HEAT TRANSFER AND PRESSURE DROP ANALYSIS IN SHELL AND TUBE HEAT EXCHANGERS: SEGMENTAL BAFFLES DESIGN

By

Eng. Ahmed Adel Hamza Mostafa

**A Thesis Submitted to the Faculty of
Engineering at Cairo University in Partial
Fulfilment of the Requirements for the Degree of
DOCTOR OF PHILOSOPHY**

In

MECHANICAL POWER ENGINEERING

**FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2017**

HEAT TRANSFER AND PRESSURE DROP ANALYSIS IN SHELL AND TUBE HEAT EXCHANGERS: SEGMENTAL BAFFLES DESIGN

By

Eng. Ahmed Adel Hamza Mostafa

A Thesis Submitted to the Faculty of Engineering at Cairo University in Partial Fulfilment of the Requirements for the Degree of DOCTOR OF PHILOSOPHY

In

MECHANICAL POWER ENGINEERING

Under Supervision of

Prof. Dr. Essam E. Khalil

Professor of Mechanical Power Engineering – Cairo University

Dr. Gamal El Hariry

Assistant Professor of Mechanical Power Engineering – Cairo University

Dr. Waleed Abdel-Samea

Assistant Professor of Mechanical Power Engineering – Cairo University

Dr. Emad M. S. El-Said

Assistant Professor of Mechanical Engineering – Fayoum University

FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2017

HEAT TRANSFER AND PRESSURE DROP ANALYSIS IN SHELL AND TUBE HEAT EXCHANGERS: SEGMENTAL BAFFLES DESIGN

By

Eng. Ahmed Adel Hamza Mostafa

A Thesis Submitted to the Faculty of Engineering at Cairo University in Partial Fulfilment of the Requirements for the Degree of DOCTOR OF PHILOSOPHY

In

MECHANICAL POWER ENGINEERING

Approved by the Examining Committee

Prof. Dr. Essam E. Khalil — **Thesis Advisors and Member**

Prof. Dr. Abdel-Wahed Fouad El-Dib — **Internal Examiner**

Prof. Dr. Osama Ezzat Abd El-Latef — **External Examiner**

Professor of Mechanical Power Engineering - Benha University

FACULTY OF ENGINEERING, CAIRO UNIVERSITY
GIZA, EGYPT
2017

Engineer's Name: Ahmed Adel Hamza Mostafa
Date of Birth: 21/12/1987
Nationality: Egyptian
E-mail: ahmedadel_1987@hotmail.com
Phone: 01005100805
Address: 707 Zahraa Nasr City- El Hay El Asher

Registration Date: 1/10/2013
Awarding Date: / /
Degree: Doctor of Philosophy
Department: Mechanical Power Department

Supervisors: Prof. Dr. Essam E. Khalil
Dr. Gamal Abdel Monem El-Hariry
Dr. Waleed Abdel Samea Marouf
Dr. Emad Mohamed Saad El Said (Assistant Professor of Mechanical Engineering – Fayoum University)

Examiners: Prof. Dr. Essam E. Khalil
Prof. Dr. Abdel-Wahed Fouad El-Dib
Prof. Dr. Osama Ezzat Abd El-Latef (Professor of Mechanical Power Engineering, Faculty of Engineering in Shoubra, Benha University)

Title of Thesis: HEAT TRANSFER AND PRESSURE DROP ANALYSIS IN SHELL AND TUBE HEAT EXCHANGERS: SEGMENTAL BAFFLES DESIGN

Key Words: CFD; heat exchangers; shell and tube; baffle spacing; turbulence model

Summary:

The shell side design of a shell-and-tube heat exchanger; in particular the baffle spacing, baffle cut and baffle orientation dependencies of the heat transfer coefficient and the pressure drop are investigated by numerically modeling a heat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD package. A set of CFD simulations is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The best turbulence model among the ones considered is determined by comparing the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Literature method results.

ACKNOWLEDGEMENT

I hereby would like to express my deep gratitude and thanks to **Prof. Dr. Essam E. Khalil** and **Dr. Gamal El-Hariry** for their support, continuous encouragement and distinctive supervision throughout the course of this work. They helped providing me with up to date technical references that were of great help in the present work. Also I would like to thank to **Dr. Waleed Abdel-Samea** and **Dr. Emad M. S. El-Said** for their supports. In addition, **Dr.Waleed** contacted CFX personnel in order to negotiate for a free academic version of ANSYS 16 CFD package. Furthermore, he encouraged me to produce several technical papers as a co-author with him and provided the required support for these papers to be accepted.

Also, I cannot express; in words; my thanks and gratitude to **my family** for their great and continuous help and support they provided me to finish this work in a suitable form.

Nevertheless, I cannot forget the support of **my colleagues** in the Mechanical Power Engineering department as well as for my Professors for their encouragement and concern throughout the scope of the work.

Finally, I should express my gratitude for the CFX® personnel for their kind interest in the current project and their support via supplying us with a free license of the ANSYS 16 package.

SUBJECT		**PAGE**

LIST OF TABLES

LIST OF FIGURES

Fig. 4.45 Contours of Temperature on Shell Side When B_C=25% of N_b=12 73

Nomenclature

Symbol	Quantity
A	Heat transfer area, m^2
Bc	Baffle cut off ratio (%)
C	Dimensionless constant of proportionality
C_p	Constant Pressure Specific Heat, J/kg K
D	Diameter of tube, m
D	Diameter of shell, m
D_{eq}	Equivalent Diameter, m
Eu	Euler Number
F	Force per unit volume, N/m^3
g	Gravitational acceleration, m/s^2
H	Enthalpy, kJ/kg
h_s	Heat transfer coefficient of shell side , W/m^2K
I	Air exchange
K	turbulent kinetic energy
k	Thermal conductivity, W/m K
L	Length of baffle spacing, m
m	Mass, kg
$\dot{m}$	Mass flow rate, kg/s
Nu	Nusselt Number
N	Number of tubes
P	Pressure, pa
Pr	Prandtl Number
Q	Heat transfer rate, W
$\dot{Q}$	Volume flow rate, m^3/hr
Re	Reynolds Number
S	Area of flow cross-section, m^2
t	Thickness of tube, m
T	Temperature, K
U	Characteristic velocity of the mean flow, m/s
U	Instantaneous velocity component in x direction, m/s
V	Instantaneous velocity component in y direction, m/s
W	Instantaneous velocity component in z direction, m/s
x, y, z	Cardinal coordinate components

Greek letters

β	Baffle angle, degree
ΔP	Pressure drop, Pa
ρ	Fluid density, kg/m^3
μ	Dynamic viscosity, $N.s/m^2$
κ	Kinetic Energy, W
ε	Dissipation rate, W/s
∇	Gradient
$\boldsymbol{\omega}$	Specific dissipation

Superscripts and Subscripts

1	Initial
2	Final
'	Fluctuating component of any property
o	Outlet
$\overline{}$	Mean Property
$\rightarrow$	Vector quantity
c	Cold
Crit	Critical
h	Hot
i,j,k	Cartesian coordinates' directions
In	Inlet
Re*f*	Reference
T	Turbulent property
w	Water

Abbreviations

2D	Two dimensional configurations
3D	Three dimensional configurations
ASM	Algebraic Stress Model
CFD	Computational Fluid Dynamics
HT	Heating Transfer
RANS	Reynolds average Navier- Stokes equations
RNG	Renormalization group
RSM	Reynolds Stress Model
SHTHEX	Shell and tube heat exchanger
STD	Standard
SIMPLEC	Semi-Implicit Method for Pressure-Linked Equations Consistent

ABSTRACT

The aim of this study is to simulate and analyze the heat transfer and friction characteristics of shell and tube heat exchangers. Case Studies are made to decrease the pressure drop and to increase the heat transfer and the ratio of heat transfer and pressure drop in shell and tube type heat exchanger by tilting the baffle angle up to which we get the minimum pressure drop using different turbulence models. The optimum results are found at baffle tilting angle of 32^o and RNG k – ε turbulence model. The ratio of Nus / Eu deals with high heat transfer and low pressure drop and that is found at angle of 32^o baffle angle 12% more than segmental baffle at 0^o baffle angle.

Other studies are carried out using a set of CFD simulations for a single shell and single tube pass heat exchanger with a variable number of baffles and different baffle cutoff. The results are observed to be sensitive to the turbulence model selection. The results are observed to be sensitive to the turbulence model selection. The best turbulence model among the ones considered is determined by comparing the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell-Delaware method results. For two baffle cut values, the effect of the baffle spacing to shell diameter ratio on the heat exchanger performance is investigated by varying flow rate. The best turbulence model among the ones considered is determined by comparing the CFD results of heat transfer coefficient, outlet temperature and pressure drop with literature data results. All studies are carried out using commercial software for computational fluid dynamics (CFD) known as ANSYS CFX 16. The CFD modeling techniques solved the continuity, momentum and energy conservation equations in addition to RNG k – ε model equations for turbulence closure.