MICROGRAVITY SCIENCES AND PROCESSES SYMPOSIUM (A2)
Poster Session (P)

Author: Dr. Shengchao Hu
Beijing Institute of Aerospace Systems Engineering, China Aerospace Science and Technology Corporation (CASC), China, 52221635@qq.com

Prof. Chengmin Ye
Beijing Institute of Aerospace Systems Engineering, China Aerospace Science and Technology Corporation (CASC), China, shissh@163.com
Mrs. Xuemei Wang
Beijing Institute of Aerospace Systems Engineering, China Aerospace Science and Technology Corporation (CASC), China, yemo1980@hotmail.com
Mr. Liming Xiao
Beijing Institute of Aerospace Systems Engineering, China Aerospace Science and Technology Corporation (CASC), China, xlm_2000@126.com

RESEARCH ON CHARACTERISTIC OF TANK PRESSURIZATION WITH DIFFUSER FOR LIQUID PROPULSION SYSTEM

Abstract

Diffuser is one of the most important parts for the liquid propulsion system. It can decrease or avoid the injected gas for the tank pressurization into liquid propellant, especially under the low-gravity condition. In order to grasp the details of liquid motion driven by the injected gas, a kind of Computational Fluid Dynamics (CFD) model, based on the volume of fluid (VOF) approach, was used to analyze the tank pressurization process of liquid propulsion system. Variational gravity magnitude was considered by User Define Function (UDF) in this model. Compared with the experimental data, the simulation result showed reasonably good agreement with visualized images which obtained with a CCD camera and outputs from the sensors in the H2 tank on the occasion of the first flight of H-IIA launch vehicle. It proves that the VOF-based computational model is able to capture the free surface and key feature of the interaction of injected gas and liquid propellant in the tank. At last, principle experimental study was carried out on the ground. Some different physical boundary condition such as mass rate of pressurization gas, the distance between diffuser out and liquid surface were applied in those experiments for investigating the diffuser performance. Due to the limitation of measure device, the CFD model introduced above was used to simulate dynamics of gas-liquid two-phase flow. In the meantime, the CFD simulation provide information about finer details of interaction of injected gas and liquid. The result can be used for diffuser design and optimization.