AIVC 12041

NUMERICAL MODELLING OF SURFACE CONDENSATIO DIFFUSER FOR COLD AIR DISTRIBUTION SYSTEMS

S. C. Hu and J. M. Barber
School of Architecture and Building Engineering
University of Liverpool
Liverpool P.O.Box 147, L69 3BX, UK

ABSTRACT

Condensation on the surfaces of diffuser and cold air dumping are the two major concerns in the application of cold air distribution brought about by the high temperature difference between supply air and room air. Condensation will form if the surface temperature of the diffuser is lower than the dew point temperature of ambient air. The presence of surface condensation can promote growth of unhealthy and smelly mold, and produce unwelcome damage of a structural and/or aesthetic nature. Cold air dumping is a major factor that detracts from thermal comfort in an airconditioned room. In this paper, a computational study of the airflow, temperature and vapour concentration for a multi-cone circular ceiling diffuser is presented. The risk of surface condensation is investigated by combining the heat and moist transfer of room air. The results show that (1) the surface condensation is most likely to happen on the inner cone of the multi-cone diffuser, (2) the surface condensation risk increases as the supply flow rate increases, (3) the surface condensation risk can be greatly reduced by providing an opening (central hole) in the inner cone to enable sufficient cold supply air to mix with the entrained room air, (4) the existence of a lip on the cones has a very significant effect on cold air dumping.

KEYWORDS

CFD, Comfort, Air flow pattern, Jets, Air Conditioning

INTRODUCTION

Multi-cone circular diffusers are commonly used for supplying air in rooms because of their effectiveness in distributing air through the room. Usually, surfaces of air diffusers in conventional systems will not suffer condensation except in extremely conditions. For example a 'hard-start' condition at the very beginning of a cooling cycle. However, surface condensation is often seen in situations where cold air as distinct to chilled air is used due to the large temperature difference existing between supply air and the room air. Also since the temperature difference between the supply air and room air is high, the associated supply flow rate is low. Therefore danger of the cold air dumping is increased. The presence of surface condensation can promote growth of unhealthy substances, and produce unwelcome damage of a structural and/or aesthetic nature. Cold air dumping is a major factor that detracts from thermal comfort in an airconditioned room.

Isothermal performance data for multi-cone circular diffusers is normally available from the manufacturer. However, non-isothermal performance data is rarely available especially for the applications where cold air is directly delivered to the room. Shioya et al. (1996) studied the thermal comfort

aspects of a cold air distribution system using a thermal manikin. Their work indicated that multi-cone circular ceiling diffusers gave a higher degree of comfort than linear slot diffusers in a similar situation. However the cold air diffusion performance as it relates to the geometric parameters of the multi-cone circular diffuser was not given.
The performance of a multi-cone circular diffuser is difficult to analysis by experimental methods because the effective outlet area is not a constant value which depends on the flow patterns and the geometry of diffuser and room . Therefore, instead of using direct measurement methods, numerical methods are adopted.

Numerical evaluation of the room air movement provided by a multi-cone circular diffuser is problematic because the computational grid in the room tends to be rectangular and therefore not well suited for a radial type of flow. The second difficulty is that the size of a multi-cone diffuser is usually much smaller than the room resulting a very large mesh aspect ratio in the location of the interface between the ceiling and the multi-cone diffuser. Therefore simple representations of the diffuser is required even if some details of the flow near the diffuser will be lost. Numerical simulations of surface condensation need to combine the heat, vapour concentration and airflow models. Heikkinen and Piira (1994) presented and tested several simple methods to simulate the airflow in rooms using a circular ceiling diffuser. Chen (1997) indicated that a body-fitted coordinate or an unstructured grid system is more suitable than a small step cylindrical coordinate system in simulation of a complex air diffuser. Sakamoto et al. (1994) investigated the surface condensation of the metal plate under various airstream velocities and humidities. Recently, Hu, Barber and Chiang have investigated the condensation problems for the a multi-cone circular diffuser, a nozzle type diffuser and a vortex diffuser. It was found that the inner cone of a multi-cone diffuser has highest risk of condensation. To date, literature about improving the airflow diffusion performance of a multi-cone diffuser seems sparse, so this paper aims to go some way to rectifying this deficiency. This involves a study of geometric parameters using the Computational Fluid Dynamics (CFD) technique.

The objectives of this study are to (1) develop a CFD model which includes the influence of air humidity and surface condensation, for assessing the diffusion performance of a multi-cone diffuser, (2) asses the influence of the geometry of a multi-cone diffuser on its air diffusion and condensation performance, and (3) to propose a method to mitigate the problems of cold air dumping and surface condensation.

THEORETICAL BACKGROUND

Airflow Model

Most airflows in test chambers and buildings are turbulent. The turbulent model used in the study is 'standard k-ε turbulence model.' With this eddy-viscosity turbulence model, the airflows, energy and species concentration transport can be described by the following time-averaged Navier-Stokes equations:

$$\frac{\partial}{\partial t}(\rho\Phi)+\mathrm{div}\,(\rho \mathbf{V}\Phi-\Gamma_{\phi,eff}\,\mathrm{grad}\Phi)=S_{\phi} \tag{1}$$

where
ρ = air density (kg/m^3)

$\Gamma_{\phi,eff}$ = effective diffusion coefficient
V = air velocity vectors (m/s)
S = source term of the general fluid property.
ϕ = any one of 1,u,v,w,k,ε, H and C.
u,v,w = velocity components in three directions (m/s)
k = turbulence kinetic energy (m^2/s^2)
ε = dissipation rate of turbulence kinetic energy (m^2/s^3).
H = enthalpy of moisture air (J/kg)
C = mean concentration of species. The species can be the water vapour of moist air or an indoor contaminant, for example Carbon Dioxide.
when ϕ=1, the general equation changes into the continuity equation. The theoretical basis has been detailed by Rosten and Spalding (1987).

For a mixture of dry air and water vapour, the dry-bulb temperature (DB) is based on the enthalpy and the mean concentration of water vapour of moisture air (ASHARE, 1997).

$$T=(H-2501)/(1+1.805C) \qquad (3)$$

The dew point of moisture air can be obtained either from Young's equation (Glasser 1989) .
Young's equation is:

$$Td = (RH)^{1/8.02} \times (Ta + 109.8) - 109.8 \qquad (4)$$

where RH is the relative humidity (%). Surface condensation is assumed to happen when the surface temperature is lower than the dew point temperature of the air adjacent to the surface.

Boundary Conditions

The boundary conditions imposed were those being standard practice in numerical computations. At the outlet station which was placed 2 m (10 neck diameter downstream of the diffuser inlet plane, the flow was assumed to be fully-developed and temperature was assumed to be 24 °C. The standard log-law function was used for the next-to-ceiling grid points. Specifically, the heat transfer wall function of Launder and Spalding was used. In absence of turbulence measurements, uniform values for turbulence kinetic energy and its dissipation rate at the inlet boundary condition were assigned according to

$k_i = 1.5\ (0.03\ U_i)^2$
$\varepsilon_i = C_\mu\ k_i\ /\ (\ 0.09 d_i\ )$,
where k_i = inlet turbulence kinetic energy,
ε_i = inlet turbulence kinetic energy dissipation rate, C_μ = 0.09
U_i = inlet velocity (neck velocity of the diffuser)
d_i = inlet diameter (neck diameter of the diffuser)
The inlet temperature was assumed to be 8 °C.

Details of the Numerical Scheme

The multi-cone circular diffuser was assumed to be circularly symmetrical to the inlet direction of the supply air. Body-fitted grids were employed in the streamwise direction and away from the diffuser toward the room The calculation domain was extended to 2.2 m (11 diameter of the diffuser) away from the centre of the diffuser. In total, a 42 × 23 grid which accommodated both memory size limitations and tolerable numerical diffusion was chosen for final production runs, as seen in Figure 2. Staggered control volumes were used. The finite-difference from of the time-averaged transport equations were obtained by adopting a semi-integral approach to discretize the equations over each control volume of the computational grid, using a hybrid-difference scheme. The line-by-line method was used to obtain converged solutions iteratively, whereas relaxation factors and false time steps were employed to promote stability of the process. These relaxation factors were 0.3, 0.3, and 0.3 for P, k and ε, respectively. These false time steps were 2.231×10^{-3}, 2.231×10^{-3}, 2.231×10^{1} and 2.231×10^{1} for V, W, T and C, respectively. The turbulent viscosity field was also underrelaxed with a value of 0.3. The iterations were terminated when all the absolute residuals (normalized by the corresponding inlet plane fluxes) were less than 5×10^{-3}. The velocity field needed about 15,000 iterations to satisfy the convergence criterion, whereas the temperature and concentration required about only 9,000 iterations. The calculations were carried out on a PC with Pentium-Pro CPU (200 HZ).

Case set-up

Table 2 summarises the cases studied. Cases 1 to 3 aim to compare air diffusion characteristics of jets created by a multi-cone circular diffuser under various supply flow rates. The objectives of case 4 and case 5 were to determine the effect of the small hole and it's size on the risk of surface condensation of diffuser. The purpose of Case 5 was to assess the effect of the cone lip on the performance of air diffusion.

Table 2. Cases studied:

case	Q(L/s)	Lip	hole dia. (mm)
1	23.6 (50cfm)	Yes	No
2	47.3(100cfm)	Yes	No
3	70.8 (150cfm)	Yes	No
4	70.8 (150cfm)	Yes	10mm
5	70.8 (150cfm)	No	10mm

RESULTS AND DISCUSSION

(1) Airflow patterns

Figures 3a to 3e show the streamline patterns for all cases studied in the computation domain. Figures 4a to 4e show the velocity vectors and isothermal lines in the vicinity of the diffuser outlet area. The streamline patterns for cases 1, 2 and 3 are different, as shown in Figure 3a, 3b and 3c. Figure 3a and Figure 3b show that the jets separate, resulting in formation of a secondary flow forms behind the main jet's envelop. Figure 3c shows the jet has a long separation distance which can extend throughout the entire computational domain. The separation distances (X_p) of case 1, case 2 and case 3 are 0.46m ,1.67m and 1.92 m, respectively. Figure 3d shows the flow pattern of case 4 in which a small hole is added at the inner cone. In general, no significant difference in flow pattern is observed for case 3 and case 4 except in the vicinity of diffuser outlet area (see Figure 3c and Figure 3d). Figure 3e shows the flow patterns of the case 5. In this case, the multi-cone diffuser has no cone lips. Figure 3e show that incoming jet is dumping directly just under the diffuser which indicating that the influence of cone lips on air diffusion performance is very significant. In this type of case, local cold drafts are expected.

(2) Temperature Distribution

Figures 4a to 4e show the velocity vectors and temperature distributions for all cases studied. The temperature distribution was presented in term of the values of the difference between dew point temperature of air at that particular point and surface temperature of cones, defined as TDS hereafter in this paper. The cones were assumed to be made of metal with high thermal conductivity. Therefore the surface temperature of the cones were taken as the same temperature as the supply air(8^{o} C). A merit of this form of presentation is that it does not only show the temperature distribution but also indicates the surface condensation risk. It should be noted that surface condensation will occur if a value of TDS is positive.
Note that the TDS values are positive at the surface of inner cone for cases 1 to

case 3 (the values are 0.16, 0.8, and 1.02 respectively) indicating surface condensation is likely to happen in this area. The values increase as the supply flow rate increases due to more room air being entrained to the inner cone (see Figures 4a to 4c). The TDS values on the surfaces of middle and outer cones is negtive because the mixing interface of room air and cold air is below the cones. Figure 4d shows the TDS distribution of case 4 which has a hole of 10mm in diameter. Note the TDS values at most of the surface of inner cone of this case is positive except at the lower part of the cone. This is because cold air is introduced through the hole of inner cone to mix with room air so the dew point temperature in this area is reduced. It was also found that the TDS value at the surface of inner cone will decrease as the diameter of the hole is increased. The manner in which the hole affects condensation will form a future paper. However, if the hole become too big, there is a danger of cold air dumping. Figure 4e show the TDS distribution at the outlet vicinity of the multi-cone diffuser which has no cone lip. It should be noted that the TDS value at all surfaces of cones are less than zero, which is because the cold air dumps directly after discharge from the diffuser without significant induction of room air. This dumping with minimal entrainment is the notable feature of these cases.

CONCLUSION

A CFD model for assessing the risk of surface condensation and airflow performance of the multi-cone circular diffuser when supplying cold air was successfully developed. Based on the results and discussion, the following conclusions have been drawn:

1. The multi-cone circular ceiling diffuser suffers an early jet separation resulting in cold air dumping in the occupied zone when handling low supply flow rates. Much surface condensation was observed on the surface of inner cone of the diffuser. Modification of the currently available circular ceiling diffuser is required for use in cold air distribution systems. This could involve optimising the cone angles and outlet effective area.
2. The surface condensation risks increase as the supply flow rate increases.
3. The surface condensation risk can be greatly reduced by providing an opening (central hole) in the inner cone to enable sufficient cold supply air to mix with the entrained room air.
4. The influence of the cone lip on airflow diffusion is very significant.

REFERENCES

ASHRAE Handbook - Fundamentals, 1997, Atalanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Chen, Q. 1997. Computational fluid dynamics for HVAC: successes and failures, ASHRAE Transactions, 103(1), 178-187.

Glaser, H. 1989. Kaltetechnik-Keinatisierung, 20(1):6-11.

Heikkinen, J and Pira K., 1994, CFD computation of jets from circular ceiling diffuser, Proceedings of Roomvent'94, Krakow Poland 1:330-433.

Hirano Takeshi ,1997, Measurement of the airflow velocity and turbulence energy around anemostat type diffuser to verify CFD results, *Proceedings of Technical Meeting, SHASE-Japan*,(B-26):510-513 (in Japanese).

Hu, S. C., Barber, J. M. Barber and Chiang, H, 1998,Comparision of room air motion and thermal comfort in a full scale environment chamber using different air diffusers with cold air distribution systems, ASHRAE Transaction (in press)

Kondo, T., 1995, Experimental study of indoor environment in cold air delivery rooms - studies of cold air delivery systems ,part 3, *Proceedings of Technical Meeting, SHASE-Japan*,Hiroshima, **(1):**357-360 (in Japanese).

Rosten, H. I. And Spalding, D. B.(1987): The Phoenics reference manual: TR 300, CHAM Ltd. London UK.

Sakamoto, Takeshi, 1994, On the experiment of condensation on the cooled metal plates - Studies on cold air delivery systems (part 1), *Proceedings of Technical Meeting, SHASE-Japan*, (D-2) 209-212 (in Japanese).

Shiyoa, M. 1996, Thermal comfort aspects of cold air distribution systems, *ASHRAE Transactions,* **3885:**61-72.

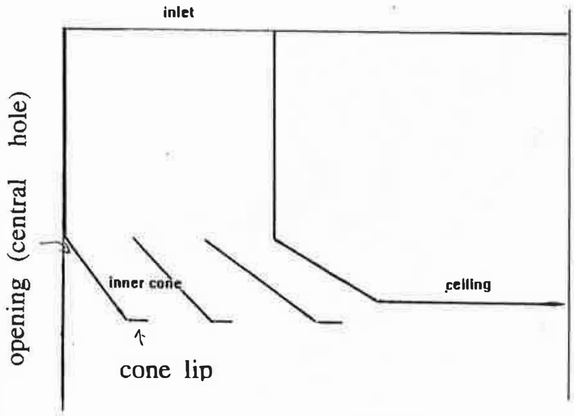


Figure 1 The multi-cone circular diffuser

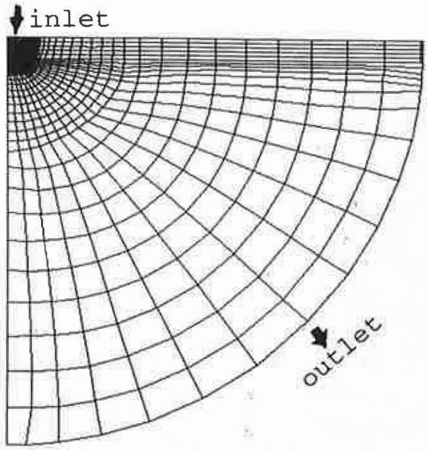


Figure 2 The mesh system.

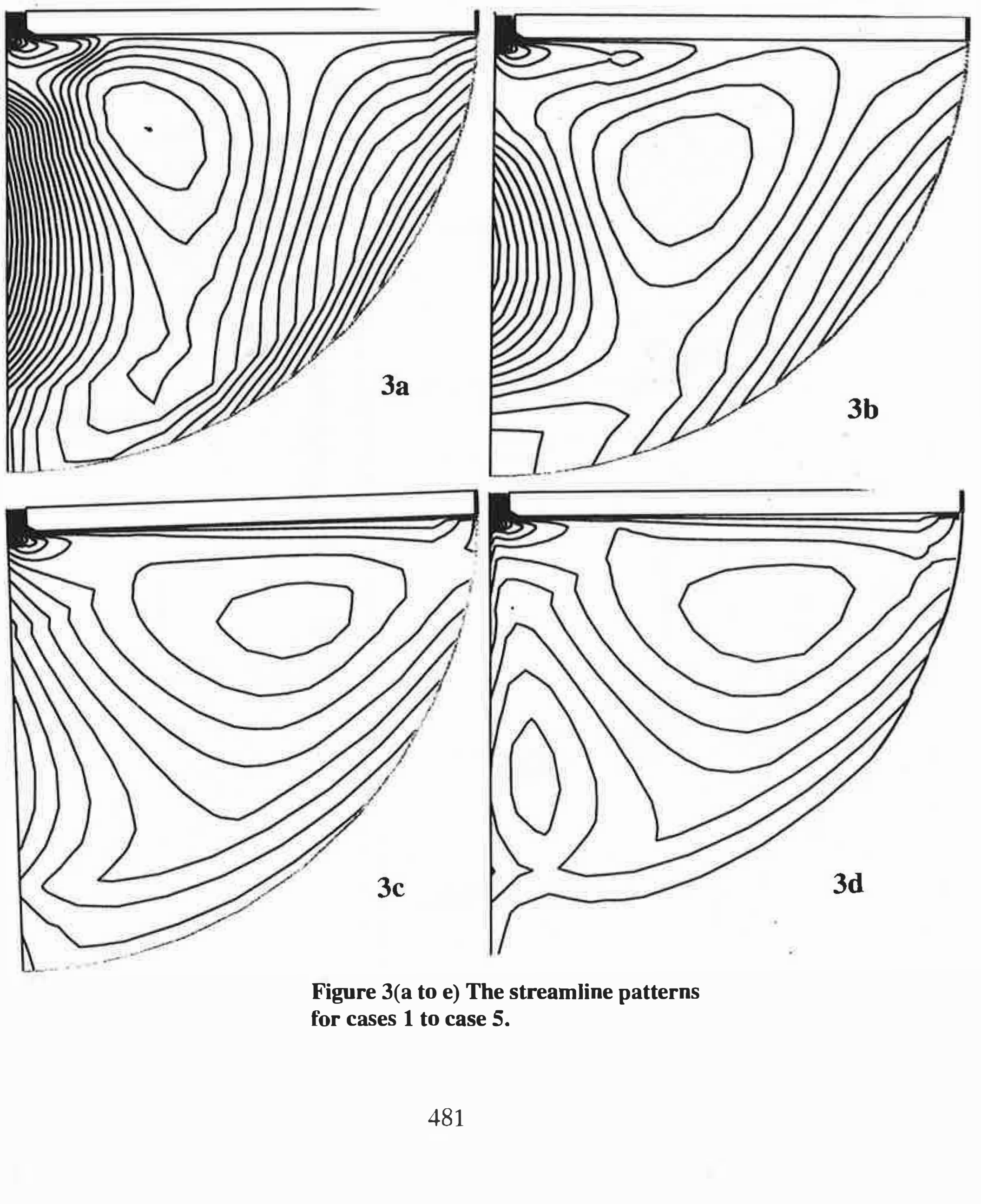
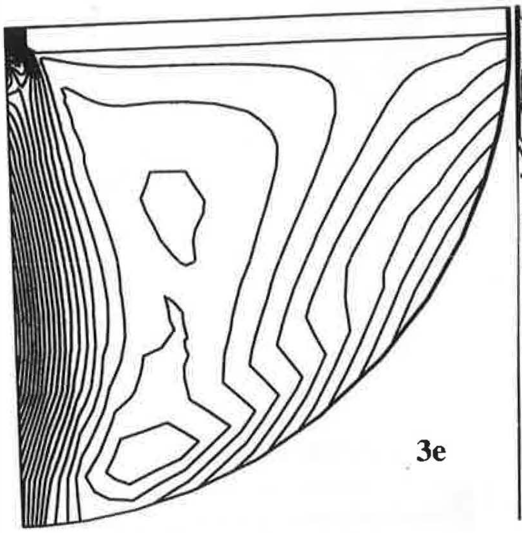
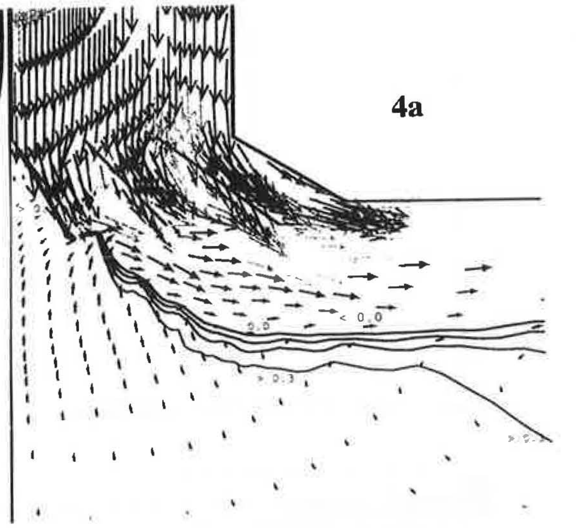


Figure 3(a to e) The streamline patterns for cases 1 to case 5.





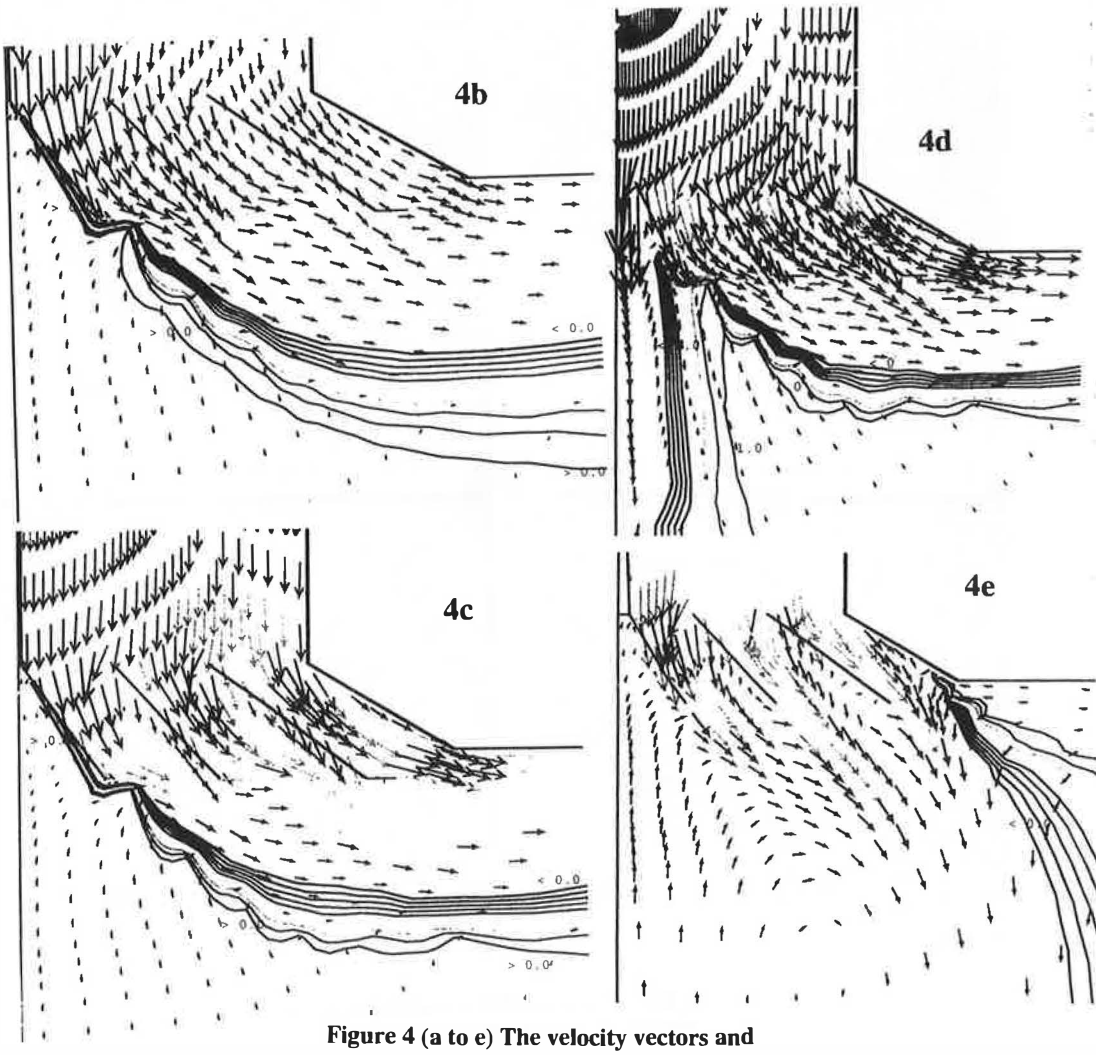


Figure 4 (a to e) The velocity vectors and temperature distributions for case 1 to case 5.