# Temperatures in Cold Rooms

**Problem presented by**

Andrew Gigiel

*Food Refrigeration and Process Engineering Research Centre (FRPERC)*

**Problem statement**

Cold rooms of varying sizes are used to chill and store food. The air in these rooms is usually cooled by a heat exchanger which draws warm air with the aid of a fan which in turn circulates cool air. The air from the fan takes the form of a turbulent jet which is obstructed by the shelving containing the food leading to a complex flow and heat transfer problem in a difficult (and time varying) geometry. A simple model of the resulting air flow and of the temperature distribution is desired which would let designers assess the impact of different room configurations in a reasonable time and which could also be used to simulate the cooling of food placed in the room.

**Study Group contributors**

J. M. Aitchison (Cranfield University)
C. J. Budd (University of Bath)
D. Fidler (Heriot-Watt University)
A. D. Fitt (University of Southampton)
N. D. Fowkes (University of Western Australia)
A. Grief (University of Oxford)
S. Khan (University of Cambridge)
D. M. Leppinen (University of Cambridge)
C. P. Please (University of Southampton)
C. Wang (Lancaster University)
A. Waugh (Heriot-Watt University)
J. F. Williams (University of Bath)

**Report prepared by**

C. J. Budd, D. M. Leppinen, J. F. Williams

# 1 Introduction

An important factor in the safe storage and distribution of food is that its temperature, when stored in a cool environment, should stay within carefully defined limits. Typically such food is kept in a cool environment in either a large room or in some form of delivery lorry. In an operational environment the configuration of the room/lorry may change with time as doors are opened and shut, food is moved around, the temperature outside the container changes or (in a worst case scenario) the fan cooling the food fails. To ensure safe operation it is thus essential to have an accurate and reliable means of simulating the temperature profile within the room or container subject to a variety of operating conditions. Ideally such a simulation should be fast and usable in a relatively simple computer environment. Currently the FRPERC has developed a package CoolVan to simulate the temperature of the food and air within a refrigerated lorry. This is a relatively simple block model which does not take into account the spatial changes of temperature within the lorry, rather treating all of the air mass as a simple unit at a constant (in space) temperature. Whilst simple this model has proved to be effective in describing the heat transfer in this relatively simple geometry and has been marketed as a commercial package.

In the more complex (and larger) geometry of a cool room this approach has difficulties, particularly in view of the fact that there is a complex flow of air within the room due to the rotating fan, and a far from uniform distribution of temperature and of the food within the room. A typical geometry of such a room is illustrated below. It comprises a fan circulating the air between shelves containing the food. The shelves are aligned parallel to the direction of the flow and can occupy up to 50% of the volume of the room. Illustrated are typical flows both looking down on the room from above and considering a side view.

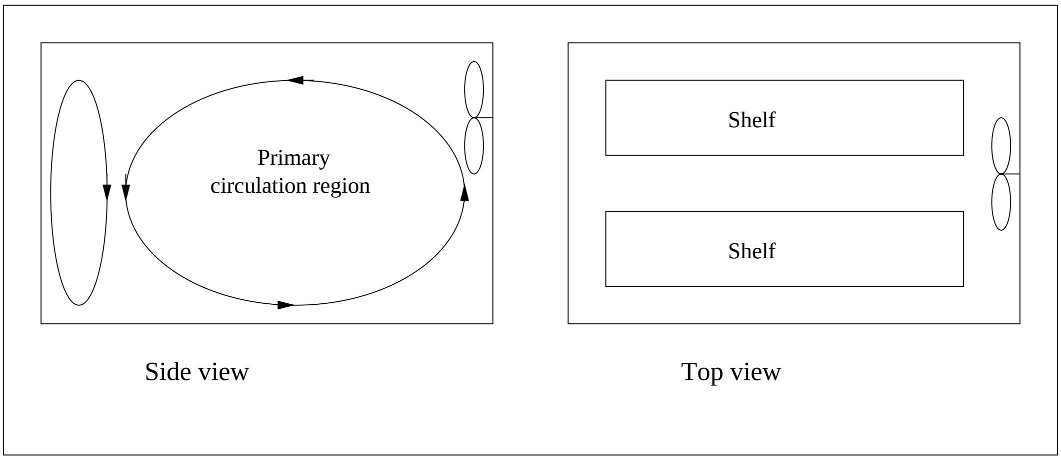


Figure 1: *Schematic of a cold room.*

Essentially this problem is one of a coupled turbulent flow and the resulting heat transfer. The basic equations governing this system are well known and (given an appropriate model for the turbulence) it is possible to model both the flow of the air and of the resulting temperature distribution using modern CFD packages. Indeed this has been done by the FRPERC who use the code Flow3D to simulate the flow of the air from the fan into a room without food in it (looking at various temperature and buoyancy models). The results of such a simulation are given below. From this it can be seen that the flow of the air within the room can be roughly described as occupying two recirculating regions, one of which includes a turbulent jet driven directly by the fan, and the other a much less strong recirculating region comprising air close to the back wall of the room.

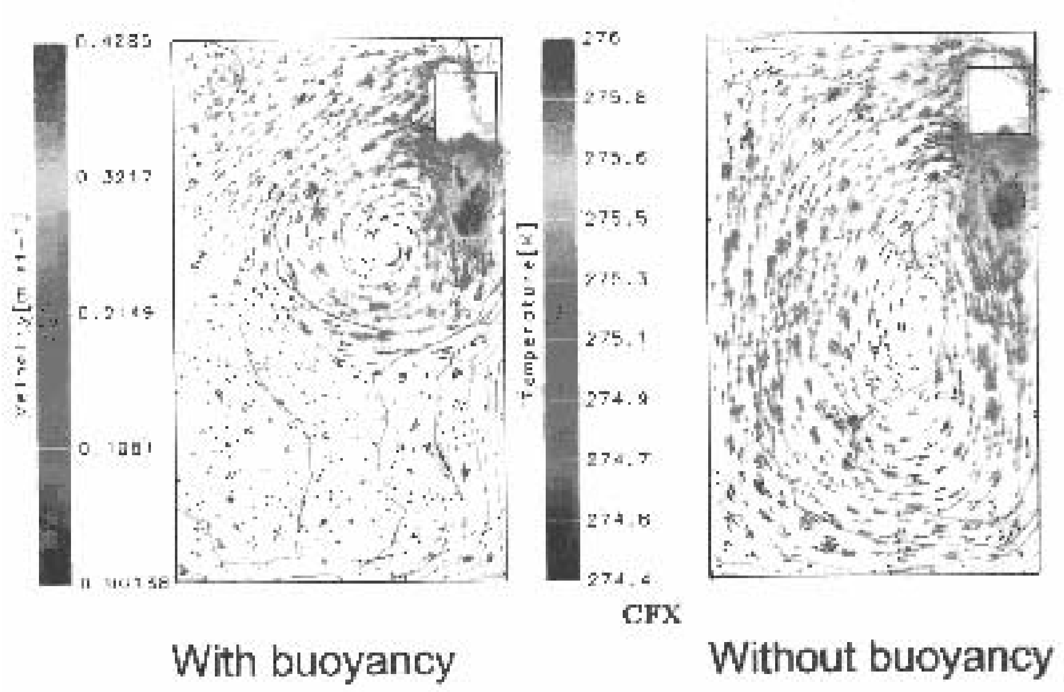


Figure 2: *Typical CFX calculation courtesy of FRPERC. Please note, in these figures the fan is directed downwards i.e. the right hand edge of the figure is in fact the* ceiling *of the room.*

Whilst in principle an accurate simulation, the CFD approach has several inherent difficulties. Each calculation is lengthy and expensive and can only be done on a large computer platform, it is difficult to simulate a variety of operating conditions and it is hard to gain insight into the effects of changing the various parameters in the system. However, it is very useful to gain insight into the overall flow of air in the room and to compare with the results of calculations with other simpler models.

The purposes of the study group investigation were two-fold. *Firstly* to obtain a simple model of the basic features of the flow (and consequent temperature distribution) in the room which could be used either as part of a larger calculation, or on its own to investigate the effects of varying parameters such as the fan strength and position. *Secondly* to consider the development of faster numerical methods for solving the existing models (such as the exploitation of explicit methods for solving the resulting differential equations and/or the use of sparse matrix methods). We will demonstrate that it is possible to derive such models which predict, for example, the size of the recirculating region close to the fan.

# 2 Model reduction

Clearly the full problem of a turbulent flow around the very complex geometry of food contained within shelves cannot be solved in a reasonable length of time (either analytically or numerically) hence a reduced model is required. In this section we state some of the simplifications that we will make for the remainder of this report. Instead of solving the full three dimensional coupled model in a complex geometry we will make the following assumptions:

1. that the temperature and velocity profiles in the air are quasi-static,
2. that the (turbulent) flow in the aisles can be considered to be essentially two-dimensional,
3. that the flow within the aisles is determined by the fan and the macroscopic geometry of the room only, and not by the shelves,
4. that the shelves can be considered to be a porous medium and the flow within the shelves is driven by a pressure gradient acting on the blocking of space by cartons of food, (Ergun's assumption)
5. that the temperature of the food does not affect the flow of the air

These assumptions allow us to calculate the temperature distribution within the cool room as a sequence of decoupled problems.

1. Model and then solve the equations for the bulk flow and pressure distribution in the aisle.
2. Using the results of this calculation solve (the linear problem) for the resulting heat transfer and the consequent air temperature.
3. Once the pressure and temperature of the air in the aisles is known we may determine the temperatures and flows into the shelving units by solving a relatively small set of nonlinear algebraic equations.
4. Finally the heat transfer with the food may be calculated.

A difficulty with this approach is that in reality the two problems 1 and 2 above are not completely decoupled. In particular it is very likely (and the CFD results seem to confirm this) that the flow of the air in the aisle is to an extent dependent upon its buoyancy and hence on its temperature. Hence the two problems 1 and 2 should be solved in parallel. In the next section we consider problem 1 and make some reference to the temperature distributions.

# 3 The air flow within the aisles

## 3.1 A turbulent jet entraining flow

In this section we will model the air flow within the aisles by approximating it by a turbulent jet entraining flow.

From the Flow3D calculations (included in the introduction) we can see that the flow in the aisle breaks into two distinct regions, a convection cell forced by the fan where the velocities are relatively large and a buoyancy driven recirculation region where the velocities are relatively small. Assuming that the fan does not sit within the shelf region, we may model this as a fitted vortex with a prescribed circulation and a region with no velocity.

Our objective is to develop a simplified model of this flow within the cool room that can be rescaled as a function of the strength of fan and room geometry. The primary flow important in the cooling of the food is the main vortex adjacent to the fan, which we assume drives the second vortex and is not affected by it. Consequently, we ignore the secondary vortex and use a turbulent entrainment model for the jet emerging from the fan. This model allows us to to estimate the throw of the air from the fan explicitly. Thus we can calculate analytically the size of the primary vortex in terms of the fan strength and position. We then model the flow in the aisles by a vortex with a geometry consistent with the throw calculated above and a velocity determined by the strength of fan. Once the velocity field of this vortex is known, the pressure field can then be determined by solving the Euler equations in two dimensions.

We initially model the air flow as a jet emerging from the fan which entrains the air beneath it. The end walls then cause a return flow which establishes the primary vortex. The turbulent entrainment model is based on the turbulent plane/jet theory detailed by J.S. Turner in [5]. The concept of an entrainment hypothesis was discussed by Morton, Taylor and Turner in [6]. We are considering entrainment into a jet in a confined region, so the problem is similar to chapter 5 of the thesis by Barnett [1].

Consider the case of a 2-D (planar) buoyant jet with initial momentum flux $m_0$ and buoyancy flux $B$. The jet length, $l_{\text{jet}}$ is[1]

$$l_{\text{jet}} = \left(\frac{m_0^3}{B^2}\right)^{\frac{1}{3}}.$$

At distance $x$ from the source with $x \ll l_{\text{jet}}$ a buoyant jet behaves like pure jet and one can ignore buoyancy. When $x \gg l_{\text{jet}}$ buoyancy dominates and flow will behave like a plume. For the current problem the maximum possible throw will be achieved as $l_{\text{jet}} \longrightarrow \infty, B \longrightarrow 0$ (i.e. the case of a pure jet). If $h$ is the height of the room, then provided $l_{\text{jet}} \gg h$ buoyancy can be ignored when determining throw. In the case where we can decouple the problems 1 and 2 described in the last section, we can use conservation to determine the throw of the jet, and we will assume this is the case.

[1] this formula is dimensionally mixed and should only be applied with the variables in the correct units

To describe this situation we consider a jet that is moving away from the fan and at a distance $x$ from the fan has a velocity $w$, a momentum flux $m$ and a width $b$ and is entraining a return flow which has velocity $v$.

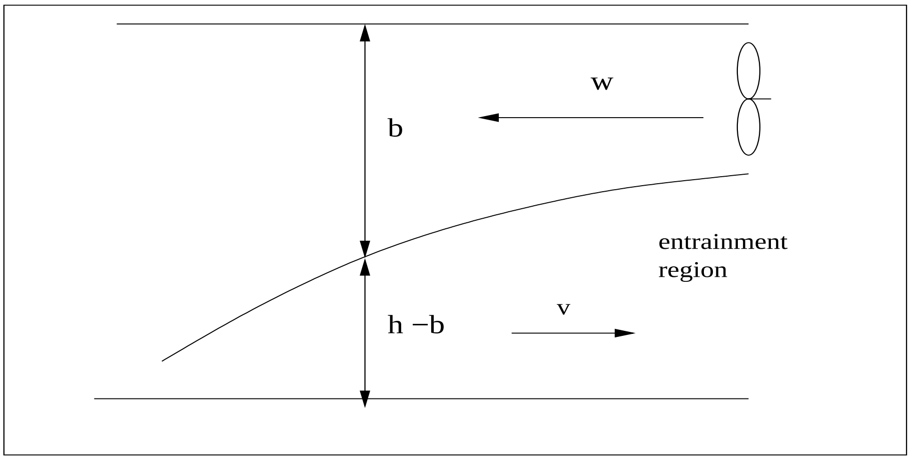


Figure 3: *Entrainment model geometry.*

Following the entrainment model proposed (with experimental support) by Barnett, conservation of volume for the line source of mass and momentum gives

$$\frac{d}{dx}(bw) = \alpha w \tag{1}$$

where the entrainment coefficient $\alpha$ is taken to be

$$\alpha \approx 0.1.$$

Conservation of momentum then gives

$$\frac{d}{dx}\left(bw^2 + (h-b)v^2\right) = 0.^2 \tag{2}$$

Global mass conservation for the total air mass then states that at all points the outward mass flux is equal to the inward mass flux, hence

$$bw + (h-b)v = 0. \tag{3}$$

From this we deduce that

$$v = -\frac{b}{h-b}w \quad \text{and} \quad v^2 = \frac{b^2}{(h-b)^2}w^2 \tag{4}$$

and hence

$$\frac{d}{dx}\left(bw^2\left(1+\frac{b}{h-b}\right)\right) = 0. \tag{5}$$

[2]This neglects the wall friction on the floor and ceiling, and the horizontal gradient of the vertically-averaged pressure. The Study Group did not investigate the circumstances in which these terms can be neglected.

By the definition of momentum flux $m = bw^2$ and of the mass flux $Q = bw$ we have,

$$m = bw^2 \Rightarrow w = \frac{m}{Q}$$

$$Q = bw \Rightarrow b = \frac{Q^2}{m}$$

hence we may rewrite our system in terms of the mass and momentum fluxes, $m$ and $Q$, to give

$$\frac{d}{dx}\left(m\left(1+\frac{Q^2/m}{h-\frac{Q^2}{m}}\right)\right) = 0, \tag{6}$$

$$\frac{d}{dx}(Q) = \alpha\frac{m}{Q}. \tag{7}$$

We can now non-dimensionalize the system by scaling it with respect to the momentum flux at the source $m_0$ and the room height $h$

$$m* = \frac{m}{m_0}, \quad x* = \frac{x}{h}, \quad Q* = \frac{Q}{(m_0 h)^{\frac{1}{2}}}, \tag{8}$$

whereupon, dropping the $*$'s we have

$$\frac{d}{dx}\left(m\left(1+\frac{\frac{Q^2}{m}}{1-\frac{Q^2}{m}}\right)\right) = 0, \tag{9}$$

$$\frac{d}{dx}(Q) = \alpha\frac{m}{Q}. \tag{10}$$

Expanding equation (9) we may derive an equation for $m$,

$$\frac{d}{dx}\left(\frac{m}{1-\frac{Q^2}{m}}\right) = \frac{d}{dx}\left(\frac{m^2}{m-Q^2}\right) = \frac{(m-Q^2)\left(2m\frac{dm}{dx}\right) - m^2\left(\frac{dm}{dx} - 2Q\frac{dQ}{dx}\right)}{(m-Q^2)^2}. \tag{11}$$

Hence,

$$\frac{dm}{dx} = -\alpha\frac{2m^2}{m-2Q^2}. \tag{12}$$

This equation breaks down when $m = 2Q^2$. At this point the jet occupies half of the room height $b = h/2$ and cannot entrain any more fluid. This determines the point of maximum throw $T$ of the jet.

Rescaling the length by the entrainment coefficient, $\alpha$,

$$z = \alpha x$$

we are left with the system

$$\frac{dQ}{dz} = \frac{m}{Q}, \tag{13}$$

$$\frac{dm}{dz} = -\frac{2m^2}{m - 2Q^2}, \tag{14}$$

subject to the initial conditions at the fan given by

$$m(0) = 1 \qquad Q(0) = \left(\frac{b(0)}{h}\right)^{\frac{1}{2}} \equiv \lambda^{\frac{1}{2}} \tag{15}$$

We may solve this system explicitly to determine the throw $T$ as a function of $\lambda$ (and hence of $m_0$ and of $h$).

Firstly we solve for $Q(m)$,

$$\frac{dQ}{dm} = -\frac{m - 2Q^2}{2mQ},$$

with

$$Q(1) = \lambda^{\frac{1}{2}}.$$

Noting that the above is a linear equation for the variable $Q^2$ we may solve it to give

$$Q(m) = \sqrt{m - (1-\lambda)m^2}. \tag{16}$$

Substituting into equation (14) gives

$$\frac{dm}{dz} = -\frac{2m}{2(1-\lambda)m - 1},$$

subject to

$$m(0) = 1.$$

Solving this gives,

$$\frac{1}{2}\ln m - (1-\lambda)(m-1) = z. \tag{17}$$

In order to determine the throw $T$ as a function of $\lambda$ we need to consider the values of $Q$ and $m$ at the throw length. This is the point $z$ such that $m - 2Q^2 = 0$. From equation (16) we have that

$$m_f = 2Q_f^2 \Rightarrow m_f = 2(m_f - (1-\lambda)m_f^2),$$

or

$$m_f = \frac{1}{2(1-\lambda)}. \tag{18}$$

Substituting this into equation (17) determines the throw, $z_f$,

$$z_f = \frac{1}{2}\ln\left(\frac{1}{2(1-\lambda)}\right) + \frac{1-2\lambda}{2}. \tag{19}$$

A figure of this function is given below. Note that $z_f$ takes its maximum when $\lambda = 0$ and is zero when $\lambda = 0.5$.

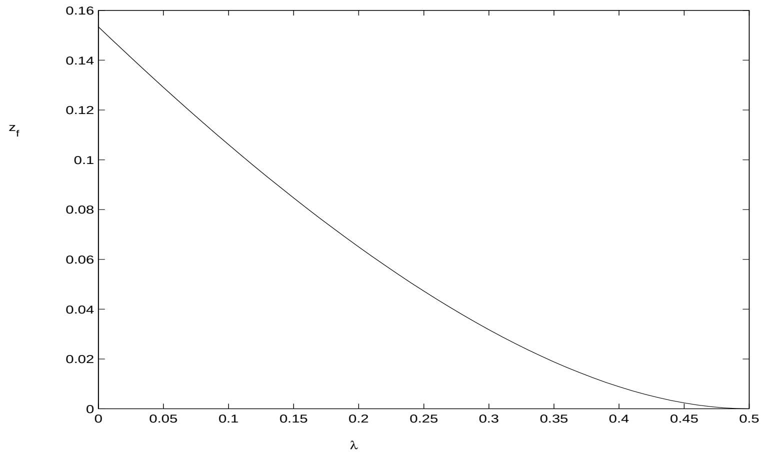


Figure 4: *The throw $z_f$ as a function of $\lambda$.*

In the original units the throw is thus

$$T = \frac{h}{2\alpha}\left(\ln\left(\frac{1}{2(1-\lambda)}\right) + \frac{1-2\lambda}{2}\right), \quad \lambda = \frac{b(0)}{h} \tag{20}$$

From this we see that the maximum throw over all values of $\lambda$ is

$$z_f(0) = \frac{1}{2}\ln\left(\frac{1}{2}\right) + \frac{1}{2} \sim .15,$$

with an entrainment factor of $\alpha \sim 0.1$ this means that

$$\max T \sim 1.5h. \tag{21}$$

regardless of the fan strength!
This is a rather surprising result and it would be of great interest to compare it with experimental measurements and/or CFD calculations.

## 3.2 Limitations of the model:

Whilst it leads to a simple conclusion, this model has certain limitations. We now detail these in the hope that a more refined calculation can improve the future model.

1. The model assumes that the flow is turbulent throughout the recirculation region.

2. The model will over-estimate the throw due to neglect of buoyancy. This is consistent with the CFD calculations made at FRPERC

3. The model assumes that all velocities are cross-sectional averages over the depth of layers. (With the aisles helping to impose a strong lengthwise direction on the flow). If

$$\lambda \neq 0$$

(i.e if the fan is a source of mass as well as momentum) then

$$Q(x = 0)$$

is non-zero, so that mass conservation requires flow in the lower layer through side wall. The implication of this is that the model will start to break down when

$$\lambda = \frac{1}{2}$$

i.e the flow can then no longer be modeled as a two-dimensional turbulent jet.

4. Finally, the analysis of the entraining jet can all be repeated using the revised entrainment model

$$\frac{d}{dx}(bw) = \alpha(w - v) \tag{22}$$

for which the entrainment is proportional to the velocity difference between the upper and lower stream. However given the experimental results reported by Barnett indicate that the equation

$$\frac{d}{dx}(bw) = \alpha w \tag{23}$$

is preferred.

### 3.3 A vortex approximation of the flow

To simplify all subsequent calculations, and in particular to obtain an estimate of the pressure variations with the primary recirculation region, the velocity field in the primary vortex can be approximated by using a simple vortex model satisfying the Euler equations. This takes the form

$$\psi = Q_0 \sin\left(\frac{\pi y}{h}\right) \sin\left(\frac{\pi x}{T}\right) \tag{24}$$

$$u = -\psi_y, \qquad v = \psi_x \tag{25}$$

where $T$ is the throw calculated previously.[3] This vortex field induces the same circulation rate as a fan with volume flux $Q_0$. The pressure field, $p$, assuming inviscid flow, can be reconstructed from the Euler equations,

$$uu_x + vu_y = -\frac{1}{\rho}p_x,$$

$$uv_x + vv_y = -\frac{1}{\rho}p_y - g.$$

Integration of these equations with the velocity field described above gives,

$$p - p_0 = \frac{\rho}{2}\left(\frac{\pi^2 Q_0^2}{h^2 T^2}\left(h^2 \cos^2\left(\frac{\pi y}{h}\right) + T^2 \cos^2\left(\frac{\pi x}{T}\right)\right) - 2gy\right). \tag{26}$$

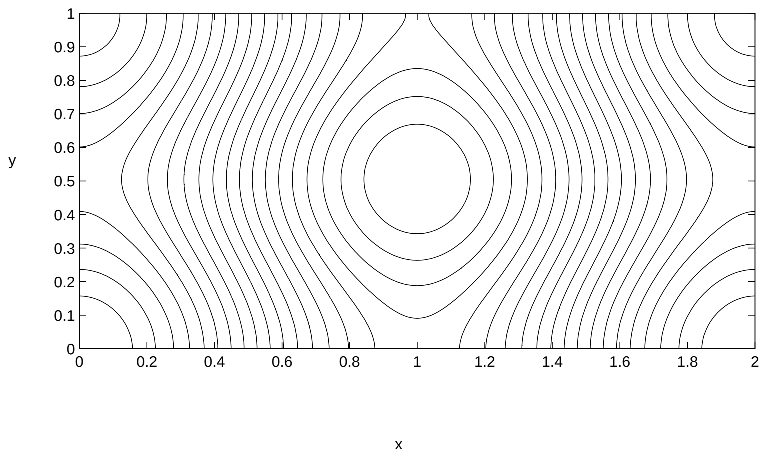


Figure 5: *The contours of the pressure in the simple vortex model.*

Typical contours for the pressure when $T = 2$ and $h = 1$ are given below:

The actual pressure field will of course also be affected by viscous effects which we have neglected here, and which will give considerable departures from the symmetry of Figure 5.

## 3.4 Heat transfer within the recirculating region

Assuming that the motion in this region is predominantly due to forced convection in which heat is advected by the flow but does not affect the flow, we can proceed to solve problem 2 of our list by solving the following (steady state) heat equation for the temperature profile,

$$u_x T_x + v_y T_y = k\left(T_{xx} + T_{yy}\right). \tag{27}$$

We have not obtained an analytical solution to this problem, however given the flow field approximation given by the vortex and a known temperature distribution at the fan and on the boundaries of the cool room it would be a relatively simple numerical exercise to solve the equation (27), well within the capabilities of a PC.

Alternately one could use equation (19) as a rule of thumb for maximum fan separation and solve the complete two dimensional Navier-Stokes equations coupled to the heat equation for the temperature and pressure profiles in the aisle before moving on to the blocking model for the shelves. This is a much more complex CFD exercise to that described above but much easier than a full Flow3D calculation.

As remarked earlier, a problem with this approach is that the heat does appear to affect the throw by changing the buoyancy of the turbulent jet.

[3]Note that $v$ now denotes the vertical component of velocity, not the return flow velocity. The sign convention for $\psi$ is not standard.

# 4 A blocking model for the flow and temperature in the shelves

## 4.1 Calculating the flow

Assuming from the calculations in Sections 3.4 and 3.5 that we know the temperature and pressure distributions in the aisle, we may solve the heat transfer problem in the shelves. We will assume that the shelving structure is sparse relative to the food and that each aisle is identical. We first divide the shelves up into blocks of uniform size; for instance horizontally along the shelves and vertically so as to make each block roughly a square. The manner this division is done is arbitrary with the blocks being as small or large as is computationally reasonable. The flow can then be simplified by considering it to be a simple mass flux in an out of each of the blocks determined by the geometry of the shelving, in particular the area between adjacent blocks (and without considering the detailed flow within the blocks or the vortices around the blocks).

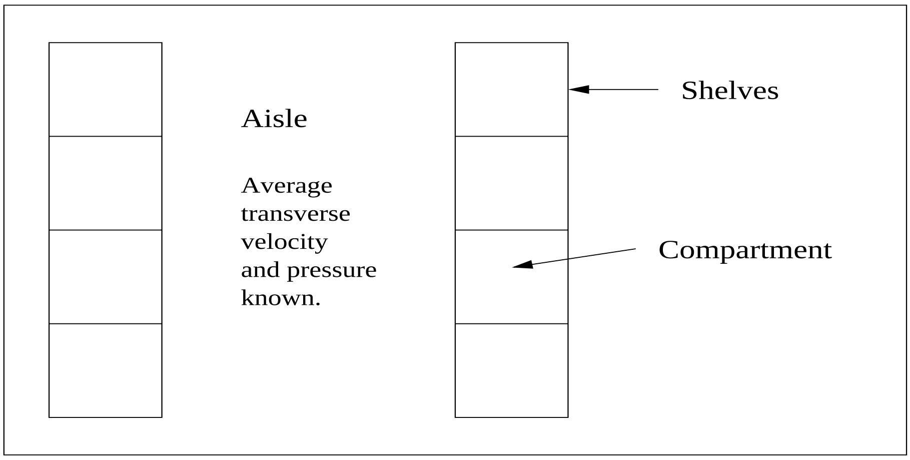


Figure 6: *Decoupled room geometry.*

Given the pressure averages over each block we assume that there is an energy loss between compartments due to the area change but that momentum is conserved. For a given compartment we need to know the area $A_i$ of each face blocked by the packages inside and the pressure $p_i$ acting on that face. All pressures *external* to the shelves are given by the vortex model above, and an objective of this section is to give a system of equations which can be solved to give the *internal* pressures and the mass fluxes
Consider first the case of the flow from a compartment of area $A_0$ into an obstructed one of area $A_1 < A_0$ with a pressure difference $p_0 - p_1$ between the two compartments and inflow and outflow velocities $v_0$ and $v_1$. Applying a momentum balance to this system (similarly to the derivation in Lighthill [2]) and ignoring buoyancy effects, we have

$$p_0 A_0 - p_1 A_1 - p_0(A_0 - A_1) = \rho A_1 v_1^2 - \rho A_0 v_0^2, \tag{28}$$

and using the mass conservation $A_0 v_0 = A_1 v_1$ gives

$$A_1(p_0 - p_1) = \rho A_1 v_1^2 (1 - A_1/A_0)$$

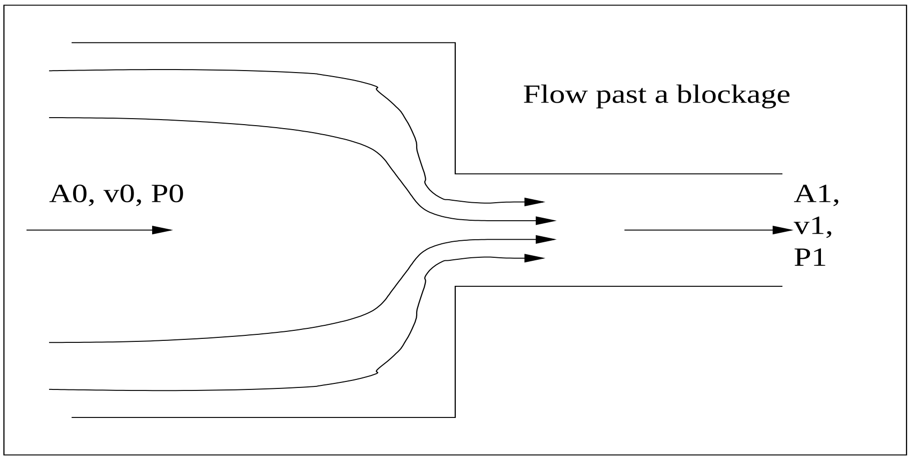


Figure 7: *Blocking model.*

hence,

$$v_1 = \sqrt{\frac{p_0 - p_1}{\rho(1 - A_1/A_0)}}. \tag{29}$$

There is some energy loss in the flow and we may suppose that $A_1/A_0$ is small. This leads to the following semi empirical formula

$$v_1 = c\sqrt{\frac{p_0 - p_1}{\rho}}, \tag{30}$$

where $c$ is a 'correction' factor typically taken to be $c \sim 0.5$. Observe that under this assumption the velocity $v_1$ is proportional to the square root of the pressure gradient (the Ergun condition) rather than the pressure gradient itself given by a Darcy law flow.[4]

Including buoyancy effects we can extend this calculation to give

$$v_1 = c\sqrt{\frac{p_0 - p_1}{\rho} + \frac{g(T_0 - T_1)\Delta y}{T_1}}. \tag{31}$$

Here $\Delta y$ is the height difference between compartments, and $T_0$, $T_1$ denote the temperatures in the compartments.

To close the system we use conservation of mass,

$$\sum Q_i = 0, \qquad Q_i = A_i v_i, \tag{32}$$

where the sum is taken over all compartments. This calculation gives a closed model for the fluxes and hence velocities in each compartment.

As an example, consider the simplified (non buoyant) case of three compartments closed at the top and ends, with external pressures $p_{0i}, \quad i = 1, 2, 3$ and compartment pressures $p_i$. Into each compartment from outside there is a mass flux $Q_{0i}$ and between each compartment there is a mass flux $Q_{ij}$.

[4]If $p_1 > p_0$ then the flow is in the opposite direction, and $v_1 = -c'\sqrt{(p_1 - p_0)/\rho}$ where the constant $c'$ may differ from $c$. This extension applies to all our equations like (30), but for simplicity we do not make the extension explicit.

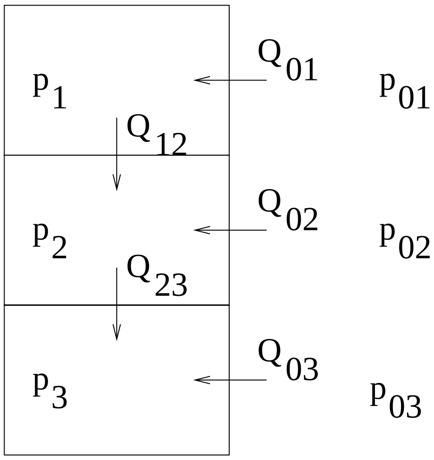


Figure 8: *Shelf model.*

Given the areas $A_{ij}$ between each compartment face and the average pressures in the aisle leads to the following system of 8 nonlinear equations for the 3 unknown pressures $p_i$ and the 5 mass fluxes $Q_{ij}$.

$$
\begin{aligned}
Q_{01} &= cA_{01}\sqrt{\frac{p_{01}-p_1}{\rho}} \\
Q_{12} &= cA_{12}\sqrt{\frac{p_1-p_2}{\rho}} \\
Q_{02} &= cA_{02}\sqrt{\frac{p_{02}-p_2}{\rho}} \\
Q_{23} &= cA_{23}\sqrt{\frac{p_2-p_3}{\rho}} \\
Q_{03} &= cA_{03}\sqrt{\frac{p_{03}-p_3}{\rho}} \\
Q_{12} &= Q_{01} \\
Q_{23} &= Q_{12}+Q_{02} \\
0 &= Q_{03}+Q_{23}.
\end{aligned}
$$

## 4.2 Calculating the temperature

Once the fluxes are known we can compute the average temperature in each compartment. Consider a compartment with two inflows and one outflow.
If the streams mix well then from the heat flux in we have

$$\rho c_p(Q_{12}T_1 + Q_{02}T_{02}) = \rho c_p(Q_{12}+Q_{02})T_2, \tag{33}$$

hence,

$$T_2 = \frac{Q_{12}T_1 + Q_{02}T_{02}}{Q_{12}+Q_{02}}. \tag{34}$$

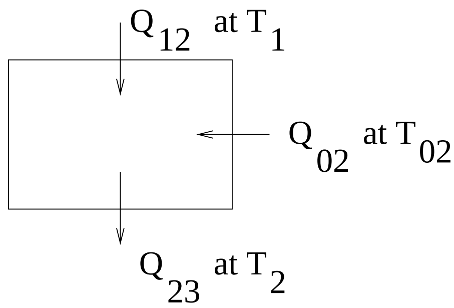


Figure 9: *Temperatures due to mixing.*

This procedure allows for efficient implementation as once all the fluxes are known the temperature in the first compartment (where all input fluxes are from the aisle) can be computed and then the temperatures in each subsequent compartment can be computed individually using known data.

## 4.3 Limitations of the blocking model

As with the turbulent jet model, we have made many assumptions with this model.

1. The model requires the decoupling of the shelves and aisles.
2. The correction factor $c$ is unknown.
3. The assumption linking the fluxes to the pressures is considerable and takes no account of complex flow (such as vortices) around the shelves.
4. It is unclear how easy it would be to solve the nonlinear equations for the fluxes easily and efficiently. Presumably it will be easiest to treat the *pressures* as the unknowns.
5. The model only works for a very specific shelf geometry comprising discrete blocks.

A possible way of overcoming the issues in item 5. above is to apply the blocking model over the whole region and to solve the (so called) Ergun equation. In this model for flow in an obstructed region the assumption (30) linking velocity to pressure gradient is applied over the whole region. So that if $p$ is the pressure and $\mathbf{u}$ the flow velocity then

$$-\nabla p = a^{-2}(\mathbf{x})\mathbf{u}|\mathbf{u}|,$$

where $a$ is a measure of the local blocking of the flow. Note that for the blocking model we can estimate $a$ in terms of the shelf areas. It will be much harder to do this for a general geometry. It follows that

$$|\mathbf{u}| = a|\nabla p|^{1/2}, \quad \text{so that} \quad -\nabla p = a^{-1}|\nabla p|^{1/2}\mathbf{u}.$$

Applying the continuity condition $\nabla.\mathbf{u} = 0$ leads to the Ergun equation

$$\nabla.\left(a\frac{\nabla p}{|\nabla p|^{1/2}}\right) = 0. \tag{35}$$

This is a continuous limit of the blocking model given in (33) This equation admits solutions which are recirculating flows. We could (as a future investigation) study the flows determined by solving (35) with boundary conditions given by the turbulent jet flow considered earlier.

### 4.4 Thermal exchanges between the air in the shelves and the food

The above model leads to an averaged temperature distribution in each of the compartments containing the food. This is *precisely* the problem considered in the CoolVan simulations. Thus to work out the heat transfer into the food itself the algorithm in CoolVan can be used as it stands.

## 5 Numerical issues

There are several ways in which the existing algorithms as implemented in CoolVan (and maybe CoolRoom) can be made to run significantly faster.

1. Larger time steps in solving for the heat transfer problem can be taken if implicit methods are used. As an example of such a method, suppose that we wish to solve the heat equation
$$T_t = T_{xx}.$$
We can do this by discretising over a a uniform mesh in time and space with spatial mesh spacing $\Delta x$ and temporal mesh spacing $\Delta t$. In particular we take $T_j^n$ as an approximation to $T(n\Delta t, j\Delta x)$. Then an implicit theta-method to calculate $T_j^{n+1}$ in terms of $T_j^n$ is given by
$$\frac{T_j^{n+1} - T_j^n}{\Delta t} = \theta \left( \frac{T_{j+1}^{n+1} - 2T_j^{n+1} + T_{j-1}^{n+1}}{\Delta x^2} \right) + (1-\theta) \left( \frac{T_{j+1}^n - 2T_j^n + T_{j-1}^n}{\Delta x^2} \right).$$
This method is *unconditionally stable* (so that an arbitrarily large value of $\Delta t$ can be used) if $\theta \geq 0.5$. It is implicit if $\theta > 0$ and requires the solution of a large set of linear equations. The method is most accurate when $\theta = 0.5$, however in practice you might wish to take $\theta$ slightly greater than 0.5 to avoid stability problems. Similar methods can be used in higher dimensions. These methods can be readily adapted to time dependent heat transfer with advection. A good reference is the text by Morton and Mayers [3].

2. For problems in which there is a nonlinear term (such as heat transfer with a reaction) for example of the form
$$T_t = T_{xx} + f(T)$$
then the theta-method can be modified to give the (so called) implicit-explicit method of the form
$$\frac{T_j^{n+1} - T_j^n}{\Delta t} = \theta \left( \frac{T_{j+1}^{n+1} - 2T_j^{n+1} + T_{j-1}^{n+1}}{\Delta x^2} \right) + (1-\theta) \left( \frac{T_{j+1}^n - 2T_j^n + T_{j-1}^n}{\Delta x^2} \right) + f(T_j^n).$$

This method retains much of the advantages in stability of the theta-method but does not involve solving any nonlinear equations.

3. The theta-method and most discretisations lead to large linear systems, the solution of which is expensive. These systems are nearly always sparse and it is essential that full advantage of sparsity is taken so that fast algorithms can be used. An account of sparsity and how it can be exploited is given in 'Numerical Recipes' [4]. Note also that packages such as MATLAB exploit sparsity directly. For the future we would strongly recommend that the (sparse) linear systems should be solved by using an iterative method such as conjugate gradient and its derivatives such as `gmres`. These algorithms can be found in public domain software .. see the guide to available mathematical software `http://gams.nist.gov`.

# 6 Conclusions and recommendations for future consideration

The main outcomes of this brief study have been to determine an exact formula for the throw of a fan given the room geometry and a direction to pursue a decoupled porous shelf model. The validity of the 2D approximation can be easily checked with a few CFD computations - in particular does the throw really depend most heavily on the fractional height of the fan? If so, then the parameter $\alpha$ can be easily determined and a rule of thumb for fan spacing firmly established. Unfortunately the meeting was too brief to explore the validity of the decoupled porous shelf model in detail. If the blocking model is reasonable in this regime, it could be coupled to the fitted vortex flow profile or to a full numerical solution of the planar problem.
Having determined the behaviour of the turbulent jet, the future calculations that need to be done to complete this model at this stage are

1. To solve for the heat transfer and resulting temperature profiles in the air given by (27)

2. To solve for the fluxes and pressures in the shelves by solving first (33) and then (33).

3. To use the CoolVan algorithm to find the resulting temperatures in the food.

As remarked earlier, it is likely that task one above does not fully decouple from the problem of finding the jet, due to the effects of buoyancy, and this link needs to be explored further.

# References

[1] S.J. Barnett, *The dynamics of buoyant releases in confined spaces,* PhD thesis, DAMTP, University of Cambridge, 1991 or 1999.

[2] J. Lighthill, *An informal introduction to theoretical fluid mechanics*, Oxford, 1986.

[3] K.W. Morton and D.F. Mayers, *Numerical solution of partial differential equations: an introduction*, Cambridge University Press, 1994

[4] W. H. Press et. al. Numerical recipes in Fortran 77 : the art of scientific computation, 2nd. Edition, CUP, (1996) (see also Numerical Recipes in C).

[5] J.S. Turner, *Buoyancy Effects in Fluids*, CUP, 1973.

[6] Morton, Taylor and Turner. Proc Roy Soc A 234 pp1-23 (1956)