Abstract—The present paper deals with the study of the stress and deformation occurred inside some parts of a storage tank made of composite materials. As the most exposed part of the tank is the sight hole, the research was focused upon the stress and deformation of this part, which was subjected to the action of the stored materials pressure. The material used for the structure consists of several layers with different properties.

Keywords—Composite materials, deformation, finite element method, layers, stress.

I. INTRODUCTION

Finite element method (FEM) becomes more and more a general method used for solving different types of complex problems concerning both stationary and non-stationary phenomena from all engineering fields but also in other activity and research areas.

As far as the stress and deformation are concerned we may observe that the internal mechanical work is linked to three components of the stress in 2D coordinates, the normal plane component of the stress does not involve the canceling of other strains or stresses.

The issue of stress distribution within the revolution bodies (axial-symmetrical) is of real importance and interest. From mathematical point of view, the problem is very similar to that of plane stress and deformation analysis, this is why the situation may be regarded as two dimensional.

By symmetry, the two components of the displacements in any 2D section of the body along the symmetry axis, completely defines the deformation state and obviously the stress state.

In order to control the complexity of the problem and “filter” the irrelevant aspects we need to accomplish a suitable mathematical model. This model should consider the fact that we are dealing with an anisotropic material, consisting of several layers and also that the loads and deformations along the contours are difficult to be obtained.

The internal stress and deformation field is locally influenced by the relative difference between the constituents’ properties, their size, shape and relative orientation as well as by the geometry of the repeating structures that form the composite material.

II. MATHEMATICAL MODELLING

A simulation process based upon a model using finite element method requires several stages. These stages are presented in fig.1.

Fig.1 Mathematical approach of FEM

The main part of the process is, as shown in the diagram, the mathematical model. This is mostly an ordinary equation or a differential one, developed in space and time. A discrete model with finite elements is generated by help of the variation form of the mathematical model. This stage is called meshing.

The FEM equations are solved using an equation solver that will provide a discrete solution.

On the left side of fig.1 an ideal physical system is presented. It may be considered as a realization of the mathematical model, while the mathematical model may be considered as an idealization of this system.

For example, if the mathematical model is represented by a Poisson equation, the physical achievement can be represented by the heat conduction in a bar or a problem of electric charge distribution. This stage is not always necessary and may be eliminated. The FEM meshing can be done without any reference to the modeled process physical aspects.

The concept of error occurs when the discrete solution is replaced in the mathematical model. This is generally named checking. The solution error represents the extent to which the
discrete solution does not check the discrete equations. This error is relatively not important when using computers and especially for linear equations systems. More relevant are the meshing errors, representing the extent to which the discrete solution does not check the mathematical model. The replacement in the ideal physical system might identify the modeling errors.

III. FEM ANALYSIS OF THE SELECTED PART

The analysis was focused upon the most exposed part of a storage tank, namely the sight hole.

The model was achieved using MSC Patran preprocessor/postprocessor and MSC Nastran processor.

In the preprocessing stage, the finite elements geometric modeling requires the finite element model, which will be finally solvable by help of the programs kit meant for this purpose.

A finite element modeling requires the material behavior modeling, selection and personalization of finite elements, finite elements structure generation, introduction of boundary conditions and loads.

The analysis and solution of the finite element model, elaborated during preprocessing requires the preliminary setting of the solving parameters and the execution of the specific program modules. During this stage, the information and error messages occurred while the program is running should be carefully monitored.

The post processing of the results obtained after solving the finite elements model assumes the visualization of the deformed and animated state of the studied structure and also the visualization of various parameters using lists, diagrams and fields. The results can be easily accessed and the input/output values of the required parameters may be identified at any point of the geometric domain.

The generation of the geometric model using elementary entities was achieved by maintaining the continuity in the passing areas between one entity and the other.

The geometric modeling previous the meshing requires the generation of closed contours consisting of lines for plane areas or surfaces. In fig.2 we presented the detailed model geometry.

The model mesh was mainly done using quad 4 type elements (quadrangle) and very few elements tria 3 type (triangle), while the meshing method was IsoMesh.

Generally the use of quadrangle elements is recommended because it offers an increased accuracy and a smaller number of finite elements for meshing. With Patran we may use four meshing methods: IsoMesh, Paver Mesh, Auto TetMesh and 2.5 D Meshing. IsoMesh method creates elements within a regular geometric region (surface) by simple subdivision of the interval.

In order to simulate the locking cap of the sight hole, we used a Multi Point Constraint (MPC) element, RBE2 type (Rigid Body Element 2), consisting of a Master node and several Slave nodes. The stiffness and displacements of the Slave nodes is replaced in the stiffness matrix by the Master node stiffness.

The mesh is obtained based on the geometric model of the sight hole, generated by MSC Patran preprocessor, as shown in fig.3. Its accuracy may be automatically chosen by the program or manually by the user.

![Fig.2 Geometric model of the sight hole](image1)

![Fig.3 Sight hole simulated by MPC](image2)

![Fig.4 Pressure exerted upon the sight hole](image3)
According to standards the water density is of 1000 kg/m$^3$. Considering a safety factor of 1.25 the water density used in calculus is of 1250 kg/m$^3$. The pressure was determined upon the 5 sectors obtained after model meshing, taking into account the length of each sector and the water volume.

Then the model will be analyzed by help of MSC Nastran processor but before running the file we need to do some previous checking in order to validate the finite elements model, as follows:

- determination of the distance between two locations or nodes;
- determination of the angle between two directions determined by three point, one of them being considered as origin;
- identification of common points;
- identification of common lines;
- identification of common nodes and joining them;
- identification of nodes belonging to a selected plane, with the possibility of moving to this plane of the nodes from the adjacent area;
- identification of the common finite elements;
- determination of a finite element distortions;
- identification of the normal in a plane finite elements group and comparing them to a given direction;
- determination of mass properties for the finite elements;
- checking the geometric boundary conditions;
- determination of the loading forces sum in a node.

Then, during post processing, the output data will be associated to both the nodes and the finite elements. The output data corresponding to the nodes, usually include the problem unknowns, like displacements, temperatures, pressures, velocities.

The output data corresponding to the finite elements are different from one element to another, for example the internal forces, strains, deformation energy.

The structure is made of 8 different layers as shown in Table I, the arrow representing the succession of the layers starting from the interior towards the exterior of the structure.

**TABLE I**

<table>
<thead>
<tr>
<th>Layers direction</th>
<th>Layer(Ply)</th>
<th>Material type</th>
</tr>
</thead>
<tbody>
<tr>
<td>interior</td>
<td>1</td>
<td>MAT600</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>MAT600</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>RT800</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>RT800</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>RT800</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>RT800</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>MAT450</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>MAT450</td>
</tr>
</tbody>
</table>

where:
- MAT 600 is fiberglass composite (short wires) in the matrix of epoxy resin with specific weight 2x600g/m$^2$, thickness 2-2, 6 mm thick;
- RT 800 is fiberglass composite (fabric) in the matrix of epoxy resin with specific weight 4x800g/m$^2$, thickness 3-2-3,6 mm;
- MAT 450 is fiberglass composite (short wires) in the matrix of epoxy resin with specific weight 2x450g/m$^2$, 1.6-2mm thick.

Also the details of the layers succession are represented in fig. 5.
The stress distribution was determined for X and Y components and also on both components (Von Mises) for each layer, some of the results are presented in the figures above. The maximum stress was 7.49MPa for the first layer on X, while for the third layer 25.6MPa on Y and for the seventh 8.03MPa on X.

This proves that MAT type material resists much better to the applied loads, especially on Y direction, but also on X the values are very small in comparison to the efforts occurred in the roving.

IV. CONCLUSION

By comparing the values obtained by help of the finite element method to the admissible values we reach the conclusion that the material could resist to 40 times more loads, proving the fact that the composite combination is suitable for the studied structure.

FEM provides a great number of benefits in comparing and checking the resistance of different materials by saving a lot of time, reducing costs and being able to assess every part of the structure for various types of loads or materials.

ACKNOWLEDGMENT

These researches are part of the grant IDEI 744 with CNCSIS Romania.

REFERENCES