Aerospace Europe Conference 2023
Joint 10th EUCASS – 9th CEAS Conference

Abstract
Preferred Topics: PROPHY
Corresponding author: Alessio Gizzi
e-mail of corresponding author: alessio.gizzi@avio.com
Type: Oral
Status of corresponding author: Regular
For student corresponding author: --

Title

DESIGN LOOP PROCESS FOR COOLING JACKET OF LIQUID ROCKET ENGINE

Authors

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)

** Corresponding author*
1viviana.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 used 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 behavior characterizing the liquid rocket engine cooling systems, such as three-dimensional effects, possible supercritical behavior of the coolant, thermo-structural behavior of the solid. 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. The loop is defined by means of the following model properly defined with respect to the focused product and involved aspects:

- Hot 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 behavior [1]. Indeed, only the coupled description of fluid stratification and roughness effects allow the correct evaluation of thermal behavior 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 2021 R2.
- 3D FEM thermo-structural non-linear analyses to obtain a description in terms of structural behavior, stress, strainof 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 behavior is available. The results allow the identification of possible design improvement. If identified, the solutions are provided as input to Hot 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 behavior.
All the models used within this loop have been validated through experimental tests and in-house code developed by AVIO. A complete description of the applied loop and models will be presented in the full paper. The paper will also report the results obtained by applying the described model to the design of a real cooling system.

References

[1] 3D Conjugate heat transfer model for simulation of Heat Transfer by High Roughness Cooling Channels, Ferretti Viviana, Sciarra Matteo, Liuzzi Daniele, Drigo Daniele