Energy Impact of Ventilation and Air Infiltration
14th AIVC Conference, Copenhagen, Denmark
21-23 September 1993

# Natural Ventilation via Courtyards: The Application of CFD

L Shao*, R R Walker**, M Woolliscroft**

** Building Research Establishment, Garston, Watford, WD2 7JR
* University of Sheffield, School of Architectural Studies, Sheffield, S10 2UJ, UK (Now at Nottingham University)

# Natural Ventilation via Courtyards: Part II - the application of CFD

by L Shao, R R Walker and M Woolliscroft.

**SYNOPSIS** Computational fluid Dynamics (CFD) is a powerful tool for analysing problems of air movement and has been increasingly widely used in applications in buildings. The emphasis has often been on its development as a replacement for the experimental approach, however, further work is needed to develop confidence in applying CFD to problems of air flow in buildings. It is suggested that CFD can be effectively used in an integrated manner together with experimental methods and that the interaction and integration of the two should be promoted in a variety of forms, and on a variety of levels within a research project. In this way the effectiveness of both the CFD and the experimental methods are enhanced. This approach has been applied to a study of the effectiveness of natural ventilation via courtyards, as part of a wider programme of work carried out at the Building Research Establishment. The CFD technique has been used to help formulate experiments, to assist in the interpretation of test results, and to perform parametric studies as well as exploring theoretical ideas. One of the important outcomes of the application of the CFD method has ben the formulation and verification of criterion for ensuring good courtyard ventilation.

## 1 Introduction

The related paper[1], Part I, describes how current understanding of natural ventilation in building courtyards was incomplete, with measurements at model scale in some cases contradicting observations at full scale. As a result of advances in computing power, CFD is now capable of modelling these air flows at sufficient resolution. This paper describes the application of CFD to this problem, as part of a wider programme of work which also involves site tests and wind tunnel experiments and described in Part I.

It is generally accepted amongst researchers that, for application to problems of air flow in buildings, CFD is not yet fully mature and results could be misleading if it is improperly used. On the other hand it is a promising technique and should not be ignored. The authors believe that by integrating the CFD approach with experimental methods and by promoting the interaction and interweaving of the two in a variety of forms and at different levels, the effectiveness of both the CFD and the experimental approach can be optimised.

This integrated approach for CFD application was adopted in this investigation which also utilised site and wind tunnel tests. The CFD technique normally assumed a supporting role. For example, it was used to estimate the air flow rates and patterns so experiments (e.g. tracer decay tests) could be better planned and set-up. Where test results were not as expected, the CFD program was run with the detailed description of the actual experimental conditions to help in judging whether the unexpected results were due to equipment/operator errors or were actual phenomena. In the latter case, the computed highly detailed flow field sometimes even allowed the causes of the unexpected results to be traced. In particular, CFD was used to explore new ideas on the causes of good or poor ventilation in courtyards. In addition, it proved effective in parametric studies of the effects of many variables including building features, building dimensions and wind characteristics, within a relatively short period of time. In these cases, CFD assisted in focusing on appropriate site and wind-tunnel tests and, subsequently, as a means for verification and validation.

This report describes the detailed modelling of the salient cases and how the results were used to provide information on the mechanism of courtyard ventilation. Additional cases, including the parametric studies, are presented in the related paper (Part I).

## 2. Description of the Computation

The computations were performed using the commercial flow simulation software FLUENT[2], which solves the Navier-stokes equations in their three dimensional form. The three dimensional version was chosen in preference to the two dimensional one, despite the penalty in computing time associated with it. Adopting the 2D version would have meant assuming that the building used in the computation is infinitely long. Consequently the air flow, in the form of wind, would have only one route around the building - over the top - instead of three, the other two being around the flanks (Fig. 1). The velocity of the air flowing up the windward facade of the building would be artificially forced higher. As will be demonstrated later, the existence of a full vortex in a courtyard depends sensitively on this velocity. Therefore adopting the 2D version would be un-realistic and could lead to fundamental errors.

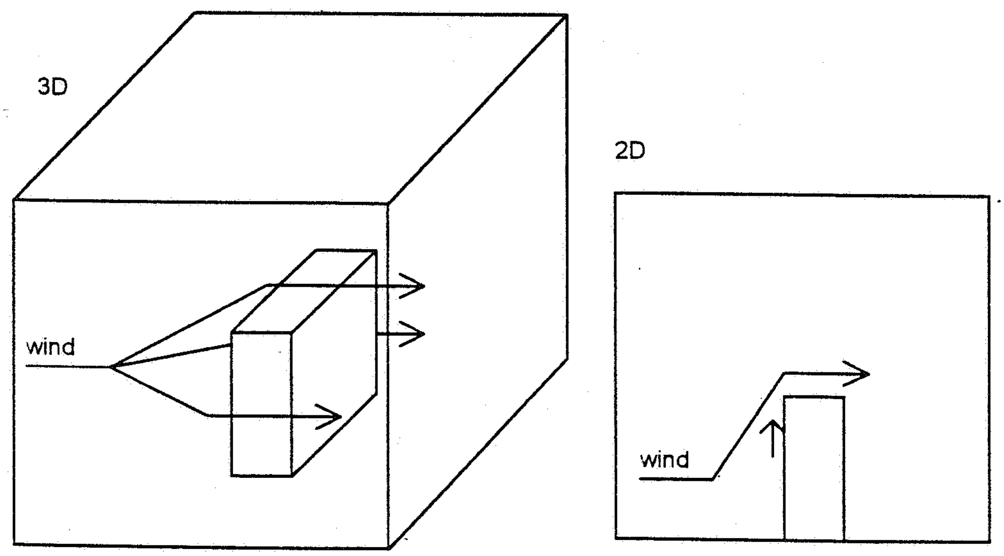



Fig. 1 Schematic of three- and two-dimensional computation domain.

### 2.1 Turbulence model

Flows around buildings are usually turbulent, which FLUENT predicts by solving the averaged Navier-Stokes equations, which additional Reynolds Stress terms. These are unknown quantities and were computed using the standard k-ε model, which links these terms, via two equations, to other flow parameters. This model is considered adequate for calculating the relatively simple flow in the study, and the model has been widely used in CFD applications in the area of air infiltration and ventilation.

## 2.2 Computation domain and boundary conditions

Calculations were carried out for a large number of cases with different computation domains. The most common is shown schematically in figure 2. The building affects the air flow only in its close surroundings, and it is this flow which affects ventilation of the building and any courtyard. Therefore only the space close to the building was included into the computation domain. However, this caused the boundary of the calculation domain to be too close to the building. Since these boundaries were treated as walls in the code, unrealistic extra shear stress was added into the flow surrounding the building, causing modelling inaccuracy. This problem was solved by using "slip wall" boundary conditions at the side and top boundaries of the calculation domain. In most cases the building model was symmetrical. This fact was utilised to reduce the amount of computer memory and calculation required by including only one half of the building in the computation domain. As a consequence, the central symmetrical plane became one of the boundaries and the symmetrical boundary condition had to be imposed, which assumes that the velocity component perpendicular to the plain is zero and all scalar gradients are zero. Other domains used in the computations are described where appropriate.

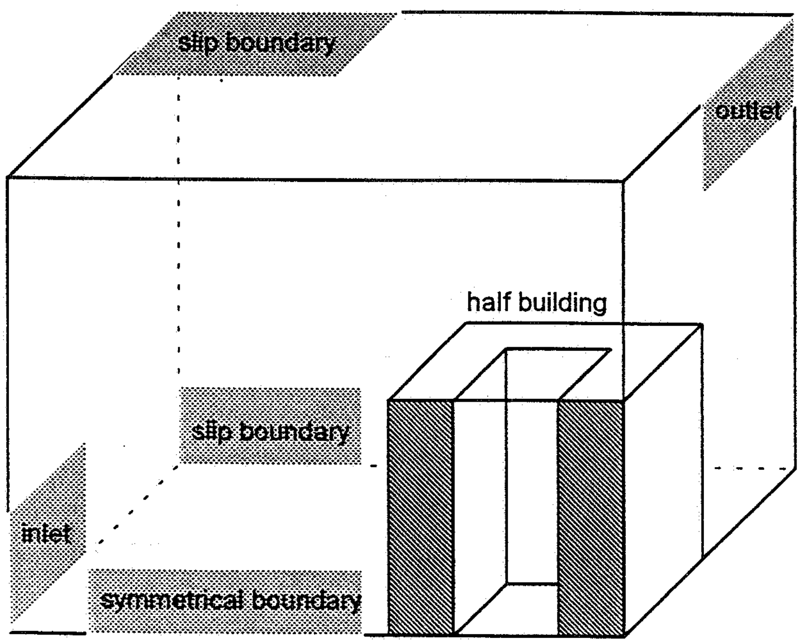



Fig. 2 A computation domain

The inlet boundary condition was defined by its velocity profile and turbulence intensity. The latter was set at 10% while the former was assigned different patterns including a uniform profile, a urban wind profile and a city wind profile. The urban wind profile typically assumes the wind velocity (u) at a point in the atmospheric boundary layer to be:

$$u \propto y^{0.28}$$

where y is the vertical distance between the point and the ground. The city wind profile, as specified in the British Standard BS-5925, assumes that

$$u \propto y^{0.33}$$

It was also assumed, for this study, that the whole computations domain was submerged in the atmospheric boundary layer.

## 2.3 Numerical scheme

The Navier-Stokes equations were discretised into finite difference equations based on a 3-D Cartesian grid. Two considerations have to be made in deciding the grid density.On the one hand the grid has to be dense enough so that gradients of flow field parameters can be accurately presented in the finite difference equations and the calculated flow is accurate and detailed. On the other hand, the grid has to be loose enough so that the number of cells is within the range that the memory capacity of a given computer can cope with and the solution converges at reasonable speed. To satisfy the two opposing requirements, denser grids were used near the walls and corners where the gradients were likely to be large and coarser grids away from walls and corners to minimise the total number of cells. The transition from the denser grids to the coarser grids was effected by means of gradual cell expansion. To promote numerical stability, the ratio of expansion from one cell to the next was limited to below 1.2 or 1.4 depending on the likely complexity of the local flow. The number of nodes for the cases computed varied from under 20,000 to over 148,000.

The finite difference equation resulting from the discretization process described above were solved using the SIMPLE[3] algorithm. The solutions convergence was accelerated by increasing the under-relaxation coefficients, up to 1, after a certain number of iterations, during which the solution had shown steady convergence.

## 3. Results and Discussion

The cases computed can be classified into three groups. Cases in Group one were computed to examine a proposed criterion for good or poor courtyard ventilation. The following discussion will be focused on these cases. The second group was used for parametric studies, i.e. examining the effects on full vortex existence of building dimensions, building features, wind strength and wind directions, etc. The third group involves computing the courtyard flow patterns for a real building. The latter two group of cases are discussed in Part I.

There are two air flow patterns concerning the natural ventilation of courtyards. One of the patterns, referred to in the following as a top vortex, involves an air flow vortex suspended at the opening of the courtyard in the building roof. The second pattern, referred to as a full vortex, is associated with a vortex occupying the full depth of the courtyard. These patterns are illustrated schematically in figure 3. A top vortex isolates the courtyard from the outside environment, resulting in poor ventilation and air quality and therefore is undesirable. A full vortex, on the other hand, ensures good ventilation by bringing fresh air into the courtyard and carrying away the stale air. Obviously, the question of "Under what conditions will the courtyard ventilation be satisfactory?" can now be transformed into a fluid dynamics question, i.e.: "Under what conditions will there be a full vortex?"

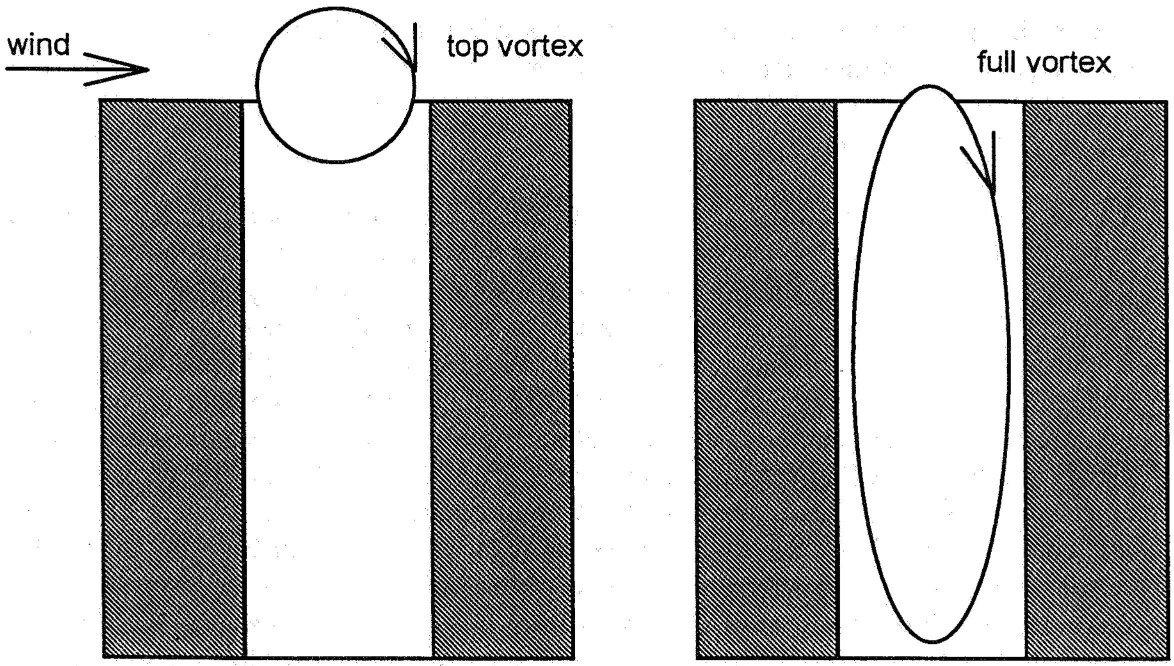



Fig. 3 Schematic of the two types of vortices

Refer to Fig 4a which shows air flow in the form of wind passing by a rectangular building. It is known that the part of the wind flowing over the top of the building will without exception, separate from the roof, resulting in a region of recirculating air - a vortex - below it. The separated flow, provided that the building is long enough, will eventually reattach to the roof. If there is a courtyard in the building and the point of reattachments is either ‘within’ or nearby downstream of the courtyard opening in the roof, as shown in Fig. 4b, the vortex (Fig. 4a) will "drop" into the courtyard and form a full vortex. In the situation that the courtyard building is not long enough and the reattachment does not happen, as shown in Fig. 4c, the vortex will remain largely on the roof, spanning the courtyard, although part of it does sink slightly into the opening. A further case is where reattachment occurs upstream of the courtyard opening, in which case a full vortex is caused by the flow shear stress.

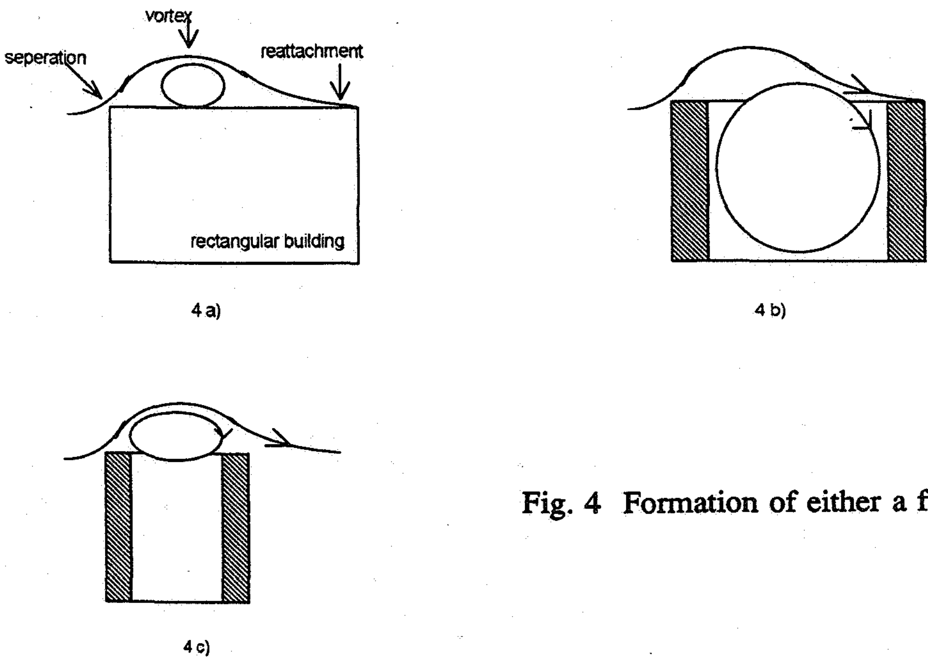



Fig. 4 Formation of either a full or top vortex

Based on the above discussion, it is proposed that a full vortex will form if reattachment occurs either upstream, within, or nearby downstream of the courtyard opening. In the cases examined, separation seems to occur when the width of the building downstream of the courtyard is less than about half the width of the courtyard. This is not a general criterion because the width downstream at which reattachment occurs is likely to be affected by several factors, wind velocity, height of building etc. and further work is necessary to attempt to produce a general criterion. It may be that a separate CFD calculation is necessary in each case. To examine the possibility for a specific criterion, six cases were computed:

- Case 1 (Fig. 5a) involved building without a courtyard, measuring 16m X 16m X 24m and with its windward facade perpendicular to the wind direction;

- Case 2 (Fig. 5b) differed from Case 1 in that the building was longer (24m instead of 16m) and

- Case 3 (Fig. 5c) was identical to Case 1 except that the building was partially sheltered and that the velocity at domain inlet was 0.5m/s up to half of the building height as shown by the first column of arrows in Fig. 6c.

The results from the cases are presented in graphic form in Fig. 5a, b, c, which show the velocity fields in the corresponding central symmetrical cross sections (also termed symmetrical boundary, as defined in Fig. 2) The arrow length and direction indicate air flow velocity and direction, respectively. Roof top velocity can be identified by the velocity arrows with opposite directions along the neighbouring horizontal grid lines close to the roofs. In all three cases, a vortex existed on top of the roof, close to its leading edge. The length of the vortex is marked by the length over which the arrows along the first horizontal grid line next to the roof remain in the direction opposite to that of the wind. Obviously, the vortex for case 1 covers the whole length of the building and probably beyond and, in consequence, reattachment does not occur. For case 2, the vortex covers only part of the roof, due to the extra length of the building. Reattachment occurs and is characterised by the reversal of arrows along the rear part of the above described grid line to the wind direction. The point of reattachment is marked by the first reversed arrow. Reattachment also occurs in Case 3, but for a different reason. In this case the shelter reduces the air flow velocity up the building windward facade, principally by reducing the total flow of air over the building. This reduces the angle between the wind and the roof where flow separates at the leading edge, and the severity of separation is less, thereby causing earlier reattachment.

Suppose that a courtyard is built into each of the buildings at the location indicated by the dotted lines in Fig. 5 a, b, c; according to the above criterion, reattachment should occur in cases 2 and 3 and cause a full vortex to form. However, in case 1, a top vortex should occur. These situations were modelled as three further cases, 4, 5 and 6, and the results are shown in figure 6. The interpretation is similar to that for Fig 5. As expected, figures 6b and Fig. 6c show a full vortex in the courtyard, with air movement reaching all places while, in contrast, the courtyard shown in Fig. 6a is dominated by a top vortex with a region of stagnant air below it.

To sum up, the above discussions show that good courtyard ventilation depends on the existence of the full vortex and that the latter is formed when the separated flow over the building roof top reattaches on the roof and the point of the reattachment is either upstream, within, or nearby downstream of the courtyard top opening. As was mentioned earlier in the cases examined this distance is about half the width of the courtyard downstream of the courtyard but further work is necessary to determine a general criterion which is likely to involve many factors. This qualitative understanding as illustrated in Part I, forms the basis for explaining many of the courtyard air flow phenomena. It should be said that the computation of the above 6 cases was only one step in developing a criterion. In fact the results from many other cases computed have been used to develop the understanding which has also been supported by experimental evidence obtained from site and wind tunnel tests.

## 4. Concluding Remarks

More than eighty cases have been computed in the course of this research project. The setting-up details of the cases have been presented. The results from a group of six cases have been analysed to begin to develop a criterion on terms of the air flow conditions under which the good courtyard ventilation is ensured. The discussion also serves to illustrate the way the detailed information afforded by the CFD method was used to explore theoretical ideas. The results from other cases are presented in Part I of this twin paper, as they are best shown in an integrated manner together with site and wind tunnel test data.

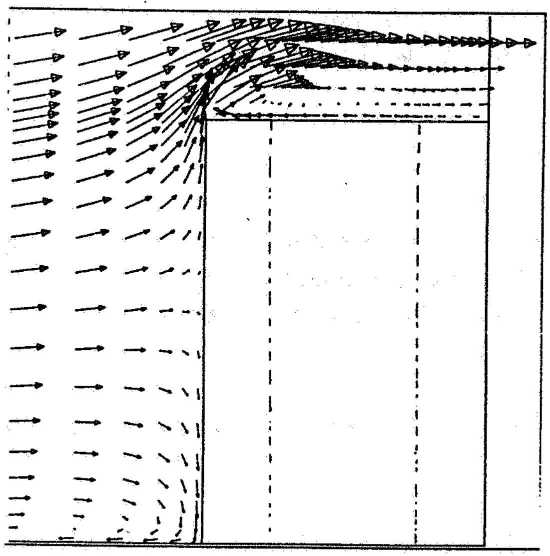
Great advances have been made in the CFD method. Yet it is not completely mature and results not always reliable. However, by using it together with the experimental method in a way that allows the interaction and integration of the two, the effectiveness of both methods as investigate tools are enhanced. In this research project, the CFD technique has been used to predict the likely flow conditions for a site test on a real courtyard so that experimental equipments can be better tuned, the results more accurate and obtained more speedily. It has also been used to assist the understanding of certain experimental results, as well as to perform parametric studies and to explore theoretical ideas.

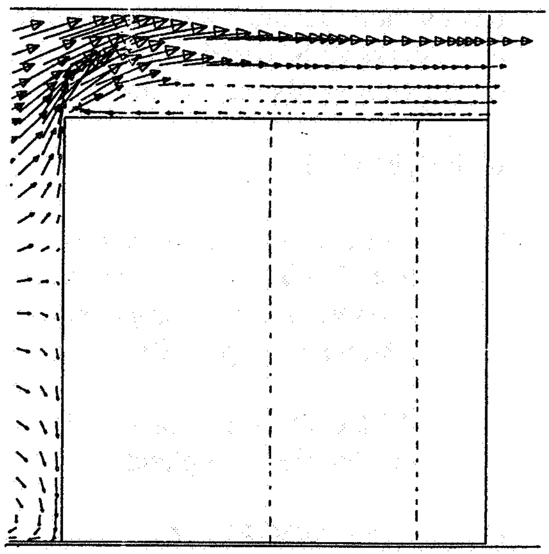
## REFERENCES


(1) Walker, R R, Shao, L and Woolliscroft, M, Natural ventilation via courtyards: part II - Theory and measurements, paper presented at the 14th AIVC Conference on Energy Impact of Ventilation and Air Infiltration, Copenhagen, Denmark, Sept 1993.

(2) FLUENT is a commercial CFD package available from Flow Simulation Ltd. of Sheffield, England.

(3) VAN DOORMAAL J.P. and RAITHBY G.D. "Enhancement of the SIMPLE method for predicting incompressible fluid flows"
Numerical Heat Transfer Vol 7 p147 1984


(a)


(b)

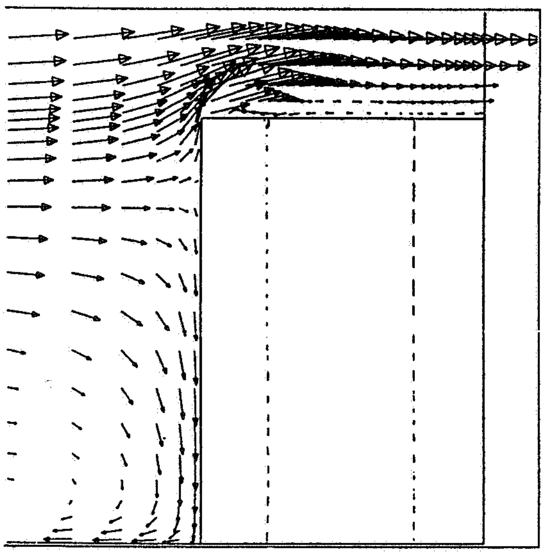

(c)

Fig. 5 Flow over building with proposed position of courtyard opening located at increasing distance from upwind edge of roof

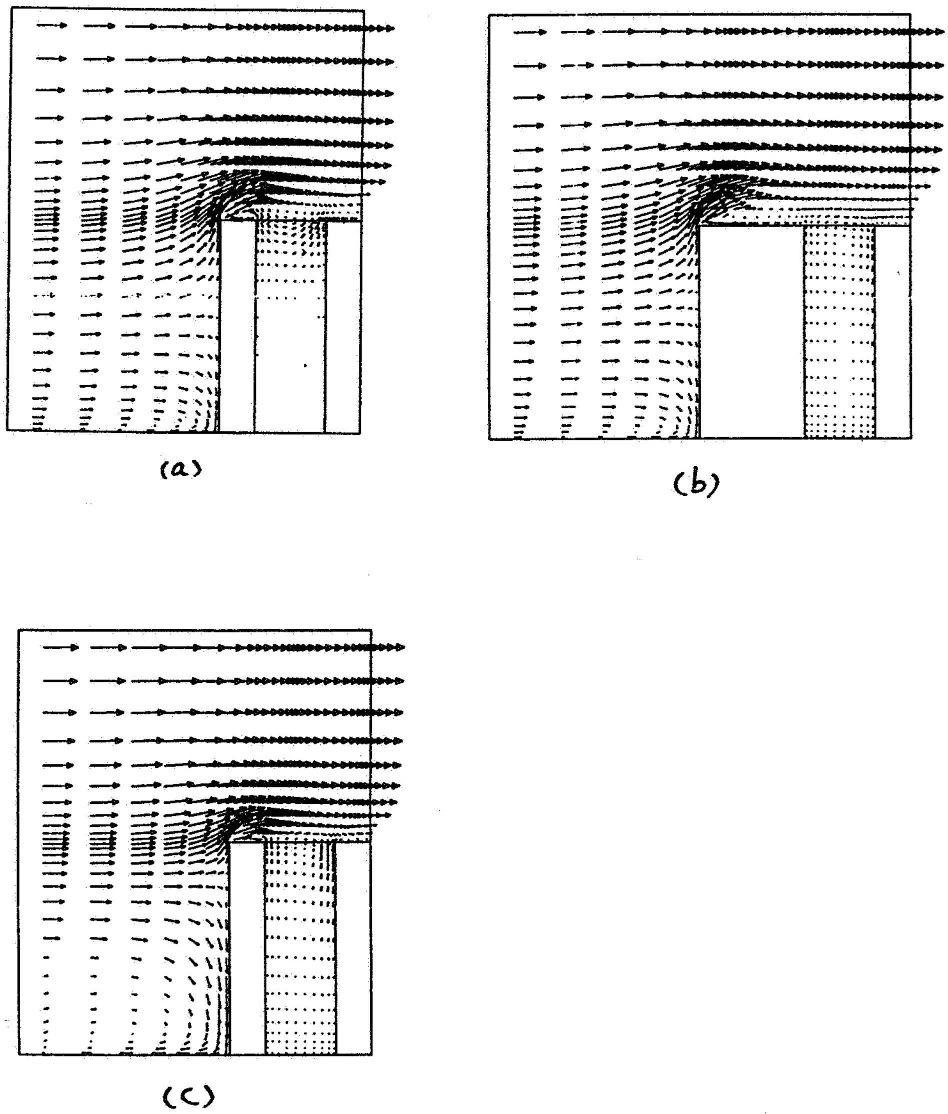



Fig. 6 Predicted flow into courtyard corresponding to Fig. 5; (a) top vortex, (b) and (c) full vortex formed.