Wall Heat Fluxes in Rocket Combustion Chamber with Porous Injector Head

V.P. Zhukov and D.I. Suslov
Institute of Space Propulsion, German Aerospace Center (DLR)
Langer Grund, 74239 Hardthausen, Germany

The accurate prediction of heat loads is one of the key problems of rocket engine design. The design of rocket combustion chamber requires an extensive trial-and-error testing while CFD (computational fluid dynamics) modelling not only helps to reduce the amount of expensive hot tests, but also makes possible the transfer of the results of sub-scale tests to full scale engines. The main targets of the current work are the wall heat fluxes, the performance, and the efficiency of the combustion chambers with a porous injector head. In the current work wall heat fluxes have been measured in a sub-scale combustion chamber and then simulated using a CFD code. The novelty of the work is the usage of a porous injector head. This concept already showed advanced features as high combustion stability and combustion efficiency [1]. The combustion chamber consists of 7 circumferential sections which allow to measure the wall heat fluxes calorimetrically. The combustion chamber is also equipped with pressure sensors distributed along the axis of the chamber. The reactive flow of hydrogen and oxygen at pressures of 60–80 bar has been simulated using ANSYS CFX 13.0 CFD software. The propellants were injected into the combustion chamber at cryogenic temperatures, therefore the real gas equation of state has been employed in the modelling. The turbulent flow in the combustion chamber has been modelled by the Reynolds-averaged Navier-Stokes equations (RANS) with the help of the shear stress turbulence (SST) model. The turbulent combustion has been modelled by Eddy Dissipation Model (EDM). The radiative heat transfer has been also taken into account in the simulations.

The experimental results are featured by the maximum of the wall heat flux at the second calorimetrical section (50–100 mm from the injector head). The initial simulations in a 2D axisymmetric computational domain [2] did not reproduce this maximum. The current simulations in a 3D domain showed the connection between the maximum of the wall heat flux near the injector plate and the arrangement of the injectors in it. Taking into account the specifics of the experiment namely that the heat flux in the section next to the injector head is underrated and in the section next to the nozzle is overrated the results of the simulations in the 3D domain can be considered as in a very good agreement with the experimental data. The numerical simulations predict accurately the combustion efficiency and the pressure distribution in the combustion chamber as well.

[1] Deeken, J., Suslov, D., Haidn, O.J., Schlechtriem, S., Combustion Efficiency Sensitivity Studies of the API Injector Concept, 49th Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, AIAA 2011-0793,Orlando, Florida USA,4-7 January, 2011.
[2] V.P. Zhukov, D.I. Suslov, and O.J. Haidn, CFD Simulation of Flow in Combustion Chamber with Porous Injector Head and Transpirationally Cooled Walls, Paper id: 29 at 4th European Conference for Aerospace Sciences, Saint Petersburg, Russia, July 4, 2011 – July 8, 2011.