The Nucleus **49**, No. 3 (2012) 171-179

The Nucleus
A Quarterly Scientific Journal of Pakistan Atomic Energy Commission
NCLEAM, ISSN 0029-5698

SIMULATION OF AIR FLOW PATTERNS INSIDE AN EXPERIMENTAL STACK

F. SHAHZAD, S. SARDAR[1], M. I. AHMAD and N. IRFAN[1]

Department of Chemical Engineering, University of Engineering and Technology, Peshawar, Pakistan

[1]Department of Nuclear Engineering, Pakistan Institute of Engineering and Applied Sciences (PIEAS), Islamabad, Pakistan

(Received August 03, 2012 and accepted in revised form September 13, 2012)

To conduct dispersion studies, an experimental facility as a 100 feet high stack is intended to be installed in the premises of a research institute i.e. Pakistan Institute of Engineering and Applied Sciences (PIEAS) in Islamabad. The purpose of this stack is to release an inert gas as a tracer and to collect its ground level concentration data around the source. This data would be helpful in order to validate the modeling results from air dispersion studies. In view of above, present study has two main objectives. First is to evaluate the discharge flow rate of a blower at the manhole of experimental stack which must be capable to produce a maximum velocity of 8m/sec for a tracer emission at the top of the stack. The second aim is to investigate the probability of reversal flow of the tracer entering into stack from an opening located at a proposed height of stack while adjusting the blower flow rate to acquire the desired exit velocity of tracer. To get aforesaid goals, a computer program named FLUENT 6.3 as a Computational Fluid Dynamics (CFD) modeling tool has been employed for the simulation of air flow patterns inside the experimental stack. The results reveal that the blower flow rate of 7650 cfm through manhole of the stack is needed to acquire the exit velocity of 8 m/sec at the top of the stack and there is no possibility of reversal flow of tracer entering from the inlet opening of the stack.

Keywords: CFD techniques, FLUENT, GAMBIT, Experimental stack

1. Introduction

Computational Fluid Dynamics (CFD) uses numerical computing techniques to produce the significant information regarding the ways in which a fluid (liquid or gas) flows under specified conditions. CFD simulation of fluid engineering systems comprises of modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters and grid generations etc). It is employed in the fields of aerospace, automotive, biomedical, chemical processing, hydraulics, marine oil and gas industries etc. [1].

The numerical simulation of industrial processes through CFD techniques is a relatively latest approach that is more powerful modeling tool for process visualization.

Now-a-days there are many advanced computer softwares based on CFD approach to simulate the flow patterns of fluids through processing facilities. These sophisticated softwares can be coupled with some graphical postprocessor for three-dimensional visualization of different flow parameters such as velocity, pressure, temperature and density etc. of flowing fluid [2].

CFD simulations are generally performed into three steps. First step is the pre-processing of the input data for a flow problem. This step involves the preparation of data using a user friendly interface and subsequently its conversion into a particular format compatible to the main CFD program. The second step is the CFD modeling. This step involves performing numerical calculations in the grid cells of computational domain to obtain the flow variables for the domain defined in the pre-processing step. The last step is the post processing of the output data. In this step results obtained from CFD modeling are shown in the form of graphs and plots such as path lines, streak lines, streamlines and vector plots [3].

CFD techniques generate numerical solutions for different kinds of geometrically complex situations. It is a general perception that CFD modeling is less expensive than experimental approaches in many cases. Major advantage of CFD modeling is that a single calculation provides "whole flow field data". In other words, we can say that it generates reliable computational results for various parameters of fluid flow at every spatial node of the geometrical model simultaneously. However, CFD modeling needs special care in order to make the numerical results more reliable.

* Corresponding author : farrukhibrahim@yahoo.com

In this regards, the accuracy of CFD simulations is highly dependent upon two input parameters such as grid resolution and iterative convergence [4].

In present study, FLUENT version 6.3 has been used as a CFD modeling tool. It is one of the most popular CFD softwares that are being used for modeling purpose. It is the standard k-ε turbulence model that was initially developed by Creare Inc. of USA in 1983. The main contributors in the development of this software were Dr. Ferit Boysan at Sheffield University in the U.K, Mr. Bart Patel, a department head at Creare Inc and Michael Engelman in Chicago, USA (http://www.fluent. com /about/history.html). FLUENT solves conservation equations for mass and momentum. For flows in which heat transfer or compressibility is involved, an additional equation for energy conservation is solved. FLUENT has been used for flow patterns simulation of a fluid inside a 100 ft high experimental stack. This experimental facility is planned to be installed at a research institute of Pakistan for conducting tracer experiments. These experiments will be conducted to collect the ground level concentration data of chemical pollutants released from this proposed experimental stack. The collected data from experiments may be used for the validation of computational results from air dispersion studies. There are two main objectives of this investigative study. First is to get a variable concentration and exit velocity (1 m/sec to 8 m/sec) of a tracer at the top of experimental stack to make it as a replica of different industrial sources existing throughout the world. In this regard, to achieve the controlled flow rate of tracer at exit, internal modeling of fluid flow was performed to get the information of variable volumetric flow rate of air through a blower required at the bottom of stack. The second objective of this study is to ensure that there would be no chance of reverse flow of tracer from two inlet openings at proposed locations of the stack while adjusting the flow rate of blower to get required exit velocity of tracer.

2. Methodology

2.1. GAMBIT

GAMBIT stands for Geometry and Model Building Intelligent Toolbox. It is an integrated preprocessor for CFD simulation. It is application software that builds complex geometry and generates a 2D or 3D mesh for it or import existing geometries from various other computer-aided design (CAD) packages. It offers wide range of meshing styles. GAMBIT also helps user to automatically select meshing schemes for structured or unstructured meshing. The meshing scheme depends on the complexity of the geometry models. It can generate high-quality triangular, tetrahedral, pyramids and prisms type meshes. It is recommended that meshing should be fine enough to resolve the physical effects occurring inside the computational domain [5].

2.2.. FLUENT

FLUENT is a state-of-the-art computer program for evaluating fluid flow properties and heat transfer for complex geometries. FLUENT supports different mesh types and also contains the capability to resolve flow problems using unstructured meshes that can be easily generated for complex geometries [6]. This model incorporates two extra transport represent the turbulent properties of the flow. The first transported variable is turbulent kinetic energy, k which determines the energy in the turbulence. The second transported variable is the turbulent dissipation, ϵ which determines the scale of the turbulence [7]. It uses a multi window pane system for displaying a variety of configuration menus and grids. FLUENT is also capable to refine or coarse grid on the basis of flow solution. It deals with compressible and incompressible flow, multiphase flow, combustion, and heat transfer etc.

2.3. Model Flow Equations

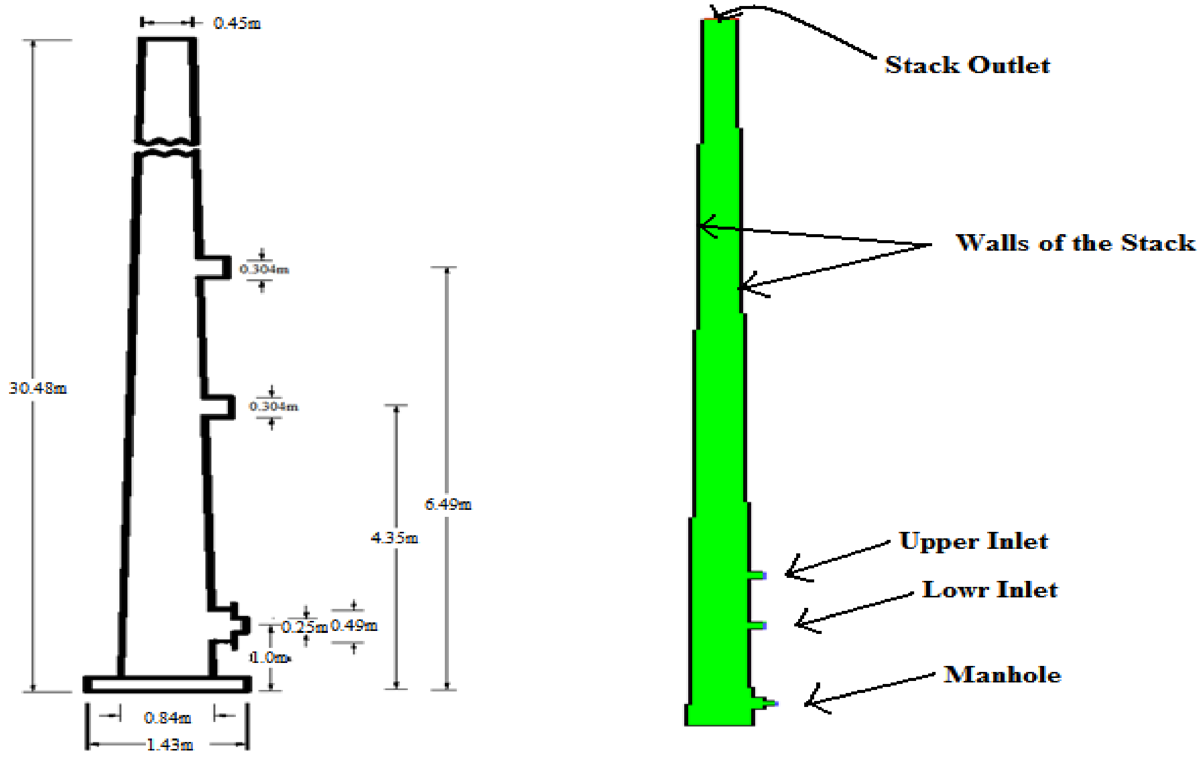
The computation of liquid or gas flow in modern CFD software is performed by numerical solution of the governing equations. These governing equations are named as conservation of mass and momentum, in differential form for incompressible, steady, isothermal, three-dimensional, turbulent flow. The general differential equation for turbulent flows known as Navier-stokes is applied to each cell and discretized can be written as:

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\emptyset dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\emptyset V \, . \, dA}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\emptyset . \, dA}_{\text{diffusion}} + \underbrace{\int_\forall S_\emptyset \, dV}_{\text{generation}}$$

and the second governing equation known as equation of continuity is

$$\frac{\partial\rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$

In these equations, independent parameters are 3 components of velocity u1, u2, u3 in the x, y and z directions, p is the average static pressure, ρ and g are the density and the acceleration of gravity. Besides these two equations, a number of



(a) **(b)**

Figure 1. (a) Dimensions of stack (b) Geometry of stack created with GAMBIT.

other model equations are implemented in the leading CFD software tools. These model equations allow for simulation and computation of compressible flows (sub-, trans- and supersonic), flows with heat transfer (including transfer by radiation), flows with cavitations, flows of a mixture of several fluids, multiphase flows, flows with chemical reactions and combustion etc.

2.4. Modeling Setup

In present work, the flow patterns of a fluid through a 100 ft high stack have been studied using CFD models, GAMBIT version 2.4.6 and FLUENT version 6.3. This stack is an integrated part of a proposed experimental facility in Pakistan. The stack would be used as a general exhaust and an experimental setup as well for conducting tracer experiments.

The first step of preparation of source data for computation of flow is the formation of a geometrical model presenting the area of the stack where the fluid flows. GAMBIT version 2.4.6 has been used with “bottom-up” approach to create geometry of the stack [8]. The dimensions of stack height, its lower and upper diameters as inputs to GAMBIT were taken as 30.484m (100ft), 0.84m (2.75ft), 0.45m (1.5ft) respectively as shown in Figure 1a. For the entrance of a tracer into stack, two inlet openings were modeled with same diameter of 0.304m (1ft) located at the heights of 4.35m (14.3ft) and 6.49m (22.6ft) from ground level. A manhole of diameter 0.496m (1.6ft) has also been provided at the height of 1m (3.3ft) from ground level for the internal inspection and blower ducting. Initially, 28 vertices (x, y) to define the outline of the stack and all the other provisions such as man hole and inlet openings were created with reference to the origin (x_0, y_0) at the left bottom corner of geometrical model. Straight edges were obtained by combining those vertices and subsequently, 4 faces were generated by connecting the 15 edges of stack boundary and 13 edges for the manhole and two inlets in a continuous loop.

To split the computational domain into a set of discrete cells, structural quadrilateral meshing scheme has been applied. Reason for the selection of this type of meshing is that this scheme generates the highest quality of structured meshes in a Cartesian coordinate system and its use reduces the computational time by half when compared with the other meshing schemes. To create structured or mapped meshing, the number of grid nodes was taken equal to those on opposite edges of the faces. As it is recommended that the grid nodes should be located denser at places where sharp changes of flow parameters occurs that is why fine meshing has been used near tracer inlets and manhole openings [5].

To set the boundary type in Gambit, the inlet openings and manhole have been specified as the inflow boundaries and top edge of the stack as outflow boundary. The Rest of the edges were considered as wall boundaries. Figure 1b represents the whole geometry of stack created by GAMBIT. All above mentioned information regarding geometry and meshing of computation domain was then exported to mesh file for FLUENT 6.3.

Starting with FLUENT, grid was first checked to avoid problems due to incorrect mesh connectivity and grid limit size etc. It was observed that there were no errors in the geometry. Then grid was smoothened and swapped to ensure the best possible grid quality for the calculation.

In the next step, under the heading of “Solver”, the options of “pressure based implicit function” and “absolute velocity formulation” were selected. As viscous model, k-ε turbulence model was selected with default settings. Moreover, the non-equilibrium wall function was selected which uses a two-layer approach to include the pressure-gradient effects. The wall neighboring cells are assumed to comprise of a viscous sub layer and a fully turbulent layer. All the simulations were first performed using the first order upwind discretization for the momentum, turbulent kinetic energy and turbulence dissipation rate while the PRESTO discretization (Pressure Staggering Option) was selected for the pressure since it is more appropriate for flow with swirl [9] and the SIMPLE method (Semi-Implicit Method for Pressure-Linked Equations) algorithm, was selected for the pressure velocity coupling. Finally the results were generated using second order of upwind discretization to make them more accurate.

The boundary conditions used for inlet flows from manhole and two inlet openings were taken as the multiples of 1730 cfm (equivalent to 1kg/s) to examine the behavior of static pressure and air velocity through the stack. The outlet flow was obtained from top of the stack. The operating pressure inside the stack has been taken as atmospheric pressure.

3. Results and Discussion

In present study, FLUENT 6.3 has been used as a CFD modeling tool for simulation of flow patterns of a fluid inside a 100 ft high experimental stack. The profiles for the flow parameters such as static pressure and velocity are discussed as follows.

Initially, some simulation cases were run in order to test the sensitivity of model predicted results on the grid meshing or special resolution of computational domain. For this purpose, three different meshing schemes i.e. coarse, medium and fine meshes were used. Medium and fine meshes were obtained by increasing the number of grid nodes by 100 and 200 % of the number of grid nodes selected for the course mesh respectively.

Figures 2a and 2b illustrate the impact of change in mesh scheme on the vertical profiles of air velocity and static pressure at the central axis of stack respectively. From both figures, it is obvious that there is an abrupt change in both parameters uptil 7.5 m height of the computational domain. The mismatch observed among the data curves within 7.5 m of stack height is because of the presence of manhole, two inlet openings and formation of eddy currents extending upto the height of inlet openings. It is mentioned here that the eddy currents are generated due to the side entry of air from manhole at the flow rate of 7650 cfm. The results of figures 2a and 2b highly support the recommendation in [5] that the grid nodes should be located denser at places where sharp changes of flow parameters occur to get more precision of the computational results. The other thing which is very clear from figures is that a significant agreement among three data curves drawn for different meshing schemes can be observed for the remaining height of stack i.e. from 7.5 m to 30.484 m. We can say that the computational results are grid independent for the remaining height of the stack. This thing is very important to ensure the good precision and convergence of the solution [5]. Keeping in view the above results, the numbers of grid points were increased reasonably at the lower part of stack extending up to 7.5 m for further simulations.

Figure 5 shows the vertical profiles of air velocity (m/sec) at central axis of stack at different flow rates (cfm) of blower air entering from man-

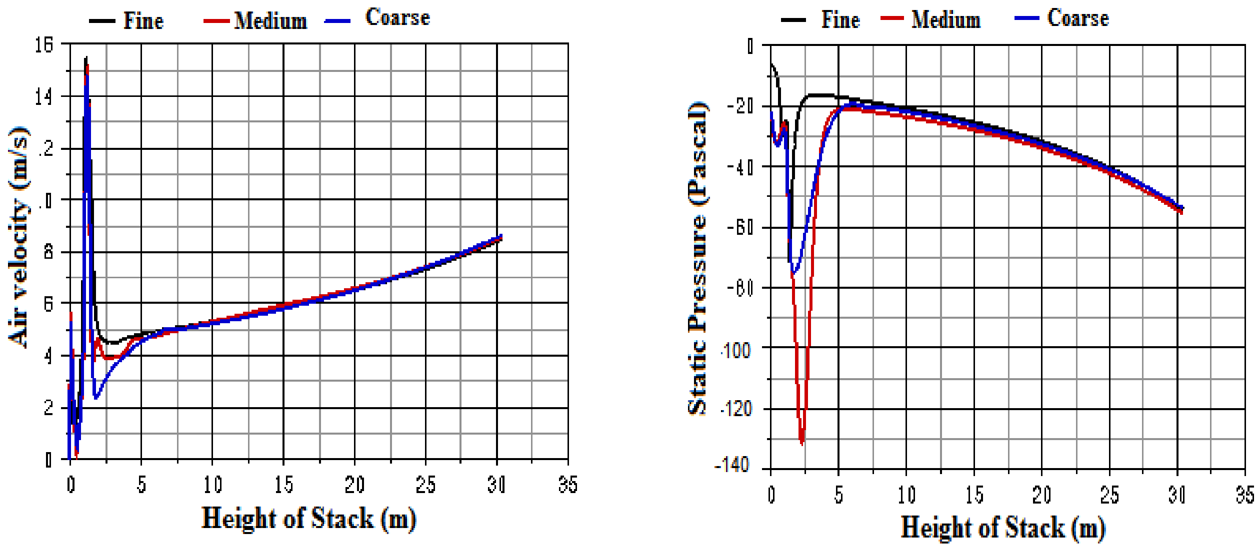


Figure 2. (a) Graph showing the sensitivity of different mesh schemes on air velocity (m/sec); (b) Graph showing the sensitivity of different mesh schemes on static pressure (Pascal).

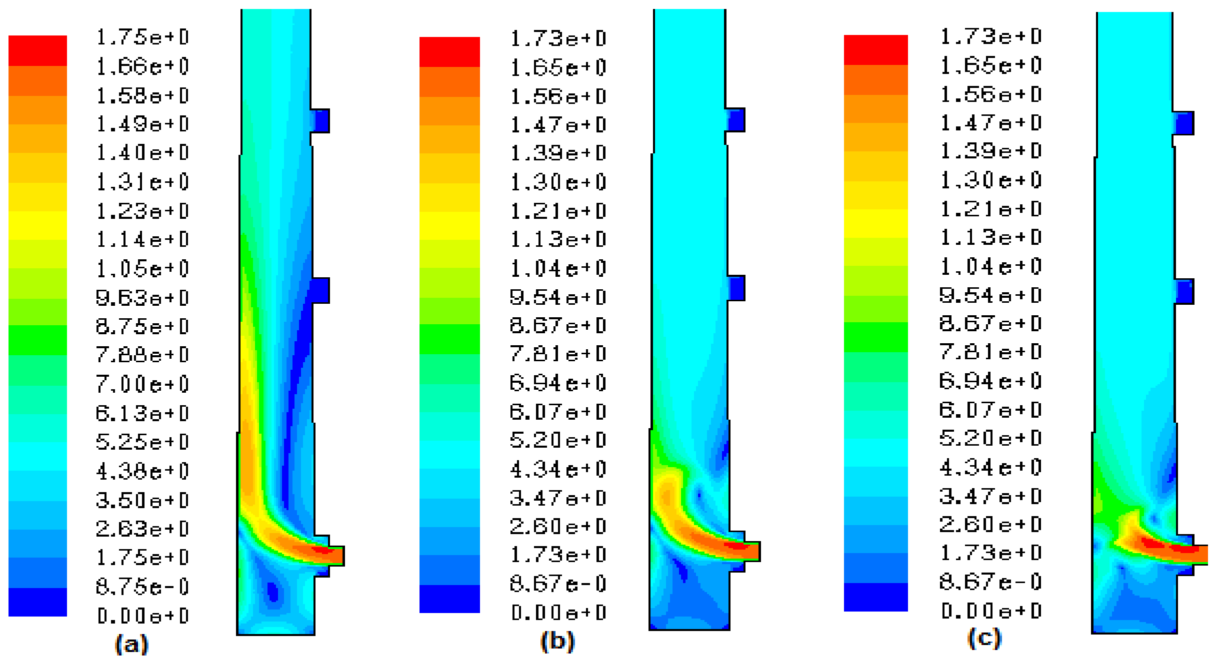


Figure 3. Air velocity (m/sec) contours showing the impact of different meshing schemes such as (a) Course mesh; (b) Medium mesh; (c) Fine mesh.

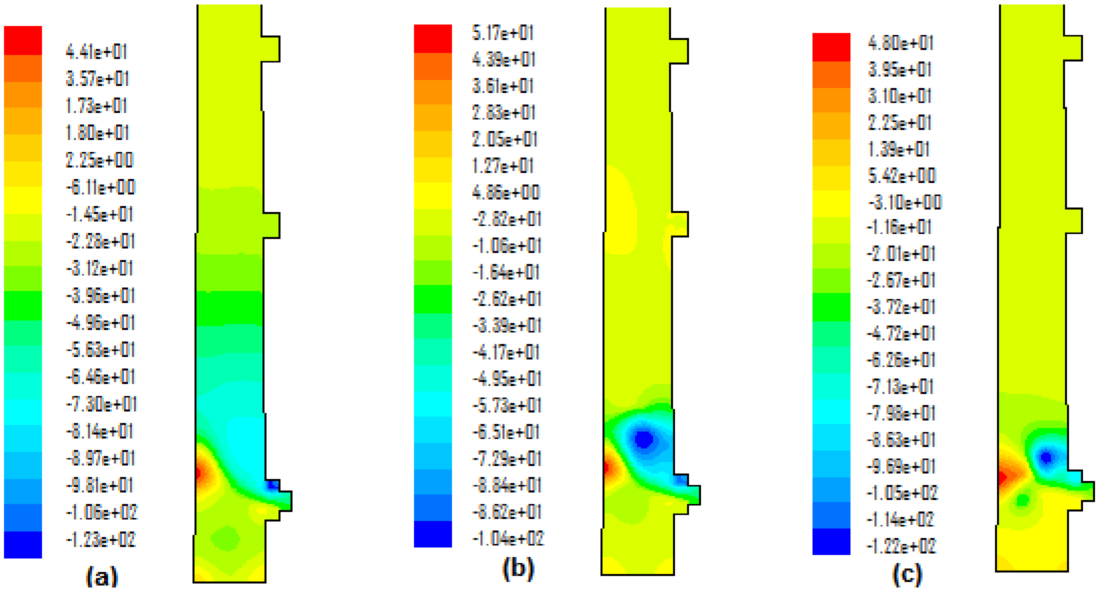


Figure 4. Static pressure (Pa) contours showing the impact of different meshing schemes such as (a) Course mesh; (b) Medium mesh; (c) Fine mesh.

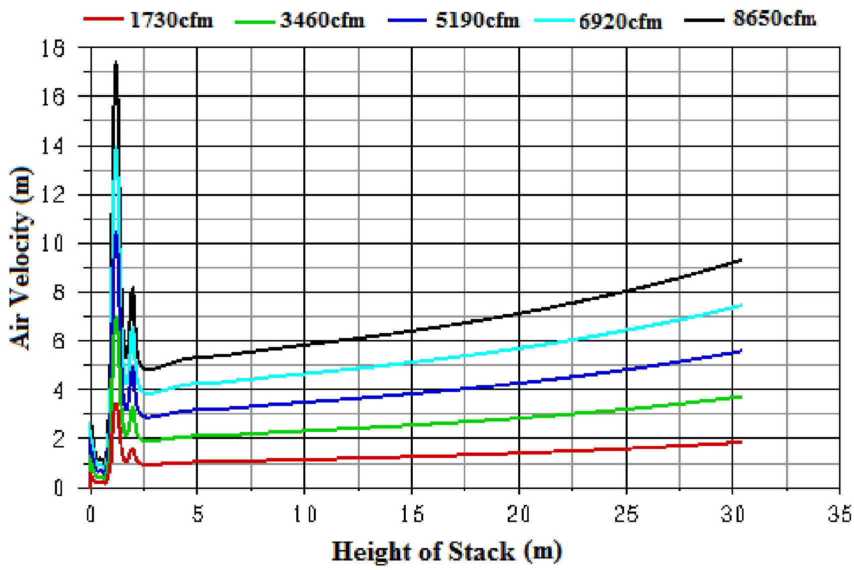


Figure 5. Air velocity (m/sec) profiles with the change in volumetric flow rate (cfm) of air from manhole.

hole. A sudden increase in velocity occurred near the manhole and after that a trend of gradual increase in the said parameter was observed throughout the height of computational domain. It is because of the reduction in cross-sectional area of stack, as the geometry of the stack is conical. It was also observed that increase in air flow rate resulted in the increase in air velocity profile.

As one of the objectives of this study was to determine the such flow rate of air at the location of manhole which results in the maximum exit velocity of 8m/sec at the top of computational domain, therefore, it is clear from figure 6 (a) that the maximum required velocity can be achieved at exit of stack by adjusting the flow rate of 7650 cfm at the location of manhole. Pressure profile at the same flow rate from manhole can be observed from Figure 6 (b).

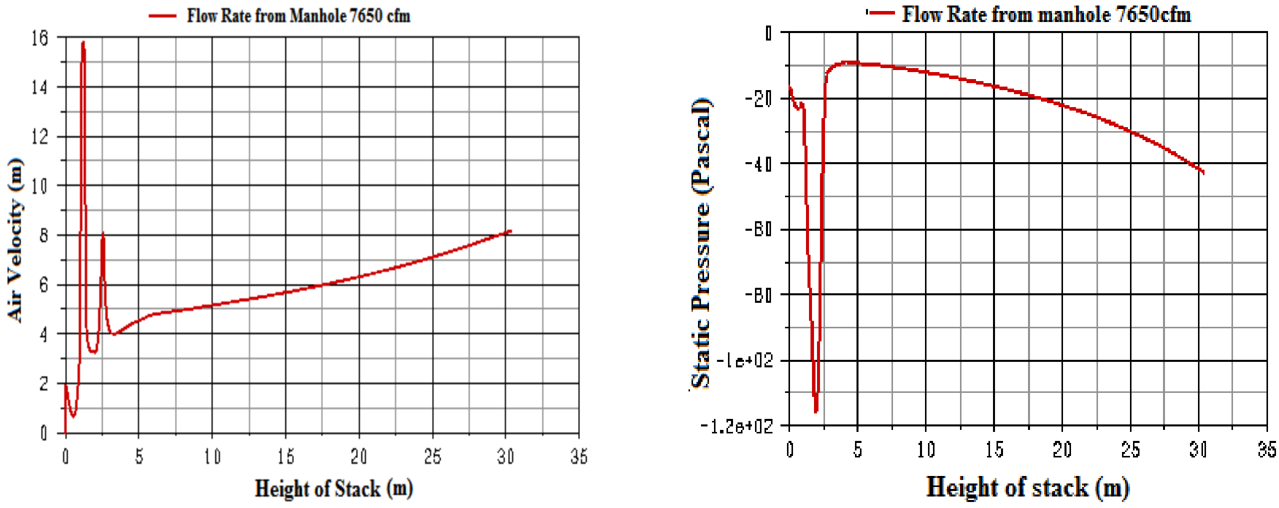


Figure 6a. Air velocity (m/sec) profile at air flow rate (cfm) of 7650 cfm from manhole; (b) Static pressure (Pa) profile at air flow rate (cfm) of 7650 cfm from manhole.

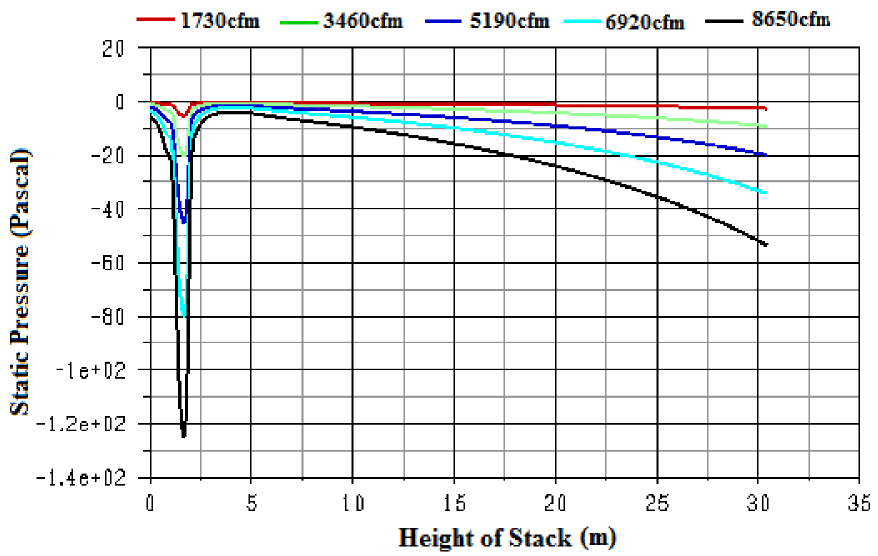


Figure 7. Static pressure (Pascal) profiles with the change in air volumetric flow rate (cfm) from manhole.

Figure 7 shows the vertical profiles of static pressure (Pa) at central axis of stack with different volumetric flow rates (cfm) of air of blower from manhole. A sudden decrease in pressure occurred near the manhole and after that a trend of gradual decrease in the said parameter was observed throughout the height of computational domain. It was also observed that increase in air flow rate decrease the pressure.

After deciding the flow rate of 7650 cfm at manhole for achieving the exit velocity of 8 m/sec at the top of stack, the next step was to ensure that there was no chance for the outflow of air through the inlet openings located above the manhole. Figure 8 shows the velocity contours and velocity vectors of air that enters from manhole and distribute within the whole domain of stack. The contours in figure 8 (a) and the direction of velocity vectors in figure 8 (b) clearly indicate that there is no outward flow of air through the inlet openings located above the manhole. Air strikes the opposite wall of the stack after entering through manhole and then move upward. A layer of greater velocity develops only along the opposite wall and the air layer in contact with the surface containing inlet openings can be observed with less fluid velocity.

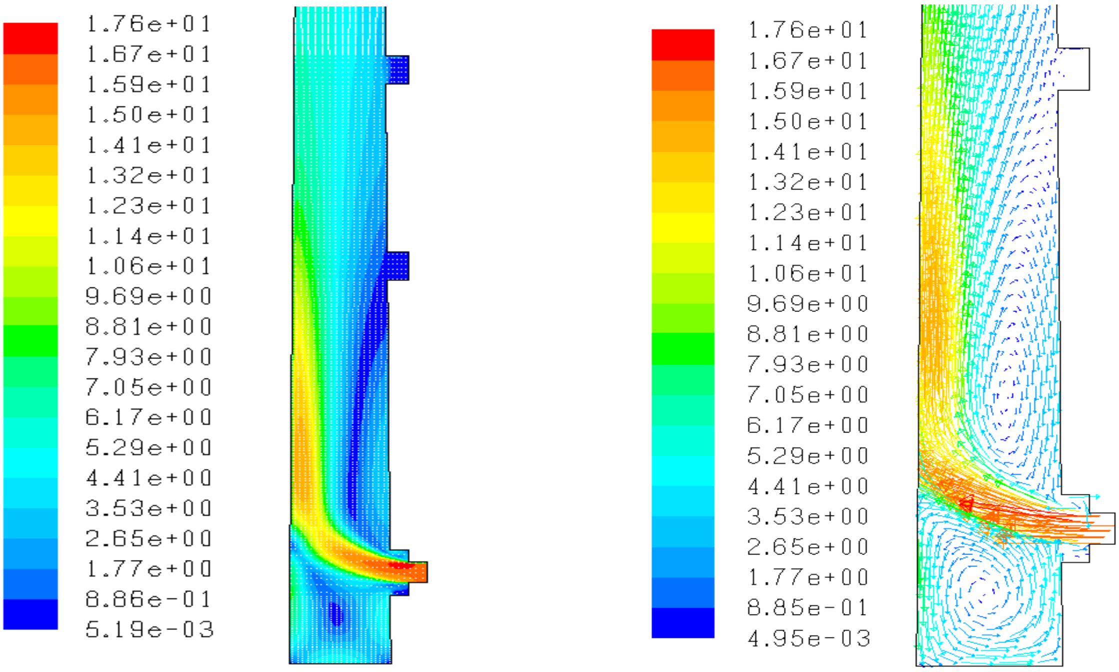


Figure 8a. Velocity contours with air flow rate of 7650 cfm from manhole; (b) Velocity vectors with air flow rate of 7650 cfm from manhole.

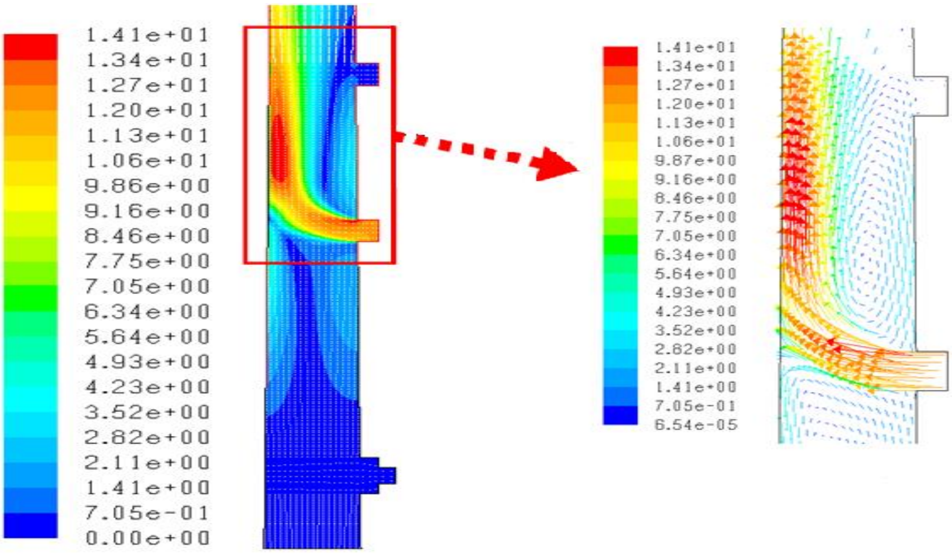


Figure 9. Velocity contours and vectors with air flow rate of 7650 cfm from one of the inlet openings.

Figure 9 shows the velocity contours and vectors of air that enters from first inlet opening just above the manhole and distribute within the whole domain of stack. The direction of velocity vectors in figure 9 once again indicates that there is no outward flow of air through the second inlet opening located above the first opening.

Figure 10 (a) illustrates the results of air velocity profile at central axis of stack with 3825 cfm flow rate of air from each of manhole and lower inlet opening. Two sharp peaks of air velocity were observed in this case due to the volumetric air flow rate given at two openings. A slight increase in air velocity was also found when compared with the results in case of total air flow of 7650 cfm only from manhole as shown in figure 6. Similarly, a more decreasing trend was observed from figure 10 (b) for the static pressure (Pa) of air with flow rate of 3825 cfm from each of manhole and inlet openings.

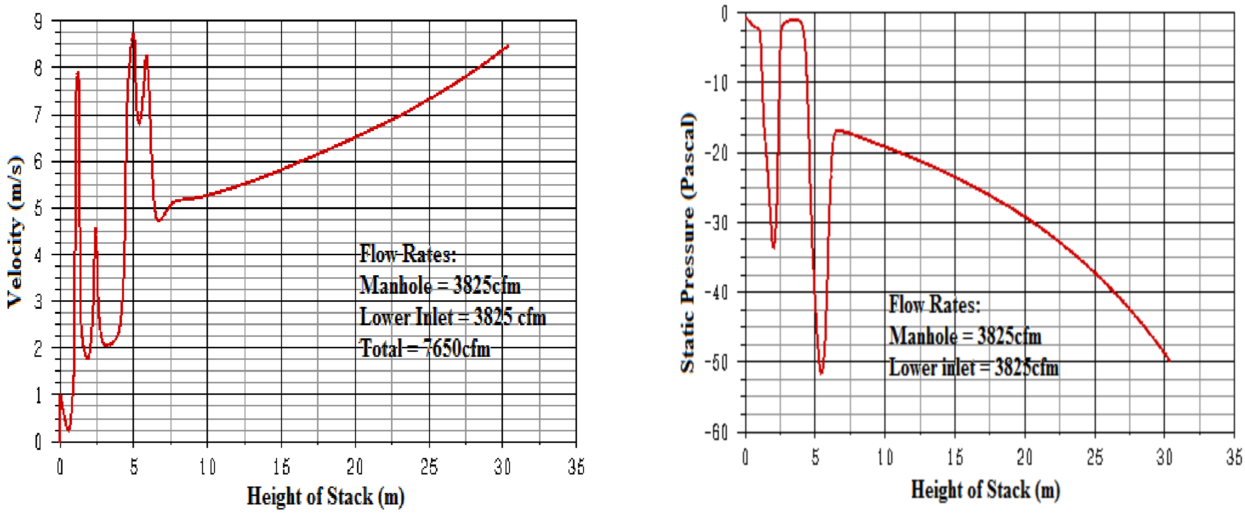


Figure 10. (a) Air velocity (m/sec) profile with 3825 cfm from manhole and lower inlet; (b) Static pressure (Pa) profile with 3825 cfm from manhole and lower inlet.

4. Conclusions

In present study, CFD based computer software FLUENT 6.3 has been used to simulate the flow patterns of air inside a 100 ft high experimental stack to be installed for conducting tracer experiments. The numerical simulation of flow problems through CFD softwares is a relatively latest approach that is more powerful tool for process visualization.

The simulations were performed to get a wide range of exit velocity from 1 m/sec to 8 m/sec at the top of stack with change of air flow rate of blower from the manhole of the stack. It was concluded that the flow rate of 7650 cfm from manhole is required to get the desirable maximum exit velocity. After deciding this air flow rate, it was also ensured that there was no chance for the outflow of air through the inlet openings located above the manhole of the stack.

Since this study was performed for practical purpose, therefore, the validation of computational results may be done after the installation of experimental stack by adjusting a variable air flow rate from blower at the location of manhole and taking the measurement of velocity at the top of stack.

Acknowledgement

The authors highly appreciate the technical assistance provided by the Pakistan Institute of Engineering and Applied Sciences (PIEAS), Islamabad. Authors are also thankful to Mr. Ajmal Sheikh, Ph.D student at PIEAS for his guidance regarding CFD modeling.

References

[1] S. G. S. Ragav, et al., FLUENT & GAMBIT ALCHEMY 2010 Department of Chemical Engineering, National Institute of Technology, Tiruchirrappalli – 620 015

[2] Integration of tracing with computational fluid dynamics for industrial process investigation Final report of a coordinate research project 2001–2003. IAEA-TECDOC-1412

[3] R. Gupta et al., Chemical Engineering Journal **144** (2008) 153.

[4] B. Blocken et al., Journal of Wind Engineering and Industrial Aerodynamics **96**, No. 10-11 (2008) 1817.

[5] N. Alexey et al., Contemporary approach for simulation and computation of fluid flows in centrifugal hydromachines, Sumy State University, Rimsky-Korsakov Str., 2, 40007, Sumy, Ukraine.

[6] Fluent Inc, 2006. FLUENT 6.3 User's Guide. Fluent Inc, Lebanon, NH, USA.

[7] B.E. Launder et al., Lectures in Mathematical Models of Turbulence, Academic Press, London, England.

[8] Fluet Inc. GAMBIT Users Guide, Published by FLUENT Inc (2006).

[9] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington DC. Publishing Corporation (1980).