1992

Numerical analysis of fluid flow and heat transfer in a power plant condenser.

Ying. Zhang

University of Windsor

Follow this and additional works at: https://scholar.uwindsor.ca/etd

Recommended Citation
https://scholar.uwindsor.ca/etd/2881

This online database contains the full-text of PhD dissertations and Masters' theses of University of Windsor students from 1954 forward. These documents are made available for personal study and research purposes only, in accordance with the Canadian Copyright Act and the Creative Commons license—CC BY-NC-ND (Attribution, Non-Commercial, No Derivative Works). Under this license, works must always be attributed to the copyright holder (original author), cannot be used for any commercial purposes, and may not be altered. Any other use would require the permission of the copyright holder. Students may inquire about withdrawing their dissertation and/or thesis from this database. For additional inquiries, please contact the repository administrator via email (scholarship@uwindsor.ca) or by telephone at 519-253-3000ext. 3208.
NOTICE

The quality of this microform is heavily dependent upon the quality of the original thesis submitted for microfilming. Every effort has been made to ensure the highest quality of reproduction possible.

If pages are missing, contact the university which granted the degree.

Some pages may have indistinct print especially if the original pages were typed with a poor typewriter ribbon or if the university sent us an inferior photocopy.

Reproduction in full or in part of this microform is governed by the Canadian Copyright Act, R.S.C. 1970, c. C-30, and subsequent amendments.

AVIS

La qualité de cette microforme dépend grandement de la qualité de la thèse soumise au microfilmage. Nous avons tout fait pour assurer une qualité supérieure de reproduction.

S’il manque des pages, veuillez communiquer avec l’université qui a conféré le grade.

La qualité d’impression de certaines pages peut laisser à désirer, surtout si les pages originales ont été dactylographiées à l’aide d’un ruban usé ou si l’université nous a fait parvenir une photocopie de qualité inférieure.

La reproduction, même partielle, de cette microforme est soumise à la Loi canadienne sur le droit d’auteur, SRC 1970, c. C-30, et ses amendements subséquents.
Numerical Analysis of Fluid Flow and Heat Transfer in a Power Plant Condenser

by

Ying Zhang

A Thesis
Submitted to the Faculty of Graduate Studies and Research
Through the Department of Mechanical Engineering
in Partial Fulfilment of the Requirement for
the Degree of Master of Applied Science
at University of Windsor

Windsor, Ontario, Canada
August, 1992

(c) 1992 Ying Zhang
The author has granted an irrevocable non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of his/her thesis by any means and in any form or format, making this thesis available to interested persons.

The author retains ownership of the copyright in his/her thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without his/her permission.

L'auteur a accordé une licence irrévocable et non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de sa thèse de quelque manière et sous quelque forme que ce soit pour mettre des exemplaires de cette thèse à la disposition des personnes intéressées.

L'auteur conserve la propriété du droit d'auteur qui protège sa thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

ISBN 0-315-78884-4
ABSTRACT

A quasi-three dimensional computer algorithm is developed to simulate fluid flow and heat transfer in the shell side of a power plant steam surface condenser. The pressure drop balance approach is used to determine the inlet mass flow rate. Both air blanket and inundation effects are taken into account. The porous medium is used to model the tube bank, and the eddy viscosity model is used to calculate the turbulent viscosity.

The governing equations are solved in primitive variable form using a semi-implicit consistent formulation in which a segregated correction linked algorithm is employed.

Sensitivity studies are carried out for four different correlations of condensation heat transfer coefficient, different inundation factors as well as different turbulent viscosity models.

The numerical predictions of an experimental steam condenser are critically assessed by comparison against available experimental data. The results indicate that solutions obtained by employing the numerical technique developed in this study are in good agreement with experimental data.
DEDICATION

To my beloved parents

Father Zhang, Wenbo; Mother Wang, Guiling
ACKNOWLEDGEMENT

It is a great pleasure for me to express my sincere gratitude to my supervisor, Dr. Chao Zhang, for her invaluable guidance, encouragement and support during the course of this work.

Special thanks go to the other committee members, Dr. Barron and Dr. Rankin, for their kind cooperation, helpful suggestions and comments.

The assistance rendered by several fellow graduate students, especially Mr. Yijing Fan, Mr. Satya Kurada and Mr. Rambod Larijani, is greatly appreciated.

The financial assistance from the University of Windsor, and Natural Sciences and Engineering Research Council of Canada is gratefully acknowledged. The facilities provided by the University of Windsor Computer Centre is also recognized.

Finally, I would like to thank my fiancee, Xing Pan, for his devoted love and support.
TABLE OF CONTENTS

ABSTRACT ................................................................. iii
DEDICATION .............................................................. iv
ACKNOWLEDGEMENT .................................................... v
LIST OF FIGURES ........................................................ ix
LIST OF TABLES .......................................................... xi
NOMENCLATURE .......................................................... xii

CHAPTERS

I. INTRODUCTION ...................................................... 1
   1.1 Motivation ....................................................... 1
   1.2 Scope of Present Work .......................................... 2

II. REVIEW OF PREVIOUS WORK .................................... 5
   2.1 Predictions of Fluid Flow and Heat Transfer in Condensers . . . . 5
   2.2 Calculations of Condensation Heat Transfer .................... 9
III. NUMERICAL MODEL AND ANALYSIS .................. 13

3.1 Quasi-Three-Dimensional Model .................. 14
3.2 Mathematical Formulation .......................... 15
   3.2.1 Governing Equations .......................... 15
   3.2.2 Auxiliary Relationships ...................... 16
3.3 Boundary Conditions ............................... 22

IV. NUMERICAL SOLUTION TECHNIQUE .................. 23

4.1 Discretization of Differential Equations ............ 24
   4.1.1 Grid Arrangement ............................. 25
   4.1.2 Discretized Equations ....................... 26
4.2 Differencing Schemes .............................. 27
4.3 Numerical Algorithm ................................ 29
   4.3.1 The SIMPLEC Numerical Formulation ............ 29
   4.3.2 Iterative Method for Linearized Algebraic Equations 31
   4.3.3 Solution Procedure ............................. 32
   4.3.4 Convergence Criteria .......................... 32

V. SIMULATION OF AN EXPERIMENTAL CONDENSER ....... 35

5.1 Computational Detail ............................... 36
5.2 Result and Discussion .............................. 37

VI. SENSITIVITY STUDY OF AUXILIARY RELATIONS ...... 63

6.1 Condensation Heat Transfer Coefficient ............. 64
6.2 Turbulent Viscosity ............................... 68
VII. CONCLUSION .................................................. 82
  7.1 Contributions .............................................. 82
  7.2 Future Work ............................................... 83
REFERENCES ....................................................... 85
VITA AUCTORIS .................................................... 91
LIST OF FIGURES

Figures                           page
4.1 Grid Arrangement             34
5.1 Configuration of the Experimental Condenser  41
5.2 Grid used for the Simulation  42
5.3 Flow Chart                   43
5.4 Distribution of Velocity for Plane No.1     44-45
5.5 Contour Map of Heat Flux       46-49
5.6 Contour Map of Air Mass Fraction     50-53
5.7 Comparison of Predicted Heat Flux along 3rd Row  
    from Bottom of Tube Bank with Experiment Data  54
5.8 Comparison of Predicted Heat Flux along 8th Row  
    from Bottom of Tube Bank with Experiment Data  55
5.9 Comparison of Predicted Heat Flux along 13th Row  
    from Bottom of Tube Bank with Experiment Data  56
5.10 Comparison of Predicted Heat Flux along 18th Row  
    from Bottom of Tube Bank with Experiment Data  57
5.11 Distribution of Condensation Rate          58
5.12 Comparison of Predicted Heat Flux along 3rd Row from Bottom  
    of Tube Bank with Al-Sanea's and Bush's Results  59
5.13 Comparison of Predicted Heat Flux along 8th Row from Bottom  
    of Tube Bank with Al-Sanea's and Bush's Results  60
5.14 Comparison of Predicted Heat Flux along 13th Row from Bottom  
    of Tube Bank with Al-Sanea's and Bush's Results  61
5.15 Comparison of Predicted Heat Flux along 18th Row from Bottom  

ix
of Tube Bank with Al-Sanea's and Bush's Results .......................... 62
6.1 Comparison of Predicted Heat Flux along 3rd Row by Different Correlations of Condensation Heat Transfer Coefficient ............ 70
6.2 Comparison of Predicted Heat Flux along 8th Row by Different Correlations of Condensation Heat Transfer Coefficient ............ 71
6.3 Comparison of Predicted Heat Flux along 13th Row by Different Correlations of Condensation Heat Transfer Coefficient ............ 72
6.4 Comparison of Predicted Heat Flux along 18th Row by Different Correlations of Condensation Heat Transfer Coefficient ............ 73
6.5 Comparison of Predicted Heat Flux along 3rd Row by Different Inundation Correction Factors .......................... 74
6.6 Comparison of Predicted Heat Flux along 8th Row by Different Inundation Correction Factors .......................... 75
6.7 Comparison of Predicted Heat Flux along 13th Row by Different Inundation Correction Factors .......................... 76
6.8 Comparison of Predicted Heat Flux along 18th Row by Different Inundation Correction Factors .......................... 77
6.9 Comparison of Predicted Heat Flux along 3rd Row by Different Turbulent Viscosity Models .......................... 78
6.10 Comparison of Predicted Heat Flux along 8th Row by Different Turbulent Viscosity Models .......................... 79
6.11 Comparison of Predicted Heat Flux along 13th Row by Different Turbulent Viscosity Models .......................... 80
6.12 Comparison of Predicted Heat Flux along 18th Row by Different Turbulent Viscosity Models .......................... 81
LIST OF TABLES

Tables

5.1 Geometrical and Operating Parameters of
the Experimental Condenser ...................................... 36

6.1 Total Condensation Rates by Using Different Correlations
of Condensation Heat Transfer Coefficient .................... 66

6.2 Total Condensation Rates by Using Different Inundation Factors . 67

6.3 Total Condensation Rates by Using Different Viscosity Models . 69
NOMENCLATURE

A  area of control volume face or heat transfer area
a  discretization coefficient
b  constant part of linearized source term in the discretized equations
c  condensation rate of nth tube row without inundation
$c_\rho$  specific heat at constant pressure
$D$  diffusivity of air in vapour
$D_h$  hydraulic diameter
$D_i$  inner diameter of tube
$D_o$  outer diameter of tube
$Fr$  Froude number = $M^2/\rho^2 g D_o$
$F_wF_v$  local flow resistance forces in momentum equations
$f_u,f_v$  friction factors
$G$  fluid mass velocity
g  gravitational acceleration
$Gr$  Grashof Number=$D_o^3 g/\mu^2 (\rho_c (\rho_c-\rho_g))$
$H$  non-dimensional number of condensation = $c_p (T_s-T_w)/Pr_c L$
$J$  diffusion flux
$k$  thermal conductivity
L  latent heat of condensation
l  mixing length
M  total number of control volumes in y-direction
\dot{M}  steam condensation rate
\dot{m}  steam condensation rate per unit volume
N  total number of control volumes in x-direction
P  tube pitch
Pr  Prandtl number = c_p\mu/k
p  pressure
p^*  estimated pressure
p^i  pressure correction
q  heat flux
R  radius of pipe or thermal resistance
Re  Reynolds number = \rho U D / \mu
ra  \rho\mu\text{-ratio} = (\rho_0 \mu_0 / \rho_1 \mu_1)^{1/2}
S  source term in the general differential equation
\bar{S}  average value of source term over control volume
S_c  constant part of linearized source
S_p  variable dependent part of linearized source
s  power of inundation correction factor
T  temperature
U  velocity vector magnitude = (u^2 + v^2)^{1/2}
u  velocity component in the x-direction

\( u^* \)  estimated velocity component in x-direction

\( u' \)  friction velocity

\( V \)  volume

\( v \)  velocity component in the y-direction

\( v^* \)  estimated velocity component in y-direction

\( w \)  velocity component in z-direction

\( w \)  total amount of condensate leaving the nth tube

\( w^* \)  estimated velocity component in the z-direction

\( x \)  main flow direction coordinate

\( y, z \)  cross-stream coordinates

**Greek Letters**

\( \alpha \)  heat transfer coefficient on steam side

\( \beta \)  local volume porosity

\( \Gamma \)  diffusion coefficient

\( \gamma \)  residue reduction factor

\( \mu \)  laminar dynamic viscosity

\( \mu_e \)  effective dynamic viscosity

\( \mu_t \)  turbulent dynamic viscosity

\( \xi_{\text{ex}} \)  loss coefficients
\( \rho \)  \text{density} \\
\( \phi \)  \text{general dependent variable} \\
\( \Phi \)  \text{air mass fraction} \\

**Subscripts**

\( a \)  \text{air} \\
\( B,T \)  \text{grid points} \\
\( b,t \)  \text{control volume faces} \\
\( c \)  \text{condensate} \\
\( cs \)  \text{steam-condensate interface} \\
\( E,W \)  \text{grid points} \\
\( e,w \)  \text{control volume faces} \\
\( N,S \)  \text{grid points} \\
\( n,s \)  \text{control volume faces} \\
\( nb \)  \text{neighbour grid points} \\
\( o \)  \text{outlet} \\
\( P \)  \text{grid point} \\
\( s \)  \text{steam} \\
\( t \)  \text{tube} \\
\( u \)  \text{parameter in x-momentum equation} \\
\( v \)  \text{parameter in y-momentum equation}
w  wall
w  cooling water
x  variables in x-direction
y,z variables in y and z-direction respectively

Superscripts

0  previous iteration
1.1 Motivation

Steam condensers play an important role in a turbine generator unit. From the simple Rankine cycle, which is the basic principal of a turbine generator unit, it can be seen that the power generated by the turbine is determined by turbine inlet steam temperature and condenser pressure. Failure and/or off-design performance of power plant steam surface condensers, as pointed out by Diaz-Tous\textsuperscript{[1]}, have caused damage or low efficiency of the turbine and pump, and resulted in substantial loss. Thus, accurate and detailed predictions of the shell-side flow distribution and heat transfer are of primary importance in performing the hydraulic design analysis of power plant condensers, particularly in what concerns flow induced vibration, heat transfer and pressure losses. Consequently, enhancement in design and performance is critically dependent upon the availability and accuracy of the required information. In the early stage of the design process, a prior
knowledge of the flow field may eliminate problems related to flow induced vibration and
flow maldistribution, and it can also identify potential problem areas such as a stagnant
corrosion damage zone, establish the appropriate corrective measures to avoid potential
structural damage, and eventually improve overall heat transfer coefficients and increase
the reliability and performance of the equipment.

Especially over the last thirty years, condenser size has increased dramatically, as noted
by Marto[2]. It has become more important and yet more difficult to design the condensers
for efficient thermal performance. This indicates that there is a clear need for a design
method which can predict the shell-side fluid flow and heat transfer more accurately. An
efficient and viable approach to improve the design tools is through the development of
advanced numerical models using fundamental conservation equations and appropriate
constitutive relations.

1.2 Scope of Present Work

The purpose of the present study is to develop a reliable model to predict three-
dimensional fluid flow and heat transfer in power plant condensers more accurately,
which may lead to improvements in the design and performance of condensers.

The main objectives of the present work may be stated as follows:

(1) develop a quasi-three-dimensional numerical model to predict the shell-side fluid flow
and heat transfer in power plant condensers,
(2) test different correlations for condensation heat transfer and their benchmarking against experimental data,

(3) study the effects of inundation on heat transfer in the shell side of condensers,

(4) study the effects of turbulent viscosity, both as fixed values and when allowed to vary with local conditions, and

(5) validate the condenser model by applying it to an experimental condenser.

The simulation method developed in this study is based on the fundamental governing equations of mass, momentum and air mass fraction conservation. A quasi-three-dimensional model including the effects of inundation and air blanket is proposed to predict the heat transfer and fluid flow in the shell side of power plant condensers. The eddy viscosity model is employed to take account of effects of turbulence. The 'porous media' principle is incorporated into the governing equations to account for the flow volume reduction, and for the distributed hydraulic and thermal resistance due to the tube bundles, baffles and other internal obstacles.

The conversion of the differential equations into equivalent finite-volume equations is carried out by integrating over a small control volume, and piece wise profiles expressing the variation of the dependent variables between the grid points are used to evaluate the required integrals. A staggered grid is used to perform the discretization. The resulting discretized equations are solved in primitive variables using the SIMPLEC algorithm\(^3\).
This thesis is organized as follows:

Chapter 1 - A brief introduction about the thesis;

Chapter 2 - Review of the previous work in the area of interest;

Chapter 3 - Description of the quasi-three dimensional-model, the governing equations and appropriate constitutive relations used for the simulation of shell-side fluid flow and heat transfer in power plant condensers;

Chapter 4 - General description of the discretization technique and numerical procedure used in this thesis;

Chapter 5 - Numerical simulation of an experimental condenser to validate the model;

Chapter 6 - Sensitivity study of different correlations of condensation heat transfer coefficient, inundation correction factors and turbulence models;

Chapter 7 - Conclusions and recommendations for future work.
2.1 Prediction of Fluid Flow and Heat Transfer in Condensers

The complex heat and mass transfer processes occurring in a condenser have led designers to rely heavily on empirical methods to predict the performance. Such methods\(^\text{[41,5]}\) are incorporated in design codes in which a mean heat transfer coefficient is obtained from an empirical equation accounting for the velocity of the cooling water and its mean temperature. These methods are based on the assessment of previous designs and in most cases take no account of the shape of the tube bank. They are useful mainly for small, simple tube bank arrangement condensers.

For large power condensers with complex tube bank arrangement, the desire to improve efficiency and hence reduce operating costs of the whole turbine unit has lead the designers to seek information regarding the fluid motion and thermal condition at a more detailed level. A number of numerical schemes have been developed to provide
information on local velocity, pressure drop, heat and mass transfer.

The numerical technique developed by Theodossiou et al. \cite{6} can be used to predict flow distribution in shell-and-tube heat exchangers. The flow is assumed to be two-dimensional, incompressible and isothermal. The tube bundles and baffles are modelled by "porous media" concepts with the distributed resistance approach incorporated into the governing equations. The resulting discretized equations are solved in primitive variables using the Semi-Implicit Pressure-Linked Equations Consistent (SIMPLEC) numerical formulation proposed by Van Doormaal and Raithby\cite{23}. The convection-diffusion flux terms are approximated using the hybrid scheme proposed by Spalding\cite{7}, and the discretized pressure correction equation which links the momentum and continuity equations is solved using a multi-grid technique. A uniform effective viscosity is assumed in the governing equations to allow for turbulence modelling.

A computational procedure to calculate two-dimensional flow in various shell-and-tube heat exchanger configurations was developed by Carlucci et al \cite{8}. The geometry and internal obstacles for different shell-and-tube heat exchangers are modelled by an isotropic porosity distribution. The authors indicated that the predictive accuracy depends not only on the fluid flow correlations used, but also on how well internal obstacles such as tube bundles and flow blocking devices are described in the numerical model. Different procedures to define the outline of tube bundle segments of arbitrary shape, and flow-blocking devices of arbitrary orientation and location were also described. The Semi-
Implicit Pressure-Linked Equations (SIMPLE) algorithm described by Patankar was employed for solving the discretized equation. This procedure has been extended by Sousa and LeBlanc, and Rhodes and Carlucci to predict the two-dimensional, isothermal shell-side fluid flow in an experimental heat exchanger.

Davidson and Rowe described a computational method to simulate condenser performance, which is based on the assumptions that there is no steam flow parallel with the tube axes in the bank, flow is two-dimensional and the diffusion terms can be neglected. The governing equations are continuity, momentum and air concentration field equations combined with relations correlating available experimental data, such as loss relations and local heat transfer coefficient correlations. Finite difference discretization of the field equations is used.

Caremoli presented a computational method to predict the shell-side flow in power plant condensers and their performance. The resulting equations are the equations of conservation of mass, momentum, and non-condensable gas mass fraction in a porous medium with flow resistance, heat and mass transfer. The viscous stress tensor and the diffusion flux vector are calculated with a fixed value of turbulent viscosity and a turbulent Schmidt number which is taken as unity. The steam is assumed to be saturated throughout the condenser shell, the steam and non-condensable gas mixture is assumed to be a perfect gas. Three-dimensional effects are not taken into account.

A mathematical model of a condenser, developed using a general purpose fluid flow
computer code, was described by Al-Sanea et al\cite{14}. The flow is treated as two-
dimensional, single-phase and steady. The equations solved are those representing the
conservation of momentum, steam concentration and mass continuity. The model
considers only the vapour phase, since the condensed water is assumed to be
instantaneously collected in the hot well. The diffusive terms are taken as fixed values,
and the inundation and three-dimensional effects are ignored.

A finite element method was used to analyze fluid flow and heat transfer in condensers
by Shida et al\cite{15}. A triangular mesh pattern is used, and all diffusion terms in the
governing equations are omitted, with the assumption that the mixture of steam and non-
condensable gas follows the perfect gas behaviour. This method was applied to one- and
two-dimensional experimental condensers.

A numerical procedure was developed to simulate the fluid flow and heat transfer
processes in power plant condensers by Zhang\cite{16}. The three-dimensional effect due to the
cooling water temperature difference has been taken into consideration by a series of step
by step two-dimensional calculations, the inlet mass flow rate of each domain is
determined by the condensation capability in that domain. A porous media concept is used
to model condenser tube bank, and fixed turbulent viscosity and mass diffusivity are
assumed. The effects of inundation and air-blanket are not taken into account.

In general, modelling is conducted under the assumption that the flow is two-dimensional,
and inundation, air-blanket and variation of diffusion terms can be neglected. This can be
reasoned in terms of computational speed and memory, and in addition the thermal hydraulic phenomena are not sufficiently understood to permit their description in a well defined set of constitutive equations. The two-dimensionality condition, however, may impose undue restrictions upon the analysis as explained by Brickell\(^1\). The shell-side flow within large power plant condensers, in general, is turbulent and three-dimensional. The flow in different sections along the cooling water flow direction will differ due to the rise of the cooling water temperature. In turn the cooling water temperature gradient will strongly affect the exhausted hood flow, which is not necessarily two-dimensional. Inundation and air-blanket are unavoidable phenomena in the shell-side condenser. They influence directly the pressure distribution and heat transfer in the tube area. A proper modelling of these terms is very important for the accuracy of numerical prediction. A practical approach is therefore needed to establish, in particular for power plant condensers, an algorithm which includes the effects of three-dimensional flow, inundation and air-blanket to predict the flow field in both untubed and tubed regions more realistically.

2.2 Calculation of Condensation Heat Transfer

Because the calculation of condensation heat transfer is a crucial part of the whole condenser simulation and of considerable importance in the accuracy of the simulation, research has been done in this field for several decades.
The starting point for the calculation of shell-side condensation heat transfer is the pioneering work of Nusselt\cite{18} in 1916 for laminar film condensation on a single horizontal tube. He idealized the problem by assuming, among other things, a pure quiescent vapour and uniform tube wall temperature. His analysis produced a well known relationship for the condensation heat transfer coefficient.

However, condensation heat transfer in an actual condenser can be quite different from the idealized condition that Nusselt assumed. In reality, vapour velocity and condensate inundation are important factors affecting shell-side condensation heat transfer and it is demonstrated that they must be taken into account. Theoretical models, based on single tube research, have been proposed from which tube bank correlations have been inferred.

For a single tube, the condensation is affected by vapour velocity in two important ways: (a). the surface shear between the vapour and the condensate, (b). the effect of vapour separation.

Grant and Osment\cite{19} proposed a modified Nusselt's equation based on solutions of the equation of motion of the condensate film only. They assumed that interfacial shear stress during condensation is analogous to a non-condensing gas flowing over a dry surface.

Shekarladze\cite{20} and Fujii et al.\cite{21} accounted for the vapour shear stress and employed two-phase laminar boundary layer equations incorporating the interface between liquid and vapour. The flow of the bulk vapour was assumed in this case to be potential. Both Shekarladze\cite{20} and Fujii et al.\cite{21} assumed there was no separation in the boundary layer.
However, Shekriladze\textsuperscript{[20]} obtained expressions which model the shear dominated condition by using an asymptotic expression for the shear stress. Fujii et al.\textsuperscript{[21]} reported more complete solutions by matching the shear stress at vapour-condensate interface and using an approximate integral treatment of vapour boundary.

Rose\textsuperscript{[22]} studied condensate film separation effect. He assumed a fixed separation angle and proposed an approximate correction of Shekriladze's equation by using the integral treatment of the vapour boundary.

Honda and Fujii\textsuperscript{[23]} treated the condensation heat transfer problem as a conjugate of the two-phase boundary layer equations and the heat conduction equation within the tube. They also assumed several separation angles in trial calculations, then found that the condensate film separation has little influence on the results.

All these equations are based on the condensation in a single tube which includes the vapour velocity effect but neglects the condensate inundation. The common treatment of inundation is to consider the effects of inundation as a separate correction factor applied to the above correlations.

By a similarity analysis of condensation equations, Kutateladze\textsuperscript{[24]} and Fuks\textsuperscript{[25]} derived a non-dimensional equation which accounted for the predominant physical mechanism.

\[
\frac{\alpha_n}{\alpha_1} = f(w/c) \tag{2.1}
\]

where \(n\) means the \(n\)th tube in a row.
Based on the experimental data, Wilson\textsuperscript{[26]} showed that

\[
\frac{\alpha_\text{w}}{\alpha_1} = (w/c)^{-s}
\]  \hspace{1cm} (2.2)

where the value of \( s \) is between 0.07 and 0.223.

While a number of correlations to calculate condensation heat transfer coefficient as mentioned above are available, only Uehara and Dilao\textsuperscript{[27]} have done some research on applying these equations to single tube calculations. There is no report in the open literature concerning the comparisons of the accuracy of predictions obtained by using different condensation heat transfer coefficient correlations for shell-and-tube type condensers.
CHAPTER III

NUMERICAL MODEL AND ANALYSIS

A great deal of research on shell-side fluid flow and heat transfer in condensers has been conducted under the assumption of two-dimensional flow, non-inundation and constant diffusive terms. However the shell-side flow within large power plant condensers is, in general, highly three-dimensional, and inundation as well as fluid property modelling have a significant influence on the accuracy of the simulation on large power plant condensers. Two-dimensionality, non-inundation and constant diffusive term assumptions may oversimplify the fluid flow and heat transfer inside the condenser.

In this study, a quasi-three-dimensional model which is based on the work by Zhang\textsuperscript{[16]} is developed to evaluate the thermal performance of condensers by using coupled heat transfer and fluid flow calculation procedures. A pressure drop balance approach is used to determine inlet mass flow rate distribution along the cooling water flow direction. Inundation and air-blanket effects have been taken into consideration. The eddy viscosity
model is used to simulate the turbulent diffusive term and Fujii\textsuperscript{[21]} condensation heat transfer coefficient correlation is employed.

3.1 Quasi-Three-Dimensional Model

Since there are always some partition plates inside a large size condenser, the condenser shell side can be subdivided into a number of domains normal to the cooling water flow direction. In each domain, the flow is assumed to be two-dimensional and the calculations are performed for the plane located half way between partition plates. The domains interact with each other through the thermal memory of the cooling water. That is, the outlet cooling water temperature of the proceeding domain is used as the inlet cooling water temperature for the successive domain. Calculations for each domain are made sequentially starting from the cooling water inlet end. The three-dimensional effect due to the cooling water temperature difference is taken into account by a series of two-dimensional calculations, each being for one domain. The inlet mass flow rate into each domain is determined by the constraint that the pressure drops from inlet to the vent for all the domains must be identical since domains share the same inlet and vent. This is the so-called pressure drop balance\textsuperscript{[4]}. The air-blanket is assumed to occur at air mass fraction $\Phi \geq 90\%$, and results in heat flux $Q=0$ and $R_{\infty}$.
3.2 MATHEMATICAL FORMULATION

The shell-side flow in the condenser is assumed to be in steady state, and the working fluid, which is treated as a mixture of non-condensable gases (air) and steam, behaves as a Newtonian fluid with constant viscosity. The steam taken as saturated and the mixture of air and vapour, for simplicity, is assumed to be a perfect gas although other equations of state can be considered. The density has been allowed to vary locally according to the perfect gas state equation. An isotropic porosity, $\beta$, which is employed to describe the flow volume reduction due to the tube bundle and baffles, is defined as

$$\beta = \frac{\text{Volume occupied by the fluid}}{\text{Total volume}}$$

3.2.1. Governing Equations

The flow is treated as two-dimensional and steady in each domain; governing equations are expressed in the Cartesian coordinate system.

The variables solved for are:

- mixture velocity in the $x$ and $y$ directions, $u$ and $v$
- steam concentration, $\Phi$, and
- pressure, $p$

The equations solved are those representing the conservation of mass, momentum, and air mass fraction.
CHAPTER 3 NUMERICAL MODEL AND ANALYSIS

Mass Conservation Equation

$$\frac{\partial}{\partial x} (\beta \rho u) + \frac{\partial}{\partial y} (\beta \rho v) = -\beta \dot{m} \quad (3.1)$$

Momentum Conservation Equation

$$\frac{\partial}{\partial x} (\beta \rho u^2) + \frac{\partial}{\partial y} (\beta \rho vu) = \frac{\partial}{\partial x} (\beta \mu e \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\beta \mu e \frac{\partial u}{\partial y}) + \beta \frac{\partial p}{\partial x} - \beta \dot{m} u - \beta F_u \quad (3.2)$$

$$\frac{\partial}{\partial x} (\beta \rho u v) + \frac{\partial}{\partial y} (\beta \rho v^2) = \frac{\partial}{\partial x} (\beta \mu e \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y} (\beta \mu e \frac{\partial v}{\partial y}) + \beta \frac{\partial p}{\partial y} - \beta \dot{m} v - \beta F_v \quad (3.3)$$

Conservation of Air Mass Fraction

$$\frac{\partial}{\partial x} (\beta \rho \Phi u) + \frac{\partial}{\partial y} (\beta \rho \Phi v) = \frac{\partial}{\partial x} (\beta \rho D \frac{\partial \Phi}{\partial x}) + \frac{\partial}{\partial y} (\beta \rho D \frac{\partial \Phi}{\partial y}) \quad (3.4)$$

where \(x, y\) - coordinates in steam main flow and cross flow directions;

\(u, v\) - velocity components in \(x, y\) directions

3.2.2. Auxiliary relationships

Physical property: The density varies locally according to the relation:

$$\rho = \frac{p}{R_m T} \quad (3.5)$$
\[ R_m = \Phi R_a + (1 - \Phi) R_s \]

where \( T \) is the saturation temperature determined by partial steam pressure, and \( R_m, R_s \) and \( R_a \) are the gas constants for the mixture, air and steam, respectively.

The concept of an effective viscosity is used, which is defined as the sum of the laminar and turbulent viscosity, namely:

\[ \mu_e = \mu + \mu_t \]  \hspace{1cm} (3.6)

The Schmidt Number has been assumed to have a value of unity, which means that turbulent diffusivity is equal to turbulent viscosity. The molecular viscosity has been assumed to be constant for steam, condensate and gas.

Turbulent viscosity \( \mu_t \) is calculated by using the algebraic eddy viscosity model\(^{[14]}\)

\[ \mu_t = \rho u' l \]  \hspace{1cm} (3.7)

where \( u' \) is friction velocity, and \( l \) is mixing length.

In the untubed region, by use of Prandtl’s assumption \( u' = l dU/dy \), the above equation can be written as

\[ \mu_t = \rho l^2 \frac{dU}{dy} \]  \hspace{1cm} (3.8)

where \( dU/dy \) is the mean velocity gradient across the lane.

In the tubed region, \( u' \) has been estimated from an empirical correlation given by
Weisman and Bowring\textsuperscript{[28]}, which is

\[ u' = \frac{aG D_h R e^{-0.1}}{\rho} \]  \hspace{1cm} (3.9)

where \( G \) - fluid mass velocity

\( D_h \) - hydraulic diameter = \( 2\sqrt{3} p^2/\pi d_o \cdot d_o \)

\( R e \) - Reynold number based on \( D_h \)

\( a \) - constant

Studies have also been carried out with fixed values of \( \mu_p \) to compare the effect of different turbulent viscosity modelling on the accuracy of the simulation.

**Momentum Source Term:** The local hydraulic flow resistance force, \( F_u \) and \( F_v \), in momentum equations due to the tube bundle and / or baffle are related to the pressure loss coefficients, \( \xi_u \) and \( \xi_v \), by

\[ F_u = \xi_u \rho u U_p \]  \hspace{1cm} (3.10)

\[ F_v = \xi_v \rho v U_p \]  \hspace{1cm} (3.11)

There is no general expression for the pressure loss in condensing tube banks, most of the existing correlations for the pressure loss were developed based on the experimental data for a specific configuration. As a first approximation, the expressions proposed by Rhodes and Carlucci\textsuperscript{[11]} for the loss coefficients, \( \xi_u \) and \( \xi_v \), are used, namely:
\[ \xi_u = 2 \left( \frac{f_u}{P} \left( \frac{P}{P - D_o} \right)^2 \left( \frac{1 - \beta}{1 - \beta_f} \right) \right) \]  

\[ \xi_v = 2 \left( \frac{f_v}{P} \left( \frac{P}{P - D_o} \right)^2 \left( \frac{1 - \beta}{1 - \beta_f} \right) \right) \]  

The friction factors, \( f_u \) and \( f_v \), are determined by:

\[
 f_u = \begin{cases} 
 0.619 Re_u^{-0.198} ; & Re_u < 8000 \\
 1.156 Re_u^{-0.2647} ; & 8000 \leq Re_u < 2 \times 10^5 
\end{cases}
\]

\[
 f_v = \begin{cases} 
 0.619 Re_v^{-0.198} ; & Re_v < 8000 \\
 1.156 Re_v^{-0.2647} ; & 8000 \leq Re_v < 2 \times 10^5 
\end{cases}
\]

where \( Re_u \) - x-direction Reynold number (\( = \rho u D_o / \mu \));

\( Re_v \) - y-direction Reynold number (\( = \rho v D_o / \mu \));

The porosity in the tube bundle region, \( \beta_i \), is defined as:

\[ \beta_i = 1 - \frac{\pi}{2\sqrt{3}} \left( \frac{D_o}{P} \right)^2 \]

where \( D_o \) - outer diameter of cooling water tube;

\( P \) - tube pitch.

In the tube bundle region, \( \beta = \beta_i \); in the untubed region, \( \beta = 1 \).
Mass Source Term: The steam mass condensation rate per unit volume, \( \dot{m} \), can be obtained by equating the phase change enthalpy with the heat transfer rate, as follows:

\[
\dot{m}LV = \frac{T - T_w}{R} A
\]

(3.14)

where:
- \( L \) - latent heat of condensation;
- \( V \) - volume of a given control volume;
- \( A \) - heat transfer area of a given control volume;
- \( R \) - overall thermal resistance;

The cooling water temperature \( T_w \) is obtained by heat balance between the steam and cooling water in each control volume. The overall thermal resistance for each control volume, \( R \), is the sum of all individual resistances which are calculated from various semi-empirical heat transfer correlations,

\[
R = R_w \frac{D_o}{D_i} = R_t + R_c + R_f + R_a
\]

(3.15)

For the water side thermal resistance, the relationship given by Ditus and Boelter\(^{29}\) can be employed,

\[
\frac{1}{R_w} = 0.023 \frac{k_w Re_{w}^{0.8} Pr^{0.4}}{D_i}
\]

(3.16)

The wall resistance for each tube is obtained with the assumption of one-dimensional, steady-state conduction;
\[
R_f = \frac{D_o \ln \left( \frac{D_o}{D_i} \right)}{2k_t}
\]

(3.17)

The fouling resistance \( R_f \) is taken as \( 3.5 \times 10^{-5} \text{ m}^2\text{K/W} \) as suggested by Naviglio et al.\(^{30}\).

The resistance from the presence of air film is evaluated via a mass transfer coefficient as reported by Berman and Fuks\(^{31}\)

\[
\frac{1}{R_a} = \frac{a D_o}{D_s} \left( \frac{P}{P - P_s} \right)^{0.5} \nu^2 \beta^2 \frac{L}{T} \frac{1}{(T - T_{\infty})^{1.5}}
\]

(3.18)

where \( a = 0.52 \) \( b = 0.7 \) for \( Re_s < 350 \)

\( a = 0.82 \) \( b = 0.6 \) for \( Re_s > 350 \)

\( D \) - turbulent diffusivity of air in vapour.

For the calculation of condensation heat transfer, the correlation given by Fujii et al.\(^{21}\) is used.

\[
\alpha_c = K \chi [1 + 0.276 / (\chi^4 FrH)]^{1/4} Re_m^{1/2} k/D_o
\]

(3.19)

where \( K = 0.8 \) for in-line arrangement

\( K = 1.0 \) for staggered arrangement

\( \chi_c = 0.9 (1 + 1.1 fr H)^{1/3} \)

\( Fr = M^2 / \rho^2 g D_o \)

\( H = c_p(T_s - T_w) / Pr_c L \)
\[ Re_m = \frac{\rho_s M D}{\rho_c \mu_c} \]

\[ ra = (\frac{\rho_c \mu_c}{\rho_s \mu_s})^{1/2} \]

\[ Pr_c = \frac{c_{pc} \mu_c}{k_c} \]

\( M \) - mass velocity of steam through maximum flow area.

The inundation correction factor \((w/c)^{-0.223}\) is applied on the Equation (3.19).

### 3.3 Boundary Conditions

The boundary conditions for inlet, vent and solid wall are:

**Inlet:** The velocity, pressure and air mass fraction are specified at the inlet boundary.

**Vent:** The velocity and air mass fraction at the vent are updated based on the mass conservation.

**Wall:** The shell walls of the condenser are assumed to be non-slip, impervious to flow, and adiabatic.
CHAPTER IV
NUMERICAL SOLUTION TECHNIQUE

The governing equations formulated in the previous chapter are highly non-linear and coupled together. They can not be solved analytically except for a few simple cases. Numerical solution of the above equations is the only viable approach. The complexity of the equations requires careful consideration in selecting a solution scheme that will be simple to incorporate and yet provide efficient and accurate results. In the present study, the finite volume method is used to solve the governing equations, in which the differential equations are discretized over the domain of interest to yield a set of algebraic equations.

Among various numerical algorithms used to solve the discretized equation, the Simple-Implicit Pressure-Linked Equations (SIMPLE) algorithm of Patankar and Spalding[9] not only provides a remarkably successful implicit method, but also has dominated the field
of numerical simulation of incompressible flows. Van Doormaal and Raithby \cite{6} proposed an improved version of the SIMPLE algorithm, so called SIMPLEC (SIMPLE Consistent) which simplifies its implementation and reduces solution costs. In this study the SIMPLEC algorithm will be used.

4.1 Discretization of Differential Equations

The general form of the governing differential equations for fluid flow and heat transfer in Cartesian coordinates is:

\[
\frac{\partial}{\partial x} (\rho u \phi) + \frac{\partial}{\partial y} (\rho v \phi) + \frac{\partial}{\partial z} (\rho w \phi) = \frac{\partial}{\partial x} \left( \Gamma \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left( \Gamma \frac{\partial \phi}{\partial z} \right) + S \tag{4.1}
\]

where the symbols indicate:

\( \phi \) - general dependent variable

\( \Gamma \) - diffusion coefficient

\( S \) - source term

This equation can be rewritten as

\[
\frac{\partial}{\partial x} (J_x) + \frac{\partial}{\partial y} (J_y) + \frac{\partial}{\partial z} (J_z) = S \tag{4.2}
\]

where \( J \) represents the total flux of convection and diffusion, i.e.
\[ J_x = \rho \mu \phi - \Gamma \frac{\partial \phi}{\partial x} \]  \hspace{1cm} (4.3)

\[ J_y = \rho \nu \phi - \Gamma \frac{\partial \phi}{\partial y} \]  \hspace{1cm} (4.4)

\[ J_z = \rho \omega \phi - \Gamma \frac{\partial \phi}{\partial z} \]  \hspace{1cm} (4.5)

The discretization procedure requires that the calculation domain should be divided into sufficiently small grids, and the distribution of the relevant variables may be expressed in terms of their values at each grid.

4.1.1 Grid Arrangement

In this study, a staggered grid used originally by Harlow and Welsh \cite{32}, and also described by Spalding\cite{7}, is adopted. In such an arrangement, the three velocity components are stored at the centre of the control volume faces, while all other variables, such as pressure, temperature, turbulence energy, and turbulence dissipation rate, are located at the centre of the control volume, as shown in Figure 4.1. N,S,E,W,T and B indicate the north, south, east, west, top and bottom neighbours. The control volume around point P has six faces marked n, s, e, w, t, b. The velocity grid points have their own control volumes around them which are also sketched in Figure 4.1.

This arrangement has the convenient feature, as mentioned by Parankar\cite{33}, that the velocity components are stored just at the points where they are required for the
calculation of their convective contribution, and the pressures are located to make it
convenient to calculate the pressure gradients present in the momentum equations.

4.1.2 Discretized Equations

The discretized equations are obtained by integrating the corresponding differential
equation for the dependent variable $\phi$ over each control volume in the calculation domain.
The piecewise linear profile expressing the variation of $\phi$ between the grid points is used
to evaluate the required integral. For example, when Equation (4.2) is integrated over the
control volume surrounding the point $P$ (see Figure 4.1), the resulting equation can be
expressed in the following general form:

$$J_e - J_w + J_n - J_s + J_t - J_b = \bar{S} \Delta V$$  \hspace{1cm} (4.6)

where $\Delta V$ is the volume of the control volume, $J$ represents the integrated total flux over
the corresponding control volume face, and $\bar{S}$ is the average value of $S$ over the control
volume.

The total fluxes $J_e$, $J_w$, etc. in Equation (4.6) can be expressed in terms of the values of
$\phi$ at point $P$ and neighbour points $E$, $W$, etc.

$$a_p \phi_p = a_E \phi_E + a_W \phi_W + a_N \phi_N + a_S \phi_S + a_T \phi_T + a_B \phi_B + b$$ \hspace{1cm} (4.7)

where $a_E, a_W$, etc. are non-negative quantities representing influences of convection and
diffusion and
\[ a_P = a_E + a_W + a_N + a_S + a_T + a_B - S_P \Delta V \quad (4.8a) \]

\[ b = S_c \Delta V \quad (4.8b) \]

The source term \( S \) in Equation (4.6) is linearized to

\[ S = S_c + S_p \phi_P \quad (4.8c) \]

Equation (4.7) can be expressed in compact generalized form by

\[ \phi_P = \sum a_{nb} \phi_{nb} + b \quad (4.9) \]

where the subscript \( nb \) indicates the neighbour points of \( P \), and \( \phi \) can stand for any one of the dependent variables. However, when a velocity control volume is considered, there must be a change in the subscript lettering to account for the off-set of the control volume location.

### 4.2 Differencing Schemes

The discretization of the convection terms in the governing equations is of considerable importance to the accuracy and efficiency of the finite volume method. The calculation of the convection flux on any face of the control volume needs the value of the dependent variable \( \phi \) on that face. Since the dependent variable is defined at the centre of the control volume, appropriate interpolation functions should be used to obtain the value at each
control volume face. It is possible to have more than one interpolation function which leads to different differencing schemes.

When the mesh Reynolds number is less than two, the central differencing scheme satisfies the requirements of accuracy and computational economy. Otherwise, central differencing may lead to unrealistic results and unboundedness\[^{132}\].

One of the most popular and effective schemes to circumvent convective instability is the upwind scheme. This scheme ensures unconditional stability and boundedness. It may introduce, however, a large truncation error which may lead to inaccurate solutions in recirculating flow situations.

The hybrid scheme described by Spalding \[^{7}\] combines the accuracy of the central difference scheme and the stability of the upwind scheme. This scheme, in general, tends to be a reasonable compromise between accuracy and computational effort, as shown in the work of Sousa and Hadjisophocleous\[^{34}\].

It has been verified by Zhang\[^{16}\] that the results obtained by using the upwind scheme are as accurate as that obtained by the hybrid scheme; and the convergence rate by using the upwind scheme is faster than using the hybrid scheme. Therefore, the upwind scheme is used as a compromise amongst stability, convergence rate, accuracy, and speed of computation.
4.3 Numerical Algorithm

4.3.1 The SIMPLEC Numerical Formulation

The discretized equation (4.9) with the E-factor (relaxation factor) can be written as follows:

$$a_p(1+\frac{1}{E})\phi_p = \sum a_{nb} \phi_{nb} + b\cdot\frac{a_p}{E}\phi_p^o$$  \hspace{1cm} (4.10)

where the superscript o denotes the values from the previous cycle.

For the purpose of giving an outline of the SIMPLEC algorithm, only the u-momentum equation is considered here.

Using the grid arrangement shown in Figure 4.1, the discretized u-momentum equation for the control volume centred at e may be derived from Equation (4.10) as

$$\alpha_e \mu_e = \sum a_{nb} \mu_{nb} + b_e + a_{e}(p_p - p_E)$$ \hspace{1cm} (4.11)

where

$$\alpha_e = \left(\sum a_{nb} - S_p \Delta V\right)(1 + \frac{1}{E})$$

$$b_e = S_c \Delta V + \frac{a_p}{E} \mu_e^o$$

For any guessed pressure distribution \( p^* \), the velocity \( u^* \) can be obtained from Equation
(4.11) as

$$a_e u_e^* = \sum a_{nb} u_{nb}^* + b_e + A_e (p_p^* - p_E^*).$$  \hspace{1cm} (4.12)

The $u_e^*$ obtained based on the guessed pressure field in general does not satisfy the continuity condition. Therefore, correction of the guessed pressure by $p^* = p - p_e^*$ is necessary to correct the $u_e^*$ field by $u^* = u - u_e^*$. Subtracting Equation (4.12) from Equation (4.11), the following relationship is obtained

$$a_e u_e^f = \sum a_{nb} u_{nb}^f + A_e (p_p^f - p_E^f).$$  \hspace{1cm} (4.13)

The term $\sum a_{nb} u_e^*$ is subtracted from Equation (4.13). Now we have

$$(a_e - \sum a_{nb}) u_e^f = \sum a_{nb} (u_{nb}^f - u_e^f) + A_e (p_p^f - p_E^f).$$  \hspace{1cm} (4.14)

The exact equation for $p^*$ (Equation (4.13) or (4.14)) is complicated and unsuitable for an economic calculation. In the SIMPLEC algorithm, to obtain a simplified relation the term $\sum a_{nb} (u_{nb}^f - u_e^f)$ on the right-hand side of Equation (4.14) is neglected since the terms $u_{nb}^f$ and $u_e^f$ should be of the same order of magnitude. Equation (4.14) can now be written as

$$u_e = u_e^* + d_e (p_p^f - p_E^f)$$  \hspace{1cm} (4.15)

where $$d_e = \frac{A_e}{a_e - \sum a_{nb}}.$$

The discretized continuity equation for the control volume centred at P (Figure 4.1) can
be derived from Equation (4.1) by using \( \phi = 1 \).

\[
(uA)_w - (uA)_e + (vA)_s - (vA)_n + (wA)_b - (wA)_i = S
\]

(4.16)

Introducing relations such as Equation (4.15) for \( u, v, \) and \( w \) into Equation (4.16) leads to

\[
d_p p'_p = (Ad)_w p'_w + (Ad)_e p'_e + (Ad)_s p'_s + (Ad)_n p'_n + (Ad)_b p'_b + (Ad)_i p'_i + b_p
\]

(4.17)

where \( b_p = (u^*A)_w - (u^*A)_e + (v^*A)_s - (v^*A)_n + (w^*A)_b - (w^*A)_i + S \)

\[
d_p = \sum (Ad)_{nb}
\]

Thus, the pressure correction \( p' \) can be obtained by solving Equation (4.17). Then the corrected pressure field is obtained by

\[
p = p^* + p'.
\]

(4.18)

Using an iterative procedure, the pressure and velocity fields will eventually satisfy both mass and momentum conservation equations.

4.3.2 Iterative Method for Linearized Algebraic Equations

The discretized equations (Equations (4.12), (4.17), etc.) are linearized in each iteration loop. The solution of the linearized algebraic equations can be obtained by the use of an iterative method. In this study, the line-by-line iteration method, which employs the Tri-Diagonal-Matrix Algorithm (TDMA)\(^{[33]}\) for each line is used.
4.3.3 Solution Procedure

The closed set of equations to be solved comprises:

a) the momentum equations;

b) the pressure correction equation derived from the continuity equation, and

c) the equations for other variables on which the flow may be dependent.

Since these equations are coupled and highly non-linear, an iterative approach is used for their solution, whereby a cyclic outer iteration comprises the following sequence of operations:

a) The momentum equations (such as Equation (4.12)) are solved to get \( u^*, v^*, w^* \) based on a guessed pressure field or a pressure field taken from the previous iteration.

b) The pressure correction equation (Equation (4.17)) is solved to get \( p^* \).

c) At the end of each outer iteration loop, pressure and velocity fields are corrected using Equation (4.18) and equations like (4.15), respectively.

d) Other dependent variables like air mass fraction, are obtained from their own discretized transport equations.

e) The corrected pressure and velocities are used to update the coefficients multiplying the nodal variables in the discretized equations, such as temperature of mixture and cooling water, density, mass source term, and momentum source term.

f) A new cycle is started unless the prescribed accuracy has been reached.

4.3.4 Convergence Criteria
The convergence criteria employed for the momentum equations and other transport equations require that,

$$\left[ \sum (\phi - \phi^o)^2 \right]^{1/2} / (N \times M) < 10^{-5}$$

where \( \phi \) - new values of \( u, v \) or \( \Phi \);

\( \phi^o \) - values of \( u, v \) or \( \Phi \);

\( M \) - total number of control volumes in x-direction;

\( N \) - total number of control volumes in y-direction;

The convergence criteria for the pressure equation suggested by Van Doormaal and Raithby\(^6\) requires the iteration to continue until

$$|r_p| / |r_p^o| < 0.10$$

where \( |r_p| \) - Euclidean norm of the residual of the pressure correction equation at present iteration level;

\( |r_p^o| \) - Euclidean norm of the residual of the pressure correction equation at the beginning of the iterative process;

Based on numerical tests, the convergence criteria for the overall computational procedure have been specified as:

(a) the error norm for the momentum equations is reduced to less than \( 10^{-5} \);

(b) the maximum absolute continuity error is reduced to less than 0.01% of the inflow.
Figure 4.1 Grid Arrangement
CHAPTER V
SIMULATION OF
AN EXPERIMENTAL CONDENSER

In order to evaluate the predictive capability of the present numerical procedure and the corresponding computer code, an experimental steam surface condenser from Al-Sanea et al.\textsuperscript{[14]} has been selected to verify the numerical procedure.

The configuration of the experimental condenser is depicted in figure 5.1. It is a small and simple steam surface condenser. It contains an internal vent, and the tube bank is composed of 20*20 tubes of triangular arrangement. The geometrical and operating parameters are listed in Table 1. Computational details, numerical prediction and comparison with experimental data are presented in the following sections.
Table 1 Geometrical and Operating Parameters for Test Condenser

<table>
<thead>
<tr>
<th>Geometrical Parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Condenser Length (m)</td>
<td>1.219</td>
</tr>
<tr>
<td>Tube Outer Diameter (mm)</td>
<td>25.4</td>
</tr>
<tr>
<td>Tube Inner Diameter (mm)</td>
<td>22.9</td>
</tr>
<tr>
<td>Tube Pitch (mm)</td>
<td>34.9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Operation Parameters</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Temperature of Cooling Water (°C)</td>
<td>17.8</td>
</tr>
<tr>
<td>Inlet Velocity of Cooling Water (m/s)</td>
<td>1.19</td>
</tr>
<tr>
<td>Inlet Pressure of Steam (Pa)</td>
<td>27670</td>
</tr>
<tr>
<td>Inlet Mass Flow Rate (kg/s)</td>
<td>2.032</td>
</tr>
</tbody>
</table>

5.1 Computational Details

Grid Distribution: The condenser is subdivided into four domains along the cooling water flow direction, and the calculation plane is located in the middle of each subdivided domain, perpendicular to the cooling water flow direction as shown in figure 5.1. Grids of 21*20, 31*34, 41*40 in the main and cross flow directions respectively have been used to test the grid-independence of the solution. The two finer grids give virtually identical results. Thus, a non-uniform grid of 31*34 in the main and cross flow directions is used in the subsequent calculations for each subdivided domain of the condenser as shown in
CHAPTER 5 SIMULATION OF AN EXPERIMENTAL CONDENSER 37

figure 5.2a.

**Calculation procedure:** The SIMPLEXC procedure described in Chapter 4 is employed to solve the mass, momentum and air mass fraction equations simultaneously. As shown in the flow chart (figure 5.3), an iterative approach is used for solution since the mass flow rate at the inlet of each domain to be given as a boundary condition was unknown when carrying out the simulation. Inlet mass flow rate for each domain needs to be amended to satisfy the equal pressure drop condition. The calculation is repeated until the maximum difference between the outlet pressure and the average outlet pressure is reduced to less than 0.01% of the inlet pressure.

**Convergence:** For each domain around 90 iterations are needed to reach convergence. For the global cycle, usually 2-3 iterations are needed to reach convergence.

5.2 RESULTS AND DISCUSSION

The results are shown in figures 5.4 to 5.6 in the form of computer generated plots of velocity vectors, and velocity magnitude, heat flux and air concentration contour maps. In figures 5.7 to 5.10 comparisons between predictions and measurements are shown for heat flux distribution along four of the tube rows through the tube bank. Figure 5.11 depicts the condensation rate of the four domains along the cooling water direction. Comparisons between the predictions by the proposed model and the best predictions from the two-dimensional models proposed by Al-Sanea et al.\textsuperscript{14} and Bush et al.\textsuperscript{35} are
shown in figures 5.12 to 5.15.

**Typical Results:** The details of the flow can be best appreciated by the consideration of the velocity vector plot and velocity magnitude contour map shown in figure 5.3 and 5.4. They show steam entering from the side, there is some deflection of the flow away from the tube bank and down the steam lane, then the flow is turned around due to the presence of the rear wall and the suction effect of tube bank. The velocities reduce to quite small values in the middle of the tube bank as the vapour condenses.

The contours of the heat flux are shown in figure 5.5. It is clear that the centre region has lower heat flux. The highest heat flux is at the entry to the tube bank, where steam velocities are highest and air concentrations are lowest. Inundation effects tend to degrade the value with lower regions corresponding to the presence of larger quantities of condensate. In the first and last planes, the average heat flux reach a maximum and a minimum, respectively which reflect the influence of three-dimensionality. It is interesting to note that the heat flux in the vent area is lower in the proceeding plane than that in the following plane although the average heat flux in the proceeding plane is higher. This can be explained by considering the higher air mass fraction in the vent area in the proceeding plane which is seen in figure 5.6.

The air mass fraction contour maps at different planes are given in figure 5.6. The figure shows a large air bubble, which reduces the heat transfer rate, in the vent area at each sector. The air mass fraction at the vent area is higher in the cold-end sector than that in
the hot-end sector. This is the result of the nonuniform condensing capability along the cooling water flow direction and has been inferred from experimental observations.

Comparison: Comparisons between predictions and measurements are made for heat flux distributions. Figures 5.7 to 5.10 compare the heat flux along four of the tube rows with the experimental data. Since the experimental heat flux at each plane is not available, the experimental values used here are mean values along cooling water flow direction. As can be seen from these figures there is good agreement between the experiments and predictions in the top half of the condenser. The size of the air bubble in the central area of the condenser tube bundle is overpredicted. Figure 5.11 depicts the steam condensation rate (\(\dot{M}\)) along the tube axial direction. The results can be explained by the three-dimensionality of the steam flow in the condenser. As expected the condensation rate is greater at the cooling water inlet end than that at the outlet end due to the varying cooling water temperature along the condenser. Since this is a small size condenser, the three-dimensional effect can only be slightly noticed. The predicted total condensation rate (2.020kg/s) is in excellent agreement with the experimental value (2.021 kg/s). The deviation is only 0.05%.

Heat flux along four of the tube rows predicted by the proposed model is compared with that predicted by Al-Sanea et al.\(^{[4]}\) and Bush et al.\(^{[25]}\). The comparisons are given in figure 5.12 to 5.15. As Al-Sanea and Bush models are two dimensional, data of plane No.2 from the proposed model are used in the comparison. It can be seen that the proposed model
gives better predictions in the top half of the condenser. In the bottom of the tube bundle, the results from the three models are similar. The results from Al-Sanea and Bush in some areas are better. This may be because Al-Sanea’s and Bush’s models take into account the two-phase (steam and water) effect which has strong influence in the bottom part as the condensate accumulates there.
Figure 5.2 Grid Used for the Simulation
START

Input Data

Assume Steam Flow Rate for Each Domain

I \rightarrow 1

Solve Momentum Eq.

Solve Pressure Correction Eq.

Update p, u and v

Solve Air mass Fraction Eq.

Update ρ, T and Source Terms

Converged

Yes

I = 4

Yes

No

P_{out} - P_{out,mean} < 0.01\%P_{T}

Yes

PRINT

STOP

I+1 \rightarrow I

Amend Steam Flow Rate for Each Domain

No
Figure 5.4  Velocity Distribution for Plane No.1
a. Velocity Vector   b. Contour Map of Velocity Magnitude
Figure 5.4 Velocity Distribution for Plane No.1
a. Velocity Vector  b. Contour Map of Velocity Magnitude
Figure 5.5 Contour Map of Heat Flux
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.6 Contour Map of Air Mass Fraction
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.5 Contour Map of Heat Flux
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.5 Contour Map of Heat Flux
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.6  Contour Map of Air Mass Fraction
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.5 Contour Map of Heat Flux
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.6 Contour Map of Air Mass Fraction
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.6 Contour Map of Air Mass Fraction
a. Plane No.1  b. Plane No.2  c. Plane No.3  d. Plane No.4
Figure 5.7 Comparison of Predicted Heat Flux Along 3rd Row from Bottom of Tube Bank with Experiment Data
Figure 5.8 Comparison of Predicted Heat Flux Along 8th Row from Bottom of Tube Bank with Experiment Data
Figure 5.9 Comparison of Predicted Heat Flux Along 13th Row from Bottom of Tube Bank with Experiment Data
Figure 5.10  Comparison of Predicted Heat Flux Along 18th Row from Bottom of Tube Bank with Experiment Data
Figure 5.11 Distribution of Condensation Rate
Figure 5.12 Comparison of Predicted Heat Flux Along 3rd Row from Bottom of Tube Bank with Al-Sanea’s and Bush’s Results
Figure 5.13 Comparison of Predicted Heat Flux Along 8th Row from Bottom of Tube Bank with Al-Sanea’s and Bush’s Results
Figure 5.14 Comparison of Predicted Heat Flux Along 13th Row from Bottom of Tube Bank with Al-Sanea's and Bush's Results
Figure 5.15 Comparison of Predicted Heat Flux Along 18th Row from Bottom of Tube Bank with Al-Sanea’s and Bush’s Results
CHAPTER VI

SENSITIVITY STUDY

OF AUXILIARY RELATIONS

In the shell side of the condenser, the fluid flow is highly turbulent and the heat transfer process is complicated. Therefore, whether the auxiliary equations which are used to calculate turbulent viscosity, condensation heat transfer coefficient etc. are reliable or not has a significant influence on the accuracy of the condenser simulation.

The main reason for this part of the work is the known fact that there are a few correlations of condensation heat transfer coefficient and models of turbulent viscosity available, but there is no report of sensitivity studies of these correlations and models on the prediction of steam surface condensers. Thus, in the present work, four correlations of condensation heat transfer coefficient proposed by Nusselt\(^{18}\), Fujii et al.\(^{21}\), Rose\(^{22}\) and Honda and Fujii\(^{22}\), have been studied. Also, the sensitivity of different inundation correction factors (c/w)\(^{y}\) and turbulent viscosity models have been discussed.
6.1 Calculation of Condensation Heat Transfer Coefficient

The four correlations used for the sensitivity study are listed below.

1. Modified Nusselt's Equation\textsuperscript{[19]}

\[
\alpha_c = 0.725(Gr/H)^{1/4}k/D_o[1 + 0.0095(Re_y)^{11.8}/\sqrt{Nu}]
\]  \hspace{1cm} (3.20)

where \(Gr = (D_o^3g/\mu_c^2)(\rho_c/\rho_g)\)

2. Fujii et al.'s\textsuperscript{[21]} Equation

\[
\alpha_c = K\chi[1 + 0.276/(\chi^4FrH)]^{1/4}Re_m^{1/2}k/D_o
\]  \hspace{1cm} (3.19)

where \(K = 0.8\) for in-line arrangement

\[
K = 1.0\text{ for staggered arrangement}
\]

\[
\chi = 0.9(1 + 1/raH)^{1/3}
\]

\[
Fr = M^2/\rho_gD_o
\]

\[
H = c_p(T_s - T_w)/Pr_cL
\]

\[
Re_m = \rho_gMD_c/\rho_c\mu_c
\]

\[
ra = (\rho_c\mu_c/\rho_g\mu_g)^{1/2}
\]

\[
Pr_c = c_p\mu_c/k_c
\]

\(M\) - mass velocity of steam through maximum flow area.
3. Rose's equation\textsuperscript{[22]}

\[
\alpha = \frac{\chi + 0.728FrH^{1/2}}{1 + 3.44FrH^{1/2} + FrH^{1/4}} Re^{1/2} k/D_o
\] (3.21)

4. Honda and Fujii's Equation\textsuperscript{[23]}

\[
\alpha = [1 - 0.27 \frac{(5FrH)^{1/2} - 1}{(5FrH)^{1/2} + 1}] (FrH)^{1/4} Re^{1/2} k/D_o
\] (3.22)

The calculations are carried out using the above four correlations with the eddy viscosity model and inundation correction factor (c/w)\textsuperscript{0.223}. Figures 6.1 to 6.4 show the comparison of predictions of the heat flux along four tube rows by the four correlations with experimental data. As the experimental heat values are mean values along the cooling water flow direction, the predicted data are taken from plane No. 2. It is interesting to note that the predictions obtained by use of Fujii et al.'s, Rose's, Honda and Fujii's equations, have similar tendency. The predictions obtained by Fujii et al.'s and Rose's equations have only little quantitative difference. However the predictions obtained by the modified Nusselt equation is quite different from the others. This might be because the modified Nusselt equation does not fully reflect the influence of the flow field.

The predicted total condensation rate by use of the above four correlations are compared in Table 6.1.
Table 6.1 Total Condensation Rates by Using Different Condensation Heat Transfer Coefficient Correlations

<table>
<thead>
<tr>
<th>Scheme</th>
<th>Predicted Result</th>
<th>Error ( %)*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental data</td>
<td>2.021 (kg/s)</td>
<td></td>
</tr>
<tr>
<td>Modified Nusselt</td>
<td>1.947 (kg/s)</td>
<td>-3.66</td>
</tr>
<tr>
<td>Fujii et al.</td>
<td>2.020 (kg/s)</td>
<td>-0.05</td>
</tr>
<tr>
<td>Rose</td>
<td>2.015 (kg/s)</td>
<td>-0.30</td>
</tr>
<tr>
<td>Honda and Fujii</td>
<td>1.972 (kg/s)</td>
<td>-2.42</td>
</tr>
</tbody>
</table>

* Error=(Prediction-Experimental Data)/Experimental Data*100

It seems that Fujii et al.'s equation gives the best prediction of the total condensation rate even though it underpredicts heat flux at the periphery of the tube bank.

As mentioned before, the inundation correction factor \((c/w)\)' is used to take the inundation effect into account. The suggested value of \(s\) is from 0.07 to 0.223. However, there is no known study about the effect of \(s\). Here, \(s=0.07\), \(s=0.153\) and \(s=0.223\) have been used respectively in the simulation of the experimental condenser where the Fujii et al.'s equation is used to calculate condensation heat transfer coefficient, and the eddy viscosity model is employed for turbulent viscosity.
The results are shown in Figures 6.5 to 6.8 (the predicted data are taken from plane No. 2). It can be seen that the smaller $s$ is, the closer the predictions are to measurements at the periphery of the tube bank. Oppositely, the bigger the $s$ is, the closer are the predictions obtained in the rest the tube bank. The comparison of total condensation rates obtained by using different values of $s$ is shown below.

<table>
<thead>
<tr>
<th>$s$</th>
<th>Predicted Result</th>
<th>Error ( %)*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental data</td>
<td>2.021 (kg/s)</td>
<td></td>
</tr>
<tr>
<td>$s = 0.070$</td>
<td>2.031 (kg/s)</td>
<td>0.49</td>
</tr>
<tr>
<td>$s = 0.153$</td>
<td>2.024 (kg/s)</td>
<td>0.15</td>
</tr>
<tr>
<td>$s = 0.223$</td>
<td>2.020 (kg/s)</td>
<td>-0.05</td>
</tr>
</tbody>
</table>

* Error=$\frac{\text{Prediction-Experimental Data}}{\text{Experimental Data}}\times 100$

When $s=0.223$ the predicted total condensation rate is the closest one to experimental data.

In general, it can be seen that the predicted condensation rate increases as the $s$ decreases. The reason is for this that when $s$ decreases, the inundation effects become small, so the
condensation rate increases.

6.2 Turbulent Viscosity

In the proposed model, it is assumed that the effective viscosity is equal to the sum of the turbulent viscosity and molecular viscosity, and Sc=1 which means turbulent diffusivity is equal to turbulent viscosity. Therefore turbulent viscosity is a critical parameter through the whole process of simulation.

Calculations\cite{14,16} have been carried out with fixed values of $\mu$, in the range of 30 times to 120 times the molecular viscosity and with values varying locally based on the eddy viscosity model. In this work, $\mu=40\mu$, $\mu=120\mu$ and the eddy viscosity model have been used in the experimental condenser simulation, while the condensation heat transfer coefficient is calculated by Fujii's equation with $s=0.223$.

From Figures 6.9 to 6.12, it can be seen that when $\mu=120\mu$, the heat flux at the periphery of the tube bank is overpredicted, the predicted heat flux distribution is quite different from the measurement; when $\mu=40\mu$, the heat flux is underpredicted. The predicted heat flux based on the eddy viscosity model agrees better with the measurement. The total condensation rate obtained by use of the three models are shown in Table 6.3.
Table 6.3 Total Condensation Rates by Using Different Viscosity Modelling

<table>
<thead>
<tr>
<th>Turbulent Viscosity</th>
<th>Predicted Result</th>
<th>Error ( %)*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental data</td>
<td>2.021 (kg/s)</td>
<td></td>
</tr>
<tr>
<td>$\mu_t = 40\mu$</td>
<td>2.009 (kg/s)</td>
<td>-0.60</td>
</tr>
<tr>
<td>$\mu_t = 120\mu$</td>
<td>2.032 (kg/s)</td>
<td>0.54</td>
</tr>
<tr>
<td>Eddy viscosity model</td>
<td>2.020 (kg/s)</td>
<td>-0.05</td>
</tr>
</tbody>
</table>

* Error=$(Prediction-Experimental Data)/Experimental Data

From Table 6.3, it is clear that when $\mu_t=40\mu$, the condensation rate is underestimated, and when $\mu_t=120\mu$, it is overestimated. Based on the comparison of heat flux and total condensation rate, the eddy viscosity model gives the best result.
Figure 6.1 Comparison of Predicted Heat Flux along 3rd Row by Different Correlations of Condensation Heat Transfer Coefficient
Figure 6.2 Comparison of Predicted Heat Flux along 8th Row by Different Correlations of Condensation Heat Transfer Coefficient
Figure 6.3 Comparison of Predicted Heat Flux along 13th Row by Different Correlations of Condensation Heat Transfer Coefficient
Figure 6.4 Comparison of Predicted Heat Flux along 18th Row by Different Correlations of Condensation Heat Transfer Coefficient
Figure 6.5 Comparison of Predicted Heat Flux along 3rd Row by Different Inundation Correction Factors
Figure 6.6 Comparison of Predicted Heat Flux along 8th Row by Different Inundation Correction Factors
Figure 6.7 Comparison of Predicted Heat Flux along 13th Row by Different Inundation Correction Factors
Figure 6.8 Comparison of Predicted Heat Flux along 18th Row by Different Inundation Correction Factors
Figure 6.9 Comparison of Predicted Heat Flux along 3rd Row by Different Turbulent Viscosity Models
Figure 6.10  Comparison of Predicted Heat Flux along 8th Row by Different Turbulent Viscosity Models
Figure 6.11 Comparison of Predicted Heat Flux along 13th Row by Different Turbulent Viscosity Models
Figure 6.12 Comparison of Predicted Heat Flux along 18th Row by Different Turbulent Viscosity Models
7.1 Contributions

A numerical algorithm has been developed to simulate shell-side flow and heat transfer in steam surface condensers. The flow is assumed to be quasi-three-dimensional and incompressible, however, density is allowed to vary with temperature and air concentration. The numerical method proposed in the present study has shown the capability of predicting the performance of condensers including three-dimensional effects due to the increase of cooling water temperature, and the existence of air blanket and inundation. The approach, strictly speaking, is only valid for those condensers with partition plates in which the steam flows normally to the cooling water tube bundle.
Calculations have been carried out for an experimental condenser. The results have successfully reproduced the main features of the flow inside the steam condensers. The comparison of the predictions against experimental data indicates that the total condensation rate is predicted very accurately and the predicted heat fluxes are in good qualitative agreement with the measured values in most regions of the tube bundle. When consideration is given to the limitations of the computational procedure, the predictions are very encouraging.

Sensitivity of condensation heat transfer coefficient correlations and inundation factors on the accuracy of the predictions has been studied. Results obtained by using Fujii et al. and Rose's correlation with inundation factor \((w/c)^{0.223}\) agree well with experimental data.

Several commonly used turbulent viscosity models have been discussed. The prediction by use of the eddy viscosity model is the closest one to experimental data.

### 7.2 Future Work

Extensive experimental data, including fluid and temperature field, and flow visualization are needed. This information will prompt future improvements of the model, particularly in what concerns the constitutive relations. This particular aspect is of primary importance, since the flooding and condensation condition will dictate the relations for the heat transfer and pressure loss parameters to be employed.
The flow in a steam condenser is a two-phase one, having a steam/air mixture and condensed water. Further the liquid phase takes several forms and affects both fluid flow and heat transfer. Therefore, a more complex model including both liquid and gas phase needs to be studied.
REFERENCES


NAME: Ying Zhang 
SEX: Female

1966 born in Chongqing, Sichuan, P.R.China

1980-1982 Chongqing No.1 High School
            Chongqing, Sichuan, P.R.China

1982-1986 Shanghai Jiao Tong University
            Shanghai, P.R.China
            B.Sc.Eng. in Power Machinery Eng.

1986-1989 Chongqing University
            Chongqing, Sichuan, P.R.China

1990-1991 Denmark Technology University
            Copenhagen, Denmark
            Guest Researcher

1991-1992 University of Windsor
            Windsor, Ontario, Canada
            M.Sc. Candidate in Mechanical Eng.