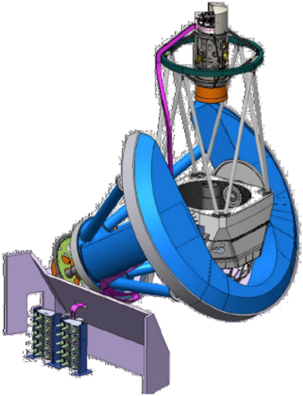
Final Review: DESI Experiment Barrel Design

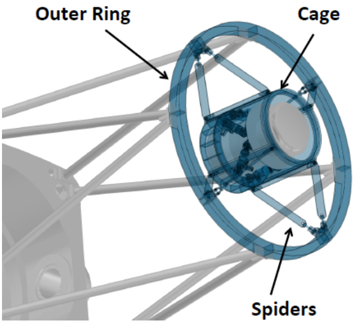
Luca De Vincentiis

University of Pisa

September 24, 2015

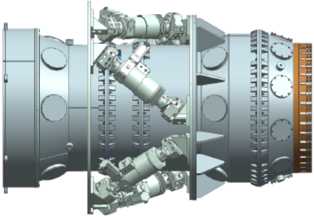
DESI experiment will run in the 4 meter telescope in Kitt Peak known as the Mayall and Fermilab has been charged with designing and building the structure that will support the optics and the fiber positioners and with rebuilding the top end of the telescope.

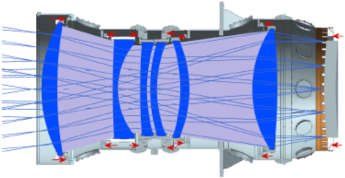




Introduction

All my tasks are related to the design of the barrel. The barrel is the scructural part of the Mayall telesope which has to support the lenses



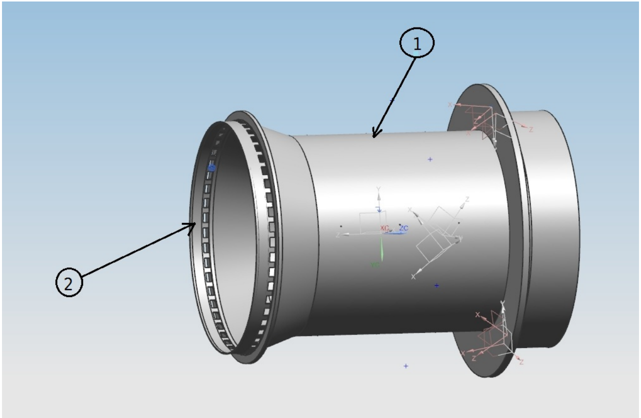


In particular I developed the following tasks:

- Cell design and FEM analysis for Test Weight
- Preliminary evaluation of stress due to Hertzian contact.

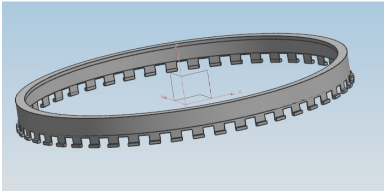
Cell design and FEM analysis

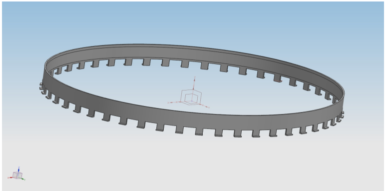
The goal of this task is to design a cell which will be used in the structural tests in which it will be evaluated the deformations of the barrel due to its weight. During the tests the barrel will be loaded in two sections with a load of 500 Kg.

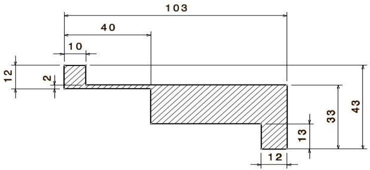


Cell design and FEM analysis

The cell, which will be used in the tests, has to have the same stiffness of the one that will be built on the telescope, but it has to be also less expensive to be manufactured.

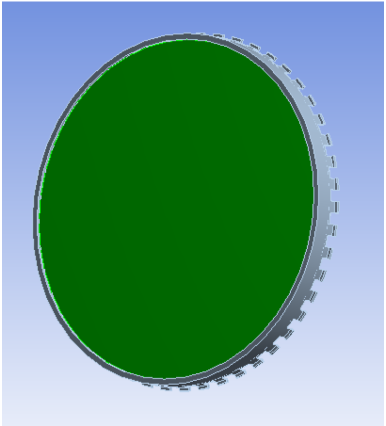


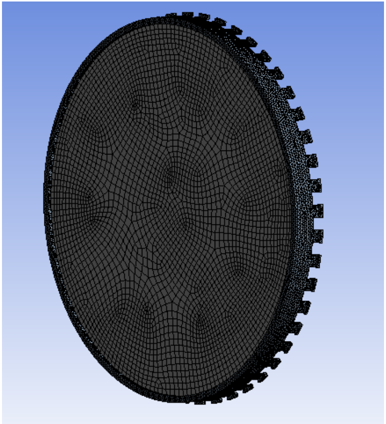




Cell design and FEM analysis

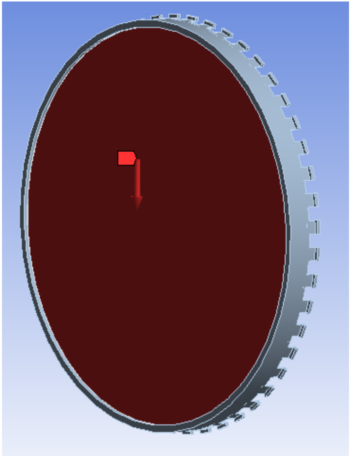
In order to understand how the different geometry of the celles affect the deformation of the barrel it is used two FEM analysis for each cells cases
The first model used for the analysis is shown in the following figure.

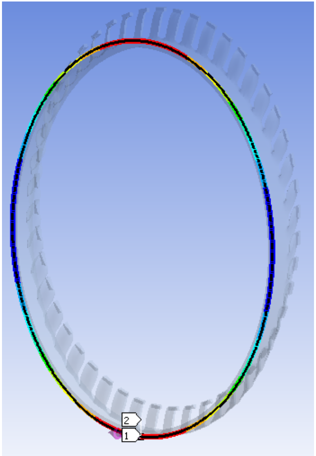




Cell design and FEM analysis

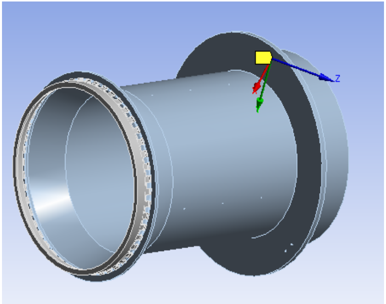
On the model is applied a force $F = 4900N$ and it is computed the displacement of the edge shown in figure

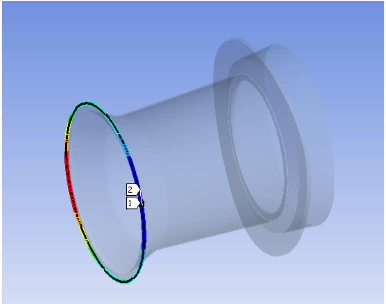




Cell design and FEM analysis

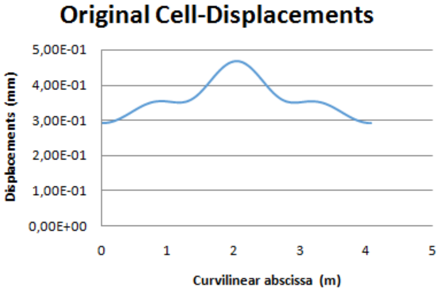
In the second model is present also the barrel. The following figures show the constraints and the displacements which are evaluated. The analysis has been performed for the two different cell and than the results has been compared

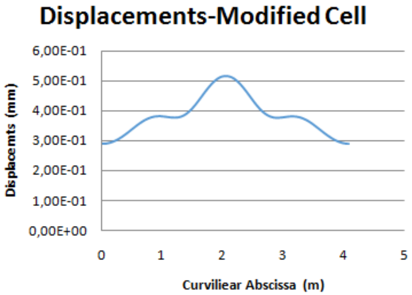




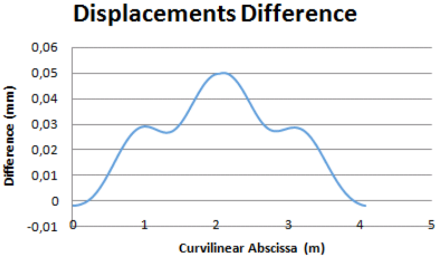
Cell design and FEM analysis

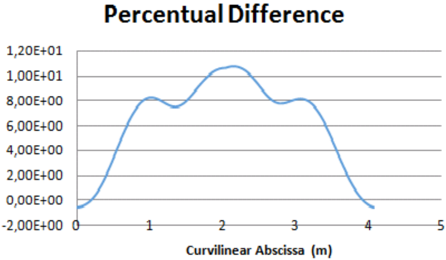
The following graphs show the values of the displacements both for the model with the original cell and with the modified cell.





Finally it is computed the difference between the displacements in the two models.

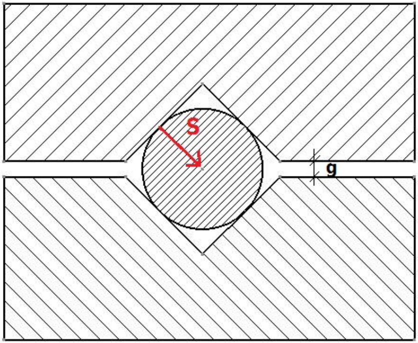




Thanks to this analysis it has been possible to define the geometry of the cell which will be used for the Weight Test.

Hertzian Contact

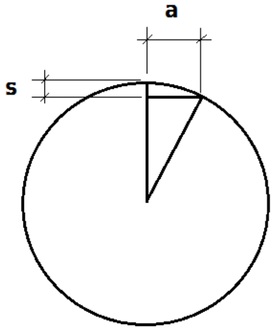
The goal of this analysis is to evaluate the stress due to the contact between the sphere and the flanges. The following images shows the model used for the analysis, where g is the distance between the flanges and s the displacement in the radial direction. The first computation is performed assuming that the flanges have no strain.



Hertzian Contact

$$s = \frac{\sqrt{2}}{2} \cdot g \tag{1}$$

$$\Delta = \frac{2(1-\nu^2)}{E} \tag{2}$$



Hertzian Contact

$$a = R \cdot \sqrt{1 - \left(1 - \frac{s}{R}\right)^2} \tag{3}$$

$$F = \frac{a^3}{0.908^3 \cdot \Delta \cdot R} \tag{4}$$

$$p = 0.578 \cdot \sqrt[3]{\frac{F}{R^2 \cdot \Delta^2}} \tag{5}$$

$$\tau_{max} \approx 0.3 \cdot p \tag{6}$$

With these equations is possible to compute the value of the force F necessary to put in contact the flanges and then the contact pressure p and τ_{max}

Hertzian Contact

The data used for the analysis are:

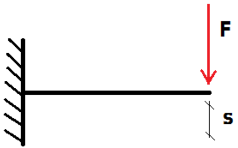
- $g = 50 \cdot 10^{-3} mm$
- $R = 7.5 mm$
- $E = 210 \cdot 10^{3} MPa$
- $\nu = 0.3$

And the results are:

- $F = 7900 N$
- $p = 7300 MPa$
- $\tau_{max} = 2200 MPa$
- $z = 0.35 mm$

Hertzian Contact

In this second model we make the hypothesis that the sphere has a very high stiffness compared to the flanges. Moreover a very simple model is used to evaluate the stress.
The flanges are compared to beams with the lenght equal to the distance between the sphere and the bolt and it is evaluated the force necessary to close the gap between the flanges.



$$F = \frac{3EJg}{2L^3} \tag{7}$$

where $L = 85mm$, $J = 18900mm^4$.

Hertzian Contact

The results are on this model are:

- $F = 490N$
- $p = 2900MPa$
- $\tau_{max} = 860MPa$
- $z = 0.14mm$

The first method gives a value of τ_{max} which is overestimated, while the second case gives a value of τ_{max} which is understimated.
In both cases the computed stress is greater than the yield strength of the material so, in the area of contact, there will be plasticity.
Moreover is possible to compute the area of contact:

- Case 1: $A = 1.3mm^2$
- Case 2: $A = 0.2mm^2$

The End