

# **SMOKE HAZARD ASSESSMENT USING COMPUTATIONAL FLUID DYNAMICS (CFD) MODELLING**

**Baldev S Kandola and Mark Morris**

AEA Consultancy Services (SRD), Thomson House, Risley, Warrington, Cheshire WA3 6AT

*Fire is a potential hazard in all buildings; industrial and residential. In both cases the fire generated heat and smoke may lead to loss of life or damage to equipment and structures. The control of fire risks requires the ability to assess them. Fire and smoke movement computer models have been under development throughout the world for a number of years, particularly since the advent of inexpensive means of automating large calculations. However, the behaviour of fire in buildings is complex and its reliable prediction poses a difficult problem. Smoke is of particular interest in complex buildings because it is pervasive and may cause deleterious effects distant from a fire. Although the possibility that a fire may occur at certain critical locations is remote, smoke from distant fires may still affect the performance of people and the operation of safety related equipment in these places. Additionally, its movement can indirectly play a role in the ventilation and hence growth of the fire.*

In the wake of such disasters as the Piper Alpha and the Kings Cross fire, there has been considerable research and development activity to predict the behaviour of smoke movement and assess its consequences. This paper discusses the results from some of the studies that have been carried out within AEA using the AEA Technology's own CFD code CFDS-FLOW3D.

Key Words: Fire, Smoke, Modelling, Computational, Offshore, Wind

## INTRODUCTION

Fire is a complex physical phenomenon occurring in nature encompassing all disciplines of scientific investigation such as heat transfer, chemistry, combustion, toxicology and fluid mechanics, to mention a few. Thermal hazards from fires are considered to be localised i.e. they have implications for structural damage or human injuries in the immediate vicinity of a fire. Smoke, on the other hand, has the potential to spread to areas remote from the seat of fire threatening life or damage to safety related equipment. Smoke, generated from most common fires usually consists of carbon monoxide (CO) and carbon dioxide (CO<sub>2</sub>), and is recognised as the biggest single life threatening hazard in building fires.

The control of fire risks requires the ability to assess them. In the wake of such fire disasters as the Kings Cross Tube Fire and the offshore Piper Alpha disaster, considerable research and development has been in progress to develop computer based predictive tools for the assessment of these hazards, in particular the spread of smoke in buildings and structures.

In general, fire induced flow is determined by a number of main factors including: fuel

geometry, fuel heat release characteristics, building leakage and the atmospheric environment (e.g. wind and stratification). In any real fire situation there is a considerable amount of randomness in each of these factors.

In an emergency situation involving fire, one of the main requirements is to provide life support for a set period until such time that a complete evacuation can be carried out. For problems of evacuation the spread of smoke is an important parameter. In large buildings the longer the evacuation time, the higher the chance for smoke to flow through escape routes. In building design it is therefore essential to optimise escape route design and to provide measures (such as the smoke control systems) to keep them free of smoke at least until evacuation is completed.

The movement of smoke within a building or structure is a complicated fluid dynamic process. The complexities are primarily due to the buoyancy of the smoke and interaction between the smoke and building leakage characteristics such as doors and windows. In addition atmospheric wind and heating and ventilation systems (HVAC) influence the smoke spread significantly.

Basically, two types of fire computer models are available: **probabilistic** and **deterministic**. The former attempt to study a range of possible fire developments while the latter allow for a single development and study the outcome in greater detail. The popularity of the deterministic models lies in the fact that they play an important role in design and are used as predictive tools. The deterministic models are further sub-divided into *zone-models* and *field-models*. This classification reflects the extent of idealisation involved and the method of solution adopted for the governing conservation equations. The zone models are semi-empirical in that they rely heavily on empirical correlations derived from laboratory scale experiments to describe the physical phenomenon involved. The field models, on the other hand, rely more on the mathematically rigorous approach and try to predict the underlying physical phenomenon.

Historically, zone models have evolved from their applications to domestic type buildings in which the fire generated flows in a single room (the fire room) are described in terms of two distinct well mixed layers (hot ceiling layer and the relatively cooler lower layer). The temperatures and the species concentration of these layers are calculated by solving the conservation equations for the layers. The field modelling approach is one in which the domain of interest (e.g. the fire room) is divided into a number of discrete cells (usually thousands in number) and the conservation equations solved for each cell to obtain the field variables such as the fluid density, species concentrations and temperature. Field models (also referred to as Computational Fluid Dynamics (CFD) models), therefore, are capable of resolving flow details in greater detail than is possible for zone models.

In this paper results of a number of smoke movement studies carried out under the AEA's research and development programme are presented. The CFD code used for this analysis is the CFDS-FLOW3D, which has been under development for a number of years. The code is very widely used and validated in many industrial applications. It incorporates a range of sub-models for turbulence and combustion.

The applications to smoke movement discussed in this paper include the smoke ingress analysis for offshore platforms, smoke movement in large single cell buildings (e.g. warehouses and aircraft hangers), and the tubular structures such as the aircraft fuselage or a submarine hull.

## OFFSHORE SMOKE INGRESS APPLICATION

In an offshore oil and gas production environment where large quantities of hydrocarbon fuels are handled, fire and smoke hazards are a serious problem. Under emergency situations it is required to maintain the breathability of air inside a temporary refuge (TR) or the accommodation quarters until such time that the escape and evacuation (E&E) is complete (usually up to 1-2 hours). In addition, it is required that smoke should not hinder, by way of reduced visibility, full and safe evacuation of the installation.

To minimise these effects and to satisfy the above mentioned requirements it is necessary to assess the permeability of the building envelope for smoke ingress so that the build up of toxicity and the reduction in visibility are kept well within the prescribed safe limits.

The flow of outside smoke laden air into a leaky building is a complex phenomenon. It is determined not only by the leakage characteristics (gaps around doors and windows) of the building but, most importantly, by the driving forces arising from buoyancy, atmospheric wind and the air conditioning system (HVAC). These factors are difficult to quantify accurately because of the uncertainties associated with their measurement. The leakage of a building for example is subject to wide variations due to workmanship, daily usage and weathering. The stochastic nature of wind and its interaction with the building structure (secondary flows) further adds to these uncertainties. The properties of smoke and its dispersion and dilution in the atmosphere are the other complicating factors making the task of assessment difficult.

Provided the properties of smoke outside a given building (such as a TR) are known, the build up of smoke concentration inside an enclosure can easily be calculated from the ingress rate. The mass rate of flow of air or gas into a given building (ingress rate) is determined, to a very large extent, by the wind induced pressure distribution on the building surfaces. In the past this information has been only possible from the wind tunnel test. However, in recent years CFD codes have proved to be very successful in predicting these pressures. Within AEA the CFDS-FLOW3D has been successfully used in the smoke ingress work. Here we present some results of a validation study involving comparison of wind tunnel and predicted results. The results for two wind directions are shown in Fig 2.1 and Fig 2.2.

### Calculation Details and Computational Grid

The flow around the cube was highly turbulent with the measured turbulence intensity around 20%. The calculations were therefore made for the steady-state, incompressible, high Reynolds number flow around the cube. The  $k-\epsilon$  turbulence model was used together with log-law wall functions to model turbulence. It should be noted that the log-law function does not take pressure gradient effects into account, and is therefore likely to be only very approximate in regions of flow separations and re-attachment.

The computational region measuring 3m x 3m x 1.5m covers most of the tunnel working section as shown in Fig 2.1. Inlet boundary conditions were specified on two sides of the domain while the other two sides were assigned the constant pressure boundary conditions. The cube flow was modelled as a wall with no slip boundary condition. The top was also modelled as a wall but with a no stress boundary condition which was used to minimise the effect of the

upper boundary on the flow.

The computational grid was generated by splitting the flow domain into 5 blocks with each block containing 8640 cells, Fig 2.2(a). A section through the grid is shown in Fig 2.2 (b), which shows that the cells were concentrated around the faces of the block.

### Boundary Conditions

At the inlet the velocity was specified using the experimental fit to the data given in equation describing the velocity profile:

$$\frac{V}{V_{\max}} = \left( \frac{z}{z_{\max}} \right)^{0.32}$$

where  $V$  is the horizontal velocity and  $z$  is the vertical coordinate. The constants  $z_{\max}$  and  $V_{\max}$  are 1m and 10m/s respectively.

This velocity field was specified on the two inlet zones in such a manner that the flow could approach a face of the cube at any angle ranging from 0 to 90°. A linear fit to the measured turbulence, allowed the turbulent kinetic energy,  $k$ , of the inflowing air to be specified. The fit was:

$$k = 1.5 V^2 (\max((0.17 - 0.125 z), 0.05))^2$$

where the form of the equation has been chosen to ensure that the turbulence level is never below a reasonable minimum value. The turbulence dissipation rate,  $\epsilon$ , was set by assuming that in the bulk of the flow the turbulence spectrum was in equilibrium, so that the production rate is equal to the dissipation rate. This leads to the following simple equation for the dissipation rate:

$$\epsilon = 0.96 k V / z$$

### Some Results and Discussion

Wind Angle 0°: The wind tunnel results taken from [Ref. 3] for this case are shown in Fig 2.3(a). Because of the way wind curls around a building with sharp corners, the results show that the pressure is positive on the windward side and negative on all other sides including the roof.

The computational results for this case are shown in Fig 2.3(b). From the comparison of Fig 2.3 (a) and (b) it is clear that the main features of the flow are correctly predicted with high positive pressures on the windward face and the negative pressure on other sides. In contrast to the experimental results the computational solution is very nearly symmetric. The main difference between the two sets of results are in the region very close to the corners of the model. The calculations show steeper gradients than were picked up in the experiments. It is possible that this kind of flow behaviour exists in reality. Further detailed experiments need to be carried out to confirm this conclusion. For practical applications to smoke ingress studies

these predictions can be used with confidence.

Wind Angle 45°: To confirm the above conclusion, another set of calculations were carried out by changing the wind angle to 45°. The measured results taken from [Ref 3]) are shown in Fig 2.4(a). It is clear that because of the complex flow pattern, the pressure distribution is such that two leading walls experience positive pressure with small regions of negative pressure appearing towards the back end of the walls. As in the case of the zero wind angle the walls exposed to the wake are subjected to negative pressures.

This general feature of the flow is predicted to an acceptable degree of accuracy by the CFDS-FLOW3D, as Fig 2.4(b) shows. Once again near the corners of the model the calculations predict very steep pressure gradients, which as remarked earlier need to be confirmed by further experiments.

### FIRE IN A SINGLE CELL ENCLOSURE

In this example the application of CFD to fires in single cell enclosures is explored. These buildings may be industrial high bay warehouses, aircraft hangers, large exhibition halls or theatres. In such buildings the positioning of fire and smoke detectors or sprinkler heads is critical for early detection and fire extinguishment. If, for example, in an aircraft hanger, a fire detector (beam type) is so placed that physical obstructions block direct path from a potential fire to the detector, the detector may fail to go off early enough for the water sprinklers either to work effectively or not operate at all. Under these circumstances the damage to contents (e.g. aircraft) or building structure could be substantial. If, however the detector is of a point type it may actually be situated in a thermally dead spot for it to function properly, resulting in similar consequences of loss.

Under such circumstances, it is essential to be able to predict the temperature distribution throughout the enclosure due to a potential fire source and the given geometry. Such an information is impossible to obtain from zone models as the underlying assumption of these models is the existence of two well mixed layers of uniform temperature. Only the CFD models can generate the required information in the form of temperature contours.

To illustrate this capability two generic fire scenarios are examined:

#### Fire in Adjacent Room

Modelling Details: This example examines the mixed natural/forced convection and associated smoke movement due to injection of hot smoke laden gas into a ventilated room such as may occur if a door were to be suddenly opened into a region where a fire was taking place.

The system modelled consists of a cuboid 10m in width, 5m in height and 10m in length with two openings forming an inlet and outlet (Fig 3.1(a)). The inlet is rectangular and is 2m in width and 2m in height located in the bottom right hand corner of the left hand face of the cuboid. The outlet is also rectangular and is 2m square located in front right hand corner of the top face, representing a roof vent. The computational grid is uniform and consists of 10000 cells (20 x 25 x 20).

At the inlet a flow of air and smoke with a temperature of 500 K and velocity of  $1 \text{ ms}^{-1}$  perpendicular to the inlet is specified. The smoke concentration is specified as  $0.1 \text{ kgm}^{-3}$  at the inlet. At the outlet only the pressure ( $1.01 \times 10^5 \text{ Pa}$ ) is specified. All the other boundaries consist of impermeable adiabatic surfaces. The cuboid is filled with air with a uniform initial temperature and pressure distribution ( $1.01 \times 10^5 \text{ Pa}$ , 288 K) with zero initial smoke concentration.

The k- $\epsilon$  turbulence model was used to avoid excessive computation times and the flow was assumed to be 'weakly' compressible ie with the density variation dependent only on temperature variations and not on pressure fluctuations. The smoke was modelled as a passive scalar.

The default linear solvers were used for this problem and a time step of 0.5 seconds was chosen. A mass residual tolerance of  $1. \times 10^{-2}$  was chosen to control convergence. All CFD3D-FLow3D runs were carried out on a Sun SparcStation IPX. The simulation required about 1.25 days.

**Results :** The calculation converged fully with the above mass source tolerance being achieved. The corresponding momentum and enthalpy residuals were  $\sim 10^{-2}$  and  $\sim 10^2$  respectively.

The temperature distributions were calculated within the enclosure at time intervals of 30s and 180s. The results of temperature contours, velocity vectors and smoke concentration in a vertical plane passing through the source and the enclosure corner near the outlet, are shown in Fig 3.2 - Fig 3.3. At time 30 seconds from the start of fire (Fig 3.2a) the results show that the flow pattern results from a buoyant jet initially moving horizontally interacting with a region of re-circulation. This is confirmed by the corresponding velocity distribution plots shown in Fig 3.2b. These results also show that a hot gas layer below the ceiling is formed but with significant mixing with the relatively cooler air in the rest of the room. The existence of this mixing process clearly demonstrate the limitations of zone models in which no mixing between the layers is allowed for.

As the fire continues the hot layer quickly grows (Fig 3.3a for  $t=180\text{s}$ ) filling a substantial fraction of the room with hot gases within about three minutes from the start of fire. The results of velocity distribution (Fig 3.3b) show that the initial plume loses its buoyancy as the temperature difference between the plume and the rest of the room is reduced. The distinction between the two layers also vanishes as the internal environment is mixed by re-circulation.

### Fire Within a Room

In this case the hot gas source was moved to the centre of the enclosure. The boundary conditions and solution parameters were the same as employed in the previous example. The residuals for mass, velocity and enthalpy had similar values to those for the previous calculation indicating full convergence. The simulation required about 4 days on a Sun SparcStation IPX. Fig 3.4-3.5 a,b and c summarise the results for temperature, velocity and smoke concentration distribution at time 30 and 480 seconds respectively from the start of fire. Here the hot buoyant plume can easily be seen with strong recirculation regions on either side of the plume (Fig

3.4b). There are no hot and cold layers but a temperature gradient exists from floor to the ceiling. By 480 seconds the smoke concentration is approaching that of the source over most of the room.

### FIRE IN A CLOSED STRUCTURE

The second of these examples examine the modelling of the development of a fire and the associated movement of combustion products within a closed system of inter-connected compartments. The example is intended to represent a confined oil spray fire.

#### Modelling Details

The structure in this case essentially consists of a closed cylinder of about 10 metres in diameter and 20 metres in length split into various compartments (Fig 3.1 (b)). The cylinder is roughly divided in half by a vertical partition which contains a rectangular opening in its upper section. The left hand side compartment is divided into sections vertically by three platforms, (which do not extend to the cylindrical walls), some of which support cuboidal structures. The right hand side of the cylinder is split into three compartments by horizontal partitions. These compartments are inter-connected by means of horizontal rectangular openings of about 1 metre square. The computational grid is body fitted and uniform and consists of about 10000 cells.

Since the system is closed there are no inlet or outlet boundary conditions and all the bounding surfaces of the flow domain, (including internal partitions), are assumed to be impermeable and adiabatic. The combustible material or fuel is introduced as a volumetric source covering 4.6 m<sup>3</sup>, (located in the bottom left hand corner of the left hand compartment), via the 'user fortran' facility in CFDS-FLOW3D. The fuel production rate is equivalent to a 1.7 MW fire on complete combustion. The system is initially filled with air at atmospheric pressure.

The fire is modelled as a gaseous combusting flow using the eddy breakup model in CFDS-FLOW3D. The code tracks the distribution of the mass fractions of fuel, oxidant, products and the mixture fraction which is a measure of the ratio the mass fractions of fuel and oxidant. The flow is modelled as being fully compressible and the k- $\epsilon$  model employed to represent turbulence.

It was found that in order to achieve convergence for all the significant variables in the flow solution use had to be made of an additional routine supplied by CFDS at Harwell. Without this routine CFDS-FLOW3D does not correctly calculate the pressure changes due to the addition of mass and enthalpy in closed systems leading to poor convergence.

As with the open room calculations the default CFDS-FLOW3D linear solvers were employed. A time step of 0.5 seconds was chosen with a transient duration of 60 seconds. A maximum mass residual tolerance of  $1 \times 10^{-2}$  was chosen to control convergence. The problem required about 10 days CPU time on a Sun SparcStation IPX.

## Results

The calculation converged fully with a mass source tolerance of  $4 \times 10^{-3}$  being achieved. The corresponding momentum and enthalpy residuals were  $\sim 10^{-3}$  and  $\sim 10^2$  respectively.

Figures 4.1 a-c show the temperature, oxidant, and combustion product distributions respectively at 60 seconds in a plane parallel to the axis of the cylinder and at a perpendicular distance of about 2.0 metres from it. The velocity distribution (Figure 4.1(d)) is shown in a similar plane 1m from the axis. The hot gases from the combustion zone form a buoyant plume which clings to the walls adjacent to the fire. In the plane in which the results are plotted this plume is interacting with the platforms and structures in the left hand compartment. Smoke can be seen to have entered the top right hand compartment forming a hot layer (Fig 4.1 (c)) with some recirculation at the far right hand wall (Fig 4.1 (d)). Comparison of Fig 4.1a and Fig 4.1c show identical distributions of temperature and combustion products as expected. The results show a peak mass fraction of products of about 10% and corresponding peak temperature rise of about 325 K after 60 seconds.

## CONCLUSIONS

In this paper we have presented some examples of the work that is being carried out within AEA Technology for the application CFD techniques in the field of fire and smoke movement. As a result of some of the validation studies which have been completed, the code has been successfully used in the assessment of fire and smoke hazards in real fire situations. Further validation work is still going on within AEA and it is expected that the CFDS-FLOW3D code will become even wider in scope as we gain more experience in its varied applications.

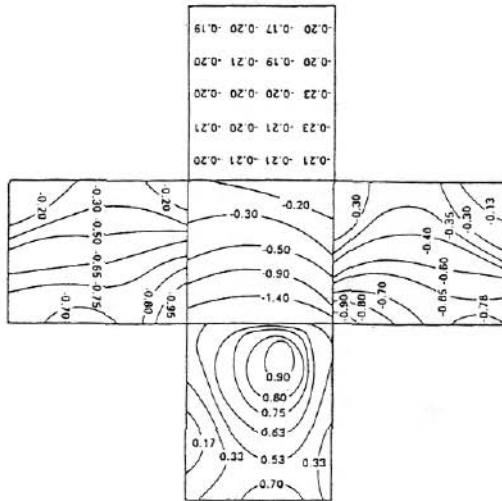
## ACKNOWLEDGEMENTS

The authors would like to acknowledge the contribution of Dr David Fletcher, formerly of the Physics Department, AEA Technology Consultancy Services, who carried out some of the CFDS-FLOW3D runs presented in this paper. The authors wish to thank AEA Technology for their permission to publish this paper. The work was funded by the Corporate Research and Applications Development Project of the AEA Technology.

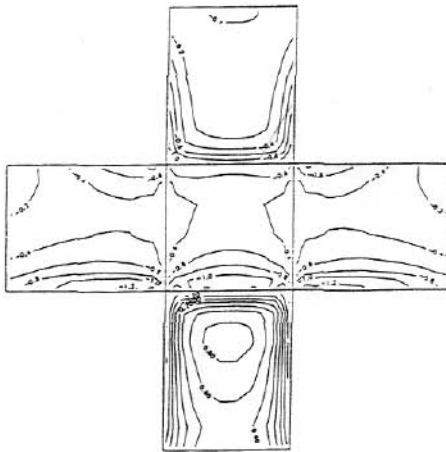
## REFERENCES

1. Fletcher, D. F. and M'Caughey, *Calculations of the Wind-induced Pressure Distribution on a Model Building*, SRD/CLM/(92) 79, also to appear in Fire Safety Journal, 1993.
2. Askari, A., Fletcher, D. F., Lockett, T. J., Sinai, Y. L., Thompson, C. P., *Fire Modelling for the Offshore Industry*, Interflam '93, Interscience Communications Ltd., Oxford 1993.
3. Kandola, B. S., *Effects of Atmospheric Wind on Flows through Natural Convection Roof Vents*, Fire Technology, 26, 106-120, (1990).



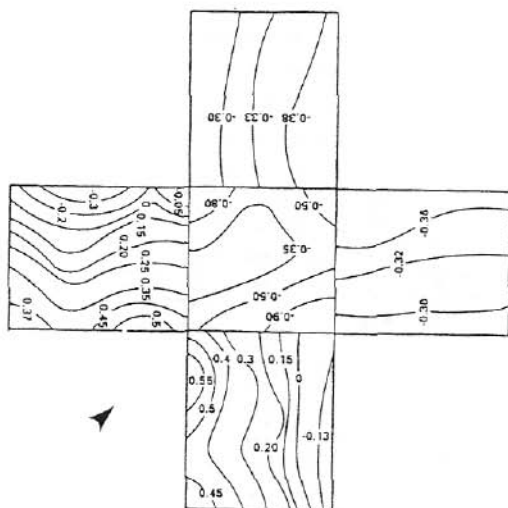


(a) Experimental Results. Taken from [Ref 3]

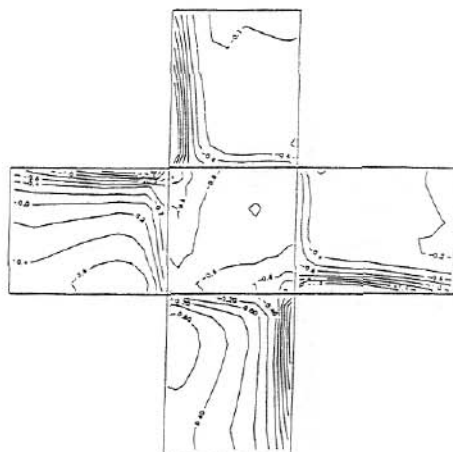


(b) Computational Results

Fig 2.1 Comparison of the experimental and calculated results of the  $0^\circ$  case.



(a) Experimental Results. Taken from [Ref 3]



(b) Computational Results

Fig 2.2 Comparison of the experimental and calculated results of the 45° case.

### Model Geometrical Details

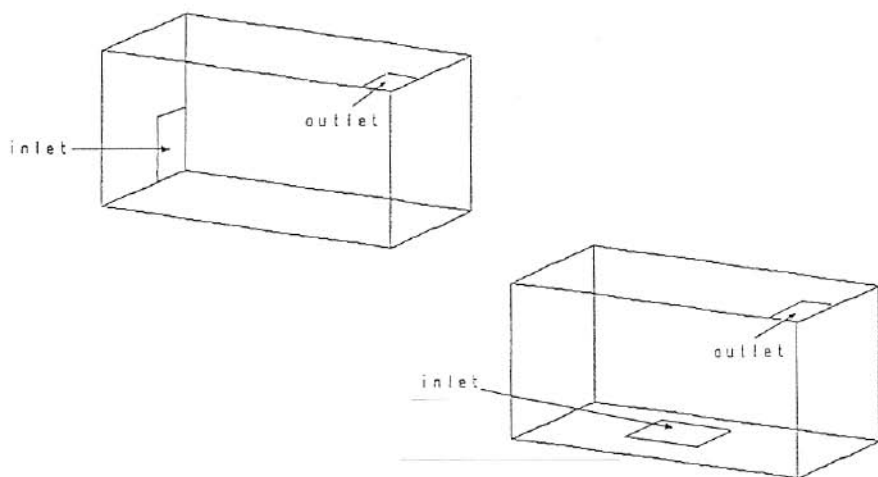


Fig 3.1(a) Cubical Model Details

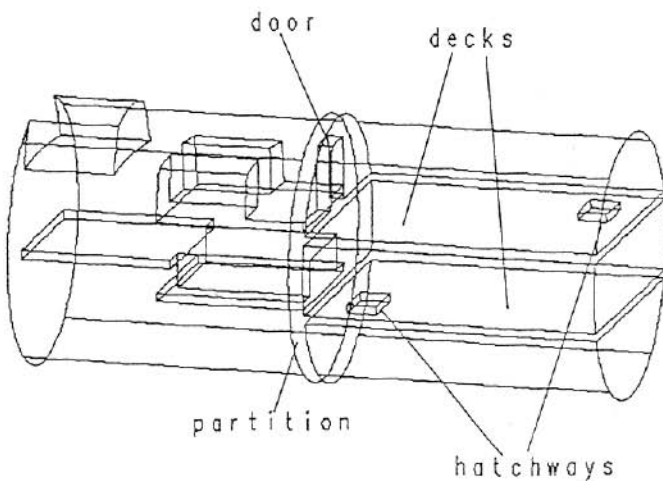
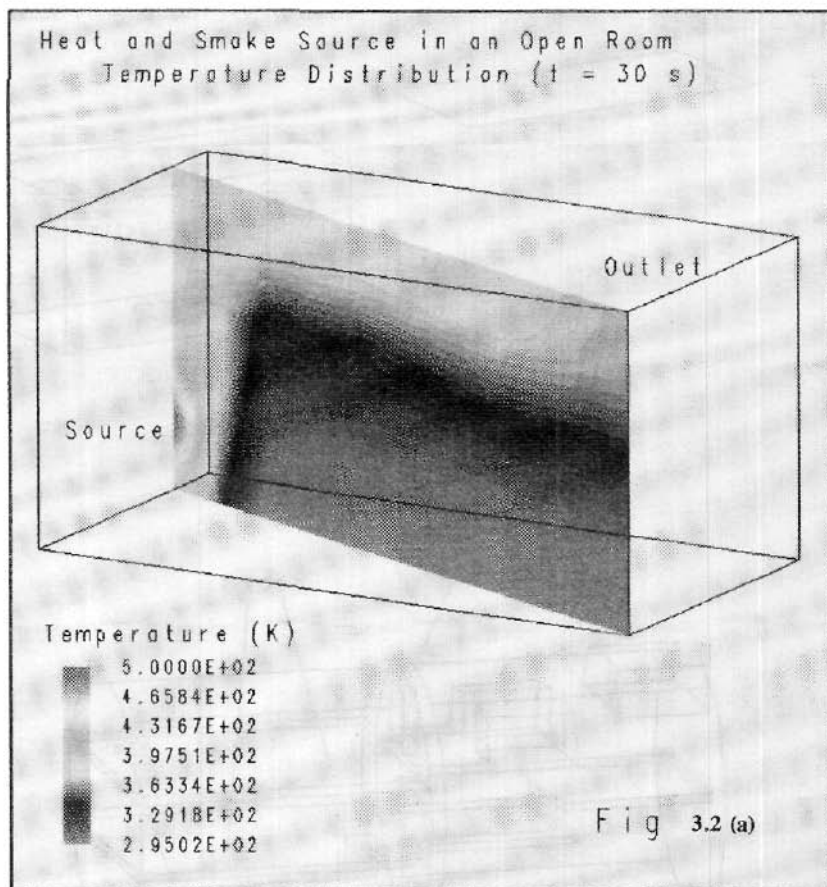
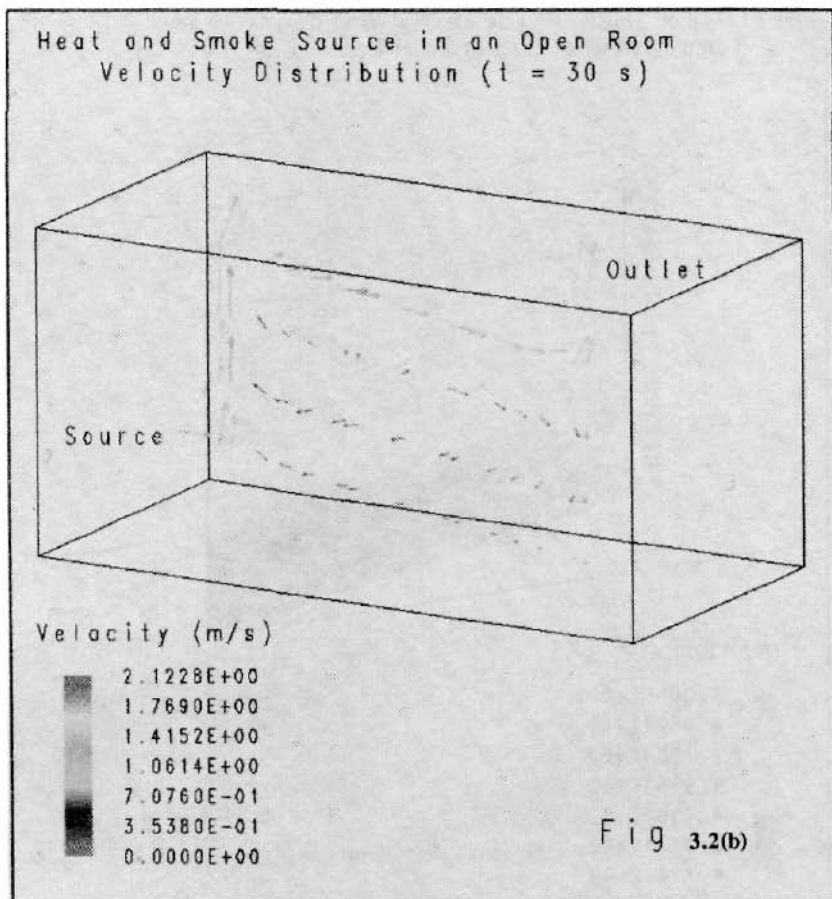
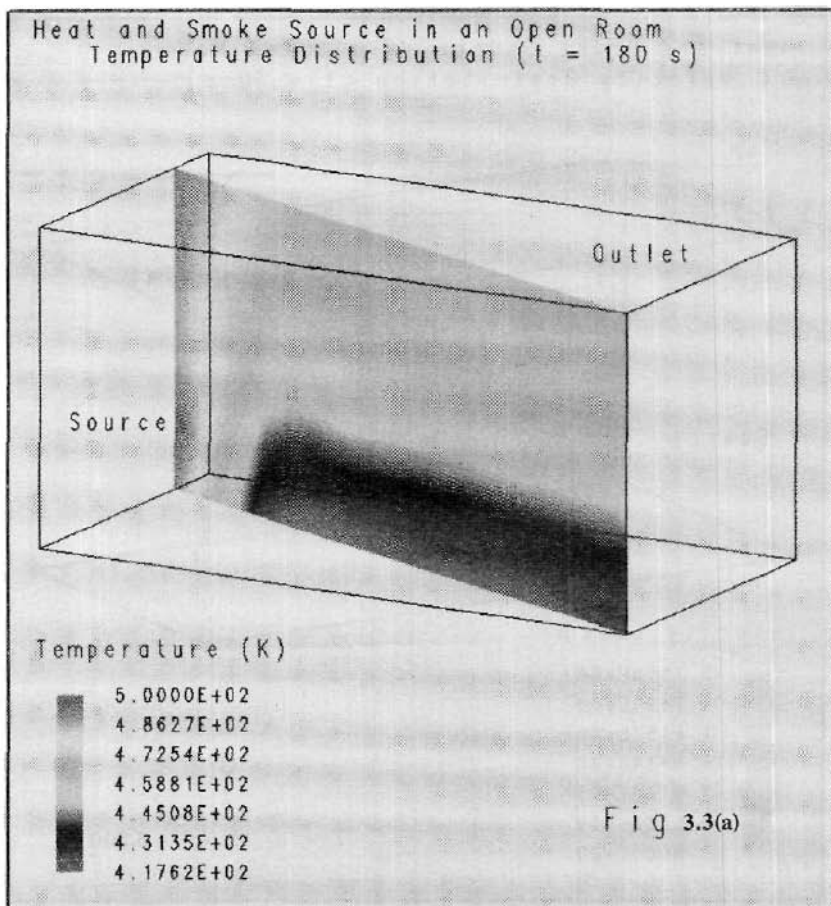
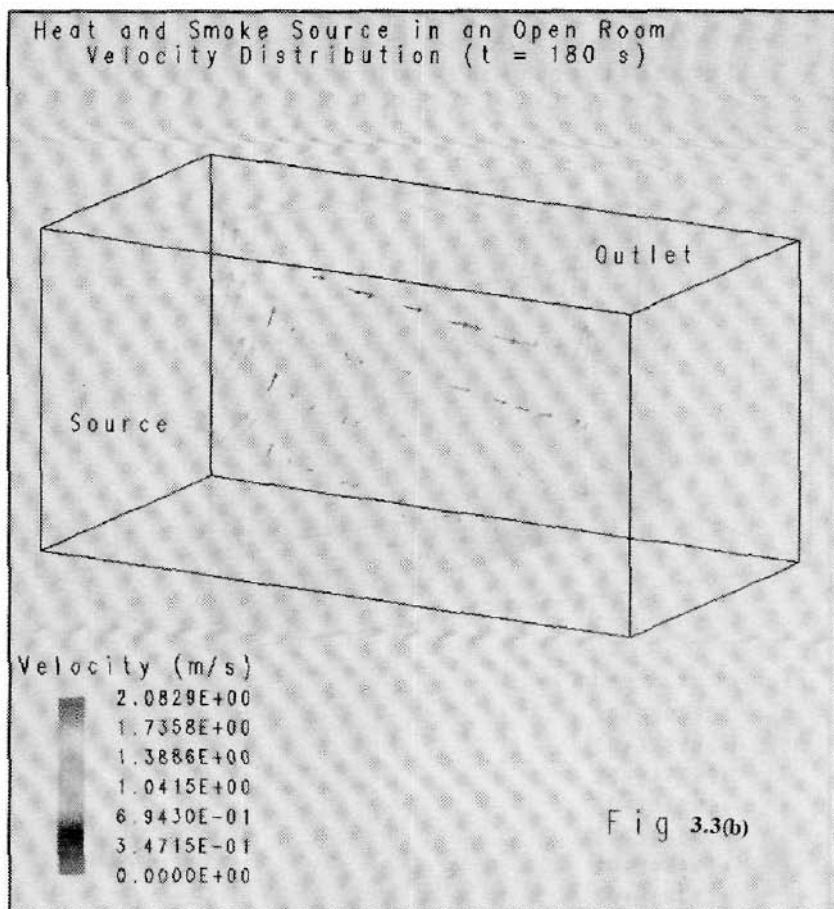


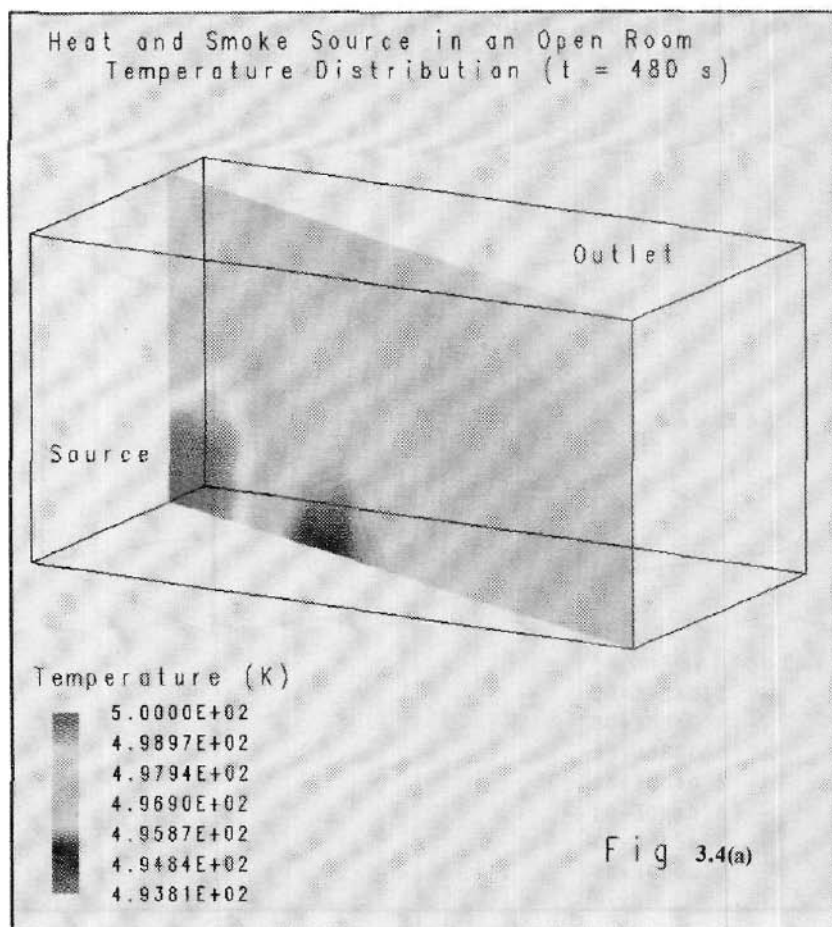
Fig 3.1 (b) Tubular Structure Details



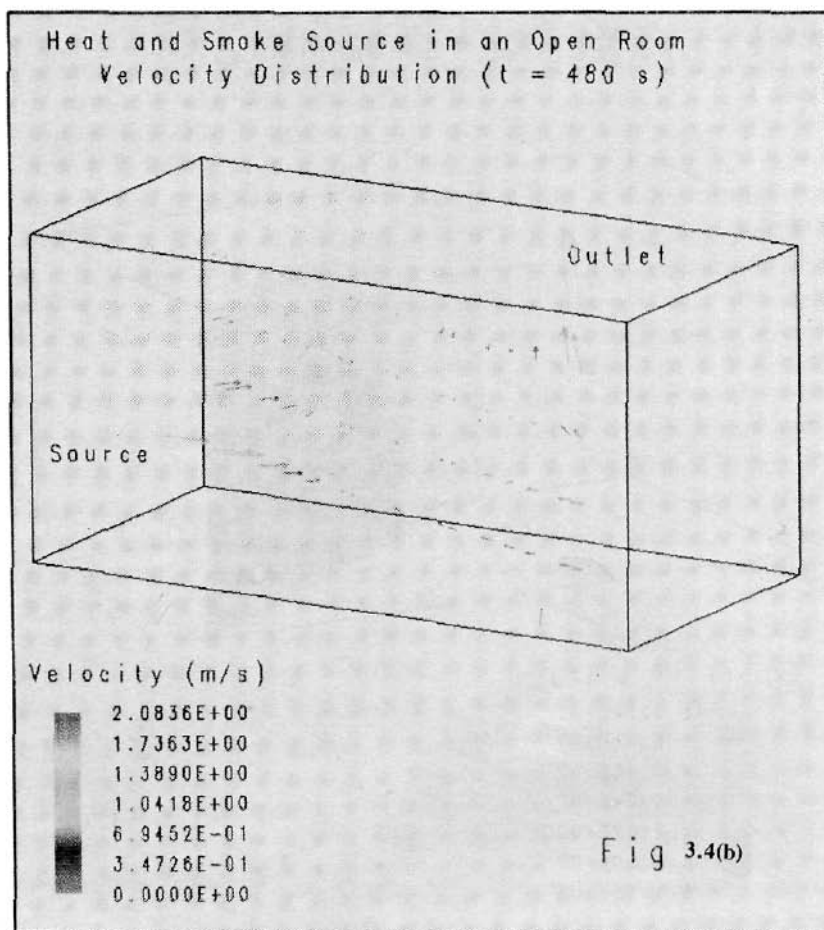


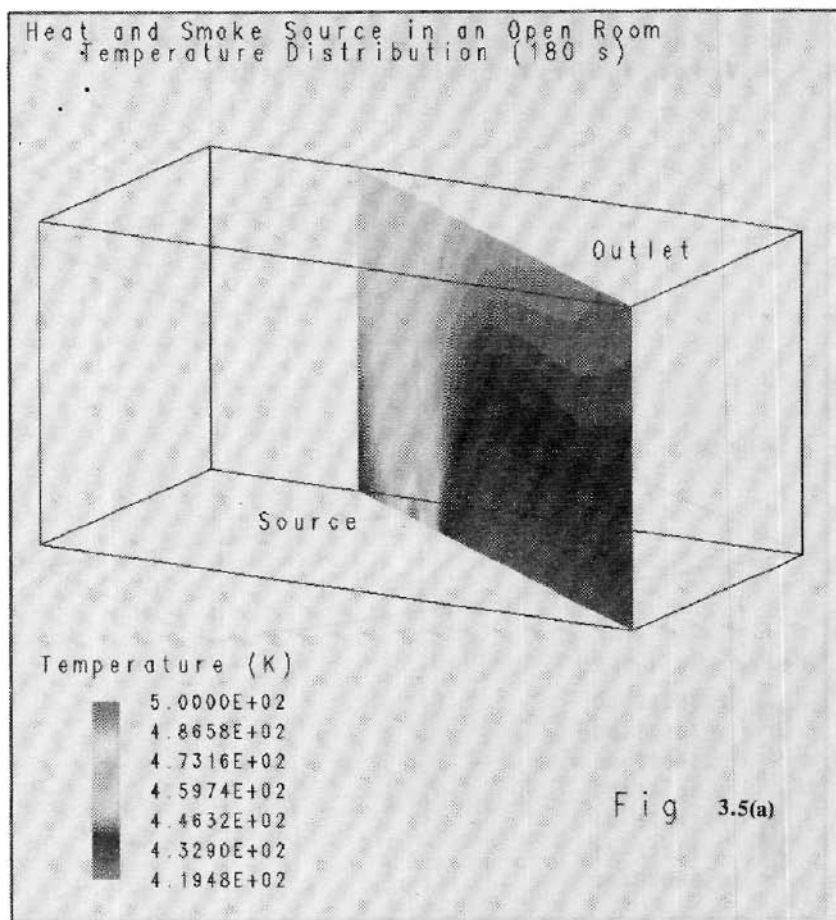


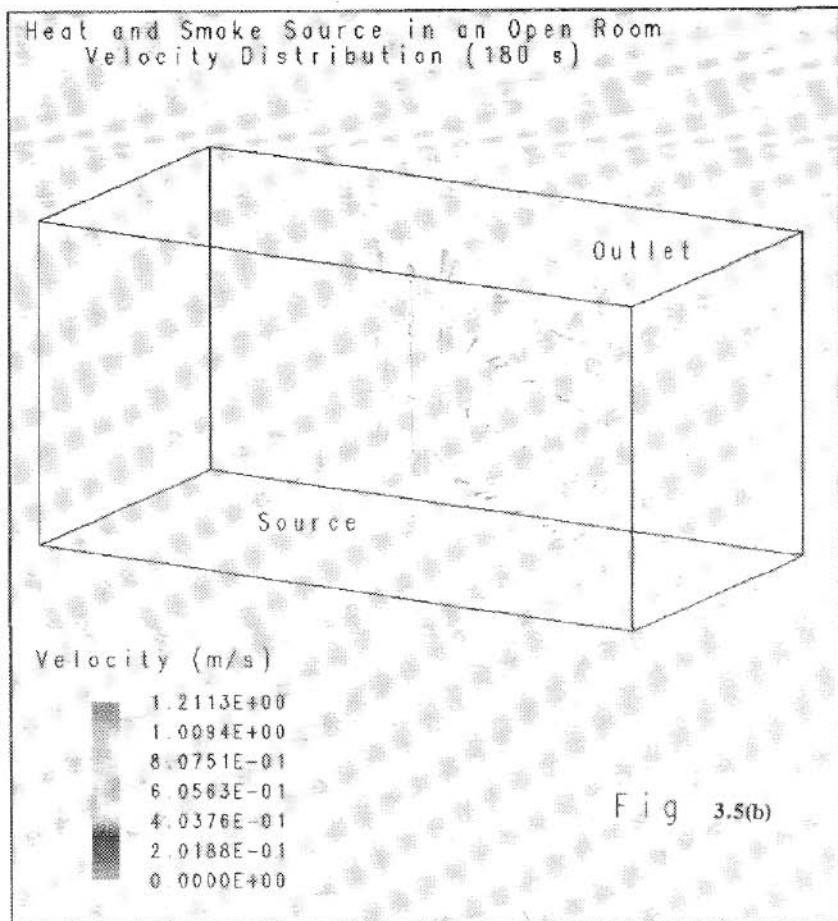












## Model Geometrical Details

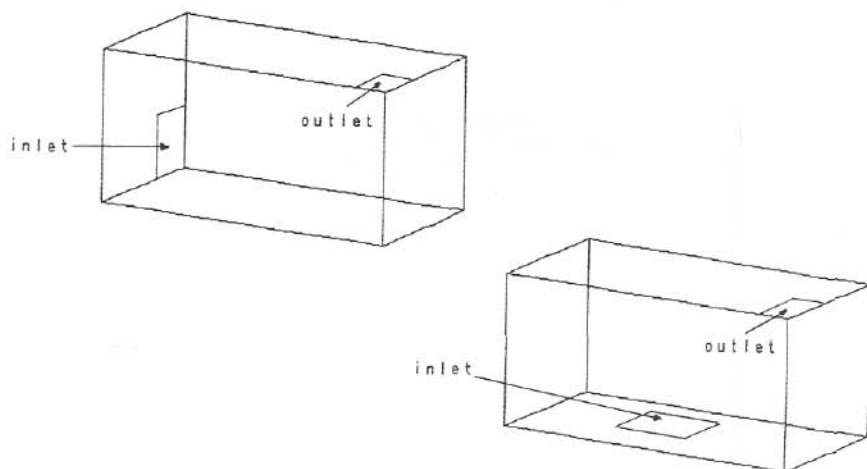


Fig 3.1 Single cell cubical model details

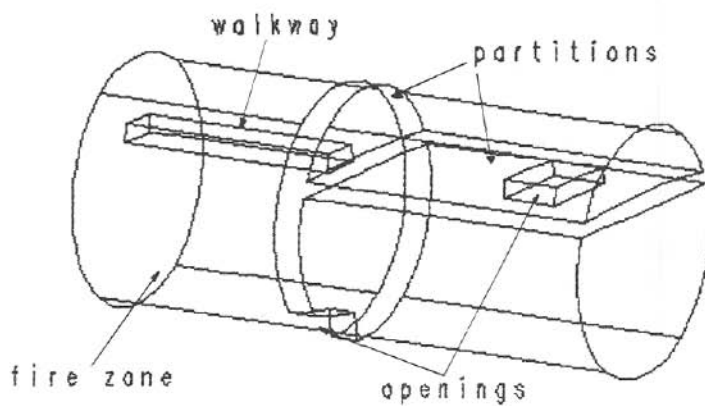
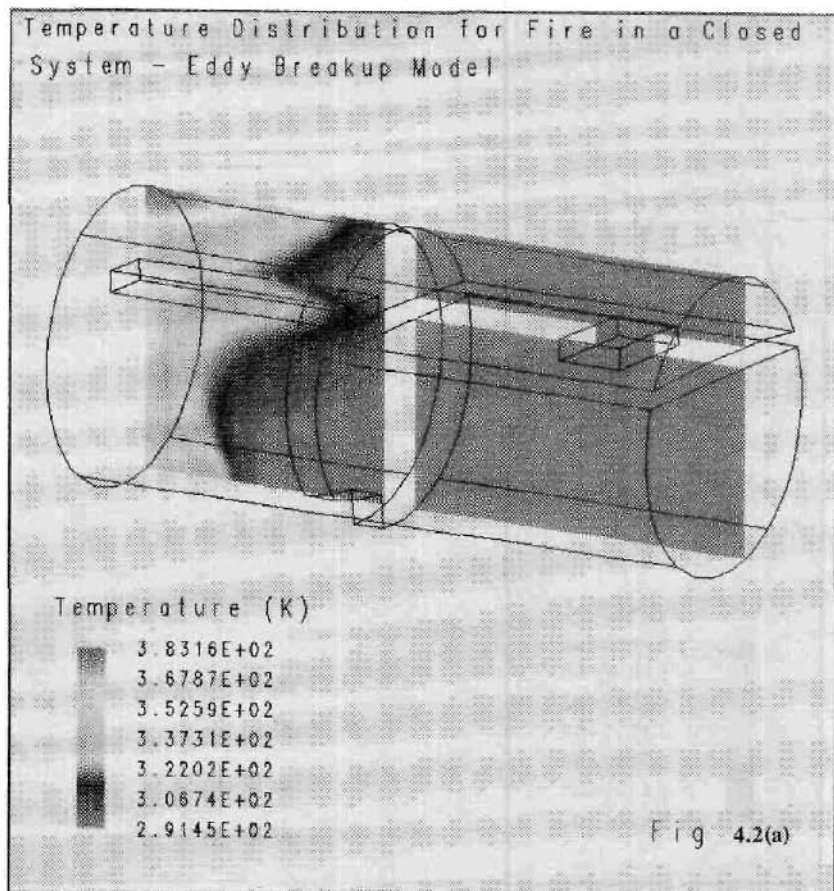
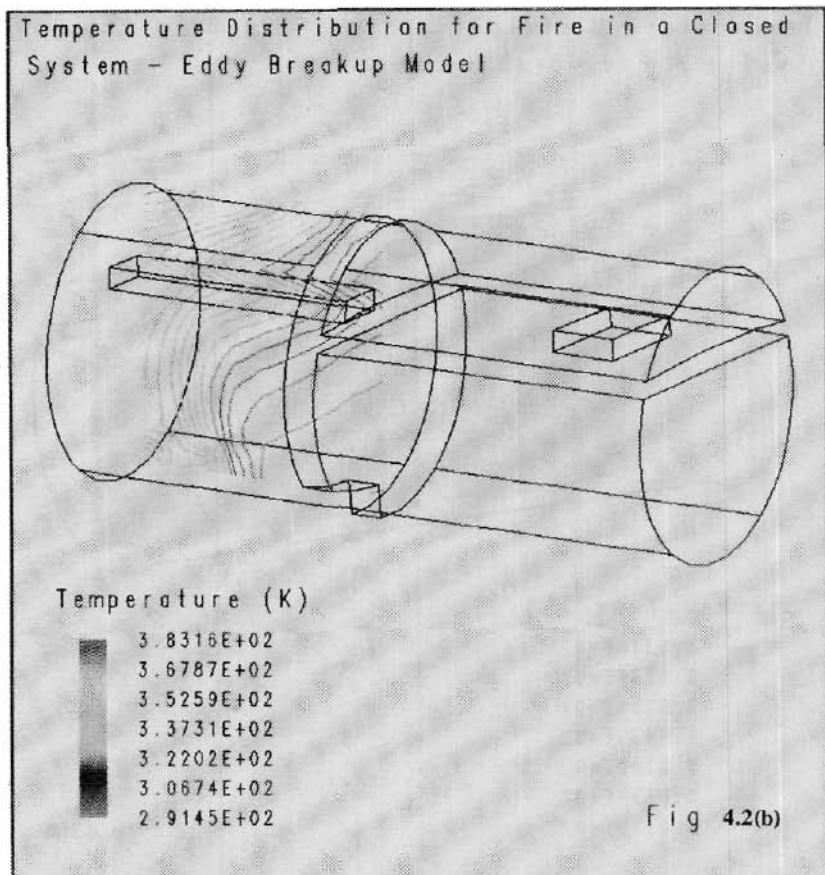
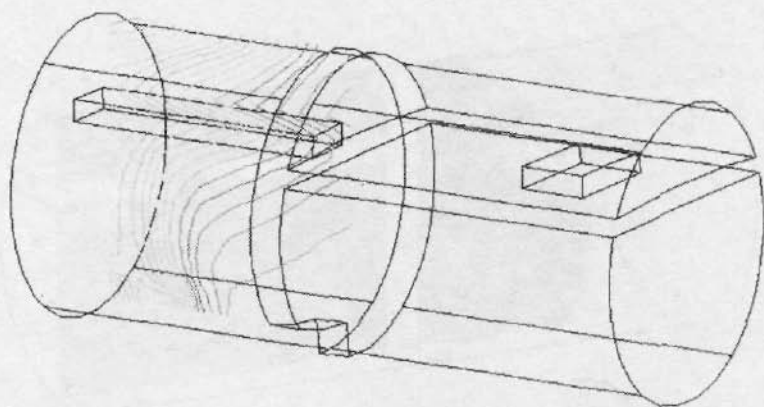


Fig 4.1 Tubular structure details





Mass Fraction of Products Distribution  
For Fire in Closed System - Eddy Breakup Model



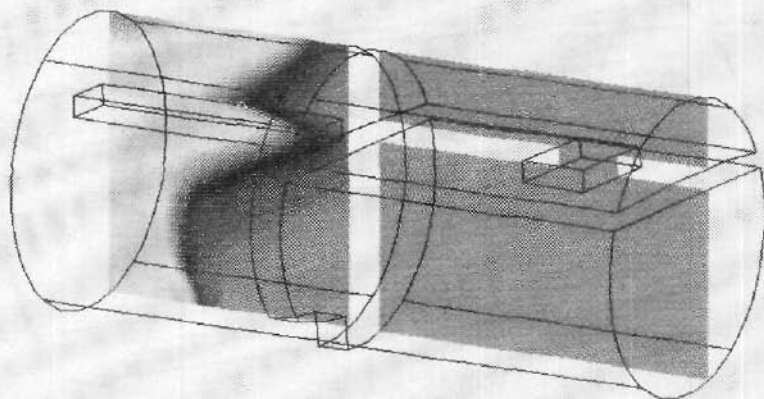
Mass Fraction



$3.4075E-02$   
 $2.8396E-02$   
 $2.2717E-02$   
 $1.7037E-02$   
 $1.1358E-02$   
 $5.6791E-03$   
 $0.0000E+00$

Fig 4.3(b)

Mass Fraction of Products Distribution  
For Fire in Closed System - Eddy Breakup Model



Mass Fraction



3.4075E-02  
2.8398E-02  
2.2717E-02  
1.7037E-02  
1.1358E-02  
5.6791E-03  
0.0000E+00

Fig 4.3(a)