RESEARCH ON THE DEVELOPMENT OF CONVECTION EMPIRICAL CORRELATION FOR FLUID FLOW THROUGH A TUBE ANNULUS BY SUPPORT OF CFD SIMULATION

Thai Ngoc Son*, Tran Thi My Linh

The University of Danang - University of Science and Technology, Vietnam

*Corresponding author: thnson@dut.udn.vn

(Received: April 13, 2025; Revised: June 03, 2025; Accepted: June 21, 2025)

DOI: 10.31130/ud-jst.2025.23(9D).574E

Abstract - The convection empirical correlation is a key concept in thermal engineering. To help students understand this concept, the authors used the ANSYS Fluent simulation tool (student version) to simulate a double-tube system with 100 input data points in the following ranges: Reynolds number $Re_f \in [10000; 18300]$, Prandtl number $Pr_f \in [3.4; 8.9]$, and $D/d_2 \in [2.00; 3.88]$. The output data includes the temperature of the flows at the outlets and the temperature of the wall, as well as experimental process data. The authors processed the simulation results and successfully derived the empirical correlation to determine the convection coefficient for forced convection turbulent flow through a tube annulus. When compared to some commonly used formulas - Mikheev, Petukhov, and Gnielinski formulas - the simulation errors fall within an acceptable range. Notably, for the Petukhov and Gnielinski formulas, the error is approximately 10%.

Key words - Criteria equation; simulation; coefficient of convection heat transfer; forced convection; tube annulus

1. Introduction

To help students better understand the method of developing the Nusselt number correlation, the Department of Heat and Refrigeration Engineering at the University of Danang – University of Science and Technology has developed a double-tube device and related experiments [1]. One of the challenges in conducting the experiments and processing the results is the temperature data of the wall. Since the experimental device is quite small and designed for educational purposes, it does not have the capability to accurately measure the wall temperature. Therefore, the authors proposed a method for calculating the wall temperature based on the assumption that the empirical correlation for fluid flow in an annular gap is accurate. From this, the wall temperature can be calculated using the heat balance equation and the empirical correlation for fluid flow in the annular tube [1].

Currently, the use of simulation software in research is quite common, and the field of heat transfer is no exception. To support teaching on developing Nusselt number correlation of convective heat transfer, we used the student version of ANSYS Fluent software to simulate the heat transfer process during turbulent fluid flow in a pipe, and then conducted experiments to verify the validity of the simulation. The results showed that the data from the simulation and the experiment were consistent, providing a basis to believe that the wall temperature predicted by the simulation is accurate to a certain degree of confidence. The following paper presents the use of simulation results to develop a Nusselt number correlation for fluid flow in an annular gap.

2. Research and Survey Results

2.1. Related works

Recently, many authors have used Computational Fluid Dynamics (CFD) to study heat transfer processes, especially radiation propagation processes. The paper [2] offers an overview of CFD simulation of heat transfer processes in fire situations. One of the reasons for using CFD is that real experiments are very costly, geometrically complex, and difficult to analyze using analytical methods or regional models. The paper reviews the 'Law of the Wall,' a method that uses wall functions to simulate convective heat transfer near surfaces in fire situations, where there is a sudden transition of heat transfer from molecular to turbulent processes near the surface, without having to resolve the viscous boundary layers. This is an important approach, especially when using a very fine mesh is not feasible. Some experimental comparisons with CFD simulations indicate that while small-scale experiments are often used for CFD validation, they may not always represent real-world conditions accurately.

The paper [3] examines the use of CFD to study the radiative effects of large-scale liquefied petroleum gas (LPG) pool fires, thereby quantifying the safety distance between LPG facilities and surrounding objects.

Another related work, [4], reviews recent advancements in simulating thermal radiation in fire environments, highlighting the application of modern non-gray models and methods for solving the radiative transfer equation (RTE) in flames, including combustion furnaces and mixed fires. This helps establish empirical correlations to describe heat dissipation processes in these environments.

Overall, modern research focuses on the application of simulations in complex heat transfer problems. For higher education purposes, [1] presents experiments and guides students in constructing Nusselt number correlations of convective heat transfer, for the simpler case of fluid flow in a tube. However, as explained in [2], it is not possible to construct many experimental models with different qualitative sizes. Furthermore, due to limitations in measuring instruments, as presented in [1], we are researching the development of convection Nusselt number correlation for fluid flow through a tube annulus with the support of CFD simulations.

2.2. Introduction to the experimental model

The experimental setup consists of a double-tube model (Figure 1), with water serving as the working fluid. Hot water flows through the inner tube, while cold water flows countercurrently in the annular gap. The outer surface of the large tube is ideally insulated. In this system, the heat transfer surface is the inner surface of the small tube, with a length of L.

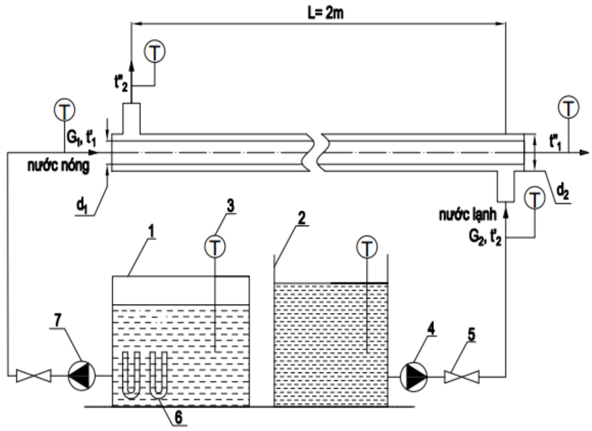

Figure 1. *Experimental model*

This experimental setup is relatively simple and forms part of the larger system depicted in Figure 2, consisting of: an inner stainless steel tube with a diameter ratio of d_2/d_1 = 17/15 mm, an outer stainless steel tube with an inner diameter of D = 34 mm; the heat transfer surface length L = 2 m. All sections of the tubes, as well as the water tank, are well insulated

Figure 2. *Experimental device*

The experiment is carried out in the following steps: Prepare hot water in tank 1 and cold water in tank 2, ensuring the required input temperatures for the fluids in the experiment. Turn on the hot water pump 7 and the cold water pump 4. Adjust the flow rate by gradually closing the valves step by step. After approximately 5 minutes, when the system stabilizes thermally, measure the temperatures of the hot water at the inlet t_1' and outlet t_1'', and the cold water at the inlet t_2' and outlet t_2'' using the TA-288 electronic thermometer with a resolution of 0.1 °C. The flow rate of the fluids is determined using an electronic flowmeter G1/2, with units in l/min (liters per minute), displayed to the nearest percent.

Each experiment is conducted with a minimum of 3 measurements, with each measurement separated by 2-3 minutes. The data for calculation is the average value based on the number of measurements. This experimental model was similarly used in [1]. In this study, the model serves to test the compatibility between simulation results and experimental data.

2.3. Simulation of the heat transfer process in the annular tube

The experimental model described above was simulated using the student version of ANSYS Fluent, which is capable of modeling both the fluid's mechanical properties and heat transfer. The physical properties of the fluid and the wall material depend on temperature

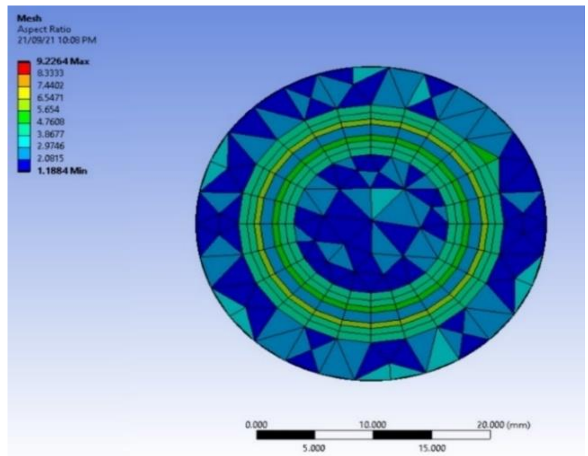

Figure 3. *Mesh at the inlet cross-section of the model*

The model was designed using ANSYS Design Modeler, and the meshing was performed using ANSYS Meshing. The Edge Sizing method was applied, dividing the circumference of the inner tube into 18 parts and the outer tube into 30 to 40 parts. To handle the boundary layer of the heat transfer process between the two fluids, the Inflation method was applied to both the inner and outer surfaces of the inner tube, each with three layers and a Growth Rate of 1.2. The bodies, including hot and cold fluids and the inner tube, are automatically meshed with an element size of approximately 20 mm along the vertical direction. The total number of elements did not exceed 512000, in compliance with the limitations of the student version of ANSYS Fluent.

The simulation problem was solved using the realizable k-ε turbulence model, the energy equation, and temperature-dependent fluid and tube physical properties. The residuals for all equations set to 10^{-6}. The inlet boundary for the fluid is defined as Mass Flow Rate, with specifications for flow rate, temperature, and flow direction. The outlet boundary was defined as Pressure outlet, with default settings that do not affect the heat transfer results in the tube. The outer wall of the tube (with outer diameter D was set in thermal boundary condition mode with a Heat flux of 0 [5].

The input data for the simulation includes:

- The geometric parameters of the model are as follows: The length of the device is L = 2 m; the inner tube has a diameter ratio of d_2/d_1 = 17/15 mm, and the outer cylindrical surface, co-axial with the inner tube, has diameters D ∈ {34; 42; 50; 58; 66} mm, resulting in a diameter ratio of $D/d_2 \in [2.00; 3.88]$.

- Physical properties of the fluid: Both the hot and cold fluids are water. We used the Water-IF 97 software [6] to determine physical properties such as specific heat capacity C_p, density ρ, thermal conductivity λ, and dynamic viscosity μ of water; the specific heat capacity C_p, density ρ, and thermal conductivity λ of the heat transfer surface material were referenced from [7]. All the aforementioned physical properties inputted into the simulation software depend on temperature.

- The inlet temperature t_1' and mass flow rate G_1 of the hot water; the inlet temperature t_2' and mass flow rate G_2 of the cold water. The outer surface of the outer tube is considered to be ideally insulated.

The simulation results are the outlet temperature of the hot fluid stream t_1'', the outlet temperature of the cold fluid stream t_2'', and the average temperature of the outer surface of the inner tube t_w. These temperature values are determined using the area-weighted average temperature function at the corresponding surfaces (areaAve (Temperature) @hot_outlet; areaAve (Temperature) @cold_outlet; areaAve (Temperature) @wall_hot_fluid_inner_pipe).

Since only one experimental model was available, the simulation results for the case where D/d_2 = 34/17 mm were compared with experimental data.

Table 1 presents some experimental and CFD simulation results. The parameters of the hot flow have the subscript 1; the parameters of the cold flow have the subscript 2.

Table 1. *Experimental and CFD simulation data of the double-tube case with D/d_2 = 34/17 mm*

№	Input parameters				Output parameters	
	$G_1.10^3$, kg/s	t_1', ºC	$G_2.10^3$, kg/s	t_2', ºC	t_1'', ºC	t_2'', ºC
Experimental data						
1	110	79.1	214	50.5	72.6	54.0
2	139	93.2	331	33.8	80.6	39.4
3	171	83.2	403	40.8	74.0	45.1
4	230	85.4	503	9.8	72.0	15.7
CFD simulation data						
1	110	79.1	214	50.5	72.8	53.7
2	139	93.2	331	33.8	80.2	39.2
3	171	83.2	403	40.8	73.5	44.9
4	230	85.4	503	9.8	71.7	16.1

Table 2. *Enthalpy change per unit time of the hot and cold fluids through experiments and CFD simulation calculations*

№	ΔI_1^{exp}, W	ΔI_2^{exp}, W	ΔI_1^{CFD}, W	ΔI_2^{CFD}, W
1	2990	3129	2898	2861
2	7371	7739	7593	7509
3	6594	7241	6952	6904
4	12913	12435	13202	13277

Table 2 presents the calculated results of enthalpy change per unit time for the hot fluid ΔI_1^{exp} and the cold fluid ΔI_2^{exp} from the experiment; and the enthalpy change per unit time for the hot fluid ΔI_1^{CFD} and the cold fluid ΔI_2^{CFD} from the CFD simulation.

The error in the enthalpy change per unit time of the hot and cold fluids in the experiment is determined by the formula:

$$\varepsilon_{\Delta I^{exp}} = \frac{\left|\Delta I_1^{exp} - \overline{\Delta I}^{exp}\right|}{\overline{\Delta I}^{exp}} \tag{1}$$

where the average enthalpy change of the two fluids $\overline{\Delta I}^{exp}$ is determined by the formula:

$$\overline{\Delta I}^{exp} = \frac{\Delta I_1^{exp} + \Delta I_2^{exp}}{2} \tag{2}$$

The value of $\varepsilon_{\Delta I^{exp}}$ can also be calculated by the formula:

$$\varepsilon_{\Delta I^{exp}} = \frac{\left|\Delta I_2^{exp} - \overline{\Delta I}^{exp}\right|}{\overline{\Delta I}^{exp}} \tag{3}$$

The error $\varepsilon_{\Delta I^{exp}}$ indicates the accuracy of the experiment. When the experimental device in Figure 2 is ideally insulated and thermally stable, then: $\varepsilon_{\Delta I^{exp}} \to 0$

Similarly, the error in the enthalpy change per unit time of the hot and cold fluids through CFD simulation is determined by the formula:

$$\varepsilon_{\Delta I^{CDF}} = \frac{\left|\Delta I_1^{CFD} - \overline{\Delta I}^{CFD}\right|}{\overline{\Delta I}^{CFD}} \tag{4}$$

$$\overline{\Delta I}^{CFD} = \frac{\Delta I_1^{CFD} + \Delta I_2^{CFD}}{2} \tag{5}$$

The error $\varepsilon_{\Delta I^{CFD}}$ indicates the convergence of the simulation. When the outer surface of the large pipe is a thermal boundary, the simulation results converge, then $\varepsilon_{\Delta I^{CFD}} \to 0$.

The error $\varepsilon_{\Delta I^{exp/CDF}}$ is determined by the formula:

$$\varepsilon_{\Delta I^{exp/CDF}} = \frac{\left|\overline{\Delta I}^{exp} - \overline{\Delta I}^{CFD}\right|}{\overline{\Delta I}^{CFD}} \tag{6}$$

The error $\varepsilon_{\Delta I^{exp/CDF}}$ indicates the level of consistency between the experiment and the CFD simulation results. The smaller the error, the higher the level of consistency between the experiment and the CFD simulation results.

Table 3 shows the error in the enthalpy change of the two fluids in both the experiment and the simulation

Table 3. *The error in the enthalpy change of the two fluids in both the experiment and the simulation*

№	$\varepsilon_{\Delta I^{exp}}$, %	$\varepsilon_{\Delta I^{CDF}}$, %	$\varepsilon_{\Delta I^{exp/CDF}}$, %
1	2.28	0.64	6.25
2	2.44	0.56	0.05
3	4.67	0.35	0.15
4	1.89	0.28	4.27

From Table 3, the error in the enthalpy change of the hot and cold fluids in the experiments $\varepsilon_{\Delta I^{exp}}$ is less than

4,7%, indicating that the experimental measurements are consistent and reliable. The error in the enthalpy change of the hot and cold fluids from the simulation results $\varepsilon_{\Delta I^{CDF}}$ is less than 1%, suggesting that the CFD model has converged well and provides accurate results. The error in the enthalpy change of the hot and cold fluids between the experimental and simulation results $\varepsilon_{\Delta I^{exp/CDF}}$ is less than 6,3%, indicating that the simulation model is consistent with the experimental model. Therefore, the simulation model can be used confidently for further analysis of heat transfer in the annular gap.

2.4. Development of the convection empirical correlation for heat transfer process in the annular gap from simulation results

The empirical correlation for convection heat transfer in the annular gap is of the form:

$$Nu_f = C.Re_f^n.Pr_f^m.\left(\frac{D}{d_2}\right)^p.\left(\frac{Pr_f}{Pr_w}\right)^{0,25} \quad (7)$$

Table 4. *Some input and output parameters of the double-tube simulation case with D/d2 = 34/17 mm*

№	Input parameters				Output parameters		
	$G_1.10^3$, kg/s	t_1', °C	$G_2.10^3$, kg/s	t_2', °C	t_1'', °C	t_2'', °C	t_w, °C
1	114	90	487	10	63.1	16.3	43.3
2	114	90	378	20	67.6	26.8	51.5
3	114	90	303	30	71.5	36.9	58.6
4	114	90	251	40	75.2	46.7	65.1
5	114	80	214	50	71.5	54.5	65.6
16	222	90	868	10	68.0	15.6	42.9
17	222	90	668	20	71.6	26.1	51.1
18	222	90	535	30	74.9	36.2	58.3
19	222	90	444	40	77.8	46.1	64.6
20	222	90	377	50	80.6	55.5	70.5

It is necessary to determine the values of C, m, n, p from the set of standard numbers Nu_f, Re_f, Pr_f, Pr_w, and the diameter ratio D/d_2.

From the input - output data of the simulation case (Table 4), the average temperature of the cold fluid is determined using the following formula:

$$t_f = 0.5(t_2' + t_2'') \quad (8)$$

That is also the temperature that allows the determination of necessary physical properties such as specific heat capacity C_p, density ρ, thermal conductivity λ, kinematic viscosity ν, as well as the Prandtl number Pr_f of water.

The Prandtl number Pr_w is determined from the average temperature of the outer surface of the inner tube t_w based on the simulation results.

The characteristic size d_{td} is determined by the following formula:

$$d_{td} = D - d_2 \quad (9)$$

The flow velocity ω is determined by:

$$\omega = \frac{4G_2}{\rho\pi(D^2 - d_2^2)} \quad (10)$$

The Reynolds number Re_f is determined by the following formula:

$$Re_f = \frac{\omega d_{td}}{\nu_f} \quad (11)$$

From the inlet and outlet temperature values t_2' and t_2'' of the cold water, the enthalpy change ΔI is calculated:

$$\Delta I = G_2 C_{p2}(t_2'' - t_2') \quad (12)$$

According to the heat balance equation, the enthalpy change of the cold fluid equals the heat flow from the wall. Since the outer surface of the tube is ideally insulated, the heat flow from the inner surface of the tube to the cold fluid is determined by Newton's law of cooling, as follows:

$$\Delta I = Q = L\pi d_2 \alpha(t_w - t_f) \quad (13)$$

This allows for the determination of the convection heat transfer coefficient α from the wall to the cold fluid. The Nusselt number Nu_f is determined:

$$Nu_f = \frac{\alpha d_{td}}{\lambda_f} \quad (14)$$

Thus, we have the set of standard numbers Re_f, Pr_f, Pr_w, and Nu_f as shown in Table 5.

Table 5. *The dimensionless numbers of the double-tube simulation case with D/d₂ = 34/17 mm*

№	Nu_f	Re_f	Pr_f	Pr_w	D/d_2
1	114.8	10172	8.53	4.05	2.00
2	99.9	10208	6.39	3.47	2.00
3	89.3	10210	5.00	3.06	2.00
4	81.5	10202	4.05	2.76	2.00
5	74.7	10135	3.42	2.74	2.00
16	183.8	17966	8.62	4.09	2.00
17	159.5	17921	6.44	3.49	2.00
18	142.7	17886	5.04	3.08	2.00
19	131.3	17967	4.08	2.78	2.00
20	122.0	18017	3.39	2.54	2.00

Similarly, we have the dimensionless number tables for cases where the diameter d_2 = 17 mm, and D ∈ {42; 50; 58; 66} mm. In total, we have 100 input data points.

To determine the values of C, m, n, p in (8), we can rearrange the formula (8) into the following form:

$$Nu_f.\left(\frac{Pr_f}{Pr_w}\right)^{-0.25} = C.Re_f^n.Pr_f^m.\left(\frac{D}{d_2}\right)^p \quad (15)$$

We denote Nu^* as follows:

$$Nu^* \triangleq Nu_f\left(\frac{Pr_f}{Pr_w}\right)^{-0.25} \quad (16)$$

Expression (16) is transformed into:

$$ln(Nu^*) = lnC + nln(Re_f) + mln(Pr_f) + pln\left(\frac{D}{d_2}\right) \quad (17)$$

Thus, the problem of determining correlation (8) becomes a multivariate linear regression problem:

$$Y = A + B_1X_1 + B_2X_2 + B_3X_3 \quad (18)$$

This problem can be solved using the Data Analysis tool in Excel, with the corresponding columns between (17) and (18). A total of 100 data points are used, with a portion shown in Table 6.

Table 6. *Multivariate linear regression data*

№	Y=ln(Nu*)	X_1=ln(Re_f)	X_2=ln(Pr_f)	X_3=ln(D/d_2)
1	4.56	9.23	2.14	0.69
2	4.45	9.23	1.85	0.69
3	4.37	9.23	1.61	0.69
4	4.30	9.23	1.40	0.69
5	4.26	9.22	1.23	0.69
26	4.73	9.44	2.16	0.90
27	4.63	9.44	1.87	0.90
28	4.55	9.44	1.63	0.90
29	4.48	9.44	1.41	0.90
30	4.45	9.44	1.24	0.90
96	5.07	9.81	2.21	1.36
97	4.97	9.81	1.91	1.36
98	4.90	9.82	1.66	1.36
99	4.84	9.82	1.44	1.36
100	4.77	9.82	1.25	1.36

The regression results are presented in Table 7 and 8.

The regression statistics (Table 7) show a very high model fit with the data ($R^2 \approx 1$), and the standard error is 0.015.

Table 7. *Statistics of multivariate linear regression*

Regression Statistics	
Multiple R	0.9973358
R Square	0.9946786
Adjusted R Square	0.9945123
Standard Error	0.0149794
Observations	100

Table 8. *Results of multivariate linear regression*

	Coefficients	Standard Error
Intercept	-3.3705772	0.066543
X Variable 1	0.7797077	0.006933
X Variable 2	0.309825	0.00451
X Variable 3	0.0859448	0.006401

From the regression results (Table 8), considering the correspondence between the formulas (8), (17), (18), we have empirical correlation of convection heat transfer for turbulent fluid flow in the annular gap within the range $Re_f \in [10000;\ 18300]$, $Pr_f \in [3.4;\ 8.9]$, $D/d_2 \in [2.00;\ 3.88]$, developed from the simulation results as follows:

$$Nu_f = 0.0343.Re_f^{0.78}.Pr_f^{0.31}.\left(\frac{D}{d_2}\right)^{0.086}.\left(\frac{Pr_f}{Pr_w}\right)^{0.25} \quad (19)$$

2.5. Comparison of the empirical correlation developed from simulation results and reference literature

The empirical correlation developed from the simulation results was compared with widely-used formulas in the field of thermal engineering, including those by Isatshenko, Mikheev, Petukhov, Avchukhov and Gnielinski. These formulas are typically used to calculate the Nusselt number of convective heat transfer for fluid flow in an annular gap, especially when the outer surface of the tube is ideally insulated.

According to Isatshenko V.P. [8], the Nusselt number of convective heat transfer is given by:

$$Nu_f = 0.017.Re_f^{0.8}.Pr_f^{0,4}.\left(\frac{D}{d_2}\right)^{0.18}.\left(\frac{Pr_f}{Pr_w}\right)^{0.25} \quad (20)$$

According to Yunus Cegel, the Nusselt number of convective heat transfer for fluid flow in the annular gap is calculated using the formula for fluid flow in a pipe, multiplied by a correction factor proposed by Petukhov B.S. and Roizen L.I. to account for the effect of the D/d_2 ratio, as follows [9]:

$$F_i = 0.86.\left(\frac{D}{d_2}\right)^{0.16} \quad (21)$$

The standard convection heat transfer equation for fluid flow in a pipe has three common forms: Mikheev, Petukhov, and Gnielinski. To calculate the convection heat transfer coefficient for turbulent fluid flow in a pipe, technical thermal engineering references typically provide the Mikheev equation [10, 11] as follows:

$$Nu_f = 0.021Re_f^{0.8}Pr_f^{0.43}\left(\frac{Pr_f}{Pr_w}\right)^{0.25} \quad (22)$$

The more complex Petukhov equation is mentioned in several references [12 - 15]:

$$Nu_f = \frac{\frac{\xi}{8}Re_f.Pr_f\psi}{1 + \frac{900}{Re_f} + 12.7\sqrt{\frac{\xi}{8}}\left(Pr_f^{\frac{2}{3}} - 1\right)} \quad (23)$$

The Gnielinski formula [12], a modern and widely adopted equation, evaluates the Nusselt number in a more accurate way for a range of Reynolds and Prandtl numbers. The equation is:

$$Nu_f = \frac{\frac{\xi}{8}\left(Re_f - 1000\right)Pr_f\psi}{1 + 12.7\sqrt{\frac{\xi}{8}}\left(Pr_f^{\frac{2}{3}} - 1\right)} \quad (24)$$

In formulas (23) and (24), ψ is the coefficient determining the effect of the heat flow direction; ξ is the frictional resistance coefficient calculated by:

$$\xi = \left(1.82lgRe_f - 1.64\right)^{-2} \quad (25)$$

According to Avchukhov V.V., the Nusselt number (Nu) for fluid flow in the annular gap is calculated using the Mikheev formula with a correction factor to account for the effect of the D/d_2 ratio [16]:

$$F_i = \left(1 - \frac{0.45}{2.4 + Pr_f}\right).\left(\frac{D}{d_2}\right)^{\zeta} \quad (26)$$

where

$$\zeta = 0.16.Pr_f^{-0,15} \quad (27)$$

We calculated the Nusselt number values, ignoring the direction of heat flow, based on the Isachenko formula

Nu^I; the Mikheev formula Nu^M with the correction factor by Petukhov; the Petukhov formula Nu^P; the Gnielinski formula Nu^G; the correction factor by Avchukhov Nu^A; and the empirical correlation we developed Nu^{CFD}. The input values are within the ranges $Re_f \in [10000; 18300]$, $Pr_f \in [3.4; 8.9]$, $D/d_2 \in [2.00; 3.88]$, with specific values of $Re_f \in \{10000; 12800; 15600; 18300\}$, $Pr_f \in \{3.4; 5.1; 6.8; 8.5\}$, $D/d_2 \in \{2.0; 2.4; 2.8; 3.3; 3.8\}$, totaling 80 calculation nodes. Table 9 shows the results for 10 of the 80 calculation nodes.

Table 9. *The Nusselt number values calculated using the formulas for 10 out of the 80 calculation nodes*

№	Re_f.	Pr_f	D/	Nu^{CFD}	Nu^I	Nu^M	Nu^P	Nu^G	Nu^A
1	10	3,4	2,0	70	50	54	61	58	57
2	10	5,1	2,0	79	59	67	73	68	69
3	10	6,8	2,0	87	66	76	81	75	78
4	10	8,5	2,0	93	72	84	89	82	87
5	15,6	6,8	2,8	126	100	108	125	119	117
6	15,6	8,5	2,8	136	109	119	137	130	129
7	18,3	3,4	3,3	117	88	91	110	106	99
8	18,3	5,1	3,3	133	104	109	130	126	119
9	18,3	6,8	3,8	147	120	123	150	144	137
10	18,3	8,5	3,8	158	131	135	164	157	152

Table 10 shows the results of the error calculation for the Nusselt number based on the simulation, compared with the Nusselt number according to the formulas mentioned above. It can be observed that the simulation and calculation results are relatively consistent with the Petukhov, Gnielinski, and Avchukhov formulas, with an average error of approximately 10%. This is particularly significant given that the Gnielinski formula is a modern formula, referenced in [12] in 2020, while the formula by Isatshenko and Mikheev appeared much earlier. This suggests that the simulation results and the methodology used to derive the empirical correlation of convection heat transfer presented in the paper are reasonable.

Table 10. *Summary of the errors for the 80 calculation nodes*

Error	Nu^I	Nu^M	Nu^P	Nu^G	Nu^A
Average	0.30	0.20	0.05	0.10	0.12
Maximum	0.41	0.31	0.14	0.22	0.23
Minimum	0.20	0.10	0.00	0.00	0.04

3. Conclusion

The authors conducted simulations of the double-tube model for heat exchange between two fluids and validated the basic simulation results with those obtained from the experimental model. The simulation results were processed to develop the standard convection heat transfer equation for the case of fluid flow in an annular tube within the ranges $Re_f \in [10000; 18300]$, $Pr_f \in [3.4; 8.9]$, $D/d_2 \in [2.00; 3.88]$. Comparing with commonly used formulas in the field of Thermal Engineering shows that the error is acceptable. The simulation model can be confidently used for determining convection heat transfer coefficients in cases where experimental measurements are difficult or impractical

The research results can be incorporated into the teaching of the Heat Transfer, Heat Exchange Equipment, and Computational Fluid Dynamics courses in the Thermal Engineering program, as well as in the Thermal Engineering courses for engineering programs across the University of Danang – University of Science and Technology.

REFERENCES

[1] T T. N. Son and B. T. H. Lan, "Manufactoring the experimental equipment to determine coefficient in the criteria equation of stable forced convection in tubes", *The University of Danang - Journal of Science and Technology*, vol. 11, no. 108, pp. 37-41, 2016.

[2] G. Maragkos and T. Beji, "Review of Convective Heat Transfer Modelling in CFD Simulations of Fire-Driven Flows", *Applied Sciences*, vol. 11, no. 11, p. 5240, 2021. https://doi.org/10.3390/app11115240.

[3] Y. Hang, Y. Feng, and Q. Wang, "Computational fluid dynamics (CFD) study of heat radiation from large liquefied petroleum gas (LPG) pool fires", *Journal of Loss Prevention in the Process Industries*, vol. 61, pp. 262-274, 2019. https://doi.org/10.1016/j.jlp.2019.06.015.

[4] S. Mazumder and S. P. Roy, "Modeling Thermal Radiation in ombustion Environments: Progress and Challenges", *Energies*, 16, no. 10, p 4250, 2023. https://doi.org/10.3390/en16104250.

[5] J. Tu, G. Yeoh, and C. Liu, *Computational Fluid Dynamics: A Practical Approach,* 2nd edition. MA: Elsevier, 2013.

[6] T. N. Son, "*Develop a Program to Calculate the Technical Properties of Water and Steam*", research project report, Basic-Level Science and Technology Research Project, The University of Danang - University of Science and Technology, Vietnam, T2010-02-86, 2010.

[7] V. A. Grigoriev and V. M. Zorkin, *Theoretical Fundamentals of Heat Engineering - Heat Engineering Experiment: Handbook,* 2nd edition. Moscow: Energoatomizdat, 1988.

[8] V. P. Isatshenko, V. A. Osipova, and A. S. Sukomel, *Heat Transfer,* 4th edition. Moscow: Energoatomizdat, 1981.

[9] Y. A. Cengel, *Heat Transfer: A Practical Approach*, 2nd edition. NY: McGraw-Hill, 2003.

[10] V. L. Erofeev, P. D. Semenov, and A. S. Pryakhin, *Thermal Engineering*. Moscow: Academkniga, 2008.

[11] T. N. Son, H. N. Hung, M. P. Hoang, L. T. C. Duyen, and B. T. H. Lan, *A Heat Transfer Textbook.* Hanoi: Science and Technics Publishing House, 2023.

[12] J. H. Lienhard IV and J. H. Lienhard V, *A Heat Transfer Textbook*, 5th edition. Massachusetts: Phlogiston Press, 2020.

[13] B. M. Khrustalev *et al.*, *Heat and Mass Transfer*. Minsk: BNTU, 2007.

[14] A. Bejan and A. D. Kraus, *Heat Transfer Handbook*. NJ: John Wiley & Sons, 2003.

[15] D. Q. Phu, T. T. Son, and T. V. Phu, *Heat Transfer, 3rd edition*. Vietnam Education Publishing House, 2007.

[16] V. V. Avtshukhov and B. Y. Paiuste, *Problem Book on Heat and Mass Transfer Processes*. Moscow: Energoatomizdat, 1986.