International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, Issue 5, May 2015

Design and CFD Investigation of Room Air Gate

Ajish Soman, V Hariganesh, Bipin P Varghese

Abstract— **Maintaining different temperature in closed region is highly different and not a cost effective option. Air gates are normally used in bigger malls and general people gathering locations, it is used for restricting air enter into the conditioned system where temperature is maintained uniformly. In this research work an attempt is made to create different temperature in a single block region through air-gate principle, here air is injected at a velocity in the dividing line to create a air-gate this air is at some pressure and temperature, the two region of interest will be maintained at a different temperature, a nozzle flow model is used to create the air-gate. Cosmos-Floworks will be used to simulate the possibilities of making the proposed system and limitations in the design. This is a Fluid heat transfer simulation involving stagnated air flow at a temperature in a region and different at another region, separated by air-gate where air in injected at a velocity with temperature.**

Keywords **– CFD, Air gate, Cosmos, Temperature**

I. *INTRODUCTION*

An air door or air gate is a device used to prevent air or contaminants from moving from one open space to another. The most common use is a downward-facing blower fan mounted over an entrance to a building, or an opening between two spaces conditioned at different temperatures. Air doors can come with or without heaters to heat the air. The fan must be powerful enough to generate a jet of air that can reach the floor. American Society of Heating, Refrigerating and Air-Conditioning Engineers define an air door as: "In its simplest application, an air gate is a continuous broad stream of air circulated across a doorway of a conditioned space. It reduces penetration of insects and unconditioned air into a conditioned space by forcing an air stream over the entire entrance. The air stream layer moves with a velocity and angle such that any air that tries to penetrate the gate is entrained. Air gate effectiveness in penetrating infiltration through an entrance generally ranges from 60 to 80%".Air Movement and Control Association defines an Air Gate as: "A directionally-controlled airstream, moving across the entire height and width of an opening, which reduces the infiltration or transfer of air from one side of the opening to the other and/or inhibits flying insects, dust or debris from passing through". Air doors are often used where doors are required to stay open for operational purposes, such as at loading docks and vehicle entrances. They can be intended to help keep flying insects out by creating forceful turbulence, or help keep out outside air, thus reducing infiltration through the opening. Cold drafts can be avoided by mixing in warm air heated by the air door. Heated air doors are commonly used when supplemental heat is needed for a space, and to reduce the wind chill factor across the opening in colder climates. Further applications include customer entryways, airplane hangars, cargo doors, drive through windows, restaurant doors, or shipping receiving doors. Non-heated air gates are often used in conjunction with cold storage and refrigerated rooms.

1.1 Effectiveness

Airflow through a door depends on wind forces, temperature differences (convection), and pressure differences. Air doors work best when the pressure differential between the inside and outside of the building is as close to neutral as possible. Negative pressures, extreme temperature differences, elevators in close proximity, or extreme humidity can reduce the effectiveness of air doors. The most effective air door for containing conditioned air inside a building with an open door will have a high face velocity at the opening, generated by top-down flow, and air recovery by a recirculating air plenum and duct return to the source fans. This configuration is feasible for new construction, but difficult to implement in existing buildings. The air door is most effective with low exterior wind velocity. At higher wind velocities, the rate of air mixing increases and the outside air portion of the total face flow increases. Under ideal conditions of zero wind, the effectiveness of the air door is at its maximum. In windy locations, air doors cannot create a perfect seal, but are often used to reduce the amount of infiltration from an opening. For industrial conditions, high face velocities are acceptable. For commercial applications like store entrances, user comfort dictates low face velocities, which reduce effectiveness of separation of exterior air from interior air.

1.2 Over Door Heaters

The UK based HEVAC Air Gate Group describes over door heaters as small electric or water heated fanned units with a low air volume flow rate. They are intended to be installed at doorways having low pedestrian footfall where the door is mainly closed and are useful in providing warmth. However, they should not be seen as an alternative to an air gate.

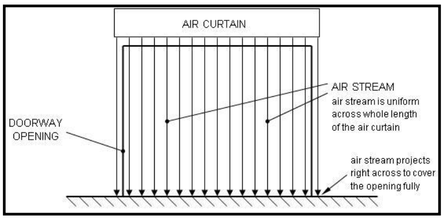


Fig.1.1 - Air flow of an air door (top-down configuration)

ISSN: 2278 – 7798

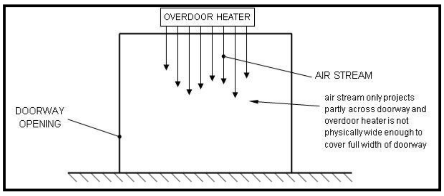


Fig.1 - Air flow of an over door heater

The main differences are: Air doors are designed to fully cover the width of a doorway, whereas over door heaters may be too small. The fans in an air door are powerful enough to provide an air stream to project across the whole doorway. Over door heaters may have less powerful fans.

The discharge nozzle on an air door is optimized to provide a uniform air stream across the whole width of the doorway, which may not be the case with over door heaters. Air gates can be used to save energy by reducing the heat transfer (via mass transfer when air mixes across the threshold) between two spaces, although a closed and well-sealed physical door is much more effective. A combination is often utilized. At the time of a door open the air door turns on, minimizing air flow from inside to outside and vice versa. An air door may pay for itself in a few years by reducing the load on the building's heating or air conditioning system.[citation needed] Usually, there is a mechanism, such as a door switch, to turn the unit on and off as the door opens and closes, so the air door only operates when the door is open. An authoritative engineering design procedure for calculating the supply air flow and thermal capacity of an air gate for an HVACR application is explained in the BSRIA Application Guide 2/97. The procedure for a ‘Building with an Air Tightness Specification’ should be followed, i.e. a practical building with some air leakage. Within the BSRIA Application Guide, Section 4.2 explains the design procedure and Section 5.2 gives worked examples for buildings with a range of air tightness specifications. This allows the engineer to calculate the supply air flow rate and thermal capacity of the required air gate for a particular application.

1.3 Design factors

The air gate should be just wider than the full width of the door opening, suitable to discharge air across the whole, height of the opening at a supply air, temperature which is acceptable for the comfort of people passing through the doorway. The heat output of the air gate must be sufficient to temper the volume of air coming in at the entrance. The air gate should be positioned as close to the opening as possible. Height of the building and its ability to create a stack (or chimney) effect should be checked before installation. The doorway locations should have the followings, air leakage which creates a pressure difference across the doorway leading to draughts should be effectively zero. A tight building envelope also reduces heat losses. The outdoor climate conditions for designing the ait-door are temperature, humidity and prevailing wind conditions. The indoor conditions for design are the surrounding area need to be heated/cooled, special conditions need to be met (e.g. work areas in proximity to the door), the obstacles on or around the door that need to be removed, special mounting requirements for air gate and other special requirements e.g. humidity, acoustic. The heating sources for design are follows, direct electric heating, low, medium or high pressure hot water, refrigerant condensers, direct & indirect gas fired steam, and renewable energy sources (wind power electric, heat pump low pressure hot water, heat pump refrigeration).

1.4 Working Principle

For winter operation air gates disrupt the natural convection (buoyant flow) effect of warm air spilling out of the top of an open doorway and being replaced by cold air coming in at the bottom. This convective flow is the primary heat loss infiltration mechanism and by minimising this flow they can save energy compared with an open doorway without an air gate. Wind and building stack effects can also cause airflow in at the doorway and whilst air gates do not necessarily act as a barrier to prevent the entry of this outside air, when used in air conditioned or industrial climate control areas (HVACR applications), they will condition the incoming air at the entrance and minimise cold draughts. The Computational Fluid Dynamics (CFD) temperature profiles below indicate the effect of air gates in doorways: Fig.1.4 shows the effect on the conditioned space without an air gate and illustrates convective heat losses, Fig.1.5 shows the effect on the conditioned space when an inadequately selected air gate is used, Fig.1.6 shows the effectiveness of a correctly installed and selected air gate.

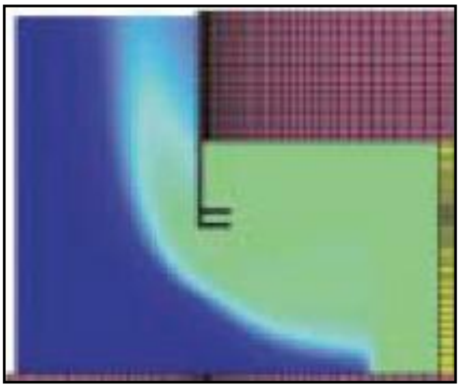


Fig.2 - Without an Air-gate

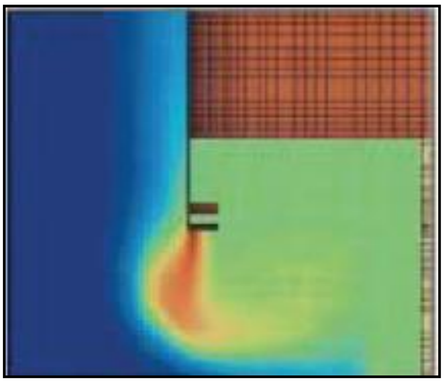


Fig.3 - Inadequately Selected Air gate

ISSN: 2278 – 7798

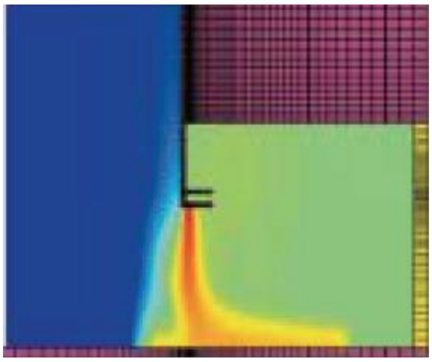


Fig.4– Properly installed Air gate

The heat output of an air gate must be sufficient to temper the volume of air coming in at the entrance. An air gate will become less effective if the velocity of the incoming air is excessive. This can occur as a result of under-pressure within the building from extract systems, stack effect with leaky or tall buildings, or wind effects on an exposed site. If conditions are not extreme, an air gate with a non-heated air stream (Ambient Air Gate) can also be effective in reducing energy losses from an air conditioned entrance by disrupting the natural convective heat transfer at the doorway. The width of an air gate discharge grille should be just wider than the doorway opening; an air gate narrower than the doorway is ineffective. Opening and closing of doors can disturb the air stream, which may take some time to re-establish. The heating capacity of an air gate can have an effect on the space temperature within the building entrance and suitable controls need to be fitted to adjust the heat output and air stream characteristics if necessary.

II. LITERATURE SURVEY

Three-dimensional effects of an air curtain used to restrict cold room infiltration [1], the aim of this study was to compare the measured effectiveness of an air curtain device at different jet velocities against a three-dimensional computational fluid dynamics model. The air curtain device was not as wide as the entrance and had a geometry that encouraged 3-D flow. By carefully setting up the air curtain an effectiveness of 0.71 was achieved compared to the initial value of only 0.31 as set by the air curtain device installer. The 3-D CFD model predicted the infiltration through the entrance with no air curtain to an accuracy of within 20–32%. The predicted effectiveness of the air curtain at different jet velocities was 0.10–0.15 lower than measured. The shape of the effectiveness curve against jet velocity was well predicted. The Effects of Orientation, Ventilation, and Varied WWR on the Thermal Performance of Residential Rooms in the Tropics [2] presents the followings, building orientation is a significant design consideration, mainly with regard to solar radiation and wind. In predominantly hot humid regions like Malaysia which receives sunlight all year around, buildings should be oriented to minimize solar gain and maximize natural ventilation. This paper describes an investigation into the effect of building orientation in view of solar radiation absorptance of exterior wall, varied area ratio of glazed window to wall and the effect of natural ventilation on the thermal performance for residential building in tropical region. Fresh Air-Han Comparison and Design [3], presented the followings, the United Arab Emirates has one of the highest wet-bulb design temperatures in the world (30.56°C), making it one of the most challenging places for controlling indoor relative humidity. Around 20% of the total building cooling load and annual energy consumption is used for the treatment of the fresh air supply needed for ventilation. For those reasons, we are always challenged to look for better and more efficient ways to treat the fresh air supply. Based on the results and assumptions of this specific analysis, when no need exists for a constant supply temperature and RH, the total energy wheel with a horseshoe heat pipe arrangement with the lowest energy consumption costs and capital costs can be used (supply temperature was found to vary between 18.9°C and 21.1°C and RH between 59% and 67%). The recommended spacing between the heat pipe coils is 1.4 m for easier cooling coil's maintenance, although this leads to having a slightly longer unit. Double energy recovery systems resulted in a better humidity control with a constant fresh air supply temperature and RH year-round regardless of fresh ambient conditions (21°C and 61% RH). Supply Air CO_2 Control of minimum outdoor air for multiple space systems[4], presented the followings, post ventilating systems have shortcomings in the control of their minimum outdoor air intake. They do not always deliver the intended ventilation to the occupants, their real performance is hard to check and they do not adjust minimum outdoor air intake as the ventilation demand varies. Demand control ventilation, based on sensing the rise in space CO_2 concentration, can address these problems but is difficult to apply to multiple space systems. Supply air CO_2 control effectively applies CO_2 technology to recirculating systems. It is inexpensive, saves energy, and helps ensure good indoor air quality. $SACO_2$ offers significant benefits for systems that recirculate from multiple spaces. These include energy savings, simple maintenance, better assurance of adequate ventilation and the ability to measure and record performance. Benefits are largest where indoor air quality is of concern, there are large variations in the number of occupants, the climate is extreme, or energy rates are high. It is not suited to single space systems, is not needed where makeup needs exceed ventilation needs, and requires modification if used with two-part ventilation rates. Simulation of Thermal Comfort of a Residential House [5], presents the following, In hot and humid climates thermal comfort can become a problem to the occupants of many residential buildings especially when they are not equipped with air-conditioning system. This paper presents outcomes of an ongoing research work to investigate thermal comfort level in a naturally ventilated residential house in Malaysia using computational fluid dynamics method. Actual measurements of the temperature distribution, relative humidity and air flow pattern were conducted. CFD simulations on the model of the house allow us to visualize the temperature distribution and air flow pattern and velocity in the house. The thermal comfort in the house was found to be well outside the limits specified by ASHRAE standards. CFD simulation was used to investigate the effects of using a ceiling fan installed in the middle of the hall section and rotating at 150 RPM. It was found that the fan produced swirling flow pattern in the hall section resulting in a more uniform temperature distribution inside the house. However, there is no significant improvement in the thermal comfort level in the house. Results of CFD simulations also show that the use of small extractor fans installed on the front and back walls has no significant effects on the thermal comfort level in the house. Application of CFD Simulation in the

Development of a New Generation Heating Oven [6] this paper deals with the application of Computational Fluid Dynamics simulation in the development of a new generation cooking appliance in Gorenje concern. As the oven is multifunctional, radiation, conduction, natural and forced convection mechanisms of heat transfer are used. The Discrete Ordinate model is used for radiation. The density of air is described by incompressible ideal gas equation in a natural convection model. The intention was to create the best possible baking conditions for different heating systems. Several discrete models were created. The influence of geometry change and boundary conditions variations to the velocity and temperature field distribution in the oven cavity was analyzed. The results of numerical simulations are validated with measurements taken from an oven prototype. The agreement was good. After successfully passing the standard tests, the oven came into serial production and was launched on the market. The velocity, pressure and temperature field in the oven cavity have been dealt with the use of CFD simulation. Natural convection, forced convection, conduction and radiation mechanisms of heat transfer were used. The comparison of five radiation models showed that only appropriate model is Discrete Ordinate. Biscuit surface emissivity didn't show significant effect on the surface temperature. Four different oven geometries and eleven variations of velocity boundary conditions were used. Control of the circulating hot air velocity in the oven cavity has emerged as the decisive impact factor on the baking quality. Predictions from numerical simulations are confirmed by the positive results from the validation and the functional testing of the oven prototypes and they allowed fast optimization of the temperature field in the oven cavity. Theoretical and numerical studies of coupling Multizone and CFD models for building air distribution simulations[7] presents the followings, multizone network models employ several assumptions, such as uniform temperature and pressure and quiescent air inside a zone, which may lead to inaccurate results in flow calculations. These assumptions can be eliminated in the zones, where the assumptions are inappropriate, by coupling a Multizone network program with a Computational Fluid Dynamics program. Through theoretical analysis, this paper proves that the solution of air distribution by using the coupled program exists and is unique. Three possible coupling methods are then discussed in the paper. The best method is pressure-pressure coupling that exchanges pressure between the Multizone and CFD since it is most stable and can always lead to a converged solution. Numerical tests were further performed to verify the theory and it demonstrated that the coupled program is able to effectively improve the accuracy of the results. The Multizone and CFD programs were coupled through the airflow rates or pressure drop at the interfaces. Through theoretical analyses, this paper has proved that the coupled program has a solution and the solution is unique with the three coupling methods. This investigation also used the Scarborough criterion to evaluate the convergent performances and to analyze the stabilities of the three coupling methods during their iterative coupling processes. Method, which exchanges pressure boundary conditions between Multizone and CFD programs, is unconditionally stable, while Methods 2 and 3 are conditionally stable. Two numerical experiments were conducted to demonstrate the theory: one was pressure gradient flow in an office suite and the other was cross ventilation in a four-zone building model. Although experimental data are either unavailable or limited for these two cases, numerical results show that the theory to be correct and valid. To further validate the numerical results, future studies of field measurements are required.

III. PROBLEM DESCRIPTION

Effective design of air gate and experiments on maintaining different temperature in a single zone is vital for high temperature region. Literature reviews provides work flow across design and simulation of cooling in a closed system and devices related to this system are presented. But a computational fluid dynamic integrated approach for design and experimental investigations of air gates to maintain different temperature is not presented and is in requirement for providing effective design of air gate. The solution contains the design and CFD based investigation of air-gate system in a bi-temperature system. In this research work, an attempt is made to design and fabricate an air-gate model for a system where two different temperatures are maintained, this is further investigated through CFD and the results are presented for further manufacturing process.

IV. DESIGN AND CFD SIMULATION

A. *Design Details*

The Air-Gate Components contains the following components.

- Base Box for Bi-Temperature Testing
- Air Inlet / Exit Cover
- Inlet Fan / Exit Fan Containers
- Heating Element
- Thermo Couples

B. *Material Property*

Category : Structural Plywood
Bending Stress : 100 Mpa
Tensile Strength : 60 Mpa
Shear Strength : 6.8 Mpa
E : 2.15e4
G : 1.075e3
Density : 546 Kg/m^3

C. *Box Model and Dimensions*

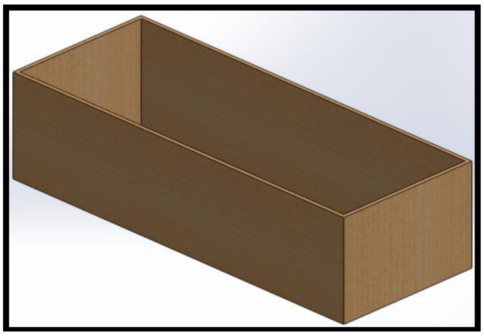


Fig.5 – Base Model

Size : 4' x 2' x 2'
Air-Capacity : 0.27461 m^3
Material : Plywood

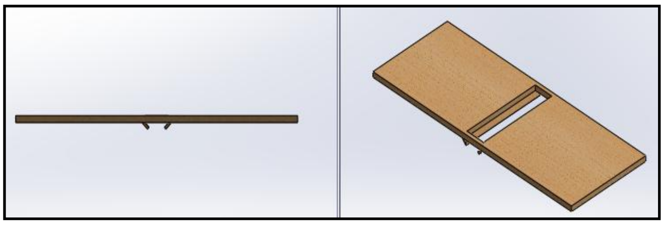


Fig.6– Top and Bottom Covers

Size : 4’ x 2’ x 0.03’
Air- Opening : 1.37’ x 0.33’
Material : Plywood

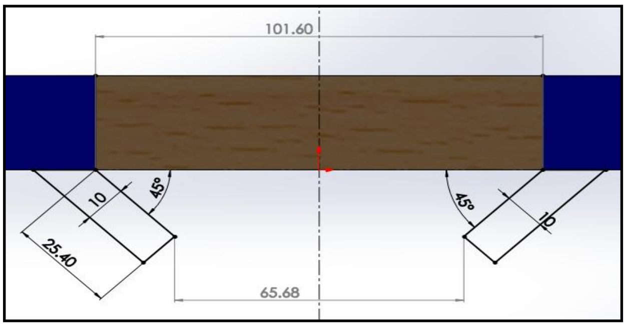


Fig.7 – Nozzle Dimensions

The following figure shows the inlet and outlet fan boxes which will be mounted at the top and bottom of the base box.

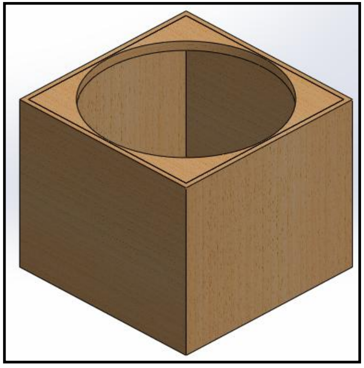


Fig.8 – Fan Box

Size: 2’ x 2’ x 2’ & Material: Plywood. The following view shows the assembly view of the bi-temperature model.

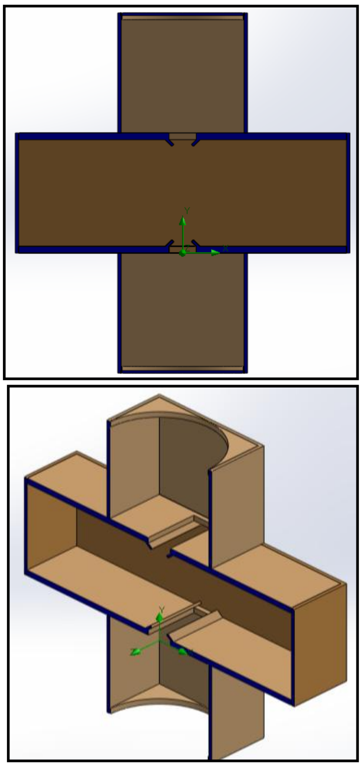


Fig.9 – Assembly View

The following figure shows the fan details of the model.

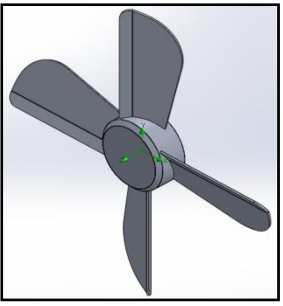
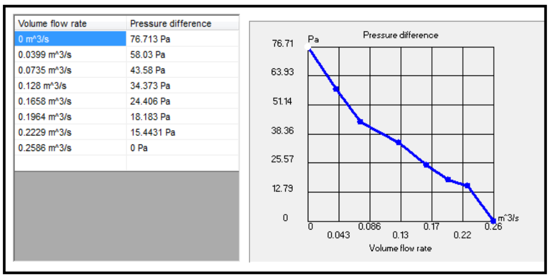


Fig.10 – Fan

Make : Visha (India)
Rpm : 2880 Max.
Power : 180 Watts
Amps : 3 Amps. (Max.)
Volt/ Frequency : 230 / 50 Hz
Diameter : 310 mm
Pitch : 23 mm
Hub Diameter : 70 mm



Volume flow rate	Pressure difference
0 m^3/s	76.713 Pa
0.0399 m^3/s	58.03 Pa
0.0735 m^3/s	43.58 Pa
0.128 m^3/s	34.373 Pa
0.1658 m^3/s	24.406 Pa
0.1964 m^3/s	18.183 Pa
0.2229 m^3/s	15.4431 Pa
0.2586 m^3/s	0 Pa

Fig.11- Performance Curve

D. Boundary Condition and Results

Inlet Fan Speed	:	2880 Rpm
Outlet Fan Speed	:	2880 Rpm
Working Pressure	:	1 Bar
Left Side Heat Source	:	100 Watts
Right Side Head Source	:	25°C

The above details shows the boundary condition used in this analysis. The left sides of the base box have a heating source of 100 Watts whereas the tight side is at ambient temperature. The entire simulation runs at atmospheric pressure. The following plot shows the pressure of the bi-temperature of the system, the peak pressure is at the top of the box which is decreased over the down of the box where it is below than the atmospheric pressure.

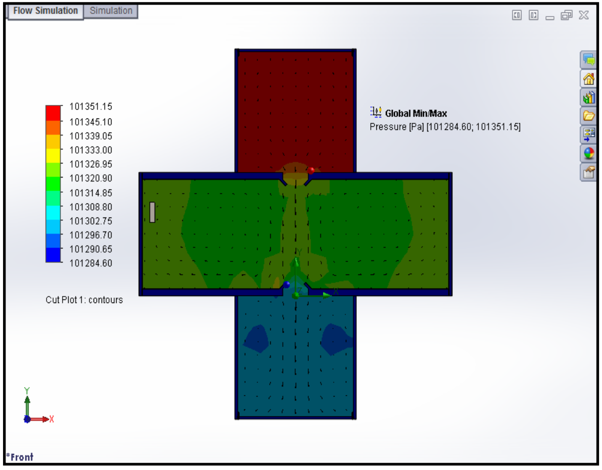


Fig.12 – Pressure Plot

The following figure shows the velocity plot of the system and it is clear that the velocities in the inlet and outlet nozzles are maintained and it shows the air-gate effectiveness.

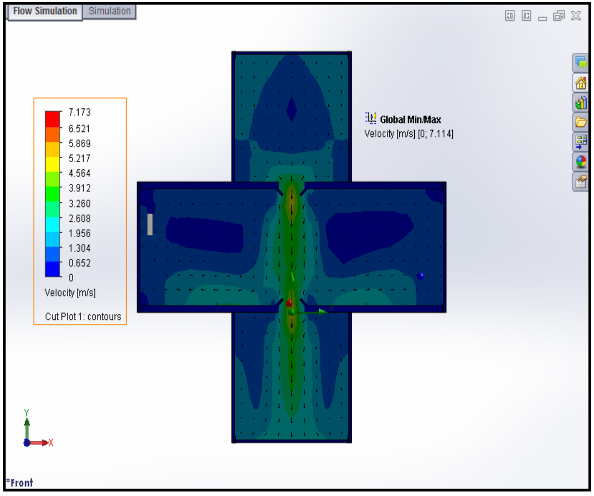


Fig.13 - Velocity Plot

The following figure shows the fluid temperature of the testing box case where the peak temperature is absorbed at the left and the room temperature is almost maintained at the right.

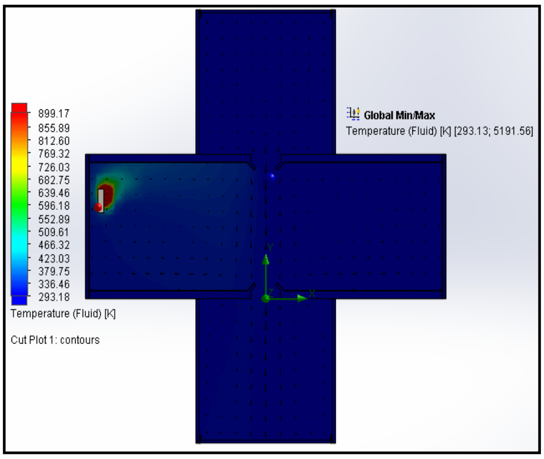


Fig.14 – Fluid Temperature

The following figure shows the solid temperature of the model, it is at ambient at the walls and peak at the heat source

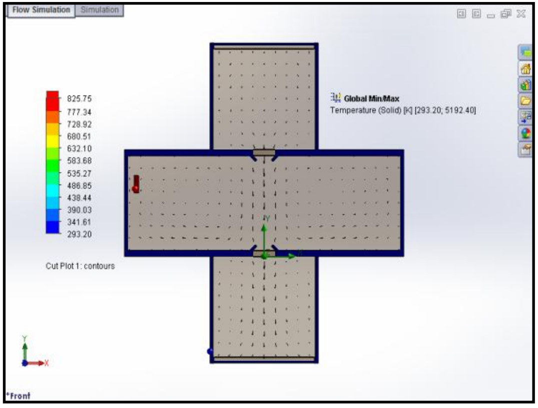
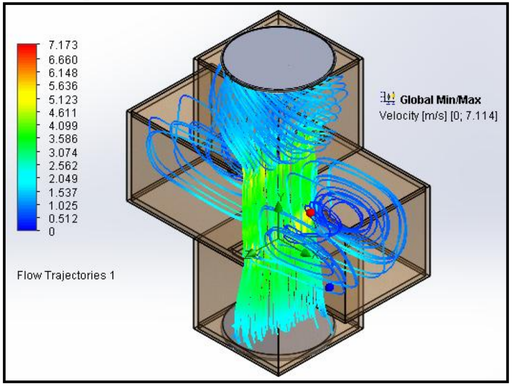


Fig.15 – Solid Temperature

The following figure shows the velocity trajectory of the air flow inside the box, the steam clearly shows the flow pattern and is maintained in the nozzle regions which are important to create a proper air-gate to maintain different temperature between two spaces.



V. CONCLUSION

The present work concludes with the followings. Objective formulation has done successfully followed by literature review to identify the design, methods of numerical analysis, types and their specifications, working principles, etc., the review provides research gap identification and solution for the same as a new research work in the form of project. The design and CFD simulation shows that the air-gate maintains different temperature between left and right side of the box, the velocity and mass flow rate generated by the fan is enough to maintain temperature in the regions, It is encouraging to go for the experimental setup and verify the results, plywood is preferred to maintain low heat transfer to atmosphere, in the next phase the experimental setup will be finished and compare with simulation to evaluate the air-gate performance.

REFERENCES

[1]. A.M. Foster, et.al. (2007), 'Three-dimensional effects of an air curtain used to restrict cold room infiltration'. Applied Mathematical Modelling 31 (2007) 1109–1123. Elsevier Inc.

[2]. Nedhal Ahmed M., et.al.(2011), 'The Effects of Orientation, Ventilation, and Varied WWR on the Thermal Performance of Residential Rooms in the Tropics' , Journal of Sustainable Development Vol. 4, No. 2; April 2011.

[3]. By George J. Berbari, et.al. (2006), 'Fresh Air-Han Comparison and Design', American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc. (www.ashrae.org). Published in ASHRAE Journal Vol. 49, Jan. 2006.

[4]. David Warden, P.Eng., Member ASHRAE (2004), 'Supply Air CO_2 Control of minimum outdoor air for multiple space systems', ASHRAE Journal, October 2004.

[5]. MasineMd.Tap, et.al. (2011),' Simulation of Thermal Comfort of a Residential House', IJCSI International Journal of Computer Science Issues, Vol. 8, Issue 5, No 2, September 2011.

[6]. Zlatko Rek, et.al. (2012), 'Application of CFD Simulation in the Development of a New Generation Heating Oven', Strojniški vestnik - Journal of Mechanical Engineering 58(2012)2, 134-144.

[7]. L.Wang, et.al. (2007), 'Theoretical and numerical studies of coupling Multizone and CFD models for building air distribution simulations', Indoor Air, 17, 348-361.

[8]. Murat Basarir, Dr. Mauro Overend (2010), 'Interim Report on the Energy Appraisal of Retail Units: Assessing the effect of open doors on energy consumption and thermal comfort', CUED/D-STRUCT/TR232, University of Cambridge, Dept. of Engg.

[9]. M/s Tim Dwyer (2007), 'Air curtains for saving energy in buildings', 10th CPD collection.

[10]. Ms.Htwe Tin,(2009),'Design of Air Conditioning System (City-multi) for Seminar Rooms of Technological University (Kyaukse)', Proceedings of the International Multi Conference of Engineers and Computer Scientists 2009 Vol II IMECS 2009, March 18 - 20, 2009, Hong Kong.

AJISH SOMAN PG scholar Thermal Engineering
RVS College of engineering and technology Coimbatore Tamil Nadu India

V HARI GANESH M.E Assistant professor
RVS College of engineering and technology Coimbatore Tamil Nadu India

BIBIN P VARGHESE PG scholar Thermal Engineering
RVS College of engineering and technology Coimbatore Tamil Nadu India