DOI: 10.13009/EUCASS2022-7281

Thermo-mechanical simulation of a small-scale liquid hydrogen fuel tank for aviation applications

George Lampeas and George Tzoumakis
Mechanical Engineering and Aeronautics Department
University of Patras Greece
labeas@upatras.gr, georget1012.gt@gmail.com

Abstract

Hydrogen is currently considered as a promising alternative fuel for the future. Provided that the transport segment comprises about one-third of the total CO_2 emissions in the European Union, hydrogen is promising for greenhouse gas reduction achievement. Specifically for the aviation sector, hydrogen when produced carbon-free, presents several advantages, as it allows for the elimination of CO_2 emissions in flight. Its usage in fuel cells allows for zero-emission propulsion, including NO_x and particles. When burnt in a turbine engine, exceptionally low particle emissions can be expected, as well as reduced NO_x emissions, provided that the combustion system is optimized.

A typical investigation of the feasibility of liquid hydrogen as aviation fuel is, among others, the Cryoplane project [1]. The key elements of the liquid hydrogen commercial aircraft technology are still at very low Technology Readiness Level and for this reason, advanced parametric modelling is required to support an optimized tank design. In this direction, the present work refers to the development of a detailed thermo-mechanical simulation model of a small-scale liquid hydrogen fuel tank for aviation applications. A tank of a typical capacity of about 100 kg of liquid hydrogen, assuming a mean density of the liquid hydrogen equal to 70.85 kg/m^3, at 2 bar design pressure, 15 bar design overpressure and 20K design temperature is investigated. The multi-parametric model developed allows different design geometries to be modelled, yet a cylindrical tank with hemispherical heads is considered.

The liquid hydrogen tank insulation must meet conflicting requirements, as it has to demonstrate very low heat losses in order to meet the boil off requirements, while at the same time to be a light-weight system, fulfilling the tank mass requirements. Considering the low operating pressure, aluminum alloy materials are used in the tank construction, as composites could lead to marginal mass reductions, at a high-cost penalty. Such a selection is in line with NASA studies e.g. [2], suggesting that aluminum alloy 2219 fulfils all requirements of a LH_2 tank design. For the insulation system a spray on foam based on polyurethane materials is currently considered.

The developed thermo-mechanical simulation methodology accounts for all the distinctive and critical parameters regarding the simulation of cryogenic metallic hydrogen tanks. The model comprises a thermo-mechanical module to account for temperature history effects on the mechanical response and a structural module to account for thermo-mechanical stress analysis. Modelling approaches combining solid elements with thin-shell element approaches are applied and parametric studies for different tank variations having variable wall thickness of the double-walled aluminum vessel and variable spacing between the walls for introduction of the foam thermal insulation have been successively performed. The developed model is used to draw conclusions about the structural performance of the tank and its mass efficiency as function of its sizing parameters.

1. Introduction

Liquid hydrogen tanks (which are usually made of cryogenic resistant materials such as aluminum alloy, nickel steel, or stainless steel) and their supporting structures (which are made of cryogenic resistant materials) are required to be designed for application in the liquefied hydrogen aircraft. To prevent excessive thermal stress at critical locations in these tanks, which are stiffened by internal members such as stringers, frames, and stiffeners, it is essential to carry out Finite Element heat transfer analysis and then thermal stress analysis to verify the strength of tank structures.

DOI: 10.13009/EUCASS2022-7281

Furthermore, Boil off gas (BOG) is caused by heat transfer into the liquefied hydrogen tank during its operation; the amount of BOG depends on the design and operating conditions of liquefied hydrogen tank.
Provided the numerous design parameters of a future civil aviation tank, the design process should be heavily based on parametric model simulations. In this frame, the objective of the present paper is to set the basic framework for the development of a thermo-mechanical simulation model of a small-scale liquid hydrogen fuel tank for aviation applications. The geometry, materials, loading and boundary conditions used in the tank model development are described in chapter 2. In chapters 3 and 4, details of heat transfer and thermo-mechanical analysis and their corresponding results are presented.
The heat transfer analysis, is performed using the finite element (FE) method, followed by FE thermomechanical stress analysis, for the strength evaluation. The methodology developed for performing steady-state heat transfer analysis for the calculation of temperature distributions and the corresponding heat transfer coefficients of the tank is presented. The predicted temperature distribution in the tank structure is used for the estimation of the boil-off rate. Consequently, thermomechanical FE stress analyses of tank and its supporting structures is performed. Design thermal loads for various loading cases including partial filling and full loading are considered together with the other mechanical loads, to determine stress distributions and to evaluate the tank structural integrity against material failure and buckling of tank structure. Conclusion and future work proposals are provided in the final section of the paper.

2. The small-scale liquid hydrogen fuel tank description

The multi-parametric model developed allows the creation of radially symmetric tanks of different geometries. The supporting structure can also be adapted as needed. Almost every design aspect is a changeable parameter, simplifying design revision or different design creation without major changes in the model.
For the present analysis, an independent cylindrical tank with spherical end caps is considered. Total length of the inner tank is 2 m with a maximum diameter of 1 m. The tank has a maximum capacity of 1308 lt, or 92.6 kg of liquid hydrogen assuming an average density of 70.85 kg/m^3. A layer of Cryogel-Z aerogel 25 cm thick is placed between the two walls, meaning that the outer tank has a length of 2.5 m with a maximum diameter of 1.5 m. The two tanks are made from aluminum alloy 2219, with the inner tank having a thickness of 3/32'' (2.38125mm) and the outer 1/32'' (0.79375 mm).

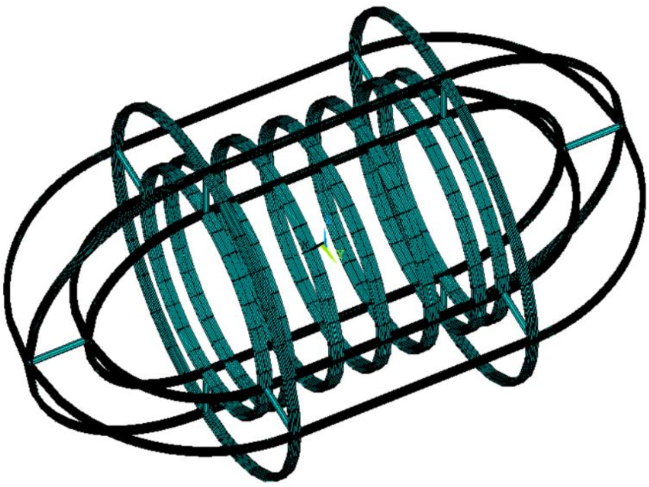

Figure 1: **Supporting Structure of the Modeled Tank**

Both tanks are supported by 4 longitudinal beams. The outer tank is supported by 2 rings while the inner tank is supported by 7 rings due to the higher load of the overpressure. All beams are made of aluminum alloy 2219 and have an I cross section, with the geometric characteristics presented in the following table.

		Flange width (mm)	Flange thickness (mm)	Web depth (mm)	Web thickness (mm)
Inner tank	Rings	50	2	25	2
	Beams	25	2	25	2
Outer tank	Rings	40	2	20	2
	Beams	30	2	20	2

Table 1: **Geometrical characteristics of the modeled tank**

DOI: 10.13009/EUCASS2022-7281

The inner tank is supported by 10 PEEK tubes of 25 mm in diameter and 2.5 mm of thickness

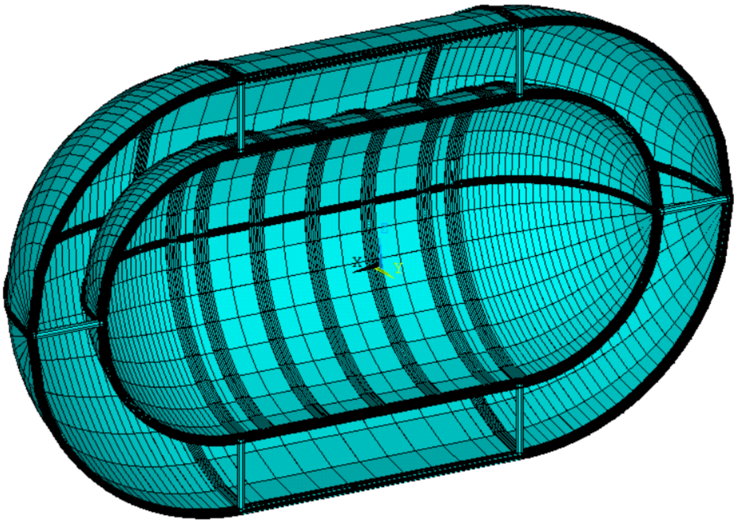

Figure 2: **Section of the Modeled Tank Showing the Tank Walls and the Supporting Structure**

The tank component masses are presented in the following table.

Component	Mass [kg]
Inner tank wall	42.373
Inner tank supporting structure	19.235
Outer tank wall	26.477
Outer tank supporting structure	10.923
PEEK tubes	1.036
Insulation	93.886
Total	193.933

Table 2: **Component Masses of the Modeled Tank**

The material properties that the model uses as input are summarized in the following table.

	Aluminum 2219-T62	PEEK	Cryogel-Z
Density	2840 kg/m^3	1320 kg/m^3	16 kg/m^3
Thermal Conductivity	120 W/mK	0.26 W/mK	0.007 W/mK
Thermal Expansion Coefficient	$2.25\cdot10^{-5}$ K^{-1}	$5\cdot10^{-5}$ K^{-1}	10^{-6} K^{-1}
Specific Heat Capacity	864 J/kgK	2.1 KJ/kgK	1.9 KJ/kgK
Modulus of Elasticity	73.1 GPa	3.5 GPa	11.26 KPa
Poisson’s Ratio	0.33	0.38	0.24
Maximum Allowed Stress	290 MPa	90 MPa	34.5 KPa

Table 3: **Material Properties of the tank and the insulation materials**

3. Theoretical Heat Transfer Analysis

3.1 Theoretical Foundation

Environmental conditions inside the fuselage part where the tank is placed are not always defined, yet some accurate assumptions can be used for the heat transfer analysis. The process for heat transfer analysis, material selection, and boil-off rate (BOR) calculation is described below.

Heat transfer within the tank and its surrounding structure depends on thermal conduction, thermal convection, and thermal radiation. Each type of heat transfer results in a specific coefficient. A methodology to define their corresponding coefficients is discussed hereafter, based on ref. [4].

DOI: 10.13009/EUCASS2022-7281

The thermal conduction comprises the heat transfer that occurs across a medium when a temperature gradient exists and can be described by Fourier's Law as:

Q = (A k/ t) (To – Ti) where :

A = area of heat contact
To = outside surface temperature
Ti = inside surface temperature
k = thermal conductivity of material
t = thickness of plate
The thermal conductivity of the tank material can be considered as a constant. The thermal conductivity of multiple insulation layers can be calculated from the insulation system ingredients.
Thermal convection is heat transfer due to the bulk movement of molecules within fluids such as gases and liquids. Convection heat transfer is classified into two types: free convection and forced convection. Free (natural) convection occurs when the flow is induced by buoyancy forces arising from density differences within the fluid. Forced convection is generally caused by external flows. As the hydrogen tank is assumed completely enclosed into the aircraft fuselage, forced convection is not the case in the present heat transfer analysis.

The transition in a free convection boundary layer depends on the relative magnitude of buoyancy and viscous forces in the fluid. Its occurrence is generally correlated in terms of the Rayleigh number (Ra) and the Nusselt (Nu) number. The Rayleigh number is a product of the Grashof (Gr) number and the Prandtl (Pr) number. The Nusselt number (which is defined as a function of Rayleigh number) provides a measure of the convection heat transfer occurring at the surface. The Rayleigh number is expressed in the following equation:

$Ra_L = Gr_L Pr = \frac{g\beta(T_s - T_b)L_c^3}{\nu^2}$ where:

g = gravity acceleration
β = thermal expansion coefficient of the bulk fluid at the average temperature
ν = kinematic viscosity of the bulk fluid at the average temperature
Ts = surface temperature
Tb = bulk fluid temperature

The convection coefficient is expressed as a function of the Nusselt number in the following equation:

$h = \frac{Nuk}{L_c}$ where:

k = thermal conductivity of the bulk fluid

The Nusselt number calculation equation differs significantly for different geometries, yet the Nusselt number can be empirically calculated for the most common ones.

For the spherical caps, the empirical equation for spheres is used. When the Rayleigh number is below 10^{11} and the Prandtl number is above 0.7, the Nusselt number is expressed in the following equation:

$$Nu = 2 + \frac{0.589 Ra^{1/4}}{[1 + (0.469/Pr)^{9/16}]^{4/9}}$$

For the cylindrical body, the empirical equation for Horizontal cylinders is used. When the Rayleigh number is below 10^{12} the Nusselt number is expressed in the following equation:

$$Nu = \left\{0.6 + \frac{0.387 Ra^{1/6}}{[1 + (0.599/Pr)^{9/16}]^{8/27}}\right\}^2$$

For the Rayleigh number calculation, the characteristic length is the diameter of the outer tank for both cases.

As far as thermal radiation is concerned, as the temperature of the outer tank is assumed to be similar to that of the surrounding parts of the fuselage, the heat transfer due to radiation can be ignored.

The heat flux into the hydrogen tank generates boil-off gas. This results in the pressure increase inside the tank and the boil off gas needs to be released to maintain the tank design pressure. Therefore, it is necessary to assess the BOR regarding the tank insulation system. With the assumption that all heat fluxes from outside into the inside of the tank generate BOG, the daily BOR can be estimated by the following equation:

BOR = Q / (ρ H V) × 3600 × 24 × 100% where :

DOI: 10.13009/EUCASS2022-7281

Q = total heat flux from outside to inside tank
ρ = fluid density
V = tank volume
H = latent heat for vaporization

For each individual fuselage, the tank geometry installed in the specific fuselage structure need to be identified and all data should be input. Environmental conditions, such as air temperatures and inner tank temperature need to be defined before any calculation. In the calculations of convection coefficients, the values of thermo-physical properties of the air such as conductivity, viscosity, and diffusivity are highly temperature dependent. Those values at a specific temperature can be approximated using linear interpolation.
Based on the above theoretical foundation, a simplified heat transfer analysis can be performed to estimate the temperatures and corresponding heat transfer coefficients the tank and its surrounding structure, as well as to calculate the boil-off rate.
To demonstrate the analytical heat transfer tank procedure a simplified tank is considered, as per the figure 3.

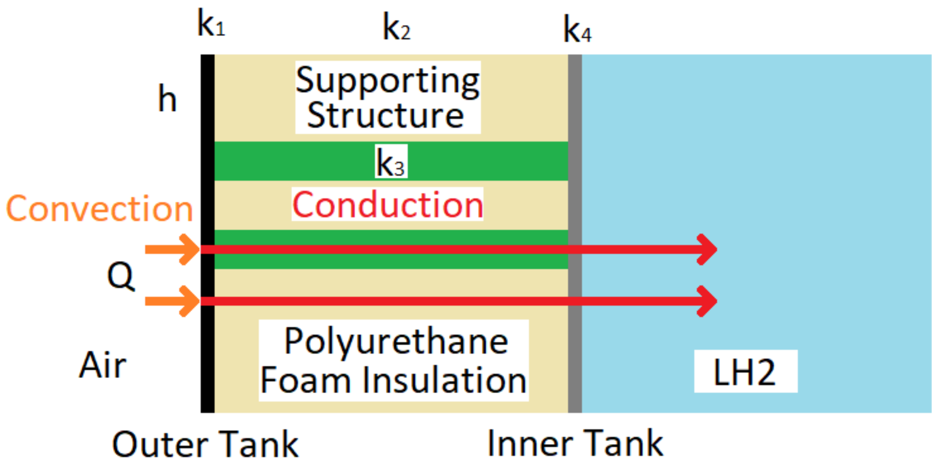

Figure 3: **Simplified Analytical Heat Transfer Model for the cryogenic Tank**

3.2 Theoretical Calculations

For the theoretical calculation of the heat losses, the tank will be divided in 3 sections, the two spherical caps and the main cylindrical body. For each section, the thermal flux will be calculated first. In order to simplify the calculations, only conduction through the insulation layer will be considered. Any losses due to conduction through the PEEK tube is not currently taken into account in the current simplified approach, although it can be significant. The theoretical calculations assume one dimensional heat transfer, ignoring any effects the curvature may have, therefore the theoretical values for heat flux and boil off rate need further detailing to become more accurate.
As described by Fourier’s Law, thermal flux is equal to:

$$\frac{Q}{A} = \frac{T_{out} - T_{in}}{L/_{k}}$$

The quantity L/k can also be referred to as thermal resistance (R_{th}). For this analysis, the equivalent thermal resistance will be calculated:

$R_{th} = \frac{t_{in}}{k_{Al}} + \frac{t_{ins}}{k_{puf}} + \frac{t_{out}}{k_{Al}} + \frac{1}{h}$ where:

t_{in} = inner tank thickness
t_{out} = outer tank thickness
t_{ins} = insulation thickness
k_{Al} = thermal conductivity of aluminum
k_{puf} = thermal conductivity of polyurethane foam
h = convection coefficient

Since the convection coefficient calculation requires the outer tank and air temperatures as an input, the assumption of the air temperature inside the fuselage to be around 27ºC or 300 K was performed. The outer tank surface temperature is not expected to drop below 0ºC (273 K). For the assumed temperatures and the current geometry the convection

DOI: 10.13009/EUCASS2022-7281

coefficients are h=8.52 for the cylinder and h=4.38 for the spherical caps. After a short study, it was found that for a temperature range around the expected surface temperatures of the outer tank (273-300 K) the convection coefficient values remain almost constant, therefore there is no need for a convergence study.
The thermal resistance of the cylindrical part is:

$$R_{th_c} = \frac{0.0015875}{120} + \frac{0.25}{0.007} + \frac{0.00079375}{120} + \frac{1}{8.52} = 35.83\ \frac{m^2K}{W}$$

The thermal resistance of the spherical caps is:

$$R_{th_c} = \frac{0.0015875}{120} + \frac{0.25}{0.007} + \frac{0.00079375}{120} + \frac{1}{4.} = 35.94\ \frac{m^2K}{W}$$

The thermal flux of the cylindrical part is:

$$\frac{Q}{A} = \frac{300-20}{35.83} = 7.815\ \frac{W}{m^2}$$

The thermal flux of the spherical caps is:

$$\frac{Q}{A} = \frac{300-20}{35.94} = 7.793\ \frac{W}{m^2}$$

The for the heat flux, the area of each section has to be calculated.
The area of the cylindrical part is:

$$A_c = 1 \cdot \pi \cdot 1.5 = 4.710\ m^2$$

The area of the cylindrical part is:

$$A_s = 4 \cdot \pi \cdot 0.75^2 = 7.065\ m^2$$

The heat flux from the cylindrical part is:

$$Q = 4.710 \cdot 7.815 = 36.8\ W$$

The heat flux from the spherical caps is:

$$Q = 7.065 \cdot 7.793 = 55.0\ W$$

Total heat flux is 91.8 W
With the latent heat of vaporization of hydrogen equal to 447 KJ/kg, the boil-off rate is:

$$BOR = \frac{Q}{\rho V H} \cdot 3600 \cdot 24 \cdot 100\% = \frac{91.8}{70.85 \cdot 1.308 \cdot 447000} \cdot 3600 \cdot 24 \cdot 100\% = 19.2\%\ per\ day$$

4. Development of thermo-mechanical simulation model of the hydrogen tank

The cryogenic hydrogen tank simulation ideally comprises an integrated multiphysics, multiscale simulation model of the as-built tank. The model should include structural, thermal and fluid dynamics modelling. In addition, details of the as-built tank should ideally included, which means that all material, structural and systems characteristics, possible defects, manufacturing anomalies, etc. should be taken into account. However, fluid dynamics (sloshing) and as-built defects of the tank will be integrated in a future simulation model upgrade; therefore, the current model focus in thermal and thermo-mechanical analysis, the overview of which is presented in Figure 4.
The analysis will be divided in two procedures, one thermal and one thermomechanical, with the individual processes described below.

4.1 Finite Element Heat Transfer Analysis Procedure

4.1.1 Heat Transfer Analysis model

Following to the simplified thermal analysis of the tank can, based on the theoretical formulations described in section 2, a Finite Element (FE) heat transfer analysis is performed for a more accurate temperature distribution calculation of the tank and especially of its supporting structure. FE heat transfer analysis involves the analysis of conduction and convection, which are the most relevant heat transfer mechanisms of the tank and its surrounding structure, to calculate the temperature distributions at the structural members. The results of the FE heat transfer analysis can be used to determine temperature distributions for the consequent FE thermal stress analysis. It can also be used to further guide material selection of the tank and its supporting structure. A flowchart of FE heat transfer analysis for the simplified tank is provided in Figure 5.

DOI: 10.13009/EUCASS2022-7281

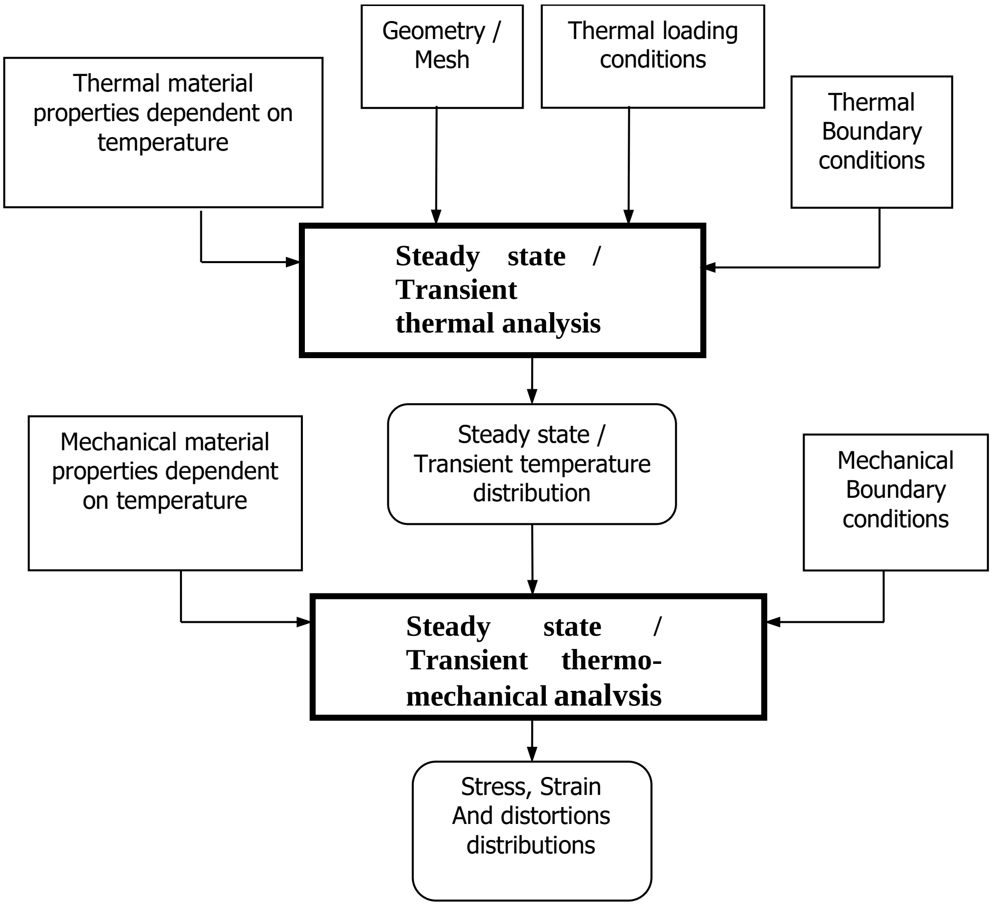

Figure 4: **Overview of the thermo-mechanical hydrogen tank simulation**

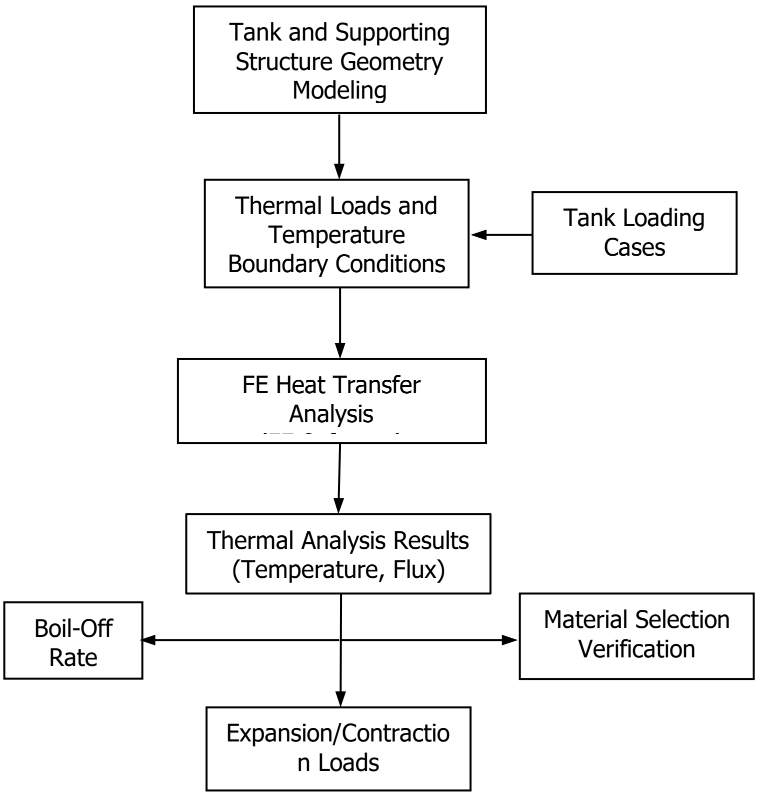

Figure 5: **FE Thermal Analysis Process**

DOI: 10.13009/EUCASS2022-7281

Thermal model mesh
The FE heat transfer model comprised the tank, the supporting structure and the necessary associated fuselage structure. Suitable element types having temperature degrees of freedom, capable to account for thermal conduction and convection behavior are selected. For the inner and outer tank walls, 4-node shell elements are used. The foam insulation is modeled with 8-node solid elements, while the supporting beams are meshed using 2-node elements, modeling the supporting beams.

Material Thermal Properties
For FE heat transfer analysis, the thermal conductivity and specific heat capacity values are needed. The magnitudes of the coefficients should be selected in accordance with design temperatures. Both coefficients should be specified for all structural members as their material properties.

Convection Loads
For main tank structural members such as the outer shell, inner hull, tank skin, and supporting structures, the convection coefficients and the associated ambient temperature are defined for each member. The convection boundary conditions consist of the bulk fluid temperature and the convection coefficient. The fuselage structure internal temperature for the present analysis has been assumed to be 27°C or 300 K, and the associated heat convection coefficients can be estimated using the heat transfer theory of section 2.

Thermal Radiation Loads
In practice, the thermal radiation effect usually can be neglected if the insulation is considered enough efficient.

Heat Flux Loads
If a cryostat is installed on the liquefied hydrogen tank, the associated heat flux loads should be properly introduced to account for the heat transfer condition. Nevertheless, this is not the case in the present analysis.

Boundary Conditions
For the inner surface of the inner tank, which is in contact with the liquefied hydrogen, the temperature is specified as the liquified hydrogen temperature (i.e. 20 K). It should be mentioned that for the steady state thermal analysis, the final heat flux equilibrium status is independent of the initial temperature, therefore, it is not necessary to have an exact initial temperature for each member of the model. An example of thermal loads and temperature boundary conditions applied to considered tank and its supporting structure is illustrated Figure 7.

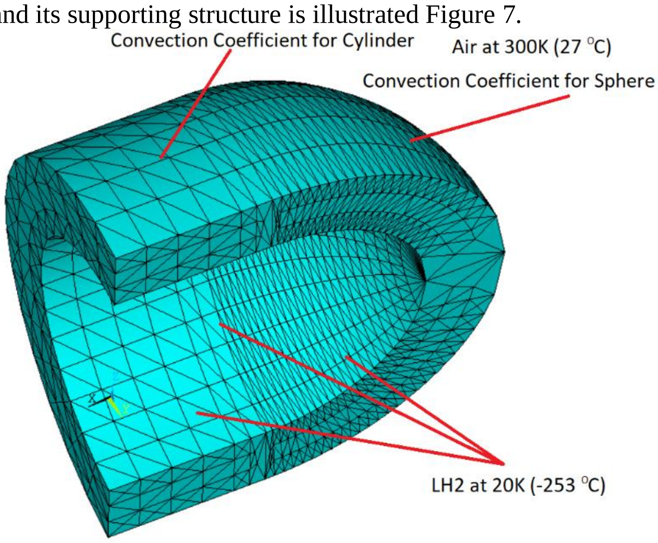

Figure 6: **Convection and Temperature Boundary Conditions**

Thermal Loading Cases Considered
Different loading cases, which include partial filling and full load, have been considered to calculate a detailed temperature distribution over all members for sequential FE thermal stress analysis. Thermal loading arising by a temperature gradient from different filling levels of a tank should be considered. To this aim, filling levels up to different horizontal levels are considered separately, i.e. for 25%, 50% and 75% partial filling, and 100% full filling. The schematic temperature profile for each loading case in a typical tank is given in Figure 7. For each filling level in this figure, the temperature of the parts of the inner tank in contact with the liquid hydrogen is set at 20 K (the boiling

temperature of hydrogen) while the temperature distribution of the rest of the inner tank surface is calculated by the model solution.

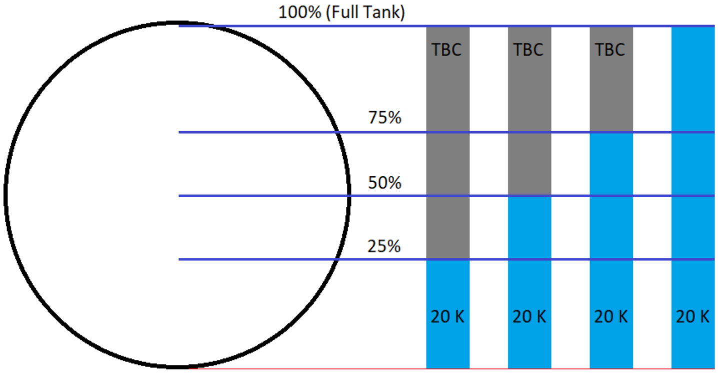

Figure 7: **Temperature distributions for different loading cases**

Convergence Study

The FE mesh density may affect heat and load transferring behaviors. A higher mesh density will produce more accurate results. Higher order elements such as 3-node beams, 8-node shells and 20-node solids can also increase the accuracy of the model. A combination of high mesh density and higher order elements can produce extremely accurate results at the expense of high memory usage and significantly long running time. A mesh convergence study is needed in order to extract results that better represent the actual conditions, although such a study may be time consuming.

4.1.2 Thermal analysis results:

Temperature Distributions

The results from FE heat transfer analysis include nodal temperatures of the entire FE model. In Figure 8 examples of temperature distributions of the simplified tank are presented.

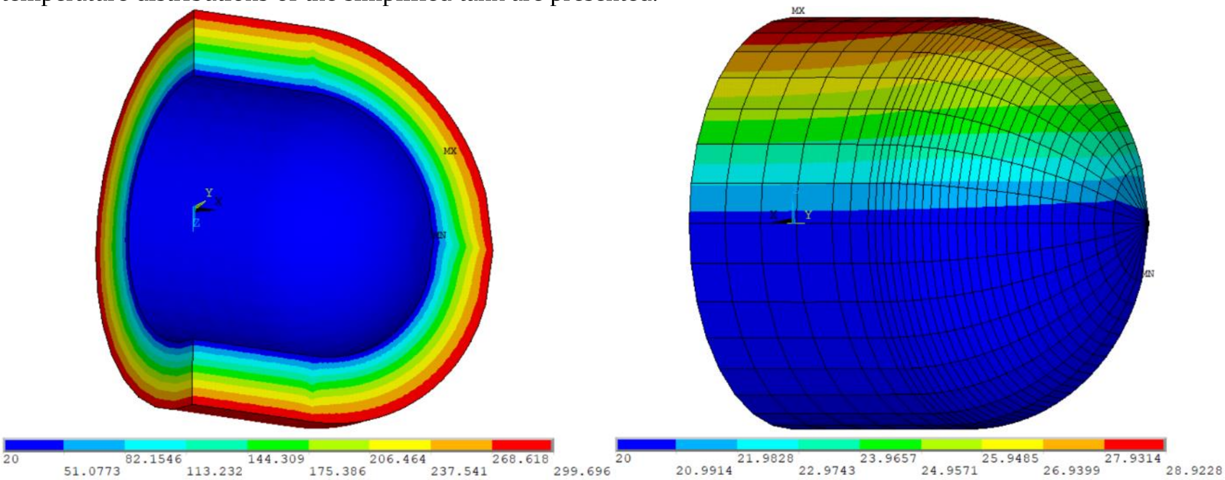

Figure 8 : **Temperature Distributions of the entire Tank for 100% loading (left) and of the Inner Tank for 50% loading (right) (units in K)**

The FE results of temperature distributions can be used as loading conditions for the sequential thermal FE stress analysis of the tank structure.

Thermal Flux

The results from FE heat transfer analysis also include thermal flux. The thermal flux is an indicator of the amount of heat transferred to the Liquid Hydrogen inside the tank through a specified area. The thermal flux can be directly translated to heat flow and thus the boil off rate. The results from FE heat transfer analysis include element thermal flux values of the entire FE model. In Figure 9 an example of thermal flux distribution on a cut of the simplified tank is presented.

DOI: 10.13009/EUCASS2022-7281

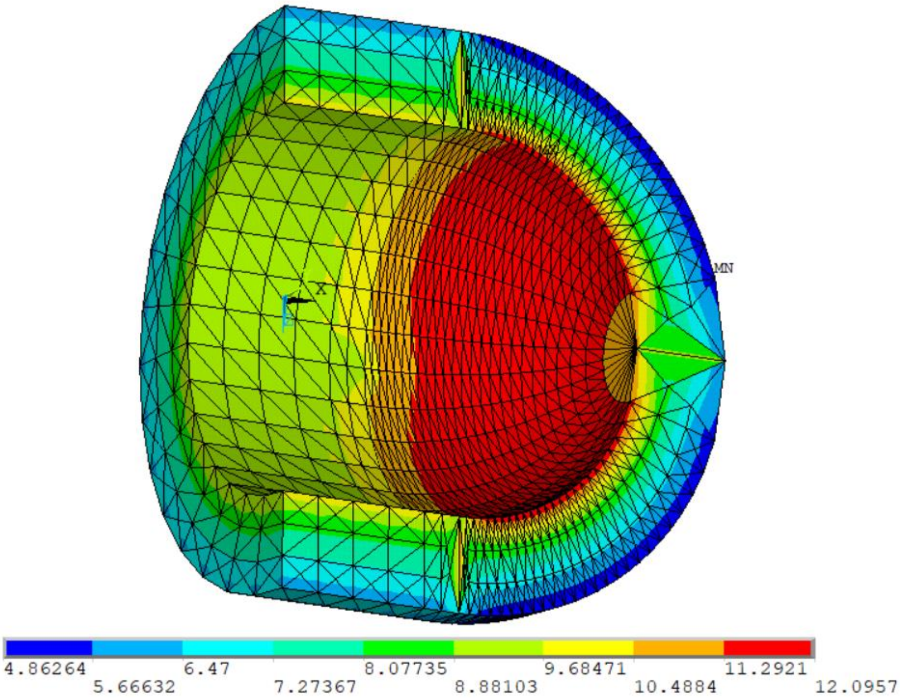

Figure 9: **Thermal Flux Distribution of the Modeled Tank (units in W/m^2)**

A comparison between analytical and FE thermal flux values is performed. The thermal flux distribution on the outer tank can be seen in figure 10.

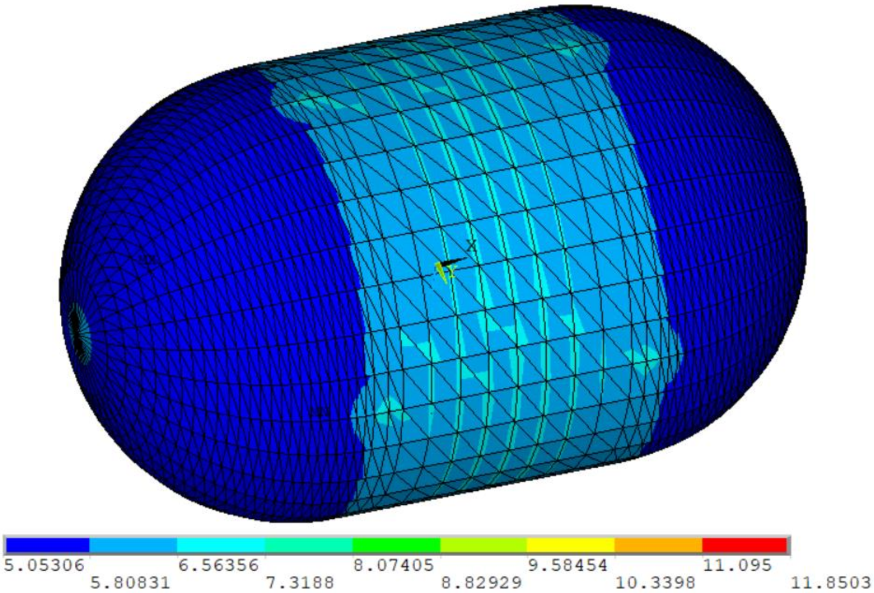

Figure 10: **Thermal Flux Distribution on the Outer Tank (units in W/m^2)**

On the cylindrical part, the FE flux values are close to 6 W/m^2, while the analytical value is 7.815 W/m^2. On the spherical caps values are close to 5.5 W/m^2, while the analytical value is 7.793 W/m^2. In the analytical calculations, curvature effects are ignored and this is the cause of the higher value of thermal flux. The FE results are much closer to reality as a three-dimensional heat transfer analysis is conducted. An analytical three-dimensional heat transfer analysis is extremely complicated and time consuming as complex partial differential equations have to be solved. This highlights the importance of FE analysis, as accurate results can be produced in a relatively short time.

DOI: 10.13009/EUCASS2022-7281

Heat Flux and Boil-off Rate

A total heat flux value of 68 W was calculated during the FE analysis. The lower value compared to the analytical is caused by the lower value of thermal flux on the spherical caps, highlighting the importance of thermal flux as an indicator of heat transfer. The boil-off rate of the FE analysis is:

$$BOR = \frac{Q}{\rho V H} \cdot 3600 \cdot 24 \cdot 100\% = \frac{68}{70.85 \cdot 1.308 \cdot 447000} \cdot 3600 \cdot 24 \cdot 100\% = 14\%\ per\ day$$

The FE thermal analysis results produce a lower boil-off rate compared to the theoretical value. As with the thermal flux, the cause of this disparity is the analytical one-dimensional heat transfer model that ignores three-dimensional geometry effects.

4.2 Thermomechanical Stress Analysis

4.2.1 Thermomechanical model description

Thermomechanical stress analysis is finally performed to the tank and its supporting structure. High local stress concentrations may occur at the inner tank wall, due to the thermal gradient around the liquid hydrogen line. Thermal loads due to temperature gradients should be combined with mechanical loads such as internal pressure for FE stress analysis and strength evaluation.

The total stresses of the tank structure under different loading cases are obtained for structural strength evaluation. Each part of the tank should be evaluated against material failure and buckling strength. Thermal stress check should also be performed on supporting structures. The detailed flowchart of FE thermal stress analysis and strength evaluation Figure 11.

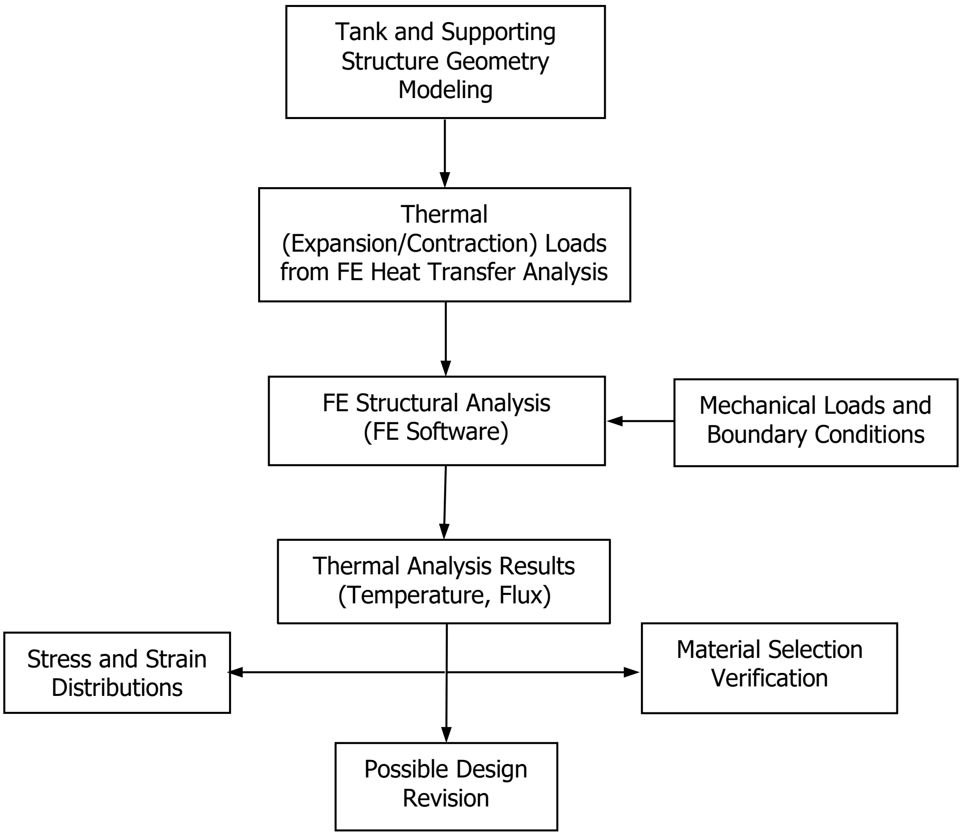

Figure 11 : **FE Structural Analysis Process**

The thermal contraction and expansion of the tank caused by the loading of liquid hydrogen can produce locally high stresses due to the geometric complexity of the structure. FE thermal stress analysis should be carried out to verify the structural integrity of a tank under thermal loads during the loading of liquid tank or initial cooling-down period, as mentioned above.

DOI: 10.13009/EUCASS2022-7281

Structural model mesh

For the structural analysis, 4-node shell elements have been used to model the inner and outer tank walls, 2-node beam elements for the beams and 8 node solid elements for the insulation. The solid elements support both thermal and structural analysis and therefore remain unchanged. For the beams and shells, structural counterparts of the thermal elements are used, with the mesh remaining unchanged.

Material Properties

The material properties like modulus of elasticity, Poison's ratio, density, thermal expansion coefficient, and reference temperature for thermal expansion coefficient have to be defined for FE thermomechanical stress analysis. When applicable, the material properties should be selected in accordance with the corresponding service temperature.

Thermal Loads

The temperature gradient of the tanks, their associated substructure members, and/or supporting structure may introduce significant thermal strains and stresses. The field temperature distributions of the relevant areas obtained from the FE heat transfer analysis of Section 4.1.2 are imported to the FE thermomechanical stress analysis FE model. The corresponding initial temperature for thermal stress analysis is assumed to be 20ºC or 293 K, but for a more accurate model, the temperature of the site where the tank was constructed must be used.

Mechanical Loads

The internal pressure of the stored hydrogen is the main load and is applied to the inner tank elements. Hydrostatic pressure loads from the liquid hydrogen, although low for a small size tank like the one examined, are also applied to the elements of the inner tank. Weight and acceleration loads can also be applied to the model, simulating mechanical loads during maneuvers. During acceleration, hydrostatic pressure of the liquid hydrogen is increased accordingly.

Boundary Conditions

For the tank structure, rigid translations and rotations have been introduced at selected nodes as boundary conditions, to account for the fuselage section loading. The tank is supported by the fuselage by means of saddle supporting structure in a manner which prevents rigid body movement of the tank under static and dynamic loads while allowing contraction and expansion of the tanks under temperature variations and fuselage deflections without undue stressing of the tanks and of the fuselage. More specifically, one end saddle support is fixed while the other end is designed to slide freely longitudinally to compensate for the contraction and expansion caused by the temperature change.

4.2.2 Thermomechanical model results

Stress analysis results

The global finite element model of a tank structure with proper boundary conditions has been used for FE thermomechanical stress analysis. The thermal expansion coefficient for the tank material is introduced in numerical modeling. Each thermal gradient specified in the loading cases together with other mechanical loads have been applied to determine the total stress at critical locations of the tank structure. The FE stress results are used for the strength evaluation of the tank structure based on the design requirements.

In Figure 12, an example of stress distribution of the tank under thermal and mechanical loads is presented. The stress results are used to evaluate the structural integrity of the tank and its supporting frame structural members.

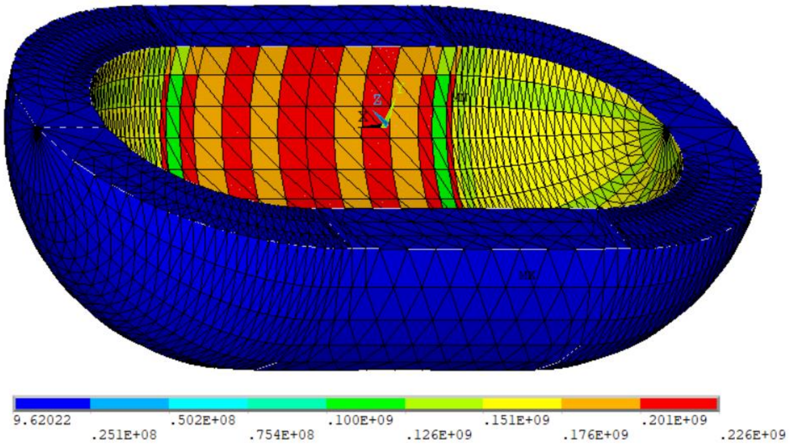

Figure 12: **Stress Distribution for 100% Loading [units in Pa]**

DOI: 10.13009/EUCASS2022-7281

Strain analysis results

The mechanical loads as well as thermal contraction due to the extremely low temperatures deform the structural members of the tank. The FE strain analysis results are used to evaluate the stiffness of the tank. Large displacements should be avoided as they can cause serious structural issues and in some extreme cases the tank or some of its components can interfere with other parts of the aircraft. Displacements are controlled by several regulations; therefore, the FE strain analysis can used to evaluate the tank design.

In Figure 13, an example of strain distribution of the tank under thermal and mechanical loads is presented.

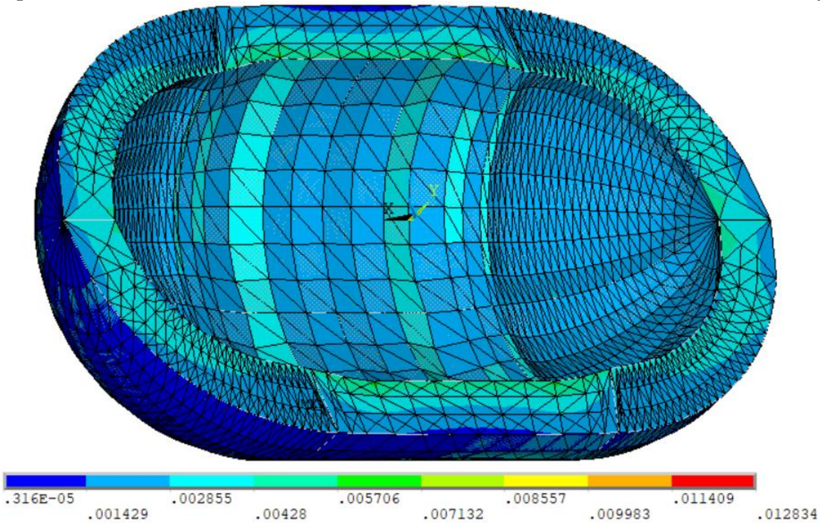

Figure 13: **Strain Distribution for 50% Loading**

4. Conclusions and Future Work

A three-dimensional finite element model capable to simulate the operating conditions of a liquid hydrogen tank, as well as predict temperature and stress distributions of any of the tank components has been developed. All the major heat conduction, convection and structural loads are simulated through a finite element (FE) thermal-structural analysis. The developed model considers temperature dependent thermal and mechanical material properties. The main numerical parameters of the model, such as mesh density and element types are defined after convergence investigations. For the verification of the model, calculated thermal and heat flux values are compared to theoretical values from the analytical models, indicating a good agreement. The developed numerical simulation is a useful tool for calculation of operating parameters such as boil-off rate, verification of the material selection, as well as design revision, contributing to identification of possible failures, and assessing the behavior of the tank during operation. The current model needs many more developments, both with respect to the heat transfer analysis, as well as to the structural analysis. In the structural model dynamic phenomena, such as movement of the liquid hydrogen inside the tank (sloshing) during maneuvers, introduction of material imperfections and defects, and other details should be introduced to improve the tank behavior prediction. The model can also be adapted to simulate an integrated tank, with the outer tank being a load bearing part of the fuselage. Fatigue cycles caused by mechanical and thermal loads can also be investigated in a future work. As the model is multi-parametric, a trade study between boil-off rate and weight, or and other design aspects, can be conducted, rendering the simulation a useful tool for design revision, resulting in more efficient tanks.

References

[1] Airbus. Liquid Hydrogen Fueled Aircraft— https://cordis.europa.eu/project/id/G4RD-CT-2000-00192

[2] Brewer, G.D. Hydrogen Aircraft Technology, 1st ed.; CRC-Press: Boca Raton, FL, USA, 1991.

[3] https://www.clean-aviation.eu/sites/default/files/2022-01/CAJU-GB-2021-12-16-SRIA_en.pdf (Accessed June 22, 2022)