DOI: 10.13009/EUCASS2022-5871
9TH EUROPEAN CONFERENCE FOR AERONAUTICS AND SPACE SCIENCES (EUCASS)

3D Conjugate heat transfer model for simulation of Heat Transfer by High Roughness Cooling Channels

Viviana Ferretti[1], Matteo Sciarra[1]**, Daniele Liuzzi[1]***, Daniele Drigo[1]*****
[1]AVIO S.p.A. Via Latina, snc (SP 600 Ariana km 5,2) 00034 Colleferro (RM) – Italia
**viviana.ferretti@avio.com, **matteo.sciarra@avio.com, ***daniele.liuzzi@avio.com, ****daniele.drigo@avio.com*

Abstract
An Ansys Fluent 3D conjugate heat transfer model has been developed for the simulation of the heat transfer problem in cooling channels characterized by high roughness. The model combines the industrial necessity to obtain results in reasonable times with a detailed description of the involved phenomena, such as propellant stratification.
The model has been validated with respect to experimental data and applied to different cooling fluid, water and methane, characterized by different stratification. The validated model has been applied to a cooling channel tested by experimental cold flow tests; the obtained wall temperature and main fluid-dynamic results are reported.

1. Introduction

A 3D conjugate heat transfer model 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.
The CFD model has been developed by using Ansys Fluent 2021 R2 [1] [2] and it has been used to simulate the cooling channel of an engine tested by cold flow campaign.
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. The approach followed for the validation is hereafter resumed:

- Comparison with respect to experimental data for water cold flow test case; the comparison allows the definition of a proper channel roughness profile to match the experimental pressure losses.
- Comparison with respect to numerical model for methane cooling with applied heat flux and with the application of roughness profile defined during previous step

The considered experimental data have been obtained by cold flow test campaign; details about the used data are reported in par. 3.
The numerical data used for the correlation have been obtained by an in-house Avio code "Cooling DL". A description of this Q1D model has been reported in par. 4.
The main outputs of the activity are represented by the CFD model setting, the definition of a proper roughness profile characterizing wall surfaces, the fluid-dynamic field global description, the wall temperatures obtained at the solid surfaces.

DOI: 10.13009/EUCASS2022-5871

2. Experimental Test

The experimental data considered for the model validation are related to Cold Flow Tests performed on a test engine. The data have been obtained by test campaign performed with counterflow water cooling.
The data have been used to validate the high roughness model; a proper roughness profile has been defined to match the experimental pressure losses. The assumed test case does not consider heat flux application and it is used to test the roughness model and to define the roughness profile.
Pressure and temperature measures along the cooling channel have been considered with reference to the stationary phase reached during the test.

3. Cooling DL Q1D Code

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 (for example CRIO9 Combustion Chamber).
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 results obtained by Cooling DL application have been used as reference for the 3D Ansys Fluent High Roughness Modelling

4. 3D Ansys Fluent High Roughness Modelling

The High Roughness model here presented has been performed by Ansys Fluent 2021 R2 [1] [2].
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 [1] [2]. It has been completed by means of high roughness wall modelling that has been properly set to be fully representative for the case of interest.

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). This model can be used for Spalart Allmaras and k-ω SST model and requires low Re number or the resolution of the boundary layer ($y^+ \sim 1$). By literature, the k-ω SST is typically somewhat more accurate in predicting the details of the wall boundary layer characteristics than the Spalart-Allmaras model. Anyway, the use of high roughness model and NIST database for fluid properties combined with k-ω SST model has not provided useful results up until now. Further analyses must be considered to evaluate the use of k-ω SST model, also considering the very higher computational effort required by this model with respect to the considered requirements. The Spalart Allmaras turbulence model has been then selected for the proposed model.

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.

4.1 Fluid properties

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 [3]. The selected thermodynamic variables for the database are fluid pressure and temperature; required properties (density, Cv, Cp, thermal

DOI: 10.13009/EUCASS2022-5871

conductivity, cinematic viscosity, enthalpy, compressibility factor, speed of sound and specific gas constant) are then obtained for the given fluid in a given state.

4.2 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 1).
The same mesh has been used for all the analysed test cases.
A fully unstructured approach has been adopted to generate the computational mesh, mainly due to the geometrical complexity in throat region and welding joint. These regions are characterized by mixing of two channel and then by 3D flows. 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 $y^+ \sim 1$ in each point of the analysed 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 2. Liner hot surface is the surface where the heat fluxes can be applied in case of hot 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 2.

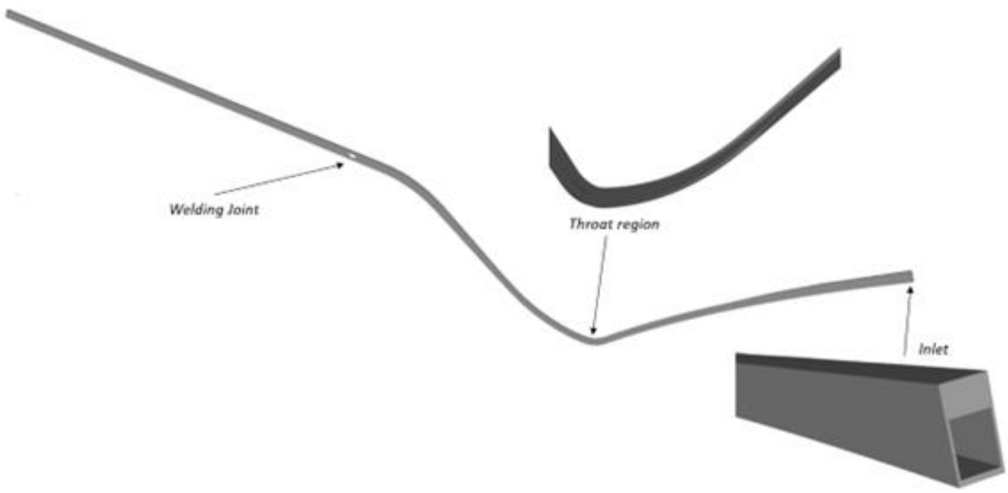

Figure 1: Cooling channel domain.

DOI: 10.13009/EUCASS2022-5871

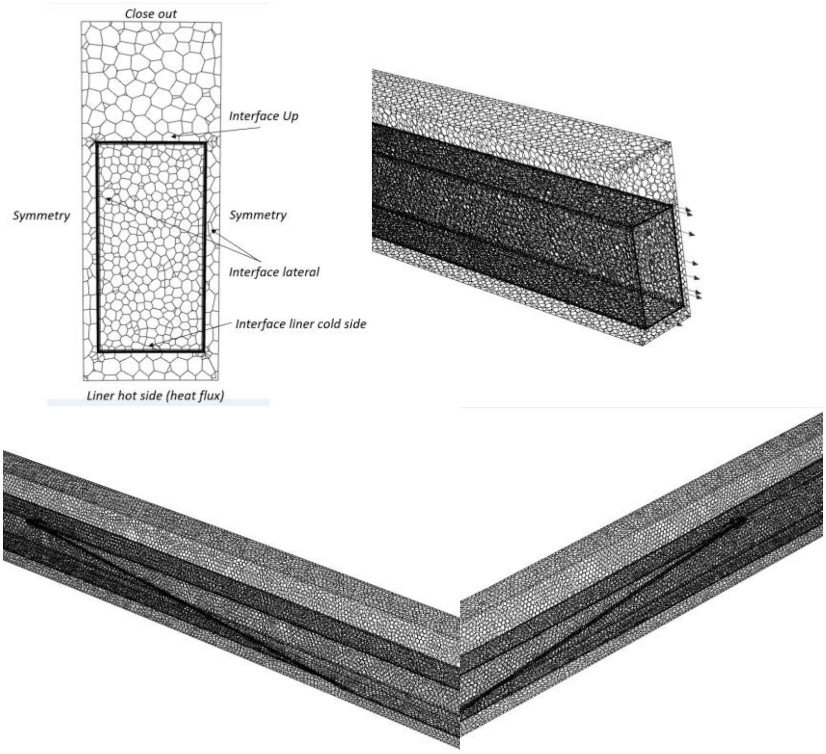

Figure 2: Cooling channel domain mesh.

5. High roughness modelling for cold flow experimental test reconstruction

As previously reported, the test case considered for the comparison has been obtained by experimental test campaign performed on test equipment with counterflow water cooling and no heat flux applied (cold flow test).
In order to test the complete model setting, the same UDF approach for fluid properties has been used also for cold water.
The main results obtained by this test case are the model validation wrt experimental data and the definition of a proper channel roughness profile, able to match the pressure losses within the channel. The obtained roughness profile must be considered representative of real surface roughness and it has been used for all successive simulations performed for the test case configuration to obtain description of cooling system performance and behaviour.
In agreement with the requirements of high roughness modelling, a maximum $Y^+ < 0.3$ has been obtained in the throat region.
The results obtained by test case reconstruction are here reported in Figure 3 in terms of bulk pressure and temperature distribution along the channel. The pressure values are scaled with respect to the channel operating pressure (P_{op}) defined as the numerical mean pressure value between inlet (P_{in}) and outlet pressure (P_{out}). The same logic has been assumed for temperature scaling (T_{in}, T_{out}, T_{op}).
Both the trends show a very good agreement between CFD numerical data and experimental ones. With respect to the pressure losses, an underestimation of 1.5% has been obtained, while for the temperature increase an underestimation of 4.5% has been obtained. With no heat flux applied, the temperature increase is only due to shear temperature increase. A resume of the comparison between cooling channel performance is reported in Table 1.

DOI: 10.13009/EUCASS2022-5871

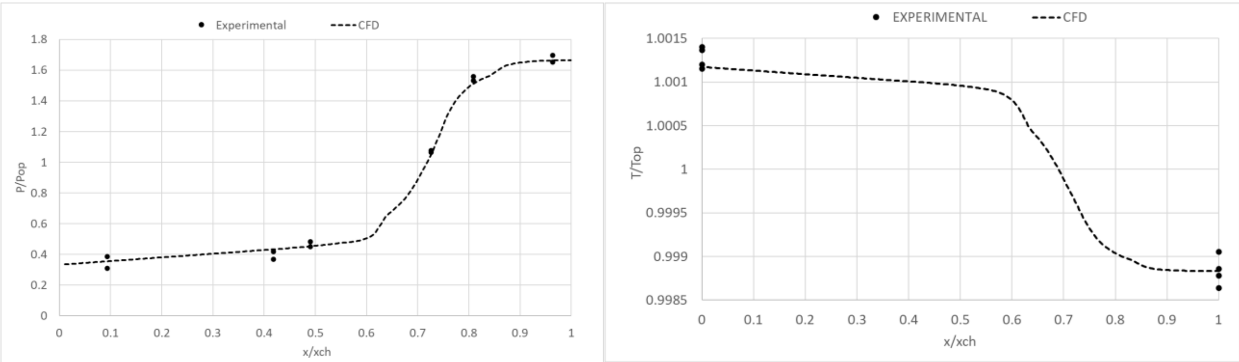

Figure 3: Cold flow test: bulk pressure and temperature along the channel.

Table 1: Cold flow test - results comparison.

	CFD	Experimental
P_{in}/P_{op}	1.663	1.675
T_{in}/T_{op}	0.9988	0.9988
P_{out}/P_{op}	0.355	0.348
T_{out}/T_{op}	1.0012	1.0013
$\Delta P/P_{op}$	1.308	1.327
$\Delta T/T_{op}$	0.0023	0.0024

The results previously reported have been obtained with the definition of the equivalent roughness profile reported in Figure 4 (also in this case the equivalent roughness profile has been scaled with respect to a mean equivalent roughness value). A maximum roughness value is obtained in the region characterize by maximum curvature (convergent-throat-divergent region), while a lower value describes the cylindrical region of the combustion chamber. The channel has been described by a piecewise linear profile. Future studies will be performed focusing on the shape of the roughness profile.
The equivalent roughness profile correctly describes the cold flow pressure losses of the channel and so provide a correct description of the channel equivalent roughness. This profile has been then used to perform a methane cooling channel simulation with heat flux application.

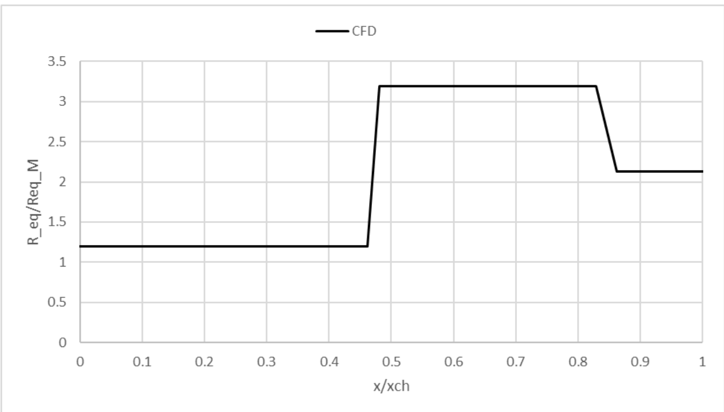

Figure 4: Cold flow test: roughness profile along the channel.

The static temperature in cross sections obtained for cylinder and throat region are reported in Figure 5. The heating effects due to shear and roughness effects can be seen in both sections (temperature scaled with respect to temperature mean value for fluid volume range).

DOI: 10.13009/EUCASS2022-5871

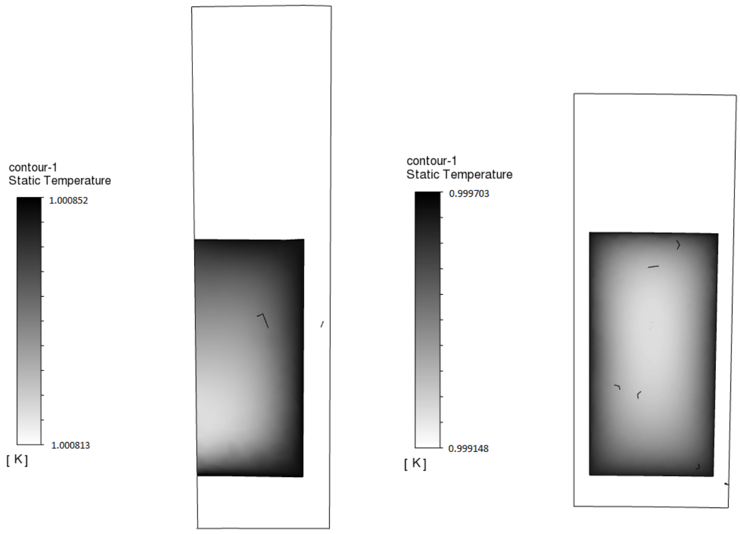

Figure 5: Cold flow test: static temperature in throat region (left) and cylinder region (right).

6. High roughness modelling for hot flow simulation

Once defined the equivalent roughness profile to be used for the channel surfaces, the model has been used to simulate the same cooling channel with counterflow methane cooling and heat fluxes applied in the combustion channel internal surfaces (heat flux surface reported in Figure 2). This configuration is quite close to a possible real regenerative cooling system.

The same mesh reported in par. 5.2 has been also used for this test case. The considered operative conditions are the engine nominal ones and the methane properties corresponding to these conditions have been defined by means of the UDF approach previously reported.

The results obtained by the high roughness model have been compared with the ones obtained by applying Q1D Cooling DL. The same boundary conditions used for Q1D simulations have been considered; the imposed heat flux profile (scaled with respect to an input/output mean value) has been reported in Figure 6.

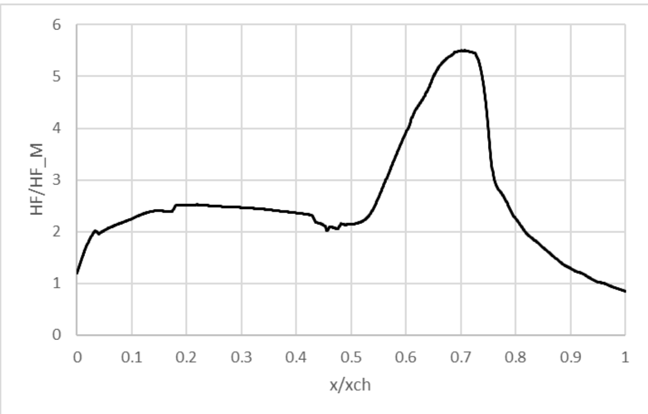

Figure 6: Methane hot test – heat flux profile.

In agreement with high roughness model requirements, the Y^+ value has been verified and a mean value of 0.35 has been obtained for methane cooling, with a maximum local value of 1.6 in the throat region.

The results obtained by CFD model and a comparison with Q1D results has been hereafter reported in Figure 7 for bulk pressure and temperature. The same approach reported in par. 6 for data post processing has been used. A very good agreement has been obtained also in this case between the two model as far as bulk properties. Slight differences are obtained in the region characterized by three-dimensional flow effects, throat and welding joint regions.

DOI: 10.13009/EUCASS2022-5871

A synthesis of cooling channel performance and a comparison with respect to Q1D results is reported in Table 2. A difference of 1.4% has been obtained in terms of pressure losses and of 0.3% in terms of obtained temperature increase.

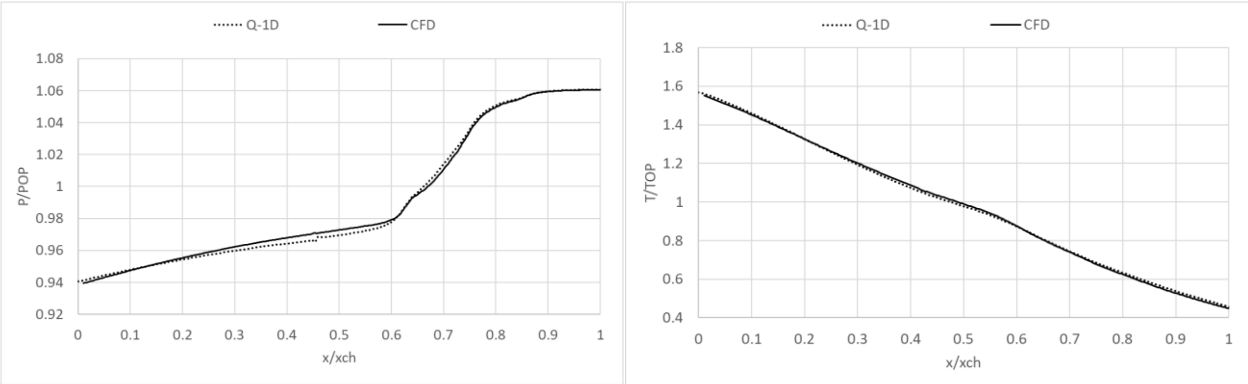

Figure 7: Methane hot test: bulk pressure and temperature along the channel.

Table 2: Methane hot test - results comparison.

	CFD	**Q1D**
P_{in}/P_{op}	1.060	1.061
T_{in}/T_{op}	0.4497	0.4583
P_{out}/P_{op}	0.940	0.941
T_{out}/T_{op}	1.5478	1.5600
$\Delta P/P_{op}$	0.121	0.119
$\Delta T/T_{op}$	1.0981	1.1017

All other bulk properties have been reported in Figure 8. The same good agreement already seen for temperature and pressure has been maintained for all other properties. Notwithstanding the good agreement of trends, slight numerical differences can be observed for velocity and density values in cylinder region, after 3D flow effect of welding joint region, and for velocity value in throat region.

DOI: 10.13009/EUCASS2022-5871

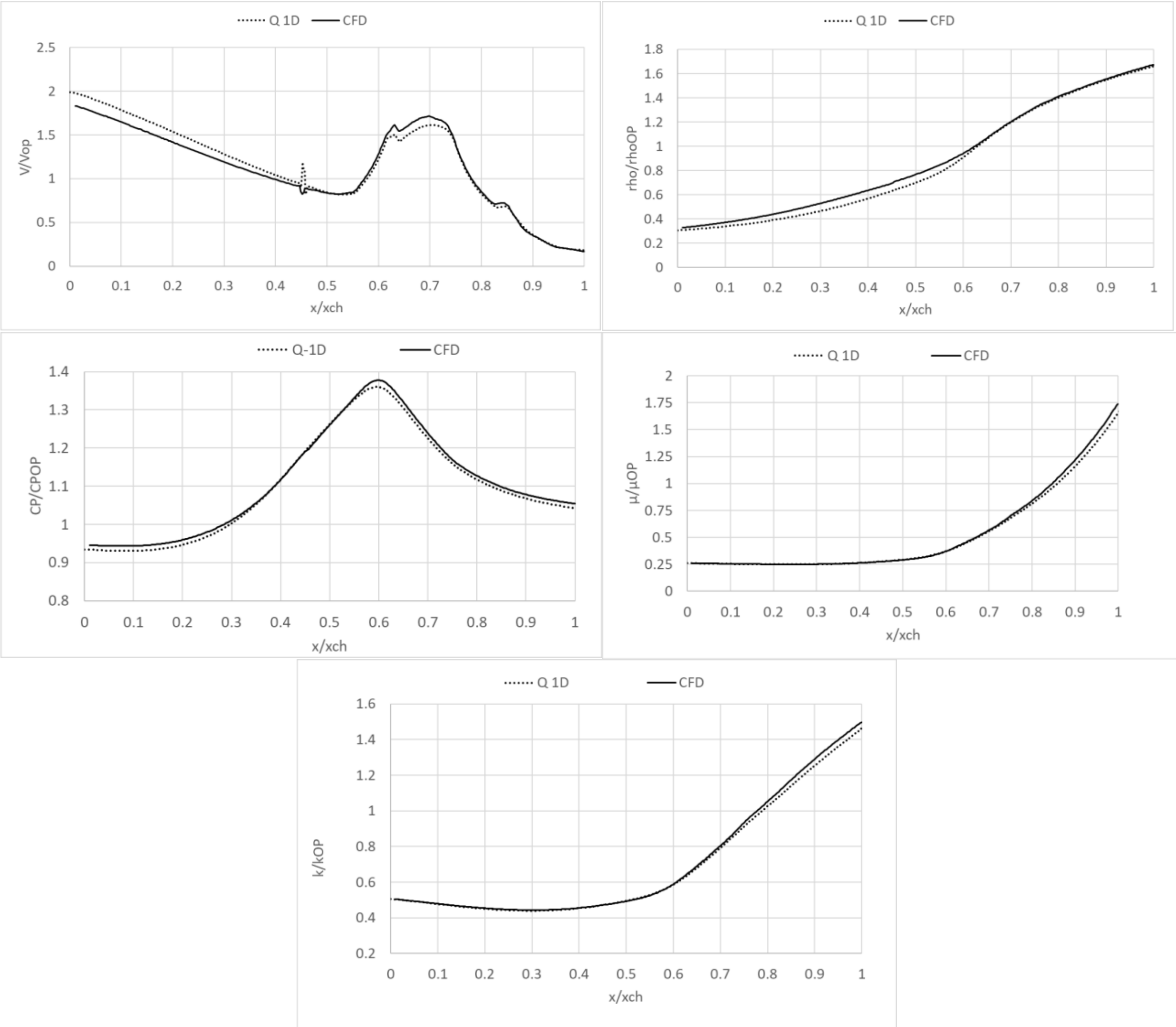

Figure 8: Methane hot test: bulk properties along the channel.

The wall temperature obtained for the liner wall is reported Figure 9. For the hot gas side, the same trend than Cooling DL has been obtained; anyway, while the same temperature value is reached in the cylinder region, a lower wall temperature in the throat region is obtained, coupled with a higher wall temperature value in the convergent region.
For the cold side, differences are obtained in the convergent region, where an overestimation of the temperature increase has been obtained, and in throat region. Except for the translation, similar trend has been obtained in the cylinder region. These differences are mainly due to high roughness model available in the CFD model and to 3D flow effects that can be modelled.
The temperature distribution on the considered surfaces has been reported in Figure 10 (temperature scaled with respect to average value within volume range). The temperature increases in throat region and in the welding joint is clearly visible. Also in this figure, a temperature value in the cylinder region close to the one of the throat one can be observed.

DOI: 10.13009/EUCASS2022-5871

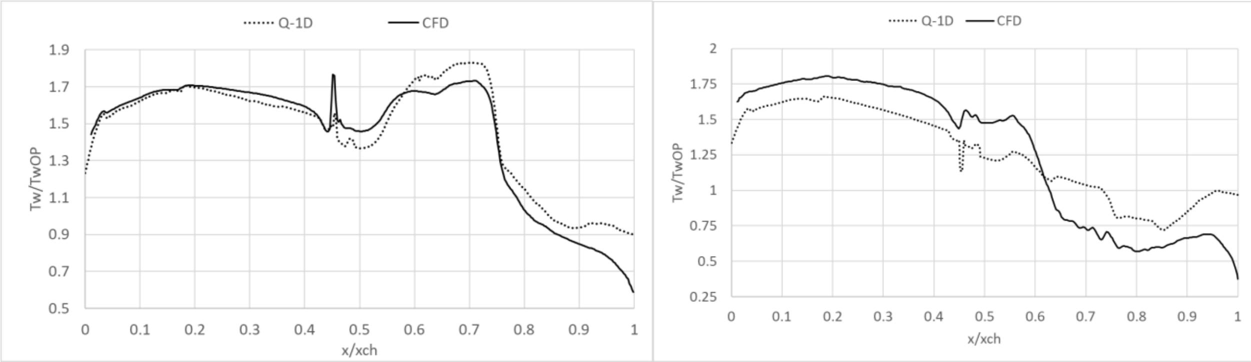

Figure 9: Methane hot test: liner temperature hot side (left) and cold side (right).

Figure 10: Methane hot test: liner temperature hot side (up) and cold side (down).

A contour plot of cross sections in throat region and in cylinder one is reported in Figure 11 and Figure 12 respectively (for each volume, temperature values are scaled with respect to the volume temperature range mean value). Notwithstanding the high temperature of liner hot side, the fluid in the throat region still has a homogeneous temperature across the section, while in the cylinder region a fluid stratification is more evident, with maximum values reached close to the liner cold surface.

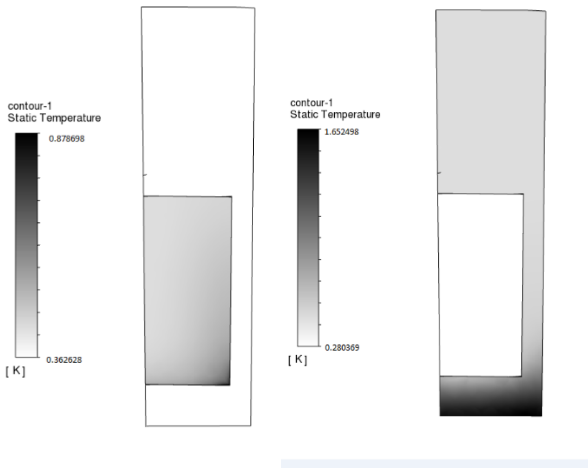

Figure 11: Methane hot test: static temperature in throat region for fluid (left) and solid (right).

DOI: 10.13009/EUCASS2022-5871

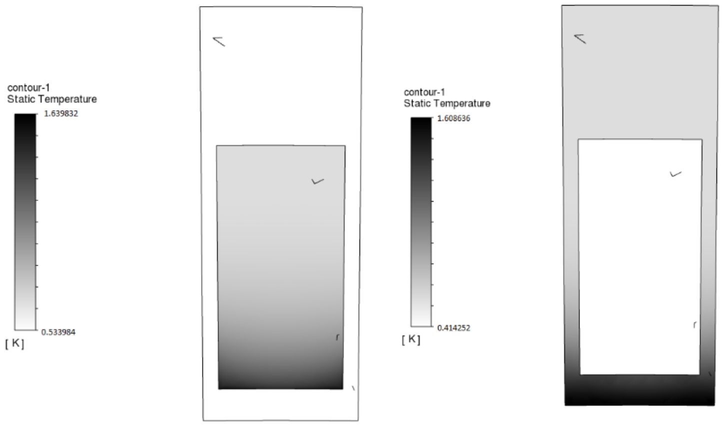

Figure 12: Methane hot test: static temperature in cylinder region for fluid (left) and solid (right).

7. Conclusions

A 3D conjugate heat transfer model has been developed for the simulation of the heat transfer problem in cooling channels characterized by high roughness surfaces. The model has been validated with respect to both experimental and numerical data considering different operative conditions and different cooling fluid.
The presented results show a very good agreement of CFD results with respect to the available data, showing a very good description of the model in terms of the involved phenomena.
Further analyses will be completed to obtain a better model functioning, for example with respect to roughness profile shape.
In order to complete the analysis of the model, in particular as far as concerned quantitative aspects, further analyses will be performed with respect to experimental data obtained from test campaign performed for counterflow methane cooling channel. The results obtained from these tests will allow a better description of quantitate aspects such as stratification and methane cooling performance.

References

[1] ANSYS Fluent Theory Guide - ANSYS, Inc.
[2] ANSYS Fluent User's Guide - ANSYS, Inc
[3] Eric W. Lemmon, Ian H. Bell, Marcia L. Huber, Mark O. McLinden 2018: REFPROP Documentation Release 10.0