BEHAVIOR OF A HVAC AIR DIFFUSER IN DIFFERENT CONDITION

Altin Dorri[1], Majlinda Alcani[2], Altin Maraj[3]

[1,2,3] *Department of Energy, Polytechnic University of Tirana, Albania*

ABSTRACT

This paper try to analyze the thermodynamic behavior of an air supply ceiling diffuser, part of an air conditioning plant. Influences of characteristic parameters of turbulence and the resolution of computational grid are also evaluated. Analyzes are made through 3-D numerical simulation of finite volumes with aim to simulate thermal and flow fields in a VAV (Variable Air Volume) type square ceiling diffuser. Simulations are performed for three different air flow rates and two turbulence models k-ε Realizable &k-εStandard and fields of temperature, velocity and turbulence intensity are studied.In conclusion, comparing the results obtained from simulations was found a relation that express the velocity as function of flow rate and position, in a symmetry plan for the computational zone.

Keywords: Air Distribution, Environmental Air Quality, Simulation, VAV (Variable Air Volume)

I. INTRODUCTION

Continued growth on environmental quality standards of indoor environments, needs to answer a series of very precise specifications that overlap between them and as a result are opportunistic design choices, especially in system design [1, 2].The design engineer of heating ventilation and air conditioning (HVAC) systems, should be able to choose between several different kinds air distribution systems.Besides mean values of temperature and relative humidity should be foreseen an accurate distribution of the air in the environment to avoid areas with high air speed and stale air areas. In this context, systems with variable air volume (VAV) have become increasingly useful the recent years compared to constant air volume systems (CAV), having smaller plant size and the potential to save energy [3].

To analyze the air distribution in an indoor environmentcomputer analysis (numerical simulation method)nowadays are representing various advantages such as: low cost, short duration, obtain complete and detailed information, simply simulate real and ideal conditions compared to experimental methods [4, 5].However, the numerical simulation method has a weakness: in most cases, found an absolute lack of quality control of the solution, when it refers to a complex situation. For this purpose, in this paper is done a numerical analysis through the CFD software Fluent, to study the thermo-fluid dynamic behavior of air flow supplied from a terminal unit (square with concentric sectors-part of VAV system) in heating conditions.

II. PHYSICAL AND MATHEMATICAL MODEL OF THE DIFFUSER

2.1 Physical model

Supply air diffuser taken in consideration (see Fig. 1) is a 4-way square ceiling diffuser, composed of anodized aluminum and different elements:

- oneparallelepiped with height equal to 100 mm and square base with sides 575 mm;

- oneparallelepiped with height equal to 100 mm and square base with sides 630 mm;
- one filter inserted in the second parallelepiped; with height of 68 mm and square base with sides 630mm;
- 6 concentric sectors turned by 45 °, 68 mm height and horizontal dimensions ranging from 120x120 mm up to 620x620mm, increasing 100 mm by a deflector to another.
- characteristic dimensions are: B = 695 mm, D = 630 mm, F = 535 mm φ_{out} = 247 mm

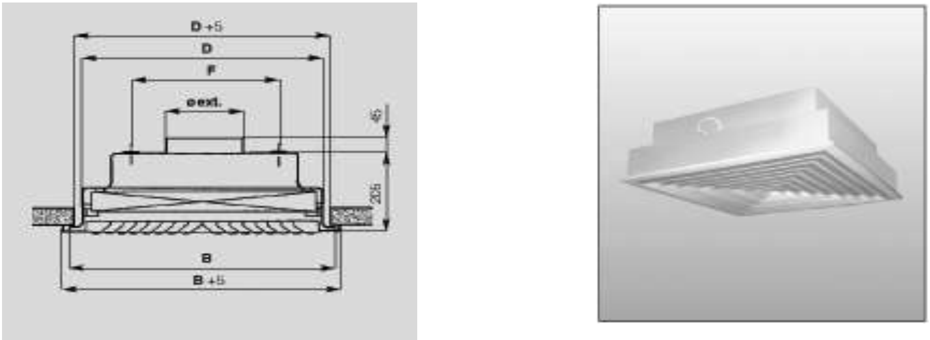
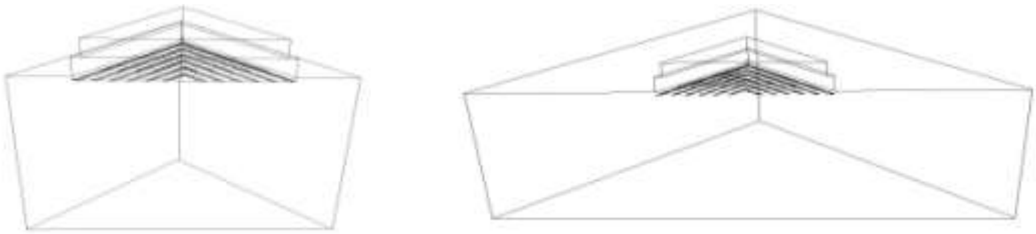


Figure 1: 4-Way Square Ceiling Diffuser With Filter

Survey is carried out for four different flows 405 m³/h, 540 m³/h, 675 m³/h 810 m³/h.Numerical simulations in the study were performed only for a diffuser sector, which is limited to the area near the diffuser.Diffuser is mounted in the center of the room ceiling, taken into consideration. Two calculations zones were created in two different dimensions, respectively size 1x1x1m and 2x2x1m, both consisting of an upper inlet and an open contour, representing continuity with the interior of the room.

In realization of the geometry is choose to model only half of the interior part of the room, considering the vertical symmetry plan, the plan that pas through the base diagonals of the diffuser (Fig. 2a &b).



(a) **(b)**

Figure 2: The Geometry of the First Calculation Zone (A) Andsecond Calculation Zone (B)

2.2 Mathematical and Numerical Model

In this study, baseline of thermo fluid-dynamic analysis is represented by the equations in stationary regime: mass conservation, movement and energy [6]:

Conservation of mass:

$$\nabla \cdot (\rho \mathbf{v}) = 0 \quad (1)$$

Whereρ is air density and **v**isvelocity vector.

Conservation of the quantity of movement:

$$\nabla(\rho \mathbf{v}\mathbf{v}) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{v}) \quad (2)$$

where p is the mean pressure and μ is the effective viscosity, given by the sum oflaminar and the turbulent viscosity.

Conservation of energy:

$$\nabla \cdot \left(\rho \mathbf{v} c_p t \right) = \nabla \cdot \left(\lambda \nabla t \right) + \dot{q} \qquad (3)$$

where c_p is the specific heat and $\dot{q}$ represents the internal heat generation.

Solving equations 1, 2, 3 gives us a full distribution of air flow parameters. In this system of equations represented is only one unknown parameter, more accurately, turbulent viscosity, which is calculated in different ways depending on the turbulence model.

For two calculation zones are considered the following boundary conditions:

- air inlet is selected a condition type *mass flow inlet*.
- filter is represented by a *porous- jump* condition, the pressure drop is equal to 140 Pa.
- air outlet is selected a *pressure- outlet*condition.
- prism side surface, which represents the continuity of the interior part of the room, is selected pressure-inlet condition.
- symmetry surface is characterized by a condition type - *Symmetry*.
- ceiling and side surfaces of the different elements that make up the diffuser, are represented by a *wall type* condition.

In the Fig.3, yellow area represents the symmetry plan of the complete area, the red area represents the basis part of the room in witch is set "*pressure outlet*" boundary condition. The blue area represents the inner continuity plan of the room in which it is used boundary condition "*pressure Inlet"*, gray surfaces represent the ceiling of the room and the air diffuser walls in which is used the boundary condition "wall" and blue surface unrestricted by walls of diffuser represents inlet flow air where is used the boundary condition "mass flow Inlet."

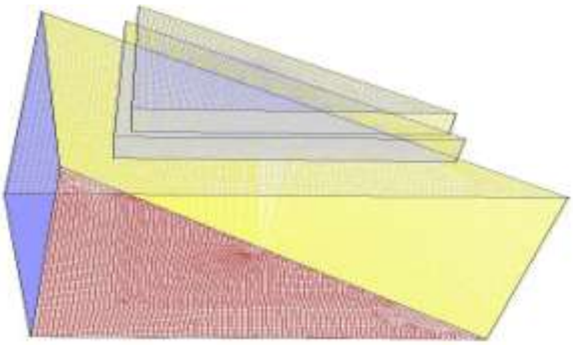


Figure 3: Three-Dimensional Computational Mesh Of The 1^{-St} Calculation Area.

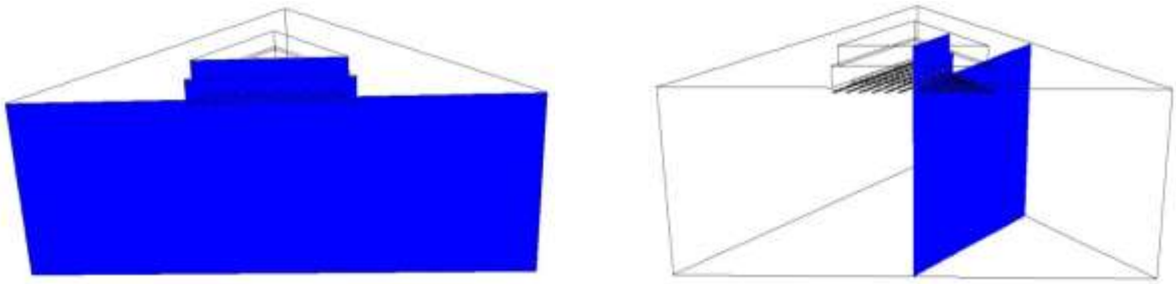
III. RESULTS AND DISCUSSION

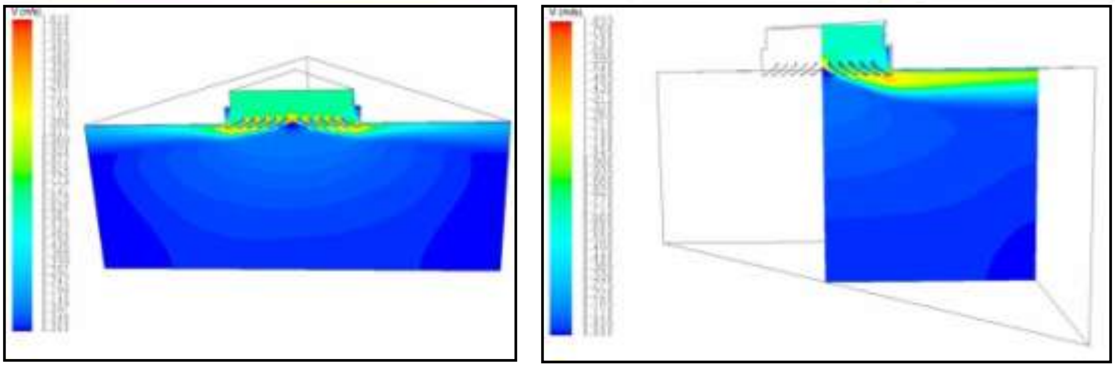
All graphical presentations shown in this study are generated based on the numerical method "segregate solver"of solving equations; discretization of convective terms is adapted for "First Order Upwind" interpolation scheme. For the pressure-velocity binomial is taken into consideration "SIMPLE" algorithm and for the turbulence investigation is used "*k-ε"* withstandard wall function [7, 8].To compare the results, are taken into consideration the performance of velocity, turbulence intensity and temperature along the horizontal and vertical axis that belong symmetry plan of three-dimensional internal space x = z, (Fig.

4a) and y = z plan which is parallel to one ofthe outside contours passing through an axis of the nozzle (Fig.4b)

(a) **(b)**

Figure 4: The Symmetry Plan X=Z (A) And The Symmetry Plan Y=Z (B)





(a) **(b)**

Figure 5:Distribution of Velocity Module in Symmetrical Plans X=Z (A) and Y = Z.Flow Rate *Q* = 810 M^3/H

Analyzing Fig. 5a, can say that also for small shift in the vertical direction the velocity decreases. The velocity will be meanwhile: large in the areas near the exit of the air, and too small in below zonesdeflector.From Fig.5b,that represents the distribution of velocity module in the symmetrical plan x = z, it can be seen that the maximum velocity can be set in the vicinity of the diffuser. Fig. 6 shows for medium flows the overlapping of the intensity turbulence profiles in the plan x = z along the horizontal and vertical axis.

Fig. 7 and 8 show the temperature distribution in different plans and distances from the diffuser for air flow of 810m^3/h. From the indicated images we can see how the temperature in the various plans represents a symmetry that coincides with the calculated space.Can also observe how the temperature decrease rapidly away from the diffuser and after reaching a maximum stabilized at temperature value at 293K ; this is a characteristic of each vertical axis parallel to the axis of the diffuser in two symmetry plans.

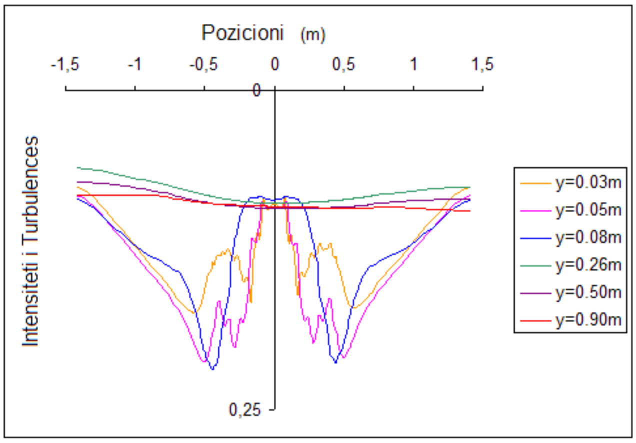


Figure 6:The Overlapping of Intensity Turbulence Profiles For Y = 0.03 M, 0.05m, 0.26 M, 0.50m and 0.90 M in Symmetric Plan X = Z. Flow Rate Q = 810 M^3/H.

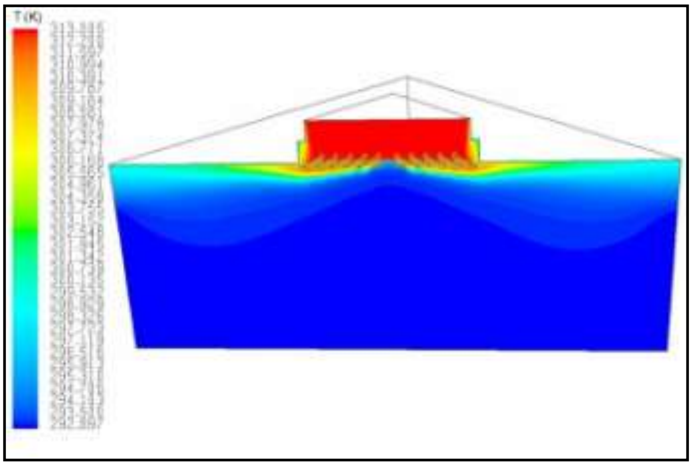


Figure 7: Temperature Distribution in the Symmetric Plane X = Z.Flow Rate Q = 810 M^3/H

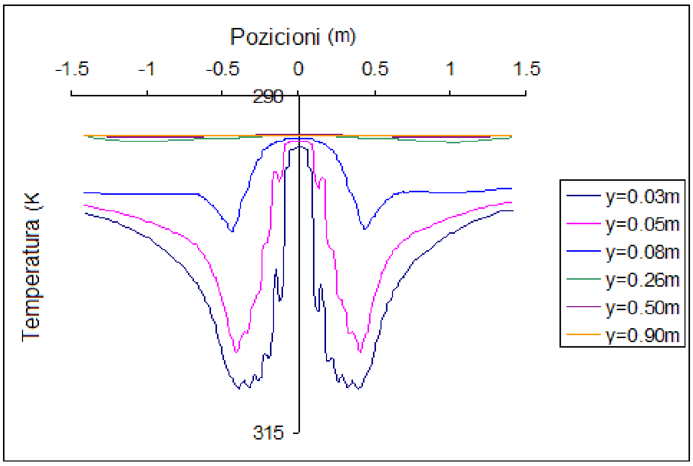


Figure 8: The Overlapping of Temperature Profiles For Y = 0.03 M, 0.05m, 0.08 M, 0.26 M 0.50 M and 0.95m in the Symmetric Plane X = Z. Flow Rate Q = 810 M^3/H.

Fig. 9 shows overlapping trendof the velocity module for two calculation zones, in the plan with equation x = z at different distances from the diffuser for air flow rate of 810m^3/h. Discern how the measured velocity module along the axis in the plane symmetric x = z until distance 0.08 m from the diffuser axis, is slightly higher for geometry "1m x 1m" in comparison with the geometry "2m x 2m".

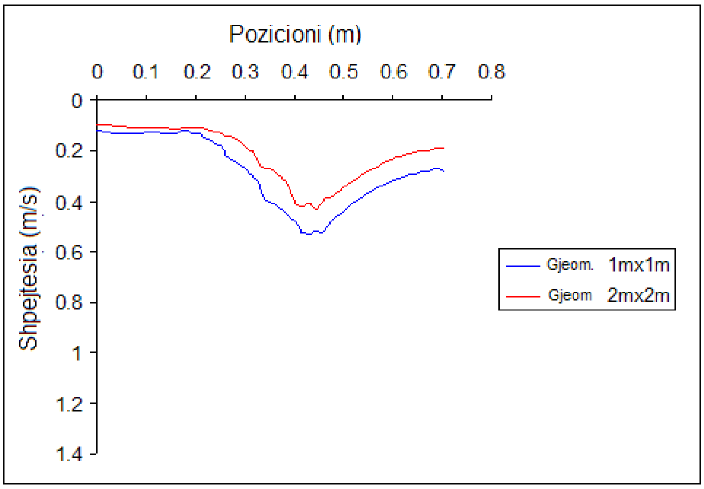


Figure 9: Overlapping Trends of Speed Module in the Plan with the Equation X = Z, At A Distance of 0.08 M From the Diffuser. Flow Rate Q = 810 M^3/H.

Fig. 10 shows the trend of the temperature in the plane x = z, where we have a similar trend of temperature for the four air flow rates.As can be seen with the increase of the air flowrate temperature distribution has not changed significantly. Is also observed a temperature increase for the four air flowratesuntil to the distance that correspond to the half of distance of the deflector center, and after reaching the maximum value, the temperature decreases moving away from the axis.

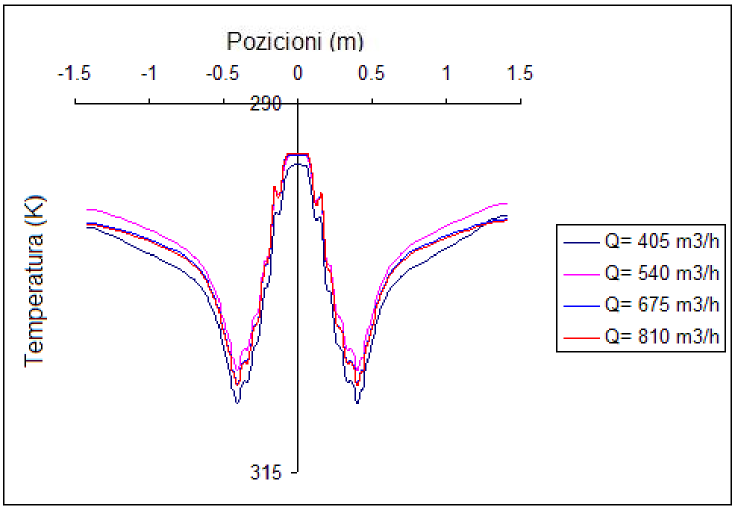


Figure 10: Overlapping Temperature Profiles For Y = 0.05 M In The Plane Symmetric X = Z For Four different Air Flow Rates.

Fig. 11 shows the trend of velocity module at distance of 0.26 m. From this figure results that since at this distance from the air diffuser, velocity module is less than 0.1 m/s for the four different air flows and the difference between maximum velocity modules reduces for this different air flows being shifted towards the end.

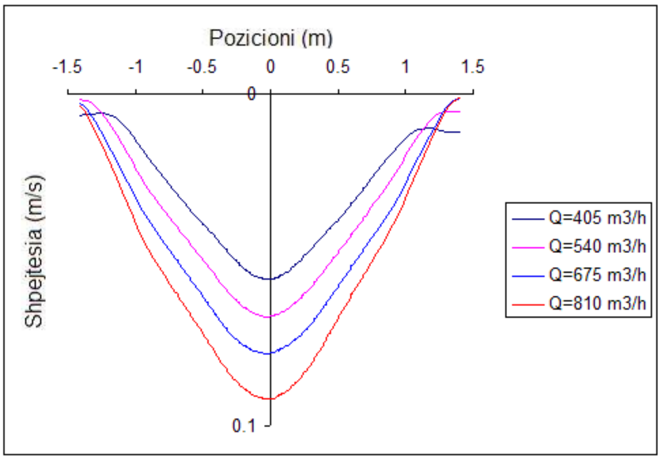


Figure 11: Overlapping Trend of Velocity Module for Y = 0.26 M in The Symmetric Plane X = Zfor Four Different Air Flow Rates.

From the represented functions of the trend of velocity $v_y$in symmetrical plan y = z (Fig. 12), at the distance of 0.7 m from the air diffuser, it can be assumed that v_y, along this axis is distributed according to a parabolic law:

$$v_y = -ax^2 + V_{max} \quad (4)$$

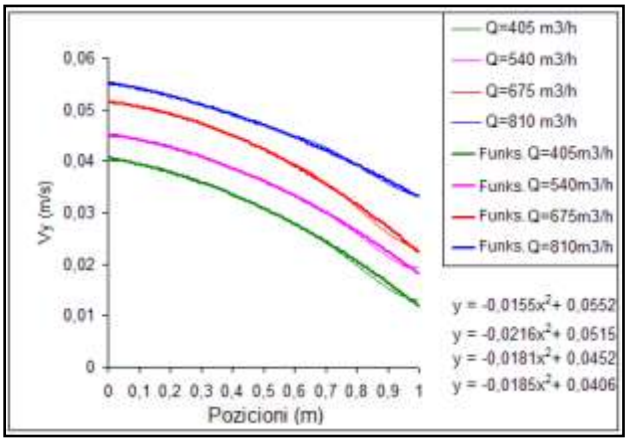
For maximum velocities achieved over the peak of parabolic curve and for minimal flow rate we can write the relation:

$$v_{max}(Q) = V_{max\, y_{rif} Q_{rif}} + c_Q \cdot (Q - Q_{rif}) \qquad (5)$$

From Figure 14, we can write that the velocity variation in function of the distance from the diffuser along the axis, can be expressed by the parabolic law:

$$v_{max}(Q,y) = a_y \cdot y^2 - b_y \cdot y + c_Q \cdot \left(Q - Q_{rif}\right) + c_{rif}. \quad (6)$$

where: $c_{rif}$is the coefficient (c) of parabolic curve wich express velocity variation in function of the distance from the diffuser for the refered flowrate. In relation (5) is introduced and the term that takes into consideration



the fact that the maximum velocity on the axis of the diffuser varies linearly with the flowrate

Figure 12: Overlapping Trend of Vertical Velocity Component and Functions that Represent the Trend of Velocity Y = 0.70 M in Symmetrical Plan Y = Z, for Four Air Flowrates.

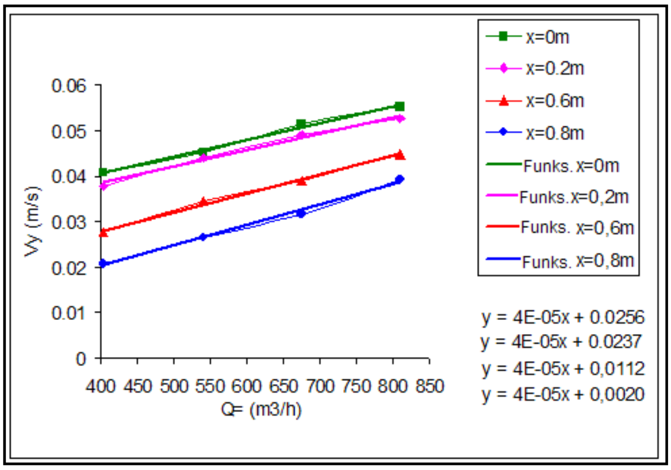


Figure 13: Overlapping of Vertical Velocity Component and Functions that Represent Trend Velocity at Some Point of Symmetry Plan Y = Z For Y = 0.7 M, For Four Air Flowrates.

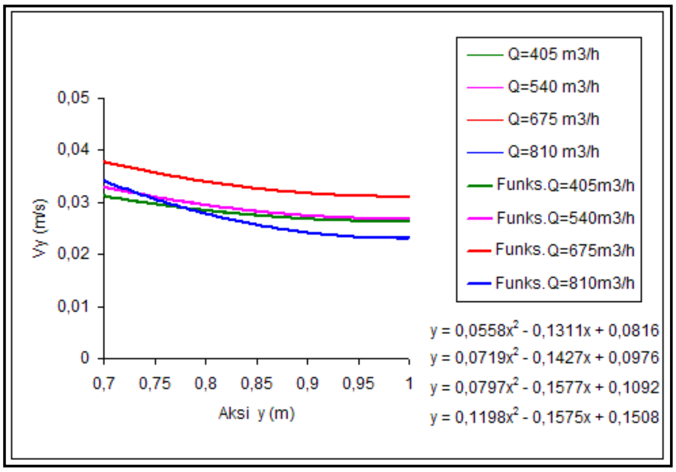


Figure 14: Overlapping of Vertical Velocity Component V_y and Functions that Represent This Trend Along the Axis of the Diffuser, For Y Ranging From 0.7 M To 1, Referred To The Fourair Flowrates.

At the conclusion analysis of the results of the trend of velocity as related to the flowrate and the position is found the following relationship:

$$v_y(Q,x,y) = -0{,}0184 \frac{\left[0{,}719 \cdot y^2 + 0{,}1573 \cdot y + 0{,}097 + 4 \cdot 10^{-5}(Q-405)\right]^3}{\left[0{,}0406 + 4 \cdot 10^{-5}(Q-405)\right]^3} x^2 + \quad (7)$$

$$+0{,}719 \cdot y^2 + 0{,}1573 \cdot y + 0{,}097 + 4 \cdot 10^{-5}(Q-405)$$

This relation can be accepted as the starting point of further studies

IV. CONCLUSIONS

In this paper is studied thermo fluid-dynamic behavior of anair diffuser type VAV through CFD Fluent software. Simulations were carried out for flow: 405 m^3/h, 540 m^3/h, 675 m^3/h and 810 m^3/h in two different calculation spaces. Are compared trends of velocity, turbulence intensity and temperature in the two symmetry plans of the air diffuser. In general, the solutions obtained are comparable and in the conclusion can be confirmed that various expansion of calculation spaces, did not significantly influence in the results of simulations.The velocity and the turbulences intensity, increases with the increase of air flowrate, and temperature distribution is less affected by increased flow. Simulations carried out with the model of turbulence "*k-ε*Realizable" and "*k-ε* Standard", gave almost identical results.Another goal of the study is that of finding a relationship that describes the velocity in function of position in the plane of symmetry y = z, perpendicular to the walls of computation space and that containing the axis of nozzle. The analysis shows that the vertical axis of the velocity has a parabolic trend over a sufficient horizontal distance from the diffuser. This is considered as the reference plan, in which the distribution of velocity component v_y is not influenced by the geometry of the diffuser and increases linearly with the trend of the air flowrates.A similar parabolic law expresses the variation of vertical velocity component along the axis of the diffuser. Combining these two effects, is found a relation that express vertical velocity component v_y in this plan as a function of the air flowrate Q and the coordinates (x, y) of the points in the plan (remember that the y axis is the vertical axis of the reference system).The analysis of the simulations, shows that the air flow doesn't come uniformly in all directions, but mainly concentrated along selected exit routes. This feature allows to avoid obstacles located aside air flow direction and developed in acceptable wide range.

REFERENCES

[1] ASHRAE Transaction, 114 (I) : SF-98-10-1, 1998.

[2] ASHRAE FUNDAMENTAL, 2001, pp. 262-398.

[3] A. BRIGANTI, Manuale della climatizzazione, Vignate (MI), Tecniche Nuove, 1992, chapter 8.

[4] Q. CHEN, P. SUTER, A numerical study of indoor air quality and thermal comfort under six kinds of air diffusion, ASHRAE Trans., 98 (1),1992, 203-217.

[5] Q. CHEN, Comparison of different k-ε models for indoor air flow computations, Numerical Head Transfer, Part B, 28,1995, 353-359.

[6] G. COMINI, Fondamenti di Termofluidodinamica Computazionale, Padova, 2008, chapter 4.

[7] H.C. CHEN, V.C. PATEL, Near-wall turbulence models for complex flows including separation, AIAA J., 26 (6), 1988, 641-658.

[8] J.J. COSTA, L.A. OLIVEIRA, D. BLAY, Test of several versions for the *k-ε*type turbulence modeling of internal mixed convection flows, Internal Journal of Head and Mass Transfer, 42,1999, 4391-4409.

[9] FLUENT 6.3 User's Guide, New Hampshire, 2006