APPENDIX 1

UNCERTAINTY ANALYSIS

In this section, the uncertainty and error associated with calculation and measurements of several performance limitations are calculated. The errors involved in the measured and resulting parameters were intended based on the accuracy/sensitivity of the determining instruments used in the current study and the least value of the measured parameters. The uncertainty method adopted is,

If an assessed quantity, ‘S’ depends on independent variables like \((x_1, x_2...x_n)\), then the uncertainty in the value of ‘S’ is valued by,

\[
\frac{\partial S}{S} = \left\{ \left( \frac{\partial x_1}{x_1} \right)^2 + \left( \frac{\partial x_2}{x_2} \right)^2 + \ldots \left( \frac{\partial x_n}{x_n} \right)^2 \right\}^{\frac{1}{2}} \tag{A 1.1}
\]

where, \(\frac{\partial x_1}{x_1}, \frac{\partial x_2}{x_2}\), etc. are the independent variables of uncertainty.

A1 ERRORS IN MEASURED QUANTITIES

A1.1 Temperature

Throughout the experiments, RTD of type PT 100 was used to measure the coil temperature at each of five different locations. The least temperature rate attained from the experiments conducted and accuracy of the apparatus is used to evaluate the extreme possible error in the temperature.

\[
\left( \frac{\partial T}{T} \right)_{Exp} = \left( \left( \frac{\partial T_{RTD}}{T_{RTD}} \right)^2 + \left( \frac{\partial T_{ind}}{T_{ind}} \right)^2 \right)^{\frac{1}{2}} \tag{A 1.2}
\]

\[
= 0.457\%
\]
**A1.2 HTF Velocity Measurement**

A digital vane anemometer (Work Zone, AVM–03 model) with ±3.0% accuracy and 0.1 m. s$^{-1}$ resolution was used to measure the air velocity.

\[ = 3\% \]

**Table A1.1 Uncertainty in measured parameters**

<table>
<thead>
<tr>
<th>S. No.</th>
<th>Measured item</th>
<th>Instruments</th>
<th>Value or limit</th>
<th>Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Inner and outer diameter</td>
<td>Vernier caliper</td>
<td>Di = 7 mm, Do = 10 mm</td>
<td>±0.006</td>
</tr>
<tr>
<td>2</td>
<td>Length</td>
<td>Measuring tape</td>
<td>L = 2500 mm</td>
<td>±0.252</td>
</tr>
<tr>
<td>3</td>
<td>Temperature</td>
<td>T sensor</td>
<td>–150 to 150°C</td>
<td>±0.457</td>
</tr>
<tr>
<td>4</td>
<td>Atmospheric temperature, Wind speed</td>
<td>Weather station</td>
<td>–</td>
<td>±0.5%</td>
</tr>
<tr>
<td>5</td>
<td>Pressure drop</td>
<td>Manometer</td>
<td>Limb difference 25cm</td>
<td>±3.5%</td>
</tr>
<tr>
<td>6</td>
<td>Velocity</td>
<td>Anemometer</td>
<td>–</td>
<td>±3%</td>
</tr>
</tbody>
</table>
APPENDIX 2

COMSOL MULTIPHYSICS VALIDATION FOR PERFORMANCE OF HEAT EXCHANGER

In this research work, the performance of the heat exchanger is analyzed using COMSOL Multiphysics simulation approach. The technique used in this research work has been evaluated for improvements in the performance of heat exchanger.

The basic model of Heat exchanger has been taken for the present numerical analysis from Engineering Heat and Mass transfer book. The model is created with the actual size of the heat exchanger component given in the textbook.

A2.1 VALIDATION CHECK 1

In an open heart surgery, under hypothermic conditions, the patient blood is cooled before the surgery and rewarmed afterward. It is proposed to use a concentric tube with a counterflow heat exchanger of length 0.5 m. It is proposed to estimate the temperature of the blood leaving the heat exchanger when water at 60 °C and 0.10 kg/s is used to heat the blood entering the exchanger at 18 °C and 0.05 kg/s. The overall heat transfer coefficient is 500 W/m². K and specific heat of the blood is 3500 J/kg. K.

A2.1.1 Given

Outer diameter of Inner tube \(D_o = 0.06\) m

Inner diameter of Inner tube \(D_i = 0.055\) m
An outer shell of inner diameter = 0.08 m
Length = 0.5 m
Blood inlet temperature $T_{ci} = 18 \, ^{\circ}C$
Mass flow rate of blood $\dot{m}_c = 0.05 \, \text{kg/s}$
Hot water inlet temperature $T_{hi} = 60 \, ^{\circ}C$
Mass flow rate of hot water $\dot{m}_h = 0.1 \, \text{kg/s}$
Specific heat of water to remain constant at $C_p = 4200 \, \text{J/kg.}^{\circ}C$

**Figure A2.1 Counter flow heat exchanger--Case 1**

**A2.1.2 Assumptions**

a) Steady operating conditions.
b) No heat exchange between the system and the surroundings
c) Changes in the kinetic and potential energies of fluid streams are negligible.
d) Heat transfer coefficients and fouling factors remain constant.
e) The thermal resistance of the inner tube is negligible as the tube is thin-walled and highly conductive.
A2.1.3 Solution

\[ C_h = \dot{m}_h C_p = (0.1\text{ kg/s}) \times (4200 \text{ J/kg.}^\circ\text{C}) = 420 \text{ W/}^\circ\text{C} \]

\[ C_c = \dot{m}_c C_p = (0.05 \text{ kg/s}) \times (3500 \text{ J/kg.}^\circ\text{C}) = 175 \text{ W/}^\circ\text{C} \]

Comparing two heat capacities, \( C_{\text{min}} = 175 \text{ W/}^\circ\text{C} \)

The ratio of two specific heat capacities,

\[ C = C_{\text{cold}}/C_{\text{min}} = 175/420 = 0.4167 \]

The number of transfer units,

\[ \text{NTU} = UA/C_{\text{min}} = U\times (\Pi d \times L)/C_{\text{min}} \]

\[ = 500\times (\Pi \times 0.055 \times 0.5)/155 = 0.2468. \]

The effectiveness of heat exchanger in counter flow;

\[ \varepsilon = 0.21 \]

Outlet temperate of blood

\[ \varepsilon = (T_{co} - T_{ci}) / (T_{hi} - T_{ci}) \]

\[ T_{co} = 18 + 0.21 \times (60 - 18) \]

\[ = 26.83 \, ^\circ\text{C.} \]

A2.2 CASE 2

Cold water enters a counter flow heat exchanger at 10 \(^\circ\text{C}\) at a rate of 8 kg/s, where it is heated by a hot water stream that enters the heat
exchanger at 70 °C at a rate of 2 kg/s. Assuming the specific heat of water to remain constant at \( C_p = 4.18 \text{ kJ/kg. °C} \), it is aimed to determine the maximum heat transfer rate and the outlet temperatures of the cold water and hot water streams for this limiting case.

A2.2.1 Given

Diameter of Inner tube \( D_o = 0.06 \text{m} \) & \( D_i = 0.055 \text{m} \)

An outer shell of inner diameter = 0.08 m

Length = 3 m

Cold water inlet temperature \( T_{ci} = 10 \) °C

Mass flow rate of Cold water \( \dot{m}_c = 8 \text{ kg/s} \)

Hot water inlet temperature \( T_{hi} = 70 \) °C

Mass flow rate of hot water \( \dot{m}_h = 2 \text{ kg/s} \)

Specific heat of water to remain constant at \( C_p = 4.18 \text{ kJ/kg. °C} \)

Figure A2.2 Counter flow heat exchanger–Case 2

A2.2.2 Assumptions

a) Steady operating conditions exist.

b) The heat exchanger is well insulated so that heat loss to the surroundings is negligible and thus heat transfer from the hot
fluid is equal to heat transfer to the cold fluid or energy balance is maintained.

c) Changes in the kinetic and potential energies of fluid streams are negligible.

d) Heat transfer coefficients and fouling factors are constant.

e) The thermal resistance of the inner tube is negligible since the tube is thin-walled and highly conductive (Cu).

A2.2.3 Solution

\[ C_h = \dot{m}_h C_p = (2 \text{ kg/s}) \times (4.18 \text{kJ/kg.}^{\circ}\text{C}) = 8.36 \text{ kW/}^{\circ}\text{C} \]

\[ C_c = \dot{m}_c C_p = (8 \text{ kg/s}) \times (4.18 \text{ kJ/kg.}^{\circ}\text{C}) = 33.4 \text{ kW/}^{\circ}\text{C} \]

\[ C_{\text{min}} = 8.36 \text{ kW/}^{\circ}\text{C} \]

Which is the smaller of the two heat capacity rates? Then the maximum heat transfer rate is determined

\[ \dot{Q}_{\text{max}} = C_{\text{min}} (T_{h, \text{out}} - T_{c, \text{in}}) \]

\[ = (8.36 \text{ kW/}^{\circ}\text{C}) (70 - 10)^{\circ}\text{C} \]

\[ = 502 \text{ kW} \]

The maximum temperature difference in this heat exchanger is \( \Delta T_{\text{max}} = T_{h, \text{in}} - T_{c, \text{in}} = (70 - 10)^{\circ}\text{C} = 60^{\circ}\text{C} \). Therefore, the hot water cannot be cooled by more than 60 °C (to 10 °C) in this heat exchanger, and the cold water cannot be heated by more than 60 °C (to 70 °C), no matter what we do. The outlet temperatures of the cold and the hot streams in this limiting case are determined to be
\[
\dot{Q} = C_c (T_{c, \text{out}} - T_{c, \text{in}}) \Rightarrow T_{c, \text{out}} = T_{c, \text{in}} + \left(\frac{\dot{Q}}{C_c}\right) = T_{c, \text{out}} = 10 \, ^\circ \text{C} + (502 \text{kW}/33.4 \text{kW}/^\circ \text{C}) = 25 \, ^\circ \text{C}
\]

\[
\dot{Q} = C_h (T_{h, \text{out}} - T_{h, \text{in}}) \Rightarrow T_{h, \text{out}} = T_{h, \text{in}} + \left(\frac{\dot{Q}}{C_h}\right) = T_{h, \text{out}} = 70 \, ^\circ \text{C} - (502 \text{kW}/8.38 \text{kW}/^\circ \text{C}) = 10 \, ^\circ \text{C}
\]

**A2.3 DISCUSSION**

**A2.3.1 Condition 1**

Note that the hot water is cooled to the limit of 10 °C (the inlet temperature of the cold water stream), but the cold water is heated to 25 °C only when maximum heat transfer occurs in the heat exchanger. This is not surprising, since the mass flow rate of the hot water is only one–fourth that of the cold water, and, as a result, the temperature of the cold water increases by 0.25 °C for each 1 °C drop in the temperature of the hot water.

**A2.3.2 Condition 2**

It is shown that the hot water would leave at the inlet temperature of the cold water and vice versa in the limiting case of maximum heat transfer when the mass flow rates of the hot and cold water streams are identical. It is also possible to show that the outlet temperature of the cold water would reach the temperature limit of 70 °C when the mass flow rate of the hot water is greater than that of the cold water.

\[
\dot{Q} = C_c (T_{c, \text{out}} - T_{c, \text{in}}) \Rightarrow T_{c, \text{out}} = T_{c, \text{in}} + \left(\frac{\dot{Q}}{C_c}\right) = T_{c, \text{out}} = 10 \, ^\circ \text{C} + (502 \text{kW}/8.38 \text{kW}/^\circ \text{C}) = 69.90 \, ^\circ \text{C}
\]
\[
\dot{Q} = C_h (T_{h, \text{out}} - T_{h, \text{in}}) \Rightarrow T_{h, \text{out}} = T_{h, \text{in}} + \left( \frac{\dot{Q}}{C_h} \right) = T_{h, \text{out}} = 70 \, ^\circ\text{C} - (502 \, \text{kW}/33.4 \, ^\circ\text{C}) = 54.98 \, ^\circ\text{C}
\]

A2.4 MODELLING AND SIMULATION

COMSOL Multiphysics software used in this study is a tool for representative the prototype model and determines all kinds of practical engineering problems based on Partial Difference Equation (PDE). Ever since the COMSOL Software is capable to interconnect with different kinds of physical phenomena into a single unit, thus this flexibility not only streamlines the modeling procedure but also drops the computational time duration. Figure A2.3 shows the methodology used for COMSOL Multiphysics analysis of counterflow heat exchanger

A2.4.1 Assumptions

The following assumptions are made for calculating the temperature of outlet for hot and cold water using COMSOL Multiphysics tool.

1. Counterflow is used in this study.
2. The structure has neglected wall thickness.
3. Conjugate heat transfer model has been selected.
4. Copper has been selected as pipe material.
5. The heat exchanger is well insulated so that heat loss to the surroundings is negligible.
A2.4.2 Model Development

Modeling procedure involves few steps which allow the user to manipulate geometry, selection of physics module, material properties, assigning values, defining boundaries, mesh size, and overall conditions to evaluate flow and heat transfer among other parameters. Subsequently, a 3D geometry is necessary to get the model as close as the actual condition.
A2.4.2  Domain

The COMSOL Multiphysics domain comprised of the purpose-built heat exchanger pipe geometry, which was built in order to carry out the numerical simulations combined with achieving direct experimental justification. The model was designed according to the specifications of the Mathematical test section integrating and matching the dimensions. Cylindrical heat exchanger pipes of the exact specification were used, which were oriented vertically to the ground. Figure A2.4 displays the schematic arrangement of the exchanger domain Diameter of Inner tube Do = 0.06m & Di = 0.055m, an outer shell diameter = 0.08 m and with a Length of 0.05 m. for both the case studies.

Figure A2.4 Schematic arrangement of the domain

A2.4.3  Selection of Physics Module

All the walls of the heat exchanger are well insulated throughout the mathematical assumptions. As far as the heat transfer performance of a counterflow heat exchanger is concerned, the heat flow structure has two fields, one solid and the other is fluid. The interaction among the hot and cold flows is through a solid pipe. The main processes that are selected in this model are laminar flow, conduction heat transfer in the solid substrate, and convective heat transfer to the fluid, defines the boundary conditions for the
conjugate heat transfer model. All relevant properties are saved as user-defined fluid and solid.

A2.4.4 Assigning Values and Defining Boundary

In assigning the input values for laminar airflow module and fluid module with heat transfer in the solid substrate, the parameters required are mass flow rates, inlet temperatures, and pipe material. While describing the boundary, the selection of pipe material, wall, and the inlet of cold and hot liquid supply, the outlet of cold and hot liquid supply is required.

Beside inlet mass flow rate (kg/s), air temperature, conjugate heat transfer module also requires properties such as density, viscosity, specific heat capacity, heat capacity ratio, and thermal conductivity. The required values have been generated in software as default input values of air, water, and Cu material properties.

A2.4.5 Boundary Conditions

It is decided to analyze the internal behavior of the fluids within a heat exchanger pipe, their thermal performance on decreasing and increasing temperatures from a hot convection stream. Table A.2.1 & A.2.2 indicates the summarized boundary conditions applied to the heat exchanger. The inlet was varied for all analyzed fluids in order to achieve a direct comparison with analytically obtained results.

| Table A2.1 Input values of conjugate heat transfer module for case 1 |
|---|---|---|---|
| **Type** | **Mass flow rate (kg/s)** | **Temperature (°C)** |
| Blood inlet | Mass flow inlet | 0.05 | 18 |
| Blood outlet | —— | —— | Desired output |
| Water inlet | Mass flow inlet | 0.1 | 60 |
| Water outlet | —— | —— | Desired output |
### Table A2.2  Table Input values of conjugate heat transfer module for case 2

<table>
<thead>
<tr>
<th>Type</th>
<th>Mass flow rate (kg/s)</th>
<th>Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hot water inlet</td>
<td>Mass flow inlet 2 and 8</td>
<td>70</td>
</tr>
<tr>
<td>Hot water outlet</td>
<td>---</td>
<td>Desired output</td>
</tr>
<tr>
<td>Cold water inlet</td>
<td>Mass flow inlet 8 and 2</td>
<td>10</td>
</tr>
<tr>
<td>Cold water outlet</td>
<td>---</td>
<td>Desired output</td>
</tr>
</tbody>
</table>

### A2.4.6  Mesh Generation (Meshing)

Mesh generation is one of the most important processes in any simulation. Meshing is creating partitions of model geometry into reduced outline or area. The quality of the mesh plays a significant role in the accuracy of results and the stability of the solution. If the mesh is too coarse then it will produce a low of component quality, which can cause a higher amount of error in the simulation results. Correspondingly, if the mesh is too finer the elucidation period for the nonlinear system will take extended computational period. An important phase is the size and quantity of mesh elements functional as these properties govern the accuracy, precision, and time taken to the simulation. Thus, it is appropriate to decide the proper types of mesh model that are extra fine, normal, coarse and extra coarse size. For the investigated domain, tetrahedron-meshing technique was applied to the channel and heat exchanger pipes. The patch independent mesh algorithm for tetrahedron elements is based on the following 3–D section procedure, which ensures refinement of the mesh where essential, but retains larger elements where feasible, therefore allowing faster computing times. The higher tenacity of mesh was used on the heat exchanger pipes (near wall mesh refinement) and in close proximity while the lower resolution was used further away from the subject in order to obtain superior precision of results. Figure A.2.5 displays the mesh generation on the computational domain.
Once the model process is completed, the simulation stage can be executed. The simulation process comprises of two stages, where the first stage is regarding of computing the heat transfer in solid and the second stage is fluid properties using laminar flow module computing. The choice of simulation time step is essential for stationary and time-dependent studies as it controls the accuracy resolution and simulation duration.

### A2.4.8 Data Analysis

COMSOL was useful as the Imaging of the problem allowed in the documentation of recirculation regions within the exceptional geometry of the heat exchanger. The results taken from the simulations are the time of heat transfer in the pipe due to airside heat transfer.

### A2.5 RESULT AND DISCUSSION

The main objective of this validation is to study and analyze the behavior of the heat exchanger performance through COMSOL Multiphysics simulation approach. The following figures illustrate the post-processed
numerically arrived results for CASE1 Figure A.2.6 and for CASE 2, Figure A.2.6 (a) & (b).

**Figure A2.6  Numerical simulation of case 1**

**Figure A2.6 (a) Numerical simulation of case 2 conditions 1**
The preliminary concentration of this validation study includes a comparison between the Analytical and Numerical results. The distribution of the temperature across the tube gives the indication of the dissipation of heat. Following the internal investigation involving flow behavior of the heat exchanger working fluids, the temperature analyses were performed along the axial length of test section for all working temperature and with the different mass flow rates tabulated in Tab.4.1 in order to obtain an in-depth comparison between the working fluids.

The numerical results for heat exchanger shown in Table 4.3 are found moreover equal to the experimental values obtained under same operating conditions. The difference between the two values should account for the inherited loses of the heat exchanger.

Figure A2.6 (b) Numerical simulation of case 2 conditions 2
Table A2.3  Comparison of the results from analytical and numerical predictions

<table>
<thead>
<tr>
<th>Type</th>
<th>Mass flow rate (kg/s)</th>
<th>Temperature (°C)</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Analytical</td>
<td>Numerical</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>inlet</td>
<td>outlet</td>
<td>inlet</td>
</tr>
<tr>
<td>CASE 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot water</td>
<td>0.1</td>
<td>60</td>
<td>51.14</td>
<td>60</td>
</tr>
<tr>
<td>Blood</td>
<td>0.05</td>
<td>18</td>
<td>26.83</td>
<td>18</td>
</tr>
<tr>
<td>CASE 2 (Condition 1)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot water</td>
<td>2</td>
<td>70</td>
<td>10</td>
<td>70</td>
</tr>
<tr>
<td>Cold water</td>
<td>8</td>
<td>10</td>
<td>25</td>
<td>10</td>
</tr>
<tr>
<td>CASE 2 (Condition 2)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot water</td>
<td>8</td>
<td>70</td>
<td>54</td>
<td>70</td>
</tr>
<tr>
<td>Cold water</td>
<td>2</td>
<td>10</td>
<td>69</td>
<td>10</td>
</tr>
</tbody>
</table>

A2.6  SUMMARY

Based on the study of validation performances of the heat exchanger through the numerical solution, it is found that the heat exchanger is performing well at all given conditions. Hence, COMSOL Multiphysics is a suitable technique to the heat exchanger for all conditions.