Dec. 2014. Vol. 5. No. 07 ISSN2305-8269
International Journal of Engineering and Applied Sciences

www.eaas-journal.org

EXPERIMENTAL AND CFD ANALYSIS OF TURBULENT FLOW HEAT TRANSFER IN TUBULAR EXCHANGER

HESHAM G. IBRAHIM

Assist. Prof., Department of Mechanical Engineering, Faculty of Marine Resources,
Al-Asmarya Islamic University, Zliten City, Libya
E-mail: hesham.ibrahim80@gmail.com

ABSTRACT

The phenomenon of forced convection with turbulent flow of industrial processes is complicated to develop analytically. The only key to the problem is empirical models and numerical solutions. Heat transfer coefficient (h) and friction factor are very important parameters for fluid flow systems due to their use in determining the heat transfer rate and the pressure drop of the system respectively. Many of coefficient formulas have been obtained from the application of simple statistical techniques to large datasets, taking into account several engine operational parameters and engine types. The resulting correlations provide reasonable estimates but perform poorly rather than extrapolated to novel concepts. So, the feasibility and accuracy of Computational Fluid Dynamics (CFD) by using FLUENT to estimate the convective heat transfer coefficients is examined. The heat transfer coefficient values are estimated by empirical correlations from literature and CFD simulation then compared with experimental data for air flow at operating conditions of the experiment. The results obtained by CFD were in good agreement with both experimental data and empirical correlations of literature. This result reveals to CFD as an accurate tools to predicting heat transfer coefficient for turbulence flow of the industrial process.

Keywords: ***Heat transfer coefficient, Turbulence flow, CFD simulation.***

1. INTRODUCTION

A common situation encountered by the chemical engineer is heat transfer to fluid flowing through a tube. This can occur in heat exchangers, boilers, condensers, evaporators, and a host of other process equipment. Therefore, it is useful to know how to estimate heat transfer coefficients in this situation. Convection is the mode of energy transfer between a solid surface and the adjacent fluid that is in motion, and it involves the combined effects of conduction and fluid motion. The presence of bulk motion of the fluid enhances the heat transfer between the solid surface and the fluid, but it also complicates the determination of heat transfer rates. In order to simplify convection phenomena complexity a lot of empirical equations were investigated. For design and engineering purposes, empirical correlations are usually of greatest practical utility. Many empirical equations, which help in convection heat transfer coefficient determination, may be found in the literature. Researches, which were based on dimensional and similarity analysis, built number of equations in different external conditions, shapes, type of convection or motion. The conventional expression for calculating the heat transfer coefficient of fully developed turbulent flow in smooth pipes is given by Dittus and Boelter [1]. This correlation 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 Sieder-Tate equation is used to reduce the error. For a comparison between the Dittus and Boelter, the Sieder-Tate and other correlations detailed discussions are given in text books by Incropera and DeWitt [2] and Holman [3] concerning this matter. All these correlations are available for calculation of the convective heat transfer coefficient for turbulent flow of a fluid in a pipe at different temperatures based on experiments data as discussed by Holman [3] to describe how correlating of the experimental data.

Empirical equations are usually derived from experimental results. There are no mathematical or physical equations of process which is analyzed.

ISSN2305-8269
International Journal of Engineering and Applied Sciences

www.eaas-journal.org

But with known boundary conditions, so a dimensional analysis allows to build mathematical, empirical equations based on Buckingham theorem. It is essential to evaluate various conditions which might impact investigated process, as in some cases omitting some of them might result with obtaining invalid (therefore useless) equation. Similitude uses research on models – in smaller scale and results may be transformed, thanks to certain dimensionless numbers. So, currently due to availability of more simulation packages such as FEMLAB, FLUENT, …etc. which may be used as tools to simulate a heat transfer within a large scale equipment with appreciated high accuracy and more efficient to experimental work. So the purpose of this paper is to demonstrate the feasibility and accuracy of using a commercial Computational Fluid Dynamics (CFD) software to calculate convective heat transfer coefficients (such as Abbas et al. [4], Jamuna and Somashekar [5]).

CFD is a modeling technique that breaks down the governing equations (continuity, momentum and energy) for fluid flow into simpler forms that can be solved using numerical techniques [6]. The mathematical resolution of the governing equations is still not fully resolved. CFD must then circumvent this by using models to approximate some components of the flow. There are still no universal rules or guidelines on the appropriateness of different models to be used in different problems. Therefore, any CFD calculation must first be validated [7]. Then, obtain a correlation for heat transfer coefficients for these large scales. Generally, the experimental rig is designated to perform a broad range of experiments related to the fluid dynamics and heat transfer phenomena for air flowing by forced convection inside a pipe at different velocities. This data acquiring from the experimental setup used to calculate heat transfer coefficients then compared with referenced correlations and to verify with the values of CFD simulation 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*.

2. MATHEMATICAL FRMULATION

2.1 Determination of Convective HTC

To evaluate the convective Heat Transfer Coefficient (HTC), the main assumptions applied for the energy balance on the tested pipe herein:

- Mass flow rate is constant.
- Heat transfer occurs at the inner surface of the pipe only.
- No shaft work is done by the fluid.
- Constant heat flux is applied at the surface.

Applying the 1st law of thermodynamics to the system, the following energy balance equation is derived;

$$Q_C = mCp(T_8 - T_9) \qquad (1)$$

where; Q_C represents the thermal power exchanged by convection from the wall to air (*W*). T_8 and T_9 represents the centerline temperature of the inlet and outlet air; *m* is the mass flow rate measured by orifice plate, (*kg/sec*); and *Cp* is a specific heat of air is determined as a function of average temperature (*J/kg.K*).

Furthermore the total heat transferred to air by forced convection is equal to the rate at which thermal energy of the fluid increases for a gas. Therefore a thermal power that is given by Newton's law of convection is;

$$Q_C = hA\Delta T_{ap} \qquad (2)$$

where; *A* is the inner surface area of the cylindrical section, (m^2), *h* convective heat transfer coefficient of air in the tube, ($W/m^2.K$) and ΔT_{ap} average wall-to-air temperature difference, (*K*), it is defined as;

$$\Delta T_{ap} = \frac{(T_2 - T_8) + (T_6 - T_9)}{2} \qquad (3)$$

Finally by equating Eqns. (1 & 2), the experimental value of heat transfer coefficient at known operating conditions is obtained as following;

$$h = \frac{mCp(T_8 - T_9)}{A\Delta T_{ap}} \qquad (4)$$

2.2 Validation by Reynolds Analogy

The Reynolds analogy allows establishing a quantitative correlation between the convective heat transfer coefficient h and the friction coefficient. The analogy is based on the physical correspondence between the two phenomena of heat transported (convection) and momentum transport (flow friction) for a viscous fluid in forced motion inside a pipe.

The assumption is made that, the following two quantities are equal for the entire section;

- The fraction of the fluid thermal energy content which is exchanged by convection.
- The fraction of the fluid momentum which is lost by friction.

This relation is given by the Reynolds analogy for tube flow [3];

Dec. 2014. Vol. 5. No. 07 ISSN2305-8269
International Journal of Engineering and Applied Sciences

www.eaas-journal.org

$$St = \frac{Nu}{Re*Pr} = \frac{f}{8} \tag{5}$$

This Eqn. may be rewriting as following;

$$Nu = \frac{f}{8} Re\ Pr \tag{6}$$

where; *Nu*, *Re* and *Pr* are dimensionless groups and defined as following;

$$Nu = \frac{h\,D}{k} \tag{7}$$

$$Re = \frac{\rho\,U\,D}{\mu} \tag{8}$$

$$Pr = \frac{Cp\,\mu}{k} \tag{9}$$

f: friction coefficient

h: heat transfer coefficient from wall to air, ($W/m^2.K$)

D : Internal diameter of tube, (*m*).

k : thermal conductivity of air, (*W/m.K*)

U: average velocity through a pipe, (*m/sec*)

ρ : Density of air, (kg/m^3)

μ: Viscosity of air, (*Pa.sec*)

Cp : Specific heat of air, (*kJ/kg.K*)

All thermal properties of air are function of the bulk temperature of air. The friction factor (*f*) can be determined with Eqn. [8];

$$f = \frac{\Delta P_t}{(L/D)\,\rho\,(U^2/2)} \tag{10}$$

where;

ΔP_t: Static pressure difference along the pipe (measured experimentally by manometer), (*Pa*).

L: Length of a pipe, (1,240 mm)

Also the average air velocity is calculated as [8];

$$U = \frac{4\,m}{\pi\,D^2\,\rho} \tag{11}$$

where;

m: Mass flow rate of the air (measured experimentally by orifice plate), (*kg/sec*).

Values of the heat transfer coefficients calculated using Eqn. (6), correspond to experimentally measured variable values.

3. EXPERMINTAL SETUP

A schematic diagram of the experimental facility is presented in Figure(1). Air is blown through one end of the pipe by a centrifugal blower (1). The air flow rate is controlled by damper (2) installed on the air inlet, and measured by flow meter (orifice plate) (3) and related to differential manometer (ΔP_d, ΔP_m). The air was delivered through a steel pipe (4) with (*ID*=64.2 *mm*) to a copper pipe (5) with (*ID*=32 *mm*). The copper pipe is heated along its circumference with constant heat flux, in which 18 electrical resistances heated over a length of copper pipe equal to1200 *mm* (6) heaters. Air is blown through the pipe and wall temperature is measured at different intervals using thermocouples. The pressure and centerline temperature of the fluid is also noted at the beginning of the copper pipe (T_8) and at the end (T_9) of the copper pipe using differential manometer (9) and thermocouples (7) respectively. A Pitot tube and related differential manometer (ΔP_p) (8) used for the measurement of local values of air velocity, the Pitot tube can be displaced in the radial direction over a range of ±10 *mm* with respect to the pipe axis. All the process controlled by a control panel including (electrical resistor power supply switch, blower power supply, thermocouple selector, three digit temperature display (resolution ± 0.5^o*C*), voltmeter and ammeter for the measurement of electrical resistors supply voltage and current, and electrical resistor power supply regulator).

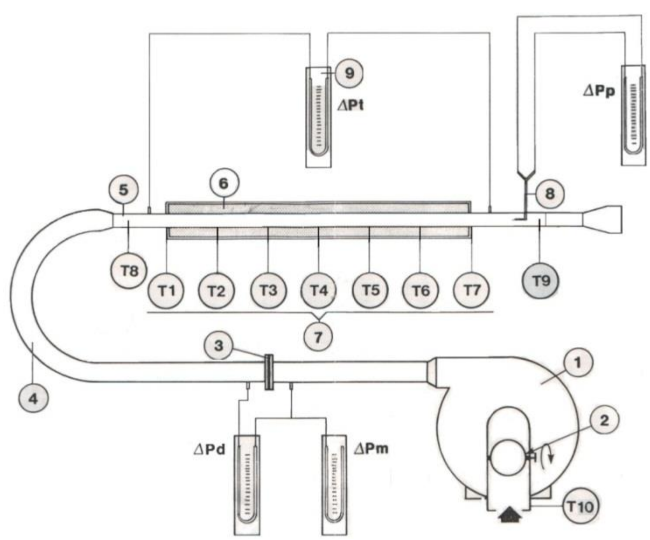


Fig-1: Experimental facility: (1) centrifugal blower; (2) damper; (3) flow meter (orifice plate *ID*=40 *mm*); (4) steel pipe; (5) copper pipe; (6) heater; (7) thermocouples; (8) Pitot tube; (9) differential manometer

Air is blown through one end of the pipe and removes heat by forced convection while the other side of the pipe is open. Heaters and the pipe are well insulated using glass wool to make sure that maximum heat flux should go inside the pipe instead of transported away. The pipe used is made of pure copper and the specifications are given as a function of the average wall temperature.

Custom made heaters were used which are band type heaters wrapped round the pipe on a length of 1,240 *mm*. heaters are made using Ni-chrome wire which provide heat when electric current is passed through it. The specification of the heaters are shown in Table.1.

Table .1: Specifications of the heater

Specification	Unit	Value
No. of heater	--	18
Length of heated section	*m*	1.24
Power of each heater	*W*	111
Total power	*W*	2,000

The used blower has a converging nozzle outlet inserted in a steel pipe working as flow development section which is then connected with a copper pipe and the blower itself is fixed on the table rig.

The apparatus consists of the components as shown in Figure 1. In the experiment, a copper pipe was used which is heated along its circumference with constant heat flux. Air is blown through the pipe and the wall temperature is noted at different intervals using thermocouples (T_1-T_7). The pressure difference (ΔP_t) and centerline temperature of the air is also noted between the copper pipe beginning and at the end of the pipe (T_8 and T_9) respectively. When steady state attained, the tests may be performed and repeated without any particular time constraint. Then the collected data is analyzed in the computer and the heat transfer coefficient is estimated.

Generally, a number of verification runs were undertaken prior to the data logging. A series of tests for air flow conducted to establish the validity of the system and test technique. The experimental results agreed well with those calculated by Kays et al. [9] for fully developed turbulent flow in a tube. Also agreed with Mosyak et al. [10], in which proved that the wall temperature fluctuations were influenced by the thermal properties and thickness of the wall but the average wall temperature and heat transfer coefficient were independent of the wall thickness.

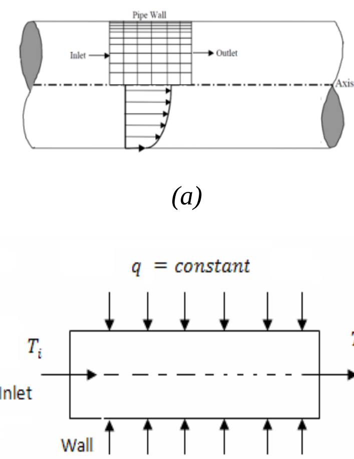
4. CFD SIMULATION

Simulation of the experimental work was done by using GAMBIT® and FLUENT 6.3.26.

4.1 Mesh Geometry in GAMBIT

The geometry and mesh created in GAMBIT®. In meshing process used edge meshing technique with high mesh cells. There are 15,000 quadrilateral cells in this case. This is because using 100 divisions in the horizontal direction and 150 divisions in the vertical direction while generating the grid. So the total number of cells is 100*150 = 15,000 as shown in Figure 2.a.

The boundaries are also specified in the GAMBIT®. From Figure 2.b, it is clear that the system is symmetric, hence symmetry form is used to shorten of the computational time.



(a)

(b)

Fig-2: Fully Developed Turbulent Pipe Flow Model with Grid

4.2 Set Up Problem in Fluent

The solver and boundary conditions are specified using FLUENT 6.3.26. The "2ddp" option is used to select the 2-dimensional, double-precision solver. The used defaults of segregated solver with assymmetric space and the k-epsilon turbulence model with the Realizable model. The Realizable k-epsilon model produces more accurate results for boundary layer flows than the Standard k-ε model. In the *Near-Wall Treatment* box, observe the Enhanced Wall Treatment option, which deals with the resolution of the boundary layer for a case study. The operating condition are constant heat flux (10,393.49 W/m^2), inlet velocity range (5-140 *m/sec*) and inlet temperature (315.15 *K*). FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. The solution will iterate until the residual for each equation falls below 1×10^{-6}.

Dec. 2014. Vol. 5. No. 07 ISSN2305-8269
International Journal of Engineering and Applied Sciences

www.eaas-journal.org

5. RESULTS AND DISCUSSION

After the steady state has been achieved (i.e. the heating transient is completed when, in two successive readouts of temperatures do not show appreciable variations). Duration of the heating transient is finally recorded when steady state conditions are reached. Each run replicated three times to avoiding any mistakes during experiments. Average values of temperatures collected during the experiment with 2 *min*. intervals are shown in Figures (3 and 4) for operating conditions Mass flux =68.797 kg/m^2.sec, Re=1.1*10^5.

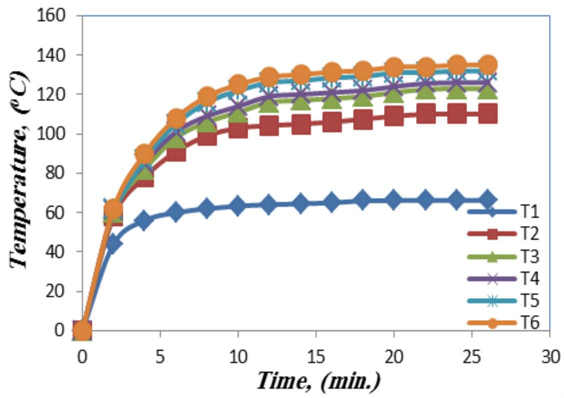


Fig-3: Wall temperature measured experimentally (Mass flux =68.797 kg/m^2.sec, Re=1.1*10^5)

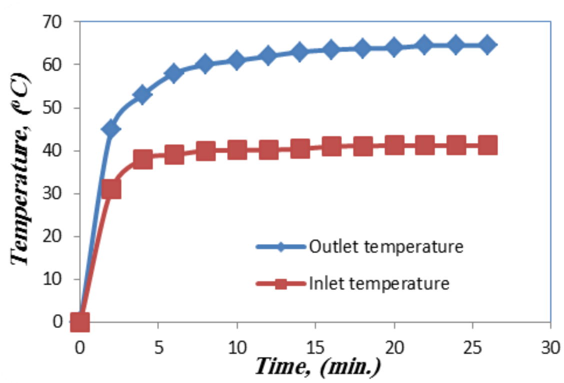


Fig-4: Temperature of bulk air measured experimentally (Mass flux =68.797 kg/m^2.sec, Re=1.1*10^5)

These figures show that, the heating transient time end after about 26 min. (this time approximately equal for each runs at different flow rates). The wall temperature of the pipe (outer surface) is recorded after reached a steady state as shown in Figure 5. The curve shows a linear response once the thermal boundary layer has developed. A good accuracy between both experimental and CFD collected data.

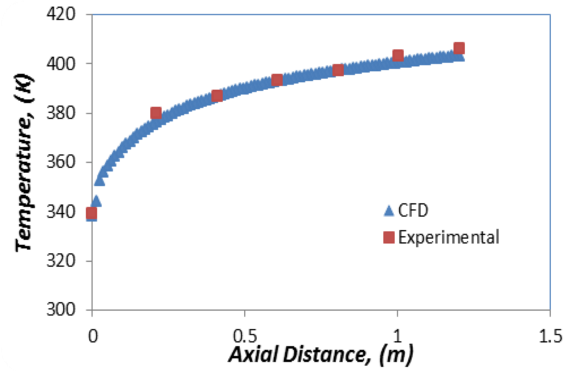


Fig-5: Comparison of the pipe wall surface temperatures between Experimental and CFD data

Also, the radial velocity profile for turbulent flow in the pipe is measured in Figure (6) where a viscous sub-layer or film is formed along the inner wall of the pipe. As shown in this satisfies the agreement between experimental data measured by Pitot tube with a CFD data.

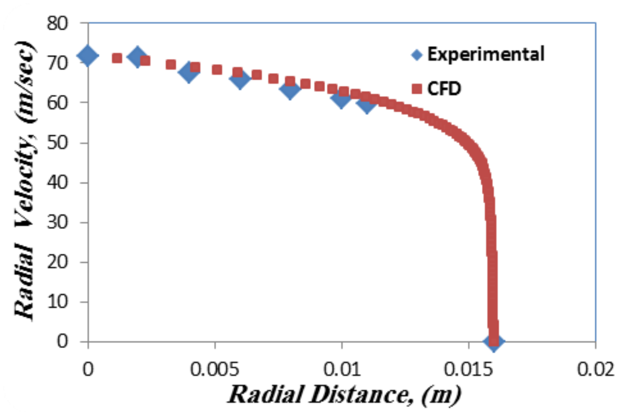


Fig-6: Comparison of radial velocity distribution for experimental and CFD data (Mass flux =68.797 kg/m^2.sec, Re=1.1*10^5)

All these appropriate consistency between experimental and CFD results shown in Figures(5, 6), reveal to that a simulation process representing the experimental setup with satisfaction mode. The average values of heat transfer coefficient estimated experimentally for different flow rate as a function of Nusselt and Reynolds numbers are shown in Figure (7)

The heat transfer coefficient increases when increasing of the flow rate, due to increase of the turbulence of the bulk that leads to decrease of the temperature gradient (driving force) between the bulk air flow and the wall of tube as a result of decreasing thickness of the boundary layer.

International Journal of Engineering and Applied Sciences

www.eaas-journal.org

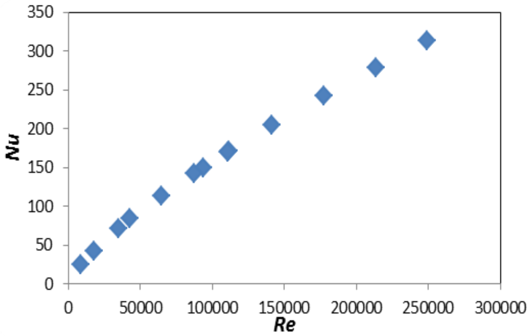


Fig-7: Experimental values of Nusselt number, by using Eqn. (4)

Values of Nusselt number that estimated experimentally by [Eqn.(4)], Reynolds analogy [Eqn.(6)] and CFD are tabulated in Table 2 with deviation error for each one compared with experimental values are present. A Nusselt number values obtained experimentally and by Reynolds Analogy are identical, that reveals to trustiness of these values and experiments, this point indicated by Pham [11]. While a CFD data for Nusselt number have error (approximately ±21%) compared with experimental data. This difference may be attributed to the changes occurs in the physical properties of the air, in which is calculated as a constant value at average bulk temperature, that may be leads to accumulative error during Nusselt number calculation. Suppose after number of experiments are conducted with measurements taken of heat transfer rates of air in turbulent flow inside for smooth tubes by varying range of the mass flow rate. Generalize the results of these experiments by arriving at one empirical equation which represents all the data. Depending on studies may anticipate that the heat transfer data dependent on Reynolds and Prandtl numbers according to the general form [3];

$$Nu = C\, Re^{m}\, Pr^{n} \tag{12}$$

A discussed procedure by Holman [3] applied to evaluate the constants of *C*, *m*, and *n* from the experimental data. Generally a final correlation usually represents the data within ±25% it will be acceptable. According the experimental data throughout Table.2 are expressed in a final correlation form;

$$Nu = 0.0315\, Re^{0.75}\, Pr^{0.333} \tag{13}$$

For this work found that for each run the *Pr* number has approximately a constant value (≈ 0.711), for air. So, Eqn. (13) may be rewriting as follow;

$$Nu = 0.0281\, Re^{0.75} \tag{14}$$

Table.2: Comparison of values of Nusselt number (experimental, Reynolds analogy and CFD)

Re (--)	*Nu* $_{Exp.}$ (Eqn.(4))	*Nu* $_{Re\,A}$ (Eqn.(6))	*Error* (%)	Nu_{CFD}	*Error* (%)
8748.76	25.398	25.39	0.0345	30.964	-21.91
17497.53	42.715	42.71	0.0128	47.733	-11.74
34995.05	71.838	71.83	0.0119	78.024	-8.61
42898.19	83.692	83.69	0.0019	89.864	-7.38
64759.34	113.980	113.98	0.00018	116.744	-2.42
87487.63	142.827	142.82	0.00546	141.088	1.22
93471.13	150.093	150.09	0.00207	155.449	-3.57
110684.1	170.379	170.37	0.0054	169.673	0.41
111446.1	171.258	171.25	0.00479	170.353	0.53
140969.8	204.267	204.26	0.00318	191.540	6.23
177313.6	242.610	242.61	0.00012	216.094	10.93
213752.4	279.117	279.11	0.00244	240.070	13.99
249425.2	313.370	313.37	0.00013	264.160	15.70

Good fitting of this correlation for experimental and CFD data as shown in Figure.8.

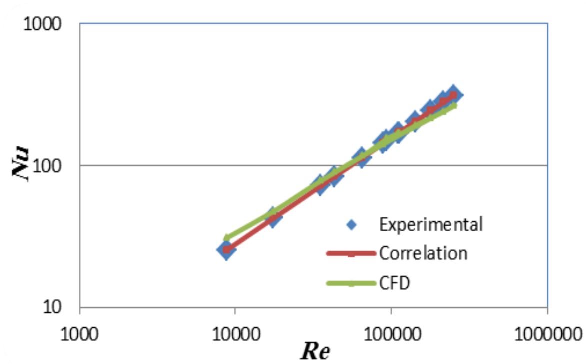


Fig-8: Comparison of Nusselt number for experimental, Correlation and CFD estimation

A traditional expression for calculation of heat transfer in fully developed turbulent flow in smooth tubes is that given by Dittus and Boelter [1] for heating of fluid;

$$Nu = 0.023\, Re^{0.8}\, Pr^{0.4} \tag{15}$$

This correlation valid for $0.6 \leq Pr \leq 160$, $Re \geq 10{,}000$ and $L/D \geq 10$ [3].

Also, a modern correlation that is given by Gnielinski [12] it is recommended for turbulent

Dec. 2014. Vol. 5. No. 07 ISSN2305-8269
International Journal of Engineering and Applied Sciences

www.eaas-journal.org

flow in smooth pipes for $3{,}000 \le Re \le 5\times10^6$, $0.5 \le Pr \le 200$ and $L/D \ge 10$.

$$Nu = \frac{(f/8)(Re-1000)Pr}{1+12.7(f/8)^{1/2}(Pr^{2/3}-1)} \quad (16)$$

The friction factor f is the Darcy friction factor, and maybe use Petukhov's formula for evaluating it;

$$f = \frac{1}{[0.79\ln(Re)-1.64]^2} \quad (17)$$

Physical properties to be used in these correlations (Eqn. (14, 15 and 16) are evaluated at average fluid bulk temperature. Comparisons between results of these correlations are shown in Figure 9. As shown there are deviations between the results obtained by this work with other classic correlations discussed. This deviation due to a generalize form of these correlations more than of the present work, in which theses correlations build on many experiments undergoes on a lot types of fluids with different diameters of tubes that effect on the final form of correlations to fitting data. While present case study only done for air. So, this deviation will be normal character for this type of work. The results refer to a correlation that given by Gnielinski [12] is more satisfied with present work more than Dittus and Boelter correlation. Generally, the deviation of the present work, Dittus and Boelter, and Gnielinski data are not exceeds ±1%, ±35% and ±18% respectively with respect to experimental data as illustrate in Table.3; these error values are within acceptable range.

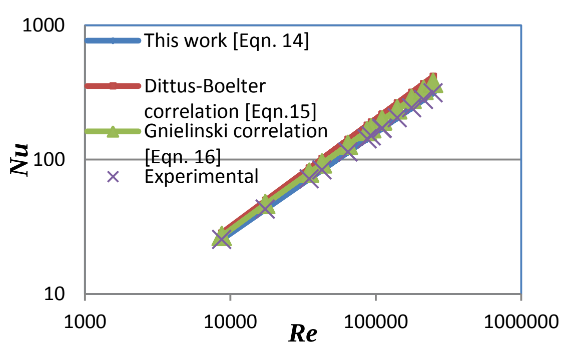


Fig-9: Comparison of *Nu* numbers as a function of Reynolds number for air

All the previous discussed results assert on the validity of using Eqn.(14) as a correlation to estimation of the heat transfer coefficient of turbulent convective flow through pipes.

Table.3: Range of errors for Nusselt number

$Nu_{Exp.}$	*Nu* Eqn.(14)	*Error* %	*Nu* Eqn. (15)	*Error* %	*Nu* Eqn. (16)	*Error* %
25.39	25.42	-0.081	28.57	-12.50	26.96	-6.16
42.71	42.75	-0.081	49.75	-16.47	46.69	-9.32
71.84	71.89	-0.081	86.62	-20.58	79.81	-11.09
83.69	83.76	-0.081	101.94	-21.81	93.40	-11.60
113.98	114.07	-0.081	141.73	-24.34	128.51	-12.75
142.83	142.94	-0.081	180.29	-26.23	162.46	-13.74
150.09	150.22	-0.081	190.09	-26.65	171.08	-13.98
170.38	170.52	-0.081	217.62	-27.72	195.31	-14.63
171.26	171.39	-0.081	218.81	-27.77	196.37	-14.66
204.27	204.43	-0.081	264.07	-29.28	236.26	-15.66
242.61	242.81	-0.081	317.26	-30.77	283.26	-16.75
279.12	279.34	-0.081	368.43	-31.99	328.60	-17.72
313.37	313.63	-0.081	416.85	-33.02	371.63	-18.59

6. CONCLUSION

A study of the turbulent flow pattern and heat transfer to air–wall flow in a horizontal pipe of 32 *mm* (*ID*) was conducted. The air average velocity varied from 5 *m/s* to 140 *m/s*.

The heat transfer coefficient was estimated from the temperatures measurements which were conducted by using thermocouples. Then theses values are verified by using Reynolds Analogy method then compared of these values with heat transfer coefficients that calculated by CFD method. Empirical correlations of the average Nusselt number were developed. From the results of the present study can be stated that it is possible, and for practical application acceptable, to use the expressed correlation in this work with high confirm for turbulent flow in the range of Reynolds number studied here for pipes. Also, the possibility of using a CFD method to evaluate the heat transfer coefficient for industrial systems with high accuracy.

International Journal of Engineering and Applied Sciences

www.eaas-journal.org

ACKNOWLEDGEMENT

The author would like to express his thanks to the lab. Technician, Eng. Fathi Abueggyla for his availability for help in the experimental work.

REFERENCES

1. F.W. Dittus, L.M.K.. Boelter, "Heat Transfer in Turbulent Pipe and Channel Flow". Engineering, University of California, Berkeley, vol.(2), 1930, pp. 443–461.
2. F.P. Incropera, D.P. DeWitt, "Fundamentals of Heat and Mass Transfer", 4th ed., John Wiley & Sons, USA, 1996.
3. J.P. Holman, "Heat Transfer", 8th edition, McGraw Hill, Singapore, 2001.
4. Q. Abbas, M.M. Khan, K. Sabir, Y.M. Khan, Z.U. Koreshi, "Numerical Simulation and Experimental Verification of Air Flow through a Heated Pipe", International Journal of Mechanical & Mechatronics Engineering, 10(2), 2010, pp. 7-12.
5. A.B. Jamuna, and V. Somashekar, "CFD Simulation and Experimental verification of Air Flow through Heated Pipe", IOSR Journal of Mechanical and Civil Engineering, 10(3), 2013, pp. 30-35.
6. B. Blocken, "Wind-driven rain on buildings"; Ph.D. thesis, Leuven: K.U. Leuven, 2004.
7. A. Neale, D. Derome, B. Blocken, J. Carmeliet, "CFD calculation of convective heat transfer coefficients and validation – Part 2: Turbulent flow", Annex 41– Kyoto, April 3rd to 5th, 2006.
8. V. Streeter, E.B. Wylie, "Fluid Mechanics", 1st ed., McGraw-Hill, Singapore, 1981.
9. W. Kays, M. Crawford, B. Weighand, "Convective Heat and Mass Transfer", 4th ed., McGraw-Hill International edition, Singapore, 2005.
10. A. Mosyak, E. Pogrebnyak, G. Hetsroni, "Effect of constant heat flux boundary condition on wall temperature fluctuations", ASME J. Heat Transfer,123 (2), 2005, pp. 213–218.
11. Q.T. Pham, "Explicit equations for the solution of turbulent pipe-flow problems", Trans I. Chem E., 57, 1979, pp. 281-283.
12. V. Gnielinski, "New equations for heat and mass transfer in turbulent pipe and channel flow", Int. Chem. Eng., 1, 1979, pp. 359-368.