Annals of Nuclear Energy 56 (2013) 227–242

Contents lists available at SciVerse ScienceDirect

Annals of Nuclear Energy

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

# How critical is turbulence modeling in gas distribution simulations of large-scale complex nuclear reactor containment?

Jianjun Xiao [a],*, John R. Travis [b]

[a] Institute of Nuclear and Energy Technologies, Karlsruhe Institute of Technology, P.O. Box 3640, 76021 Karlsruhe, Germany
[b] Engineering & Scientific Software, Inc., 2128 S. Ensenada Circle, Rio Rancho, NM 87124, USA

ARTICLE INFO

*Article history:*
Received 25 October 2012
Received in revised form 2 January 2013
Accepted 4 January 2013
Available online 21 February 2013

*Keywords:*
Computational fluid dynamics (CFD)
Turbulence modeling
Hydrogen safety
Nuclear power plant (NPP)
Containment

ABSTRACT

Uncontrolled hydrogen combustion can occur in the nuclear reactor containment during a severe accident. The energetic hydrogen combustion may threaten the integrity of the containment and lead to radioactive material being released into the environment.

In order to mitigate the risk of hydrogen combustion, the first step is to understand how the burnable hydrogen cloud develops in the containment. Turbulence modeling is one of the key elements in simulations of the physical phenomena that occur in containment. However, when a turbulence model is used, the computational time is increased in CFD simulations of large-scale reactor containment primarily due to the additional turbulent transport equations and the small time step controlled by the explicitly-treated turbulent diffusion in GASFLOW code. The purpose of this paper is to investigate how critical turbulence modeling is in the simulation of hydrogen/steam distribution in a large-scale, complex reactor containment. In other words, is it acceptable to neglect the turbulent viscosity in the momentum diffusion term in such a large-scale engineering simulation to save computational time?

The effect of turbulence models on the gas distribution in the MISTRA 2009 campaign was investigated using the CFD code, GASFLOW. The calculation results improved locally in the region near the jet source when turbulence models were used. For most of the space in the MISTRA facility, which is located away from the source, it seems that the turbulent diffusion was over-predicted by the turbulence models, and better agreements with the experimental data were obtained by simply using molecular viscosity. These results indicate that with turbulence models, more computational time is required, and the improved calculation results are local and limited. It appears that the predictions are reasonably good when only molecular viscosity is considered in the diffusion terms. Due to the limited computational resources, we must investigate the trade-offs between computational effort and accuracy, particularly in large-scale engineering applications.

## 1. Introduction

A hydrogen bubble, which could explode, was formed in the containment during the accident at the Three Mile Island (TMI-2) nuclear power plant in 1979 (Walker, 2006). The hydrogen explosions at the Fukushima Daiichi nuclear power plants (IAEA, 2011; Suzuki, 2011) in 2011 again reminded nuclear engineers, safety authorities and the public of the significance of investigating hydrogen explosion risk and mitigation measures during a severe accident in nuclear power plant.

The interaction of molten corium and water during a severe accident in a nuclear reactor produces hydrogen (Breitung et al., 1999). The total amount of readily oxidizable reactor core material (i.e., the zirconium in the fuel rods) is approximately 30 tons in the European Pressurized Reactor (EPR), which is theoretically able to generate as much as 1320 kg of hydrogen (Dimmelmeier et al., 2012). Several risk studies have shown that during core degradation, up to 2 kg/s of hydrogen can be produced, yielding more than 1000 kg of hydrogen in the containment during the first 7 h of a severe accident (Sehgal et al., 2012). Hydrogen and steam flow into the containment through a break, which could be located in either the hot or cold leg of the reactor coolant loop, along the surge line, in the pressurizer, or in the primary circuit depressurization system. Burnable hydrogen clouds may form in the upper part of the containment, and the integrity of the containment may be threatened if energetic hydrogen explosions occur (Breitung et al., 2000). Thus, the structural responses due to the pressure and thermal loads must be evaluated (Breitung and Royl, 2000).

Considerations will be given to the control of fission products, hydrogen and other substances that may be generated or released in the event of a severe accident according to the NPP safety

---

* Corresponding author. Tel.: +49 721 6082 2437; fax: +49 721 6082 4837.
*E-mail address:* jianjun.xiao@kit.edu (J. Xiao).

0306-4549/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.anucene.2013.01.016

requirements (IAEA, 2000; NRC, 1986; EUR, 2001; NNSA, 2002). The mitigation measures of hydrogen combustion risk are required to be addressed during the licensing process of NPPs.

In the past decades, several CFD codes, such as GASFLOW (Travis et al., 2011a,b), TONUS-3D (Kudriakov et al., 2006; Caroli et al., 2006), and NEPTUNE-CFD (Guelfi et al., 2007) have been developed and applied to simulate the complicated physical phenomena in reactor containments.

GASFLOW is a three-dimensional, finite volume, arbitrary Lagrangian–Eulerian (ALE, Hirt, 1974) hydrodynamic code that solves the time-dependent, compressible Navier–Stokes equations for multiple gas species. The code can model condensation, heat transfer to walls and internal structures, multiple compartments, chemical kinetics, and fluid turbulence. The GASFLOW code is currently being developed at the Karlsruhe Institute of Technology (KIT), Germany. GASFLOW has been widely used to analyze the hydrogen distribution and risk mitigation inside nuclear containments, such as EPR in Fig. 1 (Dimmelmeier et al., 2012; Movahed et al., 2003), the International Thermonuclear Experimental Reactor (ITER), as shown in Fig. 2 (Xiao et al., 2010), the German Konvoi-Type PWR, as shown in Fig. 3 (Royl et al., 2000), the VVER (Kostka et al., 2002), and the APR1400 (Kim et al., 2004, 2006).

In order to effectively mitigate the hydrogen combustion risks, firstly we need to understand how gas components are transported and mixed in the containment and how and where the burnable hydrogen cloud is formed. It is well known that turbulence modeling is one of the key elements for a successful simulation of gas mixing and transport. However, when a turbulence model is used, more computational effort is needed to solve the turbulence transport equations. Subsequently, to ensure numerical stability, the time step must be limited by turbulent diffusion if it is treated explicitly in the CFD code, as will be discussed in Section 3.4. It should be noted that the small time step is controlled by a local, strong turbulent viscosity, i.e., the region near the hydrogen/steam source, which means that most of the CPU time will be used to solve the local physical phenomena that may have a limited influence on the global parameters. For a typical NPP containment, the number of cells is approximately 200,000 for a GASFLOW geometric model, and the total time of an accident sequence can be greater than 6000 s (Royl et al., 2000). Therefore, in several cases, it becomes impractical to create a simulation for large-scale containment with CFD code due to the considerably increased computational time, i.e., several months.

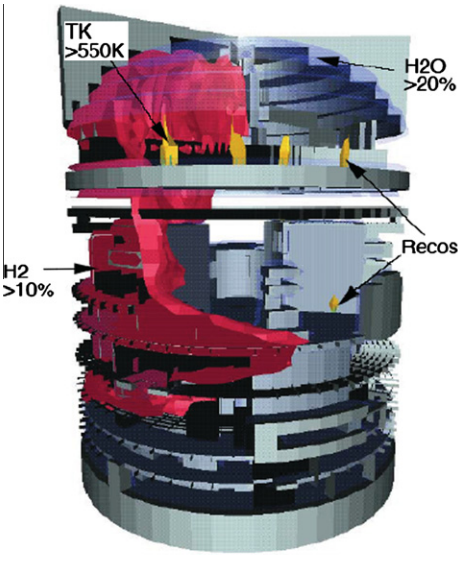

**Fig. 1.** A GASFLOW geometric model of a PWR.

After the hydrogen, steam or other gas components are injected into the containment from the coolant pipe, the gas mixtures impinge on the obstacles, such as the walls, coolant pipes and steam generators, due to the complexity of the structures and installations around the break. A common physical phenomenon is the impinging jet, as shown in Fig. 4. As a consequence of the impingement on the obstacle, the velocity of the flow becomes smaller, and thus, the flow is mainly dominated by buoyant forces.

In the geometric model of large-scale NPP containments, the cell size must be relatively coarse, i.e., 20–30 cm. The flow includes a wide spectrum of regimes, such as under-expanded jets, impinging jets and plumes, due to the complex structures in the containment. Can any turbulence model be accurate in such various flow regimes with such a coarse mesh? Is it worthwhile to simulate the local turbulence with more computational time? How can we obtain a reasonably good prediction of the global hydrogen distribution in the containment with relatively coarse mesh within acceptable computational time?

Therefore, the question arises: how critical is turbulence modeling in simulations of complex flow regimes in large-scale engineering facilities considering the computational cost and the improvement in the accuracy? In this paper, the effect of turbulence modeling on the gas distribution in the MISTRA facility (Studer et al., 2003) was investigated. A zero-equation algebraic turbulence model and a standard $\kappa$–$\varepsilon$ turbulence model in GASFLOW were used in the simulation, and the results were compared with experimental data.

## 2. Mistra 2009 campaign

The MISTRA facility is part of the CEA program related to severe accidents that occur in nuclear reactors and is focused on containment thermal–hydraulics and hydrogen safety (Studer et al., 2003). The MISTRA facility is a stainless steel vessel with a volume of 99.5 $m^3$ (Fig. 5). The internal diameter (4.25 m) and the height (7.38 m) were chosen to scale to a typical French PWR containment with a linear length scale ratio of 0.1.

The internal volume of the MISTRA vessel was divided into two distinct volumes (Fig. 6). The compartment is a vertical cylinder, which is closed at the bottom. The compartment is fitted with a ring plate. The internal cylinder diameter is 1.906 m with a height of 4.219 m. The bottom of the compartment is at an elevation of 1.245 m from the vessel bottom, and the top of the compartment is at an elevation of 5.464 m. The compartment walls are approximately 3 mm thick. The ring plate is a horizontally-placed steel ring plate at an elevation of 3.658 m with an outer radius of 1.728 m.

To perform the test series, lateral injection lines were added to the facility. Penetrations are located in the laser windows. For the test series, four 90°, equally spaced window angles placed at mid-elevation (4.341 m) are used. The inlet diameter of each injection pipe is 22.6 mm. The MISTRA 2009 campaign (Dabbene et al., 2010) is a 10-min lateral air and nitrogen injection within a stable helium stratification. To avoid the safety problems that hydrogen may cause, helium is used in the experiment. Helium flows into the vessel at ambient temperature and pressure from 0 s to 600 s. During 600 s to 1200 s, helium injection is stopped to achieve stable helium stratification in the MISTRA annular area. Lastly, for another 10 min (1200–1800 s), air is injected at the opposite side of the helium injection. For inerting purposes, nitrogen can be injected as long as air is injected by the two remaining lateral injection lines located on both sides with the air ingress.

To follow the mixture stratification set-up and the evolution of the stratified layer in the vessel, 30 transducers were installed as demonstrated in Fig. 6. The installation radius, angles and elevations of the transducers are shown in Table 1.