Contribution to the Characterization of the Kidney Ultrasound Image Using Singularity Levels

Ranjan Datta

***Abstract*—The capability and accuracy of computational fluid dynamics (CFD) software facilitates its use in all engineering fields. This is creating a need for future engineers to have knowledge of model development using CFD software by the time they acquire the degree. As supported by other studies, the incorporation of CFD into the undergraduate curriculum is possible at junior or senior level. This paper presents the main aspects of an approach used in a laboratory course to introduce students to a commercial CFD software. The approach includes the use of a previously developed CFD model of a parallel pipe heat exchanger to obtain simulation results and compare them with experimental data in order to validate the model. As a post-laboratory activity, the validated model can be used for further investigation on the heat exchanger performance without the need of running additional experiments.**

***Index Terms*— CFD model, heat exchanger, laboratory course, model validation**

I. Introduction

With advances in capabilities of commercial Computational Fluid Dynamics (CFD) software, its uses and applications have increased in all areas of engineering. These advances are creating a culture where engineers are expected to use CFD software without post-graduate education [1]. As a result, there is an increasing need to integrate CFD as a computer-assisted learning and simulation tool into undergraduate engineering courses and laboratories [1-3]. For example, Adair and Jaeger [1] presented results that confirm the effectiveness of a suitable curriculum for students to use computational fluid dynamics as part of a fluid mechanics course at intermediate undergraduate level.

Supported by previous studies, Stern et al. [2] and Adair and Jaeger [1] suggest that the use of simulation technology in education enhances the curriculum, increases learning efficiency and understanding, improves effectiveness of hands-on learning methods, is effective in a combination of physical and simulation laboratories, and produces a positive student response. A positive response comes from the fact that as CFD is used in many areas of engineering, the students find relevance to many other courses in their curriculum and build on knowledge gained in other courses such as fluid mechanics, thermodynamics and numerical methods [4].

In order to simplify the use of CFD for analysis and design, some authors have published studies on tools and teaching methodologies oriented to help students understand the physical problem and analyze the phenomena [5-6]. In this sense, Lopez et al. [7] presented a comparison between results obtained for a controlled layout simulated by both physical and computational models.

In this paper, the approach used to introduce students to CFD in a laboratory course is presented. The elements of the approach are to introduce the student to the terminology associated with CFD, give insight into the basic principles of CFD, recognize the quality of a CFD model by understanding the concept of validation, and realize the benefits of having a validated model. The approach also allows comparison of experimental results with CFD results [1]. In this order of ideas, Lopez et al. [7] suggest that CFD can also constitute a powerful tool in the educational process of learning engineering concepts since the instructor lend the inherent difficulty of the computational model to the simplicity of a case study that can be calibrated experimentally.

The previous statements are supported by CFD software developers through training that can be used by teachers or students. For example, Professor Milovan Peric [8] explains how engineering software can be integrated with traditional curriculum to improve the understanding of conduction, radiation, convection and conjugate heat transfer.

II. The Laboratory Course and Experiment

The course description for MENG 3211 Mechanical Engineering Lab II at the University of Texas at Tyler states: “Introduction to basic Thermal/Fluid sciences laboratory procedures and practices with uncertainty analysis. Experimental topics include fluid flow, heat exchanger basics, and basics of refrigeration…” One of the lab activities is based on the parallel heat pipe heat exchanger described in Section 4.

Although it is not a problem in the MENG 3211 course, Lona et al. [9] pointed out that the first problem in teaching heat exchangers is the difficulty felt by the students that are not familiar with the equipment. They suggest that the software can allow the student to analyze the effects of his/her decisions over heat exchanger parameters and operational conditions [9]. This is achieved through the approach used in this course, which is

similar to the approach used by Lopez et al. [7]. The approach can be summarized as acquiring experimental data to validate analytical equations used for heat exchanger analysis (Section 3), and then use the CFD model (Section 6) for further analysis without running additional experiments. Although the intention is to introduce CFD to the students in this course, it is expected that a much deeper physical insight can be achieved by means of a model developed with a general-purpose commercial computational fluid dynamics (CFD) software program [10] such as STAR-CCM+.

The apparatus described in Section 4 is used as the physical problem to simulate the heat exchanger performance by measuring flow rates and temperatures. The same geometry has been reproduced in a CFD model (Section 6), and temperatures are used to compute and compare the thermal heat capacity of the heat exchanger. By doing this comparison, students become aware of the importance of validation, realize the agreement between measured and computed values, and give themselves the opportunity to visualize velocity fields and temperature and pressure profiles by means of graphical displays of the CFD simulation results [7].

III. Parallel Pipe Heat Exchanger

In a parallel pipe heat exchanger, also referred to as a double pipe heat exchanger, one pipe is placed within another pipe having a larger diameter. The two fluid flows may be in the same direction (parallel flow) or in the opposite direction (counter flow) as shown in Fig. 1.

Important considerations for selecting or designing a heat exchanger are heat transfer performance, physical size, total weight, and cost. This laboratory activity is related to the heat transfer performance. For heat exchangers performance analysis, two methods are widely used: the Logarithmic Mean Temperature Difference method and the ε-NTU method [11].

A. *The Logarithmic Mean Temperature Difference Method*

The heat transfer rate (q) is computed as

$$q = UA\Delta T_{lm} \tag{1}$$

with the Logarithmic Mean Temperature Difference (ΔT_{lm}) defined as

Parallel flow:

$$\Delta T_{lm} = \frac{(T_{h2}-T_{c2})-(T_{h1}-T_{c1})}{\ln\left(\frac{T_{h2}-T_{c2}}{T_{h1}-T_{c1}}\right)} \tag{2}$$

Counterflow:

$$\Delta T_{lm} = \frac{(T_{h2}-T_{c1})-(T_{h1}-T_{c2})}{\ln\left(\frac{T_{h2}-T_{c1}}{T_{h1}-T_{c2}}\right)} \tag{3}$$

B. *The ε-NTU Method*

The Logarithmic Mean Temperature Difference method has some inherent disadvantages. The main disadvantage is when the inlet and outlet temperatures need to be determined. This generally requires an iterative procedure. An easier way is to eliminate the inlet and the outlet temperatures from the analysis which can be done by considering the ability of a heat exchanger to transfer a given amount of heat. This is called effectiveness (ε) of a heat exchanger and is defined as

$$\varepsilon = \frac{q_{act}}{q_{max}} \tag{4}$$

where is the maximum possible heat transfer. The actual heat transfer is readily determined by performing an energy balance on any of the two fluid streams. Assuming constant specific heat, the actual heat transfer is computed as

$$q_{act} = \left(\dot{m}c_p\right)_c(T_{co} - T_{ci}) = \left(\dot{m}c_p\right)_h(T_{hi} - T_{ho}) \tag{5}$$

The maximum possible heat transfer is defined as

$$q_{max} = \left(\dot{m}c_p\right)_{min}(T_{hi} - T_{ci}) = C_{min}(T_{hi} - T_{ci}) \tag{6}$$

where C_{min} refers to the lowest heat capacity $\left(\dot{m}c_p\right)$ of the two fluids.

Using the heat capacity rate ratio, $C^* = C_{min}/C_{max}$, and knowing that the Number of Heat Transfer Units is defined as $NTU = UA/C_{min}$, ε and NTU of a parallel pipe heat exchanger can be calculated using the following relationships:

Parallel flow:

$$\varepsilon = \frac{1-e^{[-NTU(1+C^*)]}}{1+C^*} \tag{7}$$

$$NTU = -\frac{1}{1+C^*}\ln[1-\varepsilon(1+C^*)] \tag{8}$$

Counter flow:

$$\varepsilon = \frac{1-e^{[-NTU(1-C^*)]}}{1-C^*e^{[-NTU(1-C^*)]}} \tag{9}$$

$$NTU = \frac{1}{1-C^*}\ln\left[\frac{1-\varepsilon C^*}{1-\varepsilon}\right] \tag{10}$$

Equations (9) and (10) are undetermined for $C^* = 1$, but applying L'Hospital's rule, ε and NTU can be written as

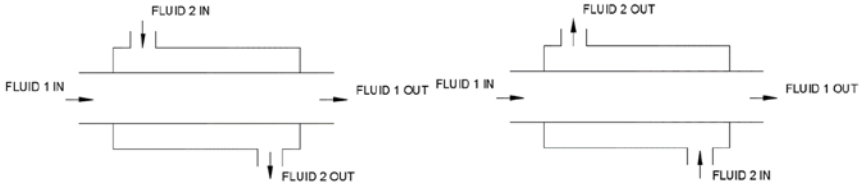

Fig. 1. Double Pipe Heat Exchanger, Parallel Flow and Counter Flow.

$$\varepsilon = \frac{NTU}{1+NTU} \quad (11)$$

$$NTU = \frac{\varepsilon}{1-\varepsilon} \quad (12)$$

It is important to understand that the heat exchanger thermal capacity (*UA*) is a parameter of the heat exchanger and is independent of the method used.

IV. The Laboratory Equipment

Fig. 2 shows the Hampden Model H-6878 apparatus used in the laboratory course. The apparatus consists of different valves to define the paths for the cold and hot water streams through the different heat exchangers. The cold and hot flow rates are measured with rotameters. Inlet and outlet temperatures are measured with type T thermocouples. In the laboratory the cold and hot water are taken directly from the supply pipes available from the building piping systems.

For this study, the copper parallel pipe heat exchanger indicated in Fig. 2 was used. The manufacturer specifies the thermal conductivity and density of the copper as 339.2 W/m-°C and 8954 kg/m3, respectively.

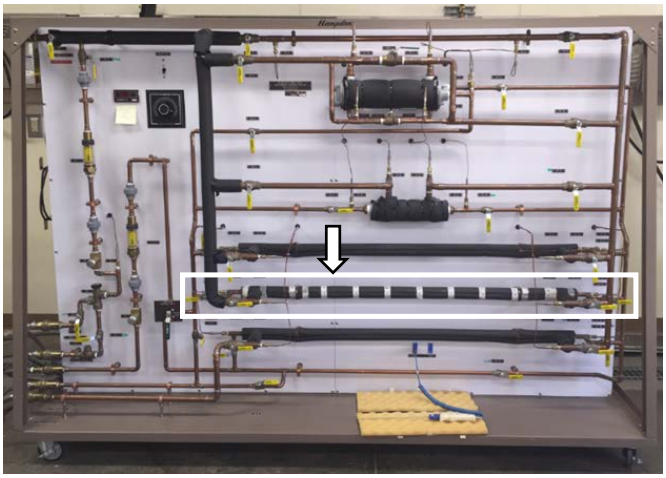

Fig. 2. Hampden Model H-6878 Apparatus.

V. Laboratory Measurements

Four sets of data are used in this study. Table 1 shows the data for the two counterflow experiments, and Table 2 shows the data for the two parallel flow experiments.

A. Uncertainty

Simulation results from the CFD model give the magnitude of temperature at the outlet of both the cold and hot fluids based on the specified inlet temperatures and flow rates, which are set identical to the experimental data. However, experimental inlet and outlet measurements have some uncertainty associated with them. Therefore, an uncertainty analysis is needed to propagate the uncertainties into the result of interest, i.e. the *UA*. The uncertainty analysis allows the ability to define a confidence interval for UA, which is used to validate the results from the CFD model (see Section 6). In this study, the *ε-NTU* method for analysis of heat exchangers was used. The approach used to estimate the uncertainty for *UA* are detailed in [12]. However, as an example of the use of the Taylor Series Method for propagation of elemental uncertainties into the result, the equations used for the counterflow are given in the Appendix. The resulting uncertainties for *UA* for the four sets of data are presented in Table 3.

TABLE I
DATA FOR COUNTERFLOW EXPERIMENTS

Experimental Data 1			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.1 (61)	25.6 (78)	57.8 (136)	50.0 (122)
Flow rate m^3/hr (GPM)			
0.454 (2.0)		0.681 (3.0)	
Experimental Data 2			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.7 (62)	23.3 (74)	57.8 (136)	46.7 (116)
Flow rate (GPM)			
0.681 (3.0)		0.454 (2.0)	

TABLE II
DATA FOR PARALLEL FLOW EXPERIMENTS

Experimental Data 3			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.7 (62)	26.1 (79)	57.2 (135)	50.6 (123)
Flow rate m^3/hr (GPM)			
0.454 (2.0)		0.681 (3.0)	
Experimental Data 4			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
21.1 (70)	30.0 (86)	74.4 (166)	59.4 (139)
Flow rate m^3/hr (GPM)			
0.681 (3.0)		0.454 (2.0)	

TABLE III
CONFIDENCE INTERVAL FOR *UA* W/K (BTU/HR-°F)

	Counterflow	
	Experiment 1	Experiment 2
Experimental	148 (280)	161 (306)
Uncertainty	10.6%	14.4%
Interval	[132-164] (250-310)	[138-185] (262-350)
	Parallel flow	
	Experiment 3	Experiment 4
Experimental	156 (295)	171 (324)
Uncertainty	11.0%	15.1%
Interval	[139-173] (263-328)	[145-197] (275-373)

VI. The CFD Model

The software used was the STAR-CCM+®, by CD-Adapco, which is a commercial software used for modeling multi-physics and complex geometries. The University of Texas at Tyler has a license that allows students to work with the software from any location through a virtual machine.

For model development in STAR-CCM+, students are introduced to the general steps shown in Fig. 3. This figure provides insights to students about the steps required to develop a model facilitating the understanding of the use of the model without actually going through the whole process.

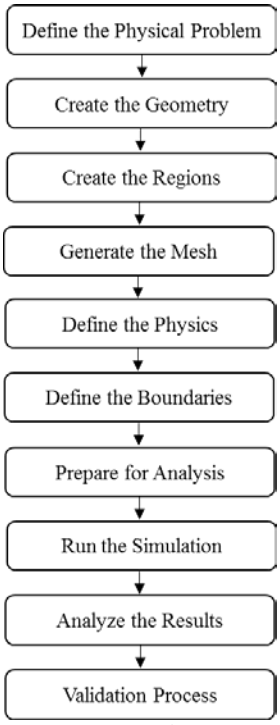

Fig. 3. General Steps for CFD Model.

The following sub-sections give relevant information for supporting the main steps on the CFD model development as in Fig. 3.

A. Create the Geometry

After defining the physical problem, i.e. the parallel pipe heat exchange, the first step in modeling the heat exchanger is to construct the geometry. A simplified model of the heat exchanger consists of a straight-end pipe within a pipe as illustrated by Fig. 4, with dimensions as given by the manufacturer and listed in Table 4. Using the CAD tools available within the software, the geometry of the heat exchanger was completed.

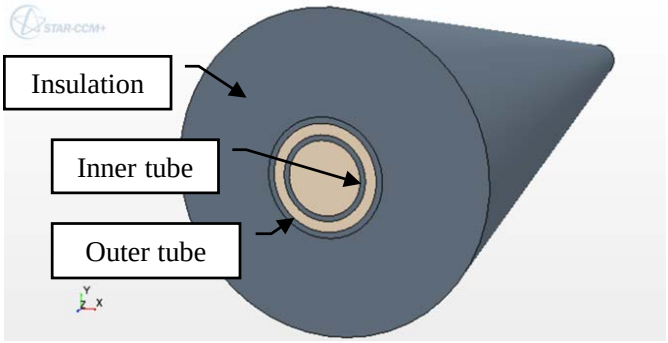

Fig. 4. Completed double pipe heat exchanger with insulation.

B. Create the Regions

Regions are the elements of a CFD model to which meshing and physical models are assigned. Different regions can have different meshing and physical models. Typically, the selections are: Create a Region for Each Part, Create a Boundary for Each Part Surface, and Create One Feature Curve for all Part Curves.

To continue with the model development, the model geometry is split into regions by part surfaces. Therefore, boundary conditions are placed at every surface that would be transferring energy. Interfaces are created between the solid and working fluid contact regions. This is very important on model development because these surfaces define the heat transfer between elements (parts of the heat exchanger). Once regions and interfaces are created, the mesh can be generated.

TABLE IV
HEAT EXCHANGER DIMENSIONS

Dimension	Units	Inner Tube	Outer Tube	Foam
Overall Length	m	1.21	1.21	1.21
	in	48	48	48
Outside Diameter	m	0.0159	0.0222	0.0603
	in	0.625	0.875	2.375
Inside Diameter	m	0.0138	0.0199	0.0222
	in	0.545	0.785	0.875

C. Generate the Mesh

Meshing is an important part of CFD model development because it defines precision and computation time. The smaller the size of the mesh, the more precise the results. However, more computation resources and time are needed. Therefore, a balance between precision and computation time is necessary for optimum management of the resources [7]. For the parallel heat exchanger, two base sizes were used to investigate the computational time. The two sizes were 0.005 m and 0.007 m. For 0.005 m the time required to run a simulation was about 5 hours, while for the 0.007 m is about half of an hour with no noticeable difference between results. Therefore, 0.007 m is used in the model. Fig. 5 shows a view of the two meshes and Fig. 6 shows the complete polyhedral mesh composed of 406,625 cells. As can be seen from these figures, the size of the cells is not the same in a radial direction. This is achieved by the fact that the different surfaces of contact between parts (fluid, tube, and insulation) are defined as interfaces (‘Create Interface’ in Star CCM+). The shape of the cells are defined by using the Prism Layer Mesher and Surface Remesher options on the meshing model selection.

D. Define the Physics

One physical continuum is created for each of the materials in order to be able to specify material properties and physical processes. These three physics continua are the Fluid, the Copper Pipe and the Foam Insulation. Table 5 to Table 7 summarize the physical models chosen from the available options given by the software.

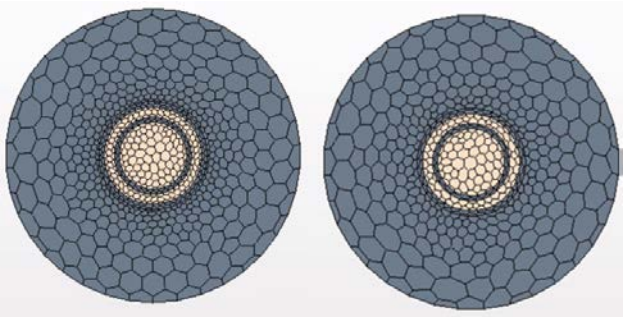

Fig. 5. Mesh 0.005 m (left), 0.007 m (right).

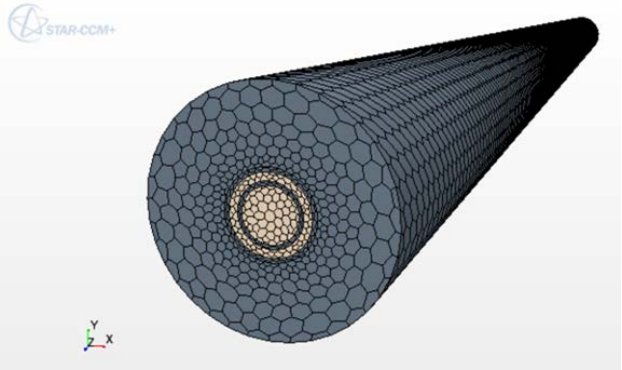

Fig. 6. Completed meshed model for the parallel heat exchanger.

E. Define the Boundaries

The energy transfer occurs between boundaries. Therefore, the notion and proper selection of boundary conditions and types of heat transfer are key definitions to reproduce heat transfer processes in analysis of heat exchangers. Table 8 summarizes the definition of the boundary conditions for the specified regions.

TABLE VI
COPPER PIPE PHYSICS CONTINUA

Models
Constant Density
Gradients
Segregated Solid Energy
Solid
Steady
Three Dimensional

TABLE VII
FOAM INSULATION PHYSICS CONTINUA

Models
Constant Density
Gradients
Segregated Solid Energy
Solid
Steady
Three Dimensional

F. Prepare for Analysis

In order to visualize the results, reports, plots and scenes are created to depict the parameters of interest, temperature in this case. As examples on how the results can be set to be visualized, Fig. 7 shows the temperature distribution for one of the four simulations performed.

TABLE V
FLUID (WATER) PHYSICS CONTINUA

Models
Constant Density
Gradients
Gravity
K-Epsilon Turbulence
Liquid
Reynolds-Averaged Navier-Stokes
Segregated Flow
Segregated Fluid Temperature
Steady
Three Dimensional
Turbulent
Two-Layer All y + Wall Treatment
Three Dimensional

G. Run the Simulation

After the model has been created, the simulation can be run to obtain results. When values of parameters or physical models are changed, previous results must be cleared before running a new simulation. This will allow starting the simulation from the first iteration and showing a progressive display of the change in magnitude of the parameters. After the simulation has been concluded, results for temperature can be obtained as the one shown in Fig. 7.

TABLE VIII
DEFINITION OF BOUNDARIES FOR THE DOUBLE PIPE HEAT EXCHANGER

Region	Physics	Boundary	Type	Thermal Specification
Inner Stream	Fluid	InletInnerStream	Mass flow inlet	Constant inlet temp
		OutletInnerStream	Pressure outlet	Constant outlet temp
		WallInnerStream	Wall	Convection
Inner Tube	Copper Pipe	InletInnerTube	Wall	Environment
		OutletInnerTube	Wall	Environment
		InsideWallInnerTube	Wall	Convection
		OutsideWallInnerTube	Wall	Convection
Outer Stream	Fluid	InletOuterStream	Mass flow inlet	Constant inlet temp
		OutletOuterStream	Pressure outlet	Constant outlet temp
		InsideWallOuterStream	Wall	Convection
		OutsideWallOuterStream	Wall	Convection
Outer Tube	Copper Pipe	InletOuterTube	Wall	Environment
		OutletOuterTube	Wall	Environment
		InsideWallOuterTube	Wall	Convection
		OutsideWallOuterTube	Wall	Convection
Insulation	Foam Insulation	InletInsulation	Wall	Environment
		OutletInsulation	Wall	Environment
		InsideWallInsulation	Wall	Convection
		OutsideWallInsulation	Wall	Environment

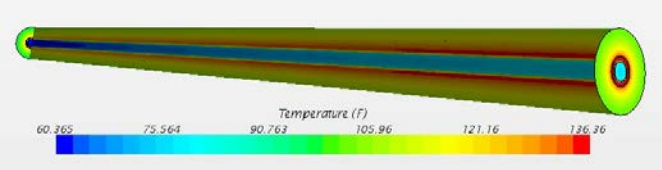

Fig. 7. Temperature distribution for data experiment 1 (counterflow).

H. Analyze the Results

The outlet temperatures were obtained for both counterflow and parallel flow. Table 9 and Table 10 present the outlet temperatures obtained using the same inlet conditions as in the experiments. As shown on the tables, most simulated results were within one degree of the experimental values. The highest difference was 1.06°C (1.9°F) for the outlet of the hot fluid in the "Experimental Data 1" of the counterflow arrangement.

Using the outlet temperatures obtained from the simulations and the *ε-NTU* method, the *UA* values were computed and are given in Table 11. These results illustrate that the *UA* values computed from simulated data fall within the uncertainty intervals. Table 11 also shows that the maximum difference between the simulated and experimental *UA* values is 10.8%, which does not exceed the uncertainty defined for the experimental data.

TABLE IX
DATA FOR COUNTERFLOW EXPERIMENTS

Experimental Data 1			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.1 (61.0)	36.2 (79.2)	57.8 (136)	51.1 (123.9)
Flow rate m^3/hr (GPM)			
0.454 (2.0)		0.681 (3.0)	
Experimental Data 2			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.7 (62.0)	23.9 (75.0)	57.8 (136.0)	46.7 (116.0)
Flow rate m^3/hr (GPM)			
0.681 (3.0)		0.454 (2.0)	

TABLE X
DATA FOR PARALLEL FLOW EXPERIMENTS

Experimental Data 3			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
16.7 (62.0)	26.3 (79.4)	57.2 (135.0)	50.9 (123.6)
Flow rate m^3/hr (GPM)			
0.454 (2.0)		0.681 (3.0)	
Experimental Data 4			
Water in tube		Water in shell	
Temperature °C (°F)			
IN	OUT	IN	OUT
21.1 (70.0)	30.2 (86.3)	74.4 (166.0)	58.8 (137.9)
Flow rate m^3/hr (GPM)			
0.681 (3.0)		0.454 (2.0)	

TABLE XI
UA W/K (BTU/HR-°F) BASED ON SIMULATIONS

	Counterflow	
	Experiment 1	Experiment 2
Simulation	160.8 (304.7)	178.7 (338.7)
Difference	8.9%	10.8%
	Parallel flow	
	Experiment 3	Experiment 3
Simulation	160.4 (304.1)	175.4 (332.4)
Difference	3.1%	2.5%

I. Validation Process

Based on definitions stated by Rykiel [13], "Verification is a demonstration that the modeling formalism is correct." In other words, "Verification is a technical matter that relates to how faithfully and accurately ideas are translated into computer code or mathematical formalisms." and verification is an implicit condition of most commercial software. On the other hand, "Validation is a demonstration that a model within its domain of applicability possesses a satisfactory range of accuracy consistent with the intended application of the model." The comparison of the simulation results with experimental results and the finding that the results fall within the uncertainty of the experimental results, demonstrates that the model possesses a satisfactory range of accuracy. This implies the validity of the CFD model developed for the parallel pipe heat exchanger of the Hampden Model H-6878 apparatus.

VII. VALIDATED CFD MODEL APPLICATIONS

As part of the approach to introduce students to the use of CFD models and understand the benefits of having a validated model, students are asked to do further analysis of the heat exchanger as a post-laboratory activity. By changing the input parameters, the students can investigate the impact of those parameters on the heat exchanger performance. This activity is intended to help students to realize that having a validated model allows them to run simulated experiments without the need of conducting further experiments in the laboratory. As an example, the performance of the heat exchanger in counterflow as a function of the hot fluid mass flow rate is investigated. The inlet temperatures for both the cold and hot fluids are kept constant at 15.6 °C (60°F) and 60.0 °C (140°F), respectively. The mass flow rate of the cold fluid is kept constant at 0.126 kg/s (0.2778 lb/s) ($\propto$ 2.0 GPM), while the mass flow rate of hot fluid is varied between 0.124 kg/s (0.2735 lb/s) ($\propto$ 2.0 GPM) and 0.248 kg/s (0.5471 lb/s) ($\propto$ 4 GPM). To be consistent with the data recorded during the experiments, the flow rate is used to compute the mass flow rate using the density of water at the given inlet temperatures, i.e. 999.6 kg/m^3 (62.40 lb/ft^3) and 983.4 kg/m^3 (61.39 lb/ft^3), respectively. After running the simulations, Table 12 shows the effectiveness of the heat exchanger obtained for the given input parameters.

TABLE XII
HEAT EXCHANGER EFFECTIVENESS AS A FUNCTION OF HOT FLUID MASS FLOW RATE

Hot fluid (in shell)		Heat exchanger
Volume m^3/hr (GPM)	Mass kg/s (lb/s)	Effectiveness (%)
0.454 (2.0)	0.124 (0.2735)	21.6
0.568 (2.5)	0.155 (0.3419)	23.1
0.681 (3.0)	0.186 (0.4103)	24.3
0.795 (3.5)	0.217 (0.4787)	25.0
0.908 (4.0)	0.248 (0.5471)	25.9

VIII. CONCLUSION

In the course MENG 3211 Mechanical Engineering Lab II at the University of Texas at Tyler, a CFD model for the parallel pipe heat exchanger of the Hampden Model H-6878 apparatus was validated and used as part of an approach to introduce students to CFD modeling. Using experimental and simulated data, students are guided to perform analysis of the heat exchanger using both the Logarithmic Mean Temperature Difference method and the ε-NTU method. The use of the model facilitates the introduction of terminology associated with CFD and gives insight into the basic principles of CFD without the need of developing the model that would be part of a CFD course. The comparison of the simulated results with the experimental results allows students to understand the concept of model validation. Further analysis of the heat exchanger using only simulated data allows students to realize the benefits of having a validated CFD model.

APPENDIX

Equations for uncertainty analysis of the counterflow arrangement are shown in Table A-I.

Temperature uncertainty: manufacturers of thermocouple type T state an uncertainty of ± 1°C or 0.75%, whichever one is largest. However, in this study reasonable uncertainties are found with 0.75% of the reading as the lowest value (0.75% reading < 1 °C).

Flow rate uncertainty: uncertainty of rotameters can be as low as 2% and as high as 10%. For this study a value of 5% of the reading is used in lack of information regarding the uncertainty of the rotameters present in the equipment.

Water properties uncertainty: the uncertainties associated to the water density and specific heat are neglected as they are taken for generally accepted values, but both are computed as a function of temperature. For the density, which is used to find the mass flow rate, the water temperature at the inlet is used. For the specific heat, which is used to find heat transfer, the average temperature between inlet and outlets are used.

REFERENCES

[1] D. Adair and M. Jaeger, "Integration of computational fluid dynamics into a fluid mechanics curriculum," *Computer Applications in Engineering Education*, vol. 22, no. 1, pp. 131-141, 2014.

[2] F. Stern, T. Xing, D. Tarbrough, A. Rothmayer, G. Rajagopalan, S. Prakashotta, D. Caughey, R. Bhaskaran, S. Smith, B. Hutchings and S. Moeykens, "Hands-On CFD Educational Interface for Engineering Courses and Laboratories," *Journal of Engineering Education*, vol. 95, no. 1, pp. 63-83, 2006.

[3] S. J. M. Cartaxo and F. A. N. Fernandes, "Educational software for heat exchanger equipment," *Computer Applications in Engineering Education*, vol. 18, no. 1, pp. 193-199, 2010.

[4] T. Barber and V. Timchenko, "Student-Specific Projects for Greater Engagement in a Computational Fluid Dynamics Course," *Australasian Journal of Engineering Education*, vol. 17, no. 2, pp. 129-138, 2011.

[5] R. A. Pieritz, R. Mendes, R. F. A. F. da Silva and C. R. Maliska, "CFD studio: An educational software package for CFD analysis and design," *Computer Applications in Engineering Education*, vol. 12, no. 1, pp. 20-30, 2004.

[6] T. C. Hung, S. K. Wang, S. W. Tai and C. T. Hung, "An innovative improvement of engineering learning system using computational fluid dynamics concept," *Computers and Education*, vol. 48, no. 1, pp. 44-58, 2007.

[7] P. A. Lopez, J. J. Mora, F. J. Martinez and J. Izquierdo, "Computational fluid dynamics (CFD) models in the learning process of Hydraulic Engineering," *Computer Applications in Engineering Education*, vol. 18, no. 2, pp. 252-260, 2010.

[8] CD-adapco, "Using Engineering Software to Teach heat Transfer," 12 April 2016. [Online]. Available: http://www.cd-adapco.com/webinar/using-engineering-software-teach-heat-transfer. (Accessed April 2016).

[9] L. M. F. Lona, F. A. N. Fernandez, M. C. Roque and S. Rodriguez, "Developing an educational software for heat exchangers and heat exchanger networks projects," *Computers and Chemical Engineering*, vol. 24, no. 2-7, pp. 1247-1251, 2000.

[10] T. Pujol, L. Montoro, M. Pelegri and J. R. Gonzalez, "Learning hydraulic turbomachinery with computational fluid dynamics (CFD) codes," *Computer Applications in Engineering Education*, vol. 21, no. 4, pp. 684-690, 2013.

[11] S. Kakac and H. Liu, Heat Exchangers, Selection, Rating, and Thermal Design, New York: CRC Press, 2002.

[12] N. Fumo, "Using Mathcad to Simplify Uncertainty Computations in a Laboratory Course," *Computer Applications in Engineering Education*, vol. 23, no. 2, pp. 250-257, 2015.

[13] E. J. Rykiel, "Testing ecological models: the meaning of validation," *Ecological Modeling*, vol. 90, pp. 229-244, 1996.

TABLE A-I
TAYLOR SERIES EXPANSION METHOD, SUMMARY OF EQUATIONS

Parameter	Uncertainty
$\dot{m} = \rho\, \dot{V}$	$u_{\dot{m}} = \rho\, u_{\dot{V}}$
$q = \dot{m}\, c_p\, (T_o - T_i)$	$u_q = \sqrt{\left(\frac{\partial q}{\partial \dot{m}} u_{\dot{m}}\right)^2 + \left(\frac{\partial q}{\partial T_o} u_{T_o}\right)^2 + \left(\frac{\partial q}{\partial T_i} u_{T_i}\right)^2}$
$C = \dot{m}\, c_p$	$u_C = c_p\, u_{\dot{m}}$
$q_{max} = C_{min}(T_{hi} - T_{ci})$	$u_{q_{max}} = \sqrt{\left(\frac{\partial q_{max}}{\partial C_{min}} u_{C_{min}}\right)^2 + \left(\frac{\partial q_{max}}{\partial T_{hi}} u_{T_{hi}}\right)^2 + \left(\frac{\partial q_{max}}{\partial T_{ci}} u_{T_{ci}}\right)^2}$
$C_r = \frac{C_{min}}{C_{max}}$	$u_{C_r} = \sqrt{\left(\frac{\partial C_r}{\partial C_{min}} u_{C_{min}}\right)^2 + \left(\frac{\partial C_r}{\partial C_{max}} u_{C_{max}}\right)^2}$
$\varepsilon = \frac{q}{q_{max}}$	$u_\varepsilon = \sqrt{\left(\frac{\partial \varepsilon}{\partial q} u_q\right)^2 + \left(\frac{\partial \varepsilon}{\partial q_{max}} u_{q_{max}}\right)^2}$
$UA = C_{min}\, NTU = \frac{C_{min}}{1 - C_r} ln\left(\frac{1 - \varepsilon\, C_r}{1 - \varepsilon}\right)$	$u_{UA} = \sqrt{\left(\frac{\partial UA}{\partial C_{min}} u_{C_{min}}\right)^2 + \left(\frac{\partial UA}{\partial \varepsilon} u_\varepsilon\right)^2 + \left(\frac{\partial UA}{\partial C_r} u_{C_r}\right)^2}$