4068

ROOM
-VENT
´90

Influence of inlet conditions on comfort in air distributed heating systems

A2

D. Blay
Laboratorie d´Etudes Thermiques, Université de Poitiers
Poitiers, France

G. Greco
Gaz de France DETN
La Plaine St Denis, France

INFLUENCE OF INLET CONDITIONS ON COMFORT IN AIR DISTRIBUTED HEATING SYSTEMS

D. Blay
Laboratoire d'Etudes Thermiques, Université de Poitiers
40, Avenue du Recteur Pineau, Poitiers, France

G. Greco
Gaz de France - DETN
361, Avenue du Président Wilson, La Plaine St Denis, France

Summary

The use of air distributed heating systems in buildings can generate uncomfort conditions mainly due to draughts and temperature stratification if the air inlet parameters are not suitably defined. For a given heat load, reducing air draughts needs to use low inlet velocities but consequently the air inlet temperature must be higher which increases ceiling temperature and thermal stratification. Then, the question to be answered is : for a given heat load, what values leading to acceptable thermal and dynamical comfort conditions can be given to the three inlet parameters (velocity , temperature, area) ? Keeping in mind this objective, we carried out a numerical study of the non isothermal air flow patterns for numerous values of the parameters. Numerical calculations were performed using a 3D finite volume code which integrates the local differential equations describing the flow (mass, momentum and energy conservation, k-ε turbulence model). Results point out the influence of the parameters on the maximum floor velocity, on the temperature difference between top and bottom as well as on the ceiling temperature . Useful correlation laws are given to improve the state of the art for the designing of air distributed heating systems.

Introduction

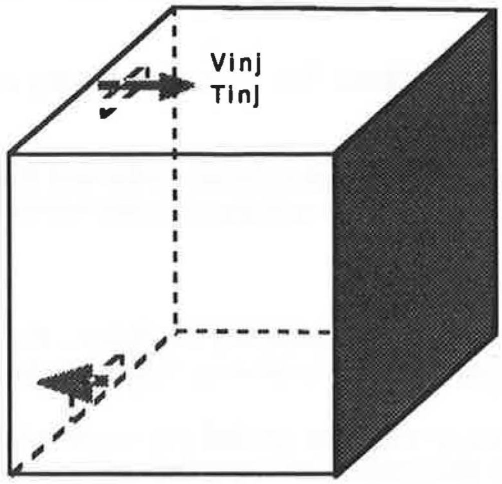
When designing warm air heating systems, much attention must be paid to inlet conditions. These conditions have a major influence on the comfort and mainly on draughts and thermal stratification. Till now, these inlet characteristics were set up on the basis of simplified analysis - using the classic laws of the 2D isothermal jet in an infinite medium- or on the basis of dispersed empirical results.

Due to free convection effects along the cooled walls and due to confining effects, it seemed to us arguable to use the 2D isothermal jet characteristics to characterize the behaviour of a three-dimensional non isothermal jet in a cavity.

In order to refine the dimensioning laws, we carried out numerical simulations of the real flow resulting from the injection of warm air into a room in presence of heat losses.

Description of the configuration

The studied configuration is sketched below :



The geometrical characteristics were the following:

- room : length: 3.6m width : 3.6m height : 2.5m

- inlet: located at 15 cm from the ceiling, height: 5 cm, variable area: 30,60,120,240 cm^2

- outlet : located at 5 cm from the bottom, height : 10 cm, area : 480 cm^2

- existence of a vertical symmetry plane

Numerical formulation

Method

Numerical simulations were made using an iterative finite volume method computer code which allows the integration of the local tridimensional equations for conservation of mass, momentum and energy. Turbulence was taken into account through a classical k-ε turbulence model and log-laws at the walls (1,2).

The grid used a variable mesh in order to calculate as accurately as possible the jet zone and the boundary layer along the cooled wall . The number of meshes resulted from a compromise between the precision of the calculation and the computational cost:
-longitudinal direction : 21 meshes
-transverse direction : 24 meshes
-vertival direction : 18 meshes

The problem was completely defined when dynamic and thermal boundary conditions at the inlet and the outlet as well as at the walls were given.

Boundary conditions

Inlet and outlet. The value of each velocity component and of the temperature was given at the inlet while the continuity of the velocity component and of the temperature was imposed at the outlet.

Wall boundary conditions. The thermal boundary conditions use for the simulation are the following:

- heat losses were uniformly distributed on the wall facing the jet
- the other walls were kept at uniform and constant temperature (19°C).

Run procedure

Thus, for given thermal wall boundary conditions, the problem was entirely defined with the three injection parameters: air velocity, V_{inj}, air temperature, T_{inj}, and inlet area, A.

A parametrical study was then carried out in order to quantify the effects of these injection parameters on the comfort indicators defined below:
- the maximum velocity, V_{max} , in the occupation zone of the room which corresponds to the volume comprised between the floor anf the horizontal plane at the height of 1.80m.
- the mean vertical temperature difference, ΔT_{mean} , in this occupation zone.
- the mean ceiling temperature, T_{top}.

The different values of these parameters were chosen in order to cover three different levels of heat load: 200w, 400w, 800w. They are summarized in the following chart:

Table 1 Set of boundary conditions

Inlet velocity m/s	(Inlet temp.- 19°C) (°C)	Area (cm^2)	Wall heat flux(w/m^2)
1.5	20	120	22.44
3	10	60	22.44
3	20	30	22.44
1.5	10	240	44.88
1.5	20	120	44.88
1.5	40	60	44.88
3	10	120	44.88
3	20	60	44.88
3	40	30	44.88
6	10	60	44.88
6	20	30	44.88
3	10	240	89.76
3	20	120	89.76
3	40	60	89.76
6	10	120	89.76
6	20	60	89.76
3	40	30	89.76

The converged solution for each case was obtained after a certain number of iterations (always greater than 2000) depending on the values of the parameters . It is interesting to note that the higher the inlet temperature and the lower the inlet flowrate, the more difficult the converging process. This was mainly due to numerical oscillations probably associated to the unsteady behaviour of the flow when buoyancy forces are important compared to inertial forces.

Results

Maximum velocity

The point of maximum velocity was searched in the volume comprised in the occupation zone apart from the walls boundary layers. For each configuration, we noted that this point was located near the floor. The velocity at this point was never higher than 0.25m/s for cases with an input heat rate not larger than 500 w. The only configuration leading to a local velocity higher than 0.25m/s was obtained for a thermal load of 1000w and an inlet flowrate of 8 vol/hr.

Thermal stratification

Results showed that for a given heat load, the best thermal comfort conditions were obtained in the case of large flowrates and low injection temperature. Nevertheless, even for the lowest flowrate situations (2 vol/hr), the thermal stratification in the occupation zone defined above never exceeded 4°C . These results, quite surprising, revealed that whatever the inlet conditions, the stratification remained small.

Ceiling temperature

The indication of the ceiling temperature is not interesting from the viewpoint of comfort but rather concerns energy savings. As it could be a priori supposed, the upper layer air temperature was strongly dependent on the air inlet temperature and for any flowrate, the lower the air inlet temperature, the lower the mean ceiling temperature.

Correlations

In order to quantify all these results, we tried to define correlation laws expressing these values in function of characteristical parameters. Taking into account the dimensional analysis relative to free jets , we expressed their adimensional values versus $\sqrt{A}$ as shown on figure 1, 2 and 3.

We obtained the following results:

V_{max} / V_{inj}	$= 0{,}54\ \sqrt{A}$	correlation coefficient: 86%
$\Delta T_{mean} / \Delta T_{inj}$	$= 1{,}98\ \sqrt{A}$	correlation coefficient: 82%
$(T_{top} - T_{mean})$	$= 1{,}90\ \sqrt{A}$	correlation coefficient: 94%

(where T_{mean} is the mean temperature inside the room)

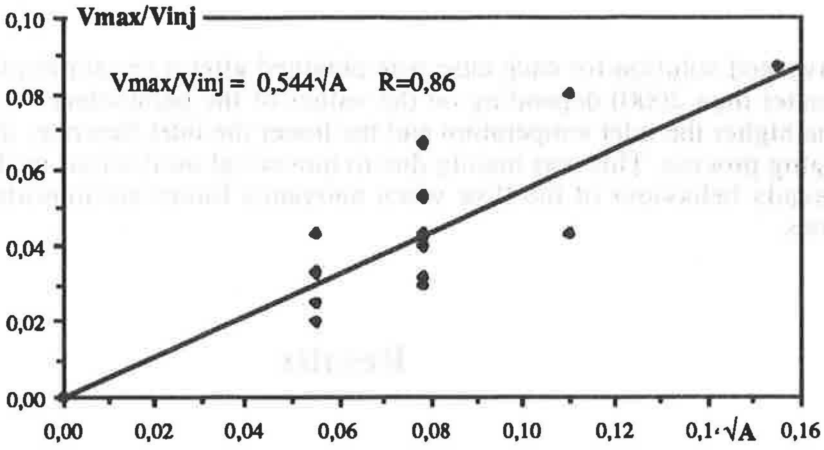


Figure 1 Adimensional maximum velocity vs √A (m)

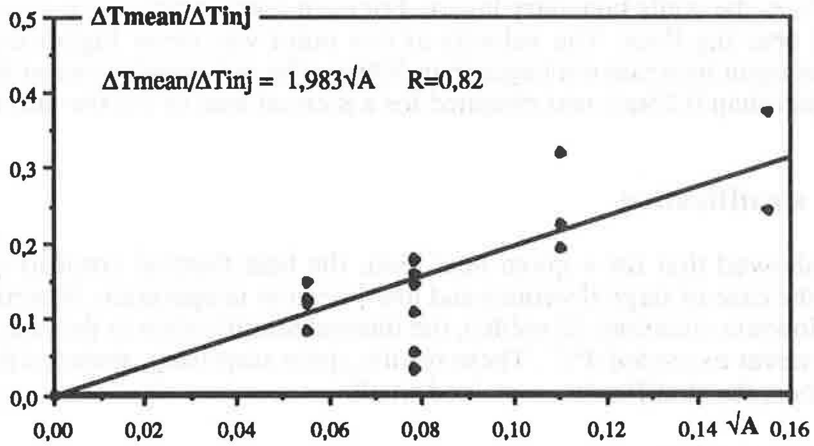


Figure 2 Adimensional mean vertical temperature difference vs √A (m)

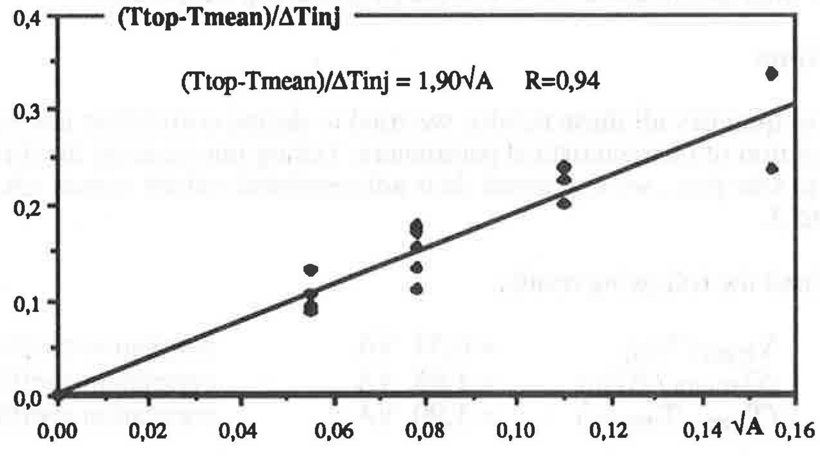


Figure 3 Adimensional mean ceiling temperature vs √A (m)

It can be noted that in spite of correlation coefficients greater than 80%, data concerning the first two quantities (fig.1 and 2) are relatively dispersed. This dispersion can be explained by two main reasons:

- the chosen parameter, $\sqrt{A}$ - representative parameter in the case of a pure jet flow- is not sufficiently relevant of the actual complex flow.
- the lack of precision of the results due to a loose grid specially in the high gradient zones (jet and boundary layers) and to the numerical method itself (choice of the turbulence model, numerical diffusion). Consequently, the absolute error in the determination of V_{max} and ΔT_{mean} is of the same order of magnitude as the quantity itself.

Comparatively, the correlation relative to the determination of the mean ceiling temperature is much better. This is due to :

- the choice of $\sqrt{A}$ as the representative parameter which is here much more appropriate because the structure of the flow in the upper part of the room is close to a jet flow structure.
- the absolute error in the determination of T_{top} which is small compared to T_{top} itself.

Maximum heat rate

Starting from these linear correlation laws, it is possible to determine what is the maximum heat load, Q_{max} , which is compatible with the comfort criteria defined below, ie:

- $V_{max} < V_{lim}$
- $\Delta T_{mean} < \Delta T_{lim}$

where V_{lim} and ΔT_{lim} are respectively the upper tolerable values of the maximum velocity and the maximum mean vertical temperature difference in the occupation zone of the room. We obtained:

$$Q_{max} = k\, \rho_{inj}\, c_p\, V_{lim}\, \Delta T_{lim}$$

where k is a dimensional constant mainly depending on the geometrical characteristics of the room. Its value here is 0.935 (S.I. units) . ρ_{inj} is the air density at the inlet and c_p the air heat capacity.

It is worthwhile noting that Q_{max} is independent of the inlet area. For the particular case where $V_{lim} = 0.25$m/s and $\Delta T_{lim} = 3$°C, the maximum heat rate would be approximately equal to 800w.

Conclusions

This numerical study mainly showed that, for a heat load up to 800w, whatever the inlet parameter configuration simulated (corresponding to flowrates from 2 to 8 vol/hr and to inlet temperatures from 30 to 60°C), the maximum air velocity in the occupation zone practically never exceeded 0.25m/s while the mean vertical temperature difference was always lower than 4°C. This could mean that for the design of an air heating distributed system, no great care has to be taken to avoid draught and thermal stratification due to inlet conditions.

Data concerning the maximum velocity, the vertical temperature difference in the occupation zone as well as the mean ceiling temperature were correlated with the square root of the inlet area (representative parameter in the case of pure jet flows). Mainly due to the choice of this representative parameter and to the precision of the numerical results, this attempt of correlation appeared to be averagely satisfactory for the two former variables, but gave good results for the latter.

References

[1] Patankar S.V. Numerical heat transfer and fluid flow. Washington : Hemisphere, 1980, 197 p.

[2] Launder B.E., Spalding D.B. Mathematical models of turbulence. London : Academic Press, 1972, 169 p.

[3] Ciofalo M., Collins M.W. Prediction of Heat Transfer in turbulent recirculating flows using an improved wall treatment. Numerical Heat Transfer, Part.B, vol. 15, p. 21-47, 1989.

[4] Gosman A.D., Nielsen P.V., Restivo A., Whitelaw J.H. The flow properties of rooms with small ventilation openings. J.of Fluid Engineering (ASME), vol.102, p. 316-323, 1980.

[5] Kuhn D., Oosthuizen P.H. Three-dimensional transient natural convective flow in a rectangular enclosure with localized heating. ASME Winter Annual Meeting, Anaheim, CA, USA, Nov.30 - Dec.5. 1986, HTD, vol.60,61,63, p. 55-62.

[6] Nielsen P.V., Restivo A., Whitelaw J.H. The velocity characteristics of ventilated rooms. Trans. ASME. Journal of Fluid Engineering. Vol.100, p. 291-298, 1978.

[7] Oosthuizen P.H., Paul J.T. Mixed convective heat transfer in a cavity. 23rd national heat transfer conference, Denver, Colorado, August 4-7. Fundamental forced and mixed convection. 1985, HTD vol.42, p. 159-169.

[8] Ozoe H., Mouri A., Hiramitsu H., Churchill S.W, Lior N. Numerical calculation of three-dimensional turbulent natural convection in a cubical enclosure using a two-equation model for turbulence. ASME. J. Heat Transfer. 1986, Novembre, vol.108, p. 806-813.

[9] Simoneau J.P., Inard C., Allard F. Numerical approach of interaction between an injection and laminar natural convection in a thermally driven cavity. ASME Winter Annual Meeting, Chicago, IL, USA, Nov.30-Dec.2. 1988, HTD vol.99, p. 45-51.

[10] Whittle G.E. Computaion of air movement and convective heat transfer within buildings. Int. J. Ambient Energy. 1986, juillet, vol.7, n°3, p. 151-164.

[11] Yoshiaki Ishizu, Kazuyo Kaneki. Evaluation of ventilation systems through numerical computation and presentation of a new ventilation model. Trans. SHASE,n° 24, Feb 1984.

[12] Blay D., Brun C. Numerical study of a non isothermal jet within a cavity. 4th Journées Internationales de Thermique, Algiers, Nov.89.

[13] Perray E., Blay D., Brun C. Air management in dwellings: interzone and intrazone air movement modelling. CLIMA 2000, Sarajevo, 1989.