Preliminary CFD Analysis for the SNL H_2 Flame Test in the Rectangular Channel

H. S. Kang,[a] J. T. Kim,[a] and S. B. Kim,[a]

a Korea Atomic Energy Research Institute, P. O. Box 105,Yuseong,Taejon, Korea, 305-600, hskang3@kaeri.re.kr

1. Introduction

A preliminary CFD (Computational Fluid Dynamics) analysis is performed against the SNL (Sandia National Laboratories) hydrogen flame test [1] to develop an analysis methodology of a hydrogen flame acceleration phenomena inside the NPP (Nuclear Power Plant) containment. The developed CFD methodology may be used to mitigate the possibilities of a flame acceleration and DDT (Deflagration to Detonation). The advantage of the CFD analysis is that it can easily generate a real grid model representing the structures inside the containment when compared to the system code analysis [2]. However, the CFD code should be validated against the test results of the hydrogen flame acceleration before the application to the NPP.

2. SNL Flame Test [1]

The SNL performed the test of a flame acceleration and DDT for the hydrogen-air mixtures in the rectangular channel (Fig. 1) varying the hydrogen mole fraction (12~30%), degree of transverse venting and the blockage ratio due to an obstacle to find out the important factor in the generation of the peak overpressure. The facility was closed on the ignition end and opened on far end. The selected test case (F-10) for the CFD analysis is a mixture of hydrogen (12.3%) and air without the obstacle and the venting place as for the first validation calculation. The measured variables in the test are the turbulent flame speed and the peak overpressure.

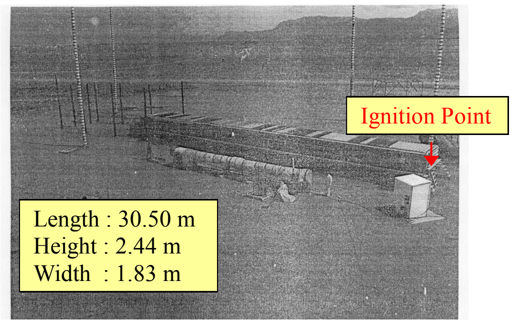


Figure 1. SNL Flame Test Facility

3. CFD Analysis

3.1 Grid Model and Boundary Conditions

A 3-dimensional grid model (Fig. 2) simulating the rectangular channel according to the dimension of the SNL test facility is developed. A total of 1,457,250 hexa mesh cell is produced to maintain the proper aspect ratio, and a dense mesh cell distribution around the center region was located to resolve a rapid propagation of a flame. An opening condition was applied to the outlet region, which allows an inflow and outflow of a fluid through the surfaces. The gas distribution of hydrogen (12.3 vol. %), oxygen and nitrogen was given to the tent volume as for an initial condition. And also, the spark ignition model [3] which is an effected region model (Fig. 3) is introduced to simulate the spark operation by the electric device in the test facility.

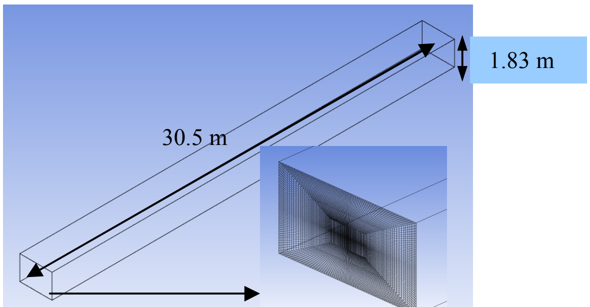


Figure 2. Grid Model

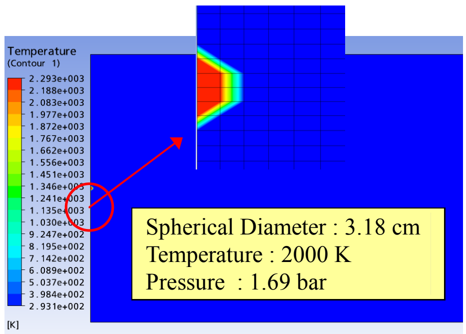


Figure 2. Spark Ignition Model
(Temperature distribution at 0.0 sec on the center plane)

3.2 Flow Field Models and Combustion Model

The hydrogen flame acceleration phenomenon in the rectangular channel was treated as a compressible flow, a combustion flow and a turbulent flow. The governing equations used in this study were the Navier-Stokes, the energy and the species transport equations with a coupled solver algorithm implemented in the CFX-11 [4]. Turbulent flow was modeled by the standard k-ε turbulent model. And also, the EDM (Eddy Dissipation Model) [4] is used for the one step combustion reaction of hydrogen and air. The global reaction rate of the EDM (Eq. (1)~Eq. (2)) is determined according to the slowest reaction rate of a fuel (Eq. (2-1)), a oxidant (Eq. (2-2)) and a product (Eq. (2-3)). Each reaction rate is proportion to a turbulent quantity ratio (ε /k), mass fraction (Y_f, Y_o, Y_p), model constants (A, B) and stoichiometric coefficient (Eq. (2-4)) [4]. And also, this calculation was performed as a transient case for about 0.2 sec with a time step of 0.001 sec..

$$\text{Fuel} + \text{Oxidant} \rightarrow \text{Product} \tag{1}$$

$$\text{Reaction Rate} = \min(R_f, R_o, R_p) \tag{2}$$

$$R_f = A\rho\frac{\varepsilon}{k}(Y_f) \tag{2-1}$$

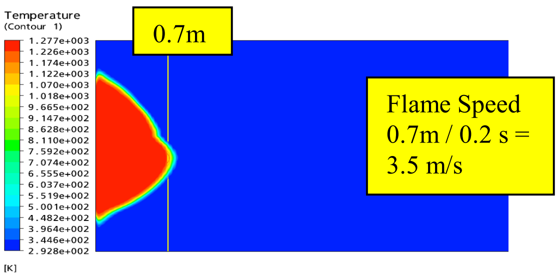
$$R_o = A\rho\frac{\varepsilon}{k}\left(\frac{Y_o}{r_f}\right) \tag{2-2}$$

$$R_p = AB\rho\frac{\varepsilon}{k}\left(\frac{Y_p}{1+r_f}\right) \tag{2-3}$$

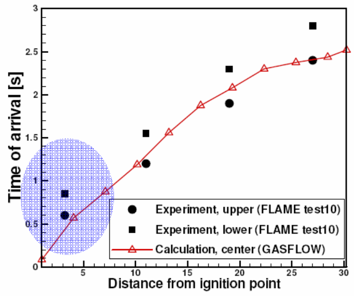
$$r_f = \frac{M_o v_o}{M_f v_f} \tag{2-4}$$

3.3 CFD Analysis Results

The preliminary CFD analysis results (Fig. 3, (a)) show that the calculated flame sped of about 3.5 m/s agree well with those of the test data (Fig. 3, (b)) only for the beginning part of the test results. In the CFD results, the flame speed is obtained based on the temperature distribution contour from which we can know the information of the flame arrival time and the distance from the ignition point. However, it is difficulty to say that the CFD results quantitatively predicted well test results because the CFD calculation was performed for the very short periods. Therefore, the further calculation should be performed to compare with the test results at least for 3 seconds.



(a) Temperature distribution at 0.2 sec. (CFD Results)



(b) Flame Speed (Test Results)

Figure 3. Flame Speed Comparison of CFD Results with Test Results

4. Conclusion and Further Research

From the preliminary CFD analysis results for the SNL hydrogen flame acceleration test results, we can know that the CFD code with the EDM combustion model can simulate the hydrogen flame acceleration phenomena in the rectangular channel. However, the CFD calculation should be performed for a longer time to quantitatively compare with the test data. And also, the CFD simulation should be conducted for the obstacle geometry inside the channel.

ACKNOWLEDGEMENTS

This work has been performed under the Nuclear R&D program supported by the Ministry of Science and Technology of Korea.

REFERENCES

1. M. P. Sherman, S. R. Tieszen and W. B. Benedick, “The Effect of Obstacle and Transverse Venting on Flame Acceleration and Transit to Detonation for Hydrogen-Air Mixtures at Large Scale”, NUREG/CR-5275, 1998.
2. J. T. Kim, et al, “Numerical Analysis of the Hydrogen-Steam Behavior in the APR1400 Containment during a Hypothetical Total Loss of Feed Water Accident”, *J. of Comput. Fluids Eng*., Vol. 10, No. 3, 2005.
3. H. S. Kang, et al, “CFD Analysis of Gas Explosion Phenomena”, Proc. of ISTP-18, Daejeon, Korea, Aug. 27-30, 2007.
4. Ansys, Inc., “CFX-11.0 Manual”, 2007.