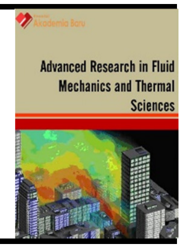
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Reducing Flow Separation in T-Junction Pipe Using Vortex Generator: CFD Study

Open Access

A Hamad[1], Syed Mohammed Aminuddin Aftab[2], Kamarul Arifin Ahmad[1,3,*]

[1] Department of Aerospace Engineering, Universiti Putra Malaysia, Selangor, 43400, Malaysia
[2] Aero Mechanical Department, Abu Dhabi Polytechnic, Al Ain, P.O. Box 66844, UAE
[3] Department of Mechanical Engineering, College of Engineering, King Saud University, P.O. Box 800, Riyadh 11421, Saudi Arabia

ARTICLE INFO

Article history:
Received 5 January 2018
Received in revised form 22 February 2018
Accepted 8 April 2018
Available online 15 April 2018

Keywords:
Computational Fluid Dynamics (CFD), Pipe flow, Vortex Generators (VG's), Water Distribution Networks (WDN)

ABSTRACT

Water Distribution Networks (WDN's) are used for providing water to residential areas. These networks have various junctions and interconnections spanning hundreds of kilometers. In the current work CFD simulation is carried out initially to determine the location of flow separation and recirculation inside a T-junction. Furthermore a parametric study is carried out to reduce this separation by implementing Vortex Generators (VG's). VG's are most commonly used in aircrafts to reduce the flow separation. A parametric analysis is carried out considering three VG configurations and later an upgrade is carried out. Simulations are carried out using ANSYS Fluent. The results show that, implementation of VG's effectively reduces the reverse flow and thus provides a solution to existing problem in Water Distribution Networks (WDN).

1. Introduction

Underground pipeline is used to distribute clean water from the water treatment plant to houses and buildings for domestic consumption. One of major problems associated with this distribution system arises, when the pipe branches into a network for supplying water to multiple areas. Few of the previous studies focused on solute mixing at the junction. Waanders *et al.*, [1] investigated the accuracy of network model and CFD simulation in predicting the mixing behaviour. The results show that network models are better for simulating the operation behaviour but lack the capability of predicting the contamination behaviour. Ho *et al.*, [2] used RANS turbulence model to simulate solute mixing in single joint and multi joint water transport network Figure 1. The simulation results matched with the experimental results. The results of cross joint indicated that smaller diameter pipes had prominent mixing compared to larger diameter pipes. The 3×3 network simulation showed difference in inlet flow rates and showed changes in downstream concentration.

* *Corresponding author.*
E-mail address: aekamarul@upm.edu.my (Kamarul Arifin Ahmad)

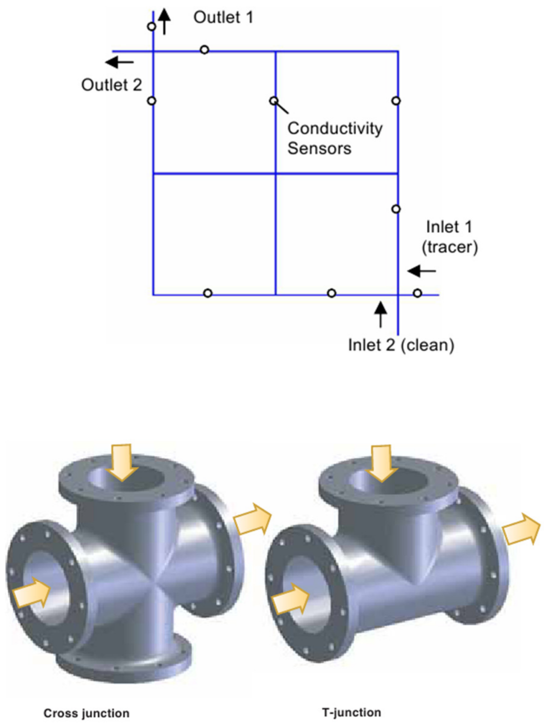


Fig. 1. 3×3 network and Cross and T- Junction [2]

Ho [3] compared solute mixing in cross joints, using CFD and other complete mixing models such as EPANET and bulk mixing model. It was concluded that complete mixing and bulk mixing models did provide an upper and lower bound. CFD models are more accurate, but are computationally expensive in studying large WDN. Song *et al.*, [4] conducted an experimental study and compared it with, EPANET complete and incomplete mixing models. It was found that incomplete mixing model provided better results, with an error of 15% compared to 66% error in case of complete mixing model. Thus, usage of CFD is only limited to gain insight into the fundamentals of the mixing in case of individual pipe junctions [3-5].

Gomez *et al.*, [6] also investigated the solute mixing in cross flow junction. Four different scenarios were tested (i) equal inflow and outflow (ii) equal outflow, varying inflow (iii) equal inflow and varying outflow (iv) varying inflow and varying outflow. The CFD study was carried out using k-ε turbulence model. The results indicate that incomplete mixing is due to bifurcating inlet flows, which reflect off with minimum contact. Ho *et al.*, [7] and Sun *et al.*, [8] further used double tee junction configurations to study the solute mixing. Shao *et al.*, [9] used simulations to study the mixing in double tee junction and later proposed an analytical model to predict the solute concentration at the outlet. The study helped in improving predicting capabilities of EPANET incorporating the Bulk Advection Model. Webb [10] utilised LES approach using Fluent to predict the mixing behaviour. Two scenarios were studied one with cross flow/double T and other with T separated by a distance of 2.5 diameters of the pipe. Later the T separation is increased to 10 D to visualise the effect of mixing. A double-sided T-Junction, T with 2.5D and 10 D separation is used to carry out the simulations. The Reynolds number is set to 40,000. Clean fluid enters from the north inlet and contaminated fluid enters from the west. Their results showed the transient nature of the mixing which was not observed by Ho *et al.*, [2] as they used RANS model to predict the mixing behaviour. Webb [11] further studied the effect of mixing at unequal flow rates in a cross junction, distribution network. It was found that mixing occurred near the wall due to large difference in momentum. A scaled experimental water distribution network model is designed by Summeren *et al.*, [12] to investigate the water quality and contamination in real world conditions and verify the numerical results.

The current work focuses on CFD simulations considering a T-junction in order to determine the initial flow separation. Later a solution to the separation problem at the junction is provided by incorporating Vortex Generators (VG's) [13-17], inside the pipe. The VG's helps by creating turbulence, their by eliminating backflow inside the pipe. Figure 2 shows a typical split i.e. T-junction with one inlet and two outlets, which is commonly used in water distribution networks.

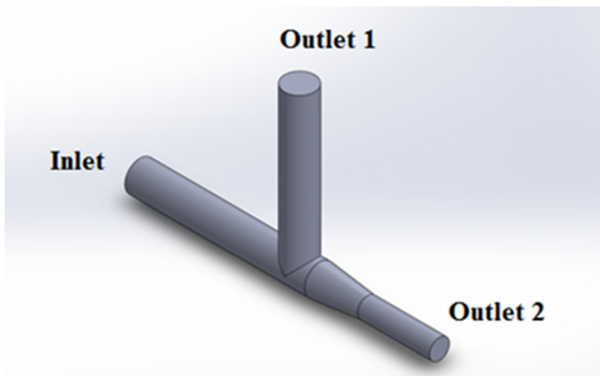


Fig. 2. T-junction

2. Turbulence Model Theory

2.1 *Reynolds Average Navier Stokes (RANS)*

In CFD, RANS is the most widely used turbulence modelling approach [19]. In this approach, the Navier Stokes equations are split into mean and fluctuating components. The total velocity U_i is a function of the mean velocity $\overline{U}_I$ and the fluctuating velocity Ui' as shown in the equation below

$$u_i = \bar{u} + u_i' \tag{1}$$

The continuity and momentum equations incorporating these instantaneous flow variables are given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial p}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_j\frac{\partial u_i}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_i}\left(-\rho\overline{u_i' u_j'}\right) = 0 \tag{3}$$

These above equations (in Cartesian tensor form) are known as RANS equations, and the additional Reynolds stress terms $-\rho\overline{u_i' u_j'}$ need to be modelled. The Boussinesq hypothesis is applied in relating the Reynolds stress and mean velocity

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij} \tag{4}$$

2.1 k-ε turbulence model

The following transport equations are used to determine the turbulent kinetic energy k and rate of dissipation ε.

$$\frac{\partial}{\partial}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right)\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{5}$$

$$\frac{\partial}{\partial}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right)\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\frac{\varepsilon^2}{k}+S_\varepsilon \tag{6}$$

where G_K is the turbulence kinetic energy generation term which is due to mean velocity gradients, G_b represents the turbulence kinetic energy due to buoyancy, Y_M represents effects of compressibility on turbulence and S_k and S_ε represent the source terms. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{\varepsilon\varepsilon}$ are constants, σ_k and σ_ε are the turbulent prandtl number for k and ε. The turbulent viscosity, μ_t is calculated as shown in the equation below

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{7}$$

The *k-ε* is one of the most widely used turbulence models in predicting the flow behaviour in pipes.

3. Case Setup

The geometry of the pipe is designed using Solid works 3D CAD software. The dimensions of the T-junction are 15m in length and 1.5m in diameter for the inlet and outlet 1. The outlet 2 diameter is 1m. There is a diffuser located before outlet 2 at a distance of 8m from the inlet of the pipe, as shown in Figure 2.

3.1 Numerical Methodology

The Numerical Methodology followed in the current study is as shown in the flow chart Figure 3. Initially the model is created using CAD software Solid Works. Later the solid model is imported into ANSYS for meshing. ANSYS meshing is utilized to mesh the model. A tetrahedral mesh is generated and the meshed model is imported to FLUENT for solving the flow. Later the results are analyzed using post processing in FLUENT.

3.2 Domain Details & Boundary Conditions

The boundary condition at the left surface is given as inlet and the right as outlet 2. The top surface of the T-junction is defined as outlet 1. The other surfaces of the pipe are defined as walls. Standard K-ε turbulence model is used for simulation and the convergence criterion is set to 10^{-6}.

SIMPLE pressure velocity coupling is implemented. Spatial discritization of the gradients is done using the Least Square Cell Based Gradient Evaluation (LSCB), as it is requires less computational time. The steady state simulation was carried out for an inlet flow velocity of 10 ms^{-1}.

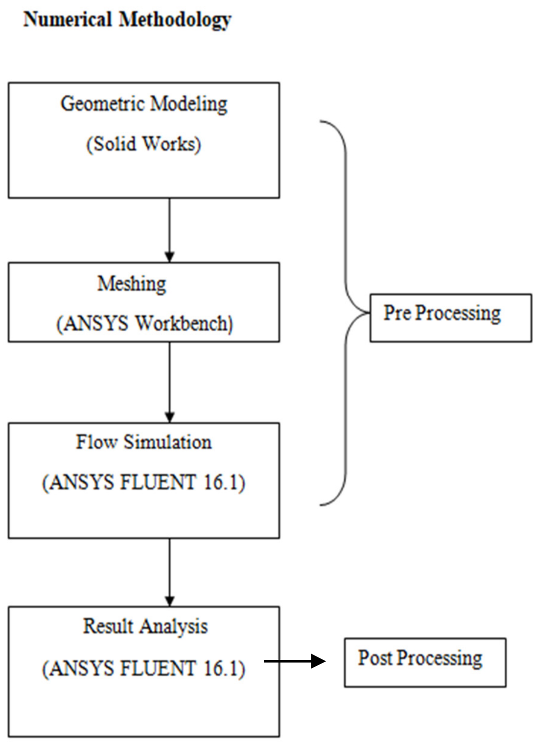


Fig. 3. Numerical Methodology flow chart

3.3 Grid Independency Check

Grid Independency check is carried out varying the no of elements and further accuracy of simulation is determined considering first and second order discritization of pressure and momentum. The Figure 4 (a) shows the overall mesh on the T-junction pipe. Three mesh cases were studied varying the element size from 0.06m, 0.03m and 0.015m which resulted in mesh size of 500k, 800k and 1.8million. The results indicated that element size of 0.015m provide good results.

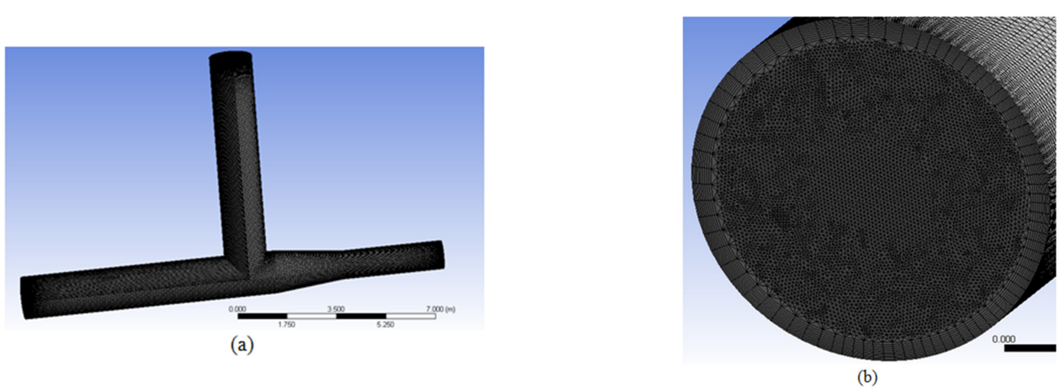
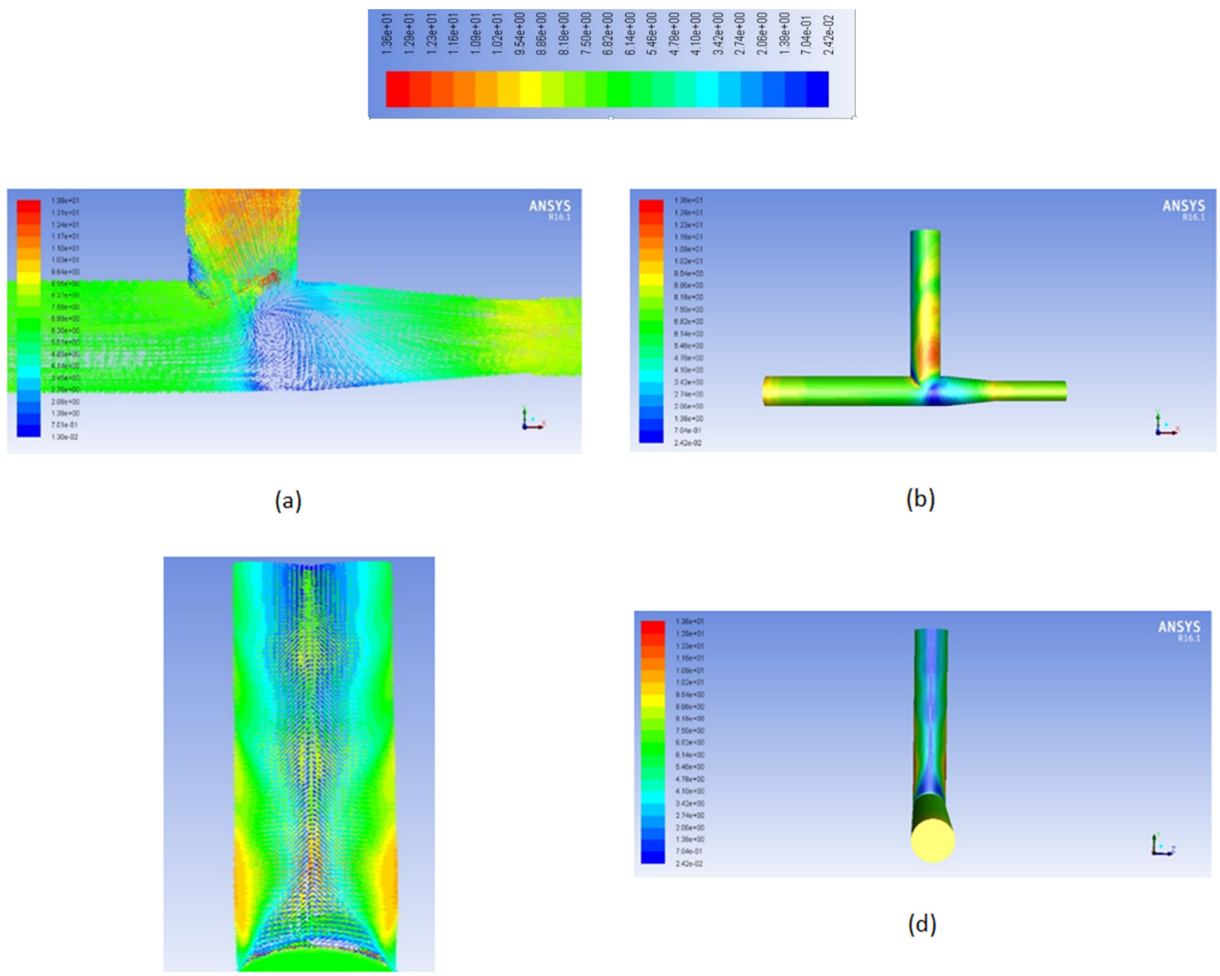


Fig. 4. Tetrahedral mesh on the T-Junction (a), Inflation layers at the inlet and two outlets (b)

Figure 4 (b) shows tetrahedral mesh elements along with 10 inflation layers, applied with growth ratio of 1.05 on the inlet and two outlets.

4. Results and Discussion

The Initial study without VG's inside the T-junction, showed that separation both in the horizontal as well as the vertical pipe Figure 5. A recirculation and separation zone is observed in the horizontal and vertical pipe of the T-junction. This baseline simulation provides the initial data, which is utilized in order to determine the VG placement.



(c)

Fig. 5. velocity vectors (a) and contours (b) representing reverse flow region in the horizontal pipe and velocity vectors (c) and contours (d) in the vertical pipe

The following aspects are essentially important while considering the placement of VG's., initial location of separation, followed by the number of VG's required and the shape of the VG.
For the current study rectangular VG's shape is used. The location is determined by the baseline study and the only parameter which has been varied is the no of VG's and its height. Aerodynamic Studies have shown that VG's reduce separation by inducing momentum in the fluid [13]-[18]. Thus,

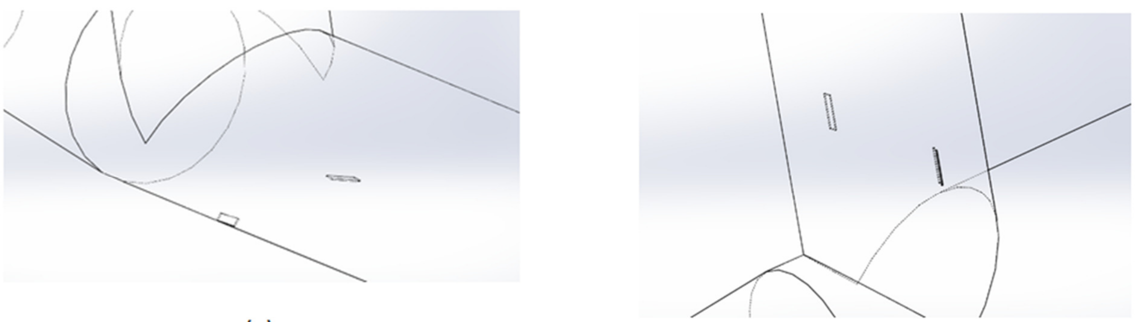
an attempt has been made to reduce the reverse flow in T-junction using VG's at the location of flow separation. The separation is initially noticed just before the diffuser. Parametric studies carried out varying the number of VG's needed and the height of the VG's.

The following VG case studies are carried out to determine the best VG configuration.
Case 1 Four VG setup, VG height 60mm.
Case 2 Six VG setup, VG height 60mm.
Case3 Eight VG setup, VG height 60mm and
Case 4 Eight VG setup with VG height 75mm.

The VG dimensions are fixed at (length) 300mm x (height) 60mm x (width) 5mm from case 1 to case 3. All four cases show very interesting results. The flow improvement is noticeable in all the cases. First three cases will determine the number of VG's required. The results of each case are further elaborated, highlighting the contour and vector plots.

4.1 Case 1

In the first case four rectangular VG's are used. Two VGs are placed inside the vertical pipe and another two inside the horizontal pipe before the diffuser as shown in Figure 6. The results showed an improvement in flow on the horizontal pipe compared to the vertical pipe. The change in the circulation at the horizontal pipe is better. Meanwhile, the vertical area seems to get worse. This may be due to insufficient number of VG's on the vertical pipe.



(a) (b)

Fig. 6. Position of VG's in hhorizontal (a) and vertical (b) pipes

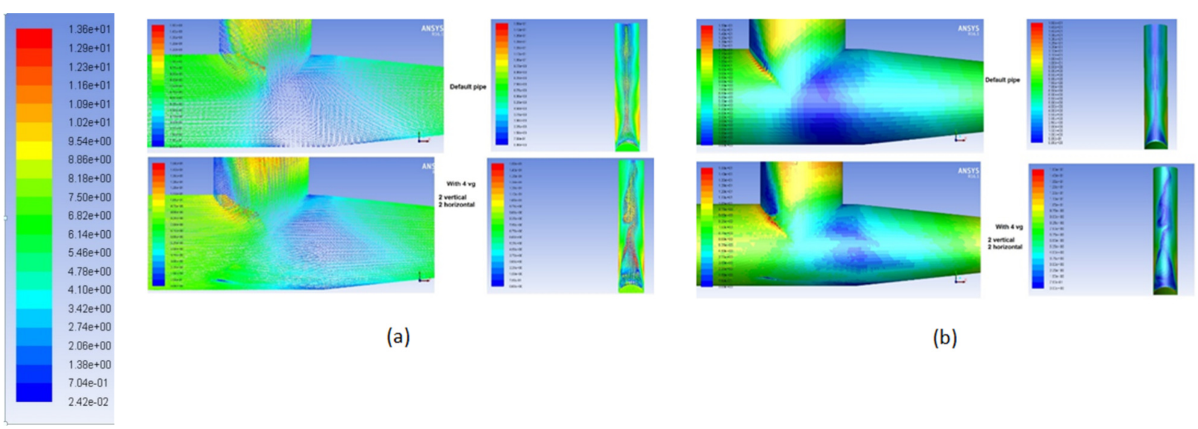


Fig. 7. Velocity vectors (a) and contours (b) for case 1

Figure 7 clearly shows a reduced turbulent zone in case of pipe with VG compared to without VG. The flow splits at the T-Junction but there is a reduced zone or recirculation due to the presence of VG. The horizontal pipe shows a reduction in separated flow compared to vertical pipe. The results clearly show that further improvement is needed for the vertical pipe.

4.2 Case 2

The second case is similar to the first one, with a slight improvement. In case 2 more VG's are added on the vertical pipe in order to improve the flow behaviour, compared to case 1. The VG's are oriented with an angle of 45 degrees to the flow direction Figure 8.

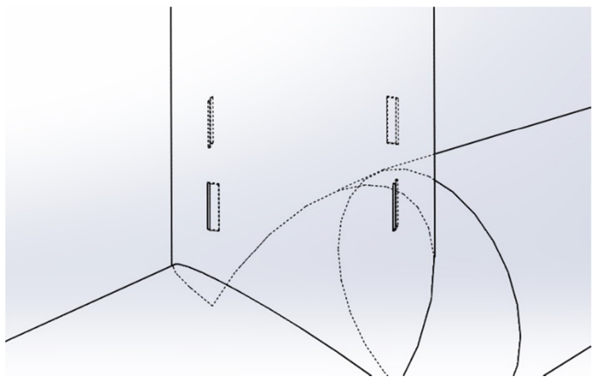


Fig. 8. Position of 4 VG's in vertical pipe

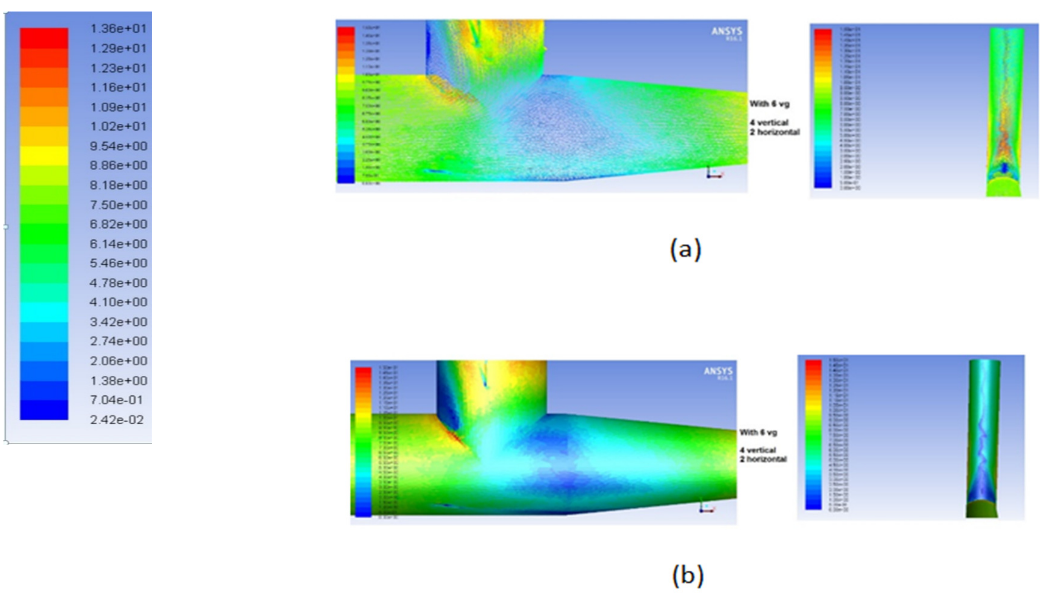


Fig. 9. Velocity vectors (a) and contours (b) for case 2

As mentioned earlier for Case 2 two more VG's are added for the vertical pipe. These additional VG's improve the flow behaviour. The VG's create a vortex which drastically reduces the separation zone as seen in Figure 9.

4.3 Case 3

In this case four VG's are placed on the vertical pipe and four on the horizontal pipe. The Figure 10 shows eight rectangular VG's inside the T-Junction.

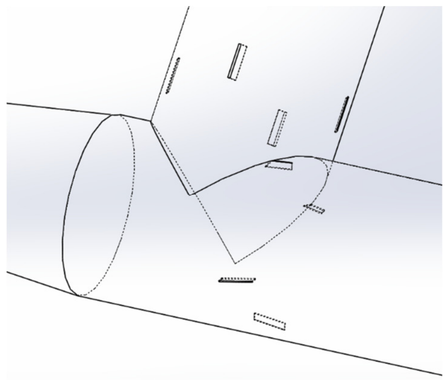


Fig. 10. Position of 8 VG's

The Figure 11 shows that this case has the best results. The VG's are effective in reducing the separation/backflow in both the horizontal and vertical pipes, thus effectively increasing the mixing at the junction.

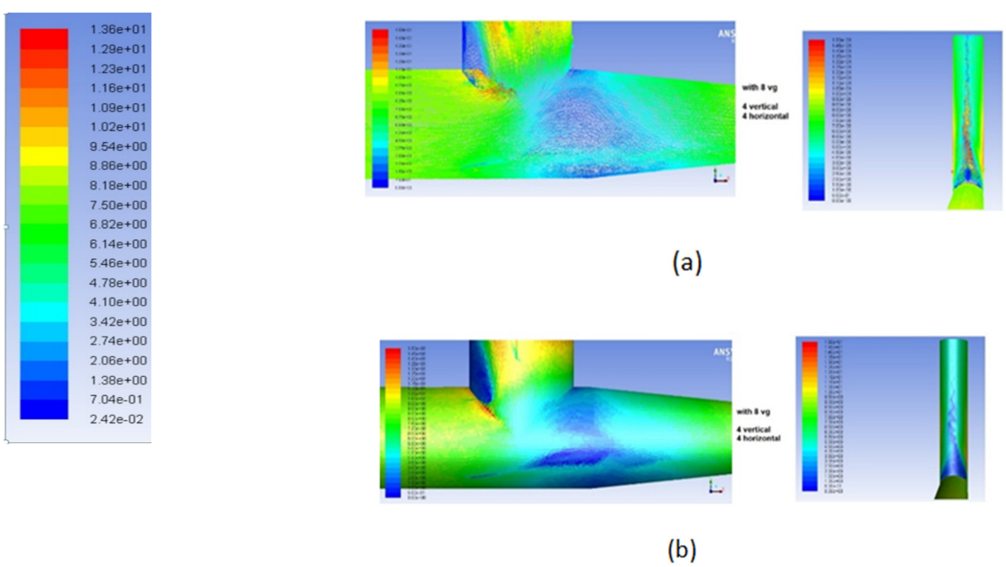
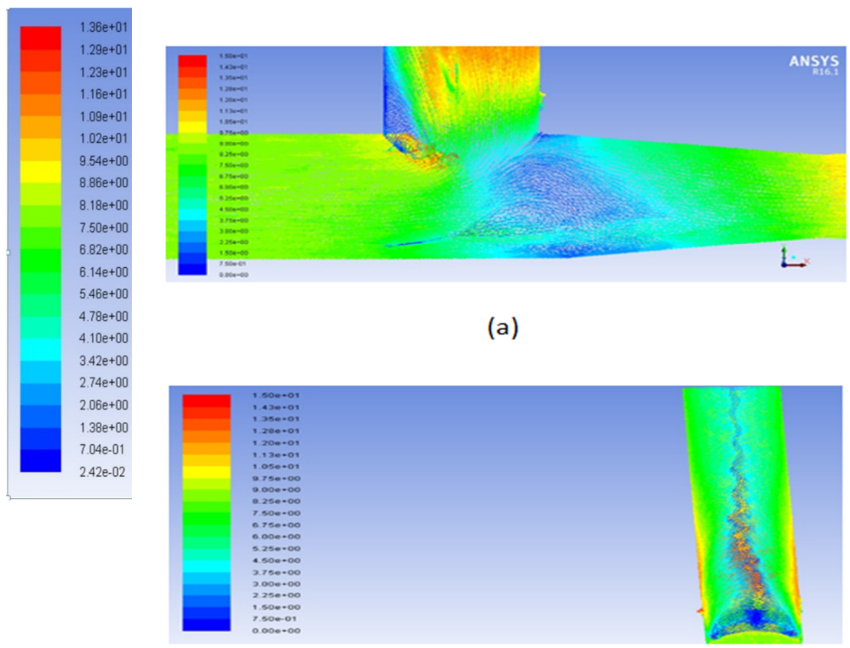


Fig. 11. Velocity vectors (a) and contours (b) for case 3

4.4 Case 4

Case 3 showed a drastic improvement compared to other cases. In case 4 a slight modification is carried out by increasing the VG height from 60mm to 75mm. Figure 12 shows the reduction in separation region for both the horizontal and vertical pipe. But the increase in VG height from 60mm to 75mm doesn't improve the mixing substantially.



(b)

Fig. 12. Velocity vectors horizontal (a) and vertical (b) pipe case 4

5. Conclusion

All the above cases show reduction in the backflow region. The best results are obtained for case 3, where in 4 VG's each are placed for the horizontal as well as the vertical pipe. The VG's increase the mixing and flow rate throughout the pipes. This qualitative CFD study shows that implementing VG can drastically improve the mixing in T-Junction in water distribution networks.

References

[1] van Bloemen Waanders, B., G. Hammond, J. Shadid, S. Collis, and R. Murray. "A comparison of Navier Stokes and network models to predict chemical transport in municipal water distribution systems." In *Impacts of Global Climate Change*, pp. 1-10. 2005.

[2] Ho, Clifford K., Leslie Orear, Jr, Jerome L. Wright, and Sean A. McKenna. "Contaminant mixing at pipe joints: Comparison between laboratory flow experiments and computational fluid dynamics models." In *Water Distribution Systems Analysis Symposium 2006*, pp. 1-18. 2008.

[3] Ho, Clifford K. "Solute mixing models for water-distribution pipe networks." *Journal of Hydraulic Engineering* 134, no. 9 (2008): 1236-1244.

[4] Song, Inhong, Pedro Romero-Gomez, and Christopher Y. Choi. "Experimental verification of incomplete solute mixing in a pressurized pipe network with multiple cross junctions." *Journal of Hydraulic Engineering* 135, no. 11 (2009): 1005-1011.

[5] Wols, Bas Anton, CFD in drinking water treatment. PhD Thesis, Delft University of Technology, 2010.

[6] Romero-Gomez, P., C. K. Ho, and C. Y. Choi. "Mixing at cross junctions in water distribution systems. I: Numerical study." *Journal of Water Resources Planning and Management* 134, no. 3 (2008): 285-294.

[7] Ho, Clifford K., and Leslie O'Rear Jr. "Evaluation of solute mixing in water distribution pipe junctions." *American Water Works Association. Journal* 101, no. 9 (2009): 116.

[8] Sun, Amy, Sean McKenna, Clifford K. Ho, Malynda Cappelle, Stephen W. Webb, Tim O'Hern, and K. Kajder. "Joint Physical and Numerical Modeling of Water Distribution Networks." (2009).

[9] Shao, Yu, Y. Jeffrey Yang, Lijie Jiang, Tingchao Yu, and Cheng Shen. "Experimental testing and modeling analysis of solute mixing at water distribution pipe junctions." *Water research* 56 (2014): 133-147.

[10] Webb, Stephen W., and Bart G. van Bloemen Waanders. "High fidelity computational fluid dynamics for mixing in water distribution systems." In *Water Distribution Systems Analysis Symposium 2006*, pp. 1-15. 2008.

[11] Webb, Stephen W. "High-fidelity simulation of the influence of local geometry on mixing in crosses in water distribution systems." In *World Environmental and Water Resources Congress 2007: Restoring Our Natural Habitat*, pp. 1-14. 2007.

[12] van Summeren, Joost, Sidney Meijering, Hendrik Beverloo, and Peter van Thienen. "Design of a Distribution Network Scale Model for Monitoring Drinking Water Quality." *Journal of Water Resources Planning and Management* 143, no. 9 (2017): 04017051.

[13] Aftab, Syed Mohammed Aminuddin, and P. Srinivasa Murthy. "Comparative Study of Vortex Generator Orientation on Wing Surface Considering Delta Vortex Generators." In *Applied Mechanics and Materials*, vol. 225, pp. 79-84. Trans Tech Publications, 2012.

[14] Zhen, Tan Kar, Muhammed Zubair, and Kamarul Arifin Ahmad. "Experimental and numerical investigation of the effects of passive vortex generators on Aludra UAV performance." *Chinese Journal of Aeronautics* 24, no. 5 (2011): 577-583.

[15] Serakawi, A. R., and K. A. Ahmad. "Experimental Study of Half-Delta Wing Vortex Generator for Flow Separation Control." *Journal of Aircraft* 49, no. 1 (2012): 76-81.

[16] Ahmad, K. A., M. Z. Abdullah, and J. K. Watterson. "CFD Simulations of Oscillating Sub-Boundary Layer Vortex Generators for Diffuser Flow Separation Control." *International Journal of Engineering and Technology* 5, no. 1 (2008): 25-35.

[17] Halim, Mohd Faizal, and K. A. Ahmad. "CFD simulation of passive vortex generator on separated flow diffuser." PhD diss., Universiti Putra Malaysia, 2013.

[18] Fouatih, Omar Madani, Marc Medale, Omar Imine, and Bachir Imine. "Design optimization of the aerodynamic passive flow control on NACA 4415 airfoil using vortex generators." *European Journal of Mechanics-B/Fluids* 56 (2016): 82-96.

[19] T. W. Yen, Y. Asako, N. A. C. Sidik and G. R. Zher, "Governing Equations in Computational Fluid Dynamics: Derivations and A Recent Review," *in Energy and Environment* 1 (2017): 1-19.