DOI: 10.13009/EUCASS2023-040
Aerospace Europe Conference 2023 – 10TH EUCASS – 9TH CEAS

Design Loop Process for Injector Dome of Liquid Rocket Engine

Alessio Gizzi[1][†], Viviana Ferretti[2]*, Michele Martini Urbano[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*†]alessio.gizzi@avio.com, [2]viviana.ferretti@avio.com, [3]micheleurbano.martini@avio.com, [4]daniele.liuzzi@avio.com, [5]daniele.drigo@avio.com

Abstract
One of the main needs in modern combustion chamber design technique is to foresee the maximum temperature condition on the components subjected to high thermal stresses.
In this frame, a model to investigate on the temperature of the injector dome is proposed and analysed in detail. The model consists of two iterative steps integrating the reacting domain and the cold one. The numerical model is composed of a hot domain, in which reacting CFD simulations are performed in the OpenFOAM environment, and of a cold domain, in which RANS compressible steady-state CFD simulations are performed by Ansys Fluent 2023 R1 ([5][6]).

1. Introduction

The iteration loop begins with the design process based on AVIO and literature state of the art in which the geometry dimensions are defined; the geometry is the one used in CFD and FEM simulation.
The first iteration loop starts with CFD simulations in which the reacting part and the steady state part share the boundaries on the firing plate of the injector head system. Firstly, the OpenSMOKE code runs reacting flow simulation using tabular properties for combustion and gives as output the wall heat flux through the firing plate surface. This output is taken as input, as the geometry, for the CFD steady state simulation performed by Ansys Fluent; the provided output is the wall temperature, to be used as boundary condition for the reacting flow simulations. The loop ends after a consistency check on both the wall heat fluxes and the wall temperatures on the firing plate.
The second step is dedicated to FEM simulations performed by MSC software to obtain feedback in terms of stress and strain to be checked with the mechanical properties. The heat flux through the firing plate, the convection heat transfer coefficient, the fluid temperature in the injector dome and the geometry are taken from the MSC software as input for thermo-structural calculations.
In the last step, the firing plate wall temperature calculated from FEM analysis and Ansys Fluent simulations is checked for consistency before to give feedback on the design and, if needed, a geometry change proposal to restart the design process verification loop.
All the three simulation codes have been validated and calibrated through experimental tests and in-house code developed by AVIO and they will be thoroughly described. The paper will also report the results obtained by applying the described model to the design of a real injector dome.

2. Design loop approach

The design loop approach, presented in this work, represents only the first iteration of the entire design loop, in order to give a detailed approach that couples different CFD numerical codes on different domains.
More in detail the “CFD reacting” simulations performed with an in-house developed code in the OpenFOAM environment are coupled with the “CFD cold” simulations developed in Ansys fluent environment and explained in the next sections.
After the 1st iteration convergence, the results coming from the CFD simulations on both sides of the domain are input of the thermo-structural simulations to finally check the overall design and to choose and improve new geometry or components.

DOI: 10.13009/EUCASS2023-040

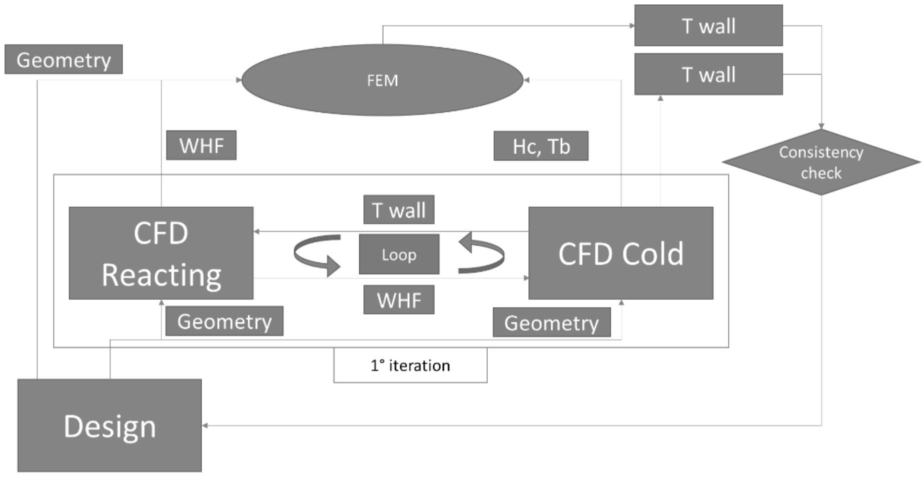

Figure 1: Design loop approach schematic.

3. CFD calculations in the reacting domain

3.1 OpenFOAM solver

This framework is based on the Low-Mach approximation that allows the use of Navier Stokes equations in incompressible form and the URANS (Unsteady Reynold Average Navier Stokes) approach for turbulence modelling. Combustion, on the other hand, is solved using non-adiabatic flamelet modelling. This allows detailed chemical kinetics mechanisms (with a large number of species and reactions) to be used to describe the structure of non-premixed flames using reasonable computational resources. With this model, it is therefore possible to simulate combustion phenomena (simulated by flamelets) and mixing phenomena separately. This subdivision of the phenomena results in a considerable computational advantage when complex chemical kinetics mechanisms are used.

3.2. Geometry of the reacting domain

The geometry of the reacting domain is a 60° degrees wedge of the whole domain, starting from the recess zone of the injector up to the end of the cylindrical section. The investigated part is the firing plate that represent the division between the cold and reacting domain.
The meshing domain is made up through a commercial opensource software able to develop a structured hexahedral mesh and to refine elements as needed; in particular, it is possible to see in Figure 2, that the mesh refinement is made up in order to have small element in the mixing zone starting from the injectors recess and in contact with the boundary pf the firing plate. In order to maintain the elements number below 1 million, due to computational limitation, a progression algorithm is applied in the axial direction for the zones far from the mixing volume.

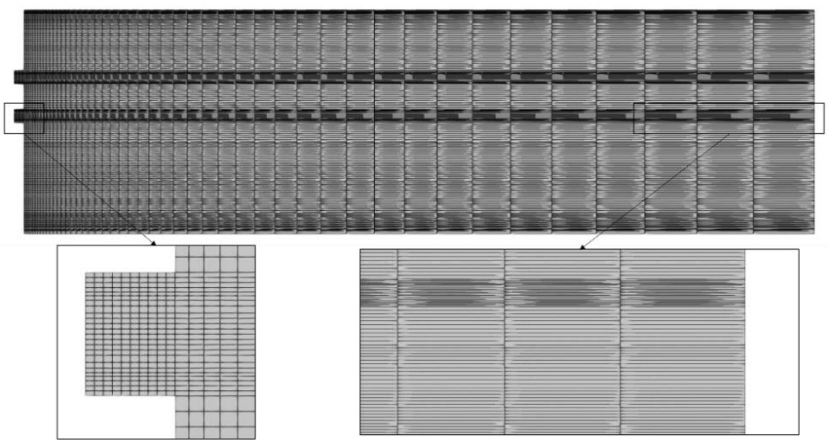

Figure 2: 2D meshing for the reacting domain.

DOI: 10.13009/EUCASS2023-040

3.3. Boundaries set up

The setup of the boundaries is addressed to the calculation of the wall heat flux on the firing plate boundary, consequently the temperature on that wall is fixed by the “cold” simulation given by Fluent and for the velocity field, a no slip condition is applied.
In the previous section (par. 3.2) the domain is axisymmetric, so the wall on the wedge has a symmetry condition and the chamber wall as a fixed temperature because it does not affect the temperature and wall heat flux distribution on the firing plate.

3.4. Convergence criterion

As the solver for the “reacting” domain is transient, it is important to fix a criterion for the convergence of the simulation, to do so the independent variables are considered during the whole computation and in the plot of Figure *3*, it is possible to see a part of the convergence plot in which the 2D field in the picture is averaged. For the convergence plot a slice very close to the firing plate is averaged and the percentage variation is plotted below timestep after timestep. As visible in the picture, when the variation of the independent variables is below the 3% the simulation is stopped, and the wall heat flux boundary of the firing plate is given to the simulation in fluent for the cold domain.

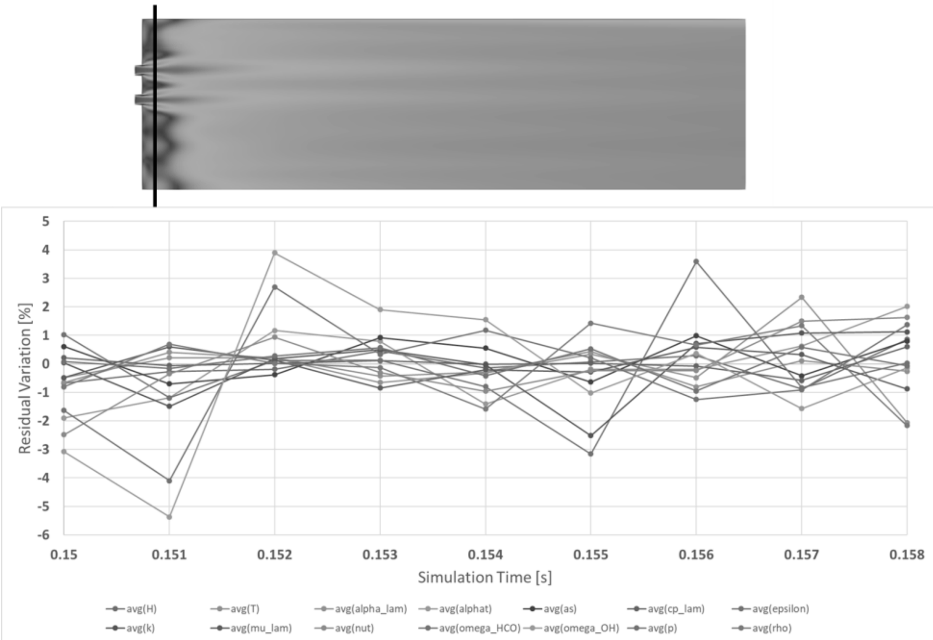

Figure 3: Convergence plot for independent variables.

3.5 Results

The plot provided in Figure 4 refers to the last loop iteration; the convergence is achieved with a variation of the wall heat flux in the reacting domain between 3% and 5%, except for the firing plate centre that has a very high heat flux up to the first injectors raw.
The 2D distribution of heat flux through the firing plate is very uniform and has some picks in correspondence of the injector holes; due to this distribution, it is possible to build a grid of high wall heat flux that connect all the injectors and it must be kept in mind during the design iterations.
Moreover, around the recess exit, an annular ring of high wall heat flux is present and dependent on the two-dimensional boundary surface of temperature given by the “cold” simulations; it can be considered part of the reality even if in the centre of the holes there are the nuts that will partially be subjected to this heat flux.
Apart from the extreme part of the domain and the simplified geometry model, the average wall heat flux found on the firing plate is reasonable and can be compared to the temperature distribution on the firing plat, right below the material limit of the copper alloy considered.

DOI: 10.13009/EUCASS2023-040

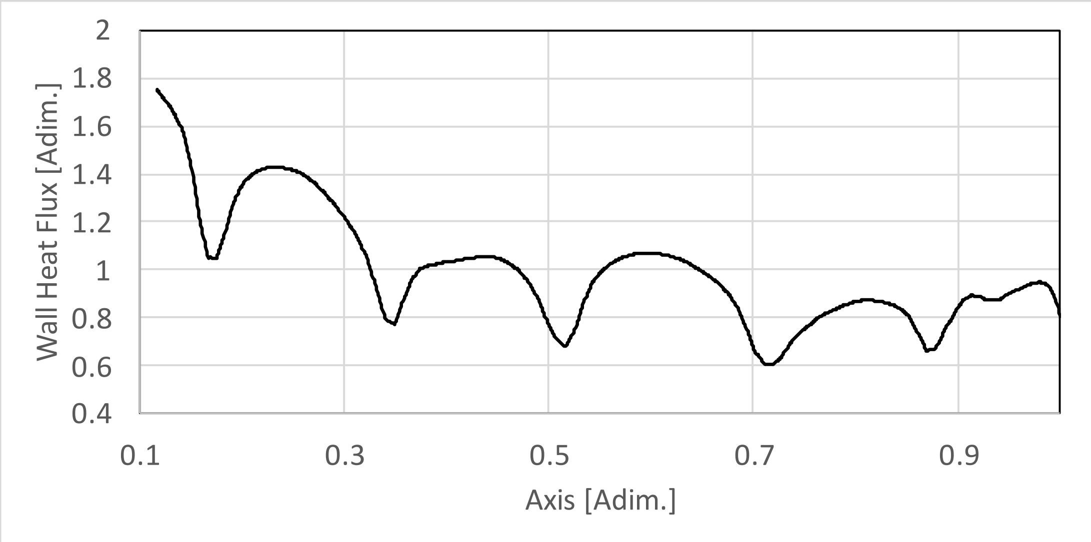

Figure 4: 2D plot dimensionless Wall heat flux results through the firing plate.

4. CFD calculations in the cold domain

The reacting domain CFD simulations provide an estimation of the wall heat flux applied to the firing plate. This map has been used as boundary condition for the CFD calculations in the reacting domain. The CFD simulation provide the temperature map obtained on the firing plate surface towards the combustion chamber. This trend is then used in the CFD calculation in the reacting domain to redefine the wall heat flux applied to the firing plate.

4.1 Model description

The simulations for the cold domain are performed by Fluent 2023 R1 ([5], [6]).
The simulation of the thermo-structural behaviour of the plate requires a 3D Conjugate heat transfer model (CHT) to model both fluid and solid domains:

- CH4 fluid domain
- Copper solid domain for the plate volume
- Inconel solid domain for the external circumferential seal

The computational method is 3D Reynolds Averaged Navier-Stokes (RANS), with realizable k-epsilon turbulence model and scalable wall functions. A standard roughness model has been considered. A second order Coupled Pressure-Based solver has been considered and a second order spatial discretization has been used.
The fluid properties taken into account have been obtained by NIST Real Gas Models database. The fluid properties have been obtained by NIST REFPROP release 10.0 database [7].

4.2 Geometry and mesh

Due to the symmetrical considered domain, a 3D domain of 180° has been considered. Due to the CHT model, a fluid volume is modelled and two solid volumes of different materials, copper and Inconel.
The mesh resolution is defined in agreement with the requirements defined by roughness model and the necessity to correctly simulate the fluid stratification occurring closely to the plate where the heat flux is applied. 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 8.5M polyhedral cells have been used (Figure 5).

DOI: 10.13009/EUCASS2023-040

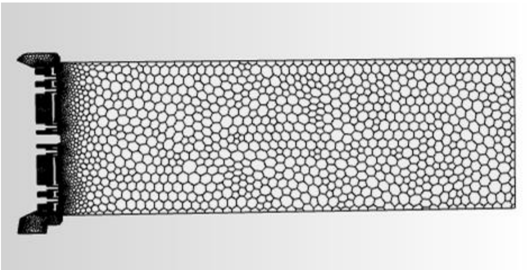

Figure 5 Solid volumes (firing plate and circumferential ring) and fluid volume.

4.3 Boundary conditions

As previously reported, the heat flux applied to the plate is obtained by CFD calculations in the reacting domain. Other boundaries conditions applied to the calculation are the methane MFR and the corresponding input pressure and temperature.
The roughness distribution has been defined by considering the roughness values related to different materials and technological aspects. In particular, the roughness value is assigned to all the surfaces with the exception of injectors, firing plate and counterplate that are characterized by lower roughness value. For the roughness values, the equivalent sand roughness used in the model has been obtained by the defined roughness R_a values.

4.4 Results

The main result obtained by CFD cold flow analyses is the temperature trend on the plate surface facing the combustion chamber. A 2D plot for the surface has been obtained and the same slice used for the reacting simulation (60°) has been extracted and provided as input for the reacting simulations of the following loop. An example of the temperature plot has been reported in Figure 6.

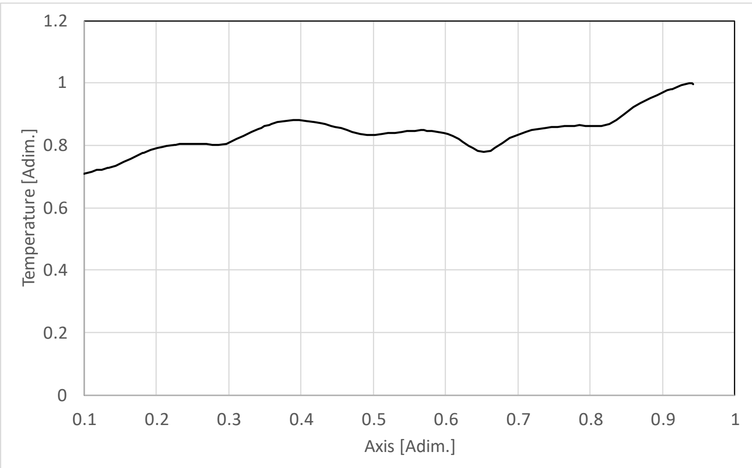

Figure 6 Temperature plot on the plate surface facing the combustion chamber.

6. FEM model

The selected configuration of firing plate 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.

DOI: 10.13009/EUCASS2023-040

6.1 Model description

The 3D FEM model is developed by using commercial software Apex 2021 [8] and Marc Mentat 2021.1 of MSC company [9], 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 firing plate, the computational domain is represented by an angular wedge of 60° containing the minimum path of injectors holes, and imposing periodic boundary conditions on the lateral walls

The total number of nodes is equal to 74821; the total number of elements is equal to 59740. The total number of firing plate is not too high because it is a part whole injector head model composed about 500k elements including contact bodies.
The following types of elements have been used:

- 59658 hexahedral elements of type hex8_7
- 82 pentahedral elements of type penta6_136

In Figure 7 is showed the detailed zooms mesh of the component modelled.

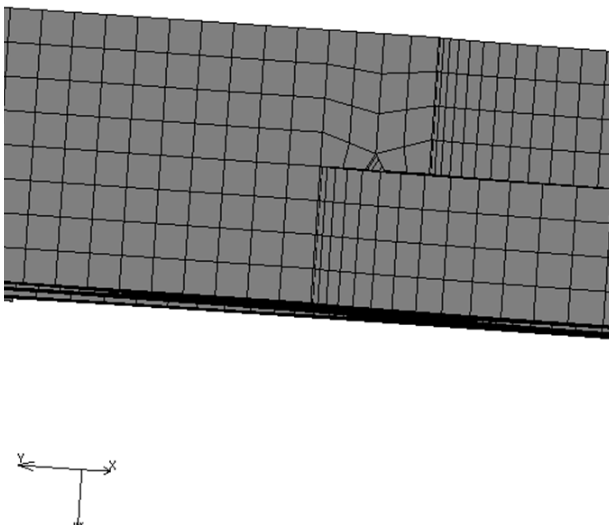

Figure 7 3D FEM Firing Plate Model Mesh

6.3 Material Properties

The material assigned to the component is Copper Based Alloy. The FEM analysis have a non-linear elasto-plastic material behaviour with temperature dependency. The material mechanical and thermal properties of Copper Based Alloy, 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.
The structure is fixed at Injector Head through nuts and touching contacts though seals.
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, 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.

DOI: 10.13009/EUCASS2023-040

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.

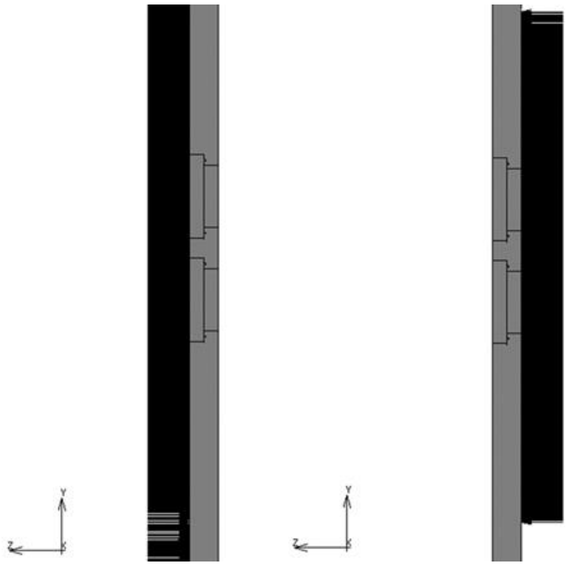

Figure 8 Hot Side Pressure (Left Side), Cold Side Pressure (Right Side)

Thermal Boundary Conditions

The heat exchange of the firing plate is obtained by assigning a peculiar value of heat transfer coefficient (HTC) as well as a sink temperature for cold side.
Instead, thermal flux is assigned to hot side because it is the only part exposed to the combustion fire. In particular, the thermal load on firing plate is applied until the combustion chamber diameter and not on the whole component, as for the mechanical load.
Such boundary conditions are provided by Reacting and Cold CFD (par. 3, par. 4).

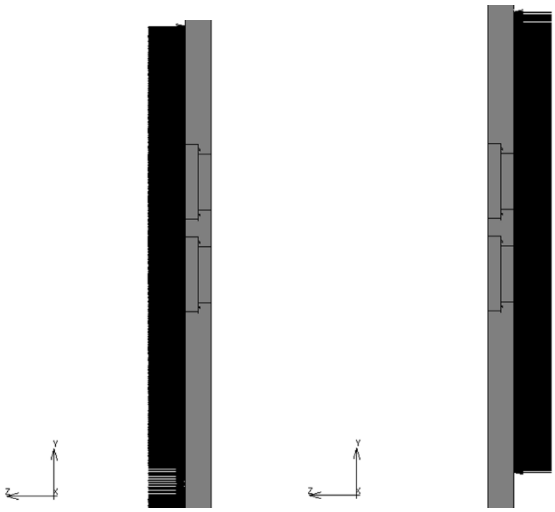

Figure 9 Hot Side Thermal Flux (Left Side), Cold Side HTC (Right Side)

DOI: 10.13009/EUCASS2023-040

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.

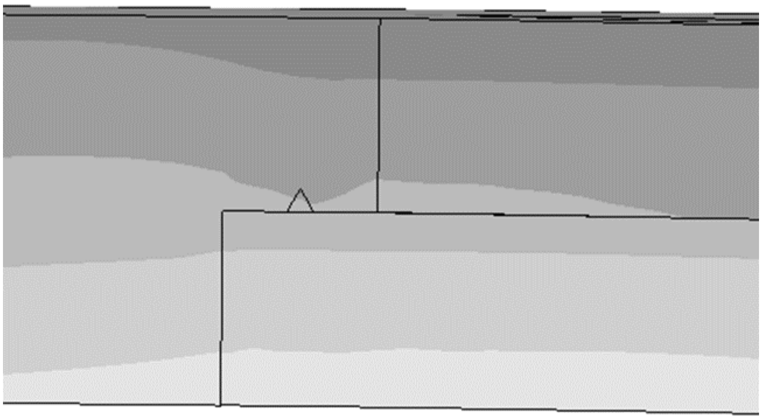

Figure 10: 3D FEM Thermo-Structural Analysis – Firing Plate

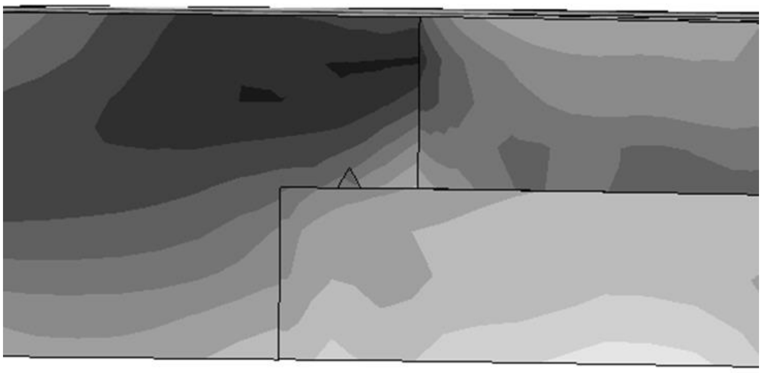

Figure 11: 3D FEM Thermo-Structural Analysis – Firing Plate Von Mises Stress

DOI: 10.13009/EUCASS2023-040

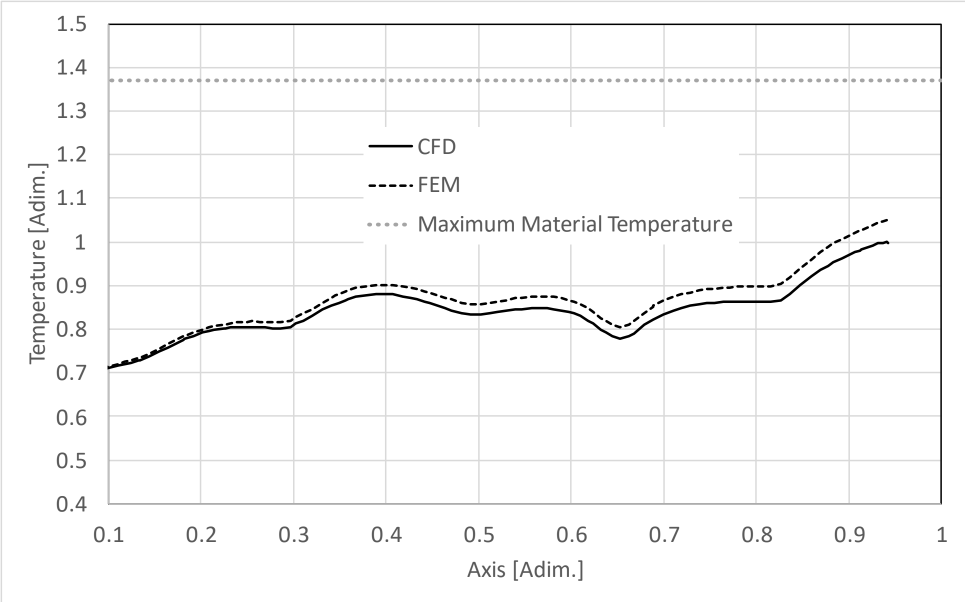

Figure 12: CFD/FEM and Maximum Material Temperature Comparison – Hot Side Firing Plate

5. Conclusions

A model to evaluate the temperature of the injector dome has been proposed and analysed in detail. The numerical model is composed of a reacting domain in which reacting CFD simulations are performed and that provide the wall heat flux through the firing plate surface for the cold domain, and of a cold domain, in which RANS compressible steady-state CFD simulations are performed and that provide wall temperatures to be used as boundary condition for the reacting flow simulations. Once obtained results for this loop, FEM simulations are performed to obtain feedback in terms of stress and strain to be checked with the mechanical properties. In the last step, the firing plate wall temperature calculated from FEM analysis and Ansys Fluent simulations is checked for consistency before to give feedback on the design and, if needed, a geometry change proposal to restart the design process verification loop.
The results obtained by applying the described model to the design of a real injector dome has been also reported. The proposed model has been verified able to correctly evaluate the thermo-structural behaviour of the analysed component with a direct coupling with the fluid-dynamic behaviour in the dome. The consistency of the used models has been also verified. The approach has been also used to optimize the dome configuration.
Further improvements of the model can be obtained by direct comparison with experimental data obtained by firing testing of the analysed dome configuration.

DOI: 10.13009/EUCASS2023-040

References

[1] Degrez, G., P. Barbante, M. de la Llave, T. Magin, and O. Chazot. 2001. Determination of the catalytic properties of TPS materials in the VKI ICP facilities. In: *3rd ECCOMAS Computational Fluid Dynamics Conference*. 162–167.

[2] Magin, T., and G. Degrez. 2004. Transport algorithms for partially ionized and unmagnetized plasmas. *J. Comput. Phys.* 198:424–449.

[3] AGARD. 1998. A selection of test cases for the validation of large eddy simulations of turbulent flows. Agard Advisory Report 345. North Atlantic Treaty Organization.

[4] Rini, P. 2006. Analysis of differential diffusion phenomena in high enthalpy flows, with application to thermal protection material testing in ICP facilities. PhD Thesis. Université Libre de Bruxelles, Faculté des Sciences Appliquées.

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

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

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

[8] Oracle APEX Release 21.2 User Manual

[9] Marc and Mentat Documentation: Release 2021

[10] A. Terracciano, S. Carapellese, G. Bianchi, D. Liuzzi, M. Rudnykh, D.Drigo, F. Del Brusco, *Additive Layer Manufacturing Technology In Avio Injector Head Design*, 7th European Conference for Aeronautics and Space Sciences (EUCASS), EUCASS2017-455

[11] 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

[12] 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

[13] 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

[14] 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