NUMERICAL STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH PROTRUDING AND FLUSH HEAT SOURCES

BY

SUSHANTA KUMAR SAHA

A Thesis Submitted to the Department of Mechanical Engineering in Partial Fulfilment of the Requirements for the Degree of

Master of Science

in

Mechanical Engineering

DECEMBER 1994

BANGLADESH UNIVERSITY OF ENGINEERING AND TECHNOLOGY
DHAKA-1000, BANGLADESH
NUMERICAL STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH PROTRUDING AND FLUSH HEAT SOURCES

A THESIS
BY
SUSHANTA KUMAR SAHA

Approved as to style and content by:

Dr. J.A. Naser (Supervisor)
Assistant Professor
Department of Mechanical Engineering
BUET, Dhaka-1000

Chairman

Dr. Md. Abdur Razzaq Akhanda
Professor and Head
Department of Mechanical Engineering
BUET, Dhaka-1000

Member (Ex-Officio)

Dr. A. K. M. Sadru Islam
Professor
Department of Mechanical Engineering
BUET, Dhaka-1000

Member

Professor Dr. M. Anwar Hossain
Department of Mechanical & Chemical Engineering
Islamic Institute of Technology
Dhaka.

Member (External)
CERTIFICATE OF RESEARCH

This is to certify that the work presented in this thesis is the outcome of the investigations carried out by the candidate under the supervision of Dr. J. A. Naser in the Department of Mechanical Engineering of Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh.

[Signatures]
Supervisor
Candidate
ABSTRACT

Numerical investigations have been performed on an incompressible fluid contained in a two dimensional enclosure. Five identical heat sources were mounted on one vertical wall of the enclosure with uniform vertical spacing. Two different types of heat sources i.e. protruding and flush were studied. The vertical wall opposite to the heated sections and the bottom wall were adiabatic. The top surface of the test enclosure was an isothermal heat sink. The horizontal and vertical dimensions of the protruding heaters were $L_2 = 9 \text{ mm}$ and $L_1 = 15 \text{ mm}$ respectively. The vertical spacing between the heaters was $L_2 = 15 \text{ mm}$. For flush heater $L_3 = 0 \text{ mm}$. Investigations were performed with different values of cavity width, resulting in the variation of aspect ratio (height-to-width ratio) and cavity width-to-protruding heater height ratios of 3.67 to 9.17 and 2.0 to 5.0, respectively. The effect of variation of heat flux from the heat sources, resulting in variation of modified Rayleigh numbers ranging from $1.9 \times 10^6$ to $4.9 \times 10^{10}$ was studied. The investigations were performed with ethylene glycol as the working fluid.

The governing transport equations for turbulent flow under investigation are closed using the two equation model of turbulence ($k$-$\varepsilon$), which includes gravity-density interaction. The equations are discretised using a finite volume technique. A non-uniform grid arrangement of 62 X 62 was used. The governing equations were solved incorporating the Pressure Implicit with Splitting of Operators (PISO) method.

Results indicate that heat transfer and fluid motion within the enclosure are influenced by the Rayleigh number, aspect ratio and heat input. At larger width ($W = 45 & 36 \text{ mm}$) a single vortex is observed occupying the flow domain and at smaller width ($W = 22.5 & 18 \text{ mm}$) the single vortex structure disintegrates and several secondary flow cells appear within it. Increase in the width results in a decrease in the difference between the temperatures of the top heated sections. The temperature gradient in the core flow region are found to be considerably small. The correlation of local Nusselt number versus local modified Rayleigh number is independent of the number of heaters in the vertical array and vertical height location of the heaters. The results show reasonably good agreement with the experimental data.
ACKNOWLEDGEMENTS

The author is highly grateful and indebted to his Supervisor, Dr. J. A. Naser, Assistant Professor, Department of Mechanical Engineering, BUET, Dhaka for his careful supervision, constant inspiration, unceasing encouragement, invaluable suggestions and untiring support throughout the research work.

The author gratefully acknowledges the assistance received from Mr. Raisuddin Khan, Ph.D. student, Department of Mechanical Engineering, BUET, Dhaka during running his computer programme.

Lastly, the author would like to express his sincere thanks to all others, who directly or indirectly have helped him in completing this work properly.
TO MY PARENTS
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Certificate of Approval</td>
<td>i</td>
</tr>
<tr>
<td>Certificate of Research</td>
<td>ii</td>
</tr>
<tr>
<td>Abstract</td>
<td>iii</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>iv</td>
</tr>
<tr>
<td>Dedication</td>
<td>v</td>
</tr>
<tr>
<td>Table of Contents</td>
<td>vi</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>ix</td>
</tr>
<tr>
<td>List of Figures</td>
<td></td>
</tr>
</tbody>
</table>

## CHAPTER - I  INTRODUCTION

1.1 General

1.2 Natural convection in an Enclosure

1.3 Motivation Behind the Selection of the Problem

1.4 Importance of Numerical Investigation

1.5 The Problems and Objectives

## CHAPTER - II  LITERATURE REVIEW

2.1 General

2.2 Natural Convection in an Enclosure

2.2.1 Natural Convection Heat Transfer in Rectangular Enclosure with Vertical Walls at Constant Temperature

2.2.2 Natural Convection Heat Transfer in Rectangular Enclosure with Discrete Heat Sources Mounted on One Vertical Wall
CHAPTER -III       GOVERNING DIFFERENTIAL EQUATIONS

3.1 The Form of Governing Differential Equations Adopted for use in this Study
3.2 Expanded Form of Governing Differential Equations
3.3 The Method of Discretization
3.4 Differencing Schemes used in this Study
   3.4.1 Central Differencing Schemes (CDS)
   3.4.2 Upwind Differencing Schemes (UDS)
3.5 Solution Procedure
   3.5.1 Grid and Variable Arrangement
   3.5.2 Calculation of Pressure
3.6 Solution Algorithm

CHAPTER -IV       RESULTS AND DISCUSSIONS

4.1 Introduction
4.2 Solution Domain and Computational Grid and Boundary Conditions
4.3 Presentation and Discussion of Results and Comparison with Experiment

CHAPTER -V       SUMMARY AND CONCLUSIONS

5.1 General
5.2 Summary of Main Findings and Achievements
5.3 Suggestions for Future Work

REFERENCES

FIGURES

APPENDIX - A
APPENDIX - B
APPENDIX - C
### NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Aspect ratio = $H/W$</td>
</tr>
<tr>
<td>$A_h$</td>
<td>Surface area of the heated section, $m^2$</td>
</tr>
<tr>
<td>$A_T$</td>
<td>Area of top surface, $m^2$</td>
</tr>
<tr>
<td>C</td>
<td>Isobaric specific heat, $J/kg\cdot K$</td>
</tr>
<tr>
<td>g</td>
<td>Acceleration due to gravity, $m/s^2$</td>
</tr>
<tr>
<td>h</td>
<td>Local heat transfer coefficient = $q/(T_h-T_c)$, $W/m^2\cdot K$</td>
</tr>
<tr>
<td>$h_T$</td>
<td>Heat transfer coefficient at the top sink surface</td>
</tr>
<tr>
<td>=</td>
<td>$q_T/(T_h-T_c)$, $W/m^2\cdot K$</td>
</tr>
<tr>
<td>H</td>
<td>Height of the enclosure, $m$</td>
</tr>
<tr>
<td>K</td>
<td>Thermal conductivity, $W/m\cdot K$</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$L_1$</td>
<td>Height of the heated section, $m$</td>
</tr>
<tr>
<td>$L_2$</td>
<td>Height of the unheated section, $m$</td>
</tr>
<tr>
<td>$L_3$</td>
<td>Protruding thickness, $m$</td>
</tr>
<tr>
<td>N</td>
<td>Heater row number (starting from the bottom)</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number = $h$ [length scale]/$K$</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number = $v/\alpha$</td>
</tr>
<tr>
<td>q</td>
<td>Convective heat flux per heated section = $Q/A_h$, $W/m^2$</td>
</tr>
<tr>
<td>$q_T$</td>
<td>Convective heat flux at the top surface = $Q_T/A_T$, $W/m^2$</td>
</tr>
<tr>
<td>Q</td>
<td>Power input per heated section, $W/m$</td>
</tr>
<tr>
<td>$Q_T$</td>
<td>Total power input, $W/m$</td>
</tr>
</tbody>
</table>
Subscripts

c  =  Cold wall

h  =  Heated section

$L_1$  =  Based on the heated section height

y  =  Based on the local height, measured from the bottom of the cavity to the mid-height of a heated section.

ij  =  Tensor subscript: terms in which a subscripts appears twice are summed overall value (1 & 2) of that subscript.

Supercritps

(')  =  Average
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure No</th>
<th>Name</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig. 1.0</td>
<td>Schematic diagram of the enclosure.</td>
<td>36</td>
</tr>
<tr>
<td>Fig. 2.0</td>
<td>Computational grid of the enclosure.</td>
<td>37</td>
</tr>
<tr>
<td>Fig. 3.1</td>
<td>Two dimensional computational cell.</td>
<td>38</td>
</tr>
<tr>
<td>Fig. 3.2</td>
<td>Control volume for U.</td>
<td>39</td>
</tr>
<tr>
<td>Fig. 3.3</td>
<td>Control volume for V.</td>
<td>39</td>
</tr>
<tr>
<td>Fig. 3.4</td>
<td>Computational grid, location and control volume (cells) of scalar variables and axial and radial velocities.</td>
<td>40</td>
</tr>
<tr>
<td>Fig. 3.5</td>
<td>Schematic presentation of CDS.</td>
<td>41</td>
</tr>
<tr>
<td>Fig. 3.6</td>
<td>Schematic presentation of UDS.</td>
<td>41</td>
</tr>
<tr>
<td>Fig. 4.1</td>
<td>Streamlines for protruding heater case at $Q = 85.7$ watts/m.</td>
<td>42</td>
</tr>
<tr>
<td>Fig. 4.2</td>
<td>Streamlines for protruding heater case at $Q = 14.3$ watts/m.</td>
<td>42</td>
</tr>
<tr>
<td>Fig. 4.3</td>
<td>Velocity profile for protruding heater case at $Q = 85.7$ watts/m.</td>
<td>43</td>
</tr>
<tr>
<td>Fig. 5.1</td>
<td>Streamlines for flush heater case at $Q = 85.7$ watts/m.</td>
<td>44</td>
</tr>
<tr>
<td>Fig. 5.2</td>
<td>Streamlines for flush heater case at $Q = 14.3$ watts/m.</td>
<td>44</td>
</tr>
<tr>
<td>Fig. 5.3</td>
<td>Velocity profile for flush heater case at $Q = 85.7$ watts/m.</td>
<td>45</td>
</tr>
<tr>
<td>Fig. 6.1</td>
<td>Turbulence intensity for protruding heater case at $Q = 85.7$ watts/m.</td>
<td>46</td>
</tr>
<tr>
<td>Fig. 6.2</td>
<td>Turbulence intensity for protruding heater case at $Q = 14.3$ watts/m.</td>
<td>46</td>
</tr>
<tr>
<td>Fig. 7.0</td>
<td>Turbulence intensity for flush heater case at $Q = 85.7$ watts/m.</td>
<td>47</td>
</tr>
<tr>
<td>Fig. 8.1</td>
<td>Turbulence length scale for protruding heater case at $Q = 85.7$ watts/m.</td>
<td>48</td>
</tr>
<tr>
<td>Fig. 8.2</td>
<td>Turbulence length scale for protruding heater case at $Q = 14.3$ watts/m.</td>
<td>48</td>
</tr>
<tr>
<td>Fig. 9.0</td>
<td>Turbulence length scale for flush heater case at $Q = 85.7$ watts/m.</td>
<td>49</td>
</tr>
</tbody>
</table>
Fig. 10.1 The temperature distribution for protruding heater case at W = 45 mm.

Fig. 10.2 Comparison of the temperature distributions for protruding heater case at W = 36 mm.

Fig. 10.3 Comparison of the temperature distributions for protruding heater case at W = 22.5 mm.

Fig. 10.4 Comparison of the temperature distributions for protruding heater case at W = 18 mm.

Fig. 11.1 The temperature distribution for flush heater case at W = 45 mm.

Fig. 11.2 Comparison of the temperature distributions for flush heater case at W = 36 mm.

Fig. 11.3 Comparison of the temperature distributions for flush heater case at W = 22.5 mm.

Fig. 11.4 The temperature distribution for flush heater case at W = 18 mm.

Fig. 12.0 The cavity width effect on heated section temperatures at various power inputs for protruding heater case.

Fig. 13.1 Heated-section Nusselt numbers (Nu_L1) as a function of Ra^{*}_{L1} for protruding heater case.

Fig. 13.2 Local Nusselt number (Nu_y) as a function of local modified Rayleigh number (Ra^{*}_{y}) for all cavity width for protruding heater case.

Fig. 14.0 Heat transfer co-efficient for protruding-heater case (based on the top cooling area) versus the heat flux based at the top cooling surface.

Fig. 15.0 Aspect ratio (H/W) effect on heated section Nusselt numbers (Nu_L1) for protruding heater case.

Appendix A Overall structure of the Mathematical Foundation.

Appendix B Overall structure of the Computer Programme.

Appendix C Properties of Ethylene Glycol.
CHAPTER - 1
INTRODUCTION

1.1 General

Motion in free convection is due to buoyancy forces within the fluid, while in forced
convection it is externally imposed. Buoyancy is due to the combined presence of a fluid
density gradient and a body force. The body force is usually gravitational. There are several
ways in which a mass density gradient may arise in a fluid but in the most common situations it
is due to the presence of imposed heat flux. The density of gases and liquids depend on
temperature, generally decreasing due to fluid expansion with increasing temperature.

Free convection currents transfer internal energy stored in fluid elements in the same manner as
forced convection currents. However the intensity of mixing is generally less in free convection
and consequently the heat transfer coefficient in free convection are lower than those in forced
convection.

The temperature distribution in natural convection depends on the intensity of the fluid currents
which is dependent on the imposed heat flux. The qualitative and quantitative analysis of
natural convection heat transfer are quite difficult. The numerical analysis of natural convection
heat transfer for most of the practical situations are rarely available in the literature. So, at this
present stage the necessity of the numerical investigation is seriously felt.

1.2 Natural Convection in an Enclosure

Natural convection in rectangular two dimensional enclosure has become a standard problem in
numerical heat transfer due to its relavance in understanding a number of problems in
engineering. With the increase of computer power the heat dissipation from the electronic
components are increasing rapidly. Effective cooling of microelectronic components by means
of natural convection has recently been accepted as a viable alternative to force cooling in some
circumstances. Cooling by natural convection provides simple, low cost, reliable, maintenance-
free and electronic-interface-free cooling (Johnson[1986]). The prediction of flow and heat
transfer in ventilated or air conditioned rooms are also examples of important uses.

Some numerical works on natural convection heat transfer in an enclosure have been done with five protruding and flush heat sources on one vertical position. Keyhani et. al. [1988b] numerically studied in a tall cavity with three flush heated sections. Kelleher et al. [1987] and Lee et al [1987] numerically studied in an enclosure with one small heater protruding from a vertical wall. Liu et al [1987] numerically studied three-dimensional, 3x3 array of protruding chips mounted on one vertical wall of the enclosure. Chu et al [1976] numerically studied the effects of heater size, location and aspect ratio on laminar natural convection for the case of a heated strip located on one side wall. Kuhn And Oosthuizen [1986] conducted numerical study in a three dimensional rectangular cavity with adiabatic horizontal and vertical walls.

But no numerical work was reported on natural convection heat transfer in an enclosure with five protruding and flush heat sources. Hence through the present investigation an effort has been made to fill up the existing gap in the information on natural convection heat transfer.

The purpose of this work is to numerically simulate the experimental results of Keyhani et al [1991] and Carmona & Keyhani [1989]. The numerical procedure developed for this purpose solves the nonlinear coupled differential elliptic equations closed with the two equation (k-ε) turbulence model. Performance of the numerical procedure for solving the governing equations, without making any assumption or estimation concerning the core flow are presented. The equations are solved for the primitive variables.
1.3 Motivation Behind the Selection of the Problem

The study of natural convection effects is important in numerous engineering applications: in designing nuclear reactors, solar collectors, electrical and microelectronic equipments containers and in many other design problems, where natural convection heat transfer is prominent.

Most of the reported numerical work for this problem has focussed upon the low Rayleigh number (Ra) laminar regime, despite the fact that turbulent conditions prevail in the majority of engineering situations. Recently however, more attention has been given to higher Ra, laminar and turbulent natural convection.

1.4 Importance of Numerical Investigation

In the recent past, the emergence of faster digital computers together with the development of more versatile and efficient numerical solution method has led to a substantial increase in the assembly of mathematical modelling of natural convection heat transfer. In the field of engineering design of flow related technology, designers are looking for computational investigations to seek the optimum design, as experiments with either model or full scale prototype are generally laborious, expensive and time dependent.

The power of prediction enables us to operate existing equipment more safely and efficiently. Prediction of the relevant process helps us in forecasting and even controlling potential danger. These predictions offer economic benefits and contribute to human well being.

1.5 The Problems and Objectives

In the present study a detailed computational investigation on the natural convection in an enclosure with protruding and flush heat sources with ethylene glycol as the working fluid will be carried out. The geometry and dimensions of the protruding and flush heat sources are shown in figures. This geometries are based in the experimental studies of Keyhani et al (1991) and Carmona & Keyhani (1989).
The Present Contribution

The specific contribution of this study are the following: Numerical prediction of

1. Velocity distribution in the flow field.
2. Turbulent intensity in the flow field.
3. Turbulent length scale in the flow field.
4. Temperature distribution in the flow field.
5. Natural convection heat transfer coefficients.
6. The dependence of the above parameters on aspect ratio.
7. The dependence of the above parameters on the imposed heat flux.
8. Assessment of the quality of numerical prediction through comparison with the available experimental data.
CHAPTER - 2

LITERATURE REVIEW

2.1 General

Free convection currents occur in a fluid enclosed in a rectangular enclosure if the temperature of the one plate (either vertical or horizontal) is higher than that of the other one. For fluids whose density decreases with the increasing temperature, an unstable state, named as 'top heavy' is observed at a very small Rayleigh number (Ra). In this state, the fluid is completely stationary and heat is transferred across the fluid layer by conduction mechanism. This unstable state breaks down at a certain critical value of Rayleigh number ($R_{ac}$). This value of $R_{ac}$ depends on the boundary conditions.

At $R_{ac}$, the stable state becomes marginally unstable as any disturbance in the fluid. This flow state is known as postconductive state. It is laminar and has nearly a hexagonal cell structure. The flow moves upward in the interior of these cells and returns downward along its rim.

At $Ra >> R_{ac}$ the flow becomes convective with the breaking down of laminar flow into turbulent flow. The resulting flow begins to take the boundary layer structure in which the resistance to heat transfer depend exclusively in two boundary layers one on each of the boundary surfaces.

Most of the reported numerical work for this problem has focussed upon the low $Ra$ laminar regime, despite the fact that turbulent conditions prevail in the majority of engineering situations. Recently, however, more attention has been given to higher $Ra$ laminar and turbulent natural convection.

2.2 Natural Convection in Enclosure

Buoyancy induced flows are complex because of the essential coupling between the flow and transport. The problems can be classified as either external ones (free convection) or internal ones (natural convection). The first unified and comprehensive review of this subject was made by Ostrach [1964]. Later summaries of free convection were presented by Ede [1967] and Gebhart [1971] and other reviews of natural convection were complied by Ostrach [1972], Catton [1982a], and Hoogendoorn [1986]. Each of the last three emphasize essentially different aspects of the subject.
It was first pointed out by Ostrach [1968] that internal problems are considerably more complex than external ones. This is because at large Rayleigh numbers classical boundary-layer theory yields the same simplifications for external problems that are so helpful in other fluid-flow problems, viz., the region exterior to the boundary layer is unaffected by the boundary layer. For confined natural convection, on the other hand, boundary layers form near the walls but the region exterior to them is enclosed by the boundary layers and forms a core region. Because the core is partially or fully encircled by the boundary layers, the core flow is not readily determined from the boundary conditions but depends on the boundary layer, which, in turn, is influenced by the core. The interaction between the boundary layer and core constitute a central problem that has remained unsolved and is inherent to all confined convection configurations, namely, that the flow pattern cannot be predicted a priori from the given boundary conditions and geometry. In fact, the situation is even more intricate because it often appears that more than one global core flow is possible and flow subregions, such as cells and layers, may be imbedded in the core. This matter, which has been discussed more fully by Ostrach [1972,1982a], and Ostrach and Hantman [1981], is not merely a subtlety for analysis, but has equal significance for numerical and experimental studies, as will be indicated later.

It is distressing to note that this crucial aspect of natural convection seems to be essentially ignored in most existing literature or is treated in a cavalier manner. The core flow is often merely assumed, estimated in an ad hoc manner, or specified from seemingly similar problems. However, experience has shown that natural convection is extremely sensitive to changes in the container configuration and the imposed boundary conditions so that use of results from "similar" problems is dangerous. In numerical studies, the entire matter generally is ignored and, as a consequence, there have been very few reliable predictive results of velocity and temperature distributions. Until the problem is resolved, numerical studies must be guided and closely coupled with experiments.

To add to all the complexity it should be recalled (Ostrach,[1964]) that there are essentially two basic modes of flow generated by buoyancy. The first, usually referred to as conventional convection, occurs whenever a density gradient (due to thermal and/or concentration effects) is normal to the gravity vector. In such case a flow ensues immediately. The second mode, called unstable convection, occurs when the density gradient is parallel but opposite to the gravity vector. In this situation the fluid remains in a state of unstable equilibrium (due to heavier fluid being above lighter) until a critical density gradient is exceeded. A spontaneous flow then results that eventually becomes steady and celluer like. If the density gradient is parallel but in the same direction as gravity the fluid is stably stratified. As if all this were not sufficiently difficult to deal with, both conventional and unstable (or stratified) modes can interact in a given configuration.
2.2.1 Natural Convection Heat Transfer In Rectangular Enclosure With Vertical Walls Held At Constant Temperature

Most of the researches in heat transfer in confined spaces have been done with the parallel plates in a vertical position. A considerable number of experiments have been conducted on air, on water and on oil and the results have been reported by investigators.

A vertical rectangular cavity is defined conventionally as an enclosure bounded by two vertical surfaces held at different temperatures. The other two parallel surfaces are often taken as insulated or with their temperature varying linearly between those of the two vertical surfaces. As predicted by Batchelor [1954] the experimental results showed that below a certain Rayleigh number and above a certain height-to-width ratio, heat was transferred from the hot to cold wall by conduction in the central part of the layer, with convection effects restricted to the corner region.

Elder [1965a] used paraffin and silicon oils which have Prandtl number around 1000 and varied the aspect ratio (height-to-width) from 1 to 60. Rayleigh numbers were up to about $10^8$. At $Ra$ less than about 1000, a weak, steady, unicellular circulation was observed, with fluid rising near the hot wall and descending adjacent to the cold one. As $Ra$ was increased further, Elder [1965a] observed secondary and tertiary flows. Secondary flows were found to arise for values around $Ra = 3 \times 10^5$ and were very weak initially. At larger $Ra$, more cells appeared. For $Ra > 10^6$, tertiary flows arise. Turbulence arose around $Ra = 10^9$, at about half-height in the cavity, and then spread to the ends at higher $Ra$.

Among the earliest measurements were those by Griffiths and Davis [1922], who measured velocity profiles in the flow adjacent to a flat vertical surface. Saunders (1939) investigated natural convection in water and mercury and gave the following heat transfer correlation for $Ra > 10^{10}$:

$$Nu = 0.17Ra^{1/3}$$

The result of turbulent heat transfer adjacent to an isothermal surface obtained by Warner and Arpaci [1968] for air show good agreement with the above correlation. On the other hand, similar experiments by Cheesewright [1968] for air appear to support Equation

$$Nu_x = 0.0246 \frac{Gr^{2/5}}{(Pr)^{7/5}} \left[ 1 + 0.494(Pr)^{2/3} \right]^{-2/5}$$
For uniform surface heat flux conditions, the results of Vliet and Liu (1969) for water, $3.6 < Pr < 10.5$, indicate the following correlation:

$$Nu = 0.568(GrPr)^{0.22} \quad \text{for } 10^{13} < GrPr < 10^{16}$$

The result of the experimental study, also in water, by Qureshi and Gebhart [1978] show good agreement with the above correlation. From data for air, Vliet and Ross [1975] gave the following relation:

$$Nu = 0.17(GrPr)^{0.25}$$

For the whole laminar and turbulent range, Churchill and Chu [1975] recommended Equations for isothermal

$$Nu^{1/2} = 0.825 + \frac{0.387Ra^{\nu/6}}{[1 + (0.492 / Pr)^{9/16}]^{8/27}}$$

and uniform heat flux surfaces,

$$Nu^{1/2} = \left(\frac{hL}{K}\right)^{1/2} = 0.825 + \frac{0.387(GrPr)^{1/6}}{[1 + (0.437 / Pr)^{9/16}]^{8/27}}$$

respectively. This two correlations are said to apply to all Pr.

Sofir Uddin et al. [1988] numerically investigated the natural convection heat transfer and fluid flow behavior for vertical walls with sinusoidal corrugation. The results showed that for the three sinusoidal corrugation with different Grashof numbers, the total heat flux becomes lower than that for plane wall, where as with corresponding Grashof numbers and for single corrugation the total heat flux becomes higher than that for straight.

Mohammad Ali [1990] numerically investigated the natural convection heat transfer and fluid flow behavior for vertical V-corrugated walls by the control volume based finite element method. The results showed that the V-corrugation with different Grashof number (Gr) the total heat flux become higher than that for straight walls.

Numerical study of transient and steady laminar buoyancy driven flows and heat transfer in a square open cavity was carried out by D. Angirasa et al. [1992]. At lower values of Gr, conduction plays a significant role in heat transfer. With increasing Gr, heat transfer is
dominated by convection and there is corresponding development of thermal boundary layers. At low Ra the heat is transferred predominately by conduction and convection picks up as Rayleigh number (Ra) is increased. If Nusselt number (Nu) is expressed as

\[ \text{Nu} = C [\text{Ra}]^n \]

where \( n = 0.296 \) \( 10^5 < \text{Ra} < 10^7 \)

For isothermal vertical flat plates, the value of \( n \) is around 0.25.

### 2.2.2 Natural Convection Heat Transfer In Rectangular Enclosure With Discrete Heat Sources Mounted On One Vertical Wall

Keyhani et al. (1988a) performed experiments on natural convection heat transfer in a tall vertical cavity with eleven alternately heated and unheated, flush mounted sections of equal height on a vertical wall. Another paper by Keyhani et al. (1988b) extended the previous work with an experimental and a numerical study of the problem in an enclosure having an aspect ratio of 4.5 and flush three heated sections. Kuhn and Ooshuizen (1986) conducted a numerical study of three dimensional, transient, natural convective flow in a rectangular enclosure with localized heating. They pointed out that the three dimensional flow increased the local heat transfer coefficient at the edge of the element and cause the average Nusselt number to be higher than that in the corresponding two dimensional flow.

Kelleher et al. (1987) and Lee et al. (1987) presented experimental and numerical results respectively for natural convection in a water-filled, rectangular enclosure with one small heater protruding from a vertical wall and the top and bottom walls were isothermal heat sink. Experiments were conducted with the heater at three different locations: near the top, in the middle, and near the bottom of the wall. Their results indicated that for a given Rayleigh number, the local Nusselt number decreased as the heater was raised in the cavity. Liu et al. (1987) numerically studied three-dimensional, convective cooling of a 3 x 3 array of protruding chips mounted on one vertical wall of an enclosure. They reported that the temperature and flow characteristics in the enclosure with a width of \( W = 30 \) mm were similar to those for \( W = 18 \) mm, except that the maximum chip surface temperature was reduced by 4°C. They also indicated that boundary layer and stratification behavior were valid for \( W/L_3 > 3.0 \). Joshi et. al. (1990) conducted an experimental study of natural convection cooling of a 3 x 3 array of heated protrusions in a rectangular enclosure filled with dielectric fluid FC75. They obtained data for a single geometry and presented a single correlation for the Nusselt numbers of the nine heated components.
Chen et al (1988) experimentally studied the flow structure and heat transfer characteristics of natural convection in an enclosure with ten protruding heaters mounted on one vertical wall and the top wall was isothermal heat sink. They reported that the transfer coefficients for all the heated sections could be represented by a single correlation when the Nusselt and modified Rayleigh numbers were calculated based on the length scale of local heater height. The local heater height was measured from the bottom of the cavity to the midheight of a given heated section. Carmona and Keyhani (1989) conducted an experimental investigation of the cavity width effect on immersion cooling of five flush heaters on one vertical wall of an enclosure. Six cavity widths were studied covering an aspect ratio range of 3.67 to 12.2. The experiments were conducted with ethylene glycol and FC75. Keyhani et al (1991) experimentally investigated the aspect ratio effect on natural convection in an rectangular enclosure with five protruding heat sources on one vertical wall. Their results indicated that cavity width variation influences the heat transfer process mainly through altered flow pattern.
CHAPTER 3

GOVERNING DIFFERENTIAL EQUATIONS

3.1 The Form of Governing differential Equations Adopted for use in this Study

For turbulent, incompressible buoyancy-driven flow of a newtonian fluid, the time average governing equations are expressed in the general orthogonal cartesian co-ordinate system for this study. The equations are

Continuity

\[ \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  

\[ [3.1] \]

Momentum

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_i \]

\[ [3.2] \]

Where \( S_i \) = External source of momentum

Scalar

\[ \frac{\partial (\rho \phi)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \phi) = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) + S_\phi \]

\[ [3.3] \]

where \( u, p, \rho \) are function of time and \( S_\phi \) is the source term for the transport of scalar property \( \phi \).

Turbulence Model for the Present Study

Despite the greater potential of the Reynolds stress models the over riding demands of economy nominates the standard k-\( \epsilon \) model (Launder and Spalding 1972) in which the unknown Reynolds stress are expressed by means of gradient transport hypothesis where the fluxes are
assumed proportional to the gradients of mean flow properties. The constant of proportionally is $\mu_t$.

According to gradient transport hypothesis (Hinge, 1959) Reynolds stress

$$-\rho u_i u_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$  \[3.4\]

$\mu_t$ turns out to be a function of turbulence energy $k$ and its dissipation rate $\varepsilon$ via $\mu_t = C_\mu \rho k^2/\varepsilon$

where $k$ and $\varepsilon$ are derived from their own transport equations, for turbulent flow, the modelled form of the $k$-equation and $\varepsilon$-equation used in the present study are

$k$-equation

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_i} \left( \frac{\mu_{eff} \partial k}{\sigma_k \partial x_i} \right) + G - \rho \varepsilon$$  \[3.5\]

$\varepsilon$-equation

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_i} \left( \frac{\mu_{eff} \partial \varepsilon}{\sigma_{\varepsilon} \partial x_i} \right) + C_{\varepsilon1} \frac{\varepsilon}{k} G - C_{\varepsilon2} \rho \frac{\varepsilon^2}{k}$$  \[3.6\]

where $G$ is the rate of production of turbulent kinetic energy defined as,

$$G = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \beta g \frac{v_i}{\sigma_k} \frac{\partial T}{\partial y}$$  \[3.7\]

The last term of equation [3.7] represents generation or destruction of turbulence due to buoyancy. For two dimensional steady plane flow, the equations [3.7] can be written as:

$$G = \mu_t \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 \right] - \beta g \frac{v_i}{\sigma_k} \frac{\partial T}{\partial y}$$  \[3.8\]

The values of empirical constants ($C_\mu$, $C_{\varepsilon1}$, $C_{\varepsilon2}$, $\sigma_k$, $\sigma_{\varepsilon}$) used in the study are taken from Launder and Spalding [1974] and given in Table 3.1.
Table 3.1: The values of empirical coefficient in the k-ε model.

<table>
<thead>
<tr>
<th>C_μ</th>
<th>C_ε1</th>
<th>C_ε2</th>
<th>ν_k</th>
<th>ν_ε</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

3.2 Expanded Form of Governing Differential Equations:

For the sake of easier manipulation, the compact forms of the governing differential equations are rewritten here for two dimensional case as

continuity equation:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0
\]  

\[3.9\]

u-momentum equation:

\[
\frac{\partial (\rho u)}{\partial t} + \frac{\partial}{\partial x} (\rho u^2) + \frac{\partial}{\partial y} (\rho uv) = -\frac{\partial p}{\partial x} + \mu_{\text{eff}} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]  

\[3.10\]

v-momentum equation:

\[
\frac{\partial (\rho v)}{\partial t} + \frac{\partial}{\partial x} (\rho uv) + \frac{\partial}{\partial y} (\rho v^2) = -\frac{\partial p}{\partial y} + \mu_{\text{eff}} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \rho g \beta \Delta t
\]  

\[3.11\]

where \( \rho g \beta \Delta t = \) buoyancy force
\( \Delta t = \) temperature difference

General differential equation:

\[
\frac{\partial (\rho \Phi)}{\partial t} + \frac{\partial}{\partial x} (\rho u \Phi - \Gamma_\phi \frac{\partial \Phi}{\partial x}) + \frac{\partial}{\partial y} (\rho v \Phi - \Gamma_\phi \frac{\partial \Phi}{\partial y}) = S_\phi
\]  

\[3.12\]
3.3 The Method of Discretization

The governing differential equations can be discretized in many ways. An overview of the discretization method for the numerical solution of the fluid flow problems is given by Patankar [1980]. In the present study the finite volume approach, as described by Gosman et. al. [1989] and others, is adopted. In this approach, the governing differential equations are discretized by integrating them over a finite number of control volumes or computational cells, into which the solution domain are divided. A typical computational cell is shown in figure (3.1). Typical discretized transport equation (3.12) will take the following quasilinear form.

\[
\left( a_p - S_p \right) \Phi_p = \sum a_{nb} \Phi_{nb} + S_c
\]  

[3.13]

where the \( a_{nb} \) are coefficients multiplying the values of \( \Phi \) at the neighbouring nodes surrounding the central node \( P \). The number of neighbour depends on the interpolation practice or differencing schemes used.

Here \( a_p \) is the coefficient of \( \Phi_p \) is given by

\[
a_p = \sum_n a_{nb}
\]  

[3.14]

\[
\sum_n = \text{summation over neighbours (N,S,E,W)}
\]

\( a_W = \rho_W u_W A_W f_W \)

\( a_S = \rho_S v_S A_S f_S \)

\( a_n, a_s, a_e, \) and \( a_w \) are the combined coefficients of convective and diffusive.

The source term [right hand terms of the equation (3.12)] are evaluated by integrating the volumetric source \( S_\Phi \) over the volume of the computational cell and expressed as,

\[
- \int S_\Phi dv = S_p \Phi_p + S_c
\]  

[3.15]

where, \( S_c \) stands for the constant part of the source term while \( S_p \) is the coefficient of \( \Phi_p \) and often a function of \( \Phi_p \).
Since the direct solution methods (i.e. matrix inversion) require very large computer storage and time and since the governing transport equations are non-linear, (the discretised governing transport equations are seemingly linears but $a_p$ being the function of $\Phi_p$ makes them virtually nonlinear) the discretised equations are solved using the PISO algorithm of R.I. Issa (1984). This scheme was selected because the other implicit schemes are iterative, can be troublesome, as convergence is not always assured and usually requires ad hoc specification of under-relaxation. However, in using PISO multiple corrector steps, whose number increased with Ra, had to be employed. In each time step, the temperature equation had to be solved twice, before and after the solution of the momentum equations. This was done to update the coefficients in the intermediate stages which helped to stabilize the solution process.

3.4 Differencing Schemes used in this study

For the present study the Hybrid Scheme is used. The name Hybrid is indicative of a combination of the Central Difference Scheme and Upwind Difference Scheme. For the range of Peclet number ($\rho u L/T$), $-2 \leq Pe \leq 2$, both the diffusion and convective term are evaluated by the Central Difference Scheme (CDS). Outside this range convective terms are evaluated using the Upwind Difference Scheme (UDS) and the diffusion terms are evaluated using CDS. In this section the differencing schemes used to evaluate convected cell face value of the dependent variable in terms of surrounding nodal values are discussed.

3.4.1. Central Differencing Schemes (CDS)

If a piecewise-linear profile of $\Phi$ is assumed between $P$ and $E$ (See Fig. 3.5), the cell face value $\Phi_c$ is given by

$$\Phi_c = \Phi_E f_p + \Phi_P (1 - f_p)$$  \hspace{2cm} [3.16]

where $f_p$ is a linear interpolation factor defined as:

$$f_p = \frac{\Delta x_P}{\Delta x_P + \Delta x_E}$$

Here $\Delta x_P$ and $\Delta x_E$ are the cell dimensions along x coordinate for $P$ and $E$ cells (See Fig. 3.1 and 3.5).
In this scheme \( a_E \) and \( a_N \) are always negative and if the convection process dominates this can cause the whole coefficient \( a_{ib} \) to assume negative value. As a result the Scarborough criteria fails and produce unbounded solutions (Spalding [1972], Rathby & Towance [1974]). At high Peclet number the CDS also violates the transportive property of employing downstream nodes in expressions given above. For these reasons application of CDS is limited to low Reynolds number problems.

### 3.4.2. Upwind Differencing Scheme (UDS)

The upwind differencing scheme (Runchal & Wolfshtein [1969]) recognizes that the weak point in the preliminary formulation is the assumption that the convective property \( \Phi_e \) at the interface is the average of \( \Phi_E \) and \( \Phi_p \) and it propose a better prescription. The formulation of the diffusion term is left unchanged but the convection term is calculated from the following assumption.

The value of \( \Phi \) at an interface (See Fig. 3.6) is equal to the value of \( \Phi \) at the grid point of the upwind side of the faces.

Thus

\[
\Phi_e = \Phi_p \quad \text{if} \quad f_e > 0 \\
\Phi_e = \Phi_E \quad \text{if} \quad f_e < 0
\]

The value of \( \Phi_w \) can be defined similarly. In this scheme all the coefficient contributing to \( a_p \) are always non-negative. As a result Scarborough criteria is satisfied. UDS also satisfies the property of transportiveness and thus the boundedness of the solution is guaranted. The scheme is sometimes said to be based on the "Tank and Tube" model (Gosman, Pun, Runchal, Spalding and Wolfshtein, 1969).

### 3.5 SOLUTION PROCEDURE

#### 3.5.1 Grid and Variable Arrangement

In the present study, the numerical solution is accomplished on a variably spaced staggered mesh [see for example Caretto et. al. [1972], Patanker [1980] in which the scaler quantities [including pressure, density, viscosity, kinetic energy (k) and energy dissipation (\( \epsilon \))] are defined at the centre and the normal velocities at cell faces as shown in Fig. 3.4. It has the advantage that the variables \( u, v, p \) are stored such that the pressure gradients which drive the velocities are easy to evaluate, and moreover the velocities are located where they are needed for the calculation of convective flux.
3.5.2 Calculation of Pressure:

The pressure gradient forming part of the source term in the momentum equations are to be obtained before the velocity field is calculated and it is the pressure field through which the continuity equation is satisfied.

The PISO method of R. I. Issa [1984] is used in the present study to obtain pressure. The differential equation presented in section 3.1 Eq.[3.2] & Eq.[3.1] can be expressed in the following discretised form.

\[
\frac{\rho}{\Delta t} (u_{i}^{n+1} - u_{i}^{n}) = H(u_{i}^{n+1}) - \Delta_{i}p^{n+1} + S_{i} \tag{3.17}
\]

and \( \Delta_{i}u_{i}^{n+1} = 0 \) \tag{3.18}

where \( n \) and \( n+1 \) denote successive time level and \( H \) is the convective and diffusive fluxes of momentum and \( \Delta_{i} \) is equivalent of \( \frac{\partial}{\partial x_{i}} \).

The corresponding pressure equation which is derived by taking the divergence of Eq. [3.17] and substituting in Eq. [3.18]

\[
\Delta_{i}^{2} p^{n+1} = \Delta_{i}H(u_{i}^{n+1}) + \Delta_{i}S_{i} + \frac{\rho}{\Delta t} \Delta_{i}u_{i}^{n} \tag{3.19}
\]

Let the superscripts \( ^{*} \), \( ^{**} \), and \( ^{***} \) denote intermediate field values obtained during the splitting process. The equations can hence be factorised as follows:

**Predictor Step.** The pressure field prevailing at \( t_{n} \) is used in the solution of the implicit momentum equations [3.17] to yield the \( u_{i}^{*} \) velocity field. Thus

\[
\left( \frac{\rho}{\Delta t} - A_{o} \right) u_{i}^{*} = H'(u_{i}^{*}) - \Delta_{i}p^{n} + S_{i} + \frac{\rho}{\Delta t} u_{i}^{n} \tag{3.20}
\]

Here \( H'(u_{i}^{*}) = H(u_{i}^{*}) - A_{o}u_{i}^{*} \)

Equation [3.20] is solved to yield the \( u_{i}^{*} \) field which, it should be noted, will not in general satisfy the zero-divergence condition [3.18].
**First Corrector Step.** A new velocity field, $u_i^{**}$, together with a corresponding new pressure field, $P^*$, are now sought such that the zero-divergence condition

$$\Delta_i u_i^{**} = 0 \tag{3.21}$$

is met. For this, the momentum Equation [3.17] is taken as

$$\left( \frac{\rho}{\partial t} - A_{o} \right) u_i^{**} - \frac{\rho}{\partial t} u_i^{*} = H'(u_i^*) - \Delta_i P^* + S_i \tag{3.22}$$

Here $H'(u_i^*) = H(u_i^*) - A_0 u_i^{**}$

By subtracting Equation [3.20] from Equation [3.22] we get

$$\left( \frac{\rho}{\partial t} - A_{o} \right) (u_i^{**} - u_i^*) = - \Delta_i (p^* - p^n) \tag{3.23}$$

Taking the divergence of Equation [3.23] and putting it into Equation [3.21], the following pressure Equation is obtained

$$\Delta_i \left[ \left( \frac{\rho}{\partial t} - A_{o} \right)^{-1} \Delta_i \right] (p^* - p^n) = \Delta_i u_i^* \tag{3.24}$$

which is readily solvable since the right hand side contains terms in the known field $u_i^*$, and this is the consequence of using the explicit form of Eq. [3.22]. The $P^*$ field obtained by solving Eq. [3.24] may be used to yield the $u_i^{**}$ field, which, it should be recalled, satisfies the zero-divergence condition equation [3.21].

**Second Corrector Step.** A new velocity field, $u_i^{***}$, together with its corresponding new pressure field, $P^{**}$, are formulated, such that

$$\Delta_i u_i^{***} = 0 \tag{3.25}$$
The operative momentum equation is now taken as the explicit type equation

\[
\left( \frac{\rho}{\partial t} - A_v \right) u_i^{***} - \frac{\rho}{\partial t} u_i^{*} = H'(u_i^{**}) - \Delta_i p^{**} + S_i
\]  \[3.26\]

The corresponding pressure equation is therefore

\[
\Delta_i \left[ \left( \frac{\rho}{\partial t} - A_v \right)^{-1} \Delta_i \right] (p^{**} - p^*) = \Delta_i \left[ \left( \frac{\rho}{\partial t} - A_v \right)^{-1} H'(u_i^{**} - u_i^*) \right]
\]  \[3.27\]

From Eq. [3.27], the \( p^{**} \) field can be readily determined since the right-hand side of the equation is known, and with this new pressure, the \( u_i^{***} \) field can be evaluated.

More corrector steps such as the above can obviously be introduced. However, the accuracy with which \( u_i^{***} \) and \( p^{**} \) approximate the exact solution \( u_i^{n+1} \) and \( p^{n+1} \) of Eq. [3.17] and [3.19] is sufficient for most practical purposes. For the present study around four corrector stages had to be employed to obtain the solutions.

### 3.6 SOLUTION ALGORITHM

A noniterative method called PISO for handling the pressure-velocity coupling of the implicit discretised fluid flow equations is used. The method (called PISO for Pressure Implicit with Splitting of Operators) utilises the splitting of operations in the solution of the discretised momentum and pressure equations are obtained, such that the field values at each time step are close approximations of the exact solution of the difference equations with a formal order of accuracy of the order of power of \( \delta t \) depending on the number of operation splittings used.

1) To apply to the coupling between variables, pressure and velocity, where by operations involving different variables are split into a series of predictor-corrector steps.

Let us superscripts *, **, and *** denote intermediate field values obtained during the splitting process.

2) Predictor Stage: A initial field value \( u_i^* \), which will not in general satisfy the zero-divergence condition, is obtained using the pressure field values of the previous iteration.
3) First Corrector Stage: A new velocity field $u_{i}^{**}$ together with a corresponding new pressure field $p^*$ are sought such that the zero divergence condition

$$A_{i} u_{i}^{**} = 0$$

is satisfied and get the $P^*$ from the equation [3.26]

4) Second Corrector Stage: In this stage new velocity field $u_{i}^{***}$ together with its corresponding new pressure field $p^{**}$ are obtained.

More corrector stages may be added following procedure used for the second corrector stage.

For the protruding heater case, the protrusion obstacle in the calculation domain can be done by inserting 'internal' boundary condition. The quantities $S_c$ and $S_p$ arise from the source term linearization of the form

$$S = S_c + S_p \Phi p$$

The quantity $S_p$ must not be positive. However, any desire value of $\Phi$ can be arranged to be the solution at an internal grid point by setting $S_c$ and $S_p$ for that point as

$$S_c = 10^{30} \Phi p, \text{ desired}$$

$$S_p = -10^{30}$$

Where $10^{30}$ denotes a number large enough to make the other terms in the discretization equation is negligible. The consequence is that

$$S_c + S_p \Phi p = 0$$

$$\Phi p = - S_c/S_p = \Phi p, \text{ desired}$$

In the present study, the convergence criteria is that, the sum of the normalized absolute residual at all computational nodes, defined as

$$R_{\Phi} = \sum N \left[ (a_{p}^r - b^{(r-1)}) \Phi p^{(r-1)} - \sum a_{nb} \Phi n_{b}^{(r-1)} - C^{(r-1)} \right] / N_f$$

should fall below a specified level

$$R_{\Phi} < 10^{-3}$$

Here $N$ is the total number of nodes, $r$ the iteration counter and $N_f$ the normalization factor.
CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Introduction

In this chapter, the results of numerical study of natural convection in an enclosure with protruding and flush heat sources, are presented and compared with the experiments of Keyhani et. al (1991) and Carmona & Keyhani (1989). The results are obtained by the numerical method described in Chapter 3. Standard k-ε model incorporated with Hybrid Scheme was employed to evaluate the diffusion terms and convection terms.

4.2 Solution Domain and Computational Grid and Boundary Conditions

The solution domain investigated here is a two dimensional rectangular enclosure shown in Fig. 2. The entire computational domain was divided into 62 x 62 grids. The distribution of these grids were non-uniform. Five identical heat sources were alternately mounted on the vertical wall of the enclosure with uniform spacing. Two types of heat sources i.e. protruding and flush were used. The other vertical wall and the bottom wall are adiabatic. The top surface of the test enclosure was an isothermal heat sink. Ethylene glycol was considered as the working fluid. The thermophysical properties of ethylene glycol (Appendix-C) were evaluated at the arithmetic average of temperatures $T_h$ and $T_c$, where $T_h$ is the average temperature of each heated surface and $T_c$ is the temperature of the top heat sink. Simulation and analysis of the flow structure is carried out at four cavity widths ($W=18, 22.5, 36 & 45\text{mm}$) and $Q = 14.3, 42.8 & 85.7 \text{ watts/m power input per heated section}$. The solution domain and boundary conditions used here are in accordance with those used in the experiments of Keyhani et al (1991) and Carmona & Keyhani (1989).

4.3 Presentation and Discussion of Result and Comparison with Experiment

Results obtained with protruding and flush heat sources for power input of $14.3, 42.8 & 85.7 \text{ watts/m per heated section}$ and cavity widths of $18, 22.5, 36 & 45\text{mm}$ are presented here. Ethylene Glycol is used as the working fluid. $Ra^*y$ ($Ra^* = g\beta q (y)^4/(\kappa \alpha)$ varied from $1.9 \times 10^6$ to $4.9 \times 10^{10}$, while the local prandtl number varied from 65 to 125.
STREAMLINES

The streamlines for the protruding heater case are presented in Fig. 4. For W=45mm and 36mm, Fig. [4.1(i) & 4.1(ii)b] show that the flow consists of single vortex which begins at a height from the bottom wall and ends near the top wall of the enclosure. The centre of this vortex is located near the upper region of the enclosure. When the power input is low [Q=14.3 watts/m, Fig. [4.2(i) & 4.2(ii)] this single vortex starts from a lower region of heater one. When the power input is increased [Q=85.7 watts/m, Fig. [4.1(i) & 4.1(ii)] the location of the bottom of the vortex moves up and reaches the middle region of the unheated section between heater one and heater two. With the decrease of width (W=22.5 & 18mm) the above mentioned single vortex structure disintegrates and several secondary flow cells appear within it [Fig. 4.1(iii) & 4.1(iv)] and [Fig. 4.2(iii) & 4.2(iv)]. Here also, the location of the bottom of the primary vortex moves up with the increase of power input. These predicted flow patterns are in good qualitative agreement with the flow visualization results observed by Keyhani et. al. (1991).

For the flush heater case of Q = 85.7 watts/m and W = 45mm, the streamlines [Fig. 5.1(i)] show that the flow consists of a single vortex which begins at a height from the bottom wall and ends near the top wall of the enclosure. The center of this vortex is located near the upper region of the enclosure. For the same power input, this vortex center moves towards the top wall as the enclosure width is reduced [Fig. 5.1(i)-5.1(iv)]. This migration of the vortex center leads to the development of secondary flow cells in the lower region of the core flow. For the case of Q = 14.3 watts/m [Fig. 5.2(i)-5.2(iv)] the streamline contours show similar behavior to that of the Q = 85.7 watts/m case. Here, the variation of Rayleigh number due to reduction in power input, is not sufficient enough to create any variation in the flow pattern.

The velocity profile for protruding and flush heater case at Q = 85.7 watts/m are shown in [Fig. 4.3 & Fig. 5.3]. These figures show that the velocity at point \( Y/H = 0.50 \) (middle position of heater 3) is always higher than that at others \( Y/H \).

TURBULENCE INTENSITY

The turbulent intensity defined here as \( k^{1/2} \) are shown in Fig. 6 & Fig. 7. As expected, higher values of turbulent intensities are found in the shear layers of the main vortex. These values seem to increase with the decrease of width and increase of power input. For the protruding heater case Fig. 6.1 & Fig. 6.2 show that high turbulence intensity values are also found in the near solid wall region of the
protruding heaters. This is due to high velocity gradients created there due to sharp streamline curvature and formation of secondary flow cells inbetween the protrusions.

TURBULENCE LENGTH SCALE

Turbulent length scales defined here as $C_k^{3/2} / e$ are shown in Fig. 8 & Fig. 9. For the protruding heater case, Fig. 8 shows that the length scale values increase with the increase of distance from the solid wall, which is desirable here. Similar behaviour is also observed for the flush heater case, Fig. 9. With the decrease of cavity width, formation of secondary flow cells in the core region and also in-between the protrusions are reflected in the length scale contours.

TEMPERATURE DISTRIBUTION

For the protruding heater case, Fig. 10 present the predicted excess temperatures ($T - T_C$), hereafter called the temperature, of the heated and unheated sections and the core fluid for various values of cavity widths and power inputs. Fig. 10.1 shows the predicted temperatures for $W = 45\text{mm}$ and $Q = 14.3 \text{ & } 85.7 \text{ watts/m per heater}$. At fixed width and both power inputs, the temperatures of the top and bottom unheated sections are lower than those of other unheated and heated sections. These temperatures at the bottom and top unheated sections are close to that of the core fluid. The top heater ($N=5$) has a lower temperature than that of heater four ($N=4$) due to its proximity to the top sink surface. The effect of an increase in power input at a fixed cavity width is the corresponding increase in the differences of temperature between the heated and unheated sections.

Fig. 10.1 also show the core temperature distribution at $X/W=0.186$ and $X/W=0.583$. These two profiles are nearly the same, which is indicative of the imposed adiabatic boundary conditions at the opposing vertical walls. Moreover, these show that the region is of nearly constant vertical temperature distribution.

For the protruding heater case, comparison of experimental and predicted temperatures for $W=36\text{mm}$ are presented in Fig. 10.2. For $Q = 85.7 \text{ & } 14.3 \text{ watts/m}$ [Figs. 10.2(i) and 10.2(ii)] show similar trend to that observed for $W = 45\text{mm}$. Here the predicted temperature of heater one is higher than that of the experimental data. The main reason of the anomaly is the two dimensional assumption of the three dimensional enclosure used in the experiments. In the three dimensional case, the
cold fluid returning along the vertical end walls (which are absent in the two
dimensional case) brings down the temperature of the fluid in the lower region of the
enclosure and hence increase the heat transfer from the first heater. This ultimately
results in lower temperatures of the first heated and unheated sections. Temperatures
of the other heated and unheated sections are in improved agreement with the
measurements. However, the temperature of the heated sections show overprediction.
This may be due to the fact that the radiation loss from the heaters are not accounted
for in the present prediction. This radiation loss, which may be calculated as,

\[ q_r = \sigma A \Delta T^4 \times \text{shape factor} \]

where \[ \sigma = 5.669 \times 10^{-8} \text{ w/m}^2\text{K}^4 \]

amounts upto 5% and ultimately leads to the lower temperature of the heated sections
in the measurements. The predicted temperatures of the unheated sections show lower
values than that of the measurements. This may be due to conduction of heat from
the heated sections to the unheated sections, which in the experiments amounts upto
9% of the heat input [Keyhani et. al. (1991)] and totally absent in the predictions.
Fig. [10.2(i)] also shows that the predicted excess core temperature distributions at
\[ X/W = 0.186 \] and \[ X/W = 0.583 \] are little higher than that of the experimental data,
which again may be due to absence of the conduction heat loss from the
experimental set up as already mentioned above. These two profiles are nearly the
same, showing that the region is of nearly constant vertical temperature distribution.
The discrepancies observed at \[ Q = 14.3 \text{ watts/m} \] [Fig. 10.2(ii)] are much lower than
that observed for \[ Q = 85.7 \text{ watts/m} \] [Fig. 10.2(i)] this may be due to lower radiation
and conduction losses at lower overall temperature for \[ Q = 14.3 \text{ watts/m} \].

Comparisons for \[ W = 22.5 \text{ mm} \] are shown in Fig. 10.3. For \[ Q = 85.7 \text{ watts/m} \], Fig.
10.3(i) shows similar trend to that observed for \[ W = 45 \text{ & 36 mm} \], but here the
predicted heater temperatures are in improved agreement with the experiments. This
improvements are due to lower radiation loss, resulting from lower temperature
difference between the heated and unheated sections. For \[ Q = 14.3 \text{ watts/m} \], Fig.
10.3(ii) shows that the core fluid temperature is no more constant, rather it gradually
decreases with the enclosure height. This trend is also observed for \[ Q = 85.7 \text{ watts/m} \]
Fig. 10.3(i) but of much lower magnitude. The experimental data were not available
for comparisons for this power input.
The comparisons for \( W = 18 \text{ mm} \) are shown in Fig. 10.4. For \( Q = 85.7 \text{ watts/m} \) [Fig. 10.4(i)] improve agreement is observed due to lower temperature difference between the heated and unheated sections. Here the core fluid temperatures shows a gradual decrease with enclosure height. For \( Q = 14.3 \text{ watts/m} \), the predicted results [Fig. 10.4(ii)] show that the magnitude and distribution of both the wall and the core fluid temperatures are very similar. But the agreement with the experiments has deteriorated. This may be due to the deficiencies of the turbulence model used in the predictions.

For the flush heater case, the predicted excess temperature \((T - T_c)\) are compared with the experiments in Fig.11. The temperatures of the heated section \((N=2, 3 \& 4)\) are in reasonably good agreement with the experiments. For \( N=1 \), the predicted results show higher values than that of the measurements. The temperatures of the first unheated section are also over predicted. The main reason of this anomaly is already explained above for the protruding heater case. Although the temperatures of the second unheated section are well reproduced, the remaining unheated section show increasingly under predicted values. This trend is more pronounced for the \( Q = 85.7 \text{ watts/m} \) case. The overall prediction for the \( Q = 14.3 \text{ watts/m} \), Fig. 11.1(ii), 11.2(ii), 11.3(ii) and 11.4(ii) show much better agreement than that obtained for \( Q = 85.7 \text{ watts/m} \). Fig. 11 also shows that the temperature distribution for the core fluid is nearly constant. For the same power input, decreasing the width results in higher overall temperature [Fig. 11.2(i), 11.3(i) and 11.4(i)].

The influence of cavity width parameter \((W/L_3)\) on the temperature of the heated sections, \( N = 3 \) and 1, at power inputs of \( Q = 14.3, 42.5, \) and \( 85.7 \text{ watts/m per heater} \) are shown in Figs.12(i) and 12(ii). Changes in \( W \) for a fixed \( Q \), and variation in \( Q \) for a fixed \( W \), each have pronounced and distinct effects on the temperature of the heated section. The data suggest that the effect of \( W/L_3 \) on \((T_h - T_c)\) at all power inputs is rather complicated for \( 2.0 \leq W/L_3 \leq 2.5 \). But, it is clear that the influence of \( W/L_3 \) on the temperature of the heated sections is negligible for \( W/L_3 > 4 \).

**HEAT TRANSFER**

The heat transfer from the heated protrusions are presented in the form of Nusselt number as a function of modified Rayleigh number and aspect ratio. The Nusselt number is defined as:

\[
Nu = \frac{h[\text{length scale}]}{K} \quad (4.1)
\]

\[
Ra^* = g\beta q \left[ \text{length scale} \right]^4 / K \nu \alpha \quad (4.2)
\]
where \( h = q/(T_h - T_c) \) and \( q = Q/A_h \). Here \( A_h \) is the total exposed surface area of each protruding heater. The length scale used in Fig. (13.1), is the height of the protruding heater (L_1) and that used in Fig. (13.2) is the vertical distance (y) of the protruding heater centre from the bottom of the enclosure.

Since \( L_1 \) is a constant, Fig. (13.1) directly shows the dependence of local heat transfer coefficient for each heater on the power input. As expected, this coefficient increases with the power input. For a particular power input, the values of \( \text{Nu}_{L_1} \), for all the five heaters are very close in magnitude. However, it is observed that heater five (N = 5) has slightly higher heat transfer coefficient.

The results based on the length scale y are presented in Fig. (13.2). It is interesting to note that for each cavity width, the log-log plot of \( \text{Nu}_y \) versus \( \text{Ray}^* \) for all the heated sections forms a straight line. The local Nusselt number of the heated sections \( \text{Nu}_y \) for each width is correlated in terms of its respective local modified Rayleigh number \( \text{Ray}^* \) in the form of:

\[
\text{Nu}_y = c (\text{Ray}^*)^m
\]

where the constant "c", exponent m and the standard errors of estimates are given in Table-4.1.

**TABLE - 4.1**

Comparison of the Coefficients (c), Exponents (m) for Protruding-Heater Case with Keyhani [1991] for Heat Transfer Correlations.

<table>
<thead>
<tr>
<th>Width (W) mm</th>
<th>Computational</th>
<th></th>
<th></th>
<th>Experimental Keyhani [1991]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>c</td>
<td>m</td>
<td>( \Sigma ) Error^2</td>
<td>Tolerance x 10^7</td>
</tr>
<tr>
<td>18</td>
<td>0.0625</td>
<td>0.249</td>
<td>16.8</td>
<td>2.59</td>
</tr>
<tr>
<td>22.5</td>
<td>0.0835</td>
<td>0.240</td>
<td>27.0</td>
<td>2.98</td>
</tr>
<tr>
<td>36</td>
<td>0.1816</td>
<td>0.213</td>
<td>44.7</td>
<td>3.57</td>
</tr>
<tr>
<td>45</td>
<td>0.1495</td>
<td>0.231</td>
<td>63.9</td>
<td>1.42</td>
</tr>
</tbody>
</table>

26
The total energy convected from the heated sections is transferred to the top sink surface. The heat transfer coefficient $h_T$ at the top sink surface defined as:

$$h_T = \frac{Q_T}{A_T(T_h - T_c)}$$

where, $Q_T$ is the total power input, $A_T$ is the top sink surface area and $T_h$ is the average temperature of all the heated sections, is plotted against the total convective heat flux at the top sink surface $q_T$ in Fig. 14. This figure shows that $h_T$ increases with decreasing cavity width.

The aspect ratio effect on the local Nusselt number ($Nu_{L_I}$) for various power input ($Q = 14.3, 42.5$ and $85.7$ watts/m) are shown in Fig. 15. This figure shows that the heat transfer coefficient decreases with aspect ratio and for all aspect ratio heater 5 has the highest heat transfer coefficient.

A comparison of predicted total heat convected from the top sink surface $Wh_T$ ($W \times h_T$) and $Q_T$ with the experiments [(Keyhani et al. (1991)) & Carmona & Keyhani (1989)] for $W = 36$ mm, are presented in Table-4.2. The predicted results are in good agreement with the measurements. The maximum deviation is 6.6%.

**TABLE - 4.2**

Comparison of $Wh_T$ for Protruding and Flush Heat Sources with Keyhani [1991] and Carmona & Keyhani [1989].

<table>
<thead>
<tr>
<th>Protruding Heater $W = 36$mm</th>
<th></th>
<th>Flush Heater $W = 36$mm</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q_T$</td>
<td>$Wh_T$</td>
<td>$Q_T$</td>
<td>$Wh_T$</td>
</tr>
<tr>
<td>71.50</td>
<td>6.67</td>
<td>71.50</td>
<td>6.50</td>
</tr>
<tr>
<td>214.00</td>
<td>8.36</td>
<td>214.00</td>
<td>8.15</td>
</tr>
<tr>
<td>428.50</td>
<td>10.14</td>
<td>428.50</td>
<td>10.90</td>
</tr>
</tbody>
</table>

27
CHAPTER - 5

SUMMARY AND CONCLUSIONS

5.1 General

In this chapter, the main findings and achievements of the present computational study, made with respect to the objective set in Chapter-1, are presented and the scope of extension and development of the present study are suggested. In Section 5.2, the summary of main findings and achievement are presented and the suggestions for future work are given in Section 5.3.

5.2 Summary of Main Findings and Achievements

Two dimensional turbulent, incompressible, buoyancy driven flow has been simulated for both protruding and flush heat sources mounted on one vertical wall of a rectangular enclosure. The main findings are summarized below:

For the larger width (W=45 & 36mm) a single vortex is observed to occupy the whole domain and for the smaller width (W=22.5 & 18mm) that single vortex disintegrates and several secondary flow cells appear within it. The influence of power input per heated sections are also studied. For the higher power input (Q=85.7 watts/m), the location of the bottom of the primary vortex moves upward.

The effects of cavity width and power input are also observed in the turbulence intensity. These values seem to increase with the decrease of width and power input.

For the all cases, the core fluid temperatures are almost constant along the vertical.

Based on the local height $y$ of each heater, all the data are well represented by a single correlation. This correlation includes an explicit aspect ratio effect & is independent of the number of heaters in the vertical array & vertical height location of the heaters.

The heat transfer coefficient $h_T$ at the top sink surface increases with the decrease of cavity width.
For a fixed power input the local Nusselt numbers \( (\text{Nu}_L) \) decreases with the increase of aspect ratio.

The prediction of flow field for the protruding and flush heater cases are in good agreement with the measure data.

The discrepancies that are evident in the predicted results, can be attributed to the deficiencies of the turbulence model used and also to the fact that, in the experiments Keyhani et. al. [1991] and Cormona & Keyhani [1989], heat loss by conduction through the enclosure walls and the unheated section, amounts to about 10% of the heat input. The heat transfer from the heat sources by radiation is about 5% of heat input. These could not be taken into account in the predictions.

5.3 Suggestions for Future Work

i. The same prediction can be carried out in the three dimensional enclosure.

ii. To obtain better results, turbulence models capable of handling this type of complicated flow can be used (e.g. Reynolds stress models, Large eddy simulation).

iii. Higher order schemes (e.g. LUDS, Quick Scheme) can be used to have better accuracy in this type of prediction.

iv. Similar study can be made different size, shape, location and power input of the heaters.

v. Similar study can be made with different angle of rotation.
REFERENCES


Fig. 1.0: Schematic Diagram of the Enclosure
(a) Protruding Heat Sources; (b) Flush Heat Sources.
Fig.2.0: Computational Grid of the Enclosure.
Fig. 3.1: Two Dimensional Computational Cell.
Fig. 3.2: Control Volume for U.

Fig. 3.3: Control Volume for V.
Fig. 3.4: Computational Grid, Location and Control Volume (cells) of Scalar Variables and Axial and Radial Velocities.
Fig. 3.5: Schematic Presentation of CDS.

Fig. 3.6: Schematic Presentation of UDS.
Fig. 4.1: Streamlines for Protruding Heater Case at $Q = 85.7$ watts/m;
(i) $W = 45\text{mm}$; (ii) $W = 36\text{mm}$; (iii) 22.5mm; (iv) $W = 18\text{mm}$.

Contour Key $(\psi/\psi_{\text{max}})$
where $\psi = \int v \, dx$

(a) = -0.10
(b) = -0.25
(c) = -0.50
(d) = -0.75
(e) = -0.95

Fig. 4.2: Streamlines for Protruding Heater Case at $Q = 14.3$ watts/m;
(i) $W = 45\text{mm}$; (ii) $W = 36\text{mm}$; (iii) 22.5mm; (iv) $W = 18\text{mm}$.
Fig. 4.3: Velocity Profile for Protruding Heater Case
at $Q = 85.7$ watts/m; (i) $W = 45$ mm; (ii) 22.5 mm.
Fig. 5.3: Velocity Profile for Flush Heater Case at $Q = 85.7$ watts/m;
(i) $W = 45$ mm; (ii) $22.5$ mm.
Fig. 5.1: Streamlines for Flush Heater Case at $Q = 85.7$ watts/m;
(i) $W = 45\text{mm}$; (ii) $W = 36\text{mm}$; (iii) $22.5\text{mm}$; (iv) $W = 18\text{mm}$.

Contour Key ($\psi/\psi_{\text{max}}$)
where $\psi = \int vdx$

- $a = -0.05$
- $b = -0.25$
- $c = -0.50$
- $d = -0.70$
- $e = -0.90$

Fig. 5.2: Streamlines for Flush Heater Case at $Q = 14.3$ watts/m;
(i) $W = 45\text{mm}$; (ii) $W = 36\text{mm}$; (iii) $22.5\text{mm}$; (iv) $W = 18\text{mm}$. 
Contour Key ($k^{1/2}$)
(a) = 0.002
(b) = 0.004
(c) = 0.006
(d) = 0.007
(e) = 0.008
(f) = 0.010

Fig.6.1: Turbulence Intensity for Protruding Heater Case at $Q = 85.7$ watts/m;
(i) $W = 45mm$; (ii) $W = 36mm$; (iii) $22.5mm$; (iv) $W = 18mm$.

Fig.6.2: Turbulence Intensity for Protruding Heater Case at $Q = 14.3$ watts/m.
(i) $W = 45mm$; (ii) $W = 36mm$; (iii) $22.5mm$; (iv) $W = 18mm$. 
Fig. 7.0: Turbulence Intensity for Flush Heater Case at $Q = 85.7$ watts/m.

(i) $W = 45$mm; (ii) $W = 36$mm; (iii) $22.5$mm; (iv) $W = 18$mm.

Contour Key ($k^{1/2}$)

(a) = 0.002
(b) = 0.004
(c) = 0.006
(d) = 0.007
(e) = 0.008
(f) = 0.010
Contour Key \((C_{\mu}k^{3/2}/\varepsilon)\)

(a) = 0.000125  
(b) = 0.000250  
(c) = 0.000500  
(d) = 0.000750  
(e) = 0.0010  
(f) = 0.0015  
(g) = 0.0020

Fig. 8.1: Turbulence Length Scale for Protruding Heater Case at \(Q = 85.7\) watts/m.  
(i) \(W = 45\)mm; (ii) \(W = 36\)mm; (iii) \(22.5\)mm; (iv) \(W = 18\)mm.

Fig. 8.2: Turbulence Length Scale for Protruding Heater Case at \(Q = 14.3\) watts/m.  
(i) \(W = 45\)mm; (ii) \(W = 36\)mm; (iii) \(22.5\)mm; (iv) \(W = 18\)mm.
Contour Key ($C_{f} k^{3/2}/\epsilon$)

- a = 0.0005
- b = 0.0010
- c = 0.0012
- d = 0.0015
- e = 0.0020
- f = 0.0025

Fig. 9.0: Turbulence Length Scale for Flush Heater Case at $Q = 85.7$ watts/m.

(i) $W = 45$ mm; (ii) $W = 36$ mm; (iii) $22.5$ mm; (iv) $W = 18$ mm.
Fig. 10.1: The Temperature Distribution for Protruding Heater Case at \( W = 45 \text{ mm} \).

\[ W = 45 \text{ mm}, Q = 85.7 \text{ watts/m} \]

\[ W = 45 \text{ mm}, Q = 143 \text{ watts/m} \]
Fig. 10.2: Comparison of the Temperature Distributions for Protruding Heater Case at \( W = 36 \text{ mm} \).
Fig. 10.3: Comparison of the Temperature Distributions for Protruding Heater Case at $W = 22.5$ mm.
Fig. 10.4: Comparison of the Temperature Distributions for Protruding Heater Case at $W = 18$ mm
Fig. 11.1: The Temperature Distribution for Flush Heater Case at $W = 45$ mm.
Fig. 11.2: Comparison of the Temperature Distributions for Flush Heater Case at $W = 36$ mm.
Fig. 11.3: Comparison of the Temperature Distributions for Flush Heater Case at $W = 22.5$ mm.
Fig. 11.4: The Temperature Distribution for Flush Heater Case at \( W = 18 \text{ mm} \).
Fig. 12.0: The Cavity Width Effect on Heated Section Temperatures at Various Power Inputs for Protruding Heater Case; (i) Heater 3; (ii) Heater 1.
Fig. 13.1: Heated-Section Nusselt Numbers (\(N_u_{L_1}\)) as a Function of \(Ra_{L_1}^*\) for Protruding Heater Case.
Fig. 13.2: Local Nusselt Number (Nu$_y$) as a Function of Local Modified Rayleigh Number (Ra*$_y$) for all Cavity Width for Protruding Heater Case.
Fig. 14.0: Heat Transfer Co-Efficient for Protruding-Heater Case (based on the top cooling area) Versus the Heat Flux Based at the Top Cooling Surface.
Fig. 15.0: Aspect Ratio (H/W) Effect on Heated Section Nusselt Numbers (Nu_L1) for Protruding Heater Case; (i) Q = 85.7 watts/m; (ii) Q = 42.8 watts/m; (iii) Q = 14.3 watts/m.
APPENDIX-A

Structure of the Mathematical foundation

- Conservation law
- Transport law
- Source law

- Differential equations
- Finite difference equations
- Solution algorithm
- Computer programme
- Predictions
APPENDIX-B

The overall structure of Computer Programme
## APPENDIX-C

### PROPERTIES OF ETHYLENE GLYCOL, C<sub>2</sub>H<sub>4</sub>(OH)<sub>2</sub>

<table>
<thead>
<tr>
<th>T°C</th>
<th>ρ</th>
<th>Cp</th>
<th>ν x 10&lt;sup&gt;6&lt;/sup&gt;</th>
<th>k</th>
<th>α x 10&lt;sup&gt;7&lt;/sup&gt;</th>
<th>Pr</th>
<th>β</th>
<th>K&lt;sup&gt;-1&lt;/sup&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Kg/m&lt;sup&gt;3&lt;/sup&gt;</td>
<td>kJ/kg°C</td>
<td>m&lt;sup&gt;2&lt;/sup&gt;/sec</td>
<td>w/m°C</td>
<td>m&lt;sup&gt;2&lt;/sup&gt;/sec</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>1130.75</td>
<td>2.294</td>
<td>57.53</td>
<td>0.242</td>
<td>0.934</td>
<td>615</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>1116.65</td>
<td>2.382</td>
<td>19.18</td>
<td>0.249</td>
<td>0.939</td>
<td>204</td>
<td></td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>1101.43</td>
<td>2.474</td>
<td>8.69</td>
<td>0.256</td>
<td>0.939</td>
<td>93</td>
<td></td>
<td>0.65 x 10&lt;sup&gt;-3&lt;/sup&gt;</td>
</tr>
<tr>
<td>60</td>
<td>1087.66</td>
<td>2.562</td>
<td>4.75</td>
<td>0.260</td>
<td>0.932</td>
<td>51</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>1077.56</td>
<td>2.650</td>
<td>2.98</td>
<td>0.261</td>
<td>0.921</td>
<td>32.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>1058.50</td>
<td>2.742</td>
<td>2.03</td>
<td>0.263</td>
<td>0.908</td>
<td>22.4</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

*Note: The table provides properties of ethylene glycol at various temperatures.*