World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:16, No:01, 2022

Numerical and Experimental Investigation of Airflow Inside Car Cabin

Authors : Mokhtar Djeddou, Amine Mehel, Georges Fokoua, Anne Tanière, Patrick Chevrier

Abstract : Commuters' exposure to air pollution, particularly to particle matter, inside vehicles is a significant health issue. Assessing particles concentrations and characterizing their distribution is an important first step to understand and propose solutions to improve car cabin air quality. It is known that particles dynamics is intimately driven by particles-turbulence interactions. In order to analyze and model pollutants distribution inside the car the cabin, it is crucialto examine first the single-phase flow topology and turbulence characteristics. Within this context, Computational Fluid Dynamics (CFD) simulations were conducted to model airflow inside a full-scale car cabin using Reynolds Averaged Navier-Stokes (RANS)approach combined with the first order Realizable k- εmodel to close the RANS equations. To validate the numerical model, a campaign of velocity field measurements at different locations in the front and back of the car cabin has been carried out using hot-wire anemometry technique. Comparison between numerical and experimental results shows a good agreement of velocity profiles. Additionally, visualization of streamlines shows the formation of jet flow developing out of the dashboard air vents and the formation of large vortex structures, particularly in the back seats compartment. These vortex structures could play a key role in the accumulation and clustering of particles in a turbulent flow

Keywords : car cabin, CFD, hot wire anemometry, vortical flow
Conference Title : ICEFSCFD 2022 : International Conference on Experimental Fluid Science and Computational Fluid Dynamics
Conference Location : Rome, Italy
Conference Dates : January 14-15, 2022