The 3rd International Conference on International Conference
″Computational Mechanics
and Virtual Engineering″
COMEC 2009
29 – 30 OCTOBER 2009, Brasov, Romania

FINITE ELEMENT METHOD USED FOR MODELING A GLASS FIBRE-REINFORCED COMPOSITE TANK

Stanciu A.[1], Purcarea R.[1], Munteanu M.V.[1], Guiman M.V.[1]
[1] Transilvania University of Brasov, ROMANIA, e-mail ancastanciu77@yahoo.com

Abstract: *The analysis of an element in a mechanical system to design using complex softwares based on the FEM, requires knowing information about the material behaviour, geometric form, links to other items, loading, functional behavior, etc. The model was developed using a preprocessor / postprocessor MSC.Patran and MSC. Nastran.*
Keywords (TNR 9 pt Bold): *geometry, mesh, layers, loads*

1. INTRODUCTION FEM ANALYSIS

Numerical methods (including FEM) analysis of physical phenomena that occur in continuous geometric domains require their replacement with idealized domains, as groups of smaller subdomains, in the FEM, called finite elements. Borders of finite elements consist of points (knots), straight lines or curves (nodal lines) and / or plane or random surfaces (focal areas).
Operation of choosing the number of nodes and type of lines or surfaces respecting the node level continuities in the finite element modeling of geometric areas, is called meshing.
The meshing can be very detailed, and as more as detailed and complex is the meshing (the meshing network) the result of the analysis is more precise and the detail is more realistic, but with this increase of the meshing also more resources from the work station are required and an increased analysis time (this 2 will involve more costs in the FEM analyze).
To adjust the density of meshing on one edge or into a corner from one body can be made with the finesse factor.
The effectiveness of a model element analysis molding, custom types of finite elements used by the correlation with the degree of fineness of the mesh, is quantified through the precision of the results obtained and through the resolution time of the associated numerical model.
Meshing scheme, which aims predefinition of sizes and number of finit elements, must be chosen so that it can obtain a minimum difference between the approximate solution (obtained with FEM analysis) and the accurate one. The approximation process of the obtained solution to the exact solution with the increasing number of finite elements is called convergence. The first stage of drawing up the model of analysis is particularly important because the shape and its dimensions have direct implications on the accuracy and the analysis cost.

Figure 1: The tank

Preprocessing steps to the geometric modeling, finite element modeling lead to the finite element model, finally solved by help of the software package for this purpose.
Finite element modeling involves modeling the behavior of the material, choice and personalization of the finite elements, generating the structure of finite elements, placing the terminal conditions and the loadings.
Analysis and resolution of finite element model established in preprocessing involves first setting the parameters for solving and execution of specific program modules. In this stage the operator seeks information and error messages that occur at run time and waits for its completion.
Postprocessing the results obtained after solving the finite element model requires the distorted and animated states and visualization of the structure with finite elements and viewing in various forms (lists, fields, charts, graphs) the obtained parameters. Options for viewing the results are different and the existing program packages allow the designer easily to identify the in and out parameter values at any point of the geometric domain.

2. TECHNICAL REQUIREMENTS

The model was developed in the MSC Patran preprocessor, through the Geometry menu, using drawings of the assembly shown in Figure 2.
Generating the geometrical model using elementary entities to obtain the finite element model is preferred to be done by maintaining the continuity in the areas of transition from an entity to another.

Figure 2: Geometry of model

Meshing model in elements has to be made in the MSC Patran preprocessor, using the Elements menu. The Elements menu can also define nodes and elements.
The model was largely meshed with elements of type quad 4 (rectangular) and very few elements of type 3 (triangular), and the method used for meshing is the IsoMesh.

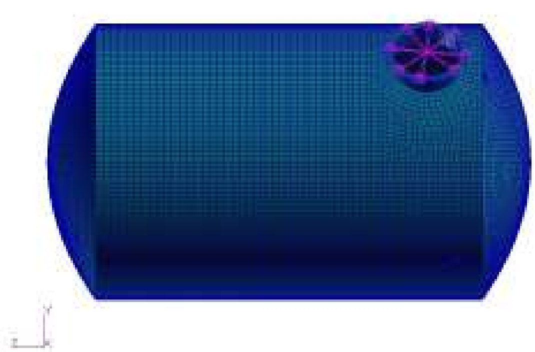

Figure 3: Side view of the meshed model

The following materials have been used:

• MAT 600 - fiberglass composite (short wires) in the matrix of epoxy resin with specific weight 2x600g / m2, 2-2,6 mm thick;

• RT 800 - fiberglass composite (fabric) in the matrix of epoxy resin with specific weight of 4x 800g / m2, thickness 3,2-3,6 mm;

• MAT 450 - fiberglass composite (short wires) in the matrix of epoxy resin with specific weight 2x450g / m2, 1.6-2mm thick.

In the module Layup / rolled plates layers must be created. For this we use LM Ply.

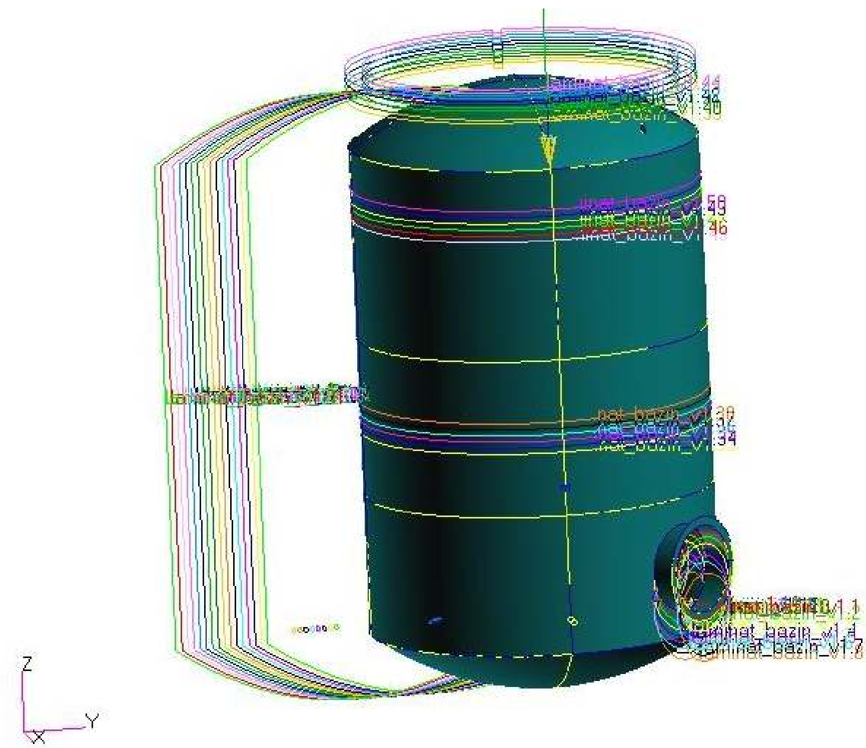

Figure 4: The sequence of layers

Then we choose the material already defined, select the area, a point on the surface for the normal, the direction from where we apply the ply in the laminate. On the label Reference Direction we'll put the direction towards we choose the tilt of fibers. On the label Reference Angle we'll put the angle towards the reference direction the ply is applied with. Sequence of layers in the laminate, and the arrow shows inside, and so they are placed from the inside to the outside, as shown in Table 1

Table 1:

Direction of the part layers	Layer (Ply)	Material type
inside	1	MAT600
	2	MAT600
	3	RT800
	4	RT800
	5	RT800
	6	RT800
	7	MAT450
outside	8	MAT450

3. RESULTS

For the results, we use the MSC.Patran preprocessor, with the Results menu.

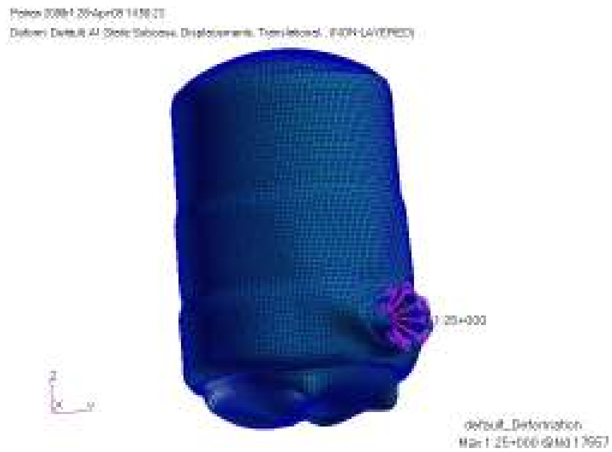

Figure 5: View (exaggerated) of deformations

To note that in the FEM analysis of the tank full of additives, a deformation of 1.25 mm appears, which exists as a maximum only at the mouth of visiting of the tank, which implies, in fact, that the material used is about 40 times more resisting.

Minimum and maximum stresses result on running in the MSC. Patran, searching for in fact: the distribution of maximum Von Misses stresses up of all layers, the distribution of minimum and maximum stresses on the X component of all layers and on and each layer separately, the distribution of maximum and minimum stresses on the Y component of all layers and on and each layer separately

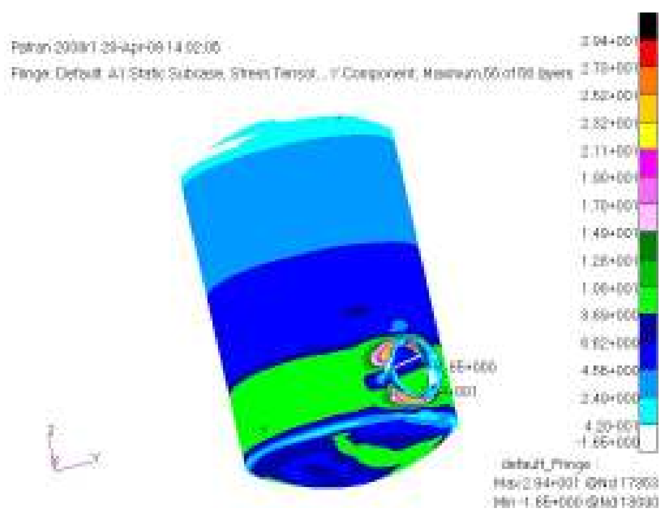

Figure 6: Distribution of maximum Von Misses stresses up of all layers

Figure 7: Distribution of maximum stresses on the component X of all layers

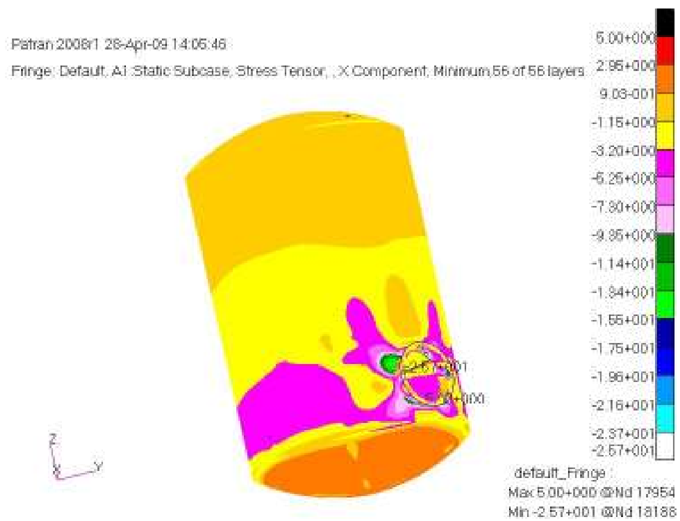

Figure 8: Distribution of minimum stresses on the component X of all layers

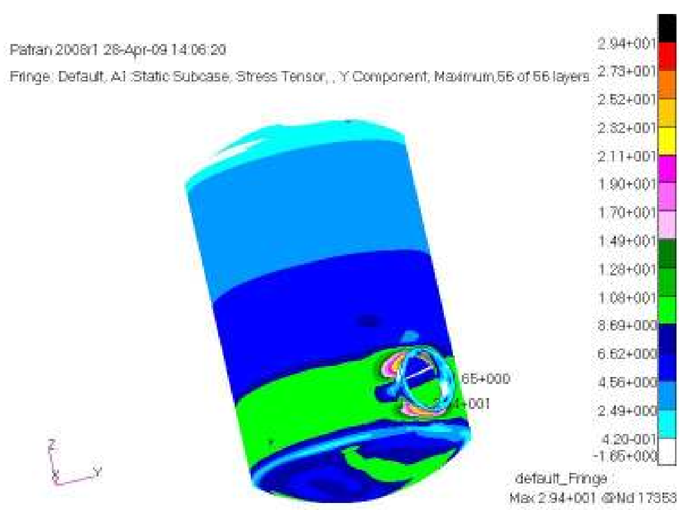

Figure 9: Distribution of maximum stresses on the component Y of all layers

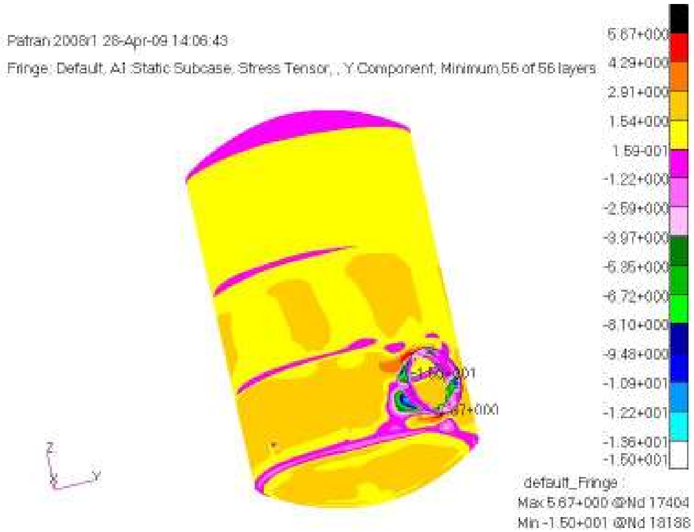

Figure 10: Distribution of minimum stresses on the component Y of all layers

Table 2 presents the results of the distribution of stresses: maximum of VM, X and Y, and also the minimum on X and Y.

Table 2:

Criteriu Criterion	Tensiune Stress [MPa]	Compresiune Compession [MPa]
σ_{VM}	29,4	-
σ_X	42,8	-25,7
σ_Y	29,4	-15

3. CONCLUSION

Finite element method has become a general method for solving various kinds of complex problems concerning both stationary and nonstationary phenomena in all branches of engineering sciences, but also in other areas.
As for the deformations and stresses, we can see that the mechanical work inside is associated with three components of the tensions on plane coordinates, the normal plane coordinate of the stress does not involve any equalization to zero of other tensions or stresses.
Similarly, the two components of the displacements in any plane section of the body along the axis of symmetry completely defines the state of deformation and of course the state of stresses.
Due to FEM, advantages are to compare and check the strentgh of a material, stresses are measured, deformations that may exist in material and are within normal limits, in order to save a lot of time, a much higher speed, lower costs, evaluating each part of the material.
Using finite element method we can upload several versions i.e.: changing loading conditions, terminal conditions, the manner of applying it on the virtual model, from which we can choose the optimal variant.

REFERENCES

[1] Vlase, S., Finite Element Analysis of the Planar Mechanisms: Numerical Aspects. Applied Mechanics - 4. Elsevier, 1992,p.90-100.
[2] N., Pandrea, Marina Pandrea, N., D., Stănescu, Dynamical absorber for Systems with Finite of Freedom Degreees. The Second Inter. Conf. of Roumania, Soc. Of Acousties on Saund and Vibration, Buckarest, 2004.
[3] Vlase,S., Tofan, M.C., Goia, I.A., Some Aspects of Finite Element Analysis of Flexible Multibody Systems.Virtual Nonlinear Multibody Systems. NATO Advanced Study Institute, Vol.I,p.224-229, Prague,2002.