Development of a Spark Ignition Model for a CFD Analysis of the JAEA Explosion Test

H. S. Kang,a S. B. Kim,a M. H. Kim,a W. J. Lee,a H. C. No b

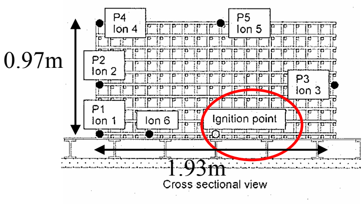
a Korea Atomic Energy Research Institute, P. O. Box 105,Yuseong,Taejon, Korea, 305-600, hskang3@kaeri.re.kr
b Korea Advanced Institute of Science and Technology, 373-1, Guseong,Yuseong,Taejon,Korea, 305-701

1. Introduction

A CFD (Computational Fluid Dynamics) analysis with a spark ignition model was performed against the JAEA explosion test [1] to develop an analysis methodology of a gas explosion and to predict a peak overpressure. The predicted peak overpressure may be used to decide a safety distance between a High Temperature Gas-Cooled Reactor (HTGR) and a hydrogen production facility [2]. In the JAEA explosion test, the electric spark device was used to ignite a mixture of methane and air, and the equivalent energy for the spark operation is reported as 40 J [1]. This value is very large when compared to the spark ignition energy of about 10 mJ in an engine combustion test [4]. The excessive energy is suspected to have an effect on the overpressure distribution and the flame speed around the spark location. Therefore, a spark ignition model representing the pressure, the temperature and the volume of the activated region due to a spark is necessary for an accurate CFD simulation of the JAEA explosion test.

2. JAEA Explosion Test [1]

JAEA performed a gas explosion test in an open space by varying the gas concentration, the ignition method and the existence of an obstacle, and measured the overpressure and the flame front arrival inside the tent where the flammable gas was located and around the tent (Fig. 1). The selected test case for the CFD analysis is a mixture of methane (9.5 vol. %) and air with an obstacle under the spark ignition.



Tent Volume : 5.6 m^3 Obstacle Volume : 0.4 m^3
Obstacle ; - Steel Tube - I.D/O.D [mm]: 15.9/ 21.3

Figure 1. JAEA Gas explosion test facility

3. CFD Analysis

3.1 Spark Ignition Model

The local phenomena of a spark and the ignition process of a flame are too complicated to model. The rapidly increased temperature and pressure of a gas cloud due to a spark are easily decayed in a short distance [3]. Therefore, a spherical activated region model based on an energy conservation (Eq. 1) under the assumption of an adiabatically confined condition is introduced. However, we have two unknown variables of the mass (M) and the hot gas temperature (T_h) in the Eq. (1) where a mass term (M) can be transferred to the volume data by the ideal gas law. A parametric study is performed by varying the temperature from 2000K to 6000K [3] to calculate the activated volume (Table 1). And then, the pressure can be approximately calculated by the ideal gas law based on the obtained temperature and volume even though an ideal gas law is not a valid for the region where the gas temperature is higher than 1000K and the density of air and methane is the data of 1273K and 573K, respectively. In order to validate the spark ignition model, a comparison work of the test data with the CFD results for the overpressure and the flame speed will be performed.

$$Q[J] = MC_p\left(T_h - T_c\right) = V\rho C_p\left(T_h - T_c\right) \qquad (1)$$

Table 1 Parametric Study of the Spark Ignition Model

	2000K	3000K	4000K	5000K	6000K
Volume 10^{-5} [m^3]	**1.69**	1.06	7.78	6.12	5.05
R [mm] $V=4\pi R^3/3$	**15.92**	13.65	12.29	11.35	10.64
Pressure [bar] $P = \rho RT$	**1.70**	2.51	3.31	4.18	5.02
Density [kg/m^3]	ρ_{air} (1bar, 1273K) = 0.268 ρ_{CH4} (1bar, 573K) = 0.3415				

3.2 Grid Model and Boundary Condition

A 3-dimensional grid model (Fig. 2) simulating the tent and its environment was developed based on JAEA's CFD work [1]. A total of 1,058,400 hexa mesh cells was produced, and a dense mesh cell distribution was located around the tent to resolve a rapid propagation of a flame. The obstacle inside the tent was directly modeled by a rectangular closed tube instead of a circular tube. An opening condition was applied to all the surrounding surfaces except for the bottom surface, which allows an inflow and outflow of a fluid through the surfaces. The stoichiometric distribution of methane (9.5 vol. %), oxygen and nitrogen was given to the tent volume for an initial condition. And also, a spark ignition model of 2000K, 1.67bar and a spherical diameter of 3.18cm are used for the initial condition simulating the spark phenomena.

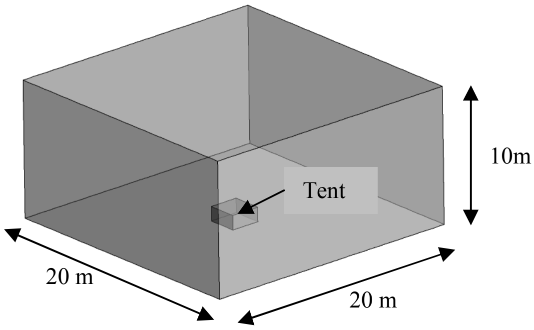


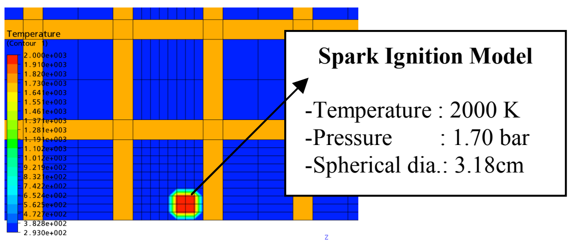
Figure 2. Grid Model for the CFD Analysis

3.3 Flow Field Models and Combustion Model

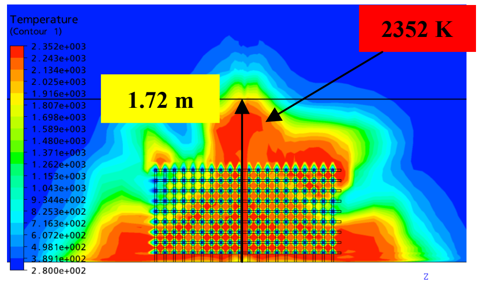
The methane combustion phenomenon in the obstacle structure was treated as a compressible flow, a combustion flow, a turbulent flow and a buoyant flow. The governing equations used in this study are the Navier-Stokes, the energy and the species transport equations with a coupled solver algorithm implemented in the CFX [4]. Turbulent flow was modeled by the standard k-ε turbulent model, and the buoyancy was modeled by the Boussinesq approximation. And also, a discrete transfer model was used for the radiation heat transfer. The Eddy Dissipation Model (EDM) was used for the one step combustion reaction of methane and air. A transient calculation for 0.01sec is performed with a time step of 0.001 sec.

3.4 CFD Analysis Results

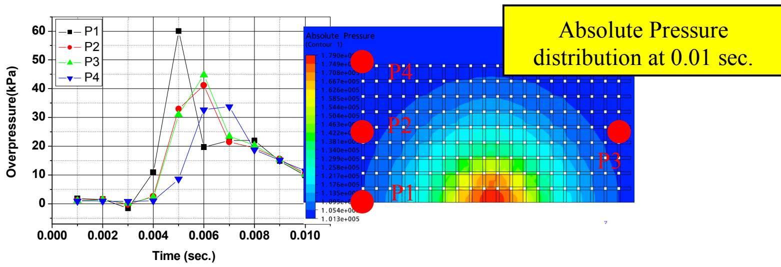
The CFD analysis results using the first parametric condition of the spark ignition model show that the turbulent flame sped (170m/s) results agree well with those of the test data (163m/s), and also the overpressure results (20~40 kPa) from 0.004 to 0.006 sec. are similar to the test data of about 30 kPa except for the maximum overpressure location. From this comparison work, we can know that the CFD analysis methodology with the spark ignition model simulates the explosion test reasonably even though an asymmetric temperature distribution should be carefully examined.



(a) Temperature distribution at 0.0 sec.



(b) Temperature distribution at 0.01 sec



(c) Overpressure Results inside Tent
(Test Data : 25.4 kPa (avg), 27.5 kPa (max))

Figure 3. CFD Analysis Results

4. Conclusion and Further Research

From the results of the comparison of the turbulent flame speed and the overpressure by the CFD and test data, we can conclude that the developed spark ignition model representing a spark operation of 40J by an electric device is performed reasonably. However, in order to increase the accuracy of it, all the parametric calculations should be conducted and the pressure value of the activated region should be recalculated instead of using an approximated value (Table 1).

ACKNOWLEDGEMENTS

This work was financially supported for the nuclear R&D program from the Ministry of Science and Technology of Korea.

REFERENCES

1. Yoshitomo Inabal, Tetsuo Nishihara, Mark A. Groethe and Yoshikazu Nitta, “Study on explosion characteristics of natural gas and methane in semi-open space for the HTTR hydrogen production system”, *NED* 232, (2004), pp. 111-1191.
2. H. S. Kang, et al, “CFD Analysis of Gas Explosion Phenomena”, Proc. of ISTP-18, Daejeon, Korea, Aug. 27-30, 2007.
3. John B. Heywood, *Internal Combustion Engine Fundamentals*, pp. 427-443, McGraw-Hill, 1988.
4. Ansys, Inc., “CFX-11.0 Manual”, 2007.