Numerical Study to Find Mechanisms Behind the Spread of Fire in a Multi-room Environment

Gong Hee Lee [a,b*], Ae Ju Cheong [a]
[a]*Nuclear Safety Research Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[b]*Nuclear and Radiation Safety Department, University of Science and Technology, Daejeon 34133, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

According to the international fire safety analysis studies, fire contributes significantly to the overall core damage frequency (CDF) for both existing and new nuclear power plants. Fire simulation models have been developed as analytical tools for a performance-based fire safety assessment. The use of calculated predictions could be considered, on the one hand, for improvements and upgrades of the fire protection by the licensees and, on the other hand, as a tool for reproducible and clearly understandable estimations in assessing the available and/or foreseen fire protection measures by the regulatory authority. However, there are still a lot of challenges in the use of validated fire simulation models that can reasonably predict the consequences of a fire in the nuclear power plants.

In this study, in order to evaluate the prediction performance of fire simulation model for the spread of fire in a multi-room environment, calculations were conducted with Fire Dynamic Simulator (FDS) 6.4 [1]. The predicted results were compared with measured data (PRS_D6) obtained from PRISME Door test series.

2. Analysis model

The PRISME Door test was carried out in the DIVA facility [2]. As shown in Fig. 1, configurations of the DIVA facility consists of 3 rooms separated by a door enabling the spread of hot gases and smoke from the fire (or source) room (room 1) towards the target rooms (room 2 and room 3). These rooms are connected in parallel to a ventilation network, which ensures an air renewal rate and the extraction of combustion products.

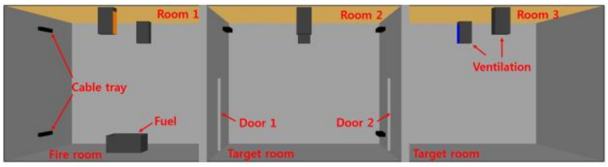


Fig. 1. Schematic diagram of DIVA facility (PRS_D6 case).

Among the test matrix of the PRISME Door, PRS_D6 was selected in this study. Summary of PRS_D6 test are as follows:

- Fire surface area : 1.0 m^2
- Air renewal rate in each room : 560.0 m^3/h
- Number of rooms : 3
- Number of open doors : 2
- Fuel : Hydrogenated tetra-propylene (TPH)
- Fuel amount : 25.098 kg
- Domain size : Width (x) × Depth (y) × Height (z) = 15.4 m × 6.0 m × 4.0 m
- Room door size : 0.8 m (y) × 2.1 m (z)
- Fire duration : 7 min
- Fire extinction mode : Lack of oxygen
- Fuel burned : 13 kg

Properties of TPH are summarize in Table I.

Table I: Properties of hydrogenated tetra-propylene

Chemical composition	Boiling point	Flash point	Density (at 20℃)
$C_{12}H_{26}$	188℃	51~61℃	0.76g/㎖

3. Numerical modeling

FDS, has been developed at the NIST (National Institute of Standards and Technology), is a computational fluid dynamics (CFD) model of fire-driven fluid flow. FDS solves numerically a form of the Navier-Stokes equations appropriate for low-speed (Mach number < 0.3), thermally-driven flow with an emphasis on smoke and heat transport from fires.

The implemented numerical algorithm is an explicit predictor-corrector scheme, second order accurate in space and time. Turbulent flow is solved by means of Large Eddy Simulation (LES), which is the default turbulence model. It is possible to perform a Direct Numerical Simulation (DNS) if the underlying numerical mesh is fine enough.

Radiative heat transfer is included in the model via the solution of the radiation transport equation for a gray gas, and in some limited cases using a wide band model. The equation is solved using a technique similar to finite volume methods for convective transport.

Combustion is modeled using a mixture faction approach, in which a single transport equation is solved for a scalar variable representing the fraction of gas originating from the fuel stream.

All solid surfaces are assigned thermal boundary conditions, plus information about the burning behavior of the material. Heat and mass transfer to and from solid surfaces is usually handled with empirical correlations.

Scalar quantities are assigned to the center of each grid cell; vector components are assigned at the appropriate cell faces. This is what is commonly referred to as a staggered grid. Its main purpose is to avoid “checker-boarding” in pressure-velocity coupling.

FDS approximates the governing equations on a rectilinear mesh. In this study, a cell size of 5 cm and total number of mesh of 308 × 120 × 80 = 2,956,800 were used. Previous studies [3] have shown that a cell size of 10 cm could produce appropriate results at a moderate computing cost.

4. Results and Discussion

4.1 General heat and flow distribution

Fig. 2 shows the distributions of flow and temperature at y = 3.0 m (center plane in depth) and t = 226 s. t = 226 s is the instant for the maximum heat release rate to occur. In fire (source) room, buoyant gases moved up to ceiling in fire plume and ceiling jet spread radially until confined by room partition. Additionally, because plume entrained surrounding air, relatively cold flow moved from target rooms (Room 2 & 3) toward fire (source) room (Room 1) through a lower part of open door.

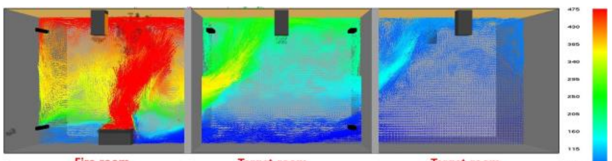
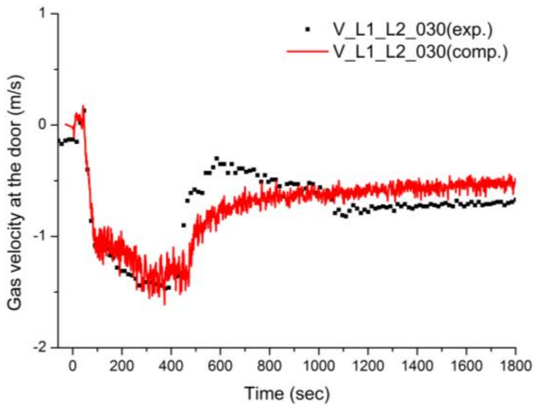


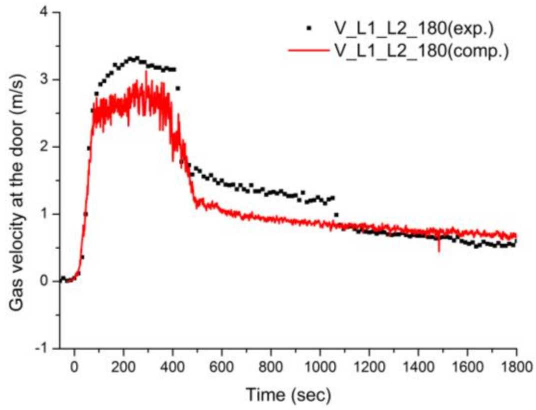
Fig. 2. Distributions of flow and temperature at y = 3.0 m (center plane in depth) and t = 226 s.

4.2 Gas velocity transiting through room door

Fig. 3 shows gas velocity at the specified locations of door between fire (source) room and target room. Gas velocity has positive value when the gas flow transits from fire (source) room (Room 1) toward target room (Room 2) through room door. On the contrary to this, gas velocity has negative value when the gas flow is entered from target room (Room 2) toward fire (source) room (Room 1).



(a) 30 cm from the floor



(b) 180 cm from the floor

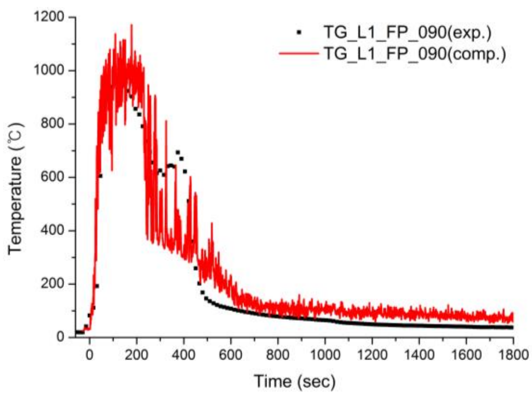
Fig. 3. Gas velocity at the door between fire (source) room and target room.

Because heat release rate (HRR) had peak magnitudes (above 1,000 kW) during t = 80 s ~ 400 s, relatively hot gas with high velocity magnitude moved from fire (source) room (Room 1) toward target room (Room 2) through an upper part of open door (see Fig. 3(b)). At a lower part of open door (z = 30 cm), the opposite result in the gas flow direction occurred (see Fig. 3(a)).

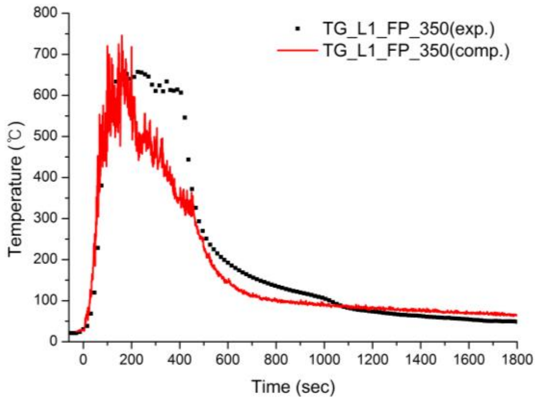
Although the predicted velocity variations with FDS were similar to those of measurement in the qualitative manner, there was a certain level of difference in the calculation result. For example, at certain time interval (t = 100 s ~ 1,100 s), FDS under-estimated the positive velocity magnitudes in comparison with the measured data (see Fig. 3(b)).

4.3 Temperature field

Fig. 4 shows flame and plume temperature at the specified locations from the bottom of fuel pan (z = 40 cm). Flame and plume temperature reached the peak temperature and then it decreased with time. The maximum temperature of flame and plume near fuel surface (90 cm from the bottom of fuel pan) was higher than that near ceiling (350 cm from the bottom of fuel pan).

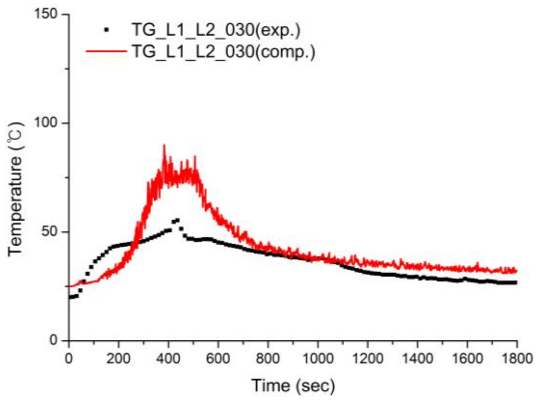


(a) 90 cm from the bottom of fuel pan (z = 40 cm)

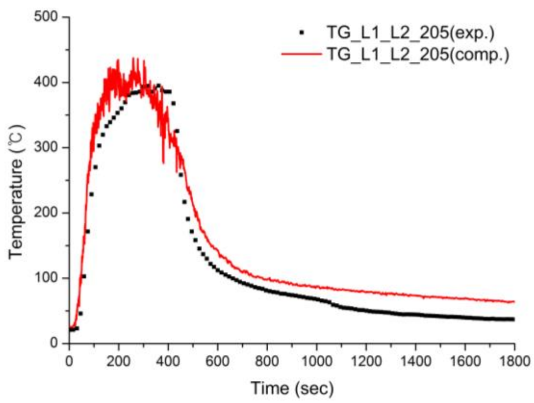


(b) 350 cm from the bottom of fuel pan (z = 40 cm)
Fig. 4. Flame and plume temperature

Around the instant for the maximum heat release rate to occur (t = 226 s), while the predicted flame and plume temperature with FDS gradually decreased, the measured data showed a certain time intervals with constant temperature.

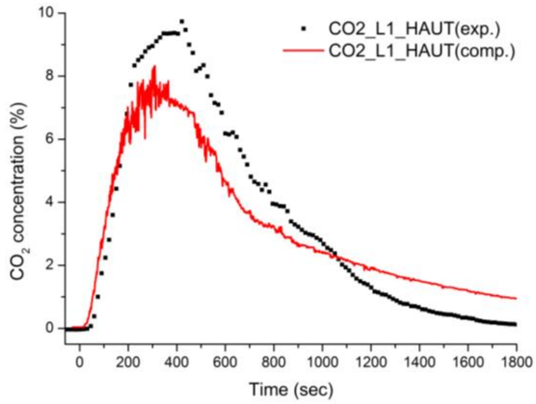


(a) 30 cm from the floor

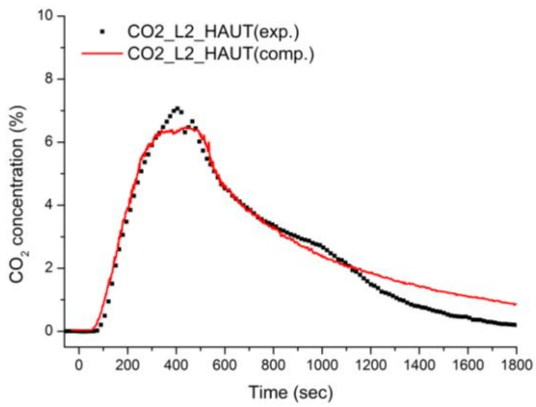


(b) 205 cm from the floor
Fig. 5. Gas temperature in doorway between fire (source) room and target room.

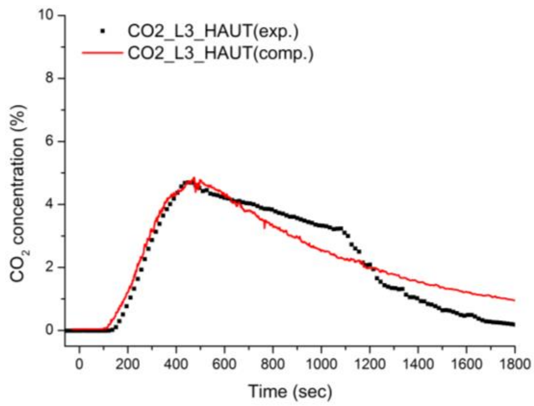
Fig. 5 shows gas temperature in doorway between fire (source) room (Room 1) and target room (Room 2) at the specified locations from the floor. As already explained in section 4.1 and 4.2, while relatively hot gas flow was distributed over an upper part of open door, relatively cold gas flow was spread over a lower part of open door. Therefore, there was a difference in the magnitude of peak temperature. Overall FDS over-estimated the temperature magnitudes in comparison with the measured data at a lower part of open door (see Fig. 5(a)).



(a) Room 1



(b) Room 2



(c) Room 3
Fig. 6. Carbon dioxide concentration at top of fire (source) room.

4.4 Gas concentrations

Fig. 6 shows carbon dioxide concentration at top of fire (source) room. Carbon dioxide concentration rapidly increased until the fire extinguishing did occur and then gradually decreased. In particular, FDS under-estimated carbon dioxide concentration in comparison with the measured data at certain time interval (t = 250 s ~ 1,000 s), as shown in Fig. 6(a).

3. Conclusions

In this study, in order to evaluate the prediction performance of fire simulation model for the spread of fire in a multi-room environment, calculations were conducted with FDS 6.4. The predicted results were compared with measured data (PRS_D6) obtained from PRISME Door test series. The major conclusion could be summarized as follows;

- In fire (source) room (Room 1), buoyant gases moved up to ceiling in fire plume and ceiling jet spread radially until confined by room partition. Additionally, because plume entrained surrounding air, relatively cold flow moved from target room (Room 2) toward fire (source) room (Room 1) through a lower part of open door.
- Although FDS could give the meaningful information to understand the thermal-flow pattern in the under-ventilated fire condition, it still had the limitation (for example, under-estimation of gas temperature in doorway) and then showed a certain level of uncertainty in the calculation result.
- Therefore, in the future, lessons learned from these benchmark simulations will be used to evaluate the sensitivities of both input variables and numerical models, implemented in FDS.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS), granted financial resource from the Nuclear Safety and Security Commission (NSSC), Republic of Korea (No. 1305002). This paper was also written by using the experimental data (CSNI2006/03) from OECD/NEA International PRISME Project.

REFERENCES

[1] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, and K. Overholt, Fire Dynamics Simulator User's Guide, http://dx.doi.org/10.6028/NIST.SP.1019, National Institute of Standards and Technology, 2016.

[2] L. Rigollet, OECD/NEA PRISME Project Application Report, NEA/CSNI/R(2012)14, 2012.

[3] W. Klein-Heßling, M. Roewekamp, and O. Riesex, Evaluation of Fire Models for Nuclear Power Plant Applications Benchamrk Exercise No. 4: Fuel Pool Fire Inside A Compartment, GRS-213, 2006.