Advanced Composite Materials
Publication details, including instructions for authors and subscription information:
http://www.tandfonline.com/loi/tacm20

Modeling and vibration analysis of a composite supporter for aerospace applications

Shaohui Zhang & Hualing Chen

Version of record first published: 02 Apr 2012.

Adv. Composite Mater., Vol. 14, No. 2, pp. 199–210 (2005)

Modeling and vibration analysis of a composite supporter for aerospace applications

SHAOHUI ZHANG * and HUALING CHEN

Institute of Vibration and Noise Control, School of Mechanical Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, People's Republic of China

Received 15 April 2004; accepted 10 November 2004

Abstract—In this paper, modeling and vibration analysis of a composite supporter newly developed for aerospace applications are presented. The composite supporter is manufactured through hand lay-up process using the combination of carbon fiber reinforced plastics and Nomex honeycombs. The dynamic behaviour of the composite supporter is investigated, both numerically and experimentally, the former using a commercially available finite element package. Good agreement between the numerical results and experimental results shows that the proposed finite element modeling and analysis procedure can be used effectively to characterize the vibration behaviour of the composite supporter. The parametric investigation is also carried out using the validated FE model to show the effects of changing orthotropic properties of the core and ply orientation on the natural frequencies of the supporter. The presented results will provide a theoretical basis for a further vibration-control design of the supporter.

Keywords: Composite; sandwich structure; vibration; finite element analysis; modal testing.

1. INTRODUCTION

Advanced fiber reinforced composites have rapidly emerged as a major class of structural materials in a wide range of engineering fields due to their superior mechanical properties compared with traditional material, such as high specific strength, specific stiffness and good damping capacities [1–12]. The demand for using composites instead of metals to produce more efficient structures is still growing, with weight saving as one of the most important considerations.

A supporter is a very common kind of aerospace structure on which electrical instruments are mounted. Here a composite supporter has been developed as a replacement for a conventional metal design. The combination of carbon fiber reinforced plastics (CFRP) and Nomex honeycombs is employed to benefit from the

*To whom correspondence should be addressed. E-mail: zsh@mailst.xjtu.edu.cn (Shaohui Zhang).

substantial mass savings and the high structural integrity. The sensitive instruments must withstand the combination of all severe dynamic environments during the period of launch and flight, so the dynamic performance of the composite supporter is one of the main points to be considered for the instruments to work properly. The main objective of this work is to present a numerical and experimental procedure to get a better understanding of the dynamic behaviour of the composite supporter newly developed. The numerical investigation has been carried out using a commercially available finite element package ANSYS 7.0. The numerical predictions for modal frequencies and mode shapes are verified with the results from experimental modal analysis up to a frequency of 2000 Hz. Then the validated FE model is used to investigate the influence of changing orthotropic properties of the core and ply orientation on the natural frequencies of the supporter. The presented results will provide the theoretical basis for a further vibration-control design of the composite supporter.

2. CONFIGURATION OF THE COMPOSITE SUPPORTER

Through the hand lay-up process and autoclave cure process, the composite supporter is produced using the combination of carbon fibre reinforced plastics and Nomex honeycombs. Figure 1 presents a photograph of the composite supporter.

Here, the upper panel of the supporter in Fig. 1 is constructed from a honeycomb core sandwich between carbon fiber reinforced composite laminate face sheets. The face sheets are made using QY8911 epoxy resin pre-impregnated unidirectional T300 carbon fiber tape and bonded to the Nomex honeycomb core named NRH-2-80. Typical material properties for the CFRP ply and honeycomb are respectively presented in Tables 1 and 2. The sandwich lay-up is symmetric with eight carbon

Table 1.
Material specifications for the T300/QY8911 CFRP ply

Thickness	Elastic modulus (Pa)		Shear modulus (Pa)	Density	Poisson's ratio
(mm)	E_{11}	E_{22}	G_{12}	(kg/m^3)	ν_{12}
0.125	1.35×10^{11}	8.80×10^9	4.47×10^9	1614	0.33

Table 2.
Material specifications for the NRH-2.0-80 honeycomb

Elastic modulus (Pa)	Shear modulus (Pa)		Density
E_{33}	G_{xz}	G_{yz}	(kg/m^3)
3.02×10^8	8.53×10^7	4.46×10^7	80

fiber reinforced plastic layers $[45°/0°/-45°/90°]_s$ on either side of the core. The outer and inner face sheet lay-ups are extended downwards and joined together to form the skirt and the base of the supporter. Ten 'T-shaped' meridional stringers are bonded to the inner surface of the skirt of the supporter using an adhesive film named J116. Then a co-curing process is performed at 130°C for 2.5 hours with a pressure of 0.4 MPa. The stringer flange and web lay-up is $[45°/0°/-45°/90°/45°/0°/-45°/90°]_s$. Because of practical requirements, two circular cutouts with the radius $r = 25$ mm are made symmetrically in the skirt of the supporter and one circular cutout with the radius $r = 15$ mm is made in the upper panel of the supporter. The typical construction dimensions of the supporter are shown in Fig. 2. The cross-sectional view of the transition corner where the outer and inner face sheet lay-ups are put together is shown in Fig. 3. The geometrical characteristics of stringers are shown in Fig. 4.

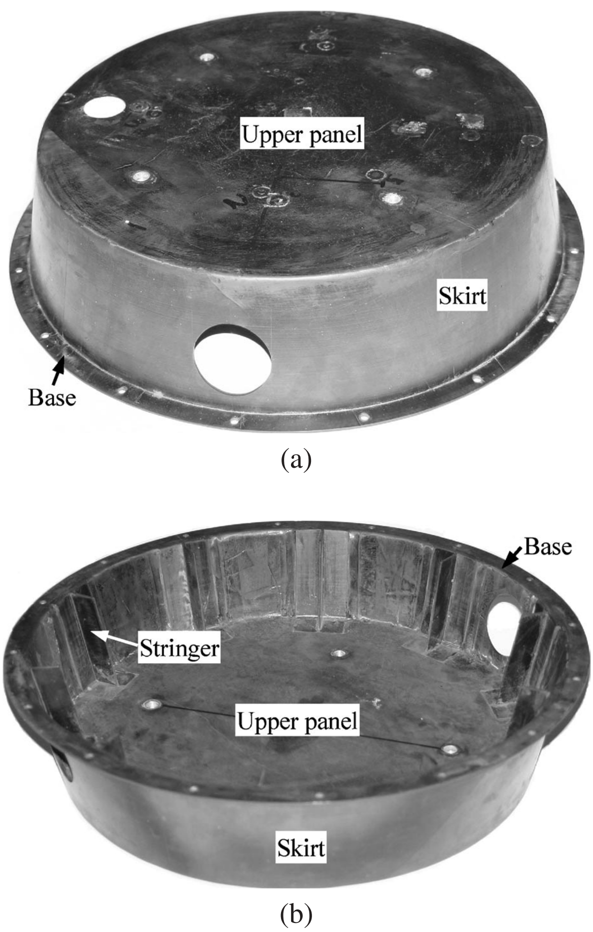

Figure 1. Photograph of the developed composite equipment supporter: (a) Exterior view; (b) Interior view.

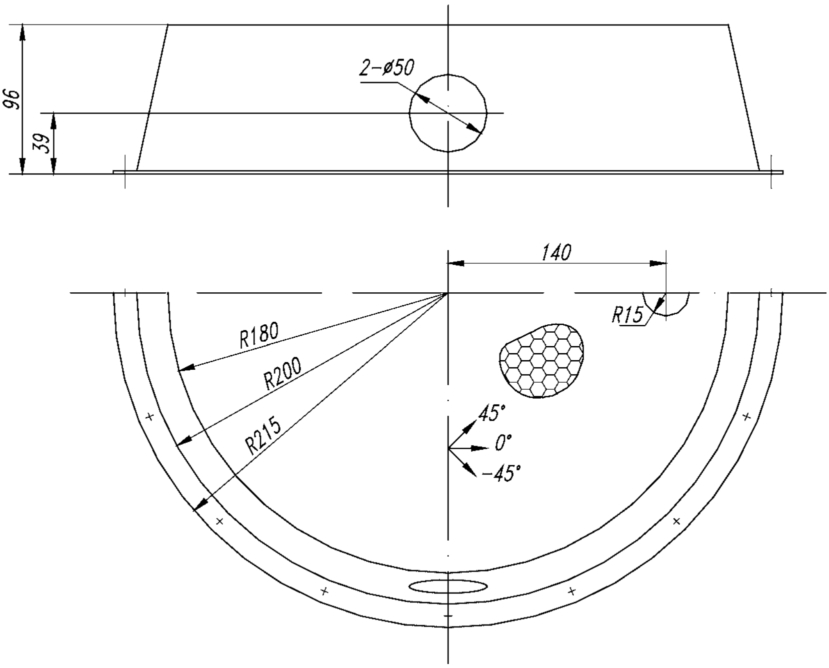

Figure 2. Dimensions of the composite supporter configuration.

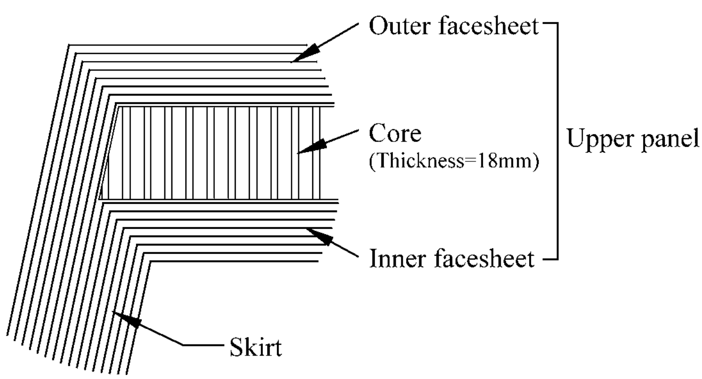

Figure 3. Cross-sectional view of the transition corner.

3. DEVELOPMENT OF THE FE MODEL

The ANSYS 7.0 finite element package, which provides the ability to model composite materials by using specialized elements called layered elements, is used to model and analyze the composite supporter developed here. Because of the geometry of the supporter and the complex lay-up details it is necessary to keep the FE model as simple as possible in order to minimize the time needed for modeling and analysis. The element SHELL91 that offers the choice of a 'sandwich option' is selected to develop a finite element model for the composite supporter. The

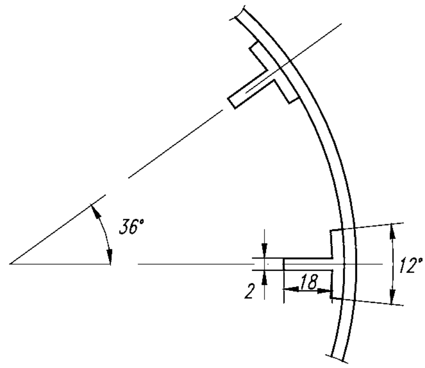

Figure 4. Stringer configuration.

middle layer of SHELL91 is assumed to be thick (greater than $5/6$ of the total thickness) and is assumed to carry all of the transverse shear when the sandwich option is turned on. The element presents six degrees of freedom at each node: three translations in the nodal x, y and z directions and three rotations about the nodal x, y and z axes. The node offset option of SHELL91 can locate the element nodes at the bottom, middle or top surface of the shell. For each SHELL91 element the real constant table is specified constituting the material property, ply orientation angle and layer thickness.

The FE model is developed based on the physical dimensions and material properties of the structure as defined in Tables 1 and 2. The solid model is built by constructing areas in the global cylindrical coordinate system with the origin positioned at the center of the base area of the supporter and the Z-direction is chosen along the centerline of the supporter. The upper panel of the supporter is meshed by using SHELL91 elements with sandwich option turned on and the remainder of the supporter is meshed by using the same element type with sandwich option turned off.

The elements of the lower flanges of stringers have the same nodes for the corresponding elements of the skin. The position of nodes reference plane is the interface plane between the skin and the lower stringer flange as shown in Fig. 5. The nodes of the elements that mesh the lower flanges are offset to bottom surface and the nodes of the elements that mesh the skin are offset to top surface. The way in which the stringer web has been joined to the stringer lower flange is also shown in Fig. 5 and the shell elements for a stringer to be displayed as solids with the layer thicknesses obtained from real constants are shown in Fig. 6. The whole finite element model of the composite supporter is presented in Fig. 7.

4. EXPERIMENTAL PROCEDURES

An experimental modal analysis is conducted to validate the finite element model. Through an impulse test, the frequency response functions are determined to allow

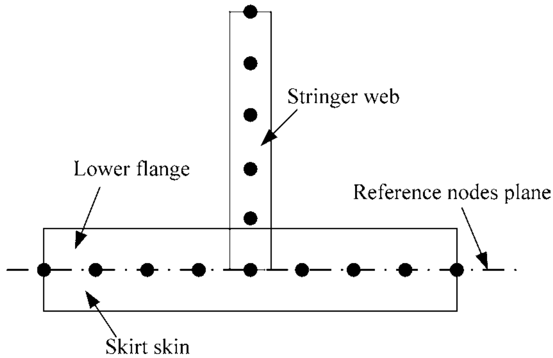

Figure 5. Reference nodes plane for ANSYS model.

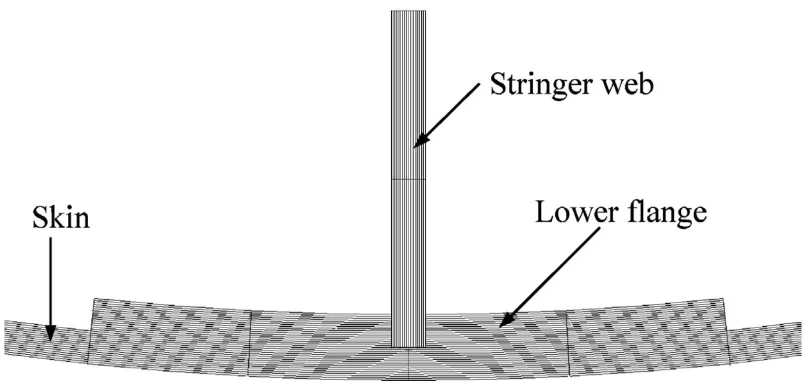

Figure 6. Detailed view of the meshed stringer.

for the determination of the natural frequencies and the modal damping ratios of the composite supporter. Since the supporter is very light, special care is taken in choosing the accelerometer to avoid undesirable influences. In the practical modal test, the specimens need to be supported in a way that will not degrade the information about the modal properties, so special care must also be taken with the boundary conditions of the supporter. Considering the practical method of installation, the supporter is clamped using joints fixed to a large block which is sufficiently massive and rigid. Figure 8 shows a schematic of the experimental apparatus used. The base of the supporter is fully fixed rigidly to the block (1). The impact hammer (3) is used to give the input load to the supporter and the response output is captured by the accelerometer (2). Both measured signals are amplified by the power amplifier (4) and then transferred into a personal computer (6) via the multi-channel data acquisition system (5). The excitation position is moved from point to point and the accelerometer remains fixed. An exponential window is used on output response and a force window on input excitation. Four measurements are recorded per position and the acceleration frequency response function is obtained and averaged over the four measurements in order to reduce the statistical variance.

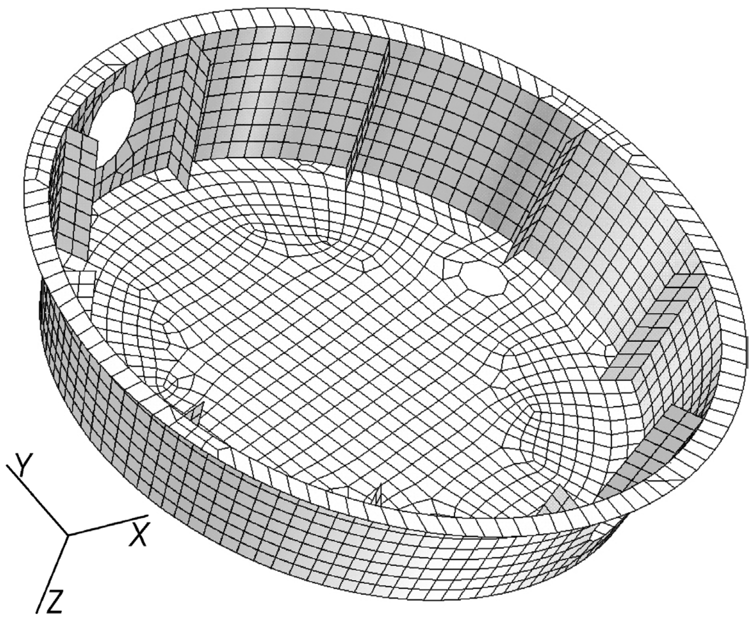

Figure 7. Finite element model of the composite supporter.

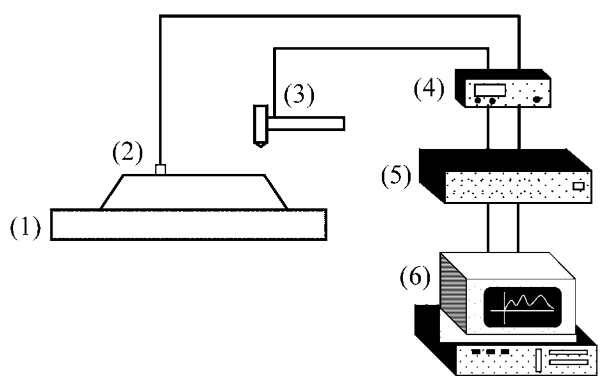

Figure 8. The scheme of experimental modal analysis.

After the measurement of the frequency response functions, the modal parameters are evaluated through the software DASP.

5. FE PREDICTIONS VERSUS EXPERIMENTAL RESULTS

Eigenvalue analysis of the composite supporter with the base fully clamped is performed using the Lanczos method in ANSYS 7.0. The adopted finite element mesh is a compromise between computational expense and accuracy. The model had 35828 active degrees of freedom with 2486 elements, and the CPU time to obtain the first four modes with frequencies up to 2000 Hz is approximately 360 s

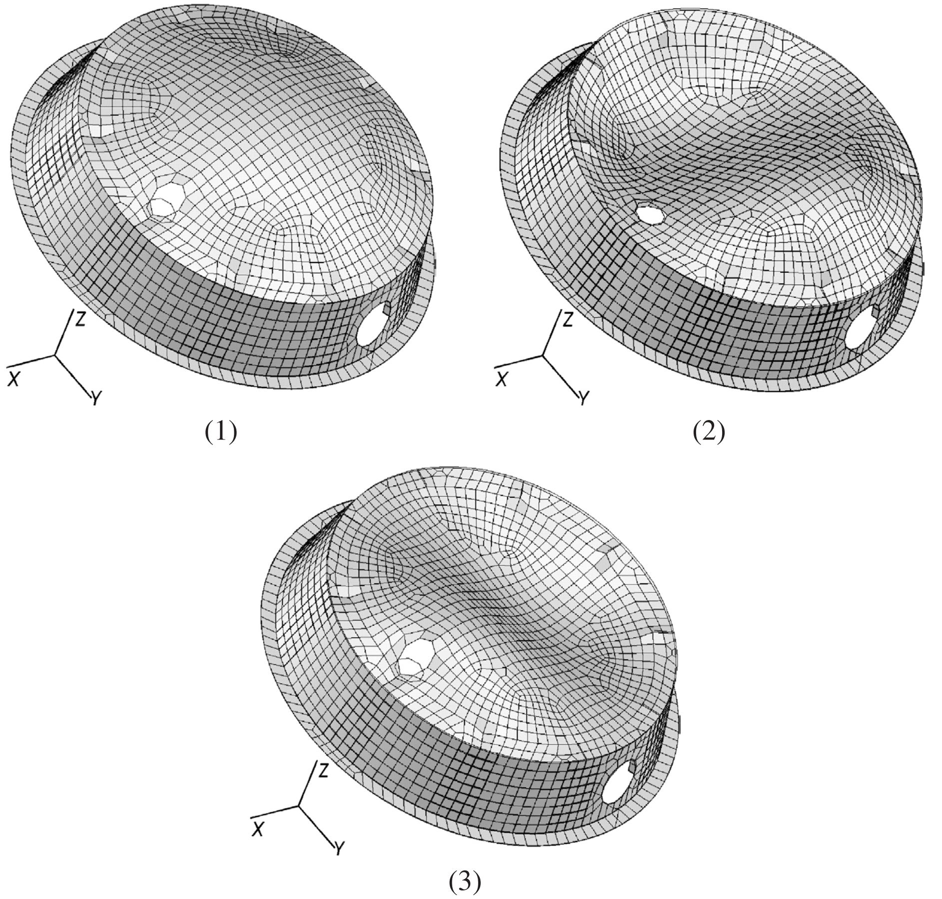

Figure 9. The first three predicted mode shapes.

on a Pentium III 850 MHz computer. The modal data are then analyzed and post-processed in the general postprocessor of ANSYS 7.0.

The comparisons between the frequencies for the first four modes predicted by the finite element calculations and the frequencies extracted from the experimental modal analysis are summarized in Table 3 and result in a good agreement. The largest relative error is 10.4%, and for the remaining modes, the errors are less than 8%. As an experimental result, modal damping ratios calculated using the half power bandwidth method are also presented in the last column of Table 3. The first three numerically predicted and experimental mode shapes for the composite supporter are respectively shown in Figs 9 and 10. It is found that the predicted ones are very similar to those measured. Thus, it can be concluded that the proposed numerical analysis procedure can be used effectively to characterize the vibration behaviour of the developed composite supporter.

With respect to the deviations of the numeric results in relation to the experimental ones, some errors may arise from the simplification of the theoretic model by using

Table 3.
Comparison of predicted and measured natural frequencies of the composite supporter

Mode	FEA f_{FEA} (Hz)	Measured f_{exp} (Hz)	Error (%)	Modal damping measured (%)
1st	818	741	10.4%	0.88
2nd	1409	1309	7.6%	1.08
3rd	1589	1528	4.0%	0.48
4th	1992	1870	6.5%	1.00

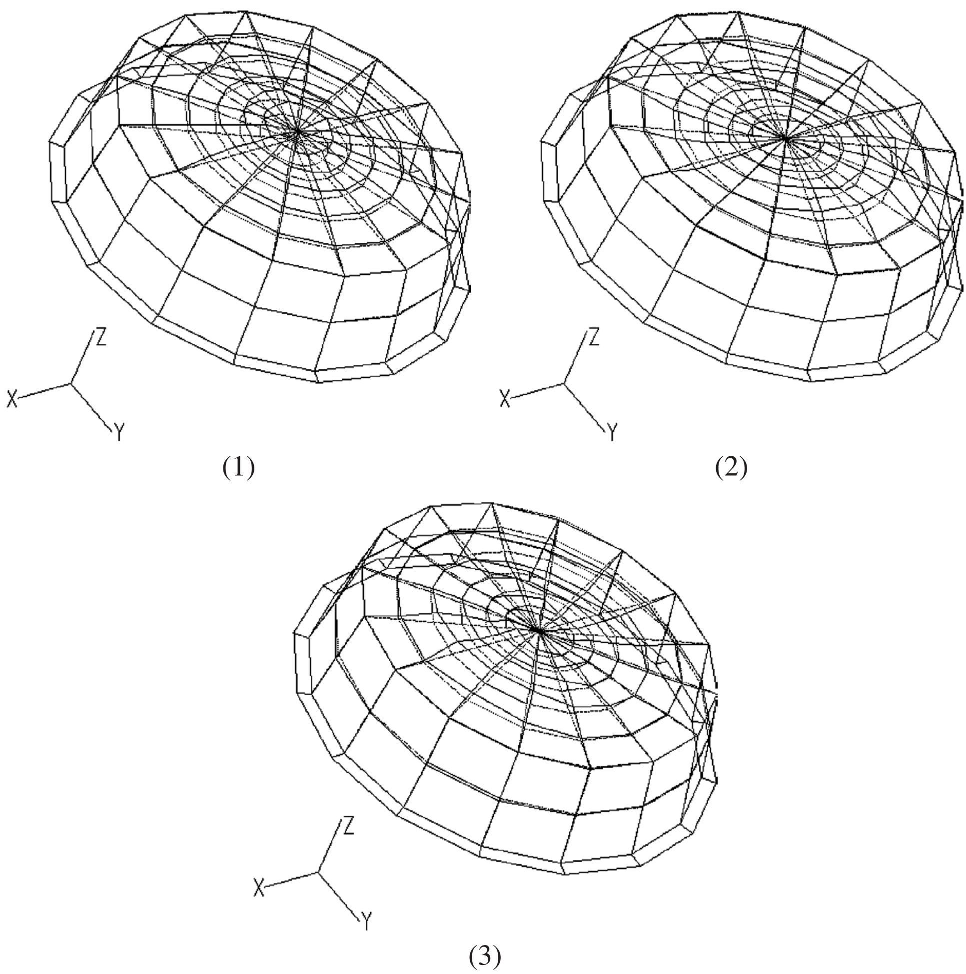

Figure 10. The first three measured mode shapes.

shell elements and measurement errors. Another aspect to be considered is the non-uniformity in the supporter materials, such as bubbles, variations in thickness and ply angle error. Such factors are not taken into account during the numeric analysis,

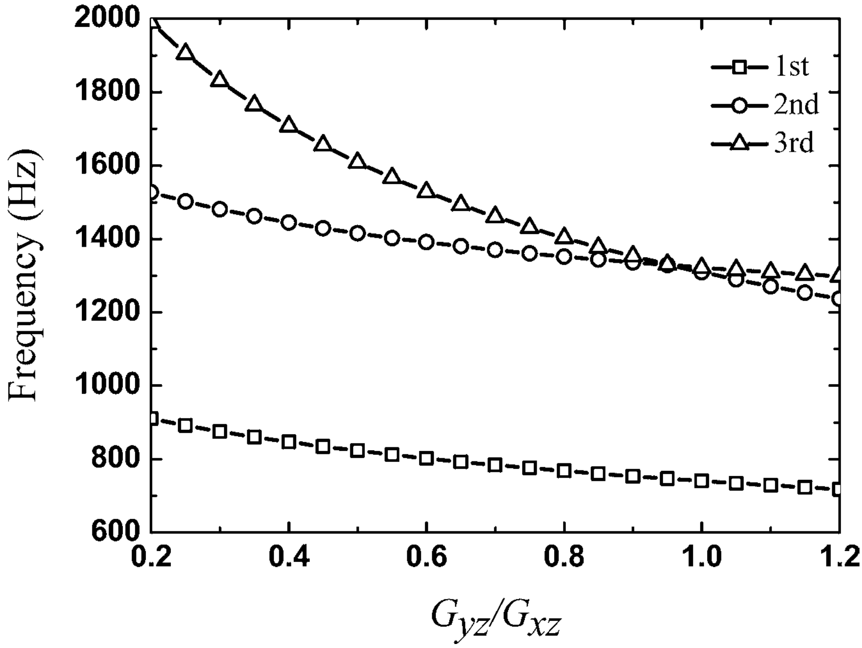

Figure 11. Effect of changing core orthotropy on the first three natural frequencies.

since the finite element model considers that the supporter has entirely perfect and homogeneous properties.

6. PARAMETER STUDIES USING THE FE MODEL

6.1. The effect of the orthotropy of the core on the first three natural frequencies

Here the orthotropy of the honeycomb core is represented by the ratio G_{yz}/G_{xz}, where G_{xz} is shear modulus of core in longitudinal direction and G_{yz} is shear modulus of core in the transverse direction. The effect of changing the orthotropy of the core on the natural frequencies of the supporter is investigated using the validated FE model. The ratio G_{yz}/G_{xz} is varied between 0.2 and 1.2 (the original ratio of $G_{yz}/G_{xz} = 0.52$) by keeping the value G_{yz} constant while varying G_{xz} between 3.72×10^7 and 22.3×10^7. All other lay-up details and material properties are kept constant. It is shown in Fig. 11 that all the first three natural frequencies decrease with increasing the orthotropic ratio, i.e. as G_{xz} is decreased. The rate of change of frequency with changing orthotropy is moderate except that the third frequency presents a sharper decrease with increasing orthotropy for the ratio values below approximately 0.8, so from the point of view of sensitivity as a whole, small changes in the core orthotropy about a certain value do not bring large changes in the natural frequencies.

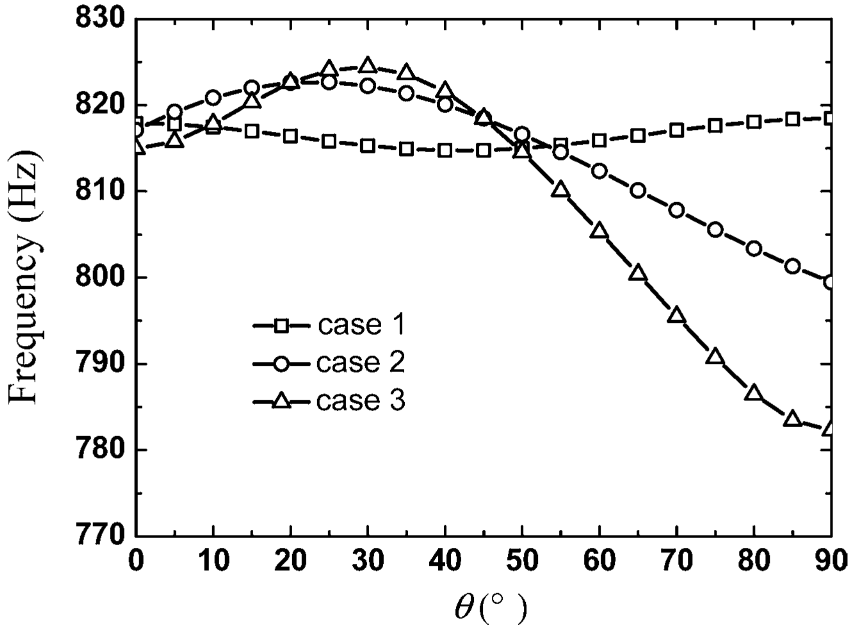

Figure 12. Effect of ply orientation on the fundamental frequency.

6.2. The effect of ply orientation on the fundamental frequency

Due to the large number of possible ply lay-ups of the face sheet for the upper panel of the composite supporter, to investigate the influence of the stacking sequence on the fundamental frequency the following three cases of ply stacking sequence for face sheet are investigated: Case 1: $[45°/0°/-45°/\theta°]_s$, changing orientation of the middle two layers of both face sheets; Case 2: $[\theta°/0°/-45°/90°]_s$, changing orientation of the external two layers of both face sheets; Case 3: $[\theta°/0°/-\theta°/90°]_s$, changing orientation of the external two and internal two layers of both face sheets. In all cases ply angle θ is varied between 0° and 90° in 5° steps. The results are plotted in Fig. 12. In case 1, the variation of frequency with ply orientation is almost negligible while in case 2 and 3 the situation is quite different. It can be concluded that the ply angle of external layers for face sheets has more remarkable effect on the fundamental frequency of the supporter than the middle layer ply angle. In detail, the fundamental frequency for cases 2 and 3 increases with ply angle between 0° and 30° and decreases with ply angle between 30° and 90°. As shown in Fig. 12, the largest change in frequency with ply angle occurs in case 3.

7. SUMMARY

This paper has given details of modeling and analysis of a composite supporter developed for aerospace applications using the combination of carbon fiber reinforced plastics and Nomex honeycombs. The finite element model of the composite supporter has been presented and validated using the experimental results. Good agreement between the numerical results and experimental results shows the proposed finite element modeling and analysis procedure can be used effectively to characterize the dynamic behaviour of the composite supporter. Finally, parameter studies have been carried out using the FE model of the composite supporter to investigate the effects of changing orthotropy of the core and ply orientation on the natural frequencies. The presented results will be useful for a further vibration-control design of the supporter.

Acknowledgements

The research work presented in this paper has been supported by the National Natural Science Foundation of China under Grant No. 10076012 and the Doctoral Science Foundation of Chinese Education Ministry under Grant No. 20010698011.

REFERENCES

1. J. Matsui, Polymer matrix composites (PMC) in aerospace, *Adv. Composite Mater.* **4**, 197–208 (1995).
2. Y. Kurihara, Polymer matrix composite materials in automobile industries, *Adv. Composite Mater*. **4**, 209–219 (1995).
3. Sh. Shibuta and M. Yoshikawa, Highly elastic carbon fiber reinforced plastic propulsion shaft, *Adv. Composite Mater*. **5**, 233–239 (1996).
4. H. G. S. J. Thuis and C. Biemans, Design, fabrication and testing of a composite bracket for aerospace applications, *Composite Struct.* **38**, 91–98 (1997).
5. K. F. Karlsson and B. T. Åströn, Manufacturing and applications of structural sandwich components. *Composite Pt. A-Appl. Sci. Manuf.* **28**, 97–111 (1997).
6. E. Sancaktar and M. Gratton, Design, analysis, and optimization of composite leaf springs for light vehicle applications, *Composite Struct.* **44**, 195–204 (1999).
7. P. R. Cunningham, R. G. White and G. S. Aglietti, The effects of various design parameters on the free vibration of doubly curved composite sandwich panels, *J. Sound Vibr.* **230**, 617–648 (2000).
8. Y. Morino, T. Shimoda, T. Morimoto, T. Ishikawa and T. Aoki, Applicability of CFRP materials to the cryogenic propellant tank for reusable launch vehicle (RLV), *Adv. Composite Mater*. **10**, 339–347 (2001).
9. A. P. Mouritz, E. Gellert, P. Burchill and K. Challis, Review of advanced composite structures for naval ships and submarines, *Composite Struct.* **53**, 21–41 (2001).
10. C. J. von Klemperer and V. E. Verijenko, Design, analysis and construction of a composite camera gimbal, *Composite Struct.* **54**, 379–388 (2001).
11. J. F. Davalos, P. Qiao, X. F. Xu, J. Robinson and K. E. Barth, Modeling and characterization of fiber-reinforced plastic honeycomb sandwich panels for highway bridge applications, *Composite Struct.* **52**, 441–452 (2001).
12. D. G. Lee, H. S. Kim, J. W. Kim and J. K. Kim, Design and manufacture of an automotive hybrid aluminum/composite drive shaft, *Composite Struct.* **63**, 87–99 (2004).