UNSTEADY STATE DYNAMİC MODELİNG OF A COUNTER-FLOW HEAT EXCHANGER

Final Report of Semester Project
03.02.2021

Mustafa ŞENTÜRK, MTME651 2020 Fall Semester

ABSTRACT

The aim of this study is to build an unsteady state dynamic model of an heat exchanger that is suitable to be improved for using on different studies and to be modified according to needs in the future.

Heat exchangers have vital role in energy systems. Simulations of heat exchangers are important for the calculation of energy efficiency of a system. In this study a simple but effective dynamic model of a counter-flow heat exchanger is studied. Most of the simulations are based on steady state analysis, however while working with a system that is supposed to maintain a nearly constant temperature level for cooling systems, a controller should be used and thus the system should be modeled as a time depended unsteady state dynamic model. For this purpose, energy conservation equations are derived based on discrete time intervals and segmentation of the heat transfer volume.

HEAT EXCHANGER

A counter flow heat exchanger is used in this study. The hot fluid flows through tubes which are surrounded with cold fluid. The walls of the tubes are considered as a uniform solid medium that conducts the heat and the section areas of the heat exchanger are considered as constants through the heat exchanger.

Diameter of side	: 0.5	meter
Diameter of a tube	: 0.0254	meter
number of tubes	: 85	
spacing	: 0.049	meter
Length	: 1.20	meter

Number of tubes are selected from charts[1] using ratio of D/spacing = 10.166 and the arrangement type is number 1 (sea figure 1).

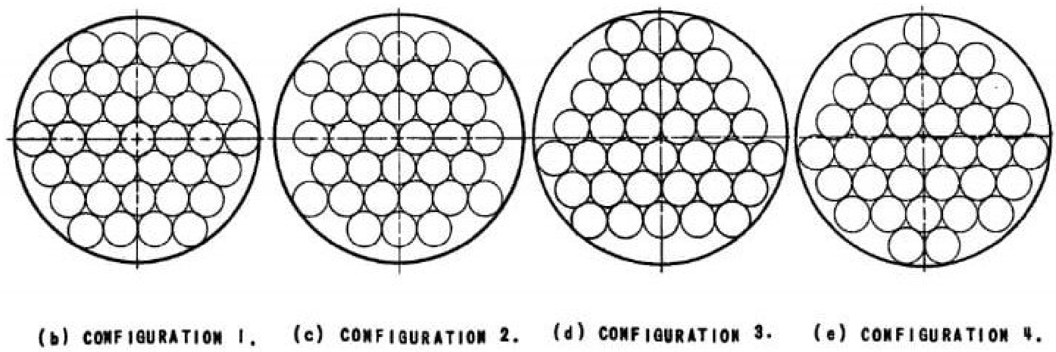

figure 1. Different arrangements for spacing of tubes inside a circular section based on spacing between centers of the tubes.

1 Fraas, A. P., Laverne M. E.; "*Heat Exchanger Design Charts*", 1952, p.16.

METHODOLOGY

The length of the Heat Exchanger is divided into N segments. Equations of energy conservation are solved in every time interval to calculate next state of the heat exchanger.

The flow of the fluids are considered as a flow of enthalpy. As the time interval is very small comparing to the flow rate of the fluids, this approximation is suitable to be used.

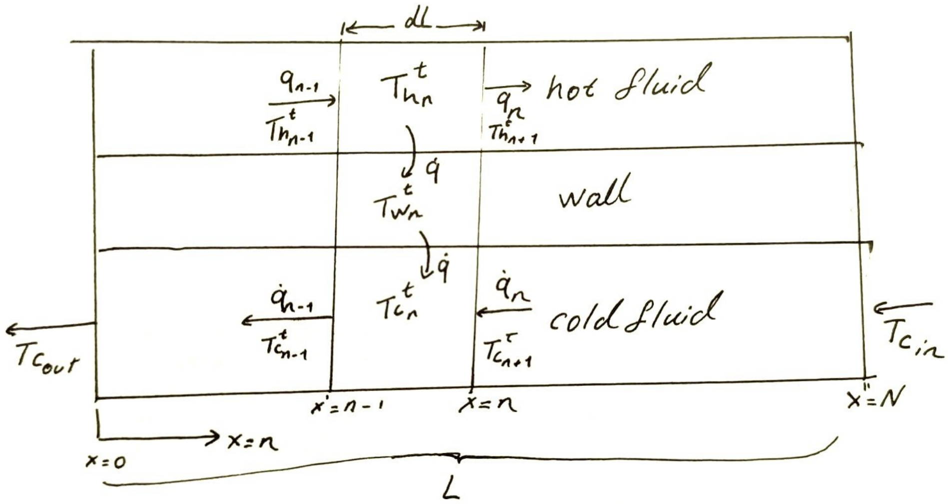

Figure 2. Enthalpy differentials for a single segment

L	: is the length of the heat exchanger through the heat transfer area
dL	: is the length of a single segments
N	: total segment number
n	: the number of the considered segments
t	: current time step
q	: heat flow for a single time interval

Assumptions

The assumptions and approximations that are made for the simplicity of the model are as follows;

- the heat exchanger is divided into segments,
- the temperature of the fluids are assumed as constant and uniform inside a single segment and through a time interval,
- conductional heat transfer inside the wall is omitted,
- heat transfers at the outlet ind inlets of the heat exchanger are omitted,
- heat loss to open air is omitted.
- there is no pressure change on the fluids,
- there is no work done on the fluids due to fractional resistance etc.

These assumptions can be improved. For example the heat loss to open air can be added easily and work done by frictional resistance may be added later.

Energy Conservation Equations

There are three mediums that has enthalpies varying due to heat transfer; the hot fluid, the wall and the cold fluid. As the energy conservation equations depends on the enthalpy, the flow of the fluids is modeled with enthalpy transfer between segments. The temperature change of a single segment in a single time interval will be equal to the enthalpy difference between the time intervals; on the other hand, the difference of the inflow and outflow energy of the fluid for the time interval. As the other type of energy flows are omitted, the heat energy is the only source of enthalpy change during time intervals.

For a single segment **energy conservation equations** are as follows:

for the wall:

$Tw_n^{t+1} = Tw_n^t + [HTC_h*A_t*(Th_n^t – Tw_n^t) + HTC_c*A_t*(Tc_n^t – Tw_n^t)] * dt * (1 / (m_w * C_w))$

where;

Tw	: temperature of the wall at the n^{th} segments
m_w	: mass of the wall for a single segment
C_w	: heat capacity of the wall at the n^{th} segments
HTC_h	: heat transfer coefficient between the hot fluid and the wall at the n^{th} segments
HTC_c	: heat transfer coefficient between the cold fluid and the wall at the n^{th} segments
Th_n^t	: temperature of the hot fluid at the n^{th} segments at the time step t
Th_c^t	: temperature of the cold fluid at the n^{th} segments at the time step t
A_t	: Area of heat transfer at the n^{th} segments
t	: time step
dt	: time interval

Here the equation, $HTC_h*A_t*(Th_n^t – Tw_n^t)$ is the heat transfer per unit time between the hot fluid and the wall while $HTC_c*A_t*(Tc_n^t – Tw_n^t)$ is the heat transfer per unit time with the cold fluid. Heat transfer coefficients HTC_h and HTC_c are dependent on the mass flow rate of the hot and cold fluids in respect.

$HTC_h = HTC_{ref} * (\dot{m}_{h/} \dot{m}_{ref})$^0.6
$HTC_c = HTC_{ref} * (\dot{m}_{c/} \dot{m}_{ref})$^0.6

Here, the HTC_{ref} is the reference value of the heat transfer coefficient at the reference mass flow rate $\dot{m}_{ref}$. This approximation is preferred due to simplification of the process. But the tables of heat transfer coefficients and interpolation methods also can be used instead of this. HTC_{ref} is supposed to be determined experimentally.

For the hot fluid:

$Th_n^{t+1} = Th_n^t + [\dot{m}_h*Cp_h(Th_{n-1}^t)* Th_{n-1}^t - \dot{m}_h*Cp_h(Th_n^t)* Th_n^t + HTC_h*A_t*(Tw_n^t – Th_n^t)] * dt * [1/(m_h*Cp_h(Th_n^t))]$

where;

Th_n^{t+1}	: temperature of the hot fluid at the n^{th} segments at the time step t
Th_{n-1}^t	: temperature of the hot fluid at the $n-1^{th}$ segments at the time step t
$Cp_h(Th_n^t)$	: heat capacity of the hot fluid at the temperature Th_n^t
$\dot{m}_h$	: rate of the mass flow of the hot fluid
m_h	: mass of the hot fluid inside a single segment; $\rho_h * A_t * dL$

$\dot{m}_h$*$Cp_h(Th_{n-1}{}^t)$* $Th_{n-1}{}^t$: transfer of enthalpy from the n-1th segment to the nth segment.
$\dot{m}_h$*$Cp_h(Th_n{}^t)$* $Th_n{}^t$: transfer of enthalpy from the nth segment to the n+1th segment.

for the cold fluid:

$$Tc_n^{t+1} = Tc_n^t + [\dot{m}_c*Cp_c(Tc_{n+1}^t)* \ Tc_{n+1}^t - \dot{m}_c*Cp_c(Tc_n^t)* \ Tc_n^t + HTC_c*A_t*(Tw_n^t - Tc_n^t)] * dt * [1/(m_c*Cp_c(Tc_n^t))]$$

where;

Tc_n^{t+1}	: temperature of the cold fluid at the nth segments at the time step t
Tc_{n+1}^t	: temperature of the cold fluid at the n+1th segments at the time step t
$Cp_c(Tc_n^t)$	: heat capacity of the cold fluid at the temperature Tc_n^t
$\dot{m}_c$	: rate of the mass flow of the cold fluid
m_c	: mass of the cold fluid inside a single segment; $\rho_h * A_t * dL$

Simulation Model and Diagram

A scenario is chosen where the temperature of a fresh water in a cooling system of a marine engine should be maintained at a reference level. So an heat exchanger is used to cool the fresh water with sea water as a coolant. To maintain the reference level of the temperature, a PID controller is used in the circuit that controls the mass flow rate of the sea water flows into the heat exchanger.

SIMULINK is used for the simulation environment. Sampling points are set in order to follow the state of the system (see figure 3). During the simulation the temperature and the mass flow rate of the fresh water comes from the engine is considered as constant. This fluid may be modeled for a variable heat load of an engine in future. For now, this simplifications are used to investigate the general behavior of the system.

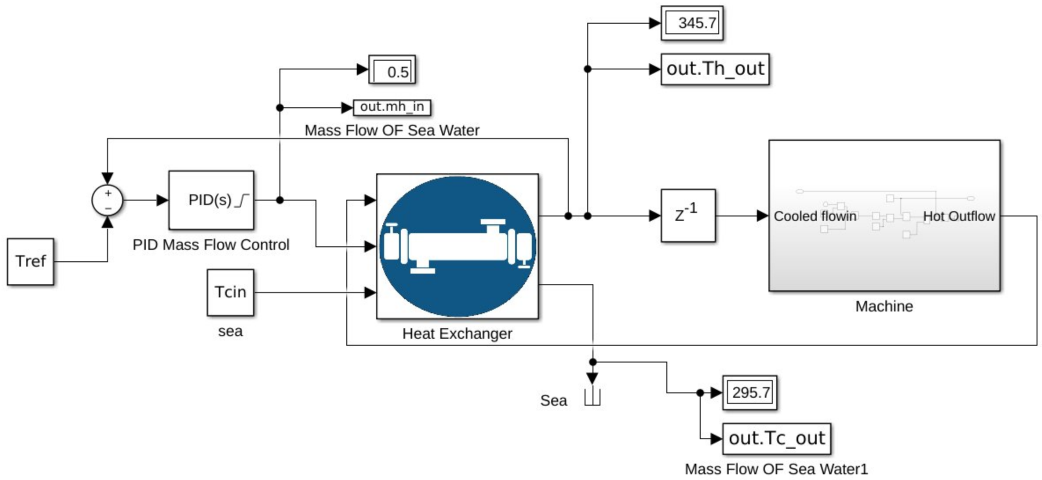

Figure 3. Simulink diagram of the circuit

Simulation Results

Initial values of the system are set before running the simulation using "*parat_heat_excahnger.m*" file (see Appendix 1). Time interval is set to 10 msec, the heat exchanger divided into 100 segments. Outflow temperature of the engine is set to 363 ^{0}K (90 $^=$C) and the reference temperature is set to 348 ^{0}K. The sea temperature is 289 ^{0}K while the temperature is 295^{0}K and the temperature inside the heat exchanger is set to the temperature of environment as a uniform temperature. Despite the equations of the energy conservation are derived with the properties of fluids which are depended on temperatures, during the simulation they have taken from charts[2] as if their temperatures are constant and the value of HTC_{ref} is set to a value preferred between 150 and 1200 based on the charts[3]. The properties of fluids can be modeled as variables using interpolation on multi dimensional charts. However, for the simplicity they considered as constants. This will not change the behavior of the simulation in general and the simulation can be easily modified to implement this flexibility.

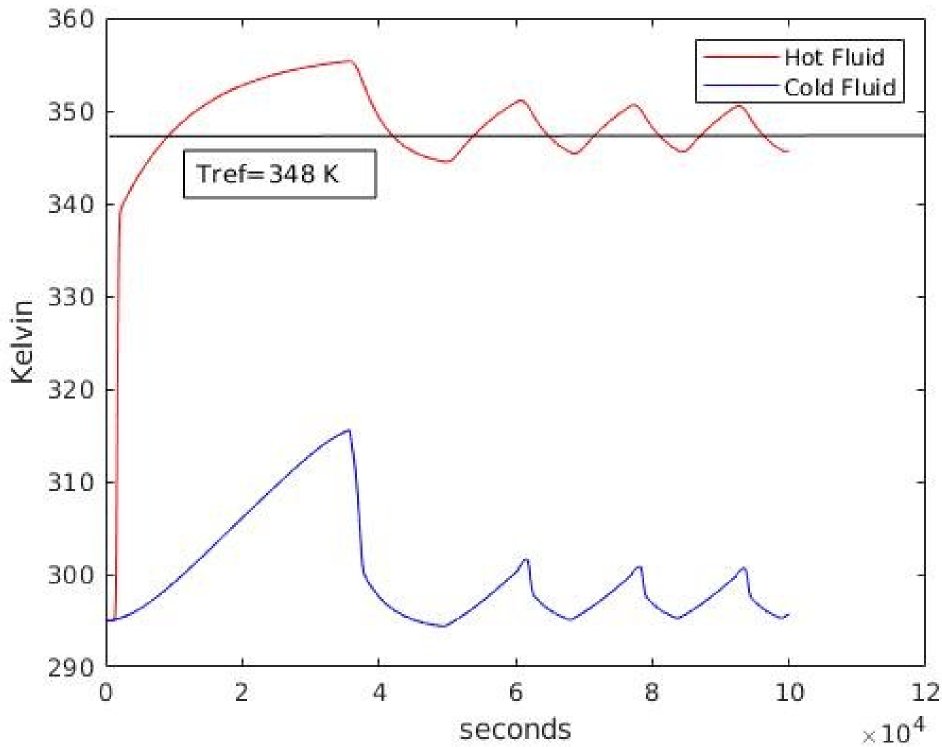

figure 4. Temperature graphs of the fluids through the time interval of the simulation

The simulation is run for 10 seconds (1000 loops of 10 msec time intervals), and the temperature is kept around 348^{0}K by the controller (see figure 4).

This simulation shows an ideal working of an heat exchanger as there is no loss due to the fractional resistance, pressure drops, heat loss to the air. Heat transfer coefficients are set with tentative values. The controller is assumed to control the flow rate of the sea water directly and with a 100% efficiency. However the model works and suitable for future studies and improvements.

2 https://www.engineeringtoolbox.com/water-thermal-properties-d_162.html
3 Jurandir Primo, PE "*Shell and Tube Heat Exchangers, Basic Calculation*", PDHonline Course M371 @https://www.pdhonline.com/courses/m371/m371content.pdf, p.13.

Appendix 1. Initializing the System
(parat_heat_exchanger.m file)

```
%Initialising Input Parameters
%-----------------------------
dt = 0.01;
 N = 100; %number of segments
nt = 85; %number of tubes (for D/spaceing = 10.166 arrangement 1 from the book

%in meter
L = 1.20; %length of the heat exchanger
dL = L / N;%length of a single segment
D = 0.5; %inner Diameter of shell
D_t = 0.0254; %outer diameter of single tube
tick_t = 0.001; %thickness of tube

%im m2
Ah = nt .* pi .* (D_t/2 - tick_t)^2; %section area of hot fluid
Ac = pi .* (D/2)^2 - nt .* pi .* (D_t/2)^2; %section area of cold fluid
Aw = Ac - Ah; %section area of the wall
At = nt .* 2 .* pi .* D_t .* dL; %heat transfer area for single segment

%in Kelvin
Tenv = 295; %temperature of environment
Thin = 363; %entering Temperature of the hot fluid
Tcin = 289; %temperature of the cold fluid reserve, sea water
Tref=  348; %temperature wished to maintaind at the exit of the tubes

%set the initial temperature of fluids
% +1 is for the inflow sections
for i=1:1:N+1
    Th_init(i) = Tenv;
    Tc_init(i) = Tenv;
end

for i=1:1:N
Tw_init(i) = Tenv;
end

%in kg / m3
density_h = 1000; %mass density of fresh water
density_c = 1025; %mass density of sea water
density_w = 8000; %mass density of stainless steel

%in kg
mass_h = density_h * Ah * dL; %mass of hot fluid in a single segment
mass_c = density_c * Ac * dL; %mass of hot fluid in a single segment
mass_w = density_w * Aw * dL; %mass of the wall in a single segment

%in j/kg.K  https://www.engineeringtoolbox.com/
Cp_h = 4187; %heat capacity of the hot fluid fresh water
Cp_c = 4005; %heat capacity of the cold fluid sea water
C_w = 490; %%heat capacity of the wall carbon steel

%in w/m2.K
HTC_ref = 350; %reference Heat transfer coefficient

%in kg/s
m_ref = 3.5; %reference flow rate of mass
mh_start = 2.5; %staarting mass flow rate of hot fluid
%-----------------------------
```

Appendix 2. Heat Exchanger Matlab Function

```
function [Th, Tc, Tw,Th_out, Tc_out] = fcn(dt,N,Th_prev, Tc_prev,Tw_prev,
Thin,Tcin,mc,mh,At,m_ref,mass_c,mass_h,mass_w,HTC_ref,Cp_h,Cp_c, C_w)

%Counter flow heat exchanger with tubes inside circular section

%this function is suitable for a heat exchanger system that the cold fluid
%flows through shell and the hot fluid flows through tubes.
%assumptions are made for this simulation:
% - temperatures at the entering points are assumed to be fixed
% - because of the counter flow, the temperature spread is assumed to be
% linear during the length of the tubes
% author Mustafa ŞENTÜRK

HTCh = HTC_ref * (mh / m_ref)^0.6;
HTCc = HTC_ref * (mc / m_ref)^0.6;

Tw = zeros(size(Tw_prev));
Th = zeros(size(Th_prev));
Tc = zeros(size(Tc_prev));

Th(1) = Thin;
Tc(N+1) = Tcin;

% n is the index of the segment
    for n=1:1:N
        i = n + 1; % indexes of Th as Th(1) is in fact Th(0)

        Tw(n) = Tw_prev(n) + ( HTCh.*At.*(Th_prev(n) - Tw_prev(n)) +
        HTCc.*At.*(Tc_prev(n) - Tw_prev(n))) .* dt / (mass_w .* C_w);

        Th(i) = Th_prev(i) + (mh .* Cp_h .* (Th_prev(i-1) - Th_prev(i)) +
        HTCh.*At.*(Tw_prev(n) - Th_prev(i))) .* dt / (mass_h .* Cp_h);

        Tc(n) = Tc_prev(n) + (mc .* Cp_c .* (Tc_prev(n+1) - Tc_prev(n)) +
        HTCc.*At.*(Tw_prev(n) - Tc_prev(n))) .* dt / (mass_c .* Cp_c);

    end

Th_out = Th(N+1);
Tc_out = Tc(1);

end
```

Annex

Experience Through the Development of the Model

For the first time I was working on the project, I was trying to solve the system of heat transfer on an heat exchanger. I got impressed and confused with the complexity of the heat system. While searching the literature about the solution for this kind of problem all I had found was efficiency analysis of heat exchangers for steady states. As I was looking for a dynamic modelling for an heat exchanger I was not able to use any of them, however I learned much.

My first attempt was to simplify the heat transfer system by linearising the temperature distribution of the system with small elements and then calculating the heat transfer for every step of the system to reach an optimum point of working state. I built the model on SIMULINK and then begun to analyze it. After a week I realized that the model is not suitable and consistent as it does not depended on a time interval. Then I begun to investigate the issue. I found that the method I had used to build my model was a finite differential method that is not suitable for a dynamic unsteady state system of heat transfer. My biggest problem was that I couldn’t know how should I have modeled the flow of the fluids according to time and heat transfer.

The last model I have built was based on a simple idea of modeling the fluids as a flow of enthalpy instead of a massive flow. Then I could model the equations of energy conversation on the imaginary segments of the heat transfer area. Enthalpy of a previous segment was flowing through the next one by the flow of the fluids, when I used a quite small time intervals for the system calculations, the flow of the enthalpy was exactly like an enthalpy transfer form one segment to another.

After all of these studies I feel that it was an amazing experience that I wouldn’t expect from myself. I learned finite differential and finite element methods, heat transfer coefficients and efficiency of heat exchangers.