World Academy of Science, Engineering and Technology
International Journal of Computer and Systems Engineering
Vol:14, No:12, 2020

Computational Fluid Dynamics Simulation Study of Flow near Moving Wall of Various Surface Types Using Moving Mesh Method

Authors : Khizir Mohd Ismail, Yu Jun Lim, Tshun Howe Yong

Abstract : The study of flow behavior in an enclosed volume using Computational Fluid Dynamics (CFD) has been around for decades. However, due to the knowledge limitation of adaptive grid methods, the flow in an enclosed volume near the moving wall using CFD is less explored. A CFD simulation of flow in an enclosed volume near a moving wall was demonstrated and studied by introducing a moving mesh method and was modeled with Unsteady Reynolds-Averaged Navier-Stokes (URANS) approach. A static enclosed volume with controlled opening size in the bottom was positioned against a moving, translational wall with sliding mesh features. Controlled variables such as smoothed, crevices and corrugated wall characteristics, the distance between the enclosed volume to the wall and the moving wall speed against the enclosed chamber were varied to understand how the flow behaves and reacts in between these two geometries. These model simulations were validated against experimental results and provided result confidence when the simulation had shown good agreement with the experimental data. This study had provided better insight into the flow behaving in an enclosed volume when various wall types in motion were introduced within the various distance between each other and create a potential opportunity of application which involves adaptive grid methods in CFD.

Keywords : moving wall, adaptive grid methods, CFD, moving mesh method
Conference Title : ICCFD 2020 : International Conference on Computational Fluid Dynamics
Conference Location : London, United Kingdom
Conference Dates : December 10-11, 2020