IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

*International Advanced Research Journal in Science, Engineering and Technology*
*Vol. 3, Issue 2, February 2016*

# CFD Analysis of a Pressure Drop in Astaggered Tube Bundle for a Turbulent Cross Flow

**Balaji.D**[1]**, L.S.S. Prakash**[2]

National Institute of Ocean Technology, Chennai, India[1, 2]

**Abstract:** Modelling and prediction of pressure drop across a staggered tube bundle for a liquid flow is an emerging development in the area of computational fluid dynamics. The present work discuss about the prediction of pressure drop using CFD and compared with theoretically estimated values using well known empirical friction factor correlations for model validation. Turbulence models such as a standard k-epsilon, a standard k-omega and a k-omega based shear stress transport were used for predicting the pressure drop in the tube bundle at same grid density. The primary objective of the present work is to find out which of these turbulence models predicts pressure drop values close to empirical correlations developed based on experiments similar to the present study. Transverse pitch ratio of2.5and longitudinal pitch ratio of 0.7 with 30$^o$ tube angle arrangement is chosen for thisstudy. For accurate prediction of pressure drop across the bundle fine grids were created around the tube portions. Grid independent study has been carried out for identifying suitable grid size for simulation.

**Keywords:** k-epsilon, k-omega, k-omega based shear stress transport, Pitch ratio, staggered tube bundle, Fluent 14.0.0.

**Nomenclature**

| | |
|---|---|
| $C_k$ | convection term in turbulent kinetic energy..$m^2.s^{-3}$ |
| d | tube diameter……………………………. ….m |
| $D_k$ | viscous diffusion in turbulent kinetic energy..$m^2.s^{-3}$ |
| k | turbulent kinetic energy …………….$m^2.s^{-2}$ |
| $P_k$ | production term in turbulent kinetic energy...$m^2.s^{-3}$ |
| $S_L$ | longitudinal pitch ..........................................m |
| $S_T$ | transverse pitch .............................................m |
| Re | Reynolds number based on inlet velocity .......... |

## 1. INTRODUCTION

Numerical simulation of flow across a tube bundle has found many practical applications suchas design of heat exchanger for predicting the heat transfer and pressure drop across bundles, in nuclear power plants for the design of cooling systems and also in the prediction of flow across the over-head cables [1].The prediction of turbulent flow in a tube bundle of staggered layout continue to attract interest because of its importance in the engineering application and also due to its complexity in prediction of flow viscous eddies which remains as a challenging problem for CFD [2].Turbulent flow inside a tube bundle exhibits three dimensional flow structures with fluctuating wake formation behind each tube accompanied by vortex shedding. Mechanism of vortex shedding and its suppression have significant effects on the various fluid-mechanical properties of practical interest such as fluid induced forces and pressure co-efficient [3]. In the present study, for the simplification of the simulation a two dimensional steady state has been chosen. It is reported in the literature that, in most of the tube bundles the steady state flow begins on the third row [3]. Pressure and velocity distribution on the inner of the bundle are not similar to that one on a single tube [4]. Re-circulation region in the rear of the tube and the intensity of the turbulence are largely governed by the relative pitches as well as the geometry of the bank. Velocity fluctuation is very intensive when the transverse pitch becomes shorter.

Literature survey indicates that correlations for predicting pressure drop in cross flow are extensively studied by many authors for the air flow compared to liquid flow. For the present study, twowell-known correlations for friction factor have been chosen fromliterature [5, 6, and 7] for the determination of pressure drop across tube bundle for liquid flow.

These empirical correlations were developed on the basis of the large number of experimental pressure drop data with arrangement similar to one that taken up for the present simulation study. In the present work transverse pitch ratio of 2.5 and longitudinal pitch ratio of 0.7 with 30$^o$ tube angle arrangement were employed fordifferent main stream velocity flow in a conduit. To perform this, three popular turbulence models such as a standard k-epsilon, a standard k-omega and a k-omega based shear stress transport were taken for simulation with same grid density. This helps in arriving to a conclusion that, which of the three models under same grid effectpredicts pressure drop close to the empirical correlation developed based on experiments. This study isalso helpful in understanding the turbulent behaviour of viscous eddiesas well asthe velocity fluctuations and resultant pressure drop variationshappening behind each row of the tube bundle.

## 2. GOVERNING EQUATION

For the present study the fluid flow is taken as two dimensional, steady, turbulent, incompressible and isothermal. The fluid is assumed to be a Newtonian havingconstant density (ρ) and dynamic viscosity (μ). The Reynolds Averaged Navier-Stokes (RANS) equation for continuity and momentum conservation is given as follows [10].

$$\frac{\partial U_j}{\partial x_j} = 0 \quad (1)$$

$$\frac{\partial}{\partial x_j}\left(\rho U_j U_i\right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu\frac{\partial U_i}{\partial x_j} - \rho\overline{u_i}\overline{u_j}\right) \quad (2)$$

ūi ūj is the Reynolds stress tensor, P is the mean flow pressure, Ui is the mean velocity component in the xi direction. To compute the Reynolds stress tensor, three turbulence models are used which are discussed below.

## 3. TURBULENCE MODELS

Commercial software package, FLUENT 14.0.0 was used for performing the analysis of pressure drop across the staggered tube bundle. Several turbulence models are available such asa standard k-epsilon, a standard k-omega and a k-omega based shear stress transport which was employed for pressure drop prediction.

### 3.1. The Standard k-Epsilon model (k-ε)

The k-ε model is a popular turbulence model which is widely accepted in the industries. This model is developed by Launder and Spalding (1994) [8]. This model was derived based on the assumption that the flow is fully turbulent and molecular viscous effects are negligible [9]. The turbulent kinetic energy, k, equation for the standard k-ε model reads as follows [1].

$$\frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + P_k - \rho\varepsilon \quad (3)$$

Where, turbulent production rate is

$$P_k = \mu_t\left(\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right) \quad (4)$$

The dissipation rate, ϵ , equation for the standard k- ϵ is given below.

$$\frac{\partial}{\partial x_j}\left(\rho U_j \varepsilon\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right) + \frac{\varepsilon}{k}\left(C_{\varepsilon 1} - \rho C_{\varepsilon 2}\varepsilon\right) \quad (5)$$

Where turbulent viscosity, $\mu_t$ is given as follows

$$\mu_t = \rho\frac{C_\mu}{\varepsilon}k^2 \quad (6)$$

Constants:

$C_{\epsilon 1}$=1.44, $C_{\epsilon 2}$=1.92,$\sigma_k$ = 1.0, $\sigma_k$ = 1.3, $C_\mu$ = 0.09

### 3.2. The Standard k-Omega model (k-ω)

The standard k-ω model is one of the most common turbulence models.This model was developed by Wilcox (1998) [11].This model consist of two extra transport equations one is for turbulent kinetic energy, k, similar to the standard k-ε model and other one is for specific dissipation, ω, which can be called as ratio of ε to k [12].The turbulent kinetic energy, k, equation for the standard k- ω model reads as follows [1].

$$\frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + P_k - \rho\beta^* k\varepsilon \quad (7)$$

The specific dissipation rate, ω , equation for the standard k- ω is given below.

$$\frac{\partial}{\partial x_j}\left(\rho U_j \omega\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right) + \alpha\frac{\omega}{k}P_k - \rho\beta\omega^2 \quad (8)$$

Where turbulent viscosity, $\mu_t$ is given as follows.

$$\mu_t = \rho\left(\frac{k}{\omega}\right) \quad (9)$$

Constants:

$\beta^*$=0.09, β =0.075,α = 5/9,$\sigma_\omega$ = 2, $\sigma_k$ = 2

### 3.3. The Shear Stress Transport k-ω (SST)

Mentor(1994)[13] developed this model which combines the capabilities of the k-ε turbulence model away from the walls and robustness of k-ω turbulence model near walls [1].

$$\frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_{k_2}}\right)\frac{\partial k}{\partial x_j}\right) + P_k - \rho\beta^* k\varepsilon \quad (10)$$

$$\frac{\partial}{\partial x_j}\left(\rho U_j \omega\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right) + \alpha\frac{\omega}{k}P_k - \rho\beta\omega^2 + (1 - F_1)2\rho\frac{1}{\sigma_{\omega_2}\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \quad (11)$$

Where the blending function, F1, is given by:

$$F_1 = \tan(\arg_1^4) \quad (12)$$

$$\arg_1 = \min\left(\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\rho k}{CD_{k\omega}\sigma_{\omega 2}y^2}\right) \quad (13)$$

$$CD_{k\omega} = \max\left(2\rho\frac{1}{\sigma_{\omega 2}\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}, 1.0 X 10^{-10}\right) \quad (14)$$

$$F_2 = \tanh(\arg_2^2) \quad (15)$$

$$\arg_2 = \max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{\omega y^2}\right) \quad (16)$$

$$S = \sqrt{\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}} \quad (17)$$

Where turbulent viscosity, $\mu_t$ is given as follows

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, SF_2)} \qquad (18)$$

Constants:
α2=0.44, σω2 =10.856, σk2 = 1, β2 = 0.0828, β* = 0.09,a1=0.31

## 4. PRESSURE DROP CORRELATIONS FOR FLOW ACROSS STAGGERED TUBE BUNDLES

Based on the literature survey two correlations have been selected for estimating the pressure drop, since these correlations are developed based on the experimental study conducted across tube bundle for the liquid flow which is similar to the present study.Zukauskas (1972) [5] proposed the following correlation for estimating the pressure drop across tube bundles.

$$\Delta p = N_L . x . \left( \frac{\rho V_{\max}^2}{2} \right) f \qquad (19)$$

Where f is the friction factor, NL is the no of rows and the correction factor x are presented graphically for staggered tube banks. Similarly Taborek (1983) [6] developed the following correlation for determining pressure drop across tube bundle for both in-line and staggered layout.

$$f = 4 b_1 . \left( \frac{1.33}{S_T / d} \right)^{b} (\mathrm{Re})^{b_3} \quad b = \frac{b_3}{1 + 0.14\,(\mathrm{Re})^{b_4}} \qquad (20)\ (21)$$

Where f is the friction factor, ST is the transverse pitch ratio; d is the outer diameter of the tube, Re is the Reynolds number, b, b1, b3, b4 are constants.

## 5. SIMULATION MODEL AND BOUNDARYCONDITIONS

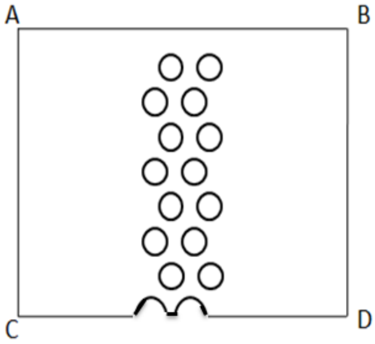

Fig.1 Simulation model without meshing

Table 1: Boundary conditions

| Sl.No | Region | Boundary condition |
|---|---|---|
| 1 | Line AB | Wall surface |
| 2 | Line AC | Velocity inlet |
| 3 | Line BD | Pressure outlet |
| 4 | All full circle tubes | Wall surface |
| 5 | Line CD | Symmetric |
| 6 | Interior zone ABCD | Fluid |

The problem is to simulate a turbulent flow over a staggered two dimensional domain of a tube bundle as shown in fig.1. The boundary condition for the simulation model is given in table.1. The bundle of tube consists of uniformly spaced tubes of diameter 0.0213m which are staggered in the direction of the cross flow fluid. The transverse pitch ratio and longitudinal pitch ratio of the bundle is 0.0514m and 0.016m respectively. Depth of the tube bundle is around 0.069m. Since the tube bundle geometry is symmetric, it is sufficient to model only a portion of the domain. A flow velocity of 1m/s is applied to the inflow boundary of the module. The properties of the water are given in the table.2.

Table 2: Material properties

| Properties | Values |
|---|---|
| Velocity at the inlet, V | 1 m/s |
| Dynamic viscosity, µ | 0.001 N-s/m$^2$ |
| Density of water, ρ | 1026 kg/m$^3$ |
| Specific heat capacity of water, Cp | 4.186 kJ/kg.K |
| Hydraulic diameter | 0.45m |
| Model width | 0.5m |
| Tube OD | 0.0213m |
| Transverse pitch, $P_T$ | 0.0514m |
| Longitudinal pitch,$P_L$ | 0.016m |

## 6. GRID INDEPENDENCY STUDY

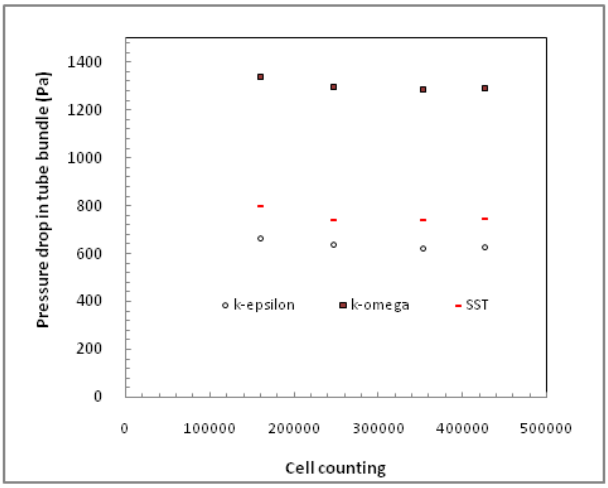

Fig.2Cell counting vs. Pressure drop in tube bundle

The section of model considered for the analysis was subjected to the grid independency study. Studies were carried out on different meshing scheme for determining how the grid size of the mesh affect the pressure drop across the tube bundle under three different turbulence models for a constantinlet flow velocity of 0.5m/sec. Grid sizes such as coarse, medium, fineand very fine grids were generated and simulation was performed on each of the grid size for determining the pressure drop.Grid independent study for each of the turbulence model with respect to cell counting is shown in fig.2, which clearly indicates that pressure drop changes by a very small amount for different cell counting for each of the model.

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
*International Advanced Research Journal in Science, Engineering and Technology*
*Vol. 3, Issue 2, February 2016*

Table.3.Comparison of grids

| Grid | Elements | Nodes |
|---|---|---|
| Coarse | 160940 | 164463 |
| Medium | 247598 | 251613 |
| Fine | 353648 | 358493 |
| Very fine | 426570 | 432721 |

Two important factors for selection of grid size includes, the time for completion of mesh generation and effect of the mesh on the pressure drop across tube bundle. Mesh generation time for finer and very fine mesh is comparatively more than that for coarse and medium mesh. For solving this problem, medium mesh was taken for present analysis work; because the medium mesh model performed faster compared to other grid sizes without significant loss in solution accuracy.

## 7. MODEL VALIDATION AND DISCUSSIONS OF RESULTS

CFD simulation has been carried out for a steady state fluid flow over a staggered tube bundle modelled in FLUENT. Simulation was done using three different turbulent models such standard k-ε, standard k-ω, SST for same grid density. The model is used to predict the pressure drop across tube bundle for different upstream velocity of flow. Different meshing scheme are generated and analysed. It was found that the medium grid density meshing gives pressure drop values at a minimum computation time without losing any solution accuracy. Quadrilateral meshing was applied for the zone around the tubes. Material properties, upstream fluid velocity and boundary conditions were kept constant for each of the turbulence models.

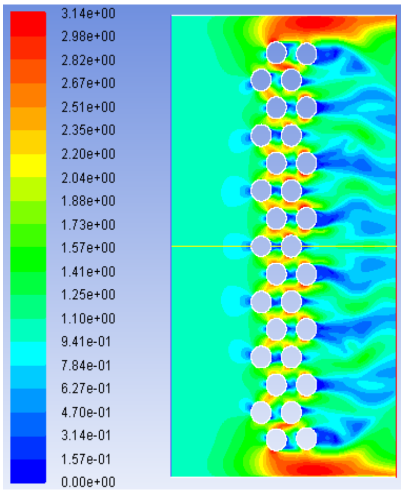

Fig.3 Velocity contour (k-ε model)

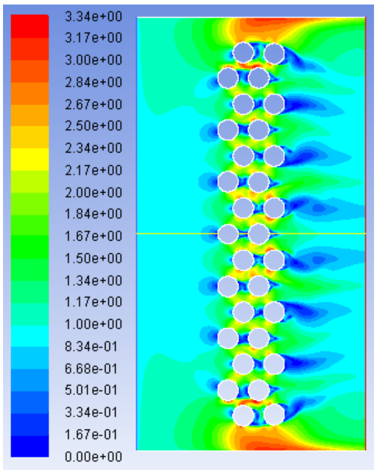

Fig.4 Velocity contour (k-ω model)

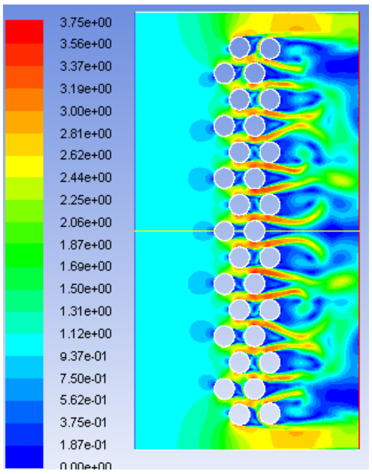

Fig.5 Velocity contour (SST model)

Cross flow over a tube exhibits a complex flow pattern. The fluid flowing towards the mid portion of tubes gets split up and encircles the tubeto form an imaginary layer called boundary layer around the tube. The fluid particle which strikes the centre of the tube is brought to rest and this point of rest is called stagnation point, where the pressure of the fluid particle increases as a result of obstruction to flow. As the flow of fluid passes around the tubes, the pressure gradually reduces and reaches to a minimum at the top curve of the tube compared to stagnation point pressure in the front side of the tube. This drop in local pressureis due to the development of separation of boundary layer that leads to a formation of wake behind each of the tubes.Velocity becomes zero at mid region of the tube perpendicular to the fluid flow and gradually increases to a maximum when fluid reaches the top side of the tube diameter. This is due to reduction in cross sectional area of the flow passage between adjacent tubes. Velocity decreases when the fluid re-joins at the rear side of the tube. The variation of the fluid velocity as a result of vortex shedding for each of the turbulence model across tube bundle wassimulated for different inlet velocity. Velocity contour for the k-ε model is shown in the fig.3 and the velocity contour for the k-ω model and SST model is shown in the fig.4 and fig.5 respectively.

Velocity contours of each of the model clearly depict the vortex shedding and its propagation extended to a fewer distance behind each of the tube rows of the bundle. The propagation of the velocity fluctuation was found to be more in SST model compared to k-ε model and k-ω model for the same fluid flow velocity. Propagation of velocity fluctuations manifested as eddies or swirl motion from last row of the tube bundle seems to be less for k-ω model compared to the other two models, since ω equations

DOI 10.17148/IARJSET.2016.3209

shows a strong sensitivity to the values of ω in the free stream outside the boundary layer [13].

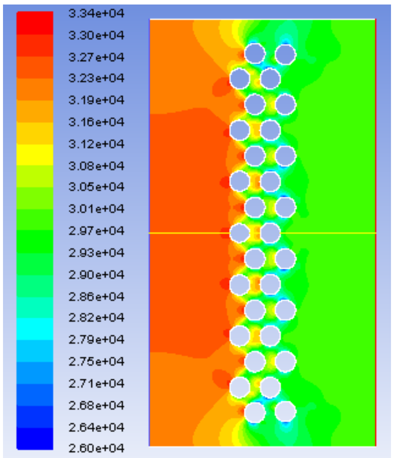

Fig.6 Pressure contour (k-ε model)

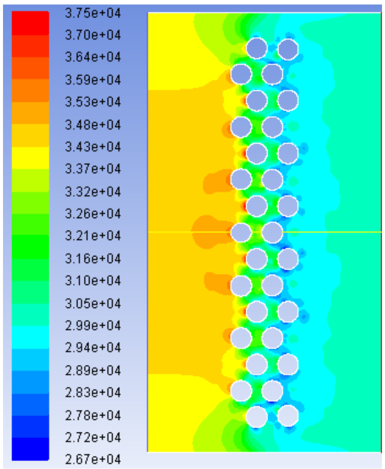

Fig.7 Pressure contour (k-ω model)

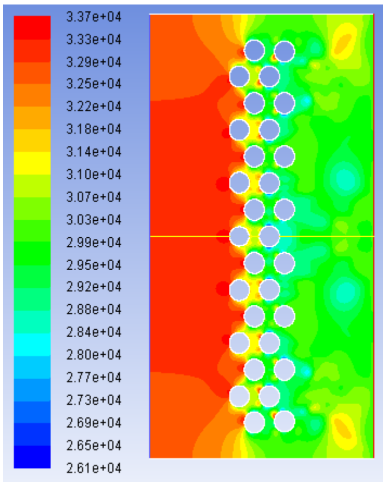

Fig.8 Pressure contour (SST model)

Performance of k-ω model in capturing the viscous eddies within the fluid flow is poor; however it shows a superior performance in the near wall region which cannot be achieved in k-ε model.Pressure contour of k-ε model matches with the SST model to some extent compared to the k-ω model for same meshing scheme and grid density as shown in fig.6, 7 and 8.Because the SST modelis inclusive of both k-ω model activated only near wall region and k- ε model activated in region away from the wall (Menter, 1992)[14].Since the k-ω model is not activated in the region away from wall, the potential errors due to free steam sensitivity can be avoided that could lead to matching of SST model contour withk-ε model moderately.However the pressure drop value predicted by the standard k-ω model is higher compared to SST and k-ε model. This isbecause of the potential errors that results from the free stream sensitivity of this k-ω model, which in turnlead to over prediction of pressure drop values.Apart from that, the grid sensitivity of the k-ω model in the near wall region due to finer grid size around tubes also has the potential for over prediction of pressure drop results compared to the other two turbulence models. Whereas the grid sensitivity in wall region for finer meshes is less in SST and k-ε model. Menteret al (2003) [15] reported that the low sensitivity to the grid spacing is important for industrial flow predictions, where typically not all walls can be resolved with fine grids and also it is stated that the SST model is less sensitive to the grid variations.

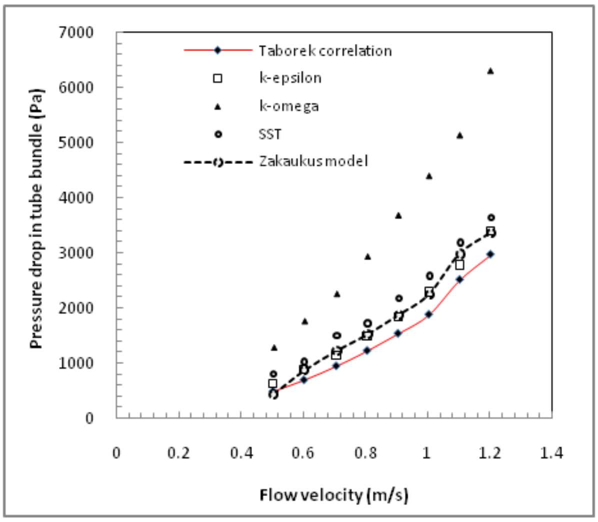

Fig.9 Flow velocity vs. tube bundle pressure drop

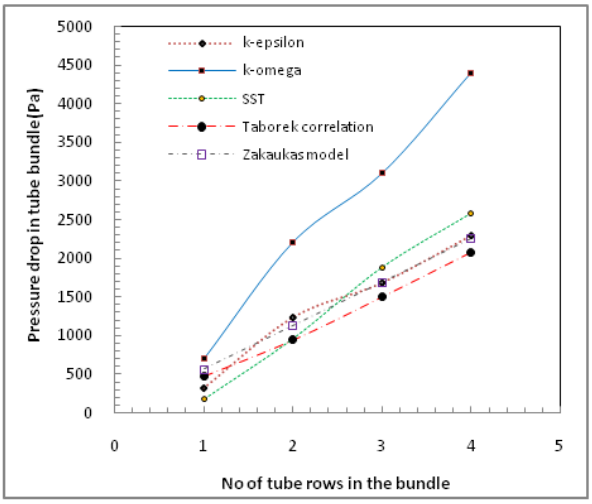

Fig.10 No of rows vs. tube bundle pressure drop

For model validation, the simulation results of tube bundle pressure drop was compared with theoretically predicted values calculated using empirical correlations such as Zukauskas (1972) and Taborek (1983). These empirical correlations were developed on the basis of the large number of experimental pressure drop data with experimental set-up similar to the present simulation study. Using these correlations the pressure drop values for different upstream flow velocity as well for different number of tube rows have been calculated and plotted in the fig.9 and fig.10 respectively. The pressure drop values across tube bundle for different numbers of tube rows are obtained for a flow velocity of 1m/sec as shown in fig.10. Pressure drop increases linearly with respect to the flow velocity at the inlet. These pressure drop values are obtained for the $30^o$ tube layout with pitch ratio of 2.5 and 0.7 in transverse and longitudinal direction respectively. The fig.9 and fig.10 clearly indicates that the simulation results follow the pattern of empirical correlations with good agreement. However the k-ω model over predicts the pressure drop value when compared to the other two model simulation results as well as correlations values. As each of the model possess its own advantages and limitations, such that the k-ω model unable to predict the viscous eddies developed within the fluid flow as well as the grid sensitivity of this model for finer wall grids might have led to over prediction of pressure drop value compared to remaining models for same meshing scheme and grid density.

## 8. CONCLUSION

CFD simulation has been carried out for a steady state fluid flow over a staggered tube bundle modelled in FLUENT software. Simulation was done using three different turbulent models namely standard k-ε, standard k-ω, SST for same grid density for predicting the tube bundle pressure drop. For validating the model, the simulation results were compared with the theoretically predicted values calculated using well known empirical correlations. From the comparison, it was found that the pressure drop results of a standard k-epsilon model shows good agreement with the correlations followed by k-omega based shear stress transport. It was also found that for the same grid density and meshing scheme the simulation results of standard k-omega model over predicts the pressure drop values compared to other two models as well as theoretical values calculated using empirical correlations.

### Greek symbols

$\varepsilon$ turbulent dissipation rate ($m^2$ s-$^3$)
$\varepsilon_k$ dissipation term in the turbulent kinetic energy ($m^2$ s-$^3$)
$\mu_t$ turbulent eddy viscosity (N s $m^{-2}$)
$\mu$ fluid dynamic viscosity (N s $m^{-2}$)
$\rho$fluid density (kg $m^{-3}$)
ω specific dissipation rate ($s^{-1}$)

## REFERENCES

[1] S.S. Paul, S.J. Ormistan and M.F. Tachie, Experimental and numerical investigation of turbulent cross-flow in a staggered tube bundle, International Journal of Heat and Fluid Flow, 29(2008) 387-414.

[2] Karnav N Shah, Kush V Mahant, Priyank D Patel and Chetan O Yadav, CFD analysis of heat exchanger over a staggered tube bundle for different angle arrangement of tube bundles, International Journal of Engineering Research and Technology, 2 (1) (2013).

[3] A.Zukauskas andJ.Ziugzda, Heat transfer of a cylinder in cross flow, Washington D.C: Hemisphere, 1985.

[4] E.S.Gaddis and V.Gnielinsky, Pressure drop in cross flow across tube bundles, International Journal of Chemical Engineering, 25(1) (1985) 1-15.

[5] A.Zukauskas, Heat transfer from tubes in cross flow,Advances in Heat Transfer, Academic press, Newyork, 8 (1972).

[6] J.Taborek, Shell and tube heat exchangers: Single phase flow, Hand book of heat exchanger design, Hemisphere, Chap.3.3.

[7] Bejan, Adrian, Convection heat transfer, Third edition,John Wiley and sons, Inc. Hoboken, NJ, 2004.

[8] Launder and Spalding, The numerical computation of turbulent flows, Computational Methods for Applied Mechanical Engineering, 3 (1994) 269-289.

[9] Launder and Spalding, The numerical computation of turbulent flows, Computational Methods for Applied Mechanical Engineering, 3 (1994) 269-289.

[10] SamyM.El-Behery, MofrehH.Hamed, A comparative study of turbulence models performance for turbulent flow in a planar asymmetric diffuser, World Academy of Science, Engineering and Technology53 (2009).

[11] Wilcox, Multiscale model for turbulent flows, AIAA J, 26 (1998) 1311-1320.

[12] Wilcox, Turbulence modelling for CFD, DCW industries, La Canada, California, 1998.

[13] Mentor, Two equation eddy viscosity turbulence models for engineering applications, AIAA J, 32 (8) (1994) 1598-1605.

[14] Mentor, Influence of free stream values on k-ω turbulence model predictions, AIAA J, 30 (6) (1992) 1657-1659.

[15] F.R.Mentor, M.Kuntz, Adaptation of eddy viscosity turbulence models to unsteady separated flow behind vehicles, Proc.Conf. The Aerodynamics of Heavy Vehicles: Trucks, Buses and Trains, Asilomar, Ca (2003).