CHEMICAL ENGINEERING TRANSACTIONS
VOL. 88, 2021
A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.cetjournal.it
Guest Editors: Petar S. Varbanov, Yee Van Fan, Jiří J. Klemeš
Copyright © 2021, AIDIC Servizi S.r.l.
ISBN 978-88-95608-86-0; **ISSN** 2283-9216
DOI: 10.3303/CET2188207

Validation of CFD Models for Double-Pipe Heat Exchangers with Empirical Correlations

Rubén Cabello*, Alexandra Elena Plesu Popescu, Jordi Bonet-Ruiz, David Curcó Cantarell, Joan Llorens

University of Barcelona, Faculty of Chemistry, Department of Chemical Engineering and Analytical Chemistry, c/ Martí i Franquès 1, 6th Floor, 08028 Barcelona, Spain
ruben.cabello@ub.edu

Heat exchangers are present in most industries as they allow the recovery of thermal energy, making processes more profitable and environmentally sustainable. The design of classical heat exchangers is performed based on well-established semi-empirical equations, e.g. Double Pipe Heat Exchangers (DPHE). Those semi-empirical equations are very susceptible to geometry changes, as well as flow conditions at the entrance of the heat exchangers. Semi-empirical correlations are based on simplifying assumptions such as constant fluid properties along the heat exchanger. A design solution to this issue is Computational Fluid Dynamics (CFD), which is widely used for the design of novel heat exchanger geometries saving experimentation time and cost. To the best of our knowledge, CFD results have not been tested against the classical semiempirical equations. Before facing future complex geometries, the proper CFD model options are identified and validated against classical equations. In this study, this analysis is performed for the case of a classical DPHE. Series of CFD simulation results are critically compared with classical for DPHE. Different turbulence models and wall treatments are tested, as well as fully developed velocity profiles at the entrance of the heat exchangers, obtained by iteration. Results show that CFD codes are able to predict and model the flow in DPHE reliably, with the same accuracy as classical correlations. CFD simulations are found to differ from semi-empirical correlations up to 0.87 % in the case of pressure drop and 7.1 % in the case of heat transfer. Consequently, CFD simulations with proper setup parameters are a good choice for assessing novel configurations. Reliable novel heat exchange configurations assessment is compulsory for heat exchange network revamping in restricted space industrial scenarios and opens novel opportunities for higher heating and cooling services savings.

1. Introduction

Double pipe heat exchangers (DPHE) are a typical configuration present in many industries. Many classical references in literature provide semi-empirical models that predict accurately its performance by estimating the pressure drop and the heat exchanged. For instance, the handbook by Sinnott and Towler (2012) provides reliable design tools for industrial practice. Those semi-empirical equations are very susceptible to subtle changes in the geometry, or operating conditions that differ from the ones assumed during the realization of the experiments. Computational Fluid Dynamics (CFD) is a tool that has been gaining popularity and as such it has suffered a rapid development in the last decade. RANS (Reynolds Averaged Navier-Stokes) methods are one of the most widely used for engineering applications, providing a balance between precision and computational time. From the literature reviewed, some authors compare CFD results with novel heat exchangers configuration experimental results, but for classical DPHE the CFD results are blindly trusted or directly rely on the well-established semi-empirical handbook equations. CFD simulations are based on numerically solving microscopic mass and energy balances (conservation principles) together with a viscosity model, the uncertainty of the results mainly is caused by the viscosity models. There are some studies that check which is the most suitable viscosity model for different units but not for DPHE. For example, different viscosity models are tested by Ajmi et al. (2020) when investigating the heat transfer enhancement of an inclined heated offset jet. Their results show that the SST k-ω model is the one that better predicts the Nusselt number. Abbasian Arani et al. (2021) found good agreements between their results using the k-ε model and analytical methods when studying single

Paper Received: 29 June 2021; Revised: 10 September 2021; Accepted: 6 October 2021
Please cite this article as: Cabello R., Plesu-Popescu A.E., Bonet-Ruiz J., Cantarell D.C., Llorens J., 2021, Validation of CFD Models for Double-Pipe Heat Exchangers with Empirical Correlations, Chemical Engineering Transactions, 88, 1243-1248 DOI:10.3303/CET2188207

and double-pass tube with helical and segmental baffles. Gavane et al. (2019) studied shell and tube heat exchangers with Realizable k-ε model and found up to 19.44 % difference in heat transfer and 13.31 % difference in pressure drop with respect to analytical methods. Petrik et al. (2018) used the Realizable k-ε model to compare empirical and CFD results in the shell side of a Shell-and-Tube Heat Exchanger. Their results found errors in heat transfer that ranged from 2.84 % to 18.47 % depending on the case studied. Sruthi et al. (2021) used the k-ε model to model a DPHE with corrugations, obtaining discrepancies with experimental studies of up to 8.42%. Pal et al. (2016) tested different turbulence models with different heat exchanger configurations, founding that Standard k-ε model was the one that fits better the experimental results (±20 % error) with less computational time compared. Their research, however, didn't take into account different wall treatments for the turbulence models, nor their combination between themselves. The aim of this project is to explore which turbulence models combined with different wall treatments describe more accurately heat transfer and pressure drop when compared against well-established semi-empirical models applied to a case with simple geometry. The novelty of the project comes from the study of the combination of the more relevant turbulent models and wall treatments.

2. Method

2.1 Simulation set up

The dimensions of the DPHE used in this study are shown in Table 1. The simplest case of this heat exchanger is two concentric tubes which share the inner pipe wall, made of a conductive material. The geometry is created by using ANSYS® SpaceClaim.

Table 1: Heat-exchanger dimensions

	Parameter	Value (cm)
Inner pipe	Inside radius (r_i)	1.425
	Outside radius (r_o)	1.510
Outer pipe	Inside radius (R_i)	2.115
	Outside radius (R_o)	2.225
	Pipe length (L)	100-1,000

All simulations are performed using ANSYS® Fluent, all codes used are found within the program: turbulence models and the energy equation. The solution algorithm chosen for these simulations is SIMPLE. Both fluid and solid zones are discretized, the turbulence models are used only in fluid zones while the energy equation is used for the whole geometry. Several meshes are created decreasing cell size, until the results do not depend on the mesh. This operation is performed once, and it is considered valid for all the methods inside its family e.g. the mesh optimization performed for k-ε standard model is considered valid for the k-Ɛ RNG and k-Ɛ Realizable models. Methods are tested with a finer mesh than the results obtained by mesh optimization to ensure no grid dependence.
The DPHE works in a parallel disposition. The operating conditions of the unit are shown in Table 2. In order to account for the difference in temperature that the fluids (water in both pipes) experiences, all properties except thermal capacity (which is fairly constant in the range of study) are dependent on temperature by using an interpolating polynomial.

Table 2: Operating conditions

	Parameter	Value
Inner pipe	Inlet velocity	0.5 m/s
	Inlet temperature	363.15 K
Outer pipe	Inlet velocity	0.5 m/s
	Inlet temperature	288.15 K

To ensure fully developed profiles, some iterations of the same tube are performed by replacing the inlet velocity value (given only in the first iteration) with the velocity profile obtained at the output of the pipe. The same procedure is performed at the outlet, where the pressure profile found in the inlet is then exported for the pressure outlet. The values of pressure drop are then estimated by area-averaging the pressure at inlet and outlet. Different turbulence models with different wall treatments are used in the simulations (Table 3). Each

turbulence model is tested with each wall treatment inside the same family. Some wall treatments are avoided for the k-ε such as Standard Wall Functions or Enhanced Wall Treatment, the first for not being a Mesh-insensitive treatment and the second for causing solution instability when the turbulent Reynolds number is around 200 at the boundary layer (Ansys, 2021). All simulations are tightly converged to a continuity and other residual values of at least 10^{-5} with the exception of the mesh optimization test with fewer nodes, that are impossible to converge and stabilized around values of 2~4 $\cdot 10^{-5}$.

Table 3: Turbulence models and wall treatments tested for the simulations

Family	Turbulence model	Treatment
k-ε	Standard	Scalable Wall Functions
	RNG	Menther-Lechner
	Realizable	
k-ω	SST	Default treatment
		Low-Re corrections

2.2 Semi-empirical models

The CFD simulations are compared with classical semi-empirical correlations. Those methods take into account the hydraulic diameter of the annular tube (outer tube of the heat exchanger) for the estimation of its pressure drop. Rothfus et al. (1950) proved that these methods tend to underestimate the Fanning Friction Factor of the system and proposed the use of a correction factor:

$$f_2 = \frac{R_i^2 - r_m^2}{R_i(R_i - r_o)} f \quad (1)$$

where r_m is the radius at which the velocity is maximal.

3. Mesh optimization results

The mesh optimization was performed by using the k-ε Standard with Menther-Lechner treatment for the k-ε family and the k-ω SST with default configuration for the k-ω family. For the optimization, various parameters are changed: the number of divisions around the perimeter of the pipes, the inflation layers as well as its growth rate, and the element size of the nodes. Table 4 shows the different meshes tested for the k-ε model, similar configurations are tested for the k-ω model. The computational time has to be taken into consideration when selecting the mesh. The final mesh obtained is the one providing more accuracy with the fewer number of nodes possible. After performing the simulations, the clue values such as pressure drop and Output Temperature are extracted and compared. The pressure drop for the k-ω SST model are shown in Figure 2.

Table 4: Mesh configurations tested for the k-ε model

Number of divisions	Element Size (m)	Inflation layers	Growth rate	Number of Nodes
50	0.006	5	1.2	1,003,879
150	0.003	5	1.1	2,322,692
200	0.002	5	1.1	3,974,133
200	0.002	10	1.1	5,402,484
200	**0.002**	**12**	**1.1**	**6,669,440**
235	0.00175	10	1.1	7,073,405
200	0.002	15	1.1	7,710,581
250	0.0015	10	1.1	8,583,650

With the data obtained from the mesh optimization (Figure 1), it has been concluded that 200 divisions, element size of 0.002, 12 inflation layers and growth rate of 1.1 are sufficient for the set purpose. The considered mesh for the following simulations is the one with $6.7 \cdot 10^6$ nodes highlighted in Table 4. Figure 1 shows that although there is a trend to results stabilization with increasing number of nodes, different number of inflation layers can give very different results due to the difference in orthogonal quality between them.

4. Results and discussion

When conducting the simulations, the temperature profile is assumed to develop rapidly along the tube, and that a constant temperature profile at the inlet should not have a great effect on the overall heat transfer coefficient (U). To prove this hypothesis, a series of simulations are performed, where fully developed temperature profiles extracted as a result from previous iterations are used as input values in the inlets of the simulations. Figure 2 shows the results comparing, three cases for the k-ε RNG with scalable wall functions: first simulation (1st) with constant profiles on inlets and outlet, second one (2nd) with fully developed profiles of velocity and pressure at the inlet and outlet, and the third (3rd) with fully developed temperature and velocity profiles at the inlet and pressure profile at the outlet.

Results from the simulations show no improvement in the solutions when fully developed temperature profiles are used, rather the opposite. All simulations tend to overestimate the overall heat transfer coefficient (U) with respect to the empirical values with a relative error of up to 11.34 % with respect to the mean empirical value in 3rd case. As for pressure drops, this model seems to fall between the empirical correlations of Sinnot et al. (2012) and the estimation of the Fanning Factor by using Colebrook's equation. All in all, simulations provide good approximation if considering that empirical correlations differ from one to another with around 5 % of error for U values and 8~24 % for ΔP. With those results, the rest of the simulations are performed by using only fully developed velocity and pressure profiles. The comparison between the studied turbulence models and the semi-empirical results for pressure drop and overall heat transfer coefficient (U) are depicted in Figures 3 and 4.

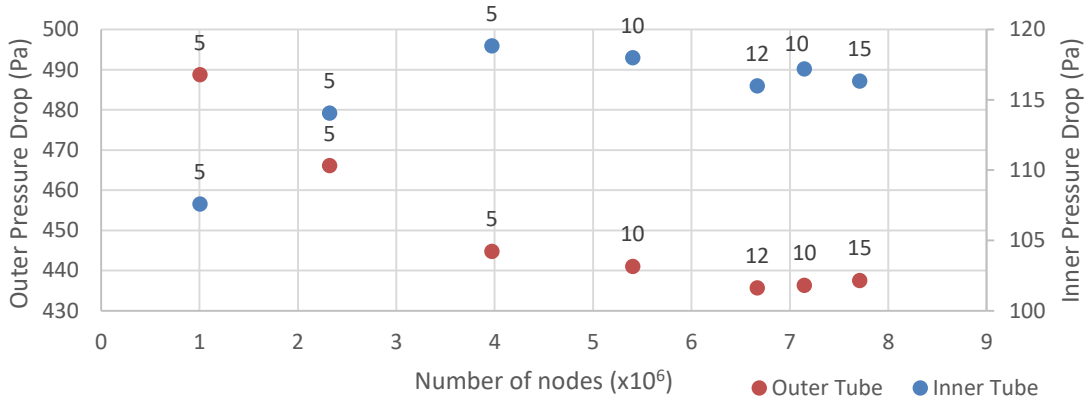

Figure 1: pressure drop for the k-ω SST model. The numbers above the dots are the number of inflation layers

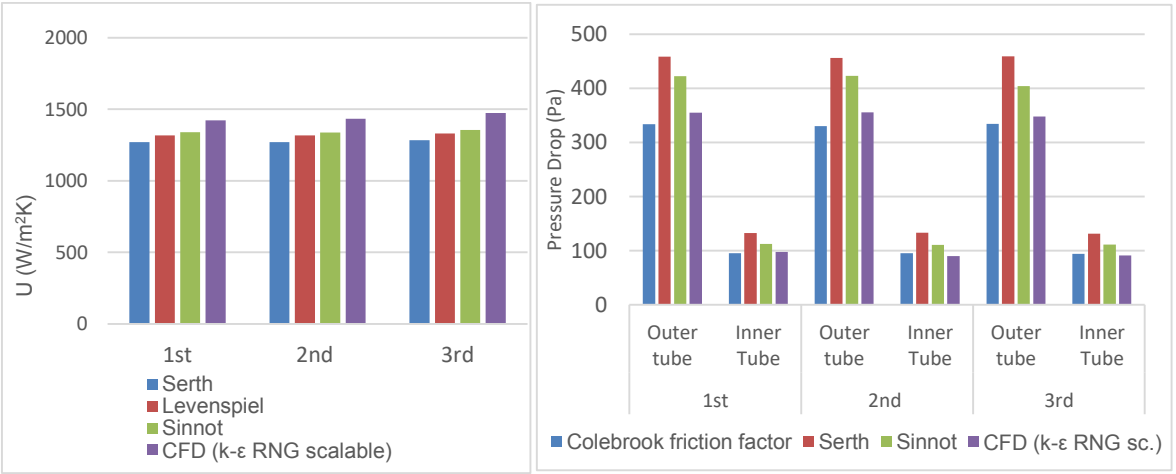

Figure 2: Comparison between three simulations with different input profiles at the boundaries. a) overall heat transfer coefficient b) pressure drop

In the case of the k-ε family, results show a major dependence on the wall treatment used that on the turbulence model itself. It also has to be taken into account that the classical equations used tend to overestimate pressure drop and underpredict the overall heat transfer coefficient for engineering purposes, as they use conservative

coefficients for the estimates. Velocity profiles at the outer tube are found to match the form of those found by Rothfus et al. (1950) for annular tubes. For pressure drop, classical correlations differ from one to another significantly: 24 % in the case of Serth (2007) compared with the corrected Colebrook friction factor equation. Sinnot's (2012) results are more conservative, differing with 7.8 % error from Serth's (2007) and 14 % from Colebrook's equation. With these considerations, it cannot be stated that there is a bad result for the CFD simulations, as each model follows one or other empirical correlation. Scalable Wall Functions are the ones that most differ from the mean value and tend to underestimate the pressure drop of both pipes, yielding results close to the ones found by the Colebrook equation corrected by Rothfus et al. (1950) in a non-isothermal regime. k-ω SST model seems to give intermediate results compared with other methods. Compared to the average semi-empirical value, k-ω SST with default settings has performed within a maximum of 16 % of relative error for the inner pipe and 2.4 % for the outer pipe. Low Re treatment for k-ω SST yields 11 % error in pressure drop with respect to the mean for both pipes.

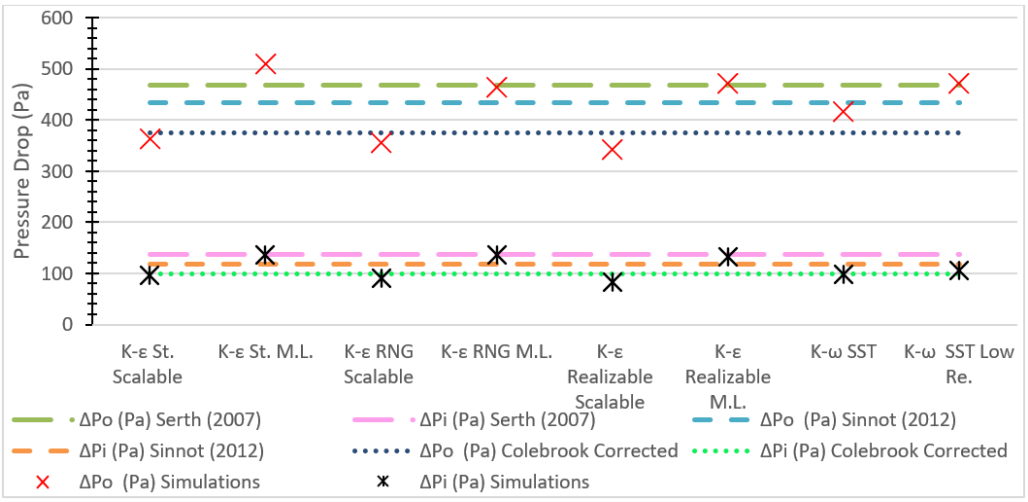

Figure 3: pressure drop comparison between empirical equations and CFD (ΔPo outer tube and ΔPi inner tube)

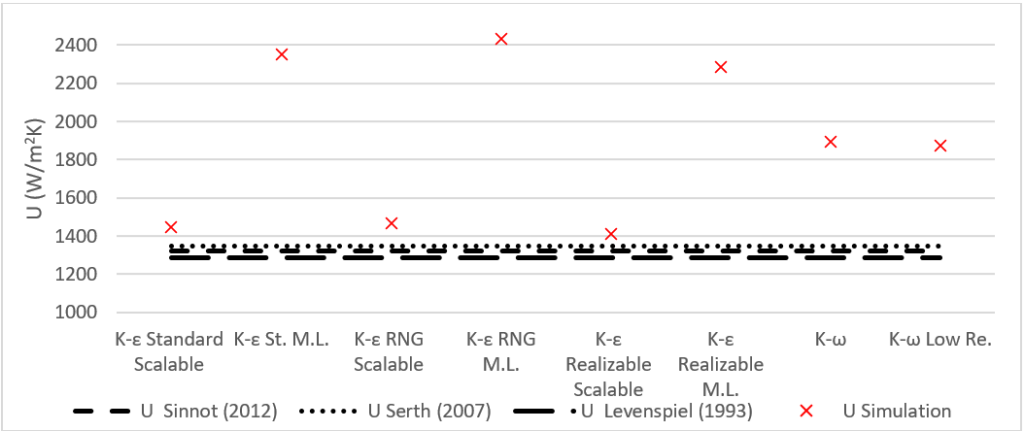

Figure 4: overall heat transfer coefficient comparison between empirical equations and simulations

All CFD results for pressure drop are valid, as they all seem to follow empirical data: Scalable Wall functions reproduce Colebrook's equation for the friction factor with k-ε Standard being the one with less relative error (~4.1 %). Menther-Lechner treatment as well as Low Re k-ω SST follow Serth's (2007) correlations with up to 2.0 % relative error in the inner pipe and 0.87 % error in the outer one for the case of the k-ε RNG model. While being less accurate at predicting pressure drop (at least in the case where they are combined with the k-ε Realizable model), Scalable Wall Functions give a far superior performance than other methods when estimating heat transfer in these simple geometries: k-ε Standard had a 9.8 % error, k-ε RNG 11 % and the k-ε Realizable model predicted U values with an accuracy of 7.1 %. SST k-ω models are found to be a good tradeoff

in accuracy between pressure drop and heat transfer but further simulations with different velocity inlet values found a correction factor of ~0.7, which is advisable when estimating the U value.

5. Conclusions

CFD methods have proven to be accurate at predicting simple heat exchanger geometries. They differ from one to another with similar errors as the classical correlations do between themselves. Scalable Wall functions combined with k-ε models provide results similar to classical equations for heat transfer between two concentric pipes, with up to 7.1 % accuracy. For pressure drop, they slightly underestimate the Colebrook corrected equation results with errors up to 16 % in the case of k-ε Realizable. The k-ω SST model offers the more balanced result as far as pressure drop goes, giving 2.4 % error with respect to the mean empiric value for the outer tube, and 11 % error for the inner tube is found with the Low Re configuration. When estimating U values, a correction factor of around 0.7 is found to be accurate for this geometry. More complex configurations can rely on the used models to properly predict the global parameters of heat-exchangers. With each method having its strengths and weaknesses, a correction factor can then be found to account for them in general cases. With these results, more simulations are to be made to prove the performance of the turbulence models tested in more complex geometries and flow regimes. It has been proven that for this heat-exchanger configuration, CFD codes are capable of predicting design variables as well as classical correlations.

Nomenclature

f -Fanning Factor
f_2 -Corrected Fanning Factor
L -Pipe Length, m
r_i -Inside radius of the inner pipe, m
R_i -Inside radius of the outer pipe, m
r_m -Radius where the velocity is maximal, m
r_o -Outside radius of the inner pipe, m
R_o -Outside radius of the outer pipe, m
U - Overall heat transfer coefficient, W/(m^2·K)
ΔPi -Inner tube pressure drop. Pa
ΔPo -Outer tube pressure drop, Pa

Acknowledgements

To Fundació Privada Mir-Puig for their financial support during the realization of this project.
Author Alexandra Elena Plesu Popescu is a Serra Húnter fellow.

References

Abbasian Arani A.A., Uosofvand H., 2021. Double-pass shell-and-tube heat exchanger performance enhancement with new combined baffle and elliptical tube bundle arrangement. International Journal of Thermal Sciences,167, 106999.

Ajmi M., Hnaien N., Marzouk S., Kolsi L., Ghachem K., Aissia H.B., Almeshaal M.A., 2020. Numerical investigation of heat transfer enhancement of an inclined heated offset jet. International Communications in Heat and Mass Transfer, 116, 104682.

ANSYS, 2021. ANSYS Fluent Tutorial Guide 2021 R1, Ansys Inc., Canonsburg, PA, USA.

Gadave N.M., Kothmire P.P., 2019. Thermo-hydraulic performance evaluation of shell & tube heat exchanger with different tube geometries. International Journal of Engineering and Advanced Technology, 9, 2125-2132.

Levenspiel O., 1998. Engineering Flow and Heat Exchange. Revised Edition. Plenum Press, New York, USA, 166-170, 353-366.

Pal E., Kumar I., Joshi J.B., Maheshwari N.K., 2016. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. Chemical Engineering Science, 143, 314-340.

Petrik M., Szepesi G.L., 2018. Shell side CFD analysis of a model shell-and-tube Heat Exchanger. Chemical Engineering Transactions, 70, 313-318.

Rothfus R.R., Monrad C.C., Senecal V.E., 1950. Velocity Friction Distribution and Fluid in Smooth Concentric Annuli. Industrial and Engineering Chemistry, 42, 2511-2520.

Serth R., 2007. Process Heat Transfer: Principles and Applications. Boston: Elsevier, 53-182, Appendix B: Dimensions of Pipe and Tubing.

Sinnott R., Towler G., 2012. Chemical Engineering Design. Fifth Edition Elsevier Limited of The Boulevard, Langford Lane, Kidlington, Oxford, United Kingdom.

Sruthi B., Sasidhar A., Surendra K.A., Sahu M.K., 2021. Comparative analysis of corrugation effect on thermohydraulic performance of double-pipe heat exchangers. Heat Transfer, 50, 4622– 4642.