Numerical Investigation of Liquid Film Cooling Performance on An Experimental Rocket Nozzle

YANIK Umutcan[1*], OZTURKMEN Musa Onur[2], TURGUL Doruk[3]

** Corresponding author*

Roketsan Missiles Inc. 06780 Elmadag Ankara,

[1]umutcan.yanik@roketsan.com.tr

[2]oozturkmen@roketsan.com.tr

[3]doruk.turgul@roketsan.com.tr

Abstract

Rocket engines are essential components of space vehicles, and their efficiency and reliability are of utmost importance in ensuring the success of space missions. One major challenge in rocket engine design is managing the high temperatures generated during operation, which can damage engine components and compromise performance. Film cooling is presently considered as one of the most effective methods, as an alternative or addition to regenerative or external cooling, for the protection of structural materials which undergo high level of heat fluxes especially for longer operation time intervals. The thermal load on the material surface coming from the hot fluid stream is dramatically decreased due to the existence of a thin liquid or gaseous film layer which act as an insulator. Cooling of turbine components, rocket combustion chambers and nozzles are among the most well-known industrial applications of film cooling[1,2].

In this study, a liquid film cooling experiment is investigated numerically using computational fluid dynamics (CFD) analyses. The simulations were performed using a three-dimensional model that was developed based on the governing equations of fluid mechanics and heat transfer. The experimental setup[3] is taken from National Advisory Committee for Aeronautics (NACA). The coolant injector was mounted between the combustion chamber and the exhaust nozzle. The injector provided a supply annulus for the coolant from which the coolant flowed through an annular slot to the inside surface of the nozzle. The hot gas is comprised of liquid ammonia and liquid oxygen combustion products. The adiabatic flame temperature is approximately 3000 K. The combustion chamber total pressure varies from 15.85 to 17.25 bar; whereas the oxidizer to fuel mass flow ratio alters from 1.38 to 1.72 and the coolant mass flow to total flow mass ratios varies from 12.7 to 23.6 for different test cases. In the experiments, water and ammonia are tested as liquid coolants.

Having mentioned the details of the experiments, a number of test cases are selected for CFD simulations. First of all, the liquid film cooling can be affected by the utilized numerical approaches. Different simulations are performed to investigate effects of numerical models, such as turbulence models. The solutions are obtained by employing the realizable k-epsilon model and k-omega SST (Shear Stress Transport) model. The computational domain is formed in three dimensions by using a 4-degree slice of the exhaust nozzle to reduce computational cost. Grid convergence study is performed by producing various meshes of different sizes for film cooling effectiveness. In addition, effect of near wall treatment is investigated by changing the value of y+. Surface temperature distribution on the combustion chamber wall and liquid film cooled length are compared with the experimental data and a reasonable mesh size is obtained.

To sum up, the main objective of this work is to validate liquid film performance with the experimental data available in the literature and to assess the impact of various coolant mass flow to total flow mass ratios, different mesh sizes and turbulence models on the film cooling efficiency.

References

[1] S.R. Shine, S. Shri Nidhi, “Review on film cooling of liquid rocket engines”, 2212-540X 2018 National Laboratory for Aeronautics.

[2] William M. Grisson “Liquid film cooling in rocket engines”, Technical Report AEDC-TR-91-1, DTIC, 1991.

[3] E. Abramson, Investigation of internal film cooling of exhaust nozzle of a 1000-pound thrust liquid ammonia liquid oxygen rocket, Technical Report, NACA RME52C26, 1951.