3-D Prediction Of Two Fluid Streams Mixing At Different Angles

J. A. Naser

CFD Consultant, CANCES-ATP, G16 National Innovation centre, ATP, Eveleigh 1430, Australia
Formerly, Associate Professor, Mechanical Engineering Department
Bangladesh University of Engineering & Technology, Dhaka-1000, Bangladesh

ABSTRACT

Three dimensional mixing, generated by two fluid streams over a splitter plate in a square duct is studied numerically. By changing the alignment of the trailing part of the splitter plate, the flow direction of the two streams, having two different temperatures are changed. Study is carried out at a Reynolds number of 80,000, which approximately corresponds to the actual situation prevailing in the mixing duct of the car air-conditioning system. The numerical procedure used for this purpose solves the governing equations using the SIMPLE algorithm. The governing equations are expressed in a general curvilinear coordinate system and are discretized in a finite volume fashion. The time-averaged governing equations are closed using the k-ε turbulence model.

In the case of two streams flowing in the same direction, mixing occurs in the limited region of the shear layer, which extends up to 16% of the flow area at the exit. For the case of splitter plate making an angle of $+30^{o}$ where the flow area for hot fluid is reduced to 1/3 and the flow velocity is increased to 3 times to that of cold fluid, mixing improves and covers a region of up to 66%. But, through the remaining flow area, cold fluid comes out at its initial temperature. For the reverse case of -30^{o}, where flow area for cold fluid is reduced to 1/3 and the flow velocity is increased to 3 times to that of hot fluid, the mixing region covers up to 66%. But now, hot fluid at its initial temperature comes out through the remaining part of flow area at exit.

NOMENCLATURE:

H	height of the square duct
k	turbulence kinetic energy
T	non-dimensionalized temperature (T_l-T_c/T_h-T_c)
T_l	local temperature
T_c	temperature of the cold fluid
T_h	temperature of the hot fluid
u_j	velocity components
ε	turbulence kinetic energy dissipation rate
θ	Angle of the splitter plate, counterclock wise +ve.

1. INTRODUCTION

Mixing of fluid streams plays a very important role in many engineering applications like air-conditioning, environmental pollution and chemical reactors. Inadequate mixing in the air conditioning ducts is a major problem being faced by the car manufacturing industries. This is due to limited length and dimensions of the ducts and also due to minimum pressure drop requirement across the ducts.

The downstream development of the flow being mixed, is governed by the large coherent structures developed in the mixing region (Brown & Roshko (1974)). The dynamics of these coherent structures is very complicated and none of the present day turbulence models can successfully reproduce the full features of this type of flow (1980-81 AFOSR-HTTM Stanford conference on complex turbulent flows, proceedings pp. 731-1400). Despite its limitations the standard k-ε (Launder & Spalding (1974)) is usually applied to predict the flows of engineering interest. To eliminate the limitations several variants of k-ε model, relevant to this study (e.g. Cho & Chung (1992) & Speziale (1987)), have been proposed. But none of these models can reveal the full flow features and the

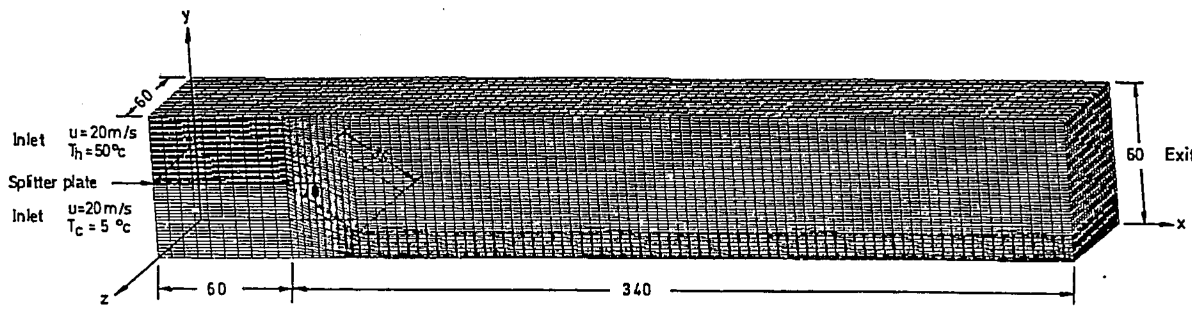


Fig. 1 Solution domain and computational grid

improvement achieved over the k-ε model is often insignificant.

The paper presents numerical study of hot and cold air mixing in a three dimensional square duct. The governing equations solved, are expressed in a general curvilinear coordinate system and closed using the standard k-ε model.

2. MATHEMATICAL MODEL AND CALCULATION METHOD

The mathematical model employed to obtain the predictions solves numerically the governing equations for the ensemble-averaged values of the components of the velocity vector, pressure, turbulence parameters and temperature. The turbulence parameters were obtained using the standard k-ε (Launder & Spalding (1974)). The governing equations for the velocity vectors take the form:

$$\frac{\partial\left(u_i u_j\right)}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \quad (1)$$

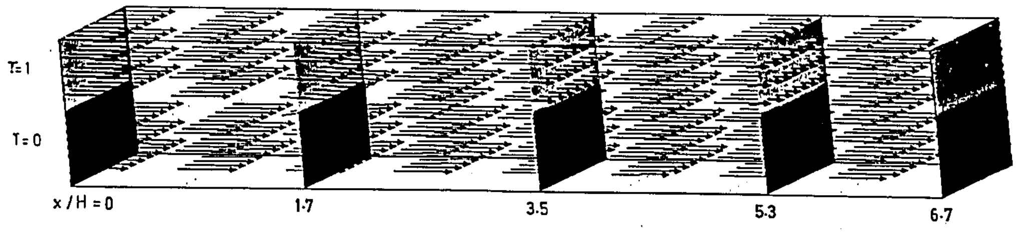
Where the effective viscosity μ_{eff} is obtained as $\mu_{eff}=\mu_{laminar}+\mu_t$ and $\mu_t=\frac{\rho C_\mu k^2}{\varepsilon}$. The turbulence kinetic energy (k), its dissipation rate (ε) and the temperature
(T) are obtained from the scalar equation given as:

$$\frac{\partial(u_j\phi)}{\partial x_j}=\frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\frac{\Gamma_\phi}{\sigma_\phi}\left(\frac{\partial\phi}{\partial x_j}\right)\right]+S_\phi \quad (2)$$

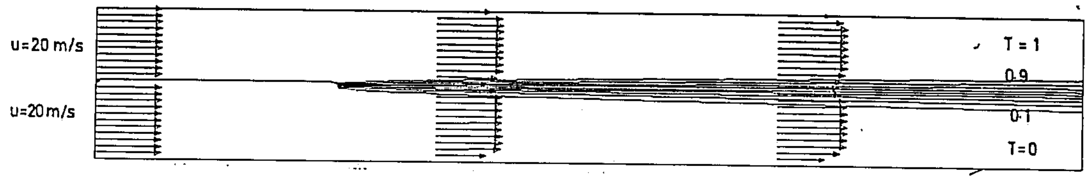
Here ϕ is a scalar property which stands for k, ε and T. Sϕ is the source term for the relevant scalar property. The equations were discretized in a finite volume fashion. Solutions were obtained using the SIMPLE algorithm. The Central Differencing scheme (CDS) and the Upwind Differencing scheme (UDS) were employed to evaluate the diffusion terms and the convective terms respectively. The software used in this study was developed by the author.

3. FLOW CONFIGURATION AND BOUNDARY CONDITIONS:

The flow configuration investigated here is a three-dimensional square duct, representing the mixing duct of a car air conditioning system. The configuration is in shown in Fig.1, along with the grid used. A splitter plate divides the hot and cold streams in the initial part of the duct. The study is performed at three different angles (θ= +30°, 0° & -30°) of the trailing part of the splitter plate. Grid dependence tests were performed. It was

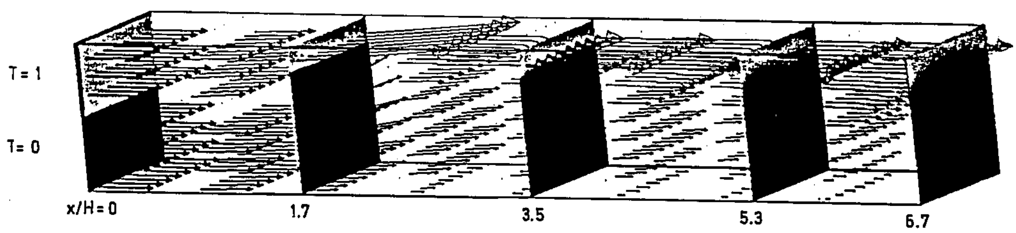


(a) 3-D velocity vector and temperature plots

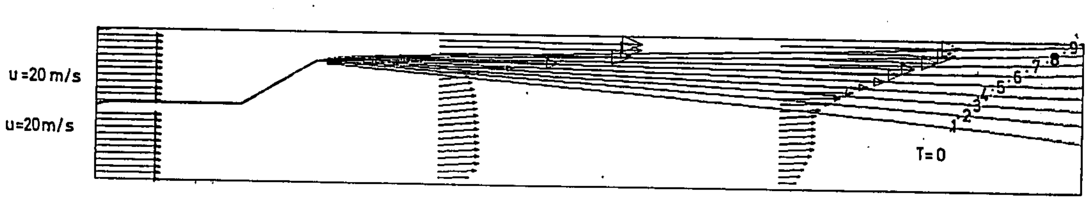


(b) Mid transverse plane velocity vector and temperature contours.

Fig. 2 Results for θ = 0°

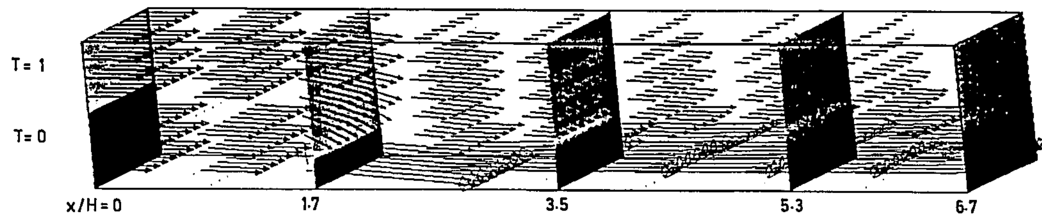


(a) 3-D velocity vector and temperature plots

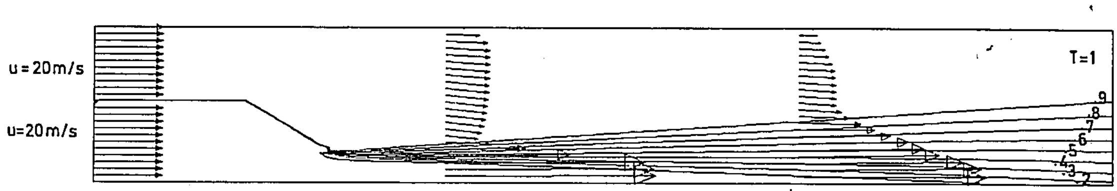


(b) Mid transverse plane velocity vector and temperature contours.

Fig. 3 Results for θ = +30°



(a) 3-D velocity vector and temperature plots



(b) Mid transverse plane velocity vector and temperature contours.

Fig. 4 Results for θ = -30°

found that 100 grids in the x-direction, 60 grids in the y-direction and 20 grids in the z-direction ensures a solution almost independent of grid distribution. Hence this grid arrangement was selected for this study.

Boundary conditions employed at inlet comprised of: uniform stream-wise air velocity u=20m/s for both hot and cold fluids; negligible turbulence kinetic energy with $k=0.01u^2$ and turbulence kinetic energy dissipation rate of $\varepsilon= C\mu\ k^{3/2}/0.1(0.5H)$, where $C\mu=0.09$ is a constant and H is the height of the square duct. The inlet temperatures for hot and for cold fluids were T=1.0 and T=0.0 respectively. At the exit plane the zero gradient boundary conditions were employed. The Reynolds number corresponding to inlet velocity and height of the duct is 80,000.

4. RESULT & DISCUSSIONS

The results are presented in the form of three dimensional velocity vector plots. The temperature distributions at some selected planes are also shown in the vector plots. The velocity vectors and the temperature contours for the mid transverse plane are presented in two dimensional plots.

For θ = 0°, shown in Fig. 2, the two streams flow in the same direction. The flow areas and the flow velocities, at the trailing edge of the splitter plate, for both the hot and the cold fluids are same. Figure shows the gradual development of velocities in the streamwise direction. Here mixing occurs in the limited shear layer region and extends up to 16% of the flow area at exit. Through the remaining available area, hot and cold fluids flow out at their inlet temperatures.

Situation improves for θ = +30° (Fig. 3), where the flow area for hot fluid is reduced to 1/3 and the average flow velocity increases to

3 times to that of the cold flow. The jet like action of issuing hot fluid creates a low pressure region in the upper part of the duct. This low pressure and the shearing action of the jet like flow leads to entrainment of cold fluid from the lower part. This ultimately leads to improved mixing. The mixing region now increases and covers up to 66% of the flow area at exit. Through the remaining part of the flow area, cold fluid flows out at its inlet temperature.

The situation reverses for $\theta = -30^{o}$ (Fig. 4). Here low pressure is created in the lower part of the duct. This low pressure and shearing action of the jet like flow leads to entrainment of hot fluid from upper part of the duct. The mixing region covers 66% of the flow area at exit. Through the remaining area hot fluid flows out at its initial temperature.

As this type of system is intended for use in a car air-conditioning system, the setting of $\theta=+30^{o}$ is recommended for summer. In summer, cold air at its inlet temperature is desirable rather than hot air at its inlet temperature. Similarly a setting of $\theta = -30^{o}$ is recommended for winter. However the mixing obtained here is not adequate.

ACKNOWLEDGMENT

The research was carried out with assistance from the Japan Society of Promotion of Science (JSPS). The assistance and suggestions by Professor N. Kasagi of the Mechanical Engineering Department of the University of Tokyo are gratefully acknowledged.

REFERENCES

Brown, G.L. and Rosko, A. , 1974, " On density effects and large structure on mixing layers", J. Fluid Mech., vol. 64, part 4, pp. 775-816.

Chung, M.K., and Cho, J. R., 1992, "A k-ε–γ equation turbulence model", J. Fluid Mech., Vol. 237, pp. 301-322.

Launder, B. E. and Spalding, D. B., 1974, "The numerical computations of turbulent flows", Comput. Meth. Appl. Mech. and Engng. , Vol. 3 , pp. 269-289

Speziale, C. G., 1987, "On nonlinear k-l and k-ε models of turbulence", J. Fluid Mech., Vol.178,pp.459-4