The 22nd Conference of Mechanical Engineering Network of Thailand
15-17 October 2008, Thammasat University, Rangsit Campus, Pathum Thani, Thailand

CFD study of flow in a natural rubber sheet smoking-cooperative: Turbulence free convection airflow

L. P. Purba[1*], P. Tekasakul[1], K. Maliwan[1], and M. Furuuchi[2]

[1] Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, Hat Yai, Songkhla 90112, Thailand
Tel: 0-6674-212893, Fax: 0-6674-212893, *E-mail: perapong.t@psu.ac.th, s4910120113@psu.ac.th
[2] Graduate School of Natural Science and Technology, Kanazawa University, Kakuma-machi, Kanazawa, 920-1192, Japan
E-mail: mfuru@t.kanazawa-u.ac.jp

Abstract

A computational fluid dynamics (CFD) method was used to investigate the velocity and temperature distribution in a natural rubber sheet smoking cooperative (RSSC). The measured velocity and temperature data at various locations were used as boundary conditions and to validate the CFD model. Simulation was performed using turbulent free convection flows where the Rayleigh number was found to be between 5.3838 x 10^{10} and 33.2003 x 10^{10}. A total of 601,999 mesh volumes were applied to the entire RSSC. This was obtained when no error on GAMBIT was present.. It was found that the results from the CFD simulation and experiment are in good agreement. The air contains smoke particles flows naturally from ventilating lids of the smoking room to the roof. Experimental and simulation results show that the thick cloud of smokes has a long residence time in the roof area. The smoke particles follow the airflow fields where some of them leave the junction of the roof and the others deposit onto the walls. The smoke particles that leave the junction of the roof then travel to the workplace areas which then make the workers feel uncomfortable and irritable. Moreover, since smoke particles contain hazardous chemical compounds, proper ventilation of the smoke inside the cooperative is necessary.

Keywords: Computational fluid dynamics, Flow simulation, Aerosol concentration, Free convection, Rubber smoking-cooperative

1. Introduction

Thailand is the largest natural rubber producer in the world [1]. One of the main products is the ribbed smoked sheet (RSS) rubber. Currently, production of RSS is shifted from large factories to small-scale community-based rubber sheet smoking cooperatives (RSSC). There are totally about 500 RSSC undergoing business in Thailand. In general, there are two models of RSSC, models 1994 and 1995. These two models differ in the size of smoking rooms and, hence, the capacity of the rubber sheet production. For model 1995, the size of the rubber sheet smoking room is 5.0 m (width) x 4.0 m (height) x 6.0 m (depth) doubling of that for model 1994. In the RSS production, rubber-wood is burned to supply heat and smoke to the rubber sheets [2].

Burning of woods results in a large portion of fine smoke aerosol particles. This is potentially harmful to workers in the factories as part of the particles is allowed to flow into the workplace area in the RSSC (Fig. 1). Improvement of the airflow is then necessary to reduce the risk to workers' health by exposing to these smoke particles. No particle concentration data have been obtained, however. Moreover, no studies of airflow inside the workplace area have been conducted so far. In this paper, velocity and temperature distributions inside of the RSSC as well as smoke particle trajectories have been investigated to determine the proper ventilation of these particles from the workplace area.

2. Theory

The numerical solution of fluid flow, and other related processes can begin with the laws governing these processes expressed in mathematical forms, generally in terms of differential equations [3]. From fluid dynamics, for the steady-state incompressible flow, the continuity equation can be written as:

$$\nabla \cdot (\rho \mathbf{u}) = 0 \qquad (1)$$

The differential equation for the conservation of momentum in a given direction takes the following form:

$$\nabla \cdot (\rho \mathbf{u} u) = \nabla \cdot (\mu \nabla \mathbf{u}) - \nabla p \qquad (2)$$

where: μ is the viscosity [kg/(m.s)], $\mathbf{u}$ is the denoting the x,y,z-direction velocity [m/s], and p is the pressure [Pa].

The energy equation can be written as:

$$\rho C_P \frac{DT}{Dt} = k\nabla^2 T + \mu\theta \qquad (3)$$

where:

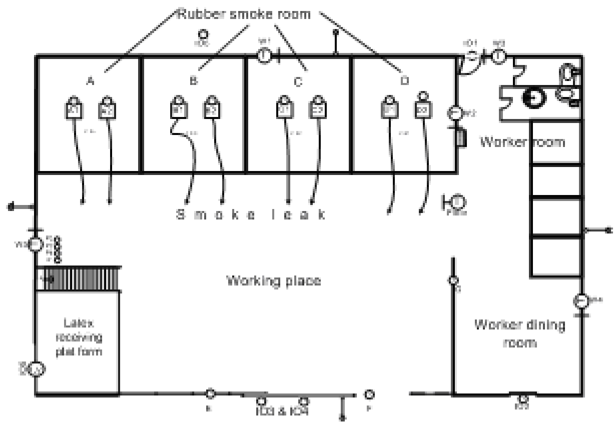

Figure 1. Diagram of measurement positions at RSSC (Top view).

$$\theta = 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] + \left[\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)^2\right] - \frac{2}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)^2 \quad (4)$$

Here $\nabla^2 T$ is the Laplacian term and k is the thermal conductivity.

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. In this work, the standard k-ε model has been used [4] because this model is one of the most widely used turbulent model. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and it bears little resemblance to its mathematically exact counterpart. In the derivation of the k-ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows.

The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (5)$$

and

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (6)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants, σ_k and σ_ε are the turbulent Prandtl numbers for k, and ε, respectively, S_k and S_ε are user-defined source terms. The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε as follows:

$$\mu_t = \rho C_\mu k^2 / \varepsilon \quad (7)$$

where C_μ is a constant. The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, σ_k, and σ_ε have the following default values [4]:

$$C_{1\varepsilon} = 1.44,\ C_{2\varepsilon} = 1.92,\ C_\mu = 0.09,\ \sigma_k = 1.0,\ \sigma_\varepsilon = 1.3 \quad (8)$$

3. Methodology

The velocity and temperature distributions, and smoke particle trajectories are obtained numerically using a commercial FLUENT CFD package version 6.2 run on a PC 2.79 GHz. The grid generation program, GAMBIT, is used to construct the grid system prior to the calculation by FLUENT. The result of meshing process of the domain RSSC geometry is shown in Table 1. A total of 601,999 mesh volumes were applied to the entire RSSC. Proper grid scheme was obtained when no error on GAMBIT was present. Velocity and temperature measured at boundaries are used as boundary conditions, while measured data at various positions inside RSSC are used to validate the simulation results.

To consider whether the flow is mixed or natural (free) convection, calculation of ratio between Grasshoff number and Reynolds numbers should be performed. According to the real measurement, values of velocity near the walls are always zero, and about 0.01 m/s at about 10 cm from the vertical walls, and average value of 0.05 m/s for free stream flows, so the fluid flows to be modeled and solved behave as natural convection and a mixed convection (when $Gr/Re^2 >> 1.0$ the flow is mixed convection). Since the Rayleigh number $(Ra = g\beta(T_s - T_\infty)H^3/\nu\alpha)$ lies between 5.3838 x 10^{10} to 33.2003 x 10^{10} (for ΔT between 1.2 to 7.4ºC), the flow is turbulent. This paper presents solution of turbulent free convection flow modeling.

In the current simulation, appropriate meshing details are shown in Table 1. Measurement positions and the 3-D diagram of the RSSC are shown in Figs. 1 and 2, respectively. The smoke flow naturally from ventilating lids into anywhere inside the cooperative: worker residential area, worker room, working place, junction of the roof, six large openings around the factory (IO1 – IO6), latex receiving platform, and walls. The smoke flow (airflow contain smoke particles) is the flow under investigation. Values of boundary conditions are shown in Table 2.

Table 1. Result of volume meshing process

Elements	Tet/Hybrid
Type	TGrid
Size	0.1 (m)
Mesh volumes generated	601,999

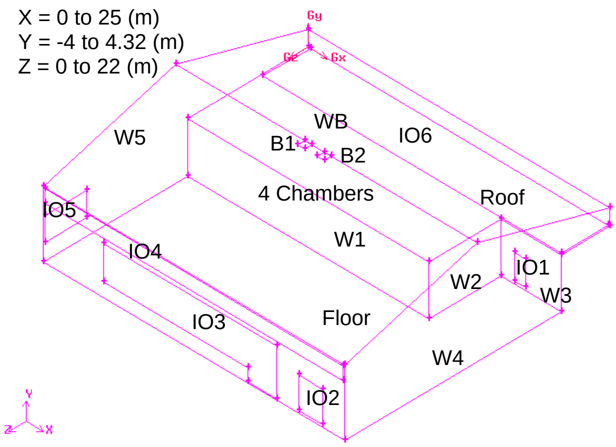

Figure 2. Schematic diagram of the RSSC indicating location of measurements.

Table 2. Boundary conditions of simulation

Inlet Boundaries		
Locations	Velocity (m/s)	Temperature (K)
B1	1.03	321.9
B2	1.02	321.7
Outlet Boundaries		
	Gauge pressure (Pa) : 0	
IO1	Temperature (K):	304.7
IO2	Temperature (K):	304.8
IO3	Temperature (K):	309.2
IO4	Temperature (K):	307.1
IO5	Temperature (K):	304.6
IO6	Temperature (K):	308.2
Solid Boundaries		
W1	Temperature (K):	320.8
W2	Temperature (K):	317.3
W3	Temperature (K):	307.1
W4	Temperature (K):	315.4
W5	Temperature (K):	305.3
WB	Temperature (K):	309.6
Roof	Temperature (K):	313
Floor	Temperature (K):	305.7

4. Results and Discussion

Results from simulation are shown in Figs. 3 to 5. The velocity and temperature contours at the plane cut across the center of the ventilating lid B1 are shown in Figs 3 and 4, respectively. Results show that hot air from the ventilating lid flows to the roof of the cooperative due to buoyant force causing the air near the roof area hotter than the lower portion. Maximum values of temperature and velocity in this area are 48.4^{o}C and 1.03 m/s, respectively. The lowest velocity of airflow containing smoke particles is located in the middle of the RSSC, indicated by dark blue color. We can see that the flow comes from IO4 to IO3 near the boundary at the right hand side of Fig. 3. From Fig. 4 we can see sample of the temperature's streamlines, temperature distributions in the area. The lowest values of temperature distribution at the plane across the center of ventilating lid B1 is around large opening of IO6 (bottom left hand side of Fig 4, located between the roof and ceiling of 4 chambers of the rubber smoke room), and IO4 (located between roof and IO3), and IO3 (right hand side of Fig.4, located between IO4 and the floor).

The trajectory of particles released from surface of the ventilating lid is shown in Fig. 6. The particles were released normal to boundaries and will be reflected if particles collide with the walls. There are 18 sampled particles released, 9 particles from B1 and 9 particles from B2. The particle diameter is uniform of 1.0 micrometer. Initially, every particle velocity is 1.025 m/s, temperature is 321.8 K, and mass flow rate is 1.5416 x 10^{-6} kg/s. It can be seen that some of the particles seem to be congested near the roof area before vented out of the cooperative. About 6 particles are escaped and 12 particles are incomplete (particles still inside of the RSSC for a given specific number of step of simulation): 2 vented out (escaped) via IO1, 1 via IO2, and 3 via IO5. The smoke particles follow the airflow fields where some of them leave the junction of the roof and some of them deposit onto the walls.

Comparison of velocity and temperature between simulation and experiment are shown in Tables 3 and 4 respectively. Error of velocity between measurement and simulation at location P is high because sometimes there are many workers and peoples came to deliver their rubber latex at the IO5 near point P. In general, results show a good agreement between the measurement and simulation values indicating a proper simulation scheme.

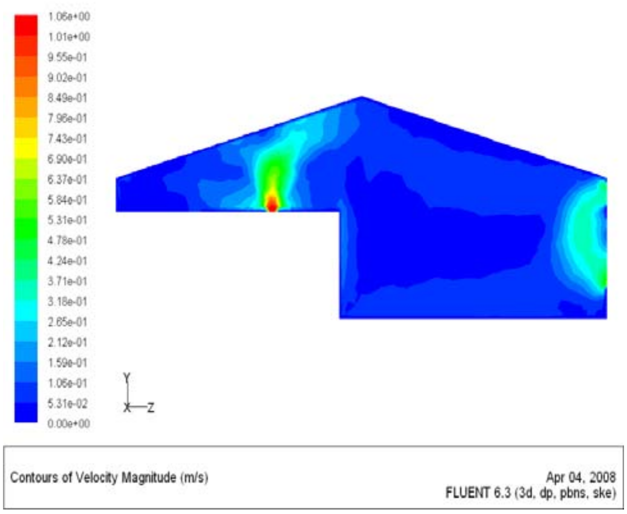

Figure 3. Contour of velocity magnitude at the plane cut across the center of B1.

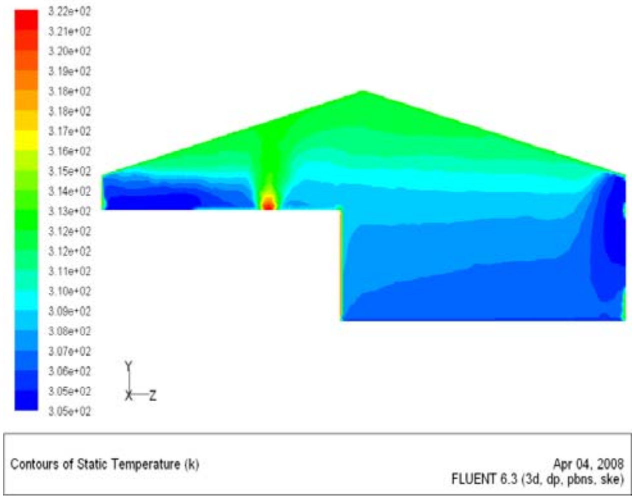

Figure 4. Contour of temperature magnitude at the plane cut across the center of B1.

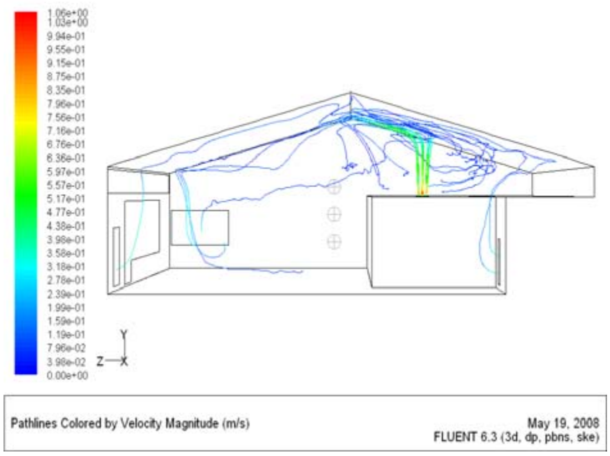

Figure 5. Trajectories of smoke particles with diameter = 1 micron released from B1 and B2 (average traveling distance of about 200 m).

Table 3. Comparison between measurement and simulation results of velocity.

Parameters	Velocity (m/s)		Error (%)
	Mea	Sim	
P	0.14	0.08	42.86
G	0.02	0.02	0.00
B1	1.03	1.03	0.00
B2	1.02	1.02	0.00
IO1	0.79	0.66	16.46
IO2	0.49	0.50	2.04
IO3	0.32	0.27	15.79
IO4	0.35	0.35	0.00
IO5	0.35	0.40	14.29
IO6	0.004	0.004	0.00

Table 4. Comparison between measurement and simulation results of temperature.

Parameters	Temperature (K)		Error (%)
	Mea	Sim	
P	305.9	307.0	0.36
G	307.0	307.8	0.26
B1	321.9	321.7	0.06
B2	321.7	321.5	0.06
IO1	304.7	307.3	0.85
IO2	304.8	306.5	0.56
IO3	308.30	305.02	1.06
IO4	307.10	305.6	0.49
IO5	304.6	307.3	0.89
IO6	308.2	306.12	0.67

5. Conclusion

Hot air (smoke, aerosol particles from the sources) flows mainly from ventilating lids to the workplace area. The Discrete Phase Model of FLUENT was applied to show the trajectories of the particles. Location of the fastest velocity measurement and simulation is at the ventilating lid. Location of the highest temperature measurement and simulation is at the ventilating lid.

Airflow field of the RSSC has been investigated. Results between CFD simulation and measurement are in good agreement. Turbulent free convection model was applied to represent the airflow containing smoke particles flows inside of the RSSC. Moreover, particles trajectories of every particle suspended in a gas (air) starting from ventilating lids as a source of particles concentration to anywhere inside of the RSSC (especially to the workplaces concentration) was well-predicted.

Acknowledgment

This research project has been supported by a grant from the NRCT-JSPS Joint Research Program.

References

1. Ministry of Commerce Thailand, http://www.ops2.moc.go.th/meeting/bb.xls, accessed January 2007
2. Ministry of Commerce Thailand, http://www.ops2.moc.go.th/meeting/XcomB10b.xls, accessed January 2007
3. S.V. Patankar, 1980. Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation USA.
4. B. E. Launder and D. B. Spalding., 1972. Lectures in Mathematical Models of Turbulence. Academic Press, London, England.
5. Fluent Inc., September 2006. ANSYS FLUENT, FLUENT 6.3 Getting Started Guide, Lebanon, NH.

6. Fluent Inc., March 2006. ANSYS FLUENT, GAMBIT 2.3 Getting Started Guide, Lebanon, NH.
7. M. Promptong, P. Tekasakul, 2007. CFD study of flow in natural rubber smoking-room: I. Validation with the present smoking-room. Applied Thermal Engineering, vol. 27, pp. 2113–2121.
8. P. Tekasakul, M. Promptong, 2008. Energy efficiency enhancement of natural rubber smoking process by flow improvement using a CFD technique. Applied Energy, vol. 85, pp. 878-895.