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

Samir Moujaes[a], Satya Gurram[a]

[a]*University of Nevada Las Vegas, Las Vegas, USA, samir.moujaes@unlv.edu*

Abstract:
A 3-D model is developed to study the behavior of a hydronic heating system in a room on the distribution of temperature and velocity profiles created by the natural convection setup in the room. This hydronic system uses a closed water loop where the fluid is heated in a boiler and circulated through pipes to a heat transfer component such as a radiator or finned –tube baseboard unit. The system can be used to effectively control the air temperature and the mean radiant temperature of an enclosure.
Two numerical models of a room one without a window and the other with a window using a flat panel baseboard heating systems were simulated using Star-Cd software. The effects of temperature on the density, viscosity and thermal conductivity were considered and suitable user subroutines were programmed. The numerical models were simulated for different conditions of inlet water temperatures.
The heat outputs for different temperature conditions were calculated. For one of the cases a comparison of the heat delivered as mentioned by the manufacturer's data was compared to the calculated one and found to be only 6% different from what was specified.
Draft Rate (DR) was calculated for both the models to check for thermal comfort. The calculated draft rate was found to be much lower than the maximum value specified by ASHRAE of 20 % for acceptable DR.
A correlation is also suggested in this study from calculations for average Nu (Nusselt number) versus Ra (Rayleigh number) for the different models and room conditions simulated as follows:
Nu = 0.0002Ra+2.48 (R^2=0.55 and a data scatter of ± 5%)

Keywords:
Computational Fluid Dynamics, hydronic heating, natural convection, Rayleigh number, Nusselt number.

1. Introduction

The systems in which water or steam carries heat to the areas to be heated are Hydronic heating systems. Hydronic systems are closed loop where water is heated in a boiler and circulated through pipes to a heat transfer component such as a radiator or finned –tube baseboard unit. Hydronic radiant baseboard heating is a common application that has been used for over 50 years in the United States.
The most commonly hydronic heating systems are floor radiators, wall panel radiators, and baseboard convectors. Floor and wall panel radiators have bank of tubes or a panel embedded in the floor or the walls through which hot water flows, where as a baseboard convector has tubes with fins running along the walls at a small height above the floor. These base board convectors [1],[2],[3] are not embedded in structure of the building but are located outside of the interior walls.

1.1 -Advantages and Limitations

Hydronic heating systems are generally recognized [4] for several advantages: provision of thermal comfort, energy savings potential, and compatibility with various energy sources, quietness and a broad scope of application . The study of hydronic radiating systems can be divided into four sub studies [5]: comfort and health, economic, architectural and limitations.

1.2- Comfort and Health

One of the major reasons for renewed interest in the hydronic heating systems is that such a system can provide a reasonable approach to thermal comfort. [6] conducted a study to quantify the thermal sensations felt by the body, local discomfort, comfort and the general acceptability of the thermal environment when heated with four different heating systems. The results showed that the occupant's preference rankings of the four systems from best to worst was floor heating, baseboard, forced air and ceiling system.

1.3-Economic

It has been reported that hydronic radiators can reduce energy costs up to 30 percent [7] or more when compared with forced air systems for the same thermal comfort.

1.4-Architectural

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

1.5- Limitations

These systems cannot maintain required thermal comfort [4] when the thermal loads are excessive. These systems are also not suitable for buildings with high ceilings as all the warm air tends to accumulate in the top of the room.

1.6- Objective of Paper

Several previous research works have been focused on under floor radiators and wall panel radiators (baseboard heaters) [8],[2] not much information is available on the baseboard convectors from a predictive point of view(CFD) in the current literature . Hence a numerical analysis on the performance and efficiency of baseboard convectors is warranted and has been performed to provide some insight in this application under different operating conditions and a relation between dimensionless quantities Rayleigh number (Ra) and its effect on Nusselt number (Nu) will be developed which can be used to predict the performance of the convector for different temperature conditions.

2. Physical Model

The baseboard heater considered for the analysis has two flat panel tubes assumed to run along the length of a room of a 3X3X3.0 m. The water from a boiler is fed into the flat panel tubes at one end .The water as it travels though the tubes looses heat to the convector which in turn looses heat to the air outside. These tubes are backed by a series of fins which help in increasing the heat transfer rate from the convector. These convectors are placed close to the floor about 0.05 m high. In most of the cases the convectors are directly placed beneath a window in the room or an outside wall to counteract the effect of downdrafts in cold climates.

Figures 1,2&3 show the different views of a commercially produced two tube baseboard convector used for the simulation

Fig. 1. Wall hung baseboard

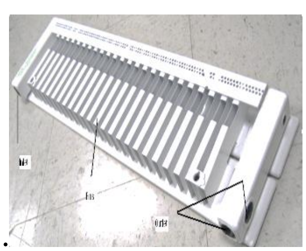


Fig. 2. Backview of baseboard heater

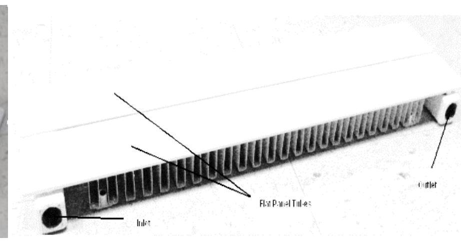


Fig. 3. Underside view of baseboard heater baseboard heater

These heaters are usually hung about 5 cm above the floor level to allow return air to pass through the underside of the unit as seen in Fig. 3 and pick up the heat and move through the channels of the heater by natural convection and then is circulated through the room. These systems are a closed circulation water system usually in one of three circulation configurations (not discussed here): series pipe system, one pipe system and a two pipe reverse circulation system.

3. Numerical Model

In order to save computational time the forced convection effect of the water inside the convector channels is not considered and a constant temperature boundary condition is used on the inside walls of the convector channels. A 3-D section of the convector is considered which includes a single modular section of a fin and the space between it and the next two fins on either side. Hence in the X-Y plane the limits of the solution extend to all the walls/ceiling/floor of the assumed room size 3x3x3 m. A plane of symmetry parallel to the X-Y plane and situated in the middle of the fine is considered. The extent of the solution field in the z-direction is 0.02m, while it is a cross-sectional area of 3x3m in the X-Y plane. The assumption here is that this modular pattern of the finned convector is repetitive all along the Z-direction of the room due to the assumed length dimension of the heater as opposed to its height. The end effect of the walls normal to the z-direction are neglected and an assumption that the temperature of the water through the convector in the section being simulated is the average temperature of the hot water between inlet and outlet temperature from supply to return to simplify the investigation of this computational problem. The convector is assumed to be placed 0.05 m above the floor along the length of one of the walls as recommended by the manufacturer. The convector has two flat panel tubes running along the length. These tubes have a series of fins along their length. The section of the convector considered for the analysis is shown in Fig. 4&5.

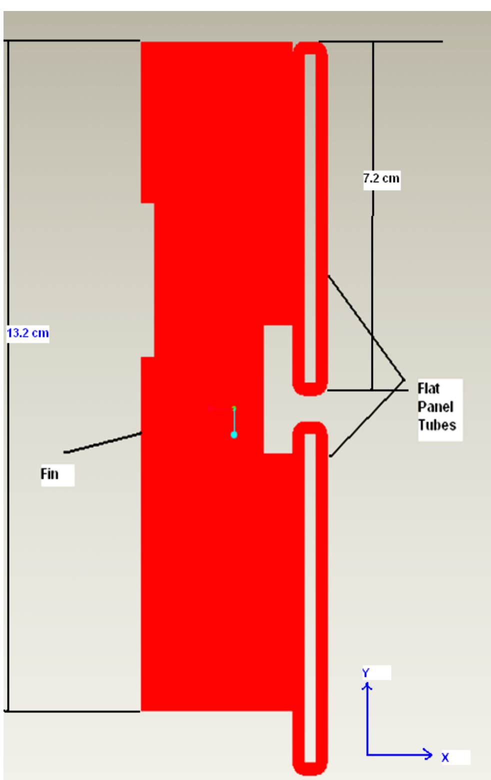


Fig. 4. Cross section of heater with13.2 cm height of fin and 7.2 cm height of one water channel

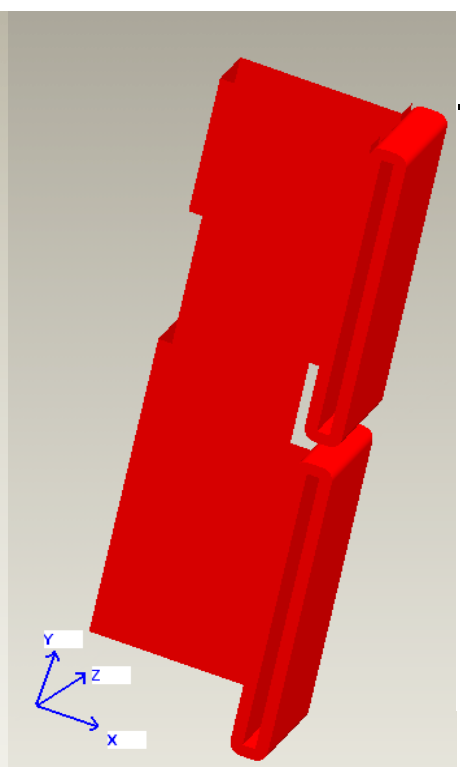


Fig. 5. Isometric view of one section of the heater simulated

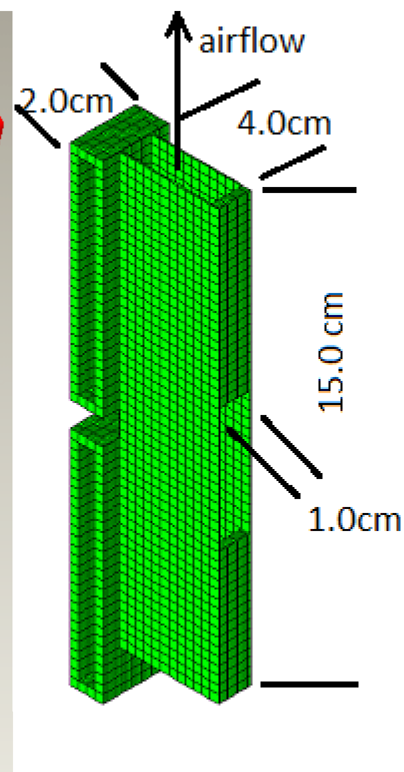


Fig 6. Schematic simulation geometry near heater

Fig. 6 shows a detailed view of the fin/channel geometry attached to the main body of the heater section. The width of the simulation field is 2.0 cm by the cross section of the room i.e. 3.0x3.0m. The channel cross section is 4.0x1.0cm and is around 15.0 cm high. The ends of the simulation are considered as planes of symmetry on both sides of the 2.0cm dimension. The grid geometry is very small (near the wall)and is of the order of less than a 0.1 cm (normal to the solid wall) to capture the gradients of velocity and temperature.

The simulation was performed using STAR-CD software [9] used on a parallel processor computer at the university. The equations employed for that purpose are the usual continuity, momentum and energy equations for 3-D Cartesian coordinate simulation. A low Reynolds number k-ε turbulent model [10] was used with the Boussinesq approximation term added in for the gravity body force [11] to take care of natural convection fluid flow. User supplied subroutines were also provided for the thermal conductivity, viscosity and density of air as a function of temperature to describe better how these properties affect the three conservation and the turbulent model equations. In general the grid density was increased near the solid boundaries of the room and the baseboard heater where it is expected that higher gradients of velocity and temperature will exist. A discussion of the grid independency will follow in section 4. A tetrahedral mesh was used to generate the grid. The numerical simulations were run in steady-state mode with a residual tolerance of 10E-4.

From the manufacturer's data it was deduced that the effect of radiation as an overall heat transfer mechanism was at its highest approximately about 8% of the total heat transfer and hence as a first approximation its effect was not considered in this simulation and hence natural convection was the only effect considered for the air and conduction heat transfer was considered through the metal of the baseboard radiator to its outer surface in contact with the air.

Two general models were used to perform parametric studies of the performance of the baseboard heater as follows:

1. Model I - An interior room, with interior wall temperatures of 20C. A second sub-model was used where one of the walls is assumed as an unheated partition wall at 14C.
2. Model II - An exterior room with a window which is centered heightwise in the room and is 0.9 m in height right above the baseboard heater where the window temperature is at 0.0C and inside surface of the outside wall temperature to have a value of 12.0C.

Both the numerical models were identical in their grid mesh except for the boundary conditions used.

4. Results

A grid independency test(as shown in Fig. 7&8) was performed by changing the grid density over the solution field to determine what is close to the minimum grid density can be used to generate the most accurate numerical results with 2-3% of the next larger grid density chosen in this study.

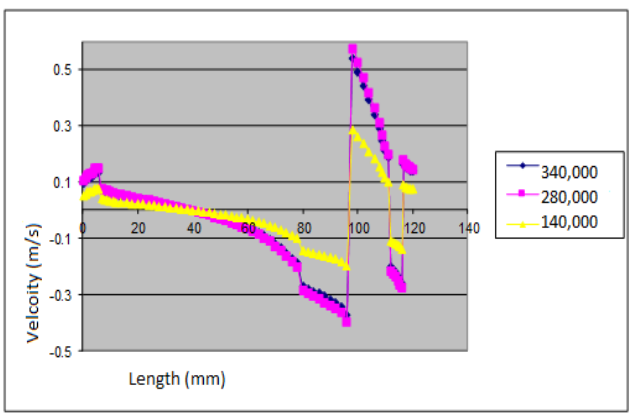


Fig. 7. Grid independency at a line parallel to the Y axis and centered in the middle of solution field plane XY

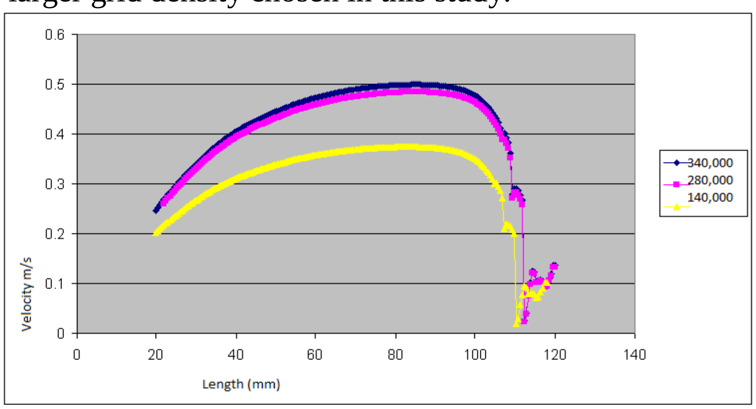


Fig. 8. Grid independency at a vertical line at 0.0762 m from the left wall where the baseboard heater is placed

The results in these figures indicate that the velocity component in the y direction has stabilized its value as there is no appreciable difference between the 280,000 and 340,000 nodes chosen. So the 280,000 node density was chosen for the rest of the simulations in this study. As shown in (11) and was validated by their experiments the model used here of low Re k-ε turbulent model is also used here as the range of Ra numbers used here was well within the range of Ra turbulent numbers used in (11). It was difficult to find studies in the literature that looked at natural convection flow in vertical channels in general. The calculation of the Ra number and any indication of what the critical Ra number is to indicate the demarcation of laminar to transitional flow was hard to find. The indication in (11) that the flow simulated was turbulent was based on the fact that the choice of the aforementioned turbulent model indicated reasonably well agreement with experimentally determined turbulent flow in that work. Also the air movement around the room would be expected to be driven mainly by turbulent natural convection which emphasizes the need to use a turbulent model.

4.1- Model I

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

Fig.8&9 show the temperature and velocity profiles respectively of the air when the temperature of the convector is at 82.5 ^{o}C. The results for other inside wall temperatures for the convector show similar trends and will not be shown here due to the limited page space available for the paper.

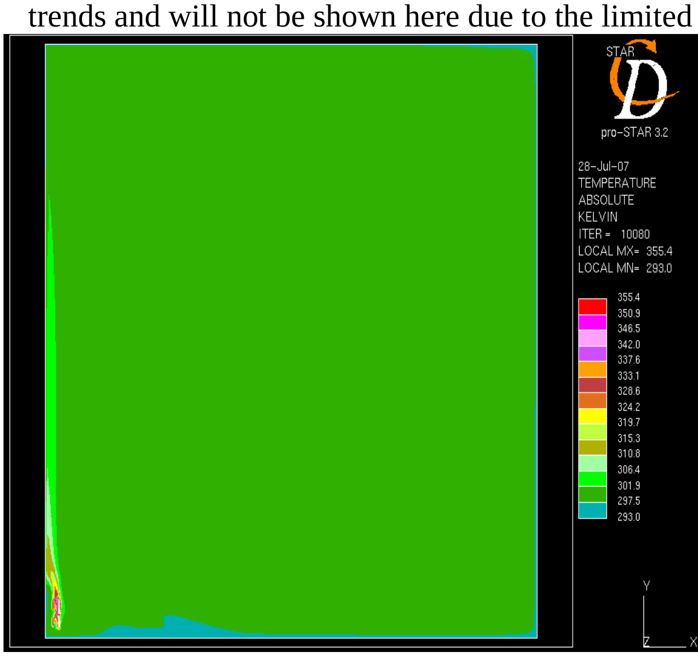


Fig. 9. Temperature profile in the XY plane

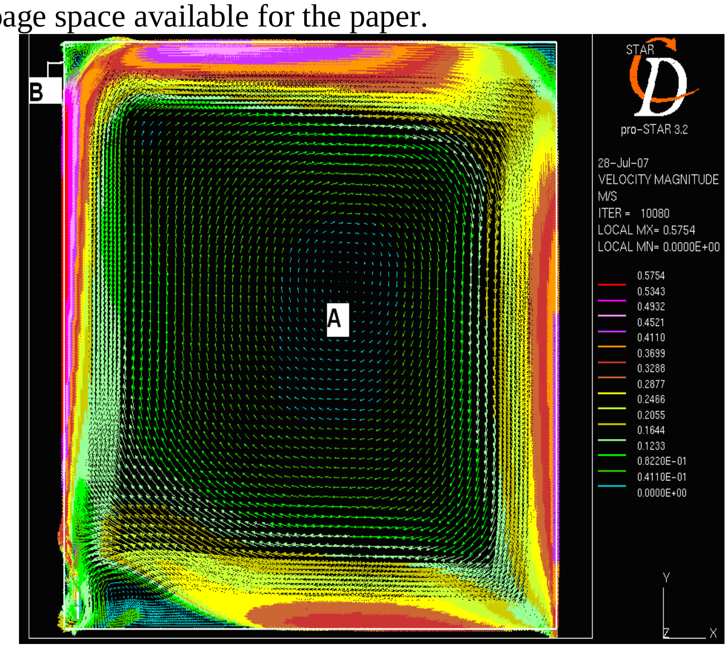


Fig. 10. Velocity profile in the XY plane

These plots are very busy and Fig. 9 indicates the majority of the room volume is maintained at 24.0C (dark green color second from bottom in color display legend). The highest temperatures as expected will be close to the baseboard heater and a plume of higher temperature (light green) than the dark green is close to the left wall where the heat is hung. That temperature is approximately 28.0C. There is a very thin belt layer all around the room with a value close to 20.0Cand is indicated in the light blue color (lowest in color display). As regards to the velocity values in Fig. 10, the red color indicates values of approximately 0.57 m/s. The lowest velocities are found close to the walls of the room (blue color) and are in the vicinity of about 0.04 m/s. The core region of the room is found to have a velocity range between 0.12-0.04 m/s. A clearer picture of the direction of motion in the room is obtained by plotting a graph of the path lines of particles in that room as shown in Fig. 11.

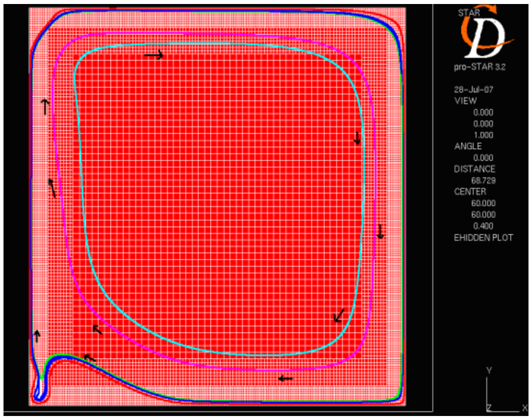


Fig. 11. Flow Pathlines in the XY plane

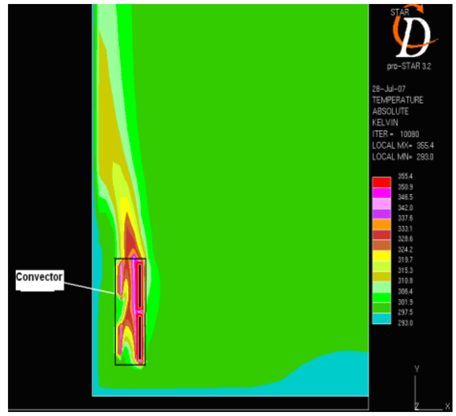


Fig. 12. Closeup of temperature distribution near the baseboard heater

One can notice a large circulation zone around the periphery of the room and is slightly disturbed at the floor of the room close the baseboard heater. This is suspected to be due to the secondary flow that is formed close to the heater but where the baseboard heater geometry cannot accommodate all the secondary flow through the heater air channels to be heated but rather is caught in a secondary flow that drags it upwards.

Fig. 12. Above shows a closer view of the temperature distribution near the baseboard heater which shows the thin cold temperature belt (i.e. 20C blue color) close to the floor and high temperatures inside the fin structure of the baseboard heater close to the metal surfaces.

Another closeup view of the velocity vector magnitude and their direction is shown in Fig. 13.

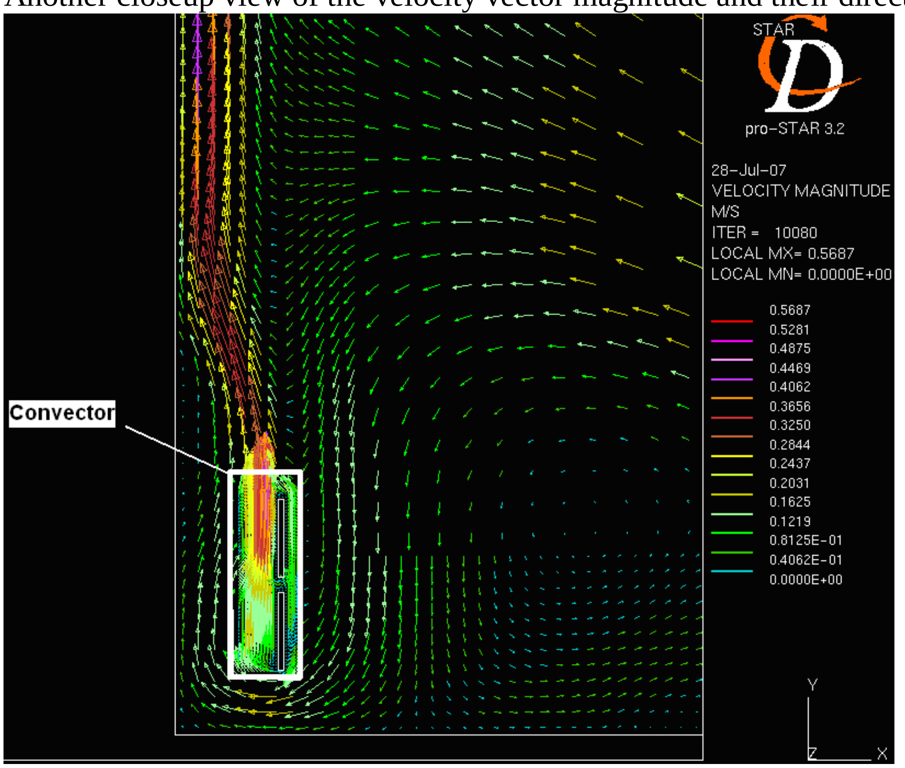


Fig. 13. A closeup view of velocity vectors in the XY plane near the baseheater

The higher velocities are expected near the hot surfaces of the baseboard heater (red color). As the flow is approaching the baseboard heater from the right hand side one notices that the upper flow in the right upper corner evades going into the heater from the its underside while the flow from the right hand side closer to the heater squeezes into that corner and enters the bottom of the heater to pick up that heat and continue on its journey upwards with an added velocity due to the decreased density of that air stream.
To test further how accurate the numerical model with respect to data provided by the manufacturer a calculation was made using the numerical data obtained for the aforementioned run with its boundary conditions. This value is a calculated heat transfer rate in W/m for that run and was found from the current simulation to be 614.0 w/m versus what was determined by the manufacturer by experiment to be 577.0 w/m, with a difference between theory and experiment of about 6% considered to be quantitatively close to each other.

Another issue investigated in this study is the value of the Draft Rate (DR). According to [12] "draft is the unwanted local cooling of the body caused by air movement " and "draft rate (DR) is the percentage of people predicted to be dissatisfied due to draft ".The acceptable higher limit for DR should be less than 20%.
This Draft Rate (DR) is given by

$$DR = ([34 - t_a] + [v - 0.05]^{0.62}) + (0.37 * v * Tu + 3.14) \qquad (1)$$

Where
t_a = ambient temperature, ^{o}C,
v = local mean air speed, m/s,
Tu = Turbulence Intensity, %.
For the present case when the temperature of the convector is at 82.5 ^{o}C, the mean ambient air temperature is calculated to be 26.22 ^{o}C and the local mean air velocity to be 0.1592 m/s.
The turbulence intensity often referred to as turbulence level [9] same here, is defined as:

$$Tu \equiv \frac{u'}{U} \qquad (2)$$

Where
U-mean velocity magnitude, m/s
u'- root-mean-square of the turbulent velocity fluctuations, m/s

$$u' \equiv \sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)} = \sqrt{\frac{2}{3}k} \qquad (3)$$

k- turbulent energy, k=0.419 for low Reynolds number [11] model.
The draft rate is calculated to be DR= 4%.This indicates that the percentage of people dissatisfied with the current setup is about 4 %, which is a fairly low percentage and can be considered acceptable from a statistical point of view. The other simulations used for the temperatures used for the inner surface of the base board heater mentioned earlier in this section have shown to have similarly low values of DR and indicate generally acceptable levels of draft by the majority of the people i.e. over 80%. The average mean value of mean air temperature and mean air velocity we calculated for the volume of the room up to a height of 1.8 m considered to be the level for which there is usually human activity in the room.
As summary of the DR and other calculated values Table 1 is presented below:

Table 1. Summary of values of mean air temperature, mean velocity and DR for different Baseboard heater temperatures simulated(Model I)

Baseboard heater Temperature ^{o}C	Mean air temperature ^{o}C	Mean air velocity m/s	Draft Rate(DR) %
82.50	26.22	0.159	4.03
76.81	25.866	0.156	4.14
71.26	25.4	0.149	4.32
65.70	24.07	0.121	5.33
60.15	23.43	0.101	6.04

For the heater output from the baseboard heater a calculation was made following the method briefly described earlier in this section and the results are summarized in Table 2.

Table 2. Calculated values of heat output for different baseboard heater values(Model I)

Temperature Of baseboard heater ^{o}C	Heat Outputs W/m
82.50	613.6
76.81	520.1
71.26	471.9
65.70	415.8
60.15	342.0

As can be noticed from that table that the heater output should decrease with a decrease of the baseboard heater surface temperature as this will decrease the value of naturally convected heat from the surface of the heater to the air.

The final general calculation made using the internal calculating facilities built into the STAR-CD software is to obtain the average values ,over the surface area of the heater exposed to the convection heat transfer by the air, of the convective heat transfer coefficient,h. These are summarized in Table 3.

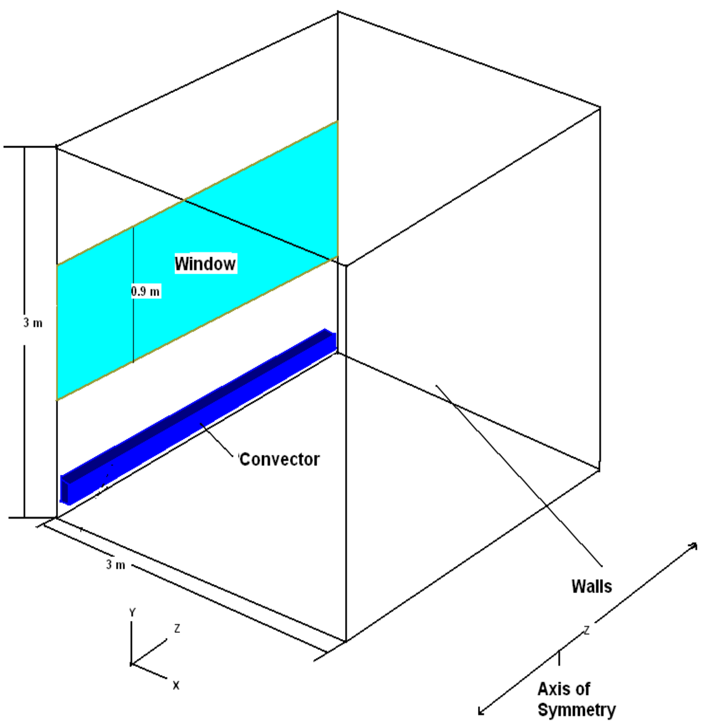
Table 3. Values of Ra (Rayleigh No.), convective heat transfer coefficient, Nusselt No. (Nu)(Model I)

Temperature Of heater °C	Ra Based on D_h	h W/m^2- °C	Nu Based on D_h
82.50	14300	8.90	4.74
76.81	13100	8.60	4.74
71.26	12700	8.30	4.57
65.70	12300	8.04	4.51
60.15	11600	7.60	4.34

The values shown for the Ra number all indicate the existence of a turbulent flow in all the cases considered as those values are within the turbulent range (11). Again as expected the values of Ra, h & Nu are expected to decrease with a decrease of the surface of the heater temperature. One can also notice the monotonous decrease of the mean room temperature as shown in Table 1 due to the decrease of the heat output from the heater.

4.2-Model II

The location of the window in the room is shown in Fig. 14 as shown below.



The room is 3x3x3 m the window is 0.9m high and extends the full length of the wall where the heater is placed. The window is centered in the middle of the wall at a height of its line of symmetry at 1.5 m from the floor.The window inner surface temperature is assumed to be at 0C. and the outer wall (inner surface)temperature is at 14C.

Fig. 14 Schematic diagram of the layout of the window in the room

Due to the limitations of pages allowed for the conference papers the detailed graphs of temperature distribution and velocity in the room will not be presented. The only graph is that of the path lines similar to Fig. 11 above. The summary of the other averaged results similar to Tables 1-3 for this case will also be presented.

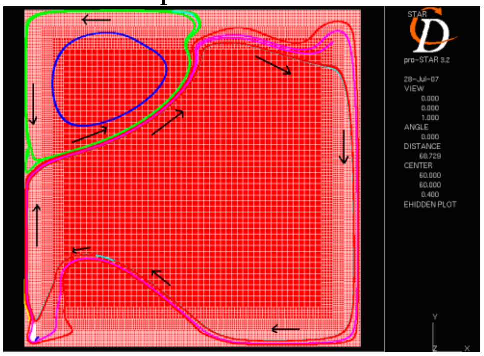


Fig. 15. Path lines shown for Model II and a heater temperature of 82.5C

A basic difference in the air flow around the room is seem in Fig. 15 compared to Fig. 11 as there is another circulating zone right above the window location. This is suspected to be due to the relatively cold temperature of the window and the wall where that window is placed which creates due to the expected increase of the density of the air next to it to start moving downwards parallel to that wall and overcome the upward air movement that was shown earlier in Fig. 11.
Tables 4,5&6 summarize similar results to the first three tables but for model II.

Table 4. Summary of values of mean air temperature, mean velocity and DR for different Baseboard heater temperatures simulated(Model II)

Convector Temperature ^{o}C	Mean air temperature ^{o}C	Mean air velocity m/s	Draft Rate(DR) %
82.50	24.19	0.117	5.84
76.81	23.79	0.100	5.38
71.26	23.07	0.092	5.18
65.70	22.35	0.090	5.35
60.15	21.04	0.084	5.41

Lower mean temperatures and mean air velocities are noticed between this Table and Table 1 which is mainly due to the lower boundary conditions values used. The DR values are somewhat higher here though and that is because of the lower mean temperatures and air velocities which make the calculations according to eq. (1) yield higher values of DR.

Table 5. Calculated values of heat output for different baseboard heater values(Model II)

Temperature ^{o}C	Heat Outputs W/m
82.50	605.8
76.81	596.5
71.26	561.48
65.70	524.08
60.15	371.40

This table shows higher values in general for the heat output from the heater as compared to Table 2. This is to be expected as the boundary conditions on the outer wall/glass window demand have more of a heat loss from them than Model I.

Table 6. Values of Ra (Rayleigh No.), convective heat transfer coefficient, Nusselt No. (Nu)(Model II)

Temperature ^{o}C	Ra Based 0n D_h	h W/m^2-^{o}C	Nu Based on D_h
82.50	14900	8.66	4.61
76.81	13600	8.34	4.60
71.26	13400	8.15	4.49
65.70	12900	7.66	4.30
60.15	12300	7.40	4.22

Here the general values of the three parameters calculated (Model II) do not seem to vary much from those values obtained for Model I earlier which can be explained partially as these are calculated parameters that would probably be most affected by the values of the baseboard heater surface temperatures whose simulated range of values i.e. from 82-60C has remained the same for both model simulations and whose mean room air temperatures has not varied by much between the two model cases.

The final result presented here is a general correlation between the Nu and Ra numbers for the difference cases considered in this study which includes models I&II.

Figure 16 shows the results of a best linear fit for the effects of Ra number on the Nu number. The Ra number calculation is based on the temperature difference of the heater surface and that of the mean air temperature calculated for the air in the room.

The correlation is Nu = 0.0002Ra+2.48. The scatter in the numerical data around that suggested linear correlation is fairly tight (5%). This correlation is valid for $11,500<Ra<15,000$ and $4.37<Nu<5.1$ as

derived from the data. Although its applicability is expected to be over a wider range of these variables as the different in the temperature between the heat surfaces and the mean air temperature changes.
The characteristic length for both Ra and Nu is D_h=1.6 cm
D_{h}- channel hydraulic radius

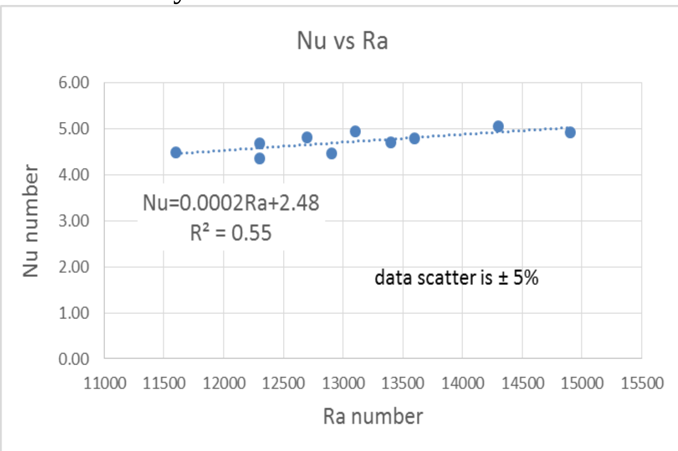


Fig. 16. Correlation of Nu number to Ra for both cases run in Model I&II

5. Discussion and Conclusions

- Two different room models were studied one without a window(Model I) and the other with a window(Model II) were simulated using CFD code. These models were simulated for a wide range of baseboard surface temperature conditions between 60.15 ^{o}C and 82.50 ^{o}C.
- The heat outputs for all the different temperature condition were calculated from the CFD code. Heat output value for one condition where the surface temperature is 82.5C is provided by the manufacturer, this value was compared with the simulated result and the percentage of error between these two values was found to be less than 6 % and is considered to be quantitatively in agreement with the experiment hence giving s higher confidence in the simulation results in general. It also ascertains the fact that with this agreement to effects of radiation on the overall heat transfer are much less than those by natural convection.
- Velocity profiles and path lines were plotted for all the simulations which gave an insight into the air flow characteristics in the room. It was found that there were two distinct flow patterns that developed in the room where we had one large circulation flow when there were only walls on the four sides of the room as opposed to two circulation zones when a window and an outside wall was introduced in the model i.e. Model II. second smaller circulation zone close to the cold wall/window is thought to exist due to the cold air circulation near that wall which counteracted to expected natural hot air rising from the baseboard heater beneath the window.
- The system was checked for thermal comfort by calculating the draft rate (DR). The DR values calculated were well below the maximum limit [12] of 20% as specified in the literature pointing to the fact that this type of heating can provide reasonably comfortable heating conditions in a space.
- The relation between Ra and Nu is obtained as Nu = 0.0002Ra+2.48 with an R^2 value of approximately 0.55 and a data scatter around that suggested linear profile is ±5% which indicates that the heat output values from this base heater can be predicted with a reasonable predictive accuracy as seen by the relatively small scatter. This can be done by deciding what the required air temperature in the room is to be and having decided on the average operating temperature of the baseboard heater one can calculate the Ra number and then the Nu number from that correlation. With the Nu number calculated and knowing the total surface area of the heater once can obtain the h value and hence total heat input of that heater without additional information

from the manufacturer. This relation also indicates that the dependency of the Nu vs. Ra is not very strong for the conditions tested as indicated by the very small numerical value of the slope correlation line. This could be attributed partially to the fact that actual conditions achieved in heat transfer effects close to the surfaces of the base heater are not drastically effected by the achieved room conditions.

References

[1] Shoemaker, R.W.(editor), Radiant Heating, 306p, 1948

[2] Olesen, Bjarne W. Heat exchange coefficient between floor surface and space by floor cooling - theory or a question of definition. *ASHRAE Transactions*, v 106 (PA, 2000, p 684-696

[3] Takemasa, Y.; Kurabuchi T.; Kamata M. Numerical Simulation of Indoor Air Temperature and Wall Heat Flow Distribution of a Heated and Cooled Room. Room Air Convection and Ventilation Effectiveness.1992, pp 99-107

[4] Leigh, S.B, An experimental approach for evaluating control strategies for radiant heating systems, Ph.D. thesis, University of Michigan 1991

[5] Raber, B.F. and F.W Hutchinson, "Optimum Surface Distribution in Panel Heating and Cooling Systems", ASHVE Transactions. Vol.50, 1944, pp. 231-266. Standard Indices of Human Response to the Thermal Humid Environment.

[6] Gagge A. P., T (John B. Pierce Foundation, New, Haven, CT, USA), *ISA*, p 157-164, 1985

[7] Buckley, N.A., "Application of radiant heating saves energy", ASHRAE Journal September, 1989, pp.17-26.

[8] Dorval, O. Hot water panel heating system to save fuel in new institution.*Heating, Piping and Air Conditioning*, v 28, n 6, June, 1956, p 82-84

[9] STAR CD user manual, version 3.24 by ADAPCO (2007)

[10] Chen Q. Comparison of different K-ε Models for indoor air flow computations. Numerical Heat Transfer, Part B, 28; 359-369, 1995.

[11] Yilmaz T., Fraser S., "Turbulent natural convection in a vertical parallel-plate channel with asymmetric heating", International Journal of Heat and Mass Transfer 50 (2007) pp. 2612–2623.

[12] ANSI/ASHRAE Standard 55-2004.Thermal Environmental Conditions for Human Occupancy