Dynamic Mathematical Model of a Heat Exchanger

By
Alok M. Patel

Departmental Honors Thesis
The University of Tennessee at Chattanooga
Chemical Engineering

Project Director:
Dr. Jim Henry

Examination Date: March 28th, 2003

Project Liaison:
Dr. David Levine

Committee Members:
Dr. Jim Cunningham
Dr. Frank Jones

Examining Committee Signatures:

Jim Henry
Frank Jones
James R. Cunningham

Chairperson, University Departmental Honors Committee

TABLE OF CONTENTS

Abstract

The paper discusses the development of a dynamic mathematical model for a heat exchanger using Lab VIEW software. The condenser and the shell and tube heat exchanger are two different units used in the chemical engineering lab at UTC. The dynamic mathematical models are based on differential equations developed for the condenser and the heat exchanger. These equations are solved using the Fourth Order Runge-Kutta method, a library function in the Lab VIEW software. The models were used to predict the outlet temperatures on the tube side of the condenser and the shell and the tube sides of the heat exchanger.

For comparing the experimental results with the model results for the condenser, a batch distillation experiment (100% reflux, 3500 watts- input to the Reboiler) was run on the distillation column. The model steady state temperature for the tube outlet was approximately equal to the experimental. The model reached steady state faster than the real experiment.

The heat exchanger model simulates the co-current flow in the heat exchanger. Therefore co-current experiments with hot water on the tube side were run on the temperature system. The results from the heat exchanger model were compared with the experimental results. The heat exchanger model reached steady state temperatures faster than the real heat exchanger unit. However the trend followed by the outlet temperatures on the shell and the tube side were similar to the experiment.

1. Introduction

1.1 Problem Definition

The primary objective of this project is to develop a dynamic mathematical model that simulates the heat exchange mechanism in the condenser and the shell and tube heat exchanger. The secondary objective was to compare the transient and the steady state temperature results for the model and the experiment.

The condenser is a part of the distillation column. It is used for condensing the methanol rich vapors produced in the process of distillation. The temperature system utilizes a shell and tube heat exchanger to demonstrate the heat transfer mechanism in fluids. The temperature system and the distillation column are located in Room 115 of Grote Hall, University of Tennessee at Chattanooga.

1.2 Background

1.2.1 Condenser:

The condenser is also a heat exchanger. The one used on the distillation column is a shell and tube type condenser. The condenser is constructed out of glass.

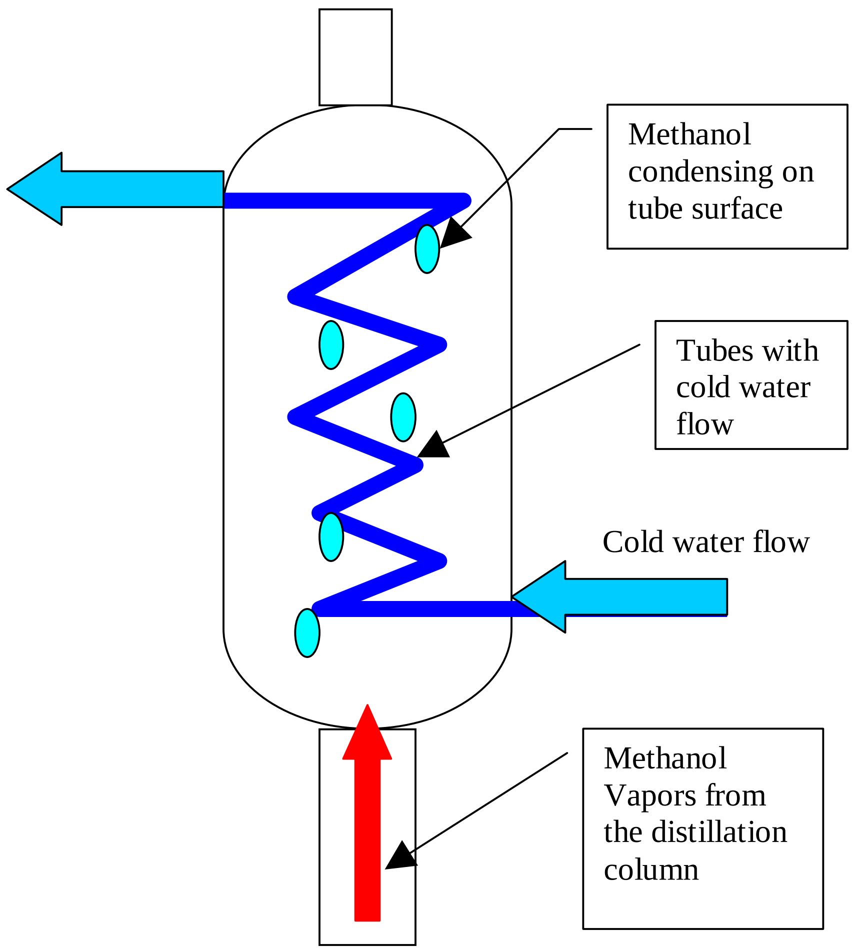


Figure 1.2.1 Condenser on the distillation column

Figure 1.2.1 shows the simple diagram of the condenser used to condense the methanol-rich vapors. The methanol-rich vapors produced during the distillation process rise up the distillation column and end up in the shell side of the heat exchanger. Cold water flows through the tubes of the heat exchanger. As the temperature of the surface of the tubes is below the dew point temperature of methanol-rich, the vapors condense on the tube surfaces and are collected as distillate. The flow of the cooling water is controlled by a manual valve.

Considering the condensing vapors on the shell side at constant temperature the condenser problem is analogous to the constant temperature plate problem. The development of the constant temperature plate equation will be discussed in detail in later section 4.1.

1.2.2 Heat Exchangers

Heat exchangers are devices used to exchange heat from one fluid to another while keeping them separate. There are many different types of heat exchangers. The one that is modeled in this project is a shell and tube heat exchanger. In shell and tube heat exchangers, the hot fluid is flowing on one side of the heat exchanger (either shell or tube) and the cold fluid flows on the other side. The transfer of heat energy takes place due to the temperature difference between the two fluids.

The heat exchange mechanism depends on many other variables such as the heat transfer area, temperature difference, flow rates of the fluids, flow pattern, etc. There are two different flow patterns possible in a shell and tube heat exchanger – co-current (parallel) flow and counter current flow. In the co-current flow pattern, both the fluids flow in the same direction whereas in counter current flow the fluids flow in opposite direction. Figure 1.2.2 shows co-current and counter current flow pattern in a concentric tube heat exchanger.

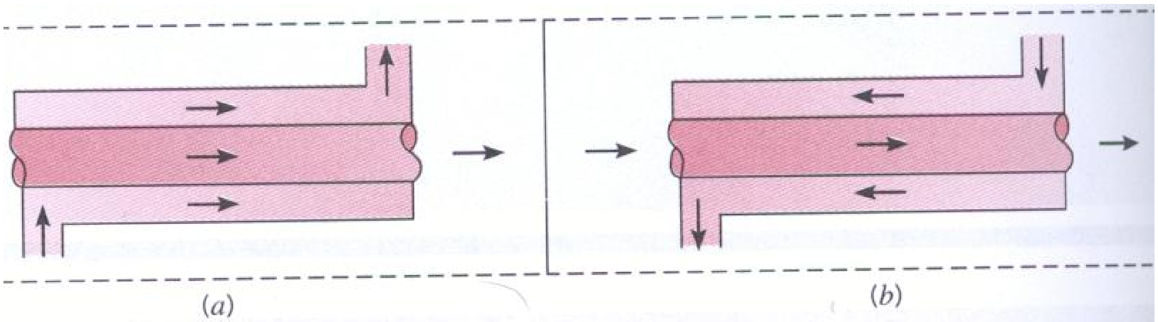


Figure 1.2.2 Concentric tube heat exchanger (a) Parallel flow (b) Counter flow

1.2.3 Temperature System

The temperature system in Grote Hall uses a shell and tube heat exchanger to transfer heat from the hot fluid to the cold fluid. Water is the fluid that flows on both the sides of the shell and tube heat exchanger on the temperature system. A conventional electric water heater is used to heat the water. Both the flow patterns co-current and the counter current are possible on the temperature system.

The temperature system can be run via Internet. Therefore, the user can run this system from any corner of the world using the Internet. Different types of experiments can be run on this system. The user can vary the flow rate of the hot water by varying the speed of the hot water pump. The cold-water flow can be varied by changing the percentage opening for the flow control valve. The hot and cold water can be run on either shell or the tube side. The output from the experiment run on this system is saved on the Internet. The user can save the output data and analyze it using Microsoft Excel.

<u>1.2.4 LabVIEW software</u>:

The mathematical model is designed using LabVIEW software. LabVIEW is the short form for **Lab**oratory **V**irtual **I**nstrument **E**ngineering **W**orkbench. LabVIEW programs are also called VI (Virtual Instruments). LabVIEW can be used for testing, measurements, data analysis and calculations. LabVIEW has a built-in mathematics kit, which can be used to perform different types of calculations. Fourth order Runge-Kutta Method is used to solve the differential equations for the dynamic model simulations developed here.

1.3 Work Scope:

Heat Exchanger Model:

The aim of the project was to develop a dynamic mathematical model for the heat exchanger on the temperature system. For developing the model, the equipment studied was a shell and tube heat exchanger. Considering the flow patterns, the project began with considering the co-current flow heat exchanger model. The counter-current flow was not modeled due to time constraints and model complexity. The model was designed initially considering no heat loss in the system. The fluid flowing on the shell and tube side of the heat exchanger was water.

Condenser:

The second aim of the project was to design a dynamic mathematical model for the condenser located on the distillation column. The condenser results will vary depending on the types of experiment and the liquid mixture used in the distillation column. Currently, methanol-water liquid mixture is used in the distillation column. As a starting point, the experimental result from one particular type of experiment was compared with the model result.

2. NOMENCLATURE:

Symbols:

Symbol		Units
Q	Heat energy	Joules
K	Thermal conductivity	W/m °C
T	Temperature	°C
L	Length	m
H	Heat transfer coefficient	W/m^2 °C
A	Area	m^2
$\dot{m}$	Mass flow rate	Kg/sec
C_p	Specific heat capacity	J/kg °C
ΔT_{LM}	Log Mean Temperature	°C
N	Number of control volumes	N/A
ρ	Density	Kg/m^3
V	Volume	m^3

Subscripts:

s	Shell; surface
m	Metal
t	Tube
surr	Surrounding
rad	Radiation
t, in	Tube inlet
s, in	Shell inlet
t, out	Tube outlet
s, out	Shell outlet

Abbreviations:

CPT	Constant Plate Temperature
HWF	Hot water flow
CWF	Cold water flow
CV	Control Volume

<u>3. Heat Transfer Theory</u>:

Heat transfer is energy in transit due to temperature difference (*Incropera*). Heat transfer can take place in three different modes – Conduction, Convection and Radiation.

Conduction is the transfer of energy across the medium. This occurs because of the temperature difference between the two surfaces of the wall. Equation 3.1 calculates the heat energy transferred through a one-dimensional plate.

$$q_{cond} = \frac{kA(\Delta T)}{L} \tag{3.1}$$

Where q is the heat energy, k is the conductivity coefficient (which depends on material characteristics), ΔT is the temperature difference, A is the surface area of the plate and L is the thickness of the plate. Figure 3.1 shows a clear picture of the process of conduction through a one-dimensional plate.

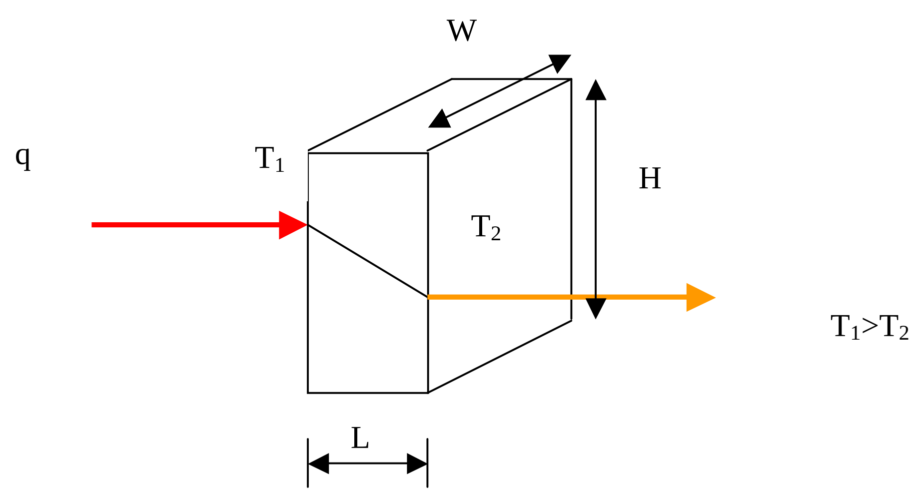


Figure 3.1 Conduction through a one-dimensional plate

Figure 3.1 shows the process of conduction through a block of height H, thickness L and width W. the temperature $T_1 > T_2$, therefore the heat energy is transferred from hot side to the cold side. The heat energy transferred through this block is given by equation 3.2

$$q_{cond} = \frac{k(W * L * H)(T_1 - T_2)}{L} \quad (3.2)$$

Convection is a mode of heat transfer where the transfer of heat energy takes place between a surface and a moving fluid. There are two different types of convection – natural convection and forced convection. For example, when cold water flows over a hot surface it absorbs heat from the surface. If an external factor such as a pump is used to increase or decrease the flow rate of water over the hot plate, the process is called forced convection.

Figure 3.2 shows the process of convection between a fluid flowing over a hot plate. The heat is transferred from the hot plate to cold fluid flowing on top of the plate.

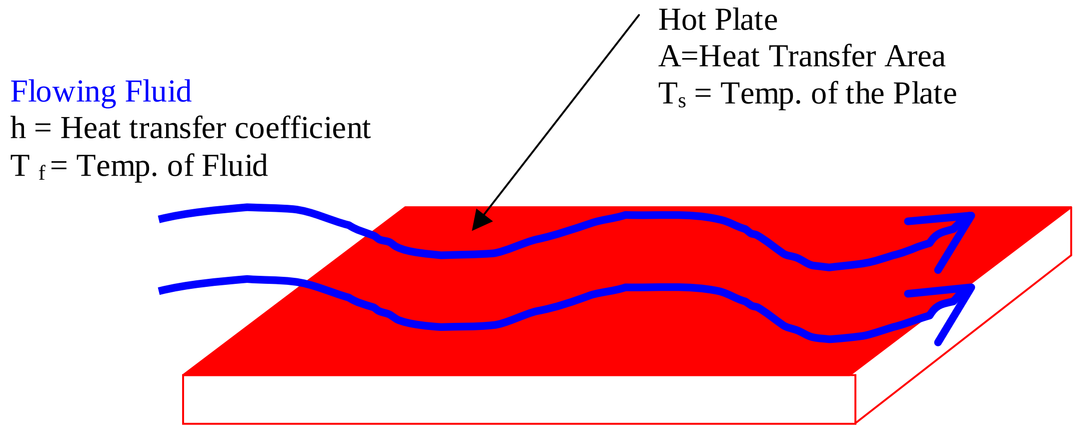


Figure 3.2 Heat transfer from the hot plate by Convection

Equation 3.3 gives the heat energy transferred by process of convection. This expression is also known as Newton's Law of Cooling.

$$q_{conv} = hA(T_S - T_f) \tag{3.3}$$

Where q is the heat transferred, A is the heat transfer area, T_S is the temperature of the surface, T_f is the temperature of the fluid, and h is the heat transfer coefficient (W/m^{2o}C).

<u>Co-Current (parallel) Flow in Shell and Tube Heat Exchangers</u>:

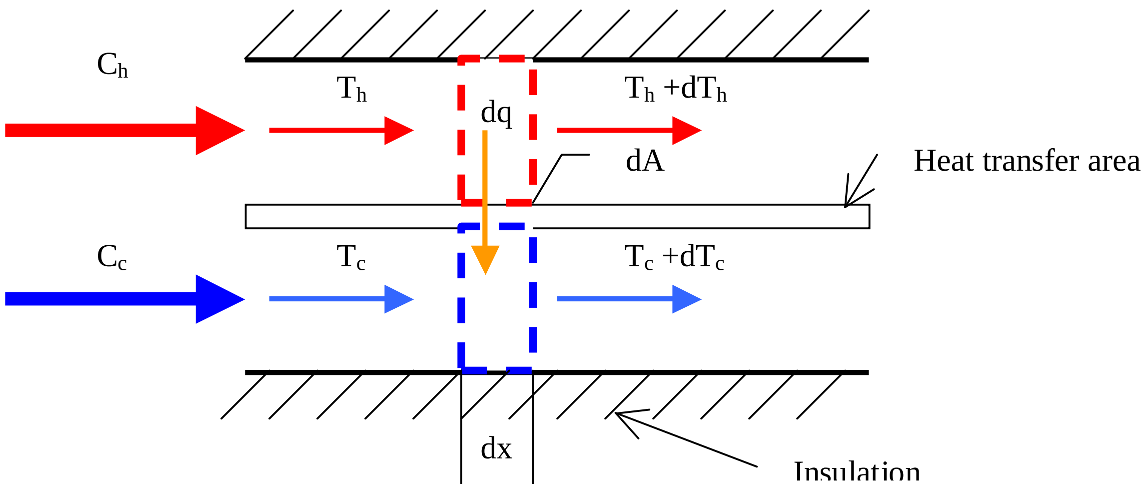


Figure 3.4 Co-current flow heat exchanger

Figure 3.4 shows a model of co-current flow in a heat exchanger (*Incropera, 588*). The heat is transferred from the hot fluid to the cold fluid through the heat transfer area. The hot and cold side is divided into sections called the control volume (CV). dq is the amount of heat energy transferred from the hot side CV to the cold side CV. Figure 3.5 shows the temperature distribution in the co-current flow on the hot and the cold side.

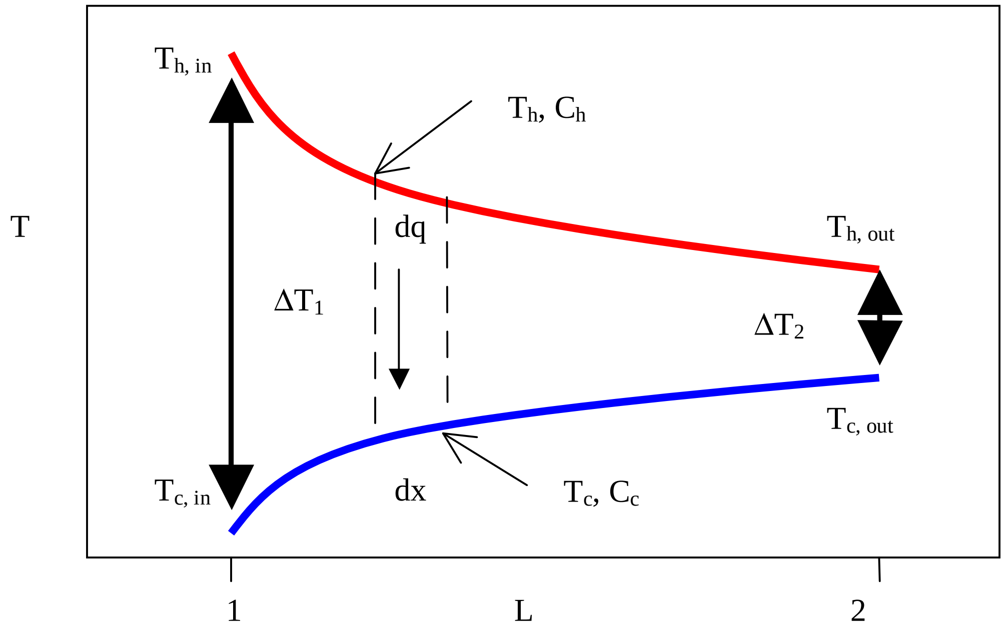


Figure 3.5 Temperature distributions in co-current flow heat exchanger

As shown in figure 3.5, the temperature of the hot fluid T_h decreases and the temperature of the cold fluid T_c increases but they never cross each other. The temperature of the cold outlet will always be less than the temperature of the hot outlet for co-current flow. In figure 3.5, C_h and C_c represent the specific heat capacity of the hot and the cold fluid and dq is the amount of energy transfer through a small section dx from the hot side to the cold. The amount of heat energy transferred from the hot side to the cold side is given by

$$q = UA\Delta T_{LM} \tag{3.6}$$

Where q is the heat energy transferred, U is the overall heat transfer coefficient, A is the heat transfer area and ΔT_{LM} is the Log Mean temperature difference. The Log Mean temperature difference can be calculated using equation 3.7

$$\Delta T_{LM} = \frac{\Delta T_2 - \Delta T_1}{\ln\left(\frac{\Delta T_2}{\Delta T_1}\right)} \tag{3.7}$$

Where $\Delta T_2 = T_{h,\,out} - T_{c,\,out}$ and $\Delta T_1 = T_{h,\,in} - T_{c,\,in}$ for co-current flow only.

4. Dynamic Mathematical Model for a Constant Temperature Plate (CTP):

4.1 Mathematical Model Development

The dynamic mathematical model was developed for studying the heat transfer mechanism from a hot constant temperature plate to a fluid flowing on top of it. This model will be analogous to the condenser. If the shell side of the condenser is at constant temperature it will act as a constant temperature plate and transfer heat to the cold water flowing in the tubes.

Figure 4.1 shows a representation of the constant temperature plate with fluid flowing over the temperature plate.

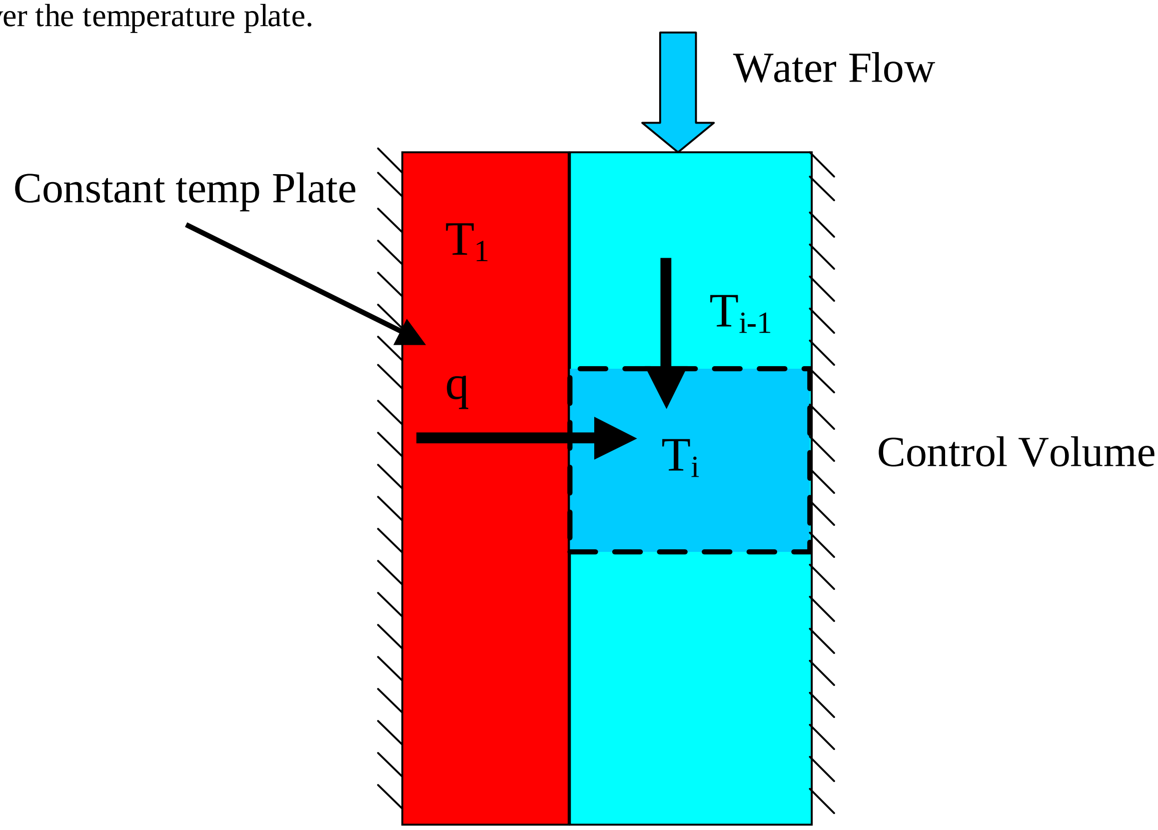


Figure 4.1 Constant Temperature Plate

The control volume method is applied to this problem to determine the outlet temperature of the water. Consider a control volume (CV) ‘i’. The energy storage rate in the CV ‘i’ is equal to the rate of energy gained from the neighboring control volumes. The energy storage rate in the CV is given by equation 4.1.1

$$\rho C_p V \frac{dT}{dt} = q + \dot{m} C_p \Delta T \tag{4.1.1}$$

Where ρ is the density of the fluid, Cp is the specific heat of the fluid, V is the volume of the control volume, $\Delta T = T_{i-1} - T_i$ is the change in the temperature across the control volume, $\dot{m}$ is the mass flow rate of the fluid, q is the energy transferred from the hot surrounding to the cold flowing fluid. Energy is transferred into the control volume due to convection, therefore

$$q = hA(T_1 - T_i) \tag{4.1.2}$$

Where h is the heat transfer coefficient, T_1 is the temperature of the surface, T_i is the temperature of the CV and A is the heat transfer area.

Substituting equation (4.1.2) into (4.1.1) gives us the equation (4.1.3)

$$r C_p V \frac{dT}{dt} = hA(T_1 - T_i) + \dot{m} C_p \Delta T \tag{4.1.3}$$

For achieving a good estimate for the outlet temperature of the fluid, we need to divide the total volume into a number of control volumes, N. The volume and the total heat transfer area are divided by N. By doing so, final equation 4.1.4 is achieved. This equation (4.1.4) is used in the Lab VIEW mathematical model to obtain the outlet temperature of the cold water in the tube section of the condenser.

$$\frac{r C_p V}{N} \frac{dT}{dt} = \frac{hA}{N}(T_1 - T_i) + \dot{m} C_p (T_{i-1} - T_i) \tag{4.1.4}$$

<u>4.2 Constant Temperature Plate (CTP) Physical Parameter Selection</u>:

The CTP model is used to simulate the results similar to the condenser used to condense the methanol vapors in the distillation column. Therefore the physical parameters of the condenser are used as the input in the CTP model equations. Equation 4.2.1 is used in the CPT model to solve for the final outlet temperature of the fluid.

$$\frac{\rho C_p V_t}{N}\frac{dT}{dt} = \frac{hA_t}{N}(T_1 - T_i) + \dot{m}C_p(T_{i-1} - T_i) \tag{4.2.1}$$

The following values were used as input parameters in the Lab VIEW program:

<u>Density of the fluid ρ (kg/m^3)</u>:

The density of water (1000 kg/m^3) was used as the density of the fluid because the fluid used in the tubes of the condenser is water.

<u>Specific Heat of the fluid C_p (J/kg ^{o}C)</u>:

The specific heat of water is used for the specific heat of the fluid flowing over the CTP. $C_{p,\, water}$ = 4180 J/kg ^{o}C

<u>Cold Water Inlet Temperature (^{o}C)</u>:

The cold water inlet temperature input for the model was equal to the temperature of the cold water supplied to the condenser during the experiment. The cold water supply temperature is measured using a temperature sensor on the water inlet of the condenser.

Volume of the tubes V_t(m^3):

The volume of the tubes was not available from the manufacturer; therefore the volume of the tubes was estimated. The tubes in the condenser are in arranged in a helical form. There are four sets of tubes wound in a helical concentric form. Each set of tube has a different inside diameter and length. The values for the inside diameter and the length were estimated and the volume of the tube section was estimated from these values. Table A-1 in the appendix shows the estimated values and the calculated volume for the tubes.

$V_t = 5.52 \text{ X } 10^{-4} \text{ m}^3$

Heat Transfer Area (m^2):

The heat transfer area is the surface area of the tubes. The surface area for the tubes is also calculated using the estimated values for the length and the inside diameter.

$A_t = 0.25 \text{ m}^2$

Volumetric Flow rate of the fluid V (m^3/sec):

The volumetric flow rate of the water is measured by a flow sensor on the cold water supply to the condenser. The flow sensor measures the volumetric flow rate of water in L/min. The volumetric flow rate is converted from L/min to m^3/sec as shown below

$$1\frac{L}{\min}\times\frac{1\min}{60\text{ sec}}\times\frac{1m^3}{1000L}=1.67\times10^{-5}\frac{m^3}{\text{sec}}$$

The flow sensor in the cold water supply line does not give reliable values for the volumetric flow rate for the water therefore the volumetric flow rate was measured manually. Section 4.3 covers the comparison between the manually measured flow rate and the flow sensor readings.

Heat transfer coefficient h (W/m^2 ^{o}C):

The heat transfer coefficient of the condenser was calculated using the data obtained from the experiments run on the distillation column. The heat transfer coefficient 'h' for the cold water in the tubes is assumed to be equal to the overall heat transfer coefficient 'U' of the condenser to simplify the calculation process. The overall heat transfer coefficient U is calculated using equation 3.6

$$q=U A_t \Delta T_{LM} \qquad (3.6)$$

The temperature for the cold water supply CWS (inlet) and the cold water return CWR (outlet) are measured using sensors at the inlet and the outlet water streams to the tubes. The temperature of the surface of the tubes is assumed to be constant.

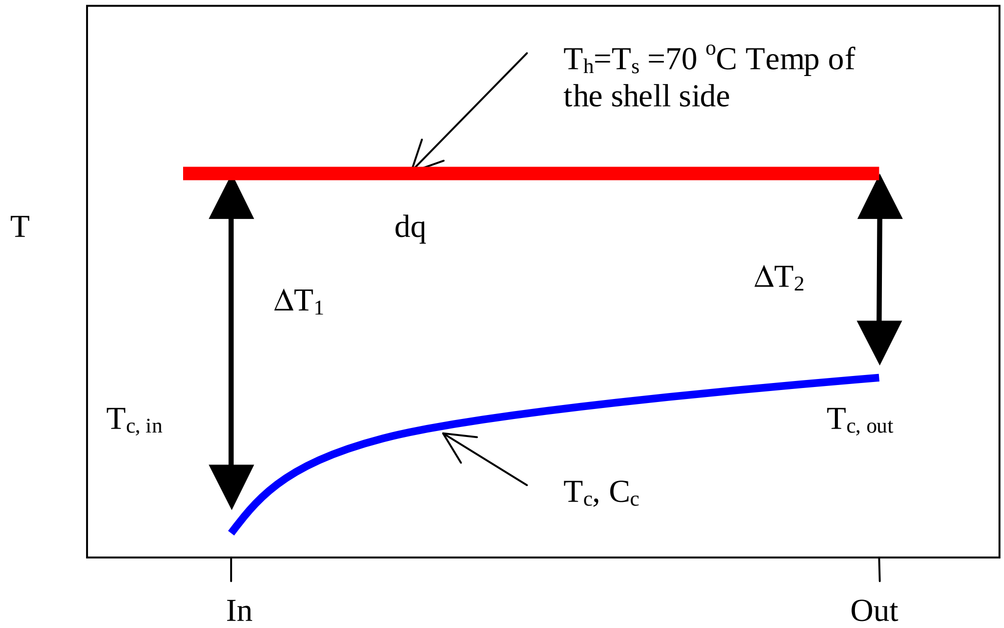


Figure 4.2.1 Temperature distribution in the condenser

Figure 4.2.1 shows the temperature distribution in the condenser. The temperature of the shell side remains constant, equal to 70 ^{o}C. The boiling point of methanol is 67^{o}C and water is 100^{o}C. The vapors have a higher concentration of methanol and therefore 70^{o}C temperature was chosen. Table A-2 in the appendix shows the calculation of the overall heat transfer coefficient of the condenser for batch distillation experiment for 100% reflux and the power added to the Reboiler equals 3500 watts. It was observed that as the flow rate of the cold water was decreased the overall heat transfer coefficient increased. Figure 4.2.2 shows the linear relationship developed between the overall heat transfer coefficient U (W/m^2 ^{o}C) and the cold water flow rate (L/min).

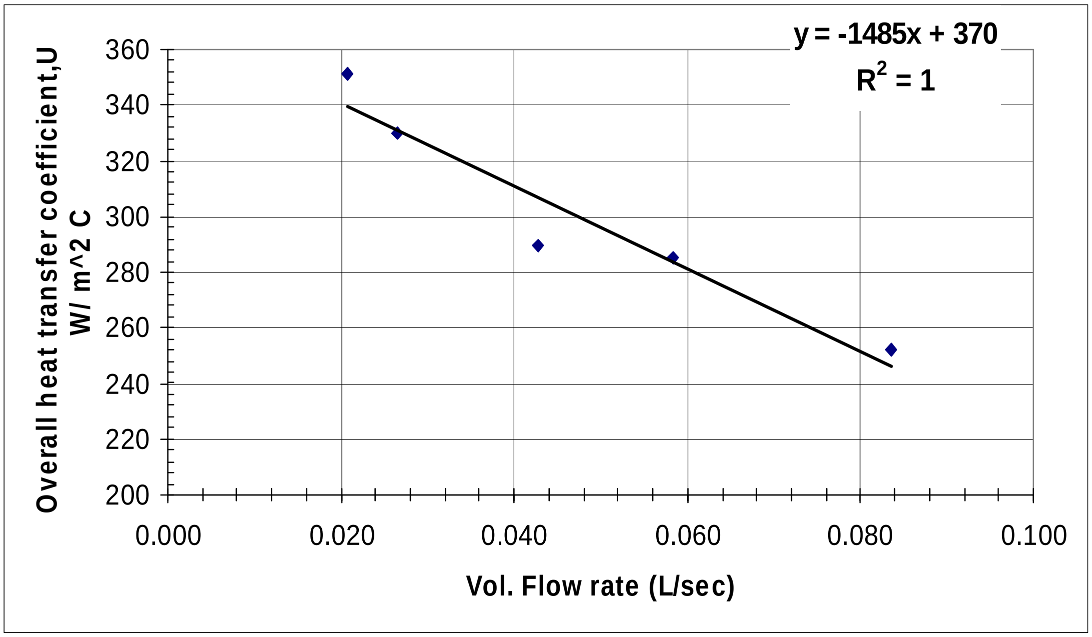


Figure 4.2.2 Overall Heat Transfer Coefficient U vs. Volumetric Flow Rate $\dot{V}$

From figure 4.2.2,

$$U = (-1485 * \dot{V}) + 370 \quad (4.2.1)$$

This relationship is only valid for a batch distillation experiment with 100 % reflux and the power added to the Reboiler equals 3500 watts.

The following table 4.2.1 shows the input values for different parameters which the user will input in the LabVIEW model for the condenser.

<u>Number of Control Volumes, N</u>:

The tube section of the condenser is divided into a number of sections – control volumes. The value for the number of the control volume depends on the user. The value for the number of control volumes should be large enough to obtain reliable results (i.e. cold water outlet temperature). N=100 is a good starting value.

Table 4.2.1 Input Values for the Condenser

INPUTS	VALUE	UNITS
Density of Fluid (water), ρ	1000	Kg/m^3
Specific Heat Capacity, $C_{p,water}$	4230	J/kg ^oC
Heat Transfer Area, A_t	0.25	m^2
Tube side volume, V_t	$5.52 \text{ X } 10^{-4}$	m^3
Volumetric flow rate, $\dot{V}$	Experiment dependent	L/min
Cold Water Inlet Temp. $T_{c, in}$	Experiment dependent	^oC
Surrounding Temp. T_{surr}	Experiment dependent	^oC
Number of CV	100	N/A

4.3 Algorithm for the Mathematical Model for Heat Exchanger and the Condenser:

The algorithm followed by the LabVIEW program is similar for both the heat exchanger and the condenser model. The following steps explain the mechanism followed by the LabVIEW program to calculate the final results.

1. **Input Physical Parameters:** The User inputs the values for the physical parameters of the heat exchanger/condenser and the initial conditions for the temperature and the flow rate. The inlet temperatures and the flow rate are assumed to remain constant through out the experiment unless the user changer them. The reason behind choosing the input values is discussed in the Physical parameter selection section 4.2 and 5.2.
2. **Choosing time start, time end and step rate for the Fourth Order RK Method:** The time start value will always be equal to 0. The time end will be equal to the step rate. The step rate for the condenser model is equal to 0.1. For heat exchanger model the step rate is equal to 0.01. Choosing a value larger value for step rate than the above discussed values sometimes produces unreliable results and makes the model unstable.
3. With these parameters fed into the LabVIEW program, the program will create the differential equations with the physical values plugged in, for the control volume(s).
4. The Fourth Order RK function will solve the differential equations simultaneously for the temperature of the control volume. This temperature

value for the first control volume will be used to solve the temperature of the next control volume. This process will continue till the temperature for all the control volumes of the heat exchanger/condenser are calculated. This array of temperatures for the control volumes will be used as the initial condition for the next time step, which is equal to the step rate specified by the user. The LabVIEW program will plot the temperature value for the last control volume (outlet temperature). The outlet temperatures will increase until they reach steady state. This process can be stopped by the user after the desirable results are obtained. The program saves the results in the form of a text file to the location specified by the user.

5. The simulations are compared with the experimental results using Microsoft excel.

<u>4.4 Flow rate analysis on the condenser</u>:

During the experiments ran on the condenser, it was discovered that the flow sensor on the cold water supply line was reading incorrect volumetric flow rate. The sensor measures the flow rate and displays it on the screen. The flow rate of the cold water was measured manually using one liter jug. The time taken to fill the one liter jug was noted. The volumetric flow rate is calculated by dividing the volume (L) by time (sec). The flow sensor readings were compared to the manually measured flow rate.

Table 4.4.1 Sensor and Actual flow rate on Cold water supply

Sensor Reading (L/sec)	Actual Flow rate (L/sec)
0.007	0.021
0.009	0.027
0.014	0.041
0.019	0.059
0.027	0.084

Comparing the sensor and actual flow rate readings, it is observed that the flow sensor is reading less flow than the actual cold water flow rate. Figure 5.4.1 shows the plot for sensor reading vs. actual flow rate.

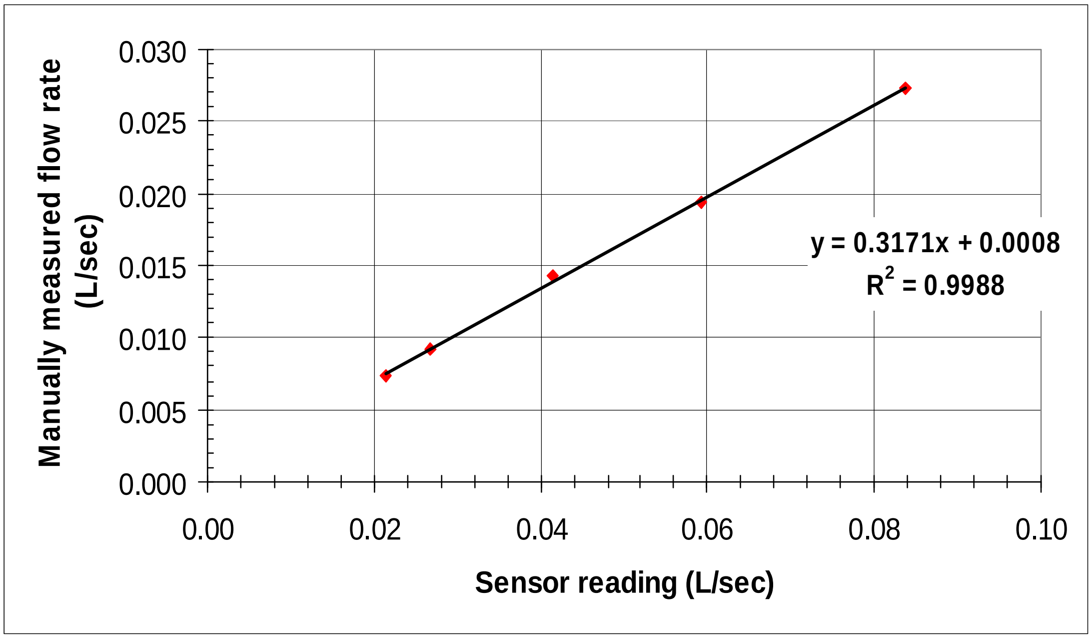


Figure 4.4.1 Sensor reading vs. manually measured flow rate

The trend line equation can be used to correct the sensor reading. The following equation 4.4.1 will correct the sensor flow rate reading and will display the actual flow rate.

Actual flow rate = (0.32 * Sensor reading) – 0.0008 (4.4.1)

Equation 4.4.1 was used to calculate the actual flow rate from the sensor reading. The actual flow rate was used in the LabVIEW mathematical model for the condenser to simulate the heat transfer and achieve results.

4.5 Comparison of Experimental and Simulation Results:

The simulation results obtained from running the Lab VIEW mathematical model are compared to the experimental data obtained from running experiments on the distillation column. The results from the mathematical model are only for one particular experiment run on the simulation column. The experiment was a batch distillation with 100% reflux (no distillate collected), and the power added to the Reboiler was 3500 Watts. The calibration of the flow meter was also performed during the same experiment. Cold water flow through the tubes was initially set low. The cold water outlet temperature was allowed to reach steady state before the flow rate was increased by changing the position of the manual valve. In the similar manner the flow of the cold water was decreased after the temperature had reached steady state. The step up and step down results are discussed below.

Step increase in CWF:

The flow rate of the cold water to the tube section of the condenser was increased during this experimental run. The cold water supply (inlet) and the return (outlet) temperature are measured by the temperature sensors and the data is saved on the computer in the form of a text file. The cold water supply temperature is assumed to be constant for the model. The average supply temperature is used in the mathematical model. Figure 4.5.1 shows the comparison between the simulation data

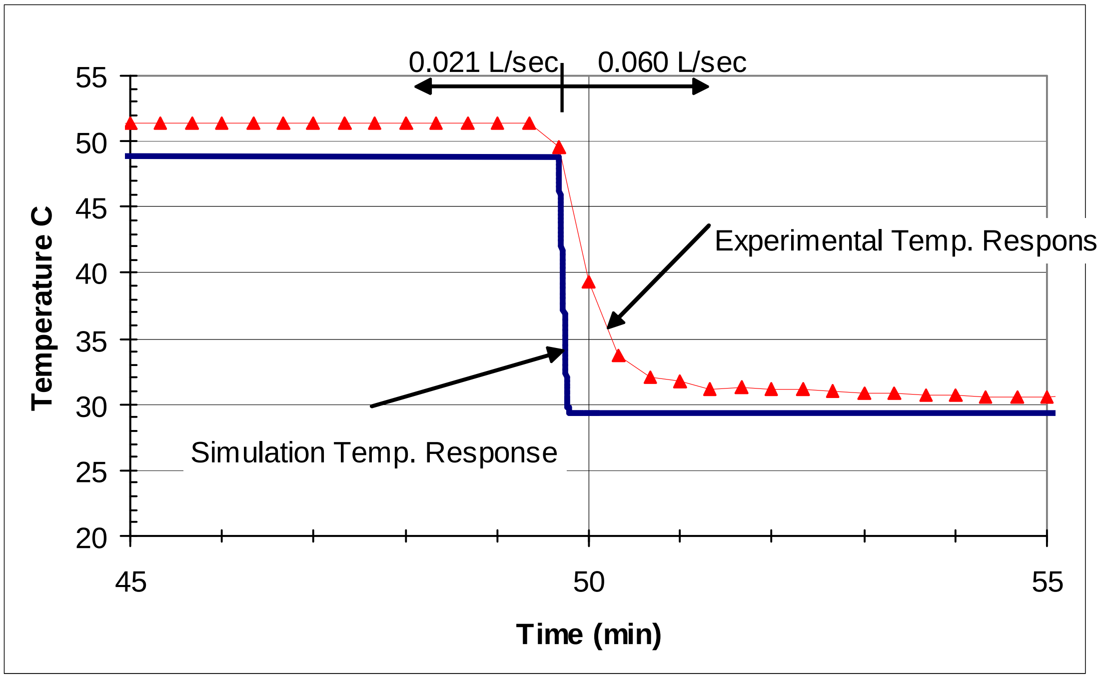


Figure 4.5.1 Step increase in cold water flow rate

In Figure 4.5.1 the cold water return temperature decreases as the flow rate of the water is increased. The model acts faster than the actual experiment. The model reaches steady state temperature in approximately 0.3 minutes and the experiment takes approximately 3 minutes to reach steady state. The difference between the experimental and the model steady state cold water return temperature is approximately 2 ^{o}C before the step occurs and 1 ^{o}C after the step.

<u>Step Decrease in CWF</u>:

The flow rate of the cold water was decreased while running the same experiment on the distillation column. After the cold water return temperature reached steady state the cold water flow rate was decreased.

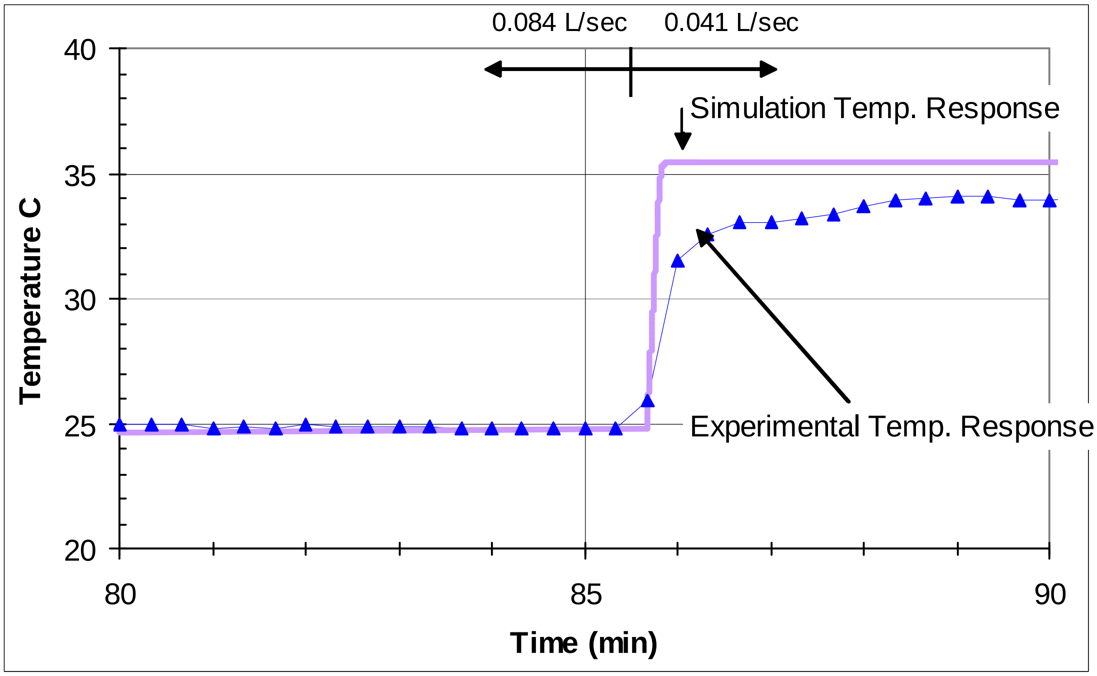


Figure 4.5.2 Step Decrease in cold water flow rate

Figure 4.5.2 shows the simulation and the experimental cold water return temperatures. It is observed that the cold water return temperature increases as the flow rate is decreased. The model output reacts much faster than the experiment, which was also observed in the step increase experiment. The model reaches steady state temperature in approx. 0.1 minutes whereas the experiment takes 4 minutes. The steady state temperature for the model and the experiment match each other before the step occurs. The temperatures differ by approximately 1.5 ^{o}C after the step.

5. Dynamic Mathematical Model for Co-current flow in Heat Exchanger:

5.1 Mathematical Model Development

Figure 5.1.1 shows the three different heat exchanger sections – shell, metal and the tube. These sections are further divided into control volumes. The user inputs the desired number of control volumes into the LabVIEW program.

The following assumptions were made while designing the mathematical model

1. The control volumes are small and assumed to have a constant temperature.
2. The heat exchanger is insulated and there is no heat loss from the heat exchanger to the surrounding.

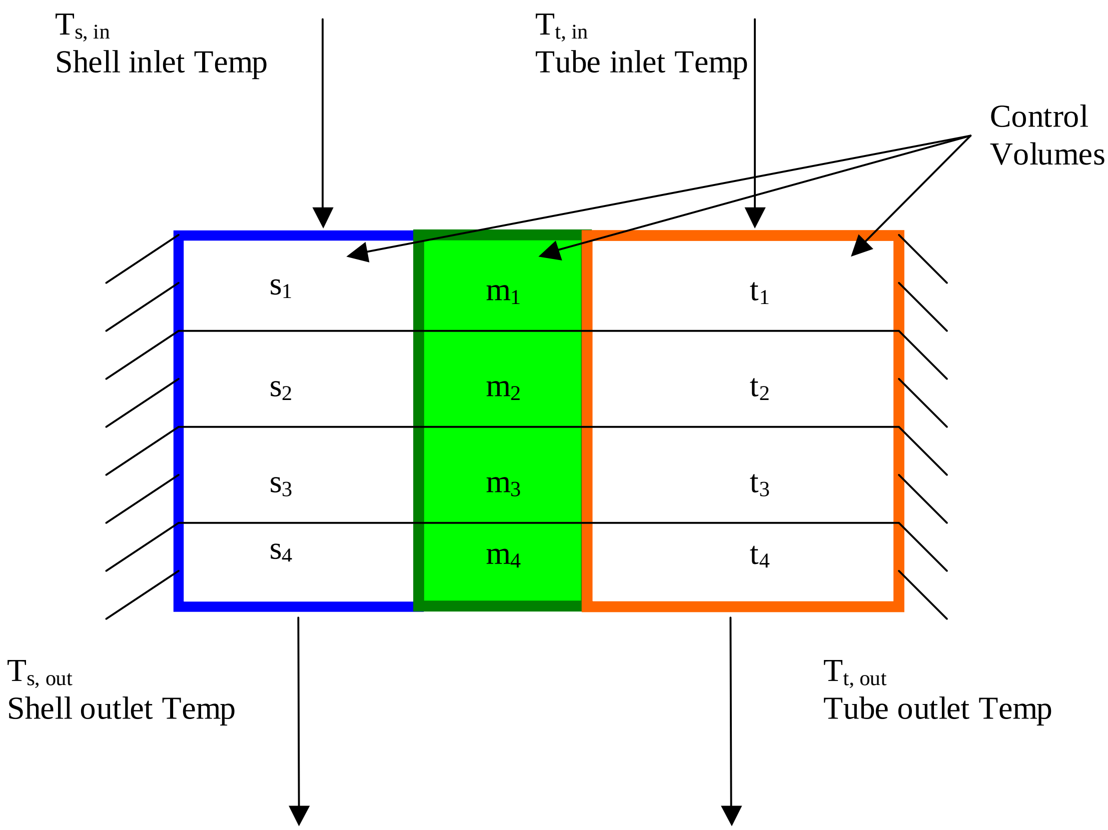


Figure 5.1.1 Shell, Metal and Tube sections of a Heat Exchanger.

The following analysis shows the development of the energy balance equations, which will help in determining the temperature of the control volumes and the outlet temperatures for the shell and tube sides of the heat exchanger. The heat transfer mechanism in the heat exchanger is studied using a set of differential equations. The outlet temperature on the shell and tube side is calculated by solving a set of finite differential equations for the control volumes on the shell, metal and the tube side. A large number of control volumes will make the outlet temperature more precise. However, the disadvantage of having a large number of control volumes is, the Lab VIEW model will run slow and there will be a lot of delay in the display of the final data.

5.1.1 Shell Control Volume (CV) Energy Balance:

Figure 5.1.2 shows the energy gain from the neighboring metal control volume due to convection and the energy gain due to fluid flowing through the shell. The diagram only represents the situation at the entrance of the heat exchanger. But the consequent control volume diagram and the energy balance equations will be similar.

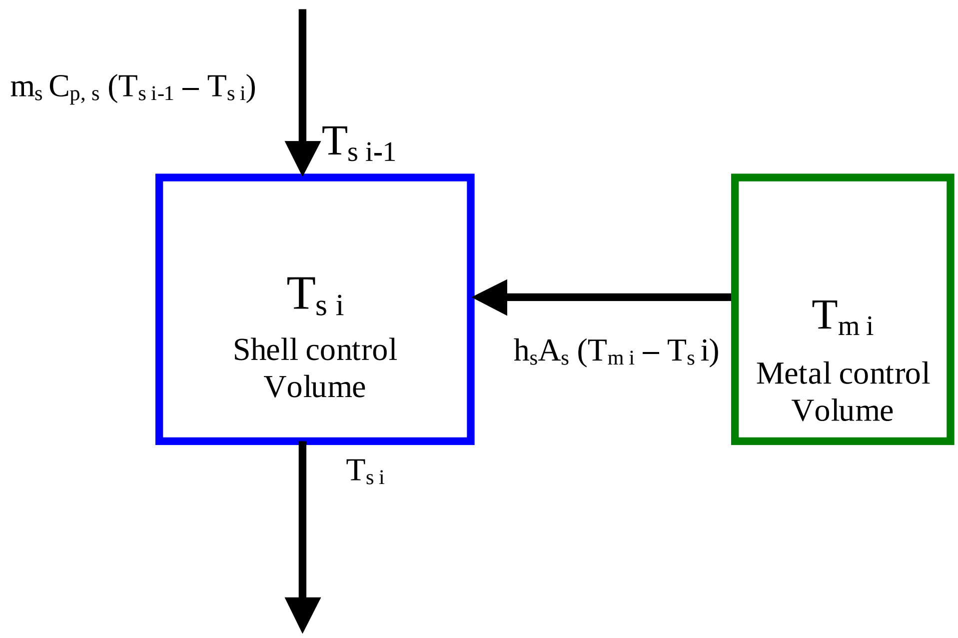


Figure 5.1.2 Energy gain in the shell CV from the neighboring CV

Let's consider the energy storage rate in the shell control volume S_0. It is given by

$\frac{(rCV)_s}{N} * \frac{dT_{si}}{dt}$ where ρ_s is the density of the fluid in the shell, V_s is the volume of the shell, C_s is the specific heat capacity of the fluid in the shell, and $\frac{dT_{si}}{dt}$ is the change in temperature of the shell control volume i with respect to time. As the fluid flows in and out of the control volume, the energy gained by the control volume due to the change in temperature of the fluid flow is given by $\dot{m}_s C_s (T_{si-1} - T_{si})$. $\dot{m}_s$ is the mass flow rate of the fluid in the shell, C_s is the specific heat capacity of the fluid in the shell, T_{si-1} is the temperature of the fluid at the entrance of the shell control volume and $T_{s\,i}$ is the temperature of the fluid at the exit point of the control volume. The shell control volume also gains energy by the process of convection due to the temperature difference between the shell fluid and the metal. The energy gained by convection is given by $\frac{h_s A_s}{N}(T_{mi} - T_{si})$. h_s is the heat transfer coefficient of the fluid in the shell, A_s is the heat transfer area on the shell side, T_{mi} is the temperature of the metal control volume i. The convection term is divided by N, the number of control volumes, because the area and the volume of the heat exchanger are divided into a number of sections called the control volume. The final equation for the energy balance on the shell control volume is given by equation 5.1.1

$$\underbrace{\frac{(\rho CV)_s}{N} * \frac{dT_{si}}{dt}} = \underbrace{\dot{m}_s C_s (T_{si-1} - T_{si}) + \frac{h_s A_s}{N}(T_{mi} - T_{si})} \qquad (5.1.1)$$

Rate of energy stored in the CV= rate of gain of energy from the neighboring CV

T_{si-1} and $T_{s\,i}$ are the temperatures of the shell control volume i-1 and i, and i =1…n. where T_{si-1}=$T_{s,\,in}$ when i=1. The subscripts ‘m’ stands for metal, ‘t’ stands for tube and ‘s’ stands for shell.

5.1.2 Metal Control Volume (CV) Energy Balance:

Figure 5.1.3 shows the energy transfer from the neighboring CVs into the metal CV. Heat energy is gained from the neighboring shell and the tube CVs by the process of convection whereas the gain from the neighboring metal CV is due to the conduction within the metal.

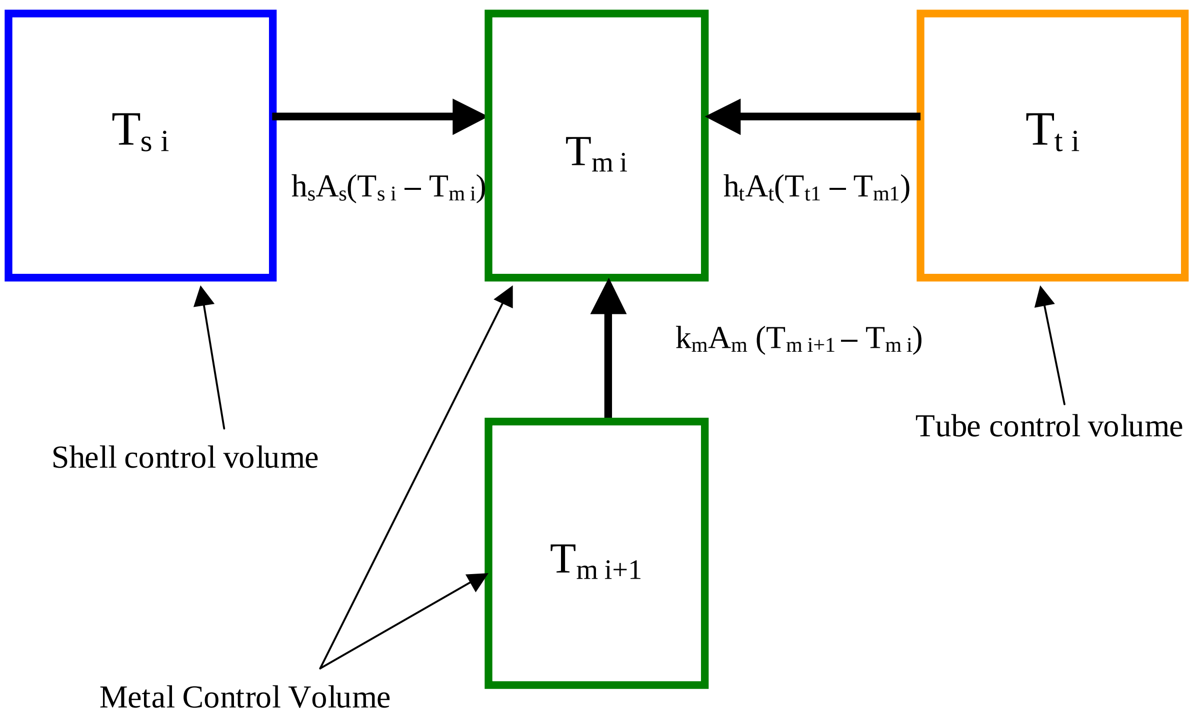


Figure 5.1.3 Energy gain in the metal CV from the neighboring CV

The energy storage rate in the metal control volume is given by $\frac{(rCV)_m}{N} * \frac{dT_{mi}}{dt}$. The metal control volume gains or loses energy from the shell and the tube control volumes due to convection. The energy gained by convection from the shell and the tube side is given by $\frac{h_s A_s}{N}(T_{si} - T_{mi})$ and $\frac{h_t A_t}{N}(T_{ti} - T_{mi})$. The metal control volume also gains energy by conduction within the metal, which is given by $\frac{kA_m}{L}(T_{mj} - T_{mi})$ where k is the conductivity coefficient of the metal. Other terms in the equation are defined in the shell control volume energy balance section. The following equation (5.1.2) is derived by performing the energy balance on the metal CV.

$$\underbrace{\frac{(rCV)_m}{N} * \frac{dT_{mi}}{dt}}_{} = \underbrace{\frac{h_s A_s}{N}(T_{si} - T_{mi}) + \frac{h_t A_t}{N}(T_{ti} - T_{mi}) + \frac{kA_m}{L}}_{}(T_{mj} - T_{mi}) \qquad (5.1.2)$$

Rate of energy stored in the CV= rate of gain of energy from the neighboring CV

In equation 5.1.2 i =1…n, and j= i+1 or i-1.

The energy gained by the conduction will be ignored in the process of calculating the outlet temperatures for the fluid in the shell and the tube section. This term is ignored because the temperature difference between the two metal nodes will be negligible. This will make the conduction term very small compared to other terms.

5.1.3 Tube Control Volume Energy Balance:

The energy balance on the tube control volume is analogous to the energy balance on the shell control volume. Figure 5.1.4 shows the energy gained by the fluid in the metal control volume.

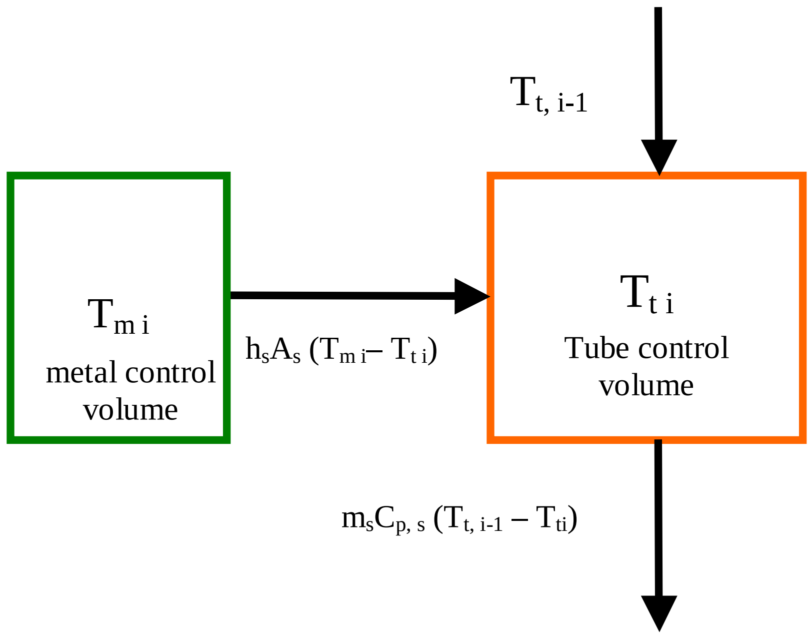


Figure 5.1.4 Energy gain by the tube CV from neighboring metal CV

The energy balance equation is developed in the same manner as the equation developed for the shell control volume. The final differential equation for the rate of energy stored in the tube control volume is given by

$$\frac{(rCV)_t}{N} * \frac{dT_{ti}}{dt} = \dot{m}_t c_t (T_{ti-1} - T_{ti}) + \frac{h_t A_t}{N}(T_{mi} - T_{ti}) \qquad (5.1.4)$$

Where i=1…n.

Differential Equations used in the Heat Exchanger Simulation:

The following three differential equations developed for the shell, metal and tube control volumes are used in the LabVIEW simulation program.

Shell CV

$$\frac{(rCV)_s}{N}\frac{dT_{si}}{dt} = \dot{m}_s C_s (T_{si-1} - T_{si}) + \frac{h_s A_s}{N}(T_{mi} - T_{si}) \qquad (5.1.5)$$

Metal CV

$$\frac{(rCV)_m}{N}\frac{dT_{mi}}{dt} = \frac{h_s A_s}{N}(T_{si} - T_{mi}) + \frac{h_t A_t}{N}(T_{ti} - T_{mi}) \qquad (5.1.6)$$

Tube CV

$$\frac{(rCV)_t}{N}\frac{dT_{ti}}{dt} = \dot{m}_t c_t (T_{ti-1} - T_{ti}) + \frac{h_t A_t}{N}(T_{mi} - T_{ti}) \qquad (5.1.7)$$

The three above differential equations (5.1.5, 5.1.6, and 5.1.7) are solved using the Fourth Order Runge-Kutta Method. The Runge-Kutta Method is a built-in mathematical VI, which is used to solve differential equations.

5.2 Fourth Order Runge-Kutta Method:

The Forth Order Runge-Kutta (FORK) method is used to solve the differential equations for the mathematical model. FORK is used to solve Ordinary Differential Equations (ODE). The mathematical models for both the heat exchanger and the constant temperature plate have differential equations to be solved in order to achieve results. The mathematical model for the heat exchanger has three differential equations to be solved simultaneously to calculate the outlet temperatures for the shell and the tube side. Therefore the FORK function is a powerful tool which will be used in solving the differential equations.

The general form of the FORK is represented by equation 5.2.1

$$y_{i+1} = y_i + \left[\frac{1}{6}(k_1 + 2k_2 + 2k_3 + k_4\right] \bullet h \tag{5.2.1}$$

Where

$$k_1 = f(x_i, y_i) \tag{5.2.2}$$

$$k_2 = f\left(x_i + \frac{1}{2}h, y_i + \frac{1}{2}hk_1\right) \tag{5.2.3}$$

$$k_3 = f\left(x_i + \frac{1}{2}h, y_i + \frac{1}{2}hk_2\right) \tag{5.2.4}$$

$$k_4 = f(x_i + h, y_i + hk_3) \tag{5.2.5}$$

Where y_{i+1} is the value of the function after the step, y_i is the value of the function from the previous time step, and h is the step rate.

<u>5.3 Heat Exchanger Physical Parameter Selection</u>:

The heat exchanger mathematical model is designed to simulate similar results like the shell and tube heat exchanger used in the temperature system. Therefore the physical input values are the same as the ones for the heat exchanger system. To validate the mathematical model with the results of the temperature system, different types of experiments were ran on the temperature system. As the mathematical model does not consider the heat loss the experiments run on the temperature system were limited to hot water on the tube side to minimize the heat loss to the surrounding. It is assumed that the heat loss from the shell and tube heat exchanger is zero.

Using the same input parameters from the experiment the mathematical model was run to study and compare the transient and the final steady state results. The following Values were used as input to the simulation.

<u>Mass flow rate of fluid on shell/tube side (kg/sec)</u>: The values for the mass flow rate of the fluid were equal to the values for the mass flow rate of water on the shell/tube side from the experiment ran on the temperature system. While running the model, the input for the mass flow rate on the shell/tube side was changed by the user after the temperature output reached steady state, with the VI was still running. The value of the mass flow rate was dependent on the user. The user can vary the hot water flow rate by varying the speed of the pump. The cold water flow rate can be varied by changing the flow control valve on the cold water line. While running experiments on the temperature system it was discovered that the flow sensors are not reading the correct flow rate. Therefore the flow rate of the hot water and the cold water were measured at their outlet and compared it with the flow sensor readings. Section 5.4 discusses the comparison between the manual flow readings and the sensor readings.

Density of the Fluid r (kg/m^3): The density of water (1000 kg/m^3) is used as the density of the fluid as water is used on both the shell and tube side of the heat exchanger. The value for density is assumed to remain constant through out the experiment.

Density of Metal r$_m$ (kg/m^3): The construction material for the shell and tube heat exchanger is stainless steel. Therefore the density of the steel is used in the simulation.

ρ_m = 8238 kg/m^3

Specific Heat Capacity of the Metal $C_{p, m}$ (J/(kg ^{o}C)): The value for the specific heat capacity of steel is used in the simulation program.

$C_{p, m}$ = 468 J/kg ^{o}C

Specific Heat Capacity of the fluid C_p (J/(kg ^{o}C)): Water is the fluid used on both the shell and tube side of the heat exchanger on the temperature system. Therefore the specific heat of the water is used in the simulation program as the specific heat of the fluid on both the shell and the tube side. The specific heat of the water is 4230 J/kg ^{o}C.

Shell Side Heat Transfer Area $A_s(m^2)$: The shell side heat transfer area is calculated using the physical data available for the shell and tube heat exchanger on the temperature system. The calculated value for the A_s is used in the mathematical model.

$$A_s = p \times L_{total\ tube} \times D_{Tube\ outside}$$
$$A_s = p \times 27.94m \times 0.0032m = 0.281m^2 \quad (5.3.1)$$

Tube Side Heat Transfer Area $A_t(m^2)$: The tube side heat transfer area is also calculated from the physical data available for the shell and tube heat exchanger on the temperature system.

$$A_t = p \times L_{total\ tube} \times D_{Tube\ inside}$$
$$A_t = p \times 27.94m \times 0.00288m = 0.253m^2 \quad (5.3.2)$$

Shell Side Volume V_s (m^3): Given in the physical data provided by the manufacturer.

$V_s = 2.62 \times 10^{-4}\ m^3$

Tube Side Volume V_t (m^3): The volume of the tube is the total volume of all the tubes in the heat exchanger.

$V_t = 1.43 \times 10^{-4}\ m^3$

Volume of Metal V_m (m^3): The volume of the metal is calculated using the physical data provided by the manufacturer.

$$V_m = \frac{p}{4}(D^2{}_{Tube\ outside} - D^2{}_{Tube\ inside}) \times L_{total\ tube}$$
$$V_m = \frac{p}{4}(0.0032^2 - 0.00288^2) \times 27.94 = 4.27 \times 10^{-5} m^3 \quad (5.3.3)$$

Cold Inlet Temperature (^{o}C): The average temperature of the cold inlet for the experiment run on the heat exchanger system is used as the inlet cold temperature for the mathematical model. The cold water is run on the shell side to minimize the heat loss.

Hot Inlet Temperature (^{o}C): Temperature of the hot inlet for the mathematical input is equal to average hot inlet temperature to the shell and tube heat exchanger. The hot water is run on the tube side of the heat exchanger to minimize the heat loss to the surrounding.

Heat Transfer Coefficients for the shell and the tube side h_s and h_t (W/m^2 ^{o}C): The heat transfer coefficients for the shell and the tube side depend on many factors such as the construction of the heat exchanger, the flow rates, temperature, lay-out pattern of the tubes, etc. The shell and tube heat exchanger on the temperature system also has baffles in the shell section to increase the heat transfer. This makes the flow pattern in the shell side complicated. Therefore the heat transfer coefficients for both the shell and the tube side are assumed to be equal. The values for the heat transfer coefficients were calculated using the following equation 4.3.4

$$\frac{1}{UA} = \frac{1}{h_s A_s} + \frac{1}{h_t A_t} \qquad (5.3.4)$$

Where U is the overall heat transfer coefficient (W/m^2 ^{o}C), h_s and h_t are the heat transfer coefficients on the shell and the tube side (W/m^2 ^{o}C), A_s and A_t are the heat transfer areas on the shell side and the tube side, and 'A' the overall heat transfer area A=0.27m^2 (provided by the manufacturer). The values for the areas are assumed to be equal to simplify the calculations further. The overall heat transfer coefficient U was calculated using equation

$$q=UA_t\Delta T_{LM} \qquad (3.6)$$

The temperature and the flow rate data from co-current experiments on the shell and tube heat exchanger was used to calculate the overall heat transfer coefficient. Table A-3 in the appendices shows the calculated overall heat transfer coefficient U for 3 different co-current experiments with different cold water flow rate. The average value for U is used for calculating the heat transfer coefficient on the shell and the tube side. Equation 4.3.4 simplifies down to the following equation after the assumption of equal area on the shell and the tube side.

$$\frac{1}{U_{average}}=\frac{1}{h_s}+\frac{1}{h_t} \qquad (5.3.4)$$

Substituting $U_{average}$=1081 W/m^2 ^{o}C in equation 4.3.4, we obtain h_s=h_t=2162 W/m^{2o}C. The values obtained for the heat transfer coefficient on the shell and tube side from the above analysis will be used in the mathematical model as inputs.

Number of Control Volumes, N:

The value for number of control volumes is equal to 50. Value greater than 50 yielded same results and response as N=50. Choosing a larger value for control volume will make the mathematical model for the heat exchanger slower.

Table 5.3 lists all the values that the user has to input in the heat exchanger program.

Table 5.3 Input Values for the Heat Exchanger Mathematical Model

INPUTS	VALUE	UNITS
Density of Fluid (water), ρ	1000	Kg/m^3
Density of Metal (steel), ρ_m	8238	Kg/m^3
Specific Heat Capacity, $C_{p,\ metal}$	468	J/kg ^oC
Specific Heat Capacity, $C_{p,water}$	4230	J/kg ^oC
Shell Heat Transfer Area, A_s	0.281	m^2
Tube Heat Transfer Area, A_t	0.253	m^2
Shell side volume, V_s	2.62×10^{-4}	m^3
Volume of Metal, V_m	4.27×10^{-5}	m^3
Tube side volume, V_t	1.43×10^{-4}	m^3
Heat transfer coeff. Shell h_s	2162	$W/m^{2o}C$
Heat transfer coeff. Tube h_t	2162	$W/m^{2o}C$
Mass flow rate, $\dot{m}$	Experiment dependent	Kg/sec
Cold Water Inlet Temp. $T_{c,\ in}$	Experiment dependent	^oC
Hot Water Inlet Temp. $T_{h,\ in}$	Experiment dependent	^oC
Number of control volume	50	N/A

<u>5.4 Flow Rate Analysis on the Heat Exchanger</u>:

Since this model was to be compared with experimental system, careful measurements on the system were made. After analyzing some experiments on the temperature system it was noticed that the flow sensors on the hot and cold water streams were reading incorrect flow rate. Therefore the flow rate was manually measured on the hot and the cold water outlet. The volume of water flowing at the outlet was measured using one liter measuring jug. Time taken to fill the one liter jug was noted down. The volumetric flow rate (L/sec) was equal to the mass flow rate (kg/sec) (only for water) because one liter of water weighs one kg.

<u>Hot water flow rate</u>:

Table 5.4.1 Hot Water Sensor reading vs. manual readings

% Pump Input	Sensor Reading kg/sec	Actual measured Flow kg/sec
10	0.000	0.000
30	0.0088	0.018
40	0.017	0.022
50	0.026	0.030
60	0.034	0.038
70	0.042	0.047
80	0.050	0.056
90	0.061	0.065
100	0.073	0.076

Table 5.4.1 shows the flow sensor readings and the manual flow rate readings. It is observed that the manual readings and the sensor readings do not match. The plot for % pump input vs. the flow rate gives a clear picture of the mismatch.

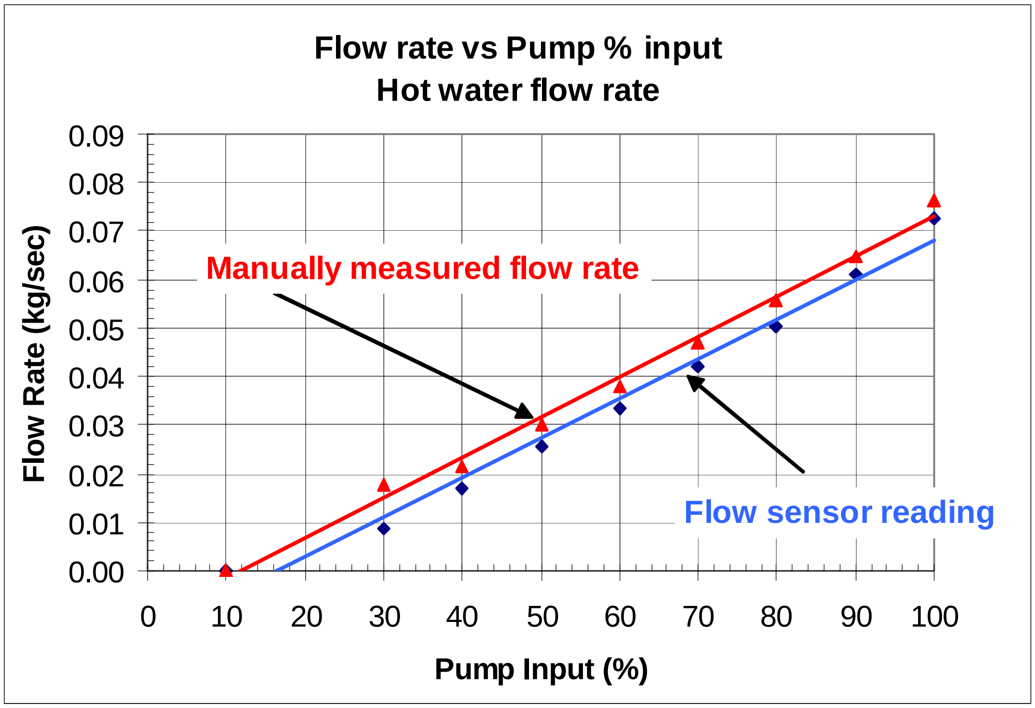


Figure 5.4.1 Comparison between the sensor and manually measured flow rates

Figure 5.4.1 shows that the manual and the sensor flow rates do not match. This implies that the flow rate measured by the flow sensor is unreliable. From the plot we can conclude that the sensor is reading a lower hot water flow rate than the actual flow rate measured manually. The sensor reading can be adjusted to read the actual flow rate by using the following equation

$$\text{Adjusted flow rate} = 0.98 * \text{sensor flow rate} + 0.0051 \qquad (5.4.1)$$

Figure 5.4.2 shows the adjusted flow rate and the actual flow rate.

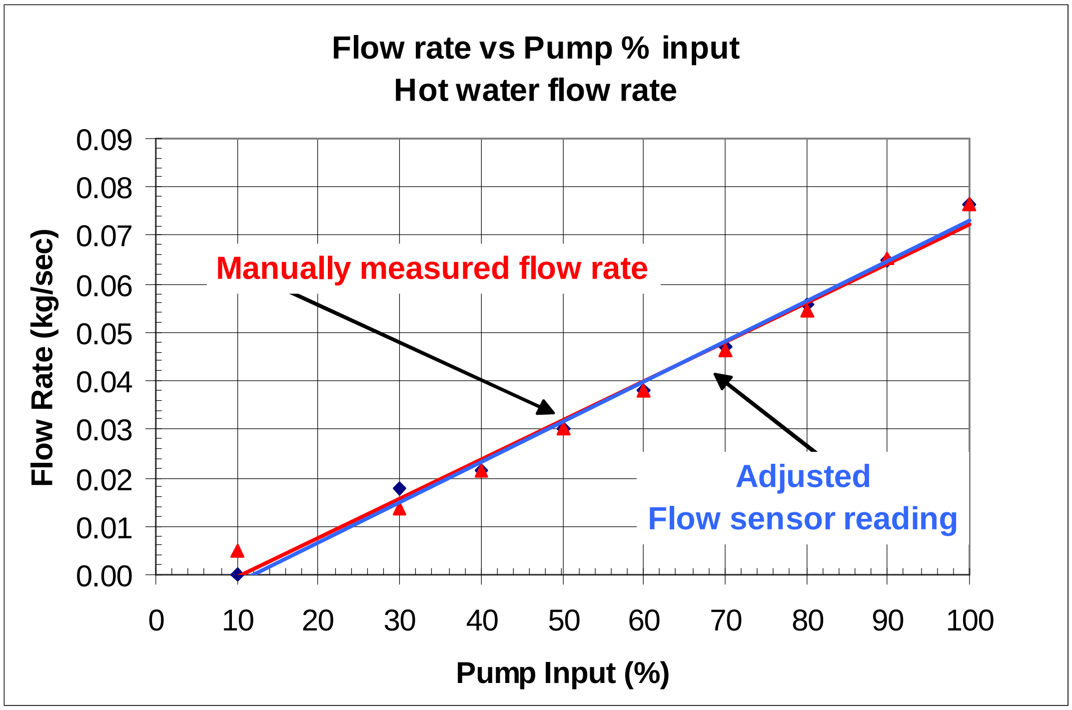


Figure 5.4.2 Adjusted flow rate and manually measured flow rate

From Figure 5.4.2 it is observed that the adjusted flow rate and the actual manually measured flow match each other satisfactorily.

<u>Cold water flow rate</u>:

Table 5.4.2 Cold water flow rate – sensor vs. manual readings

% Pump Input	Sensor Reading kg/sec	Actual measured Flow kg/sec
0	0.019	0.000
20	0.036	0.029
30	0.042	0.036
40	0.042	0.041
50	0.047	0.047
60	0.048	0.046
70	0.049	0.049
80	0.049	0.050
90	0.049	0.050
100	0.049	0.050

Table 5.4.2 shows the sensor and the manual flow rate readings for the cold water flow. The cold water flow rate is varied by varying the % valve opening of the flow control valve on the cold water supply line. It can be observed from the table 5.4.2 that sensor readings match the actual flow rate for 40% to 100% valve opening. But for valve position less than 40% the flow sensor reading does not match the actual cold water flow rate.

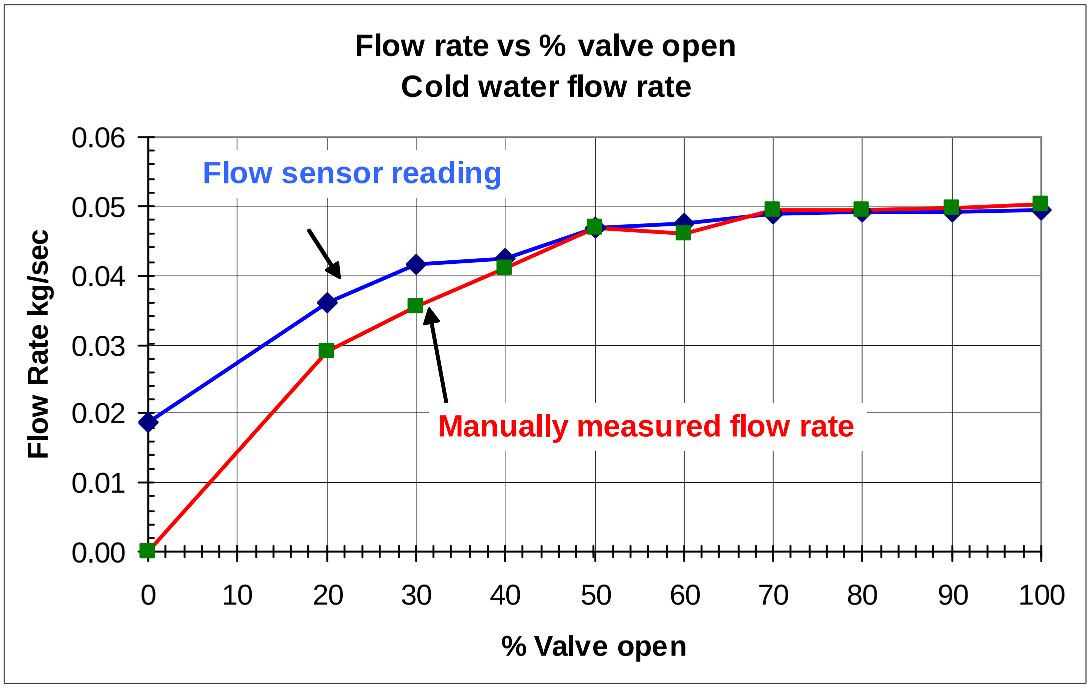


Figure 5.4.3 Comparison between sensor and manually measured flow rate

From Figure 5.4.3 it is observed that the flow sensor reading and the actual flow rate do not match each other for valve position less than 40%. The flow sensor readings can be adjusted using the following relationship for valve position less than 40%.

Adjusted sensor flow rate = (1.58 * sensor reading) – 0.03 (4.4.2)

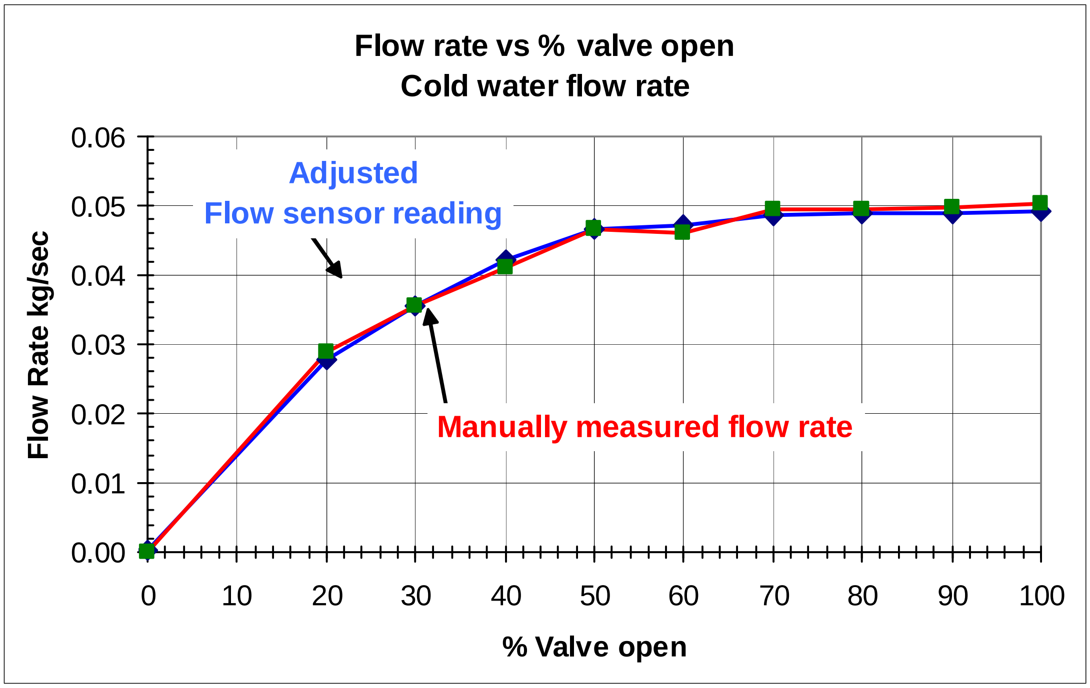


Figure 5.4.4 Adjusted flow rate and manually measured cold water flow rate

In Figure 5.4.4, the sensor reading of valve position less than 40% are adjusted using equation 5.4.2. It is observed that the adjusted sensor readings and the actual flow rate match each other for valve position from 0% to 100%.

5.5 Comparison of experimental and simulation results for the Heat Exchanger:

Several different experiments were run on the temperature system to study the heat transfer mechanism of the heat exchanger. Four different types of co-current experiments are on both the temperature system and mathematical model.

1. Step increase hot water flow; constant cold water flow
2. Step decrease hot water flow; constant cold water flow
3. Step increase cold water flow; constant hot water flow
4. Step decrease cold water flow; constant hot water flow

All the above experiments were performed with hot water in the tubes; the flow pattern was co-current flow and both the solenoid valves on the cold water line were in open position to maximize the cold water flow. The hot water flow is increased or decreased by changing the pump speed. The cold water flow is varied by changing the flow control valve position. The pump speed and the cold water flow control valve position can be changed from the computer station. The temperature experiments can also be run from the internet at http://chem.engr.utc.edu/labs/ . The data from these experiments are saved on the web. The experimental and the simulation data were compared using Excel.

Step increase in hot water flow (HWF); constant cold water flow:

The hot water flow rate was initially 0.03 kg/sec at 50% pump speed. The temperatures at the outlet were allowed to reach steady state before the flow rate was increased. The flow rate was increased to 0.054 kg/sec (80% pump speed). The flow rate values for the hot water side were obtained from the hot water flow rate analysis (section 5.4). The cold water flow rate was measured manually because the flow rate analysis on the cold water side was only for one solenoid valve open and not for both valves open. The average cold and hot water inlet temperatures from the experimental data are used as inputs in the model. The heat transfer coefficient input to the model on both the tube and the shell side was equal to 2162 W/m^2 ^{o}C. The input parameters are given in the physical parameter selection section (4.3)

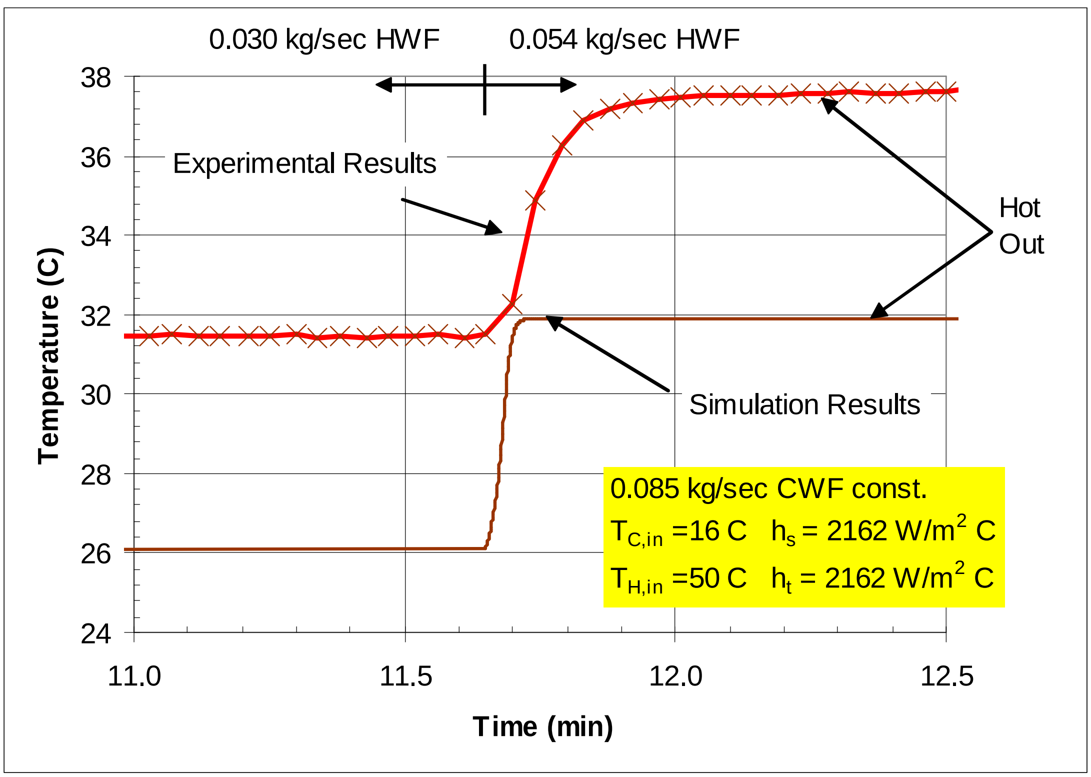
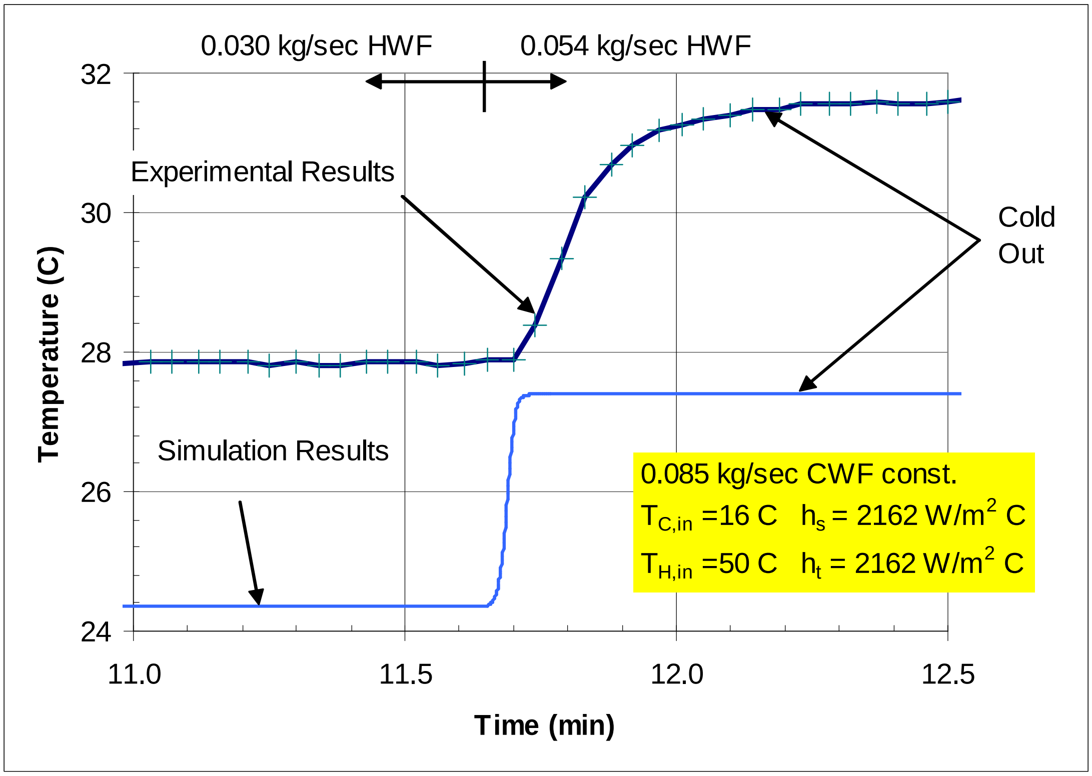


Figure 5.5.1 Step increase in HWF – Hot outlet temperature response

Figure 5.5.1 shows the hot outlet temperature for a step increase in the hot water flow. The simulation hot outlet temperature behaves in the same pattern as the experimental hot outlet temperature response. As the flow rate of the hot water is increased, the outlet temperature for the hot water increases. The model reacts faster to the change in HWF than the real experiment. The model takes 0.06 minute to reach steady state whereas the experiment takes 0.4 minute. The temperature difference between the simulation and the experiment is approximately 5.8 °C on the hot outlet.



<u>Figure 5.5.2 Step increase in HWF – Cold outlet temperature response</u>

Figure 5.5.2 shows the cold outlet temperature response for a step increase in the hot water flow rate. The temperature of the cold outlet increases as the hot water flow rate is increased. The model reaches steady state temperature in approx. 0.06 minute whereas the experiment takes 0.6 minute to reach steady state.

Step decrease in hot water flow (HWF); constant cold water flow:

The hot water flow rate was decreased from 0.054 kg/sec to 0.030 kg/sec. The cold water flow rate was held constant at 0.085 kg/sec.

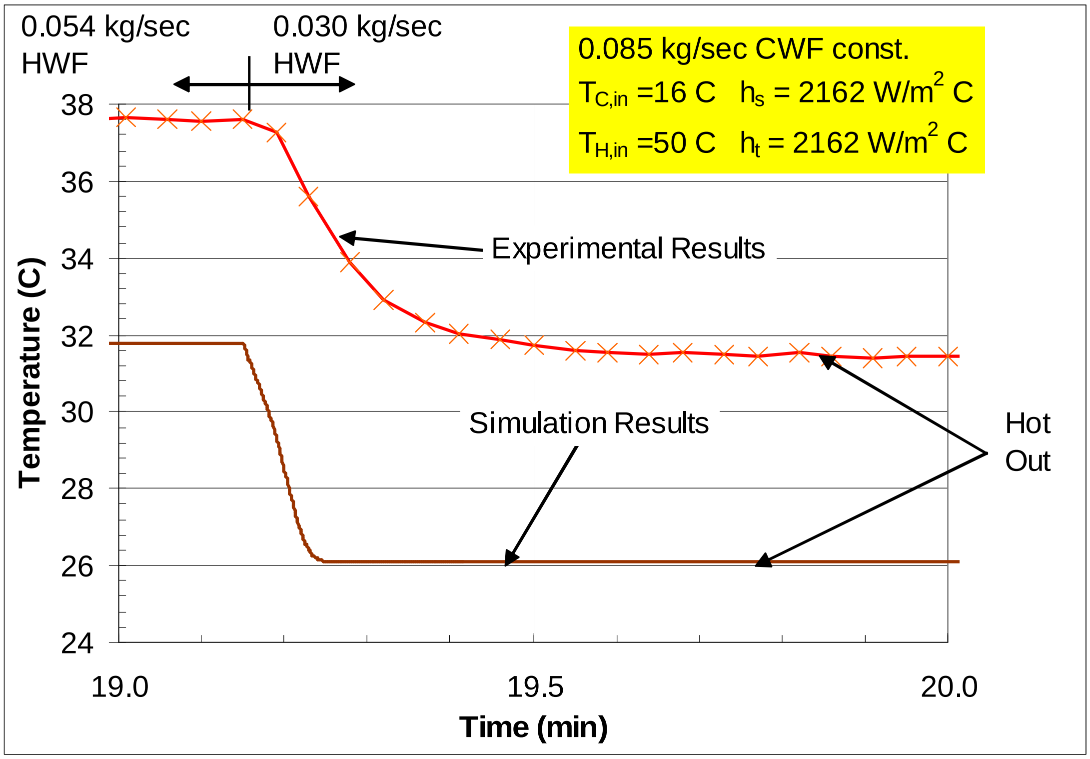


Figure 5.5.3 Step decrease in HWF - Hot outlet temperature response

Figure 5.5.3 shows the experimental and the simulation results for the hot outlet temperature. The hot outlet temperatures for both the simulation and the experiment decreased as the flow rate of the hot water was decreased. The model takes 0.1 minute to reach steady state whereas the experiment takes 0.6 minute to reach steady state temperature. The difference in the steady state temperature is approx. equal to 5.8 ^{o}C.

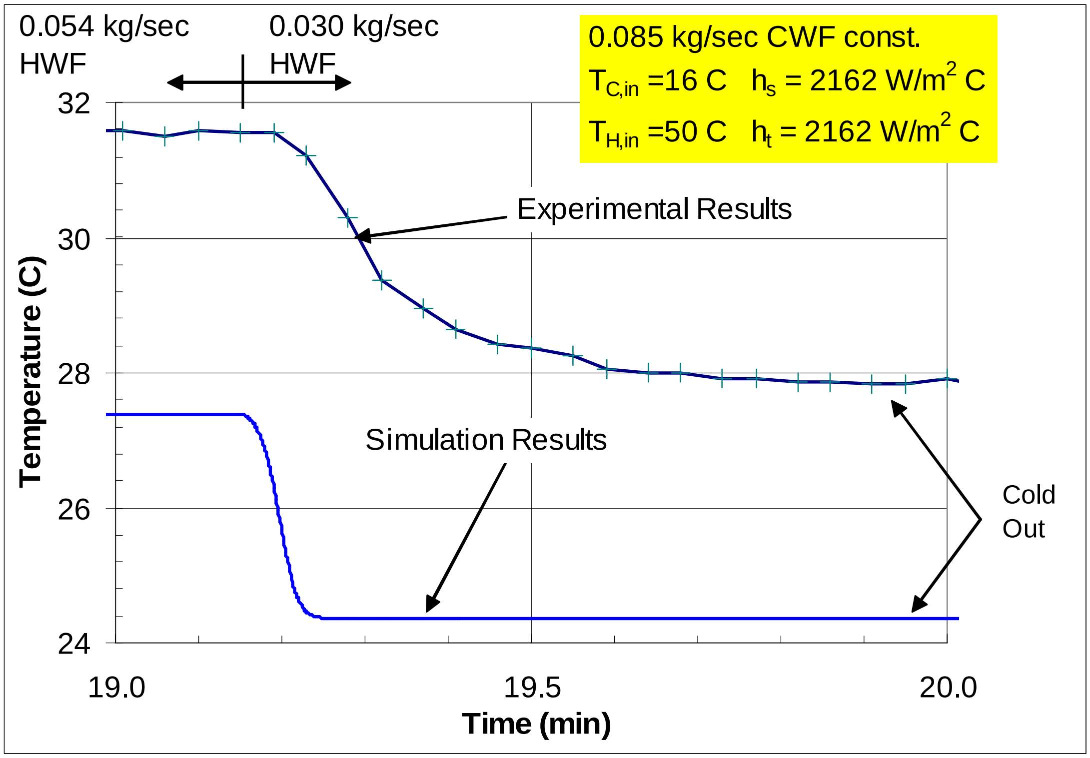


Figure 5.5.4 Step decrease in HWF - Cold outlet temperature response

Figure 5.5.4 shows the temperature results for the experiment and the model for the cold outlet. The cold outlet temperature decreased as the flow rate of the hot water was decreased. The model reached steady state faster than the experiment. The model took 0.1 minute to reach steady state whereas the experiment reached steady state in approx. 0.7 minute. The difference in the steady state temperatures for simulation and the experiment is approximately equal to 4 ^{o}C.

Step increase in cold water flow (CWF); constant hot water flow:

In this experimental run, the hot water flow rate in the tubes was constant at 0.076 kg/sec and the cold water flow rate was changed by changing the flow control valve position from 50 % (0.06 kg/sec) open to 100 % (0.085 kg/sec) open.

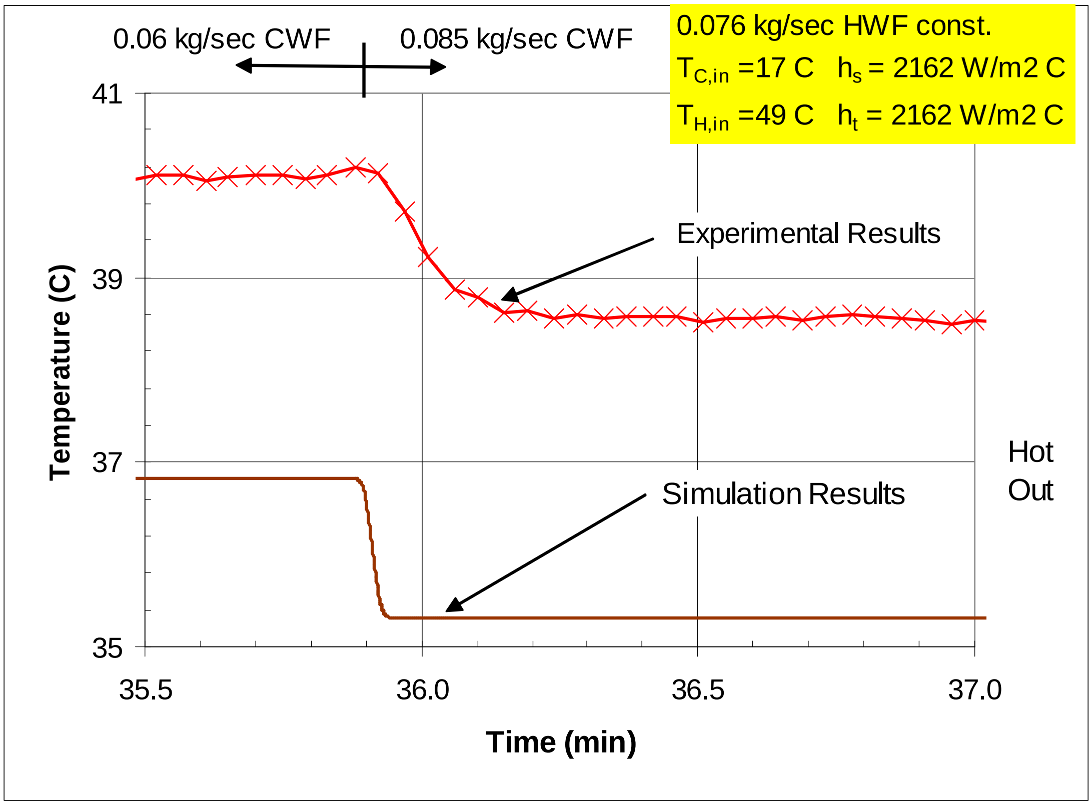


Figure 5.5.5 Step increase in CWF - Hot outlet temperature response

Figure 5.5.5 shows the hot outlet temperature results for the experiment and the simulation. The experimental and the simulation hot outlet temperature decrease as the cold water flow is increased. The model reacts faster than the experiment to a step change in cold water flow rate. The model reaches steady state temperature in approx. 0.1 min whereas the experiment takes 0.4 min to reach steady state. The steady state temperature difference between the simulation and the experimental results is approx.

3°C before the step increase, and increases to 3.5 °C after the step increase in cold water flow rate.

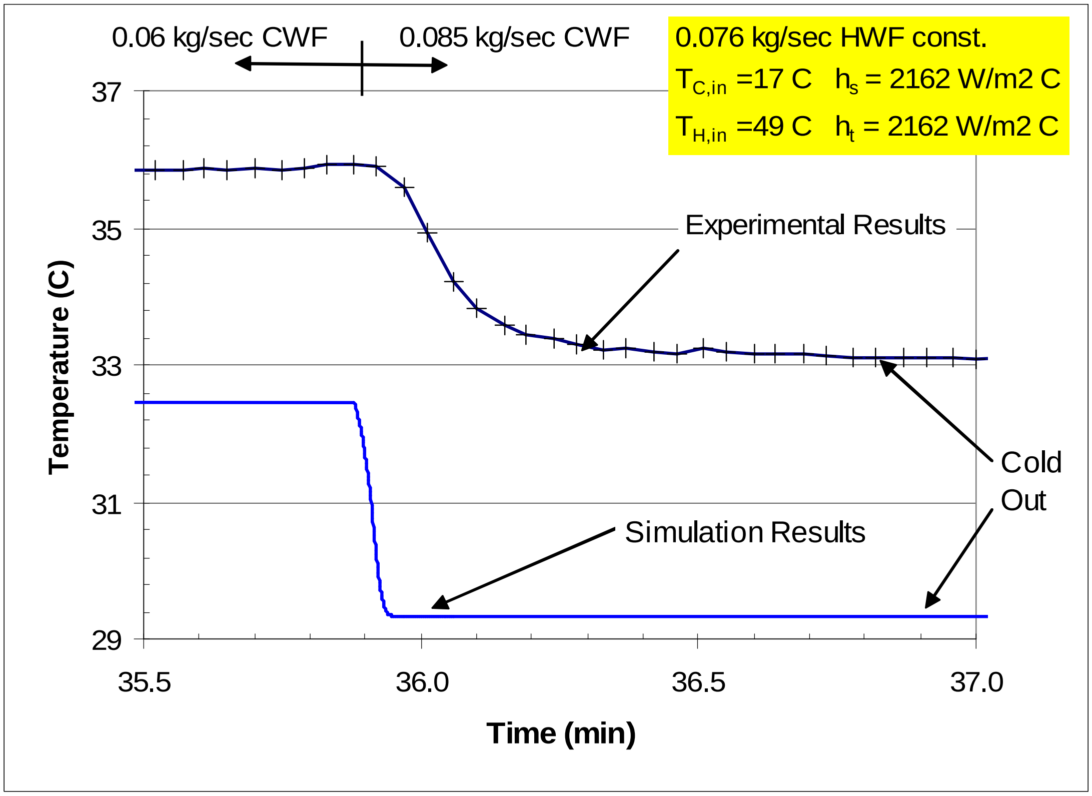


Figure 5.5.6 Step increase in CWF - Cold outlet temperature response

Figure 5.5.6 shows the cold outlet temperature response to the change in the cold water flow rate. The same pattern as the hot outlet temperature results is observed here. The simulation reacts faster than the experiment. It takes approx. 0.1 min to reach steady state temperature and the experiment takes approx. 0.5 min to reach steady state. It is observed that the model reacts much faster than the experiment. The temperature for the cold and the hot out decrease as the cold water flow rate is increased. The steady state temperature difference for the cold outlet changes from approximately 3°C before the step to 4°C after the step.

Step decrease in cold water flow (CWF); constant hot water flow:

The cold water flow was decreased from 0.085 kg/sec to 0.06 kg/sec. The hot water flow rate is held constant at 0.076 kg/sec.

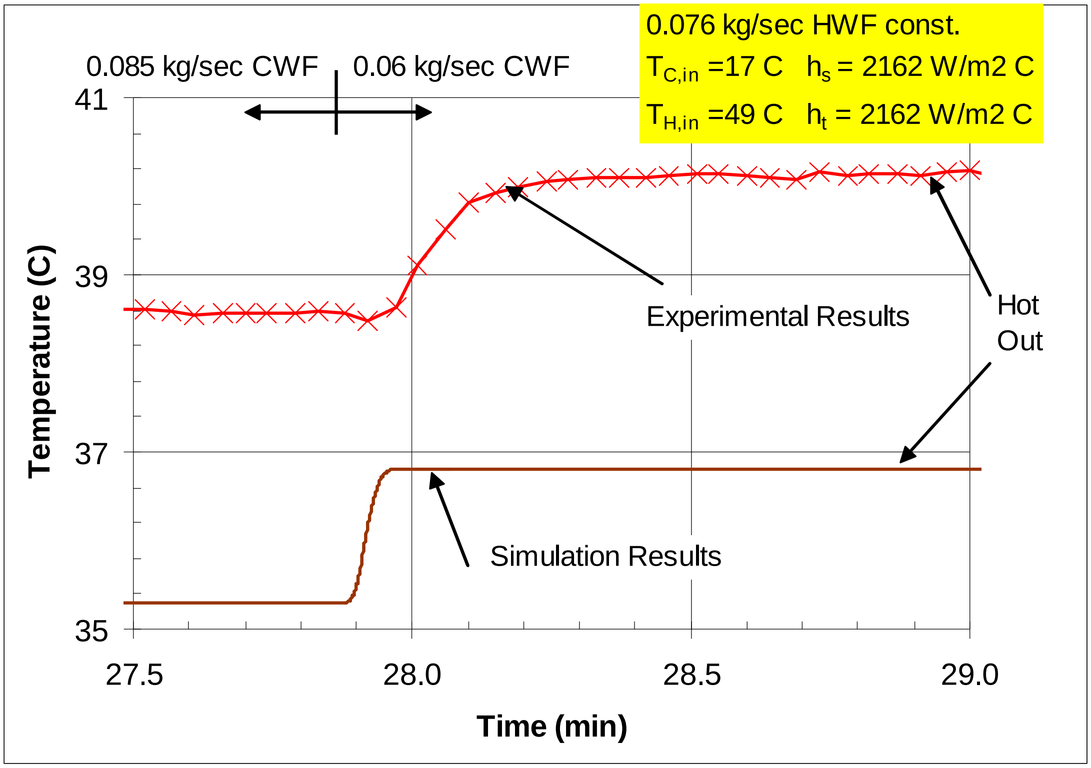


Figure 5.5.7 Step decrease in CWF - Hot outlet temperature response

Figure 5.5.7 shows the hot outlet temperature results obtained from the experiment and the simulation. The hot outlet temperature increases as the flow rate of the cold water is decreases. The steady state temperature difference between the experimental and the simulation results is approx. 3°C. The model is faster than the experiment and reaches steady state in approx. 0.1 minutes. The experiment takes approx. 0.5 minutes to reach steady state.

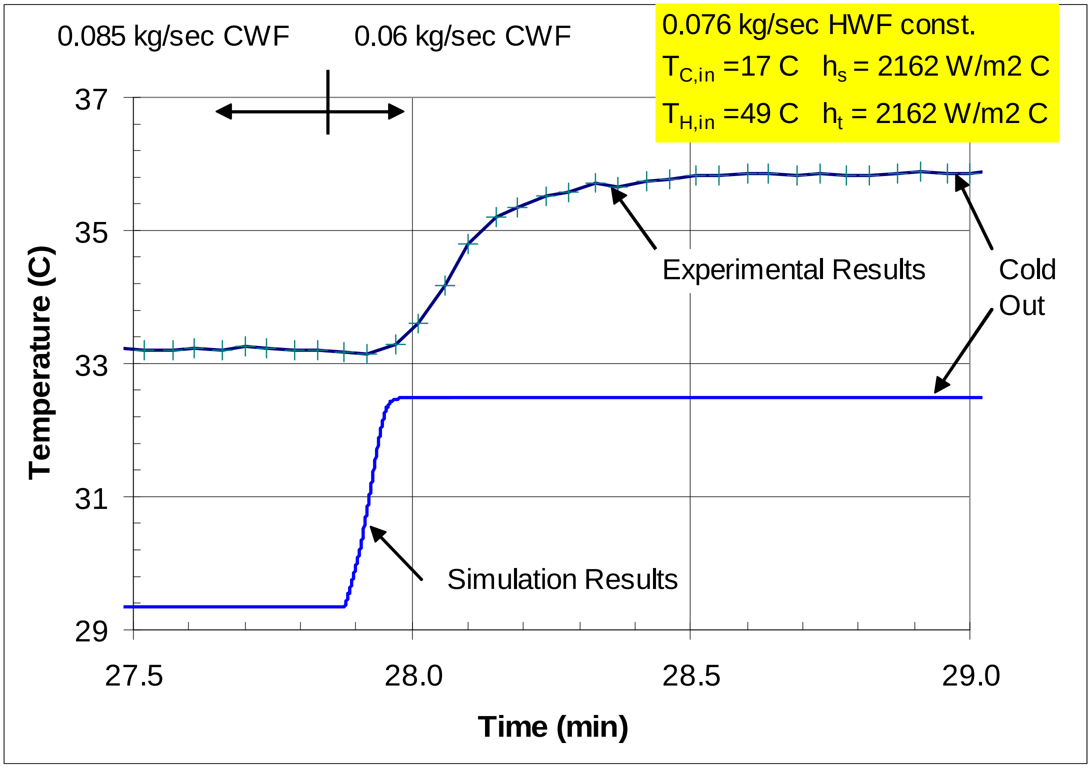


Figure 5.5.8 Step decrease in CWF – Cold outlet temperature response

Figure 5.5.8 shows the cold outlet temperature results for the experiment and the simulation. The cold outlet temperature increases as the flow rate of water is decreased. The cold water removes more heat at a higher flow rate. The model response is faster than the experiment. It reaches steady state in approx. 0.1 minutes whereas the experiment takes 0.7 minutes to reach steady state temperature. The steady state temperature difference is approx. 4^{o}C before the step and decreases to 3^{o}C after the step. As the cold water flow is decreased the temperature of the hot and the cold out temperatures increase. The model reaches steady state faster than the experiment.

5.6 Investigation of the Dynamic Response of the Model:

The dynamic response of the model was faster than the experiment. The values for the input parameters such as the heat transfer coefficient (h_s and h_t) and number of control volumes were changed to study the impact on the dynamic response.

Heat Transfer Coefficient:

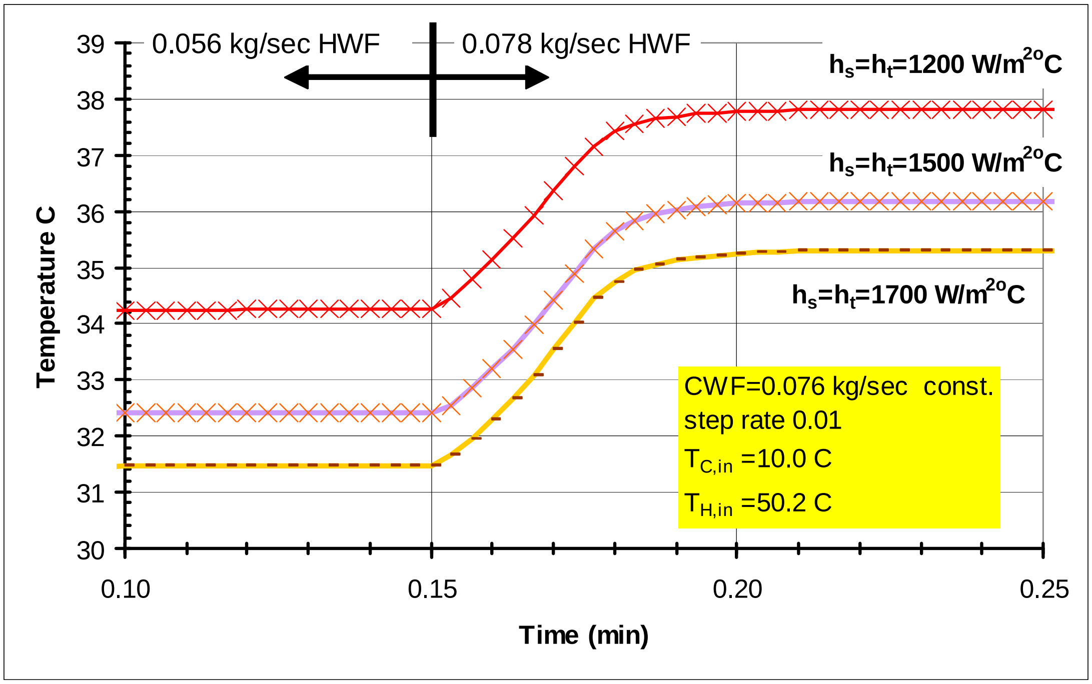


Figure 5.6.1 Dynamic Response – Hot Outlet

Figure 5.6.1 shows a plot for the dynamic response of the model for the hot outlet. The hot water flow rate was changed from 0.056 kg/sec to 0.078 kg/sec. The cold water flow rate was held constant at 0.076 kg/sec. The inlet temperatures were assumed to be constant $T_{C,\,in}$ =10.0 ^{o}C and $T_{H,\,in}$ =50.2^{o}C. From figure 5.6.1 shows the dynamic response of the model for different values of heat transfer coefficient. The change in heat transfer coefficient only has an impact on the outlet temperature. For all the three different values of the heat transfer coefficient, the model takes 0.05 minutes to reach steady state temperature.

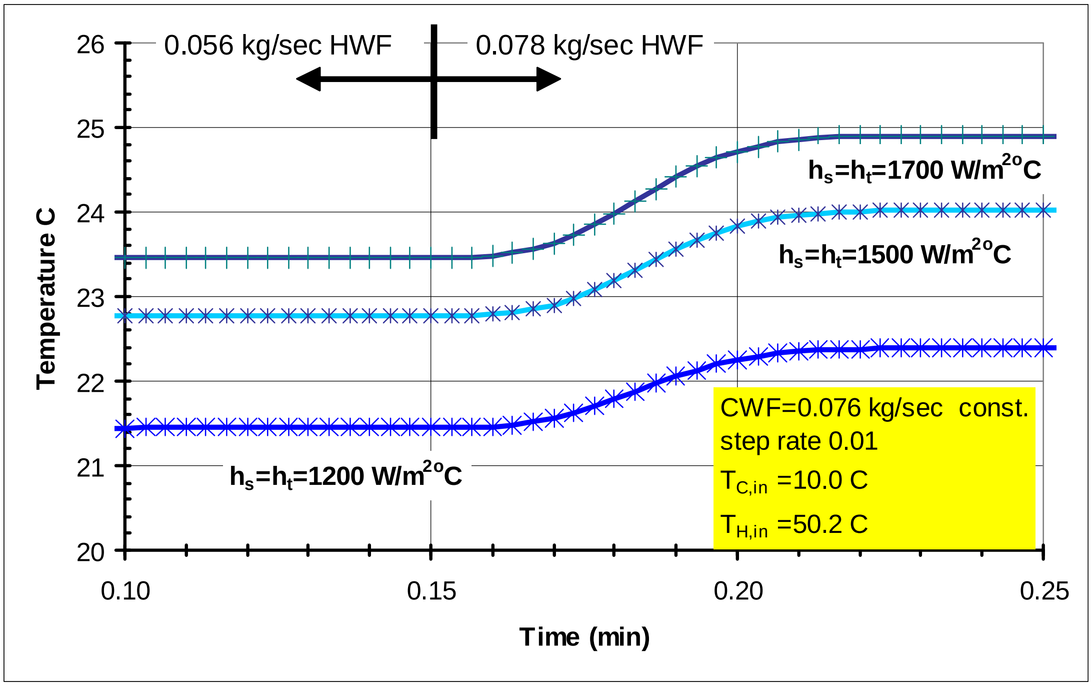


Figure 5.6.2 Dynamic Response – Cold Outlet

Figure 5.6.2 shows the temperature response for the cold outlet for different values of heat transfer coefficient. The model reaches steady state temperature in approx. 0.07 minutes. The dynamic response of the model is similar for the range of heat transfer coefficients used. From figures 5.6.1 and 5.6.2 it can be concluded that the heat transfer coefficient has no effect or minor effect on the dynamic behavior of the model.

Number of Control Volumes:

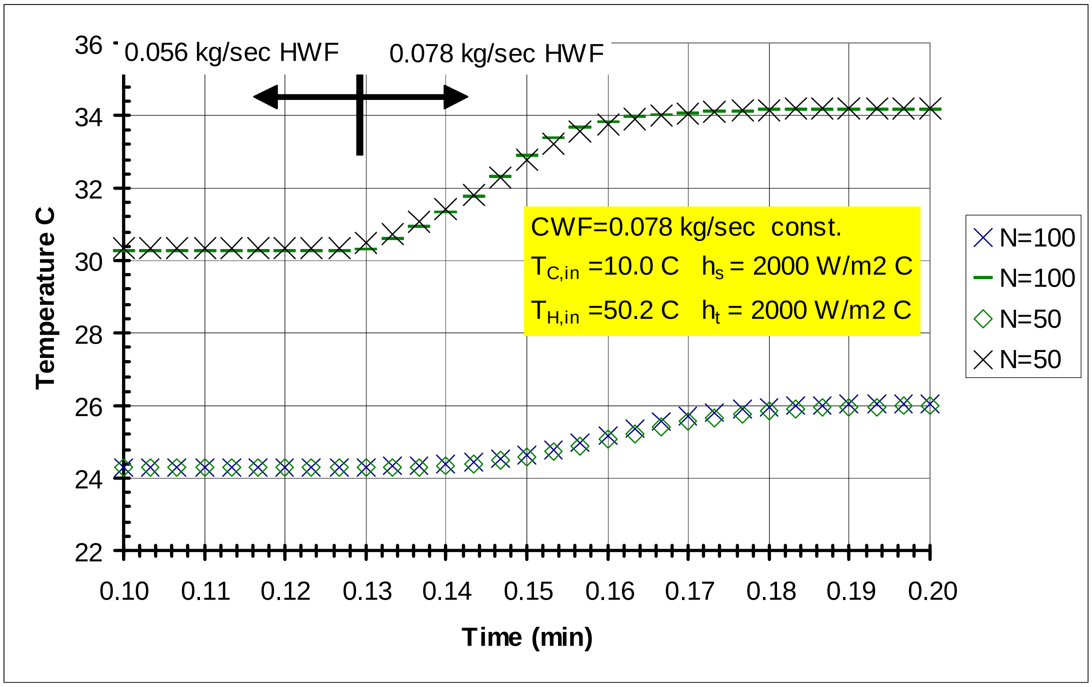


Figure 5.6.3 Dynamic Response – Changing number of Control Volume

Figure 5.6.3 shows the dynamic response of the model for hot and cold outlet temperatures. The hot water flow rate was increased from 0.056 kg/sec to 0.078 kg/sec while the cold water flow remained constant at 0.078 kg/sec. The hot and the cold inlet temperatures were assumed to be constant at $T_{C,\ in}$=10.0 ^{o}C and $T_{H,in}$=50.2^{o}C. The heat transfer coefficients for the shell and the tube h_s and h_t are equal to 2000 W/m^{2}C. The number of control volume N was changed from 50 to 100. The hot outlet temperature response takes 0.04 minutes to reach steady state and the cold outlet temperature takes approx. 0.06 minutes to reach steady state. From figure 5.6.3, it is observed that the change in the control volume does not have a major impact on the dynamics of the model.

6. Conclusions:

Condenser:

The objective of this project was to design the mathematical model to simulate the condenser, which is used to condense methanol vapors on the distillation column. The mathematical model was designed successfully for the condenser. The results from a batch distillation experiment (with 100% reflux and 3500 watts added to the Reboiler) were compared to the model results. The model and the experimental results were compared using the overall heat transfer coefficient (U) for this particular experiment. Both step up and step down experiments results were compared. The model reacted faster to a step change in flow rate than the actual experiment. The model reached steady state in approx. 0.1-0.3 minutes whereas the experiment reached steady state in 3-4 minutes. The difference between the experimental and the model cold outlet temperature was equal to 1-2^{o}C. Therefore the model can be used to estimate the cold water outlet temperature for this particular experiment.

Heat Exchanger Model:

The objective of this project was to develop a dynamic mathematical model which will simulate the results similar as the shell and tube heat exchanger on the temperature system. The milestone of developing the mathematical model was completed successfully using LabVIEW software. The temperature patterns of the results produced by the model are similar to the real heat exchanger. For example, as the flow rate of the water is increased in the step experiment for the cold water, the hot and the cold outlet temperatures decrease.

The model reacts faster than the experiment to reach steady state temperatures. The model reaches steady state in approximately 0.1 minutes whereas the experiment reaches steady state temperature in approximately 0.4-0.7 minutes for the experiments studied. The difference between the experimental and the steady state temperature was in between 3-6 ^{o}C. One of the reasons for this difference is the assumption that the heat transfer coefficients are equal on both the shell and the tube side. From the dynamic investigation it was concluded that the heat transfer coefficient and the number of control volumes (within the range studied) did not have major impact on the dynamic behavior of the model.

7. Recommendations:

Condenser:

- The first objective would be to investigate the dynamic response of the model as compared to the experimental response. Different input parameters should be changed to study the transients. The dynamic response of the model should be made to match the experimental response for any experiment.
- The current model results are only for one particular experiment. More experiments (different reflux % and watts added to the Reboiler) should be run on the distillation column to compare the experimental results with the model results.
- The flow sensor on the cold water supply line was reading incorrect flow rate. Calibration of the flow sensor should be performed to obtain the correct flow rate readings.

Heat Exchanger:

- The main objective would be similar to that of the condenser. Further study should be carried out to investigate the dynamic behavior of the model. Other input parameter such as the volume of the shell, tube and metal should be varied to study the effect on the dynamic response of the model.
- Currently the heat transfer coefficients for the shell and the tube side used in the model are not precise. More experiments and calculations should be

performed on the heat exchanger to find the heat transfer coefficients for the shell and the tube side of the heat exchanger.

- From the flow rate analysis on the temperature system it was discovered that the flow sensors were reading incorrect flow rate. The flow sensors should be calibrated so that the results obtained from running experiments on the heat exchanger are more reliable and correct.
- The mathematical model for the heat exchanger was only designed for co-current flow due to time constraints. In future a mathematical model for the heat exchanger could be designed for a counter current flow pattern.

8. Annotated Bibliography:

- à Incropera, Frank P., and DeWitt. David P. Fundamentals of Heat and Mass Transfer.4th ed. New York: John Wiley and Sons, 1996

  The above reference was used to study the heat transfer theory and to find different physical parameters.

- à Felder, Richard M., and Ronald W. Rousseau. Elementary Principles of Chemical Processes. 3rd ed. New York: John Wiley & Sons, Inc. 2000

  Physical parameter values were found from this reference.

- à Chapra, Steven C., Raymond P. Canale. Numerical Methods for Engineers. 3rd ed. New York: McGraw Hill Inc., 1998

  The above mentioned reference was used for understanding the Runge-Kutta method used for solving differential equations.

- à Cultip, Micheal B., and Mordechai Shacham. Problem Solving in Engineering with Numerical Methods. Upper Saddle River, NJ: Prentice Hall, 1999

  The above reference had many different chemical engineering problem solved using numerical methods and computer software.

- à http://Chemical .caeds.eng.uml.edu/onlinec/white/sdyn/s6/s6mod1.html

  This website link had a similar problem for heat exchanger solved using Math Lab software.

9. Appendices:

Tables:

Table A-1 Volume and Surface Area of the Tubes in the Condenser

distance from the center to the tubes	diameter of the tubes	radius of the tubes	# of circles	length of the tubes	Surface area	Volume
X (cm)	D (cm)	r (cm)	Y	2(PI)XY	2(PI)rL (cm^2)	(PI)*r^2 * L (cm^3)
1.3	0.5	0.25	10	82	128	16
2.5	0.7	0.35	12	188	415	73
3.8	0.9	0.45	11	263	743	167
5	1	0.5	12	377	1184	296
					2470	552
					Surface area (m^2)	Volume (m^3)
					0.25	5.52E-04

Table A-2 Overall Heat Transfer Coefficient 'U' for the condenser

CWS (C)	CWR (C)	diff T	CWF (L/min)	Kg/min	Watts absorbed by condenser	delta T LM	U (W/m^2 C)
18.5	51.4	32.83	0.44	1.24	2840	32.3	352
17.1	30.5	13.4	1.16	3.50	3270	45.9	285
16.0	25.0	8.91	1.64	5.01	3110	49.4	252
16.2	34.2	17.94	0.86	2.56	3200	44.2	290
16.7	44.7	28.04	0.55	1.59	3100	37.6	330

Table A-3 Overall Heat Transfer Coefficients for different co-current experiments

Co current flow, hot water=tubes, 100% HWF					
CWF	Watts				
%	q_h	q_c	q average	delta T_{LM}	U
50	-3506	5838	4672	16	1070
70	-3381	5898	4639	16	1085
85	-3318	5789	4553	15	1089
			Average U --->		1081

Dynamic Mathematical Model of a Heat Exchanger

By
Alok M. Patel

Departmental Honors Thesis
The University of Tennessee at Chattanooga
Chemical Engineering

Project Director:
Dr. Jim Henry

Examination Date: March 28th, 2003

Project Liaison:
Dr. David Levine

Committee Members:
Dr. Jim Cunningham
Dr. Frank Jones

Examining Committee Signatures:

Jim Henry
Frank Jones
James R. Cunningham

Chairperson, University Departmental Honors Committee