Reinforcing Conceptual Content in Undergraduate Heat Transfer through the use of CFD

David Calamas and Gyunay Keten
Department of Mechanical Engineering, Georgia Southern University

Abstract

It is extremely difficult for students to conceptualize the various modes of heat transfer as the transport of thermal energy is often invisible to the human eye. With the widespread availability of Computational Fluid Dynamics (CFD) software the ability to visualize the transport of thermal energy has vastly improved. While undergraduate engineering students are often perfectly capable of solving problems out of a textbook they often miss out on the physical significance of the equations that they are using. Common misconceptions about heat transfer, which could be alleviated through the use of proper visualization techniques, often exist even upon course completion. A commercially available CFD software was used in an undergraduate heat transfer course to encourage students to understand the underlying physics associated with the various modes of heat transfer. Quantitative and qualitative results from the following CFD simulation topics were used to increase student understanding of the transport of thermal energy: external forced convection over a tube bank and internal forced convection in a double-pipe heat exchanger. Visualization of the aforementioned topics and their relation to textbook content are presented.

Keywords

Heat Transfer, CFD, Visualization, Learning Modules

Introduction

Undergraduate heat transfer contains numerous topics that require effective visualization to convey the physical significance of the various thermal transport processes taking place. As experimental visualization techniques may be unavailable, or impractical to use due to class sizes, it may be desirable to utilize Computational Fluid Dynamics (CFD) to illustrate the transport of thermal energy. It is not expected, however, that undergraduates will have experience with CFD. Even introductory CFD courses are often first offered at a graduate level. As such, the learning modules presented in this paper do not require a background in CFD. Students will be assigned pre-configured learning modules to help them better understand the transport of thermal energy. These learning modules have already been validated with theoretical and/or experimental solutions and have been verified for grid independence. The mesh in the learning modules has already been generated and the appropriate boundary conditions imposed. In addition, all post-processing (e.g. contour plots, vector plots, etc.) has been completed prior to student interaction. Interaction with the CFD software will be limited to visualization and changing of pre-defined input parameters to study various temperature fields. Several learning modules will be utilized in undergraduate fluid mechanics through the curriculum. Two of the learning modules are shown for brevity. The CFD examples can be found in an introductory

CFD tutorial textbook by Matsson[1] and accompany theoretical solutions found in an introductory heat transfer text by Cengel and Ghajar[2].

Student and Course Background

Heat Transfer is offered during the sixth semester of the undergraduate Mechanical Engineering program at Georgia Southern University. Thus, the course is composed primarily of second semester juniors. Fluid Mechanics, Thermodynamics, Calculus III, and Differential Equations are pre-requite courses. Students must pass pre-requisite courses with a grade of C or better. The textbook utilized for the course is *Heat and Mass Transfer: Fundamental and Applications* by Cengel and Ghajar[2]. McGraw-Hill Connect is used for electronic book access as well as for homework assignments.

Learning Module 1: Flow across Tube Banks

Cross-flow over tube banks is frequently encountered in practice. In shell-and-tube heat exchangers one fluid flows internally through the tubes while a second fluid flows externally over the tubes in a perpendicular, or cross-flow, arrangement. Flow visualization in shell-and-tube heat exchangers is extremely important because the number and configuration of tubes affect the flow pattern and turbulence level downstream. The tubes in a shell-and-tube heat exchanger are typically arranged either in-line or staggered. The arrangement of the tubes is characterized by the transverse pitch, S_T, longitudinal pitch, S_L, and diagonal pitch, S_D, which is defined as

$$S_D = \sqrt{S_T^2 + S_L^2} \tag{1}$$

When the fluid enters the tube bank the flow area decreases because the fluid must flow between the tubes. As the flow area decreases between the tubes the velocity increases. In tube banks, the flow behavior is thus dependent upon the maximum velocity, V_{max}, rather than the approach velocity, V. The Reynolds number that is used to characterize the flow in a tube bank is thus dependent on the maximum velocity, the external tube diameter, D, and the kinematic viscosity of the fluid, v and is defined as

$$Re_D = \frac{V_{max}D}{v} \tag{2}$$

where the maximum velocity for an in-line arrangement is defined as

$$V_{max} = \frac{S_T}{S_T - D} V \tag{3}$$

Once the Reynolds number is known an appropriate Nusselt number correlation can be chosen and used to find the heat transfer coefficient and subsequently rate of heat transfer.

While students may be able to solve tube bank problems without much difficult they may not understand the impact of the number of tubes or tube arrangement on the rate of heat transfer and pressure drop. As such, a CFD model of a tube bank can help students visualize the impact of the

tubes on the velocity, temperature, and pressure fields within a shell-and-tube heat exchanger. An overview of the geometry used to computationally analyze flow behavior in a tube bank can be seen in Fig. 1. The tube bank consists of 6 tubes in the flow direction and 2 rows of tubes. The dimensions of the tube bank can be seen in Table 1.
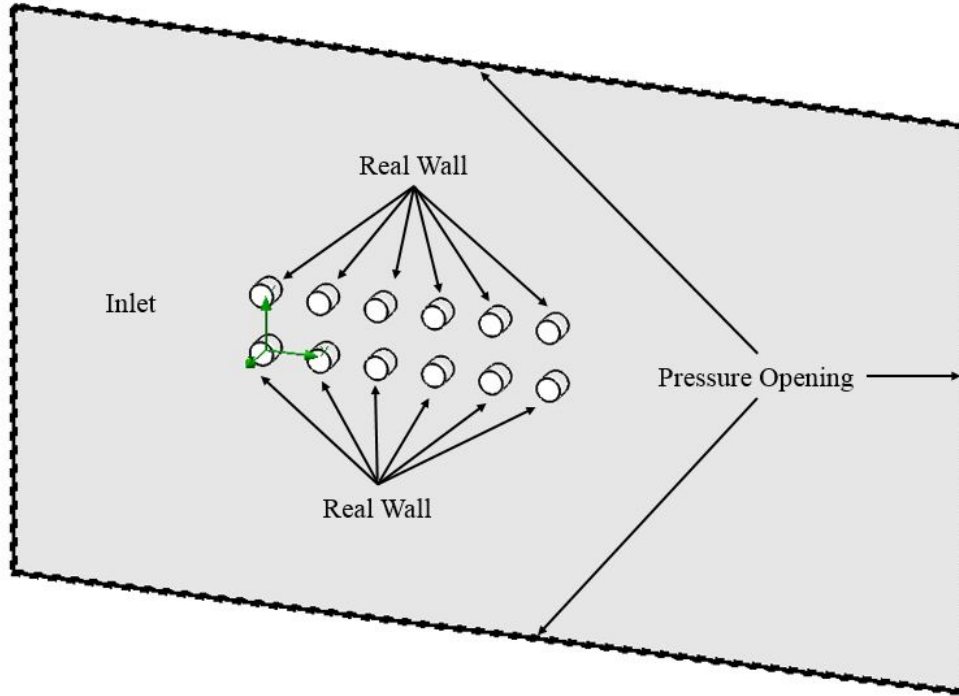

Figure 1. Computational Domain and Boundary Conditions for Flow in a Tube Bank

Table 1. Tube Bank Model Geometry

Arrangement	D (mm)	S_T (mm)	S_L (mm)	S_D (mm)
In-Line	20	50	50	70.71

The boundary conditions utilized in the computational model can be seen in Table 2.

Table 2. Computational Model Boundary Conditions

Boundary Condition	Value
Inlet	Uniform velocity of 4 m/s
Pressure Openings	Static pressure of 101,325 Pa
	Temperature of 293.2 K
Real wall	No-slip condition
	Temperature of 373.2 K

By opening the pre-configured simulations students are able to visualize various aspects of the flow field around the tube banks. For example, a student can visualize a contour plot of the velocity field, temperature field, or pressure field as shown in Figures 2-4 respectively.
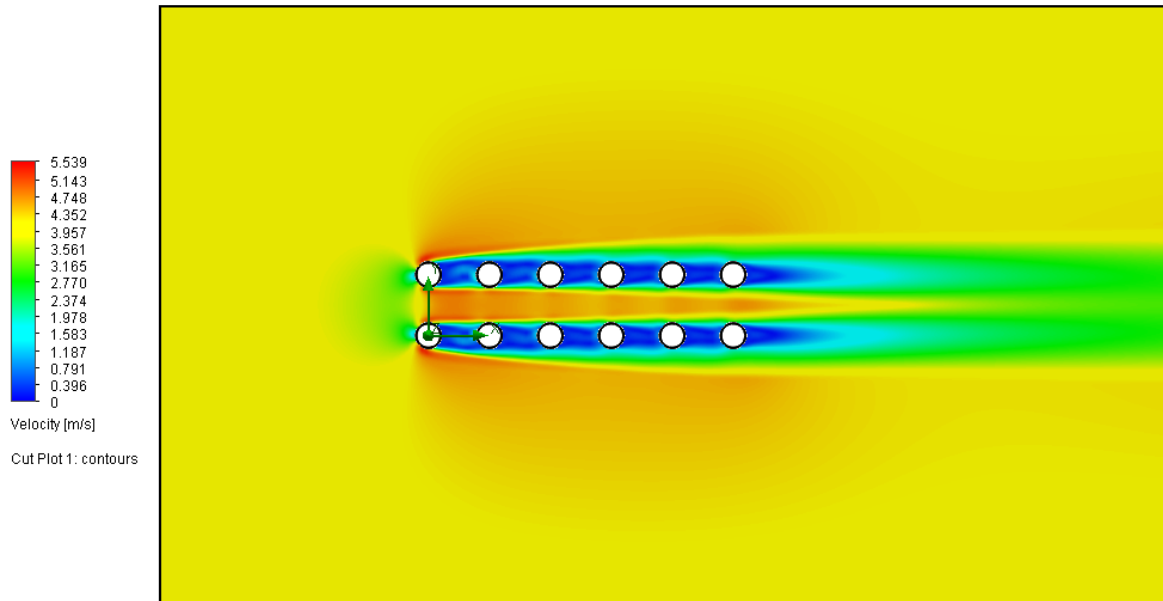

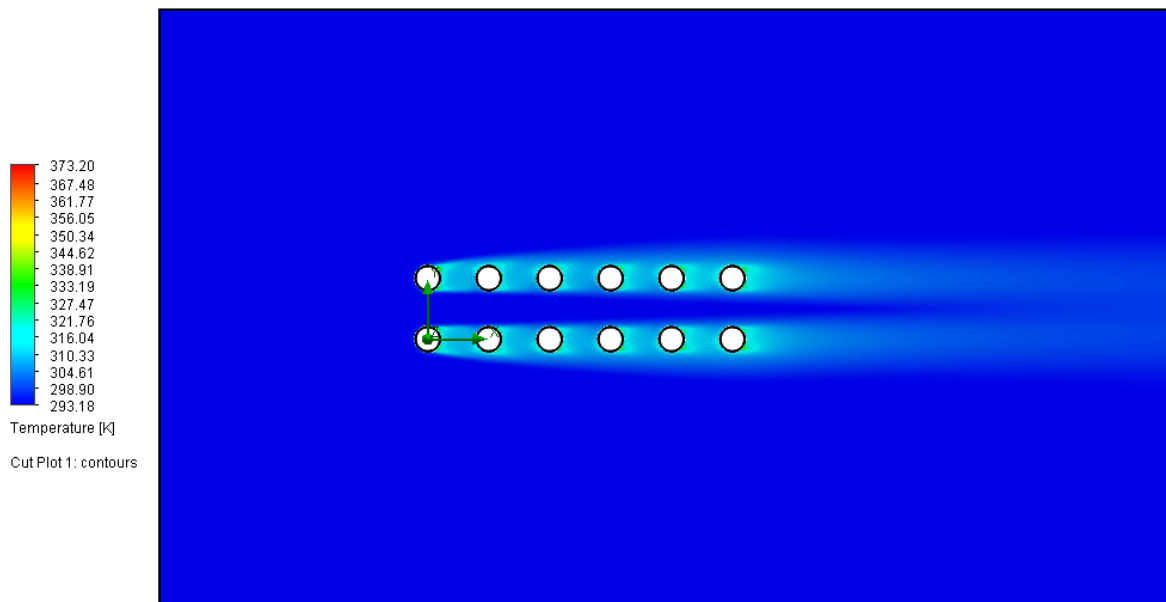**Figure 2. Velocity Field for External Flow over a Tube Bank**

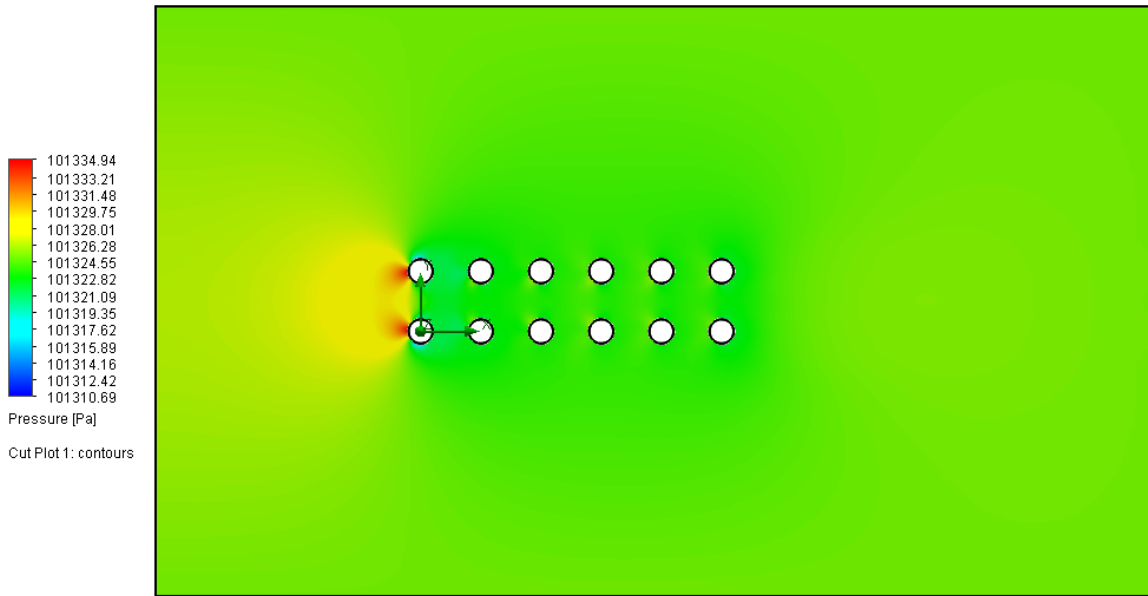**Figure 3. Temperature Field for External Flow over a Tube Bank**

Figure 4. Pressure Field for External Flow over a Tube Bank

Using the results of the simulation the students may calculate the Reynolds number based on the maximum velocity and then choose an appropriate Nusselt number correlation to calculate the heat transfer coefficient. For example, the maximum velocity based on the current in-line configuration and approach velocity can be calculated using Eq. 3. The Reynolds number can then be calculated using Eq. 2. The appropriate Nusselt number based on the Reynolds number is

$$Nu = 0.027 F Re_D^{0.63} Pr^{0.36} \left(\frac{Pr}{Pr_s}\right)^{0.25} \tag{4}$$

where Pr and Pr_s are the Prandtl numbers evaluated at the film and surface temperature, respectively, and F is a correction factor used when the number of tubes in the longitudinal direction, N_L, is less than 16. In the in-line tube configuration used in the computational model there are 6 tubes in the longitudinal, or flow, direction which corresponds to a correction factor of 0.945. Once the Nusselt number has been determined the heat transfer coefficient can be calculated as

$$h = \frac{k}{D} Nu \tag{5}$$

where the thermal conductivity, k, of the fluid is evaluated at the film temperature. Once the heat transfer coefficient is known the exit temperature of the fluid, T_e, can be calculated as

$$T_e = T_s - (T_s - T_i) e^{\left(-\frac{A_s h}{\dot{m} c_p}\right)} \tag{6}$$

where the mass flow rate, $\dot{m}$, is defined as

$$\dot{m} = \rho A_c V \tag{7}$$

The cross-sectional area, A_c, is a function of the number of tubes in the transverse direction, N_T, and is defined as

$$A_c = N_T S_T L \tag{8}$$

Students may compare the exit temperature of the fluid as calculated by the CFD simulation to the theoretical exit temperature as calculated in Eq. 5.

Table 3. Comparison of CFD and Theoretical Fluid Exit Temperature

$T_{e,CFD}$ (°C)	$T_{e,theory}$ (°C)	%$_{difference}$
30.23	30.20	0.1

After students evaluate the rate of heat transfer following the steps presented in their textbook[2] they are then encouraged to alter the inlet velocity boundary condition (and thus Reynolds number) and re-run the simulation to observe the effect of approach velocity on flow behavior. It is hoped that through CFD visualization students will have a better understanding of the impact of tube banks on the rate of heat transfer and pressure drop.

Flow across Tube Banks Concept Survey

The objective of the learning modules presented are to increase student understanding of conceptual content and the physical significance of thermal energy transport in an undergraduate heat transfer course. As such, a concept survey will be given before and after each learning module to assess if the learning modules increased student understanding. The following Fundamentals of Engineering (FE) Exam style concept questions obtained from Cengel and Cimbala[2] will be assigned before and after the first learning module that demonstrates flow across a tube bank:

1. In tube banks the Reynolds number is defined on the basis of
    a. approach velocity.
    b. upstream velocity.
    c. downstream velocity.
    d. maximum velocity.
2. The average heat transfer coefficient for an entire tube bank is determined from experimental correlations based on the dimensionless
    a. Reynolds number.
    b. Euler number.
    c. Fourier number.
    d. Nusselt number.
3. When calculating the heat transfer coefficient a correction factor is often required. The correction factor is a function of
    a. the number of tubes in the longitudinal direction.
    b. the number of tubes in the transverse direction.

      c. the spacing between tubes.
      d. the diameter of the tubes.
4. In flow across tube banks the heat transfer coefficient increases with row number due to turbulence caused by upstream rows.
      a. True
      b. False
5. In tube banks, flow characteristics are dominated by the approach velocity rather than the maximum velocity.
      a. True
      b. False

Learning Module 2: Double-Pipe Heat Exchanger

The simplest type of heat exchanger consists of two concentric pipes and is known as a double-pipe heat exchanger. In a double-pipe heat exchanger one fluid flows through the internal pipe while the other fluid flows through the annular space between the two pipes. There are two flow configurations in a double-pipe heat exchanger. In a parallel-flow configuration both fluids flow in the same direction while in a counter-flow configuration the two fluids flow in opposite directions. There are two methods used in the analysis of heat exchangers. The Log Mean Temperature Difference (LMTD) method is used to select a heat exchanger that will achieve a specified temperature change in a fluid of known flow rate. The Effectiveness-NTU (Number of Transfer Units) method is used to determine the outlet temperatures of the two fluids in a specified heat exchanger. In this learning module a double-pipe heat exchanger has been selected and will be analyzed using the Effectiveness-NTU method. Flow visualization in double-pipe heat exchangers can help students understand the difference between parallel and counter-flow configurations.

When analyzing heat exchangers it is common practice to combine the product of the mass flow rate, $\dot{m}$, and the specific heat, c_p, into a single quantity known as the heat capacity rate, C. The heat capacity rate for a hot and cold fluid is defined as

$$C_h = \dot{m}_h c_{ph} \ and \ C_c = \dot{m}_c c_{pc} \tag{9}$$

In the learning module the hot fluid flows through the tube while the cold fluid flows through the shell. The heat transfer rate in a double-pipe heat exchanger will reach its maximum value when the cold fluid is heated to the inlet temperature of the hot fluid or alternatively the hot fluid is cooled to the inlet temperature of the cold fluid. In addition, the fluid with the smaller heat capacity rate will have a larger change in temperature across the heat exchanger. The fluid with the smaller heat capacity rate will thus reach the maximum temperature first. The maximum possible heat transfer rate can thus be calculated as,

$$\dot{Q}_{max} = C_{min}(T_{hi} - T_{ci}) \tag{10}$$

where the mass flow rate in the tube, $\dot{m}_t$, is defined as

$$\dot{m}_t = \rho_t A_c V_t \tag{11}$$

and the cross-sectional area, A_c, of the tube is

$$A_c = \frac{\pi}{4} D_{it}^2 \tag{12}$$

where D_{it} is the diameter of the inside of the tube. The mass flow rate was a pre-specified inlet boundary condition and with knowledge of the cross-sectional area allows the mean velocity, V_t, in the tube to be calculated as

$$V_t = \frac{\dot{m}_t}{\rho_t A_c} \tag{13}$$

Once the mean velocity is known, the Reynolds number, Re, for the tube flow can be calculated from

$$Re = \frac{\rho V_t D_{it}}{\mu_t} \tag{14}$$

where μ_t is the dynamic viscosity of the tube fluid and ρ_t is the density of the tube fluid. Based on the pre-defined inlet mass flow rate the Reynolds number at the tube inlet is turbulent. The flow is hydrodynamically and thermally fully developed if the hydrodynamic, L_h, and thermal, L_t, entrance lengths are much less than the total length, L, of the pipe. For turbulent flow, the hydrodynamic and thermal entrance lengths can be approximated as

$$L_h = L_t \cong 10D \tag{15}$$

For smooth tubes, the friction factor, f, for turbulent flow can be determined from the first Petukhov equation given as

$$f = (0.790 \ln Re - 1.64)^{-2} \tag{16}$$

Once the friction factor is known, the Nusselt number, Nu, can be calculated using Gnielinski's equation which is defined as

$$Nu = \frac{\left(\frac{f}{8}\right)(Re - 1000)Pr}{1 + 12.7\left(\frac{f}{8}\right)^{0.5}\left(Pr^{\frac{2}{3}} - 1\right)} \tag{17}$$

The heat transfer coefficient for the tube, h_t, can be determined from the Nusselt number correlation as

$$h_t = \frac{k_t}{D_{it}} Nu \tag{18}$$

where k_t is the thermal conductivity of the tube fluid. For the shell fluid, the mean velocity is a function of the hydraulic diameter, D_h. For an annular space, the hydraulic diameter is defined as

$$D_h = D_{is} - D_{ot} \tag{19}$$

where D_{is} is the diameter of the inside of the shell and D_{ot} is the diameter of the outside of the tube. Using the definition of the mass flow rate again, the mean velocity of the shell fluid can be defined as

$$V_s = \frac{\dot{m}_s}{\rho A_c} \tag{20}$$

where the cross-sectional area for the annual space is a function of the hydraulic diameter and is defined as

$$A_c = \frac{\pi}{4} D_h^2 \tag{21}$$

The Reynolds number of the shell is defined in the same manner as the Reynolds number of the tube but is a function of the hydraulic diameter.

$$Re_s = \frac{\rho_s V_s D_h}{\mu_s} \tag{22}$$

The flow in the annular space inside the shell based on the pre-defined inlet mass flow rate is also turbulent and the Nusselt number can be obtained once again from the Gnielinski equation. However, the accuracy of the Gnielinski equation can be improved for annular flow through the use of a correction factor, F, as recommended by Petukhov and Roizen. The correction factor for an adiabatic outer wall is

$$F = 0.86 \left(\frac{D_{ot}}{D_{is}}\right)^{-0.16} \tag{23}$$

Applying the correction factor to the Nusselt number allows the heat transfer coefficient for the annular flow through the shell, h_s, to be determined as

$$h_s = \frac{k_s}{D_h} F Nu_s \tag{24}$$

The total thermal resistance, R, for a double-pipe heat exchanger consists of two convection resistances (one for the tube fluid and one for the shell fluid) and one conduction resistance (for the tube wall). The total thermal resistance can thus be calculated as

$$R = R_t + R_{wall} + R_s \tag{25}$$

where the convection resistances are

$$R_t = \frac{1}{h_t A_t} \; and \; R_s = \frac{1}{h_s A_s} \tag{26}$$

and the conduction resistance for the tube wall is

$$R_{wall} = \frac{\ln\left(\frac{D_{ot}}{D_{it}}\right)}{2\pi k_{wall} L} \tag{27}$$

Once the total thermal resistance for the heat exchanger is known the overall heat transfer coefficient, U, can be determined from

$$\frac{1}{UA} = \frac{1}{U_t A_t} = \frac{1}{U_s A_s} = R \tag{28}$$

The NTU (Number of Transfer Units) is a function of the overall heat transfer coefficient and is defined as

$$NTU = \frac{UA}{C_{min}} \tag{29}$$

The effectiveness, ε, of a double-pipe heat exchanger is a function of the NTU as well as the heat capacity ratio. The heat capacity ratio, c, is the ratio of the tube and shell heat capacity rates and is defined as,

$$c = \frac{C_{min}}{C_{max}} = \frac{C_t}{C_s} \tag{30}$$

Finally, the effectiveness of a parallel-flow shell and tube heat exchanger can be determined from

$$\varepsilon_{parallel} = \frac{1 - \exp[-NTU(1 + c)]}{1 + c} \tag{31}$$

and for a counter-flow heat exchanger can be determined from

$$\varepsilon_{counter} = \frac{1 - \exp[-NTU(1 - c)]}{1 - c \exp[-NTU(1 - c)]} \tag{32}$$

While students may be able to solve heat exchanger problems without much difficult they may not understand the reason why a double-pipe heat exchanger in a counter-flow arrangement is more effective than a double-pipe heat exchanger in a parallel-flow arrangement. As such, a CFD model of a double-pipe heat exchanger can help students visualize the impact of the flow arrangement on the heat transfer rate. Students will also be able to compare the effectiveness of the heat exchanger as calculated using the equations above to the effectiveness of the heat exchanger as calculated using results of the simulation. An overview of the geometry used to

computationally analyze flow behavior in a double-pipe heat exchanger can be seen in Fig. 5. The dimensions of the double-pipe heat exchanger can be seen in Table 4.
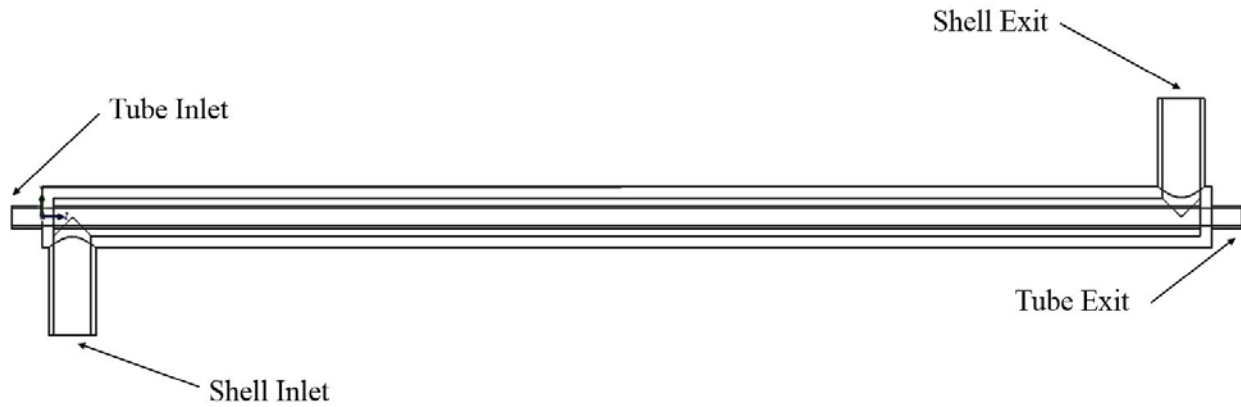

Figure 5. Double-Pipe Heat Exchanger in Parallel-flow Arrangement

Table 4. Double-Pipe Heat Exchanger

Component	Diameter (mm)	Thickness (mm)	Length (m)	Material	Fluid
Tube	19 OD	2mm	1.2	Stainless Steel 321	Water
Shell	32 ID	10 mm	1.0	Stainless Steel 321	Water

The boundary conditions utilized in the computational model in the parallel-flow arrangement can be seen in Table 5.

Table 5. Computational Model Boundary Conditions

Boundary Condition	Value
Tube inlet	Mass flow rate of 0.2 kg/s Temperature of 343.2 K
Shell inlet	Mass flow rate of 0.8 kg/s Temperature of 283.2 K
Tube outlet	Static pressure of 101,325 Pa Temperature of 293.2 K
Shell outlet	Static pressure of 101,325 Pa Temperature of 293.2 K
Interface of tube and shell	Conjugate heat transfer No-slip condition
Shell outer wall	Adiabatic

By opening the pre-configured simulations students are able to visualize various aspects of the flow fields in double-pipe heat exchangers. It should again be noted that these learning modules do not require any input from the students. The results can be pre-loaded by the instructor. This allows students to focus on visualization. For example, students can visualize a contour plot of tube and shell fluid temperatures for a parallel-flow arrangement as seen in Figures 6 and 7 respectively and for a counter-flow arrangement as seen in in Figures 8 and 9 respectively.
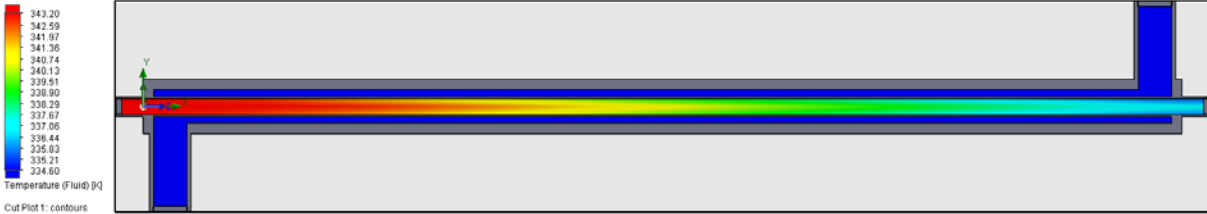

Figure 6. Temperature Profile of Tube Fluid in Parallel-flow Arrangement

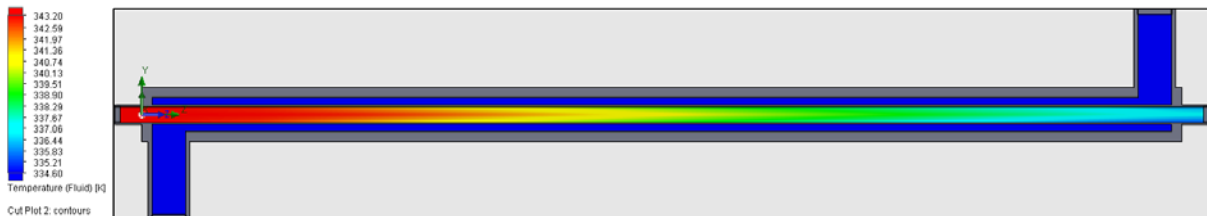**Figure 7. Temperature Profile of Shell Fluid in Parallel-flow Arrangement**

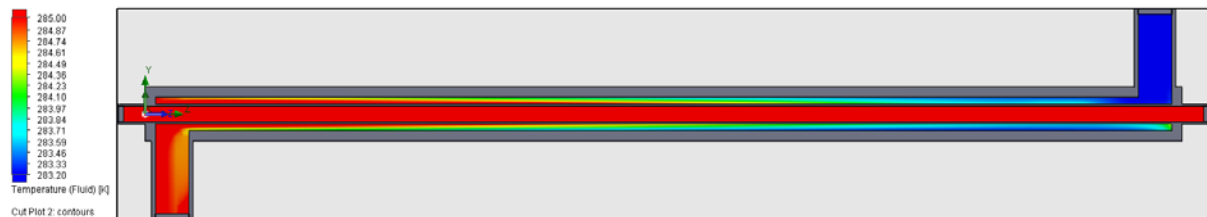**Figure 8. Temperature Profile of Tube Fluid in Counter-flow Arrangement**

Figure 9. Temperature Profile of Shell Fluid in Counter-flow Arrangement

It is hoped that through CFD visualization students will have a better understanding of the impact of flow configuration on double-pipe heat exchanger effectiveness. In addition, students may compare the effectiveness as calculated by the results from the simulation to the theoretical effectiveness as seen in Table 6. Students may alter the input boundary conditions such as the tube and shell mass flow rates or inlet fluid temperatures to see how this impacts performance.

Table 6. Comparison of CFD and Theoretical Effectiveness

ε_{CFD}	ε_{theory}	$\%_{difference}$
0.122	0.131	6.2

Double-Pipe Heat Exchanger Concept Survey

The following Fundamentals of Engineering (FE) Exam style concept questions obtained from Cengel and Cimbala[2] will be assigned before and after the second learning module that demonstrates parallel and counter-flow in a double-pipe heat exchanger:

1. Consider a shell-and-tube water-to-water heat exchanger with identical mass flow rates for both the hot and cold-water streams. The overall heat transfer coefficient and the inlet temperatures are also identical. Now the mass flow rate of the cold water is reduced by half. The effectiveness of this heat exchanger will
   a. increase.
   b. decrease.
   c. stay the same.
   d. not enough information
2. Consider a heat exchanger that has an NTU of 4. Now double the size of the heat exchanger and thus double the NTU to 8. The effectiveness of the heat exchanger will
   a. increase.
   b. decrease.
   c. remain the same.
   d. not enough information
3. For a specified fluid pair, inlet temperatures, and mass flow rates, what kind of heat exchanger will have the highest effectiveness?
   a. Double-pipe parallel-flow
   b. Double-pipe counter-flow
   c. Cross-flow
   d. Multipass shell-and-tube
4. Consider two double-pipe heat exchangers that are identical except that one is twice as long as the other. The heat exchanger with twice the length is more likely to have a higher effectiveness.
   a. True
   b. False
5. The temperature of the hot fluid can drop below the inlet temperature of the cold fluid in a heat exchanger:
   a. True
   b. False

Conclusion

Computational Fluid Dynamics (CFD) can be utilized in an undergraduate heat transfer course to help students visualize various thermal transport processes. In this paper two learning modules were presented to help students visualize external cross-flow over cylinders in a tube bank and internal forced convection in a double-pipe heat exchanger. The learning modules presented do not require prior CFD knowledge or simulation experience. The modules are intended to help students understand the relationship between what they are solving for in a problem to the physical significance of the result they obtained. Pre-learning module and post-learning module concept quizzes were developed and will be implemented to assess the effectiveness of the modules

References

1   Matsson, J., An Introduction to SolidWorks Flow Simulation 2015, SDC Publications, Mission, KS, 2015.
2   Cengel, Y., and A. Ghajar, Heat and Mass Transfer: Fundamental and Applications, 5th Edition, McGraw-Hill, New York, NY, 2015.

David Calamas

David Calamas is an Assistant Professor in the Mechanical Engineering Department at Georgia Southern University. He received a BS in Mechanical Engineering from Clemson University in 2010. He received a MS and PhD in Mechanical Engineering from The University of Alabama in 2012 and 2013 respectively. He currently teaches heat transfer and fluid mechanics. His research interests are in the in the areas of biologically-inspired heat transfer, fluid and thermal transport phenomena, active and passive thermal management, solar energy, as well as engineering education.

Gyunay Keten

Gyunay Keten is a Graduate Research Assistant within the Department of Mechanical Engineering at Georgia Southern University. He is currently pursuing a Master of Science in Applied Engineering. His research is on passive and active thermal management of electronic devices using biologically-inspired fractal geometries.