CFD Development of Engineering Measures for Fire Fighting Automatic Sprinkler Systems

Mahmoud Fouad*, Ghada El-Sawah**

* Professor, Faculty of Engineering, Cairo University, Egypt
Email: ***mahmoudfouad@gawab.com***

** Assistant Prof, Higher Technological Institute, Tenth of Ramadan City, Egypt.
Email: ***g_gareebh@gawab.com***

Abstract

Fires may lead to great loss of both life and property. For that rationale, it has been of great importance to develop engineering measures for fire fighting. The development of international standards that govern this process has been based on sound Engineering principles, test data, and field experience. Automatic sprinkler system is one of the imperative systems in the profession. Excellent fire fighting records, that have been established by the sprinkler systems, have been documented in these codes and standards. However, meeting changing technological changes need more research and tools in that respect.

In determining the position, location, spacing and proper use of sprinklers to achieve their function, the codes and standards have relied on previously stated principles. However, the physical existence of obstructions will cause serious changes. These may include structural beams, HVAC ducts, utility pipes etc… Obstructions may prevent the flow pattern of sprinkler from fully developing. They also may prevent sprinkler discharge from reaching the hazard. Therefore, it is of vital importance to fully comprehend and understand the mechanism of interference through which these obstructions work against the flow pattern of sprinklers. CFD analysis may represent a new and useful tool towards this understanding. It helps visualizing and identifying the mechanism, leading to the development of engineering practice suitable to address this problem.

In the present work, flow simulations of the problem, through the use of CFD analysis is attempted. Different configurations, while guided by previously built up experience, are studied.

Keywords

Fire sprinkler, two phase flow, CFD, water flow pattern obstructions.

Introduction

Sprinkler systems are now widely used in buildings to protect against fire due to their proven effectiveness as safe and reliable fire fighting tools. However, as greater reliance is placed upon such systems for fire safety it becomes more important that we understand and able to predict their behavior. Since the mechanism of the water spray pattern produced by a conventional sprinkler system is vital importance to be fully developed and works against the obstructions, more attention has been paid to investigate this pattern numerically. Many factors such as water flow rate, spray nozzle type, droplet size, location of sprinkler head, ventilation condition, and obstructions in the way, have grate effects on the water spray flow pattern.

An insufficient knowledge is available for a quantitative assessment for such combination of factors. Such knowledge when found will lead to an optimal design of water spray system. The engineering design of water spray systems imposes stringent requirements to develop a quantitative approach to estimate and predict the effectiveness/performance of the water spray system for various fire scenarios.

Computational fluid dynamics (CFD) has already been proven to be a useful and powerful tool in fire safety science. A number of CFD studies on fire growth, spread and smoke movement have been reported in many investigations. Some of the computational models have been validated against experimental data and have been proven to be successful in predicting smoke flow patterns, smoke temperature distributions in rooms, and fire extinguishing process.

Chow and Fong, (1) were investigated three-dimensional numerical model to examine sprinkler spray interaction with fire-induced hot smoke layer. They considered only the coupling of momentum and energy between the gas and water droplet phases. To simplify the problem, the sprinkler discharge pattern was treated as a constant hollow cone, and was assumed to be unaffected by the airflow. Hence, no droplet trajectories were calculated. The effect of water spray on the airflow was accounted for by the additional source terms in the momentum and energy equations. An early work on modeling fire suppression with sprinklers was carried out by Alpert (2). The study modeled the gas/water droplet flows in two dimensions and took into account three modes of interactions between fire plume and water droplet (namely mass, momentum, and energy transfer).

Nam (3) investigated the penetration capability of commercial sprinkler sprays by numerical simulation. In his work, steady-state simulations were conducted to study the interaction of fire plume and water spray. In order to avoid modeling the complex chemical reaction in the fire combustion, the fire was treated simply as a prescribed heat source. Hassan (4) also conducted steady-state simulations on the use of fine water sprays to extinguish fires in computer cabinets. In his study, the fire extinction model took into account some aspects of chemical reaction in fires. However, from a realistic point of view, since any fire extinction process is time dependent, the assumption of steady state may not be able to capture all the relevant aspects.

Hoffmann and Galea (5, 6) performed, by extending the field-fire modeling technique, transient simulations of the interaction between water spray and fire plume. Unfortunately, they considered only the momentum and thermal interactions between the water spray and fire and disregarded the effect of chemical reaction in fire combustion

Currently, quantitative approaches to estimate the performance and effectiveness of sprinkler systems have not been developed to a stage where they can be used to optimize the design for obstructions found in the field. In the present work, a numerical investigation is introduced to provide a quantitative analysis of such complex interactions occurring between sprinkler flow pattern and obstructions. The approach adopted is a computer modelling using CFD technique.

Since it is necessary to consider both air and water droplet flows (no fire situation), the problem becomes one of two phases, a continuous (air) phase and a discrete (water droplet) phase. Lagrangian discrete phase model which follows the Euler-Lagrange approach was used in the numerical modelling. The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of droplets through the calculated flow field. The dispersed phase was enabling to exchange momentum, mass, and energy with the fluid phase.

A fundamental assumption was made in the model that the dispersed second phase occupies a low volume fraction, even though high mass loading. The dispersion of particles due to turbulence in the fluid phase was predicted using the stochastic tracking model. The stochastic tracking model was included the effect of instantaneous turbulent velocity fluctuations on the particle trajectories, as the simulations were carried for a turbulent flow.

The procedure for setting up and solving a coupled steady-state discrete-phase model was started first by solving the continuous-phase flow which impacts the discrete phase then creating the discrete-phase injections, and solve it to calculate the effect of this discrete phase trajectories on the continuum and finally coupling these two solutions to have a solved coupled flow, (figure 1).

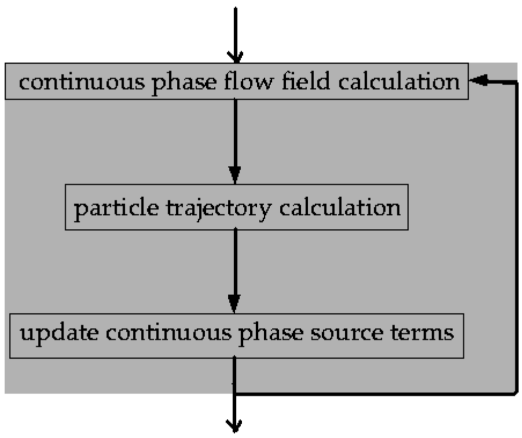

Figure (1): Coupled Discrete Phase Calculations.

Model Mathematics

The interaction between the water spray and the air were studied by using a numerical model includes the two main fluid phases. The gas flow will represent the air in the numerical model, while the liquid phase represent the water droplets injected from the spray nozzle. The used numerical modelling approach to solve such two phase flow problem was chosen to be the Eulerian- Lagrangian method. In this method the gas phase is regarded as a continuum while the water droplet phase is treated as individual particles and are traced using the lagrangian approach. The momentum, heat and mass transport of these discrete particles are calculated by taking into account the various interacting forces with the gas phase. The effect of the particles on the gas phase is taken into account by introducing appropriate source terms in the conservation equations for the gas phase.

Gas phase Modelling

The gas phase is created in the model as an isothermal turbulent flow simulation. The gas is described by the conservation equations of mass, momentum, energy and species along with K-ε model for turbulence.

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0, \tag{1}$$

Where i=1, 2 and 3 denote x, y and z (vertical direction) directions, respectively, in the Cartesian coordinate system.

Momentum Equation in (x) Direction:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial \rho}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \rho g_i + \frac{\partial}{\partial x_j}\left(\mu_{\text{eff}} \frac{\partial u_i}{\partial x_j}\right). \tag{2}$$

General Transported Equation (e.g. smoke concentration, temperature):

$$\frac{\partial}{\partial t}(\rho \phi) + \frac{\partial}{\partial x_j}(\rho u_j \phi) = \frac{\partial}{\partial x_j}\left(\Gamma_{\text{eff},\phi} \frac{\partial \phi}{\partial x_j}\right) + S. \tag{3}$$

The standard two-equations for k– ε model have been widely used to estimate the turbulence characteristics of the gas phase flow. By solving these two equations, the turbulence kinetic energy and its dissipation rate are obtained and then used to calculate the turbulent effective diffusion coefficient.

Water Spray Modelling

The water spray discrete phase is modeled by creating an injection plane surface with a reflect boundary condition for the particles. The initial position of the spray nozzle surface, velocity of the water, temperature, and size of individual particles are defined.

These initial conditions are used to initiate the trajectory calculations. The trajectory calculations are based on the force balance on the particle using the local continuous phase conditions as the particle moves through the gas flow. The predictions of the discrete phase patterns are including of the effect of the discrete phase on the continuum. In this coupled approach, the continuous phase flow pattern is impacted by the discrete phase (and vice versa), and the model alternated the calculations of the continuous phase and discrete phase equations until a converged coupled solution is achieved.

The particles trajectory predictions of the discrete phase were performed by integrating the force balance on the particle, which is written in a Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the particle, as:

$$\frac{du_p}{dt} = F_D(u-u_p) + \frac{g_x(\rho_p-\rho)}{\rho_p} + F_x \qquad (1)$$

Where:

$$F_D = \frac{18\mu}{\rho_p d_p^2}\,\frac{C_D \mathrm{Re}}{24} \qquad (2)$$

$$\mathrm{Re} \equiv \frac{\rho d_p |u_p - u|}{\mu} \qquad (3)$$

Here, u is the fluid phase velocity, u_p is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_p is the density of the particle, and d_p is the particle diameter. Re is the relative Reynolds number. The drag coefficient, C_D, is taken as,

$$C_D = a_1 + \frac{a_2}{\mathrm{Re}} + \frac{a_3}{\mathrm{Re}^2} \qquad (4)$$

Where a_1, a_2, and a_3 are constants, that apply for smooth spherical particles over several ranges of Re given by Morsi and Alexander (7).

$$F_x = \frac{1}{2}\frac{\rho}{\rho_p}\frac{d}{dt}(u-u_p) \qquad (5)$$

The Numerical Model

A two-dimensional CFD model is developed to investigate the interaction between water spray flow pattern and the obstructions in the field. The schematic diagram of the computational domain section is shown in Figure (2). The simulation was carried out for tunnel square section with 9 m^2 area. An automatic fire sprinkler Recessed Pendent type was used to inject the water spray from the middle of the ceiling. The sprinkler orifice diameter was 10 mm diameter with a flow rate 1 kg/sec. The water spray nozzle of the sprinkler was simplified to be a ceiling opening representing the orifice of this Recessed Pendent type with a deflector of 5 cm diameter and 2.5 cm vertical distance downward the nozzle opening.

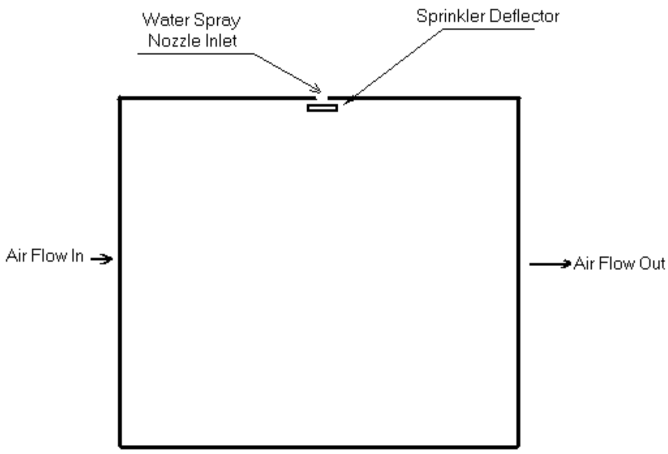

Figure (2), Schematic diagram of the computational domain section.

Results and Discussions

2D simulation was carried for the numerical model with 90295 quadrilateral cells and gravitational force effect included. The steady-state simulation results of the fluid phase were shown in Figs. (3, 4), where the velocity, static pressure predictions were illustrated. This was taken to be the initial state from which the coupled simulation developed.

Figure (3) shows the fluid phase velocity vectors. The air was supplied from the left side of the tunnel with a velocity normal to boundary of 0.8 m/sec. Figure (3) (B) shows the effect of the sprinkler deflector obstruction to the flow streams. The static pressure distribution contours were shown in figure (4) agreed with the expectations especially around the sprinkler deflector giving an increasing in the pressure value than the domain at the leading edge of the deflector followed by a decreasing in the pressure value at the trailing edge.

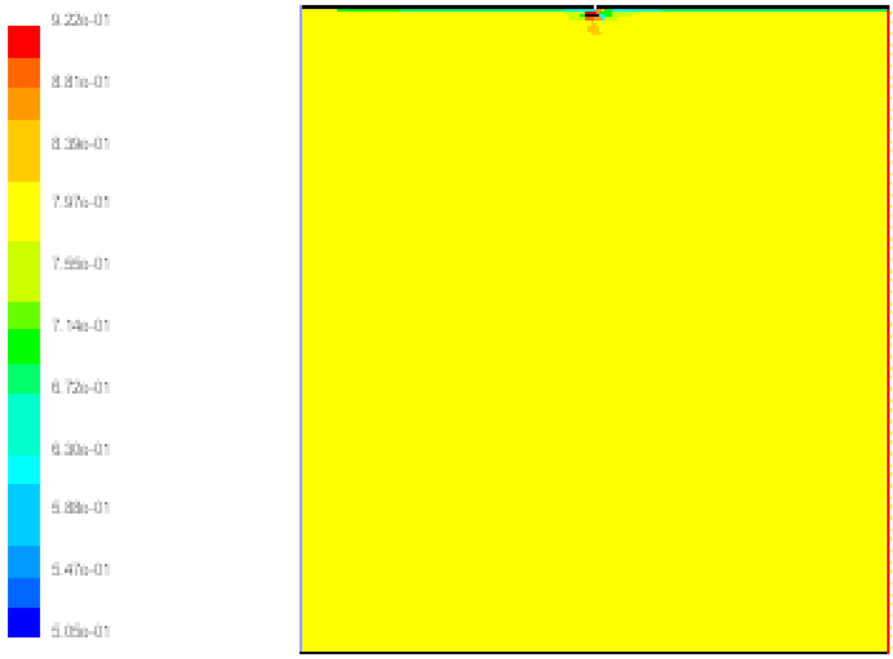

Figure (3), (A) Velocity Vector of the Continues Phase.

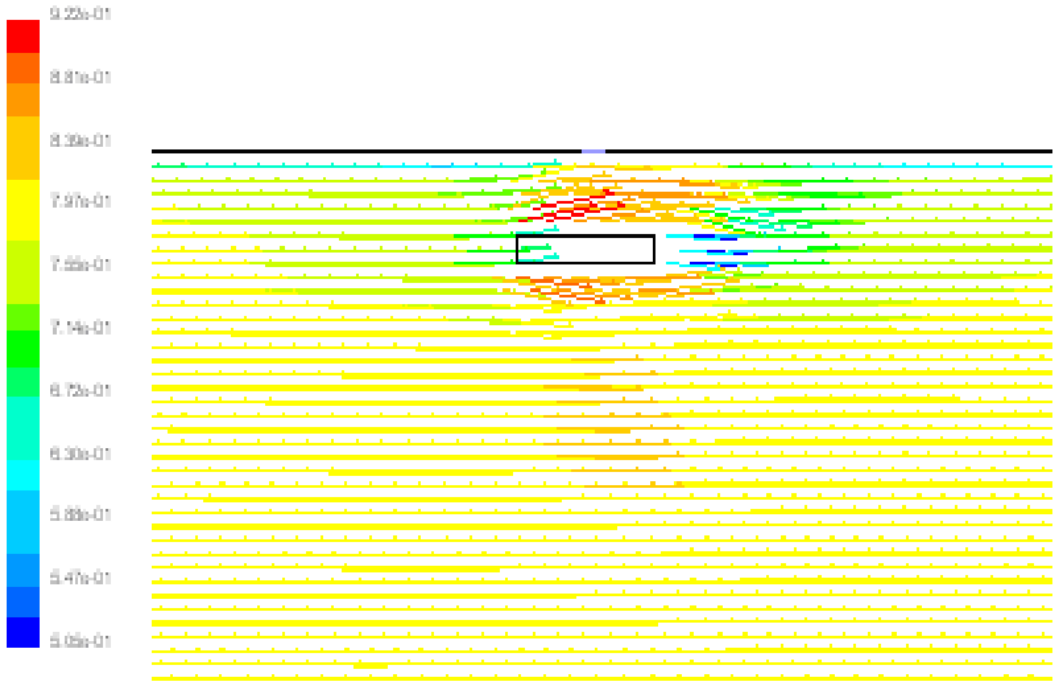

Figure (3), (B) Zoom In For the Velocity Vector around the Sprinkler Deflector.

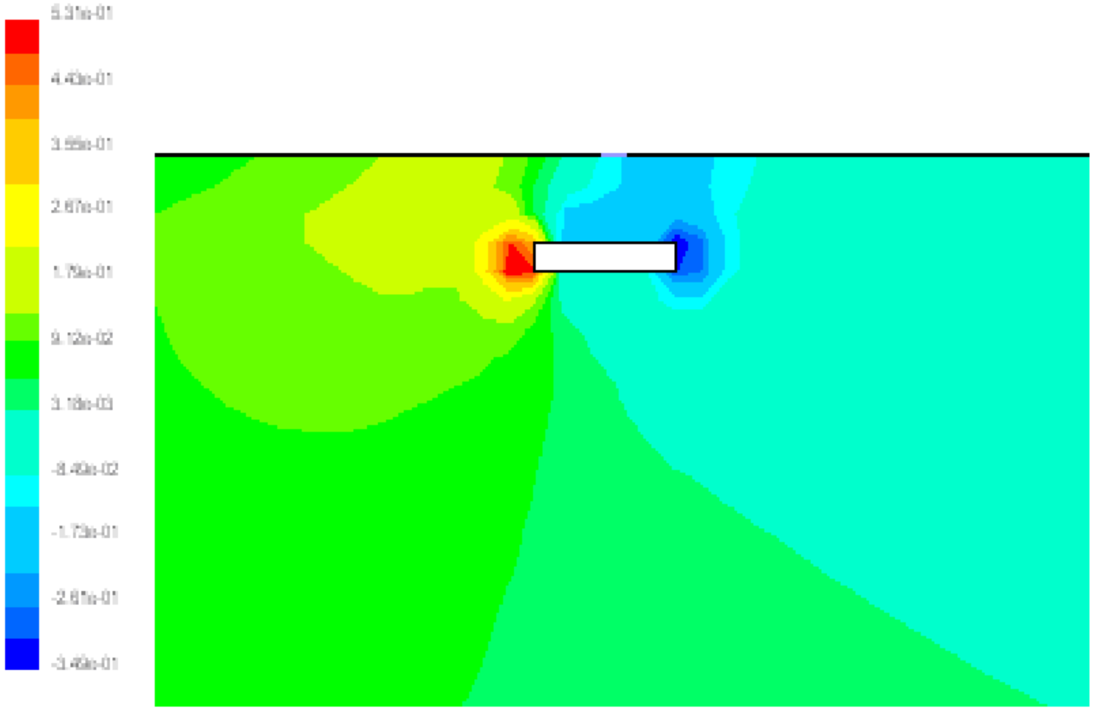

Figure (4), Static pressure contours of The Continues Phase.

For better understanding to the interaction between the two phases, the numerical simulation predictions have been shown the effect of the water injection on the fluid phase velocity vectors. The water droplets shoot out at a high velocity of 12.8 m/s. Obviously, because of this high speed, many recirculation zones were performed in the domain mainly around the inlet of the spray nozzle. Figure (5) shows the velocity vectors of the fluid phase when the water spray is activated from a surface injection type with a discrete phase model simulation.

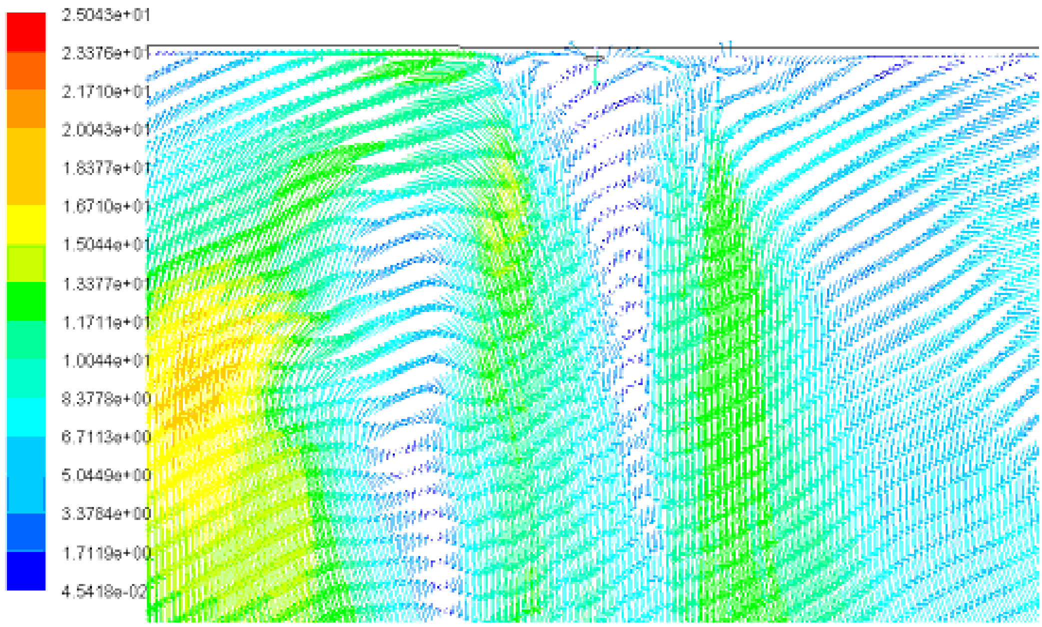

Figure (5), Velocity Vector of The fluid phase in a coupled calculations.

The water spray parameters used in the simulations were uniform distribution for a particle size diameter of 100 μm and inlet water mass flow rate of 1 kg/sec. Different inlet gauge pressure values were simulated to investigate their effect on the water spray flow pattern dispersion. The simulation results for the used type fire sprinkler (Recessed Pendent) show that the value of the inlet pressure has a grate effect on that dispersion. Figure (6) (A) shows the particle trajectory for an inlet pressure zero gauges. The water spray pattern shows approximately 1.5 m^2 area of dispersion 2 m downward from the ceiling.

Figure (6) (B) shows a close view to the injection point. The particle trajectories colored by the velocity value which was about 13 m/sec except for some points. The figure shows deflection in the pattern in the air flow direction as the inlet gauge pressure was zero.

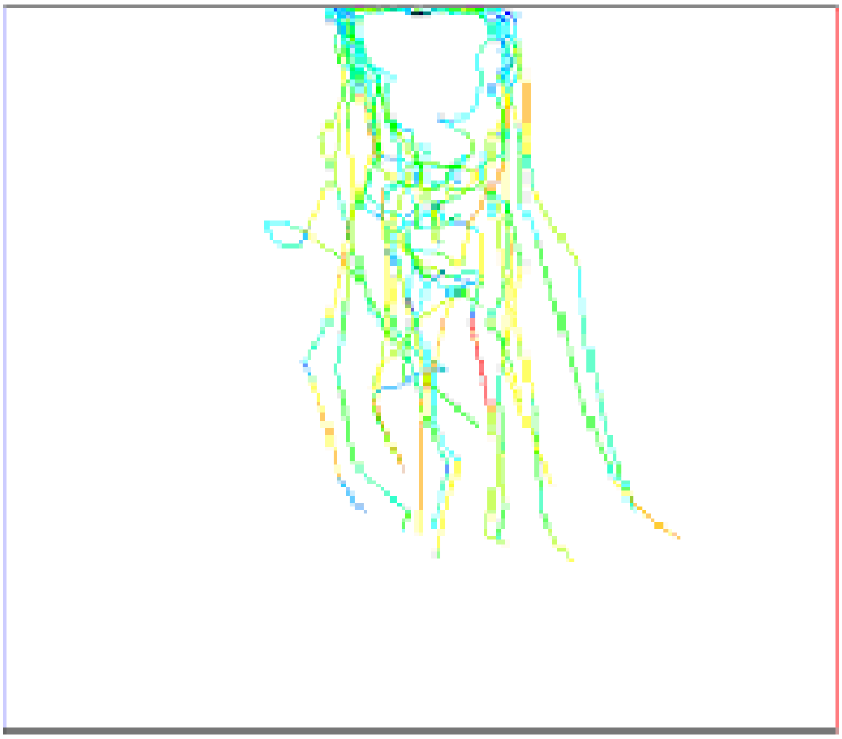

Figure (6), (A) Water spray flow pattern at inlet pressure 0 gauge.

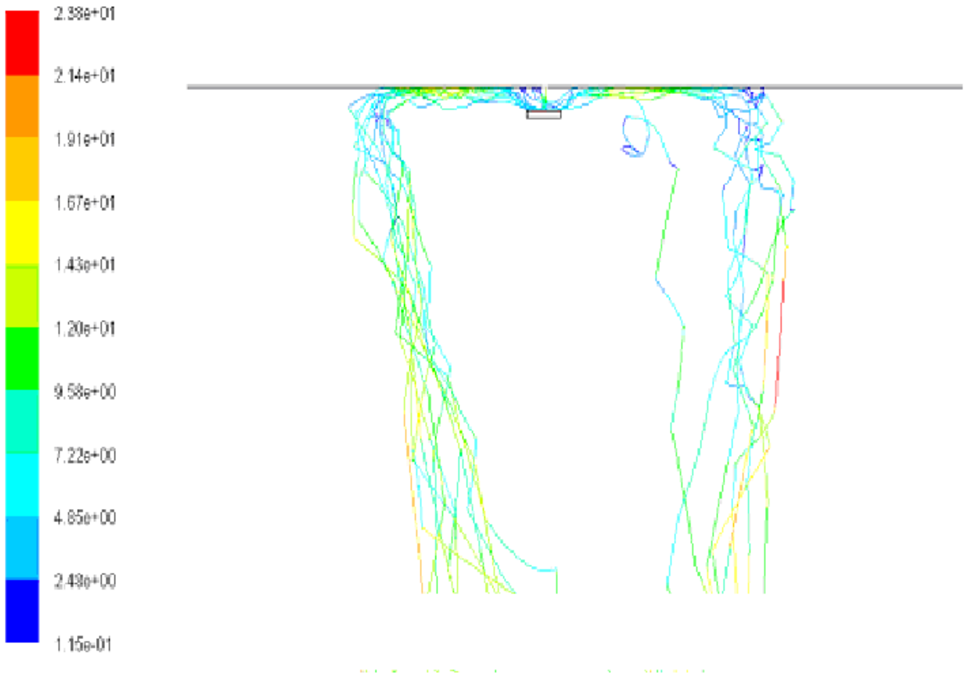

Figure (6), (B) Zoom in For Water spray flow pattern at inlet pressure 0 gauge.

By increasing the inlet gauge pressure to 2 bar for the same inlet mass flow rate a complete change in the velocity vectors occurred. As the water flow interred the domain with a higher pressure, a direct wide dispersion for the particles was performed. The velocity vectors of this case are shown in figure (7), a very strong effect of the discrete phase on the fluid phase was observed.

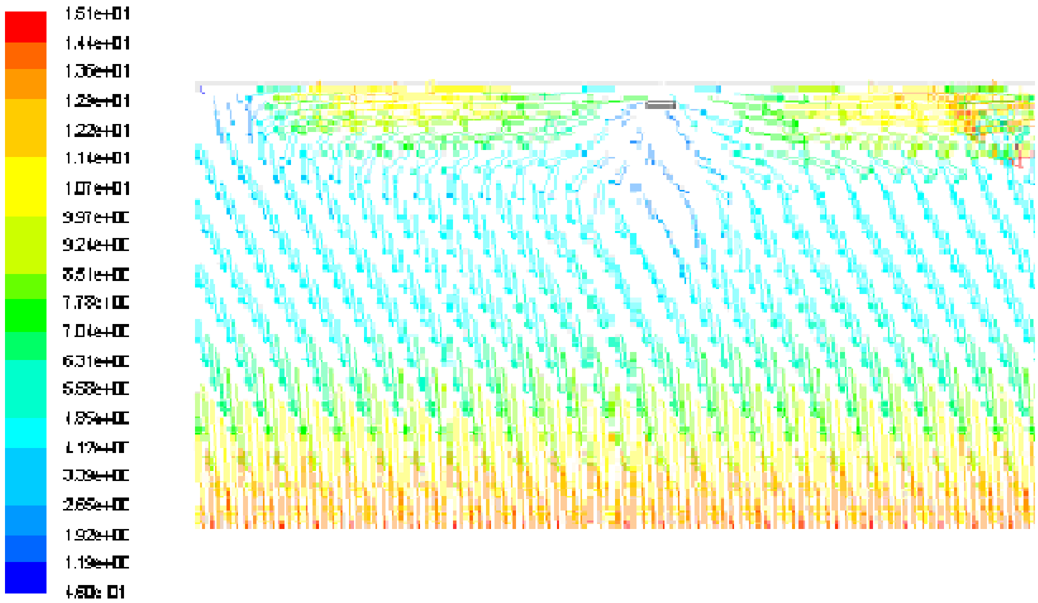

Figure (7), Velocity Vector for an inlet gauge pressure of 2 bars.

The increasing of the inlet gauge pressure to 2 bar had increased the dispersion of the water spray as shown in figure (8) (A). This result leads to an improvement in the fire sprinkler operation. The simulation results show a dispersion of 5.5 m^2 area 2m from the ceiling vertically downward.

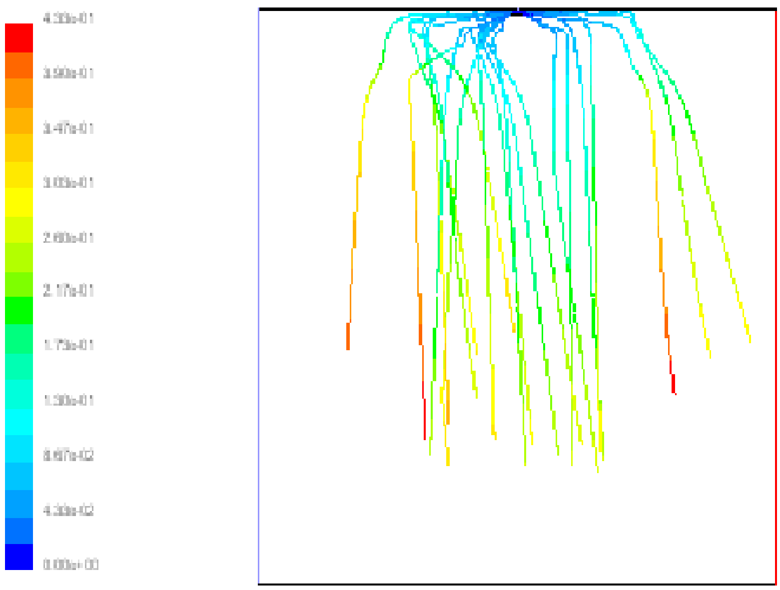

Figure (8), (A) Water spray flow pattern at an inlet gauge pressure 2 bars.

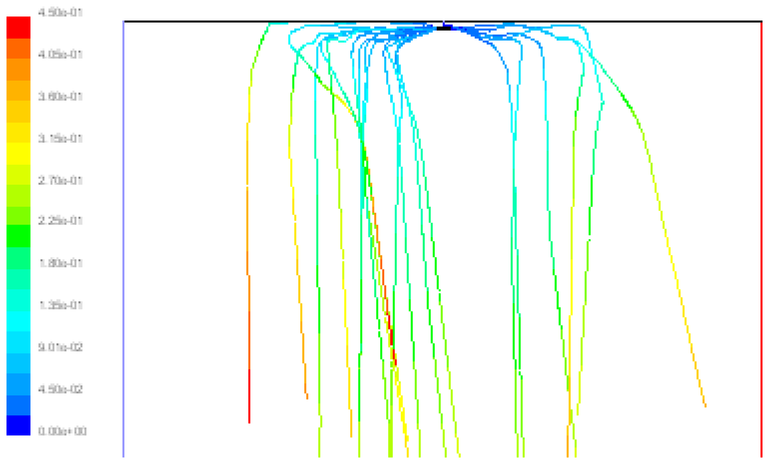

Figure (8), (B) Zoom in For Water spray flow pattern at inlet pressure 2 bars gauge.

From these results it is very clear the effect of the inlet pressure of the water flow rate on the operation of the fire sprinkler as the dispersion increased by more than 100% and so the extinguished area in a fire situation.

As the CFD simulation results gave a complete vision of the operation, many obstruction configurations, as seen in figure (9), could be examined against the predicted water spray pattern resulted from the numerical solution of the two phase flow to avoid the obstruction and inshore a best performance during the fire extinguishing operation.

The CFD numerical approach also will give the ability to simulate different types of automatic fire sprinklers with a wide range of operating and geometrical parameters as the shape of the deflector, the inlet mass flow rate, ventilation rates and so on.

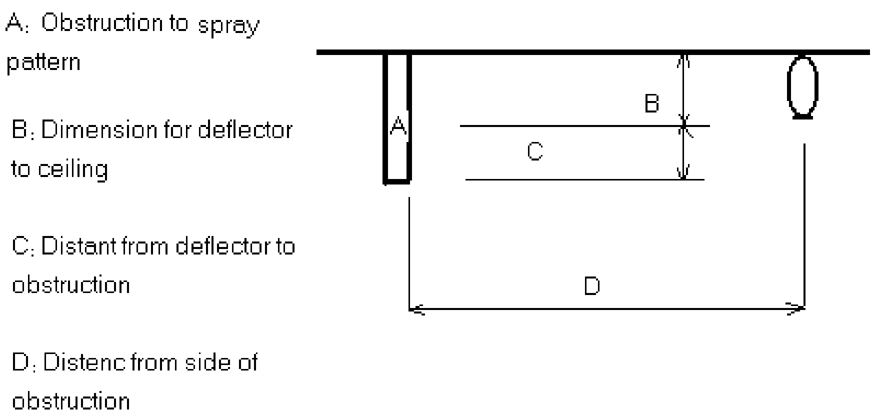

Figure (9), Fire sprinkler obstructed by a structural beam.

Conclusions

In the present work, two phase flow numerical simulations have been carried out to solve the problem of the water spray fire sprinkler flow pattern obstruction in fire fighting systems. The spray was assumed to release from a surface injection fire sprinkler to represent an automatic fire sprinkler flow pattern and its operation in no fire situation. The simulation results have been implemented for two phase flow in which the water treated as a second phase with a coupled solution.

The predictions show that the water inlet pressure has a grate effect on the flow pattern dispersion. Also, these predictions are used to solve the problem of the flow patter obstruction. In this study, the CFD is used successfully as a powerful tool to improve the performance of the fire sprinkler by visualizing and identifying the mechanism and the pattern of the spray at various operating conditions and input parameters.

References

1. W.K. Chow and N.K. Fong , "Numerical simulation on cooling of fire-induced air flow by sprinkler water sprays". *Fire Saf J* **17** (1991), pp. 263–290.

2. R.L. Alpert , "Numerical modelling of the interaction between automatic sprinkler sprays and fire plumes". *Fire Saf J* **9** (1985), pp. 157–163.

3. S. Nam , "Development of a computational model simulating the interaction between a fire plume and a sprinkler spray". *Fire Saf J* **26** (1996), pp. 1–33.

4. M.A. Hassan , "A theoretical simulation of fire extinction by water spray in a computer cabinet". *Appl Math Modelling* **20** (1996), pp. 804–813.

5. N.A. Hoffmann and E.R. Galea , "Mathematical modeling of fire sprinkler systems" *Appl Math Modelling* **13** (1989), pp. 298–306.

6. N.A. Hoffmann and E.R. Galea , "An extension of the fire-field modelling technique to include fire-sprinkler interaction" I. The mathematical basis. *Int J Heat Mass Transfer* **36** (1993), pp. 1435–1444.

7. S. A. Morsi and A. J. Alexander. "An Investigation of Particle Trajectories in Two-Phase Flow Systems". *J. Fluid Mech.*, 55(2):193-208, September 26 1972.