INTERNATIONAL JOURNAL OF HYDROGEN ENERGY 40 (2015) 8230–8238

Available online at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/he

Emergency blower ventilation to disperse hydrogen leaking from a hydrogen-fueled vehicle

Hong Xie, Xuefang Li, David M. Christopher[*]

Key Laboratory for Thermal Science and Power Engineering of Ministry of Education,
Thermal Engineering Department, Tsinghua University, Beijing 100084 China

ARTICLE INFO

Article history:
Received 23 December 2014
Accepted 29 March 2015
Available online 9 May 2015

Keywords:
Hydrogen safety
Fuel cell vehicles
Emergency response procedures
Vehicle accidents

ABSTRACT

The effect of blower ventilation to disperse hydrogen leaking from a hydrogen fueled vehicle in an emergency was studied numerically with comparison with experimental data to investigate the practicability of rapid hydrogen dispersion to mitigate dangerously high hydrogen concentrations. The hydrogen gas was assumed to leak at 2000 NL/min at the center of the vehicle chassis while the blower was placed on the left side of the vehicle. Steady-state calculations showed that the blower significantly reduced the hydrogen concentrations around the vehicle with the hydrogen concentrations in most of the area below the lower flammability limit. Transient calculations showed that the hydrogen concentration just downstream of the vehicle had a high concentration peak shortly after the ventilation began with the hydrogen then being rapidly dispersed with additional ventilation. More cases with various air capacities and blower shapes were analyzed to find a better blower geometry with a higher blower efficiency. The results show that both a larger air capacity and a smaller blower with a higher air velocity improved the hydrogen dispersion, with the smaller blower shape being the more efficient method because the greater air entrainment into the jet for higher jet velocities improved the hydrogen dispersion, especially when the leak position was known.

Introduction

Hydrogen is a clean, transportable, efficient fuel for vehicles, so hydrogen is one of the most promising energy carriers for future transportation systems [1]. Many vehicle companies are promoting their new hydrogen power vehicles and the number of hydrogen-fueled vehicles is increasing rapidly.

The companies want to design hydrogen fuel cell vehicles with the same or even better safety standards than the current well-developed hydrocarbon fueled vehicles. There have been many theoretical and experimental studies of hydrogen safety. Experiments can directly assess the safety of hydrogen fuel systems, but such tests are dangerous and costly. Simulations using computational fluid dynamics (CFD) models have many advantages since these studies cost much less and provide more detailed information about the leakage and dispersion such as the extent of the flammable region and how the hydrogen concentrations change at any location, which is very difficult for experimental studies because of the finite number of sensors. However, CFD results need to be validated by comparing with experimental data.

---

[*] *Corresponding author*. Tel.: +86 (10) 6277 2986.
E-mail address: dmc@tsinghua.edu.cn (D.M. Christopher).

http://dx.doi.org/10.1016/j.ijhydene.2015.03.146

Choi et al. [2] simulated hydrogen dispersion from a fuel cell vehicle in an underground parking garage. Papanikolaou et al. [3] compared experimental and numerical results for hydrogen leakage inside a naturally ventilated garage using various codes with different models. Merilo et al. [4,5] experimentally studied hydrogen leakage in a garage and also in passenger vehicle compartments. Most of these studies have focused on vehicles in parking garages, which is a significant concern because the hydrogen disperses more slowly and will concentrate in a small space. Another concern is how first responders should approach a vehicle accident involving hydrogen leakage. First responders need to know where the hydrogen concentrations exceed the lower flammability limit and need to be able to quickly and safely disperse the hydrogen. Salva et al. [6] used a CFD model to simulate hydrogen leakage inside a vehicle. Maeda el al [7,8] experimentally investigated hydrogen release and dispersion from underneath a vehicle. Tamura et al. [9] experimentally investigated the effectiveness of wind provided by a blower for reducing the hydrogen concentration with the results showing that a blower can effectively disperse the hydrogen to reduce or eliminate hazards for a fuel cell vehicle leaking hydrogen.

The present article also focuses on hydrogen leaking from the underside of a single vehicle. The analysis uses CFD models with the results compared to recent experiments by Tamura et al. [9]. The result show that a single hydrogen leak with a volumetric flow rate of 2000 NL/min can be dissipated by winds that are at least 10 m/s; thus, a blower can effectively disperse the leaking gas. This CFD results compare well with the experimental data and provide more details about the hydrogen concentration distribution and dispersion. The transient calculations show that the hydrogen concentration rises to a high peak near the vehicle after the fan is started; thus the first responders need to wait for a short period of time before rescuing the people in the vehicle. Various velocities and blower shapes were evaluated to identify the key factors that affect the blower performance. The results show that the blower shapes significantly affects the dispersion efficiency. Although both the air capacity and the blower shape influenced the performance, changes in the blower shape are a better way to disperse the hydrogen rather than increasing the blower air flow capacity. This method is especially useful if the leak location is already known.

Problem description

Geometry description

The vehicle modeled in the simulation was based on the experiments by Tamura et al. [9]. The size and shape of the vehicle are shown in Fig. 1. The hydrogen was released through a nozzle with an internal diameter of 4 mm, which is much smaller than the vehicle. The model included many simplifications to reduce the complexity of the geometry for the simulation. Many elements of the vehicle geometry were assumed to have little effect on the hydrogen dispersion so they were eliminated. Other shapes were simplified to allow use of a more structured grid, such as the tires were modeled as hexagons and the circular nozzle and the fan were modeled as squares.

The unsymmetrical shape of the vehicle was simplified to a hexagon with the front part of the car assumed to have little effect on the flow from the side. The geometry used in the simulation is shown in Fig. 2. The vehicle was 4.670 m × 1.695 m × 1.685 m while the total computational region was 8 m × 6 m × 3 m, large enough that the boundaries have little effect on the flow around the vehicle, especially underneath. The computational region was set to 8 m long because the downstream region behind the vehicle is the most dangerous part, with the experiments having several monitors there to detect the hydrogen concentration. The computations also assumed that the geometry was symmetrical, with only half of the chamber and vehicle modeled in the calculations. A more complicated model was also used with more details about the vehicle shape, but the calculations showed that the shape of the upper part of the vehicle had little effect on the hydrogen movement. Thus, the simplified geometry in Fig. 2 was used to obtain better grid quality.

The hydrogen leaking point was right below the center of the vehicle with a 2000 NL/min flow rate. The hydrogen flowed vertically out of the tube. This hydrogen leak rate is the maximum flow rate for an ordinary passenger car which can provide approximately 200 kW from the fuel cell [6]. The fan air capacity was 740 m^3/min with the fan modeled as a square with sides 0.710 m long. The fan was placed on the left side with the flow direction adjusted to ensure that the center of the flow stream hit the center of the gap between the bottom of the vehicle and the ground. The mesh contained 1,266,658 cells after improving the grid in the vicinity of the hydrogen leak point.

Boundary conditions and monitor configuration

FLUENT 14.5 was used for the simulations. The fan was set as a velocity inlet with a velocity of 24.5 m/s and the direction pointed down 20.10° towards the center of the space below the vehicle. Due to the high volume flow rate of the hydrogen (2000 NL/min) from a very small nozzle, the hydrogen escape velocity was supersonic. Thus, the Fluent mass-flow-inlet type would normally be a better choice for the hydrogen inlet vent boundary condition. However, some tests showed no significant differences between the results for the mass-flow-inlet and the velocity-inlet, so the velocity-inlet was chosen for hydrogen leaking point to correspond to the fan inlet boundary condition. The calculations then solved the three-dimensional Navier–Stokes equations with the energy and species equations using the pressure-based solver and the standard k-ε turbulence model with the standard wall functions. The air–hydrogen mixture was modeled as an incompressible ideal gas with the gravitational constant set to 9.8 m/s^2. The flow and turbulence equations were all discretized using the second-order upwind method in Fluent. The diffusion was modeled using the species transport with the diffusion energy source. All the vehicle surfaces were set as wall boundary conditions. The downstream outlet boundary condition was set as outflow.

With the symmetry in this model, the highest hydrogen concentration should be on the symmetry plane shown in