JOURNAL OF AIRCRAFT
Vol. 42, No. 3, May–June 2005

High-Order Finite Elements Reduced Models for Use in Flutter Design Tool

Paolo Lisandrin* and Michel van Tooren†
Delft University of Technology, 2625 HH Delft, The Netherlands

A method is presented to build reduced (equivalent) models of thin-walled composite structures to be used in the modal analysis of composite aircraft wings. The technique is developed to allow finite element modeling and analysis using a single-type, three-dimensional orthotropic *p* element. The use of a single element guarantees speed and flexibility in the (re)modeling of the structure and reduces the modeling and analysis errors connected to finite element analysis in preliminary-design/multidisciplinary-optimization environments. The method is tested on a sandwich plate. The dynamic behavior of sandwich plates is determined by the in-plane extensional and shear stiffness of the facings just as the behavior of the wing box is determined by the extensional and shear stiffness of skin panels and spars. Numerical results obtained with solid orthotropic *p* elements show the validity of the method. Comparison with results from traditional linear shell finite elements models shows that with the new method more accurate results can be obtained with fewer degrees of freedom.

Nomenclature

$\boldsymbol{A}$ = matrix of the extensional and shear stiffenesses of the laminate
$\boldsymbol{B}$ = matrix of the bending–extension coupling stiffnesses of the laminate
$\boldsymbol{D}$ = matrix of the bending and torsion stiffnesses of the laminate
$E_{1,2}$ = Young's modulus along the in-plane material principal directions 1 and 2 of an orthotropic plate
E_{33} = Young's modulus along the material principal direction 3, thicknesswise
G_{12} = shear modulus for an orthotropic plate, in-plane material principal directions
G_{13} = shear modulus for an orthotropic plate, material principal directions 1, in-plane, and 3, out-of-plane
G_{23} = shear modulus for an orthotropic plate, material principal directions 2, in-plane, and 3, out-of-plane
$k^0_{x,y}$ = laminate's middle-surface curvatures around x and y axes
k^0_{xy} = laminate's middle-surface twist curvature
$N_{x,y}$ = normal forces per unit length along x and y axes
N_{xy} = shear force per unit length in the xy plane
Q_{ij} = components of the lamina's stiffness matrix in the material principal frame of reference
$\bar{Q}_{ij}$ = components of the lamina's stiffness matrix in a general frame of reference
z_k = position of the kth lamina of the laminate respect to the laminate midplane
γ_{xy} = shear strain of the lamina in a general frame of reference
γ_{12} = shear strain of the lamina in the 12 plane, material principal frame of reference
$\epsilon_{x,y}$ = normal strain of the lamina in a general frame of reference
$\epsilon^0_{x,y}$ = laminate's middle-surface strains along x and y axes
ϵ^0_{xy} = laminate's middle-surface shear strain
$\epsilon_{1,2}$ = normal strain of the lamina in the directions 1 and 2 of the material principal frame of reference
ν_{12} = Poisson's modulus between the in-plane material principal directions 1 and 2 of an orthotropic plate
ν_{23} = Poisson's modulus between the material principal directions 2, in-plane, and 3, out-of-plane, of an orthotropic plate
ν_{31} = Poisson's modulus between the material principal directions 1, in-plane, and 3, out-of-plane, of an orthotropic plate
$\sigma_{x,y}$ = normal stresses along general x and y directions
$\sigma_{1,2}$ = normal stresses along directions 1 and 2 of the material principal frame of reference
τ_{xy} = shear stress component in the xy plane
τ_{12} = shear stress component in the 12 plane, material principal frame of reference

I. Introduction

NEW generation aircraft have an ever increasing percentage of composite materials, built from unidirectional laminas or fabrics. Appropriate tools to support the design and analysis of composite structures in the preliminary design stage are, therefore, becoming more important. The present paper deals with the definition of reduced (equivalent) models for structural elements made of fiber-reinforced materials. The method is based on the use of high-order, p-formulation, solid, that is, brick, geometry finite elements. Higher-order finite elements have been implemented in the so-called p formulation by Babuska et al.[1] In this particular formulation, convergence is achieved increasing the order of the shape functions, built through the use of so-called hierarchical polynomials, such that every time the order of a polynomial is increased new degrees of freedom are added, in terms of internal nodes.

A first advantage of the proposed approach stems from the use of finite elements based on a fully three-dimensional formulation for the displacement field. The lack of any a priori assumptions on the nature of the structure makes this element suitable for representing any kind of structural part, that is, both thin and thick shells and beams.[2] Therefore, only one kind of finite element can be used to model and analyze a complete structure, with the potential of bringing a major improvement in speed and flexibility in the (re-)modeling phase of the structural analysis. This improvement is particularly important in preliminary design context, in which a lot of time is spent in the preparation, modification, and updating of the models when modifications are introduced in the design. These operations are very time consuming and, normally performed by hand, restrict the time dedicated to the computational phase.

Received 5 April 2004; presented as Paper 2004-2002 at the AIAA/ASME/ASCE/AHS/ASC 45th Structures, Structural Dynamics, and Materials Conference, Palm Springs, CA, 19–22 April 2004; revision received 8 July 2004; accepted for publication 28 July 2004.

*Ph.D. Student, Faculty of Aerospace Engineering, Kluyverweg 1. Member AIAA.

†Full Professor, Faculty of Aerospace Engineering, Kluyverweg 1. Member AIAA.

Using only one kind of finite element instead the meshing phase can be easily automated. No choice between different elements is needed, thus, reducing the modeling errors to a minimum.

Other authors used solid finite elements either for both dynamic and static analysis, but the analyses are based on assumptions on the behavior along the thickness that limit the applicability of these methods. For example, a solid plate finite element for modeling static and fracture behavior of thin shells structures made of composite materials has been developed by Hashagen and de Borst[3]; however, the behavior along the thickness of the plate is limited to a quadratic approximation and it still suffers from shear locking problems.

Another advantage of the present approach comes from the use of a finite element based on high-order shape functions, in the form of the p formulation. It is known that with the commonly used linear interpolation shape functions, a locking phenomenon can occur. It is due to the inability of the lower-order shape functions to interpolate the displacement field correctly, which leads to an overestimation of the stiffness of the element. Most of the finite element codes propose as remedies the use of so-called reduced or selective integration methods that mistakenly integrate the shear terms of the stiffness matrix. In the present work, the use of high-order shape functions directly eliminates the occurrence of this phenomenon without recourse to any numerical treatment.

A third advantage stems from that, in the p elements, the order of the shape function is not fixed; it can be varied in principle to any value. Adaptive analyses can be performed, deciding a starting order for the polynomials to be used and increasing it until a certain convergence criterion has been satisfied or when a maximum polynomial order has been reached. Convergence checks on complex structures such as a whole aircraft become feasible, whereas by using common linear elements, or using other higher-order formulations where the order of the polynomial is fixed, as by Morino et al.,[4] this would not be possible without regenerating the complete mesh several times. In industry practice, just one final mesh is generated with the number of elements defined a priori by experienced people, or especially for modal analyses, reduced models made of beams and lumped masses are created. These models are not always affordable, especially in structures made of composite materials where the reduction process is even more diffcult. P-formulation elements make the convergence check process easily automatable, and this is particularly important in preliminary design/multidisciplinary-optimization (MDO) environment, where no experimental data are available and the results must rely only on the numerical calculations.

The capabilities offered by the use of solid p elements have been translated into an element independent modeling and analysis tool, where a fully automated and adaptive analysis can be carried out. The tool is part of a design and engineering engine (DEE)[5] currently under development at the System Integration Aircraft group of Delft University of Technology. The DEE consists of a set of properly interconnected toolboxes (Fig. 1) such that fully automated multidisciplinary analysis and optimization for an aircraft configuration is feasible. The core element of the DEE is a multimodel generator[6,7] in which the description of the design concept to be analyzed by the different disciplines is done using the capabilities offered by the parametric modeling and knowledge-based engineering features.

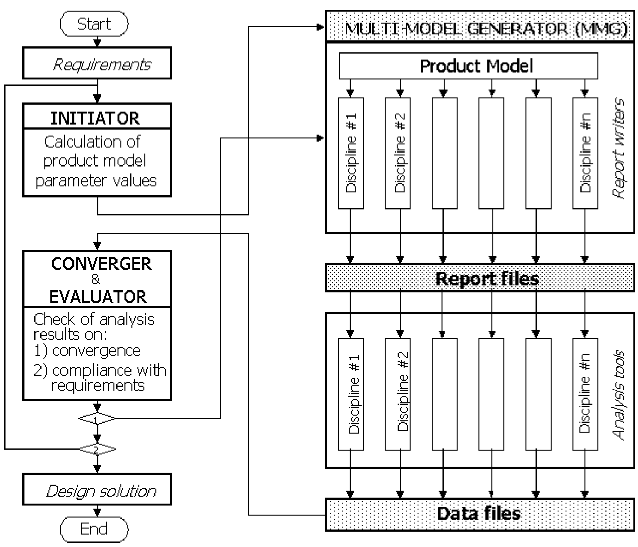


Fig. 1 Paradigm of a DEE.

A preliminary evaluation of this modeling approach has been performed to evaluate the modal and static behavior of a reference wing box, made of isotropic material, and it has been shown[8] that eight-node brick finite elements with cubic shape functions can be used to model a wing trunk, obtaining results comparable with the model based on standard linear shell elements.

In the following sections, a method for the modeling of a composite structure using solid p elements will be defined and tested, using the p elements available in the commercial finite element package NASTRAN.[9]

II. Solid Orthotropic Model for Composite Plates

A unidirectionally reinforced or a woven lamina can be reasonably considered as an orthotropic material loaded in plane stress.[10] The elastic behavior of the lamina can then be defined by a stiffness matrix or a compliance matrix, the latter having the closest relation to the engineering constants of the material. The stiffness matrix is defined by the following stress–strain relations, in the material principal frame of reference, Eq. (1), and in a general frame of reference, Eq. (2). Thus,

$$\begin{bmatrix}\sigma_1\\ \sigma_2\\ \tau_{12}\end{bmatrix}=\begin{bmatrix}Q_{11} & Q_{12} & 0\\ Q_{12} & Q_{22} & 0\\ 0 & 0 & Q_{66}\end{bmatrix}\begin{bmatrix}\epsilon_1\\ \epsilon_2\\ \gamma_{12}\end{bmatrix} \tag{1}$$

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix}=\begin{bmatrix}\bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16}\\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26}\\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66}\end{bmatrix}\begin{bmatrix}\epsilon_x\\ \epsilon_y\\ \gamma_{xy}\end{bmatrix} \tag{2}$$

From laminate theory, the relations between force and deformations of a thin laminate made of a stack of N laminas having different orientations are

$$\begin{bmatrix}N_x\\ N_y\\ N_{xy}\end{bmatrix}=\begin{bmatrix}A_{11} & A_{12} & A_{16}\\ A_{12} & A_{22} & A_{26}\\ A_{16} & A_{26} & A_{66}\end{bmatrix}\begin{bmatrix}\epsilon_x^0\\ \epsilon_y^0\\ \epsilon_{xy}^0\end{bmatrix}+\begin{bmatrix}B_{11} & B_{12} & B_{16}\\ B_{12} & B_{22} & B_{26}\\ B_{16} & B_{26} & B_{66}\end{bmatrix}\begin{bmatrix}k_x\\ k_y\\ k_{xy}\end{bmatrix} \tag{3}$$

$$\begin{bmatrix}M_x\\ M_y\\ M_{xy}\end{bmatrix}=\begin{bmatrix}B_{11} & B_{12} & B_{16}\\ B_{12} & B_{22} & B_{26}\\ B_{16} & B_{26} & B_{66}\end{bmatrix}\begin{bmatrix}\epsilon_x^0\\ \epsilon_y^0\\ \epsilon_{xy}^0\end{bmatrix}+\begin{bmatrix}D_{11} & D_{12} & D_{16}\\ D_{12} & D_{22} & D_{26}\\ D_{16} & D_{26} & D_{66}\end{bmatrix}\begin{bmatrix}k_x\\ k_y\\ k_{xy}\end{bmatrix} \tag{4}$$

where the terms of the $\boldsymbol{A}$, $\boldsymbol{B}$, and $\boldsymbol{D}$ matrices are defined as

$$A_{ij}=\sum_{k=1}^{N}(\bar{Q}_{ij})_k(z_k-z_{k-1}) \tag{5}$$

$$B_{ij}=\frac{1}{2}\sum_{k=1}^{N}(\bar{Q}_{ij})_k\left(z_k^2-z_{k-1}^2\right) \tag{6}$$

$$D_{ij}=\frac{1}{3}\sum_{k=1}^{N}(\bar{Q}_{ij})_k\left(z_k^3-z_{k-1}^3\right) \tag{7}$$

The preceding mathematical representation is the result of assumptions that eventually eliminate the dependence of the displacements from the thicknesswise coordinate of the laminate. However, in our case the solid modeling reintroduces this dependence. We could then follow two approaches to tackle the problem: model each lamina of the laminate as a solid, but this is too computationally expensive for our goals, or better, consider the laminate as a general anisotropic and homogeneus material. Its properties can be derived either by experimental tests or, as done in this paper, just using Eqs. (1–4) in a backward manner, because from the components of the matrices $\boldsymbol{A}$, $\boldsymbol{B}$, and $\boldsymbol{D}$, using Eq. (5), we can calculate back the stiffness matrix $\boldsymbol{Q}$ for the solid model. In particular, in the hypothesis that the laminate is made of a one-layered homogeneus and orthotropic material we have

$$A_{ij} = \bar{Q}_{ij} t, \qquad B_{ij} = 0, \qquad D_{ij} = \bar{Q}_{ij} t^3 \big/ 12 \qquad (8)$$

From Eq. (8), it can be seen that for the determination of the components of the matrix $\boldsymbol{Q}$ we have two equations in one unknown. As a consequence, it should be decided from which equation to determine the Q_{ij} values, thus, choosing whether the solid model should match the membrane/shear ($\boldsymbol{A}$ matrix) or the bending/torsion ($\boldsymbol{B}$ matrix) behavior. In any case, the behavior of the one-layer solid laminate and the layered laminate will be different. For a wing skin or a fuselage panel, a reasonable choice is to derive the Q_{ij} values from the $\boldsymbol{A}$ matrix because they carry mainly membrane and shear loads. From the $\boldsymbol{Q}$ values, the engineering constants E_1, E_2, G_{12}, and ν_{12} can be determined. Because the laminate is modeled as a solid, we will need the other five missing constants, E_3, G_{13}, G_{23}, ν_{13}, and ν_{23}, that have to be derived from physical consideration because the laminate model is determined only by the four constants mentioned earlier. The determination of the offplane elastic properties can be done through the following physical considerations.

1) In the case of a the unidirectional lamina, the E_{33} value is determined by the elasticity of the resin, so that it is equal to the E_2 value of the lamina itself.

2) The ν_{23} and ν_{31} values have been made equal to the ν_{12} value because a sensitivity analysis can show that the Poisson values have almost no influence on the modal frequencies.

3) The G_{13} and G_{23} values have been set equal to the G_{12} value. For thin-walled structural elements, it is not important to have the exact value for these constants because they behave as shear indeformable elements, so that the only important thing is not to give values zero.

III. Applications

The equivalencing method is tested on the flat sandwich panel that has been extensively studied in Ref. 11. Indeed the dynamic behavior of sandwich panels is determined by the in-plane extensional and shear stiffness of the facings just like the behavior of the wing box is determined by the extensional and shear stiffness of skin panels and spars.

The sandwich panel is made of composite facings and a titanium honeycomb core. Geometrical and elastic properties for the components are taken from Ref. 11 and reported in Tables 1 and 2. Lamination sequence is $[45, -45, 90, 0]_s$. In Ref. 11, no data are given for Poisson's number of the honeycomb core, and so zero values have been assigned to the three Poisson values because they can be either positive or negative. Only the ν_{31} value has been set equal to the value of the titanium alloy used to make the honeycomb, as stated in Ref. 12.

In Sec. III.A, finite element models will be built for a composite plate having the same elastic properties as the facings of the sandwich panel. These models will be used to show some obvious but important consequences of the assumptions made for the equivalent model and to demonstrate some guidelines in finite element modeling and analysis using solid p elements.

In Sec. III.B, a convergence analysis of the modal frequences of the sandwich panel will be done, comparing the different kinds of equivalent and original models.

Table 1 Material properties of the titanium honeycomb core (from Ref. 11)

Property	Value
Weight density ρ, N/m^3	1066.5
Young's modulus $E_{11} = E_{22}$, Pa	1.65E+05
Young's modulus E_{33}, Pa	1.03E+09
Shear modulus G_{13}, Pa	4.82E+08
Shear modulus G_{23}, Pa	4.82E+08
Shear modulus G_{12}, Pa	3.38E+08

Table 2 Material properties of the Im7/Peti5 layers used for the facings (from Ref. 11)

Property	Value
Young's modulus E_{11}, GPa	151.92
Young's modulus E_{22}, GPa	9.646
Shear modulus G_{12}, Pa	2.564
Poisson's ratio ν_{12}	0.34
Weight density ρ, N/m^3	15750
Ply thickness t, m	1.397E–04

Table 3 Frequency of first vibration mode for different models of unconstrained facing

Model	Frequency, Hz
Laminated plate, 1	2.968980
One-layered plate, 2	5.046190
One-layered solid, 3	5.045440

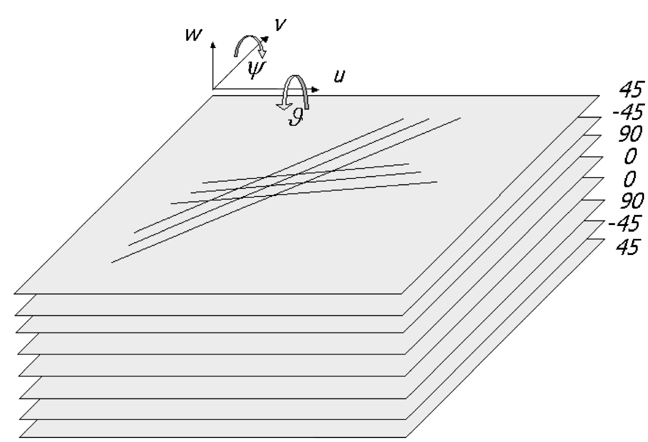


Fig. 2 Full multilayered plate model: degrees of freedom per node and stacking sequence.

A. Unconstrained Facing Analysis

The technique for building equivalent models of laminates is tested first on a composite plate, which has the properties of the facings of the sandwich panel. The modal analysis is carried considering three different finite element models: Model 1 (Fig. 2), regarded as the reference, is made of four-node plate linear elements (CQUAD4), whose elastic properties are specified assigning each ply and its orientation through a PCOMP card in NASTRAN. Model 2 (Fig. 3) again is made of CQUAD4 elements whose properties are assigned considering the plate a single-layered orthotropic material. Model 3 (Fig. 4) is made of eight-node brick p elements (CHEXA, p formulation), whose elastic properties are again assigned considering it a single-layered orthotropic material. In the latter two cases, the engineering constants of the orthotropic materials are determined using the equivalencing procedure described in Sec. II. The basic assumption of matching only the membrane and shear behavior of the facing generates differences between the equivalent and the reference model. As an example, in Table 3 the frequencies determined for the three models for a 12×12 meshing (considered in Ref. 11 as converged values) are reported. As can be seen, there are large differences between model 1 and models 2 and 3. It is clear also that there is almost no difference in the values of the frequencies predicted by the equivalent plate and solid finite

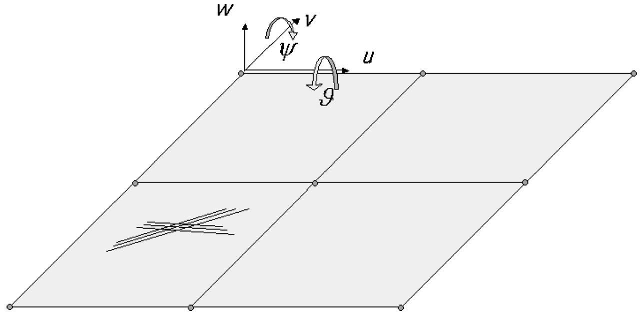


Fig. 3 Single-layered orthotropic plate model: sample mesh with the degrees of freedom per node.

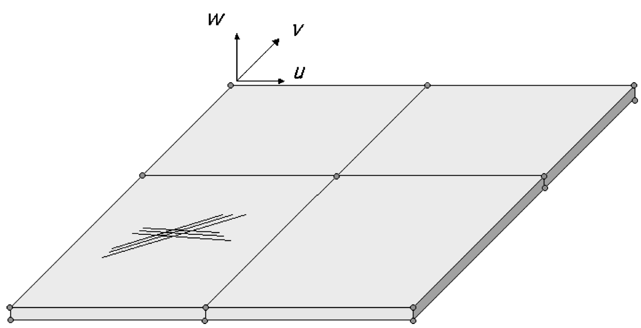


Fig. 4 Single-layered orthotropic brick model: sample mesh with the degrees of freedom per node.

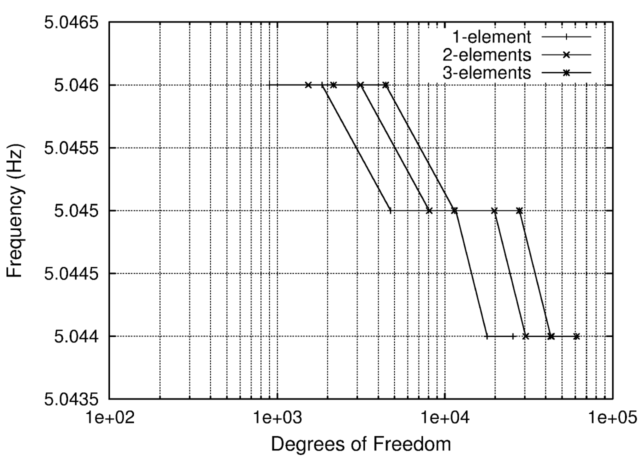


Fig. 5 Sensitivity analysis to numbers of elements along the thickness; first frequency vs degrees of freedom.

element models 2 and 3. This just confirms the ability of the solids to behave as thin-plate elements.

When the facing is meshed with solids, the number of elements along the thickness must be specified. The small thickness of the facing should infer that the results do not depend on the number of elements. For this reason, a sensitivity analysis has been done. Figures 5 and 6 show the trend of the first two frequencies vs the number of degrees of freedom for one, two, and three elements along the thickness. As can be seen, the solution does not depend at all on this number; thus, in the following calculation only one element will be put along the thickness of the panel.

In addition, p formulation allows the choice of the polynomial order of the shape functions along the three edges of a solid element. Because of the small thickness of the plate, the solution is not sensitive to the choice of the polynomial order along the thickness provided that it is high enough to prevent locking of the element. Thus, a sensitivity analysis on the choice of the order of the polynomials along the element thickness has been carried out. Figures 7 and 8 show the trend of the first four frequencies vs the number of

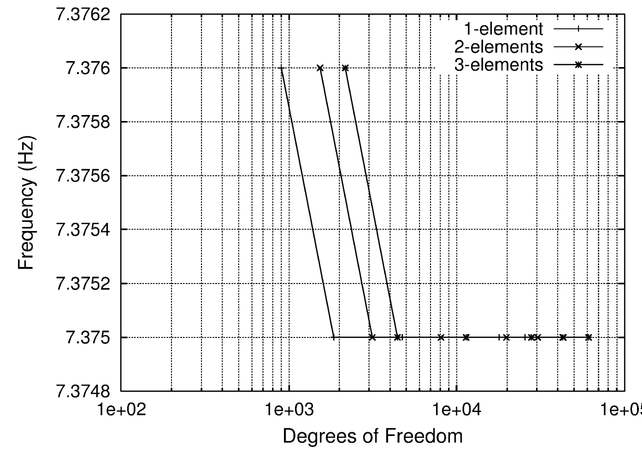


Fig. 6 Sensitivity analysis to numbers of elements along the thickness; second frequency vs degrees of freedom.

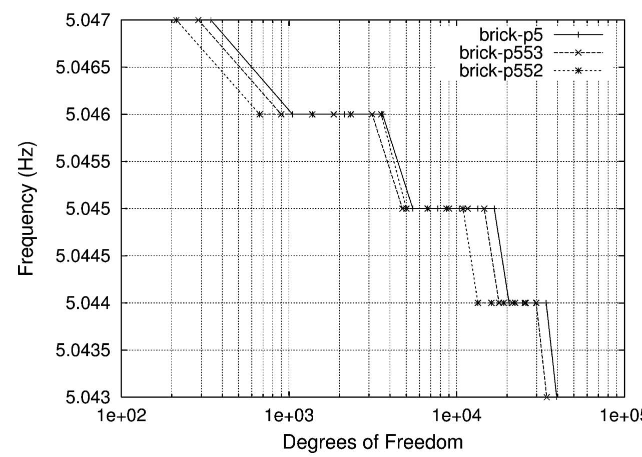


Fig. 7 Sensitivity analysis on polynomial expansion order along the thickness; first frequency of unconstrained facing.

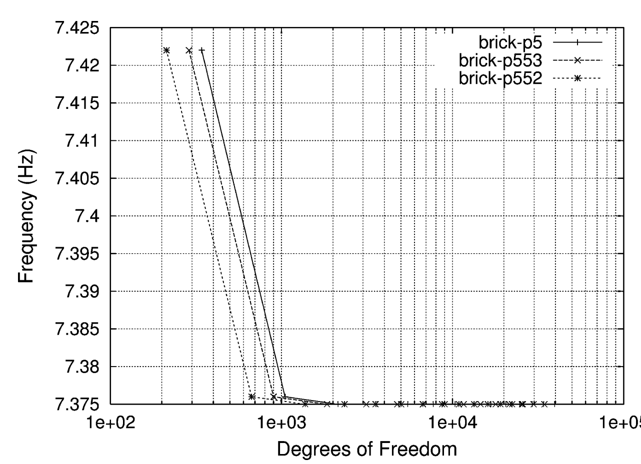


Fig. 8 Sensitivity analysis on polynomial expansion order along the thickness; second frequency of unconstrained facing.

degrees of freedom of the system, for single-layered solid model, considering three different choices for the shape functions order: fifth order for the two in plane directions and fifth, third, and second order along the thickness, labeled respectively brick-p5, brick-p553, and brick-p552. Figures 7 and 8 show that when the order of the polynomials along the thickness is lowered, the solution is not affected at all, that is, the curves are shifted on the left-hand side, which indicates reduction of the size of the system, thus, a reduction in the computational time. Following these results, the calculations using solid p elements have been done with one element and second-order polynomials along the thickness.

B. Unconstrained Flat Sandwich Panel

The technique for building equivalent models of laminates is finally tested on the flat sandwich reported in Ref. 11. As done for the facing analysis, three different models have been considered. All of them have the core modeled using solid eight-node finite elements (CHEXA) and the facings modeled in three different ways: Model 1 (Fig. 2), considered as reference, is made of four-node linear elements (CQUAD4) the elastic properties of which are specified assigning each individual ply and its orientation through a PCOMP card in NASTRAN. Models 2 and 3 (Figs. 3 and 4) are made of four-node linear plate (CQUAD4) and eight-node brick p elements (CHEXA, p formulation), respectively, whose elastic properties are determined using the equivalencing method introduced in Sec. II.

Model 3 is, thus, entirely made of solid finite elements, but the number of the elements along the thickness is kept equal to 1, and the polynomial order of the shape functions is kept equal to 2, eliminating any dependence of the model behavior on the thickness. For the p elements, the choice of the polynomial order is free. We decided to select three different orders for the in-plane edges, in particular, third, fifth and seventh, resulting in three full solid models.

A convergence analysis of the modal frequencies of the sandwich panel has been performed for each of the models considered, by increasing the number of elements along the in-plane edges. Mode shapes of the first four modes for model 3 are shown in Fig. 9. (They are the same for the other models.)

Table 4 reports the converged values for the different models, where the shell laminated refers to model 1, shell equivalent refers to model 2, and brick-p332, brick-p552, and brick-p772 refer to the third-order, fifth-order, and seventh-order choices for model 3.

As can be seen, all of the models give the same value (to the third digit): The differences among the reference model and the several equivalent models have disappeared. This confirms the validity of the equivalencing method introduced earlier at a wing-box level: It is sufficient to match only the membrane and shear behavior of the full model.

The trend of the models toward the converged values is presented in Figs. 10–13. The number of degrees of freedom are reported on the x axis and the calculated frequencies on the y axis. Figure 14 shows for each model (solid or plate) the number of finite elements used and the corresponding number of degrees of freedom.

Figures 11–13 show that the plate models have an oscillating behavior such that a converged value is not reached even with very fine meshes. It would be meaningless to put the relative error to the converged solution on the y axis, and so the frequency value is reported. (The last runs for models 1 and 2 are done using 150 elements in the in-plane edges.)

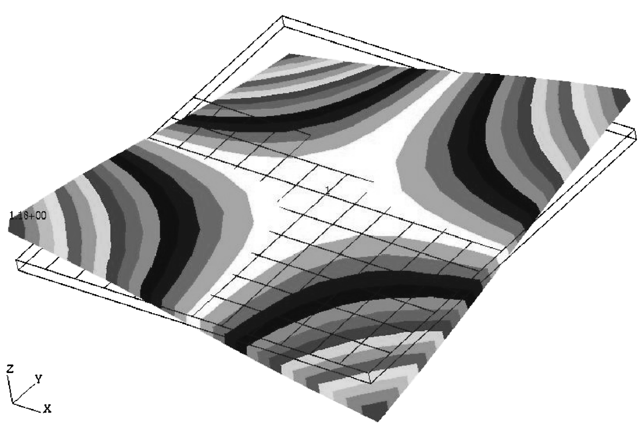
From the results obtained, the following observations can be made:

1) The solid models attain their converged value faster than the shell models, especially for solids using fifth- and seventh-order polynomial orders. Specifically, when the seventh-order polynomial expansion is used, a converged result can be obtained with one or two solid elements.

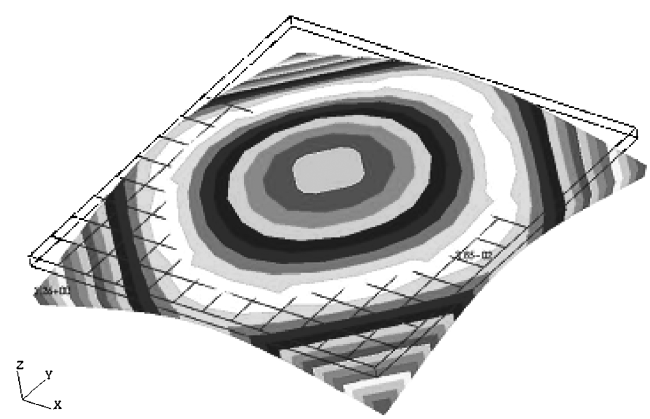
2) The solid p formulation always tends monotonically to a converged solution, whereas the plate formulation does not. This can

Table 4 Converged values of modal frequencies for different models of unconstrained flat sandwich panel

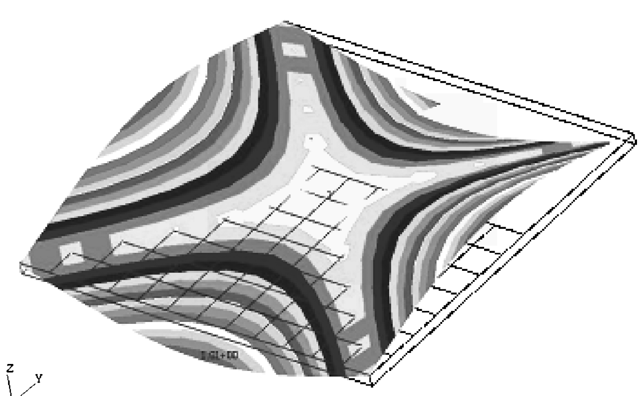
Mode	Shell laminated, Hz	Shell equivalent, Hz	brick-p332, Hz	brick-p552, Hz	brick-p772, Hz
First	153.14	153.16	153.19	153.19	153.19
Second	224.67	224.70	224.71	224.71	224.71
Third	285.32	285.14	285.16	285.16	285.16
Fourth	388.64	388.64	388.62	388.62	388.62



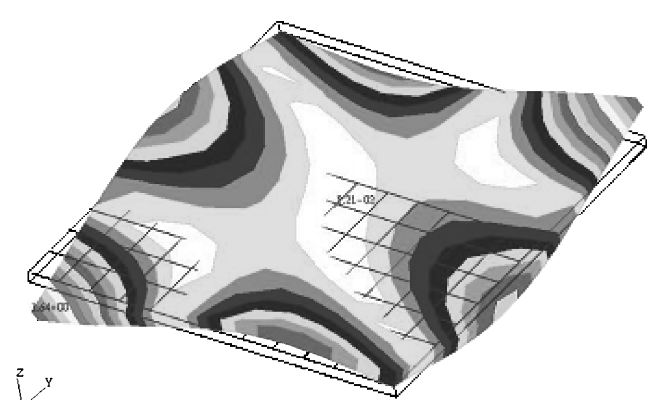
a) First mode



c) Third mode



b) Second mode



d) Fourth mode

Fig. 9 First four vibration mode shapes of the sandwich plate.

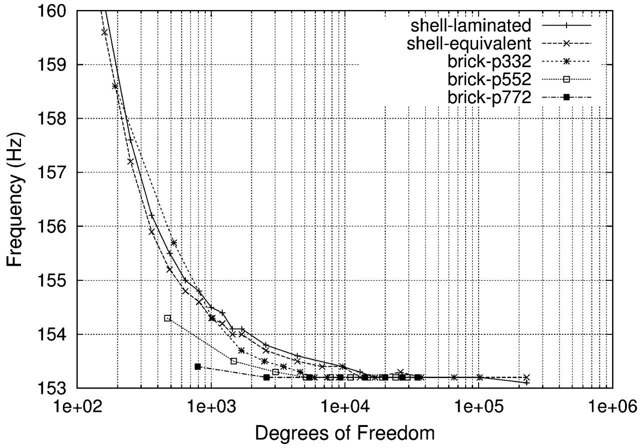


Fig. 10 Convergence rate of shell and solid models of sandwich panel; first frequency vs degrees of freedom.

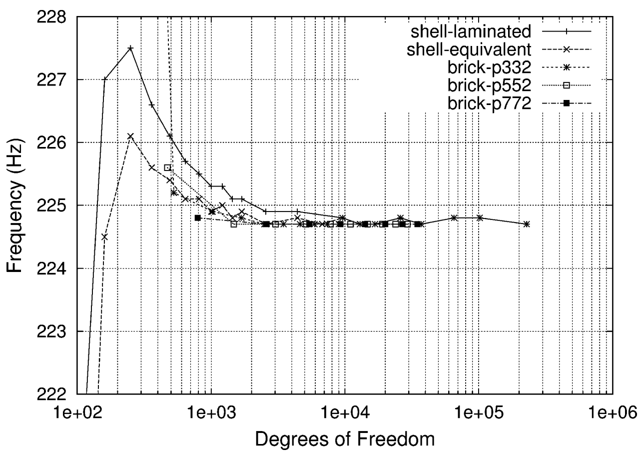


Fig. 11 Convergence rate of shell and solid models of sandwich panel; second frequency vs degrees of freedom.

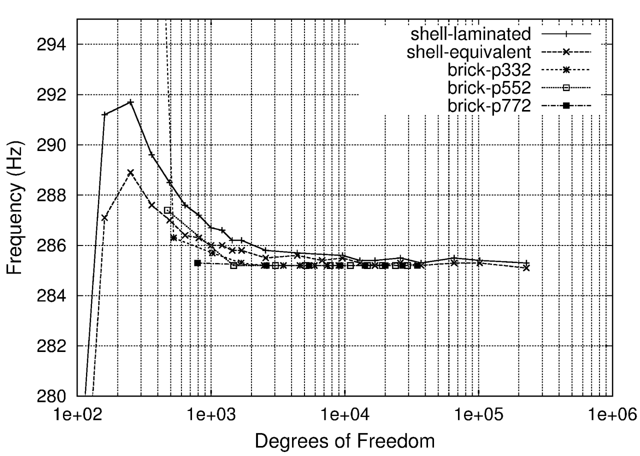


Fig. 12 Convergence rate of shell and solid models of sandwich panel; third frequency vs degrees of freedom.

be useful for the definition of the converged values through some extrapolation technique such as Richardson's (see Ref. 13) when a standard convergence analysis is performed, that is, increasing the number of elements.

C. Discussion

Differently from the others, the p-formulation solid elements give the possibility to achieve convergence just by the increasing of the order the polynomials without modifying the existing mesh.

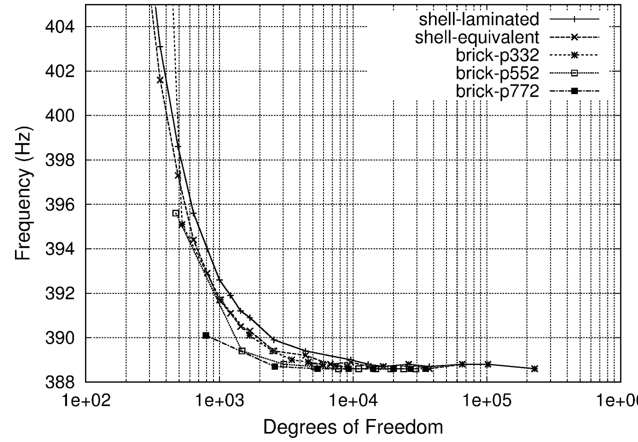


Fig. 13 Convergence rate of shell and solid models of sandwich panel; fourth frequency vs degrees of freedom.

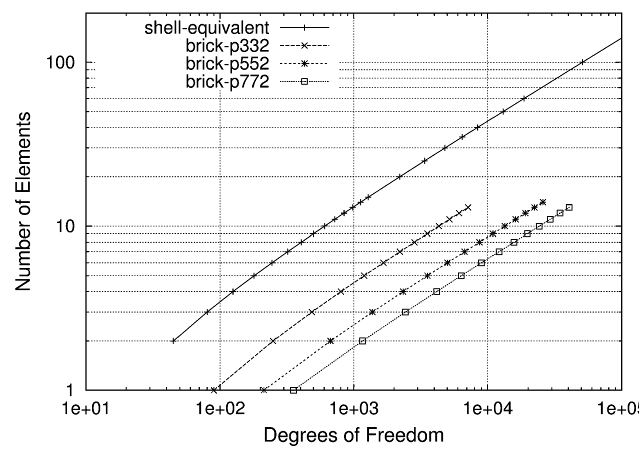


Fig. 14 Number of elements vs degrees of freedom for different models.

In Figs. 10–13, it is sufficient to connect the points corresponding to a fixed number of elements for each of the different order's curves. It can be seen that, using two elements, convergence can be achieved starting from the third order and stopping at the seventh. As stated before, this is a quite remarkable feature because it makes a convergence check feasible for complex structures, without the need of regenerating the mesh, an operation done mostly by hand.

Note that, with the same number of degrees of freedom, the computational time of a solid model is longer than a plate model. The time is as longer as the order of the polynomials used increases because the mass and the stiffness matrices are denser than those for the plate model.

This problem could be overcome using reduction techniques, such as the quasi-static reduction of Guyan.[14] Guyan reduction defines a master and a slave group of degrees of freedom, splitting the analysis in two dependent problems, reducing the size of the problem to be solved. In the specific NASTRAN implementation of the higher-order solid elements, in which the increase in the order of the polynomials is associated with the addition of internal nodes between the corner nodes, Guyan reduction has no beneficial effect because it is not possible to define a master class of nodes and a slave group, inasmuch as all of them have the same importance. Other formulations, such as the one from Morino et al.,[4] which uses the derivatives of the displacement field at the nodes to increase the order of the polynomials used, are particularly suited to the application of Guyan reduction because the derivatives are associated with higher-order modes; thus, considering them as slave degrees of freedom does not affect the quality of the solution.

As an addendum, note that all of the analyses have been carried using a consistent mass matrix model. Linear finite elements

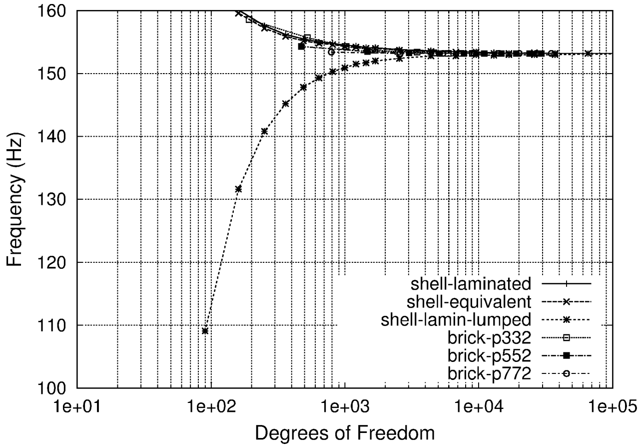


Fig. 15 First frequency of sandwich panel, comparison between lumped and consistent mass matrix formulation.

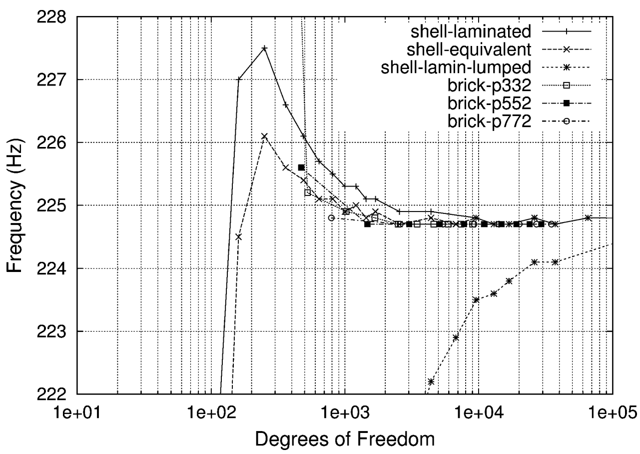


Fig. 16 Second frequency of the sandwich panel, comparison between lumped and consistent mass matrix formulation.

are available in the lumped mass matrix formulation, and in industrial practice they are used in this form to take into account the nonstructural masses. NASTRAN solid p elements allow only the formulation with consistent mass matrix. To the authors' knowledge, at present it is unknown whether it is possible to include the nonstructural masses in this formulation, but for the sake of clarity Figs. 15 and 16 show the same analysis done earlier including the plate model created using a lumped mass matrix. As can be expected, the lumped mass matrix model is much slower than the consistent one, enforcing the conclusion drawn earlier.

IV. Conclusions

The solid p elements can be used to model thin-walled structural elements, made of composite materials. Equivalent models based on the use of a single-layered, homogeneus, and orthotropic material matching of the extensional and shear stiffness of the structural element to be modeled can be defined. The equivalent models give the same results as the full multilayered models. Equivalent models made of solid p elements give much accurate results using fewer degrees of freedom than models made of traditional linear shell elements.

The capability of solid p elements in representing the behavior of orthotropic sandwich panels can be exploited in the modeling of stiffened panels. In fact, such a panel can be reduced to a sandwich, defining a suitable reduction technique that spreads the additional stiffness introduced by the stiffners over the entire panel considered as orthotropic. This way the mesh (re-)generation is again handled in a very flexible way, and any detail can be easily taken into account in the structural analysis.

References

[1]Babuska, I., Szabo, B. A., and Katz, I. N., "The P-version of the Finite Element Method," *SIAM Journal on Numerical Analysis*, Vol. 18, No. 3, 1981, pp. 515–545.

[2]Szabo, B., and Sahrmann, G., "Hierarchic Plate and Shell Models Based on P-Extension," *International Journal for Numerical Methods in Engineering*, Vol. 26, No. 8, 1988, pp. 1855–1881.

[3]Hashagen, F., and de Borst, R., "Numerical Assessment of Failure Mechanisms in Fibre Metal Laminates," *HERON*, Vol. 43, No. 2, 1998, pp. 141–159.

[4]Morino, L., Mastroddi, F., Bernardini, G., and Piccirilli, M., "A Hermite Interpolation Finite Element for Structural Dynamics," *XVI Congresso Nazionale AIDAA*, Rome, 2001.

[5]Van Tooren, M. J. L., Krakers, L. A., La Rocca, G., and Beukers, A., "Design and Technology in Aerospace, Flying High," *Onderzoek Integraal Ontwerpen Architectuur & Techniek, Verliefdheid of Hartstocht*, Delft, The Netherlands, April 2002, pp. 75–109.

[6]La Rocca, G., Krakers, L., and van Tooren, M. J. L., "Development of an ICAD Generative Model for Aircraft Design, Analysis and Optimisation," *International ICAD User Group Conference*, 2002 [online database], URL: http://www.iiug.com/presentations-2002/tudelft-iiug-paper.pdf [cited Jan. 2003].

[7]La Rocca, G., Krakers, L. A., and van Tooren, M. J. L., "Development of an ICAD Generative Model for Blended Wing Body Aircraft Design," *Proceedings of the 9th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization*, 2002; also AIAA Paper 2002-5447, Sept. 2002.

[8]Lisandrin, P., and van Tooren, M. J. L., "Generic Volume Element Meshing for Optimisation Applications," *Proceedings of the 9th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization*, 2002; also AIAA Paper 2002-5647, Sept. 2002.

[9]*MSC/NASTRAN Encyclopedia*, Ver. 70.5, MacNeal–Schwendler Corp., Santa Ana, CA, 1998.

[10]Jones, R. M., *Mechanics of Composite Materials*, 2nd ed., Taylor and Francis, Philadelphia, 1999, Chaps. 2, 4.

[11]Barisciano, L. P., "Broadband Transmission Loss due to Reverberant Excitation," NASA CR-1999-209687, Nov. 1999.

[12]Zenkert, D., *The Handbook of Sandwich Construction*, Engineering Materials Advisory Service, Ltd., Cradley Heath, England, U.K., 1997, Chap. 2.

[13]Press W. H., Teukolsky S. A., Vetterling W. T., and Flannery B. P., *Numerical Recipes* [online books], URL: http://www.nr.com/ [cited May 2003].

[14]Guyan, R. J., "Reduction of Stiffness and Mass Matrices," *AIAA Journal*, Vol. 3, No. 2, 1965, p. 380.