DOI: 10.13009/EUCASS2023-039
Aerospace Europe Conference 2023 – 10TH EUCASS – 9TH CEAS

Design Loop Process for Cooling Jacket of Liquid Rocket Engine

Viviana Ferretti[1], Michele Martini Urbano[2]*, Alessio Gizzi[3]*†, Daniele Liuzzi[4]*, Daniele Drigo[5]**
**AVIO S.p.A. Via Latina, snc (SP 600 Ariana km 5,2) 00034 Colleferro (RM, Italia)*†*

[1]viviana.ferretti@avio.com, [2]micheleurbano.martini@avio.com,[3*†]alessio.gizzi@avio.com,
[4]daniele.liuzzi@avio.com, [5]daniele.drigo@avio.com

Abstract

One of the most stressed components of a liquid rocket engine is the cooling systems of the combustion chamber. A complete 3D model loop for the cooling system thermal dimensioning has been developed and here presented. The main objective of the model is a complete and detailed description of coupled solid/coolant behaviour characterizing the liquid rocket engine cooling systems, such as three-dimensional effects, supercritical behaviour of the coolant, thermo-structural behaviour of the solid. The model combines the industrial necessity to obtain proper results in reasonable times with a detailed description of the involved phenomena and specific characteristics of the configuration.

1. Introduction

The evaluation of the cooling system performance is important to identify possible criticalities related to the system dimensioning. The modelling of the cooling system requires a complete and detailed description of coupled solid/coolant behaviour, considering all the physical aspects involved in the cooling effects, such as three-dimensional effects, possible supercritical behaviour of the coolant and thermo-structural behaviour of the solid walls. All these aspects have been considered to develop a complete 3D model loop used for the cooling system thermo-structural dimensioning.

The loop is defined by means of the following models properly defined with respect to the focused product and involved aspects:

- Reacting CFD simulation performed by Q1D reduced order model to define the heat fluxes of the combustion chamber acting on the cooling system; the Q1D model has been validated with respect to 2D axisymmetric simulations performed with AVIO internal code.
- Cold CFD simulation: RANS compressible steady-state CFD simulations with a 3D conjugate heat transfer model. A proper high roughness model is used, able to describe the effects of high roughness wall on the internal flow and on the cooling channel performance in terms of thermal behaviour [1]. Indeed, only the coupled description of fluid stratification and roughness effects allow the correct evaluation of thermal behaviour of the cooling systems. They directly affect the wall temperature, the corresponding heat flux coefficient, and the fluid bulk temperature. The analyses have been performed by Ansys Fluent 2023 R1.
- 3D FEM thermo-structural non-linear analyses to obtain a description in terms of structural behaviour, stress, strain of the cooling system. The thermo-structural analyses are used to obtain a consistency check with respect to the wall temperatures obtained by Cold CFD analyses and to evaluate possible design actions required to respect the limits imposed to the design process (for example geometry change proposal to restart the design process verification loop). The analyses have been performed by MARC MSC 2021.

Once closed a loop, a complete description of fluid/structural behaviour is available. The results allow the identification of possible design improvement. If identified, the solutions are provided as input to Reacting CFD simulations to obtain the new heat fluxes acting on the cooling system. The use of this loop allows a proper dimensioning of the cooling system with an optimization of both fluid/structural behaviour.

DOI: 10.13009/EUCASS2023-039

All the models used within this loop have been validated through experimental tests and in-house code developed by AVIO.

2. Design loop approach

As previously explained, the thermo-structural dimensioning of a cooling system requires a complete 3D model loop able to describe the coupled fluid/structural behaviour. This description has been obtained by means of different numerical models used in a multidisciplinary approach to simulate all the different physical aspects to be considered. A scheme with the logic of the loop has been reported in Figure 1.

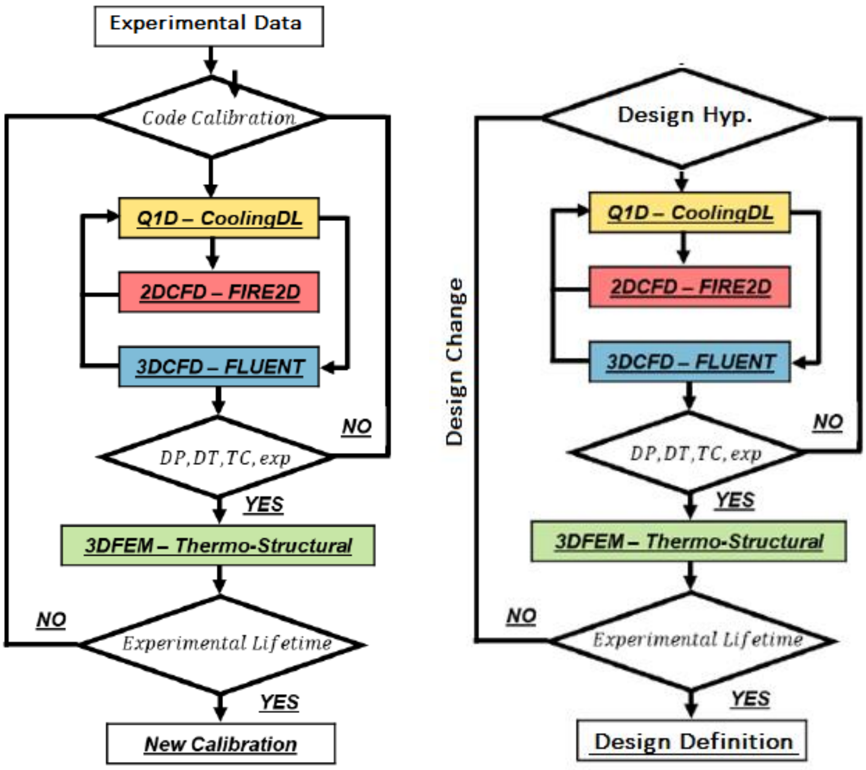

Figure 1: Design Loop Schematic: loop calibration and design analysis.

The model considered for the approach are here resumed:

- Fire 2D: 2D axisymmetric simulations performed with AVIO internal code and used to validate the Q1D model (details in par. 3)
- Cooling DL Q1D simulations: Q1D reduced order model to define the heat fluxes of the combustion chamber acting on the cooling system. The model is validated with respect to Fire 2D and provide the input data to be used for the 3D CFD Cold Simulations. (details in par. 4)
- Fluent 3D Cold simulations: 3D CHT simulations for the evaluation of channel performance and to obtain a description of fluid/structural behaviour. The input data are obtained by Q1D Cooling DL and it provides outputs for the 3D FEM analyses. (details in par. 5)
- 3D FEM thermo-structural analyses: inputs are obtained by Fluent 3D simulations and they provide assessments on lifetime and thermo-structural verification of the analysed configuration (details in par. 6)

The loop of these models has to be repeated in two different phases:

- Loop calibration: the loop is used for a calibration with respect to the available experimental data. The experimental data taken into account are related to both reacting and cold test performed for the considered liquid rocket engine. The final output of a loop is the verification of the configuration in terms of obtained lifetime with respect to experimental one.
- Design definition and thermo-structural behaviour: the loop is used to analyse the design and the thermo-structural behaviour. The final output of a loop is the verification of the configuration in terms of expected

DOI: 10.13009/EUCASS2023-039

lifetime and behaviour. In case of not successful results, the loop will be repeated with introduced design modifications. Moreover, the loop can be considered concluded when the wall temperature obtained by CFD and FEM analyses are in agreement in terms of values and trends (par.6).

3. Reacting simulations

The analyses to evaluate the heat transfer coefficient in the combustion chamber have been performed with an "in-house" AVIO code based on CEA, namely Fire2D, that simulates the reactant flow inside the combustion chamber. The main outcome of this code is the $h_{c,Gas}$ that describes the heat flux applied to the internal wall of the cooling system. The output of this code is used to calibrate the following model Cooling DL that simulates both reacting side and cold side, as described in par. 4. In particular, the heat transfer coefficient estimation through the Bartz equation, implemented inside CoolingDL, has been modified in order to consider the so-called "2D effects"; these effects have been introduced through a "shape function" that modifies the trend of the "original" Bartz correlation, in order to follow the heat transfer coefficient obtained by dedicated CFD-2D simulations, performed by Fire2D, of the reactant flow inside the combustion chamber.

4. Q1D model – "Cooling DL"

Cooling DL is in-house Avio code used for the analysis of cooling channels of regenerative cooled combustion chambers. Code "CoolingDL" is used to solve the coupled problem of heat exchange inside a thrust chamber with a regenerative cooling system, between the combustion chamber and the cooling channels that surround it. It is based on a *Quasi-1D* model of the flow inside the cooling channels, using the conservation equations of mass, momentum and energy; it includes a modelling of the friction and heat exchange phenomena along the channel walls. The fluid variable used in Cooling DL are obtained with the NIST REFPROP program and loaded with the program.

The code is validated using experimental data provided by different Test Campaigns performed by Avio on various test article.

Cooling DL is used during design and dimensioning phase, to estimate cooling channel performance, and during experimental data post processing to better understand the real cooling performance and the involved physical phenomena.

The Q1D model is calibrated with respect to dedicated CFD-2D simulations, performed by Fire2D. In particular, the heat transfer coefficient estimation through the Bartz equation, implemented inside CoolingDL, has been modified in order to consider the so-called "2D effects"; these effects have been introduced through a "shape function" that modifies the trend of the "original" Bartz correlation, in order to follow the heat transfer coefficient obtained by dedicated CFD-2D simulations, performed by Fire2D, of the reactant flow inside the combustion chamber.

The $h_{c,Gas}$ estimation inside the CoolingDL Code is then given by the multiplication of three different contributions:

- the Bartz correlation.
- a shape function obtained through correlation of dedicated CFD simulations.
- a k_{hcgas} coefficient, function of P_{cc} and MR.

5. Cold CFD model

The main output provided by reacting simulations are the hot heat flux to be applied to the internal surface of the cooling channel facing the motor combustion chamber. The heat flux distribution along the channel longitudinal axis is provided by Cooling DL (see par. 4). This trend, in addition to the channel mass flow rate, inlet temperature and pressure have been used as boundary conditions for 3D CFD analyses performed to evaluate the wall temperatures of the channel and the $h_{c,cold}$ to be provided as boundary conditions for the FEM analyses (see par. 6)

5.1 Model description

The 3D CFD analyses have been performed by means of Fluent 2023 R1 (□[2], □[3]). A 3D conjugate heat transfer model (CHT) has been developed for the simulation of the heat transfer problem in cooling channels characterized by high roughness surfaces. The main objective of the model is a complete and detailed description of coupled solid/coolant behaviour characterizing the liquid rocket engine cooling systems. Details of the used model can be found in [1].

Different objectives have been considered for the model development: to describe the behaviour of coolant within the channel (three-dimensional effects and possible supercritical behaviour of the coolant) and to describe the solid thermal behaviour. The development of this model combines the industrial necessity to obtain proper results in reasonable times with a detailed description of the involved phenomena and specific characteristics of the configuration.

Main aspects characterizing the presented model are:

- the high roughness description (High Roughness Modelling) able to describe the effects of high roughness wall on the internal flow and on the cooling channel performance in terms of thermal behaviour. For example, the proper modelling of these aspects allows a correct description of possible fluid stratification close to the wall. Indeed, only the coupled description of fluid stratification and roughness effects allow the correct evaluation of thermal behaviour of the cooling systems because they directly affect the wall temperature, the corresponding heat flux coefficient, and the fluid bulk temperature.
- the proper fluid description obtained by a dedicated UDF (User Defined Function) to obtain real gas modelling and to assign to the cooling fluid a proper and accurate material properties database.

The high roughness model has been validated with respect to both experimental and numerical data [1].

The model is a 3D conjugate heat transfer model coupling a RANS Spalart-Allmaras turbulence model for the coolant flow with Fourier equations for the thermal conduction in the solid volume.

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modelled transport equation for the kinematic eddy (turbulent) viscosity. The use of this turbulence model is mainly due to the following aspects:

- the model is simple and fast computing, comparable with other results already available by literature
- this turbulence model allows the use of an improved high roughness model (see par. 6.2). A second order Coupled Pressure-Based solver has been considered and a second order spatial discretization has been used.

The proposed model has been validated with respect to numerical data and experimental results also for previous engine configuration.

The fluid properties taken into account have been obtained by NIST Real Gas Models database. A dedicated User Defined Function (UDF) has been written and implemented to assign to methane a proper and accurate material properties database and to improve some restriction related to NIST database implemented on Fluent. The fluid properties have been obtained by NIST REFPROP release 10.0 database [4].

5.2 Geometry and mesh

The model here proposed is used to model only one of the cooling channels, without considering the inlet and outlet manifold (domain reported in Figure 2).

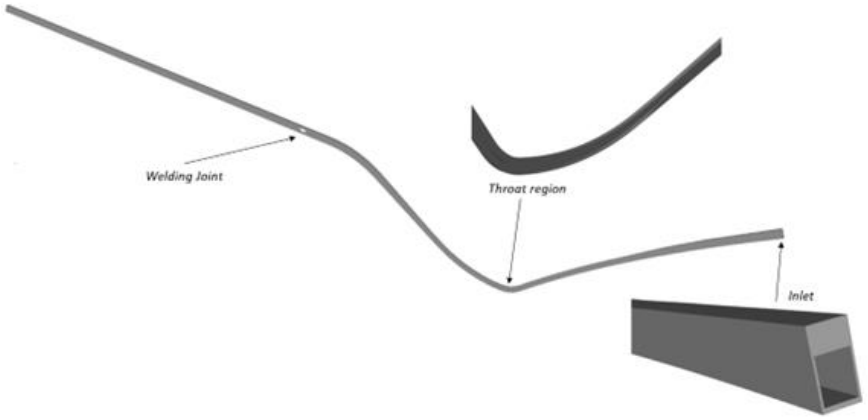

Figure 2: Cooling channel domain.

A fully unstructured approach has been adopted to generate the computational mesh, mainly due to the geometrical complexity presented in different regions of the channel (e.g. the throat). The computational domain is first discretized using tetrahedral elements and then, the resulting discretization is transformed into a polyhedral mesh. This approach provides several advantages as a greater accuracy in the computation of flow gradients and a better

quality in terms of mesh skewness, usually resulting in an increased convergence speed. A final mesh of 9.5M polyhedral cells has been obtained, starting from a tetrahedral mesh size of 21M cells.
Due to the requirements of high roughness model, the boundary layer has been modelled with a mesh assuring the requirements of the selected model in each point of the domain.
The interface between solid and fluid domain has been divided in three different interfaces named liner (liner cold surface), lateral and up surface as reported in Figure 3. Liner hot surface is the surface where the heat fluxes can be applied in case of reacting simulation. The close out surfaces are imposed as adiabatic surfaces.
A global overview of the computational meshes employed for the simulations is reported in Figure 3.

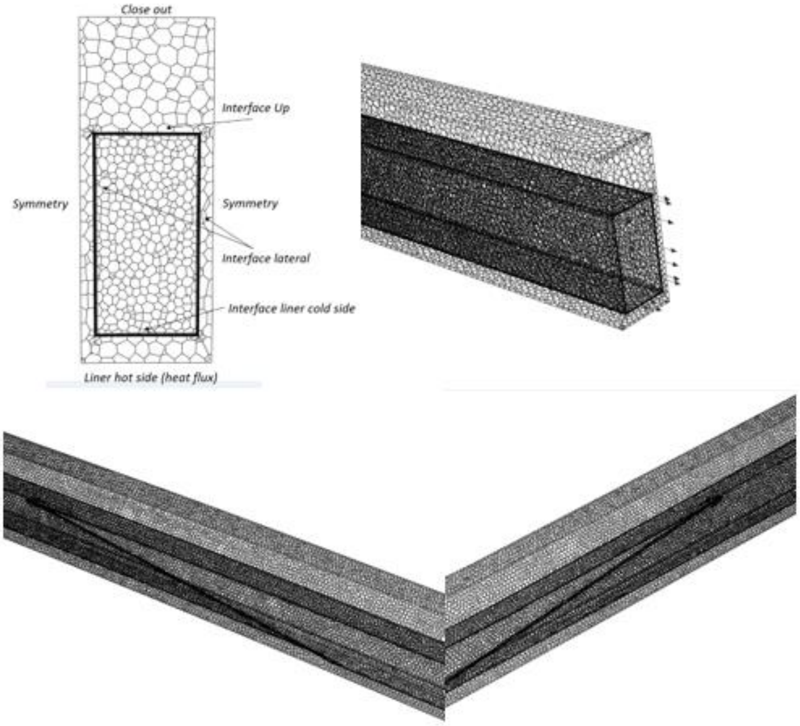

Figure 3: Cooling channel domain mesh.

5.3 Boundary conditions

The considered operative conditions are the Pin, Tin, MFR and the imposed heat flux profile (scaled with respect to an input/output mean value) has been reported in Figure 4.

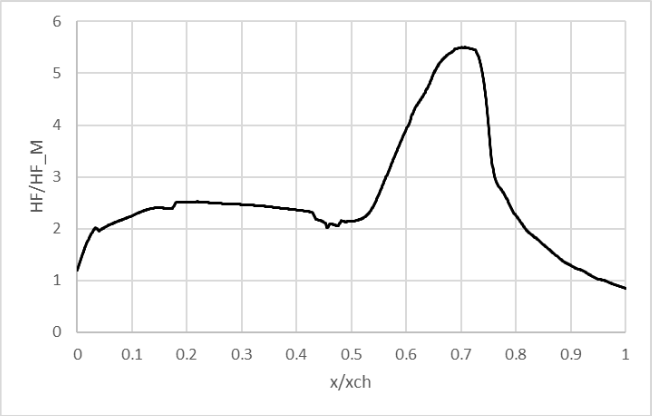

Figure 4: Methane hot test – heat flux profile.

The roughness profile to be applied to the channel wall has been obtained by correlation with respect to experimental cold flow tests (heat flux applied); the correlation has been considered successful when the experimental pressure

DOI: 10.13009/EUCASS2023-039

evolution along the channel has been matched. The obtained roughness profile must be considered representative of real surface roughness and it has been used for all successive simulations, both cold and reacting cases, performed for the test case configuration to obtain description of cooling system performance and behaviour.

5.4 Results

The results obtained by cold CFD analysis are the temperature distribution on the considered surfaces and the fluid-dynamic behaviour of the fluid within the channel. An example of the wall temperature distribution along the surfaces is reported in Figure 5.
The input data provided to FEM analyses are represented by the obtained T_{wall} along each solid surface of the channel and the corresponding h_c values. An example of the provided trends is reported in Figure 5.

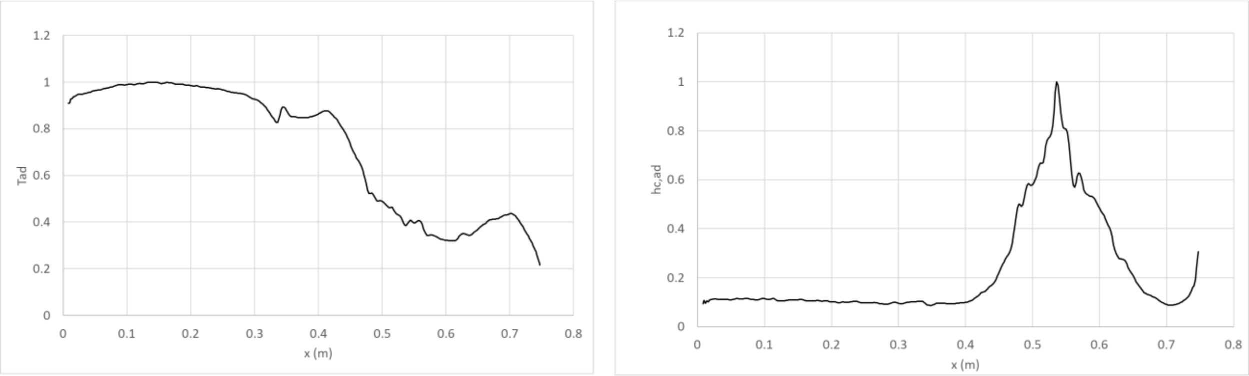

Figure 5: Methane hot test: liner temperature hot side (left) and h_c cold side (right).

6. FEM model

The selected configuration of cooling system is subsequently verified by means of FE thermo-structural analyses in order to provide the maximum wall temperature and lifetime estimation.
Due to elevated temperatures involved, linear structural analyses are not satisfactory and for this reason, elasto-plastic non linear models are necessary to assess the evolution of stress-strain process during several firing cycles.

6.1 Model description

The 3D FEM model is developed by using commercial software Apex 2021 [5] and Marc Mentat 2021.1 of MSC company [6], widely employed inside AVIO Spa and for this reason adopted for this specific case study.

6.2 Geometry and mesh

For the analysis involving the whole cooling system, the computational domain is represented by an angular wedge containing one cooling channels along the initial part and final part of the cooling system and half cooling channel in the throat zone, and imposing periodic boundary conditions on the lateral walls, as shown in Figure 2.

The total number of nodes is equal to 247348; the total number of elements is equal to 187368.
Following types of elements have been used:

- 186968 hexahedral elements of type hex8_7
- 400 pentahedral elements of type penta6_136

In Figure *6* is showed the detailed zooms mesh of the component modelled.

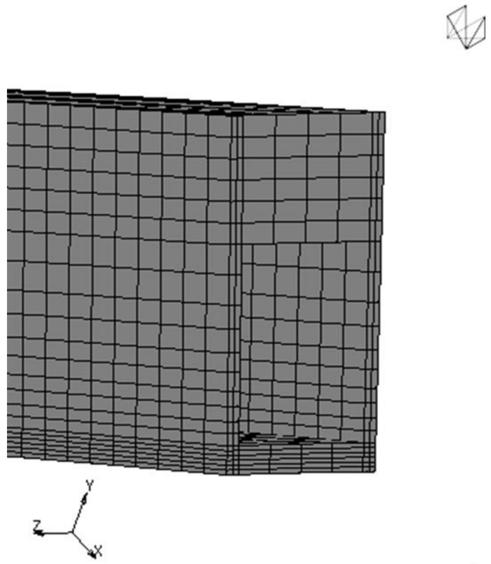

Figure *6* 3D FEM Cooling Channel Mesh

6.3 Material Properties

The material assigned to the component is ALM Inconel 718. The FEM analysis have a non-linear elasto-plastic material behaviour with temperature dependency. The material mechanical and thermal properties of ALM-Inconel718, used for the actual simulation, was obtained experimentally by AVIO Spa through dedicated material characterization.

6.4 Boundary conditions

Structural Boundary conditions
Two boundary conditions have been imposed on the global structure as shown in Figure 7. The structure is fixed at X direction in the highlighted zone in left side of Figure 7. This would avoid the displacement of the structure at X direction.
In order to define symmetric condition for the geometry, the displacements of two lateral surfaces have been set to zero at perpendicular directions to each surface, as shown in right side of Figure 7. For the purpose two different reference systems have been created and the d.o.f. of the nodes laying on the symmetry faces are expressed via those systems.

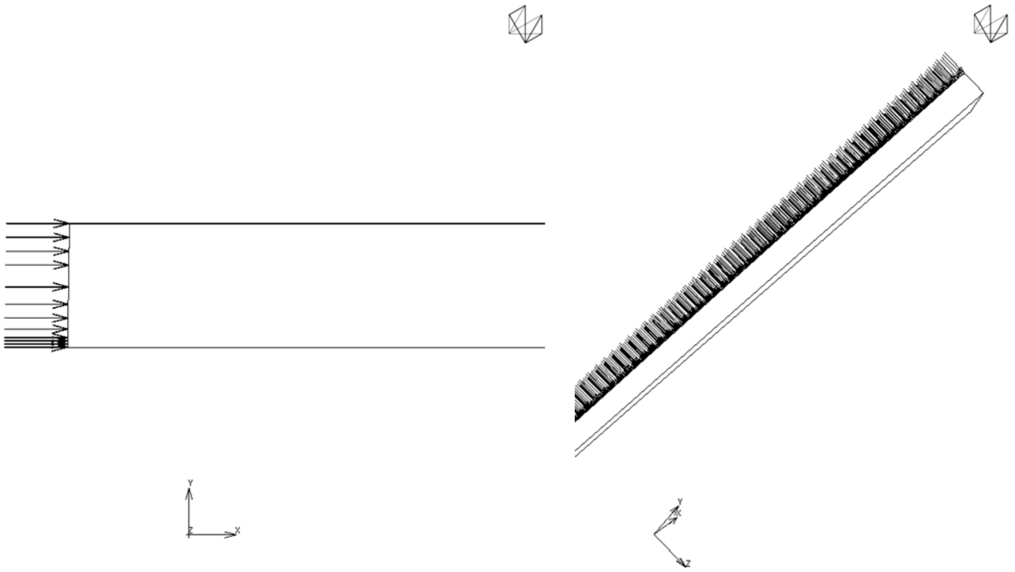

Figure 7 Constrained Zone at X-direction (Left Side), Symmetry on one one lateral side of the structure.

Mechanical Boundary Conditions

The mechanical interactions between the gases flowing inside the structure and the structure itself are translated into pressure. Such pressures, provided by CFD analyses, are different both axially and from hot side, inside combustion chamber, and cold side, inside cooling channel.

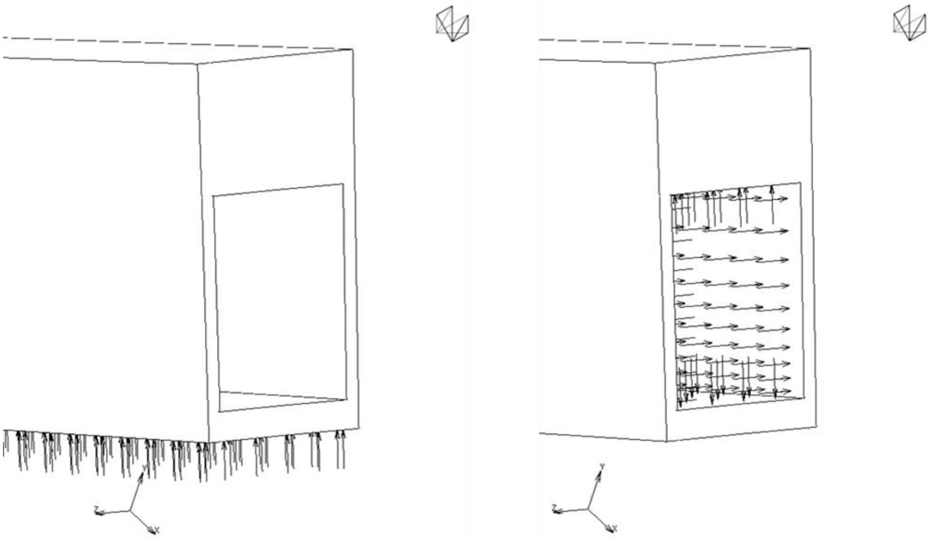

Figure 8 Hot Side Pressure (Left Side), Cold Side Pressure (Right Side)

Thermal Boundary Conditions

The heat exchange of the cooling channel is obtained by assigning to each region/surface a peculiar value of heat transfer coefficient (HTC) as well as a sink temperature for cold side. In particular, the cold side is subdivided into three regions Liner, Ribs and CloseOut in order to be closer to Cold CFD results.

Instead, thermal flux is assigned to hot side because it is the only part exposed to the combustion fire.

Such boundary conditions are provided by Reacting and Cold CFD.

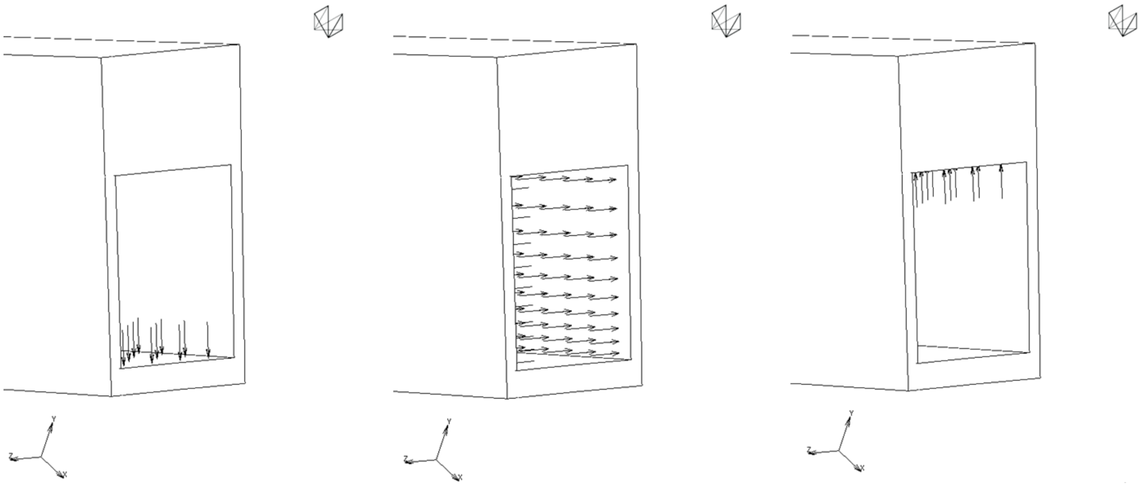

Figure 9 Liner Cold Side HTC (Left Side), Rib Cold Side HTC (Middle Side), CloseOut Cold Side HTC (Right Side)

6.5 Results

The models involved in the proposed design loop have described in detail. The loop has been considered concluded where the obtained FEM and CFD temperature evolution are in agreement in terms of values and trends. Moreover, the final output of a loop is the verification of the configuration in terms of expected lifetime and behaviour. In case of not successful results, the loop will be repeated with introduced design improvements.

An example of the Temperature, comparing CFD, FEM and maximum material temperature, and Von Mises Stress distribution has been reported in the following figures.

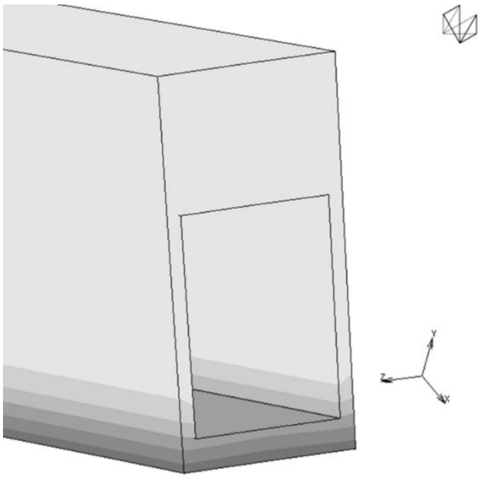

Figure 10: 3D FEM Thermo-Structural Analysis – Temperature Distribution

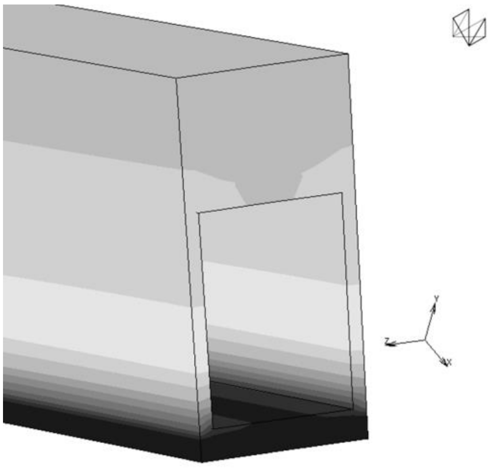

Figure 11: 3D FEM Thermo-Structural Analysis – Von Mises Stress Distribution

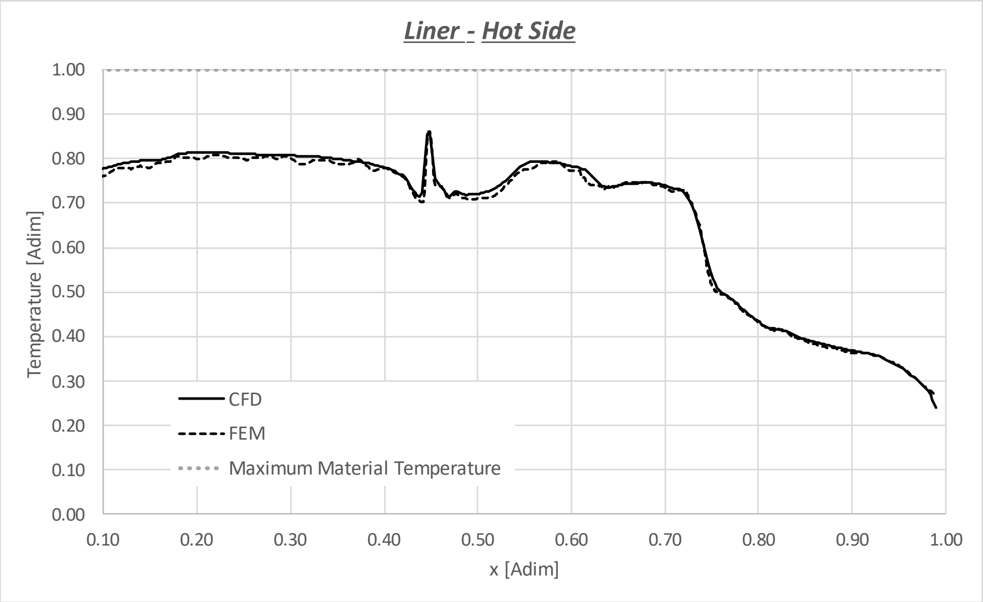

Figure 12: CFD/FEM and Maximum Material Temperature Comparison – Liner Hot Side

DOI: 10.13009/EUCASS2023-039

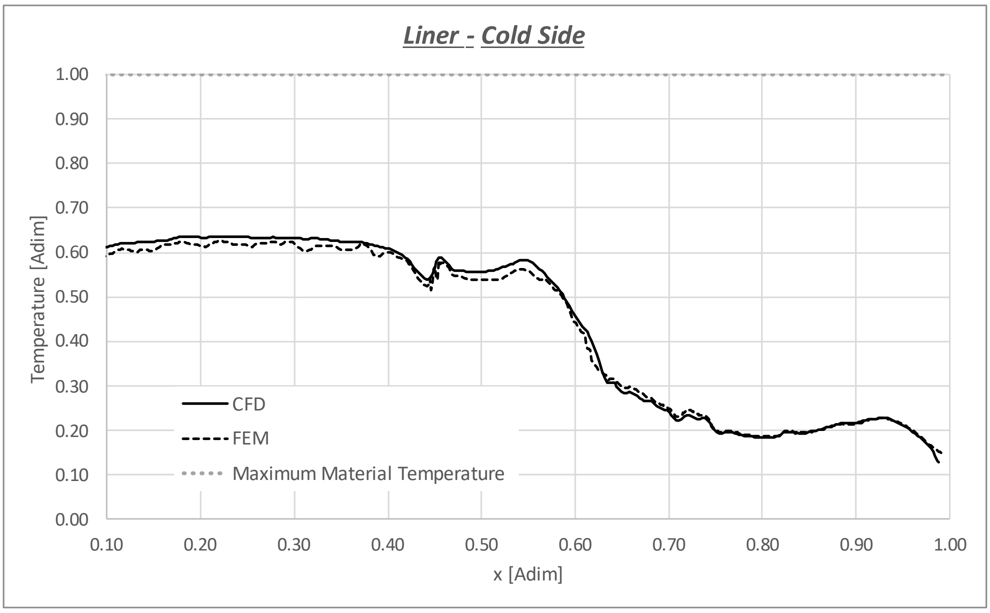

Figure 13: CFD/FEM and Maximum Material Temperature Comparison – Liner Cold Side

7. Conclusions

A complete 3D model loop for the cooling system thermal dimensioning has been developed and presented. The model has shown its capability to correctly describe both the thermo-structural behaviour of the solid and the fluid-dynamic behaviour of the methane within the channel, analysed in a coupled way. This description has been obtained by means of different numerical models used in a multidisciplinary approach to simulate all the different physical aspects to be considered.

The proposed approach has been validated wrt available cold and hot experimental data. The obtained final output of the loop is the verification of the configuration in terms of obtained lifetime with respect to experimental one.

The approach has been then used to analyse the design and the thermo-structural behaviour, in terms of expected lifetime and behaviour. The closure of the loop also verifies the consistency between the used models.

References

[1] V. Ferretti1, M. Sciarra, D. Liuzzi, D. Drigo, *3D Conjugate heat transfer model for simulation of Heat Transfer by High Roughness Cooling Channels*, 9th European Conference for Aeronautics and Space Sciences (EUCASS), EUCASS2022-5871

[2] ANSYS Fluent Theory Guide - ANSYS, Inc.

[3] ANSYS Fluent User's Guide - ANSYS, Inc

[4] Eric W. Lemmon, Ian H. Bell, Marcia L. Huber, Mark O. McLinden 2018: *REFPROP Documentation Release 10.0*

[5] Oracle APEX Release 21.2 User Manual

[6] Marc and Mentat Documentation: Release 2021

[7] B. Betti, D. Liuzzi, F. Nasuti, M. Onofri, *Development of Heat Transfer Correlations for LOX/CH4 Thrust Chambers*, 9th European Conference for Aeronautics and Space Sciences (EUCASS), 2015

[8] A. Terracciano, D. Liuzzi, A. Pascucci, *2D Thermo-Structural Code Development for Regenerative Combustion Chamber Analysis*, 7th European Conference for Aeronautics and Space Sciences (EUCASS), EUCASS2017-453

[9] D. Liuzzi, M. Sciarra, G. Bianchi, M. Rudnykh, D. Drigo, *Firing Test and Program Progress of the SMSP regenerative combustion chamber*, 7TH European Conference for aeronautics and Space Sciences (EUCASS), EUCASS2017-483

DOI: 10.13009/EUCASS2023-039

*[10]*A. Terracciano, R. Gelain, G. Bianchi, D. Liuzzi, D. Scarpino, M. Rudnykh, D. Drigo, L. Arione,R. Pellegrini, E. D'Aversa, *Experimental Tests Results of Transpiration Cooling Subscale Injector Head in the frame of LYRA Program*, 8TH European Conference for aeronautics and Space Sciences (EUCASS), EUCASS2019-660
*[11]*Liuzzi, D., Rudnykh, M.; Drigo, D.; Ierardo, *Architecture Trade-Off for the VEGA-E upper stage LOX/CH4 engine,* 7th EUropean Conference for Aeronautics and Space Sciences (EUCASS) 2017, EUCASS2017-484