Numerical Simulation of Air Flows - an Essential Tool of Comfort Optimization of Modern Buildings and HVAC Systems

Blazej Ochocinski
ROM Zentralbereich Ingenieurtechnik
Tilsiter Straße 162, 22111 Hamburg,
Germany

ABSTRACT

The paper deals with the CFD (computational fluid dynamics) application to the comfort optimization of some complex architectural projects in which the physical interaction effects between the building and its environment affect strongly the building's comfort conditions and the HVAC system behaviour. The author reports on his practical experience of modelling aerodynamic and thermodynamic interaction effects between internal and external air flows. The selected examples range from the investigation of the pollutant transport around several skyscrapers in the city of Frankfurt to the simulation of internal air flows in very complex large spaces like modern atria or shopping malls.

Keywords: Computational Fluid Dynamics (CFD), building technology, practical design, air flow, heat transfer, pollutant transport, HVAC systems, simulation

INTRODUCTION

Accurate predictions of air flows in and around buildings are necessary for the comfort optimization of modern buildings and heating, ventilating and air conditioning (HVAC) systems. A lot of research papers on air flow simulation methods were published in the last years (representative lists of references may be found e.g. in [1] to [12]). The majority of them deals mainly with air flows in single enclosures with well-defined boundary conditions. In contrast, the main objective of the present paper is to discuss some practical aspects of the numerical modelling of air flows in extremely complicated architectural configurations, each of which being exposed to different environmental conditions.

CFD-AIDED INTEGRATED APPROACH

The integrated planning of buildings means generally a simultaneous design work which is performed by architects, civil engineers and engineers specialized in HVAC and other building systems. The main objectives of the integrated design are to plan, to build and to operate an energetically optimized building while respecting all required comfort criteria.

For several years, more or less complex building simulation models have been developed throughout the world for reliable assessment of the building and its HVAC system. The integrated design tool which was applied by the author in his present investigations includes an appropriate CAD system for the interactive building and HVAC system design as well as a couple of other personal computer codes, developed e.g. for the dynamic simulation of the building's thermal behaviour or for the simulation and energy evaluation of alternative HVAC systems. As well, the numerical air flow simulation, which is usually performed quasi separately on high performance computer systems by means of very complex, general purpose computational fluid dynamics (CFD) codes, represents a very important essential part of the whole integrated approach. More detailed description of the whole system can be found in the reference [4].

The main objective of the numerical air flow simulation by means of CFD methods is to assist all the cooperating groups of architects and engineers with the comfort optimization of buildings and HVAC systems with regard to required air velocities, air temperatures, allowed turbulence intensities, air exchange rates, pollutant concentrations etc. These flow variables, in turn, are generally subject to both steady (time independent) and transient (time dependent) fluid transport phenomena.

Modern computers and advanced numerical investigation methods enable coupled and/or decoupled solution of the following classes of the building aerodynamics' problems:

Global simulation of the external air flow field for the whole building of interest as well as for a relevant part of the surrounding area (to a scale of 1:1).

Simplified computation of the global infiltration effects in the buildings of interest (zonal models).

Detailed simulation of air flows and comfort conditions in enclosures which are exposed directly to the action of the external air flow field.

Detailed simulation of air flows and comfort conditions in enclosures which do not experience directly the action of the external air flow field.

The necessity of the coupled numerical treatment of the sub-topics mentioned above depends on the intensity of the aerodynamic and thermodynamic interaction effects between the buildings or single enclosures and their outdoor and/or indoor environment.

SIMULATION METHODOLOGY AND TOOLS

All the simulations and results' visualizations presented in the following have been performed by means of the general purpose commercial CFD software CFX 4.1 (for details refer to [13,14]) and of the advanced CFD post-processor FIELDVIEW [15]. However, any other CFD software of a similar complexity and flexibility could be used alternatively for simulating the same problems.

Following physical models have been utilized for setting up the air flow simulation runs:

quasi steady or transient Navier-Stokes equations for modelling the transport of momentum,

SIMPLEC correction for pressure,

energy equation with enthalpy as the primitive variable,

k-ε turbulence models based on different mathematical formulations (standard, low Re and RNG models),

full or weak compressibility option for modelling the air density as a function of both pressure and temperature,

mass fraction equations for modelling the pollutant transport within the flow area.

Most of the results discussed in this paper represent quasi steady state asymptotic solutions which were obtained by means of the real time or pseudo time step-by-step integration of the flow equations (refer to [14] for more details on the theoretical and numerical background of the applied simulation software). However, the purely transient behaviour of the coupled building-environment system was also analysed very carefully in order to estimate the effect of time-dependent changes in the flow behaviour upon the building's comfort conditions (e.g. room temperature drop after opening windows on a cold and windy day in winter or external pollutant transient spreading along the building's facades).

For the 3-dimensional spatial descretization of the flow problems under consideration, blockwise structured body-fitted computational grids with up to 650000 control volumes had to be generated for the coupled internal/external computations as well as for the global simulations of the external air flow fields. The simulations of indoor air flows were performed by means of computational grids with up to 350000 cells. Depending on the flow behaviour, the time increment for transient or pseudo transient integration of the full system of the flow equations was varied between 0.2 and 1.0 s.

Provided that all required physical and numerical criteria has been formally satisfied by the generated simulation model, the proper set up of initial and boundary conditions becomes crucial both to the solution's numerical convergence and to the physical reliability of the simulation results. In the examples presented in the following, results of the simulation of the building's thermal behaviour have been utilized for the definition of realistic thermal boundary and initial conditions of the flow simulations. Realistic inlet and outlet conditions have been defined by making use of measured data from wind tunnel experiments or tests carried out in an air conditioning laboratory. As well, a multi-level analysis method with successive refinements of computational grids and flow domain subdivisions has been applied in order to accelerate the convergence behaviour and to increase the solutions's accuracy.

The internal air flow problems with up to 350000 grid cells could be simulated successfully by means of single processor workstations with up to 512 MByte main memory capacity. The run times were strongly dependent not only on the absolute number of control volumes itselves but also on the number of grid blocks into which a given grid structure was subdivided. The run times range between two CPU days and one CPU week for a single computational case (one set of boundary conditions) and depend strongly both on the kind of simulation (transient or steady) and on the convergence behaviour. The external or coupled internal/external air flow computations with a greater number of control volumes and flow variables required using single processor workstations with up to 1 GByte main memory. Significant spead-ups could be observed on parallel computers while performing parallel simulation runs on four to eight processors. However, as the latter machines had to be shared by the author with other users, no reliable speed-up factors can be presented in this paper.

PRACTICAL EXAMPLES

In this section, different practical aspects of the numerical approach to the air flow investigation are discussed briefly in some detail with respect to four selected CFD-aided design cases. Note that none of the large objects of interest could be investigated in such detail by making use of the traditional model test experiments. This is due to too strong model scale effects which would affect significantly the quantitative insight into the real air flow details.

However, dimensionless-made experimental results may be used sometimes as a rough reference for the evaluation of the global reliability of the simulation results as well as for the definition of flow boundary conditions in very complicated geometrical system configurations, e.g. for the estimation of the deformation of undisturbed wind velocity profiles by the earth surface roughness and buildings. In the latter case, full-scale test data can hardly be found for a quite specific urban scenery.

A new skyscraper in Frankfurt on the Main

Fig. 1 shows a geometrically simplified simulation model of a new skyscraper (the highest building of the group) which is to be built in the downtown of Frankfurt on the Main. The tower of interest is surrounded by other skyscrapers and several (up to 8 storeys high) residential and office buildings. Seven exhaust pipes of a cogeneration power station are to be situated near the tower's base at a height of only 30 m above the street level instead of on the tower's top. In this way, the total investment costs could be significantly reduced, provided that the exhaust fume concentrations do not exceed allowable threshold values in the vicinity of the neighbouring facades.

Fig. 1 demonstrates also the relative exhaust fume concentration distribution on a critical summer day with no action of wind. The flow area, which is affected by the contaminant transport (Fig. 2), follows the main direction of the air motion. In summer, the latter is primarily induced by the action of thermal buoyancy. The relative pollutant concentrations in the vicinity of the affected facades do not exceed 15 % of the emission concentration at the outlets of the exhaust pipes. As the absolute concentrations at the outlets just fulfil the German law regulations, the mixing of the exhaust fumes with the surrounding air contributes significantly to the further reduction of the pollutant concentrations far below threshold values required by law.

The exhaust fume concentrations in the air decrease rapidly as soon as the wind starts to blow the exhaust fume cloud away from the critical area. The wind velocity boundary conditions on the windward and leeward sides of the flow domain, Figs. 3 and 4, have been derived semi empirically by making use of both meteorological data and dimensionless wind velocity profiles determined from the wind tunnel experiments. The latter have been measured in reference planes which coincide with the outer boundaries of the simulated flow domain. As the mathematical functions, that have been derived to describe the boundary conditions mathematically within the simulation model set up, do take into account the wind profile deformations due to the presence of the surrounding buildings, there was no need for modelling directly wide upstream and downstream flow areas around the building complex of interest. Instead, more cells could be spent on the finer discretization of the flow region which might be affected by the exhaust fumes. The rapid concentration decrease and the global pollutant spreading due to the action of wind are visualized in Figs. 5 and 6, respectively.

In general, the air flow simulation has shown that it is not necessary to install the cogeneration power station's exhaust pipes on the tower's top in order to avoid too high pollutant concentration at the neighbouring buildings' facades. Thus it was possible to save appr. 1.6 millions U.S. $ of the total investment costs.

The simulated pressure and velocity fields in the vicinity of the tower of interest were used further as input data for the analysis of the global infiltration effects due to wind pressure and shaft effect as well as for the 3-dimensional simulation of the natural ventilation of typical offices in the round tower.

A new business quarter in the city of Stuttgart

An existing complex of several older office buildings and courtyards in the heart of Stuttgart (Fig. 7) is being converted into a modern business quarter and shopping district. One of the three courtyards is to be covered with a glazed roof the sides of which, however, are to be fully open towards the external air flow field (the atrium's simulation model is shown Fig. 10). The air flow in the occupied zones of the two open courtyards as well as the comfort conditions in the glazed atrium have been analysed by means of the numerical air flow simulation.

Similarly as in the case of the skyscraper in Frankfurt, the business quarter is situated in the middle of an urban district (simulation model - see Fig. 8). Many surrounding buildings affect strongly the wind-induced air velocities and pressures within the building complex of interest.

As no experimental data were available for the realistic definition of wind velocity profiles in the neighbouring streets, real wind tunnel tests had to be replaced by "numerical wind tunnel'' experiments. At first, a macro analysis of a large part of the downtown area has been performed in order to estimate the deformed wind velocity profiles within the global flow's subdomain around the interesting business quarter. The computational model developed for the first stage of analysis is shown in Fig. 8. Approximately 550000 control volumes were needed for the discretization of the global flow field. An example of the global air flow pattern is also shown in Fig. 8.

On the second level of the investigation, a more refined computational model of the business quarter itself has been developed (Fig. 9) the boundary conditions of which have been extracted from the macro analysis results by means of numerical interpolations. The CFD model comprises now all relevant building details such as entrance gates, roofs, open connections between the courtyards and the neighbouring streets, walls of the neighbouring houses etc. The open courtyards form a system of large "driven cavities'' which interact with each other both aerodynamically and thermodynamically. An example of the internal air flow pattern in Fig 9 shows that these complex physical interactions may lead to strong draughts in the pedestrian zones which, in turn, could never be predicted a priori by means of any simplified design method. This example demonstrates once more the usefulness of the CFD computations for the global building analysis.

At the third step, a detailed computational model of the glazed atrium has been developed for the simulation of the local comfort conditions. The atrium's simplified geometry, the computational grid with approximately 300000 control volumes as well as the indoor air flow patterns are shown in Figs. 10 to 13. The atrium's most critical region is situated in front of the round bistro where air velocities up to 4 m/s can occur on windy days due to the air flow acceleration along the bistro's curved walls. After the analysis of the detailed results, some architectural changes in the atrium's interior (additional walls and doors, entrance locks in the gates etc.) have been developed in order to reduce such strong draughts.

Air flows in atria with open office galleries

In many practical cases, the effectiveness of natural ventilation is strongly determined not only by the interaction effects between the building and its outdoor environment but also by the enclosure-to-enclosure interactions which occur within the building [12]. The latter may be very important for the air flow behaviour in a glazed atrium with open office galleries, as that shown e.g. in Fig. 14.

The atrium is completely closed and equipped with a fully mechanical ventilating system. Fig 15 shows the block-structured grid developed for the simulation of the air flow within this complex system of several coupled enclosures. Sample temperature and velocity distributions are plotted in Figs. 16 and 17, respectively.

On cold winter days, the comfort conditions in the atrium are not acceptable due to cold downdraughts which are induced by heat losses through the glazing of the atrium's pyramid (top parts of Figs. 16 and 17). As predicted further by the numerical simulation, these can be avoided by additional heating of the cold glazed walls (bottom parts of Figs. 16 and 17).

On warm summer days, the air flow in the atrium is strongly affected not only by the action of the heat sources which are situated both in the atrium and in the adjacent offices (e.g. persons, PCs, printers, lamps etc.) but also by the "greenhouse effect'' due to the solar radiation through the glazing. After the analysis of the simulation results which were obtained analogously for a critical summer day, some proposals could be made for improving the indoor air conditions on hot summer days. (See also reference [12] for further examples of atria with open office galleries.)

Can the natural ventilation of office galleries - and thus the office air quality - be improved noticeably by opening some windows towards the building's external environment?

Reference [12] describes an other building which is similar in form and size to the previous example. In this case, the building's open office galleries are not only connected directly with the glazed atrium, but also have openable windows located in the buildings external facades. The simulation results show that the atrium experiences a very intensive outdoor air infiltration as soon as some windows are opened. Unfortunately, the incoming external air flow causes strong draughts in the atrium's occupied zones. This danger increases with the growing open window area. However, such situations can be avoided successfully by the windows opening control, provided that the critical window opening schemes are determined in advance by means of the numerical simulation.

Natural ventilation of a shopping mall

The author's practical experience shows that the numerical air flow simulation is the best and most efficient way of determining the distribution and the size of windows, doors and other openings in naturally ventilated large enclosures of arbitrary geometry and functional requirements (see e.g. [12]). This approach is demonstrated impressively by the following (and last) example of the present paper.

A glazed shopping mall (Fig. 18) is to be ventilated naturally via two large openings situated in its two gable walls. In the initial design concept, the two large windows are located in ca. 6 m height above the floor level. The top plot in Fig. 19 shows strong draughts in the mall's occupied zone, regardless of the building's groundplan shape. The air flow pattern in the occupied zone can be improved significantly by locating the open windows in ca. 15 m height above the floor level, by reducing their open area by ca. 50 % as well as by changing the mall's groundplan shape from a parallelogram to a rectangle (compare the bottom plot of Fig. 19).

CONCLUSIONS

Some aspects of the numerical analysis of air flows in buildings have been discussed placing special emphasis on the interactive nature both of the physical behaviour and of the design of ventilating systems for modern buildings. A consistent analysis method has been presented which takes into account the action of external and internal air flow fields.

The examples included in this paper as well as many other interesting applications, as those discussed e.g. in [4 to 12], demonstrate clearly that the numerical simulation of air flows

does not restrict the architect's design freedom and creativity,

is no absolute alternative, but only a reasonable support to the traditional design and experimental methods,

is very informative in almost all conceivable practical design cases and

becomes an essential for the Integrated Planning of large modern buildings and their HVAC systems.

Unfortunately, CFD-aided projects are still affected by numerous limitations. First of all, the proper ambient, boundary and initial conditions as well as the proof of the grid independence of the simulated air flow patterns are very difficult to define prior to starting the numerical computations. The prediction of interaction effects within the building-environment system is therefore worthy of further numerical research and experimental validation.

REFERENCES

[1] Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within Buildings, Ottawa, 24-27 September 1991, Vol. 1-3

[2] Proceedings of the 4th International Conference on Air Distribution in Rooms ROOMVENT´ 94, Vol. 1-2, June 15-17, 1994 Kraków, Poland

[3] Proceedings of the 5th International Conference on Air Distribution in Rooms ROOMVENT´ 96, Vol. 1-3, July 17-19, 1996 Yokohama, Japan

[4] Glück. B., "Komplexe Simulation - Voraussetzung zur Energieoptimierung'', HLH Heizung-Lüftung/Klima-Haustechnik, Bd. 46 (1995), Nr. 3 - März, S. 143-148

[5] Ochocinski, B., "Entwicklung von raumlufttechnischen Konzepten auf Basis der Solarenergienutzung in Gebäuden bei Einsatz durchströmter Elemente der transparenten Wärmedämmung und Formulierung von industriell einsetzbaren Entwurfsmethoden; Teilbereich B : Simulation von Raumluftströmungen'', Abschlußbericht zum BMFT/ BEO-Forschungsverbundvorhaben Nr. 0329016 A, RUD. OTTO MEYER, Hamburg, Oktober 1993

[6] Ochocinski, B., "Energieökonomische Optimierung von Raumluftströmungen'', Abschlußbericht zum BMFT/BEO-Forschungsverbundvorhaben Nr. 0329016 D, RUD. OTTO MEYER, Hamburg, März 1994

[7] Ochocinski, B., "Numerical Simulation of Air Flows in Buildings in the Practical Design, Selected Examples of CFD-aided HVAC Projekts, ECCOMAS'92'', First European Fluid Dynamics Conference, Brussels, Belgium, Sept 7-11, 1992

[8] Ochocinski, B., "Rechnerische Raumströmungssimulation, Konferenzband zum Seminar: Raumluftströmung - Gestaltung und Optimierung," FH München, 4. Februar 1993

[9] Ochocinski, B., "Numerical Simulation of Indoor Air Flows Using FLOW3D'', First CFDS International Users Conference held in Eynsham Hall, Oxford, UK, on May 6-7, 1993

[10] Glück, B. and Ochocinski, B., "Strahlplattenleistung unter realen Einsatzbedingungen, Simulation der Luftströmung um Strahlplatten in Prüfräumen und Hallen'', HLH Heizung-Lüftung/Klima-Haustechnik, Bd. 44 (1993), Nr. 3 - März, S. 158-165

[11] Ochocinski, B., "Simulation von Raumluftströmungen in der Technischen Gebäudeausrüstung (Projektvorstellung)'', FIA News, Forschungs-Informations-Austausch, Fachinstitut Gebäude-Klima e.V., 9.Ausgabe, August 1995, S. 3-6

[12] Ochocinski, B., "Coupled Simulations of Internal and External Air Flows for Building Technology'', CFDS European User Group Meeting held at Shoppenhangers Manor, Windsor, UK, on 27-28 June 1995

[13] CFX 4.1 User Guide, AEA Technology, Computational Fluid Dynamics Services, Harwell Laboratory, Oxfordshire, UK, 1995

[14] CFX 4.1 Flow solver: User Guide, AEA Technology, Computational Fluids Dynamics Services, Harwell Laboratory, Oxfordshire, UK, 1995

[15] FIELDVIEW User's Manual, Software Release Version 5.5, Intelligent Light, Lyndhurst, NJ, USA

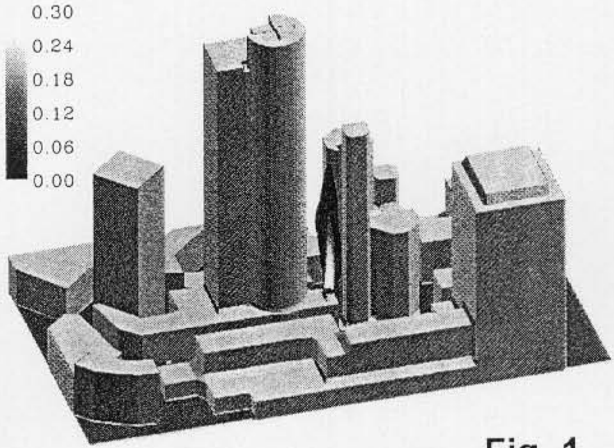

Fig. 1

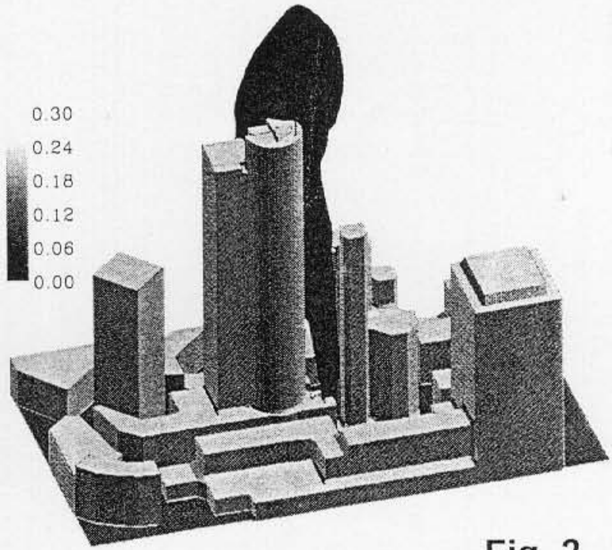

Fig. 2

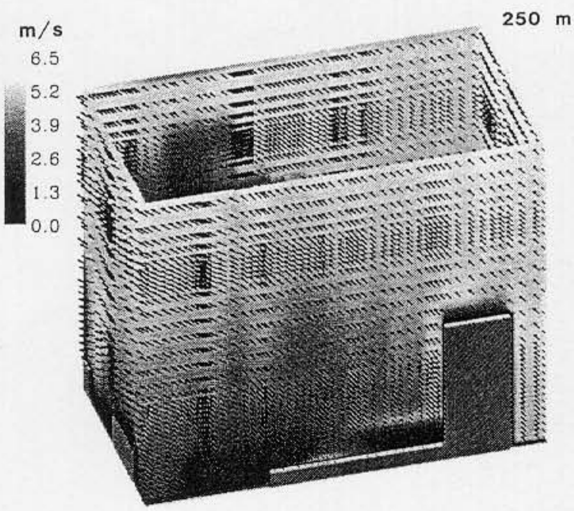

Fig. 3

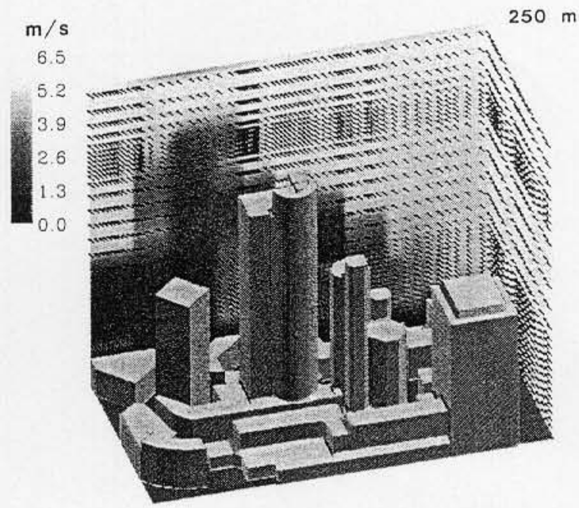

Fig. 4

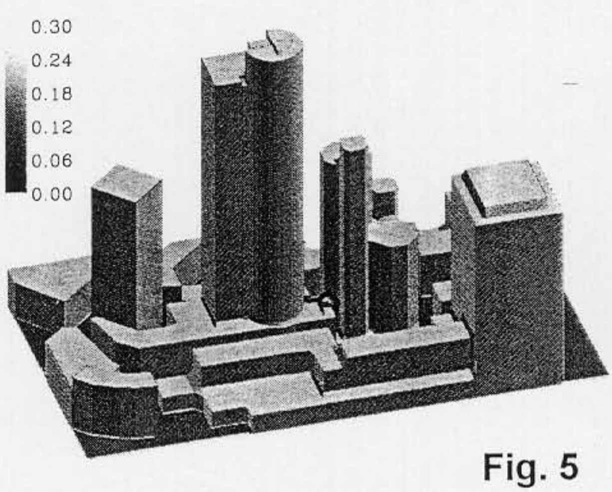

Fig. 5

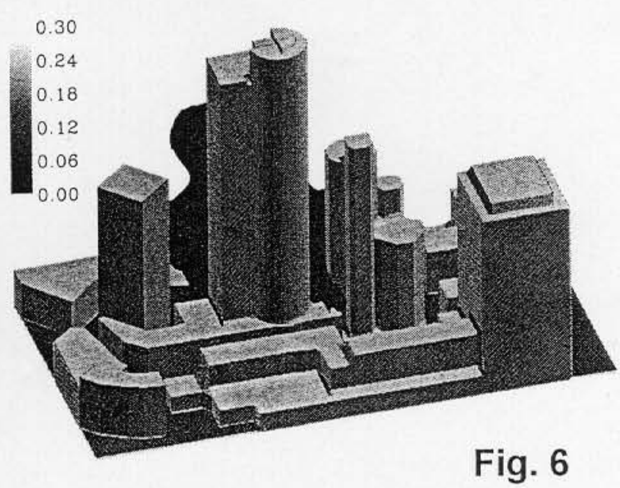

Fig. 6

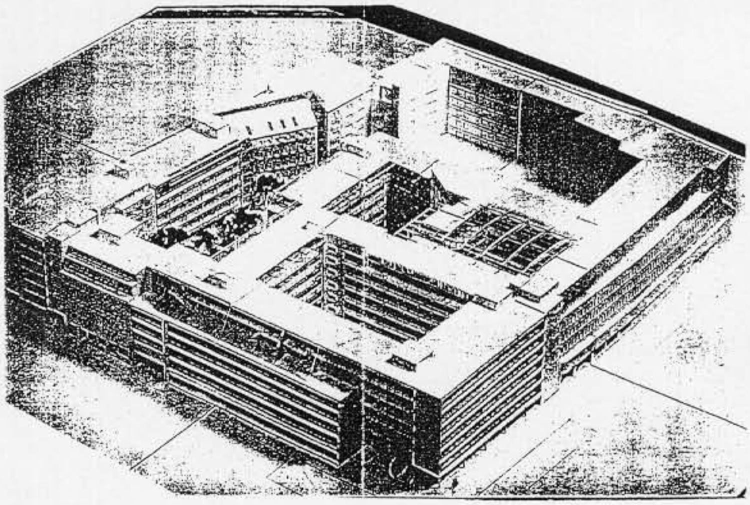

Fig. 7

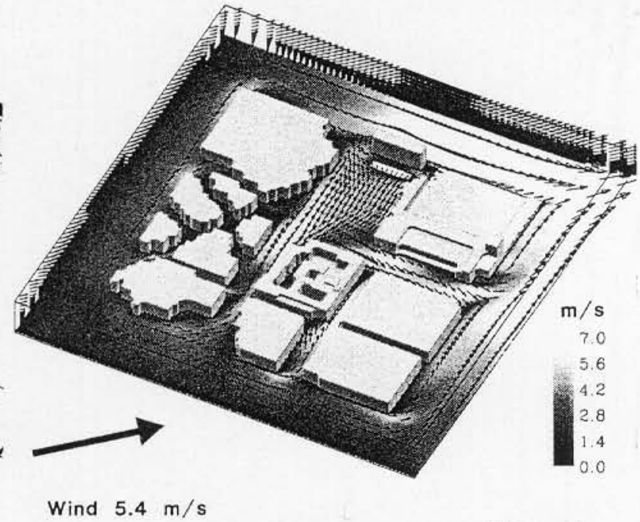

Fig. 8

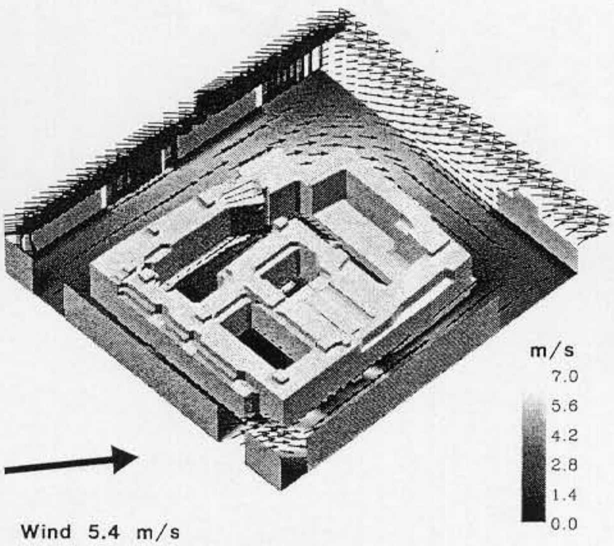

Fig. 9

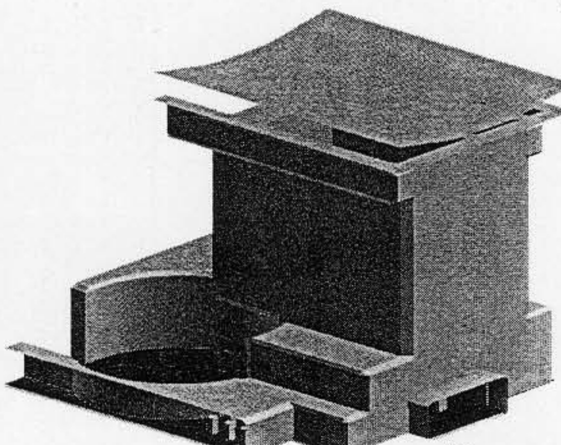

Fig. 10

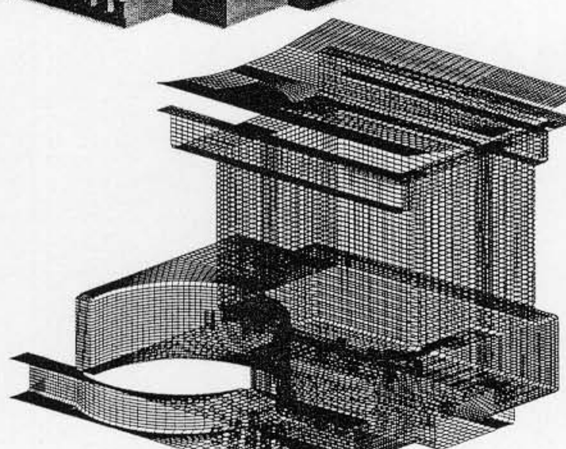

Fig. 11

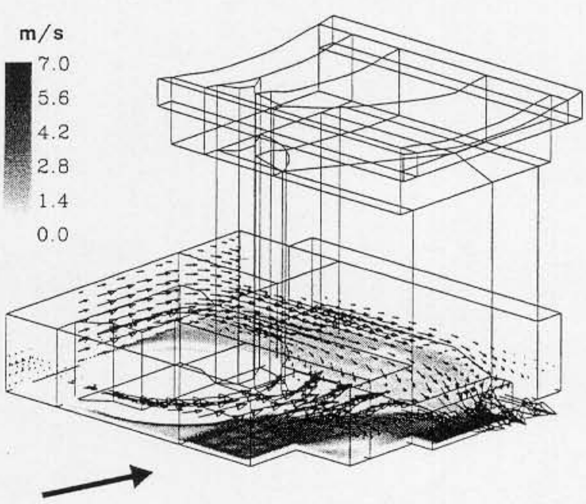

Fig. 12

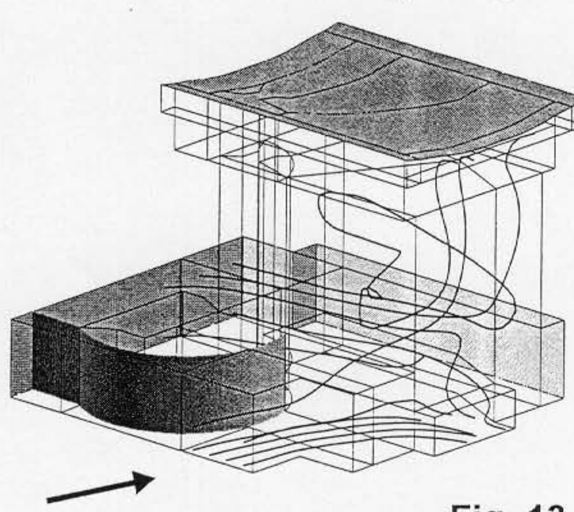

Fig. 13

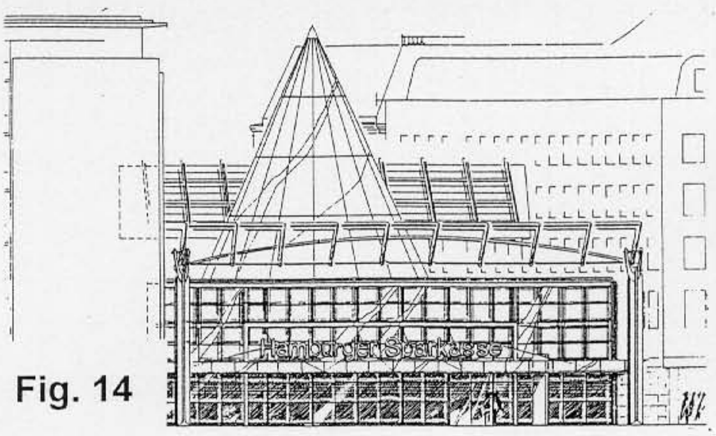

Fig. 14

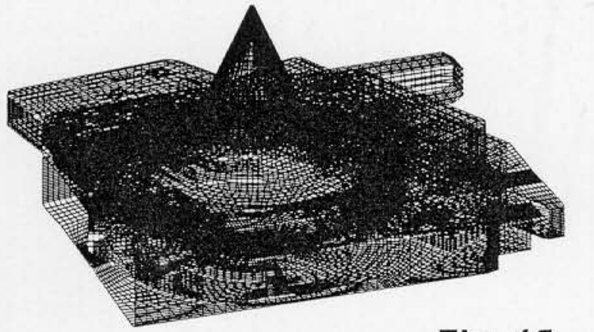

Fig. 15

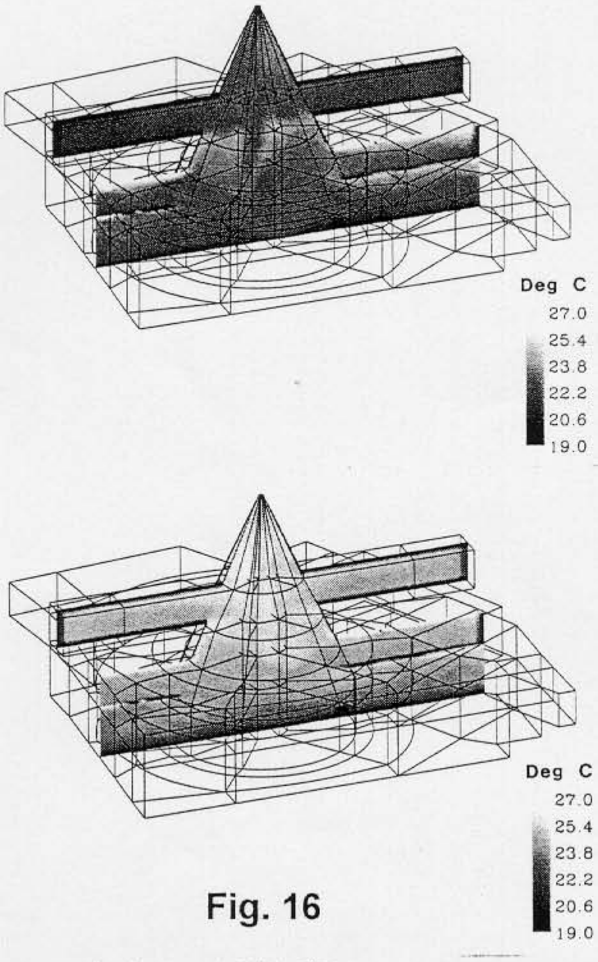

Fig. 16

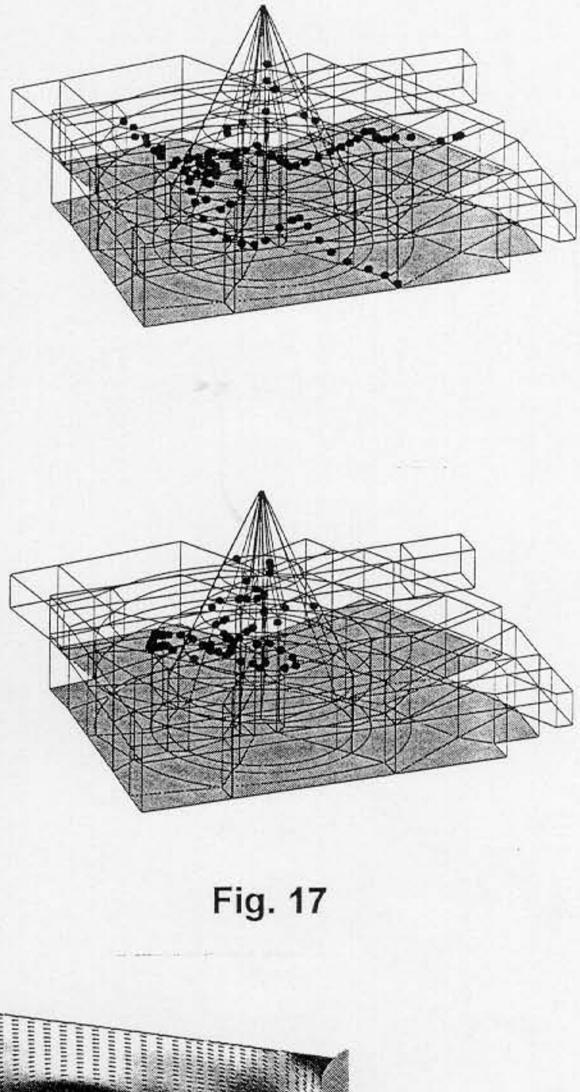

Fig. 17

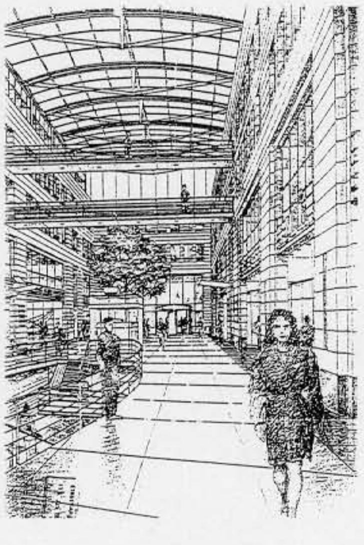

Fig. 18

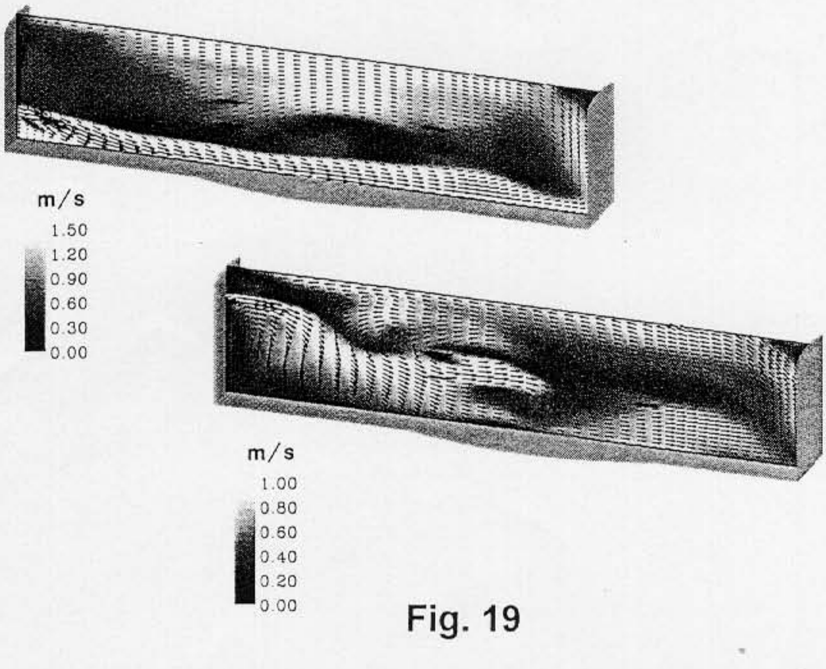

Fig. 19