Sensitivity analysis of the design of large scale fire calorimeter for fire safety engineering research and education

Mohamad Lutfi Ramadhan, Muhammad Agung Santoso, Yulianto Sulistyono Nugroho*

Department of Mechanical Engineering, Universitas Indonesia

Research regarding fire phenomenon had been rapidly developed as fire incident could occurs frequently. In order to assess the fire incident itself, a measuring test rig that could simultaneously measure heat release rate and smoke release from various fire phenomenon could be implemented. Fire safety engineering research group within the Department of Mechanical Engineering Universitas Indonesia has been allocated a new space for future laboratory development. Therefore it is necessary to design a compact test rig arrangement that capable for carrying measurement of various fire phenomena and for education purposes. In designing a Large-Scale Fire Calorimeter, we need to incorporate safety and security aspect. One of those aspects is generated smoke that could spread throughout the laboratory. To reduce the risk associated with smoke spread, we utilize a fan with specification fits the smoke production yielded during fire test. A ducting hood with dimension of 5 x 5 m also installed to collect smoke prior to the duct line. This research analysed the sensitivity of the flow field of smoke movement by suction power of fan during various fire load and observed its capability in controlling smoke from fire test by carrying series of fire simulation using Fire Dynamic Simulator (FDS) version 6.

Keywords: Fire calorimeter, fire phenomenon, ISO 9705, smoke distribution simulation, computational fluid dynamics, gas temperature measurement

1. Introduction

Fire safety engineering research group within the Department of Mechanical Engineering Universitas Indonesia has been allocated a new space for future development. The size of the laboratory floor is 16 x 24 m. A fraction of the floor is an atria having the size of 7 (width) x 14 (length) x 8 (height) m. Due to a limited space, it is necessary to design a compact design of large calorimeter capable to arranges various measurement setup and enables various studies of full scale fire. Furthermore, on the basis of current and future challenge in term of fire engineering analysis, it is important to have a laboratory facility that proficient in verifying and validating fire modelling study.

In designing a measuring device in the form of Large-Scale Fire Calorimeter, we need to incorporate safety and security aspect to ensure the sustainability of the research using this device. One of those aspects is fire – generated smoke that could spread throughout the laboratory. To reduce the risk associated with smoke spread, we utilize an exhaust fan with specification fits the smoke production yielded during fire test.

By paying attention to numerous risk associated with smoke spread, it is important to improve fire prevention and smoke ventilation strategies. This study proposed a ventilation mode in the form of ducting which incorporated mechanical ventilation by an exhaust fan. The exhaust hood is operated at a capacity of approximately 3.5 m³/s in accordance with the ISO 9705 Room/Cornet Test standard. In addition, a short discussion on the phenomenon of the smoke flow will be indicate.

2. Simulation setup

2.1 Physical model

This study used combustion chamber test model with 5 x 5 m hood (note: for future use, the size of hood could be upsized) and 0.6 x 0.6 m ducting with 2 m length. To compare with the standard, we set the exhaust hood volume flow was variated as 3.33 m³/s and 6.66 m³/s. Hood was placed in 3.5 m above the burner. The physical model of the large scale fire calorimeter used in this study can be clearly observed in Fig. 1a.

2.2 FDS setup

Simulations were carried out using Fire Dynamic Simulator (FDS) of NIST version 6.0. FDS is a computational fluid dynamics (CFD) model of fire-driven fluid flow. The modeled geometry of the large scale fire calorimeter for the numerical simulation has a 1:1 ratio (same size) with the physical model, and it was shown in Fig. 2. Fire source was positioned in the floor. The fire heat release rate (HRR) was variated as 0.75 MW, 1 MW, 2 MW, 3 MW, and 4 MW. There was no fire growth rate because we assume that the fire would be the same for each second. The simulation time was set as 1200 seconds and the fire source burned as long as the simulation time. This approach was taken into account to make sure that the specified exhaust fan either capable or not to control the smoke
distribution. In other words, this approach was taken to observe the sensitivity of the exhaust fan being considered in controlling smoke flow at relatively steady state condition.

The cell dimensions in all of the simulation were set as 0.1 m in x, y, and z coordinates direction. The mesh resolution which is the ratio between characteristic diameter of the fire of the fire calculated from Eq. (1) is considered sufficient based on its value that is ranged between 4–16. The validation study sponsored by US Nuclear Regulatory Commission stated that the value between 4–16 is sufficient for the FDS mesh resolution according to the conducted validation experiments. In this paper, the calculated mesh resolution with characteristic diameter of the fire of the fire calculated from Eq. (1) is different for each heat release rate.

\[
D^* = \left( \frac{q}{\rho c_p T_\infty g \alpha} \right)^{2/5}
\]  

(1)

2.3 Operating parameters

List of various conditions which were studied in this paper is tabulated in Table 1. The magnitude of heat release rate and volume flow were specified as previous explanations. It can be seen from Table 1, in Cases 1 to 5, fire loads were variated and applied in each volume flow of exhaust fan.

<table>
<thead>
<tr>
<th>Case</th>
<th>Fire Load (kW)</th>
<th>Volume Flow (m³/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>750</td>
<td>3.33</td>
</tr>
<tr>
<td>2</td>
<td>1000</td>
<td>3.33</td>
</tr>
<tr>
<td>3</td>
<td>2000</td>
<td>3.33</td>
</tr>
<tr>
<td>4</td>
<td>3000</td>
<td>3.33</td>
</tr>
<tr>
<td>5</td>
<td>4000</td>
<td>3.33</td>
</tr>
</tbody>
</table>

3. Results and discussion

3.1 Smoke distribution

Particulate smoke is a product of incomplete combustion. It is generated in both smouldering and flaming combustion, although the nature of the particles and their modes of formation are very different. Thus, smoke distribution was noted in this research. In the following discussion, the observation will occasionally refer to the case number listed in Table 1 in discussing various impacts of volume flow and fire load. Figures 2(a) – 2(d) show the smoke distribution in the hood with volume flow 3.33 m³/s. The smoke distribution for volume flow 6.66 m³/s was depicted in Figs. 3(a) – 3(d). The rates of smoke thickness were represented by the black circle within Fig. 2 and 3.

Figures 2(a) and 3(a) represent the smoke distribution in Case 1 with volume flow of 3.33 m³/s and 6.66 m³/s. It can be observed that there is no accumulating of smoke on the hood and the smoke remains thin at the bottom of hood. Figures 2(b) and 3(b) depict the result of simulation of Case 2 with volume flow of 3.33 m³/s and 6.66 m³/s respectively. The work of exhaust fan is still able to suck the smoke inside the hood, but there is change in the accumulated smoke inside the hood. The exhaust fan is still able to suck the smoke from the hood, but there is change in the accumulated smoke inside the hood. After the fire load is added until it reaches 2 MW, the result of smoke distribution is shown in Figs. 2(c) and 3(c). There is a quite significant difference in the volume flow of 3.33 m³/s and
6.66 m$^3$/s. In Fig. 2(c), it can be noted that the smoke is quite thick on hood and it is not focused on the ducting above it. In Fig. 3(c), it can be observed that the smoke distribution is quite thick, but it is completely sucked through the exhaust fan. Figures 2(d) and 3(d) explain that the smoke distribution when the 3 MW HRR is given. The flame height rises, has a little accumulated smoke inside the hood, and also it is danger to applied in 5 x 5 m hood. The same phenomenon occurs in the Case 5 which generates the flame overgrowth and affect to the exhaust fan. For Case 4 and 5, the resulting smoke and fire could damage the exhaust fan provided inside the duct.

### 3.2 Gas temperature

It is of considerable importance to the fire protection engineer to be able to roughly predict the hot gas temperature in a fire compartment. This knowledge can be used to assess when hazardous conditions for humans will occur, when flashover may occur, when structural elements are in danger of collapsing, and the thermal feedback to fuel sources or other objects. To compare all of the gas temperature data, Figs. 4 and 5 represent the upper layer gas temperature in every condition. Upper layer means the area near the exhaust fan. We can see that in volume flow of exhaust fan 3.33 m$^3$/s, the highest gas temperature created is in the 4 MW heat release rate. In the same condition, the highest gas temperature in volume flow 6.66 m$^3$/s also in the 4 MW heat release rate. For the 4 MW heat release rate on 3.33 m$^3$/s and 6.66 m$^3$/s, the highest gas temperature are reaching 850°C and 750°C respectively. These graphs also prove that the heat release rate will affect the gas temperature in the combustion process.

### 3.3 Gas velocity

Gas velocity could influence the work of exhaust fan. The flow rate of fire highly depends on the capacity of flow from exhaust fan. For Case 1, there is a difference that is quite obvious in each volume flow. In volume flow of 3.33 m$^3$/s, the formed gas velocity is around 10.5 – 14.7 m/s, while for volume flow of 6.66 m$^3$/s, the velocity reaches 16 – 28 m/s.

There is a comparison between 3.33 m$^3$/s and 6.66 m$^3$/s, the flow of gas after suction process of exhaust fan in 3.33 m$^3$/s is slower than the 6.66 m$^3$/s volume flow. It means that the highest volume flow of the exhaust fan, the higher flow rate of gas extracted. There is no significant difference between Case 1 and Case 2 in terms of gas velocity. However, in Case 3, there is a decrease of gas velocity. The decrease was caused by the excessive smoke, so the flow capacity from exhaust fan must work harder to suck the formed smoke. For the extreme condition, Case 4 and 5, the velocity created by the gas are dropped into about 11.6 – 13.2 m/s because of stacking phenomenon. From the result above, it could be inferred that the higher fire load would affect the gas velocity inside the hood.
4. Conclusion

From the series of conducted simulation, it can be concluded that there are many connection between fire load and volume flow of exhaust fan. The maximum value of heat release rate that is applied into the hood is about 2 MW. Smoke distribution in the hood is safe because it still could be extracted by both 3.33 m$^3$/s and 6.66 m$^3$/s exhaust fan. The gas temperature must be concerned because it would affect the work of exhaust fan. Lastly, to maintain the gas velocity, the smoke stacking phenomenon would influence the gas flow rate in the hood.

Acknowledgements

The authors would like to thank fellow researchers at Fire Safety Laboratory Universitas Indonesia for sincere support and advices during the experiment.

Reference