Investigation of flow field around flanged hoods with Rectangular and Circular cross-sections

M. T. Shervani Tabar[1], M. H. Javareshkian[1], K. Zamzamian[2], R. Dadvand[2]

School of Mechanical Engineering, Tabriz University, Tabriz, Iran
msherv@tabrizu.ac.ir E-mail:

Abstract

In order to control the concentration of pollutants in the industrial workrooms and laboratories, local ventilation systems are widely used. Hood is the most important part of this system that has various types. Investigating the flow field around the hood and the induced velocity at a specified distance from the hood opening is of special importance in designing of hood. In this paper, the airflow around flanged hood's opening with both circular and rectangular cross-sections has been modeled using computational fluid dynamics (CFD). Numerical modeling is accomplished assuming inviscid flow and the result is compared with that of potential correlations exist for circular and rectangular openings. The comparisons show good agreement. At the next step the flow field has been analyzed using $\kappa - \varepsilon$, RNG turbulence model. Results of this case show that although there are differences in velocity vector magnitudes near the opening, generally there is a very small deviation from that of potential flow at centerline. But for points far from the centerline, deviation is significant because of the viscosity and boundary layer effects near the wall. Therefore, considering the real flow field and turbulence for modeling airflow around flanged hoods opening is inevitable.

Key words: Hood, Local Ventilation Systems, CFD, Turbulence, Potential Solution

Introduction

In the industrial workrooms and laboratories, the indoor air quality (IAQ) control must comply with two requests: to provide healthy working conditions and to protect the industrial products from damages. In order to satisfy these requirements, important design instructions and guidelines are defined. In this direction, particular attention is paid to the air ventilation, widely used to contain the airborne contaminants concentrations to sufficiently low levels. The local exhaust ventilation (LEV) systems achieve this aim by generating, towards the exhaust opening, an airflow field that removes the airborne contaminants decreasing their spreading over the workplace air. These systems include a suction hood that is

[1] Assistant Professor
[2] Ph.D studentd

placed at the vicinity of contaminants generating sources.

Hoods are used for controlling pollutants. They control the pollutants by collecting them at same points where they are dispersed. Generally hoods are classified to various types, some of which have been presented in figure 1. Two main types of hoods are flanged and unflanged hoods. In the flanged hood, a solid plane which acts as a wall and is used in order to further control the local suction at the desired area, is placed around the hood opening (fig. 2). In addition the shape of hood opening is one of the important parameters in its design and performance. Various hoods with different opening shapes such as: circular, triangular, rectangular and elliptic have been designed and constructed. In this research, rectangular and circular hood openings are considered, which are most applicable in industry. The Navier-Stokes equations describing the air flow field induced by a suction hood are complex and their exact analytical solutions are provided only in very simple cases and with lots of assumptions. Therefore classical designs of local ventilation systems are based on empirical correlations. One of the important estimations in analytical solution of local ventilation systems is based on potential flow of air. In this model, assuming the fluid to be inviscid and irrotational, the flow field around hood opening is solved using analytical correlations governing potential flow. For some of the hood types, such as circular and rectangular hoods, complete solutions based on potential solution are available. An alternative method, recently developed, is the use of computational fluid dynamics (CFD) to analyze flow field around hood openings. Nowadays, this method can be used for calculating the performance of different local suction ventilation systems and designing new systems with high performance. In this research, circular and rectangular flanged hoods have been considered. Special type of rectangular opening hoods is considered, in which the aspect ratio (a/b) is large. This type of hood, often called slot, is largely diffused, since it includes a wide capture action towards large containments sources.

Numerical modeling has been accomplished assuming the potential flow and the results have been well compared with available analytical solutions. Then the flow field around the openings of slot and circular hood has been analyzed using $k-\varepsilon$, RNG turbulence model. The flow field, particularity the velocity in the centerline of hood opening which is the most important design parameter, investigated. Finally comparison between potential and turbulence modeling solutions shows that, although the velocities obtained from these solutions are different in the regions far fram the hood axis, there are slight differences in the two solutions for hood centerline. Therefore, it is noteworthy that the potential solution for analyzing the flow field around hood opening can be used as a simple, applicable and almost accurate, at the centerline, solution where the actual solution is complex and unavailable.

Potential flow field due to suction in hood opening

In this section, the velocity field assuming potential flow around the suction hood opening is given. The governing equations considering inviscid and irrotational flow are as fallows:

$$\begin{cases} \nabla^2 \phi = 0 \\ \vec{v} = \nabla \phi \end{cases} \quad (1)$$

Where ϕ is the potential function of the flow field. Boundary conditions due to flanged opening are:

$$\begin{cases} \partial \phi / \partial n = 0 \\ \nabla \phi = -v_0 \end{cases} \quad (2)$$

Where v_0 is the suction velocity at hood opening. The complete solution is given by Chen et al. (1997):

$$\vec{V}(\vec{R}) = -\frac{v_0}{2\pi}\int_S \frac{(\vec{R}-\vec{\xi})}{\left|\vec{R}-\vec{\xi}\right|^3}dS \qquad (3)$$

Where $\vec{V}(\vec{R})$ is the velocity vector in terms of location in flow field, stated by $\vec{R}$ and $\vec{\xi}$ vectors.

For the hood with rectangular opening the analytical solution by Taylor and Shepelev (1970) has been given as [2]:

(4)

$$V_x(x,y,z) = -\frac{V_0}{2\pi}\ln[\frac{y+h/2+\sqrt{(x-w/2)^2+(y+h/2)^2+z^2}}{y+h/2+\sqrt{(x+w/2)^2+(y+h/2)^2+z^2}} \times\frac{y-h/2+\sqrt{(x+w/2)^2+(y-h/2)^2+z^2}}{y-h/2+\sqrt{(x-w/2)^2+(y-h/2)^2+z^2}}]$$

$$V_y(x,y,z) = -\frac{V_0}{2\pi}\ln[\frac{x+w/2+\sqrt{(x+w/2)^2+(y-h/2)^2+z^2}}{x+w/2+\sqrt{(x+w/2)^2+(y+h/2)^2+z^2}} \times\frac{x-w/2+\sqrt{(x-w/2)^2+(y+h/2)^2+z^2}}{x-w/2+\sqrt{(x-w/2)^2+(y-h/2)^2+z^2}}]$$

$$V_z(x,y,z) = -\frac{V_0}{2\pi}\{\tan^{-1}\frac{(x+w/2)(y+h/2)}{z\sqrt{(x+w/2)^2+(y+h/2)^2+z^2}} -\tan^{-1}\frac{(x-w/2)(y+h/2)}{z\sqrt{(x-w/2)^2+(y+h/2)^2+z^2}} -\tan^{-1}\frac{(x+w/2)(y-h/2)}{z\sqrt{(x+w/2)^2+(y-h/2)^2+z^2}} +\tan^{-1}\frac{(x-w/2)(y-h/2)}{z\sqrt{(x-w/2)^2+(y-h/2)^2+z^2}}\}$$

In the above correlations, the rectangular cross section opening hood, is in x-y plane and with $w \times h$ dimensions. The velocity in the centerline is [1]:

$$V_z(z) = -\frac{2V_0}{\pi}\tan^{-1}(\frac{wh}{2z\sqrt{w^2+h^2+4z^2}}) \qquad (5)$$

For circular opening flanged hood, the following correlation is available for centerline velocity (Drakel, 1971) [1]:

$$V_z(z) = -V_0(1-\frac{z/D}{\sqrt{0.25+(z/D)^2}}) \qquad (6)$$

Where D is the diameter of circular hood opening.

Grid generation

A 3D geometry is modeled by a grid generator GAMBIT. As it can bee seen in fig. 3, the air duct along with a portion of hood opening surrounding have been simulated to investigate the hood's suction effects. Dimensions of $(0.6\times 0.3)m^2$ for rectangular opening hood and a radius of 0.15 m for circular opening hood have been considered. One essential feature in any CFD work is grid independency of the solution method. To accomplish this, three types of grid namely; fine, medium and coarse grids have been considered and represented in table 1.

In the rectangular hood geometry, the hood opening is located at y-z plane and and its center is located at the coordinate center. In the circular one, the opening is located at x-y plane and z-axis is perpendicular to it. Noting the great importance of centerline in analyzing flow field of hoods, the grid is finer around the centerline and the hood opening. In addition, to investigate the effects of boundary layers and walls on turbulence solution, a finer grid is used near the flange.

To investigate the suction effects, a constant velocity condition has been applied in the exit opening of air duct. The wall boundary condition has been considered for duct and flange walls and a fixed pressure boundary condition is assigned at free stream surfaces. In rectangular opening hood grid the elements type is Hexahedron and the grid type is Submap, while in circular one the elements type is Hexahedron/wedge and the grid type is Cooper.

Numerical modeling

Numerical modeling in this research is performed by the commercial code FLUENT 6.0.12. In order to compare with analytical potential solution, inviscid model has been applied. Then the turbulence flow field has been modeled using the $k-\varepsilon$, RNG turbulence model. The adopted fluid is air. For rectangular opening hood with $0.6 \times 0.3 m^2$ dimensions the suction velocity at the exit is about 2 m/s and for circular one with a radius of 0.15 m the suction velocity is about 1.14 m/s, so that the $\mathrm{Re}_{D_H} = 2.28 \times 10^4$ in both cases. The segregated implicit solver and QUICK discretisation scheme are used. The momentum and continuity equations are coupled via the SIMPLE algorithm. Standard wall functions were adopted in the turbulence model. To determine fluctuations in boundaries, the known turbulence intensity and hydraulic diameter are used. At the exit of the suction duct the following correlation is used to determine the percentage of the fluctuation's intensity [6]:

$$I = \frac{u'}{u_{ave}} \cong 0.16(\mathrm{Re}_{D_H})^{-1/8} \qquad (7)$$

Results

In the first section of this research, assuming the flow field to be inviscid, velocity and pressure values in the two assumed geometries have been obtained. Since there is no rotation of flow in the centerline of the hood opening, (i.e. vorticity = 0) it is expected that the results of inviscid solution via Fluent agree well with those of analytical potential flow modeling.

The grids used in numerical solution involve 90,000 and 90200 elements for rectangular and circular hood, respectively. The velocity values in centerline obtained from inviscid solution by Fluent have been compared with those obtained from equations (5) and (6) and as it can be seen in figures 4 and 5, there is good agreement between them. Fig 6 and 7 show the static pressure and velocity magnitude contours at x-z plane for rectangular hood.

After modeling the inviscid flow and comparing the results with those of analytical potential solution, the $k-\varepsilon$, RNG turbulence model is used to model the suction flow around the hood opening. In order to accomplished the grid independency, results on a line far from the hood axis and parallel to it, using three different grids (table 1) have been compared with each other as are shown in figures 8 and 9. The static pressure and velocity magnitude contours at x-z plane for circular opening hood have been shown in figures 10 and 11. To compare the turbulence solution with that of potential one the velocity vector magnitude in centerline and on a line out of the hood axis and parallel to the centerline of a rectangular opening hood (on line y=0.3, z=0.02) have been compared in figures 12, 13 and 14 respectively. According to these figurers it can be said that in the centerline of the hood opening, although there is slight differences between velocity values near the opening (fig. 15), in general small differences can be seen between turbulence solution and potential correlations (5) and (6). But for the eccentric case (fig. 14) due to viscous effects and boundary layer formation near the wall, there is big difference between results of the two solutions.

Conclusion

In this research, the hydrodynamic field around the opening of circular and rectangular flanged hoods was investigated. The analytical approximations based on the potential flow solutions are available in Tyaglo and Shepelev (1970) and Drakel (1971) for rectangular and circular openings respectively. The numerical modeling is investigated assuming inviscid flow field and the obtained results from this method in the centerline of hood opening are compared with available potential solution. It is found that there are good agreement between these results. This is due to the irrotational flow in the centerline. At the next step to model this phenomena accurately, the $k-\varepsilon$, RNG turbulence model is used. The static pressure and velocity magnitude contours and velocity magnitude diagrams along the centerline and along a line out of and parallel to the hood axis are reported. The results show that although there are small differences between velocity magnitude values near the opening, in general slight differences can be seen between turbulence solution and those obtained from potential formulations of Tyaglo & Shepelev (1970) for rectangular opening and of Drakel (1971) for circular opening in the centerline. For the line out of the hood axis however, due to viscous effects and boundary layer formation near the walls there is significant difference between potential and turbulence solutions. Strictly speaking, at similar conditions one can use the potential flow in the centerline of the hood opening, provided that there is no possibility to perform accurate turbulence modelings. But to obtain the values of the flow properties at the entire computational domain, numerical solution of turbulence flow is recommended.

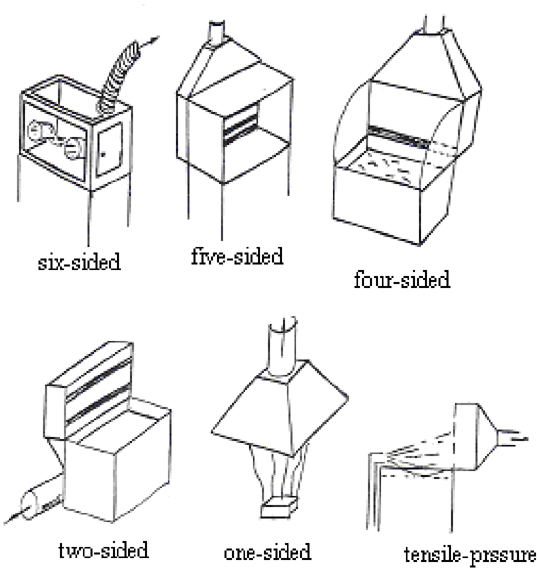

Fig. 1: The shapes of various types of hoods [5].

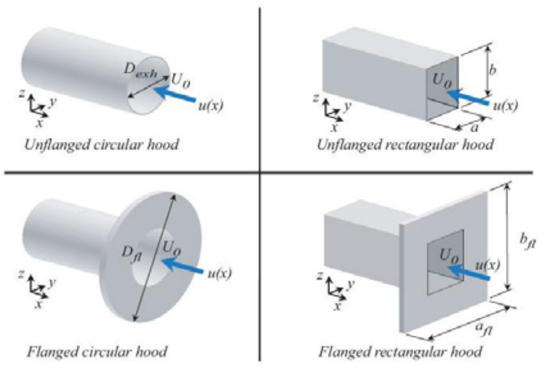

Fig. 2: Flanged and unflanged hoods [1].

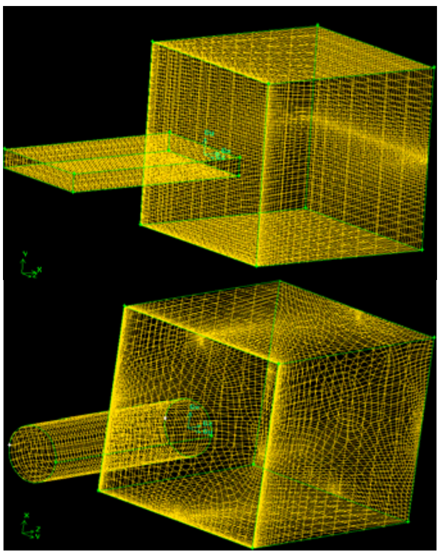

Fig. 3: Computational domain and the grid generated by GAMBIT.

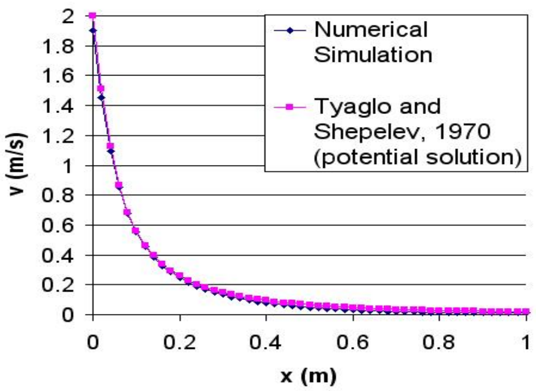

Fig. 4: Comparison between numerical inviscid and analytical potential solutions in the centerline of the rectangular opening hood.

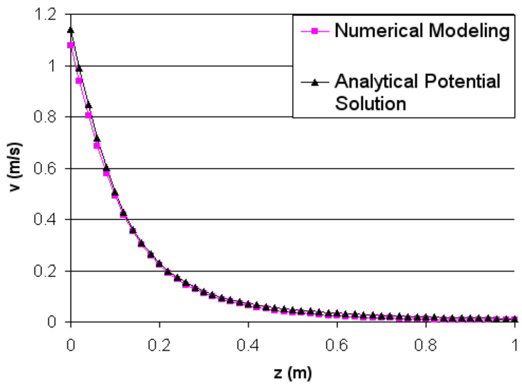

Fig. 5: Comparison between numerical inviscid and analytical potential solutions in the centerline of the circular opening hood.

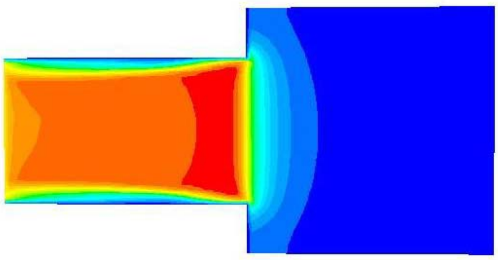

Fig. 6: Velocity magnitude contours at x-z plane for rectangular opening hood.

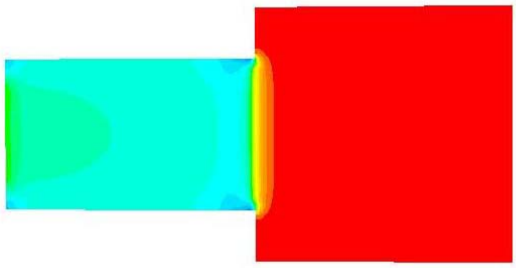

Fig. 7: Static pressure contours at x-z plane for rectangular opening hood.

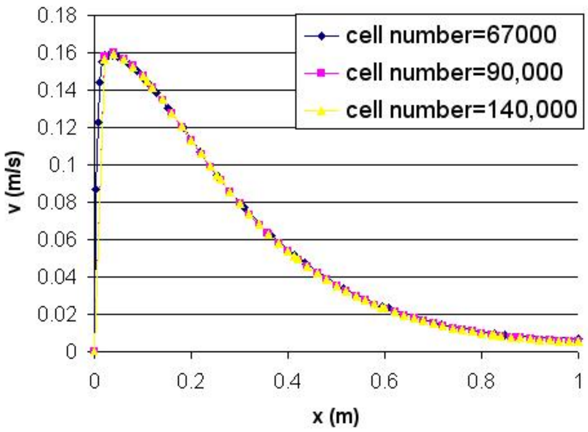

Fig. 8: Assessment of grid independence for numerical modeling of rectangular opening hood.

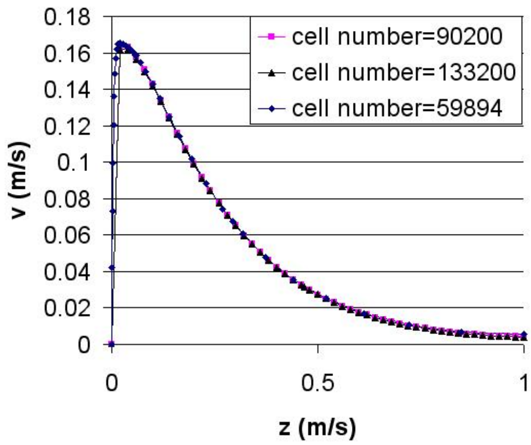

Fig. 9: Assessment of grid independence for numerical modeling of circular opening hood.

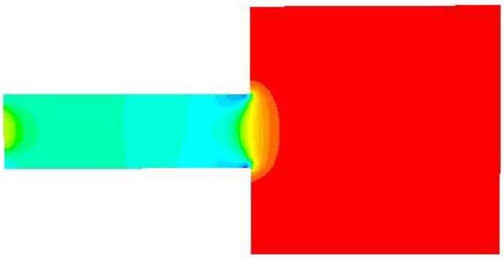

Fig. 10: Static pressure contours at x-z plane for the circular opening hood.

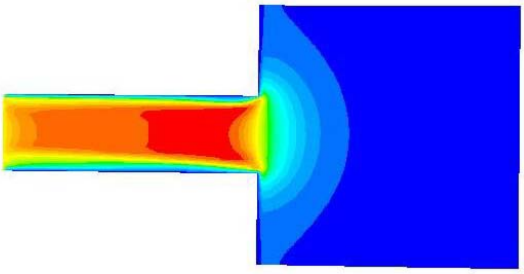

Fig. 11: Velocity magnitude contours at x-z plane for thecircular opening hood.

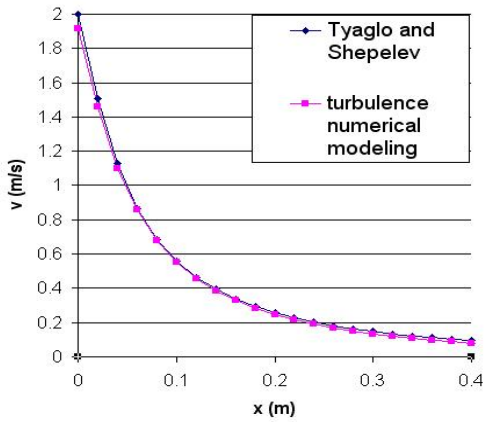

Fig. 12: comparison of velocity magnitudes in centerline obtained from analytical potential and numerical turbulence solutions for the rectangular opening hood.

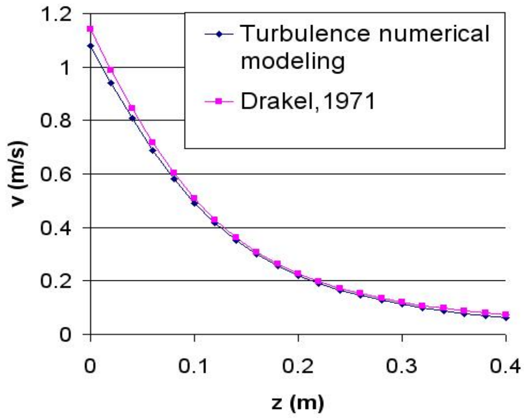

Fig. 13: Comparison of velocity magnitudes in the centerline obtained from analytical potential and numerical turbulence solutions for the circular opening hood.

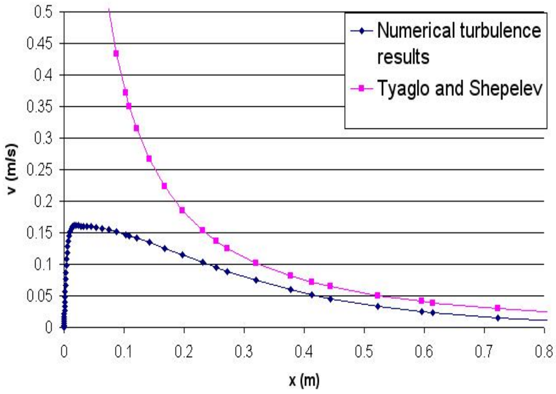

Fig. 14: Comparison of velocity magnitudes on line (y=0.3, z=0.02) obtained from analytical potential and numerical turbulence solutions for the rectangular opening hood.

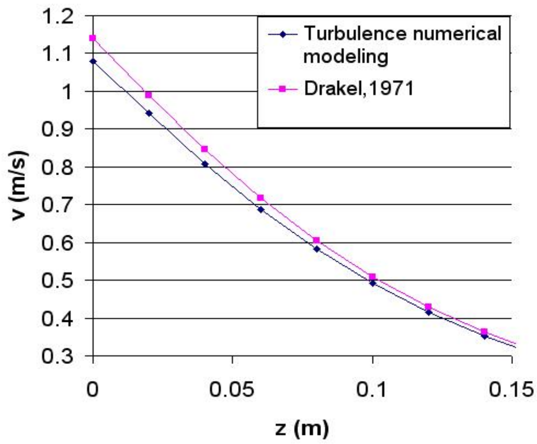

Fig. 15: Comparison of velocity magnitudes in centerline obtained from analytical potential and numerical turbulence solutions for the circular opening hood, near the opening.

Table 1: Properties of the applied grid.

rectangular opening hood		
coarse grid	medium grid	Fine grid
67000	90000	140000
Hexahedron	Hexahedron	Hexahedron
Submap	Submap	Submap
circular opening hood		
coarse grid	medium grid	Fine grid
59894	90200	133200
Hex/Wedge	Hex/Wedge	Hex/Wedge
Cooper	Cooper	Cooper

References

1. Gubler, D., 2002, "REEXS-Reinforced Exhaust System Optimization of Operating and Design Parameters", Doctoral Thesis, Swiss Federal Institute of Technology, Zurich.
2. Chen, Y. K., Chung, K. C., Yeh, W. Y., Chen, C. W., 1998, "The Potential Flow Field Generated by the Flanged Hood Openings of Various Shapes", Institute of Occupational Safety and Health, Council of Labor Affairs, Department of Mechanical Engineering, National Yunlin University of Science and Technology.
3. Kulmala, L., 1998, "Jet Enhanced Exhaust Hood", Tamper, Finland.
4. Wen, X., Ingham, D. B., 2000, "Theoretical and Numerical Predictions of Two-dimensional Aaberg Slot Exhaust Hood", Ann. Occup. Hyg., Vol. 44, No. 5, PP. 375-390.
5. Pur-Mahabadian, M., 1990, "Industrial Ventilation", in Persian, Jahad-e-Daneshgahi Press, Teran, Iran.
6. Soltani, M., Rahimi Asl, R., 2003, "Computational Fluid Dynamics Using FLUENT Software", in Persian Tarrah Press, Tehran, Iran.