Excerpt from the Proceedings of the 2011 COMSOL Conference in Boston

Double Pipe Heat Exchanger Modeling – COMSOL Uses in Undergraduate Education

Louis Desgrosseilliers* and Dominic Groulx
Mechanical Engineering Department, Dalhousie University
*Corresponding author: P.O.Box 15000, Halifax, Nova Scotia, Canada, B3H 4R2, Louis.D@dal.ca

Abstract: A double pipe, counter-current, single-phase heat exchanger with water is used in the undergraduate chemical engineering lab. We prepared a simple 2D axisymmetric model in COMSOL Multiphysics to be used as a teaching tool for students to learn the internal and physical nature of the heat exchanger. Foremost was to show the impact of purely turbulent, linear heat exchanger on the model prediction of the outlet temperatures. This approach was of course roughly equivalent to using the Seider-Tate equation for straight pipe heat exchange and the LMTD method; however, the COMSOL model responded differently to the low-Re transition annular flow and was not limited to the LMTD temperature distribution. Adding baffles to the model where there were bends in the lab provided enhanced heat transfer, especially with smaller open area and with more baffles in series. External convection had little impact on heat exchange in the COMSOL model.

Keywords: Double pipe heat exchanger, counter-current, turbulent flow, undergraduate education, baffles.

1. Introduction

A cornerstone of Chemical and Mechanical Engineering undergraduate programs the world over is the experimental and theoretical study of heat exchange. Graduating engineering students gain some appreciation in their lab course by comparing empirical correlations combined with the thermodynamics of heat exchange with the real operation of a counter-current, double pipe, single-phase heat exchanger. They vary water flow rates and entrance temperatures to achieve new outcomes to study. The hot stream is conditioned prior to entering the heat exchanger by passing through a vertical, saturated-steam jacketed, double pipe heat exchanger, while the cold stream is supplied directly from the building's potable water.

Students succeed in gaining some new perspectives of heat exchange, but the process is not whole. Often times, more complex phenomena that cannot be easily predicted empirically are encountered, like mixed-regime heat exchange ($2{,}100 < Re < 6{,}000$) [1] and nucleate boiling, and as such, will be left unfulfilled in the context of the course. An equivalent double pipe heat exchanger model in COMSOL Multiphysics can be used to quantify the different contributions in the heat exchanger beyond the accuracy provided by the available empirical correlations and contribute to intimate understanding of heat exchange in the classroom.

The objective of the current work is to develop and validate a simple, 2D axisymmetric, finite element model of the single-phase, counter-current, double pipe heat exchanger as a proof of use of COMSOL Multiphysics as part of an undergraduate heat transfer course. The model's strength would be to enable the students to add or omit features affecting the flow and heat exchange of both fluids so as to quantify individual contributions.

2. Experimental Set-up

The double pipe heat exchanger housed in the undergraduate chemical engineering unit operations laboratory at Dalhousie University is constructed in serpentine (Fig. 1) so as to provide sufficient heat exchange area, but limit its vertical length. Each of the four segments measures 2.159 m, in which there is a 2" sch.40 PVC pipe with a 1" ID (1 mm thick) copper tube in the centre. The flows are arranged for the hot water to pass through the centre, copper tube, while the cold water flows through the annulus. Each segment is connected by approximately 30 cm long, 1" ID tubing with two 90° bends.

Flow rates are monitored and controlled by means of two independent glass-tube rotameters with integral throttling valves, measuring in the range of 0 – 5 USGPM (only accurate in middle 90% of the range, so 0.5 – 4.5 USGPM). All of the flow rates in the copper tube provide Reynolds number in the range $4{,}600 < Re_D < 17{,}000$; whereas, the annulus flow

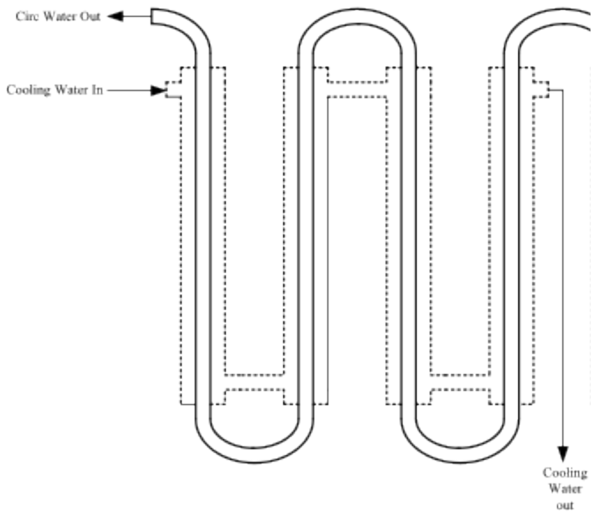

Figure 1. Single-phase, double-pipe heat exchanger layout.

rates were within 800 < Re_D < 3,100. The steady state temperatures were monitored by means of electronic display from thermocouples placed in the turbulent bends entering and exiting the heat exchanger.

3. Physical Model

Heat transfer theory offers only empirical equations in the way of predicting heat exchange between two non-laminar fluids in a double pipe heat exchanger. The Seider-Tate correlation (Eq. (1)) for turbulent flow (Re_D > 6,000) is easily the most recognizable and works accurately for straight channel systems (can be non-circular channels), but works poorly if physical features (e.g. bends of baffles) of the heat exchanger generate local variations in the turbulence [1]:

$$Nu = \frac{hD}{k} = 0.023 Re_D^{0.8} Pr^{1/3} \tag{1}$$

The average *Nu* is calculated for the flow with Re_D and *Pr* evaluated at the average temperature of the inlet and outlet of the flow, where *h*, *D* and *k* are the corresponding heat transfer coefficient, hydraulic diameter and thermal conductivity coefficient respectively. The heat flows are estimated for each stream with an initial guess of the outlet temperatures and the solution is obtained by iteration of Eq. (1) with the individual stream heat balances and overall heat transfer equation using the LMTD method (assuming the system is perfectly insulated). The resulting solution would be expected to overestimate the overall rate of heat transfer in a straight pipe heat exchanger due to the fully turbulent assumption of the annular flow.

Since the temperatures in both streams in the heat exchanger were both quite low, radiation heat transfer was neglected.

However, one refinement to this approach would be the result obtained from the Gnielinski equation for non-laminar flows [2]:

$$Nu = \frac{\frac{f}{2}\left(\mathrm{Re}_D - 10^3\right)\mathrm{Pr}}{1 + 12.7\sqrt{\frac{f}{2}}\left(\mathrm{Pr}^{2/3} - 1\right)} \tag{2}$$

where *f* is the Fanning friction factor. For two 90˚ bends (or a 180˚ return bend), *f* can be approximated by the head loss coefficient, K_{180°, equal to 1.5 [1]. From this method, the approximate value of *f*, assuming 1” ID bends that are 12” long, is:

$$\begin{aligned} K_{180^\circ}\frac{v^2}{2} &= 2fv^2\frac{L}{D} \\ f &= 1.5\frac{1}{12} = 0.125 \end{aligned} \tag{3}$$

Equation (2) would simply indicate that turbulence enhancing features elevate the local value of *Nu*, implying that the lower value obtained from using Eq. (1) is inaccurate in representing the whole heat exchanger.

4. Use of COMSOL Multiphysics

For model simplicity, the four series heat exchanger segments were generated as one single 8.636 m long heat exchanger of equivalent heat exchange area (i.e. without connecting bends). The model of the entire counter-current, single-phase heat exchanger sections was prepared using 2D axisymmetric quadrilateral elements, along with boundary layer elements in COMSOL Multiphysics 4.2. The solid pipe walls consisted of quadrilateral elements only, since heat conduction was the only physics of interest. A single non-isothermal flow physics (turbulent flow) was applied to both fluids, and gravity was neglected since it has no effect on the heat transfer in such systems. Although both flows were simulated as fully turbulent, the annular flow did not necessarily meet this criterion everywhere; however, this initial assumption should overestimate the heat transfer coefficient in the annulus for the case of a real, perfectly linear flow.

One single set of experimental conditions was tested, while various aspects of the model were adjusted in order to quantify the individual contributions with the purpose to demonstrate them to students. The fixed boundary conditions on both internal flows were set to superficial velocities of 0.56 m/s (4.5 USGPM) for the hot water – entering at 33.4˚C (from the experiment), and 0.15 m/s (3.6 USGPM) for the cold water – entering at 5.8˚C (again, from the experiment). The outlet boundary conditions were both specified to 0 Pa downstream pressure. Since the outer PVC pipe of the real heat exchanger was not at all insulated, an external convection boundary condition was applied, specifying vertical natural convection with ambient air at room temperature (20˚C) and the characteristic dimension set only to 2.159 m due to each segment being in contact with air over those lengths.

The basic model was only linear, with no turbulence enhancing features included, but baffles were later added to simulate the flow turning at the connecting bends between segments. This was especially important to reproduce the effect of turbulence at the bends on enhancing the convection heat transfer coefficient of the cold water in the annulus, which was otherwise considered to be in laminar or transition flow in the experiments and always the limiting resistance to the overall heat transfer. Poorly conducting baffles (same properties as PVC: C_p = 1,360 J kg^{-1}K^{-1}, ρ = 1,300 kg m^{-3}, k = 0.147 W m^{-1}K^{-1}) [3] were arranged in single pairs (only one in the center tube and annulus respectively) at intervals between each segment of heat exchanger, in two pairs 30 cm apart (with the center of the span at the same segment intervals), and in three pairs (same span as for the two pairs, with the third in the middle). The model also studied the effect of baffle open area (equivalent circular cross-section) and orientation (both flows directed against (Fig. 2a) or away (Fig. 2b) from the contact surface or opposing surfaces (Fig. 2c)). The relevance is that the higher velocity stream next to the heat exchange surface produced by the baffles was believed to provide enhanced local heat transfer.

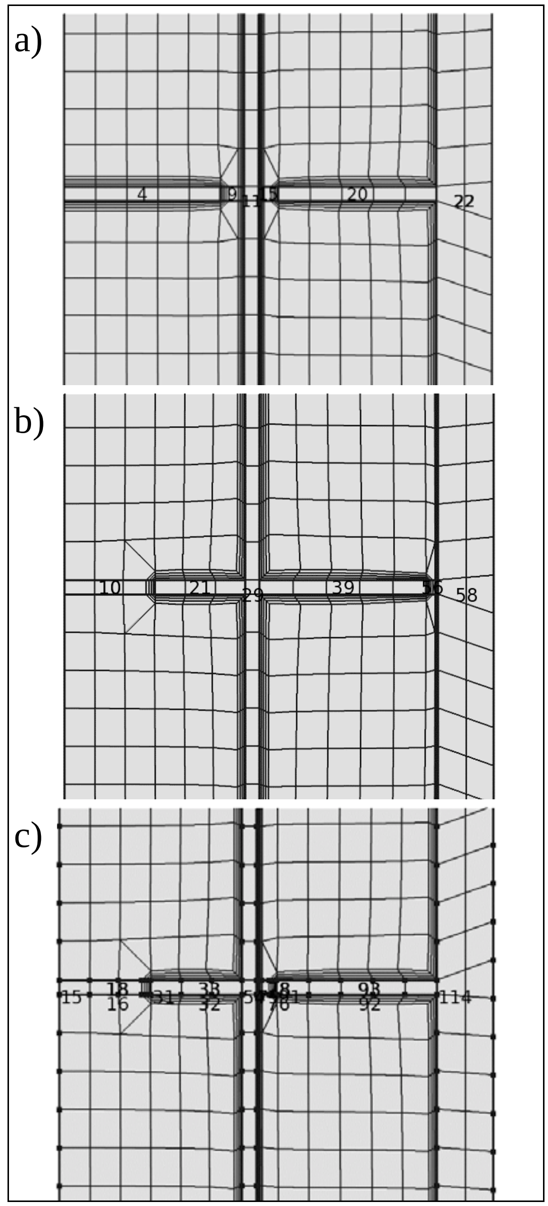

Figure 2. Baffle arrangements: a) baffle flows against the heat exchange surface; b) baffle flows away from the heat exchange surface; c) baffle flows on opposing surfaces.

The measure of overall heat exchanger performance was taken as the mixed stream average temperature at each outlet of the heat exchanger at steady state (i.e. stationary solution), which was computed in COMSOL Multiphysics as the area average temperature (T_{ave}) over the cross-section of flow:

$$T_{ave} = \int_{r_1}^{r_2} \frac{2\pi r T}{S} dr \quad (4)$$

where T is the local temperature at coordinate (r,z), and S is the cross-section area of the flow. Both exit stream T_{ave} became the basis on which to compare the model predictions.

4.1 Mesh Convergence Study

Mesh convergence was studied in the range of models containing 51,000 to 358,000 elements for the simple straight pipe heat exchanger, and 87,000 to 322,000 elements in the case of the single baffle pair model. The outlet temperatures were examined for convergence in Fig. 3.

Since both models showed little variation in outlet temperatures, and larger and smaller meshes could not be considered for practical reasons (too few boundary layer elements or too many elements causing memory saturation), the 67,683 element mesh was chosen for the straight pipe heat exchanger model and the 87,345 element mesh was chosen for the baffled heat exchanger. All other baffle arrangement would be prepared with the same mesh parameters (i.e. same quadrilateral and boundary layer element distributions).

5. Results and Discussion

The various models and their respective results as obtained from Eq. (4) are summarized in Table 1. The experimental conditions yielded the highest overall rate of heat transfer (taking the hot water flow as the basis) and of course showing the largest change in both outlet temperatures.

All three models with the straight pipe assumption only (no baffles, including the Seider-Tate/LMTD method) predicted outlet temperatures that were 4 to 5°C different from the experiment, thus highlighting the significance of turbulence enhancing features on the overall performance of the physically installed heat exchanger. Regarding the validation of the straight-pipe COMSOL models, reasonable agreement was obtained with the Seider-Tate model, within the degree of accuracy of the empirical model and assumptions (e.g. uniform heat transfer coefficients over the length of each stream and LMTD). For students, these models represent those with limited accuracy, meaning error ≥ 20%.

Both attempts at modelling in COMSOL without external convection produced no measureable change in either of the outlet temperatures. Students should be wary of this as

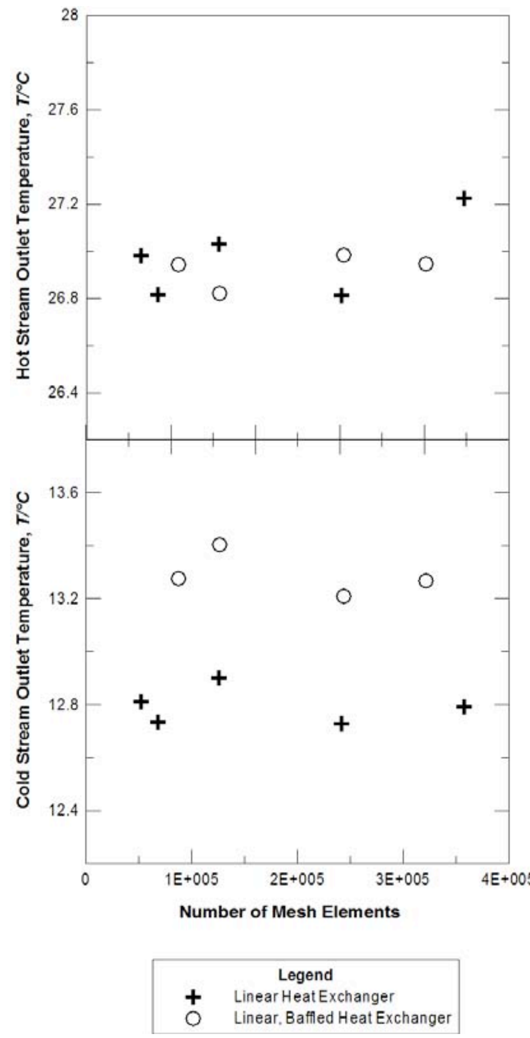

Figure 3. Mesh convergence studies: (top) hot stream outlet temperature results, and (bottom) cold stream outlet temperature results.

Table 1. Simulation results.

	External Convection	Baffles Pairs	Baffle Equivalent ID	Baffle Arrangement	Hot Outlet (°C)	% Error	Cold Outlet (°C)	% Error	Total Rate of Heat Transfer (kW)
Experimental	Y				**22.4**		**18.2**		13.0
Seider-Tate	N	N			**26.7**	19.0	**14.2**	22.1	8.0
COMSOL - Straight Pipe	N	N			**27.3**	22.1	**13.0**	28.5	7.2
COMSOL - Straight Pipe	Y	N			**27.3**	22.1	**13.0**	28.5	7.2
COMSOL - Baffled	Y	1	0.25"	Against	**26.2**	17.0	**14.0**	23.0	8.5
COMSOL - Baffled	Y	1	0.5"	Against	**26.6**	18.6	**13.7**	24.6	8.1
COMSOL - Baffled	Y	1	0.5"	Away	**26.9**	20.3	**13.3**	27.1	7.6
COMSOL - Baffled	Y	2	0.5"	One Against, One Away	**25.6**	14.4	**14.9**	18.1	9.2
COMSOL - Baffled	N	2	0.5"	One Against, One Away	**25.6**	14.4	**14.9**	18.3	9.2
COMSOL - Baffled	Y	2	0.5"	One Against, One Opposite	**26.0**	15.9	**14.6**	19.5	8.8
COMSOL - Baffled	Y	2	0.75"	One Against, One Away	**26.1**	16.7	**14.2**	22.2	8.6
COMSOL - Baffled	Y	3	0.5"	Two Against, One Away	**24.6**	9.8	**16.0**	12.4	10.4

a significant source of error in the empirical calculations, especially since the capacity for heat exchange from the cold water annulus stream to the ambient air becomes limited as the water gains temperature.

The principal factor that did improve the accuracy of the COMSOL model was the inclusion of baffles, which still varied amongst themselves. Two separate simulations were run with equivalent baffle flow area different from the nominal 0.5” ID that was chosen. The simulation with 0.25” equivalent ID baffles (only one pair) improved accuracy by 1.6% for both the hot water outlet and cold water outlet temperatures over the 0.5” ID model; whereas the simulation with 0.75” equivalent ID baffles (two pairs) decreased accuracy by 2.3% and 4.1%.

The most favourable baffle orientation, either both ‘against’, both ‘away’ or ‘opposite’, was shown to be both ‘against’ and this was true for single baffle pairs and for 2 and 3 baffle pairs. In the single pair arrangements, this was due to both accelerated flows (before and after) being channelled along the shared heat transfer surface, but further gains in heat transfer rates with 2 and 3 baffle pairs was only achieved by including an ‘away’ baffle pair before one ‘against’ baffle pair or alternating between any two ‘against’ baffle pairs. Flow turning causing enhanced turbulence (Fig. 4) played an important role in these multiple baffle pair arrangements, for which Eqs. (2) & (3) estimated the local heat transfer coefficients in each stream exiting the connecting bends 7 to 13 times greater than from Eq. (1). The real overall heat transfer coefficients would have a value somewhere between the results of Eqs. (1) and (3).

COMSOL models with two baffle pairs at each interval represented those with moderate accuracy, 14% < error < 23%; the model with three baffle pairs obtained the highest accuracy, with an error < 13%.

Following that the heat transfer enhancements were achieved through primarily inward facing (‘against’) baffles and adding alternating inward/outward facing baffle pairs in series and finally reducing their respective open area to flow, further manipulation of these parameters would be expected to produce a high accuracy model of the double pipe heat exchanger.

6. Conclusions

Undergraduate engineering students of the unit operation labs should now achieve better understanding of the heat exchanger by visualizing its internal physical features affecting heat transfer (although simplified in 2D axisymmetric space) and determining their relative contributions to the experimentally measured heat exchange.

Contributions in the models varied the accuracy of the simulations from > 20% error to less than 13%, of which the largest contributions came from flow turning and accelerating features (i.e. baffles to represent bends) and little came at

Figure 4. Velocity surface plot of the three baffle arrangement showing flow turning (Baffles in white and direction of the flow indicated by the gray arrows).

all from the external natural convection.

Future simulations should also look at the effect of surface roughness on the overall heat transfer between the two flows.

7. References

1. Warren L. McCabe, Julian C. Smith, and Peter Harriott, *Unit Operations of Chemical Engineering*, 7th ed., McGraw-Hill, p. 347-382 (2005)

2. Adrian Bejan, *Heat Transfer*, John Wiley & Sons Inc., New York, p.316 (1993)

3. M.F. Ashby, *Materials and the Environment: Eco-Informed Material Choice*, Elsevier, Oxford (2009)

8. Acknowledgements

We are very grateful for the opportunity afforded by Dr. Gianfranco Mazzanti, of the Process Engineering and Applied Science Department at Dalhousie University, who taught the undergraduate chemical engineering unit operations lab during the winter of 2010. Also, we extend a sincere thanks to the students of the chemical engineering class of 2010 at Dalhousie University.

The authors are grateful to the National Science and Engineering Research Council of Canada (NSERC), the Canadian Foundation for Innovation (CFI), Dalhousie Research in Energy, Advanced Materials and Sustainability (DREAMS) program and the Institute for Research in Materials (IRM) for their financial support.