MIXING IN PIPELINES WITH SIDE-TEES

Zahid Khokhar[1], Habib D. Zughbi[2], and Rajendra Sharma[3]

1: Graduate student, Department of Chemical Engineering, KFUPM, Dhahran, 31261
2. Assistant Professor, Department of Chemical Engineering, KFUPM
3. Professor, Department of Chemical Engineering, KFUPM

E-mail: hdzughbi@kfupm.edu.sa

ABSTRACT

Numerical and experimental investigations of mixing in pipelines with side-tees are carried out to determine the quality of mixing in such pipelines. Temperature is measured experimentally to quantify the degree of mixedness. Numerically, the temperature field is calculated and then compared with experimental results to validate the models.

The computational fluid dynamics (CFD) package FLUENT is used to solve the governing equations, namely the equations of continuity, motion and energy. Numerical results showed good agreement with experimental results. The mesh size is selected do that the numerical solution is independent of mesh size. Turbulence is modelled using the standard k-ε model and the more involved Reynolds stress model. The pipe length required for achieving 95% mixing is found to be a function of the ratio of the velocities of the side and main streams.

Keywords: *Mixing, pipeline, side-tee, opposed-tee, numerical simulation*

الملخص

تم إجراء بحوث عددية ومعملية للخلط في أنابيب مزودة بأنابيب جانبية وذلك لتقييم نوعية الخلط في هذه الأنابيب وطول الانبوب اللازم لتحقيق ٩٥% من الخلط. وقيست الحرارة معملياً لتحديد درجة الخلط، وتم حلّ حقل الحرارة عددياً ثم قورنت قيم الحرارة العددية بالقيم المعملية وذلك للتأكد من فعالية ودقة النموذج العددي. استعمل برنامج متعدد الأهداف لديناميكا الموائع العددية FLUENT لحل المعادلات الرياضية التي تتحكم بانتقال المادة والطاقة وسريان الموائع.

وفي البحث الحالي تم استعمال السائل ذاته في الانبوب الرئيسي والانبوب الجانبي. وقد أظهرت النتائج العددية تطابقاً جيداً مع النتائج المعملية ووجد أن طول الأنبوب اللازم لتحقيق ٩٥% من الخلاط يعتمد على نسبة سرعة السائل في الانبوب الجانبي والانبوب الرئيسي

1. INTRODUCTION

Mixing problems, such as the design and scale-up of a mixer and quantification of mixing, have been traditionally tackled by developing empirical design equations mainly due to the complexity of the fluid dynamics of mixing. Although this approach has proven to be satisfactory for many applications, it is rather limited because it neglects the complexity of flow in most mixing applications.

Applications where pipeline mixing with tees is used include low viscosity mixing such as wastewater treatment and blending of some oils (injection of additives) and petrochemical products. Other applications include blending of fuel gas, and mixing of feed streams for catalytic reactors. A tee is formed by two pipe sections joined at a right angle to each other. One stream passes straight through the tee while the other enters perpendicularly at one side as shown in Figure 1. This flow arrangement is known as the side-tee. However other flow arrangements may be used, such as having the two opposing streams entering co-axially and leave through a pipe, which is perpendicular to the entering direction. This is known as the opposed-tee. A review of various flow arrangements is presented by Gray (1986). A survey of the literature shows that simulation using CFD of pipeline mixing with tees have been carried out by Cozewith et al. (1991) and Forney and Monclova (1994).

The main interest in this paper concentrates around the side-tee shown in Figure 1. For all designs of pipe tees, mixing takes place in shorter distances compared with distances required for mixing in a pipe with undisturbed turbulent flow. Reviews of pipeline mixing with tees has been presented by (Simpson, 1974), (Gray, 1986) and (Forney, 1986).

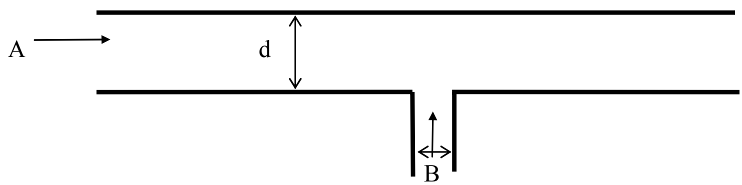

Figure 1: A schematic diagram of a pipeline side tee

The flows generated by a tee mixer have been studied by (Moussa et al., 1977) and (Crabb et al., 1981). Cozewith et al.(1991) simulated tee mixing characteristics with and without a reaction for a tee with d/D = 0.188 over the range of side stream/main stream velocity ratios from 1.2 to 6.5. A three-dimensional model was constructed and the k-ε model was used to model turbulence. Literature recommends and uses the k-ε model especially for non-circulating flows. Cozewith et al. (1991) compared their numerical results with the experimental results of Cozewith and Busko (1989) and got reasonable agreement for concentration trajectory for x/D > 0.7. Concentration trajectory is defined as the locus of

Top

maximum concentration. Other comparisons also showed qualitative agreement between experimental and numerical results.

Forney and Monclova (1994) simulated pipeline side-tee mixing quality with the commercially available fluid flow package PHOENICS. The $k-\varepsilon$ model was used to model turbulence. They compared numerical results with the experimental results of Sroka and Forney (1989) and obtained reasonable agreement.

Both of the above numerical models solved the conservation equations for mass and momentum in primitive variables for steady turbulent flow of a single-phase fluid with an inert tracer introduced at the injection point. Both models also used a mixing criteria based on the standard deviation of the component mixed and the mean value of the tracer over the pipe cross sectional area $\overline{C}$. The use of CFD, despite the two above-mentioned papers, has still a lot to offer in analyzing and understanding mixing at pipeline tees. Simulation of variations of tees mixers and opposed flow tee have not been reported in literature.

2. MODEL EQUATIONS

The differential equations representing mass, momentum and energy conservation can be written in the general form:

$$\frac{\delta\left(R_i \rho_i \phi_i\right)}{\delta t} + div\left(R_i \rho_i U_i \phi_i - R_i \Gamma_{\phi_i} grad\phi_i\right) = R_i S\phi_i$$

Transient Convection Diffusion Source

Where R_i is the volume fraction of phase *i*, ϕ_i, is any conserved property of phase *i*, U_i, is velocity vector of phase *i*, Γ_{ϕ_i} is the exchange coefficient of ϕ in phase *i*, S_{ϕ_i}, is the source rate of ϕ_i. Thus, the continuity equation for phase *i* becomes:

$$div\left(R_i \rho_i U_i - R_i D_i grad R_i\right) = m_i$$

where D_i, is the diffusivity of phase *i*, m_i, is mass per unit volume entering phase *i*, and ρ_i is the density of phase *i*. The conservation of momentum for variable ϕ_i becomes:

$$div\left(R_i \rho_i U_i \phi_i - R_i \mu_{eff} grad \phi_i\right) = R_i S_{\phi_i}$$

where μ_{eff} is the effective viscosity and S_{ϕ_i} is the source of ϕ_i per unit volume

Top

3. THE EXPERIMENTAL APPARATUS

An experimental apparatus was built to quantify mixing in a pipeline with a side-tee. Temperature is used as the measured variable. Hot water is injected from the side tee and is mixed with the same liquid flowing in the main pipe at a lower temperature. A one-inch main pipe is shown in Figure 2. This Figure shows a schematic diagram of the experimental apparatus. A ¼ inch side-tee is also shown. Eight thermocouples are inserted at various positions of the main pipe in order to measure the temperature of the flow. These thermocouples are connected via an OMEGA data-logging card to a PC as shown. Flow through the side passes through a heater that can raise the temperature of the side stream significantly above that of the main stream.

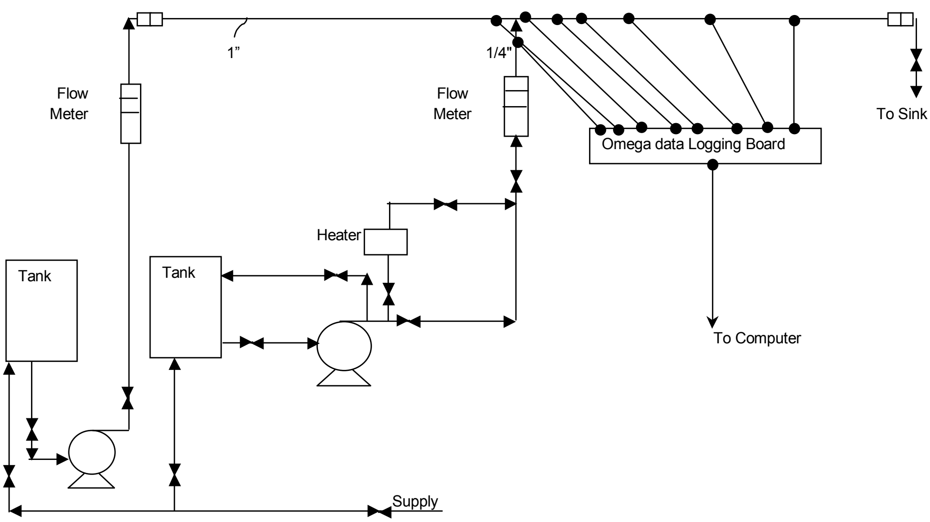

Figure 2: A schematic diagram of the experimental rig used to investigate mixing a pipeline with a side-tee

4. THE NUMERICAL MODEL

Flow in pipeline is simulated by solving the mass and momentum conservation equations. The degree of mixedness is investigated by solving for the energy equation and by monitoring the temperature at various positions along the flow. The general-purpose three-dimensional computational fluid dynamics package FLUENT is used to solve the governing equations. This allows the investigation of a range of conditions and geometries quite efficiently once a general model has been established and validated against experimental results.

Top

A basic three-dimensional numerical model representing a 15 in section of a main pipe with a side-tee located at 2 in from the front end of the pipe has been constructed. The grid is shown in Figure 3. An unstructured tetrahedral grid was chosen. To test the dependence of the numerical solution on the grid size and to also test different models to simulate turbulence, one case with Q_j of 7 lit/min and Q_m of 9 lit/min has been chosen.

In this study, the pipe length required to achieve 95% mixing is numerically and experimentally determined. This is the length from the jet inlet to the location along the pipe centreline where the value of the measured quantity anywhere in the pipe is less than 5% of the step input. The step input is defined as the difference between the initial value and the final mean value. In this study, the 95% mixing is defined and used to quantify mixing. Distinction should be made between mixing length and blending length. Blending length is when the flow through the side pipe (hot fluid) is started exactly at the same time as the flow through the main pipe. Other papers may refer to mixing length, which is defined in a similar way except the flow through the side-tee is started not when the flow in the main pipe starts but after the flow reaches steady state.

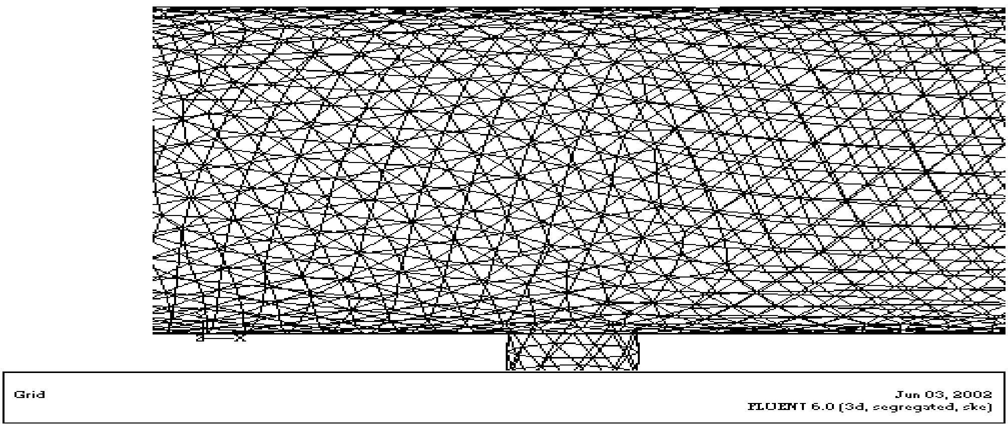

Figure 3: The computational grid of a piece of main pipeline with a side tee used in these simulations

In terms of a concentration tracer, *m* can be defined as:

$$m = \left| \frac{c - \bar{c}}{\bar{c}} \right| \langle \ 0.05$$

Where $\bar{c}$ is the equilibrium concentration and *c* is the concentration at any monitoring point at any time. When the above condition is met at all monitoring points in a cross sectional plane of the main pipe, it can then be said that concentration at any point of the pipe after that length has reached 95% or more of the equilibrium concentration. For this case the initial value of *m* before the addition of the tracer is considered to be 0.

Top

In the present study, the flow in the main pipe before the jet inlet is set initially at a certain temperature. The flow through the side-tee is set at a higher but know temperature. Thus the equilibrium temperature, $\overline{T}$, can be calculated. The 95% blending is reached when the temperature anywhere across a plane inside the pipe is within the range of $((\overline{T} \pm (\overline{T} - T_{im}) * 0.05)$ where T_{im} is the initial temperature of the fluid in the main pipe, i.e. before the inlet of the side tee. The length required for the hot fluid to blend is then measured according to this criterion that means that the maximum temperature difference between any two points across a cross sectional area of the pipe should not exceed a certain value which is a function of the initial temperatures and the flow rates of the fluids in the main and side pipes.

5. RESULTS

5.1. Numerical Results and Validation of Numerical Model

Numerical and experimental results are presented in this paper. Figures 4 and 5 show the velocity and temperature contours in a plane along the pipe axis for a velocity ratio (U_j/U_m) of 17.1.

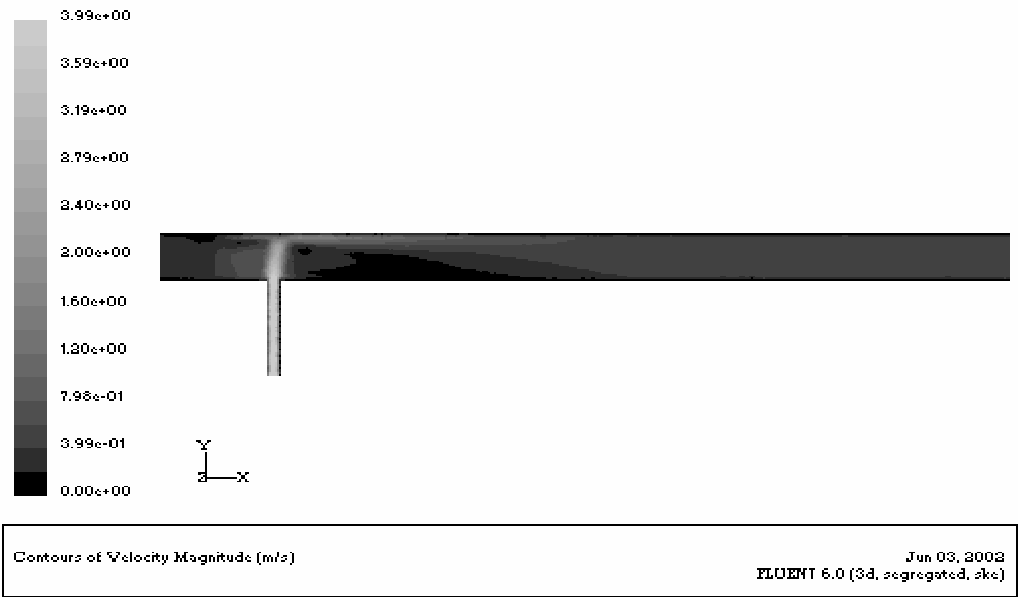

Figure 4: Velocity contours in a plane passing through the centreline for a mesh size of 2

Top

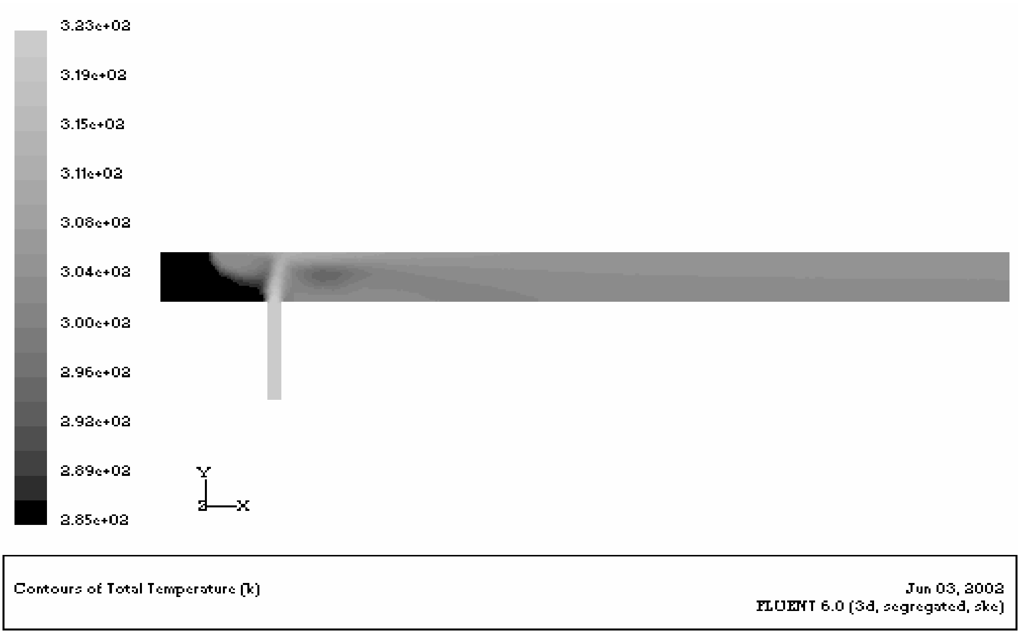

Figure 5: Temperature contours in a plane passing through the centreline for a mesh size of 2

A mesh size of 2 is used and turbulence is modelled using Reynolds Stress Model (RSM) or the *k*-*ε*. Model. Figure 4 shows clearly that the jet impinges on the opposite wall of the pipe. Figure 5 shows that the distance for 95% mixing to be achieved is about 9 inches. In order to analyze the results quantitatively, values of temperature versus location along the pipe axis are plotted. In order to validate the numerical model, these numerical values are compared with experimental values measured at exactly the same locations.

Figure 6 shows a plot of experimental and numerical values of temperature versus location along the main pipe axis. Good overall agreement is observed between numerical and experimental results especially regarding the distance required to achieve 95% mixing. Some differences are observed in the vicinity of the jet coming through the side-tee. The final (equilibrium) temperatures and the distance required for 95% mixing are almost identical. The difference in the value of temperature in the vicinity of the jet can be explained by the high sensitivity of temperature to the location of the thermocouple. A difference of a couple of mm could result in a significant difference in temperature.

The side jet impinges on the opposite wall of the pipe and this creates a region of backflow. This region could be significant and it could explain some problems faced by some process industries. These problems are corrosion related and could be due to this zone of low velocity.

Top

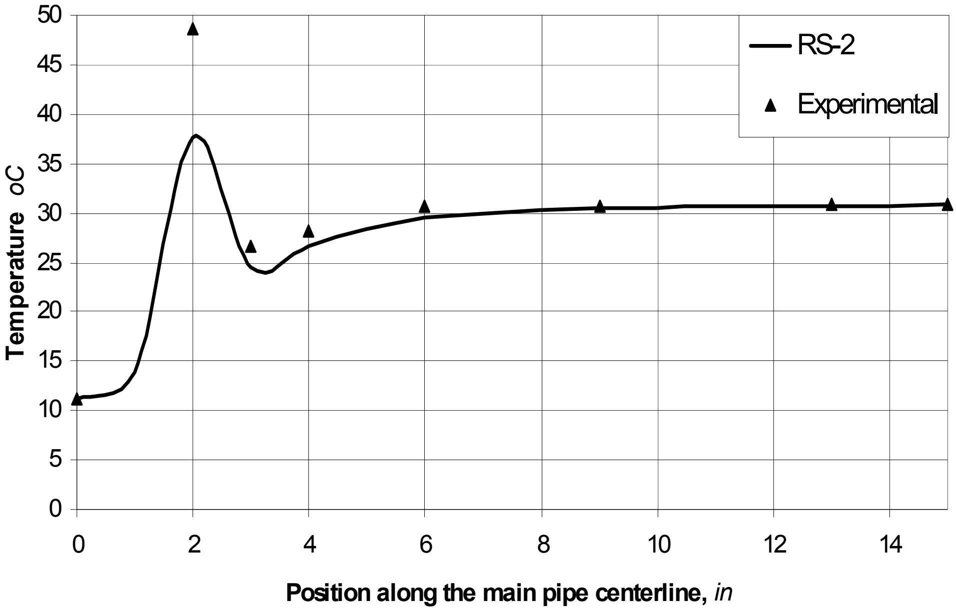

Figure 6: Simulated and experimental temp vs. position along a centreline for the same case of mesh size 2

5.2. Dependence of Solution on Grid Size

In order to quantitatively compare results with different mesh sizes, a plot of temperature versus position along a centreline of the main pipe is shown in Figure 7. Mesh sizes of 4, 3 and 2 have been tested. The number of cells used in each case is shown in Table 1.

Table 1: Number of cells for mesh sizes of 2, 3 and 4

Mesh Size	2	3	4
Number of cells	162367	56463	18610

Figure 7 shows a comparison of the temperature versus location along a centreline for mesh sizes of 2 and 3. It is clear that the solution changes with a mesh size, although the difference between solutions of mesh size of 3 and 2 is not very significant. The number of cells for a mesh size of 2 is relatively very high, however, since the solution still shows some change a mesh size of 1 was attempted. This attempt could not be completed, because the time required to perform the meshing of the computational domain is prohibitively excessive. A mesh size of 2 was used for all the main runs in this study.

Top

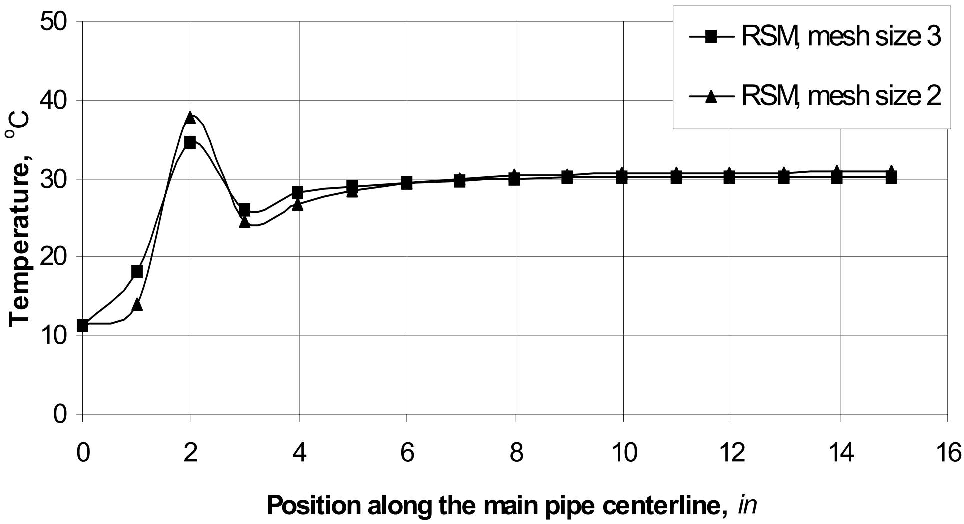

Figure 7: Comparison of temperature versus location along a centreline for mesh sizes of 2 and 3.

5.3. Dependence of Solution on Turbulence Model

Plots of the temperature versus location along the centreline of the main pipe obtained using the *k-ε* model and the Reynold Stress Model (RSM) are shown in Figure 8. It is noted that the mixing length required to produce 95% mixedness is exactly the same for both cases. However, differences are observed in the vicinity of the jet where high turbulence intensity is observed.

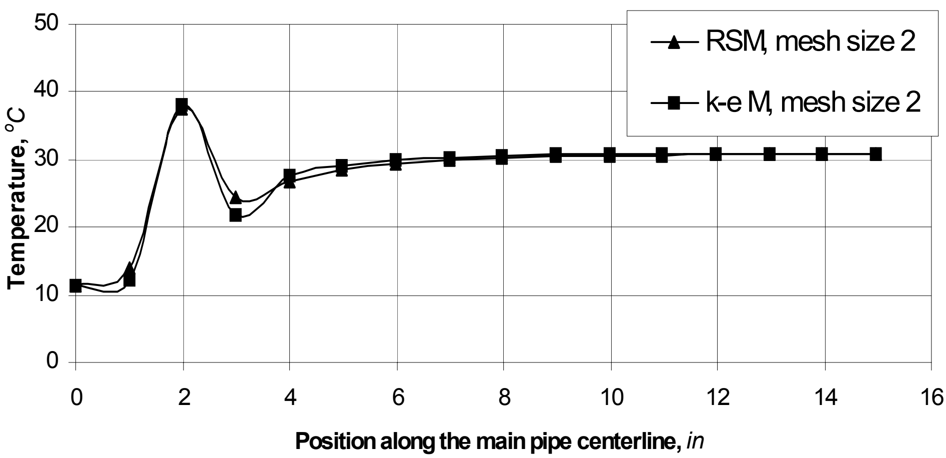

Figure 8: Plots of temperature versus location along the centre line of the main pipe for cases with *k-ε* and RSM

The time required for the above case to converge was 8 hours when the RSM model is used compared to 3 hours only needed for convergence when the k-ε model is used. Since the jet impingement area is of interest in this work, the RSM model is used despite it being computationally more expensive.

5.4. Experimental Results

Experimental runs have been carried out with water flow rates through the main pipe and the side tee as 7.02, 12.28 and 19.30 liters/min and water flow rate through the side pipe of 7.5, 5.0 and 3.0 liters/min. Details of these runs together with the corresponding values of the velocity in the side stream U_j, velocity in the main pipe, U_m, ratio of U_j/U_m, values of Reynolds number in the side pipe and main pipe before and after the tee are given in Table 2.

Figure 9 shows plots of the temperature measured by the thermocouples versus the location along the centreline of the main pipeline for the three cases with U_m of 7.02 lit/min. From the Figure, the pipe lengths required for 95% mixing of hot and cold fluids can be deduced. These values and other values for the remaining six cases are summarized in Table 3.

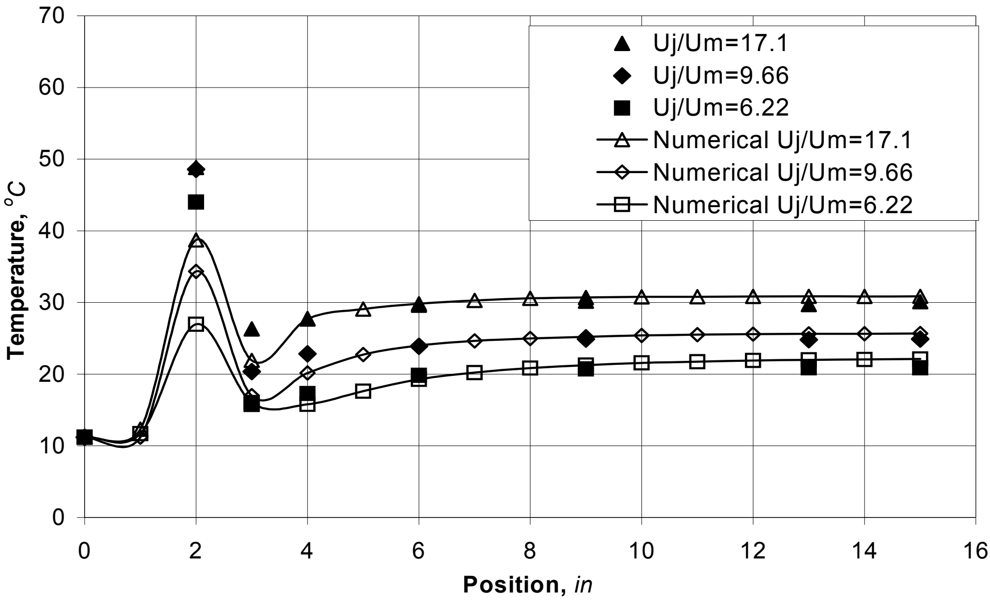

Figure 9: Experimental and simulation, using k-ε model, values of temperature versus location along the centreline of the main pipe for cases with Q_m of 7.02, 12.28 and 19.3 lit/min and Q_j of 7.5, lit/min

Top

Table 2: Values of variables for certain experimental runs

Q_j lit/min	Q_m lit/min	V_j m/s	V_m m/s	U_j/U	Re_j	Re_m	R_m (after the tee)	Distance required for 95% mixing (diameters)
7.5	7.02	3.9471	0.23082	17.10	250637.7	16122.9	22390.3	9
7.5	12.28	3.9471	0.40394	9.77	250637.7	10260.0	16475.3	11
7.5	19.30	3.9471	0.63476	6.22	250637.7	5862.9	12130.9	13

Table 3: Comparison of experimental and numerical results

Q_j lit/min	Q_m lit/min	U_j/U_m	Distance (experimental) required for 95% mixing (diameters)	Distance (numerical) required for 95% mixing (diameters)
7.5	7.02	6.22	13	13
7.5	12.28	9.77	11	11
7.5	19.30	17.10	9	9

6. COCLUSIONS

Mixing in pipelines with side-tees has been experiemntally and numerically investigated. Temperature is measured and used to quantify mixing. Good agreement between experimental and numerical results is observed especially when the final temperatures and the distance required to achieve 95% mixing are considered. Some differences are iobserved in the values of teperature in the vicinity of the jet incoming through the side-tee. This could be due to the high sensisitivity of such value to the position of the the thermocouple. A small difference in position results in a significant difference in the value of temperature. The mesh size was chosen such that the solution is made independent of the mesh size. The Reynolds Stress Model (RSM) and the k-ε model were used to account for turbulence and gave similar results except in the vicinity of the jet impingement region. Results showed that the pipe length required to achieve 95% mixing depends on the ration of Uj/Um.

Top

REFERENCES

1. Cozewith, C. and Busko, M., 1989, "Design Co-relations for Mixing Tees", *Industrial and Engineering Chemistry Research*, 28, pp. 1521-1530.
2. Cozewith, C., Ver Strate, G., Dalton, T. J., Frederick, J. W. and Ponzi, P.R., 1991, " Computer Simulation of Tee Mixers for Non-reactive and reactive Flows", *Industrial and Engineering Chemistry Research,* 30, pp. 270-275.
3. Crabb, D., Durao, D., and Whitelaw, J.H., 1981, "A Round Jet Normal to a Cross Flow", *Journal of Fluids Engineering*, 103, pp. 143-151.
4. Forney, L.J., 1986, Encyclopaedia of Fluid Mechanics, Vol. II, Ch. 25, N. P. Cheremisinoff, ed., Gulf Publishing Co., Houston USA.
5. Forney, L.J. and Monclova, L.A., 1994, "Numerical Simulation of Pipeline Tee Mixers: Comparison with Data", Industrial Mixing Technology: Chemical and Biological Applications, Gaden, E.L., Tatterson, G.B.Calabrese, R.V. and Penney, W.R., eds., pp. 141-143.
6. Gray, J. B., 1986, Mixing: Theory and Practice, Ch. 13, Vol. III, J.B. Gray and V. W. Uhl, eds., Academic Press, USA.
7. Moussa, Z., Trischka, J. and Eskinazi, S., 1977, "The Near Field in the Mixing of a round Jet with a Cross Stream", J*ournal of Fluid Mechanics*, 80 pp. 49-58.
8. Ottino, J. M., 1990, "The Kinematics of Mixing: Stretching, Chaos and Transport", Cambridge University Press, Cambridge, UK.
9. Simpson, L. L., *Chem. Eng. Prog.,* 70, 77 (1974)
10. Sroka, L. M. and Forney, L. J., 1989, "Fluid Mixing with a Pipeline Tee: Theory and experiment". Journal of the American Institute of Chemical Engineers, 35(3) pp. 212-218.
11. Sroka, L. M. and Forney, L. J., 1989, "Fluid Mixing in a 90° Pipeline Elbow", *Industrial and Engineering Chemistry Research,* 28(6), pp. 850-856.
12. Yao, W., Takahashi, K. and Koyama, K., 1998, " Theoretical Tool for Optimum Design of Mixer, and Visualisation and Quantification of Mixing Performance", *Journal of Chemical Engineering of Japan*, 31(22), pp. 220-227.