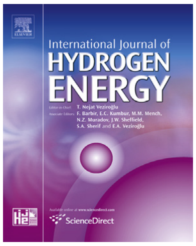
INTERNATIONAL JOURNAL OF HYDROGEN ENERGY 40 (2015) 11762–11770

Available online at www.sciencedirect.com

ScienceDirect

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



Large eddy simulation of hydrogen dispersion from leakage in a nuclear containment model

Jungwoo Kim [a,*], *Eunbum Jung* [b], *Seongwon Kang* [b,*]

[a] *Department of Mechanical System Design Engineering, Seoul National University of Science and Technology, Seoul 139-743, Republic of Korea*

[b] *Department of Mechanical Engineering, Sogang University, Seoul 121-742, Republic of Korea*

ARTICLE INFO

Article history:
Received 16 February 2015
Received in revised form
16 April 2015
Accepted 27 April 2015
Available online 6 June 2015

Keywords:
PANDA
Hydrogen dispersion
Hydrogen leakage
Nuclear safety
Large eddy simulation

ABSTRACT

Hydrogen leaking inside a nuclear power plant poses a critical nuclear safety issue, and accurate prediction of the dispersion process of hydrogen has become an important topic of research. In the present study, temporal evolution of hydrogen dispersion in a test facility (called PANDA) for nuclear safety is analyzed using high-fidelity simulation techniques such as large eddy simulation (LES), high-order discretization methods, immersed boundary (IB) method, etc. An important topic in this study is how a turbulent buoyant jet of a released gas-air mixture interacts with a stratified layer formed near the ceiling of a containment. In this study, the interaction of jet penetration with the stratified layer is characterized as the "slow erosion process". The height of the jet penetration is limited by light gas at the top due to the negative buoyant effect. From a viewpoint of the safety of the containment, the present study shows that by a continuous release of hydrogen, the flammable region expands significantly over time. However, the change of the detonable region over time is much smaller compared to that of the flammable region.

Copyright © 2015, Hydrogen Energy Publications, LLC. Published by Elsevier Ltd. All rights reserved.

Introduction

During a severe accident in a nuclear power plant, hydrogen can be generated and released into the containment due to oxidation of zirconium and radiolysis of water during the progression of an accident leading to core degradation [1]. Because hydrogen has a relatively large flammable region and a fast flame speed, local hydrogen accumulation sufficient for hydrogen explosion may result in increasing the containment pressure if there is an ignition of the released gas, and eventually affecting its integrity.

The risk due to hydrogen was first realized when a large quantity of hydrogen was released into the containment during the Three Mile Island accident in 1979. Since then, the thermal–hydraulic processes such as the mixing and distribution of hydrogen playing an important role on the safety of a nuclear power plant have been investigated via international co-works in nuclear research societies. However, a hydrogen explosion in the Fukushima Daiichi accident in March 2011 showed that the control and mitigation of the hydrogen risk remains as one of the key safety issues in nuclear power plants.

In order to control and mitigate the risk associated with hydrogen, it is important to understand the hydrogen dispersion process inside a containment during a severe accident.

Among many previous studies related to this topic, most of them were experimental studies, and only a few were

---

* *Corresponding authors.* Tel./fax: +82 2 705 7972.
E-mail addresses: kimjw@seoultech.ac.kr (J. Kim), skang@sogang.ac.kr (S. Kang).

http://dx.doi.org/10.1016/j.ijhydene.2015.04.156

0360-3199/Copyright © 2015, Hydrogen Energy Publications, LLC. Published by Elsevier Ltd. All rights reserved.

performed by numerical simulations [2,3]. To fully understand temporal evolution of the hydrogen dispersion in the containment, three-dimensional transient data inside a containment is necessary, which is not accomplished easily in experiments. On the other hand, using CFD, the hydrogen dispersion over time and space inside a containment can be analyzed relatively easily. In that respect, there has been a consistent need for CFD. However, most of CFD studies until now for thermal-hydraulic problems in containments have been based on the Reynolds-averaged Navier-Stokes (RANS) models (e.g. Refs. [4,5]) for turbulence. Due to relatively large uncertainty of the RANS models, important physical phenomena such as mixing contributed by small-scale fluctuations can be predicted with only limited accuracy. Hence, accurate prediction of turbulent mixing still remains as one of the key elements for a successful simulation of gas mixing and transport inside a containment [6]. In this regard, a large eddy simulation (LES) technique is advantageous over the conventional RANS approaches because it is able to predict turbulent fluctuation relatively better and possibly can predict the complex turbulent phenomena related to the gas dispersion in a containment more accurately.

In the present study, we investigate gas dispersion in a test facility referred to as PANDA using high-fidelity simulation techniques such as the large eddy simulation, high-order discretization methods, immersed boundary (IB) method, etc. There are two objectives: (1) to make a quantitative analysis based on detailed simulations of gas dispersion and (2) to understand the temporal evolution of gas dispersion related to the safety issue in a containment. A particular attention is paid to the understanding the erosion of a stratified layer by a buoyant jet in a large vessel. In Sec. 2.1, the test facility and configuration are described, followed by an introduction of the numerical methods in Sec. 2.2. In Sec. 3, the dynamic behavior of gas dispersion and the related turbulence statistics are presented. Finally, conclusions are given in Sec. 4.

Computational details

Description of the PANDA test facility

The PANDA facility is a large-scale thermal-hydraulics test rig located at the Paul Scherrer Institute (PSI), Switzerland. With the PANDA facility, many experiments have been done so far to understand the large-scale gas mixing and stratification in the containment as well as to improve the accident management and mitigating measures [2,7,8].

Recently, PSI started experimental studies together with OECD in a given condition with an aim to provide accurate data for validating CFD simulation results. This international activity is named as the third International Benchmark Exercise (IBE-3) [9]. To obtain appropriate data for CFD validation purposes, the PANDA experiment considered in the IBE-3 is designed assuming the absence of the effects of complex geometries existing inside a real containment, the condensation of the steam released together with the hydrogen and so on. Therefore, suitable data for CFD validation is readily available by considering the PANDA configuration. The images of PANDA considered in this study are shown in Fig. 1. As shown in Fig. 1(b), the total height of PANDA is 8 m and total diameter is 4 m.

The gas dispersion problem in a containment depends primarily on the transient scenario and break location [3]. A typical accident scenario consists of two phases: In the first phase, the released gas produces a stratified layer at the upper part of the containment, and in the second phase, the stratified layer formed in the first phase is eroded by the gas released subsequently. Therefore, the gas transport, stratification build-up, stratification disruption and erosion are important thermal-hydraulics phenomena during an accident. Among them, in the IBE-3, the stratification disruption and erosion are the main topics of research. For these phenomena, a stratified layer at a uniform temperature of around 20 °C and at about 1.0 bar is set up as the initial condition following suggestions by Ref. [9] (see Fig. 1(b) and (c)). As shown in Fig. 1(c), prior to the test stratified air/helium layer is formed in the test vessel. On the other hand, the break location is located off the center of the containment, as shown in Fig. 1(b).

In the PANDA experiment, helium is used as a working fluid instead of hydrogen, because an experiment of hydrogen leakage is dangerous due to a possibility of hydrogen deflagration. Also, instead of steam released with hydrogen from a nuclear reactor, air is released with helium from an injection tube shown in Fig. 1(b). In order to validate the present numerical methods, a simulation with helium is performed in the present study. Then, the safety issue related to hydrogen dispersion is investigated using a simulation with hydrogen. Also, the similarity and difference of the results between the cases of helium and hydrogen are assessed. By comparing three non-dimensional numbers (Reynolds number, Froude number and Schmidt number) existing in the present problem, it is found that the cases with helium and hydrogen show the maximum 7% difference.

Numerical methods

The flow field inside the PANDA facility is characterized as a combination of several complex flow phenomena such as a turbulent jet, fountain effect through a helium (or hydrogen) layer, a large-scale circulatory flow, etc. In order to simulate turbulent flows with the buoyancy force, we considered the mass and momentum conservation for unsteady, viscous flows:

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho u)=0, \tag{1}$$

$$\frac{\partial\rho u}{\partial t}+\nabla\cdot(\rho uu)=-\nabla p+\nabla\cdot\mu S+\rho g, \tag{2}$$

where ρ is the density, t is the time, u is the velocity vector, p is the pressure, μ is the dynamic viscosity, and S is the shear-stress tensor defined as

$$S=\left[\nabla u+(\nabla u)^{T}\right]-\frac{2}{3}I\nabla\cdot u, \tag{3}$$

where I is the Kronecker delta. Mass conservation of the leakage gas (i.e. hydrogen or helium) is satisfied by a scalar transport equation:

Download English Version:

https://daneshyari.com/en/article/7714283

Download Persian Version:

https://daneshyari.com/article/7714283

Daneshyari.com