

CFD analysis of flow past staggered tube array using 2D RANS turbulence models

Nikhil Sharma

Department of Mechanical Engineering
Malaviya National Institute of Technology Jaipur, Rajasthan India

Abstract

The aim of this project is to utilize periodic boundaries to simulate flow past a staggered tube bank array using the open source CFD package OpenFOAM. The flow is simulated using pimpleFoam solver and the results are compared with experimental results from the ERCOFTAC database as well as the available computational results. Performance of different RANS models in predicting the flow is also compared.

1. Introduction

A tube bank consists of tube bundle bathed in a fluid, generally utilised to either cool or heat the material flowing through the tube. Flow through a tube bundle has many industrial applications. Common applications can be seen in air conditioning, refrigeration systems, and chemical processing plants where a material is to be cooled or heated efficiently. Using tube bank increases surface to volume ratio thus improving the rate of heat transfer, thus are commonly employed as design elements in heat exchangers.

A tube bank is characterized by stream-wise and transverse pitch to diameter ratios. Tube banks can be classified as compact or widely-spaced depending on product of the two pitch to diameter ratios[2]:

$$\begin{aligned} \text{for compact tube banks: } \frac{S_L}{D} * \frac{S_T}{D} &< 1.25 \\ \text{for widely-spaced tube banks: } \frac{S_L}{D} * \frac{S_T}{D} &> 1.25 \end{aligned} \tag{1}$$

There are two basic tube bank patterns referred to as inline and staggered. The inline tube bank involves a rectangular primitive unit while a staggered tube bank involves rhombic primitive unit[1] as shown in figure 1 below. In this case study, flow past a staggered tube bank will be studied.

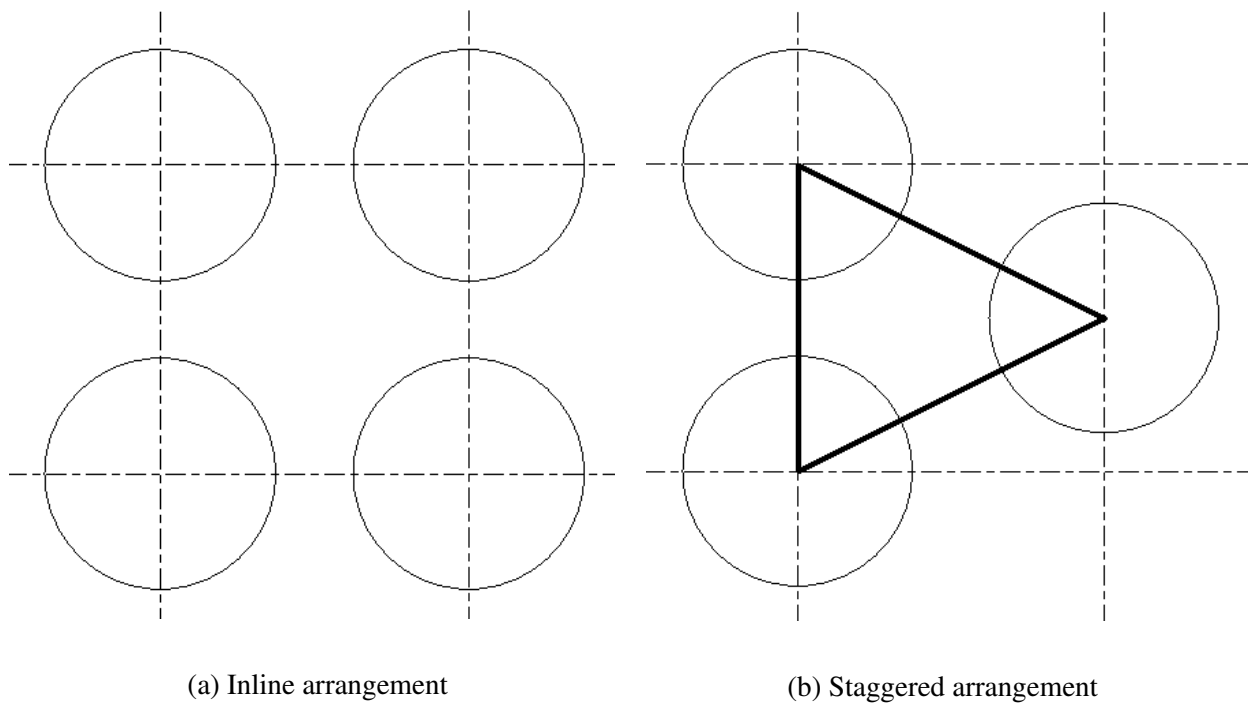


Figure 1: Different tube bank patterns

2. Problem Statement

Experimental studies on an iso-thermal flow through a staggered tube bank were performed by Simonin and Barcouda[5]. Experimental database of the work was released through the second ERCOFTAC(European Research Community on Flow Turbulence and Combustion) IAHR Workshop on Refined Flow Modeling in June 1993. Over the years the database has become a benchmark and a number of experimental and computational studies have been performed on the same.

In the experimental results, the flow is observed to reach a periodic state after flowing past initial tube rows. It has been suggested that flow periodicities in tube banks may be due to a number of different mechanisms rather than just one[4]. This has led to development of two different schools of thought. Many researchers suggest exploiting symmetry of the geometry and utilizing periodic boundaries in both transverse and stream-wise directions. On the other side, researchers advocate not using periodic boundaries in stream-wise direction as that would consider infinite number of tube rows and include effects of wake shed by one tube row impinging on the previous one [3].

The objective of this project is to simulate flow past staggered tube bank array utilized by Simonin and Barcouda[5] using different 2D RANS model and validate it against their experimental results as well as computational results obtained by N. Kulasekharan and B. Prasad [3]. Figure 2 shows the experimental domain. Periodic boundary condition will be applied in both transverse and stream-wise direction and geometry will be modelled as a "unit cell" consisting of one tube in the centre and parts of four tubes in the surrounding.

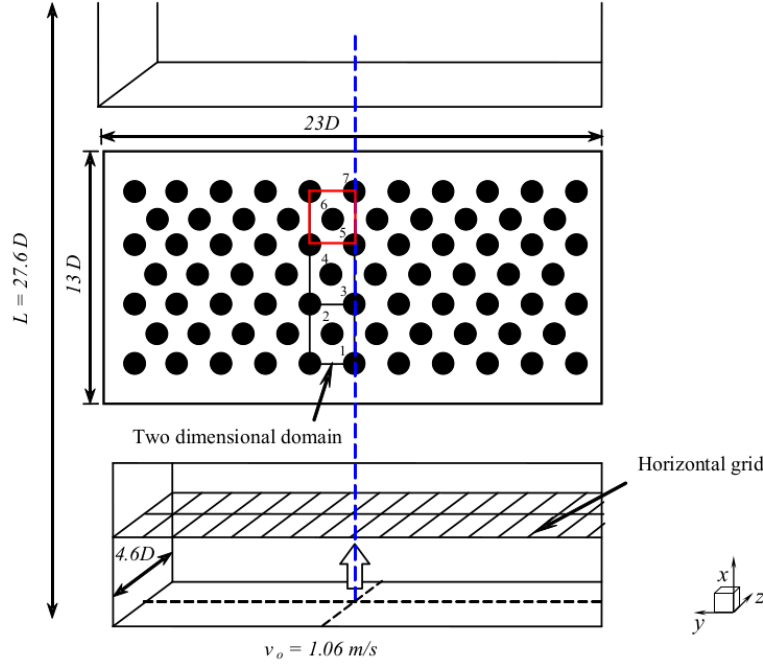


Figure 2: Experimental domain of Simonin and Barcouda[5]

3. Governing Equations

pimpleFoam equations

Following governing equations are solved by the pimpleFoam solver:

$$\nabla \cdot U = 0 \quad (2)$$

$$\frac{\partial U}{\partial t} + \nabla \cdot (U \otimes U) - \nabla \cdot \mathbf{R}^{\text{eff}} = -\nabla p + \mathbf{S}_u \quad (3)$$

Here, \mathbf{R}^{eff} is the stress tensor and \mathbf{S}_u is the momentum source. Reynold's stress is modelled by different RANS models available. The transport equations for the turbulence models are not repeated here for the sake of conciseness.

Equations for Cyclic boundary

In cases where periodic boundaries are employed, pressure field is divided into two components:

$$\begin{aligned} p(x) &= \beta x + \tilde{p}(x) \\ \beta &= \frac{p(x) - p(x + L)}{L} \end{aligned} \quad (4)$$

In equation 4, the term βx relates to the global mass flow and $\tilde{p}(x)$ which is the reduced pressure responsible for local motion. From the OpenFOAM implementation it is observed that pressure field in equation with cyclic boundary represents reduced pressure.

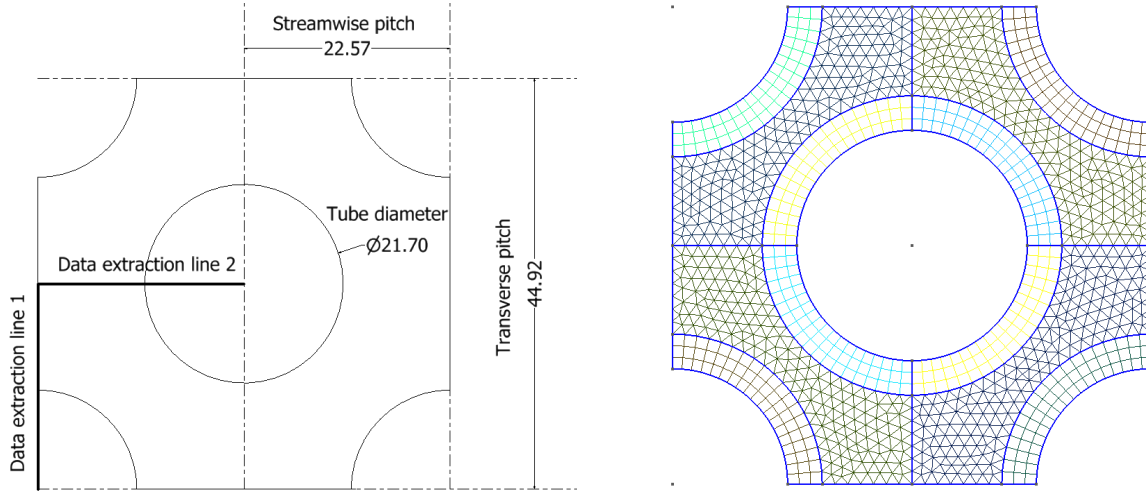
4. Simulation Procedure

Setting up a case in OpenFOAM requires three directories namely *0*, *constant* and *system*. Simulation is run by typing the commands in terminal. Transient, incompressible, turbulent solver *pimpleFoam* is used for the simulations. Simulation is run till a fully developed profile is achieved. Table 1 below shows the different flow parameters. The parameters have been taken from the ERCOFTAC database. Two different RANS model: standard $k - \epsilon$ and $k - \omega$ SST will be utilized and separate cases for the two models will be made.

Parameter	Present simulation
Fluid	Water
Velocity at the inlet (ms^{-1})	1.06
Hydraulic diameter (m)	0.0217
Fluid dynamic viscosity ($\text{kgm}^{-1}\text{s}^{-1}$)	7.98×10^{-4}
Fluid density (kgm^{-3})	996
Tube diameter, D (m)	0.0217
Stream-wise pitch to diameter ratio ($\frac{S_L}{D}$)	1.04
Transverse pitch to diameter ratio ($\frac{S_T}{D}$)	2.07

Table 1: Flow parameters used in simulation

4.1 Geometry and Mesh



(a) Geometry of the unit cell

(b) Computational domain

Figure 3: Specification of geometry and Computational domain

Figure 3 shows the geometry of 2D computational domain and the mesh. All the dimensions shown in figure 3a are in millimeters. The mesh shown in figure 3b is a hybrid mesh consisting of hexahedral cells near the tube surface and triangular cells everywhere else. Meshing was done using

Gmsh version 4.6.0. After converting the mesh to OpenFOAM format using *gmshToFoam* command, boundary patch type and separation between the cyclic patches are specified in the *boundary* file located in *constant/polyMesh* directory. The mesh has been refined such that the first cell near the wall lies in the log-law region.

4.2 Boundary Conditions

Boundary conditions used in the two cases are as follows:

- **U**

```
inlet : cyclic
outlet: cyclic
wall : noSlip
top : cyclic
bottom : cyclic
frontAndBack : empty
```

- **epsilon**

```
inlet : cyclic
outlet: cyclic
wall : epsilonWallFunction
top : cyclic
bottom : cyclic
frontAndBack : empty
```

- **p**

```
inlet : cyclic
outlet: cyclic
wall : zeroGradient
top : cyclic
bottom : cyclic
frontAndBack : empty
```

- **omega**

```
inlet : cyclic
outlet: cyclic
wall : omegaWallFunction
top : cyclic
bottom : cyclic
frontAndBack : empty
```

- **k**

```
inlet : cyclic
outlet: cyclic
wall : kqRWallFunction
top : cyclic
bottom : cyclic
frontAndBack : empty
```

- **nut**

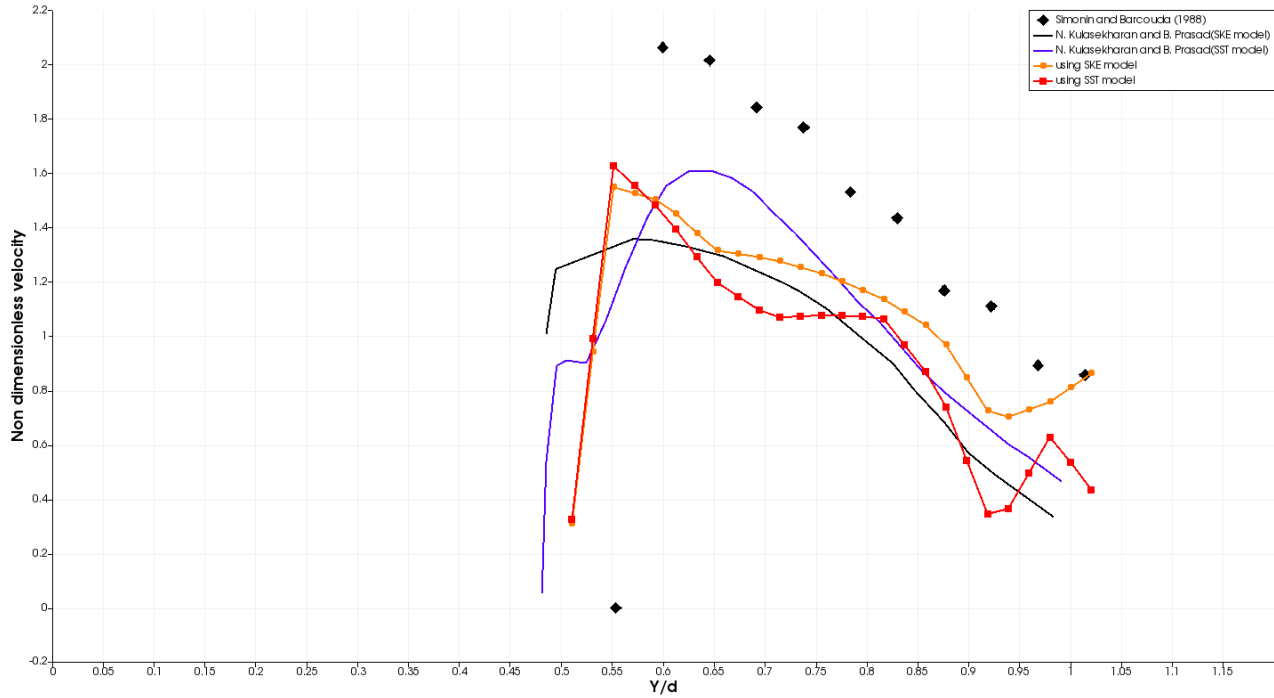
```
inlet : cyclic
outlet: cyclic
wall : nutKWallFunction
top : cyclic
bottom : cyclic
frontAndBack : empty
```

4.3 Solver

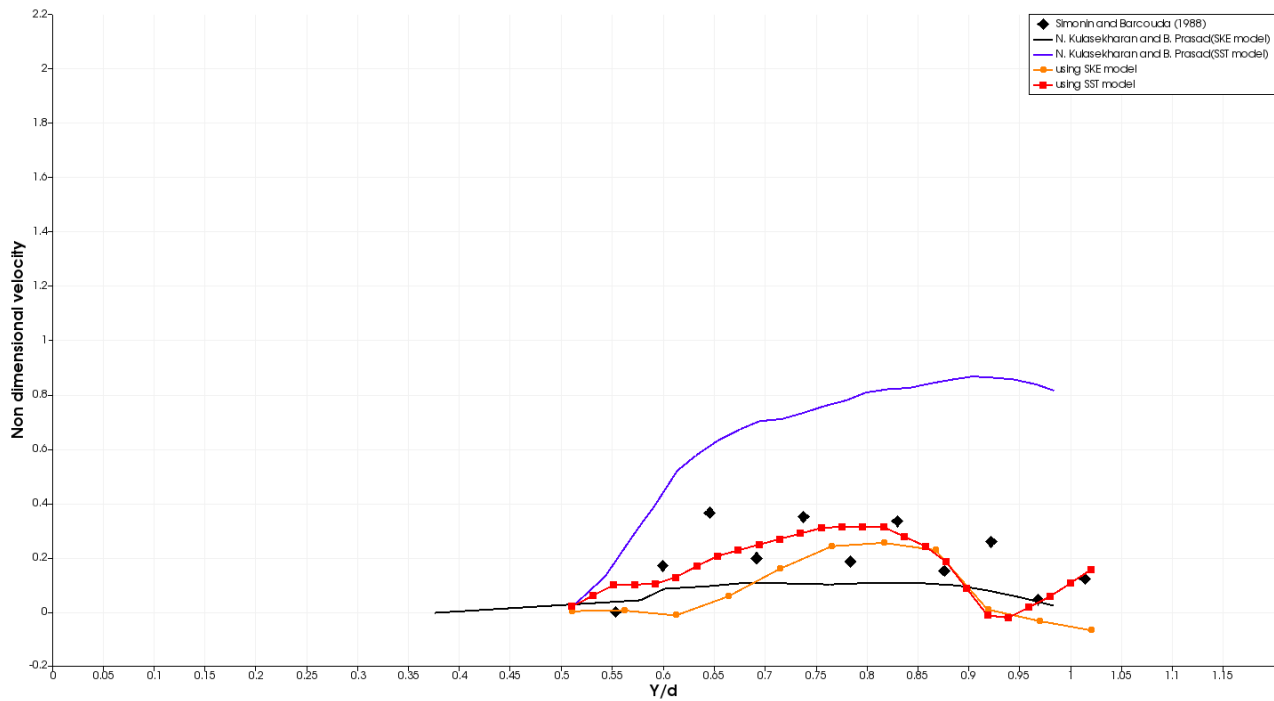
To analyze a transient, turbulent flow with cyclic boundary in our case, *pimpleFoam* solver is used. The solver utilizes **PIMPLE** algorithm which is a combination of **PISO** (Pressure Implicit with Splitting of Operator) and **SIMPLE** (Semi-Implicit Method for Pressure-Linked Equations). The additional source term needed to drive the flow is defined by *fvOptions* utility specifying single region momentum source namely "*meanVelocityForce*".

5. Results and Discussion

5.1 Plot of Non-dimensional velocity

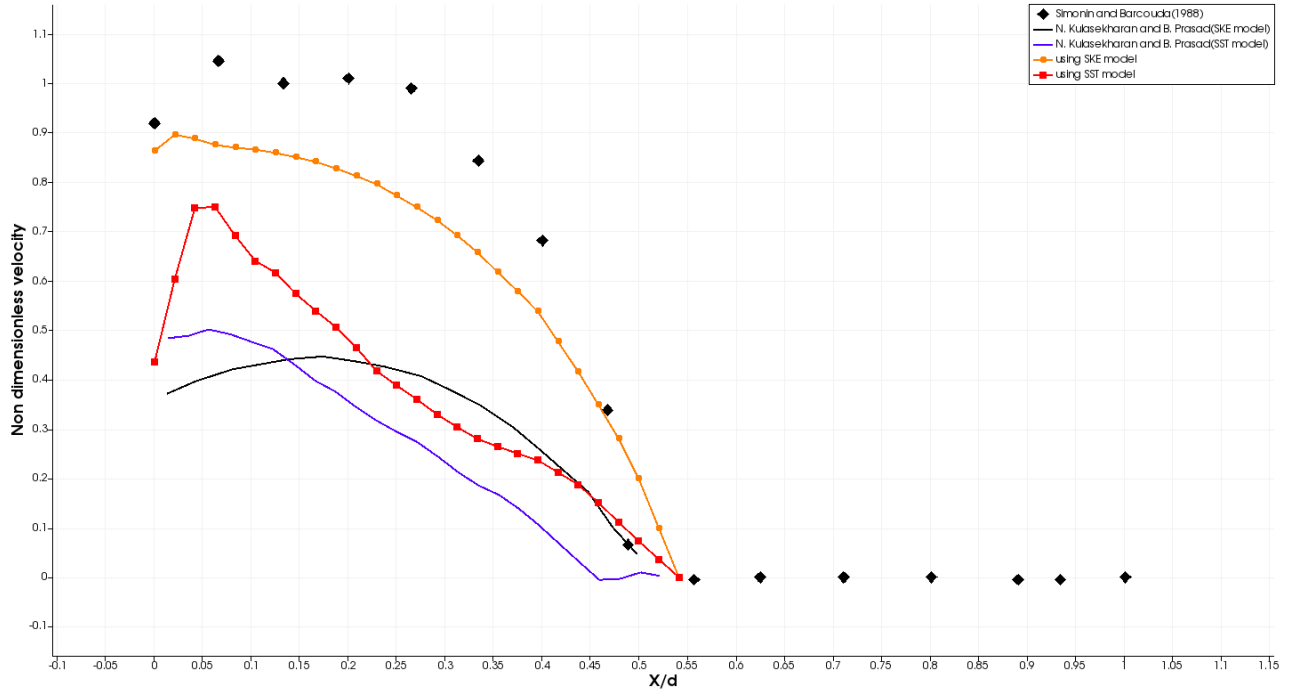


(a) Streamwise velocity

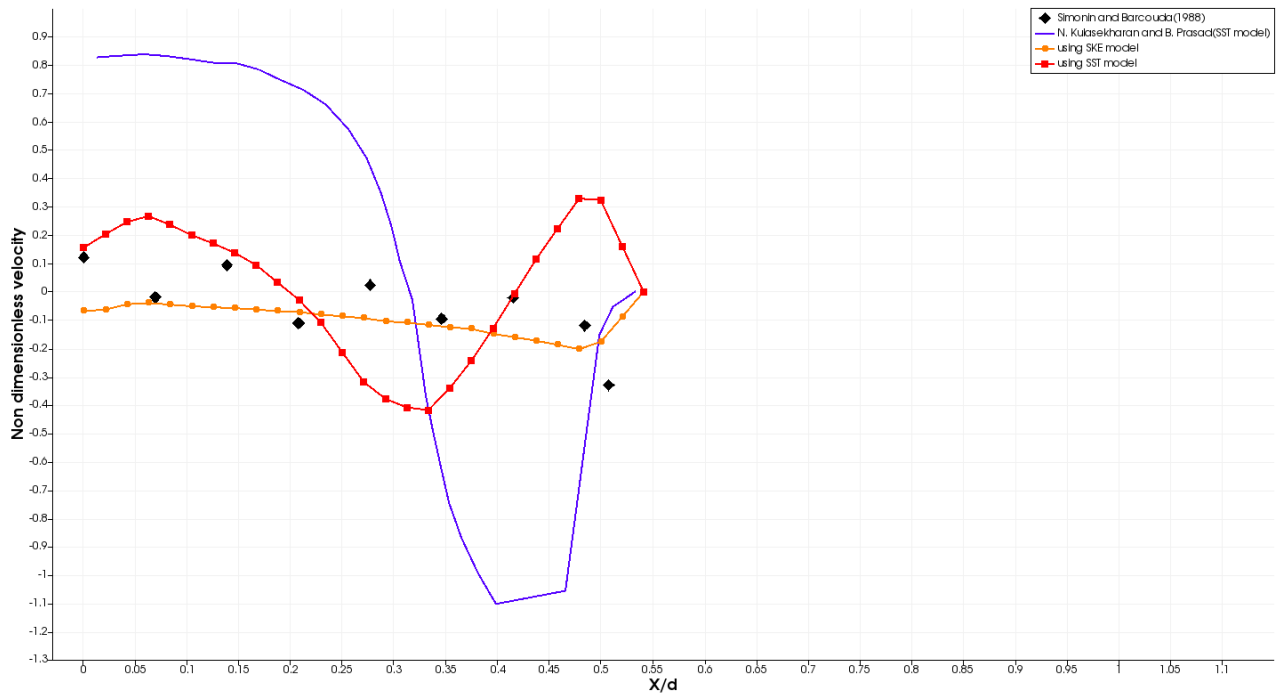


(b) Transverse velocity

Figure 4: Plot of non dimensional velocity along data extraction line 1



(a) Streamwise velocity



(b) Transverse velocity

Figure 5: Plot of non dimensional velocity along data extraction line 2

In the original experiment performed by Simonin and Barcouda(1988), data was extracted along five different lines using LDA technique. The results are available in the Classic Collection Database of ERCOFTAC. However in the present case, data is extracted only along two lines ($x=0$ and $y=22.5\text{mm}$ line) as shown in figure 3a. Present results are compared with the experimental data as well as the computational results obtained by Kulasekharan and Prasad(2009) who considered periodicity only

in the transverse direction.

Both transverse and stream-wise components of the velocity are made non dimensional by dividing it with bulk mean velocity and are plotted against non-dimensional distance along the data extraction line. Figures 4 and 5 show the plot of non dimensional velocities along the two data extraction lines. From the plots it is seen that both flow predictions made by standard $k - \epsilon$ and $k - \omega$ SST models are not very accurate. The average percentage error along the data extraction lines are shown in table 2 below.

		Standard $k - \epsilon$ model		$k - \omega$ SST model	
		Present case	Kulasekharan & Prasad	Present case	Kulasekharan & Prasad
Line 1	Stream-wise velocity	26.19	40.78	38.67	28.79
	Transverse velocity	77.92	55.62	40.73	404.51
Line 2	Stream-wise velocity	43.41	57.75	52.61	74.00
	Transverse velocity	196.35	-	451.67	1589.95

Table 2: Percentage errors in non dimensional velocity components

From the above table, it is seen that percentage error in present results is much less than Kulasekharan and Prasad's result, which establishes utilizing periodicity in all directions as a more accurate model of a large tube bank such as the one in present case.

5.2 Boundary Layer Separation Point

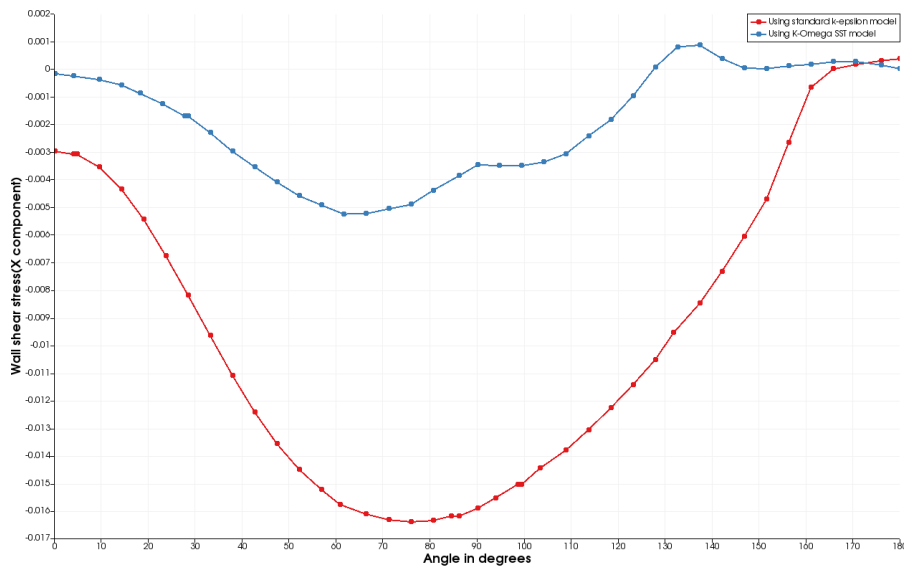


Figure 6: Variation of stream-wise component of shear stress at the walls of centre tube

Point of flow separation is calculated as the point on the wall of center tube at which the shear stress is zero. From figure 6 it is seen that flow separation angle is predicted to be around 170° for

$k - \epsilon$ model and 127° for $k - \omega$ SST model. The angle of separation is measured along the stream-wise direction from the front stagnation point.

Conclusions

We performed in-compressible, transient simulations over 2D staggered tube bank array using two RANS models: standard $k - \epsilon$ and $k - \omega$ SST. From the error values (Table 2) it is seen that 2D RANS models used in our case are not suitable for satisfactorily predicting the flow features.

Kulasekharan and Prasad[3] have also arrived at similar conclusion. They performed simulations using nine different RANS models: Spalart-Allmaras, standard $k - \epsilon$ with standard wall function, standard $k - \epsilon$ with enhanced wall treatment, RNG $k - \epsilon$, Realizable $k - \epsilon$, standard $k - \omega$, $k - \omega$ SST, Reynolds Stress Transport and v^2f models. It was found that none of these 2D RANS models is suitable for accurately predicting the flow features.

Therefore, a simulation using LES or DNS techniques may be attempted for accurately predicting the flow.

References

- [1] S. B. Beale. Tube banks, crossflow over. *Thermopedia™ Article*, 2011.
- [2] D. Butterworth. The development of a model for three-dimensional flow in tube bundles. *International Journal of Heat and Mass Transfer*, 21(2):253–256, 1978.
- [3] N. Kulasekharan and B. Prasad. Performance of 2-d turbulence rans models for prediction of flow past a staggered tube bank array. *Engineering Applications of Computational Fluid Mechanics*, 3(3):386–407, 2009.
- [4] S. Price, M. Pardoussis, and B. Mark. Flow visualization of the interstitial cross-flow through parallel triangular and rotated square arrays of cylinders. *Journal of Sound and Vibration*, 181(1):85–98, 1995.
- [5] O. Simonin and M. Barcouda. Measurements and prediction of turbulent flow entering a staggered tube bundle. In *Proceedings of Fourth International Symposium on Applications of laser anemometry to fluid mechanics (1988)*, 1988.