Journal Homepage: -www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)

Article DOI:10.21474/IJAR01/16187
DOI URL: http://dx.doi.org/10.21474/IJAR01/16187

ISSN NO. 2320-5407

RESEARCH ARTICLE

OPTIMIZATION OF CONTROL PARAMETERS OF THERMAL PLUME INTERACTION WITH ENVIRONMENT

Z. Yahya and Norah M. Alturki
Department of Physics, College of Science, Qassim University, Buraidah, Saudi Arabia.

……

Manuscript Info

Manuscript History
Received: 29 November 2022
Final Accepted: 30 December 2022
Published: January 2023

Key words:-
Guided Thermal Plume, CFD Analysis, Mathematical Modeling, Simple Algorithm,Plume Source, Optimized Flow Behavior

Abstract

The present work is interested in the mathematical and numerical modeling of the effects of different parameters such as the angle of the cone, the length of the cone, the hot source radius, and the hot source length on thermal plume flow in interaction with its surrounding. The behavior of the guided thermal plume flow can give us more knowledge about the control parameters of different applications relevant to the environment and industry. This study investigates the thermal plume in interaction with its surrounding environment. To determine the control parameters that influence this flow we must study at the beginning mathematically and numerically the studied flow thru a natural convection flow generated by a circular hot source at the entrance of a vertical cone (Figure 1). In this work, the flow is natural convection, turbulent, steady, two-dimensional, incompressible flow. Numerical results are obtained using CFD analysis based on mathematical modeling of the flow and ANSYS-FLUENT software. The introduction of the hot source with a different radius causes an important change in the flow structure. The profiles show a very strong gradient in the temperature in the zone near the plume source for a specific cone angle. The comparison of different profiles indicates the optimized control parameters of the flow.

Copy Right, IJAR, 2023,. All rights reserved.

……

Introduction:-

The turbulent thermal plume in interaction with its environment that we consider in our study is found in many important applications, for example in fires, in combustion chambers, and dispersion of pollutants. This phenomenon characterizes the interaction of two flows, the first flow is the thermal plume induced by the hot plume source the second flow is the thermosiphon flow caused by the heated walls around the plume source.

Many authors have experimentally studied the thermal plume to characterize the general structure of this flow. Agator et al. [1] and Brahimi et al. [2] studied the turbulent thermal plumes issuing, respectively, from a hollow portion of a sphere and a circular disc heated at a constant and uniform temperature of 500 °C. These works indicated the appearance of two vertical zones. The first zone, just above the plume source, characterizes the flow development region. This zone is followed by a second zone where the dynamic and thermal profiles are self-similar. This plume behavior is confirmed by Jaluria and Gebhart [3]. However, other researchers [4, 5] have reported the appearance of a small zone of transition between the two zones previously found. The transition regime

Corresponding Author:- Z. Yahya
Address**:-** Department Of Physics, College of Science, Qassim University, Buraidah, Saudi Arabia, z.yahya@qu.edu.sa

between laminar and turbulent thermal plume flow was determined by Yang [6]. Thereafter, the thermal and dynamic structure of two thermal plumes induced by a hot disc and by a hot vertical cylinder was studied by Bouzinaoui [7]. They have shown that the flow structure of the plume, in the two configurations, is axisymmetric.

Researchers investigated the effect of different parameters on the thermal plume in channels. Jouinietal.[8] investigated numerically the effect of the height of the channel, they found that the dimensionless height Z^* depends on the shape ratioA, at $A < 2.5$ the averaged velocity at the outlet of the channel increases linearly with the height of the channel, and where the value of A is around 3 the exit velocity remains constant. Then for the shape ratio A is 3.5 or more the velocity continues to increase with the height of the channel. Seung and Bum [9] examined experimentally and numerically the influences of the length, the diameter, and the Prandtl number on the natural convection heat transfer inside open vertical pipes. They found that the heat flux increase with increasing the diameter and decreases with increasing the length of the channel.

Zinoubi et al. [10] focus on an application of chimney problems, which studied experimentally the influence of the shape factors on fluid flow. The device consists of a hot disk placed at the entrance of the open-ended vertical cylinder. They found that the ideal configuration of the cylinder is the length L=20cm and the radius r=7.5cm with a shape ratio A=0.375 witch improves the flow rate inside the cylinder.

The analysis of previous works shows the lack of numerical investigation interest in the control parameters that influence the interaction between a thermal plume, whose air entrainment is simultaneously horizontal and vertical, and inclined walls despite the relative diversity of the walls used in these studies. The study of the development of a thermal plume inside a cone is justified by its role in improving the understanding of the mechanisms involved in practical applications in the real world and industry. Besides its importance from the fundamental viewpoint, the utilization of the cone allows the hot air to be strongly evacuated at the exit of the flow for possible future applications [11]. In the present investigation, the vertical contribution of the air entrainment was favored by the use of a circular cone, with small opening angles, placed around the hot source. The aim of the present work is to explore the mechanisms of buoyancy flow development and the effects of the angle of the cone (θ), the length of the cone (H), the hot source radius(R_s) and the hot source length (L) on the studied flow. This work starts by studying the general structure and the thermal and dynamic behavior of the flow. Thereafter, we examined in this study the effect of different parameters to determine the control parameters of the studied flow. The studied flow is natural convection, turbulent, steady, two-dimensional, incompressible flow. Mathematical modeling based on the Navier-Stokes equations was carried out. The velocity-pressure coupling was resolved using the SIMPLE algorithm. Results were obtained using ANSYS-FLUENT software.

Physical characteristics and geometriy:-

A problem of two-dimensional, steady, turbulent natural convection flow of a viscous, incompressible, Newtonian fluid evolving in a vertical cone with the heated source at the center of the cone inlet, which heated uniformly by joule effect at a constant temperature. The internal walls of the conical channel are heated by the thermal radiation effect emitted from the hot source.

The analysis of this study is based on the following assumptions as shown in figure 1.
An axi-symmetric cylindrical coordinates (r,z) where z measures the distance along the surface of the cone from the inlet(z=0), and the r measures the distance horizontally.
ais the radius at the inlet, (z=H)the high of the cone.

The hot source is the disc with a radius R_s located at the center of the inlet (z=0).
$T_w(z)$is the walls temperature heated uniformly by the heated disc, and T_0 is the fluid temperature inside the cone, at $t > 0$ then $T_w(z) > T_0$.

All the fluid properties are assumed constant except the density that changes with the difference of the temperature in the gravitational force, which is playing the main role in natural convection because it produces the buoyancy force. From Boussinesq approximation for buoyancy-driven flows the density is given by

$$\boldsymbol{\rho(T) = \rho_0\big(1 - \beta(T - T_0)\big)\mathrm{inthe}\rho g\mathrm{term}} \quad (1)$$

In the **otherterms**$\boldsymbol{\rho}(\mathbf{T}) = \boldsymbol{\rho_0}$

The density in the governing equations is always constant except in the z-momentum equation that is affected by gravity.

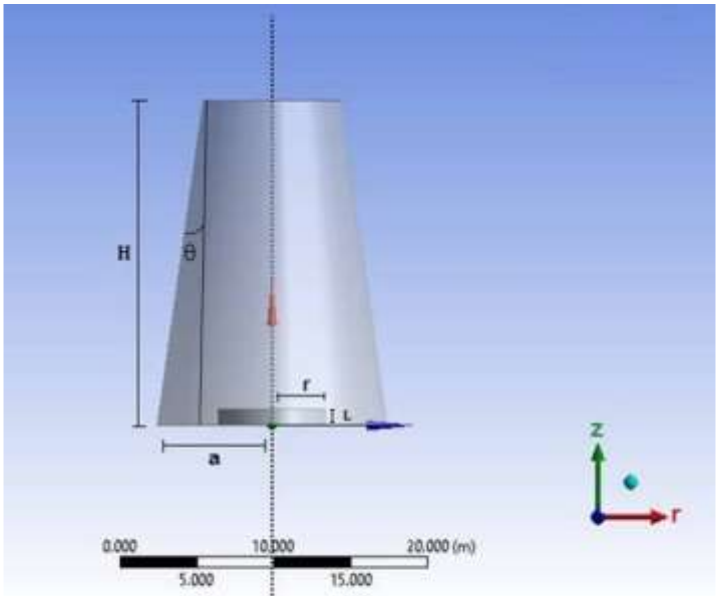


Figure 1:-The conical channel geometry.

Governing equations:-

Under the above assumptions, the conservation equations for two-dimensional turbulentincompressible steady floware:

Mass conservation equation

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r)+\frac{\partial(v_z)}{\partial z}=0 \quad (2)$$

z-momentum conservation equation

$$\rho_0\left(v_r\frac{\partial v_z}{\partial r}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial z}\left[2\mu\frac{\partial v_z}{\partial z}-\frac{2}{3}\mu\left(\frac{\partial v_z}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}(rv_r)\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial v_r}{\partial z}+\frac{\partial v_z}{\partial r}\right)\right]+\rho_0 g\beta(T-T_0) \quad (3)$$

r-momentum conservation equation

$$\rho_0\left(v_r\frac{\partial v_r}{\partial r}+v_z\frac{\partial v_r}{\partial z}\right)=-\frac{\partial p}{\partial r}+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v_r}{\partial z}+\frac{\partial v_z}{\partial r}\right)\right]+\frac{\partial}{\partial r}\left[2\mu\frac{\partial v_r}{\partial r}-\frac{2}{3}\mu\left(\frac{\partial v_z}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}(rv_r)\right)\right]+\frac{2\mu}{r}\left(\frac{\partial v_r}{\partial r}-\frac{v_r}{r}\right) \quad (4)$$

Energy conservation equation

$$\rho_0 c_p\left(v_r\frac{\partial T}{\partial r}+v_z\frac{\partial T}{\partial z}\right)=v_z\frac{\partial p}{\partial z}+v_r\frac{\partial p}{\partial r}+\frac{1}{r}\frac{\partial}{\partial r}\left(r\lambda\frac{\partial T}{\partial r}\right)+\frac{\partial}{\partial z}\left(\lambda\frac{\partial T}{\partial z}\right)+\phi_v (5)$$

where ϕ_v is the viscous dissipation function for an incompressible fluid in (r-z) dimension, which defined as

$$\phi_v=\mu\left[2\left(\frac{\partial v_z}{\partial z}\right)^2+2\left(\frac{\partial v_r}{\partial r}\right)^2+2\left(\frac{v_r}{r}\right)^2+\left(\frac{\partial v_z}{\partial r}+\frac{\partial v_r}{\partial z}\right)^2-\frac{2}{3}\left(\frac{\partial v_r}{\partial r}+\frac{\partial v_z}{\partial z}\right)^2\right] (6)$$

Boundary conditions:-

To close the system the initial and boundary conditions for our study, the dimensional boundary conditions are given by [12,13].

At $\mathbf{t}=\mathbf{0}$: $v_r=0,\ v_z=0,\quad T=T_0=T_w$ for all r, z.

Where the temperature at the cone region is equal to the temperature of the surrounding because the thickness of the thermal boundary layer is negligible at the entrance of the cone.

At $\mathbf{t}>0$ at the walls (r=a), and for the no-slip condition, the flow variables are $v_r=0,\ v_z=0,\ T=T_w(z)$

On the axis of the cone and from the symmetry (r=0), the variation of the velocity components and temperature are $\frac{\partial v_r}{\partial r}=0,\ \frac{\partial v_z}{\partial r}=0,\ \frac{\partial T}{\partial r}=0$

At the entrance of the cone (z=0) with a hot source in the center, then

$0\le r\le r_s$; $v_r=0,\ v_z=0,\quad T=T_s$

$$r_s \leq r \leq a\,;\ \ v_r = 0,\ \ \frac{\partial v_z}{\partial z} = 0, \qquad T = T_0$$

At the outlet of the cone (z=H), the variation of the quantities is zero because the flow becomes fully developed, then

$$\frac{\partial v_r}{\partial z} = 0,\ \ \frac{\partial v_z}{\partial z} = 0\,,\ \ \frac{\partial p}{\partial z} = 0\ \ ,\frac{\partial T}{\partial z} = 0$$

Results and Discussions:-

A numerical study has been carried out for natural convection evolving in a conical channel and the effect of the following parameters:

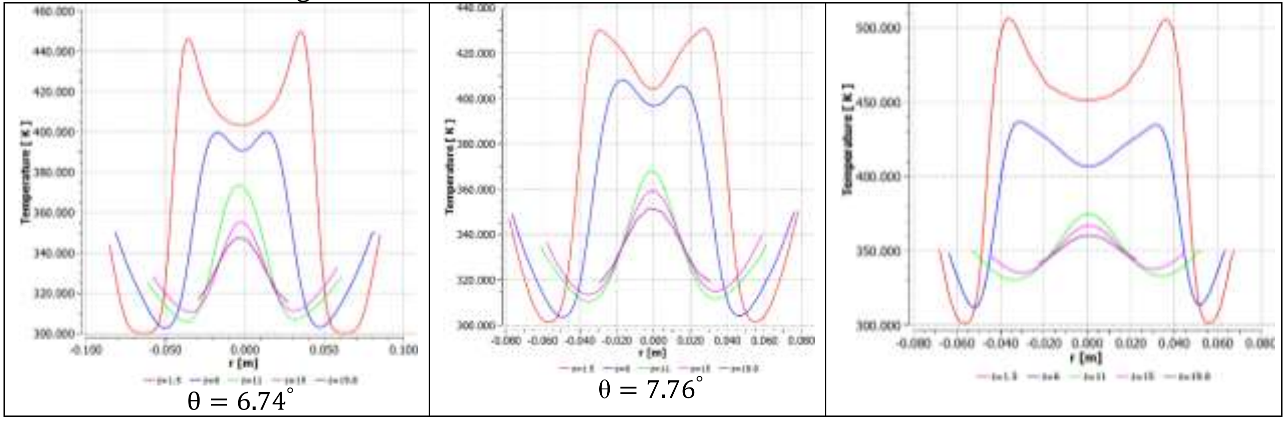
1. the angle of the cone [θ]
2. the length of the cone [H]
3. hot source radius [R_s]
4. hot source length [L]

Different parameters in different values are shown intable 1.

Table 1:-The control parameters values in the study.

General simulation study of the flow	Control Parameters	
θ=8°	θ=7.76°	θ=6.74°
H= 20 cm	H= 28 cm	H= 36 cm
R_S= 4.5 cm	R_S= 5.5 cm	R_S=6.3 cm
L=0.5 cm	L= 1 cm	L= 1.5 cm

The effect of the cone angle: -



$\theta = 6.74^\circ$ $\theta = 7.76^\circ$

Figure 2:- Radial distributions of the dimensional air temperature of the plume with different angles.

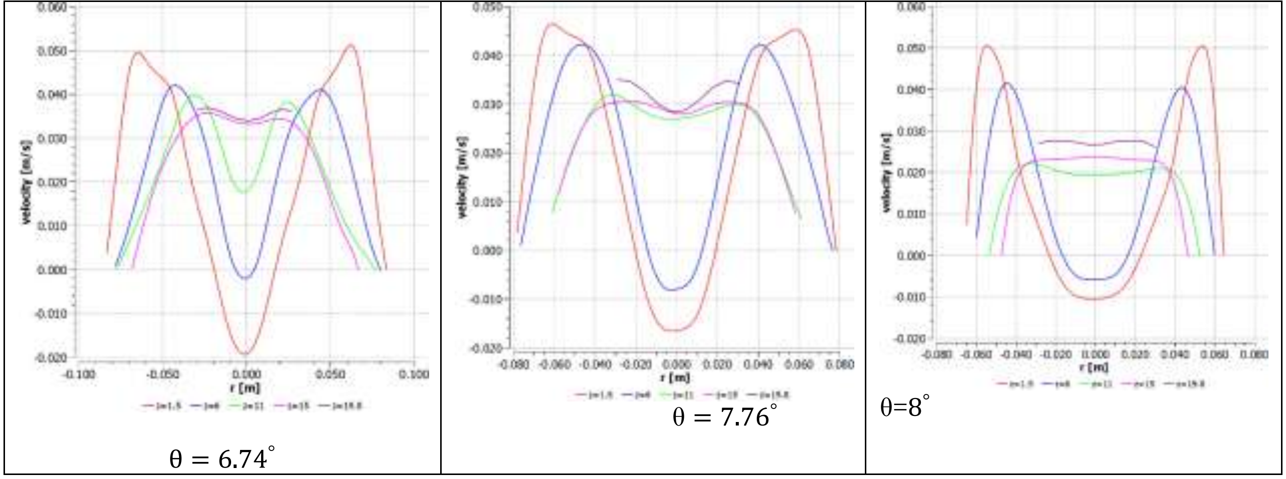
Fig. 2 shows the general behavior of the radial distributions of the dimensional air temperature of the plume inside the cone with different angles. At the angle 8°, the profiles show a very strong gradient in the temperature in the zone near the plume source.

In the zone near the plume source ($Z \leq 6$), the profiles showed a three extrema structure with a minimum on the axis and two maxima on either side of the plume axis**.** The entrainment of cool air to feed the heat source at its axis explained why the temperature on the axis dropped**.** As level Z increased, the maximum temperature dropped and drew closer to the plume axis**.**

The gradients persisted in importance until the three extrema profile vanished, pointing to the emergence of a new regime. Profiles showed one maximum in the intermediate regions ($Z \geq 11$) this behavior is decelerated in prior studies [11].

The radial distributions of the dimensional air velocity of the plume are presented in Fig. 3. These profiles illustrate the same general behavior of the thermal flow indicated in Fig2.

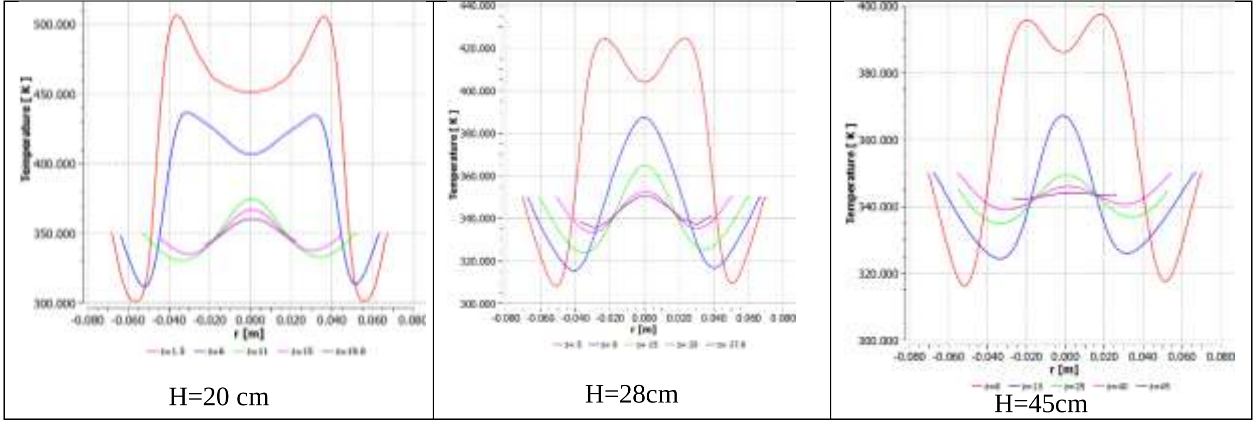
Fig3 illustrates that the cone angle is an essential parameter for the domination of the interaction of the flow and its surrounding presented by the three extrema structure.



$\theta = 6.74^{\circ}$ $\theta = 7.76^{\circ}$ $\theta=8^{\circ}$

Figure 3:- Radial distributions of the dimensional air velocity of the plume with different angles.

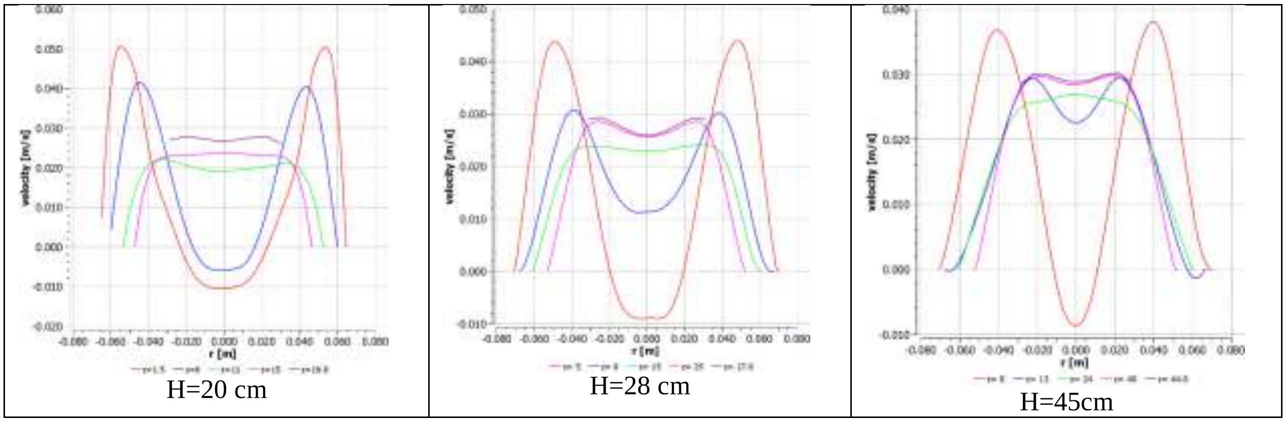
The effect of the length of the cone on the studied flow: -



H=20 cm H=28cm H=45cm

Figure 4:- Radial distributions of the dimensional air temperature of the plume with different heights.

Figures 4 and 5 show that as the cone length decreased, the temperature increased and, the dynamic structure was intense in the plume zone.

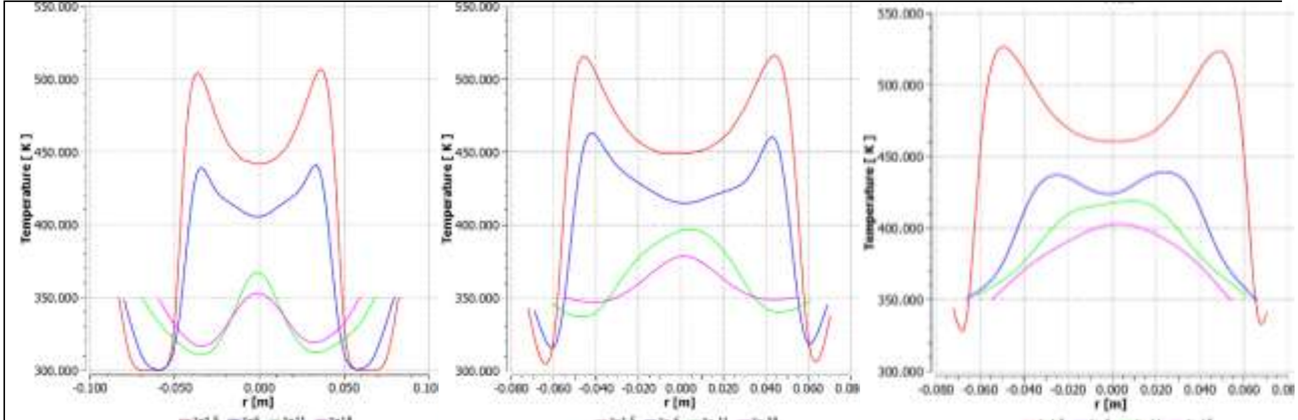


H=20 cm H=28 cm H=45cm

Figure 5:- Radial distributions of the dimensional air velocity of the plume with different heights.

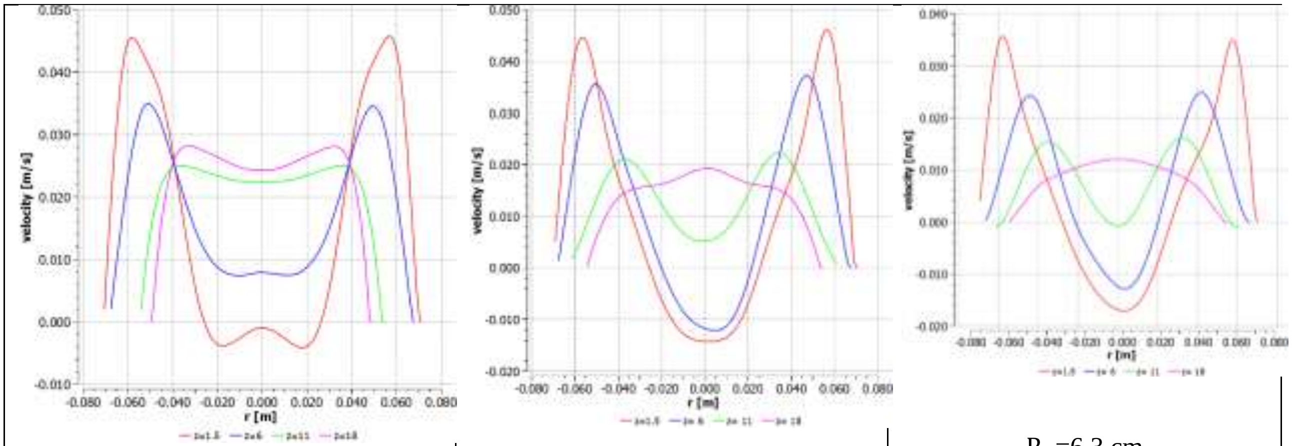
The effect of the plume source radius:-

The figures 6 shows that the increase of the plume source radius increases the maximum of the temperature profiles in the zone ($Z \geq 11$) andhas a significant effect on the structure of the flow in the region of the plume because it leads to an intensification of the phenomena of recirculation at the level of this zone. However, it does not affect the boundary layer flow at the wall. As shown in figure7 the dimensionalvelocity of the flow follows the same evolution as that of the temperature.



R_s= 4.5 cm | R_S=5.5 cm | R_S=6.3 cm

Figure 6:- Radial distributions of the dimensional air temperature of the plume with different plume source radius.

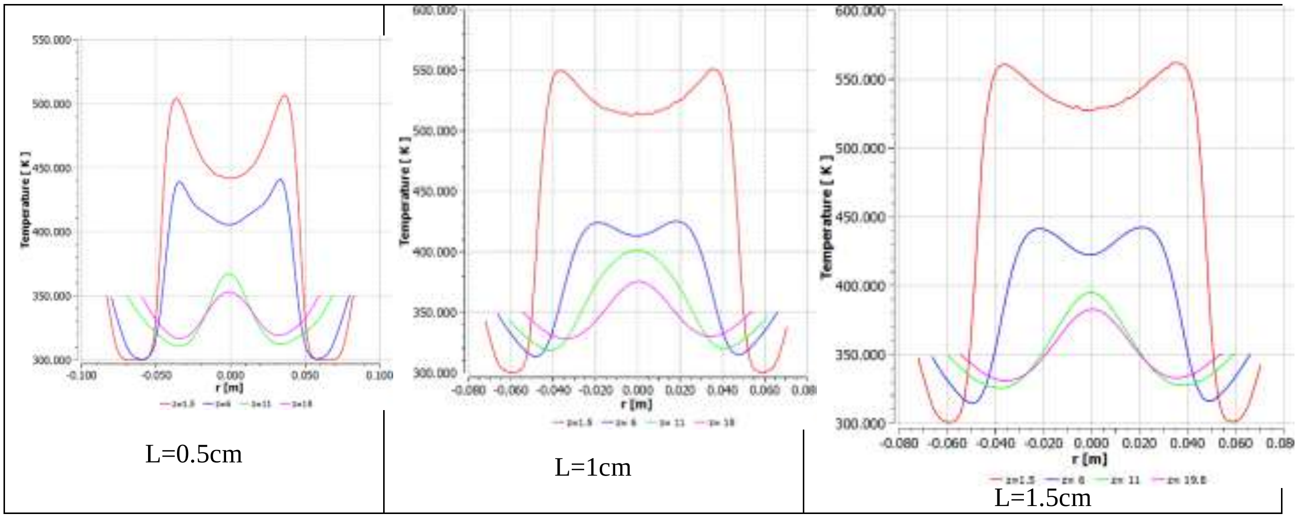


R_s= 4.5 cm | R_S=5.5 cm | R_S=6.3 cm

Figure 7:- Radial distributions of the dimensional air velocity of the plume with different plume source radius

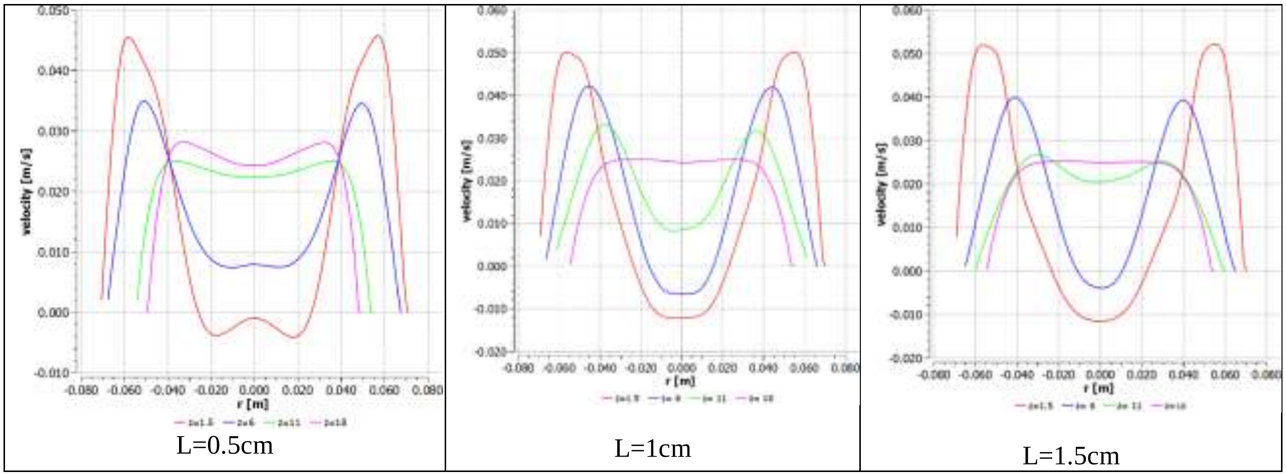
The effect of the heat source length: -

The profiles of figures 8 and 9 indicate an improvement of the thermal and dynamic characteristics in the plume zone for plume source length (L=1.5 cm). However, the heat source length does not affect the boundary layer flow at the wall.



L=0.5cm L=1cm L=1.5cm

Figure 8:- Radial distributions of the dimensional air temperature of the plume with different plume source lengths.

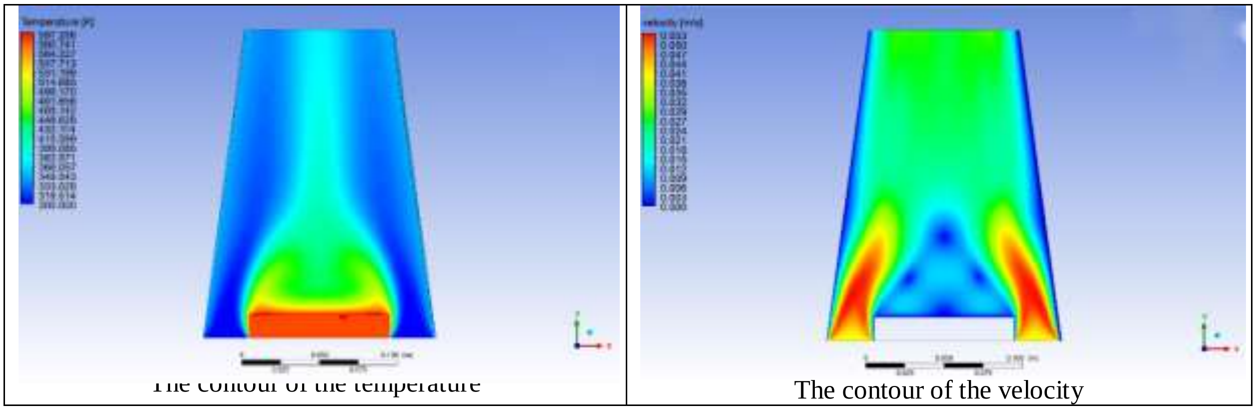


L=0.5cm L=1cm L=1.5cm

Figure 9:- Radial distributions of the dimensional air velocity of the plume with differentplume source lengths

The Optimized Control Parameters of the studied flow:-

Figure 10 determines the control parameters that improve the behavior of the thermal plume flow in interaction with its environment.



The contour of the temperature The contour of the velocity

Figure 10:- The temperature and velocity contours for the control parameters θ=8°; R_S= 4.5 cm; H=20 cm and L=1.5cm

Conclusion:-

Turbulent, steady, incompressible natural convection flow evolving in avertical conical channel was studied in this work. The main objective of this work is to study the dynamical and thermal behavior of the guided natural thermal plume to improve the efficiency of this flow in different applications in the environment and industry.

The guided natural thermal plume was simulated by the presence of the heat source in the middle of the cone inlet. We presentedthe flow by the fundamental physical principles and the governing equations, that governed the fluid flow.

To simulate the studied problem mathematically and numerically, the selection of methods was necessary as the k-ε model and the Finite-volume method, and the SIMPLE algorithm.

Results are obtained using the CFD analysis and the ANSYS-FLUENT software and compared with previous studies.

The comparison of profiles using different values indicates the optimized control parameters that improve the efficiency of the thermal and dynamical flow behavior.

References:-

[1] J. M. Agator, Contribution à l'étude de la structure turbulente d'un panache à symétrie axiale. Thèse, Université de Poitiers. (1983).

[2] M. Brahimi, D. K. Son, Experimental investigation and numericalprediction of the mean flow of a turbulent pure plume. Arch. Mech. 38 (1986) 5-6.

[3] Y. Jalura, B. Gebhart, On the buoyancy-induced flow arising from a heated hemisphere, Int. J. Heat Mass Transfer. 18 (1975) 415 – 431.

[4] B. Guillou, Etude numérique et expérimentale de la structure turbulente d'un panache à symétrie axiale. Thèse, Université de Poitiers. (1984).

[5] B. Guillou, M. Brahimi, D. K. Son, Structure turbulente d'un panache thermique : Aspect dynamique. J. Méc. Théo. Et Appl. 2 (1986) 371- 401.

[6] H. Q. Yang, Buckling of a thermal plume. Int. J. Of Heat and Mass Transfer. 35 (1992) 1527 – 1532.

[7] A. Bouzinaoui, R. Devienne, J. R. Fontaine, An experimental study of the thermal plume developed above a finite cylindrical heat source to validate the point source model. Experimental Thermal and Fluid Science. 31 (2007) 649-659.

[8] H. Nakagome, M. Hirata, The structure of turbulent diffusion in an axisymmetric thermal plume, in:Proc. Int. Sem. Tur. Buoy. Conv., Dubrovnik, Yugoslavia, PP: 361-375, (1976).

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

[10] J.Zinoubi, R. B. Maad, A.Belghith, Experimental study of the resulting flow of plume–thermosiphon interaction**:** application to chimney problems, Applied Thermal Engineering, 25(4), 2005, 533–544.

[11] A. M. Mahmoud, Z. Yahya, Experimental Investigation of a Thermal Plume's Air Entrainment in a Circular Cone. International Journal of Thermofluid Science and Technology, 8 (4), 080403 ,2021.

[12] G. Palani, E. J. Lalith Kumar, and K.-Y. Kim, Free Convection Effects on a Vertical Cone with Variable Viscosity and Thermal Conductivity. Journal of Applied Mechanics and Technical Physics, 57(3), 2016, 473–482.

[13] B. Pullepu, K. EkambavananandA.J.Chamkha, Unsteady Laminar Free Convection from a Vertical Cone with Uniform Surface Heat Flux. Nonlinear Analysis: Modelling and Control, 13(1), 2008, 47–60.