Numerical Analysis of Turbulent Flow and Convection Heat Transfer

Nand Kishore Yadav[1] **Devesh Kumar**[2]
[1]M. Tech Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]M.M.M. University of Technology, Gorakhpur, India

Abstract— This paper describes about numerical analysis of turbulent flow in a tube, the variation of turbulent intensity, surface heat transfer coefficient and other parameter with Reynolds number is analyzed using standard k epsilon model of turbulent flow, it is assumed that tube is fully insulated, heat transfer parameters are hugely influenced by turbulence.

Keywords: Turbulence Models, Reynold Number, Heat Transfer, CFD, Numerical Analysis

I. INTRODUCTION

Convection heat transfer occurs in liquids and gases and it does not occur at all in solids, convection is the phenomena which occurs because of transport of energy and is associated with the transport property of fluids. the dimensionless number namely reynold number and grashoff number are highly significant in the concern of convection heat transfer, reynold number is highly important for analysis of forced convection while grashoff number is highly important for free convection., Free convection results in very low rate of heat transfer and if we want to increase the rate of cooling or heating of fluids then this can be achieved by the means of increasing mass flow rate of fluid creating artificial pressure difference or by the means of pump or blower this kind of convection is called forced convection.

$$Q_{\text{convection}} = hA\,(t_s - t_\infty)$$

h is called transport property of fluid,the heat transfer rate is more in turbulent flows as compared with laminar flows because of its more mixing rate (1); so the forced convection is practiced in industries because of higher transfer rate of heat as compared with free convection.

Mass flow rate of fluid plays vital role for convection heat transfer coefficient, specific heat of fluid conductivity of fluid, fluid density, fluid flow geometry and surface roughness are crucial factors for forced convection coefficient, in wall bounded flows, flow geometry governs the turbulence (2,3).

Temperature variation, pressure field, distribution of velocity and turbulence field are the chief parameters of hydraulic and thermal charecteristics of fluid flow, in turbulent analysis of wall bounded flow the turbulent parameters are greatly affected by wall functions (4,5).for investigation of these fields in fluid flow commercial cfd software ANSYS- FLUENT software was used.

II. METHODOLOGY

- Collection of geometrical data of simulation set up and cration of 2 D model of the tube carrying fluid and having heat and momentum transport in it using geometry cell of fluent module.
- Fixing the boundary conditions.
- Generation of grid using ICEM CFD.
- CFD solution of the fluid flow and heat transfer.
- Analysis using Ansys fluent
- Collecting flow field results

III. PROBLEM DESCRIPTION

A hollow tube of aluminium of length 200 mm and 32 mm inner diameter and 37 mm outer diameter is carrying flow of air with flow rate ranging from 0.912 m/s to 7.296 m/s corresponding to local Reynold number 2000 to 16000 at entrance of tube and the analysis of surface heat transfer coefficient, friction factor, average turbulent kinetic energy, average turbulent intensity, average turbulent viscosity and average turbulent viscosity ratio is to be performed.

A. *Solution Methodology*

The CFD solution of the above problem is to make 2 d geometry of the tube carrying the fluid and make computational domain by the means of meshing the geometry of the fluid carrier.

B. *Computational Domain of the Problem*

The above computational domain is consist of 32098 elements and 16892 nodes having triangular mesh elements.

C. *Material Properties and Boundary Conditions*

1) *Material:*
FLUID: Air
CASING MATERIAL: Aluminium
2) *Boundary Conditions:*
INLET:
Velocity: 0.912 m/s to 7.296 m/s
Turbulent intensity: 5 %
Hydraulic diameter: .032 m
OUTLET:
Pressure: 1 atm. (gauge pressure =0)
WALL:
Temperature: constant wall temperature of 400 kelvin
3) *Solution Method:*
SCHEME: Simple
SPATIAL DISCRETIZATION: Green gauss cell based
PRESSURE: Second order
MOMENTUM: Second order upwind
TURBULENT KINETIC ENERGY: Second order upwind
TURBULENT DISSIPATION RATE: Second order upwind

IV. RESULT AND DISCUSSION

This study entails numerical analysis of fluid flow and heat transfer charecteristics of forced convection (focusing over turbulent intensity in flow) employing standard k epsilon model.
Figure below represents the results generated by FLUENT.

A. Turbulent intensity

Turbulent intensity in flow in circular tube is being shown below for different reynold number (2000,6000, 10000, 16000) at inlet.

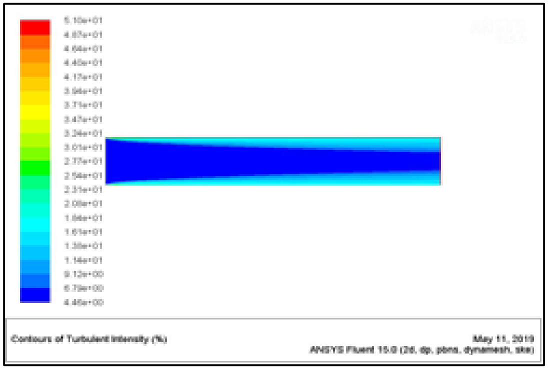

Fig. 1: Turbulent intensity distribution in tube for reynold number 2000 at inlet

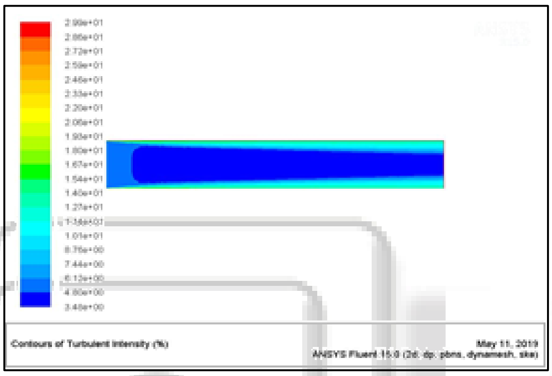

Fig. 2: Turbulent intensity distribution in tube for reynold number 6000 at inlet

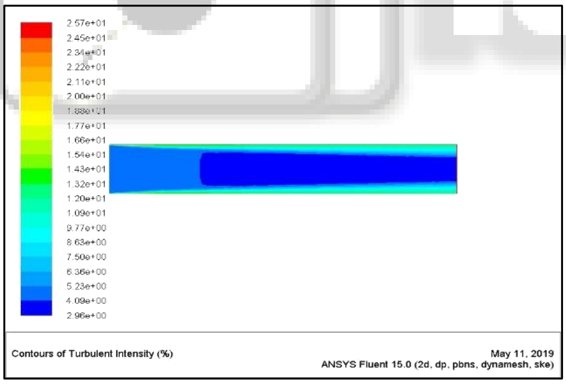

Fig. 3: Turbulent intensity distribution in tube for reynold number 6000 at inlet

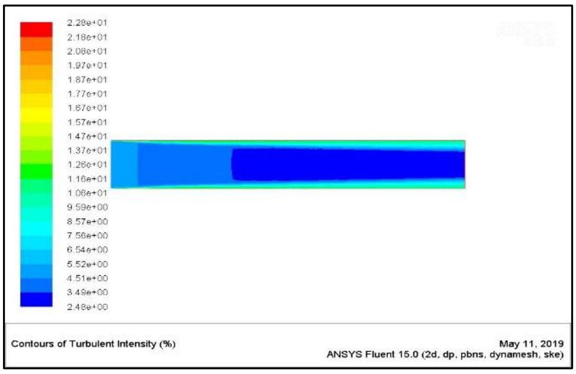

Fig. 4: Turbulent intensity distribution in tube for reynold number 16000 at inlet

The turbulent intensity is the ratio of variation of fluid velocity with mean velocity and mmean velocity. every point in a fluid flow domain posses a unique value of turbulent intensity. the distribution of turbulent intensity in fluid domain is presented in the above figures. the turbulent intensity decreases as at every cross section of fluid domain when the reynold number increases.

B. Graphical Representation of Turbulent Intensity

The variation of turbulent intensity with y coordinate in fluid domain at x= 100 mm is being shown in the below figure at different reynold number(2000,6000,10000,16000)

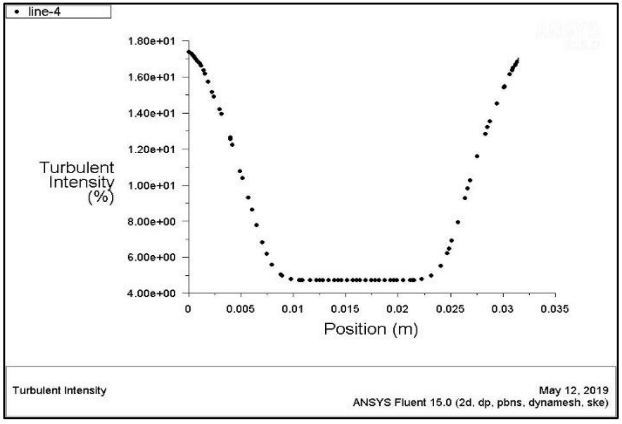

Fig. 5: variation of turbulent intensity with abscissa at X/L= 0.5 for reynold number 2000 at velocity inlet

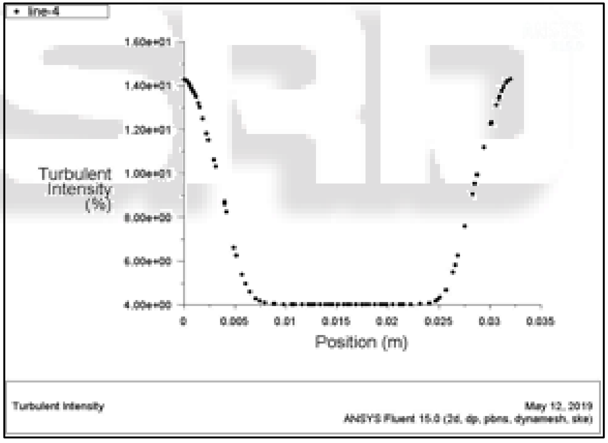

Fig. 6: variation of turbulent intensity with abscissa at X/L= 0.5 for reynold number 6000 at velocity inlet

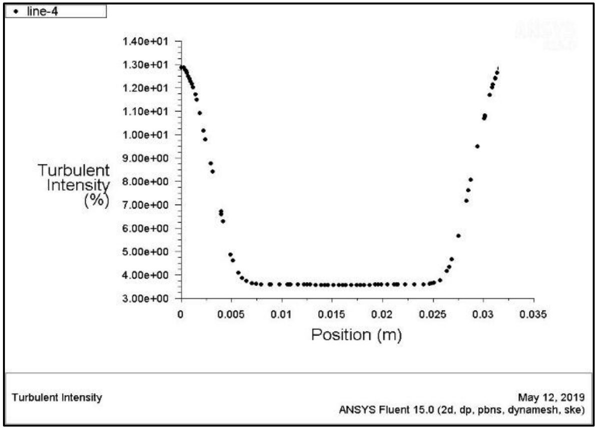

Fig. 7: variation of turbulent intensity with abscissa at X/L= 0.5 for reynold number 10000 at velocity inlet

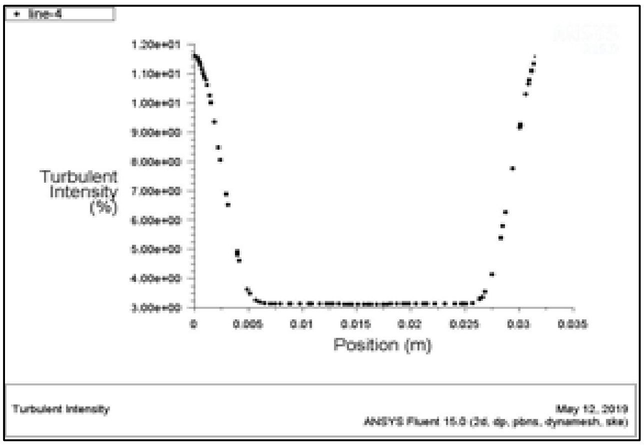

Fig. 8: variation of turbulent intensity with abscissa at X/L= 0.5 for reynold number 16000 at velocity inlet

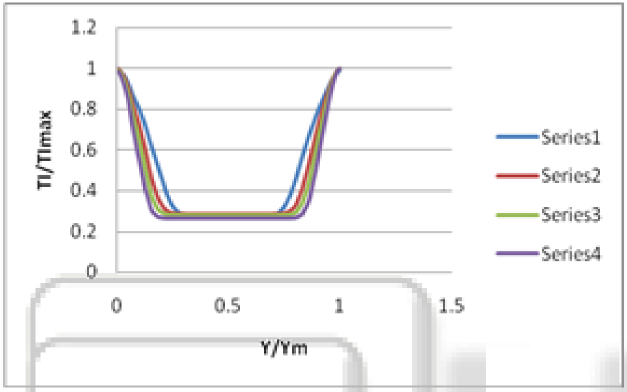

Fig. 9: Non dimensionalized graph between nondimensionalized turbulent intensity and radial length Y/Ym for reynold number 2000,6000,10000 and 16000 (series 1,2,3 and 4 respectively) at X/L =0.5

One may think that turbulent intensity increases with increase of reynold number pertaining to flow, however the results are absolutely opposite. turbulent intensities are higher at wall of the tubes and decreases toward the centreof the tube.

C. Impact of Reynold Number over Turbulent Intensity in Tube Flow

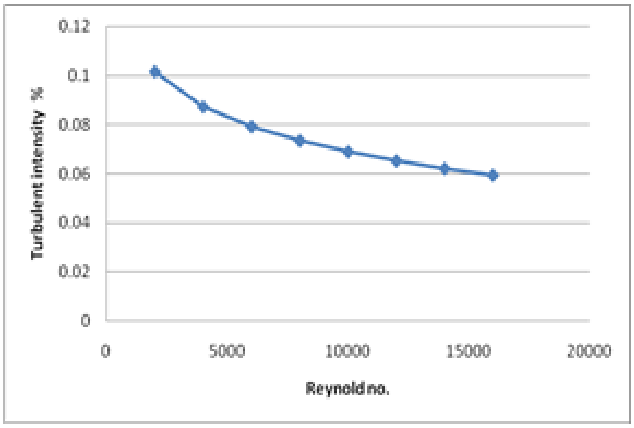

Fig. 10: variation of turbulent intensity with reynold number at X/L= 0.5

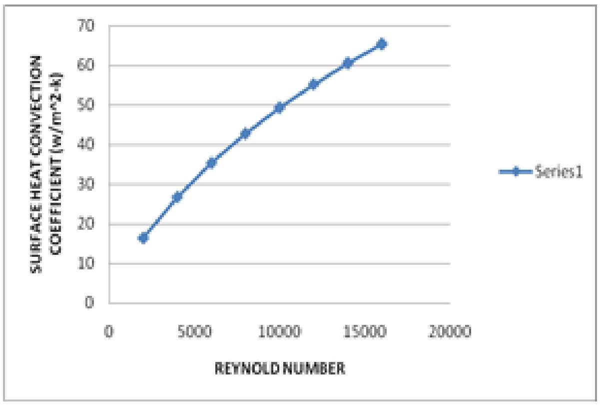

Fig. 11: variation of surface heat transfer coefficient v/s reynold number

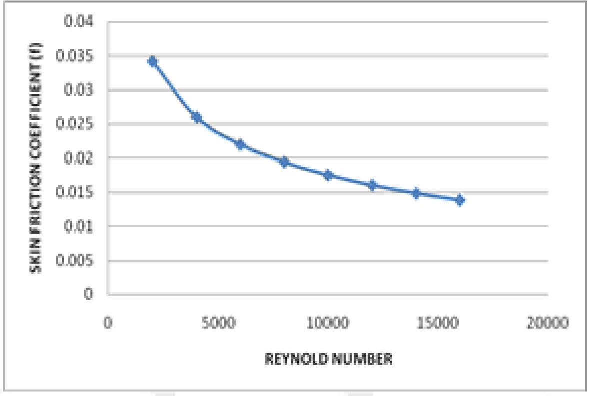

Fig. 12: variation of skin friction coefficient v/s reynold number

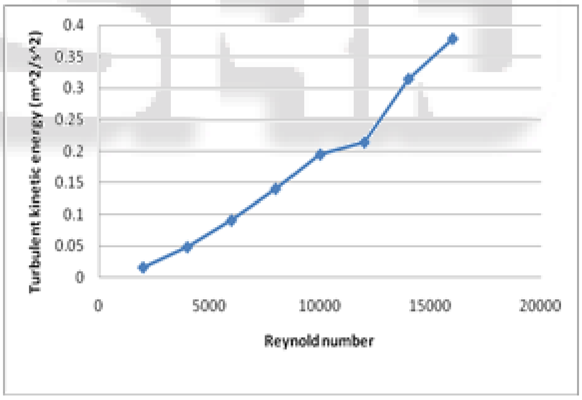

Fig. 13: variation of Turbulent kinetic energy v/s reynold number

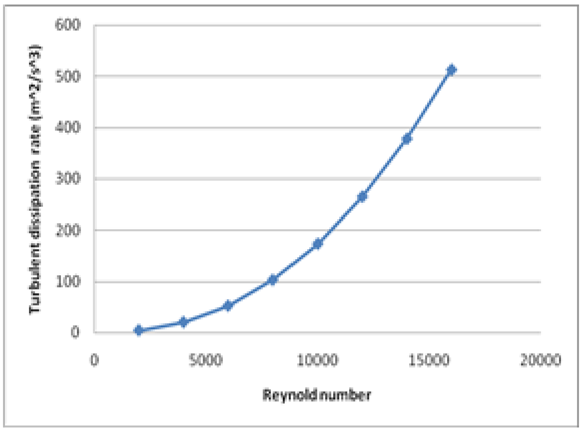

Fig. 14: variation of Turbulent dissipation rate v/s reynold number

V. CONCLUSION

This work focuses to investigate the thermo hydrodynamic charecteristics in turbulent forced convection flows. It is found that,

1) ratio of fluctuation of velocity of fluid particles with average velocity(turbulent intensity) decreases as the reynold number increases, the relation of turbulent intensity with reynold number also speaks the same,there may be a doubt that increase of reynold number increases the turbulence intensity.
2) convection coefficient increases as expected with increase of reynold number or in other word when the velocity of fluid flowing over the surface increases, convection coefficient increases.
3) increase in turbulent kinetic energy and increase in turbulent dissipation rate causes molecular friction (viscous dissipation) which results in increase in convection coefficient.
4) The above results are provided by cfd code (ansys-fluent)which solves the mass conservation,momentum and energy conservation equation starting with initial guess of parameters at every cell.

REFERENCES

[1] T.Kobayashi,M. Yoda,Modified k –e model for swirling flow in a straight pipe,JSME Int.J. 30 (259) (1987) 66-71

[2] O. Kitoh, Experimental study of turbulent swirling flow in a straight pipe, J.Fluid Mech.225 (4) (1991) 445-479

[3] H. Li, Y. Tomita, Charecterisics of swirling flow in a circular pipe, J. Fluids Eng.Trans. ASME 116 (1994) 370-373

[4] C.M.Hrenya, E. J Bolio, D.Chakrabarti,J.L Sinclair, Comparision of low Reynold number k-e turbulence models in predicting fully developed pipe flow, Chem. Eng.Sci. 50 (12) (1995) 1923-1941.

[5] S.Vijiapurapu, J.CUI,SIMULATION OF turbulent flow in ducts with symmetric and axisymmetric rough walls. Chem Eng J 74 (3) (1999)147-153