Numerical Simulation of a Ceiling Jet Fire in a Large Compartment


School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney 2052, Australia
Australian Nuclear Science and Technology Organisation (ANSTO), PMB1, Menai, NSW 2234, Australia
Department of Civil and Architectural Engineering, City University of Hong Kong, Tat Chee Avenue, Kowloon Tong, Hong Kong, PRC

Abstract

The ceiling jet phenomenon of a 15 m long test hall fire with 1.5 MW was reconstructed by an in-house large eddy simulation (LES) model which incorporates fully coupled sub grid-scale (SGS) turbulence, combustion and radiation models. Fire spread of an enclosure fire is typically caused by the ceiling jet behavior when radially-outward gas motion produced by impingement of a rising fire plume on a horizontal ceiling allowing hot gas to travel along the ceiling transfer energy to the lower portion of the hall by gas radiation. The non-equilibrium combustion caused by microscopic mixing processes was modeled by a scalar dissipation conditioned combustion model coupled with SGS turbulence model. The performance of the two sub grid-scale (SGS) turbulence models, Smagorinsky SGS model and Wall Adapting Local Eddy Viscosity (WALE) model, were assessed by comparing predicted transient gas temperatures and velocities at various spatial locations.

© 2013 The Authors. Published by Elsevier Ltd. Open access under CC BY-NC-ND license. Selection and peer-review under responsibility of School of Engineering of Sun Yat-sen University

Keywords: Large Eddy Simulation, Compartment Fire, Ceiling Jet

1. Introduction

Tunnels or long corridor fires are nowadays greatly concerned in the research field of fire safety science. In tunnels or corridors, the combustion products of fire are confined to spread in one or two directions. There are 3 main areas in a fire, i) the plume, prior to impingement, ii) the impingement region; and iii) the ceiling jet region. Ceiling jet is formed when hot gases from the fire rise above the burning fuel and impinges on the ceiling. The flow spreads radially away from a stagnation point where it hits the ceiling in any direction horizontally along the ceiling. With the rapid development of Computational Fluid Dynamics (CFD) and computing technology, field modeling techniques have become the convenient way of fire studies. Motevalii[1] presented calculations based on fire field model. However, the velocity profile in the ceiling jet was taken from empirical correlation and introduced into the model as an input parameter. Chow et al [2-3] reported field study results of ceiling jet flows. However they have not been substantially validated against experimental data[4]. Motevalii[5] used the computer model LAVENT to predict heat transfer and velocity in confined ceilings. Recently, O’grady and Novozhilov[6] used large eddy simulation (LES) CFD model to predict the interaction between sprinklers with ceiling jet. The results indicated good accuracy of LES approach in application to fire design problems.

In this paper, the ceiling jet phenomenon of a 15 m long test hall fire with 1.5 MW was reconstructed by an in-house large eddy simulation (LES) model which incorporates fully coupled sub grid-scale (SGS) turbulence, combustion and...
radiation models. The non-equilibrium combustion caused by microscopic mixing processes was modeled by a scalar dissipation conditioned combustion model coupled with SGS turbulence model. The performance of the two sub grid-scale (SGS) turbulence models, Smagorinsky SGS model and Wall Adapting Local Eddy Viscosity (WALE) model, were assessed by comparing predicted transient gas temperatures and velocities at various spatial locations.

2. Description of ceiling jet in hall fire

An experimental study was carried out by Ingasson and Olsson[7] where a detail set of gas temperature and airflow results at various horizontal locations were measured. They performed a series of compartment fire test at the Swedish National Testing and Research Institute. The objective was to investigate the interaction between sprinklers and fire vents and also to provide useful data for fire field model validation of a typical ceiling jet compartment fire scenario.

The fire test was conducted in a single storey rectangular compartment of internal dimension 15 m long by 7.5 m wide with a ceiling height of 6 m. The fire test room ceiling and upper walls were constructed of 9.5 mm-thick Navilite N boards while 13 mm-thick gypsum was applied for the lower wall boards. At -7.5 m along the x-direction illustrated in Fig. 1 (a), one end of the room was partially-opened where the bottom half from height level at 0 m to 3 m was opened, while the other end of the room was fully opened at x-direction of +7.5 m. A 1 m by 1 m propane fuel burner with a heat release rate of 1.5 MW fire was placed 1 m away from the partially-opened end (i.e. at x-direction of -6.5 m). During the fire test, air was entrained mainly from the half opening end providing sufficient oxygen for combustion of fuel at the fuel bed sustaining the fire source. The hot plume produced by the fire source then rose vertically until it impinged the ceiling and diverged horizontally towards the two openings. As hot plume was filling up the internal volume, a hot gas layer was gradually formed. This hot gas layer escapes from the compartment at the full opening end while the flow was blocked at the half opening end. Hence, the dominant ceiling jet flow indicated in Fig. 1 (a) and (b) was demonstrated from the half opening towards the full opening.

Velocities and gas temperatures were measured using thermocouples and velocity measurement probes positioned on instrument trees at points K, B and A showed in the Fig. 1 (b). The measurements were taken at heights of 5.935 m, 5.85 m, 5.725 m, 5.575 m, 5.1 m and 4.5 m from the floor level. The measurement errors reported in the test were 2 K (maximum conservative error) for temperatures and less than 10% for gas velocities and heat release rates.

![Diagram](image)

(a) (b)

Fig.1. Fire and thermocouple locations of ceiling jet hall fire: (a) Plan view, (b) height levels of vertical layer measurements (extracted from [6])

3. Mathematical Formulations

Fire phenomena consist of non-interactive chemical and physical processes including turbulence, combustion and radiation. The LESF3D code [8] developed on the basis of the fire field model reported by Yeoh et al. [9] applying various computational models to replicate fluid flow and ceiling jet behavior which formed the basis of the in-house CFD Fire model was adopted in this study. It utilizes LES turbulence model directly which resolves the dynamic behaviors of large eddies whilst indirectly models the SGS eddies. For combustion, the laminar flamelet model, also named as mixture fraction model is employed, assuming an infinitely fast combustion of diffusion flames through a single step chemical reaction. Laminar flamelet model offers the feasibility of incorporating detailed chemical kinetics with minimal computational costs for computing turbulent flames. Radiation is modeled by using the Discrete Ordinates Method (DOM) with the Weight Sum
of Gray Gases Model (WSGGM) for radiative absorption and emission from the combustion products. Soot is modeled according to the semi-empirical based on Moss et al [10]. The performance of LESF3D [7] have been tested on large scale compartment fires reported by Häggland et al [11] and Chow et al [9].

The mathematical formulations of the LES were based on the Favre filtering (density-weighted) of the conservation equations with the Smagorinsky SGS Turbulence [12]. The soot and radiation models from the works by Cheung et al [13] and Moss et al [10] were also adopted. Due the limited scope of the pages in this paper, these shall not be presented. In this work, the Wall Adapting Local Eddy Viscosity SGS Turbulence (WALE) model implemented by the authors was employed to compare with the performance of a more traditional Smagorinsky SGS turbulence model. The detailed formulations for these 2 SGS turbulence models implemented in the current study are described below.

3.1. Smagorinsky SGS Turbulence

The SGS momentum stress was modeled according to Smagorinsky [13]:

$$\tau_{k}\approx 2\mu T \left( B_{i} - \frac{1}{2} \tau_{ij} \right) - \frac{1}{2} \tau_{kk} \delta_{ij}$$  \hspace{1cm} (1)

where \( \tau_{ij} = \frac{1}{2} \left( \frac{\partial \omega_i}{\partial x_j} + \frac{\partial \omega_j}{\partial x_i} \right) \) and \( \tau _{kk} = 2C_T \rho \Delta \left| 2 \tilde{S}_{ij} \tilde{S}_{ij} \right|^{1/2} \).

\( \tau_{kk} \) may be ignored when \( C_T \ll C_s \) (Erlebacher et. al. [14]). The turbulent viscosity \( \mu_T \) in Equation (1) is then given by

$$\nu_T = \beta (C_s \Delta)^2 \left| \Pi \right|$$  \hspace{1cm} (2)

The sub grid length \( \Delta \) can be expressed as \( \Delta = \frac{1}{\sqrt{\Delta x \Delta y \Delta z}} \) and the Smagorinsky constant \( C_s \) is taken to be a value of 0.2 [15] while the turbulent Prandtl and all the scalar turbulent Schmidt numbers are prescribed at values of 0.3, \( \Delta x, \Delta y \) and \( \Delta z \) are edge sizes of the structured grid in x, y and z direction respectively. Erlebacher et al [14] suggested that \( \tau_{kk} \) may be ignored since \( C_T \ll C_s \). Similar to the laminar counterparts, the thermal, scalar flux vectors, soot particulate number density, and soot volume fraction due to SGS motions are also correlated according to the turbulent Prandtl and Schmidt numbers.

3.2. Wall Adapting Local Eddy Viscosity SGS Turbulence

The WALE model is a sub grid scale model based on the square of the velocity gradient tensor that accounts for behavior near the wall [16]. The WALE turbulent eddy viscosity is:

$$\nu_T = \Delta z^2 \frac{(S_{ij}^2)^{3/2}}{\left( (S_{ij}^2)^{1/2} + (S_{ij}^2)^{1/2} \right)^2}$$  \hspace{1cm} (3)

Where the sub grid length \( \Delta = \sqrt{\Delta x \Delta y \Delta z} \), \( S_{ij}^2 = \frac{1}{2} \left( \bar{g}_{ij}^2 + \bar{g}_{ij}^2 \right) - \frac{1}{3} \delta_{ij} \bar{g}_{kk}^2 \), \( \bar{g}_{ij} = \frac{\partial \omega_i}{\partial x_j} \) . The term \( \tilde{S}_{ij} \) is the rate-of-strain tensor for the resolved scale defined by

$$\tilde{S}_{ij} = \frac{1}{2} \left( \tilde{g}_{ij} + \tilde{g}_{ij} \right)$$  \hspace{1cm} (4)
The WALE constant $C_w$ is designated the value 0.325 \(^{[17]}\).

4. Numerical implementation and model parameters

An extended region of 2 m x 6 m x 7.6 m were applied on both sides of the openings as shown in Fig. 2 by shaded regions to resolve correctly and account for the air entrainment and outgoing flow structures. The total volume of the computational domain was 19 m by 6 m with a 7.5 m height. Since extended regions were applied, a blockage was placed for the half opening to block away flow from escaping the domain.

![Isometric view of the computational domain in tecplot360](image)

Fig. 2 Isometric view of the computational domain in tecplot360

The initial boundary conditions were implemented to initialize the simulation in the appropriate modeling conditions. Interior gas and surrounding temperatures were both defined at 290 K (i.e. 27°C). The fire was modeled by an inlet with fuel injection mass flow rate of 0.0323 to replicate the 1.5 MW fire. The boundary conditions details are summarized in Table 2.

Table 1 Boundary conditions for the ceiling jet study

<table>
<thead>
<tr>
<th>BOUNDARY CONDITIONS</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat release rate (MW)</td>
<td>1.5</td>
</tr>
<tr>
<td>Reference temperature (K)</td>
<td>290</td>
</tr>
<tr>
<td>Fuel type</td>
<td>Propane</td>
</tr>
<tr>
<td>Heat of combustion of fuel (kJ/g)</td>
<td>46.45</td>
</tr>
<tr>
<td>Area of fuel boundary (m2)</td>
<td>1</td>
</tr>
<tr>
<td>Mass flow rate (kg/s)</td>
<td></td>
</tr>
<tr>
<td>4.1.1. Mass flow rate (kg/s)</td>
<td>4.1.1. 0.0323</td>
</tr>
<tr>
<td>4.1.2. Volume flow rate (m3/s)</td>
<td>4.1.3. 0.01746</td>
</tr>
<tr>
<td>4.1.4. Velocity of fuel flow (m/s)</td>
<td>4.1.5. 0.01746</td>
</tr>
</tbody>
</table>

5. Grid Sensitivity Analysis

In order to determine the suitable grid cell size, three non-uniform mesh systems with overall 15 cm, 10 cm and 7.5 cm grid size were tested. A 5 cm refinement was applied at the fire region where the height of the fire refinement was determined by the theoretical flame height which is approximately 5 m high. Within the refined region, the predictions of the second order gradients of each gas species were improved thus a greater accuracy of heat release rate could be achieved. The main focus of this study was to investigate the ceiling jet phenomenon for a large compartment fire scenario. Therefore,
ceiling refinements at 1 m height from the ceiling were applied for various uniform mesh systems as summarized in Table 2. This allows the model to capture the hot gas layer produced by the ceiling jet.

Table 2 Summary of refinement grid sizes for 15 cm, 10 cm and 7.5 cm mesh size systems

<table>
<thead>
<tr>
<th>Mesh size(cm)</th>
<th>x</th>
<th>y</th>
<th>z</th>
<th>Ceiling refinement(cm)</th>
<th>Fire refinement(cm)</th>
<th>Number of grids</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>120</td>
<td>97</td>
<td>63</td>
<td>2.5</td>
<td>5</td>
<td>5.1.1.733,320</td>
</tr>
<tr>
<td>5.1.2.10</td>
<td>5.1.3.185</td>
<td>5.1.4.116</td>
<td>5.1.5.72</td>
<td>5.1.6.2</td>
<td>5.1.7.5</td>
<td>5.1.8.1,545,120</td>
</tr>
<tr>
<td>5.1.9.7.5</td>
<td>5.1.10.232</td>
<td>5.1.11.154</td>
<td>5.1.12.72</td>
<td>5.1.13.1</td>
<td>5.1.14.5</td>
<td>5.1.15.2,572,416</td>
</tr>
</tbody>
</table>

These simulations were aimed to evaluate the influence of mesh resolution on results. The Smagorinsky SGS model was employed for turbulence with turbulent Prandtl number and Schmidt number of 0.3. The simulation total time was set to 100 s, a 60 s pre-burn period in which the flame was allowed to reach a steady state and induced enough flow to develop the ceiling jet. The gas temperature and velocity results were taken after 60 s. These results were time-averaged for over 40 second. The three different non-uniform meshes were assessed and simulation results in term of heat release rate and computational time are summarized in Table 3.

Table 3 Average heat release rate and computational time

<table>
<thead>
<tr>
<th>Mesh Size(cm)</th>
<th>5.1.16. HRR( kW)</th>
<th>5.1.17. Computational time, hrs in real time per 1 s in simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1.18. 15</td>
<td>5.1.19. 1252.86</td>
<td>5.1.20. 9.6</td>
</tr>
<tr>
<td>5.1.21. 10</td>
<td>5.1.22. 1252.18</td>
<td>5.1.23. 17.5</td>
</tr>
<tr>
<td>5.1.24. 7.5</td>
<td>5.1.25. 1131.50</td>
<td>5.1.26. 26</td>
</tr>
</tbody>
</table>
The simulation results were unable to show the most effective mesh for analysis. While the 7.5 cm grid produced the best accuracy velocity, the time required for the simulations was too high to be considered. While the computational time of the 15 cm mesh takes 3 times and 5 times less than the 10 cm and 7.5 cm mesh systems respectively, the 15 cm mesh was considered the most effective mesh system and was adopted in this study.

6. Results and Discussions

Generally for compartment fires, a hot gas layer is introduced by the ceiling jet effect and the gas temperature at this region is significantly greater than room temperature. Moreover, soot particulates generated through incomplete combustion travels along with the hot gas layer. Soot particles within the hot gas layer will act as a heat transfer medium where it absorbs and emits thermal energy via radiation. Hence, coupling of combustion, soot and radiation models is essential when considering a ceiling jet fire scenario. The hot gas layer prediction by LESF3D model along the long hallway are demonstrated by vertical gas temperature (Fig. 4a to 4c) and u-velocity (Fig. 5a to 5c) tree results at locations K, B and A respectively which are 1.5 m, 9.5 m and 11.5 m, respectively, away from the fuel bed towards the full opening respectively. All graphical results in the following section are presented by comparing the numerical results of the two examining SGS turbulence models against experimental measurements.

Fig. 4a to 4c depict the vertical gas temperatures from 6 m to 4.4 m height at locations K, B and A respectively. Heat energy aggregates at the ceiling due to the rising hot plume by the fire. The gas temperature measured reached its peak value near the wall and decreased slightly along with the height and the hot gas layer height was about 1 m. Over time, the hot gas layer became thicker as it descended downwards, increasing in volume as well as in temperature. In general, the overall trends of the predictions agree well with the experimental results. The temperature predictions by the Wale SGS turbulence model agreed better than the smagorinsky model to the measured thermocouple results. Simulation results near the fire compare better with the experiment than those further away from the fire.

One of the several reasons for large deviation was due to the under predicted heat release rates as mentioned previously. This might be due to the limitations of the fast chemistry combustion model or inadequate resolution of the fire region. Besides, other sources for the discrepancies might be attributed to wall numerical treatments. The change in mesh sizing for the non-uniform mesh between refinement regions (i.e. change in meshes increments $\Delta x$, $\Delta y$, and $\Delta z$) might have caused oddly shaped cells in less important area or abnormal aspect ratios thus might be a source of error. Furthermore, the
absorption and emission coefficient for radiation at the walls were set as 1 implying perfect black body. However, this assumption might not be applicable to the wall materials used in the experiments.

Fig. 4 Gas temperature predictions for SGS models against experiment at measurement (a) tree K, (b) tree B and (c) tree A, respectively.

Fig. 5a to 5c depicts the vertical gas temperatures from 6 m to 4.4 m height at locations K, B and A respectively. At location K, the u-velocity compares well against the experiment for the two SGS turbulence models. The peak velocity deviations for the two SGS models are less than 0.5 ms\(^{-1}\). Nevertheless, the predicted ceiling velocity for locations B and A is under-predicted owing to similar model limitations mentioned in the previous section.

With regards to thermocouple and airflow probes at A, B, and K, the relative error from simulation to experimental values varied from 21 % to 22.45 % for temperature predictions, and from 10% to 196% for velocity predictions, over the height of the measurement tree. All of the temperatures are under-predicted close to the fire and over-predicted away from the fire (measurement trees A and B). Velocities are generally under-predicted away from the fire at measurement trees A and B and mixed under- and over-predictions at location K. The results suggested that the WALE SGS turbulence model was more superior in modeling the ceiling jet regions than Smagorinsky model.
Fig. 5 Velocity predictions for SGS models against experiment at measurement (a) tree K, (b) tree B and (c) tree A, respectively.

Fig. 6 gives the gas temperature contours indicating the development of ceiling hot gas layer. Initially, the hot gas travelled along the ceiling towards the full opening at around 6 s of the simulation. A transition period around 20 s to 40 s was observed where the hot gas aggregated at the ceiling and the layer slowly descended resulting in shrinking of the cold air region. The hot gas layer height reached a steady-state condition at about 60 s as the hot gas outflow rate is approximately equivalent to the generation rate at the fuel bed. A fluctuation of the fire source can also be observed from various changes of fire structures (i.e. approximately demonstrated by highest temperature legend level of 390 K). This further proves the flameless combustion model is able to replicate the randomized fluctuation behavior of fire.

Fig. 7 illustrates a three-dimensional iso-surface gas temperature plot utilizing two surface temperature levels of 700 °C (i.e. 973 K) and 50 °C (i.e. 323 K) respectively. Typical visible flame is at around 700 °C hence the first iso-surface level at 700 °C (i.e. 973 K) elucidates the shape of the developing fire during the simulation at incremental time instances. The three-dimensional shape of the fire was frequently changing owing to the fluctuation term in combustion. The hot gas layer is represented by 323 K or 50 °C temperature level. It is observed that this layer reached the fully-open end and became constant in height after 60s where the structure of iso-surface maintained a similar shape.

\[ t = 6.492 \text{ s} \quad \text{and} \quad t = 19.9831 \text{ s} \]

\[ t = 40.3818 \text{ s} \quad \text{and} \quad t = 62.6198 \text{ s} \]

\[ t = 88.1349 \text{ s} \quad \text{and} \quad t = 109.223 \text{ s} \]

Fig. 6 Gas temperature contours over time cutting at the mid-plane at \( z = 3.75 \text{ m} \)

\[ t = 6.492 \text{ s} \quad \text{and} \quad t = 19.9831 \text{ s} \]
7. Conclusions

In enclosure fires, the ceiling jet region controls the smoke filling rate and determines the level of fire hazard. An investigation had been conducted to determine how accurately the boundary layer flow can be modeled by using Large Eddy Simulation. The model includes a fast chemistry combustion model, radiation from combustion products and soot. In these simulation studies, two different sub grid scale turbulence models, i.e. Smagorinsky, WALE approach were examined. The simulation results with 1.5M.W fire in a hall were compared with experimental results from full-scale fire test reported for the validation of the numerical models. Comparison between different LES models on ceiling jet flows has not been attempted.

The grid sensitivity analysis was performed. A moderate sensitivity was found on the grid refinements on the height of the model. Refinements on the y-axis have showed more accurate predictions on the flame as well as temperature and velocity predictions. Results reported included the gas temperatures and tangential flow velocity in the ceiling jet. The average prediction error from WALE model ranged from 15 to 30% while the Smagorinsky model ranged from 20 to 50%. These comparisons may suggest that the WALE SGS turbulence model was more superior in modeling the ceiling jet regions than Smagorinsky model. However, there are still rooms for further improvements of the WALE model for closer agreement with the experimental results. Also, experimental study which may provide more good quality data for validation of numerical model should be encouraged.

Acknowledgements

The authors acknowledge the full financial support of this research work by the Research Grant City U 122612 from the Research Grant Council, HKSAR.

References


