SPACE PROPULSION SYMPOSIUM (C4)
Interactive Presentations (IP)

Author: Mr. Guangxu Wang
The 11st Institute of the Sixth Academy, China Aerospace Science and Technology Corporation (CASC), China, wangguangxu1010@126.com

Ms. Jinhua Hu
National Key LAB of Science and Technology on LRE Xi'an, China, China, wangguangxu1010@126.com
Mr. Xiaobo Shi
National Key LAB of Science and Technology on LRE, China, wangguangxu1010@126.com
Mr. Jianwen Yang
National Key LAB of Science and Technology on LRE, China, wangguangxu1010@126.com
Mr. Zhanyi Liu
National Key LAB of Science and Technology on LRE, China, wangguangxu1010@126.com

WALL TEMPERATURE PREDICTION OF REGENERATIVE COOLED ROCKET ENGINE BASED ON A COUPLED HEAT TRANSFER MODEL

Abstract

The continuous demand for higher performance engines has resulted in higher pressure and temperature for the combustion chambers. In order to ensure structural integrity of thrust chamber, there is a need to perform a comprehensive thermal analysis and predict the distribution of wall temperature, raised by large heat flux coming from the combustion gases. Considering the tremendous computational cost of multidimensional CFD simulations of fluid flow and heat transfer both on hot-gas-side and coolant side, thermal analysis of rocket engines is often relied on separated calculations for convection heat transfer from hot-gas to the wall and convection from the wall to the cold coolant. Such one-dimensional-model-based methods often assume an ideal mixing of the thermal energy into the coolant channel cross-section. This implies that when a significant radial thermal stratification takes place, like in the case of high aspect ratio cooling channels, a significant error arises. To overcome these limitations and yet still maintain the computational simplicity of the one-dimensional approximation, an integrated quasi-2D coupled heat transfer model (Pizzarelli) has been introduced. This paper will try to understand its accuracy of prediction on coolant temperature distribution at different cross section along the cooling channel. The effect of turbulent Reynolds based on turbulent heat conduction coefficient will be discussed. Fully three dimensional CFD method will be also introduced as a validation way. Different channels with all kinds of across sections will be chosen as the research objects.