Journal Homepage: - www.journalijar.com
INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)
Article DOI: 10.21474/IJAR01/16601
DOI URL: http://dx.doi.org/10.21474/IJAR01/16601
ISSN NO. 2320-5407

RESEARCH ARTICLE

IMPROVEMENT OF THE SECURITY SYSTEM AGAINST POLLUTANTS USING A CIRCULAR CONE

Z. Yahya
Department Of Physics, College of Science, Qassim University, Buraidah, Saudi Arabia.

……………………………………………………………………………………………………....

Manuscript Info

Manuscript History
Received: 31 January 2023
Final Accepted: 28 February 2023
Published: March 2023

Key words:-
Thermal Plume, Security System, Numericall Modeling, Fluent Software, Vertical Cone

Abstract

This study focuses on the improvement of a security system against the pollutants dispersing from Industrial chimneys. In order to simulate the system, we study numerically a turbulent flow generated by a heated open cone at the extremities. Different control parameters were used to improve the thermal and dynamic behavior of the considered flow. In this study, the interaction of pollutants with its surrounding environment simulated by the cone walls was investigated. This study was focused on convection, which is natural, turbulent, steady, 2-D, incompressible flow passing through a vertical conical channel with a heated source at the center of the cone inlet that performed uniform heating at a constant temperature. The interior of the conical wall channel was reheated via the thermal radiation emitted by the hot source. The studied system was mathematically presented by the continuity, Navier stokes, energy and K-epsilon equations. The computational fluid dynamics analysisand ANSYS-FLUENT software were used. The optimal cone can provide further control of the pollutant's velocity at the exit. The thermal and dynamic profiles exhibitedgood stability and homogeneity at the cone exit.

……………………………………………………………………………………………………....

Introduction:-

The dispersion of pollutants considered in this study is an application of the interaction of a turbulent thermal plume with its environment. This phenomenon is presented in this study by the thermal plume induced by the hot plume source and surrounded by a heated cone.

Many scholars have experimentally studied thermal plumes to characterize the general structure of this flow. Although a large amount of research has been conducted to simulate the natural convection between two vertical plates, there is, however, a crucial lack of numerical investigations on natural convection in a conical channel. C. F. Kettleborough [1], has described transient natural convectionin open-ended vertical channels. Results are obtained for Pr = 0.733 and Grashof number Gr=100 and 10000. He showed that the maximum temperature on the centerline at the channel's exit increases as the Grashof number decreases, he found that for larger Grashof numbers the flow reaches the steady state faster. Laminar flow between two parallel vertical walls heated symmetrically was studied by V. I. Terekhov et al. [2]. They aimed to determine the effect of aspect ratio on natural convection. The study was for Pr=0.7, and Rayleigh number ranges$\mathrm{Ra} = 10$ to10^5.Navier-Stokes equations were solved by the finite volume method. Their findings revealed that the temperature of the flow inside the channel increases with the increase of the aspect ratio while the Nusselt number decreases. Moreover, Reynold (Re) number increases when the Rayleigh number and aspect ratio (AR) increase due to the buoyancy effect. Finally, as Rayleigh and Reynolds number values

Corresponding Author:- Z. Yahya
Address:- Department Of Physics, College Of Science, Qassim University, Buraidah, Saudi Arabia.

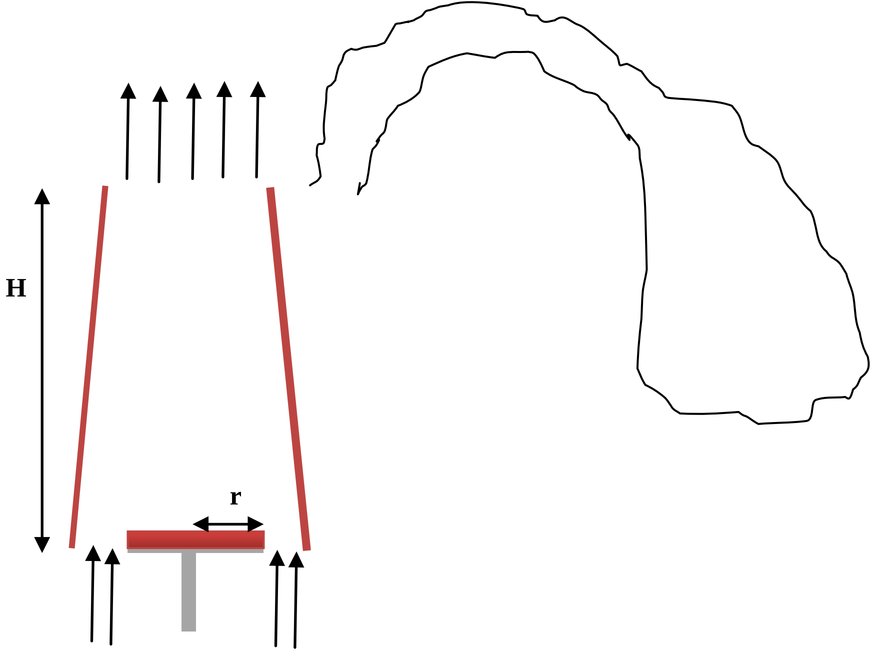
increased the Nusselt number value increased. A laminar buoyant plume created by a heated sphere inside a cylinder was simulated by S. K. S. Boetcher et al. [3]. Their simulation was for air Pr=0.7 and Grashof number varying between 50 to $5\text{x}10^6$. The effect of the height of the solution domain was examined for domain heights Z=11, 20, and 30. They showed that the velocity of the plume increases with elevation but approaches a fully developed state at sufficiently high elevation and the temperature decay with the increase of elevation and attains a self-similar shape at high elevation. Finally, they found that the similarity solutions exist at high elevations above the heat source, and the approach to reach the similarity solution is faster at low Grashof numbers. Seung-Min Ohk et al. [4] examined the influences of different parameters (length, diameter, and Prandtl number) on the natural convection flow inside a vertical pipe. The study was conducted for diameters varying between 0.003m to 0.03m, the length varying between 0.2m to 1m, Prandtl number Pr=0.7, 20, 1 and, local Grashof number $\mathrm{Gr_L} = 3.4 \times 10^8 \text{to } 4.2 \times 10^{10}$.Numerical analysis was done using FLUENT 6.3. They found that, for all Prandtl numbers, as the diameter increase the heat flux increased and, as the length of the channel increased the heat flux decreases. The experimental study of the global structure of the flow of natural convection generated by a hot source placed at the entrance of an open vertical cylinder was carried out by A. O. M. Mahmoud [5]. The author has shown that the vertical evolution of the flow can be divided into three zones. A first zone is characterized by the formation of rotating rolls above the hot source, a second zone of the turbulence transition and, a third zone where the flow becomes established in the upper part of the cylinder. The region at the wall vicinity was not discussed in this study.

This study aimed to investigate numerically the improvement of the security system against the dispersion of pollutants using the cone shape, the development of a thermal plume inside a cone is justified owing to its role in improving the understanding of the mechanisms involved in this application in industry. In addition to its importance from a fundamental perspective, the utilization of a cone allows hot air to be strongly evacuated at the exit of the flow [6]. The flows investigated in this study were natural convection, turbulent, steady, two-dimensional (2-D), and incompressible flows. Further, mathematical modeling based on the Navier-Stokes equations was conducted and the velocity-pressure coupling was resolved using the SIMPLE algorithm. The general structure and thermal and dynamic behavior of the flow were obtained using ANSYS-FLUENT software.

Numerical Modeling and Geometric Configuration:-

This section mainly presents the problem of the study system, wherein the natural convection flow is turbulent. This resulted in the governing equation of a turbulent flow from which we deduced the mean dimensionless equations. Subsequently, through discussions on a k-ε model, this study focused on K-epsilon (2 equation) and deduce that the k–ε equations must render them in a dimensionless form.

Consequently, we present discretization equations followed by an explanation for a meshing step, which affects the fluid flow by partitioning a continuous geometry into discrete control volumes and swiftly calculating the physical quantities. Finally, we used SIMPLE algorithm to solve them.



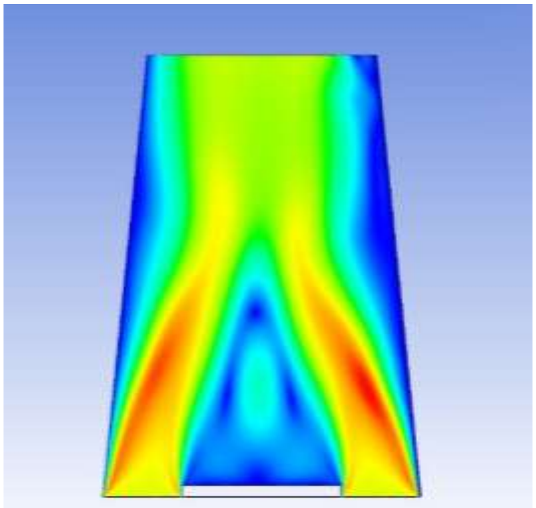


Figure1:- The proposed security system

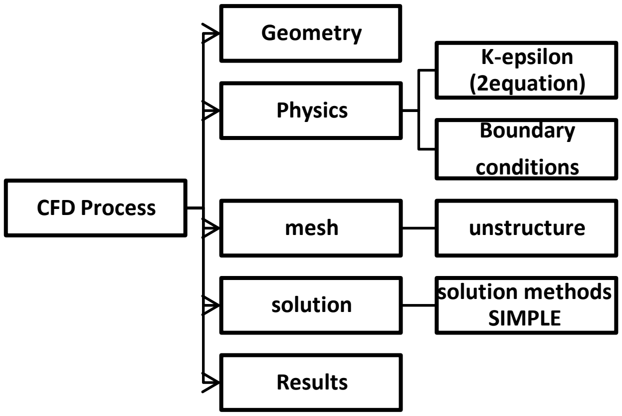


Figure 2:- The study's problem plan.

Governing equation of turbulent thermal plume:-

Dimensionless equations

In fluid mechanics problems, the governing equations are rewritten in a dimensionless form to simplify them, where the quantities in the dimensionless equations no longer have units.

The dimensionless governing equations for turbulent flow is obtained as the equation of continuity:

The equation of continuity:

$$\frac{1}{R}\frac{\partial}{\partial R}(RV_R)+\frac{\partial}{\partial Z}(V_Z)=0 \qquad (1)$$

The equation of r-momentum:

$$\left(V_R\frac{\partial V_R}{\partial R}+V_Z\frac{\partial V_R}{\partial Z}\right)=-\frac{\partial P}{\partial R}+Pr\frac{\partial}{\partial Z}\left[\left(\frac{\partial V_R}{\partial Z}+\frac{\partial V_Z}{\partial R}\right)\right]+Pr\frac{\partial}{\partial R}\left[2\frac{\partial V_R}{\partial R}-\frac{2}{3}\left(\frac{\partial V_Z}{\partial Z}+\frac{1}{R}\frac{\partial}{\partial R}(RV_R)\right)\right]+Pr\frac{2}{R}\left(\frac{\partial V_R}{\partial R}-\frac{V_R}{R}\right) \qquad (2)$$

The equation of z-momentum:

$$\left(V_R\frac{\partial V_Z}{\partial R}+V_Z\frac{\partial V_Z}{\partial Z}\right)=-\frac{\partial P}{\partial Z}+Pr.Ra.T+Pr\frac{\partial}{\partial Z}\left[2\frac{\partial V_Z}{\partial Z}-\frac{2}{3}\left(\frac{\partial V_Z}{\partial Z}+\frac{1}{R}\frac{\partial}{\partial R}(RV_R)\right)\right]$$

$$+Pr\frac{1}{R}\frac{\partial}{\partial R}\left[R\left(\frac{\partial V_R}{\partial Z}+\frac{\partial V_Z}{\partial R}\right)\right] \qquad (3)$$

The equation of energy:

$$\left(V_R\frac{\partial T}{\partial R}+V_Z\frac{\partial T}{\partial Z}\right)=\frac{\partial}{\partial Z}\left(\frac{\partial T}{\partial Z}\right)+\frac{1}{R}\frac{\partial}{\partial R}\left(R\frac{\partial T}{\partial R}\right)+\frac{Ec}{Ra}\left[\frac{1}{Pr}\left(V_Z\frac{\partial P}{\partial Z}+V_R\frac{\partial P}{\partial R}\right)+\Phi_v\right] \quad (4)$$

where

$$\Phi_v=\left[2\left(\frac{\partial V_Z}{\partial Z}\right)^2+2\left(\frac{\partial V_R}{\partial R}\right)^2+2\left(\frac{V_R}{R}\right)^2+\left(\frac{\partial V_Z}{\partial R}+\frac{\partial V_R}{\partial Z}\right)^2-\frac{2}{3}\left(\frac{\partial V_R}{\partial R}+\frac{\partial V_Z}{\partial Z}\right)^2\right] (5)$$

The $\boldsymbol{k}$-equation:

$$\frac{1}{R}\frac{\partial}{\partial R}\left[R\left((\overline{V_R}K)-\left(Pr+\frac{\mu_t}{\alpha_k}\right)\frac{\partial K}{\partial R}\right)\right]+\frac{\partial}{\partial Z}\left[(\overline{V_Z}K)-\left(Pr+\frac{\mu_t}{\alpha_k}\right)\frac{\partial K}{\partial Z}\right]=-RaPr\lambda_t\frac{\partial\overline{T}}{\partial Z}-\epsilon$$

(6)

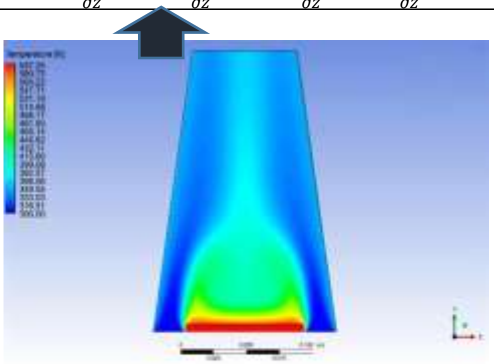
The $\boldsymbol{\varepsilon}$-equation:

$$\frac{1}{R}\frac{\partial}{\partial R}\left[R\left((\overline{V_R}\epsilon)-\left(Pr+\frac{\mu_t}{\alpha_\varepsilon}\right)\frac{\partial\epsilon}{\partial R}\right)\right]+\frac{\partial}{\partial Z}\left[(\overline{V_Z}\epsilon)-\left(Pr+\frac{\mu_t}{\alpha_\varepsilon}\right)\frac{\partial\epsilon}{\partial Z}\right]=-\frac{\epsilon}{K}\left[RaPrC_{\varepsilon1}\lambda_t\frac{\partial\overline{T}}{\partial Z}-C_{\varepsilon2}\epsilon\right] \qquad (7)$$

Boundary conditions:-

To obtain an exact solution, first, the initial and boundary conditions must be identified.

<table>
<tr><td colspan="3">At the outlet of the cone (z=L),
$\frac{\partial v_r}{\partial z}=0,\ \frac{\partial v_z}{\partial z}=0,\ \frac{\partial p}{\partial z}=0,\ \frac{\partial T}{\partial z}=0 \qquad if V_2>0$</td></tr>
<tr><td>At the walls
(r=a)

$v_r=0,$
$v_z=0,$
$T=T_w(z)$</td><td></td><td>On the axis of the cone

$\frac{\partial v_r}{\partial r}=0,$
$\frac{\partial v_z}{\partial r}=0,$
$\frac{\partial T}{\partial r}=0$</td></tr>
<tr><td colspan="3">t the entrance of the cone (z=0)
$0 \quad \leq r_s;\ v_r=0,\ v_z=0, \qquad T=T_s$
$r_s\leq r\leq a;\ v_r=0,\ \frac{\partial v_z}{\partial z}=0, \qquad T=T_0$</td></tr>
</table>

Figure 3:Boundary Conditions of the Geometry.

Results:-

We used in this numerical study the critical parameters indicated by A.O.Mahmoud et al. [6] in their study. Their experimental results show that the critical parameter that controls the thermal and dynamic behavior of the flow is the ratio of the cone radius C=0.65, the radius ratio R*=0.6, and the shape ratio A=0.75 which is defined as the inlet diameter divided by the height of the cone. In this study, the parameters are A=0.75, C=0.6, and the R*=0.6 with Prandtl number of the air 0.72 and Rayleigh number $\boldsymbol{Ra} > \mathbf{10^{10}}$ also Grashof number $\boldsymbol{Gr_a} > \mathbf{1.5} \times \mathbf{10^{9}}$ for turbulent flows.

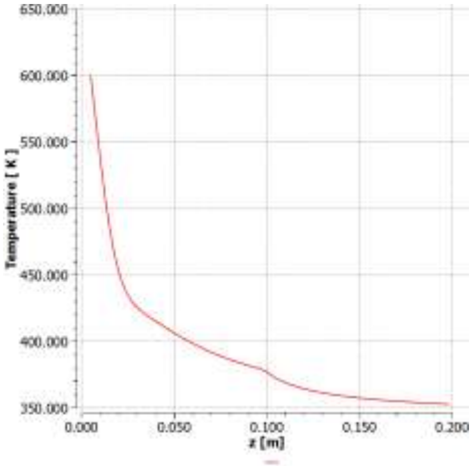


Figure 4:- Axial development of the temperaturein thecone.

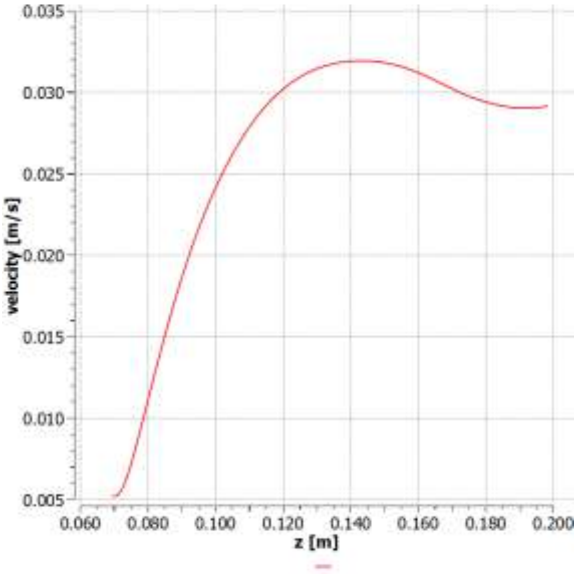


Figure 5:- Axial development of the velocity in the cone.

The profiles of figures 4 and 5 show three variations in the temperature and velocity of the air along the plume axis.

At $Z \leq 0.1$, the position of the heat source at the cone inlet significantly impacted the plume flow.

The air temperature and velocity fluctuations along this zone were relatively minimal. This little recirculation region focused on the axis of the source, where the air did not actively contribute to the flow growth, causing the low variance.

In the second zone ($Z \leq 0.14$), the variations in the air temperature and velocity and the flow mixture began to activate.

When the forces of viscosity and buoyancy were balanced at Z = 0.14, the maximum axial velocity was attained. A slowdown process started in this part. These profiles were similar to the study [6].

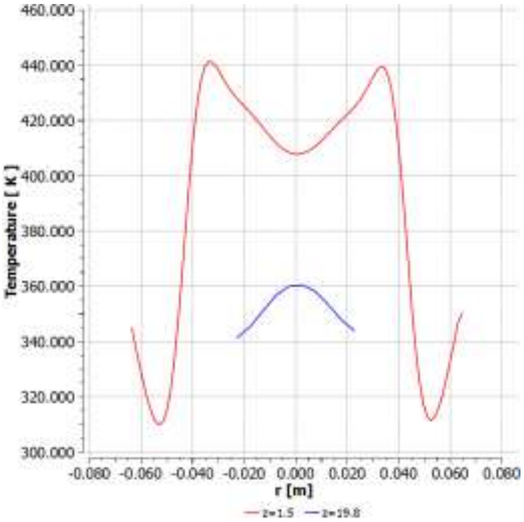


Figure 6:- Comparison of the dimensional air temperatures at the entrance and the exit of the proposed system.

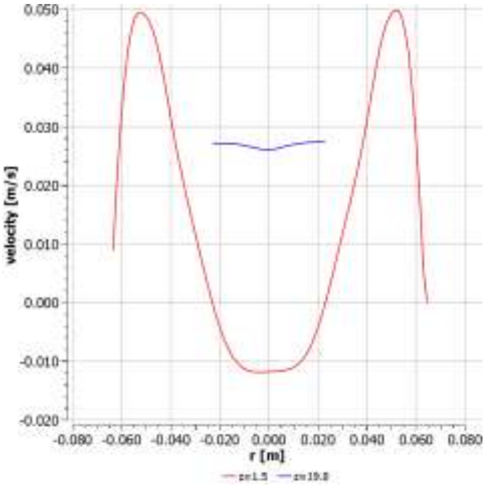


Figure 7:- Comparison of the dimensional air velocity at the entrance and the exit of the proposed system.

The inlet (z=0.03) and outlet (z=0.19) zones were compared in figures 6 and 7. The zone near the hot source appears the temperature was very high above the hot source and low temperature both sides of the source, with raises in the height the temperature decreases, the confinement effect on the flow rate and introduces the homogeneity of the velocity and the temperature, the cone walls confined the airflow, which gives the stable temperature at the exit.

Conclusion:-

Turbulent, steady, incompressible natural convection flow evolving through the conical channel was studied. This work aimed to study the dynamical and thermal evolution of this flow to improve and develop system securities against the industrial emissions that cause air pollution.

We showed gradually the fundamental physical principles and the mathematical governing equations in the conservative general form for all physical quantities that governed the fluid flow.

To simulate the studied problem mathematically and numerically, the selection of methods was necessary as the $\mathbf{k}$- $\boldsymbol{\varepsilon}$ model and the Finite-volume methodfor discretization.

esults obtained using the software ANSYS-FLUENT indicated that the cone shape and the parameters A=0.75, C=0.6, and R*=0.6 with Prandtl number of the air 0.72 and Rayleigh number $\boldsymbol{Ra > 10^{10}}$improve the thermal and dynamic behavior of the flow.Consequently, improve the security system.

References:-

[1] C. F. Kettleborough,. Transient laminar free convection between heated vertical plates including entrance effects. International Journal of Heat and Mass Transfer, (1972). 15(5), 883-896.

[2] V. I.Terekhov, A. L.Ekaid, andK. F. Yassin, The effect of aspect ratio on the laminar free convection inside a vertical channel heated isothermally. In Strategic Technology (IFOST), 11th International Forum (2016, June).pp. 619-624. IEEE.

[3] S. K. S.Boetcher, andE. M. Sparrow, Simulation of Laminar Axisymmetric Buoyant Plumes: A Case Study of the Anatomy of Similarity Solutions. Numerical Heat Transfer, Part A: Applications, (2007). 52(5), pp 399-415.

[4] S. M. Ohk, andB. J. Chung, Natural convection heat transfer inside an open vertical pipe: Influences of length, diameter and Prandtl number. International Journal of Thermal Sciences,(2017). pp 115, 54-64.

[5] A. O. Mohamed Mahmoud, Etude de l’interaction d’un panache thermique à symétrie axiale avec un écoulement de thermosiphon, Thèse de Doctorat, Université Tunis II, 1998.

[6] A. M. Mahmoud, Z. Yahya, Improvement in the Performance of a Solar Hot Air Generator Using a Circular Cone. International Review of Mechanical Engineering (I.RE.M.E.), 13(8), August 2019, 1970 – 8734.