Stress analysis of toroidal shell

Cristian PURDEL*, Marcel STERE**

*Corresponding author
Department of Aerospace Structures
INCAS - National Institute for Aerospace Research "Elie Carafoli"
Bdul Iuliu Maniu 220, Bucharest 061126, Romania
cpurdel@incas.ro
**Aerospace Consulting
Bdul Iuliu Maniu 220, Bucharest 061126, Romania
sterem@incas.ro
DOI: 10.13111/2066-8201.2010.2.4.27

Abstract*. Stress analysis for a toroidal shell used for a flying platform. We will be comparing the results obtained by classical approach and by numerical method (finite element analysis).*

Key-Words: toroidal shell, FEM, stress

1. INTRODUCTION

The toroidal shell will be analyzed from a classical perspective and by a finite element analysis approach. The tools which will be used are MATLAB for the classical approach, CATIA and ANSYS for FEM. Numerical simulations will be made on the theoretical model, as well as on a modified version of the toroid, which will resemble the shape of the project.

Preliminary remarks. We will use three descriptions for the geometry of the torus. *Real torus* is our physical geometry for the flying prototype. *Theoretical torus* is the classical perfect torus on which we will run numerical simulations in MATLAB, CATIA and ANSYS. *Approximate torus* is the approximate shape for the real torus, on which we could run an accurate simulations in CATIA and ANSYS.

MATLAB is going to be used just for the implementation of the theoretical formulas, while also trying to present the results in comparison to CATIA and ANSYS. The geometry of the model will be made in CATIA, and then exported into ANSYS Workbench. We used also CATIA Structural Analysis to see if there were major discrepancies between the results obtained in ANSYS, which has a more established reputation as a structural solver.

The material considered for the first prototype is polyethylene.

Physical properties

Density (ρ) = 950 kg/m^3
Young's Modulus (E) = 1.1e9 Pa
Ultimate Tensile Strength (UTS) = 3.3e7 Pa
Poisson's Ratio (υ) = 0.42
Coefficient of thermal expansion (α_{TE}) = 2.3e-4 C^{-1}

Loading

Internal pressure at 4 000m (p_1) = 125.66 N/m^2

Geometry of the real torus

Dimensions of the real torus:
Height = 1930 mm

Inner distance = 680 mm
Outer distance = 3000 mm

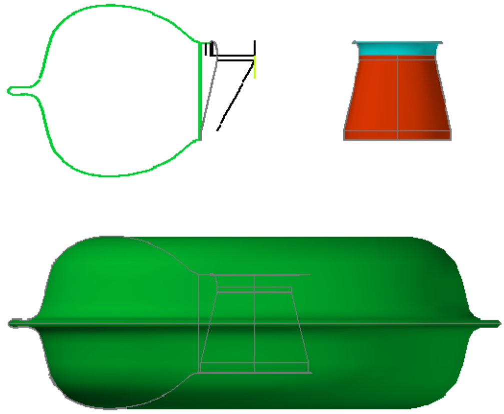

Fig. 1 Real Torus shell (upper left – cross section, upper right – propeller housing, bottom – real model)

A better view of the cross section of the toroidal shell can be seen below.

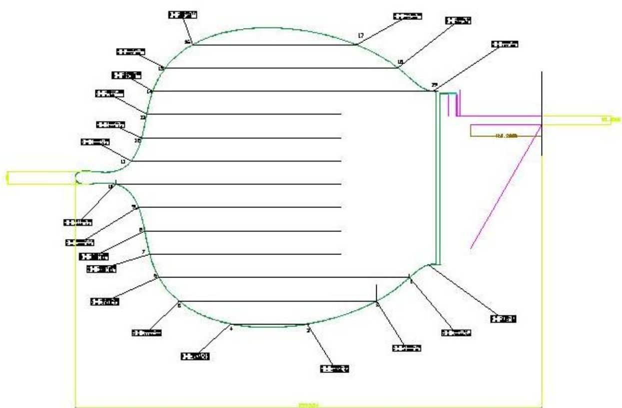

Fig. 2 Cross section of the toroidal shell

Geometry of the theoretical torus. From [4] and [5] we have the following notations on the correspondent figures:

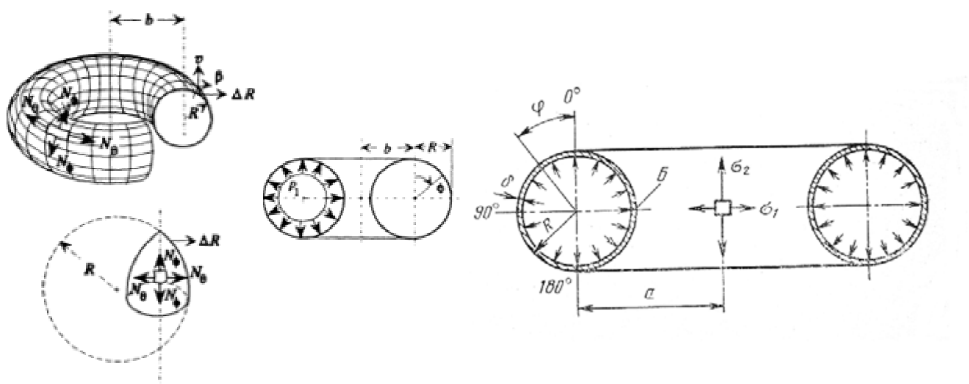

Fig. 3 Theoretical Torus as taken from [4] (left) and [5] (right)

Thickness of the shell (h, δ) = 0.15 mm
The arm of the torus (b, a) = 2.161 m
Meridional angle of point of interest (□, φ)
Latitudinal angle of interest (θ)

Geometry of the approximate torus. Due to the geometry of the structures, while running CATIA and ANSYS simulations the closest approximation that we could obtain from the real model is shown in the figure below. For other geometries which were not as smooth as a perfect circle, and had more inflections as the real model, the results diverged and they were not accurate.

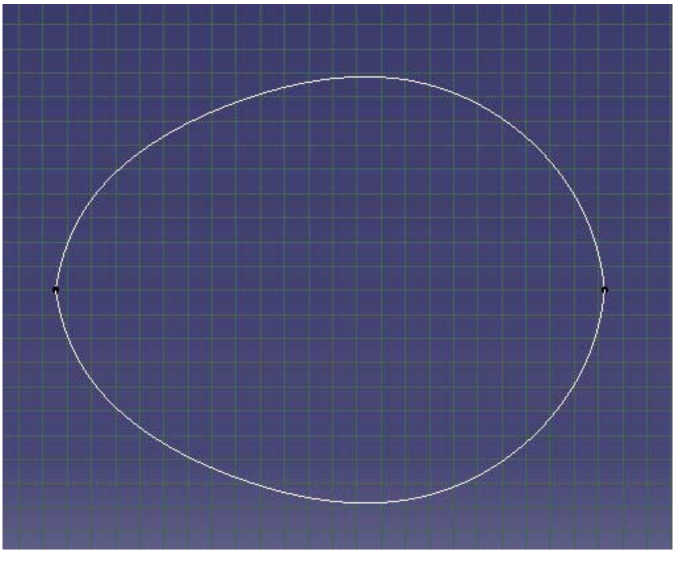

Fig. 4 Cross-sections of the approximate model

2. STRESS ANALYSIS OF THE TOROIDAL SHELL

We will perform simulations on two models: the theoretical torus and the approximate torus.

Stress analysis of the theoretical torus. For the theoretical torus, the first approach will be classical one, using classical analysis formulas.

The second approach will be a finite element analysis running in CATIA and ANSYS.

Classical approach using MATLAB. We shall use the relations given in reference [4], [5]

$$\sigma_1 = \sigma_\phi = \frac{p_1 R}{2h}\left(\frac{2b + R\sin\phi}{b + R\sin\phi}\right) \tag{1}$$

$$\sigma_2 = \sigma_\theta = \frac{p_1 R}{2h} \tag{2}$$

$$\sigma_{eqv} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}} \tag{3}$$

Since σ_3 is negligible compared to σ_1 and σ_2

$$\sigma_{eqv} \cong \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + \sigma_1^2 + \sigma_2^2}{2}} \tag{4}$$

$$\Delta R = \frac{p_1 R^2}{2Eh}[\frac{b}{R}(1-2\nu)+(1-\nu)\sin\phi] \tag{5}$$

Equivalent stress von Mises (σ_{eqv})
Maximum principal stress ($\sigma_1,\sigma_{\square}$)
Middle principal stress (σ_2,σ_θ)
Displacements (ΔR)

Finite element analysis using ANSYS and CATIA. Meshing in ANSYS was done on a specified thickness surface. In CATIA the mesh was done on a solid body with the specified thickness. We tried meshing in ANSYS with solid body and in CATIA with a surface, but the results were not accurate. Below are the meshes from MATLAB, ANSYS and CATIA. In our results, we used only one eighth of the models due to their symmetries and also to increase the resolution of the mesh.

Because of the lack of uniformity of ANSYS and CATIA not producing uniform results, and of the quite inaccurate meshing models, we will define three regions on this model. The reason is that CATIA and ANSYS find maximum or minimum in singular points, which are not relevant for the entire area surrounding that points. We've introduced these regions so that we can have a better view of the overall stress values that we are going to be analyze further on. From Fig. 5, from the MATLAB mesh, we are going to define the inner region, the region around the inner radius of the theoretical torus (the shorter region with the blue color). The upper region will be the highest region (z axis) of the theoretical torus (the middle of the red region). The outer region will be near the outer radius of the theoretical torus (the longer region with the blue color). We used this approach since the model has three planes of symmetry, and all the values are in the regions that we are covering.

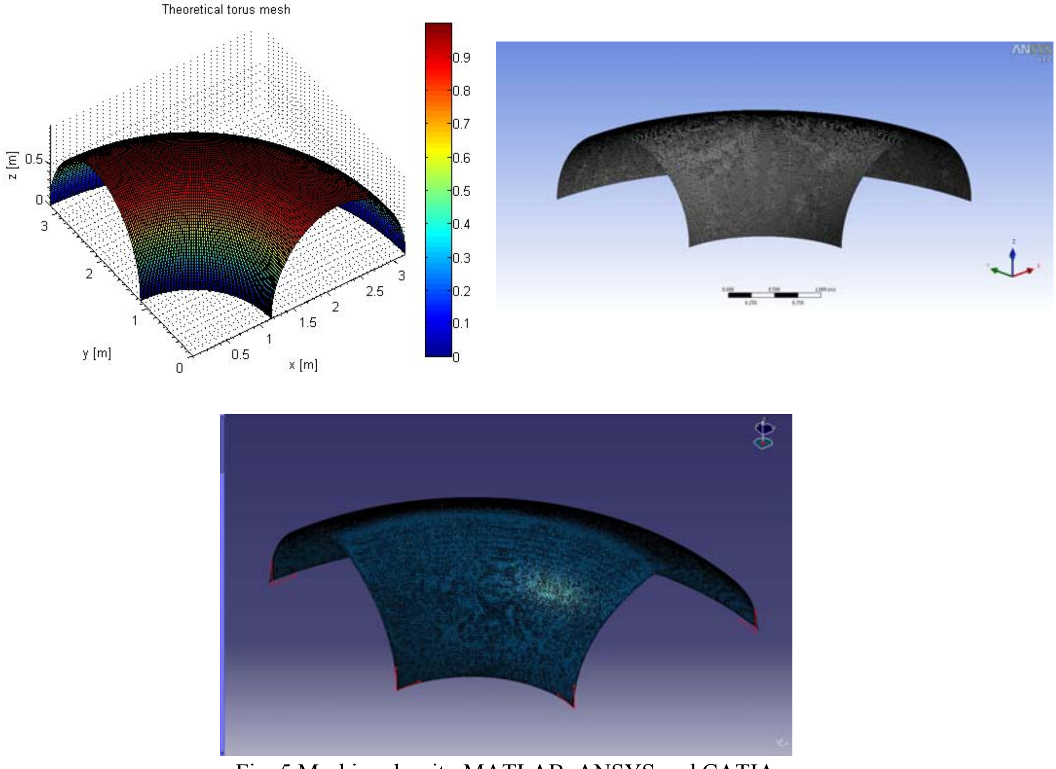

Fig. 5 Meshing density MATLAB, ANSYS and CATIA

The mesh comparison is presented in Table 1:

Table 1 Theoretical torus mesh density comparison for MATLAB, ANSYS and CATIA

Theoretical torus mesh	Number of nodes	Number of elements	Type of elements
MATLAB	100		
ANSYS	55086	54792	Triangles/Quadrilaterals
CATIA	335129	165889	Tetrahedrals

Boundary conditions. Since we used one eighth of the real model, the bounding conditions were placed on the planes of symmetry, restricting only the displacement in the direction perpendicular to the corresponding plane of symmetry. In Fig. 5, in the CATIA model, there is a better representation of the imposed constraints.

Equivalent stress von Mises of the theoretical torus. As stated before, we will present also the values from the three regions, together with the maximum and minimum values from the solvers.

These values are taken not on the boundary, but on the vicinity, so that the boundary conditions don't have a great influence on them.

The values from these regions are arbitrary, but they are meant to better represent the values along the regions of interest.

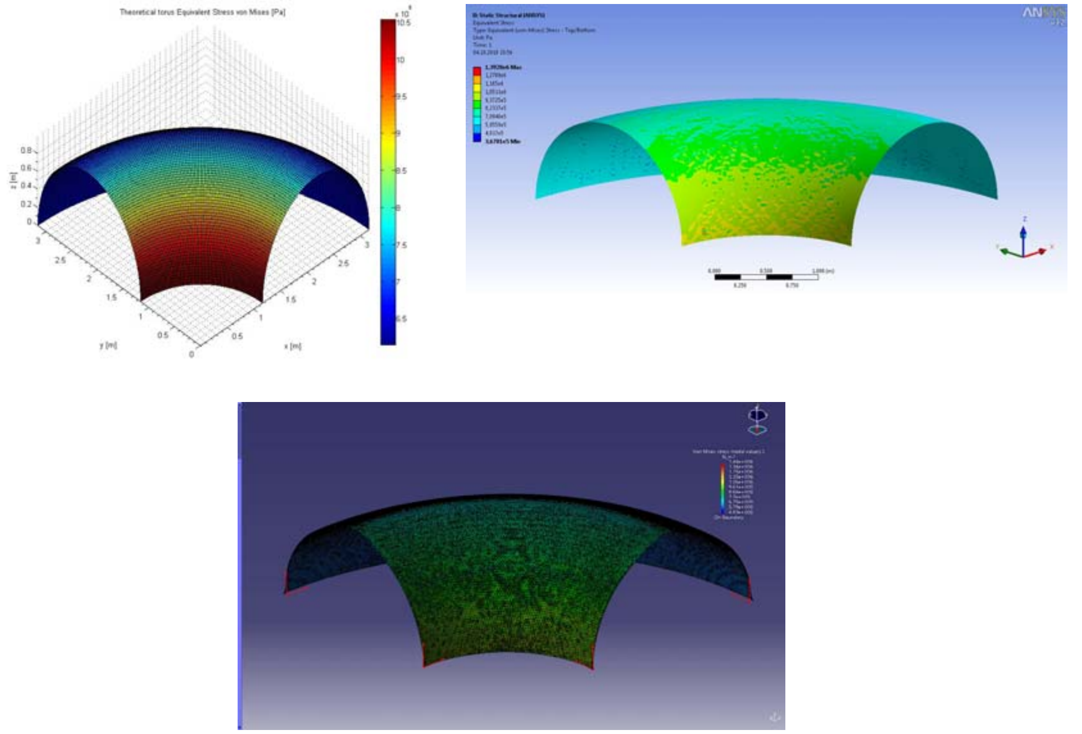

Fig. 6 Equivalent stress von Mises in MATLAB, ANSYS, CATIA

As can be seen more clearly from Fig. 6 and Table 2, all three solvers values are in the same range.

All values are under the UTS specified for polyethylene.

Table 2 Equivalent stress von Mises in MATLAB, ANSYS, CATIA

Equivalent stress von Mises [MPa]	Maximum	Minimum	Inner region	Upper region	Outer region
MATLAB	1.06	0.615	1	0.775	0.65
ANSYS	1.39	0.367	1.05	0.75	0.61
CATIA	1.44	0.483	1.02	0.74	0.62

Total Deformation of the theoretical torus

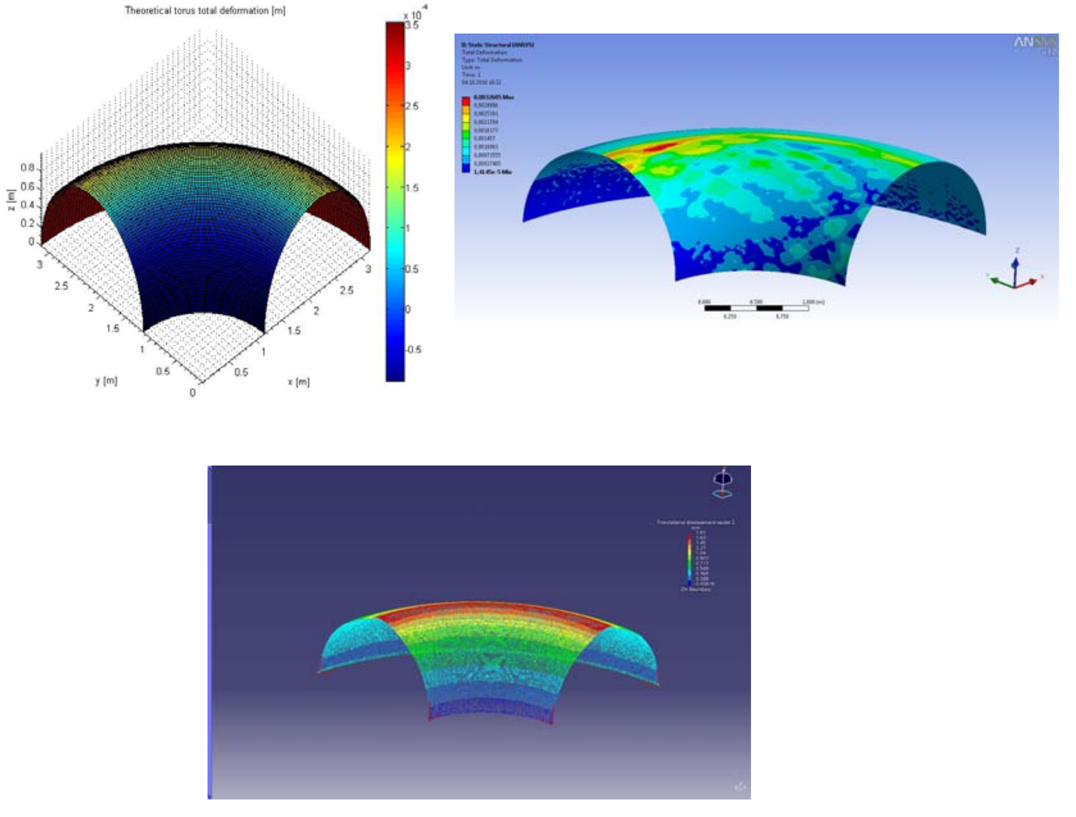

Fig. 7 Total Deformations in MATLAB, ANSYS, CATIA

The discrepancy between the maximum and upper region values may be attributed to how each program computes the total displacements.

ANSYS gives the entire displacement, while CATIA plots the modulus vector, and also the direction.

Overall, the values aren't too high and the polyethylene will not tear.

Table 3 Total Deformation in MATLAB, ANSYS, CATIA

Total Deformation [mm]	Maximum	Inner region	Upper region	Outer region
MATLAB	0.35	0.05	0.13	0.33
ANSYS	3.26	0.05	1.7	0.35
CATIA	1.81	0.07	1.7	0.3

Maximum Principal Stress of the theoretical torus

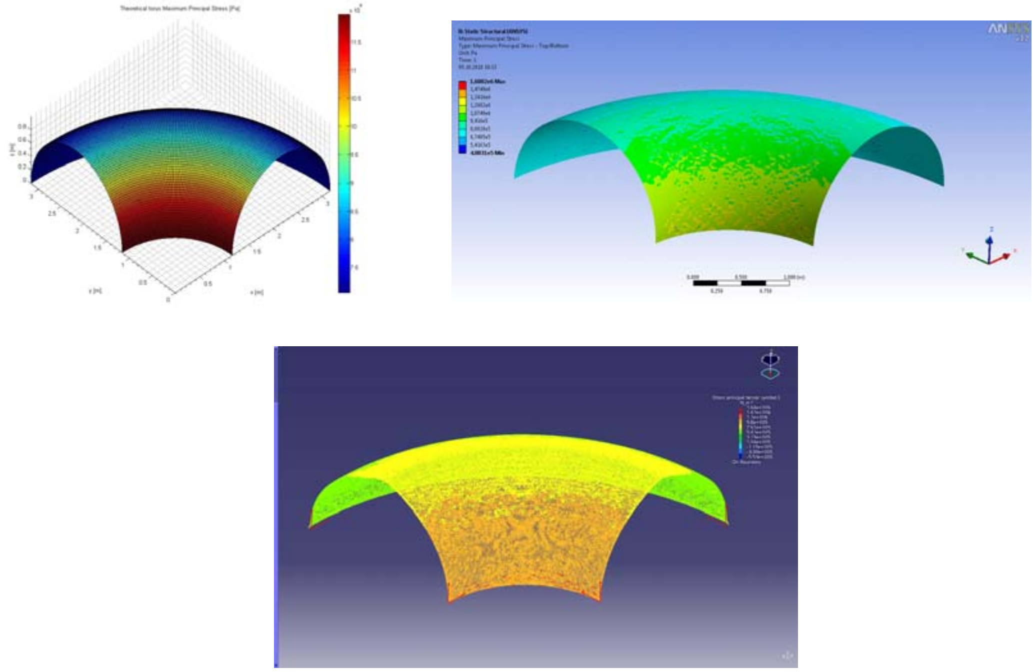

Fig. 8 Maximum Principal Stress in MATLAB and ANSYS, Stress Principal Tensor in CATIA

All the values from within the three regions are more or less in the same range (Table 4). While ANSYS plots the values for each of the principal stress individually, CATIA plots all three values in the same graph.

Table 4 Maximum Principal Stress in MATLAB, ANSYS, CATIA

Maximum Principal Stress [MPa]	Maximum	Minimum	Inner region	Upper region	Outer region
MATLAB	1.2	0.71	1.15	0.85	0.75
ANSYS	1.636	0.497	1.18	0.83	0.7
CATIA	1.64	0.553	1.15	0.8	0.71

Middle Principal Stress of the theoretical torus

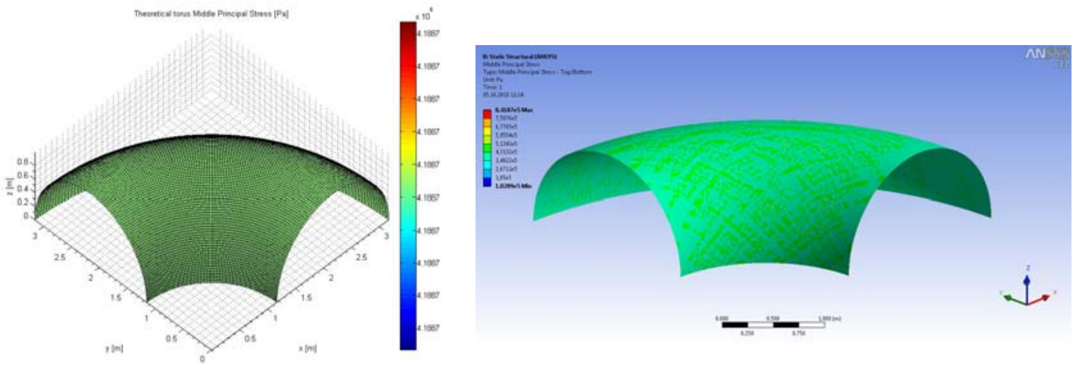

Fig. 9 Middle Principal Stress in MATLAB and ANSYS

As it can be seen from Fig. 9 and (2), the middle principal stress is constant in MATLAB, while in ANSYS and CATIA it varies a lot. However, the overall values are within the same interval, and we also found that the average for the CATIA middle principal stress values is almost the same value as for the MATLAB middle principal stress up to the 4th digit.

Table 5 Middle Principal Stress in MATLAB and ANSYS

Middle Principal Stress [MPa]	Maximum	Minimum	Inner region	Upper region	Outer region
MATLAB	0.418				
ANSYS	0.841	0.102	0.41	0.425	0.42
CATIA	0.925	0.013			0.43

Minimum Principal Stress of the theoretical torus

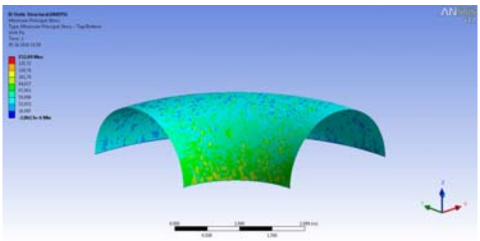

Fig 10 Minimum Principal Stress in ANSYS

As stated before, the minimum principal stress can be ignored if we look at the values of the other principal stresses, especially in ANSYS. Values for CATIA don't seem to be following any pattern, and it should be considered not relevant since they vary so much, and don't have the same pattern as the other two principal stresses.

Table 6 Middle Principal Stress in ANSYS

Minimum Principal Stress [Pa]	Maximum	Minimum	Inner region	Upper region	Outer region
ANSYS	152.6	-2.06e-6	30	40	20
CATIA	687468	-553405			

Stress analysis of approximate torus. After running the simulations on the theoretical model, we are going to run them on the approximate model. The reason we chose this approximate model is due to the fact that both ANSYS and CATIA give erroneous results if the shape is not smooth enough. Thus, we tried using as few inflection points as possible. However, the approximate model can be improved to resemble our real model.

Mesh comparison. Below are the meshes from ANSYS and CATIA. As with the theoretical torus, we used only one eighth of the models because of its symmetries.

Table 7 Approximate torus mesh density comparison for ANSYS and CATIA

Approximate torus mesh	Number of nodes	Number of elements	Type of elements
ANSYS	94787	94368	Triangles/Quadrilaterals
CATIA	279238	165889	Tetrahedrals

Equivalent stress von Mises of approximate torus. Like in the previous simulations, in CATIA we used a solid mesh, while in ANSYS we used a surface mesh. As with the theoretical torus, we will present results from the three regions (inner, upper and outer).

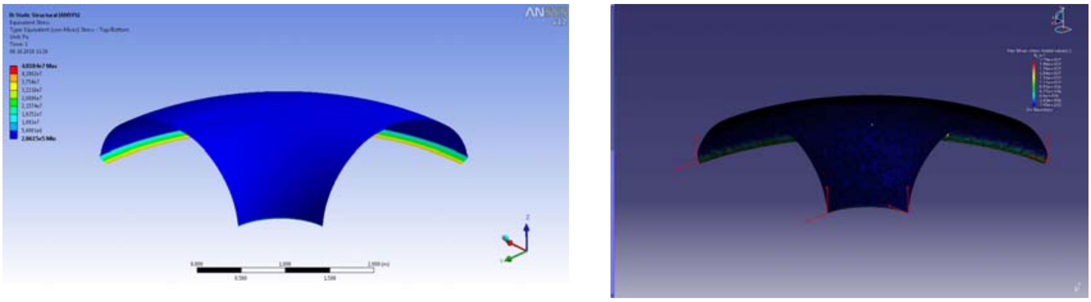

Fig 11 Equivalent stress von Mises in ANSYS and CATIA

The values for the equivalent stress von Mises tend to be higher than for the theoretical model, and the critical part is the outer region, but this is due to the boundary conditions. The maximum equivalent stress exceeds the ultimate strength, but this is influenced by the boundary conditions, and given the results from the theoretical model, the material should not brake.

Table 8 Equivalent stress von Mises in ANSYS and CATIA

Equivalent stress von Mises [MPa]	Maximum	Minimum	Inner region	Upper region	Outer region
ANSYS	48.1	0.28	1.31	1.0	5
CATIA	21.9	0.259	1.4	1.1	6

Total Deformation of approximate torus

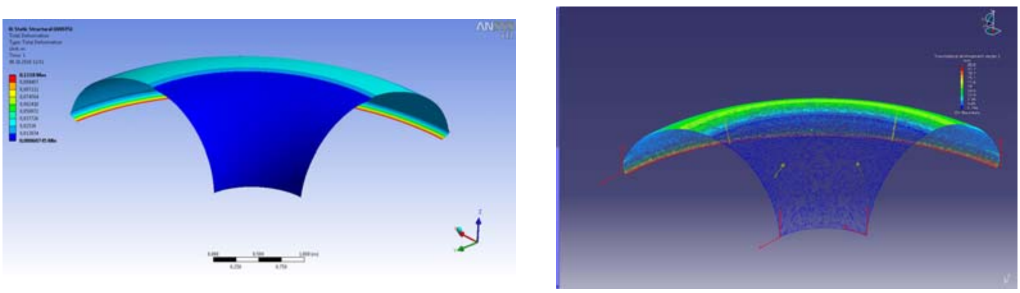

Fig 12 Total Deformation in ANSYS and CATIA

The deformations are not that high, and are within limits. The critical area remains the outer region, and this is because of the boundary conditions we imposed on the model.

Table 9 Total Deformation in MATLAB, ANSYS, CATIA

Total Deformation [mm]	Maximum	Inner region	Upper region	Outer region
ANSYS	111.8	1.11	25	17
CATIA	35.8	1	15	5

Maximum Principal Stress of approximate torus

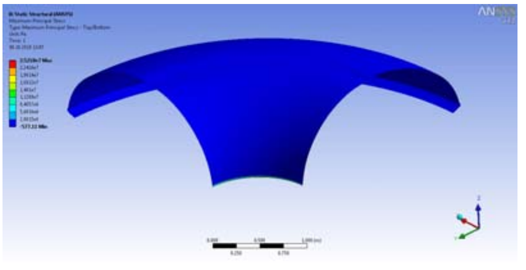
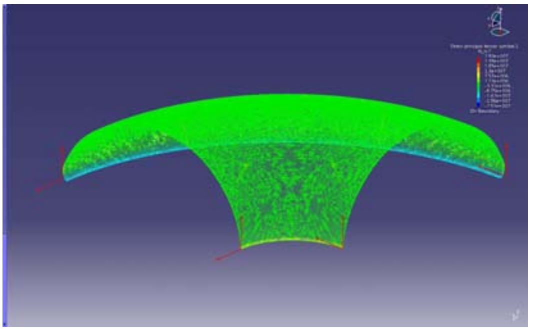

Fig 13 Maximum Principal Stresses in ANSYS and Stress Principal Tensor in CATIA

Values in CATIA were not as continuous as they were in ANSYS, but they were in the same range as the values for the theoretical model. Like in the case of the Equivalent Stress von Mises, the values are higher than the theoretical model (Table 10).

Table 10 Maximum Principal Stress in ANSYS and CATIA

Maximum Principal Stress [MPa]	Maximum	Minimum	Inner region	Upper region	Outer region
ANSYS	25.21	-0.0006	1	1.5	0.9
CATIA	29.34	-16.19	1	1.15	0.7

3. CONCLUSIONS

We tried to find a suitable model that could resemble our real model. But because of a not so smooth geometry, there were problems running the simulation. Also, having such a thin shell, the solid model started giving errors since it could fit one degenerated tetrahedral. Another issue was the boundary condition, which we had to impose in that particular way to take advantage of the symmetries and have a more refined mesh order. For a better accuracy, there could be a more refined approximate model, but the inflexions on the geometry have to be taken into account. Nonetheless, the results from the theoretical models provided by MATLAB, CATIA and ANSYS gave similar results, and since the equivalent stress on the material is below the ultimate tensile strength, for that particular pressure, there are no signs of material tearing.

REFERENCES

[1] *** *MATLAB* 7.9
[2] *** *ANSYS* 12.0.1
[3] *** *CATIA* R18
[4] W. D. Pilkey, *Formulas for stress, strain and structural matrices.* Second Edition, John Wiley & Sons Inc, 2005.
[5] V. T. Lizin, *Proektirovanie tonko-stennih konstruktii,* Moskva, Masinostroenie, 1985 V. A. Piatkin.