CFD SIMULATION AND EXPERIMENTAL VERIFICATION OF AIR FLOW THROUGH HEATED PIPE

Lohith .A.G,[1] Dr. K. V. Sreenivasa Rao[2] & H.V.Harish[3]

[1]I.T.Analyst, Tata Consultancy Services, Bangalore.

[2]Professor & PG Coordinator, Department of Mechanical Engineering, Siddaganga Institute of Technology, Tumkur.

[3] Assistant Professor, Department of Mechanical Engineering, Raja Reddy Institute of Technology, Bangalore.

E-mail: lohithag@gmail.com

Abstract:

The phenomenon of forced convection with turbulent flow is complex and hard to develop analytically. The problem can be solved through experimentally and CFD simulation. Here we are evaluating the value of heat transfer coefficient by experimentally for 3 different heat inputs (Q=VI). Further at Tmean temperature (obtained experimentally) we are calculating the heat transfer coefficient by using correlation (Dittus-Boelter equation).

Finally by CFD simulation for three different heat inputs we are finding the value of heat transfer coefficient which will give the value nearer to experimental and correlation method.

The aim of our project is to validate the Dittus-Boelter equation, a correlation used to find the value of heat transfer coefficient 'h' for turbulent flow in many fluid transfer systems. Heat transfer coefficient 'h' is very important parameter because they determine rate of heat transfer .This project uses three methods to determine these parameters: Correlations, CFD simulation and experiment. This project will discuss how the Dittus-Boelter equation is applied to the problem of circular pipe. In CFD simulation we are using ICEM CFD for modeling and CFX13 for analysis. Results of CFD simulation will be obtained by CFD-POST. Finally, we are comparing heat transfer coefficient value by correlations, experiments and CFD simulations.

1. INTRODUCTION

In the study of thermodynamics the average heat transfer coefficient, ***h***, is used in calculating the convection heat transfer between a moving fluid and a solid. This is the single most important factor for evaluating convective heat loss or gain. Knowledge of ***h*** is necessary for heat transfer design and calculation and is widely used in manufacturing processes, oil and gas flow processes, air-conditioning and refrigeration systems. The heat transfer coefficient is critical for designing and developing better flow process control resulting in reduced energy consumption and enhanced energy conservation.

When heat transfer occurs between a flowing fluid and a surface, if the flow is caused by fan, blower or pump or forcing jet, the process is called Forced convection. If the flow is caused by the buoyant force generated by heating or cooling of the fluid by the surface, the process is called as natural or free convection.

Heat flux by convection was determined by **$q=h(T_s-T_a)$** ,where q is heat flux in w/m^2, T_s is the surface temperature and T_a is the ambient temperature. Hence the unit of heat transfer coefficient is w/m^2 k. convective heat transfer coefficient is influenced by fluid properties like density, viscosity and other thermal properties like specific heat and thermal conductivity.
It is also influenced by flow velocity and surface geometry. It may be noted that the physical or thermal properties of the surface material play no part in the process of convective heat transfer. As the fluid properties vary with the temperature and location the value of convective heat transfer coefficient varies from point to point, this leads to the situation that analytically derived equation are applicable only to a limited extent. Heat is not a state of the material , it is the transfer of energy caused by temperature difference. When heat is passed from one body to another the internal energy of the bodies change. heat transfer is thus , by definition, caused by temperature differences.

1.1 : Dittus Boelter relation

The conventional expression for calculating the heat transfer coefficient in fully developed turbulent flow in smooth pipes is the Dittus Boelter equation

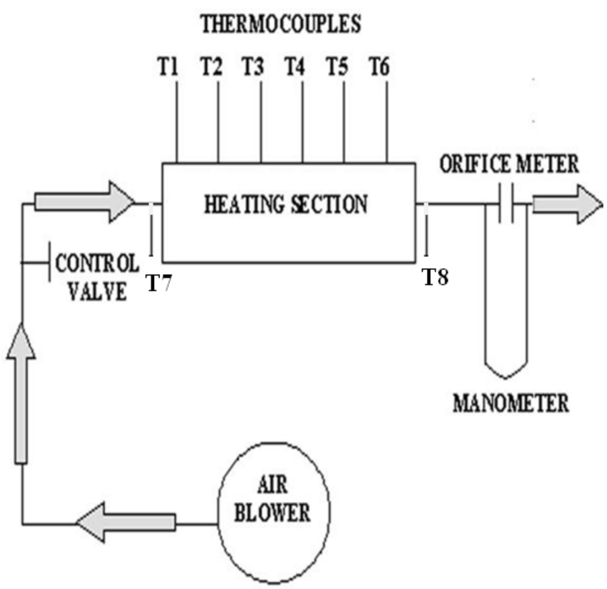
$$Nu = C\, Re^{m}\, Pr^{n}$$

Where C, m and n are constant determined experimentally. We will use these values C= 0.023, m = 0.8 and
n = 0.4 for heating of the fluid
n = 0.3 for cooling of the fluid
Properties of this relation have been calculated at the average fluid bulk temperatures. Equation is valid for single phase heat transfer in fully developed turbulent flows in smooth pipes for fluids with Prandtl number ranging from 0.6 to 100 at low heat fluxes. At high fluxes the fluid properties changes resulting in higher errors. At higher heat fluxes the Sieder-Tate equation is used to reduce the error. The work presented is about designing and acquiring data from an experimental setup to verify the Dittus-Boelter empirical relation by finding the heat transfer coefficient. The value of the heat transfer coefficient has been calculated from theory, experiment and CFD simulation to compare the results. This research work uses different approaches to calculate two very important performance considerations, heat transfer rate and pressure drop in turbulent duct flow, using Re, Pr and Nu.

For laminar flow in pipes the heat transfer coefficient (***h***) can be found by solving the governed differential equations analytically. But in the case of turbulent flow things become chaotic and it is impossible to solve differential equation analytically.
For turbulent flow we rely on experiments and develop correlations. One such correlation is *Dittus-Boelter equation.*

2. Experimental Setup:

The experimental set up consists of a blower unit fitted with the section pipe .The test section pipe is copper which is surrounded by band heater. Six thermocouples are embedded on the test section and two thermocouples are placed in the air stream at the entrance and exit of the test section to measure air inlet and outlet temperature. Test pipe is connected to the delivery side of blower along with orifice to measure flow of air through the pipe. Input to the heater through dimmerstat and measured by voltmeter and ammeter. Air flow is controlled by gate valve and is measured with the help of orificemeter and the manometer fitted on board.



Layout of Experimental setup

Specification:

Length of hallow pipe = 500 mm
Pipe inner diameter = 0.038 m.
Orifice diameter = 0.025 m.
T1 to T6 = Surface temperature of test cylinder
T7 and T8 = Air inlet and outlet Temperatures

Experimental heat Transfer Co-efficient,:

$$h_{exp} = \frac{Q}{A_s\,(T_s - T_A)}\ ,\ \frac{W}{m^2\,k}$$

Where,

Q = Rate of heating = V * I watts
A_s = Surface area of test tube = π * D * L
V = Voltage (volts)
I = Current (Amps)
D = Inner Dia.of cylinder (m)
L = length of cylinder (m)
T_s = Mean Temp. of Test cylinder surface

$$T_s = \frac{T_1 + T_2 + T_3 + T_4 + T_5 + T_6}{6}$$

Where, T1 to T6 = Surface temp. of test cylinder in ° c

T_A = Ambient Temperature.

$$T_A = \frac{T_7 + T_8}{2}$$

Where, T_7 = Air inlet temperature ° c
T_8 = Air outlet temperature ° c

Rate of heating : Q = V * I watts

V = 200 volts {Obtained from experiment}
I = 0.40 amps {Obtained from experiment}
Q = 200 * 0.40
Q = 80 watts.

Surface area of test tube : A_s = π * D * L
D = 0.038 m
L = 500 mm
A_s = π * 0.038 * 0.5
A_s = 0.05966 m^2

Head of water , h = 20 mm. {Obtained from experiment}

Temperatures obtained from experiment :
T_1 = 85 °c, T_2 = 88 °c, T_3 = 90 °c, T_4 =91°c, T_5=92 °c, T_6 = 92 °c, T_7 =38 °c, T_8 =41 °c.
T_s = Mean Temperature of Test cylinder surface

$$T_s = \frac{T_1 + T_2 + T_3 + T_4 + T_5 + T_6}{6}$$

$$T_s = \frac{85 + 88 + 90 + 91 + 92 + 92}{6}$$

T_s = 89.66 °c.

T_A = Ambient Temperature.

$$T_A = \frac{T_7 + T_8}{2}$$

$$T_A = \frac{38 + 41}{2}$$

T_A = 39.5 °c.

$$h_{exp} = \frac{Q}{A_s\,(T_s - T_A)}\ ,\ \frac{W}{m^2\ k}$$

$$h_{exp} = \frac{80}{0.05966\ (89.66\ -\ 39.5\ )}$$

$$h_{exp} = 26.55\ \frac{W}{m^2\ k}\ .$$

3. Analytical method :

Bulk mean temperature,

$$T_M = \frac{T_7 + T_8}{2}$$

Where, T_7 = Air inlet temperature ° c

T_8 = Air outlet temperature ° c

$$T_M = \frac{38+41}{2}$$ {Obtained from experiment}

T_M = 39.5 °c.

Properties of air for Tmean:

Density, ρ = 1.110 kg/m^3 ,
Kinematic viscosity, ν := 16.9×10^{-6} m^2/s
Thermal conductivity, K = 0.027 W/ mK
Prandtl number, Pr = 0.69

Discharge through orifice

q= C_d* A_o* $\sqrt{2gh_a}$

Where,

C_d = Coefficient of discharge = 0.65,

A_o = Area of orifice, (dia. =.025 m)

= πd^2 / 4

= π * 0.025^2 / 4

= 4.9 x10^{-4} m^2

h_A = (density of water × head of water) / density of air.

h_A = (1000×0.020) / 1.2

h_A = 16.66 m (head of air).

q= C_d* A_o* $\sqrt{2gh_a}$

q= 0.65 × 4.9 x10^{-4} √2×9.81×16.66

q= 5.76×10^{-3} m^3/s (discharge in orifice)

velocity:

v = q/A

where, A = inner surface area of pipe

A = πd^2 / 4

A = π * 0.038 ^{2} / 4

A = 1.13×10^{-3} m^2

V = 5.76×10^{-3} / 1.13×10^{-3}

V= 5 m/sec.

Reynolds number:

$$Re = \frac{v D_H}{\nu}$$

D_H = inner dia of pipe = 0.038 m

ν is kinematic viscosity obtained from data hand book properties of air = 16.96×10^{-6}

$$Re = \frac{5 \times 0.038}{16.96 \times 10^{-6}}$$

Re = 11515

From heat transfer data hand book corresponding to this value of Reynolds number, correlation has been choosen that is Dittus-Boelter equation

$$Nu = 0.023\,Re^{0.82}\ Pr^{0.4}$$

$$Nu = 0.023\ 11515^{0.82}\ 0.69^{0.4}$$

$$Nu = 35.8$$

Nusselt number

$$Nu = \frac{hD_h}{k_f}$$

where:
D_h : pipe inner diameter (m).
k_f : Thermal conductivity of the fluid W/m K
h :convective heat transfer coefficient W/ m^2 K

$$h = \frac{\mathrm{Nu}\, k_f}{D_h}$$

$$h = \frac{35.8 \times 0.027}{0.038}$$

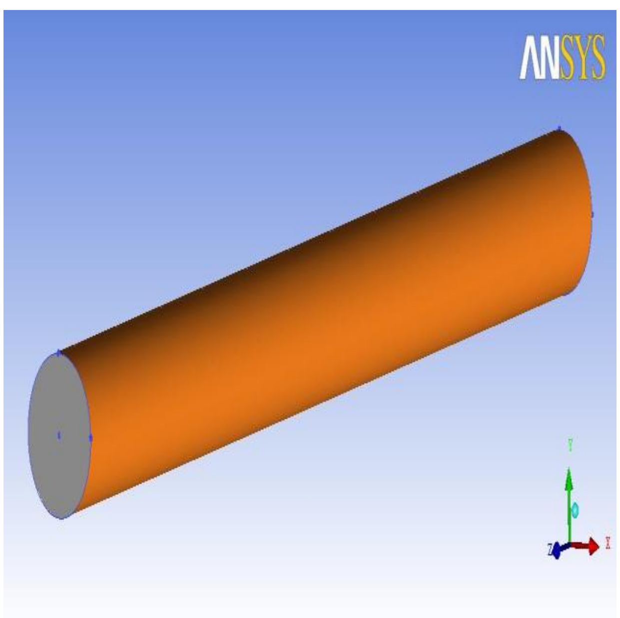
$$h = 25.5 \ \frac{W}{m^2\, k}$$

4 . CFD SIMULATION :

CFD Analysis of pipe:
There are four basic steps involved in the CFD analysis of pipe. They are given as below

- Creating geometry and meshing
- Pre-processing
- Solver
- Post processing.

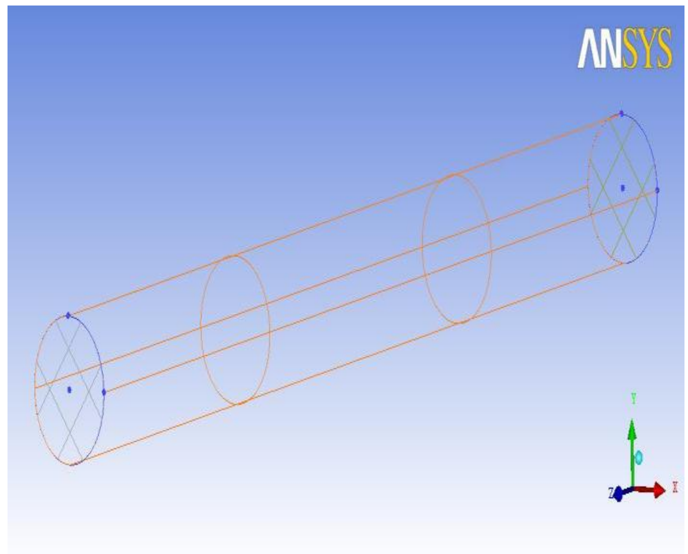
4.1 Modeling of pipe using ICEM CFD



Modeling of pipe

4.2 Geometry Model

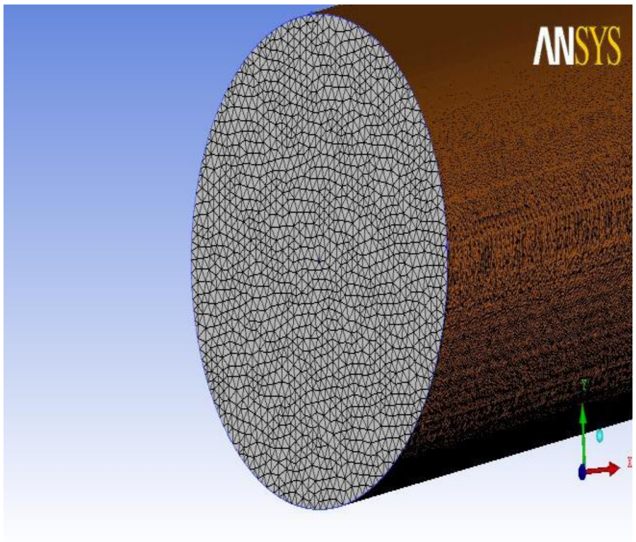
ANSYS ICEM is used to build a 3D geometry for CFD analysis is shown in fig. 7.1 The geometry is created and Meshed. Here we are using ANSYS ICEM CFD 12.1 tool to create the pipe geometry. Modeling has been done in ICEM CFD. The length of pipe is 500mm and inner dia is 0.038m.



Wire Frame of Pipe using ICEM CFD

4.3 Meshing: In ANSYS CFX, geometry can be imported from most major CAD packages using native format, and the mesh of control volumes is generated automatically. The partial differential equations that govern- fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller sub domains. The governing equations are then discretized and solved inside each of these sub domains.

Typically, one of three methods is used to solve the approximate version of the system of equations: finite volumes, finite elements, or finite differences. The sub domains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. The objective is to produce a mesh for input to the problem definition i.e. pre-processor.



Mesh Generated Using ICEM

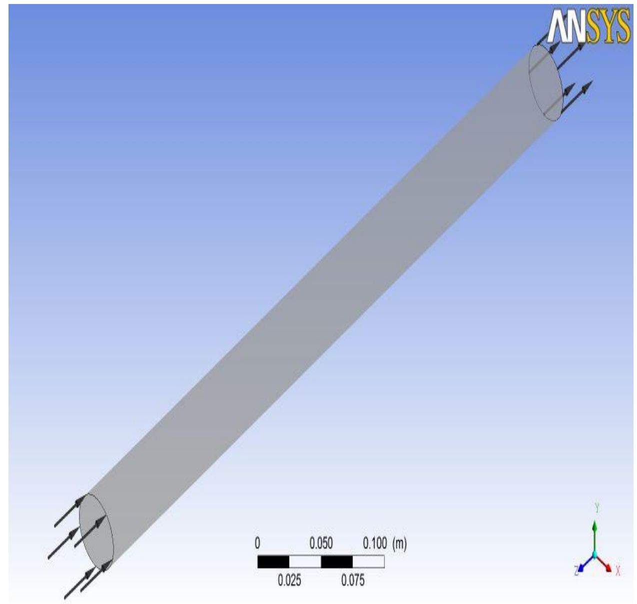
4.4 CFD Simulation in CFX-13

The third method is the CFD simulation. All of our work is done in ICEMCFD andCFX-13. These are very powerful tools with many advanced features. We have created the geometry and mesh in ICEMCFD and CFX-13. In meshing we have used unstructured meshing technique with high mesh concentration where the gradients are large. The boundaries are also specified in the CFX-13.

The solver and the boundary conditions are specified in CFX-13 We have used segregated solver with axsymmetric space and k-epsilon for turbulent modeling. The working conditions are given below

INLET VELOCITY =5 m/s

INLET TEMPERATURE = 311K



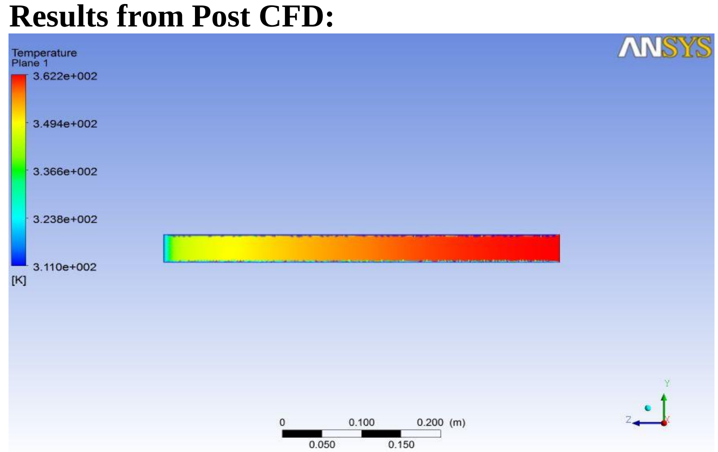
Boundary conditions in CFX-13

Model imported from ICEM-CFD to CFX and applied boundary condition

- Location inlet :
  Boundary details : subsonic
  Normal speed = 5m/sec.
  Medium intensity=5%
  Heat transfer: Isothermal
  Static temp=311k
- Location out let
  Boundary details : subsonic
  Relative pressure = 1pa
  Wall : no slip wall
  smooth wall

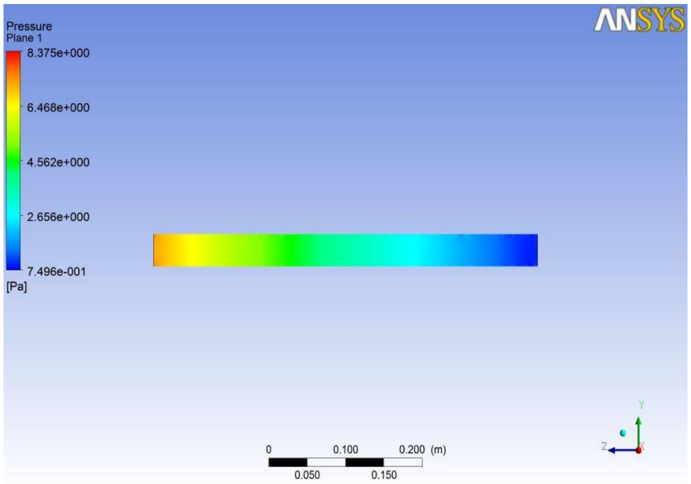
5. Results & Discussion

Results from Post CFD:



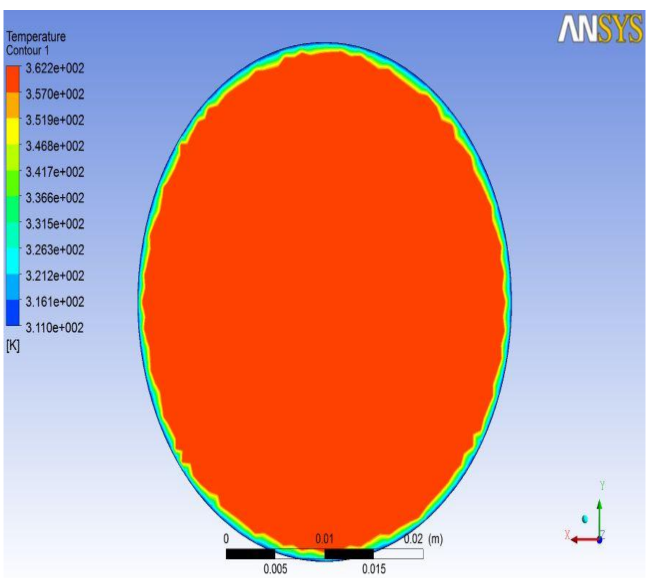
Temperature Plane

From the figure above, we can see the temperature variation inside the cylindrical duct. Where temperatures along with the duct from inlet to outlet is increasing linearly.



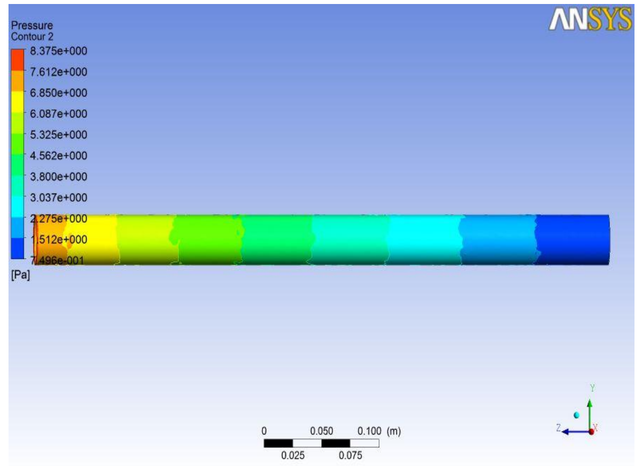
Pressure Plane

From the figure ,we can see the pressure variation inside the cylindrical duct. Where pressure along with the duct from inlet to outlet is increasing linearly.



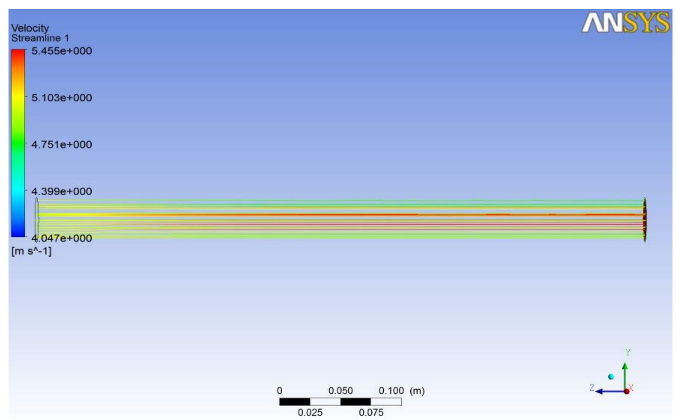
Temperature Outlet Countour

From figure,we can see the Temperature Outlet countour variation inside the cylindrical duct. Where Temperature increases circumferentially



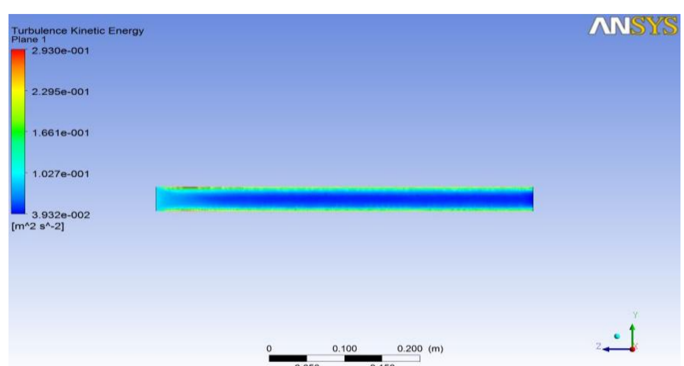
Pressure Countour

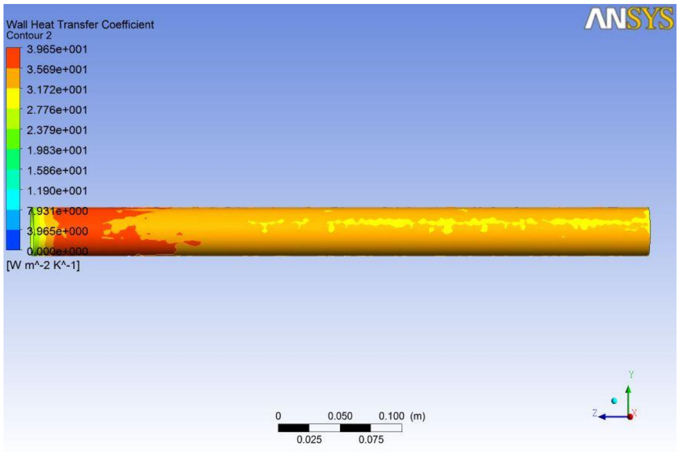
From the figure , we can see the pressure countour inside the cylindrical duct. Where pressure increases circumferentially.



Velocity Streamline

From the figure , we can see the velocity variation inside the cylindrical duct. Where velocity along with the duct from inlet to outlet is decreases linearly.





Heat Transfer Coefficients

From the figure , we can see the heat transfer coefficient is increasing along the length of the duct, on an average w can take 32.00 as heat transfer coefficient.

Comparison of experimental ,CFD and correlation:

Method	Model 1
Experimental	26.55, $\frac{W}{m^2\ k}$
CFD	32.0, $\frac{W}{m^2\ k}$
correlation	25.55 , $\frac{W}{m^2\ k}$

Comparison of experimental, CFD and correlation

we can see that the value obtained by CFD is very nearer to correlation and experimental. Hence **Dittus-Boelter equation is validated**.

6. Conclusion

By carrying out experiment, CFD simulation and analytical calculation, we conclude that these correlations gives an error well within the range 25% and usually less in controlled environment. The results clearly validate the use of Dittus-Boelter correlation in many industrial applications.

There are many reasons which contribute to the errors. The heat loss does account for errors, but the value of heat loss is negligible as compared to total heat flux. The heaters are local made and do not provide constant surface heat flux although we have tried to control wattage of each heater. Secondly all sensors (temperature) are attached in a close proximity of 220 V AC lines coming from heaters. Therefore signals from sensors are distorted and give an error. If somehow these problems are overcome results may improve. Another thing is correlation results must be more but here it is less because we are taking the property of air at mean temperature of inlet and out let obtained by experiment.

REFERENCES

1) **Dittus, F. W.,** and **L. M. K. Boelter**. *University of California (Berkley)* *Pub. Eng.*, vol. 2., p. 443, 1930.

2) **Holman, J. P**., *Heat Transfer*, 9th edition, Mc Graw Hill, p. 267-271.

3) **Diane L. Linne, Michael L. Meyer, Tim Edwards, and David A. Eitman.** *Evaluation of Heat Transfer and Thermal Stability of Supercritical JP–7 Fuel.* NASA Technical Memorandum 107485.

4) Incropera, *Fundamentals of Heat and Mass Transfer*, 5th edition, John **Wiley and Sons**

5) **J. R. Lamarsh & Anthony B.**, *Intro to Nuclear Engineering*, 3rd edition, Wiley

6) **Munson, Young, Okishi**, Fundamentals of Fluid Dynamics, 4th edition, Wiley

7) **Tichakorn,** *Forced Convection*, research paper, Cornell University, Nov 2004

8) Mechanical and Aerospace Engineering Laboratory, *Forced Convection Heat Transfer*, State University of New Jersey

9) Application note, *Active Filter Design*, Texas instrument

10) Application note, *AN699*, *AN823*, *AN990*, *AN682*, *AN685*, *AN687*, *AN684* Microchip Technology Inc

11) **Hilpert, R**. "Heat Transfer from Cylinders," Forsch. Geb. Ingenieurwes, 4:215. 1933.

12) **Incropera F. and D. De Witt.** Fundamentals of Heat and Mass Transfer, 5th Edition, USA. 2002

13) **Fand, R. M. and K. K. Keswani**. "A Continuous Correlation Equation for Heat Transfer from cylinders to Air in Crossflow for Reynold's Numbers from 10-2 to 2(10)5," International Journal of Heat and Mass Transfer, 15:559-562. 1972.

14) **Fand, R.M. and K. K. Keswani**. "Recalculation of Hilpert's Constants," Transactions of ASME,pp. 224-226. 1973.

15). **Zukauskas**, **A.** "Heat Transfer From Tubes in Crossflow," Advances in Heat Transfer, 8:87-159.1987.

16). **Churchill, S.W. and H. H. S. Chu.** "Correlating Equations for Laminar and Turbulent Free Convection From a Horizontal Cylinder," International Journal of Heat And Mass Transfer,18:1049-1053. 1975.

17). **Churchill, S. W. and M. Bernstein.** "A correlating Equation for Forced Convection from Gases and Liquids to a Circular Cylinder in Crossflow," J. Heat Transfer, 99:300-306. 1977.

18). **Morgan, V.** "Heat Transfer from Cylinders," Advances in Heat Transfer, 11:199-264. 1987.

19). **Manohar, K., Yarbrough, D. W. and Booth, J. R**. "Measurement of Apparent Thermal Conductivity by the Thermal Probe Method," Journal of Testing and Evaluation, 28(5):345-351.2000.

20). **Coleman, H. W. and Steller, W. G**. "Experimentation and Uncertainty Analysis for Engineers,"2nd Ed., John Wiley and Sons, New York, pp. 47-64 (1999).