JASP Journal of Aerospace Power 航空动力学报

Hide Expanded Menus

Numerical simulation of axial liquid film cooling in rocket combustor

投稿时间： 2012-02-16

DOI：

中文关键词： liquid rocket engine　　liquid film cooling　　heat flux　　numerical simulation　　turbulent flow

英文关键词：liquid rocket engine　　liquid film cooling　　heat flux　　numerical simulation　　turbulent flow

基金项目：

作者	单位	E-mail
YANG Wei	School of Astronautics, Beijing University of Aeronautics and Astronautics, Beijing 100191, China	maggieyangxie@sa.buaa.edu.cn
SUN Bing	School of Astronautics, Beijing University of Aeronautics and Astronautics, Beijing 100191, China	
ZHENG Li-ming	School of Astronautics, Beijing University of Aeronautics and Astronautics, Beijing 100191, China	

中文摘要：

　　Numerical simulation has been done for liquid film cooling in liquid rocket combustor.Multiple species of axial Navier-Stokes equations have been solved for liquid-film/hot-gas flow field,and k-ε equations have been used for compressible turbulent flow.The results of the model agree well with the results of software FLUENT.The results show that:(1) Liquid film can decrease the wall heat flux and temperature effectively,and the cold border area formed by the film covers the whole combustor and nozzle wall.(2) The turbulent viscosity is higher than the physical viscosity, and its biggest value is in the border area of the convergent area in nozzle.The effect of turbulent flow on the whole simulation field can not be ignored.(3) The mass fraction of kerosene at the film inlet is 1,but it decreases along the nozzle wall and achieves its lowest value at the outlet.However,the mass fraction of kerosene near the wall is the biggest at any axial location.

英文摘要：

　　Numerical simulation has been done for liquid film cooling in liquid rocket combustor.Multiple species of axial Navier-Stokes equations have been solved for liquid-film/hot-gas flow field,and k-ε equations have been used for compressible turbulent flow.The results of the model agree well with the results of software FLUENT.The results show that:(1) Liquid film can decrease the wall heat flux and temperature effectively,and the cold border area formed by the film covers the whole combustor and nozzle wall.(2) The turbulent viscosity is higher than the physical viscosity, and its biggest value is in the border area of the convergent area in nozzle.The effect of turbulent flow on the whole simulation field can not be ignored.(3) The mass fraction of kerosene at the film inlet is 1,but it decreases along the nozzle wall and achieves its lowest value at the outlet.However,the mass fraction of kerosene near the wall is the biggest at any axial location.