Design and CFD Modeling of a Guarded Hot Box for Thermal Performance of Building Assemblies

By

Dhrumil A. Shah

09MMET15

DEPARTMENT OF MECHANICAL ENGINEERING

AHMEDABAD-382481

May 2011

Design and CFD Modeling of a Guarded Hot Box for Thermal Performance of Building Assemblies

Major Project

Submitted in partial fulfillment of the requirements

For the degree of

Master of Technology in Mechanical Engineering (Thermal Engineering)

By

Dhrumil A. Shah

09MMET15

DEPARTMENT OF MECHANICAL ENGINEERING

AHMEDABAD-382481

May 2011

Declaration

This is to certify that

i) The thesis comprises my original work towards the degree of Master of Technology in Mechanical Engineering (Thermal Engineering) at Nirma University and has not been submitted elsewhere for a degree.

ii) Due acknowledgement has been made in the text to all other material used.

Dhrumil A. Shah

Certificate

This is to certify that the Major Project entitled "Design and CFD Modeling of a Guarded Hot Box for Thermal Performance of Building Assemblies" submitted by Dhrumil A. Shah (09MMET15), towards the partial fulfillment of the requirements for the degree of Master of Technology in Mechanical Engineering (Thermal Engineering) of Nirma University of Science and Technology, Ahmedabad is the record of work carried out by him under my supervision and guidance. In my opinion, the submitted work has reached a level required for being accepted for examination. The results embodied in this major project, to the best of my knowledge, haven't been submitted to any other university or institution for award of any degree or diploma.

Dr. V. J. Lakhera
Guide, Professor,
Department of Mechanical Engineering,
Institute of Technology,
Nirma University, Ahmedabad

Prof. V. R. Iyer
Professor and Head,
Department of Mechanical Engineering,
Institute of Technology,
Nirma University, Ahmedabad

Dr K Kotecha
Director,
Institute of Technology,
Nirma University, Ahmedabad

Acknowledgements

I owe thanks to all people who helped and supported me during the course of the dissertation work.

My deepest thanks to Prof. Dr. V J Lakhera the guide of the project for guiding me with attention and care.

I express my thanks to the Head of the Department, Mechanical Engineering Prof. V R Iyer, for extending his support.

I would also like to thank all faculty members in the department without whose support this project would have been a distant reality. I also extend my heartfelt thanks to my family and well wishers.

- Dhrumil A Shah
09MMET15

Abstract

In building elements, substantial energy is wasted by heat losses due to higher values of overall heat transfer coefficient. Hence an estimation of the same is important for making decisions influencing the energy savings in buildings and structures. For measuring this value of overall heat transfer coefficient and other important thermal properties of the building elements, a Guarded hot box can be utilized. A hot-box measures either thermal transmittance (or U-value) which determines the overall heat transfer through a structure; or it measures thermal conductance and thermal resistance, which are concerned with heat flow within the structure.

The present study undertaken the CFD analysis of a Guarded Hot Box using FLUENT software and comparing the results of those obtained experimentally by Fang et al[1]. The numerical values obtained by CFD analysis of the Guarded Hot Box using FLUENT validate the experimental results within 10%.

Contents

List of Figures

NOMENCLATURE

A - Metering area, m^2
Gr - Grashof number
h - Convectional heat transfer coefficient, W/m^2K
k - Thermal conductivity, W/mk
L - Length, m
N - Number of thermocouples
Nu - Nusselt number
Pr - prandtl number
Q, q - Heat flux, W
Ra - Rayleigh number
Re - Reynolds number
T_h - Temperature on hot side, K
T_c - Temperature on cold side, K
U - Overall heat transfer coefficient, W/m^2K
v - Air velocity
μ - Kinematic viscosity, kgs/m

Chapter 1

Introduction

1.1 General

The energy demand of the world is increasing day by day. In order to cope up with this, either more energy should be produced or the available resources should be conserved. Due to limited energy resources (fossil fuels) it is important that the focus should be to conserve the available energy resources.

Buildings are large consumers of energy in all countries. In regions with harsh climatic conditions, a substantial share of energy goes to heat and cool buildings. This heating and air-conditioning load can be reduced through many means, for example, by proper use of thermal insulation. The proper use of thermal insulation in buildings does not only contribute in reducing the required air-conditioning system size, but also in reducing the annual energy cost. Additionally, it helps in extending the periods of thermal comfort without reliance on mechanical air-conditioning especially during inter-seasons periods. The magnitude of energy savings as a result of using thermal insulation vary according to the building type, the climatic conditions at which the building is located, as well as, the type of the insulating material used. As climate modifiers, buildings are usually designed to shelter occupants and achieve thermal comfort in the occupied space backed up by mechanical heating and air-conditioning systems as necessary. Significant energy savings could be realized in buildings if they are properly designed and operated. As a least cost energy strategy, conservation should be supported in the energy future. For every unit of energy saved by a given measure of technology, resources will be saved, and the annual operating costs associated with producing that unit of energy will be reduced. Therefore, building designers can contribute to solving the energy problem if proper early design decisions are made regarding the selection and integration of building components. Substantial saving of energy can be achieved by minimising the heat transfer through the different parts of the room such as windows, doors and walls. Thermal insulation is a major contributor and obvious practical and logical first step towards achieving energy efficiency especially in envelope-load dominated buildings located in sites with harsh climatic conditions. Space air-conditioning can have a big share of energy used to operate buildings. For example, in the average American home, for example, space heating and cooling account for 50-70 % of its energy use[2]. This percentage could be higher in other parts of the world specially India, with more harsh climatic conditions and less energy efficient buildings. The thermal performance of building envelope is determined by the thermal properties of the materials used in its construction characterized by its ability to absorb or emit solar heat in addition to the overall U-value of the corresponding components including insulation. The placement of insulation material within the building component can affect its performance under transient heat flow. Heat loss through building is mostly related to the thickness and type of the construction material used and its ability to delay heat transfer through the building structure over a period of time.

1.2 Heat loss through building elements

In building elements, substantial energy is wasted by heat losses due to higher values of overall heat transfer coefficients associated with the elements such as walls, glass panes, etc. Hence an estimation of the same is important for making decisions influencing the energy savings in buildings and structures. Figure 1.1 shows the heat loss from different elements in a building.

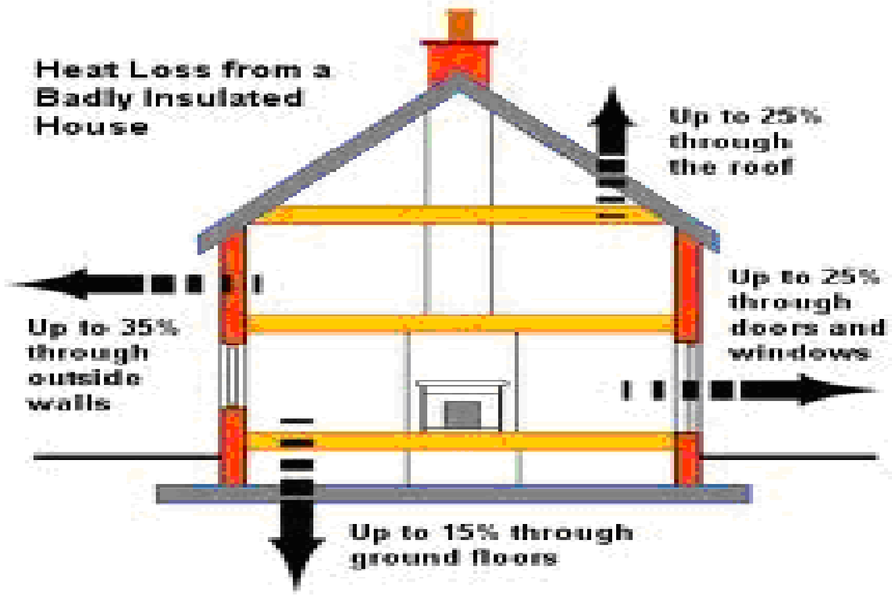

Figure 1.1: Heat loss from different building elements

From the figure it is clear that maximum heat loss in a building takes place through walls. By using different materials as well as a combination of different materials, the heat loss can be reduced. In the figure 1.2, it can be observed that by using different type of materials and its combinations, the overall heat transfer coefficient (U-value) is affected.

By using a guarded hot box, we can measure such different properties of the material (U-value) and by using proper material (and proper combination) we can minimize the heat loss occurring through the wall and also from other parts of the building.

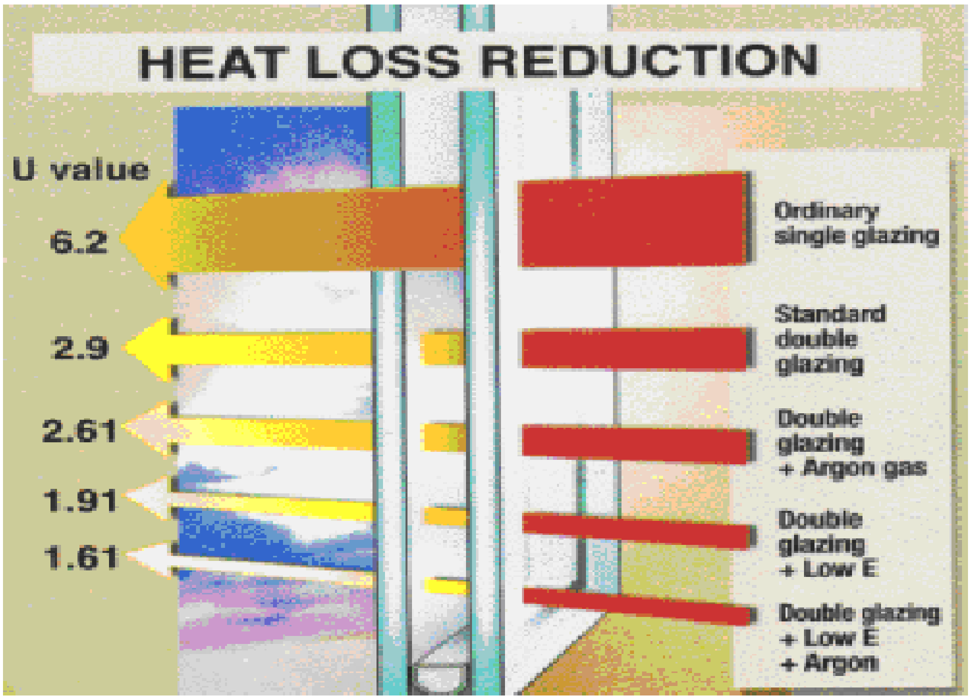

Figure 1.2: Effect of different material (and combinations) on overall heat transfer coefficient (U-value)

1.3 Benefits of minimising heat loss

By using material having low overall heat transfer coefficient (U-value) and also by minimizing losses occurring from different parts of a building lot of energy can be saved.

- Economic benefits- By minimizing the energy losses from the walls and other parts of the room and also by minimizing heat transfer from walls by using proper material we can save a large amount of electricity by reducing air-conditioning load.

- Environmental benefits- Due to combustion of fossil fuel for electricity generation, large amount of pollutants are generated which can be reduced by saving on energy requirements.

- Thermally comfortable room- By using proper material and minimizing the heat loss from the room we can create thermally comfortable rooms for both summer and winter conditions.

1.4 Heat transfer coefficient

The heat transfer coefficient, in thermodynamics and in mechanical and chemical engineering, is used in calculating the heat transfer, typically by convection or phase change between a fluid and a solid.
heat transfer in convection[3] can be measured by using following equation.

$$q = hA(T_h - T_c) \tag{1.1}$$

where,
q = heat flow,
h = heat transfer coefficient,
A = heat transfer surface area,
T_h and T_c = temperature on hot and cold side,

Now from above equation value of q and temperature difference can be calculated by using hot box and we can calculate heat transfer area A which will be equal to metering area. For calculating heat transfer coefficient we need to use different correlation and that correlation depends on different condition. Its value also changes on different shape and size of the specimens also for different material as well as different types of flow. By using equation of Nusselt number (Nu), the heat transfer coefficient can be calculated.[3]

$$Nu = hL/K \tag{1.2}$$

where,

Nu = Nusselt number,

L = Length,

k = Thermal conductivity.

In the above equation length can be directly measured while the thermal conductivity of the material is either known or else it can be directly measured. Here the value of the Nusselt number changes according to different conditions, in different flows, in different geometry of the object, etc. For example in case of forced condition Nu depends on Reynolds number (Re) and Prandtl number (Pr) which can be shown[3] using equation as.

$$Nu = 0.332Re^{1/2}Pr^{1/3} \tag{1.3}$$

In case of free convection flow, Nusselt number (Nu) depends on Rayleigh number (Ra) while this Rayleigh number depends on Grashof number (Gr) and Prandtl number (Pr).[3]

$$Nu = CRa^{n} \tag{1.4}$$

where Rayleigh number (Ra) can be given by equation

$$Ra = GrPr \tag{1.5}$$

In case of flow on the vertical plane, the above equation will be modified as[3],[4]

$$Nu = 0.68 + 0.670Ra^{1/4}/[1 + (0.492/Pr)^{9/16}]^{4/9} \tag{1.6}$$

The above equation can be applied for the entire range of Ra and this equation has been recommended by Churchill and Chu[4] and it is suitable for most engineering calculations.

The overall heat transfer coefficient for a wall or heat exchanger can be calculated as[3]

$$1/UA = 1/(h_1A_1) + dx_w/(kA) + 1/(h_2A_2) \tag{1.7}$$

where,
U = the overall heat transfer coefficient, W/m^2 K
A = the contact area for each fluid side, m^2
k = the thermal conductivity of the material, W/m K
h = the individual convection heat transfer coefficient for each fluid, W/m^2 K
dx_w = the wall thickness, m

1.5 Guarded Hot Box

Hot-boxes measure the overall heat transfer through large, inhomogeneous structures [5]. A hot-box measures either thermal transmittance (or U-value) which determines the overall heat transfer through a structure; or it measures thermal conductance and thermal resistance, which are concerned with heat flow within the structure.

This test method is suitable for building construction assemblies, building panels and other applications of non homogeneous specimens at similar temperature ranges. It may also be applicable for homogeneous specimens.

By using this method we can find out the overall heat transfer coefficient (U) of the given material and also other properties like conductance (C), thermal resistance (R) and convection coefficient (h).

The measurement principle is straightforward: a large specimen panel is well instrumented on each surface with temperature sensors arranged to provide a representative temperature distribution across the surface (figure 1.3 and 1.4). It is placed between hot and cold chambers operating at controlled fixed temperatures and air flow conditions. Temperature sensors are placed at positions approximately opposite to those in the specimen to obtain the corresponding air temperatures. The DC power required to maintain the temperature difference across the external environments is measured together with the air temperatures, the temperatures of the surfaces in radiant exchange with the specimen and in some circumstances, the specimen surface temperatures (for thermal conductance or resistance)[6].

In the guarded hot-box, a central metering box that covers a representative area of the panel is surrounded by an outer guard box. The temperature and air-flow conditions in the guard box are arranged to reduce the heat transfer through the metering box walls to negligible levels. The total measured power into the metering box from all sources is taken as the power through the specimen.

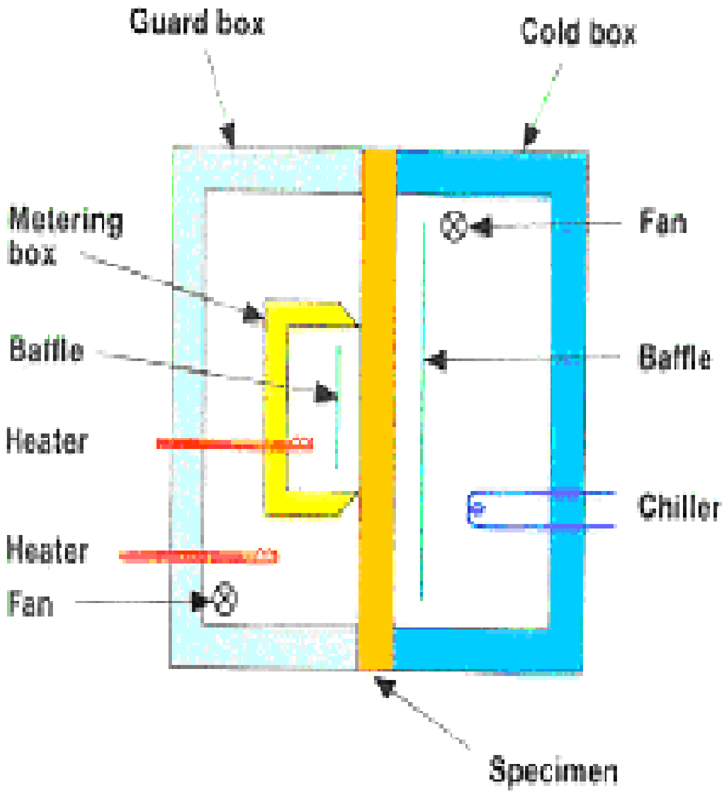

Figure 1.3: Schematic of a Guarded hot box

1.6 Organization of Thesis

This thesis has been divided into five main topics as listed below,

a. The Chapter number One provides an outline of energy utilization, heat loss through buildings and an introduction to Guarded Hot Box.

b. The Chapter number Two includes the basic theory behind the guarded hot box

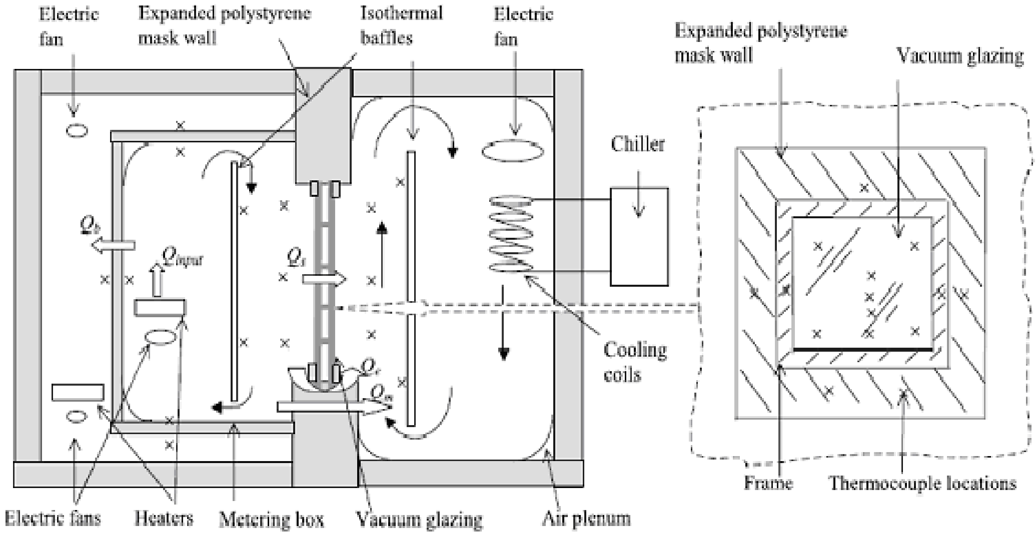

Figure 1.4: Section of a guarded hot box

and description about its different components. It also discusses the previous works done related to guarded hot box and the objective for the present study.

c. The Chapter number Three provides an introduction to CFD and includes about designing and modeling of a guarded hot box using CFD tools.

d. The Chapter number Four provides the CFD results and its discussion with reference to the objectives of the study.

e. The Chapter number Five provides conclusion from the present work and the scope for future work.

Chapter 2

Literature Review

2.1 Overview of Guarded Hot Box

Guarded Hot Box is an apparatus which is used for measuring thermal properties of insulating material. By using it the overall heat transfer coefficient and other important parameters like convection heat transfer coefficient can be estimated. The construction and design of a Guarded Hot box is summarized from the relevant ISO and ASTM standards[5][6] in this section.

In the guarded hot box (figure 2.1), the metering box is surrounded by a guard box in which the environment is controlled to minimize the lateral heat flow in the specimen Φ_2, and heat flow through the metering box walls Φ_3. Ideally, when a homogeneous specimen is mounted in the apparatus and when both inside and outside the metering box the temperatures are uniform and furthermore when cold side temperatures and surface coefficients of heat transfer are uniform, a temperature balance for air both inside and outside the metering box would imply a balance on the specimen surface and vice versa[6], i.e Φ_2=Φ_3=0. The total heat flow through the specimen will then be equal to the heat input to the metering box. In practice, for each equipment and each specimen under test, there will be a limit in detecting imbalance.

In practice, even with homogeneous specimens, the local surface coefficients of heat transfer are not uniform, especially close to the borders of the metering box. As a consequence, neither the specimen surface temperature nor the air temperatures are uniform close to the periphery of the metering box both inside and outside. This has two consequences:

a. It can be impossible to reduce to zero at the same time both the lateral heat flow, Φ_2, through the specimen, and the heat flow, Φ_3, through the metering box walls.

b. The temperature non uniformity close to the metering box, on the specimen surface, and in the air, respectively, imbalance resolution.

The apparatus shall be designed and operated in such a way as to obtain an optimum heat flow balance as indicated in a) above, i.e. apparatus geometry and guard air space and air flow speed so that does not exceed 10 percentage of Φ_p.[6]
Inhomogeneities in the specimen will enhance non uniformities in local surface coefficients and in specimen surface temperatures. Heat flow imbalance through the metering box wall and in the specimen shall be evaluated, and when necessary corrected for. For this purpose the metering box walls shall be equipped to serve as a heat flow meter. Additionally, a thermopile across the metering area periphery can be mounted on the specimen surfaces. In routine testing, imbalance detection can be simplified by calibration and calculation.
The main three parts of the guarded hot box are:

- Metering box
- Guard box
- Cold box

The above are further described in following sections.

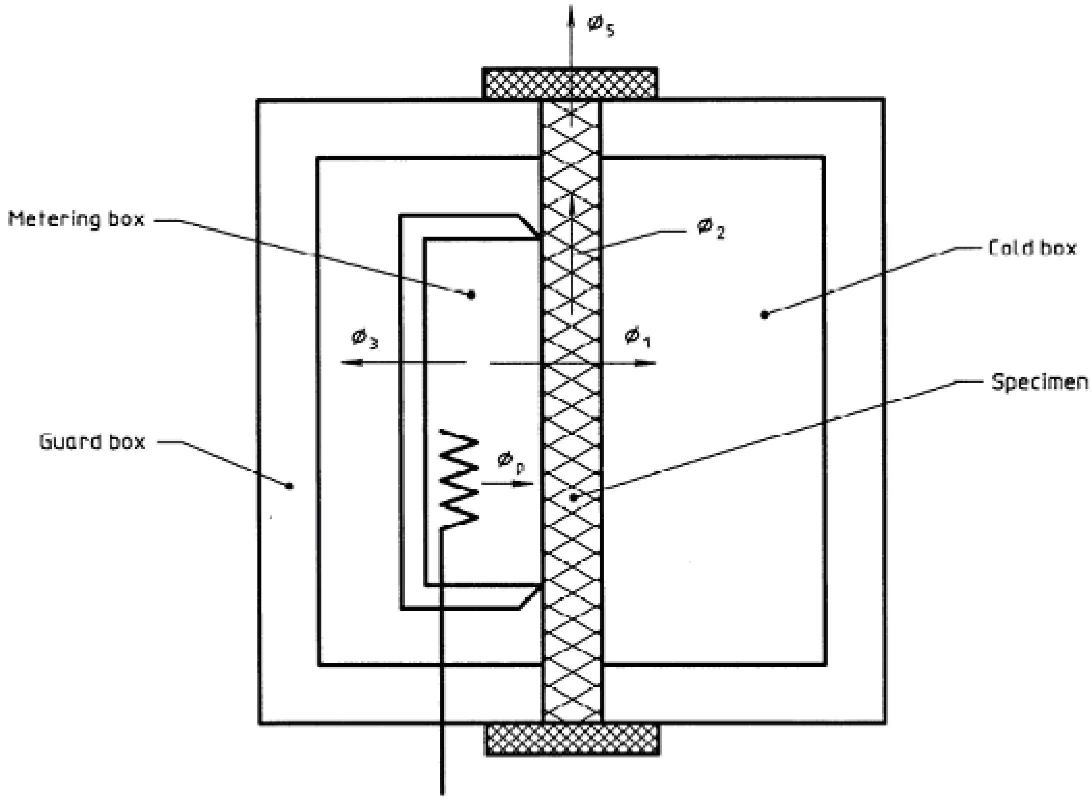

Figure 2.1: Guarded hot box

2.1.1 Metering box

- SIZE - The size of the metering box is largely governed by the metering area required to obtain a representative test area of panel. For example, for panels incorporating air spaces or stud spaces, the metering area, preferably should exactly span an integral number of spaces. The height of metering box should not be less than the width[5] and is subject to the limitations like for the vertical specimens with air spaces that significantly effect thermal performance, the metering box height should ideally match the construction height. If it is not possible, horizontal convection barriers must be installed to prevent air exchange between meter and guard areas, unless it can be shown that the omission of such barriers does not significantly affect results. The depth of the metering box should be not greater than that required to accommodate its necessary equipment.

- THERMAL RESISTANCE - The metering box walls shall have a thermal resistance of not less than 0.83 m^2 K / W.[5] In order that the resistance of the box wall shall be uniform over the entire box area, a construction without internal ribs shall be used. The edge in contact with the panel shall, if necessary, be narrowed on the outside only, to hold a gasket not more then 13 mm wide.[5] If necessary a wood nose piece can be used to carry the gasket. The metering area of the panel shall be taken as the area included between the center lines of the gaskets.

- GASKETS - The contact edges of the metering box should ensure, by a gasket or other means, a tight air seal against the surface of the test panel. The metering box should be pressed tightly against the panel by suitable means. Some gasket materials age with time and service. Periodic inspection of gaskets is recommended in order to confirm their ability to provide a tight seal under test conditions.

- HEAT SUPPLY AND AIR CIRCULATION - The arrangement of equipments should be proper to assure an even, gentle movement of air over the metering area of the panel.[5],[6] The electrical heaters are mounted in a housing with walls of resistance not less than 0.83 m^2 K/W[5]. In this arrangement air is continuously circulated by a small fan upward through the cylindrical housing and downward between the baffle and the panel in accordance with the motion that would result from natural convection forces. A baffle is placed some distance above the outlet of the cylindrical housing to prevent impingement of a jet of heated air against the top inner surface of the metering box. For large meter boxes the cylindrical housing may cause concentrations of air flow. To direct the air properly across the specimen other fan arrangements may be preferable.

A curved vane is mounted at the top of the baffle to smooth the entrance of air into the baffle space. In a hot box apparatus used for testing panels in a vertical position only, the moderate circulation of air resulting from natural convection may be sufficient without the use of a fan. The change in temperature of the air as it moves along the surface of the panel will, in general be greater with natural circulation than with the fan. If a fan is used, its motor should be within the metering box[5], its electrical input should be as small as feasible, and the input should be carefully measured. If it is necessary to locate the motor outside the metering box, the heat equivalent of the shaft power must be accurately measured, an air leakage into or out of the metering box around the shaft must be zero.

- TEMPERATURE CONTROL - To obtain reliable test results, accurate temperature control equipment must be utilized and the temperature controllers must be capable of controlling temperature[5],[6] within 0.25 K during the test period. The heater should be open wired type of a minimal heat capacity and lag.

- HEAT FLUX TRANSDUCER - To equip the metering box walls to serve as a heat flux transducer, a means of detecting temperature difference across the metering box walls or the heat flux through the metering box walls shall be provided[5]. One method found satisfactory for this purpose is to apply a number of differential thermocouple connected in the series to the inside and outside surfaces of the metering box walls to form a thermocouple. Precautions must be taken when determining the number of differential thermocouples.

2.1.2 Guard box

- SIZE - It is recommended that the guard box be large enough so that there is a clear distance between its inner wall and the nearest surface of the metering box of not less then the thickness of the thickest panel to be tested, but in no case less than 150 mm[5].

- THERMAL CONDUCTANCE - To assure that there shall be a temperature difference of no more then a few degrees between the guard box air and its inner surfaces, the walls shall have a thermal conductance not greater then 0.6 W / (m^2K)[5] a low conductance is also desirable for operating reasons, to assure that the heat flow into or out of the guard box from outside will be only a small fraction of the heat flow through the guard area of test panel.
- HEAT SUPPLY AND AIR CIRCULATION - One or more reflective surface cylindrical heater units with the fan may be used to supple heat to the guard box air and also to circulate the air to avoid stratification[5],[6]. The fan air intake of at least one such heater unit should be located in the lowest point in the guard box, to prevent pulling of cool air at the bottom. The air discharged from the heater cylinder shall not impinge directly against either the metering box or the test panel.
- TEMPERATURE CONTROL - The guard box air temperature and heat input can be controlled by a differential thermopile such that used on the metering box for a heat flow meter, or by a sensitive bridge circuit with opposed temperature sensitive arms located in the guard and metering boxes[5],[6].

2.1.3 Cold box

- SIZE - The of the cold box is governed by the size of the test panel or by the arrangement of boxes used[5],[6].
- INSULATION - The cold box should be heavily insulated to reduce the require capacity of refrigerating equipment.
- TEMPERATURE CONTROL - The cold box may be cooled in any manner that is capable of the close control of air temperature necessary during test.

An arrangement of equipment similar to that in the metering box may be used with a fan to force air downward through the enclosed refrigerating coils and upwards through the space between a baffle and the test panel. An alternative method is to use an exterior located refrigeration system and insulated ducts to supply chilled air to the cold box, liquid nitrogen in connection with solenoid valve regulating its flow may also be used. For fine control of the cold box, installation of open wire electrical heaters and the blower duct or other fast moving parts of the air circulation system and controlling these heaters by the sensor located in the discharge of the air circulation system is recommended. The use of desiccants to remove excessive moisture in the recirculating cold air may be useful. Temperature controllers for steady state test must be capable of controlling temperatures[5],[6] within 0.25 K.

- AIR CIRCULATION - High air velocities are permissible when their effect upon heat flow is to be determined. This may be accomplished by directing the airflow either parallel or perpendicular to the specimen cold surface. Velocities commonly used to simulate cross wind conditions are 3.35 m/s for summer conditions and 6.70 m/s for winter conditions[5].

2.1.4 Temperature measurement

Thermocouples of wire not larger in size than 0.25 mm (No. 30 AWG gage)[5],[6] are recommended for measuring surface temperatures in the apparatus. For this purpose the thermocouple junction and the adjoining lead wires for a distance at least 100 mm[5],[6] should be taped, or preferably cemented tightly to the surface.

If the specimen is uniform, or nearly so over the area and thus the surface temperatures vary only slightly at lower air velocities, a minimum number of thermocouples spaced uniformly and symmetrically over the surface is sufficient. This minimum number depends upon specimen size. The required minimum number of thermocouple, N, can be determined from the relation[5]:

$$N = A/(0.07 + 0.08A^{0.5}) \tag{2.1}$$

where A is the metering area in m^2

If the specimen is of non-uniform construction, the number of thermocouples specified in the above equation will be sufficient. In this case the thermocouples shall be judiciously located to represent each of the construction elements. Such representations shall be distributed approximately uniformly and systematically over the specimen.

If the surface temperatures are accepted to be greatly non-uniform, additional thermocouples must be used to sample adequately the different temperature areas so that reliable weighted mean temperatures may be obtained.

With some non homogeneous walls, such as concrete, it may be advisable to use copper shim stock under the thermocouples to average the temperature. Large aggregates in the concrete can create biased temperature readings.

At least two surface thermocouples shall be placed on the guard area of the specimen at suitable locations to indicate the effectiveness in the guard area[5],[6].

Surface temperatures on the cold side of the test panel shall be measured by surface thermocouples placed directly opposite to those on the warm side[5],[6].

Air temperatures shall be measured by the thermocouples, temperature sensitive resistance wires, or other sensors. Air thermocouples shall be made of wire not larger than 0.51 mm (No. 24 AWG)[5],[6].

If the thermocouples or other point sensors are used, they shall be located in the metering box area in the same quantity and spacing as that specified for surface thermocouples measured from the equation for the number of thermocouples. The thermocouples shall be located midway between the face of the panel and the baffle if one is used, but in no case less then 75 mm from the face of the panel[5].

Thermocouples shall also be placed in the guard space at suitable locations, to indicate the degree of uniformity of guard space air temperatures, preferably one should be placed opposite each guard area surface thermocouple, but not less then 75 mm from the panel[5].

Air temperatures on the cold side of the panel shall be measured by one thermocouple placed directly opposite each of the warm side air temperature thermocouples and located in a panel parallel to the specimen surface and space far enough away that they are unaffected by temperature gradients in the boundary layer. The thermocouples shall be located midway between the face of the panel and the baffle, if one is used. For low velocities a minimum spacing of 75 mm for the specimen surface is required. At higher velocities the required minimum spacing is less but in no case less then 20 mm. No thermocouples need to be placed in the cold space opposite guard space thermocouples remote from the panel surface[5].

It is recommended that the surface temperature of the baffles on the hot and cold sides be measured by placing thermocouples on all surfaces the specimen can see[5],[6].

2.2 Previous work related to guarded hot box

Fang et al[1] worked on the experimental validation of a numerical model for heat transfer in vacuum glazing using a guarded hot box calorimeter to measure their heat transfer coefficients. They measured the temperature and heat transfer rates and found good agreement with a finite element method.

Ghazi Wakili and Tanner[7] determined the U-value of a dried wall made of perforated clay bricks by using a guarded hot box and compared the results with those obtained by numerical analysis. The numerical models were refined to obtain more acceptable values.

Nussbanmer et al[8] performed a thermal analysis of a wooden door system with two integrated vacuum insulation panels experimentally using a guarded hot box and nu-

merically using the finite difference method. The goal of this study was to determine the impact of damaged vacuum insulation panels on the thermal performance of overall door system. Since the metal fittings act as thermal bridges in a highly insulating system, special attention was given to this effect. Additionally, an infrared imaging system was used to validate the computed temperature distribution on the surface.
Rose and Svendsen[9] analysed typical lightweight walls containing different types of linear thermal bridges using a guarded hot box and validated the results of numerical calculations generated by two simulation programs. The deviations ranged from approximately 1 to 5 %
The ISO standard 8990, "Thermal insulation determinations of steady-state thermal transmission properties calibrated and guarded hot box" deals with the principles for the design of the apparatus and minimum requirement that shall be met for determination of the laboratory steady-state thermal transmission properties of building components and similar components for industrial use. It does not, however, specify a particular design since requirements vary, particularly in terms of size, and also to a lesser extent in terms of operating conditions. This International Standard describes also the apparatus, measurement technique and necessary data reporting. Special components, for example windows, need additional procedures which are not included in this International Standard. Also excluded are measurements of the effect on heat flow of moisture transfer or redistribution but consideration shall be given in the design and operation of the equipment as to the possible effect of moisture transfer on the accuracy and the relevance of test results. The properties which can be measured are thermal transmittance and thermal resistance. Two alternative methods are included: the calibrated hot box method and the guarded hot box method. Both are suitable for vertical specimens such as walls and for horizontal specimens such as ceilings and floors. The apparatus can be sufficiently large to study full-scale components.
The ASTM standard C 236, "Standard test method for steady-state thermal performance of building assemblies by means of a guarded hot box" covers the measure-

ment of the steady-state thermal transfer properties of panels. In distinction to test method C 177, which is primarily applicable to homogeneous samples, the guarded hot box method provides for the evaluation of thermal performance of building assemblies, building panels, and other applications for nonhomogeneous specimens at similar temperature ranges. It may also be used for homogeneous specimens. This test method may be applied to any building construction for which it is possible to build a reasonably representative specimen of size appropriate for the apparatus.

2.3 Objective of the present study

The literature reviewed on the topic related to guarded hot box indicates that further work need to be conducted for optimizing the design of guarded hot box using CFD tools for finalising the optimized guarded hot box model. It is necessary that the CFD modeling work be undertaken and the results analysed. In this regard, a CFD model is prepared using FLUENT and validated with experimental results as obtained by Fang et al[1].

Chapter 3

Design and modeling of Guarded hot box

3.1 Introduction to CFD

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research, however, yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is performed using a wind tunnel with the final validation coming in flight tests.

3.1.1 Discretization methods

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are:

a. Finite Volume Method

The finite volume method (FVM) is a common approach used in CFD codes. The governing equations are solved over discrete control volumes. Finite volume methods recast the governing partial differential equations (typically the Navier-Stokes equations) in a conservative form, and then discretize the new equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature, there is no guarantee that it is the actual solution. The finite volume equation yields governing equations in the form,

$\frac{\delta}{\delta t} \int\int\int QdV + \int\int FdA = 0$

Where, Q is the vector of conserved variables, F is the vector of fluxes, V is the volume of the control volume element, and is the surface area of the control volume element.

b. Finite Element Method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach. However, FEM can require more memory than FVM. In this method, a weighted residual equation is formed:

$Ri = \int\int\int WiQdV^e$

Where Ri is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, Wi is the weight factor, and V^e is the volume of the element.

c. Finite Difference Method

The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries, making these codes highly efficient and accurate. Other ways to handle geometries include use of overlapping grids, where the solution is interpolated across each grid.

$$\frac{\delta Q}{\delta t} + \frac{\delta F}{\delta x} + \frac{\delta G}{\delta y} + \frac{\delta H}{\delta z} = 0$$

Where Q is the vector of conserved variables, and F, G, and H are the fluxes in the x, y, and z directions respectively.

d. Boundary Element Method

In the boundary element method, the boundary occupied by the fluid is divided into a surface mesh.

e. High Resolution Discretization Schemes

High-resolution schemes are used where shocks or discontinuities are present. Capturing sharp changes in the solution requires the use of second or higher-order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

3.1.2 Theoretical considerations in CFD

The CFD codes are arranged by the numerical algorithm accordingly, so that the fluid flow problem can be tackled.There are three main elements of CFD codes in the CFD packages which consist of pre-processor, solver and post-processor.

Pre-processor

The pre-processor contains all the fluid flow inputs for a flow problem. It can be seen as a user-friendly interface and a conversion of all the input into the solver in CFD program. In this stage, quite a lot of activities are carried out before the problem is being solved. These stages are listed as below:

- Definition of the geometry
  The region of interests which is the computational domain.

- Grid generation
  The subdivision of the domain into a number of smaller and non overlapping domains. The grid mesh of cells is carried out for the geometry.

- Selection of the physical and chemical properties
  The geometry to be modeled.

- Definition of the fluid properties

- Specifications of correct boundary conditions
  This is done at model's cells. The solution of the flow problem such as temperature, velocity, pressure etc. is defined at the nodes insides each cell. The accuracy of the CFD solution governed by the number of cells in the grid and is dependent on the fineness of the grid.

Solver

In the numerical solution technique, there are three different streams that form the basis of the solver. There are finite differences, finite element and finite volume methods. The differences between them are the way in which the flow variables are approximated and the discretization processes are done.

- Finite difference element, FDM Describes the unknown flow variables of the flow problem by means of point samples at node points of a grid coordinate. By FDM, the Taylor's expansion is usually used to generate finite differences approximation.

- Finite element method, FEM Use the simple piecewise functions valid on elements to describe the local variations of unknown flow variables. Governing equation is precisely satisfied by the exact solution of flow variables. In FEM, residuals are used to measure the errors.

- Finite volume method, FVM It was originally developed as a special finite difference formulation. The main computational commercial CFD codes packages using the FVM approaches involves PHOENICS, FLUENT, FLOW 3D and STAR-CD. Basically, the numerical algorithm in these CFD commercial packages involved the formal integration of the governing equation over all the finite control volume, the discretization process involves the substitution of variety FDM types to approximate the integration equation of the flow problem and the solution is obtained by iterative method.

Discretization in the solver involves the approaches to solve the numerical integration of the flow problem. Usually, two different approaches have been used and once at a time.

- Explicit approach Usually, this is the most approach that makes sense. It is relatively simple to set up and program. The limitation is that for a given Δt and Δx, must be less than some limit imposed by stability constraints. In some cases, Δt must be very small to maintain the stability and consequently long running time required for the calculation over a given time interval, t.

- Implicit approach For this approach, the stability can be maintained over a large value of Δt and fewer time steps required making calculation. Thus resulting less computer time. Adversely, it is complicated to set up and program. The computer time per time step is much larger than the explicit approach due to the matrix manipulation which is required for each time step. This approach is very accurate to follow the exact transients i.e. the time variations of the independent variables.

Post-processor

A FLUENT package provides the data visualization tools to visualize the flow problem. This includes - vectors plots, domain geometry and grid display, line and shaded counter plots, particle tracking etc. Recent facilities aided with animation for dynamic result display and also have data export facilities for further manipulation external to the code.

3.1.3 Problem solving

In the computational fluid dynamics, using the FLUENT codes provide to solve the problem numerically. The fundamental involves determining the convergence, whether the solution is consistent and stable for all range of flow variables.

- Convergence - is a property of a numerical method to produce a solution that approaches the exact solution of which the grid spacing, control volume size is reduced to a specific value or to zero value.

- Consistent - to produce the system of algebraic equations, which can be equivalent to the original governing equation.

- Stability - associates with the damping of errors as a numerical method proceeds. If a technique chosen is not stable, even the round-off error in the initial data can leads to wild oscillations or divergence.

3.1.4 Governing equations

The equations for conservation of mass, conservation of momentum, and energy equation are given as:

$$\nabla \cdot (\rho \overrightarrow{\nu}) = S_m \tag{3.1}$$

$$\nabla \cdot (\rho \overrightarrow{\nu} \overrightarrow{\nu}) = -\nabla p + \nabla \cdot (\overline{\tau}) + \rho \overrightarrow{g} + \overrightarrow{F} \tag{3.2}$$

$$\nabla \cdot (\nu(\rho E + p)) = \nabla \cdot (\lambda_{eff} \nabla T - \sum h_j J_j + (\overrightarrow{\tau}_{eff} \cdot \overrightarrow{\nu})) + S_h \tag{3.3}$$

Where, λ_{eff} is the effective conductivity (l+lt, where lt is the turbulence conductivity) and J_j is the diffusion of species j.

The stress tensor is given by,

$$\overline{\tau} = \mu[(\nabla \overrightarrow{\nu} + \nabla \overrightarrow{\nu}^T) - \frac{2}{3} \nabla \cdot \overrightarrow{\nu} I] \tag{3.4}$$

where m is the molecular dynamic viscosity, I is the unit tensor, and the second term on the right-hand side is the effect of volume dilatation. The first three terms on the right hand side of equation represent heat transfer due to conduction, species diffusion, and viscous dissipation. $S_h$is a source term including the enthalpy formation from the chemical reaction of the species, if any.

3.2 Design of the guarded hot box

From the literature review, different available designs were considered and after it the present design was selected for its flexibility for sample and also the ease for mounting of the specimen.

The size of design was modified according to need and space available. The size of the guarded hot box selected is 1875 x 1700 mm[9],[10],[8], while the size of the metering box is 1325 x 575 mm[10],[8], the size of the guard box is 1050 x 1700 mm and the size of the cold box is 750 x 1700 mm. For insulation is preferred option because temperature in hot side will not exceed $60^{o}C$ so polystyrene is safe as well as has low thermal conductivity. The baffle is kept between specimen and wall of the metering box for proper circulation of the hot air in one direction inside the metering box. For heating of the metering box one electric heater of capacity 60 W[9] can be used and for air circulation a fan of 20 W capacity is used. For cooling of cold box, chilled air is supplied from an external chiller of capacity 1 tonne. For measurement of temperature inside the apparatus, T-type of thermocouple is selected. For calculating number of thermocouples placed between the baffle and specimen is calculated by the equation[5]

$$N = A/(0.07 + 0.08A^{0.5}) \tag{3.5}$$

where A is the metering area in mm.

From this equation, the number of thermocouples calculated for metering area are 10. According to ASTM the minimum distance between the specimen and thermocouple should be 75 mm. In the present design the distance between specimen and thermocouple is around 144 mm, so it's greater than the specified value as per ASTM. The distance between two thermocouples is kept around 120 mm while according to ASTM the minimum distance between two adjacent thermocouples should be 100 mm. For measuring air temperature in the guard box as well as cold box thermocouples are used and they are placed according to requirement. Figure 3.1 shows the design of the guarded hot box.

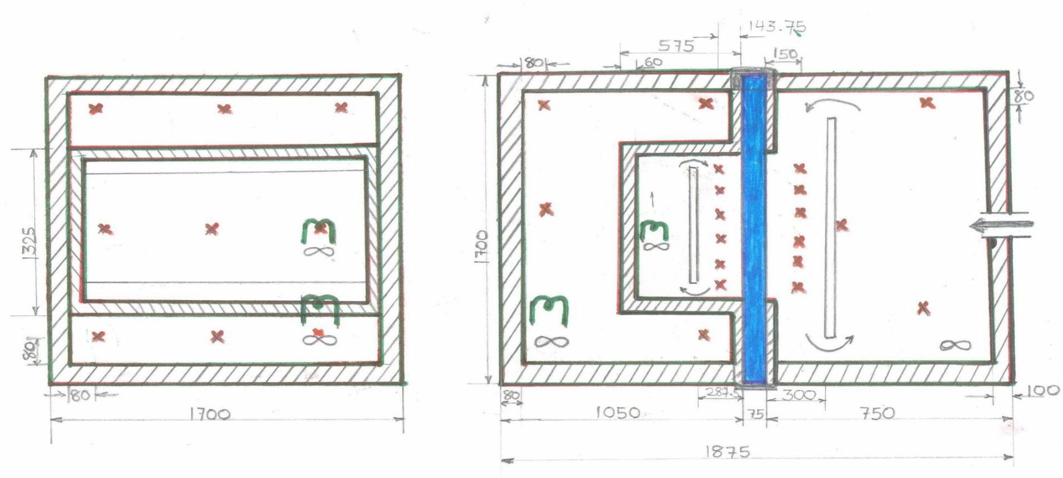

Figure 3.1: Design of the proposed Guarded Hot Box

3.2.1 Instrumentation

PID controller

A proportional-integral-derivative controller (PID controller) is a generic control loop feedback mechanism (controller) widely used in industrial control systems - a PID is the most commonly used feedback controller. A PID controller calculates an "error" value as the difference between a measured process variable and a desired set point. The controller attempts to minimize the error by adjusting the process control inputs.

The PID controller calculation (algorithm) involves three separate parameters, and is accordingly sometimes called three-term control the proportional, the integral and derivative values, denoted P, I, and D. Heuristically, these values can be interpreted in terms of time: P depends on the present error, I on the accumulation of past errors, and D is a prediction of future errors, based on current rate of change. The weighted sum of these three actions is used to adjust the process via a control element such as the position of a control valve or the power supply of a heating element.

Thermocouple

A thermocouple consists of two dissimilar metals, joined together at one end, which produce a small voltage when heated (or cooled). This voltage is measured and used to determine the temperature of the heated metals. The voltage for any one temperature is unique to the combination of metals used.

Thermocouples are available in different combinations of metals, usually referred to by a letter, e.g. J, K, T etc. Each combination has a different temperature range and is therefore more suited to certain applications than others. Although it is worth noting that the maximum temperature varies with the diameter of the wire used in the thermocouple. Temperature range and different data for different thermocouples are as given below.
Type B thermocouples can be used up to 1600C with short term excursions up to 1800C. They have a low electrical output, therefore are rarely used below 600C. In fact the output is virtually negligible up to 50C, therefore cold junction compensation is not usually required with this type.
Type E thermocouples are often referred to as Chromel-Constantan thermocouples. They are regarded as more stable than Type K, therefore often used where a higher degree of accuracy is required.
Note - Constantan is Copper-Nickel. Type J thermocouples degrade rapidly in oxidising atmospheres above 550C. Their maximum continuous operating temperature is around 750C though they can with stand short duration excursions to 1000C. They are generally not used below ambient temperature due to condensation forming on the wires leading to rusting of the iron.
Type K is the most widely used thermocouples in the Oil and Gas, and refining industries due to their wide range and low cost. They are occasionally referred to as

Chromel-Alumel thermocouples. Note that above about 750C oxidation leads to drift and the need for recalibration.

Type N thermocouples can handle higher temperatures than type K, and offer better repeatability in the 300 to 500C range. They offers many advantages over Type R and S at a tenth of the cost, therefore prove to be popular alternatives. Type R thermocouples cover similar applications as Type S but offers improved stability and a marginal increase in range. Consequently, Type R tend to be used in preference to Type S.

Type S thermocouples can be continually at temperatures up to 1450C. They can with stand short duration excursions up to 1650C. They need protection from high temperature atmospheres to prevent metallic vapour ingress to the tip resulting in reduction of emf generated. Protection commonly offered is high purity recrystallised alumina sheath. For most industrial applications, thermocouples are housed in a thermowell.

Type T thermocouples are rarely used in industrial applications, and lend themselves more to use in laboratory situations. In this type of thermocouple temperature range is from-50C to 370C.

For the present design the thermocouples selected are of type T, and wire diameter less than 0.25mm.

Heat flow meter

One another important instrument to be used in this design is heat flow meter. It is kept on the wall of the specimen which is useful for measuring the heat transferring from the metering box to the specimen so by using it the value of q can be measured. Heat flow meter is a device which is to be placed on the area where it is required to measure the heat transfer rate. So it directly gives us the value of q from that surface.

Digital anemometer

Anemometer is a commonly used device for air velocity measurement so in this present design, a digital anemometer is proposed to be used.

Digital regulator

In the design we are using a fan for flow of air and for its control a digital type of regulator can be used. For this, a regulating circuit is connected with the fan so the air velocity can be set and test can be conducted for different air flow conditions.

3.2.2 Procedure

The specimens shall be representative of the construction to be investigated but may be modified if necessary for test purpose. The usual pre-test conditioning is in ambient air long enough to come to practical equilibrium. Assemblies that may have significant moisture content, which can influence test results, must be allowed to reach steady-state moisture conditions. Since the specimen size will probably preclude oven drying, concrete wall specimens may require 6 to 8 weeks of room temperature aging[5].
Test conditions of temperature and orientation should be chosen to correspond as closely as possible to the circumstances of use of the construction to be tested. This test method is primarily designed for the temperatures encountered in normal building use, however it is recognize that the method may find application in testing conditions that are outside this normal range. It is recommended that a minimum temperature differential of approximately 25 K be maintain for accurate measurement[5].
Impose steady state condition for at least 4 hr prior to final data collection. This condition is satisfied when, over 4 hr period[5],[6], the average surface temperature does not vary by more than 0.06 C and the data do not change unidirectionally[5]. During this period data shall be collected at intervals of 1 hr or less. After these conditions has been satisfied, the test procedure is continued for at least 8 hrs, but the test is not terminate the test until two or more successive 4 hr periods produce

results that do not differ by more than 1 percentage. The average data for the two or more successive 4 hr periods that agree within 1 percentage are used in calculating the final results[5].

3.3 Modeling of Guarded hot box

For the modeling work of the guarded hot box the data was collected from respective paper by Fang et al[1]. The dimensions of the guarded hot box used by them are as described in the following section.
The wall thickness of the expanded polystyrene guarded hot box used was 150 mm. The thickness of the mask wall was 300 mm. The height and depth of the metering box were 1600 mm and 550 mm, respectively. A 1100 mm by 1100 mm by 2 mm thick matt black copper baffle was installed parallel to the central mask wall within the inner metering box to radiatively shield the surface of the sample. This prevented local radiative heat exchange with the box walls and from the heater and electric fan causing a non-uniform radiant temperature inside the box.A 60 W DC electric resistance heater was installed behind the baffle.

A rubber window sealant was used to provide a perimeter seal for the metering box to minimize errors due to air and moisture transfer, and to ensure that the box walls, perimeter seal and sample formed a sealed enclosure. A 6000 mm long helical cooling pipe was connected to a chiller to maintain the air temperature in the cold chamber between $0°C$ and $5°C$. The cooling pipe was installed behind a matt black copper baffle of 1100 mm by 1100 mm by 2 mm to minimize the influence of the radiative heat transfer from the pipe to the test sample. A 20W electric fan installed above the cooling pipe provided air circulation.

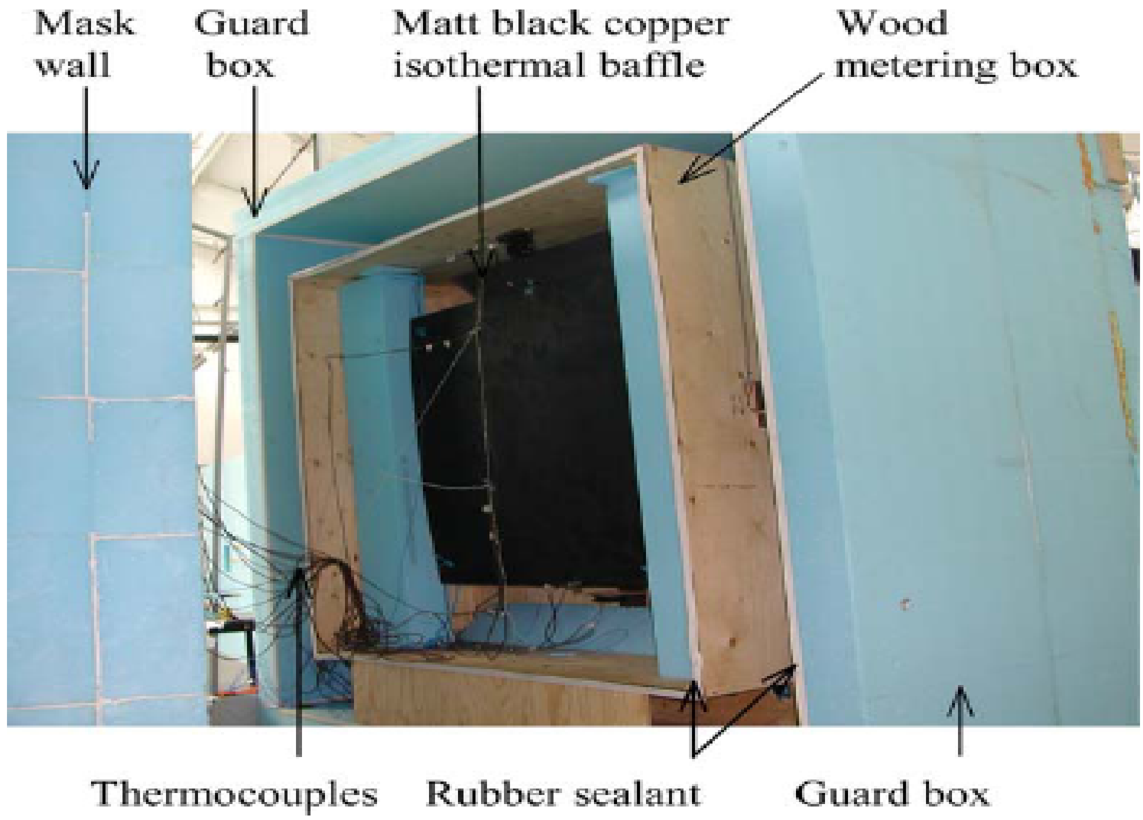

Figure 3.2: A photograph of guarded hot box by Fang et al

3.3.1 Computational modeling of guarded hot box

For preparing the computational model of the guarded hot box the design data was collected from the work by Fang et al[1]. According to the data given in the paper the same model was prepared in the GAMBIT and its 3d model was prepared(figure 3.3,3.4).

For analysis of the same, some simplifying assumptions were made.

The main requirement for keeping same temperatures in guard box and metering box is to make steady state condition and there should no heat transfer between the

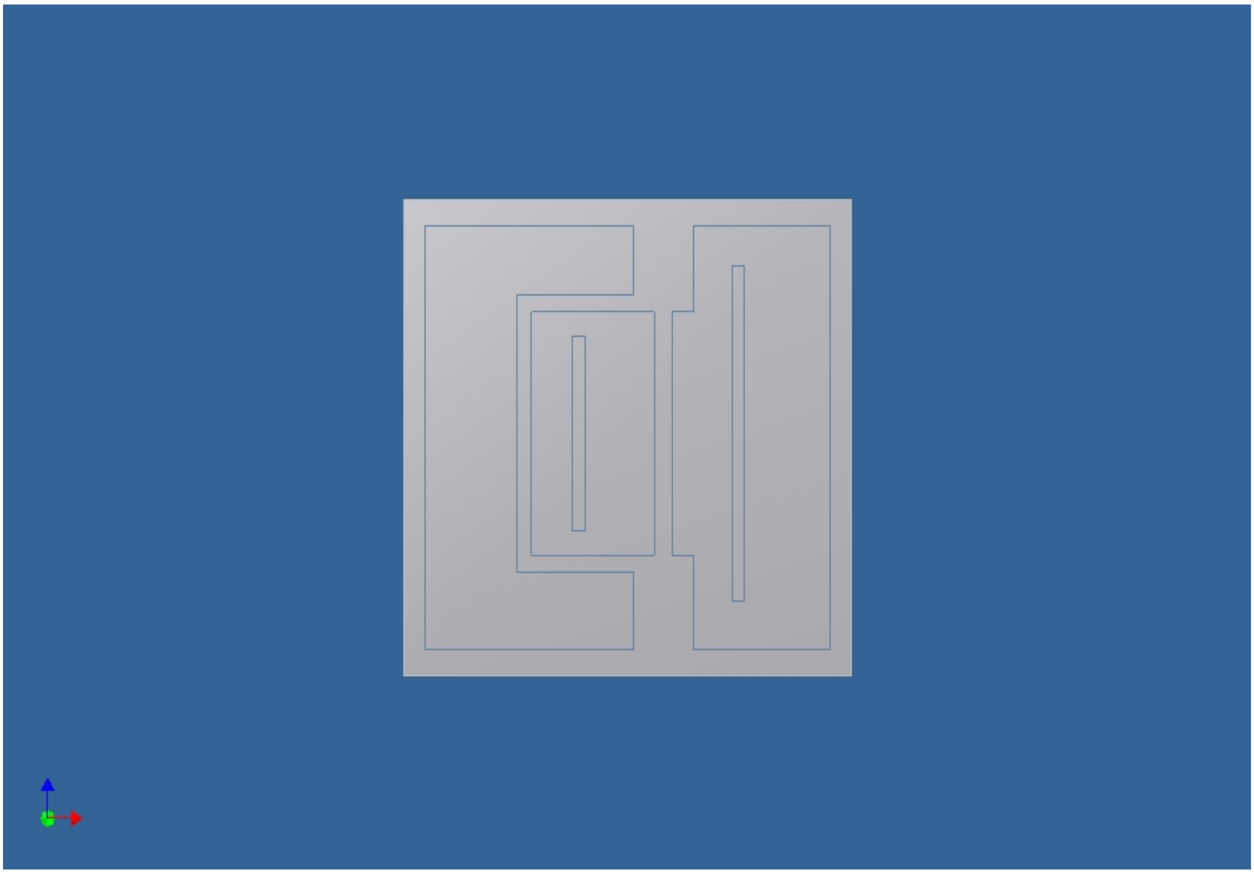

Figure 3.3: Front view of the model

guard box and metering box. So the wall between guard box and metering box can be considered as adiabatic such that no heat transfer occurs between the guard box and the metering box. Hence we can neglect the guard box by applying boundary condition at the metering box wall with adiabatic condition. For that purpose the model was created in GAMBIT in neglecting the guard box.

For the model there are following assumptions to be considered:

- 2 - Dimensional flow
- Adiabatic walls
- Laminar flow

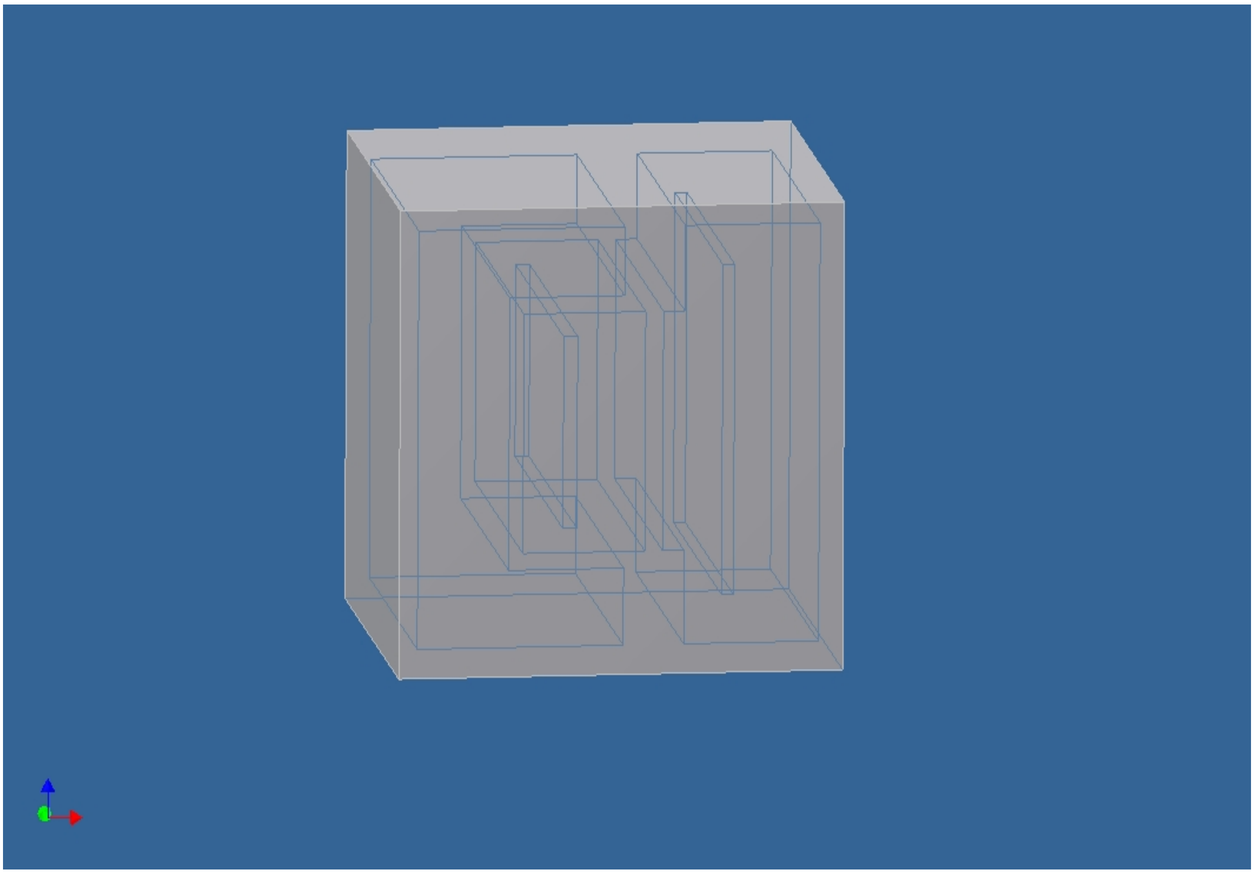

Figure 3.4: 3d model

- Single phase ideal gas flow
- Electromagnetic radiation is neglected
- Newtonian flow
- Fouriers law for the heat transfer

3.4 Boundary conditions and Initial conditions for different parts of guarded hot box

The guarded hot box has different parts, so first of all while making the model in GAMBIT, the guarded hot box was divided as different parts which are as follows:

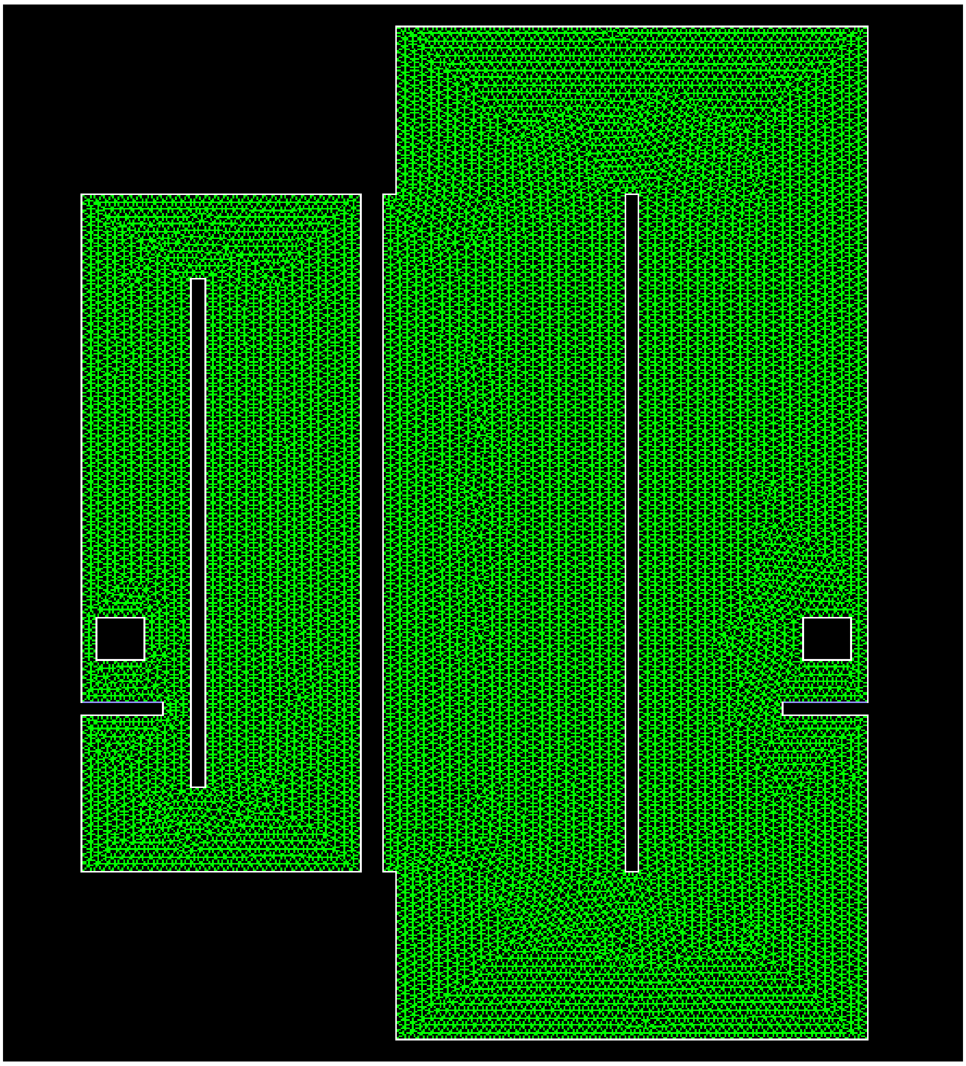

Figure 3.5: Meshed model of guarded hot box

- Metering box
- Hot and Cold baffle
- Heater
- Hot inlet
- Cold box
- Chiller

- Cold inlet

For each individual parts different boundary conditions were classified in FLUENT, which are as stated as following.

Metering box

Walls of the metering box have been considered and all of them were grouped and named as metering box. For this the boundary condition is taken as "Wall" for the metering box and at initial condition it is set at general room temperature.

Baffle

Baffle is used in both hot and cold side for proper circulation of the air flow inside the box so for hot side it is named as hot baffle and vice versa for the cold side it is named as cold baffle and both of them are kept under boundary condition of wall and initially it is taken as adiabatic because there is no heat transfer across the baffle.

Heater

Heater is located inside the metering box and it is used for heating of the metering box. Heater is kept under boundary condition of wall and at initial condition "heat flux" is considered for the heater.

Hot inlet

For proper air circulation of air inside the box there should be installation of any fan or any area from where the flow at proper velocity can be entered. For metering box one single plate is selected below heater for proper flow of air and which will also help for heating of the metering box.

Cold box

Similar to the metering box the walls of cold box is also grouped and kept under boundary condition of "Wall". Initially, the temperature of wall is set as general room temperature.

Chiller

The chiller unit is used for the cooling of the cold box. For this the surface area is considered and kept under the boundary condition of wall.

Cold inlet

For the proper circulation of air and for cooling of cold box there should be proper air flow, so for that purpose one plate below the chiller is considered and kept under wall boundary condition and at initial condition the velocity given in reference is considered.

3.5 Parameters to be considered

Model defining

- solver ⇛ segregated
- space ⇛ 2D
- velocity formulation ⇛ absolute
- gradient option ⇛ cell-based
- formulation ⇛ implicit
- time ⇛ steady
- porous formulation ⇛ superficial velocity

- energy equation $\Rrightarrow$ enabled
- viscous model $\Rrightarrow$ laminar and viscous heating
- radiation model $\Rrightarrow$ off

Material

- Material type $\Rrightarrow$ fluid
- Density $\Rrightarrow$ 1.225 kg/m^3 constant
- Viscosity $\Rrightarrow$ $1.7894e^{-05}$ constant

Relaxation factor

- Pressure $\Rrightarrow$ 0.3
- Density $\Rrightarrow$ 1
- Body forces $\Rrightarrow$ 1
- Momentum $\Rrightarrow$ 0.7
- Energy $\Rrightarrow$ 1

Discretization

- Pressure $\Rrightarrow$ standard
- Momentum $\Rrightarrow$ first order upwind
- Energy $\Rrightarrow$ first order upwind
- Pressure-Velocity coupling $\Rrightarrow$ SIMPLE

Chapter 4

Results and Discussion

The results obtained by CFD simulation using FLUENT are compared with the experimental results of the work by Fang et al[1]

4.1 Temperature variation

The results are tabulated in following table 1

Table I: A comparison between simulated temperatures and experimental results as obtained by Fang et al

Heat Rate	Experimental results from paper				Simulated results			
(W)	Air Temp(^oC)		Specimen Temp(^oC)		Air Temp(^oC)		Specimen Temp(^oC)	
	Hot	Cold	Hot	Cold	Hot	Cold	Hot	Cold
20.7	36.5	3.3	36.4	3.5	38.0	3.2	35.0	3.6
18.4	32.5	2.3	30.7	3.7	32.1	2.1	29.1	3.9
14.8	26.4	2.2	25.0	3.3	25.2	2.0	24.1	3.0
12.1	24.1	2.8	22.8	3.8	24.8	2.5	21.2	3.4

The temperature profile as obtained in FLUENT are shown in figures from 4.1 to 4.4

Table II: The percentage variation between CFD results and experimental results.

Heat Rate	Air Temperature		Specimen Temperature	
(W)	Hot	Cold	Hot	Cold
20.7	4%	3%	4%	3%
18.4	1%	9%	2%	5%
14.8	1%	9%	4%	9%
12.1	4%	10%	7%	10%

Figure 4.1: The temperature contours at a heat rate of 21 W for guarded hot box

From the figures 4.1 to 4.4 it is clear that in the average temperature on hot side is slightly higher in the region nearer the heater region because of constant heat generation from that part and also due to constant air velocity generated from the air inlet below the heater. On the cold side of the guarded hot box, the temperature is slightly higher nearer to the area close to the specimen because of heat transfer occurring through specimen. On the hot side the maximum temperature occurring was 319 K which was generated nearer to the heating area.

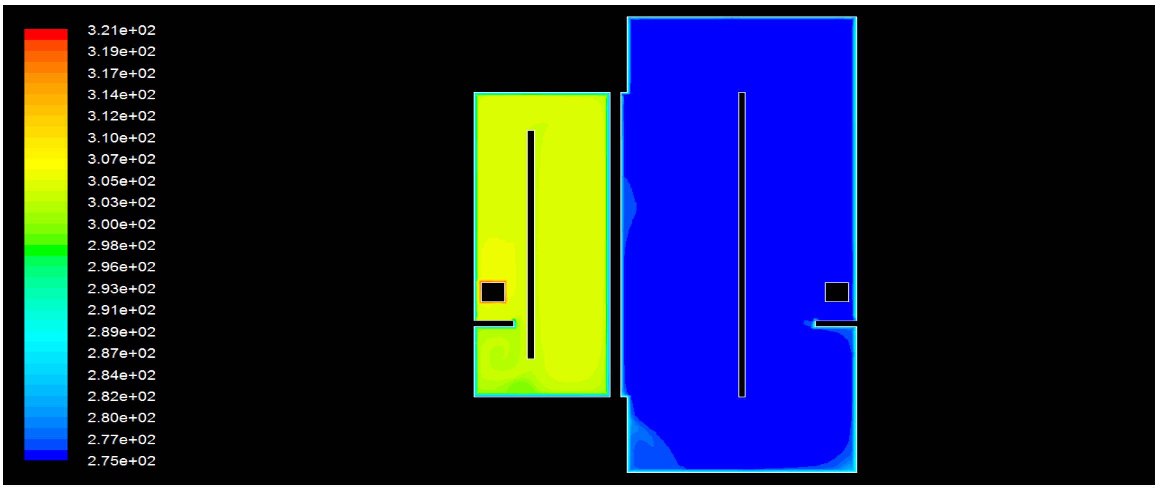

Figure 4.2: The temperature contours at a heat rate of 18 W for guarded hot box

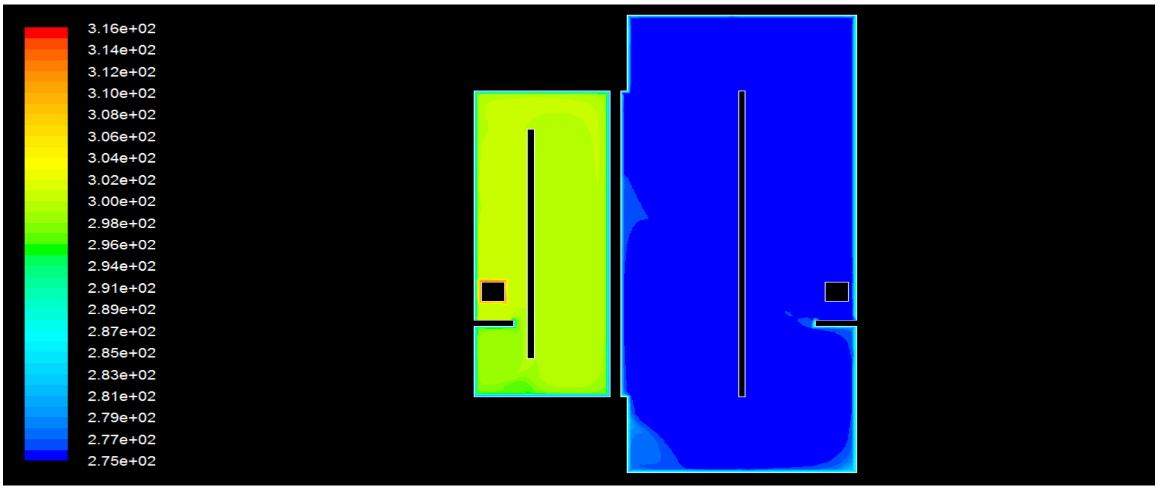

Figure 4.3: The temperature contours at a heat rate of 15 W for guarded hot box

The variation of temperatures between the simulated and experimental values (of Fang et al)are shown in figures 4.5 to 4.8.

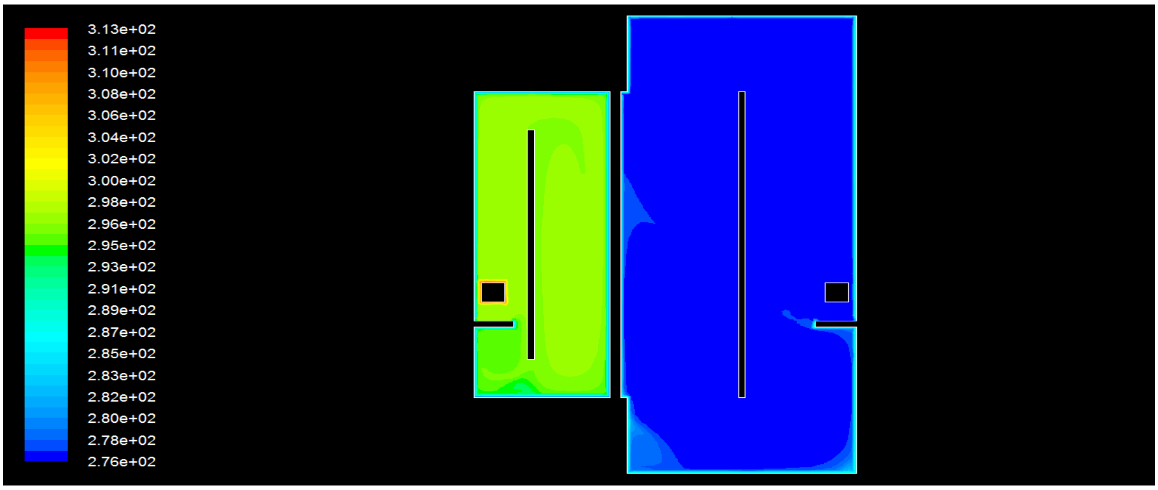

Figure 4.4: The temperature contours at a heat rate of 12 W for guarded hot box

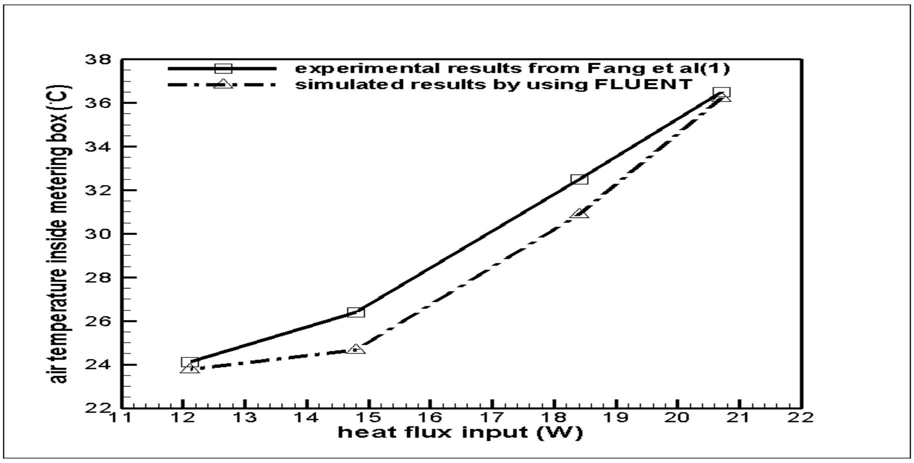

Figure 4.5: Comparison between present CFD simulation and experimental results as obtained by Fang et al for hot side air temperature

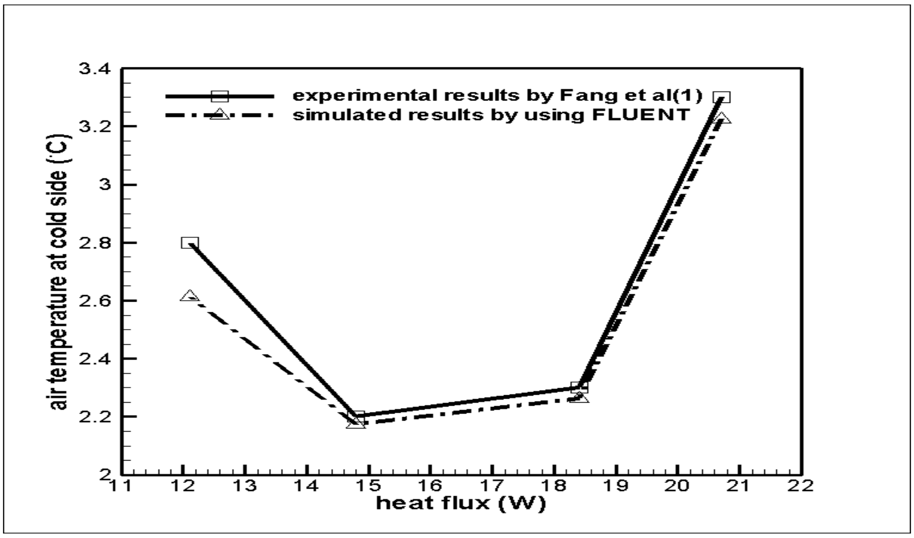

Figure 4.6: Comparison between present CFD simulation and experimental results as obtained by Fang et al for cold side air temperature

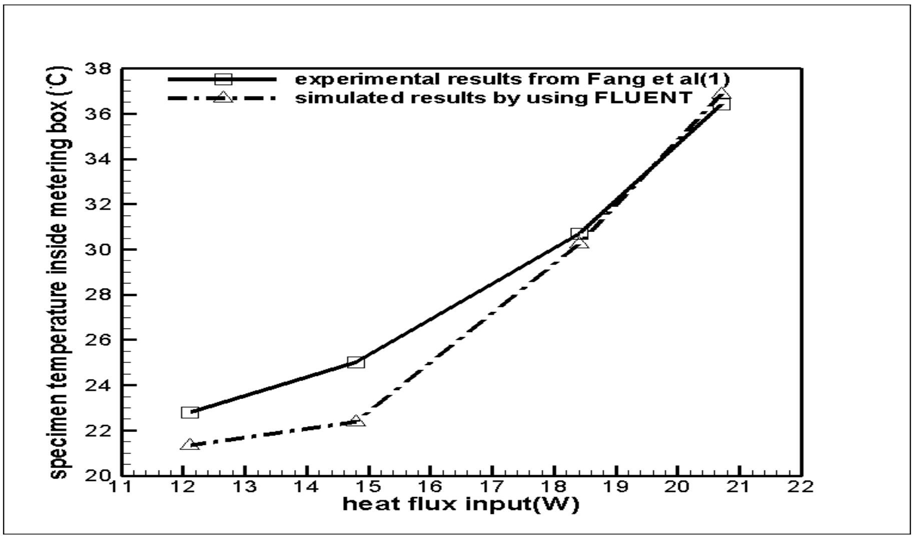

Figure 4.7: Comparison between present CFD simulation and experimental results as obtained by Fang et al for hot side specimen temperature

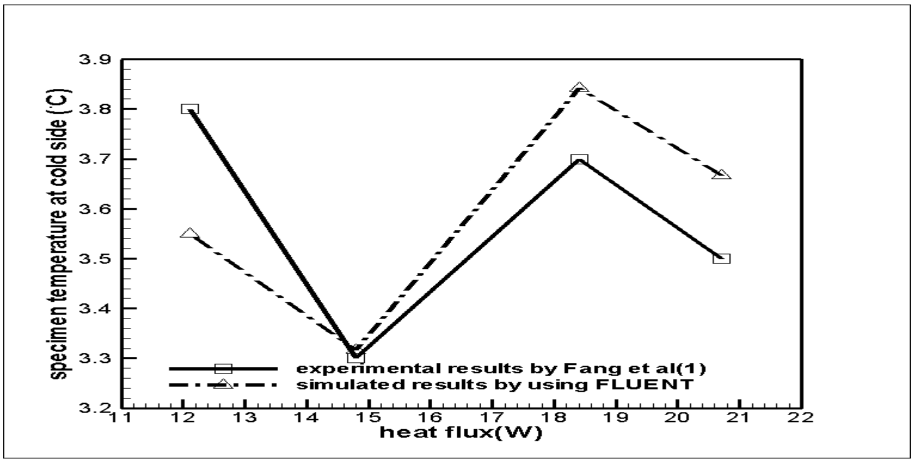

Figure 4.8: Comparison between present CFD simulation and experimental results as obtained by Fang et al for cold side specimen temperature

4.2 Velocity variation

The velocity profile results as obtained from the present simulation using FLUENT are shown in figure 4.9

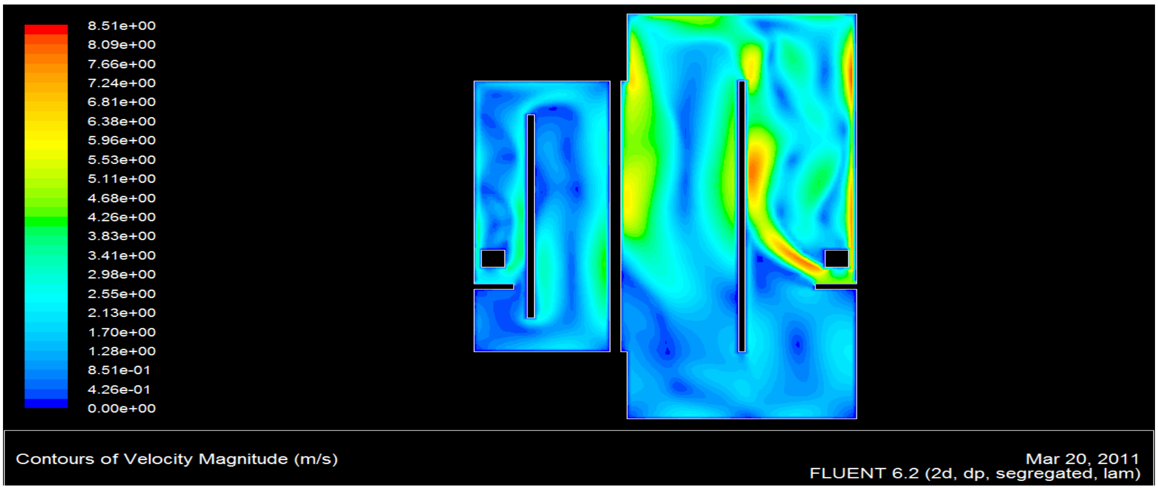

Figure 4.9: The velocity contours for the guarded hot box

From the figure 4.5 it is clear that in the guarded hot box the maximum velocity is achieved on the cold side of the box which is found to be 6.81 m/s in some regions while the average velocity is maintained close to 3.41 m/s in the cold side. In the metering box the maximum velocity noted was around 3.83 m/s while the average air velocity in hot side was around 1.28 m/s. From the figure it is clear that the flow is properly guided by using baffles.

4.3 Convection heat transfer coefficient

Based on the temperature profile obtained from the present simulation, the convection heat transfer coefficient are evaluated and compared with those obtained by Fang et al[1]

Table III: A comparison between simulated results for convection heat transfer coefficient and experimental heat transfer coefficient(1)

Heat rate	Experimental value		CFD results	
(W)	h_{hot} (W/m^2K)	h_{cold}(W/m^2K)	h_{hot}(W/m^2K)	h_{cold}(W/m^2K)
20.7	8.38	11.72	8.10	11.20
18.4	8.42	11.70	8.21	11.16
14.8	8.50	11.76	8.39	11.21
12.1	8.78	11.69	8.45	11.09

Table IV: The percentage variation in the predicted heat transfer with the experimental values

Q (W)	h_{hot} (w/m^2K)	h_{cold}(w/m^2K)
20.7	3%	5%
18.4	2%	4%
14.8	1%	4%
12.1	3%	5%

The experimental and CFD predicted values for the hot side and cold side convection heat transfer coefficients are shown in figure 4.10 and 4.11.

Using the convection correlations as discussed in chapter 2, the convection heat transfer coefficients for given average conditions were found to be

- h_{hot} = 9.12 W/m^2K
- h_{cold} = 11.5 W/m^2K

The calculations for above are shown in Appendix A.

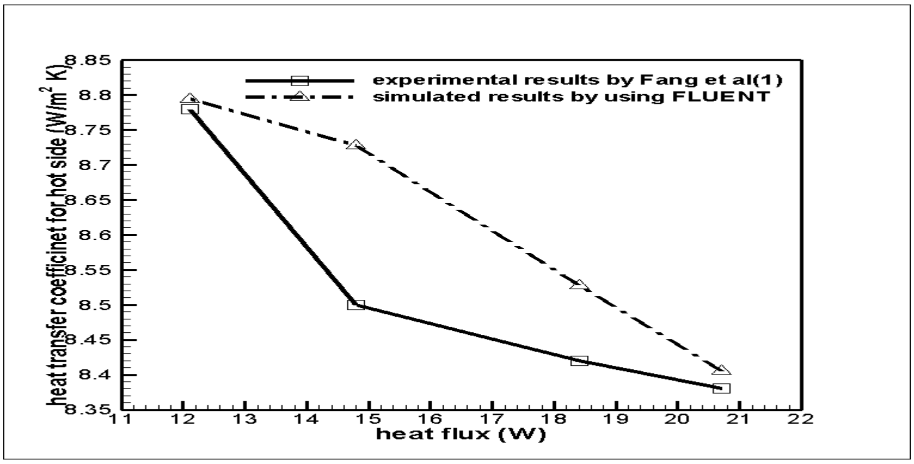

Figure 4.10: Comparison between present CFD simulation and experimental results(1) for h_{hot}

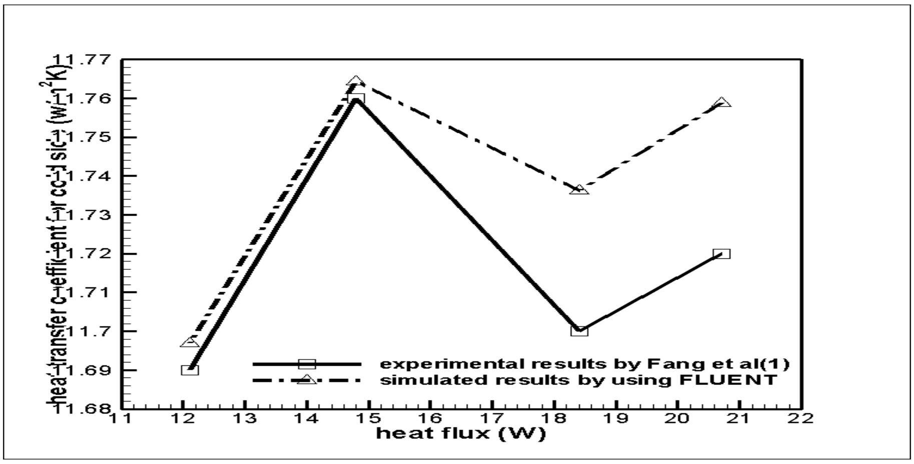

Figure 4.11: Comparison between present CFD simulation and experimental results(1) for h_{cold}

Chapter 5

Conclusions and scope for future work

5.1 Conclusions

From the present study, the following conclusions are drawn:

a. A Guarded Hot Box is a useful apparatus for determining the thermal transmittance (U-value) for building elements.

b. The Guarded Hot Box can be designed using ASTM C 236 and ISO 8990 which provides the various necessary details necessary for its effective design. Based on the study of these standards, a Guarded Hot Box design was finalized.

c. A model of the Guarded Hot Box using GAMBIT was prepared based on the work by Fang et al[1] and further analyzed using FLUENT software. The result obtained there in was compared with the experimental results obtained by Fang et al[1].

d. The variation in the results based on the present FLUENT simulation matched within 10% of these obtained experimentally by Fang et al[1].

5.2 Scope for future work

a. Based on the proposed design as per ASTM and ISO standards a Guarded Hot Box may be fabricated and tested preliminarily.

b. The model may be created using GAMBIT and analyzed using FLUENT and experimental results validated.

c. The design may be further optimized using FLUENT and the practical setup modified, if necessary.

Appendix A

Appendix

A.1 Theoretical calculation for calculating convectional heat transfer coefficient

For the flow of air over the flat plate Nusselt number can be calculated[3] using equation:

$$Nu = 0.0296 Re^{4/5} Pr^{1/3} \tag{A.1}$$

where, Re = Reynolds number.
Pr = Prandtl number.

Reynolds number can be calculated[3] using equation:

$$Re = vL/\mu \tag{A.2}$$

Applying given velocity condition by Fang et al[1] Re was calculated as 62500, and Prandtl number at given condition by Fang et al[1] is 0.712. By applying these two values in equation A.2 Nusselt number can be calculated as 172.72. Nusselt number can also be calculated by using equation:

$$Nu = hL/k \tag{A.3}$$

where, h = Convectional heat transfer coefficient, W/m^2K
L = Length of specimen, m
k = Thermal conductivity, W/mK
For given specimen length is selected as 500mm and thermal conductivity of air is 0.0264 W/mK at present condition, by applying these values in to equation A.3 convectional heat transfer for hot side can be calculated as 9.12 W/m^2K. Similar calculation can be done for cold side of the specimen and convectional heat transfer coefficient calculated was 11.5 W/m^2K.

References

[1] Yueping Fang Trevor J. Hyde Philip C. Eames, Brian Norton. Experimental validation of a numerical model for heat transfer in vacuum glazing. *Solar Energy, 80*, pages 564–577, 2006.

[2] The US department of energy. Insulation fact sheet with addendum on moisture control. *DOE/CE-0180, USA*, 2002.

[3] Frank P. Incropera and David P. DeWitt. *Introduction to Heat Transfer*. Wiley, 1996.

[4] S. W. Churchill and H. H. S. Chu. Correlating equations for laminar and turbulent free convection from a vertical flat plane. *International Journal Heat Mass Transfer, 18*, page 1049, 1975.

[5] ASTM C 236. *Standard test method for steady-state thermal performance of building assemblies by means of a guarded hot box*. 1993.

[6] ISO 8990. *Thermal insulation - determination of steady-state thermal transmission properties - calibrated and guarded hot box*. 1994.

[7] Ch. Tanner K. Ghazi Wakili. U-value of a dried wall made of perforated porous clay bricks hot box measurement versus numerical analysis. *Energy and Buildings, 35*, pages 675–680, 2003.

[8] Ch. Tanner T. Nussbaumer, K. Ghazi Wakili. Experimental and numerical investigation of the thermal performance of a protected vacuuminsulation system applied to a concrete wall. *Applied Energy, 83*, pages 841–855, 2006.

[9] Svendsen Jorgen Rose. Validating numerical calculations against guarded hot box measurements. *Building Physics, vol. 4*, 2004.

[10] Ch. Tanner H. Muehlebach T. Nussbaumer, R. Bundi. Thermal analysis of a wooden door system with integrated vacuum insulation panels. *Energy and Buildings, 37*, pages 1107–1113, 2005.