International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 02 | Feb-2016 www.irjet.net p-ISSN: 2395-0072

Computational Analysis of Turbulent flow heat transfer and pressure loss in Duct or Pipe with Segmental Baffles

Prakash Chand[1], Y.P Banjare[2]

[1]*M.Tech Scholar, Mechanical Engineering Department*
[2]*Professor, Mechanical Engineering Department*
[1,2]*Government Engineering College Jagdalpur, Chhattisgarh, India*

---***---

Abstract – *Turbulent Flow in pipes and ducts are one of the major research domains which considerably receives great attention due to its Importance in many engineering applications such as heat exchanger, nuclear reactor, Solar heater, and many industrial cooling machine and devices. Therefore, it has been the subject of interest for researchers. Local heat transfer and pressure drop on the pipe and Baffles take place in order to enhance the flow characteristics and thermal performance segmental baffles are implemented in the form of extended surface. Due to implementation of Baffle leads to significant turbulence which increases friction with the pipe or duct and improve heat transfer and furthermore leads to momentous pressure drop.*

In the present work local heat transfer and pressure drop along the length are examined and comparative analysis has been done between smooth and segmental Baffle pipe. It has observed that on implementing the Baffles improves the thermo hydraulic performance of the pipes and ducts. The analysis has been done with the help of Finite element volume tool ANSYS- Fluent, where simulation is being done. During the simulation wide range of Reynolds number in both laminar and turbulent flow regimes are considered. The FEV results are validated with well published results in the literature and furthermore with experimentation. The FEV and experimental results show good.

Key Words: (Size 10 & Bold) Key word1, Key word2, Key word3, etc (Minimum 5 to 8 key words)…

1. INTRODUCTION

Turbulent Flow over rough surfaces or even extended surface in form of segmented Baffles or Fins has been a subject of increasing interest in the areas of heat transfer and fluid dynamics over the last few years. Flows of this nature can be found in Engineering systems of significant technological interest such as turbine blade internal cooling, advanced gas cooled nuclear reactor, heat exchangers and cooling of microelectronic devices

Heat transfer enhancement techniques can be classified whichever as passive, which involve no direct application of external power, or as active, which require external power. The effectiveness of both types of techniques is strongly reliant on the mode of heat transfer. This may range from single-phase free convection to dispersed- flow film boiling. Segmented Baffles and fins work on the principle of passive enhancement technique. One of the ways of heat transfer enhancement is to implement segmented baffles of fins create in the flow. In ducts, heat transfer augmentation by means of secondary flow is obtained either by means of Segmental Baffles or an insert (e.g. twisted tape) in a straight duct or by using a twisted duct. Segmented Baffles used in heat exchanger not only enhances the heat transfer inside the duct but also outside it (duct to duct space of heat exchanger bundle). Baffles or extended surface in form of fine induced flow inside the duct or pipe is expected to enhance the heat transfer coefficient by an amount similar to that of twisted tape or turbulator inserts.

2. Literature Review

Kirmse 1979 measures the velocities and pressure gradients in pulsating turbulent pipe flow by means of Laser-Doppler Velocimeter and inductive pressure transducers. [2] It has found that the temporal and spatial velocity field in the pulsating turbulent pipe flow could be adequately determined for the complete flow field with the exclusion of the region extremely close to the wall ($y < 0.0065$, where y expresses the ratio of wall distance to pipe radius).

Quader and Wilkinson 1980 investigate Isothermal and non-isothermal flow rate-pressure drop result in turbulent flow through smooth pipes for non-Newtonian fluids.[3] 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.

Liou and Kao 1988 use laser-Doppler velocimetry for measuring flow past double-sided wall obstacles in a rectangular duct with aspect ratio. The velocity overshot are examined and studied the flow characteristics for upstream and downstream.[4]

Pollard et. al 1989 by means of blade manipulator devices the effect on developing turbulent pipe flow has been investigated computationally and experimentally aiming to reduce transmission loss within pipeline. [5]Through nine manipulator configurations Wall Pressure and Mean Axial Velocity profiles have been obtained. Using finite volume method Numerical calculations have been performed for two of these manipulator configurations at low Reynolds number model of turbulence. The results revealed that nett drag reduction may be possible in this flow.

Chevrin et. al 1992 examine Reynolds stress in the near-wall region of a fully developed turbulent pipe flow. Glycerine is adopted as working fluid Because of closed circuit tunnel and Two laser Doppler velocimeter (LDV) systems has been employed for measuring the two point spatial correlation, R 12, between the Stream wise and Radial velocities in a radial plane of the pipe.[6]

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. [7] The results are compared with simple numerical turbulent flow and heat transfer model.

Pavelyev et. al 2003 conducts experiment to evaluate resistance coefficients at critical Reynolds number from Blasius formula for laminar flow and from the Prandtl formula for turbulent flow are identical and the effect of insert has also been investigated. From this it can be concluded that insertion of the rod led to a decrease in the critical Reynolds number.[8]

Galinat et. al 2005 analyzed the drop fragmentation process caused by a cross-sectional restriction in a pipe. Using Trajecto-graphy Drop break-up downstream of the restriction has been studied and found that mean drop diameter downstream of restriction linearly increases as a function of the inverse of the square root of the pressure drop.[9]

Abou et. al 2010 examined the the pressure drop for an Fractal-shaped orifices and give measure for pressure recovery at different stations downstream the orifice. It has found that for same flow area pressure drop across the fractal-shaped orifices is lower than that from regular circular orifices.[11]

Teng et. al 2011 discusses the pressure drop in circular pipes of Nano-fluid for both Laminar and Turbulent flows at different temperatures and Weight fractions. It has found that nano-fluid causes enhancement, but on increasing temperature reduces pressure drop. Moreover the proportional increase in pressure drop for turbulent flow has been lower than that for laminar flow.[13]

Nouri et. al 2013explorethe bubble effect on the pressure drop caused by frictional losses in upward pipe flow. It has found that on increasing void fraction the bubbles effect on the frictional stress increases and decreases the pressure drop in the pipe and revealed that the bubble injection into pipe flow can be used to decrease the flow transfer costs.[14]

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. [15]

Nimwegen et al 2015 perform Direct Numerical Simulation of turbulent flow integrated with an arbitrary roughness topography using a combined momentum–mass source immersed boundary method. This method incorporated into a standard second-order finite-volume code with a Fast Fourier Transform solver. The obtained results are compared with laminar and turbulent flow over walls with sinusoidal undulations. [16]

3. Mathematical Modelling

The **Navier–Stokes equations** can be written in the most useful form for the development of the finite volume method:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \quad (1)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \quad (2)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \quad (3)$$

Governing equations of the flow of a compressible Newtonian fluid

Continuity $$\frac{\partial \rho}{\partial x} + div(\rho u) = 0$$

x-momentum

$$\frac{\partial(\rho u)}{\partial x} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \quad (4)$$

y-momentum

$$\frac{\partial(\rho v)}{\partial y} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \quad (5)$$

z-momentum

$$\frac{\partial(\rho w)}{\partial z} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \quad (6)$$

Energy

$$\frac{\partial(\rho i)}{\partial t} + div(\rho i u) = -p div u + div(k grad T) + \Phi + S_i \quad (7)$$

Using various correlation FEV results are been compared analytically

$$h_f = f \frac{LV^2}{D_h 2g}$$

Where,
f is the friction factor for fully developed laminar flow
L: length of the pipe
V: mean velocity of the flow
d: diameter of the pipe
f is the friction factor for fully developed laminar flow:

$f = \frac{64}{Re}$ (For Re<2000) $Re = \frac{\rho u_{avg} d}{\mu}$

C_f is the skin friction coefficient or Fanning's friction factor.

For Hagen-Poiseuille flow: $C_f = \tau_{wall} l \frac{1}{2}\rho u_{avg}^2 = \frac{16}{Re}$

For turbulent flow: $\frac{1}{\sqrt{f}} = 1.74 - 2.0\log_{10}\left[\frac{\varepsilon_p}{R} + \frac{18.7}{Re\sqrt{f}}\right]$ Moody's Chart

R: radius of the pipe

ε_p: degree of roughness (for smooth pipe, $\varepsilon_{p=0}$)

$Re \rightarrow \infty$: Completely rough pipe

4. Methodology

The ANSYS 14.5 finite element program was used for analyzing Pipe flow. For this purpose, the key points were first created and then line and spline segments were formed. The lines were combined to create an area. Finally, this area was extruded a We modeled the Pipe with Segmented Baffles.

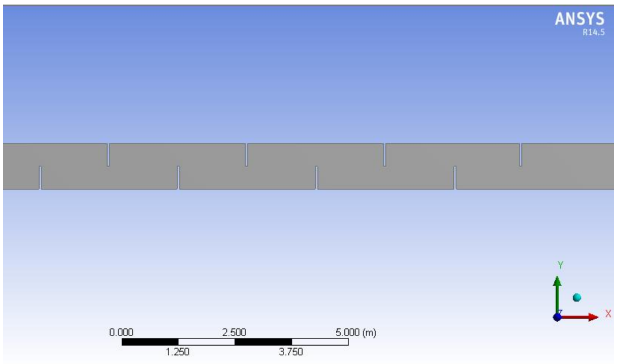

Fig. 1 Model Geometry

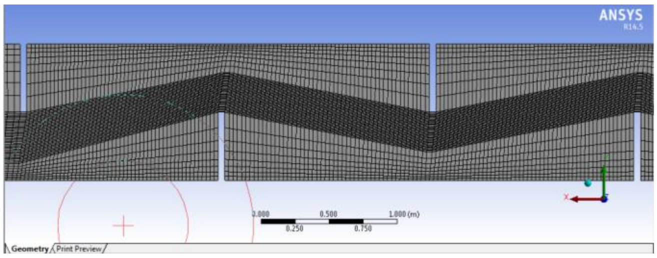

Fig. 2 Mesh Model

A 20-node three-dimensional structural solid element was selected to model the Baffled pipe. The Pipe with Segmented Baffles was discretized into 49841 elements with 50726 nodes. The Pipe surface boundary conditions can also be provided in mesh section through naming the portion of modeled Pipe i.e Inlet, Outlet, Top wall, Bottom Wall, Baffles.

5. Result and Discussion

The governing equations of the problem were solved, numerically, using a Element method, and finite Volume method (FVM) used in order to calculate the Hydrodynamic characteristics of a Pipe with Segmented Baffles. As a result of a grid independence study, a grid size of 10^6 was found to model accurately the Hydrodynamic performance characteristics are described in the corresponding results.

The accuracy of the computational model was verified by comparing results from the present study with those obtained by Ahsan [15], McGovern [12] Mushtak [10], Experimental, Analytical and FVM results.

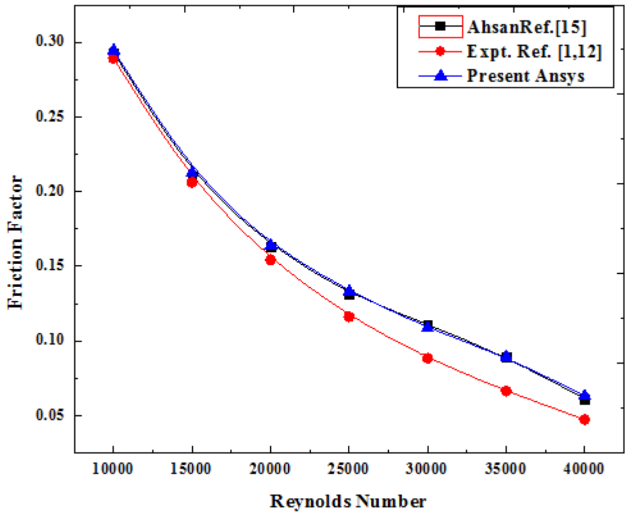

Fig. 3 Validation of Friction Factor with respect to varying Reynolds Number

Fig. 3-4 shows the validation of FEV result obtained from the ANSYS tool. It has been seen that the obtained result shows good agreement with the analalytical, Experimental and FEV of avalilable literature.

The small variation is results is due to variation in grid sizing, operating condition, material properites, etc. but the obtained result shows the same trend so that the results are suitably verified.

Fig. 7 to 16 shows the contour plot of an pipe which illustrates the flow pattern across the Straight smooth pipe and pipe with segemented baffles. On the basis of this thermo hydrodynamic characteristics various graphs has been ploted and dissued there significance in thermohydrodynamic performance has been stated.

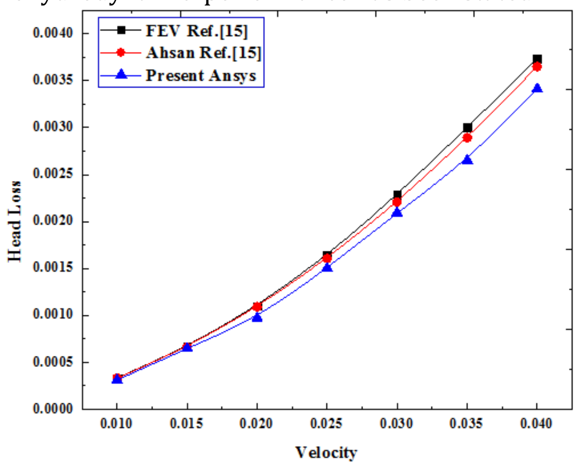

Fig. 4 Validation of Head loss with respect to varying

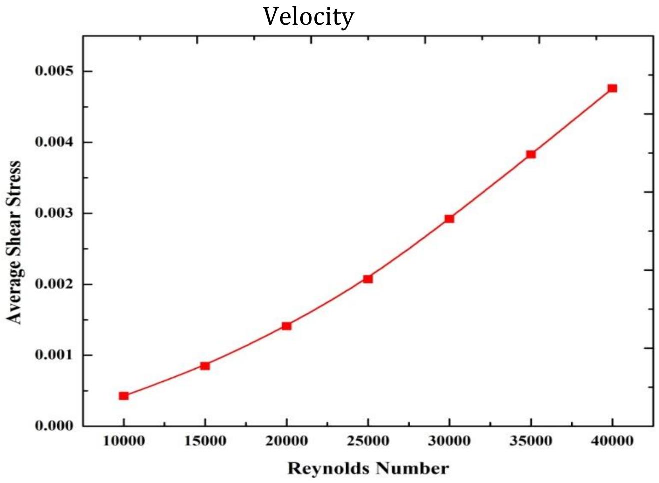

Fig. 5 Variation of Shear Stress with varying Reynolds Number

In Fig. 9 it can be shown that there is only changes in pressure in the axial direction. This represents that the pressure in the center of the pipe is equal to the pressure on the walls of the pipe. There is not an augmentation in the static pressure because the flow is not going during this stagnation process anywhere. It is worth noting that there is an increase in the dynamic and total pressures at the pipe center relative to the walls of the pipe due to higher velocity, however again, the static pressure is constant because the flow isn't going through a stagnation process.

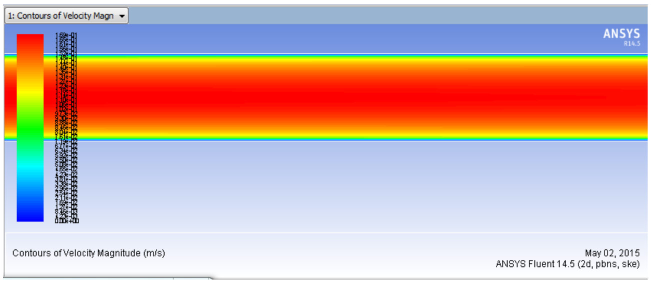

Fig. 6 Contour Plot of Velocity Magnitude of Smooth Pipe

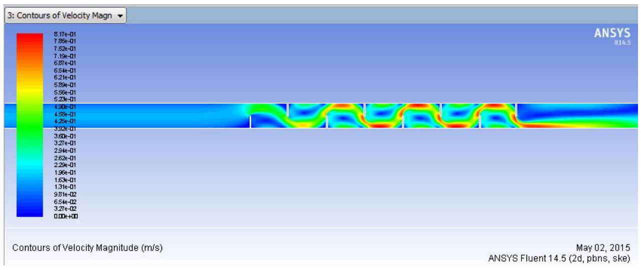

Fig. 7 Contour Plot of Velocity Magnitude of Segmented baffles Pipe

Due to pressure of baffels pressure drop significantly incrases as reynolds number increases. This baffles creates turbulence in fluid flow asvelocity raises and cause friction with leads to increases in friction factor. Moreover, increaes in friction factor correspongly affects the shear stress across the wall and generates heat and improve thermo hydrodynamic performace.

Thus it is widly been used in heat exchanger, where extra surface area has been alocated in form of baffles and at high reynolds number the influnce of baffles has been seen in improve nusselt number and heat transfer coefficient.

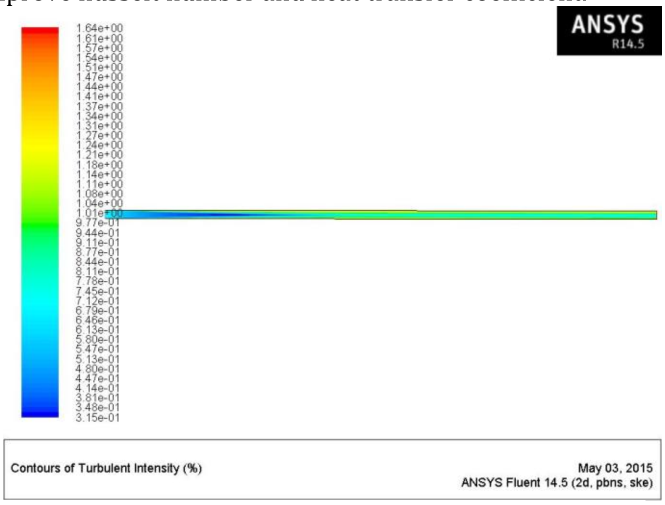

Fig. 8 Contour Plot of Turbulence Intensity of Smooth Pipe

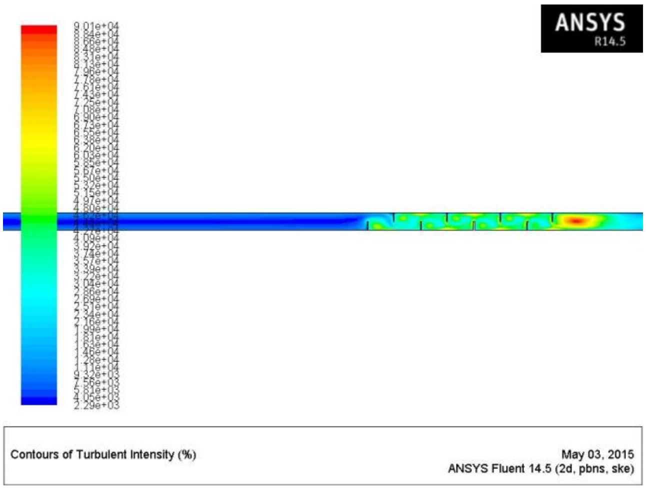

Fig. 9 Contour Plot of Turbulence Intensity of Segmented baffles Pipe

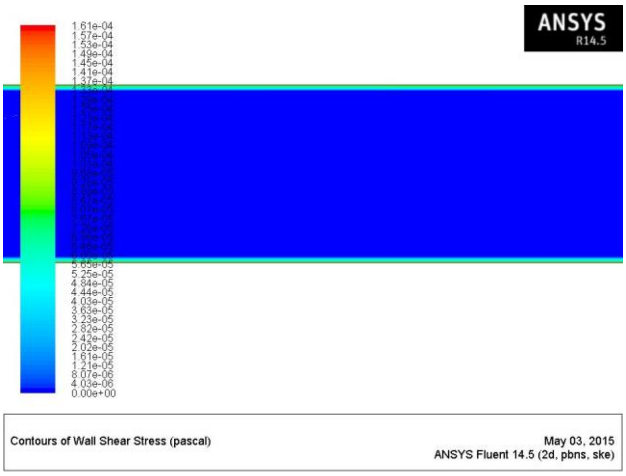

Fig. 10 Contour Plot of Wall Shear Stress of Smooth Pipe

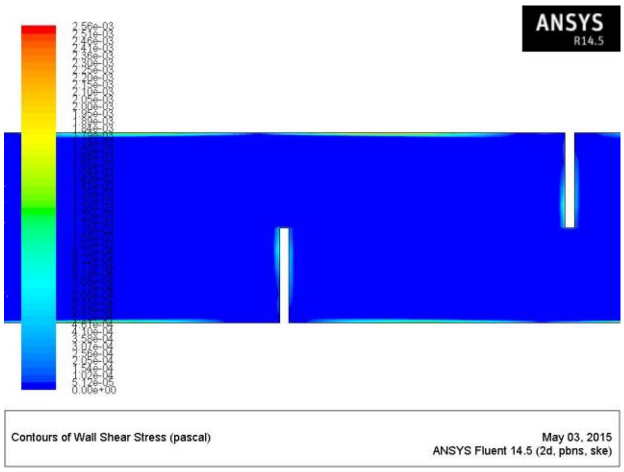

Fig. 11 Contour Plot of Wall Shear Stress of Segmented baffles Pipe

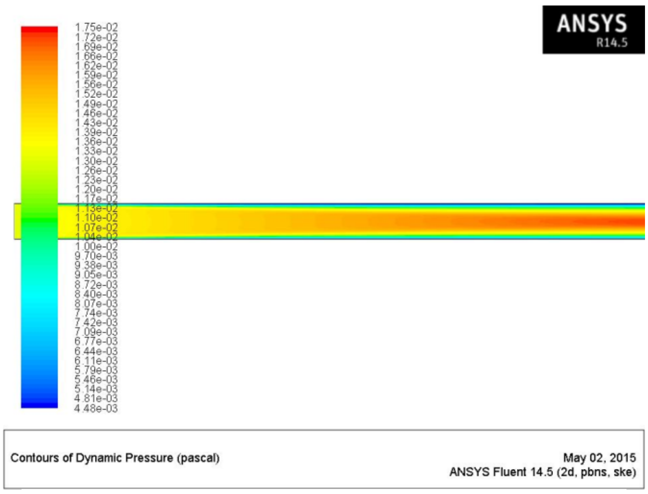

Fig. 12 Contour Plot of Dynamic Pressure for Smooth Pipe

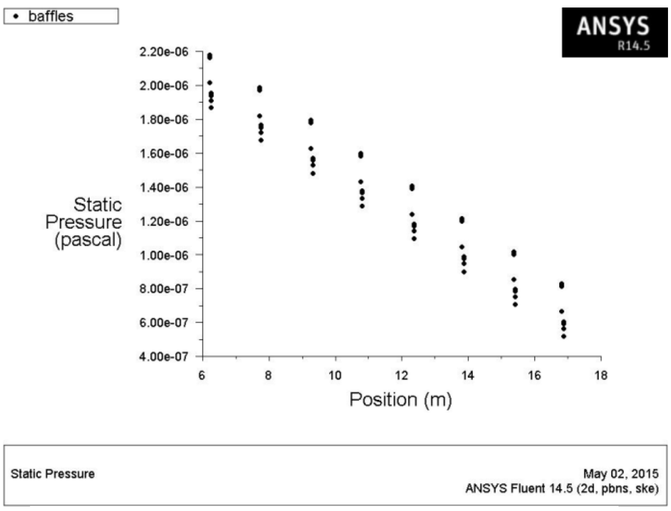

Fig. 13 Static Pressure distributions throughout the Segmented Baffles pipe

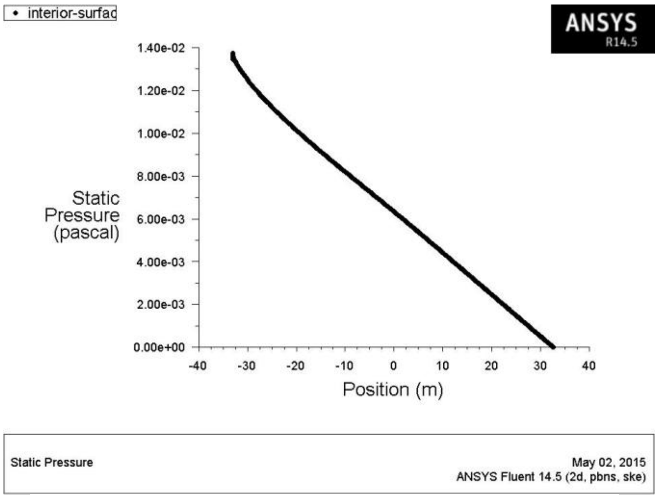

Fig. 14 Static Pressure distributions throughout the Smooth pipe

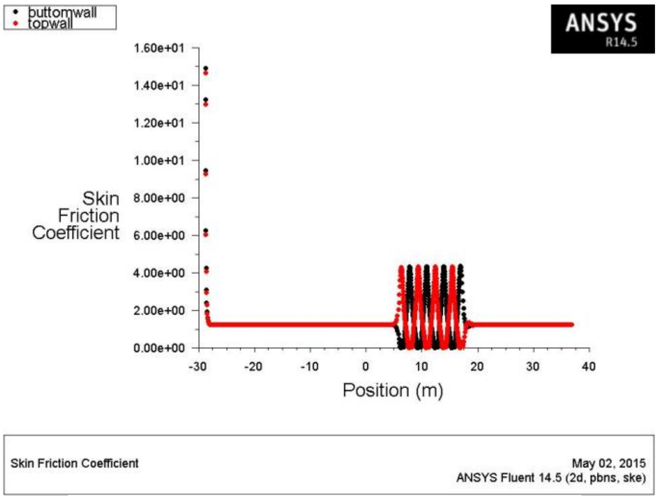

Fig. 15 Skin Friction Coefficient distributions throughout Segmented Baffles pipe

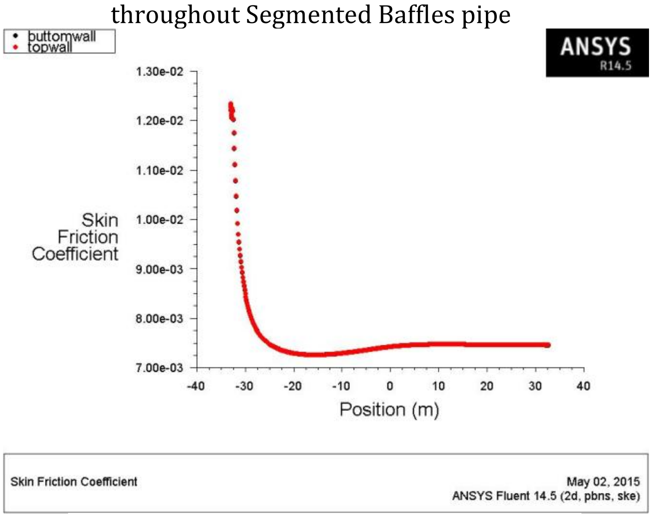

Fig. 16 Skin Friction Coefficient distributions throughout Smooth pipe

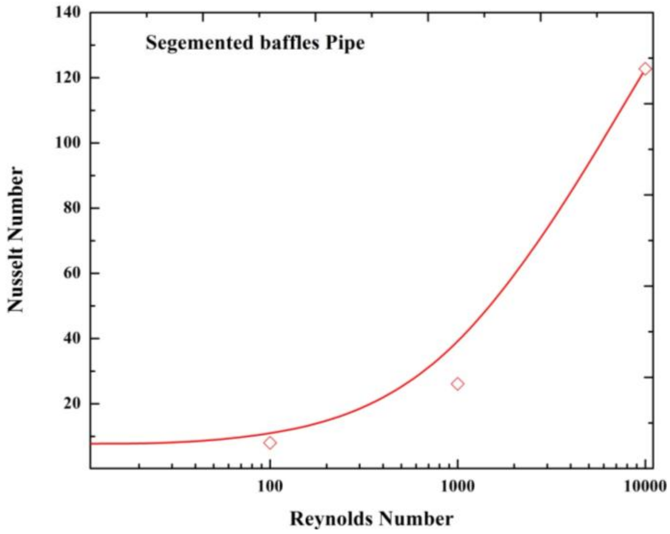

Fig. 17 Variation of Nusselt number with respect to Reynolds Number

Fig. 17 shows the variation of Nusselt number with respect to Reynolds Number. It has observed that on increasing Reynolds number nusselt number significantly increases. Nusselt number shows the ratio of convective to conductive heat transfer across (normal to) the boundary (baffles and wall within the pipe). In laminar flow the variation in nusselt is often quite less and very negligible here. Therefore, to gain advantage of better heat transfer turbulent flow is preferred and recommended.

Fig. 18 shows the variation of Stanton number with respect to Reynolds Number. It has observed that on increasing Reynolds number Stanton number significantly increases.

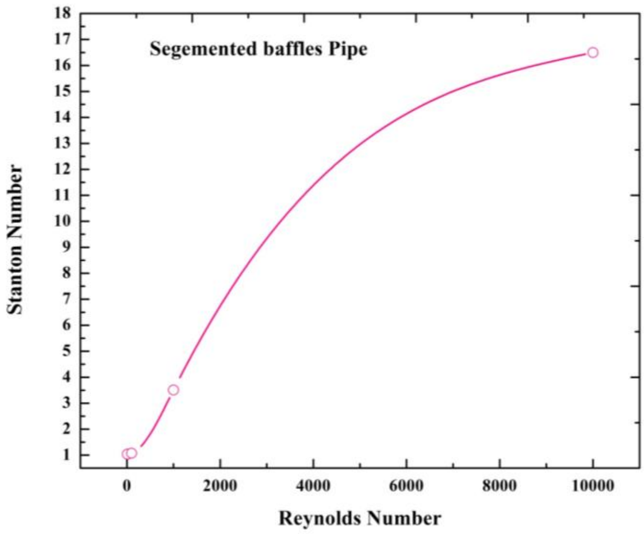

Fig. 18 Variation of Stanton number with respect to Reynolds Number

Stanton number shows the ratio of heat transferred into a fluid to the thermal capacity of the fluid and it also shows the relationship between the shear force at the baffles and wall (due to viscous drag) and total heat transfer at the wall (due to thermal diffusivity).

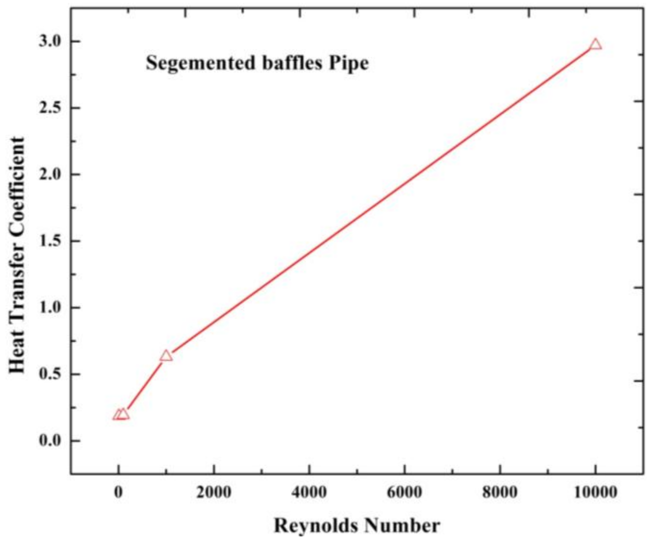

Fig. 19 Variation of Heat Transfer Coefficient with respect to Reynolds Number

Fig. 19 shows the Variation of Heat Transfer Coefficient with respect to Reynolds Number. It has observed that increasing flow rate heat transfer coefficient linearly increases. Due to wall shear, friction and extended surface in form of baffles are the other crucial features that amplify the heat transfer coefficient.

3. CONCLUSIONS

On integrating segmental baffles in pipes and ducts heat transfer and pressure significantly increases as compared with the straight pipe.

Turbulent intensity momentously increases as Reynolds number increases

Segmental Baffles increases turbulence with increases friction factor and plays crucial role in optimizing heat transfer.

Dynamic pressure for segmental Baffles is more as compared with smooth pipe or duct

On implementing Segmented Baffles in pipe pressure drop significantly increases in comparison with straight pipe.

Dynamic pressure goes increases as Reynolds number increases but on comparison with straight pipe dynamic pressure is more significant in baffled one.

Wall shear stress drastically enhances more in baffled as Reynolds number increases.

Heat transfer increases as reynold number increases.

REFERENCES

[1] Lewis F. Moody. Friction factors for pipe flow. Transactions of the A.S.M.E., pages 671–684, November 1944

[2] R. E. Kirmse, "Investigations of Pulsating Turbulent Pipe Flow", Journal of Fluids Engineering |1979 Volume 101 | Issue 4

[3] 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

[4] T.-M. Liou, C.-F. Kao "Experimental measurements of flow past double-sided wall obstacles in a rectangular duct", Experimental Thermal and Fluid Science, Volume 1, Issue 2, April 1988, Pages 135-146

[5] A. Pollard, A. M. Savill, H. Thomann, " Turbulent pipe flow manipulation: some experimental and computational results for single manipulator rings", Applied Scientific Research, July 1989, Volume 46, Issue 3, pp 281-290

[6] P. -A. Chevrin, H. L. Petrie, S. Deutsch, "The structure of Reynolds stress in the near-wall region of a fully developed turbulent pipe flow", Experiments in Fluids, October 1992, Volume 13, Issue 6, pp 405-413

[7] 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

[8] A. A. Pavelyev, A. I. Reshmin, S. Kh. Teplovodskii, S. G. Fedoseev, "On the Lower Critical Reynolds Number for Flow in a Circular Pipe", Fluid Dynamics, July 2003, Volume 38, Issue 4, pp 545-551

[9] S. Galinat, O. Masbernat, P. Guiraud, C. Dalmazzone, C. Noı¨k, "Drop break-up in turbulent pipe flow downstream of a restriction", Chemical Engineering Science, Volume 60, Issue 23, December 2005, Pages 6511-6528

[10] Mushtak T. Al-Atabi and S. B Chin, "Pressure Drop in Laminar and Turbulent Flows in Circular Pipe with Baffles – An Experimental and Analytical Study", International Journal of Fluid Mechanics Research, Vol. 33, No. 4, 2006

[11] 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

[12] Jim McGovern, "Technical Note: Friction Factor Diagrams for Pipe Flow, Dublin Institute of Technology ARROW@DIT

[13] 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

[14] N.M. Nouri, S. Yekani Motlagh, M. Navidbakhsh, M. Dalilhaghi, A.A. Moltani, "Bubble effect on pressure drop reduction in upward pipe flow", Experimental Thermal and Fluid Science, Volume 44, January 2013, Pages 592-598

[15] Muhammad Ahsan, "Numerical analysis of friction factor for a fully developed turbulent flow using keε turbulence model with enhanced wall treatment", b e n i - s u e f university journal of basic and applied sciences xxx (2014) 1e9

[16] A.T. van Nimwegen, K.C.J. Schutte, L.M. Portela"Direct Numerical Simulation of turbulent flow in pipes with an arbitrary roughness topography using a combined momentum–mass source immersed boundary method" , Computers & Fluids, Volume 108, 15 February 2015, Pages 92-106