Numerical Simulation of a Regenerative Cooling System for a Rocket Engine

Esteban Fernández Babaglio[1,2,3], Ana Scarabino[1], Federico Bacchi[1]

[1]Grupo Fluidodinámica Computacional GFC – Universidad Nacional de La Plata,
[2]Grupo Ensayos Mecánicos Aplicados, GEMA – Universidad Nacional de La Plata
Calle 116 entre 47 y 48 – (1900) La Plata, Argentina
[3]Comisión de Investigaciones Científicas de la Provincia de Buenos Aires
esteban.fernandez@ing.unlp.edu.ar, www.gfc.ing.unlp.edu.ar

ABSTRACT

Heat transfer in the thrust chamber is of great importance in the design of liquid propellant rocket engines. Regenerative cooling is an advanced method which can ensure not only the proper running but also higher performance of a rocket engine. In the context of rocket engine design, this is a configuration in which the propellant is passed through tubes or channels around the nozzle to cool the engine. The heated propellant is then injected directly into the main combustion chamber for combustion there [1]. The heat exchange capacity of this system depends on the propellant flow, its thermodynamic properties, the wall thermal conductivity and the geometry of nozzle and channels. It is thus necessary a careful study of the involved variables in order to optimize the overall heat transfer [2-4].

The objective of this work is to predict the heat transfer and temperature of a liquid propellant rocket engine, with a numerical model of its regenerative cooling system. The model considers heat transfer from the combustion chamber and nozzle through the nozzle wall to the cooling fuel, which flows through rectangular channels embedded in this wall. Simulations were carried out with Fluent 6.3. Different turbulent models and meshes were carefully tested in order to achieve the best results for both velocity and temperature profiles for the well-established experimental results of pipe flow in circular ducts with heat exchange. The boundary condition for the internal nozzle wall was set as a heat flux, which was computed following Bartz model [1-5]. The tested turbulence models, each with an adequate meshing near the walls, were: k-ε realizable with enhanced wall treatment, k-ω standard with"transitional flow correction" and k-ω SST with"transitional flow correction.

Results show the temperature distributions for the nozzle wall and the propellant under different operating conditions. The model has proven to be a valuable tool for optimizing the coolant propellent flow, the number of channels needed for an adequate cooling, and the channel geometry.

REFERENCES

[1] Dieter K. Huzel and David H. Huang: "Modern Engineering for Design of Liquid-Propellant Rocket Engines", AIAA, 1992.

[2]Wang T. and Luong V. "Hot-Gas Side and Coolant-Side Heat Transfer in Liquid Rocket Engine Combustors", Journal of Thermophysics and Heat Transfer. 1994

[3]M. E. Boysan, A. Ulas, K. A. Toker, B. Seckin, "Comparison of Different Aspect Ratio Cooling Channel Design for a Liquid Propellant Rocket Engine". Middle East Techical University. 2007

[4] "CFD analysis of transcritical methane in rocket engine cooling chanels" Marco Pizzarelli, Francesco Nasuti, Marcello Onofri. Dipartamento di IngegneriaMeccanica e Aerospaziale. 2011

[5] D. R. Bartz, "A Simple Equation for Rapid Estimation of Rocket Nozzle Convective Heat Transfer Coefficients," *Jet Propulsion,* 37, 1, January 1957, pages 49-51.