*Transactions of the Korean Nuclear Society Autumn Meeting*
*Gyeongju, Korea, October 29-30, 2015*

# Turbulence-resolved Numerical Simulation for Hydrogen Safety in a NPP Containment

Jongtae Kim [a*], Seong-Wan Hong [a]
[a]*Severe Accident & PHWR Safety Research Division, Korea Atomic Energy Research Institute,*
*Daeduk-daero 989-111, Daejeon, Korea*
[*]*Corresponding author: ex-kjt@kaeri.re.kr*

## 1. Introduction

In order to understand accident progress in a nuclear power plant (NPP), experimental and numerical methods have been applied. The experimental approach is fundamentally applicable to a prototype. But because of economical or environmental limitations, small-scale experiments with simulant materials are commonly conducted especially in the research of severe accident of nuclear power plant (NPP). The current numerical approach for the severe accident analysis strongly relies on models and correlations which are developed by analytical and experimental works.

Traditionally, the numerical method is split into a lumped-parameter (LP) method and a multi-dimensional method. In the frame of multi-sale approach proposed by Yadigaroglu [1], LP method is thought to be useful for system scale (macro scale) analysis. On the contrary, computational fluid dynamics (CFD) is applicable to component (meso scale) or detail (micro scale) analysis.

Recently the boundary of the two LP and CFD methods becomes more and more obscure. LP codes such as Melcor [2] and Relap [3] have been continuously improved in their capabilities. Melcor has been added convection terms in two-phase momentum equations [4]. Relap-3D [5], a 3-dimensional version of the Relap code, has a multi-dimensional nodalization module similar to a CFD code. One of long-standing containment analysis codes GOTHIC [6] was developed as a LP code originating Cobra-TF [7]. Now, it can simulate a containment three-dimensionally using a Cartesian or cylindrical coordinate-based nodalization. So, it is believed that the LP codes can now resolve 3-D behavior of flows in a component such as a reactor or containment. GASFLOW [8] is a famous containment analysis code based on 3-D nodalization. It has some lumped models for PAR (passive auto-catalytic recombiner) and sink and source of mass and energy to reduce the number of nodes and efficiently simulate their thermal hydraulic phenomena.

In the author's point of view, it is not likely that LP-based code is only applicable to a system-scale or macro scale analysis. So, it is thought that a new measure to distinguish the numerical analysis methods is needed. Here, "turbulence-resolved method" is proposed as a new measure to distinguish the analytical methods. If GOTHIC is used for a containment analysis only with correlations or lumped model (e.g. not a localized model), it is a turbulence-unresolved approach. On the other hand, when a turbulent flow is simulated by GOTHIC with a turbulence model, it can be a turbulence-resolved approach. In the Melcor or Relap-3d codes, turbulence effect is lumped into a k-factor of pressure drop. So, these belong to a turbulence-unresolved approach even with a very fine nodalization.

It is believed that the two approaches must be applied carefully for an accident analysis in NPPs. If turbulent characteristics in a flow field are not well resolved by the turbulence-resolved approach, the flow field may be poorly distorted, and sometimes it is worse than the correlation-based turbulence-unresolved solution. On the contrary, in the case that turbulent characteristics are well lumped into correlations used in the turbulence-unresolved approach, the solution can be better than a poorly resolved turbulent solution.

In two-phase flows, it is still difficult to get accurate solutions by the turbulence-resolved approach. That is mainly because of limited applicability of currently available models required for the two-phase flow simulations.

In a containment safety analysis, multi-dimensional characteristic in thermal hydraulics is very important because the flow path is not confined in a large free volume of the containment. It is also because of a difference in length scales between a characteristic length of the flow and representative length of a compartment in the containment. In order to implement the turbulence-unresolved approach with multi-dimensional nodalization to the containment safety analysis, parameters of correlations must be specified, but it is difficult because of the transient nature of an accident progress and very few experimental data to validate the correlations. From 90's, the turbulence-resolved approach has been applied for the containment safety analysis. Royl et al. [9] used GASFLOW to analyze the hydrogen–steam distributions in the Konvoi-type NPP. Houkema et al. [10] compared the results from a LP code and the commercial CFD code CFX, and suggested that a three-dimensional analysis is necessary to predict a non-uniformly distributed hydrogen concentration. Recently, Jiang et al. [11] applied CFD method for an analysis of PCCS (passive containment cooling system) implemented in AP1000.

Recently, detailed thermal hydraulic experiments in large containment vessels have been conducted in the world among which PANDA [12] and THAI [13] test facilities are most contributed for the model validations. From the review of the current research in multi-dimensional containment safety analysis, it is thought that the turbulence-resolved approach is useful to understand thermal hydraulic phenomena in a containment during an accident and support nodalization and parameter setting of the turbulence-unresolved LP approach though validation and improvement of models are still required.

For the evaluation of hydrogen safety in domestic NPPs, KAERI imported GASFLOW and COM3D codes developed by KIT. GASFLOW is used for an analysis of hydrogen behaviors in a containment and COM3D is used for a combustion analysis with a distribution of hydrogen obtained from a GASFLOW analysis. Though GASFLOW and COM3D are well developed for a real NPP containment analysis, there exist shortcomings in nodalization and turbulence models. They are based on a Cartesian or cylindrical mesh generation, so it is impractical to refine a mesh locally in a region with a physical or geometrical complication. Recently it is known that jet flow of a released gas from RCS (reactor cooling system) is strongly affected in initial distribution and mixing of hydrogen. If the turbulent convection of the released gas augmented by jet momentum and buoyancy force is not well resolved, then the hydrogen distribution during an accident may not be conservative.

COM3D is also one of matured codes for an analysis of hydrogen combustion in an NPP containment. But its applicability is limited because it allows only uniform Cartesian meshes. Unfortunately it is impossible to conduct an experiment of hydrogen flame propagation in a real-scale NPP containment to validate the hydrogen combustion codes. It is only possible to validate turbulent combustion models and numerics by using experiments conducted in modeled small-scale geometries. In the case of applying the analysis codes for an NPP containment, a code-to-code comparison is useful by using codes with different numerical and physical models to understand characteristics of flame propagation in an NPP containment.

In order to supplement the current framework of hydrogen safety evaluation with GASFLOW-COM3D, a new turbulence-resolved approach founded on modern CFD technology is introduced. The new code is developed based an open-source CFD tool OpenFOAM [14] with robust mesh generation and manipulation. Another reason to choose the OpenFOAM library is easy modification and addition of physical models such as turbulence.

In this paper, recently conducted research for the development of the turbulence-resolved analysis code is introduced.

## 2. Turbulence-Resolved Approach

In this section some numerical modeling and results of the turbulence-resolved approach are described.

### *2.1 Turbulent mixing of a stratified gas*

When hydrogen generated from a fuel-cladding oxidation in a nuclear reactor is released into the containment, it can be stratified in the upper region of a compartment such as a containment dome. It is very important to evaluate how long it takes for the stratified hydrogen mixture cloud to be well mixed with ambient gas. During a severe accident, hot steam is continuously released into the containment even after the hydrogen release. A jet flow of the steam easily becomes a buoyant jet or plume when it loses its momentum by a turbulent shear or occurrence of jet impingement on an obstacle or compartment wall. The turbulent buoyant flow of the steam enhances mixing of the highly concentrated hydrogen mixture cloud developed in the upper region, which is called an erosion of stratified hydrogen. This phenomenon is revealed in the frame work of international collaboration research HYMERES [15] operated by PSI and CEA. Recently one of the erosion experiments HP1-6 was chosen as a blind benchmark problem. Here, the results of the blind benchmark simulation are presented.

In the HP1-6 test, the vessels are initially filled with steam and helium as a simulant of hydrogen is stratified in the top region of vessel 1 (the left vessel in Fig. 1) with a nominal mole concentration of 25 %. Steam is injected from a vertical pipe inside the vessel 1 and the steam jet is impinged on the circular disk installed 1 m away from the pipe exit.

In order to simulate the HP1-6 test using a desktop PC with 6 cores, a computational mesh had to be carefully generated to control the size of the mesh. For the blind benchmark, the vertical circular pipe of steam injection was modeled as a square pipe with an equivalent sectional area but the mesh was refined in the region of the jet flow to resolve a turbulent shear as shown in Fig. 1.

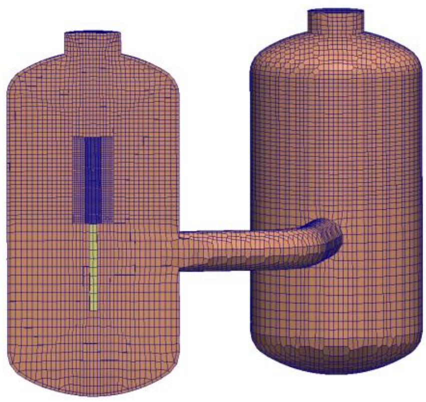

Fig. 1. Mesh generated for a simulation of HP1-6 test

A standard k-e turbulence model with modifications for generation of kinetic energy by buoyancy and prevention of kinetic energy build-up at a stagnation point [2] was used.

$$G_b = -\frac{C_\mu}{\mathrm{Pr}_t}\frac{k^2}{\varepsilon}\mathbf{g}\cdot\nabla\rho \qquad (1)$$

$$\widetilde{P} = \min(P, 10\rho\varepsilon) \qquad (2)$$

In order to consider heat loss from the vessel wall in a transient mode, it is necessary to model a heat capacity of the solid structure. It is possible to simulate conductive heat transfer in the vessel wall by a general conjugate heat transfer model. Here, 1-dimensional conductive heat transfer model is applied as a vessel wall boundary condition to reduce a computing time.

$$\rho C_p \frac{\partial T_s}{\partial t} = q''_{ex} - q''_w \qquad (3)$$

where $q''_{ex}$ is heat flux coming in from outer vessel wall and $q''_w$ is heat flux calculated on the vessel inner wall. $T_s$ is a lumped vessel solid temperature. After getting $T_s$ from Eq. (3), vessel inner wall temperature is obtained on the assumption of parabolic temperature profile.

Fig. 2 shows helium distribution at 195 s after steam injection was initiated. Because the injected steam density is lighter than the ambient cold steam but heavier than the stratified helium mixture, it is not likely to penetrate the helium layer. The helium layer is slowly eroded by turbulent shear at the boundary of the helium layer. The detailed comparison of the numerical results with the experimental data is omitted here but will be presented in the workshop

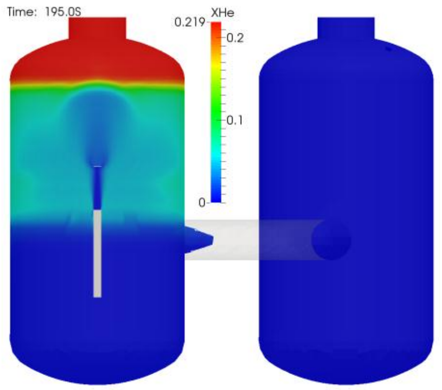


Fig. 2. Helium distribution from the results of HP1-6 test simulation

## *2.2 Hydrogen burning in a containment*

As mentioned in the introduction of this paper, a flame acceleration in a compartment of the containment is evaluated by using the COM3D code. But because COM3D allows only a uniform Cartesian mesh, the mesh size is quickly increased to model a complicated geometry of a containment. And it is thought that a new code based on modern CFD technology is needed to compare the COM3D results for a NPP containment.

Recently a new method to map a GASFLOW solution on an OpenFOAM mesh was developed. Here, mapping means a solution transfer between the two codes with different meshes.

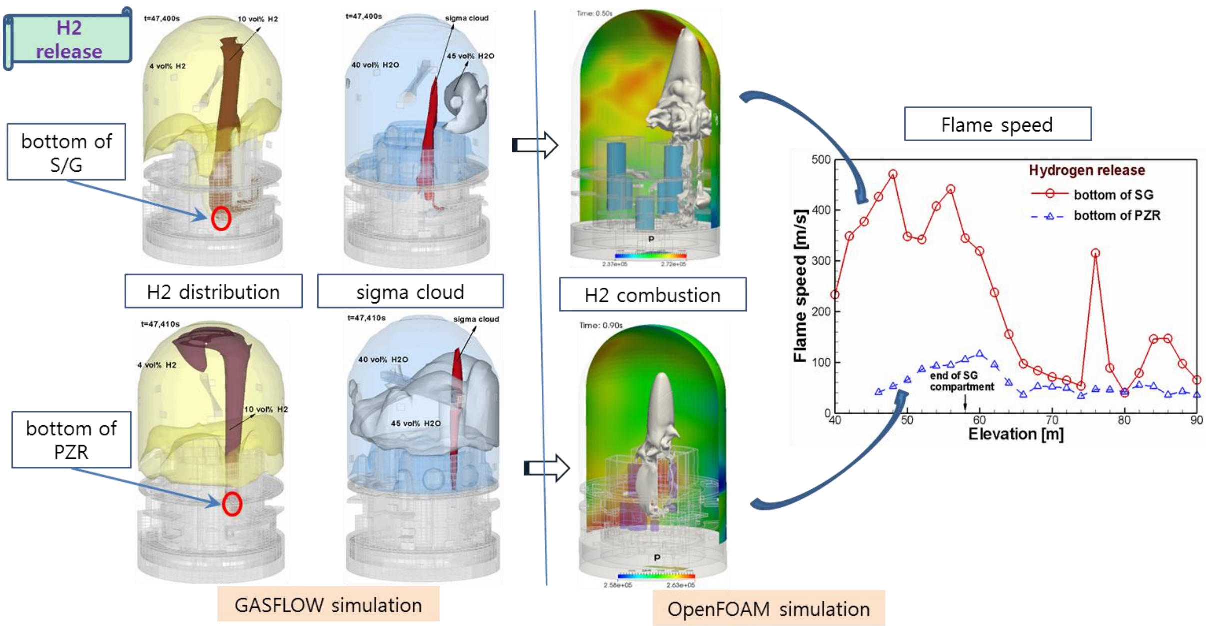


Fig. 3. Comparison of the characteristics of hydrogen flame acceleration in a steam generator compartment depending on hydrogen release location in APR1400 containment

In this paper, a coupled analysis method of the GASFLOW [9] code for hydrogen distribution and the OpenFOAM [10] code for hydrogen combustion is applied for an assessment of the possibility of hydrogen flame acceleration during a station blackout accident in the Shin-Ulchin 1&2 plants.

In this study, the main RCS components such as reactor vessel, steam generator, pump and pressurizer are modeled as cylinders with their equivalent volumes and heights. The size of the mesh used for APR1400 containment is about 3 million polyhedral cells.

The left figures of the vertical center line show the hydrogen distributions and sigma clouds obtained from GASFLOW analysis and the right figures of the vertical center line are results of hydrogen combustion analyses. Two cases with different hydrogen release locations were studied. One is the case with a hydrogen release at the bottom of a steam generator, and the other case is a hydrogen release at the bottom of a pressurizer. In the figure of the flame speed comparison, it is seen that a hydrogen flame can be more accelerated in the lower release case. It is also found that the hydrogen flames are decelerated after leaving the steam generator compartment.

### 3. Summary

In this paper, a new measure to distinguish numerical analysis methods is proposed, which is based on how to resolve turbulent characteristics in nuclear thermal hydraulics. Even though 3-dimensional thermal hydraulic equations are used, it belongs to turbulence-unresolved approach if turbulence effect is lumped into correlations. Nowadays the turbulence-resolved approach becomes more important, and it is expected that the traditional LP method is supported by the turbulence-resolved method.

Here, a new tool based on the turbulence-resolved method is introduced to simulate hydrogen distribution and combustion in a containment. A continuous research on the turbulence-resolved method will improve credibility of a hydrogen safety evaluation in a NPP containment.

### REFERENCES

[1] G. Yadigaroglu, Computational Fluid Dynamics for Nuclear Applications: from CFD to multi-scale CMFD, Nuclear Engineering and Design, Vol.235, p.153, 2005.

[2] R.O. Gauntt, et al., “MELCOR Computer Code Manuals”, NUREG/CR-6119, SAND2001-0929P, 2001.

[3] K.E. Carlson, et al., “RELAP5/MOD3 Code Manual”, NUREG/CR-5535, EG&G Idaho, Inc., 1990.

[4] F. Martin-Fuertes, et al., A Melcor-based Model to Calculate Vapor Natural Circulation in Pressurized Water Reactor High-Pressure Severe Accident, Nuclear Technology Vol. 127, p.141, 1999.

[5] “RELAP5-3D Code Manual”, The RELAP5-3D code development team, INEEL-EXT-98-00834, 2005.

[6] F. Rahn, et al., “GOTHIC Containment Analysis Package Technical Manual”, EPRI, 2009.

[7] C.Y., Park, et al., Analysis of FLETCH-SEASET 163 Rod Blocked Bundle Data Using COBRA-TD”, NUREG-CR-4166, 1986.

[8] J. R. Travis, et al., GASFLOW: A Computational Fluid Dynamics Code for Gases, Aerosols, and Combustion, LA-13357-M, FZKA- 5994, 1998

[9] P. Royl, H. Rochholz, W. Breitung, and J. Travis, G. Necker, “Analysis of steam and hydrogen distributions with PAR mitigation in NPP containments,” Nucl. Eng. and Design 202, pp. 231–248 (2000).

[10] M. Houkema, E.M.J. Komen, N.B. Siccama, and S.M. Willemsen, “CFD analyses of steam and hydrogen distribution in a nuclear power plant,” NURETH-10, Seoul, Korea (2003).

[11] X.W. Jiang, E. Studer and S. Kudriakov, “A simplified model of Passive Containment Cooling System in a CFD code,” Nucl. Eng. and Design 262, pp. 579-588 (2013).

[12] D. Paladino and J. Dreier, “PANDA: A Multipurpose Integral Test Facility for LWR Safety Investifations,” Sci. and Tech. of Nucl. Inst. 2012, ID 239319 (2012).

[13] OECD/NEA THAI Project “Hydrogen and Fission Product Issues Relevant for Containment Safety Assessment under Severe Accident Conditions,” NEA/CSNI/R(2010)3, 2010.

[14] H. Weller et al., OpenFOAM: The Open Source CFD Toolbox User Guide, http://www.openfoam.org , 2014.

[15] D. Paladino et al., “OECD/NEA HYMERES Project: PANDA Test HP1_6_2 specifications for the blind benchmark with initial and boundary conditions,” PSI, 2015.
Radiation Detection and Measurement, John Wiley & Sons, New York, pp.612-613, 1999.