Annals of DAAAM for 2010 & Proceedings of the 21st International DAAAM Symposium, Volume 21, No. 1, ISSN 1726-9679
ISBN 978-3-901509-73-5, Editor B. Katalinic, Published by DAAAM International, Vienna, Austria, EU, 2010
Make Harmony Between Technology and Nature, and Your Mind will Fly Free as a Bird
Annals of DAAAM International

ANALYSIS AND OPTIMIZATION OF JUNCTION STRUCTURES MADE FROM LAYERED COMPOSITES USING FEA

TUDOR, D[aniela] I[oana]; VLASCEANU, D[aniel]; DINU, G[abriela] & PETRESCU, H[oria] A[lexandru]

Abstract: *This paper highlights analysis and optimization method of junction layered composites structures using finite elements analysis (FEA). In practice, there are many junction structures made from alloy or composite materials. The authors was effectuated a study for one type of structures made from layered composites material namely a junction between a tube and a flange. The optimization process was accomplished in two steps: the first step was the static analysis of the structure and the second step was the optimization process aiming at the weight reduction of the entire structures.*
Key words: *FEA, composites, junction, optimization, flange*

1. INTRODUCTION

The modelling and analysis, using a numerical calculus, of layered composites structures be made aiming at the determination of the local stresses states scope, in the structures areas where stresses variations are high. Example: supports areas, loading areas, geometrical discontinuity areas, junctions etc. For junctions the main problem which should be considered is to develop constructive and technological solutions aiming at the interpenetration of composite layers so as to ensure a maximum strength in junction area, for applied stresses.

Given the wide variety of industrial junction structures type and constructive and technological possibilities of execution do the practical significance of these which justify the research interests for this class of problems. (Hadar et al., 2004)

Modelling and analysis the junction areas of structures made from layered composites should allow the stresses determination in all layers and at the interface layers too.

The used method for static analysis and optimization was the finite elements method (FEM). This method, in speciality literature, represents a useful method for these types of complex problems. (Constantinescu et al., 2006)

The types of geometrical junction layered composite structure analyzed are showed in Figure 1.

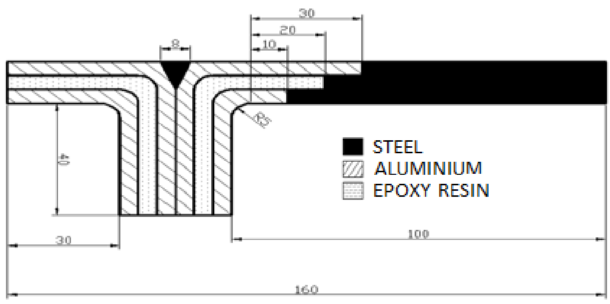

Fig. 1. The geometry of junction type structure

2. METHOD

In the optimization process is necessarily to effectuate, in first step, a static analysis of the structure taking into account the real behaviour of these.

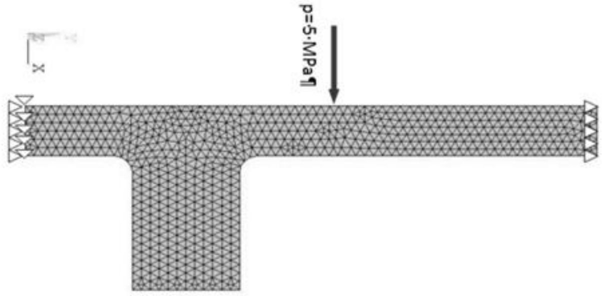

Fig. 2. The analyzed model

The static analysis was realized creating the geometrical model using the initial dimensions (see in Figure 1). The thickness of aluminium and epoxy resin was considerate at 5 mm. The structures was loaded with internal pressure p=5 MPa and constraint on Ox and Oy at the base of the flange and Oy at the end of the tube as shown in Figure 2.

Composite material used has three layers for tube and six layers for flange. To increase the strength and rigidity of the flange was placed inside a steel ring (Figure 1).

The elastic properties for each material are: steel (E=210000 MPa, ν=0.3), aluminium (E=70000 MPa, ν=0.35) and epoxy resin (E=2240 MPa, ν=0.46).

The meshing of this structure was made using a finite element type PLANE 82. This element provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy in comparison with others. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries. The 8-node element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element or as an axial-symmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.(ANSYS Manual)

Mesh structure was performed on each layer in order, to take account of the properties of each compound material. The size finite elements were imposed with value of 2.5 mm. After meshing the structure were obtained 2747 nodes and 1286 elements.

For the optimization, when elaborating the model with finite elements, the user must take into consideration the following aspects:

- The main problem of the junction is its local character. That's why the model of the junction will be elaborated for a sub model or for one of its substructures;

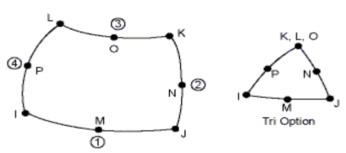

Fig. 3. The PLANE 82 geometry

- The model will be defined parametric, so all its dimensions will be defined by words. Some parameters will have constant values in the optimization, and others are design variables and are declared as such;

- Constant values of the parameters and limits of variation of design variables must be set with discretion, so the junction do not degenerate, namely geometric configurations with overlap, gaps, outliers distortions etc.; (Iliescu et al., 2009)

- Definition of model geometry (points, lines, surfaces, volumes etc..) must be made in order to highlight the composite layers as they are of different materials and results obtained from finite element analysis should provide information on the tensions and strains on interfaces between layers;

- Meshing the model should be done by generating nodes and elements along each layer, as distinct group of elements with different physical, elastic and mechanical properties.

Other aspects of modelling, analysis and optimization, are the usual known. (Hadar et al., 2007)

The objective of the optimization was weight minimization of entire structure. The parameters which were varied: the thickness of the aluminium between 2 and 8 mm and the thickness of epoxy resin between 2 and 8 mm. Restriction that was imposed as the maximum equivalent stress does not exceed 80 MPa. This value was chosen because, in static analysis was seen as maximum equivalent stress has a value of 46.144 MPa.

3. RESULTS

After static analysis revealed that the equivalent stress, calculated according vonMises criterion, has a maximum of 46.114 MPa in the ring of steel and the stresses, in layers of resin, are minimal.

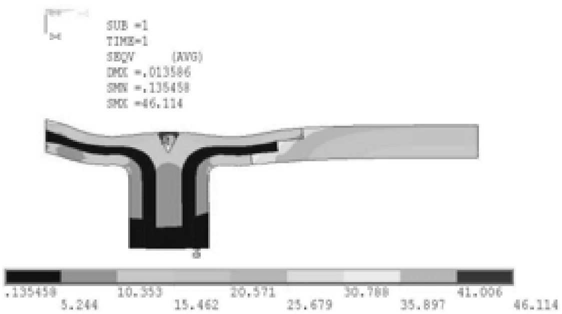

Fig. 4. The variation of vonMises stresses (static analysis)

In following Figure are presented the geometry of entire structure after optimization process. Analyzing this Figure we observed that the layers thicknesses are decreased.

Fig. 5. The new structure after optimization process

The convergence was reached after 13 iterations, and in Table 1 is presented the values of layer thicknesses (variables) and the value of volume which was the objective function.

The notations in Table 1 represent: S_{MAX} - the maximum vonMises stress (state variable); T_1, T_2, T_3 thicknesses of steel, aluminum and epoxy resin layers (design variables); T_{VOL} represent the total volume (the objective function).

	Name	Initial value	Final value
S_{MAX}	State variabiles	42.85 MPa	79.69 MPa
T_2	Design variabiles	5 mm	2.27 mm
T_1	Design variabiles	5 mm	2.58 mm
T_3	Design variabiles	5 mm	2.04 mm
T_{VOL}	Objective function	1.51E+06	6.43E+05

Tab. 1. The values of variables

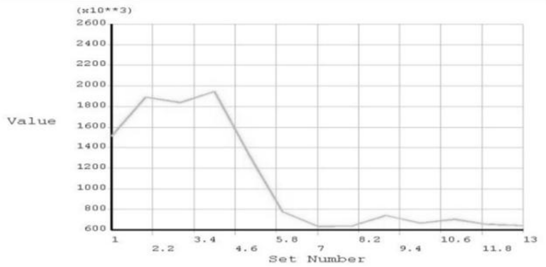

Fig. 6. The volume variation depend the number of iterations

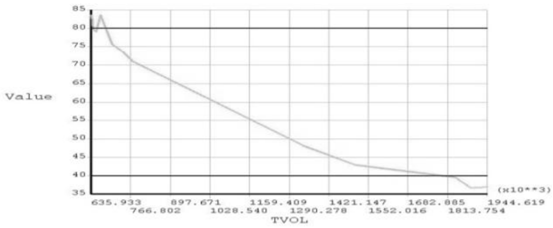

Fig. 7. The S_{MAX} variation depend the volume variation in optimization process iterations

4. CONCLUSION

The criterion for optimization is chosen based on the conditions that the structure must fulfil. Each criterion lead to a certain result; there is no ‘absolute’ criterion of optimization.

After the optimization, the volume was decreased by 42.58% and the thickness decreased by 50% which leading to reduce the material costs.

5. ACKNOWLEDGEMENTS

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Romanian Ministry of Labour, Family and Social Protection through the Financial Agreement POSDRU/88/1.5/S/60203.

6. REFERENCES

Constantinescu, I.N., Sorohan, St, Pastrama, St, (2006) “*The Practice of Finite Element Modeling and Analysis*”, PRINTECH Publishers, Bucharest

Hadar, A., Constantinescu, I.N., Jiga, G., Ionescu, D. S., (2007), “*Some Local Problems in Laminated Composite Structures*”, Mat. Plast. , 45 (4), Bucharest, 2007

Hadar, A., Constantinescu, I.N., Motomancea, A., (2004), “*About the Theoretical and Practical Analysis of the Junctions of Layered Composite Structures*”, 1st International Conference “From Scientific Computing to Computational Engineering”, pp. 230-233 8-10

Iliescu, N; Hadar, A; Pastrama, St., (2009), “*Combined Researches for Validation of a New Finite Element for Modelling Fiber Reinforced Laminated Composite Plates*”, Mat Plast., 46, Bucharest,

*** ANSYS Manual