NUMERICAL SIMULATION AND DESIGN OF COMPLEX COMPOSITE STRUCTURES

Afonso LEITE, Instituto Superior de Engenharia de Lisboa, Lisbon, Portugal

Zuzana DIMITROVOVÁ, Instituto Superior de Engenharia de Lisboa, Lisbon, Portugal

João M. TRAVASSOS, Instituto Superior de Engenharia de Lisboa, Lisbon, Portugal

1 ABSTRACT

The aim of this paper is to present a detailed parametric study of complex fiber reinforced composite – metallic structure static behavior. Advantages versus disadvantages of two different finites element (FE) models used in numerical simulation performed by general purpose FE software ANSYS are discussed in detail. Numerical results are verified by experimental measurements using HBM/Spider, 8-600Hz. Detailed parametric study exploits APDL - ANSYS Parametric Design Language. Obtained results allow making important conclusions about optimization of the present composite-metallic structure with the aim to reduce the manufacturing cost and weight while keeping the required structural resistance.

2 INTRODUCTION

Fiber reinforced composite – metallic structure studied in this paper, which we are going to name as shell base, forms the supporting part of the impact experiment device. Therefore it must be capable to absorb and translate into the ground high impact pressure. Due to repeated character of the load, besides good impact resistance it has also to exhibit high level of fatigue resistance. In addition, shell base should be sufficiently light, as it might be used in locations with poor vehicle access.

The impact experiment device mentioned above has capacity to apply impact load not only with high precision and long trajectory; but moreover, impact load application is flexile in terms of energetic levels, impulse duration and directions of application. The device can also be used to combat forest fires or to other applications of mechanical engineering. The study presented in this paper was initiated in [Travassos et al, 1994] and [Travassos, 1994].

Only static load is examined in this work, nevertheless obtained results already allow making important conclusions about optimization of the base shell with the aim to reduce the manufacturing cost and weight, while keeping the required structural resistance. However, the main objective of this study is to choose and establish requirements on finite element model for numerical simulation, giving results of high confidence for static load and consequently allowing to be used in dynamic/impact type of load, which is the principal load the structure is exposed to.

3 SHELL BASE BEHAVIOR AND ITS MAIN PARTS

A general view of the shell base is presented in Figure 1.

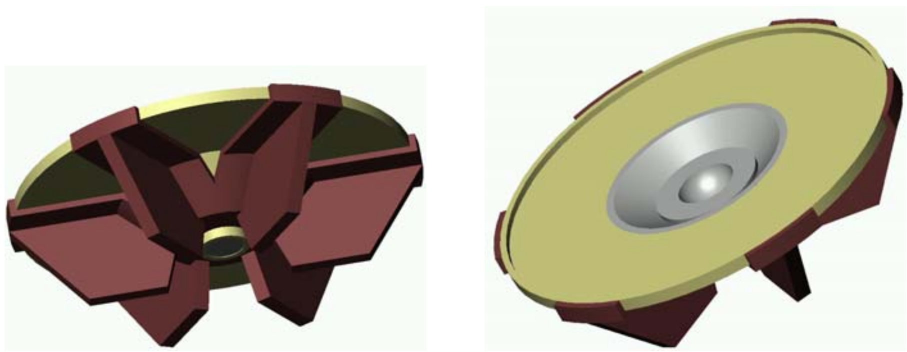

Figure 1. General view of the shell base.

It can be seen, that it is composed from four major parts:

- glass fiber reinforced plate-type shell,
- mainly carbon fiber reinforcing ribs,
- aluminium stable alveolus,
- high performance steel gyratory alveolus.

Each part has its own purpose. The only part, which enters directly in contact with the applied load, is the steel gyratory alveolus. Its aim is to fix in right position and direction the load, which the secondary structure is exposed to, and to reduce the pressure transmitted to the aluminium stable alveolus. Then the aluminium component serves for distribution as much as possible uniform to the largest composite part, the glass fiber reinforced plate-type shell. Glass fiber reinforcement is used due to its very good impact properties. The component aim is to partly absorb and partly translate into the ground the impact energy. Carbon fiber reinforcing ribs have very good fatigue resistance and their main purpose is to ensure stable positioning of the full equipment on the ground and to prevent instability failure of the plate-type shell.

Composite parts use epoxy resin with high performance. They are autoclaved together, even with the aluminium part, in order to ensure good adhesion among them. Carbon fiber reinforcing ribs in fact implement also glass fibers in its middle. In summary, the following reinforcements and layouts (specified in Tables 1-2) can be found in the composite components of the base shell.

The four major parts of the base shell with the simplified designation from Table 1 are presented in Figure 2 in separation, altogether in cut are shown in Figure 3; different colors correspond to distinct materials and local coordinate systems. Here some geometry details are omitted, as these models form base of one of the FE models, where material parts not influencing the numerical results may not be modeled.

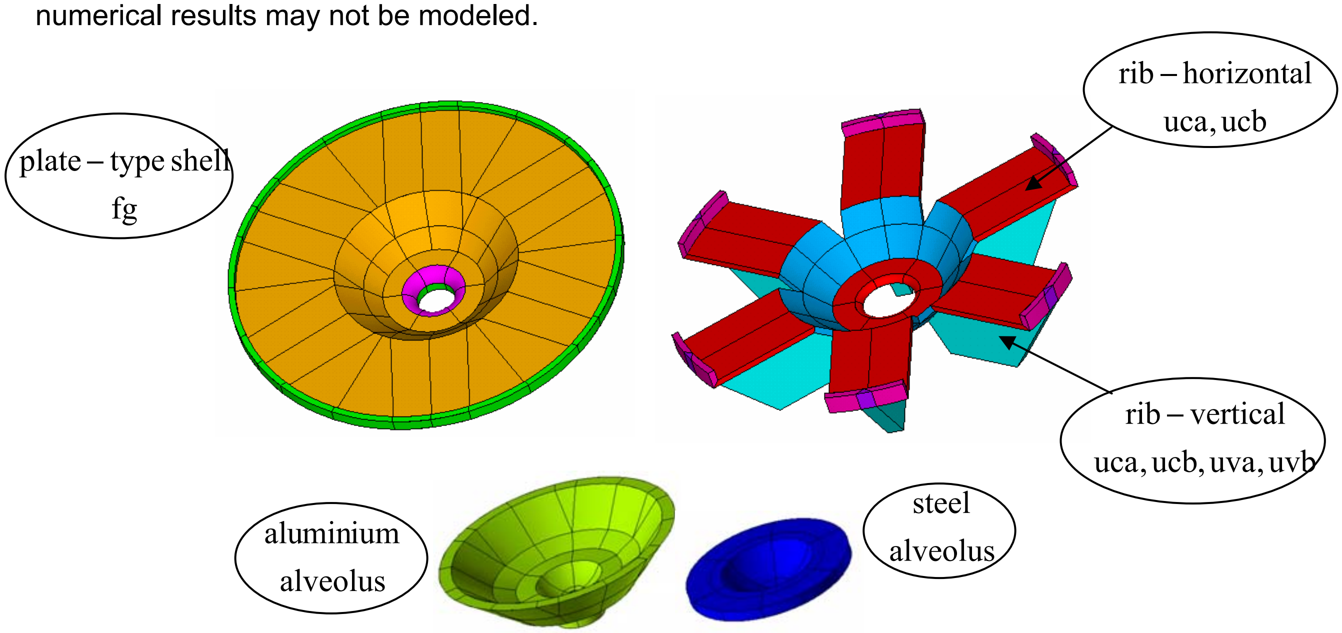

Figure 2. Principal components of the base shell.

Table 1. Fiber reinforcements used in composite components.

Simplified designation	Commercial designation	Type	Nominal thickness [mm]	Fiber volume fraction
fg	1581 ES 67 - 42%	Textile glass	0,195	58%
uva	VEE_2 20 R368	Unidirectional glass	0,130	60%
uvb	VEE_0 25 R368	Unidirectional glass	0,163	65%
uca	CTE_3 12 R368	Unidirectional carbon	0,112	57,5%
ucb	CTE_2 35 R367	Unidirectional carbon	0,327	60%

Table 2. Layouts used in composite components.

Component	Layout	Nominal thickness [mm]
Plate-type shell	$\left[\left(0_{fg}/45_{fg}\right)_5/0_{fg}\right]_S$	4,290
Rib - horizontal	$\left(90_{uca}/90_{ucb}\right)_{11}$	4,829
Rib - vertical	$\left[\left(90_{uca}/90_{ucb}\right)_{11}/\left(0_{uva}/0_{uvb}\right)_5\right]_S$	12,588

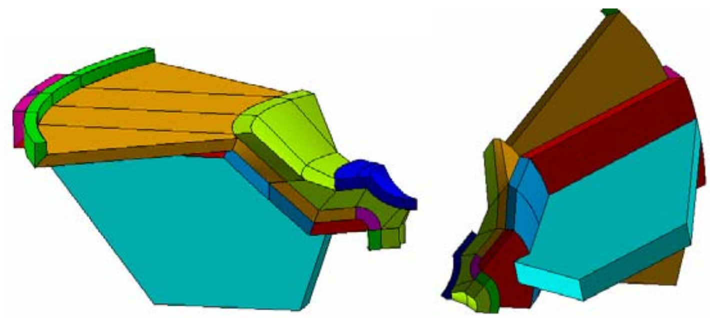

Figure 3. Principal components of the base shell in cut.

4 NUMERICAL SIMULATION

Advantages versus disadvantages of two distinct FE models used in numerical simulation performed by general purpose FE software ANSYS [ANSYS, 2000] are discussed in detail. First model is build of (20 node) solid brick elements. In this way it is possible to use the same kind of elements also in metallic parts, ensuring correct model of the interfaces between the different materials, which is important for the load transition. On the other hand, laminate composite properties must be introduced in their mean values, in our case they are calculated using classical theory of laminates, [Daniel and Ishai, 1994], [Reddy, 1997]. Calculations are programmed in software Maple, allowing arbitrary posterior alteration of the input data, which is important for optimization. Plate-type shell due to the adopted layout (Table 2), belongs to quasi-isotropic material and thus its behavior is approximated by in-plane isotropy.

Second model exploits multi-layered shell elements in composite laminates parts. This brings the advantage to model exactly the layout specified in Table 2. However, now the interface between the metallic and laminate parts must be modeled in simplified form.

Both FE models must state correctly principal directions of material orthotropy, which is done by specifying local coordinate systems, in summary, cylindrical, spherical and toroidal systems are implemented in the model.

Preliminary results correspond to concentric vertical static pressure of total value of 50kN applied at the steel alveolus. These results were already presented at national event [Pedradas, Dimitrovová and Travassos, 2001]. Next figures review some of them. In Figure 4 the FE mesh is presented, while Figures 5-6 show some of the numerical results.

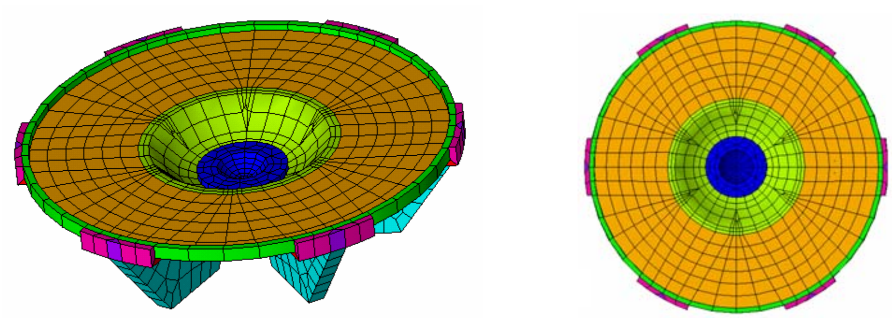

Figure 4. FE mesh used in preliminary studies.

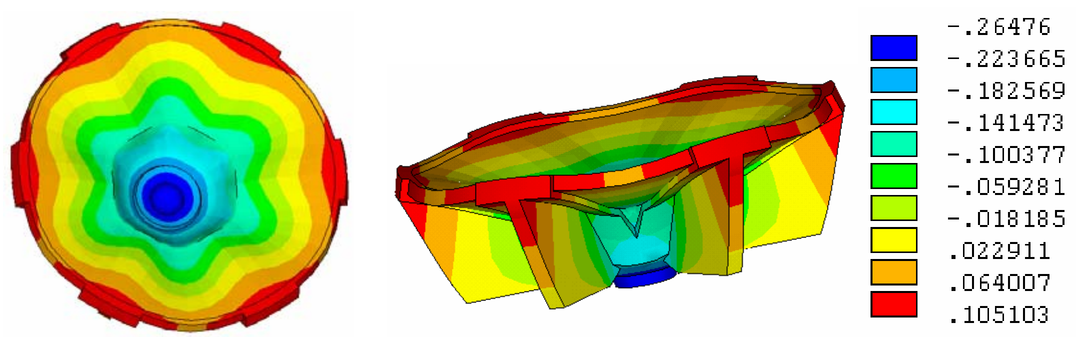

Figure 5. Amplified vertical displacement on the full base shell [mm].

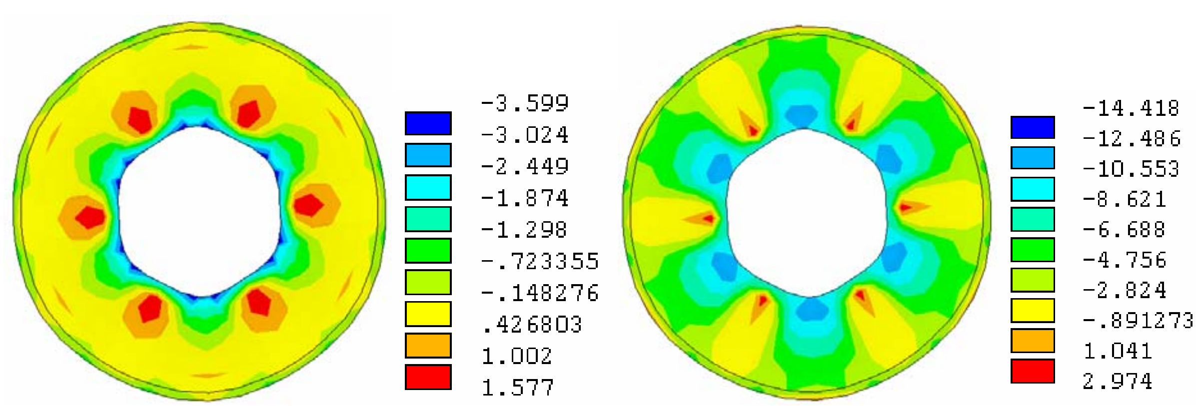

Figure 6. Radial normal tension and tangential normal tension at the horizontal surface of the plate-type shell [MPa].

In the first approach only static load is examined. Optimization of the shell base structure with the aim to reduce the manufacturing cost and weight while keeping the required structural resistance must be originated by determination of the maximum load of the onset of the first failure. With this purpose different kinds of possible failures must be taken into account. It can occur either delamination or ply failure in composite components, either cohesion or adhesion failure at some interface, instability of some composite part exposed to high compression and also yield in metallic parts should be considered. Anyway, in determination of the maximum load corresponding to the first possible failure, linear analysis can be used.

Even in linear static analysis many input factors might be determined with some error, which can influence the final results. Regarding this fact, first studies omitted the ground properties to be used in elastic foundation constants, and only rigid support at bottom part of the base shell is simulated. Sensitivity of the results to constitutive properties or geometry details are studied in parametric analysis exploiting ANSYS Parametric Design Language.

5 EXPERIMENTAL EXAMINATION

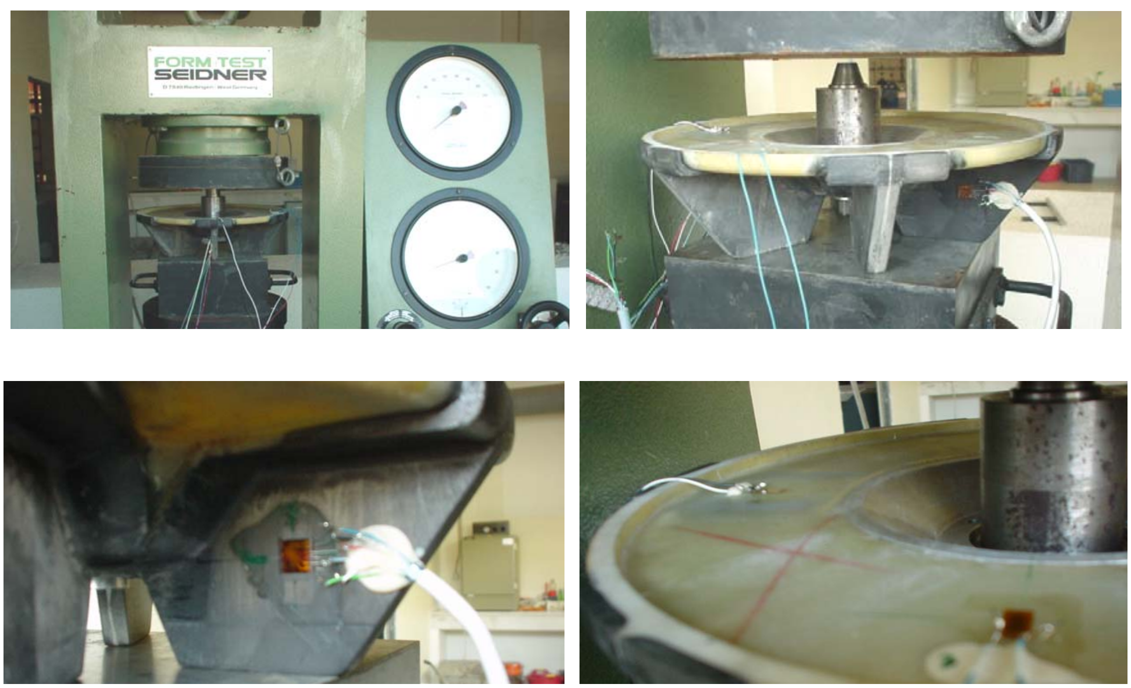

Figure 7. Hydraulic press and gauges implementation.

Numerical results are verified by experimental measurements using HBM/Spider, 8-600Hz from Hottinger Baldwin Messtechnik equipped with software Spider8 Control. In the execution of the experimental testing hydraulic press FORM TEST from Seidner is used. It is shown in Figure 7 together with the shell base with gauges implemented.

6 CONCLUSION

Complex structures consisting of distinct materials can be found in many applications. Therefore, detailed studies of their behavior under static and especially under dynamic loads, implementing correct model of the material interfaces permitting load transition form distinct material components, and yielding possible manufacturing cost saving, are very important. Present study belongs to such kind of works. Although the examination of the structure behavior is in first studies simplified by omitting the ground properties and considering only the static loads, obtained results already allow making important conclusions about optimization of the base shell with the aim to reduce the manufacturing cost and weight, while keeping the required structural resistance. However, the main objective of this study is to choose and establish requirements on finite element model for numerical simulation, giving results of high confidence for static load and consequently allowing to be used in dynamic/impact type of load. Contribution of this work consists also in presenting sensitivity analysis of the results, which is the principal study required in modern approaches. It is shown that sensitivity analysis can be done by parametric study automatized by ANSYS Parametric Design Language.

7 REFERENCES

1. **ANSYS (2000),** Tutorials of the version 5.7, Swanson Analysis Systems IP, Inc.

2. **Daniel I.M., Ishai O. (1994),** Engineering Mechanics of Composite Materials, Oxford University Press.

3. **Reddy, J.N. (1997),** Mechanics of Laminated Composite Plates, CRC Press.

4. **Travassos, J.M. et al (1994),** "Reinforced Shells Made of Hybrid Epoxy Composites", Proceedings of 10^{th} Internacional Conference on Experimental Mechanics, Recent Advances in Experimental Mechanics, Lisbon, Portugal, pp 1055 - 1060.

5. **Travassos, J.M. (1994),** "Produção de Materiais Compósitos de Elevado Desempenho Mecânico", PhD Thesis, IST, Lisbon, Portugal, 349fl.

6. **Pedradas, A., Dimitrovová, Z., Travassos, J.M. (2001),** “Simulação Numérica do Comportamento de Estruturas em Materiais Compósitos”, Proceedings of 4º Encontro Nacional de Análise Experimental de Tensões e Mecânica Experimental, Bragança, Portugal, pp. 33-34.