

Improvement of Performance of Anti-Splash Device for Cargo Oil Tank Vent Pipe Using CFD Simulation

Authors : Sung-Min Kim, Joon-Hong Park, Hyuk Choi

Abstract : This study is focused on the comparative analysis and improvement to grasp the flow characteristic of the anti-splash device located under the P/V valve and new concept design models using the CFD. The P/V valve located upper deck to solve the pressure rising and vacuum condition of inner tank of the liquid cargo ships occurred oil outflow accident by transverse and longitudinal sloshing force. Anti-splash device is fitted to improve and prevent this problem in the shipbuilding industry, but the oil outflow accidents are still reported by ship owners. Thus, 4 types of new design model are presented by this study, and then comparative analysis is conducted with new models and existing model. Mostly the key criterion of this problem is flux in the outlet of the anti-splash device. Therefore, the flow and velocity are grasped by transient analysis, and then it decided optimum model and design parameters to develop model. Later, it is needed to develop an anti-splash device by flow test to get certification and verification using experiment equipments.

Keywords : anti-splash device, P/V valve, sloshing, CFD

Conference Title : ICMEM 2014 : International Conference on Mechanical Engineering and Mechatronics

Conference Location : Barcelona, Spain

Conference Dates : October 27-28, 2014