TFM089

The 19th Conference of Mechanical Engineering Network of Thailand
19-21 October 2005, Phuket, Thailand

CFD and Experimental Study of the Temperature Distribution in a Present Rubber Smoking Room

M. Promtong*, P. Kirirat, J. Taweekun, P. Tekasakul
Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University,
Hat Yai, 90112, Thailand
*E-mail: machimontorn@hotmail.com

Abstract

Computational Fluid Dynamics (CFD) and experimental studies of the temperature distribution in a present rubber smoking room of cooperatives were carried out. FLOVENT V5.2 program was used for simulation and input conditions were obtained from measurements. Temperatures in an empty room were measured at 15 points for 64 hours to determine a near-steady period. Results from measurements were then compared with those from CFD simulation. It was found that the temperature difference is 0.12-2.43ºC for the front plane, 0.20-1.54ºC for the middle plane and 0.23-4.46ºC for the rear plane of the room. These results are used as a benchmark for future study of the smoking room in which the rubber sheets are present.

1. Introduction

Rubber sheets are an important exporting product of Thailand. Their value is about 12,500 million baths per year. Most of the ribbed smoked sheets (RSS) are produced at the entrepreneur groups in the community level. At present, about 500 rubber cooperatives are undergoing the RSS production throughout the country, particularly in Southern Thailand. The rubber smoke room is a critical component in the rubber smoking process. From previous work [1], it was found that variation of the temperature distribution in the rubber smoke room is as large as 15ºC. This resulted in a non-uniform drying of the rubber sheets, and hence, the quality of the dry sheets is affected. Improvement of the temperature and velocity distributions in the rubber smoke room is then necessary to increase uniformity of rubber sheet and efficiency of fuel usage.

Computational Fluid Dynamics (CFD) is an essential tool in solving problems involving fluid flow and heat transfer. This technique can be used to study flow and temperature characteristics in rubber smoke room. CFD involves discretization of the conservation of mass, momentum and energy. Study of literature indicated that CFD technique has been widely used by many researchers to analyze problems of fluid dynamics, both for indoor and outdoor environments.

Nielsen [2] was one of the first to apply the CFD technique for the numerical prediction of the indoor air flow. The validity of the results however remains an issue of concern given the necessary discretization and the numerical input parameters [3, 4].

However, many flows of engineering interest are turbulent. The prediction of turbulent flow needs additional terms called the Reynolds stresses that require additional equations to solve the problem. This is called turbulence modeling. There are several different kinds of turbulence models [5]. The turbulence models most often used in the built environment are variations of the standard k-epsilon ($k-\varepsilon$) model. This model is a time average representation of turbulence that is considered very robust, thus provides accurate results with low computational overhead [6].

In this research, CFD technique is used for simulation of the present empty rubber smoke room and experimental results will be benchmarked against the values obtained for the CFD technique. This will lead to the methods of the rubber smoke room improvement by the CFD technique.

2. Governing equations and turbulent modeling

The fluid flow can be described by the conservation of mass, momentum and energy. Given the boundary conditions, the resulting flow and temperature pattern are determined by solving these equations all together. The governing equations, based on the low-*Re* $k-\varepsilon$ model for natural convection flows, are given by:

Continuity equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho \overline{u_i}\right)=0 \tag{2.1}$$

Momentum equation

$$\frac{\partial \rho \overline{u_i}}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho \overline{u_j u_i}\right)=-\frac{\partial \overline{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j}+\frac{\partial \overline{u}_j}{\partial x_i}\right)-\rho\overline{u_i' u_j'}\right]$$
$$-\rho g_i \beta\left(\overline{T}-\overline{T}_{ref}\right) \tag{2.2}$$

Energy equation

$$\frac{\partial \rho \overline{T}}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho \overline{u_i T}\right)=\frac{\partial}{\partial x_i}\left[\frac{\mu}{\Pr}\frac{\partial \overline{T}}{\partial x_i}-\rho\overline{u_j' T'}\right] \tag{2.3}$$

Where, $\overline{u}_i$ is the mean velocity components (u, v, w), $\overline{u_i'}$ is the velocity fluctuation and $\overline{P}$ is the pressure. The diffusion term is indicated by viscosity μ.

Here, x_i is the coordinate axis (x, y, z), ρ is the density, g_i is the gravitational acceleration vector and β is the thermal expansion coefficient.

The Boussinesq approximation is employed in the last term of Eq. (2.2) where, $T_{ref} = \frac{1}{2} T_h + T_c$ is the reference temperature and $\overline{T}$ is the mean temperature, $\overline{T'}$ is the temperature fluctuation.

The averaging process results in new unknown terms, $-\rho \overline{u_i' u_j'}$ and $-\rho \overline{u_i' T'}$, so called Reynolds terms. The first term is called the Reynolds stress (τ_{ij}). The latter can be considered as a diffusion term for the enthalpy.

The correlation of the Reynolds terms to the mean flow field is resolved by turbulence models. One of the most widely used turbulent models is the standard $k-\varepsilon$ model [7]. The equations for the kinetic energy of turbulence (k) and its dissipation rate (ε) are given by

k Equation:

$$\rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + P + G - \rho\varepsilon \qquad (2.4)$$

ε Equation:

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_1 \frac{\varepsilon}{k}(P + C_3 G) - C_2 \rho \frac{\varepsilon^2}{k} \qquad (2.5)$$

The model constants are [7]

$$C_\mu = 0.09, C_1 = 1.44, C_2 = 1.92, C_3 = 1.0, \sigma_k = 1.0, \sigma_\varepsilon = 1.217$$

The last term in equation (2.4), $\rho\varepsilon$, is the destruction rate, and P is the shear production and G is the buoyancy production term, which are given by:

$$P = \mu_t \frac{\partial u_i}{\partial x_j}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_i}{\partial x_i}\right) \qquad (2.6)$$

$$G = g_i \beta \frac{\mu_t}{\sigma_t}\left(\frac{\partial \overline{T}}{\partial x_i}\right) \qquad (2.7)$$

Where μ_t is called the turbulent viscosity and is given by

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon} \qquad (2.8)$$

3. Experimental works

Experimental works have been carried out to obtain the temperature distribution in the rubber smoke room for a long period of time. A portion of the results when the temperature is nearly constant was selected for the steady-state case representation.

The experiments have been done with an empty room at Ban Tai - Prik Tok Rubber cooperatives, Sadao district in Songkhla province. The smoke room dimension is 2.6 x 6.2 x 3.7 m. The room floor has 12 four-inch-diameter inlet ducts which are used for hot air introduction. Two 0.6x0.6-m ventilating lids are installed at the ceiling for air outlet. An 8-inch-diameter and 8-m-long chimney is used for gas exhaust.

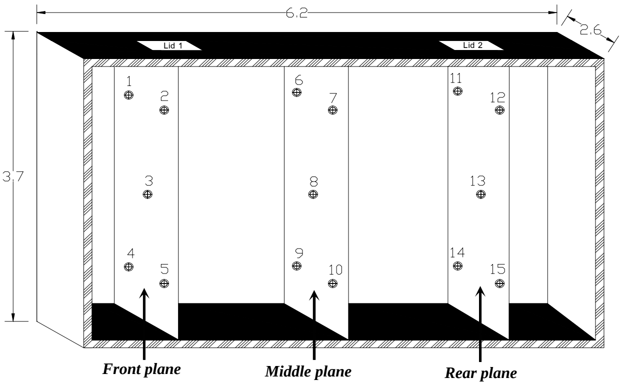

Figure 1. Positions of temperature probes on each plane shown from the side view of the rubber smoke room.

Type-K thermocouples were used for temperature measurements at 15 positions shown in Fig. 1. Moreover, ambient temperature and temperature of hot gas inside the chimney were measured. Data logger (DataTaker, DT 500) was used to record the temperatures at 2-minute interval to ensure continuous reading.

Air Velocity at burner inlet and the velocity of exhaust gas at the ventilating lids were also measured for comparison purpose. Hot-wire type anemometer (Airflow, TA400T) is used for the velocity measurement.

Fuel wood of known mass was fed to the burner to supply heat to the smoke room. Its moisture content was determined on dry basis by drying a sample in a laboratory oven at 105ºC until totally dried. Heat supply by the wood can then be determined from the heating value of the wood which is dependent on the moisture content [9].

In this experiment, the moisture content of wood and the ambient temperature are uncontrollable parameters. The steady state is only an approximation of the period that the standard deviation of the temperature is sufficiently low.

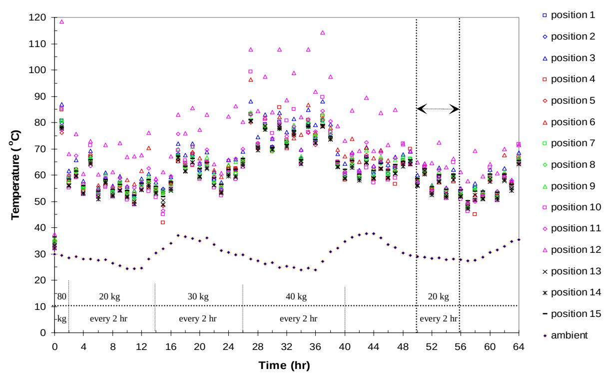

Figure 2. Temperature history at 15 positions in the room and ambient.

Results of the temperature at 15 positions in the smoke room are shown in Fig. 2. Total time for this experiment is 64 hours. The near-steady state that is used for benchmarking is the period during the hours of 50-56 hours. The rate of wood supply in this period is 20 kg per 2 hour. The temperature used in benchmarking is the average values at all positions in 6 hours.

4. Simulation

Computational Fluid Dynamics technique can be used for improvement of the temperature and velocity distributions in the rubber smoke room because it is quick, inexpensive and effective. In this work, Flovent V 5.2 that uses a finite volume method was employed for the flow and temperature study in an empty smoke room.

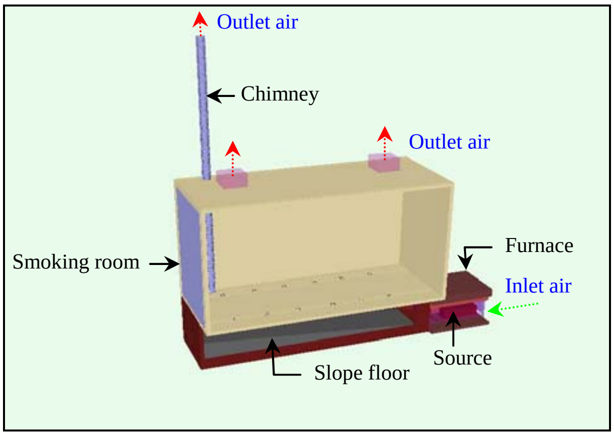

Figure 3. The rubber smoke room model.

The model of rubber smoke room is shown in Fig 3. Constant static pressure boundary condition was used to represent the system surrounding. Outside temperature was set constant at 26.7ºC. Geometry and material properties of the smoke room are shown in Table 1.

Table 1. Details of component, material, size and material property of the rubber smoke room.

Component	Material	Size	Material property		
			Thermal conductivity (W/mK)	Density (kg/m^3)	Specific heat (J/kgK)
Furnace door	Iron	0.6x0.8 m., thickness 5 mm.	80.2	7,870	447
Furnace wall	Brick & cement	1.0x1.9x1.3 m., thickness 0.25 m.	1.0/0.72	2,645/ 1,860	960/780
Wall of supply gas room	Brick & cement	2.0 x6.2x1.1 m., thickness 0.25 m.	1.0/0.72	2,645/ 1,860	960/780
Slope floor	Concrete (stone mix)	1.1x6.0 x0.5 m.	1.4	2,300	880
The smoke room floor	Cement	2.4x6.0 m., thickness 0.1 m.	0.72	1,860	780
Supply ducts	Iron	Diameter 4”, thickness 2 mm.	80.2	7,870	447
Enclosure of the smoke room	Brick	2.6x3.7x6.2 m., thickness 0.1 m.	1.0	2,645	960
Door of the smoke room	Iron	2.4x3.3 m., thickness 3 mm.	80.2	7,870	447
Chimney	Iron	Diameter 8”, long 8 m., thickness 3 mm.	80.2	7,870	447
Ceiling	Ceiling tiles	2.4x6.0 m., thickness 5 mm.	0.056	380	1,000
Ventilating lid	Ceiling tiles	0.6x0.6 m., thickness 5 mm.	0.056	380	1,000

Localize grid scheme was used for setting fine grids where high accuracy is needed and coarse grids were used elsewhere. Local grid of the smoke room has maximum size of 10 centimeters. Locations of fine grids include air inlet, heat source, air supply ducts and air outlets. The total number of grids of the current model is 559,569.

5. Results and Discussion

5.1 Heat Source

In the experiment, the moisture content of fuel wood is about 60.5% dry basis. Average heat supply rate of wood in 6 hours is 28.78 kW. However, some heat loss in the room through leaks at the front door and ceiling causes the lower actual heat supply to the rubber sheets. From iteration, it was found that a heat source of 16.0 kW is appropriate and results in minimal error between experimental and simulation results. This is corresponding to a 56% overall thermal efficiency.

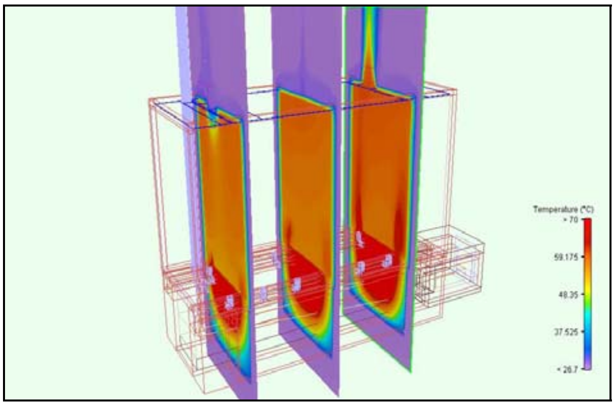

Figure 4. The temperature contours on the front, middle and back planes of the model room.

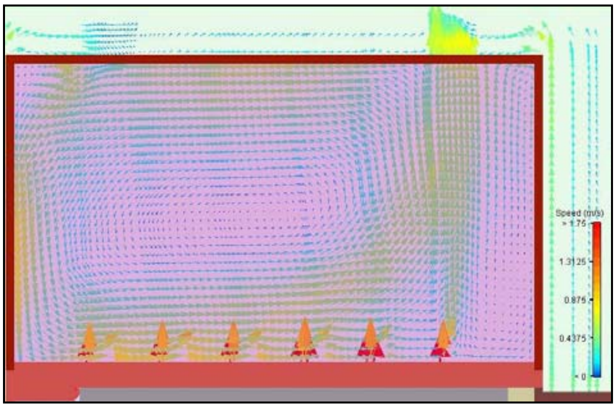

Figure 5. The velocity vector plane on the supply duct line in the model room.

5.2 Temperature

The temperature contours of the front, middle, and rear planes of the model room are shown in Fig. 4. The figure shows that the highest temperature took place at the bottom edge of each plane, since this area is near the heat source. The temperatures at the rear plane are highest where the front plane temperatures are lowest. It can also be seen from the figure that there is a back flow to the room through the ventilating lid at the front plane. This is confirmed in Fig. 5 where the velocity vectors are shown. This may result from spatial variation of velocity at hot air supply ducts. The supply ducts near the front end have lower velocity than the rear end as shown in Fig. 5. Comparisons of the results of the temperature between experiment and simulation are shown in Figs. 6-8 for each individual plane. In these figures, temperatures are plotted against the positions on each plane as indicated in Fig. 1.

Temperature difference between the experiment and simulation is 0.12-2.43ºC on the front plane (Fig. 6), 0.20-1.54ºC on the middle plane (Fig. 7) and 0.23-4.46ºC on the rear plane (Fig. 8). Highest deviation is about 4.03%. Agreement between the experimental and simulation results is quite good considering that the flow is natural and uncontrollable. Summary of the temperatures at all positions is given in Fig. 9.

Deviations at positions 4, 5 and 15 are larger than others positions because these positions are close to the hot gas inlet this results in high temperature gradient. Increase of grids near these locations would enhance the accuracy of the simulation. Part of the deviation may result from the unsteady behavior of the experiment.

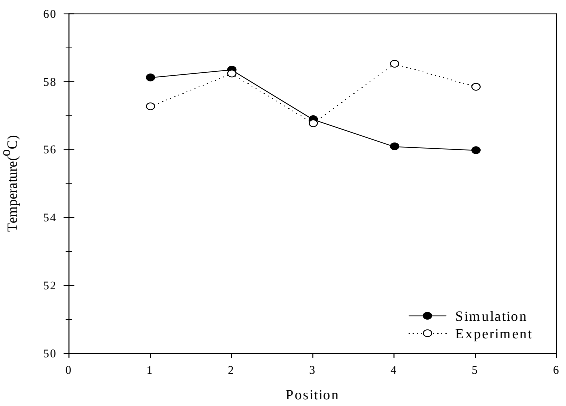

Figure 6. Comparison of temperature between the experiment and simulation at the front plane.

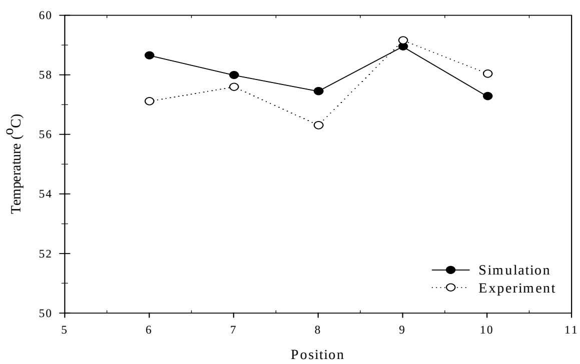

Figure 7. Comparison of temperature between the experiment and simulation at the middle plane.

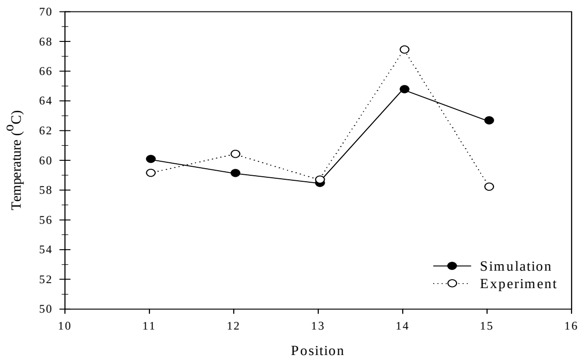

Figure 8. Comparison of temperature between the experiment and simulation at the rear plane.

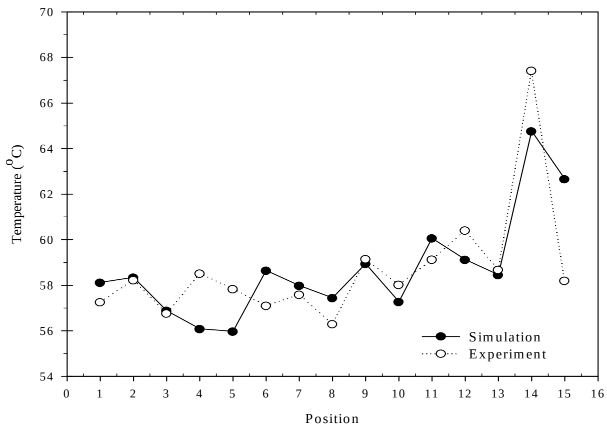

Figure 9. Comparison of temperature between the experiment and simulation at all positions.

5.3 Velocity

Comparison of velocities is shown in Table 2 for 3 positions. Agreement at all positions is quite good. The back flow at the front ventilating lid is confirmed by the experimental result.

Table 2. Comparison all positions of velocity between experiment and simulation.

Position	Experiment	Simulation
Air inlet (m/s)	2.62	2.84
Ventilating lid 1 (m/s)	-0.20	-0.22
Ventilating lid 2 (m/s)	0.62	0.63

6. Conclusion

Results from benchmarking of the velocity and temperature in the rubber smoke room indicate that the agreement between the experiment and simulation are quite good. Overall thermal efficiency for this case is about 56%. Therefore, it is possible to use the CFD technique for modeling the present rubber smoke room. Future work includes the simulation of the smoke room that contains rubber sheets. Improvement of the room can be carried out subsequently by this method as well.

7. Acknowledgments

This research was financially supported by the Graduate School, Prince of Songkla University. Thank also go to Ban Tai-Prik Tok rubber cooperative at Sadao district, Songkhla province for allowing to use the facility for the experiments.

8. References

[1] W. Kalasee, Y. Pongpariyawatee, W. Tantaprapa, S. Suwanjaras, P. Kirirat and P. Tekasakul, “Fuel Efficiency Enhancement in Rubber Smoking for Rubber Cooperative in Southern Thailand”, The 17 th Conference on Mechanical Engineering Network of Thailand, Prachinburi, Thailand, 2546.

[2] P.V. Nielsen, Flow in Air Conditioned Rooms. thesis, Nordborg, Denmark, 1974.

[3] G.E. Whittle, “Evaluation of measured and computed test case results from annex 20”, subtask 1, Proc. 12th AIVC Conference, Ottawa, Canada, 1991, pp. 27-55.

[4] Q. Chen, Computational Fluid Dynamics for HVAC: Successes and Failures. ASHRAE Transactions, vol.103 part 1, 1997, pp.178-187.

[5] J.C. Tannehill, D.A. Anderson, and R.H. Pletcher, Computational Fluid Mechanics and Heat Transfer, 2 nd ed., Taylor & Francis, London, 1997.

[6] C.W. Sohn, A. Solberg and T. Gonsoulin, Analysis of Numerical Models for Dispersion of Chemical/Biological Agents in Complex Building Environments. USA CERL report, 2005.

[7] B.E. Launder and D.B. Spalding, “The numerical computation of turbulent flows”, Comp. Meth. Appl. Mech. Energy, vol.3, 1974, pp.269-289.

[8] Q. Chen, Indoor Airflow, Air Quality and Energy Consumption of Buildings. thesis, Delft University of Technology, Delft, The Netherlands, 1988.

[9] C. Khullar, “Effects of Chemical Additives on the Reduction of Particulate Emissions and Combustion Deposits in Solid Fuel Fired Systems”, European - ASEAN Conference on Combustion of Solids and Treatment of Products, Hua Hin, Thailand, 16-17 February 1995, pp. 362-386.