International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
ISNCESR-2015 Conference Proceedings

A Review on Turbulent flow Through Pipe- "A State of Art"

Ashu Kumar Pandey[1]
M.Tech CAD/CAM ROBOTICS
Mechanical Engineering Department
Christian College of Engineering and Technology
Bhilai, India

Rahul Patil[2]
Assitant Professor
Mechanical Engineering Department
Christian College of Engineering and Technology
Bhilai, India

***Abstract*— In this paper a review of turbulent flow through a pipe using experimental or computational approach is presented. Vast researches has been carried out by the researches in the flied of fluid flow within pipes or channel in order to optimized the flow performance characteristic. Navier stokes and Euler energy continuity equations are the basis governing equation for the fluid flow .During flow within pipe due to friction, sudden expansion, pipe bending significant losses has been seen which can be optimized by implement roughness in form of baffles, ribs, dimples, etc are practiced from optimizing the thermal as well as flow characteristics.**

Keywords- CFD, Turbulent flow, pressure drop

I. INTRODUCTION

In a field of fluid dynamics and turbulent flow heat transfer over rough surfaces has been a area of increasing interest. This type of flow can be examined in various engineering applications such as, nuclear reactor, heat exchangers, turbine blade, fluid catalytic cracking wind tunnel, and air foil. In rough pipes the examination of fluctuating velocity spectra is employed to determine the turbulence profile in all coordinate directions. A significant scrutiny of this study was that the nature of the solid boundary has negligible effect on the flow in the innermost part of the pipe. On the other side the flow near the wall is dependent on the nature of the solid boundary.

One of the most widely used heat exchangers in industry is the shell-and-tube type due to its relatively straightforward construction and multi-purpose applications for gaseous and liquid media over pressure and large temperature ranges. Several types of baffles have been utilized in shell-and-tube heat exchangers over the years to enhance fluid performance and the rate of heat transfer and to provide structural support. The most commonly used baffles, the segmental baffles, dimples, cause the shell side fluid to flow transversely to the tube bundle, thus advancing heat transfer by enhancing local mixing on the shell side of the exchanger, but at the cost of an increase in pressure drop . Besides heat exchangers, circular pipes fitted with segmental baffles also find varied applications such as in labyrinth seals, chemical reactors, desalination and filtration. Pipe with segmental baffles also has been used as an idealized model of the human cystic duct to simulate the flow of bile from human gallbladder to the common bile duct. Flow visualization and pressure drop measurements have been involved to study the effects of baffle configuration on the flow and on heat transfer. However, several analytical approaches are still going on to predict the phenomena.

II. LITERATURE REVIEW

In 1973 Richman and azad perform numerical and and experimental investigation for analyzing steady incompressible developing turbulent flow behavior in smooth pipes. A modified Van driest viscosity model is adopted to solve governing equation to determine vorticity transport and stream function. The obtained result is verified with the experiential result at different Reynolds number.

Quader and Wilkinson 1980 investigate Isothermal and non-isothermal flow rate-pressure drop result in turbulent flow through smooth pipes for non-Newtonian fluids. It has observed that friction factor plays a significant role with Reynolds number during fluid flow. And other papramters such as pipe diameter, mean velocity, Bowen's correlation are experimentally derived parameters which characterize the fluid.

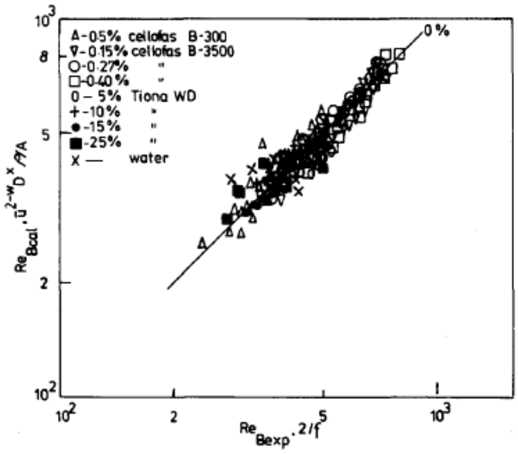

Figure 1Re_{Bcal}-Re_{Bexpt}, Plot for non-isothermal pressure drop results [2]

In 1994 Linden and Hoogendoorn perform experimental to visualize the effect of a pressure wave on the turbulent flow and heat transfer in rectangular air flow channel. Hot film sensor, hot wire and pressure transducer are used for instant measuring heat flux, velocity and pressure. It has been found that heat transfer is function of thermal boundary layer thickness and Reynolds number. The results are compared with simple numerical turbulent flow and heat transfer model.

Habib et.al 1999 experimentally investigates the Heat Transfer characteristics of pulsated turbulent pipe flow under different conditions of pulsation frequency, amplitude and Reynolds number. Uniform flux was kept at the pipe wall. Reynolds number was varied from 5000 to 29000 while frequency of pulsation ranged from 1 to 8 Hz. The results show an enhancement in the local Nusselt number at the entrance region. The rate of enhancement decreased as Re increased. Reduction of heat transfer coefficient was observed at higher frequencies and the effect of pulsation is found to be significant at high Reynolds number. It can be concluded that the effect of pulsation on the mean Nusselt numbers is insignificant at low values of Reynolds number. At Reynolds numbers below 2000, after at a high level of the inlet pulsations an almost laminar flow had developed in the pipe, a perturbation was introduced by inserting a diametrically oriented cylindrical rod with a diameter 10–20 times smaller than the pipe diameter. In these experiments, at Reynolds numbers higher than 1000, at a distance from the rod equal to 50 pipe diameters the axial to mean-flow velocity ratio was less than 2, approaching this value again at large distances from the rod. The insertion of the rod led to a decrease in the

Heuydong et.al 2001 investigates the effectiveness of an orifice system in producing pressure drops and the effect of compressibility on the pressure drop; computations using the mass-averaged implicit Navier-Stokes equations were applied to the axisymmetric pipe flows with the operating pressure ratio from 1.5 to 20.0. The standard k- ε turbulence model was employed to secure the governing equations. Numerical calculations were carried out for some combinations of the multiple orifice configurations. The present CFD data showed that the orifice systems, which have been applied to incompressible flow regime to date, could not be used for the high operating pressure ratio flows. The orifice interval did not strongly affect the total pressure drop, but the orifice area ratio more than 2.5 led to relatively high pressure drops. The total pressure drop rapidly increased in the range of the operating pressure ratio from 1.5 to 4.0, but it nearly did not increase when the operating pressure ratio was over 4.0. In the compressible pipe flows through double and triple orifice systems, the total pressure drop was largely due to shock losses.

David et.al 2008 performed a comprehensive measurement of local heat transfer coefficients in a square channel with a perforation baffle by means of the transient liquid crystal themography. The varied parameters were the Reynolds number, the baffle height, and the hole numbers on the perforation baffle. The results showed that the enhancements of local heat transfer appeared in the leading edge of the baffle due to the impinging effect, which was more significant when Reynolds number became larger or the baffle height became higher. Additionally, the heat transfer coefficients off center were better than those in the center at downstream of the baffle. It might be resulted from two secondary flows, which appeared off center after the airflow passed through the baffle. Baffles with various hole numbers but having same total hole area were also studied to find the heat transfer enhancement. The results depicted that the back facing step flow which had characteristics of backflow and flow reattachment had an important effect on the heat transfer characteristics at downstream of the baffle. Finally, the correlation for the location of the flow reattachment point (X_r) was proposed.

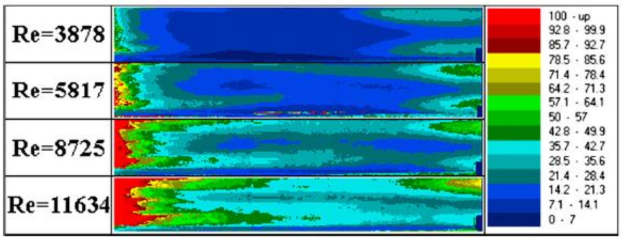

Figure 2 Distribution of heat transfer coefficient of h/H=0. [6]

Abou et. al 2010 Fractal-shaped orifices are thought to have a significant effect on the flow mixing properties downstream a pipe owing to their edge self-similarity shape. Here, we investigate the pressure drop after such fractal orifices and measure the pressure recovery at different stations downstream the orifice. A direct comparison is made with the pressure drop measured after regular circular orifices with the same flow area. Our results show that the fractal-shaped orifices have a significant effect on the pressure drop. Furthermore, the pressure drop measured across the fractal-shaped orifices is found to be lower than that from regular circular orifices of the same flow areas. This result could be important in designing piping systems from the point of view of losses. It looks promising to use the fractal-shaped orifices as flowmeters as they can sense the pressure drop across them accurately with lower losses than the regular circular-shaped one

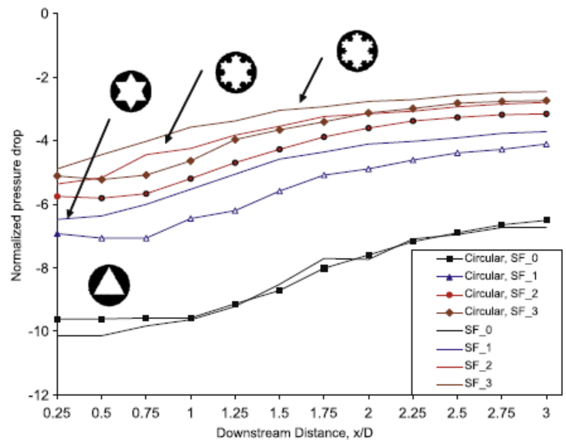

Figure 3 Normalized pressure drop 2Dp=qV2 inlet for SF-type fractal-shaped orifices and the equivalent circular orifices [7]

Teng et. al 2011 elaborates the pressure drop in circular pipes of TiO_2/water nanofluid for both turbulent and laminar flows at different temperatures and TiO_2 weight fractions. In their study shows that TiO_2/water nanofluid causes enhancement, but temperature rise reduces pressure drop. The comparative increase in pressure drop for turbulent flow is lower than that for laminar flow. The traditional equation for pressure drop fails to accurately estimate the pressure drop for laminar and turbulent flows. Accordingly, this study developed new empirical equations for the friction factor for both laminar and turbulent flows, and the maximum deviations between calculated and experimental results were reduced to within the ranges of −6.17% to 3.55% and −3.08% to 3.81%, respectively, that is, for TiO_2/water nanofluid, the correlations apply better to turbulent than to laminar flow.

Riccardo et. al presented modeling of break-up in an inhomogeneous flow which develops in concentric restriction in a pipe. Euler-Lagrange is used for simulations of the drop motion of interface deformation model. Turbulent flow downstream is solved by direct numerical simulation firstly then single drop trajectories are calculated. Simultaneously, the interface deformation is computed by Rayleigh–Lamb type oscillator forced. The flow conditions and fluid properties is taken to match with the experimental studies. Turbulent flow

covenant with experimental data.

Kang and yang 2012 investigates the flow instability in presence of baffles in channel. The simple geometry of baffles is adopted from heat exchanger for analyses. In investigation parametric analyses has been done to verify the effect of baffles aspect ratio, however the effect of baffle interval and variable Reynolds number has also carried out. The main aim for stability analysis in order to study the secondary instability through which time-periodic two-dimensional flow bifurcates into three-dimensional flow The obtained results are compared with computational result and showing good agreement.

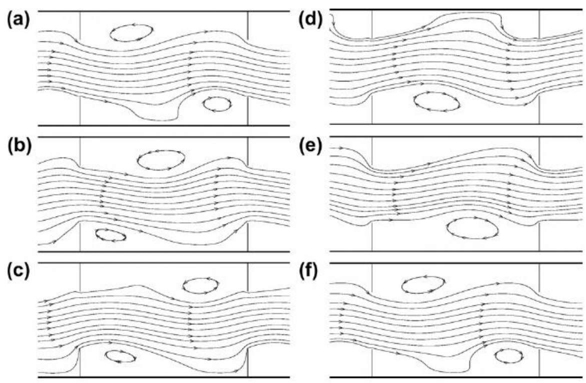

Figure 4 Streamlines of the unsteady flow during one period for RB = 1.456 and Re = 130; (a) t = 0T, (b) t = 1/5T, (c) t = 2/5T, (d) t = 3/5T, (e) t = 4/5T and (f) t = 5/5T. [10]

Fedelea and Dutykhb 2013 studied dynamics of a interruption to the laminar state in non-rotating axisymmetric with in Poiseuille pipe Flows. The Navier stokes equation is employed as governing equation which is coupled with Camassa-Holm type equations. Due to presence of peakons bifurcate from smooth solitary waves as their celerity enhances. Peakons are support singular inviscid travelling waves with wedge type singularities. In physical state corresponds to vortexons or toroidal vortices. Which lead disturbances during flow at transition regimes.

Wael 2014 perform CFD analysis to study pressure drop and heat transfer characteristic using water based Nano fluid flowing inside the heat exchanger. In analysis realizable k–ε turbulent model with enhanced wall treatment has been adopted. Parametric analysis has been carried out for Nano particles, coil diameter inner tubes and annulus side flow rates .it is found that heat transfer coefficient increases on increasing coil diameter volume contraction. However friction flows also increases with increase in curvature ratio and pressure ratio .since pressure drop is considered to be negligible with increase in volume concentration of Nano fluid. At last conventional correlations are used for predicting heat transfer and friction factor in turbulent regime

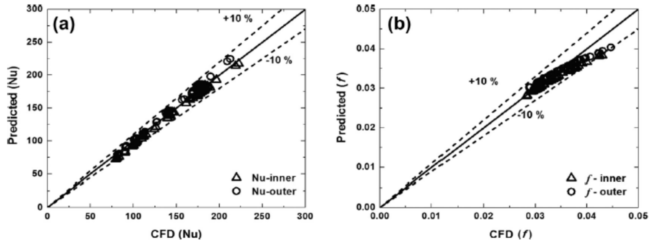

Figure 5 Comparison of the simulated values of (a) Nusselt number (b) friction factors, [11]

Ahsan 2014 perform CFD analysis for simulating fully turbulent flow in a pipe at higher Reynolds number. keε turbulence model has been adopted to solve the governing equation of fluid flow at high Reynolds number with enhanced wall treatment. A Numerical result has been presented to illustrate the effects of Reynolds number on turbulence intensity, average shear stress and friction factor. Friction factor is used to investigate the pressure drop along the length of the pipe. The contours of wall function are also presented to investigate the effect of enhanced wall treatment on a fluid flow. A maximum Reynolds number is also found for which the selected pipe length is sufficient to find a full developed turbulent flow at outlet. The results of CFD modeling are validated by comparing them with available data in literature. The model results have been shown good agreement with experimental and correlation.

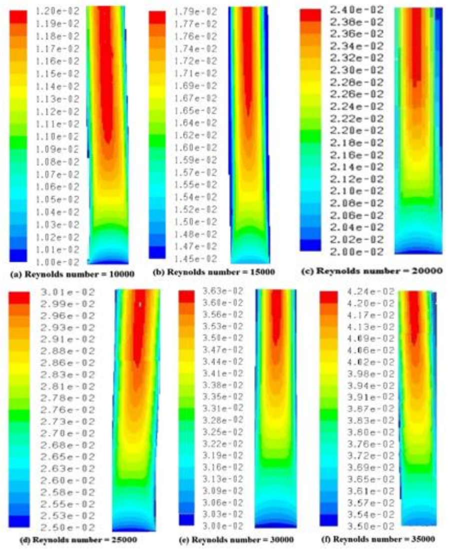

Figure 6 Contours of velocity magnitude (m/s) at various Reynolds numbers [12]

III. CONCLUSION

It is found that the pressure drop across pipe with baffles Flow visualization recommended that the baffles induce turbulence-like flow structure at Reynolds number much lower than the critical value where flow transition is expected to occur. On increasing Reynolds number pressure drop significantly increases. The model accounted for the geometry of the baffles and used well known correlations to combine the effects of turbulent and laminar flows. System in nominally laminar flow condition.

On inserting perforated taps or Insert in pipe pressure drop increases.

Heat transfer is depending on thermal boundary layer thickness and thus on the Reynolds number.

The implementing roughness the heat transfer and pressure drop significantly affected.

REFERENCES

[1] J. W. Richman, R. S. Azad, "Developing turbulent flow in smooth pipes" Applied Scientific Research,1973, Volume 28, Issue 1, pp 419-441

[2] A.K.M.A. Quader, W.L. Wilkinson, " Correlation of turbulent flow rate-pressure drop data for non-newtonian solutions and slurries in pipes", International Journal of Multiphase Flow, Volume 6, Issue 6, December 1980, Pages 553-561

[3] R. J. van der Linden, C. J. Hoogendoorn, "Transients in flow and local heat transfer due to a pressure wave in pipe flow", Applied Scientific Research, June 1994, Volume 52, Issue 4, pp 371-399

[4] M. A. Habib, S. A. M. Said,A. A. Al-Farayedhi,S. A. Al-Dini,A. Asghar,S. A. Gbadebo, "Heat transfer characteristics of pulsated turbulent pipe flow" Heat and Mass Transfer February 1999, Volume 34, Issue 5, pp 413-421

[5] Heuydong Kim, Toshiaki Setoguchi, Shigeru Matsuo, S. R. Raghunathan, "Pressure drop control using multiple orifice system in compressible pipe flows", October 2001, Volume 10, Issue 4, pp 309-317

[6] K. David Huang, Sheng-Chung Tzeng, Tzer-Ming Jeng, Jee-Ray Wang, Sen-Yao Cheng, Kuo-Tung Tseng "Experimental study of fluid flow and heat transfer characteristics in the square channel with a perforation baffle",International Communications in Heat and Mass Transfer, Volume 35, Issue 9, November 2008, Pages 1106-1112

[7] A. Abou El-Azm Aly, A. Chong, F. Nicolleau, S. Beck, "Experimental study of the pressure drop after fractal-shaped orifices in turbulent pipe flows" Experimental Thermal and Fluid Science, Volume 34, Issue 1, January 2010, Pages 104-111

[8] Tun-Ping Teng, Yi-Hsuan Hung, Ching-Song Jwo, Chien-Chih Chen, Lung-Yue Jeng "Pressure drop of TiO2 nanofluid in circular pipes",Particuology, Volume 9, Issue 5, October 2011, Pages 486-491

[9] Riccardo Maniero, Olivier Masbernat, Eric Climent, Frédéric Risso, "Modeling and simulation of inertial drop break-up in a turbulent pipe flow downstream of a restriction",International Journal of Multiphase Flow, Volume 42, June 2012, Pages 1-8

[10] Changwoo Kang, Kyung-Soo Yang, "Flow instability in baffled channel flow",International Journal of Heat and Fluid Flow, Volume 38, December 2012, Pages 40-49

[11] Wael I.A. Aly, "Numerical study on turbulent heat transfer and pressure drop of nanofluid in coiled tube-in-tube heat exchangers", Energy Conversion and Management, Volume 79, March 2014, Pages 304-316