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 INECTOR DOME OF LIQUID ROCKET ENGINE

Authors

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)

** Corresponding author*
*[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 need 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 analyzed in detail. The model consists of two iterative steps integrating the hot domain and the cold one. The numerical model is so composed:

- Hot domain: reacting CFD simulations performed in the OpenFOAM environment by the integrated OpenSMOKE solver
- Cold domain: RANS compressible steady-state CFD simulations performed by Ansys Fluent 2021 R2 [1]
- Hot/Cold domain connected: FEM thermo-structural simulations by MSC MARC 2021.

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 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 iteration 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 in the full paper. The paper will also report the results obtained by applying the described model to the design of a real injector dome.

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