IAF SPACE TRANSPORTATION SOLUTIONS AND INNOVATIONS SYMPOSIUM (D2)
Launch Services, Missions, Operations, and Facilities (2)

Author: Mr. Chuangchuang Yu
Beijing Institute of Aerospace Testing Technology, China

Mr. Chuangchuang Yu
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China
Mr. Jiaxian Zhang
Beijing Institute of Aerospace Testing Technology, China
Mr. Jiaxian Zhang
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China
Dr. Jia Zhang
Beijing Institute of Aerospace Testing Technology, China
Dr. Jia Zhang
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China
Mrs. Ruimin Liu
Beijing Institute of Aerospace Testing Technology, China
Mrs. Ruimin Liu
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China
Mr. Sheng Qu
Beijing Institute of Aerospace Testing Technology, China
Mr. Sheng Qu
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China
Dr. sifeng yang
Beijing Institute of Aerospace Testing Technology, China
Dr. sifeng yang
Beijing Engineering Research Center of Aerospace Testing Technology and Equipment, China

NUMERICAL INVESTIGATIONS OF THERMAL CIRCUMSTANCE OF LAUNCHING SITE'S WATER-COOLING DEFLECTOR

Abstract

Computational fluid dynamic (CFD) modeling is a useful tool for studying the thermal performance of a large launching vehicle's launching site. However, traditional models are mostly single phase model, neglecting the cooling effect of spraying water of the deflector. Actually, there is strong heat and mass transfer between cooling water and exhausted gas with high temperature, which makes the flow field at the bottom of the deflector extremely complex. In this paper, numerical investigations were conducted by using a multiphase CFD model, aiming to improve the thermal performance of the launching site. The multiphase model directly takes into consideration the strong heat and mass transfer. A geometry model is built, including a water-cooling deflector and a rocket stage with two liquid hydrogen/liquid oxygen rocket engines. A single-step hydrogen-oxygen (H-O) reaction model and standard k-ε turbulence model were employed to obtain the flow field, discrete phase model (DPM) was applied to simulate the effect of cooling water. Subsequently, the design parameters of launching site were studied and compared with the verified model. Results showed that, deeper depth of the deflector, more mass flow rate of the

cooling water and larger distribution proportion of cooling water in the central area could effectly help in improving the thermal circumstance at the bottom of the deflector. P1 modified model was used to simulate the radiation heat transfer between high temperature gas and the circumstance. The predicted gas temperature near the engine increase significantly after loading the radiation model, regardless of whether the effect of cooling water is considered or not. Follow-up research will be carried out on the influence of the water-spray system parameters (including the pressure of spray water, spray direction angle, and distribution of sprinkler area) on the thermal circumstance.