. Modeling with advanced shell elements

Due to the poor performance of standard solid elements in bending dominated problems and the limitations of modeling shell structures with conventional shell elements, a high number of Shell Theories have been developed in the last thirty years, see [5].

2.4.1. Three dimensional shell elements

Shell-like models based on three dimensional shell formulations preserve the computational efficiency of classical shell finite elements and incorporate additional mechanical features that allow more accurate estimations of the full set of representative stress components to be performed. One of the most attractive shell formulations is the so-called 7-parameter model originally developed in [6], and further extended for composite structures in [7]. This formulation proposed a kinematic description that regarded the displacement of the reference middle surface together with the so-called difference vector to update the normal shell vector along the deformation process, in this way circumventing the use of complex co-rotational parameterizations for large displacement applications. Low-order finite elements based on this underlying shell model additionally require the use of numerical techniques that alleviate the different locking pathologies. In this concern, a combination of Enhanced Assumed Strain (EAS) and the Assume Natural Strain (ANS) techniques are, following the scheme carried out in [6], employed.

2.4.2. Continuum shell elements and solid shell elements

Continuum shell elements discretize the entire three-dimensional solid. They have 8 nodes (for the quadrilateral linear case) having only translation degrees of freedom (rotations are defined by the displacement of the nodes on the top and bottom of the shell). A continuum shell mesh looks like a solid element mesh, but their formulation is founded on identical hypothesis than conventional shells. These elements include the effects of transverse shear deformation and thickness change, they can be stacked through thickness, and allow the prediction of transverse stress. Nevertheless, convergence may not be monotonic, the thickness strain mode may yield a small stable increment of load for thin shells, and care has to be taken to reproduce the boundary conditions associated to rotational degrees of freedom.

Solid shell elements are similar to those denominated as continuum shell elements. The main difference between them is that while continuum shell elements work in the space of resultant forces and moments along the thickness, solid shell elements work in the space of stresses.

ABAQUS® continuum shell elements can be stacked. The outputs are section forces and moments; stresses: σ_{11}, σ_{22}, σ_{12} (1 and 2 being the in-plane directions); strains: ε_{11}, ε_{22}, ε_{12}; curvatures: κ_{11}, κ_{22}, κ_{12}; strains: ε_{13}, ε_{23}, ε_{33}; and the thickness average stresses: S_{11}, S_{22}, S_{33}, S_{12}, S_{13}, S_{23}. See Section 29.6.8 of [2] for details. ABAQUS® does not implement solid shell elements.

3. Aplications

In this communication a 200×200 mm^2 rectangular plate, 2 mm thickness, has been chosen for accomplishing the computations. Notice that the smaller the thickness is (in this case thickness is 1/100 times the characteristic plane dimension), the smaller the transverse stresses are, comparatively speaking with the in-plane stresses. The two load cases presented can be solved very accurately (displacements and shell forces) using a 20×20 linear shell mesh, i.e. 400 shell elements. However, the corresponding solid model needs about 40000 linear elements (and this considering only 4 elements along the thickness, which means bricks elements of size 0.5×2×2 mm^3).

The material employed is a symmetric laminated composite, with lamina properties given in Table 1, and symmetric stacking sequence [45/−45/0/90/0/−45/45].

Properties	E_1 [Gpa]	E_2 [Gpa]	G_{12} [Gpa]	G_{13} [Gpa]	G_{23} [Gpa]	ν_{12}
Value	131	9.75	4.65	4.65	3	0.3

Table 1. Properties of the lamina.

3.1. Bending

First of all, the analysis of a simply supported plate (considered infinite in one direction) is presented. The load is being applied normal to the plate on the upper surface.

Mesh size has been chosen using a mesh refinement procedure in which element size was divided by 2 every step, the process ends when the relative differences between the computed in-plane stresses and the analytical reference value were less than 0.1%. Table 2 shows the element size resulting with this rule for every model considered.

Model	Solids	Classic shells	Continuum shells	Shell-solid-shell
Element size [mm]	3.3×3.3×0.29	6.7×6.7	3.3×3.3×0.29	Shell: 4×4 Solids: 4×4×0.29
DOF	75600	5400	75600	37500
CPU time [s]	188.06	8.64	30.50	19.05

Table 2. Element sizes resulting from the mesh refinement procedure. Bending case.

Figure 1 shows the distribution of stresses (1 and 2 refer to the local direction of each lamina) through the thickness at points (0,0) and (a/4,a/4). Results corresponding: to the corrected analytical solutions given in [4], to ABAQUS® models using SC8R (continuum shell),

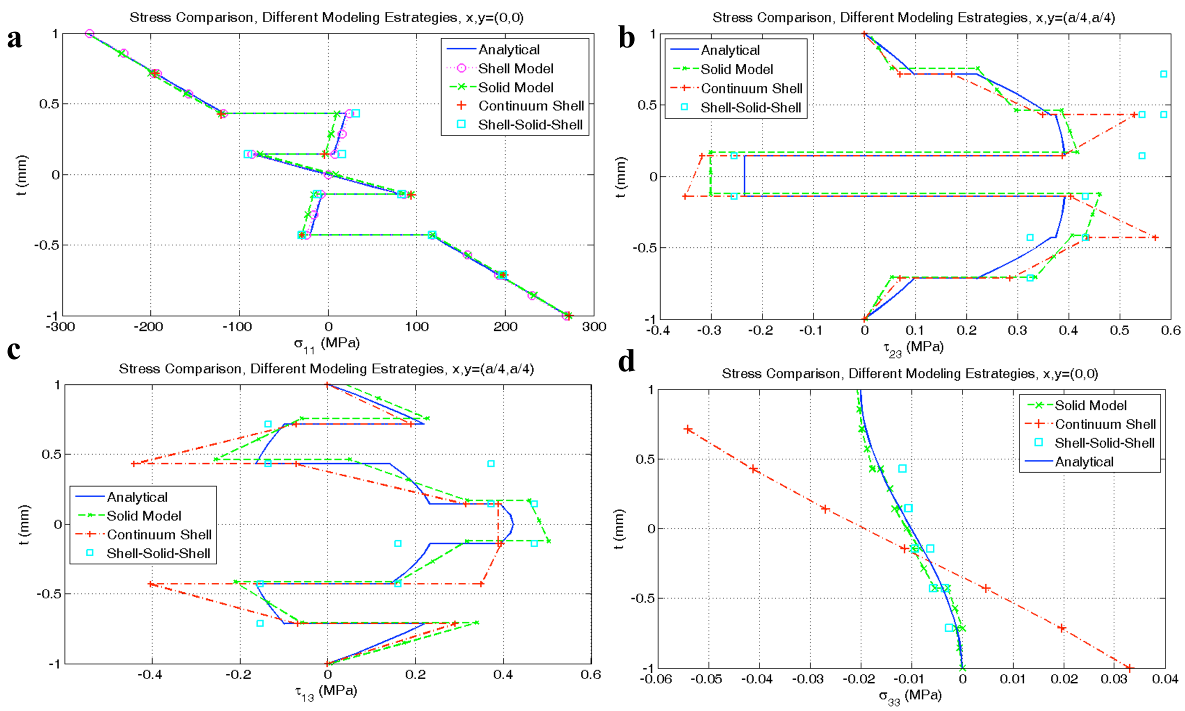

Figure 1. Stress distributions through the thickness of the plate for different modeling strategies.

C3D20R (quadratic solid element), and to shell-solid-shell models (one model for each position of the solid layer) are presented.

Regarding the in-plane stress components, all procedures lead to satisfactory results. Differences appear in normal and tangential transverse stresses. On the one hand, and in spite of the relatively large size (only one element along the thickness of each lamina), solid element model provides an accurate result with respect to the analytical estimation (remember that analytical values were obtained assuming some approximations). On the other hand, the shell-solid-shell model mesh has to be relatively fine to get satisfactory results, with the consequent increment of the computational cost. Finally, results with continuum shell elements do not fulfill boundary conditions by far (see Figure 2d), which made them under question.

3.2. Postbuckling

Secondly, a new loading case, a plate subjected to a compression load higher than the critical buckling load (around five times higher) is presented. To perform, a geometrically non-linear analysis of the problem is needed. Moreover, due to the fact that there is no immediate analytical solution available to compare with, the comparison of the different modeling approaches is made with respect to a local model solution.

The resulting element sizes are shown in Table 3, where a fast convergence of the in-plane stresses has been obtained. Nevertheless, finer meshes are required in order to approximate accurately the tangential transverse stresses, mainly due to the non-linearity of the problem.

Model	Solids	Classic shells	Continuum shells	Shell-solid-shell
Element size [mm]	2.5×2.5×0.29	1.32×1.32	2.5×2.5×0.29	Shell: 1.72×1.72 Solids:1.72×1.72×0.29
DOF	134400	138624	134400	201804
CPU time [s]	84618	927	6718	8185

Table 3. Element sizes resulting from the mesh refinement procedure, postbuckling case.

Stress results along thickness at the point ($a/4$,$a/4$) are shown in Figure 2, where it can be seen that the estimations of in-plane stresses are quite accurate. However, the peeling stresses do not reach the complete convergence, even with the small size of elements employed. This indicates that a local model analysis is needed.

4. Summary and Conclusions

Different approaches for the assessment of the three-dimensional stress analysis of a layered shell, making use of FEM commercial software ABAQUS, are herein presented.

From the obtained results, several conclusions can be reached. In general, it is found that the most accurate approach is to use solid elements, but it has the disadvantage that it is very expensive computationally. Nevertheless, in complex or large problems an option to decrease computational time is to use coarse elements (conventional shells) in the whole domain but in the region in which the three-dimensional stress field is required to use fine meshes of solid elements.

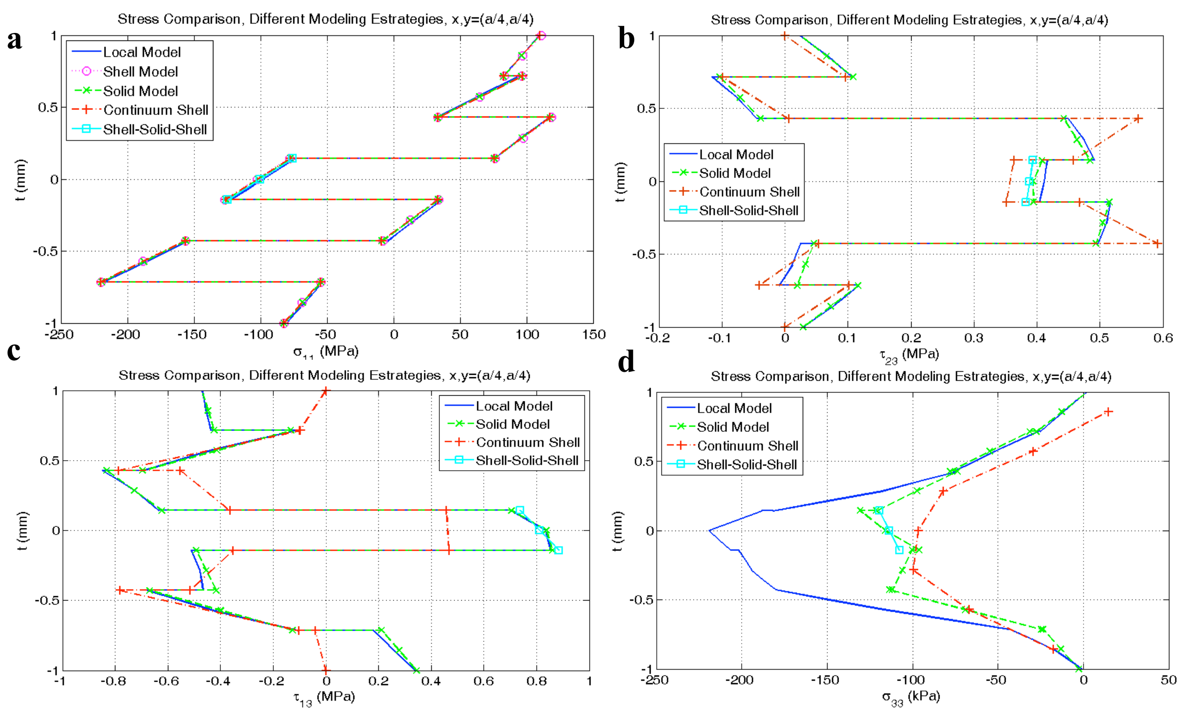

Figure 2. Stress distributions through the thickness of the plate for different modeling strategies.

Furthermore, it can be shown that continuum shell elements estimate with a satisfactory level of accuracy the shear stresses but not the peeling ones, since the elements do not include transverse normal stress directly in equations; on the contrary they were estimated through post-processing the results. Hence, the application of continuum shell elements gives rise to a qualitative solution, but not accuracy enough is reached to be considered as a final solution. Nevertheless, it can be used to estimate the location and size of the area susceptible for a local analysis. A similar reasoning could be applied to shell-solid-shell approach.

Beside the development of presented options, it is important to consider other possibilities: analytical results for plate bending could be improved, and other non-conventional shell elements, such as the 7-parameter shell element introduced in section 2.5, could be tested and assessed in these structural applications.

Acknowledgements

This research is supported by the Ministerio de Economía y Competitividad of Spain (Project DPI2012-37187) and the Consejería de Economía, Innovación y Ciencia of Junta de Andalucía (Project TEP2011-07093).

References

[1] P.L. Gould. *Analysis of shells and plates*. Upper Saddle River, N.J. Prentice Hall, 1999.
[2] *ABAQUS Documentation*. Dassault Systèmes, Providence, RI, USA, 2012.
[3] J. Reinoso, A. Blázquez, A. Estefani, F. París, J. Cañas, E. Arévalo and F. Cruz. Experimental and three-dimensional global-local finite element analysis of a composite component including degradation process at the interfaces. *Composites: Part B*, volume 43:1929-1942, 2012.

[4] R. Tanov and A. Tabiei. Adding transverse normal stresses to layered shell finite elements for the analysis of composite structures. *Composite Structures*, volume 76:338-344, 2006.

[5] M. Bischoff, WA. Wall, K-U. Bletzinger and E. Ramm. *Chapter 3: Models and Finite Elements for Thin-Walled Structures*, in: Encyclopedia of Computational Mechanics (E. Stein, R. de Borst, T.J.R. Hughes, eds), Volume 2: Solids and Structures, John Wiley & Sons, Ltd. (2004)

[6] M. Bischoff and E. Ramm Shear Deformable Shell Elements for Large Strains and Rotations". Int. J. Numer. Meth. Engng. 40:4427--4449, 1997.

[7] J. Reinoso Study of composite stiffened panels in postbuckling regime. PhD dissertation, University of Seville, 2012.