Enhancement of Heat Transfer in a Circular and Non-Circular Duct using CFD Simulation

Dundayya Hiremath[1] Basanagouda B. Patil[2]
[1]M. Tech Student [2]Assistant Professor
[1,2]Department of Thermal Power Engineering
[1,2]The Oxford College of Engineering, Bangalore

Abstract— Analytical study of forced convection phenomenon with turbulent flow is complex. The aim of the project is to enhance heat transfer by optimizing the design of domain. In this study forced airflow for heat input will be carried. Naturally convection depends on fluid parameters and geometry of the domain through which fluid flows. The aim of our project is to find the value of heat transfer coefficient 'h' for turbulent flow in heat transfer systems. In convection important parameter is, heat transfer coefficient 'h' because, it determines rate of heat transfer. A study of literature on heat transfer coefficients shown very little work carried out for different non circular duct. This project uses analysis of flow both for circular and non-circular duct based experimental results to determine heat transfer coefficient, and comparison of heat transfer coefficient for circular, ellipse, and triangular duct will be carried out. For the enhancement of heat transfer of circular, ellipse, and triangular duct passive method is used, and the duct which gives maximum heat transfer coefficient is be optimized. For modelling and meshing ICEM-CFD has been used. For analysis CFX has been use and for results CFD-POST has been used.

Key words: Convection, heat transfer coefficient ICEM-CFD, CFX, CFD-POST

I. INTRODUCTION

It is important to have understanding of the characteristics of the forced convective heat transfer in turbulent Newtonian flow through circular pipe and non-circular ducts in order to exercise proper control over the performance of heat exchanger. Forced convection heat transfer of Newtonian and non– Newtonian fluids through ducts have been the subject of several studies from the past, because of the extensive range of uses such as heat exchangers and petrochemical industries, that are commonly used that includes condenser and boiler in petrochemical and steam power plants. Forced convection heat transfer through ducts involves different aspect of problems. This variety of problems comes from possibly geometry characteristic of ducts, nature of fluid flow, kind of fluid, etc. In this work, numerical study is performed to analyse the turbulent forced convective heat transfer of Newtonian fluids. Convective heat transfer is the conduction of heat into a moving fluid. An internal flow, such as a flow in a pipe, is one for which the fluid is restricted by a surface. Hence the boundary layer is incapable to develop without eventually being constrained. In this work, heat transfer is investigated experimentally for circular ducts and through CFD for non-circular ducts. Further heat transfer characteristics are evaluated by dimensional analysis and mathematical formulation under different thermal boundary conditions. Some researchers have proposed important correlations for laminar, transition and turbulent flow in plain tube. Sieder and Tate (1936) studied heat transfer and pressure drop of liquid in tube. `Hausen reported heat and pressure drop studies in circular tube for transition flow using water as working fluid. Dittus and Boelter suggested an empirical correlation for heat transfer in fully developed turbulent flow in smooth tubes.

II. LITERATURE SURVEY

Many authors studied different methods of heat transfer from the past years. Forced convection heat transfer has many application like air-condition, refrigeration system, process industry, oil and gas industry, petrochemical industry etc. Forced convection heat transfer coefficient is important parameter which determines rate of heat transfer. There are many techniques available for enhancement of heat transfer like use of inserts, screw tapes, use of liquids, increase in velocity, surface modification etc. Since convection heat transfer coefficient is important parameter, will study the research work of different authors and their conclusions. Authors like Date, Hussein, Qaiser, S.K.Saha,etc

Date, A. W [1] he has formulated and solved numerically the problem of fully developed, uniform property flow in a tube containing a twisted tape. Steele and Coleman, [2] they have noted that the uncertainties associated with the experimental data are calculated on the basis 95% confidence level. Krishpersad & Kimberly, [3] they have worked out on correlation of heat transfer coefficients for an external flow at different velocity. Saha, [4] He has investigated pressure drop and heat transfer on laminar flow of viscous fluid through horizontal tube under a uniform wall heat flux conditions, tube fitted with regularly spaced twisted tapes..Sundar, L.S, [5] all have reported enhancements with Al_2O_3 nano-fluid and twisted tape insert in a circular tube subjected to constant heat flux boundary condition in a turbulent range. A maximum enhancement of 28% has been observed when flowing with nano fluid with tape insert when compared with water flowing in the plain tube at the same mass flow rate. Qaiser M, [6] They have presented the work about designing and acquiring data from an experimental setup to verify the Dittus-Boelter empirical relation by finding the heat transfer coefficient. Patil S.V & Vijay P V, they have shown the heat transfer coefficient increases with insertion of twisted tape in a square duct.

III. METHODOLOGY

A. *Finite Volume Method*

In Finite Volume formulation, computations are carried out in the physics flow domain. Computational domain is divided into network of finite volumes or cells. The main advantage of FVM is its flexibility in treating arbitrary geometries efficiently. Nowadays it has become very popular for 2-D and 3-D flow computation. In this approach governing equations are considered in their integral forms.

IV. COMPUTING PLAT FORM

All commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine results provide easy access to their solving power. Hence all CFD codes contain three main elements:

- Pre-processor
- Solver
- Post processor

V. EXPERIMENTAL SET UP OF FORCED CONVECTION

Fig. 1: Forced Convection Apparatus

Experimental setup of forced convection consists of blower, orifice, mercury manometer, dimmerstat, test pipe (copper) and thermocouples. There are 7 thermocouples mounted on the test pipe, which indicate air and surface temperature. Orifice is connected to the entry section of the pipe, to measure the flow of air. Valves used to control the flow of air. Dimmerstat is used control the heat input. Mercury manometer indicates the difference of pressure head between inlet and outlet of orifice. Voltmeter and ammeter indicates the voltage and current. Test pipe is connected to the deliver side of the blower.

Specimen	Copper tube.
Size of specimen	I D 25mm*300mm long
Centrifugal blower	single phase, 230V, 50hz, 3000 RPM
Manometer	U-Tube with mercury as working fluid
Orifice dia	20mm
G.I pipe dia	40mm I D and 1m long
Ammeter	Digital type 0 to 20 amps
Voltmeter	Digital type 0 to 300 volts
Heater	Externally heated, Nichrome wire Band
Dimmerstat for heating coil	0 - 230 V, 2 amps
Thermocouple Used	7 no

Table.1 Specification

Sl No	Heat input (watts)			Difference in manometer readings (mm)	Temperature readings (0C)						
	Volts (V)	Amps (I)	Q=V*I	hm	T1	T7	T2	T3	T4	T5	T6
1	80	0.31	24.8	15	42.1	47.5	46.2	45.7	47.5	47.7	41.5

Table 2: Experimental Readings

A. *Calculations (circular model)*

1) *Calculation of heat transfer coefficient by Experimental results:*

Heat input (Q)

$Q = V*I$ (1.1)

$= 80*0.31$

$Q = 24.8$ watts.

Forced convective heat transfer coefficient:

$Q = h*A*dt$ (1.2)

$= h*(\pi*D*L)*(T_s-T_\infty)$

$24.8 = h*(\pi*0.025*0.3)*(45.11-32)$

$h = 80.28\ w/m^2K$.

2) *Calculation of heat transfer coefficient by Correlation method:*

Average surface temperature: (T_s)

$T_s= T_1+T_2+T_3+T_4+T_5+T_6$

$= (42.1+46.2+45.7+47.5+47.7+41.5)/6$

$T_s = 45.11^0$

Bulk mean temperature: (T_{Bulk})

$T_{Bulk} = (T_{in} + T_{out}) /2$ (1.3)

$= (32 + 47.5) / 2$

$T_{Bulk}= 39.75^0$

3) *Properties of air (from data hand book) at bulk mean temperature*

Kinematic viscosity (ν) $= 16.936*10^{-6}\ m^2/s$

Prandtle number (P_r) $= 0.6698$

Thermal conductivity (K) $= 0.02754$ w/m K

Discharge at the orifice :

$Q = Cd*A*\sqrt{(2*g*h_a)}$ (1.4)

$= 0.62*\pi/4*0.02^2*\sqrt{(2*9.81*176.31)}$

$Q = 0.01145\ m^3/s$

Velocity of air:

$Q = A*V$ (1.5)

$0.01145 = \pi/4*0.025^2*V$

$V = 23.32$ m/s

Reynolds Number (Re):

$Re = (V*D)/\nu$ (1.6)

$= (23.32*0.025)/ 16.936*10^{-6}$

$Re = 34423.71$

For fully developed flow, DittusBoelter equation is (from data hand book)

$Nu = 0.023*Re^{0.8} * P_r^{0.4}$ (1.7)

$= 0.023*34423.71^{0.8} * 0.6698^{0.4}$

$Nu = 83.48$

$Nu = (h*D)/K$ (1.8)

$83.48 = (h*0.025)/0.02754$

$h = 91.96\ w/m^2\ K$

From correlation we got heat transfer coefficient value as $91.96\ w/m^2K$. This is almost near to the theoretical value, which is $80.28\ w/m^2\ K$.

VI. VALIDATION OF CORRELATION BY CFD SIMULATION

Boundary conditions

At Inlet:

Boundary details

Heat flux = $1052\ w/m^2$

Temperature = 305 K

Solid values

Heat flux= $1052\ w/m^2$.

Copper

At Outlet:

Solid value
Copper
Heat flux = 1052 w/m^2.
At Wall:
Boundary wall = no slip wall
Roughness = smooth wall
Heat flux = 1052 w/m^2

A. Model 1 Circular pipe with smooth surface

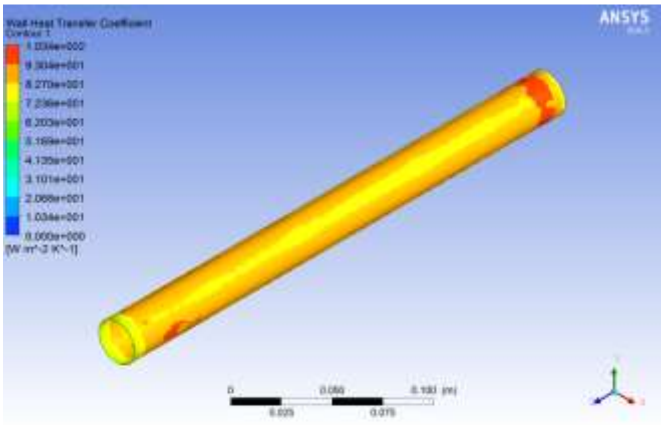

Fig. 2: Circular pipe with smooth surface

It is seen from the figure, wall heat transfer coefficient obtained is 85W/m^2 k.

Wall heat transfer coefficient obtained from cfd is 85 w/m^2 K. It is nearer to the values obtained by correlation and theoretical. i.e. 91.96 w/m^{2}K and 80.28 w/m^2 K. Hence Dittus-Boleter correlation is validated through CFD.

B. Model 2: Circular pipe with surface roughness

Enhancement of heat transfer coefficient by passive method for circular pipe.

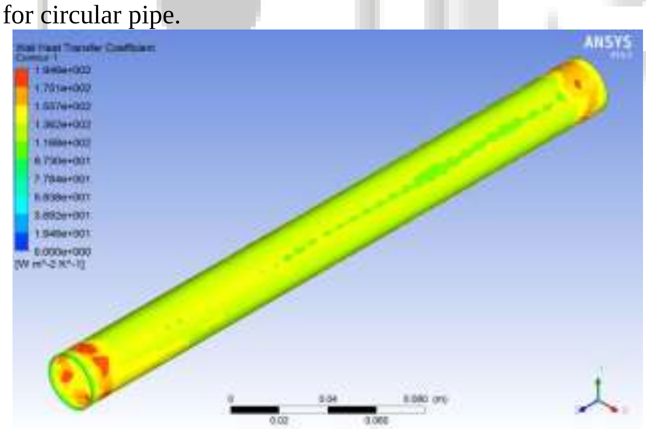

Fig. 3: Circular pipe with rough surface

It is seen from the figure, wall heat transfer coefficient obtained is 120 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient with surface roughness is more, compared to with smooth surface.

C. Model 3 Triangular duct with smooth surface

After circular pipe geometry is changed to triangular duct, to check the heat transfer rate. Model is created and meshed in ICEM-CFD and simulated in CFX with same boundary condition, which are taken form experiment.

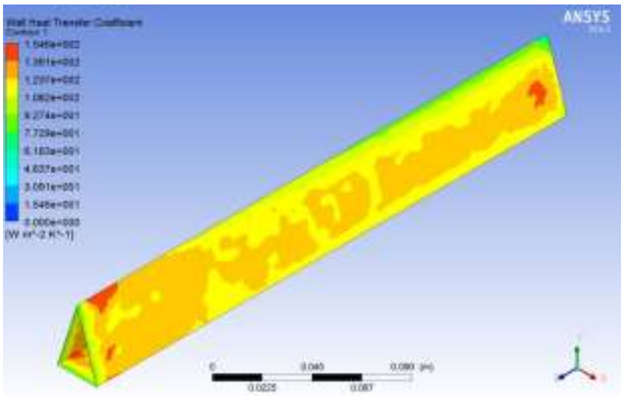

Fig. 4: Triangular duct with smooth surface

It is seen from the figure, wall heat transfer coefficient obtained is 130 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient of triangular duct is more, compared with circular duct.

D. Model 4 Triangular duct with surface roughness

Enhancement of heat transfer coefficient by passive method for Triangular duct.

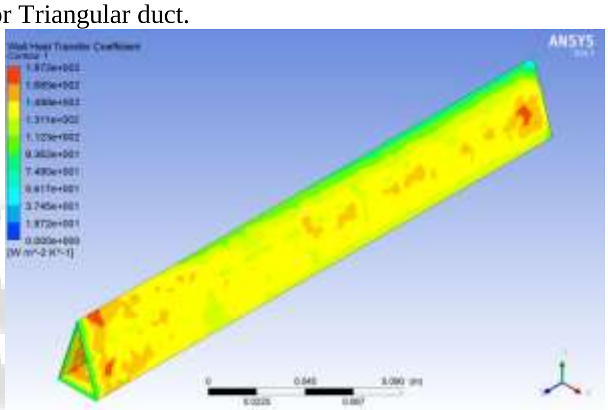

Fig. 5: Triangular duct with rough surface

It is seen from the figure, wall heat transfer coefficient obtained is 135 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient of triangular duct with surface roughness is more, compared with triangular duct with smooth surface.

E. Model 5 Elliptical Duct

Geometry of circular pipe is again changed to ellipse, to check the heat transfer rate. Model is created and meshed in ICEM-CFD and simulated in CFX with same boundary condition, which are taken form experiment.

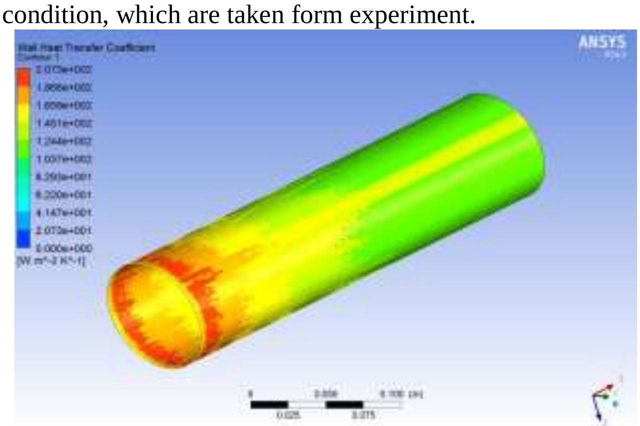

Fig. 6: Elliptical duct with smooth surface

It is seen from the figure, wall heat transfer coefficient obtained is 145 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient of elliptical duct is more, compared with circular and triangular duct, due to more surface area.

F. Model 6 Elliptical duct with surface roughness

Enhancement of heat transfer coefficient by passive method for elliptical duct.

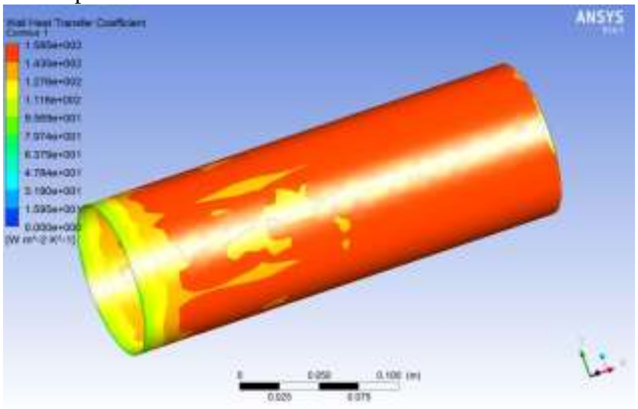

Fig. 7: Elliptical duct with rough surface

It is seen from the figure, wall heat transfer coefficient obtained is 155 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient with surface roughness is more, compared with smooth surface.

G. Model 7 Optimization of Geometry

Since the elliptical duct gives maximum heat transfer coefficient, compared to circular and triangular duct, the geometry is optimized with passive method. For optimization of geometry, threads are inserted in the elliptical duct and analysis is done.

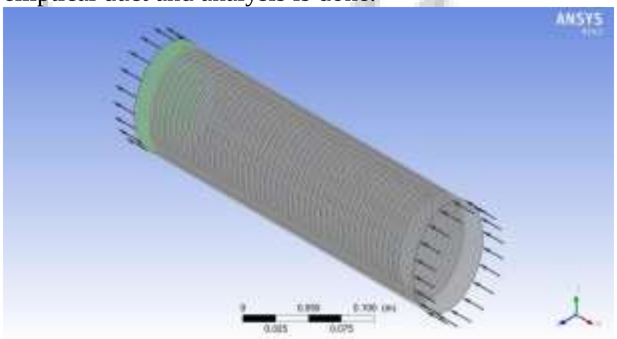

Fig. 8: Elliptical duct with thread inserted

The above figure shows the elliptical model with thread inserted. Arrows indicates the application of boundary condition. Boundary conditions are taken form experiment.

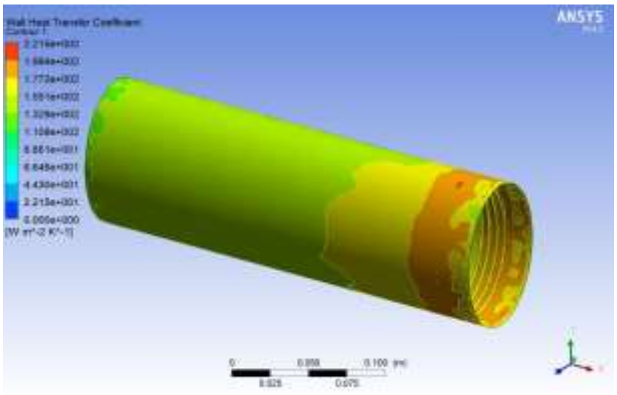

Fig. 9: Elliptical duct with smooth surface

It is seen from the figure, wall heat transfer coefficient obtained is 165 W/m^2 k.

This value is obtained according to colour coding.

It is observed that heat transfer coefficient with thread inserted is more, compared with smooth and rough surface due to the more surface area.

VII. Results and Discussions

The wall heat transfer coefficient was studied for circular, circular roughness, triangle, ellipse, ellipse with roughness and ellipse with thread insertion.

The results of all geometries consolidate in below table.

Sl No	Heat transfer coefficient of circular pipe with 0^0 inclination (w/M^2.K)	Heat transfer coefficient of circular pipe ROUGHNESS (w/M^2.K)	Heat transfer coefficient of TRINGULAR DUCT (w/M^2.K)	Heat transfer coefficient of TRINULAR DUCT WITH ROUGHNESS (w/M^2.K)	Heat transfer coefficient of ELLIPSE (w/M^2.K)	Heat transfer coefficient of ELLIPSE WITH ROUGHNESS (w/M^2.K)	Heat Transfer Coefficient of ELLIPSE THREADED (w/M^2.K)
1	85	120	130	135	145	155	165

Table 3: Heat transfer coefficients

Values in the above table shows wall heat transfer coefficient for different geometry of ducts.

It is seen form the values that, elliptical duct with thread inserted gives maximum heat transfer coefficient, for same heat input and boundary conditions. The reason is more surface area of contact between air and heated surface.

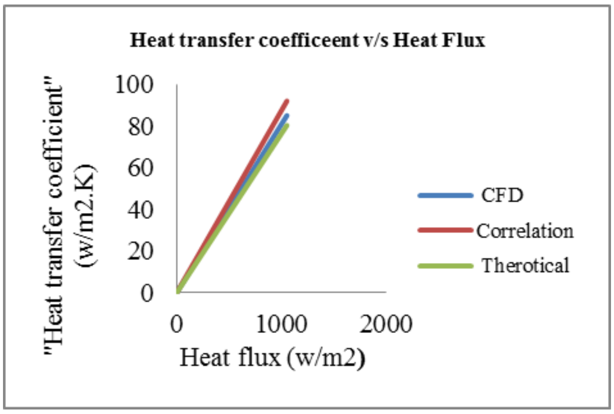

Fig. 10: Graph of Heat transfer coefficient v/s Heat Flux

From the above figure it is clear that CFD value(85 w/m^{2}K) is nearer to the theoretical and correlations values (i.e.80.28 and 91.96 w/m^{2}K).

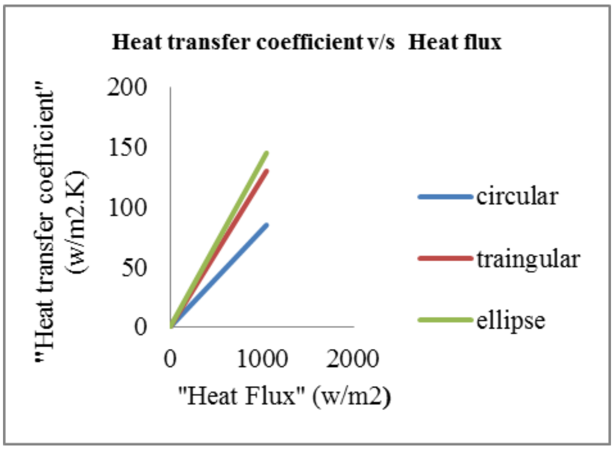

Fig.11: Graph of Heat transfer coefficient v/s Heat Flux of circular, triangular and ellipse

Above figure shows that wall heat transfer coefficient is more for elliptical duct compared to triangular and circular duct.

VIII. CONCLUSIONS

The following conclusions can be drawn from the analysis:

1) It has been observed that irrespective of inclination, the heat transfer coefficient depends on geometry of the domain and interaction of fluid with the surface.
2) Heat transfer depends on heat flux also.
3) Heat transfer enhancement has been achieved by passive technique (wall roughness, thread insertion).
4) Clearly the boundary layer formation has been observed more in triangular domain which was responsible for heat transfer enhancement.
5) Due to boundary layer phenomenon pressure is more towards wall side.
6) Maximum wall heat transfer coefficient is obtained for elliptical tube.
7) Finally depending on the wall heat transfer coefficient geometry has been optimized by insertion of thread.

REFERENCES

[1] Date, A. W. (1974). “Prediction of a fully developed flow in a tube containing twisted tapes”,Int. J. Heat Mass Transfer,17, 845–859.

[2] Steele,W.G.,Coleman,H.W.(1989).”Experimental and Uncertainty Analysis for Engineers” 1st edition, Wiley, New York.

[3] Dr.Krishpersad Manohar& Kimberly Ramroop.”A Comparison of Correlations for Heat Transfer from Inclined Pipes” International Journal of Engineering (IJE), Volume: 4, Issue: 4,2001

[4] Saha, S. K., and Mallick, D. N. (2005). “Heat Transfer and Pressure Drop Characteristics of Laminar Flow in Rectangular and Square Plain Ducts and Ducts With Twisted-Tape Inserts”, Trans. ASME, J. Heat Transfer, 127, 966-977.

[5] Sundar, L. S., Sharma, K. V., and Ramanathan, S. (2007). “Experimental investigation of Heat Transfer Enhancements with Al2O3 Nanofluid". World Academy of Science, Engineering and Technology. Vol: 6 2012-08-24.

[6] Qaiser Abbas, M. Mahabat Khan, Rizwan Sabir, Yasir Mehmood Khan,Zafar Ullah Koreshi. “Numerical Simulation and Experimental Verification of Air Flow through a Heated Pipe”. International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol:10 No:02 2010

