SIZING OF A METALLIC AND COMPOSITE COMBUSTION CHAMBER

TFMASA Course

COURSE UNIT SIMULATION AND DESIGN OF STRUCTURES-4TSA706U

Under

PROFESSOR CHRISTOPHE BOIS

By

SANGHVI, Mahek Atul (22210896)

TABLE OF CONTENTS

1. INTRODUCTION

Combustion chamber analysis is performed for hybrid propulsion system. It is made of a structure named nude combustion chamber used to support the internal pressure and a thermal protection used to fight against erosive gas flow higher than 2800 K.

2. OBJECTIVES

The design requirements are based on targeted functionalities and performances of the overall propeller. The main requirements are summed up below:

- Service pressure: 30 bar, pressure for sizing criterion: 60 bar,
- Compression load applied on skirts: 8000 N,
- Minimal volume: 35 L,
- Maximal external diameter: 250 mm,
- Length: 840 ± 5 mm,
- Neat mass without thermal protection: 5,2 kg (with thermal protection: 9,7 kg),
- Exposition to combustion gas: temperature: 2800 K, duration: 5 s near the ferrule at the interface between the propellant and the thermal protection, 30s at others areas.

3. ASSUMPTIONS

For an analysis to run efficiently and to perform experiment close to real life world application there are some assumptions made which makes negligible perturbations in the analysis. The assumptions made are:

- The model made is entirely designed using axis-symmetric which is a 2d model valid for design for analysis since it is symmetric.
- The cap attached to the dome's is fixed and doesn't move freely during entire study.
- The composite layer made is directly attached to the ferrule making it rigid body as a tie constraint.
- Friction considered is very less near the tooth or any other parts attached to each other and Interactions made are smooth without much of friction or any other type of forces for the cap, and the tooth.
- The tooth holds the entire structure for holding the high pressure to support the structure.

- Pressure force acting on the internal walls is evenly distributed throughout the inner walls of combustion chamber.
- The materials used are homogeneous and evenly distributed throughout.

4. DESCRIPTION OF MODEL

4.1 DESIGN

There are 5 parts containing the front dome, rear dome, ferrule, cap and composite layer. As the entire model is cylindrical type hence, we use axisymmetric model which revolves and forms a 3D shape. The 5 parts include 2 caps for top and bottom dome, top dome, bottom dome and the composite layer. The parts design has been shown in the figures.

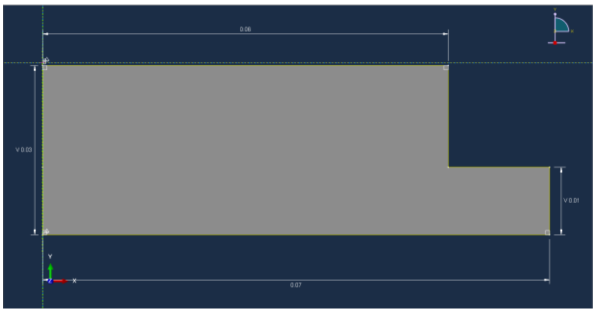


Fig. 1 Design of Bottom Cap

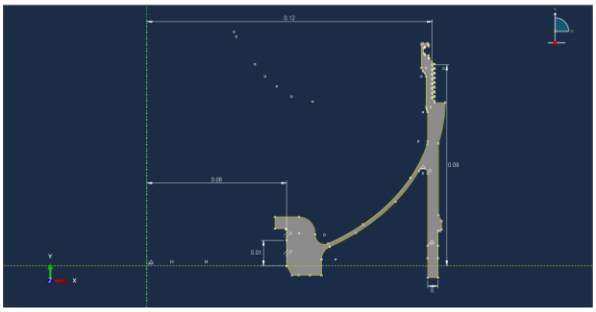


Fig. 2 Design of Bottom dome

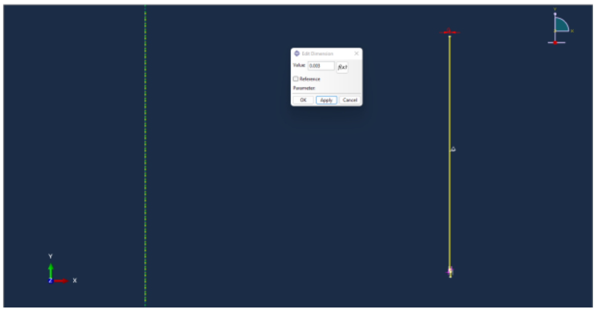


Fig. 3 Design of Composite Layer

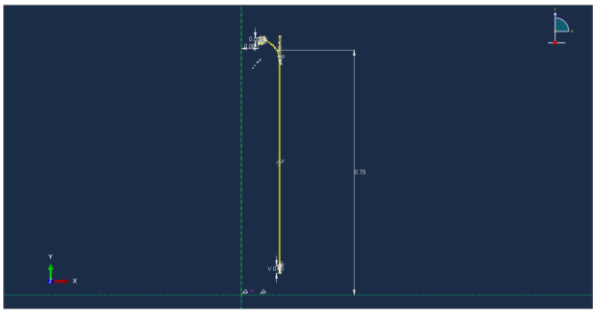


Fig. 4 Design of Top dome

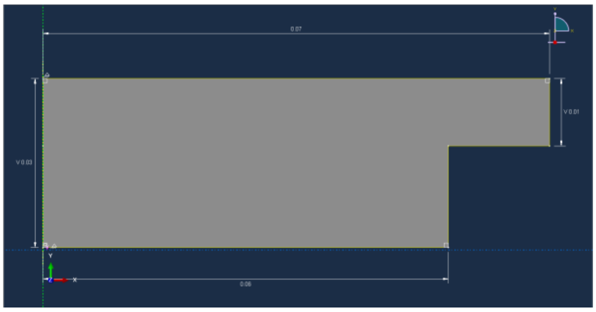


Fig. 5 Design of Top Cap

Fig. 1 shows the design of bottom cap, Fig.2 shows the design of bottom dome, Fig.3 shows the design of composite layer, Fig.4 shows the design of front dome and Fig.5 shows the design of top Cap.

4.2 MATERIAL

The material used to model the ferrule, caps and the rear dome is Aluminium alloy AL5083 H111 due to its good characteristics for welding. This material also has plastic behaviour due to which when unloading occurs there might be some energy loosed due to plasticity. The material used for the front dome is Al 7075 T651 due to its high strength. The aluminium alloys used are isotropic in nature. The caps have been assigned the material Al2024. The material for the composite layer is a composite having its fibre direction in z-direction and properties need to be aligned according to this axis also it is an orthotropic type of material in nature. The structure is to be designed in cylindrical shape with different materials based on mechanical properties required for the chamber to handle pressure load on its wall without it being leaked even at high temperatures and feasible to use those parts in a simulation for optimum results. The entire study has been done in statics for the preliminary study of structures to validate the model and various constraints.

The combustion chamber is made of several materials for and having different materials for each part. The details of materials have been tabulated in Table 1.

Table 1: Material properties Used in the model

1. Al2024	
Young's Modulus	Poisson's Ratio
73100000000 Pa	0.33
2. Al 5083 H111	
Young's Modulus	Poisson's Ratio
68224000000 Pa	0.33
3. Al7075 T651	
Young's Modulus	Poisson's Ratio
71100000000 Pa	0.33

4. Composite: assigned to composite

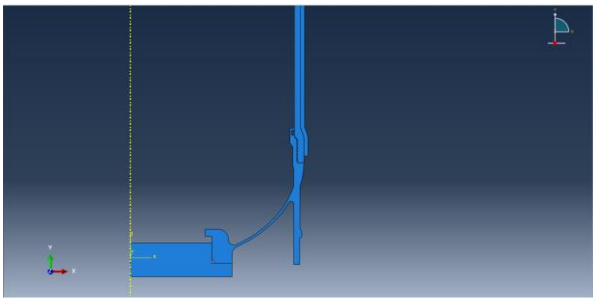
E1	E2	E3	Nu12	Nu13	Nu23	G12	G13	G23
7.6 GPa	7.6 GPa	163.1 GPa	0.34	0.016	0.016	2.6 GPa	3.8 GPa	3.8 GPa

4.3 MODELLING

The part has been created and sketch is imported from the cad files for the ferule, front and rear dome. The sketch of the composite layer and cap is created using the drawing sheet to perform the sketch. The cap has to be made on each end hence we need to create 2 parts and invert one of them. After creating parts, all the parts need to be assembled. To assemble by keeping some points align with other parts points or by using option of joining the edges of the parts together in the assembly to assemble.

4.4 ASSEMBLY

The model has different parts such as ferrule, cap's, front dome, bottom dome and composite layer. The Fig. 6 and Fig. 7 shows the assembly of all the parts of combustion chamber.



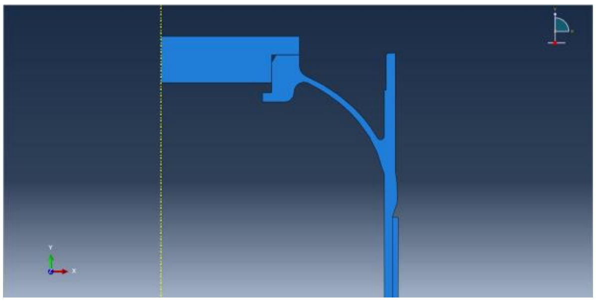


Fig. 6 Assembly of top Dome, cap & composite **Fig. 7 Assembly of Bottom dome, Cap**

4.5 INTERACTIONS

To analyse any type of assembly interaction plays an important role to define the motion of each against the other when various forces or displacement is acting on the system.

There are 2 types of interaction has been used for this model. A constraint and interactions used in this model in Abaqus.

The constraints applied are:

- A tie contact constraint is applied between the composite and the ferrule as both are fixed together.
- A tie contact between the cap's upper surface which will avoid it to open the system.

The interactions made are:

- Surface-to-surface penalty contact with friction in the thread region of the front and rear dome and we apply a friction co-efficient of 0.2.
- Surface to surface hard contact which allows free transition in the cap sides which allows to open the caps.

The constraints and interactions can be observed in the Fig. 8 and Fig. 9 respectively.

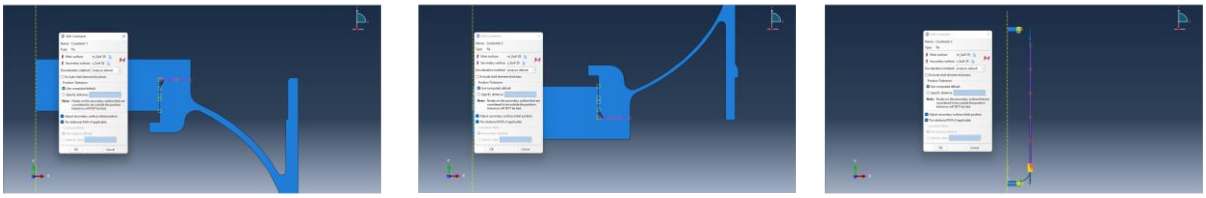


Fig. 8 Tie Constraints in cap's upper surface and composite layer to the ferrule

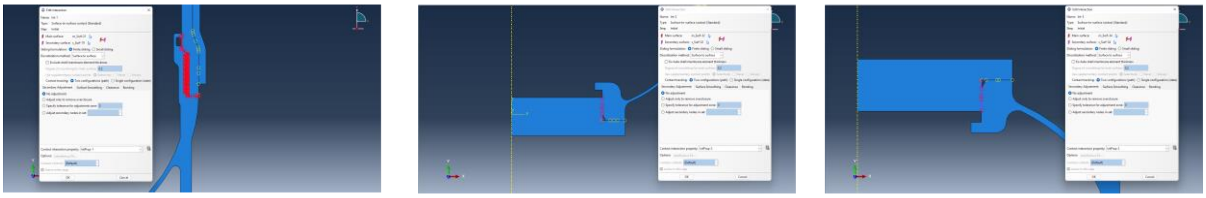


Fig. 9 Interactions in tooth with friction and hard contact between cap's side surfaces and domes

4.6 MESH

The Structure has been meshed with global size of 0.001 and near the tooth due to its complex geometry has been given mesh size of 0.0001 with triangular mesh type to give better results.

4.7 LOADING AND BOUNDARY CONDITIONS

A pressure load is applied in inner part of the structure which it must bear to maximum of 6Mpa or 60 bar for the case 1 and for individual study a pressure 100 bar or 10 MPa. Also surface traction force of 8000N has been applied on the outer skin edge of the ferrule at the cap's ends. Entire geometry in real life is not fixed but in simulations the software displaces entire body due to which the front dome is given a boundary condition at a point which lies in the central axis has been encastered.

5. DESCRIPTION OF RESULTS

The combustion chamber has been designed and the results have been run. The results obtained are based on the sizing criteria to predict the failure and feasible design of the structure.
The Fig. 10 shows the results of the analysis Mises in front dome, stress S11 in welding area, strain E11 in composite, Plastic strain in the ferrule and the displacement between seal and ferrule.

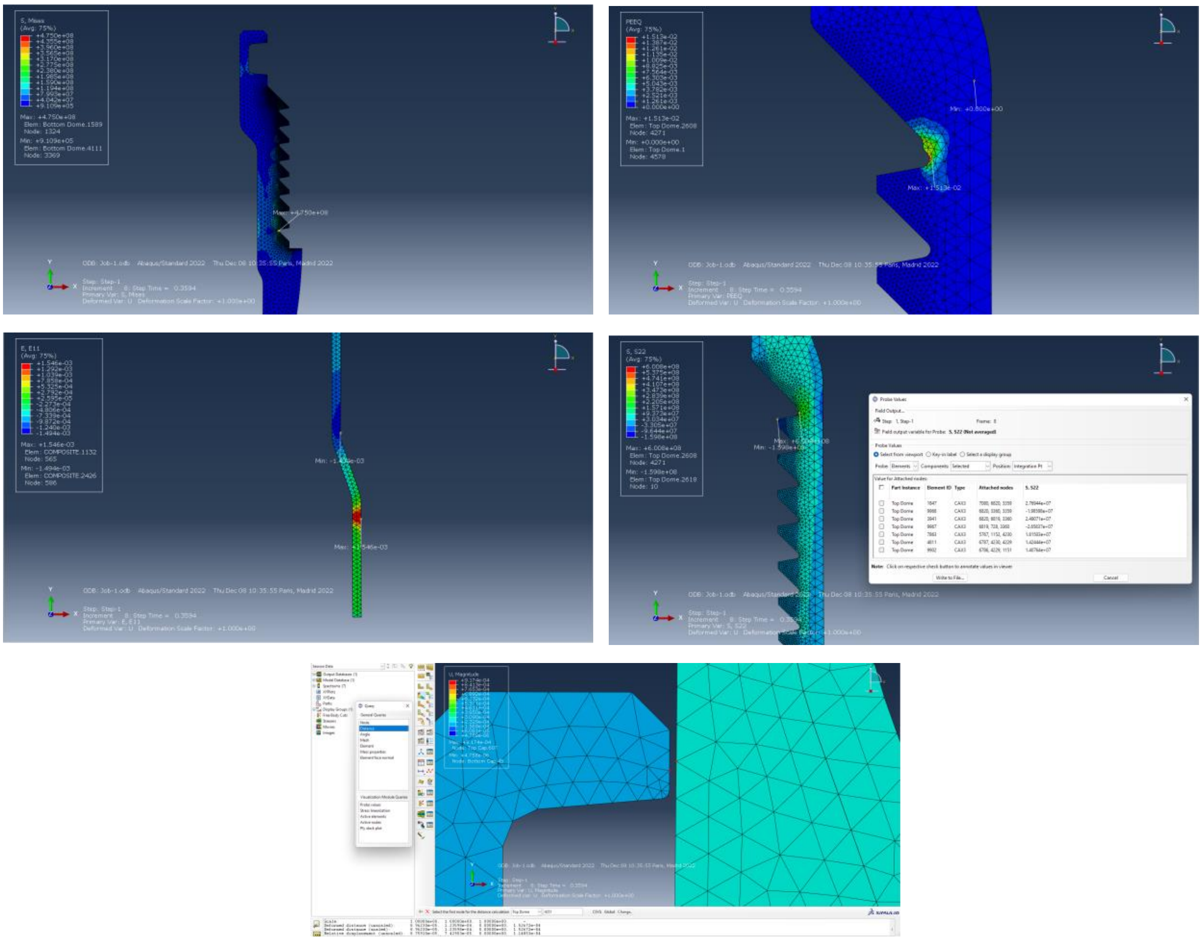


Fig. 10 Contours of Mises in front dome, stress S11 in welding area, strain E11 in composite, Plastic strain in the ferrule and the displacement between seal and ferrule respectively

The Stress obtained in front dome is little higher than the sizing criteria which shows that front dome has to be given some strength by adding material or adding supporting structure to resist that stress easily. The Strain obtained in the composite structure is feasible a per the sizing criteria which make worth of adding support to ferrule for less weight. The Plastic strain in the bottom dome is very close to the value needed to be obtained based on the sizing criteria which will allow it to regain its original shape even after high stress concentration. The Stress obtained in the welding region is also very good in response to the sizing criteria which allows it to have less stress where manufacturing process can add a weld to join the dome and the ferrule. The Displacement between the seal and ferrule I also very less hence will avoid it to fail during maximum pressure.

6. SIZING CRITERIA

There are 2 failure modes of the structure which are:

1. Burst failure due to the internal pressure

To check for burst condition the values of the list below must be satisfied.

- Strain in fibre direction of the composite must be lower than the strength limit.
- von Mises stress must be lower than the elastic limit of the material of the front dome.
- Cumulated plastic strain in the ferrule must be lower than 1%.
- Longitudinal stress in the ferrule in the welding area must be lower than the strength.

2. Leakage of hot gas Sizing criteria associated to failure mode (a) are:

To check for leakage, condition the values of the below must be satisfied.

- Gap in front of the seal between the front dome & the ferrule must remain lower than 1%.

In the above case we can see that all the results obtained are satisfying the sizing criteria, and are also satisfying the failure modes, these results conclude that this model can further be used for the individual case study, i.e., the strain in the fibre direction of the composite is less than strength limit of 1.7%, the mises stress in the front dome is less than its elastic limit of that material which is 502 MPa, longitudinal stress near the welded region is less than the strength limit defined from dedicated specimens based on the material provide to the ferrule which is 250 MPa, Cumulative elastic strain is less than 1% and the other case of failure where the gap between ferrule and front dome should be less than 0.3mm.

7. INDIVIDUAL STUDY

Individual study includes changing the material to titanium TA6V for the ferrule in order to support an internal pressure of 100bar.

The material Titanium TA6V is Elastoplastic by nature which has elastic and perfectly plastic properties meaning material having same stress acting for different values of strain which is also called as No Hardening shown in the table 2.

Table 2: Perfect Elastoplastic Properties of Titanium TA6V

Elastic Properties	
Young's Modulus	**Poisson's Ratio**
110000000000	0.3
Plastic Properties	
Yield Stress	**Plastic Strain**
980000000	0
980000000	0.5

7.1 PROCEDURE

The entire procedure performed is similar but there were some changes made to match the final result for sustainable and good design based on criteria developed to be followed.

The material to be changed as Titanium TA6V for the ferrule and the pressure of 100 bar which is around 1×10^7 Pa.

As the changes were made and the simulation were performed but the results observed showed a very high value in stress which could break the front dome, high plastic strain and the relative displacement between the seal and the ferrule increased due to which many criteria for design change was thought of with the reasons are mentioned below:

1. Increasing the diameter of the cap: Rejected method

Increase in diameter of cap can increase the surface area which might make entire structure of the rocket or aeroplane more complicated with more size

It can also have less pressure chamber which can burst due to external pressure.

2. Increasing the thickness of the ferrule: Accepted method

Increasing ferrule size will add little bit of material which can add some weight but will make structure feasible for the purpose.

3. Increasing the thickness of the composite: Accepted method

Adding composite will not add much of weight but might solve the problem of high stress generation.

4. Changing the material of the front dome: Rejected Method

As per the requirement we do not have the option to change the material as per our requirements. Hence changing material is not an option to be considered.

5. Increasing the size of tooths: Rejected Method

As the tooths are designed very complicatedly and further size increment will not only make manufacturing difficult but also increase size of all the parts.

6. Increasing the friction co-efficient.

Friction Co-efficient as a structural engineer can try upon but not predict the exact value if not provided. And hence has to go through process of increasing and analysing each and every time which increases computation time and also not guarantee of optimum results.

7. Addition of stiffeners to support the structure: Rejected method

Stiffeners make structure more stable but decrease the area, complex design and more material and weight hence it had to be rejected.

7.2 STEPS PERFORMED FOR THE INDIVIDUAL STUDY

1. Material of Titanium TA6V had replaced with the previous aluminium alloy.
2. Pressure had been increased to 100 bar.
3. The thickness of composite was increased from 0.1mm to 0.2mm and later to 0.3 mm. Further increase to composite was difficult from design perspective and also composite would become weaker as the fibres are perpendicular to the plane of the chamber.
4. Even after increasing the thickness of the composite optimum results as required could not be obtained.
5. The thickness of the ferrule was increase from 2mm to 5mm and finally to 8mm which is a very higher value and due to this high thickness plastic strain had become negligible but still the stress in front dome could not be achieved as per requirements but other results could be achieved with increase up to 5mm itself.
6. Due to increase in the ferrule diameter the surface traction force acting had to be changed since Abaqus take surface traction force with Force per unit area.

The results for 0.5mm and 0.8mm have been tabulated in Table 3.

Table 3: Design Criteria obtained after analysis

Parameters	5mm Ferrule	8mm Ferrule
Strain-E11: In Composite	0.5919%	0.2434%
Stress-Von Mises: In Front Dome	1890 MPa	1597 MPa
Plastic Strain: PEEQ: In Bottom Dome	0.2148%	0%
Stress-S11: In Welding Area	6-10 MPa	6-10 MPa
Distance-U: Between Seal & Ferrule	0.00576675mm	0.00386685mm

Some Figures have been shown of the analysis done for the 0.8mm ferrule. The Fig. 11 shows the Strain E11 in the composite part. The Fig. 12 shows the Von Mises stress in the front dome. The Fig. 13 and Fig. 14 shows the Plastic strain PEEQ in the front dome for 0.5 mm and 0.8 mm ferrule thickness since it is the only part containing plastic properties. Fig. 15 shows the stress S11 in welding area. Fig. 16 shows the distance between seal and ferrule.

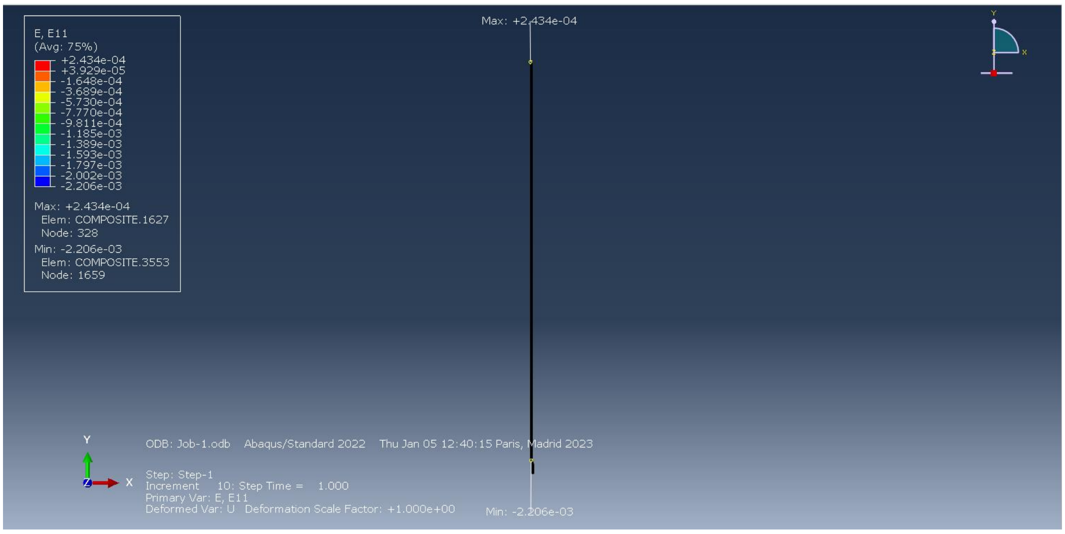


Fig. 11: Strain-E11: In Composite

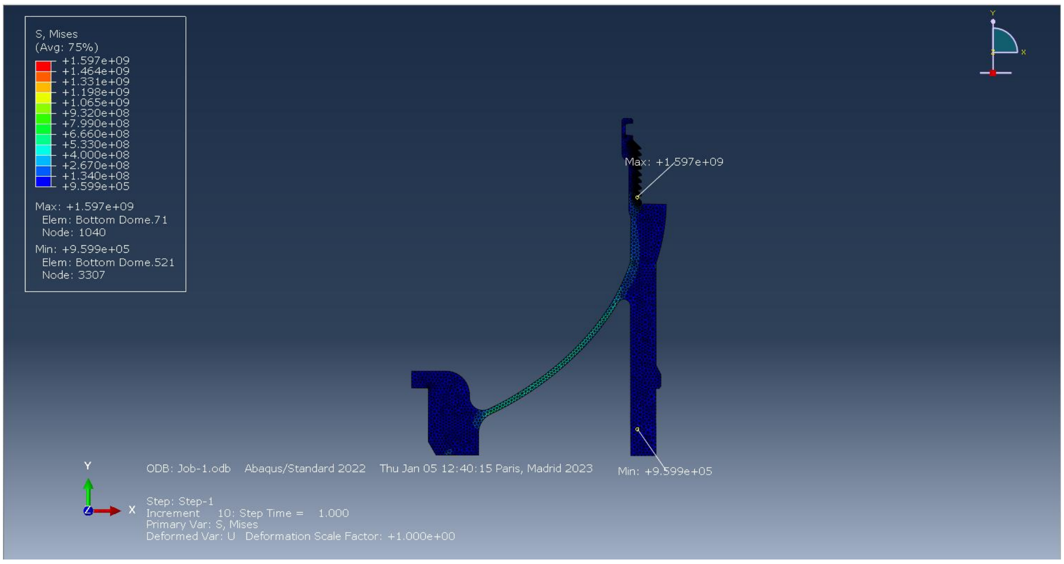


Fig. 12: Stress-Von Mises: In Front Dome

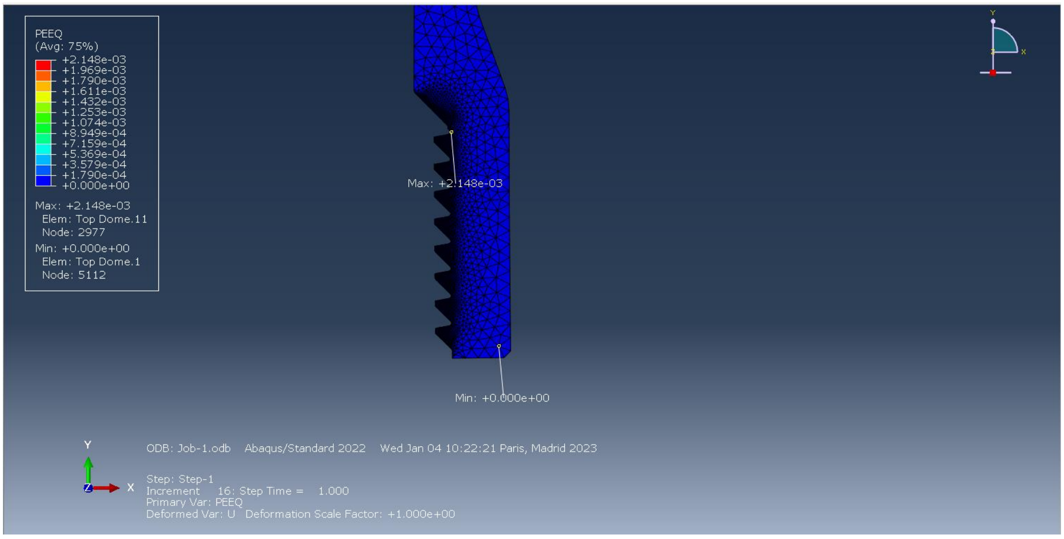


Fig. 13: Plastic Strain: PEEQ: In Bottom Dome for 0.5mm thickness of ferrule

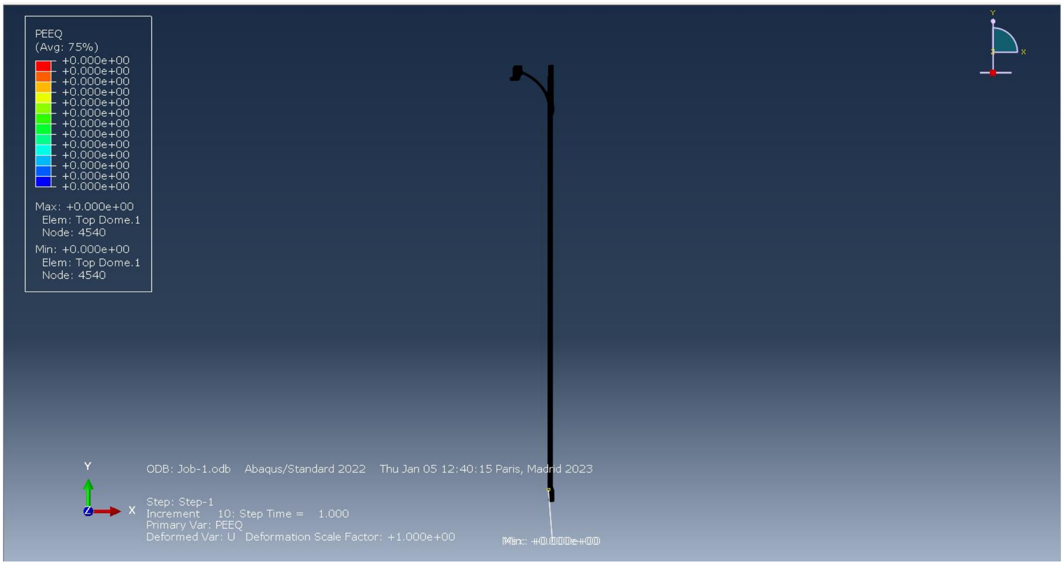


Fig. 14: Plastic Strain: PEEQ: In Bottom Dome for 0.8mm thickness of ferrule

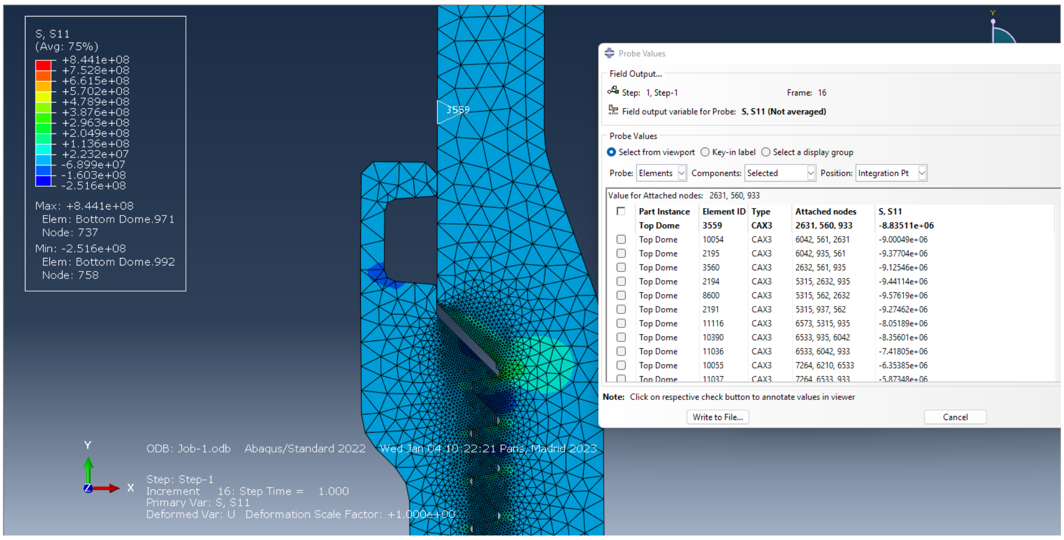


Fig. 15: Stress-S11: In Welding Area

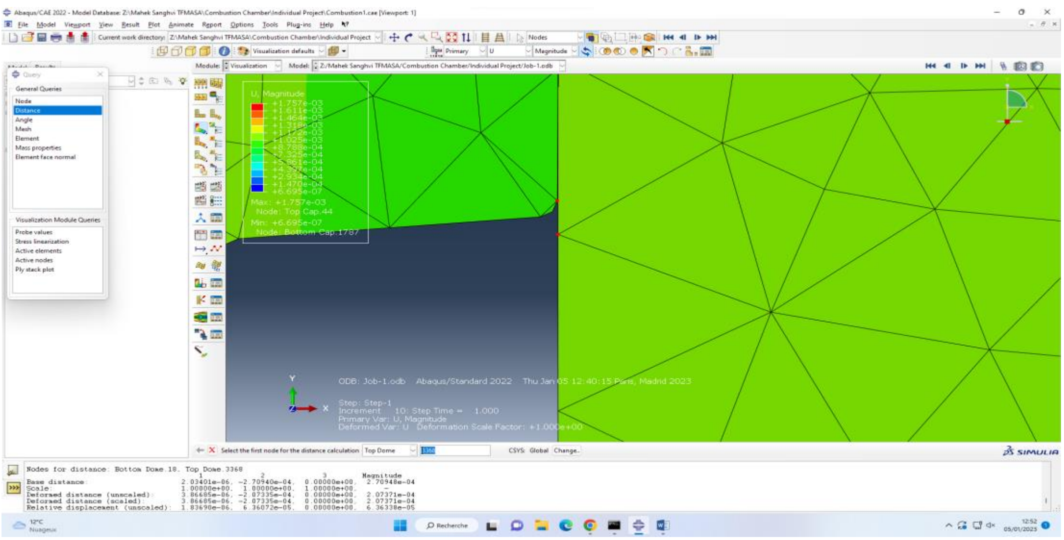


Fig. 16: Displacement between seal and ferrule

8. CONCLUSION

Analysis of a complex design like a combustion chamber analytical solving is very much not possible. Using simulations and fixing the sizing criteria a designer can perform the design and give the results accordingly through model design and analysis. In this study using the interactions, loads and boundary conditions we reach very similar to the results of real life. In the common study we could match most of the sizing criteria requirements and perform very good and close results.

Similarly, an individual study was performed by changing the material of ferrule to titanium which is a perfectly elasto plastic material and changing pressure to 100 bar. An optimization has been made to get results very close to sizing criteria requirements by changing the thickness of the composite and ferrule thickness through steps. The results match very similar to the sizing criteria though some more addition the model analysed can be validated and manufacturing can be possible.