ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

Conjuring Misdiscrepancies of k-ε and Reynolds Stress Turbulence model in Cross Jet Vicinity Collaborating Side Injection Experiments and CFD Simulations

Zahid H Khokhar*+; Mumdouh Al-Harthi[1]; Habib Al-Ali[1]; Abdurraheem Abulazeez[2]; Habib Zughbi[3], BS Yilbas[4]

[1]Department of Chemical Engineering, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia, [2]Research Institute, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia, toazeez@gmail.com, [3]Petroleum, UNSW, Sydney, Australia [4]Mechanical Engineering, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia, *zahidhkhokhar@yahoo.com, +Corresponding email: zahedhk@gmail.com

Abstract— In This piece of work, simulations following experiments are done of side jet into pipeline flow. Standard k-ε model is used to establish computational fluid dynamics model to study the process. Good agreement is found at downward flow between k-ε model predicted values with experimental values. In vicinity of jet into pipeline discrepancies are found. To conjure these misdiscrepancies of k-□ , Reynolds stress model is runto the same model to explore these discrepancies of k-ε model to predict the values of variables at jet vicinity. It is found that it is passed to resolve jet injection vicinity. It is better to use both models one by one where turbulence is more and any type of expanse (eddies, swirl and back mixing with jet impingement etc. to mention) is involved to prevent any type of loss in the process by mystification.

***Index Terms*— Turbulence model, Jet, Injection, CFD, Simulations, Combustion.**

I. INTRODUCTION

Combustion processes, chemical reactions, heat transfer operations and mixing are stimulated with turbulence. Effective use of turbulence creates small contagious expanse which controls time for molecular mixing and reaction significantly reducing the cost of producing many chemicals and preventing any type of loss in process industries. Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. In jet vicinity accurate turbulence modeling benefits plant design and modification to save overall cost and to get better process product.

Computational Fluid Dynamics has been adopted by a whole range of industries, including chemical, petrochemical, oil, automotive, built- environment, food processing and many others enabling the process engineer to begin to understand in greater detail the internal operation of individual units by relating an analysis of the flow field and other transfer processes with observed phenomena and thereby identifying the cause of the problem and evaluate solutions. Moreover, it has steadily spread from research groups to the design and development departments. Reference [1] reviewed the developments in industrial computational fluid dynamics. In present study, the concern is to investigate turbulent k-□ model and Reynolds stress model working with establishing a computational system of analysis based on physical experiments done at laboratory pilot scale using a side-injection pipeline rig. To use both models one by one it is desired to establish and define parts of fluid flow based on geometry, flow and its turbulence quality. Wherever the model is adequate, it may not be suitable for some other case and whenever it is not suitable for a case, it may give promising results near or far from that investigating position under considered. Therefore, it is very essential to study detail of models for broad ranges.

A pipe with side injection forming a tee like shape is a simple device formed by two pipe sections joined traditionally at a right angle to each other. One stream passes straight through the side injection pipe while the other enters perpendicularly at one side. However, other flow arrangements may be used, such as having the two opposing streams entering co-axially and leaving through a pipe, which is perpendicular to the entering direction.

A review of various flow arrangements is presented by [2] stating that for all designs of pipetees, mixing takes place in shorter distances compared with distances required for mixing in a pipe with undisturbed turbulent flow. However, turbulence may introduce other pipe material related problems as well as fluid back mixing and swirling. A review of pipeline mixing with pipetees has been presented by [3] and [4]. It is clear from surveying the literature that numerical simulation of mixing in pipeline with tees has still a lot to offer towards better understanding of pipeline mixing. No single turbulence model is universally accepted as being superior for all classes of problems. The choice of a turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for a certain application, one needs to understand the capabilities and limitations of the various options. The standard *k-ε* model has become widely used in practical engineering flow calculations. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. The standard *k-ε* model is a semi-empirical model based on model transport equations for the turbulent kinetic energy (*k*) and its dissipation rate (ε). The model transport equation for *k* is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. From the literature review it is seen that many researchers have recommended and used the k-ε model to model turbulence in mixing studies. The flow generated by a teemixer has been studied by [5] - [7]. Simulation studies appeared somewhat later. Reference [8] simulated teemixing characteristics in both the absence and presence of a reaction. The flow and pressure fields for a teemixer were solved using the TEACH-T flow code which uses the SIMPLE algorithm of [9]. A three-dimensional model was constructed and the k-ε model was used to model turbulence. Reference [10] while evaluating four models of turbulence through comparisons of their extensive turbulent conical wall-jet data observed that k-ε model successfully predicted most of their flows. Reference [11] simulated pipeline sidetee mixing with the k-ε model to model turbulence. Reference [12] studied modeling of a single confined turbulent slot jet impingement using various k-ε turbulence models. Reference [13] predicted room air motion by Reynolds-stress models showing anisotropic turbulence and secondary recirculation existing in a room airflow for which the k-ε model was not suitable. However, the Reynolds-stress models require additional computing effort. Reference [14] did a comparison of linear and nonlinear RNG-based k-ε models for incompressible turbulent flows suggesting that the linear RNG k- model can yield significant improvements over the standard k-ε model for recirculatory flows, because of its less dissipative nature. Reference [15] did an experimental study and CFD modeling of barium sulphate precipitation in a closure pipe previously proposed by computing flow field using the k-ε model Reference [16] researched on the gravitational terms of the k-ε and other turbulence models and stated that it has significant importance in some cases. Reference [17] applied k-ε turbulence models to study two-dimensional turbulent wall jets and compared their results with measurements and numerical results found in the literature. Reference [18] used Reynolds stress model in the prediction of confined turbulent swirling flows giving adequate agreement between the experimental data and the simulations. Reference [19] explored the characteristics of cold three-dimensional wall jets using Reynolds stress model predictions and full-scale measurements. They found that the quadratic model predicted values on the turbulent variables with closer agreement to the measurements than the linear model**.** Reference [20] studied steady-state mixing in a t-junction and performed calculations with ANSYS-CFX-10 using the k–ε , SST and BLS Reynolds stresses models. They found that both turbulent mixing and turbulent momentum transport down stream of the side-branch connection are underestimated by these models. They got better results by increasing the model coefficient C_{μ} in the k–ε model. In this work side injection experiments are done and the simulations are carried out for collaboration to conjure up misdiscrepancies in the jet injection vicinity. Hot water is injected at various speeds to a cold main stream fluid in pipeline. In the vicinity of jet that is when side injection is entering into main fluid stream calculations of value of any gradient is difficult. Turbulence is introduced in it and at higher velocities different types of flow motion are evolved and ordinary measurement becomes difficult. k-ε and RSM model are applied to study the jet behavior in this vicinity. Discrepancies are observed in the jet vicinity.

II. EXPERIMENTAL SET UP

The experimental apparatus is shown schematically in Figure 1. An assembly consisting of a main horizontal PVC pipe 3m long is employed as the main part of the rig. The rig has a replaceable facility (unions at both ends of a replaceable horizontal pipe) so that different diameters of main pipe may be used. Experiments are carried out using 25.4mm diameter main pipe and side pipe of 6.35mm. 'X' is showing the main part of cross mixing.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

Experiments with different velocities were also carried out. Tests were done in Reynolds number range of 5000-50000. Suitable pumps are chosen to supply main fluid and side fluid respectively. Thermocouples are available and a desktop computer having data logging software OMEGA with suitable hardware to connect thermocouples. Output data from thermocouples is fed to desktop computer for data logging and storage. The commissioning of the experimental set up was done prior to the first experimental run. Some leakages found were sealed and other faults were rectified. A major electrical faulty connection to pump was found and corrected. Flow meters were calibrated and thermocouple reading for cold water supply (main) pump and hot water supply (side) pump was logged. Results showed that thermocouples were working correctly and data logging software OMEGA was also running correctly. To control heat losses from main pipe, insulation of the part where the two streams (main and side) were being mixed was done with lead wool. Less storage tank capacity of main supply slows down the main flow rate as the level of fluid in the tank comes down during operation. This cause was overcome by doubling the main supply fluid storage capacity.

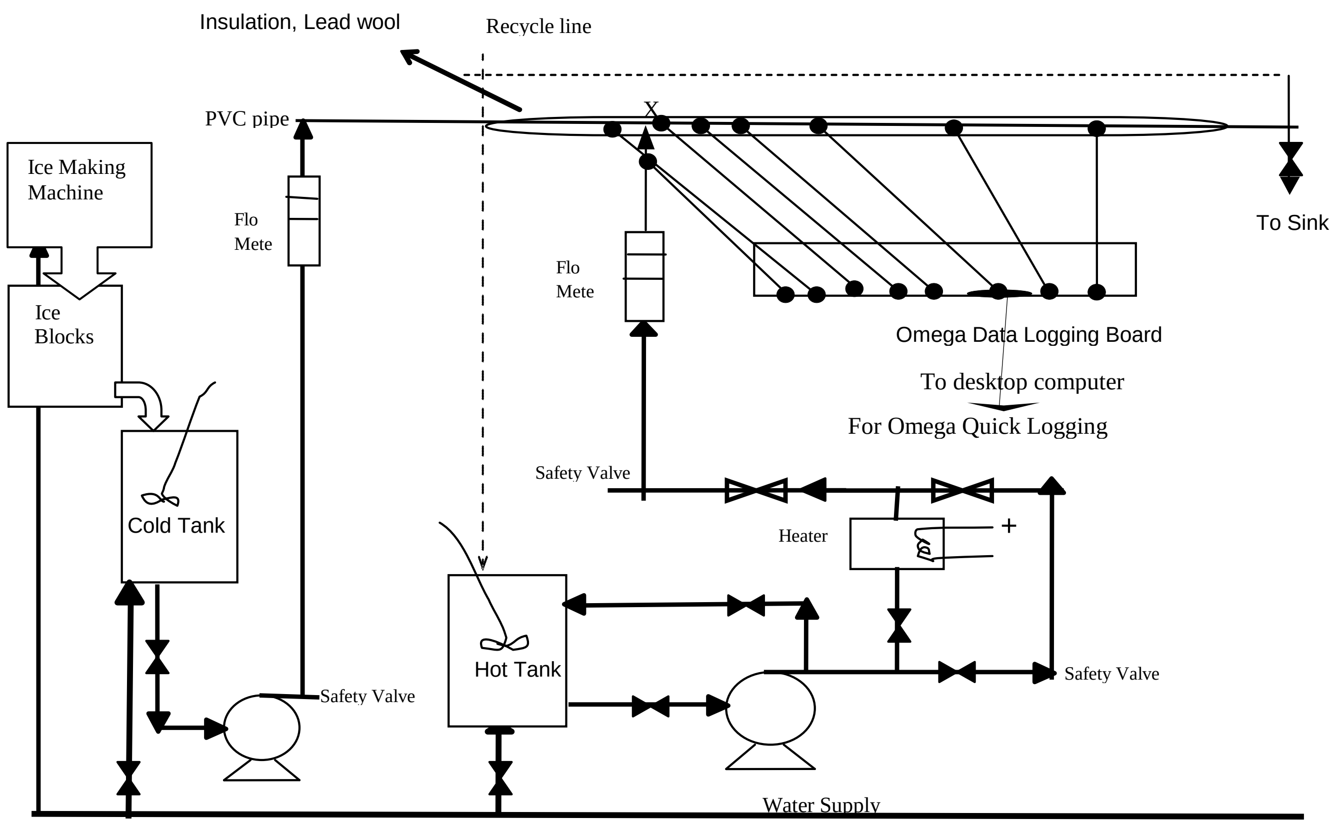


Fig 1: Schematic Diagram, 'X' Shows Main Part of Cross Mixing

III. MODELING & SIMULATION

Flow in pipeline is simulated by solving the mass and momentum conservation equations using CFD Package FLUENT. It allows the investigation of a range of conditions and geometries quite efficiently once a general model has been established and validated against experimental results. A segregated solver was used to solve the Reynolds Averaged Navier-Stokes equations along with the energy equation. With this numerical method, computational work is reduced by comparison to fully coupled solvers but higher convergence criteria have to be chosen. In this cross jet injection mixing study, k-□ and RSM model are used for simulations and comparison is done with experimental results to discuss jet vicinity discrepancies of k-□ model. Standard k-□ model is used with C□ □ □ □ □ □ □ □ □,C1□ = 1.44, C2□ = 1.92, TKE, TDR, Energy and Wall Prandtl number are 1, 1.3, 0.85, and 0.085 respectively. Fluid is water-liquid with boussinesq parameter and operating conditions of 101325 Pascal and operating temperature 288.16K. Gravity is taken -9.8m/s in y direction with geometry reference location at 0, 0, 0.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

RSM, seven equations model constants are taken as C□ □ □ □ □ □ □ □ □ □ , C1□ = 1.44, C2□ = 1.92, $C1_{PS}$ = 1.8, $C2_{PS}$ = 0.6, $C1'_{PS}$ = 0.5, $C2'_{PS}$ = 0.3, and TKE, TDR, Energy and Wall Prandtl number are 1, 1.3, 0.85, and 0.085 respectively. Iteration time for different cases is shown in Table 1 for both models. RSM is taking more iteration to resolve the same case.

Table 1: Iteration Histories of Turbulence Models

Sr	Velocity Ratio, Uj/Um	Turbulence Model		Difference	%age increase
		K-e Model	RSM Model		
1	1.57	106	191	85	80.19
2	6.31	107	164	57	53.27
3	14.73	117	437	320	273.50

A three-dimensional geometry representing a main pipe with a side-injection was created and meshed. An unstructured tetrahedral grid was chosen. Two different types of mesh, *tgrid* and *hexcore*, are simulated and compared at jet center line as shown in Figure 2. No significant effect is found with change of mesh type. In general, the k-ε model proved to be satisfactory especially when used for non-circulating flows. However, in the jet vicinity of pipeline flow its use is not favorable to extract results. Reynolds stress model is better in this area as shown in results. Following experiments simulations are carried out using the k-ε model and the Reynolds Stress model (RSM).

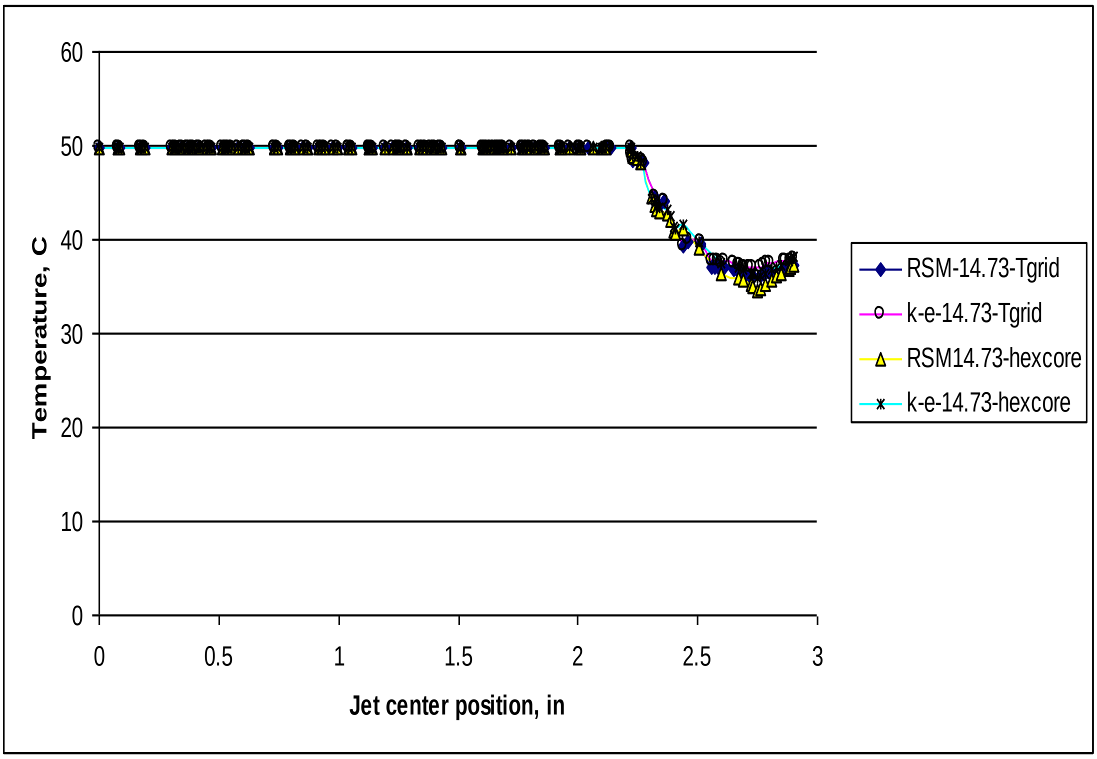


Fig 2: Temperature Plots Of Jet at Centerline Of Side Jet Using K-E and RSM Tetrahedral/Hybrid For Different Mesh Types Of *Tgrid* and *Hexcore* Uj = 14.73 M/S With Um = 0.40 M/S.

Figure 3 shows comparison of numerical results for a case of U_j = 14.73 m/s with U_m = 0.40 m/s using both of two models of turbulence. It shows that both RSM and k-□ model are good before and after jet mixing. However, for

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

the results in the vicinity of that incoming jet, a somewhat significant difference in the results is observed. On average, the RSM in FLUENT requires 60% more CPU time per iteration compared to the k–ε models. For certain cases of jet mixing, the increase in CPU iteration when using RSM was about 300% compared to that when using the standard k–ε. Further more, 15-20% more memory is needed. Change of machine for a case or running any other work window during any simulation effects the computation in any way. So, dedicated machines were used to run simulations.

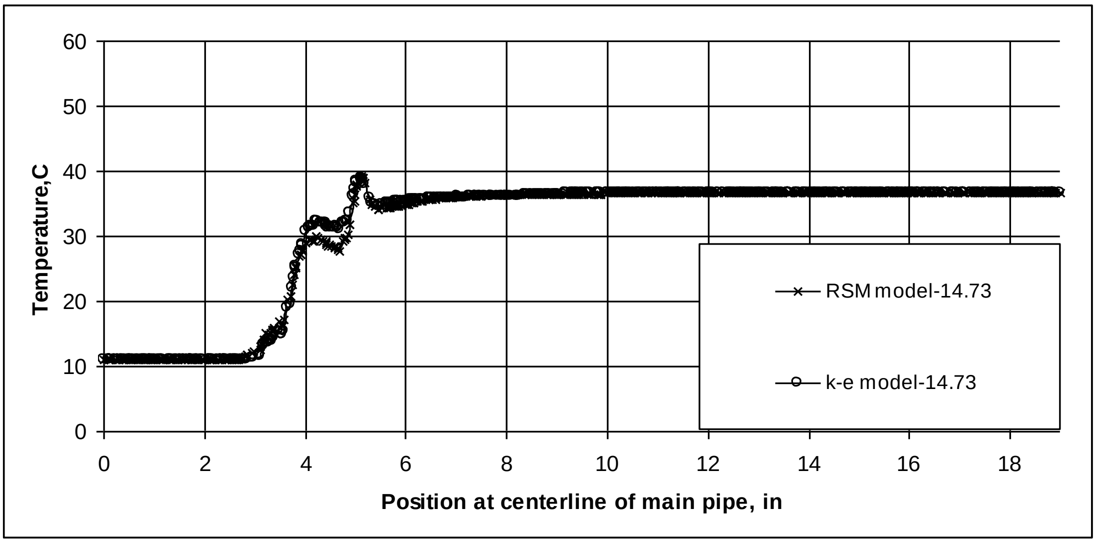
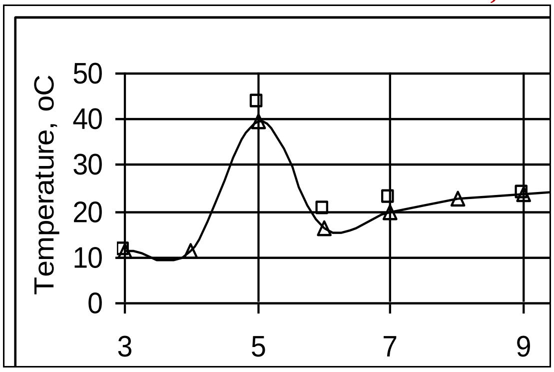


Fig 3: Comparison of Temperatures Values at Centerline Of Main Pipe With K-□ Model and RSM Model Along Main Flow Direction For U_j = 14.73 M/S With Um = 0.40 M/S, Injection Is At X=5.

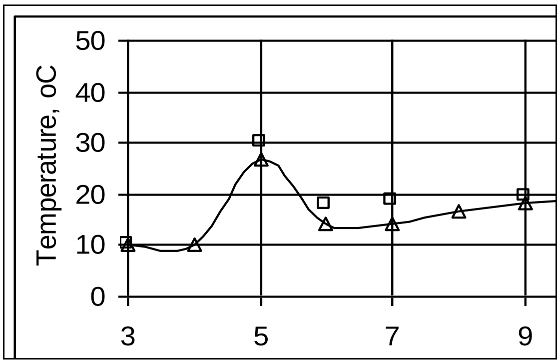
IV. RESULTS AND DISCUSSION

In order to analyze the results quantitatively and conjure both models, values of temperature versus location along the pipe axis are plotted. These numerical values are compared with experimental values measured at exactly the same locations. Experimental results are compared with simulated cases for 6.35mm side-injection using k-ε turbulence model. Velocity of the main-fluid is also varied. The temperature readings are along the centerline of the main pipe. The discrete points are the experimental values whereas the lines show the simulated results. Differences between experimental and numerical results are observed in the vicinity of the incoming jet due to the complexity of flow in that region. The temperature peak coincides with the hot side-fluid injection into the cold mainstream. Following the injection, mixing of hot and cold fluid takes place. For all cases, past the injection point, there is a temperature bouncing, which is due to the fact that hot fluid-jet is fluctuating due to main fluid resistance and inner wall friction and is not along the centerline of main pipe, but it is away from the center towards the opposite wall and with the momentum bouncing within mixed fluid. For a 6.35mm right-angle injection with 25.4mm main pipe, experiments are done for a set taking Uj = 3.94, 2.63 m/s with Um = 0.40 m/s as shown in Figure 4. Temperatures averaged over a few seconds are used for the main- and side-fluids. Flow rates are measured using calibrated rota-meters for velocity calculations. Temperature of fluid along the centerline of the main pipe is plotted versus position. All the experimental cases are simulated. Experimental and numerical results show good agreement. Line plots show the simulation results of temperature at centerline of main pipe whereas, square points show experimental results as logged via thermocouple. A higher temperature peak shows the side hot-fluid entrance. The temperature then decreases as the cold fluid of the main pipe is mixed with the hot fluid of side-pipe. The k-ε model is used to model turbulence and as mentioned earlier, it tends to underestimate the temperature peaks. At around in the vicinity of side stream entrance some discrepancies exist. These discrepancies were reduced by using the RSM model. Figure 5 show temperature plots along main pipe centerline using RSM for velocity ratios of 6.22, 9.77. These plots show much better agreement with experiments. Figure 6 shows a column chart comparison of k-□ model results along experiments and RSM model results alongexperiments. It is clear that RSM is very good model at jet entrance to other fluid as well as in jet vicinity. Its agreement is above 60 percent with experiments whereas k-□ is notlooking good within jet vicinity.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

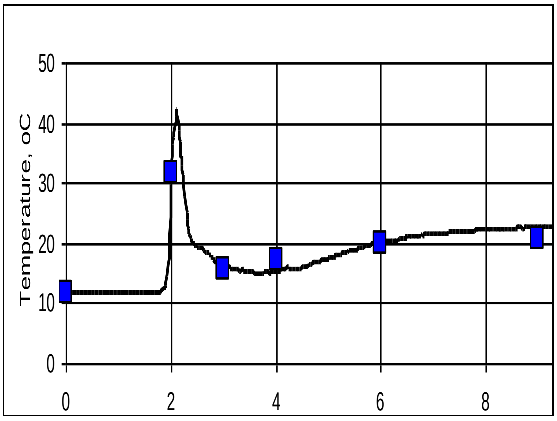


(a) U_j/U_m = 9.77, U_m = 0.40 m/s

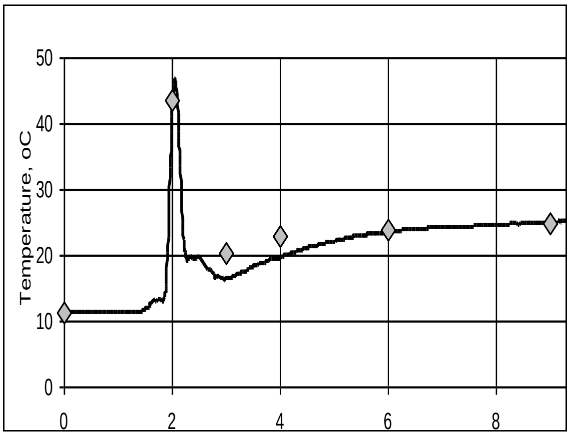


(b) U_j/U_m = 6.51, U_m = 0.40 m/s

Fig 4: Comparison of experimental and simulation results for 6.35mm side pipe, (a) U_j = 3.94 m/s U_j/U_m = 9.77, (b) U_j = 2.63 m/s U_j/U_m = 6.51, at centerline of main pipe, pipe length at x-axis is in diameter of main pipe (using k-ε model) injection at 5, □ Experimental, — Simulation



(a) U_j/U_m = 6.22, U_m = 0.40 m/s



(b) U_j/U_m = 9.77, U_m = 0.40 m/s

Fig 5: Comparison of experimental (points) and simulation (line) results for 6.35mm side-pipe, for U_j/U_m = 6.22 (U_j = 3.94 m/s, U_m = 0.63 m/s), and U_j/U_m = 9.77 (U_j = 3.94 m/s, U_m = 0.40 m/s), at centerline of main pipe, pipe length at x-axis is in diameter of main pipe (using RSM), Injection at 2, □ Experimental, — Simulation

Above results clearly show four zones as: 1. before injection of side fluid into main fluid, 2. zone of injection of side fluid, 3. zone immediately after injection of side fluid and this zone further can be by parts into four sub-zones, and the 4. far downward calm zone. The third zone which is immediately after injection is divided into further four sub-zones as: 1. very adjacent to injection, 2. adjacent to first sub-zone, 3. sub-zone between second sub-zone and fourth sub-zone, and 4. adjacent sub-zone to far downward calm zone. In this fourth zone and the adjacent sub-zone of third zone have less circulations, in other words less turbulence to account for choosing turbulence model. The k-ε model for both geometries is unable to predict the highest peak point accurately and also showing discrepancies after jet vicinity zone whereas RSM is successful in the second zone of jet vicinity. Unaware of experimental values in first zone is a lack in this study but obviously at higher jet velocities when back flow occurs there will be a second peak. To read this value experimentally, installment of so close thermocouple was very difficult. However, varying velocities it is clear that RSM is very good model at jet entrance to other fluid. After jet vicinity k-ε model behavior is fluctuating. It is predicting less or more but in acceptable range. RSM model is predicting well in last two sub-zones of third zone but its predictions are below to the mark in the first two sub-zones of third zone. The RSM gives better estimate of the temperature profile in the vicinity of the jet when compared with experimental results. At cost of higher machine computation time, RSM model is always preferable in jet vicinity. At least 60 percent of k-ε model accuracy in the jet vicinity zone is acceptable for working out problems far away from jet injection points.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

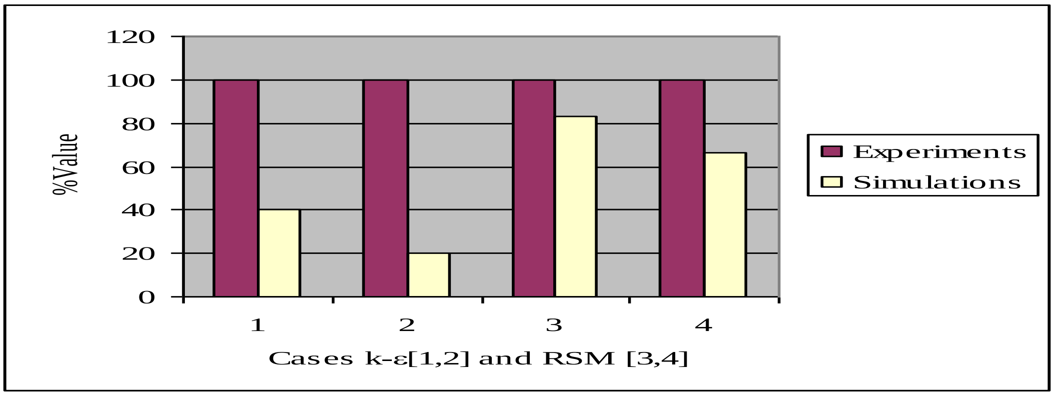


Fig 6: Chart shows comparison of k-ε model results along experiments and RSM model results along experiments. It is clear that RSM is very good model at jet entrance to other fluid as well as in jet vicinity. Its agreement is above 60 percent with experiments whereas k-ε is less promising with in jet vicinity.

V. Conclusion

Collaboration of numerical and experimental work in pipelines of side-injection was carried out. The designed experimental facility responded excellently to data generation. The temperature of the mixing streams was chosen in such a way which allowed the use of temperature as a measured variable. Good agreement between experimental and numerical results is observed in far down pipe. Discrepancies are observed in the values of temperature in the vicinity of the jet incoming through the side-injection. This is due to the more complex nature of the flow at this place. A better agreement in this regime was obtained by using RSM turbulence model instead of the k-ε modelto conjure these misdiscrepancies. Therefore, it is better to use it where turbulence is more and any type of expanse (eddies, swirl and back mixing with jet impingement etc. to mention) are involved to prevent any type of loss in the process by mystification. Different velocities and phases for different geometries may also be investigated for turbulence model conjuring for jet vicinity.

Acknowledgment

The support of department of chemical engineering of, and King Fahd University of Petroleum and Minerals, Dhahran-31261, Saudi Arabia is acknowledged. Also thanks to Prof. MB Amin, Dr. AA Shaikh, Dr. SU Rehman, Prof. RN Sharma, Dr. KF Loughlin, Prof. MA Gondal and Prof. H Redhvi, advisor and chairman chemical engineering department.

REFERENCES

[1] Gosman, A. D., “Developments in Industrial Computational Fluid Dynamics”, Trans I Chem E, 76 (Pt A), 53-161, 1998.

[2] Gray, J. B., Mixing: Theory and Practice, III, J.B. Gray and V. W. Uhl, Academic Press, 1986.

[3] Simpson, L. L., “Turbulence and Industrial Mixing,” Chem. Eng. Prog., 70, 77, 1974.

[4] Forney, L. J., Encyclopedia of Fluid Mechanics, Vol. II, Ch. 25, N.P. Cheremisinoff, Gulf Publishing Co., Houston ,1986.

[5] Moussa, Z., J. Trischka, and S. Eskinazi, “The near Field in the Mixing of a round Jet with a Cross Stream”, J. Fluid Mech., 80, 49, 1977.

[6] Crabb, D., D. Durao, and J. H. Whitelaw, “A Round Jet Normal to a Cross Flow”, J. Fluids Eng., 103, 103, 1981.

[7] Andreopoulos, J., “Heat Transfer Measurements in a Heated Jet-Pipe Flow Issuing into a Cold Cross Stream”, Phys. Fluids, 26, 3201, 1983.

[8] Cozewith, C., G. VerStrate, T. J. Dalton, J. W. Frederick, and P.R. Ponzi, “ Computer Simulation of tee mixers for Non-reactive and Reactive Flows”, Ind. Eng. Chem. Res., 30, 270-275, 1991.

[9] Patankar, S. V., Spalding, “DBA Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows”, Int. J. Heat Mass Trans., 15, 1787-1806, 1972.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

[10] Sharma R. N. and S. V. Patankar, "Numerical Computations of Wall-Jet Flows", Int. J. of Heat and Mass Transfer, 25(11), 1982.

[11] Forney, L. J. and L. A. Monclova, "Numerical Simulation of Pipeline tee mixers: Comparison with Data", Industrial Mixing Technology: Chemical and Biological Applications, E. L. Gaden, G. B. Tatterson, R. V. Calabrese, and W. R. Penney, 141-143, Academic Press, NY, 1994.

[12] Seyedein S. H., Hasan M., and Mujumdar A. S., "Modelling of a single confined turbulent slot jet impingement using various k-e turbulence models", Appl. Math. Modelling, , Vol. 18, October, 526, 1994.

[13] Chen Q., "Prediction of Room Air Motion by Reynolds-Stress Models", Building and Envt, Vol. 31, No.3, 233-244, 1996.

[14] Papageorgakis G. C. and Assanis D. N., "Comparison Of Linear And Nonlinear Rng-Based k-ε Models For Incompressible Turbulent Flows", Numerical Heat Transfer, Part B, 35:1- 22, 1999.

[15] Baldyga, J. and W. Orciuch, "Barium sulphate precipitation in a pipe -an experimental study and CFD modeling", Chem. Eng. Sci., 56, 2435-2444, 2001.

[16] Hassid S., "On the gravitational terms of the k-ε and other turbulence models",Ocean Dynamics 52: 169-178, 2002.

[17] Kechiche J, Mhiri H, Le Palec G, Bournot P, "Application of low Reynolds number k-ε turbulence models to the study of turbulent wall jets", International Journal of Thermal Sciences 43, 201-211, 2004.

[18] Vatistas G. H., Jawarneh A. M., "Reynolds Stress Model in the Prediction of Confined Turbulent Swirling Flows", Journal of Fluids Engineering, ASME, November, Vol. 128 / 1377, 2006.

[19] Tornstrom T. and Moshfegh B., "RSM predictions of 3-D turbulent cold wall jets", Progress in Computational Fluid Dynamics, an International Journal Volume 6, Number 1-3 /, 110 - 121, 2006.

[20] Walker, C., et al., "Steady-state RANS-simulations of the mixing in a T-junction". Nucl. Eng. Des. doi:10.1016/j.nucengdes.2010.05.056, 2010.

AUTHOR BIOGRAPHY

Mamdouh Ahmad S Al-Harthi
Chemical Engineering, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia
Email: mamdouh@kfupm.eud.sa

Zahid Hafeez I Khokhar
CHEM-3399, Professional Engineer, Chemical Engineering, Pakistan Engineering Council
P-2420, St # (19), Farooq West C Rd. Faisalabad-38060, PAKISTAN, zahidhkhokhar@yahoo.com, 24-2012
Corresponding email: zahedhk@gmail.com

Abdulazeez Abdurraheem
Research Institute, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia
toazeez@gmail.com

Habib Al-Ali
Department of Chemical Engineering,
King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia

Habib Daoud Zughbi
Petroleum Engineering, UNSW, Sydney, Australia

Bekir S Yilbas
Department of Mechanical Engineering, King Fahd University of Petroleum & Minerals, Dhahran-31261, Saudi Arabia