A numerical prediction of the thermal environment of a room heated with a hydronic heating system

Satya Kiran C Gurram

University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation
Gurram, Satya Kiran C, "A numerical prediction of the thermal environment of a room heated with a hydronic heating system" (2007).
UNLV Retrospective Theses & Dissertations. 2169.
https://digitalscholarship.unlv.edu/rtds/2169

This Thesis is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Retrospective Theses & Dissertations by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
A NUMERICAL PREDICTION OF THE THERMAL ENVIRONMENT OF A ROOM
HEATED WITH A HYDRONIC HEATING SYSTEM

By
Satya Kiran C. Gurram
Bachelor of Technology
Jawaharlal Nehru Technological University, Andhra Pradesh, India
April 2005

A thesis submitted in partial fulfillment
of the requirements for the

Master of Science Degree in Mechanical Engineering
Department of Mechanical Engineering
Howard R. Hughes College of Engineering

Graduate College
University of Nevada, Las Vegas
August 2007
Thesis Approval
The Graduate College
University of Nevada, Las Vegas

August 3, 2007

The Thesis prepared by
Satya Kiran C. Gurram

Entitled
A Numerical Prediction of Thermal Environment in a Room Heated with a
Hydronic Heater

is approved in partial fulfillment of the requirements for the degree of
Master of Science in Mechanical Engineering

Examination Committee Chair

Dean of the Graduate College

Examination Committee Member

Examination Committee Member

Graduate College Faculty Representative
ABSTRACT

A Numerical Prediction of the Thermal Environment of a Room Heated with a
Hydronic Heating System

by

Satya Kiran C. Gurram

Dr. Samir Moujaes, Examination Committee Chair
Professor, Mechanical Engineering
University of Nevada, Las Vegas

The systems in which water or steam carries heat to the areas to be heated are
Hydronic heating systems. Hydronic systems are closed loop where water is heated in a
boiler and circulated through pipes to a heat transfer component such as a radiator or
finned –tube baseboard unit. The systems can be used to effectively control the air
temperature and the mean radiant temperature of an enclosure.

Two numerical model of a room one without a window and the other with a window
using a flat panel baseboard heating systems were simulated using StarCd. The effects of
temperature on the density, viscosity and thermal conductivity were considered and
suitable user subroutines were programmed. The numerical model was simulated for
different conditions of inlet water temperatures.

The power outputs for different temperature conditions were calculated and a relation
between Ra and Nu has been developed for the flow field. A parametric study was
conducted for both the models for a wide range of temperatures to get a reasonable
understanding of the system under various temperature conditions.
Draft rate was calculated for both the models to check for thermal comfort. The calculated draft rate is much below the maximum value specified by ASHRAE of 20%.
# TABLE OF CONTENTS

ABSTRACT ........................................................................................................ iii

LIST OF FIGURES ................................................................................................ vii

LIST OF TABLES ..................................................................................................... x

ACKNOWLEDGEMENTS .......................................................................................... xi

CHAPTER 1 INTRODUCTION .................................................................................. 1
  1.1 Hydronic heating Systems ............................................................................ 2
  1.2 Advantages and Limitations ........................................................................ 2
    1.2.1 Comfort and Health ................................................................................ 2
    1.2.2 Economic ................................................................................................. 2
    1.2.3 Architectural ........................................................................................... 2
    1.2.4 Limitations .............................................................................................. 3
  1.3 Objective ........................................................................................................ 3

CHAPTER 2 BASEBOARD HEATERS ...................................................................... 4
  2.1 Baseboard Heaters ....................................................................................... 4
  2.2 Piping Systems for Baseboard Convectors ................................................ 6

CHAPTER 3 MODEL DESCRIPTION AND NUMERICAL METHOD ............... 10
  3.1 MODEL DESCRIPTION ................................................................................. 10
    3.1.1 Setup I .................................................................................................... 13
    3.1.2 Setup II ................................................................................................. 14
  3.2 Introduction to CFD simulations .................................................................. 15
  3.3 Numerical Model .......................................................................................... 20

CHAPTER 4 RESULTS AND DISCUSSIONS ...................................................... 26
  4.1 Grid Independency Results ......................................................................... 26
  4.2 CFD Simulation Results .............................................................................. 28
    4.2.1 Model I .................................................................................................. 28
      4.2.1.1 Power Output .................................................................................... 33
      4.2.1.2 Thermal Comfort ............................................................................ 33
      4.2.1.3 Raleigh Number ............................................................................. 35
      4.2.1.4 Nusselt Number ............................................................................. 35
    4.2.2 Model II .................................................................................................. 46
      4.2.2.1 Power Output .................................................................................... 50

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
4.2.2.2 Draft Rate................................................................. 51
4.2.2.3 Raleigh and Nusselt numbers................................ 51
4.3 Parametric study results for the Model I................ 60
4.4 Parametric study results for model II....................... 67
4.5 Generalized correlation between Nu and Ra.............. 73

CHAPTER 5 CONCLUSIONS ............................................... 75

REFERENCES ................................................................. 78

VITA .................................................................................... 80
LIST OF FIGURES

Figure 2.1  A Two Panel Base Board Convector ................................................................. 5
Figure 2.2  Rear views of the convector .............................................................................. 6
Figure 2.3  Bottom view of the convector ............................................................................ 6
Figure 2.4  Series loop hydronic heating system ................................................................. 7
Figure 2.5  One-Pipe System ............................................................................................... 8
Figure 2.6  Two-Pipe Reverse System ................................................................................ 9
Figure 3.1  Side view of the convector section ................................................................. 11
Figure 3.2  Isometric view of the convector section .......................................................... 12
Figure 3.3  Room with out the window ............................................................................. 14
Figure 3.4  Room with a window ...................................................................................... 15
Figure 3.5  The Numerical Model of the room .................................................................. 21
Figure 3.6  The mesh section showing the location of the convector .............................. 21
Figure 3.7  Isometric view of the convector section ......................................................... 22
Figure 3.8  Front view of the convector section ............................................................... 22
Figure 3.9  Isometric view of the complete mesh of the room indicating the location of the boundaries for Model I .................................................................. 24
Figure 3.10 Isometric view of the complete mesh of the room indicating the location of the boundaries for Model II .......................................................... 24
Figure 4.1 Grid Independency along the Y axis in the middle of the room ................. 27
Figure 4.2 Grid Independency along the Y axis 0.0762 m from the left wall ............. 29
Figure 4.3 Temperature Profile ....................................................................................... 30
Figure 4.4 Velocity Profile ............................................................................................... 31
Figure 4.5 Temperature Profile around the convector ..................................................... 32
Figure 4.6 Velocity Profile around the convector ............................................................ 33
Figure 4.7 Pathlines along the flow field ......................................................................... 33
Figure 4.8 Temperature Profile when the convector is at 76.81 °C .............................. 36
Figure 4.9 Velocity Profile when the convector is at 76.81 °C ....................................... 40
Figure 4.10 Pathlines when the convector is at 76.81 °C .................................................. 40
Figure 4.11 Velocity Profile when the convector is at 71.26 °C ..................................... 41
Figure 4.12 Temperature Profile when the convector is at 71.26 °C .............................. 41
Figure 4.13 Pathlines when the convector is at 71.26 °C .................................................. 42
Figure 4.14 Velocity Profile when the convector is at 65.7 °C ........................................ 42
Figure 4.15 Temperature Profile when the convector is at 65.7 °C ............................... 43
Figure 4.16 Temperature Profile when the convector is at 65.7 °C ............................... 43
Figure 4.17 Velocity Profile when the convector is at 60.15 °C ...................................... 44
Figure 4.18 Temperature Profile when the convector is at 60.15 °C ............................. 44
Figure 4.19 Pathlines when the convector is at 60.15 °C ............................................... 45
Figure 4.20 The section of the room where average temperature and velocity values are calculated ......................................................................................... 46
Figure 4.21 Velocity Profile of air when the convector is at 82.5 °C ..................................49
Figure 4.22 Temperature Profile of air when the convector is at 82.5 °C .................................50
Figure 4.23 Velocity Profile at the Convector ...........................................................................51
Figure 4.24 Temperature Profile at the Convector .....................................................................51
Figure 4.25 Pathlines when the Convector is at 82.5 °C ...........................................................52
Figure 4.26 Temperature Profile when the convector is at 76.81 °C ........................................54
Figure 4.27 Velocity Profile when the convector is at 76.81 °C ............................................54
Figure 4.28 Pathlines when the convector is at 76.81 °C ..........................................................55
Figure 4.29 Temperature Profile when the convector is at 71.26 °C ........................................55
Figure 4.30 Velocity Profile when the convector is at 71.26 °C ................................................56
Figure 4.31 Pathlines when the convector is at 71.26 °C ..........................................................56
Figure 4.32 Temperature Profile when the convector is at 65.7 °C ..........................................57
Figure 4.33 Velocity Profile when the convector is at 65.7 °C ................................................57
Figure 4.34 Pathlines when the convector is at 65.7 °C ...........................................................58
Figure 4.35 Temperature Profile when the convector is at 60.15 °C ...........................................58
Figure 4.36 Velocity Profile when the convector is at 60.15 °C ................................................59
Figure 4.37 Pathlines when the convector is at 60.15 °C ..........................................................59
Figure 4.38 Temperature Profile when the left wall is at 14 °C ..................................................62
Figure 4.39 Velocity Profile when the left wall is at 14 °C ..........................................................62
Figure 4.40 Pathlines when the left wall is at 14 °C .................................................................63
Figure 4.41 Temperature Profile when the right wall is at 14 °C ...............................................63
Figure 4.42 Velocity Profile when the right wall is at 14 °C ........................................................64
Figure 4.43 Pathlines when the right wall is at 14 °C ...............................................................64
Figure 4.44 Temperature Profile when the ceiling is at 14 °C .....................................................65
Figure 4.45 Velocity Profile when the ceiling is at 14 °C ............................................................65
Figure 4.46 Pathlines when the ceiling is at 14 °C .................................................................66
Figure 4.47 Temperature Profile when the floor is at 14 °C .......................................................66
Figure 4.48 Velocity Profile when the floor is at 14 °C ...............................................................67
Figure 4.49 Pathlines when the floor is at 14 °C .................................................................67
Figure 4.50 Temperature Profile when the outside wall is at 12 °C and the window is at 0 °C ..........................................................................................................................68
Figure 4.51 Velocity Profile when the outside wall is at 12 °C and the window is at 0 °C .................................................................69
Figure 4.52 Pathlines when the outside wall is at 12 °C and the window is at 0 °C .......................69
Figure 4.53 Temperature Profile when the outside wall is at 14 °C and the window is at 0 °C ..........................................................................................................................70
Figure 4.54 Velocity Profile when the outside wall is at 14 °C and the window is at 0 °C .................................................................70
Figure 4.55 Pathlines when the outside wall is at 14 °C and the window is at 0 °C .......................71
Figure 4.56 Temperature Profile when the outside wall is at 12 °C and the window is at 3 °C ..........................................................................................................................71
Figure 4.57 Velocity Profile when the outside wall is at 12 °C and the window is at 3 °C ......................72
Figure 4.58 Pathlines when the outside wall is at 12 °C and the window is at 3 °C.............72

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.59  Temperature Profile when the outside wall is at 12 °C and the window is at -3 °C ................................................................. 73

Figure 4.60  Velocity Profile when the outside wall is at 12 °C and the window is at -3 °C ................................................................. 73

Figure 4.61  Pathlines when the outside wall is at 12 °C and the window is at -3 °C ................................................................. 74

Figure 4.62  Plot between Nu and Ra .................................................. 75
LIST OF TABLES

Table 4.1 Mean Temperature, Velocity and DR values for Model I ......................... 43
Table 4.2 Mean Heat Outputs for Model I ................................................................. 44
Table 4.3 Ra, h and Nu values for Model I ............................................................... 44
Table 4.4 Mean Temperature, Velocity and DR values for Model II ..................... 58
Table 4.5 Heat Output Values for Model II .............................................................. 58
Table 4.6 Ra, h and Nu values for Model II ............................................................. 59
ACKNOWLEDGEMENTS

I would like to sincerely thank Dr. Samir Moujaes, advisor and committee chair, for overseeing my progress as a graduate student. I would like to thank Dr. Mohamed Trabia, Dr. Nabil Nassif and Dr. Yahia Baghzouz for having accepted the task of serving on my thesis committee. I would like to thank my friends Dhandapani Selvaraj and Ashok Kumar for their support.

Finally I would like to thank my parents and sister for their consistent support and encouragement throughout my life.
CHAPTER 1

INTRODUCTION

1.0 Hydronic Heating Systems

The systems in which water or steam carries heat to the areas to be heated are Hydronic heating systems. Hydronic systems are closed loop where water is heated in a boiler and circulated through pipes to a heat transfer component such as a radiator or finned–tube baseboard unit. Hydronic radiant baseboard heating is a common application that has been used for over 50 years in the United States (NBPM 2001).

The most commonly hydronic heating systems are floor radiators, wall panel radiators, and baseboard convectors. Floor and wall panel radiators have bank of tubes or a panel embedded in the floor or the walls through which hot water flows, where as a baseboard convector has tubes with fins running along the walls at a small height above the floor. These base board convectors are not embedded in structure of the building but are located outside of the interior walls (Shoemaker 1948; Bjarne 2000; Takemasa et al, 1992).

1.1 Advantages and Limitations

Hydronic heating systems are generally recognized for several advantages: provision of thermal comfort, energy savings potential, and compatibility with various energy sources, quietness and a broad scope of application (Leigh 1991). The study of hydronic radiating systems can be divided into four sub studies: comfort and health, economic, architectural and limitations (Raber et al 1947).
1.1.1 Comfort and Health

One of the major reasons for renewed interest in the hydronic heating systems is that such a system can provide approach to thermal comfort. Berguland and Gagge (1985) conducted a study to quantify the thermal sensations felt by the body, local discomfort, comfort and the general acceptability of the thermal environment when heated with four different heating systems. The results showed that the occupant’s preference rankings of the four systems from best to worst was floor heating, baseboard, forced air and ceiling system.

1.1.2 Economic

It has been reported that hydronic radiators can reduce energy costs up to 30 percent or more when compared with forced air systems for the same thermal comfort (Lenman 1988; Buckley, 1989).

1.1.3 Architectural

All radiant hydronic systems provide an alternative to large-scale air-handling systems. This impacts many aspects of the building design including the required plenum sizing, boiler/chiller sizing, ducting, etc (NBPM 2001).

1.1.4 Limitations

These systems cannot maintain required thermal comfort when the thermal loads are excessive (Leigh, 1991). These systems are also not suitable for buildings with high ceilings as all the warm air tends to accumulate in the top of the room.

1.2 Objective

Several previous research works have been focused on under floor radiators and wall panel radiators (Dorval 1956; Olesen 2000), not much information is available on the
baseboard convectors in the current literature (Li et al.). So, a complete numerical analysis on the performance and efficiency of baseboard convectors has been performed and a relation between dimensionless quantities Rayleigh number (Ra) and Nusselt number (Nu) has been developed which can be used to predict the performance of the convector for different temperature conditions.
2.1 Baseboard Heaters

The most common types baseboard heaters are the finned-tube convector and radiant convector, both of which heat cold air at the floor level of the room and induce an upward convective current. This is extremely effective in reducing downdrafts at cold facades and under windows. These systems provide heat through a combination of convection and radiation. Another application is the panel, or flat pipe, radiator.

These systems are applicable in all areas experiencing extreme cold, and are especially effective in areas of significant heat loss, such as entryways or under windows. The advantages of baseboard heaters are:

- Hydronic systems can decrease or eliminate the need for mechanical air-handling systems.
- The systems are low maintenance.
- They can be more fuel efficient than forced air systems and use less power to move heat through the air conditioned space.
- They are quiet.
- Cold downdraffs at outside walls and windows are minimized by suitable use of these convectors (NBPM 2001).
The baseboard heater considered for the analysis has two flat panel tubes running along the length of a room. The water from the boiler is fed into the flat panel tubes at one end. The water as it travels through the tubes loses heat to the convector which in turn loses heat to the air outside. These tubes are backed by a series of fins which help in increasing the heat transfer rate from the convector. These convectors are placed close to the floor about 0.05 m high, and run along a wall's length in a room. In most of the cases the convectors are directly placed beneath a window in the room. This is done in order to overcome the cold downward draft from the cold window surface.

A commercially available boiler is used to heat the water required for convector circulation. The Figures 2.1, 2.2, 2.3 show the different views of a commercially produced two tube baseboard convector.
Figure 2.2 Rear views of the convектор

Figure 2.3 Bottom view of the convектор
2.2 Piping Systems for Baseboard Convectors

i) Series loop:

Most residential systems use a series loop or one pipe system layout for each zone. It is a continuous run of constant sized pipe from a supply connection to a return one. Figure 2.4 shows a series of two loops on a supply and a main return.

In this system the circulating hot water passes through each consecutive baseboard unit in series and the cooled water returns to the boiler. A limitation of using this system is that if one baseboard unit is shut off it prevents the flow to the remainder of the loop.

Figure 2.4 Series loop hydronic heating system
ii) One-Pipe System:

Figure 2.5 illustrates a one pipe system arrangement consisting of one pipe that is looped around the building and is both the supply and return main. At each baseboard unit a supply and return tee are installed on the main. A special diverting tee is used on the return side of the baseboard which creates a pressure drop in the main flow to divert part of the flow through the terminal unit and its branch circuit.
iii) Two-Pipe Reverse System:

Figure 1.6 illustrates a basic two pipe reverse return system. In this system the return main flows in the same direction as supply main flow and the return main returns all the water to the boiler after the last heater is fed. A reverse return system seldom need balancing valves because water flow distance to and from the boiler is virtually same through any unit (Clifford 1990; Whitman et al 2000).
CHAPTER 3

MODEL DESCRIPTION AND NUMERICAL METHOD

This Chapter provides the information on the working theory behind the simulation package used to model the convector. The chapter is split into two sub chapters, the first chapter gives a detailed description of the physical model of the convector used for simulation and the second part describes the numerical model and the computational method used for the simulation of the problem.

3.1 Model Description

In order to save computational time the forced convection effect of the water entering the convector is not considered and a constant temperature boundary condition is used on the inside walls of the convector. A 3-D section of the convector is considered which includes a single modular section of a fin and the space between it and the next two fins on either side. Hence in the X-Y plane the limits of the solution extend to all the walls/ceiling/floor of the assumed room size 3x3x3 m. A plane of symmetry parallel to the X-Y plane and situated in the middle of the fine is considered. The extent of the solution field in the z-direction is 0.02m, while it is an area of 3x3m in the X-Y plane. The assumption here is that this modular pattern of the finned convector is repetitive all along the Z-direction of the room. The end effect of the walls normal to the z-direction are neglected and the assumption that the temperature of the water through the convector
from supply to return does not vary by more than 20 °C which can be neglected to simplify the investigation of this computational problem. Hence the temperatures chosen for the water in the different simulations can be considered as an average value between supply and return water streams to the convector.

The convector is placed 0.05 m above the floor along the length of one of the walls as recommended by the manufacturer. The convector has two flat panel tubes running along the length. These tubes have a series of fins along their length. The section of the convector considered for the analysis is shown in Figures 3.1 and 3.2.
Figure 3.1 Side view of the convector section
Water from a conventional boiler enters these tubes at one end and returns to the boiler at the other end. The water enters the boiler at a temperature of 60°C and leaves at a temperature of 82.2°C resulting in a temperature difference of 22.8°C. This heat is lost from the convector to the air in the conditioned space through natural convection. A
small percentage of it is radiated to the surroundings in the room of about 8% when compared with the total heat output given by the manufacturer.

The material of the convector considered for the analysis is carbon steel. The properties of carbon steel are

Density = 7832 kg/m$^3$

Conductivity = 63.9 W/m-K

Specific Heat = 434 J/kg-K

Two different setups were considered for the study.

1) Setup I - An interior room with no window, and

2) Setup II - An external room with one external wall and a window included on that wall.

3.1.1 Setup I

In this setup all the walls of the room are assumed to be at a constant temperature (partitions) and the convector is located at a height of 0.05 m from the floor. A physical sketch of the setup is represented in the Figure3.5.
3.1.2 Setup II

In this setup a window of 0.9 m height is assumed to run along the length of the room is directly located on the wall directly above the convector. The temperature of the window is assumed to be constant at 0 °C. Eventually a parametric study was performed by varying the temperatures of the window and the external wall. A physical sketch of the setup is represented in Figure 3.6.

Figure 3.5 Room with out the window

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
3.2.1 Introduction to CFD simulations

All the simulations were performed using STARCD. This software comprises of the main analysis code, and the pre-processor and post-processor codes. STAR-CD is a powerful CFD tool for thermo fluids analysis and has been designed for use in a CAE environment. Its many attributes include:

Figure 3.6 Room with a window
1) A self-contained, fully-integrated and user-friendly program suite comprising pre-processing, analysis and post-processing facilities

2) A general geometry-modeling capability that renders the code applicable to the complex shapes often encountered in industrial applications

3) Extensive facilities for automatic meshing of complex geometries, either through built-in tools (such as Proam) or through interfaces to external mesh generators such as ICEM CFD Tetra built-in models of an extensive and continually expanding range of flow phenomena, including transients, compressibility, turbulence, heat transfer, mass transfer, chemical reaction and multi-phase flow

4) Fast and robust computer solution techniques that enhance reliability and reduce computing overheads

5) Easy-to-use facilities for setting up and running very large CFD models using state-of-the-art parallel computing techniques

6) Built-in links with popular proprietary CAD/CAE systems, including PATRAN, IDEAS and ANSYS.

STAR operates by solving the governing differential equations of the flow physics by numerical means on a computational mesh. PROSTAR is an interactive, command driven, combined pre-processor and post-processor whose main functions include geometry modeling, mesh generation, problem specification, results manipulation and display, and links to external CAD/CAE systems.

The governing equations used by STAR to solve the problem are given below. The mass and momentum conservation equations solved by STAR-CD for general
incompressible and compressible fluid flows and a moving coordinate frame (the ‘Navier Stokes’ equations) are, in Cartesian tensor notation.

Mass Conversation:

\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} \rho \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u}_j \right) = s_m
\]

Momentum Conservation:

\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} \rho u_i \right) + \frac{\partial}{\partial x_i} \left( \rho \bar{u}_j u_i - \tau_{ij} \right) = - \frac{\partial p}{\partial x_i} + s_i
\]

Where

t-time

\(x_i\) -- Cartesian coordinate \((i = 1, 2, 3)\)

\(u_i\) -- Absolute fluid velocity component in direction

\(\bar{u}_j\) -- Relative velocity between fluid and local (moving) coordinate frame

\(p\) -- Piezometric pressure \(= p_s + \rho g \text{m} \), where \(p_s\) is static pressure, \(\rho_0\) is reference density, \(g\) are gravitational field components and the are coordinates from a datum, where \(\rho_0\) is defined

\(\rho\) -- Density

\(\tau_{ij}\) -- Stress tensor components

\(S_m\) -- Mass source

\(S_i\) -- Momentum source components

\(\sqrt{g}\) -- Determinant of metric tensor

This specialization of the above equations to a particular class of flow involves:

- Application of ensemble or time averaging if the flow is turbulent.
• Specification of a constitutive relation connecting the components of the stress tensor to the velocity gradients.

• Specification of the source, \( s_n \), which represents the sum of the body and other external forces, if present.

For turbulent flows, \( u_n, p \) and other dependent variables, including, assume their ensemble averaged values (equivalent to time averages for steady-state situations) giving, for Newtonian fluids:

\[
\tau_{ij} = 2\mu s_{ij} - \frac{2}{3} \mu \frac{\partial u_k}{\partial x_k} \delta_{ij} - \rho \bar{u}_i' \bar{u}_j'
\]

Where the \( u' \) are fluctuations about the ensemble average velocity and the overbar denotes the ensemble averaging process. The rightmost term in the above represents the additional Reynolds stresses due to turbulent motion. These are linked to the mean velocity field via the turbulence models.

Heat transfer in STAR-CD is implemented through the following general form of the enthalpy conservation equation for a fluid mixture

\[
\frac{1}{\sqrt{g}} \left( \sqrt{g} \rho h \right) + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_j h - F_{h,j} \right) = \frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} p \right) + \frac{\partial}{\partial x_j} \left( \bar{u}_j p \right) - p \frac{\partial \bar{u}_j}{\partial x_j} + \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} + s_i
\]

Here, \( h \) is the static enthalpy defined by

\[
h \equiv \bar{c}_p T - \bar{c}_p T_0 + \sum m_m H_m = h_i + \sum m_m H_m
\]

And,

\( T \) — temperature

\( m_m \) — mass fraction of mixture constituent \( m \)
$H_m$ — heat of formation of constituent $m$

$C_p$ — mean constant-pressure specific heat at temperature $T$

$F_{h,i}$ — diffusional energy flux in direction

$S_h$ — energy source

$h_t$ — thermal enthalpy

It should be noted that the static enthalpy $h$ is defined as the sum of the thermal and chemical components. The chemical components are neglected here as they are not related to the current analysis.

The thermal enthalpy is given by:

$$\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} \rho h_t \right) + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_j h_t - F_{h,j} \right) =$$

$$\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} p \right) + \frac{\partial}{\partial x_j} \left( \bar{u}_j p \right) - p \frac{\partial u_j}{\partial x_j} + \tau_{yj} \frac{\partial u_j}{\partial x_j} + s_b - \sum H_m S_{e,j}$$

Here, $h_t$ is the thermal enthalpy, defined by

$$h_t = \bar{c}_p T - c''_p T_p$$

The total thermal enthalpy is given by the sum of mechanical energy conservation and static enthalpy, and is represented as

$$\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} \rho H \right) + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_j H - F_{h,j} - u_j \tau_{e,j} \right) = \frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} p \right) - \frac{\partial}{\partial x_j} \left( \bar{u}_j p \right) + s_j u_j + s_h$$

Where

$$H = \frac{1}{2} u_j u_j + h$$
In problems involving natural or free convection, redistribution of energy is mainly due to the force of gravity acting on a fluid of non-uniform density and causing fluid motion [StarCd Manual 2003].

3.2.2 Numerical Model

Two numerical models were considered. They are

1. Model I - An interior room, and
2. Model II - An exterior room with a window.

Both the numerical models were identical except for the boundary conditions used. The 3-D numerical model consists of 270,000 nodes. The 3-D mesh was created in StarCd using the pro-Star module. A tetrahedral type of mesh was used to generate the mesh. The Figures 3.8 to 3.10 show the generated mesh. As the velocity of the air particles is maximum along the walls and the draft from the window collides with the warm air at the left wall, a high grid density is used along the four walls of the room.
Figure 3.7 The Numerical Model of the room

Figure 3.8 The mesh section showing the location of the convector
Figure 3.9 Isometric view of the convecto section

Figure 3.10 Front view of the convecto section
Both models are simulated in a steady state mode. The boundary conditions for the model are as follows

1) Convecto: The inside walls of the convecto are maintained at a constant temperature. This boundary condition represents the flow of the hot water in the convecto. The different average water temperatures that were used are 82.5 °C, 76.8 °C, 71.26 °C, 65.7 °C and 60.15 °C. This covers a relatively wide range of temperatures for the operation of the convecto.

2) Walls: All the walls of the room are maintained at a constant specified temperature of 20°C because they are considered interior walls i.e. partitions with adjacent rooms heated in a similar way.

3) Window: A boundary condition to represent a window was added to the model. This window is maintained at a constant temperature lower than the ambient room air temperature to represent the winter climatic conditions where in general windows have a higher heat transfer coefficient than walls and hence exhibit a lower temperature than adjacent walls.

The Figures 3.11 and 3.12 indicates the boundary conditions.
Figure 3.11 Isometric view of the complete mesh of the room indicating the location of the boundaries for Model I.
The numerical scheme used for both the modes is as follows

1) As natural convection is the dominant form of heat transfer through the air a Boussinesq approximation was considered for the density in the momentum equation. The basic approach in this approximation is to treat the density as a constant in the continuity equation and the inertia term of the momentum equation, but allow it to change temperature in the gravity term [Latif 2006]. The Boussinesq approximation is given by the equation
\[ \frac{D\vec{V}}{Dt} = -\beta \vec{g}(T - T_s) - \frac{1}{\rho_s} \nabla (p - p_s) + \nu \nabla^2 \vec{V} \]

Where,

- \( \vec{V} \) -- Velocity,
- \( T \) -- Surface temperature,
- \( T_s \) -- Free stream temperature,
- \( \beta \) -- Coefficient of thermal expansion,
- \( \nu \) -- Kinematic viscosity,
- \( \rho \) -- Density,
- \( (p - p_s) \) -- Pressure difference,
- \( g \) -- Gravitational constant

2) Both conductivity and molecular viscosity play an important role in the heat transfer in a fluid. These properties for air vary between 5-20 \(^\circ\)C for the temperature range between 0-100 \(^\circ\)C. Considering these values to be constant will result in some discrepancies in the results obtained. So, suitable subroutines or programs must be written during the computational analysis defining them as the functions of temperature.

3) A low Reynolds number model k-\( \varepsilon \) turbulence model is used as it considered most suitable for natural convection problems in an enclosure [Chen 1995].

4) PISO algorithm was used to solve the buoyancy effects.

5) AMG (Algebraic Multigrid) approach was used for solving matrix equations.

6) The numerical simulations were run in steady-state mode with a residual tolerance of 10E-4.
7) Each simulation was run on multiple processors using STAR-HPC.
CHAPTER 4

RESULTS AND DISCUSSIONS

This chapter provides grid independency results and a detailed overview of all the results from simulations performed on the numerical models of the room section in question with and without the window placement in the room.

4.1 Grid Independency Results

In order to save computational time a grid with the least number of cells giving the most accurate results must be selected for the analysis. To achieve this layout grids with different densities are created and the results are compared with the results obtained from the highest density grid. The lower of two grid densities with a sufficiently small difference of about 2-3 % in their solution accuracy of some of the variables involved is selected for the analysis.

The grid independency check was carried out for the Y components of velocity along the vertical axis in the center of the solution field and also at a distance of 0.07m from the left wall where the velocity values are expected to be highest. The simulations were performed on three different grids of different sizes. The numbers of cells in each of the grids were:

Mesh 1 -Finest: 310,000
Mesh2 -Fine: 270,000
Mesh3 -Coarse: 140,000
It was found out the average variation in velocity for the Y component in the middle of the room between mesh-1 and mesh-2 is about 2% from Figure 4.1 and the average variation of velocity for the Y component is about 3% with a distance of 0.0762 m from the left wall from Figure 4.2. As there is little variation the results of grid-2 considered to give sufficiently accurately numerical results for all the numerical calculations are used from now on in future simulation results.

Figure 4.1 Grid Independency along the Y axis in the middle of the room
4.2.0 CFD Simulation Results

A steady state analysis was conducted ignoring the radiation effects as it has already been mentioned in section 3.1.1 that the percentage of heat transfer through radiation when compared with the heat transfer by natural convection is only about 8%. The first section of the discussions shows the effect that the temperature of the convector has on the velocity and temperature distribution in an interior room i.e. Model I. The second section of the discussions shows the resultant effects of having a window directly above the convector at a height of 0.9 m from the floor on the velocity and temperature distribution i.e. Model II. A residual tolerance of 10e-4 was used for all the simulations.

4.2.1 Model I

The simulation results show the temperature and velocity profiles of the air enclosed in the room and the effect of variation of the temperature of the convector on the above two factors. Different values of temperatures that were used on the inside walls of the convector are 82.5 °C, 76.8 °C, 71.26 °C, 65.7 °C and 60.15 °C. The walls of the room are
kept at a constant temperature of 20 °C. A parametric study was performed by changing the temperature of one of the walls to 14 °C indicating an outside wall, while keeping the other walls at a constant temperature of 20 °C. The initial temperature of the air before the analysis is assumed to be 20 °C.

Figures 4.3 and 4.4 show the temperature and velocity profiles of the air when the temperature of the convecto is at 82.5 °C.

From Figure 4.3 it can be seen that the average temperature at the center of the room is 24.5 °C indicating the rise in temperature of 4.5 °C. It is observed that the temperature
is almost uniform throughout the room. The only variation that occurs is along the left wall of the room resulting from the warm upward draft from the convector.

![Figure 4.4 Velocity profile](image)

As the temperature of air increases it rises upward due to the buoyancy effect and the flow of the air can be observed in Figure 4.4. From Figure 4.4 it can be clearly observed that the air gains heat from the convector and moves upward towards the ceiling and flows along the walls and comes back to the convector. We can see that there is a recirculation zone in the center of the room resulting from the low temperatures of the walls at A and a smaller recirculation at B.
Figures 4.5 and 4.6 show the temperature and velocity distribution in the proximity of the convector.

![Figure 4.5 Temperature profile around the convector](image)

The enclosure marked with the black line in Figure 4.5 indicates the location of the convector. From figure 4.6 it can be observed that some of the air particles returning to the convector collide with the warm upward stream where as some of the air particles return back to the convector completely, get heated and form the upward current.
The Figure 4.7 represents the pathlines at a certain points in the flow. Pathlines are the trajectory that a fluid particle would make as it moves around with the flow. These pathlines give a clear indication of the path traversed by the different air particles. The arrows in Figure 4.7 indicate the direction of the flow of air particles.
4.2.1.1 Power Output

The power output from the convector is obtained from the simulation results as 613.6 W/m. The power output rating given by the manufacturer experimentally is 576.77 W/m when the inlet water temperature is 82.5 °C. The percentage difference between the two values is about 6 indicating that the results obtained are fairly accurate. No details on how the complete testing procedure was done by the manufacturer is available, so it is not known as to what exactly were the surround wall temperatures in the test chamber used.
4.2.1.2 Thermal Comfort

According to ASHRAE standards-55 2004, "draft is the unwanted local cooling of the body caused by air movement " and "draft rate (DR) is the percentage of people predicted to be dissatisfied due to draft ".

This Draft Rate (DR) is given by

\[ DR = ([34 - t_a] + [v - 0.05]^{0.62}) + (0.37 * v * Tu + 3.14) \]

Where

\( t_a \) = ambient temperature, °C,

\( v \) = local mean air speed, m/s,

\( Tu \) = Turbulence Intensity, %.

For the present case when the temperature of the convector is at 82.5 °C, the mean ambient air temperature is calculated to be 26.22 °C and the local mean air velocity to be 0.1592 m/s.

The turbulence intensity \( l \), also often referred to as turbulence level [9], is defined as:

\[ Tu = \frac{\bar{u}}{U} \]

Where

\( U \) = Velocity magnitude

\( \bar{u} \) - root-mean-square of the turbulent velocity fluctuations

\[ u' = \sqrt{\frac{1}{3}(u_x^2 + u_y^2 + u_z^2)} = \frac{\sqrt{2}}{\sqrt{3}} k \]

k- Turbulent energy, \( k=0.419 \) for low Reynolds number model.

The draft rate is calculated to be \( DR= 4 \% \). According to ASHRAE standards 55-2004 the acceptable value for DR should be less than 20%.
This indicates that the percentage of people dissatisfied with the current setup is about 4%, which is a fairly low number.

4.2.1.3 Rayleigh Number

Rayleigh number for a fluid is a dimensionless number associated with the heat transfer within the fluid. When the Rayleigh number is below the critical value for that fluid, heat transfer is primary in the form of conduction and when it exceeds the critical value, heat transfer is primarily in the form of convection.

\[ Ra = \frac{g\beta}{\nu\alpha} (T_s - T_{\infty}) L^3 \]

Where,
Ra = Rayleigh number

\( g = \) acceleration due to gravity
L = Characteristic length
Ts = Surface temperature
Tr = Quiescent temperature
\( \nu = \) Kinematic viscosity
\( \alpha = \) Thermal diffusivity
\( \beta = \) Thermal expansion coefficient

The Rayleigh number for the current simulation when the inlet water temperature is at 82.5 °C is calculated to be Ra=1.049E11

4.2.1.4 Nusselt Number

Nusselt number (Nu) is a dimensionless number used to measure the enhancement of heat transfer when convection takes place.
\[ Nu_i = \frac{hL}{k_i} \]

Where,
\( L \) = characteristic length,
\( k_i \) = thermal conductivity of the fluid
\( h \) = convection heat transfer coefficient (Incropera et al, 1998).

Heat transfer coefficient is calculated using the equation

\[ h = \frac{Q}{A(T_s - T_x)} \]

Where,
\( Q \) = Heat output,
\( A \) = Area,
\( T_s \) = Surface temperature and
\( T_x \) = Average ambient temperature.

Nusselt number for the current simulation is calculated to be \( Nu = 43.47 \)

The same analysis was conducted using different temperatures for the convector which fall under its operating range. The different temperatures that were used are 76.8 \(^\circ\)C, 71.26 \(^\circ\)C, 65.7 \(^\circ\)C and 60.15 \(^\circ\)C. The results obtained were quite similar to the results shown in the previous analysis.

The velocity and temperature profiles for these simulations are represented in the following figures from Figure 4.8 to Figure 4.19.
Figure 4.8 Temperature profile when the convector is at 76.81 °C

Figure 4.9 Velocity profile when the convector is at 76.81 °C
Figure 4.10 Pathlines when the convector is at 76.81 °C

Figure 4.11 Velocity profile when the convector is at 71.26 °C
Figure 4.12 Temperature profile when the convector is at 71.26 °C

Figure 4.13 Pathlines when the convector is at 71.26 °C
Figure 4.14 Velocity profile when the convector is at 65.7 °C

Figure 4.15 Temperature profile when the convector is at 65.7 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.16 Pathlines when the convector is at 65.7 °C

Figure 4.17 Velocity profile when the convector is at 60.15 °C
Figure 4.18 Temperature profile when the convector is at 60.15 °C

Figure 4.19 Pathlines when the convector is at 60.15 °C
Average temperature and velocity values were calculated for all the simulations in the section of the cells shown in Figure 4.20. This section approximately represents the living space in the room i.e., 1.8 m below the floor. The temperature, velocity and the corresponding draft rate values are given in Table 4.1

Figure 4.20 The section of the room where average temperature and velocity values are calculated
Table 4.1 Mean Temperature, Velocity and DR values

<table>
<thead>
<tr>
<th>Convector Temperature °C</th>
<th>Mean air temperature °C</th>
<th>Mean air velocity m/s</th>
<th>Draft Rate(DR) %</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>26.22</td>
<td>0.159</td>
<td>4.03</td>
</tr>
<tr>
<td>76.81</td>
<td>25.866</td>
<td>0.156</td>
<td>4.14</td>
</tr>
<tr>
<td>71.26</td>
<td>25.4</td>
<td>0.149</td>
<td>4.32</td>
</tr>
<tr>
<td>65.70</td>
<td>24.07</td>
<td>0.121</td>
<td>5.33</td>
</tr>
<tr>
<td>60.15</td>
<td>23.43</td>
<td>0.101</td>
<td>6.04</td>
</tr>
</tbody>
</table>

The heat outputs for the convector for different temperatures calculated from the CFD simulations are given in the Table 4.2

Table 4.2 Mean Heat Outputs

<table>
<thead>
<tr>
<th>Temperature °C</th>
<th>Heat Outputs W/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>613.6</td>
</tr>
<tr>
<td>76.81</td>
<td>520.1</td>
</tr>
<tr>
<td>71.26</td>
<td>471.9</td>
</tr>
<tr>
<td>65.70</td>
<td>415.8</td>
</tr>
<tr>
<td>60.15</td>
<td>342.0</td>
</tr>
</tbody>
</table>
Table 4.3 Ra, h and Nu values

<table>
<thead>
<tr>
<th>Temperature °C</th>
<th>Ra</th>
<th>h W/m²-°C</th>
<th>Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>1.04E11</td>
<td>8.90</td>
<td>43.47</td>
</tr>
<tr>
<td>76.81</td>
<td>9.09E10</td>
<td>8.60</td>
<td>42.09</td>
</tr>
<tr>
<td>71.26</td>
<td>8.18E10</td>
<td>8.30</td>
<td>40.30</td>
</tr>
<tr>
<td>65.70</td>
<td>8.01E10</td>
<td>8.04</td>
<td>39.90</td>
</tr>
<tr>
<td>60.15</td>
<td>7.06E10</td>
<td>7.60</td>
<td>36.52</td>
</tr>
</tbody>
</table>

Rayleigh Number (Ra), heat transfer (h) coefficient and Nusselt Number (Nu) for the different temperature ranges were calculated and tabulated in the Table 4.3.

4.2.2 Model II

The simulation results shows the temperature and velocity profiles of the air enclosed in the room with a window and the effect of variation of the temperature of the convector on the above two factors. Different values of temperatures used on the inside walls of the convector are 82.5 °C, 76.81 °C, 71.26 °C, 65.7 °C and 60.15 °C. The walls of the room are kept at a constant temperature of 20 °C.

The window is maintained at a temperature of 0 °C. A parametric study was performed by changing the temperature of the outside wall (The wall with the window.) to a range of temperatures from 5 °C to 14 °C while keeping the other walls at a constant temperature of 20 °C. The initial temperature of the air before the analysis is assumed to be 20 °C to start the simulation process.
Figures 4.21 and 4.22 show the temperature and velocity profiles of the air when the temperature of the convector is at 82.5 °C.

![Figure 4.21 Velocity profile of air when the convector is at 82.5 °C.](image)

From the Figure 4.21 it can be observed that there is a cold downward draft caused by the low temperature of the window. This downward draft collides with the warm air moving upwards from the convector and gets diverted. This effectively results in keeping the part of the room which is below the window warm.
Figure 4.22 Temperature profile of air when the convектор is at 82.5 °C.

From Figure 4.22 it can observe that the temperature distribution is not uniform. The hot air moving upwards collides with the cold downward draft and gets diverted laterally resulting in the temperature distribution shown in the Figure.

The Figures 4.23 and 4.24 show the temperature and velocity profiles around the convектор.
Figure 4.23 Velocity profile at the Convector

Figure 4.24 Temperature profile at the Convector
Figure 4.25 shows the pathlines along the flow field. It is clearly observed that there is a recirculation region of cool air in the top left-side of the room.

![Pathlines when the Convector is at 82.5 °C](image)

Figure 4.25 Pathlines when the Convector is at 82.5 °C

4.2.2.1 Power Output

The power output from the convector is obtained from the simulation results as 605.8 W/m. The power output rating given by the manufacturer is 586 W/m when the inlet water temperature is 82.5 °C. The percentage difference between the two values is about 5 % and hence it can be conclude that the results obtained are fairly accurate.
4.2.2.2 Draft Rate

The average temperature and velocity values for the analysis were found to be 24.1 °C and 0.11 m/s respectively. The draft rate is calculated to be 5.84 %.

This means that 5.84 % of the population will be dissatisfied with this setup which is reasonable.

4.2.2.3 Rayleigh and Nusselt Numbers

Using the methods described in sections 4.1.2.3 and 4.1.2.4 Rayleigh and Nusselt number are calculated for the current simulation as \( Ra = 1.089E11 \) and \( Nu = 44.73 \).

The same analysis was conducted using different temperatures for the convector which fall under its operating range. The different temperatures that were used are 82.5 °C, 76.8 °C, 71.2 °C, 65.7 °C and 60.15 °C. The results obtained were quiet similar to the results shown in the previous analysis.

The velocity and temperature profiles for these simulations are represented in the following figures from Figure 4.26 to Figure 4.37.
Figure 4.26 Temperature profile when the convector is at 76.81 °C

Figure 4.27 Velocity profile when the convector is at 76.81 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.28 Pathlines when the convector is at 76.81 °C

Figure 4.29 Temperature profile when the convector is at 71.26 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.30 Velocity profile when the convector is at 71.26 °C

Figure 4.31 Pathlines when the convector is at 71.26 °C
Figure 4.32 Temperature profile when the convector is at 65.7 °C

Figure 4.33 Velocity profile when the convector is at 65.7 °C
Figure 4.34 Pathlines when the convector is at 65.7 °C

Figure 4.35 Temperature profile when the convector is at 60.15 °C
Figure 4.36 Velocity profile when the convector is at 60.15 °C

Figure 4.37 Pathlines when the convector is at 60.15 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Average temperature and velocity values were calculated for all the simulations in the section of the cells shown in Figure 4.20.

The temperature, velocity and the corresponding draft rate values are given in Table 4.4.

**Table 4.4 Mean Temperature, Velocity and DR values**

<table>
<thead>
<tr>
<th>Convecter Temperature °C</th>
<th>Mean air temperature °C</th>
<th>Mean air velocity m/s</th>
<th>Draft Rate( DR) %</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>24.19</td>
<td>0.117</td>
<td>5.84</td>
</tr>
<tr>
<td>76.81</td>
<td>23.79</td>
<td>0.100</td>
<td>5.38</td>
</tr>
<tr>
<td>71.26</td>
<td>23.07</td>
<td>0.092</td>
<td>5.18</td>
</tr>
<tr>
<td>65.70</td>
<td>22.35</td>
<td>0.090</td>
<td>5.35</td>
</tr>
<tr>
<td>60.15</td>
<td>21.04</td>
<td>0.084</td>
<td>5.41</td>
</tr>
</tbody>
</table>

**Table 4.5 Heat Output Values**

<table>
<thead>
<tr>
<th>Temperature °C</th>
<th>Heat Outputs W/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>605.8</td>
</tr>
<tr>
<td>76.81</td>
<td>596.5</td>
</tr>
<tr>
<td>71.26</td>
<td>10.48</td>
</tr>
<tr>
<td>65.70</td>
<td>524.08</td>
</tr>
<tr>
<td>60.15</td>
<td>371.40</td>
</tr>
</tbody>
</table>
The heat outputs for the convector for different temperatures calculated from the CFD simulations are given in the table 4.5.

Ra, heat transfer coefficient and Nu for the different temperature ranges were calculated and tabulated in the table 4.6.

<table>
<thead>
<tr>
<th>Temperature °C</th>
<th>Ra</th>
<th>h W/m²·°C</th>
<th>Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.50</td>
<td>1.089E11</td>
<td>8.66</td>
<td>44.7</td>
</tr>
<tr>
<td>76.81</td>
<td>9.466E10</td>
<td>8.34</td>
<td>42.0</td>
</tr>
<tr>
<td>71.26</td>
<td>8.626E10</td>
<td>8.15</td>
<td>39.9</td>
</tr>
<tr>
<td>65.70</td>
<td>7.774E10</td>
<td>7.66</td>
<td>38.7</td>
</tr>
<tr>
<td>60.15</td>
<td>6.982E10</td>
<td>7.40</td>
<td>34.4</td>
</tr>
</tbody>
</table>

4.3 Parametric study results for the Model 1 (Room without the window)

A parametric study was conducted on model 1 keeping the temperature of the convector constant at 65.7°C and the temperatures of the left, right, top and bottom walls were changed to 14 °C for each simulation while keeping the other walls at a constant temperature of 20 °C, which means that the adjacent rooms to these walls are unoccupied and unheated at that time.

The results (velocity and temperature profiles) for each simulation are given in the following sections.
Figure 4.38 Temperature profile when the left wall is at 14 °C

Figure 4.39 Velocity profile when the left wall is at 14 °C
Figure 4.40 Pathlines when the left wall is at 14 °C

Figure 4.41 Temperature profile when the right wall is at 14 °C
Figure 4.42 Velocity profile when the right wall is at 14 °C.
Figure 4.43 Pathlines when the right wall is at 14 °C

Figure 4.44 Temperature profile when the ceiling is at 14 °C

Figure 4.45 Velocity profile when the ceiling is at 14 °C
Figure 4.46 Pathlines when the ceiling is at 14 °C

Figure 4.47 Temperature profile when the floor is at 14 °C
Figure 4.48 Velocity profile when the floor is at 14 °C

Figure 4.49 Pathlines when the floor is at 14 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
From figures 4.38 to 4.46 i.e., for the simulations where the temperatures of left, top and right walls are changed to 14 °C, the temperature, velocity profiles and the pathlines are almost similar to each other. For the simulation where the temperature of the floor is at 14 °C, it is observed that when the air particles move down along the right wall they collide with the cool air particles on the floor and tend to move upward due to the difference in densities between the hot and cold air particles. This phenomenon can be observed in the velocity profile in Figure 4.48 and the pathlines in Figure 4.49.

4.4 Parametric study results for model II (Room with the window)

A parametric study was performed on this model by changing the temperature values of both the left (outside wall) and the window. Different combinations of temperature are used to get a reasonable understanding of the behavior of the system under different conditions. The results for the different conditions are given in the following sections.

Figure 4.50 Temperature profile when the outside wall is at 12 °C and the window is at 0 °C

68
Figure 4.51 Velocity profile when the outside wall is at 12 °C and the window is at 0 °C

Figure 4.52 Pathlines when the outside wall is at 12 °C and the window is at 0 °C
Figure 4.53 Temperature profile when the outside wall is at 14 °C and the window is at 0°C

Figure 4.54 Velocity profile when the outside wall is at 14 °C and the window is at 0 °C
Figure 4.55 Pathlines when the outside wall is at 14 °C and the window is at 0 °C

Figure 4.56 Temperature profile when the outside wall is at 12 °C and the window is at 3 °C
Figure 4.57 Velocity profile when the outside wall is at 12 °C and the window is at 3 °C

Figure 4.58 Pathlines when the outside wall is at 12 °C and the window is at 3 °C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.59 Temperature profile when the outside wall is at 12 °C and the window is at -3 °C

Figure 4.60 Velocity profile when the outside wall is at 12 °C and the window is at -3°C

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.61 Pathlines when the outside wall is at 12 °C and the window is at -3°C

From figures 4.49 and 4.61 it can be observed that the intensity of downward draft and the mean temperature varies to a large extent on the temperature on the window and the temperature of the left (outside) wall. From figures 4.60 and 4.61 when the temperature of the window is at -3°C and the left wall at 12 °C, a maximum downward draft can be observed. The velocity of the cold downward draft is almost the same as the upward draft from the convector. From the pathlines it can be observed that this extreme downward draft is effectively blocked completely entering the room by the hot air from the convector.
4.5 Generalized correlation between Nu and Ra

The data obtained from the two models is combined to obtain a correlation between Nu and Ra. The graph between Nu and Ra is shown in Figure 4.62.

\[ y = 0.0001x^{0.4976} \]

\[ R^2 = 0.906 \]

The relation obtained between Nu and Ra is

\[ Nu = 0.0001Ra^{0.4976} \]

This generalized equation can be used to predict the heat output values for any temperature condition for this system.
CONCLUSIONS

- Two different models of a room one with a window and the other without a window were simulated using CFD code. These models were simulated for a wide range of inlet water temperature conditions between 60.15 °C and 82.50 °C.

- The heat outputs for all the different temperature condition were calculated from the CFD code. Heat output value for one condition when the water inlet temperature is 82.5 is provided by the manufacturer, this value was compared with the obtained results and the percentage of error between these two values was found to be less than 6 %.

- Velocity profiles and pathlines were plotted for all the simulations which give an insight into the flow characteristics is air in the room.

- The system was checked for thermal comfort by calculating the draft rate (DR). The DR values calculated were well below the maximum limit of 15% as specified by ASHRAE Standard 55-2004.

- A parametric study has been performed on both the models for a wide range of temperature conditions to get a reasonable understanding of the system under different temperature conditions.
The relation between $Ra$ and $Nu$ is obtained as $Nu = 0.0001Ra^{0.4976}$ with an $R^2$ value of 0.906, which indicates that the heat output values can be predicted with an accuracy of over 90% using the formulated equation for a variety of operating conditions.
REFERENCES


12. STAR CD user manual, version 3.24. ADAPCO.


VITA

Graduate College
University of Nevada, Las Vegas

Satya Kiran C. Gurram

Local Address:
969 E Flamingo Rd Apt # 167,
Las Vegas NV, 89119

Degrees:
Bachelor of Technology, Mechanical Engineering,
JNTU, Hyderabad, India.

Thesis Title:
A Numerical Prediction of the Thermal Environment of a Room Heated with a
Hydronic Heating System

Thesis Examination Committee:
Chairperson, Dr. Samir Moujaes, Ph.D. P.E
Committee Member, Dr. Mohamed Trabia, Ph.D.
Committee Member, Dr. Nabil Nassif, Ph.D.
Graduate Faculty Representative, Dr. Yahia Baghzouz, Ph.D.