Using a Constant Volume Displacement Ventilation System to Create a Micro Climate in a Large Airport Terminal in Bangkok

Peter Simmonds
Flack + Kurtz
New York

Wayne Gaw
Flack + Kurtz
San Francisco

Abstract

In order to conserve energy and create a comfortable climate for both passengers and workers at a new large international airport in Thailand, a design concept was created where only the first 2m above the occupied zone is conditioned. The temperature of the air outside of this area is allowed to rise above normal conditions. The idea was to let this temperature rise so that it was either equal to or higher than the outdoor temperature, thus reducing heat gain. Computer simulation programs were used to define parameters for the CFD program. Once the boundary conditions were defined, the process of design analysis began. This paper will outline the steps taken to set up the CFD program. Secondly, the exploration taken to obtain an optimal climate, and thirdly, how the many results were used to explain to both fellow engineers and the architects what had been achieved. The conclusion of this analysis was the design of special supply air grilles to meet the design criteria.

Introduction

This large international airport, which is to be built in Bangkok, Thailand will have a total floor area of 550,000 m^2. The construction of the building envelope will be glass and PTFE membrane material. The airport will function 24 hours a day for both domestic and international passengers. The spaces within the airport must remain comfortable at all times. A tremendous amount of heat can be admitted to the space through the envelope of be directly emitted into the space from occupants, lighting and electronic equipment. Because of the nature of the airport there is a large diversity factor. Many computer simulation programs were used throughout the design process to assess the various dynamic characteristics of the building and its HVAC equipment. CFD analysis was used to investigate the effectiveness of a displacement ventilation system. A displacement ventilation system pours ventilation air into a space across the floor. The temperature difference between the supply air and the room air ensures that the air remains at ground level. When this air comes into contact with occupants or heat sources it rises up, removing heat and contamination from the occupied zone.

Background

There have been numerous designs applied to airport ventilation and conditioning systems. Kansai airport in Japan uses a buffer between the envelope and the indoor environment. CFD analysis as used to assess the air flow characteristics. Stanstead airport in England and Schiphol airport in The Netherlands uses a fountain system Air is supplied into the space at about 2m above the floor, because this air is at a lower temperature than the space temperature, and therefore this air will fall downwards in a fountain effect to the floor. Munich and Frankfurt airports in Germany use a combination of jets and displacement ventilation. Terminal 4 at Heathrow airport, England uses a displacement ventilation system. This displacement system is unique and it not only ventilates, but cools the occupied space.

Design Parameters

The occupied space must remain within a set of temperature limits 24 hours a day (20 - 24°C). Air must be supplied to the space so that each occupant can receive 36m 3/h of fresh air. Occupants must not be subjected to drafts, air velocities in the space will be between 0.6 to 0.2 m/s. The design of a supply air diffuser has many limitations it should be large enough to supply the necessary air volume into the space. Architects don't like large objects, therefore the diffuser should remain small, but effective. By decreasing the size of the diffuser, the supply velocity increases. Decreasing the velocity decreases the air flow. High velocities also produce unwanted acoustics. Somewhere between these limitations there is an optimum design.

CFD Modeling

Before the CFD can be modeled the boundary conditions must be determined. This was done by using a dynamic thermal simulation program developed by Holmes (1988). The same building dimensions were used for both the thermal and the CFD input. Once the boundary conditions such as surface temperatures and heat fluxes had been defiled the CFD model could be constructed.

There were two designs that had to be evaluated, the Concourse and the main terminal.

Concourse

This model had a semi circular roof construction. Body fit coordinates were used to flow the roof contours. Several different configurations of supply and return grilles were simulated. Simulation results were investigated for a true representation of temperatures within the space. The CFD results were compared to the thermal simulation results. Once the temperature profiles of both programs correlated, then the air flows were analyzed for their effectiveness.

Terminal

Although the shape of this building was rectangular, its size and diversity of usage provided sufficient exercise in both thermal the CFD modeling.

Results

Concourse

Figure 1 shows the ventilation air distribution through the space. The near constant temperature distribution through the space was not the required result. The air flow shows certain blind spots, places where the air is not fully distributed through the space. This would not provide the designed ventilation patterns and effectiveness.

Horizontal diffusers were then modeled. Four diffusers would be placed across the section through the Concourse, this is shown in Figure 2. It was assumed that air would be returned behind the two side wall diffusers. Figure 3 shows the results of this simulation. Air flow and distribution through the occupied space as within the prescribed units. However, the turbulence at the side of the curve, above the two side wall diffusers, did cause concern. (It was viewed that occupants would or could stand near the perimeter to view outside). The return grilles were then moved to the walkway. The results improved dramatically (see Figure 4). This last model was then adopted to show a high level return grille at the second level. Figure 5 and 6 show the results. It can be seen that the temperatures in the occupied zone remain within the required limits. The air flow through the space is laminair and there are no blind spots. By moving the model through a 3 dimensional mode, the effect of the air flow across the floor can be studied.

Terminal

As stated before the terminal had a rectangular shape so it wasn't the geometry that proved to be difficult. However, the mix-match of displacement diffusers, and a few air jets did prove to be interesting. The objective was to see how the space would be ventilated and to investigate if the air jets would induce the displacement flow which would decrease the ventilation effectiveness.

Figure 7 shows the results of supplying too much air into the space; the stratification level is too high. Figure 8 shows the results when reducing the air flow to the space, the temperature profile is lower. This system would consume less energy than the system simulated in Figure 7.

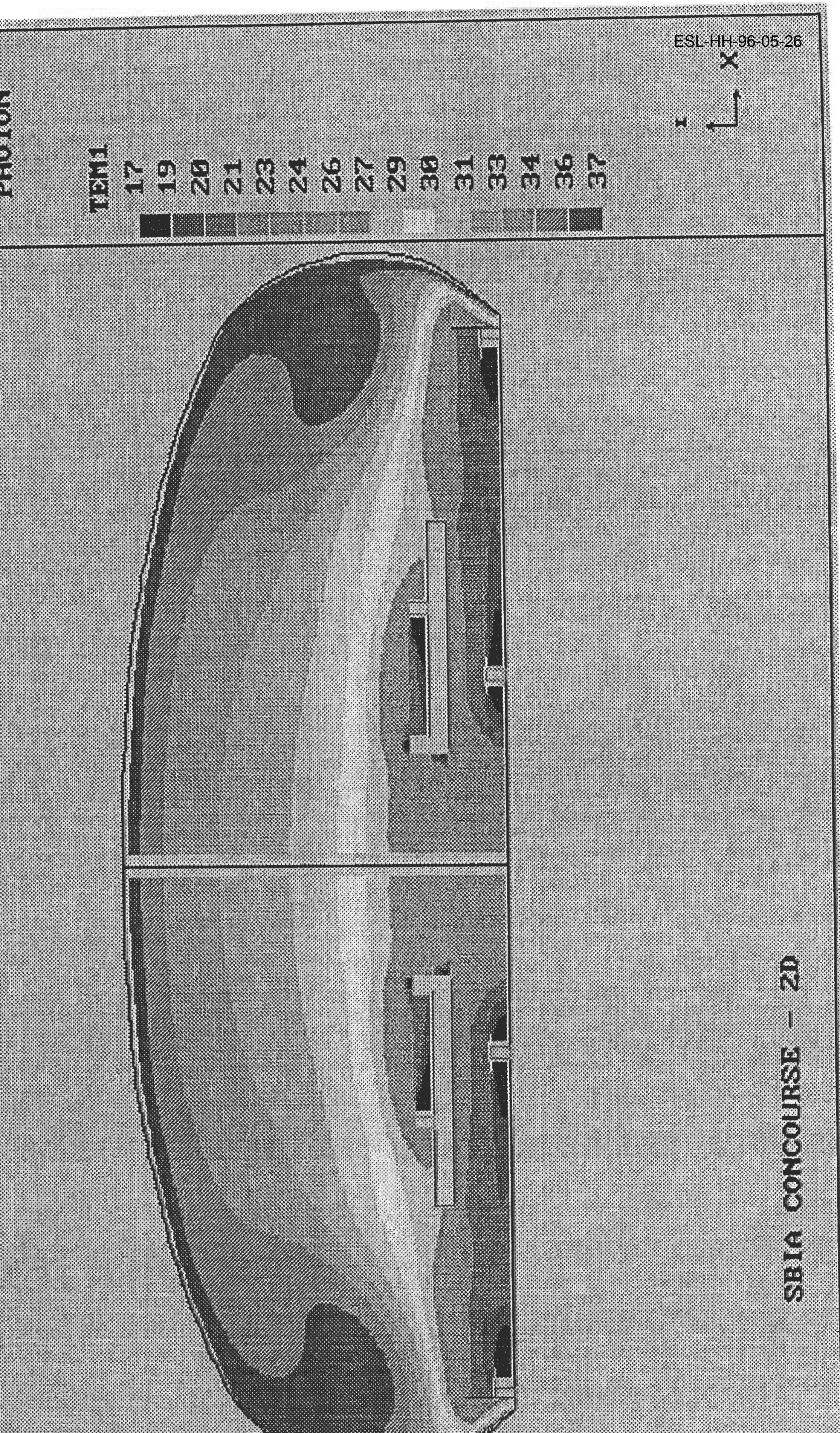
Conclusions

The development of a specialized ventilation system together with the application of CFD techniques into a well established consulting engineers did have its moments. Once the model had been built in the program, and the parameters were checked and the first results were attained, the most painful process started, explaining to experienced engineers what they were looking at, and how this should be visualized in a real world application. The results are often confusing and not logical, but what should not be forgotten, is that a series of simulation results should be compared to each other in order to evaluate the effectiveness of the system.

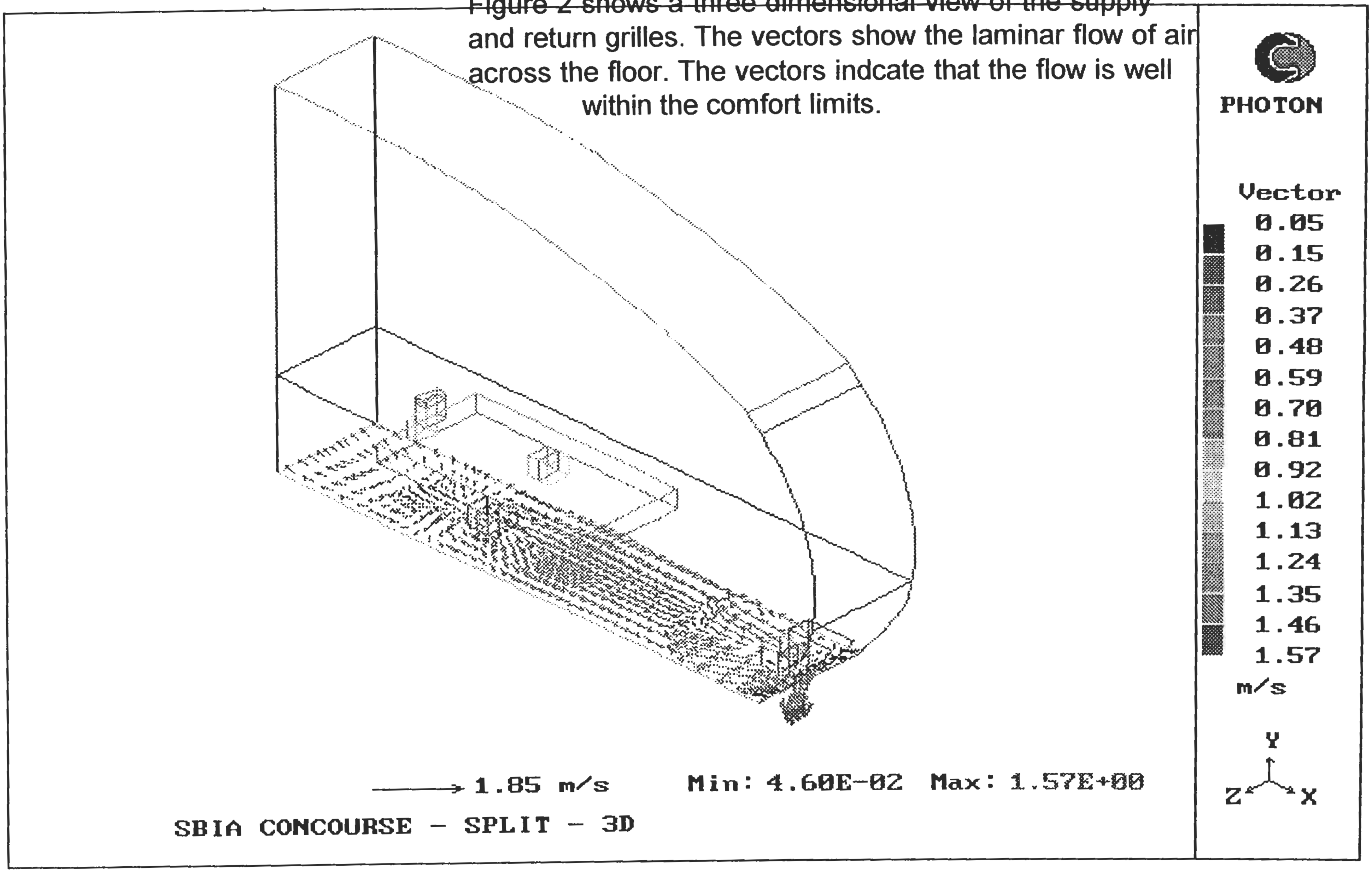
The overall effect was very useful, when following the whole design process the characteristics of each system were apparent. The CFD results are going to be used as a reference. When the specially designed diffusers are being factory tested, smoke will be used to show the path of air, the flow patterns of smoke and CFD will be compared in order to assess the performance of the diffusers.

Proceedings of the Tenth Symposium on Improving Building Systems in Hot and Humid Climates, Fort Worth, TX, May 13-14, 1996

Figure 1 shows the temperature profiles across the concourse when supplying air through dislacement diffusers and returning air at the perimeter.



~~Figure 2 shows a three dimensional view of the supply~~ and return grilles. The vectors show the laminar flow of air across the floor. The vectors indcate that the flow is well within the comfort limits.



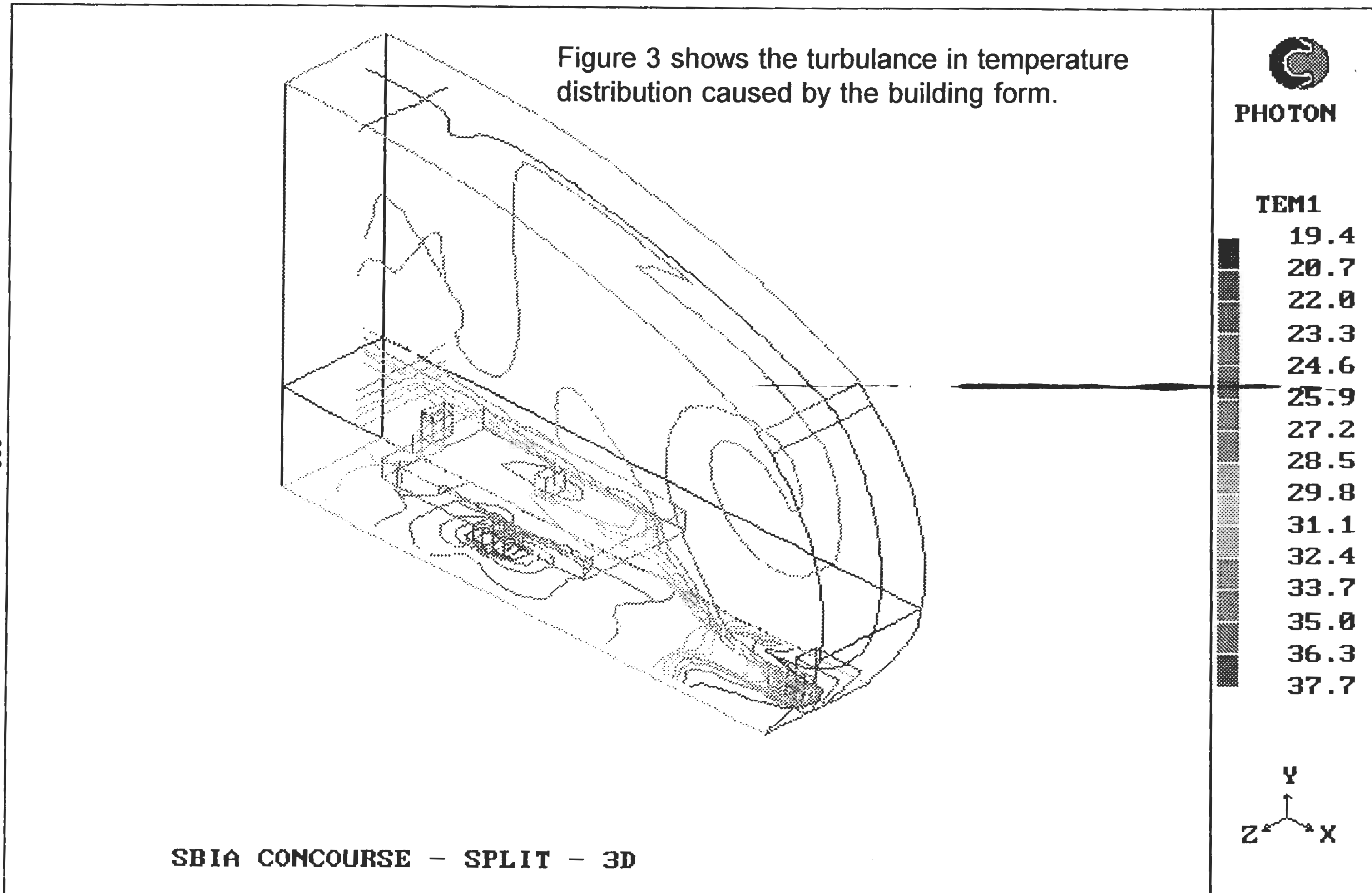


Figure 3 shows the turbulance in temperature distribution caused by the building form.

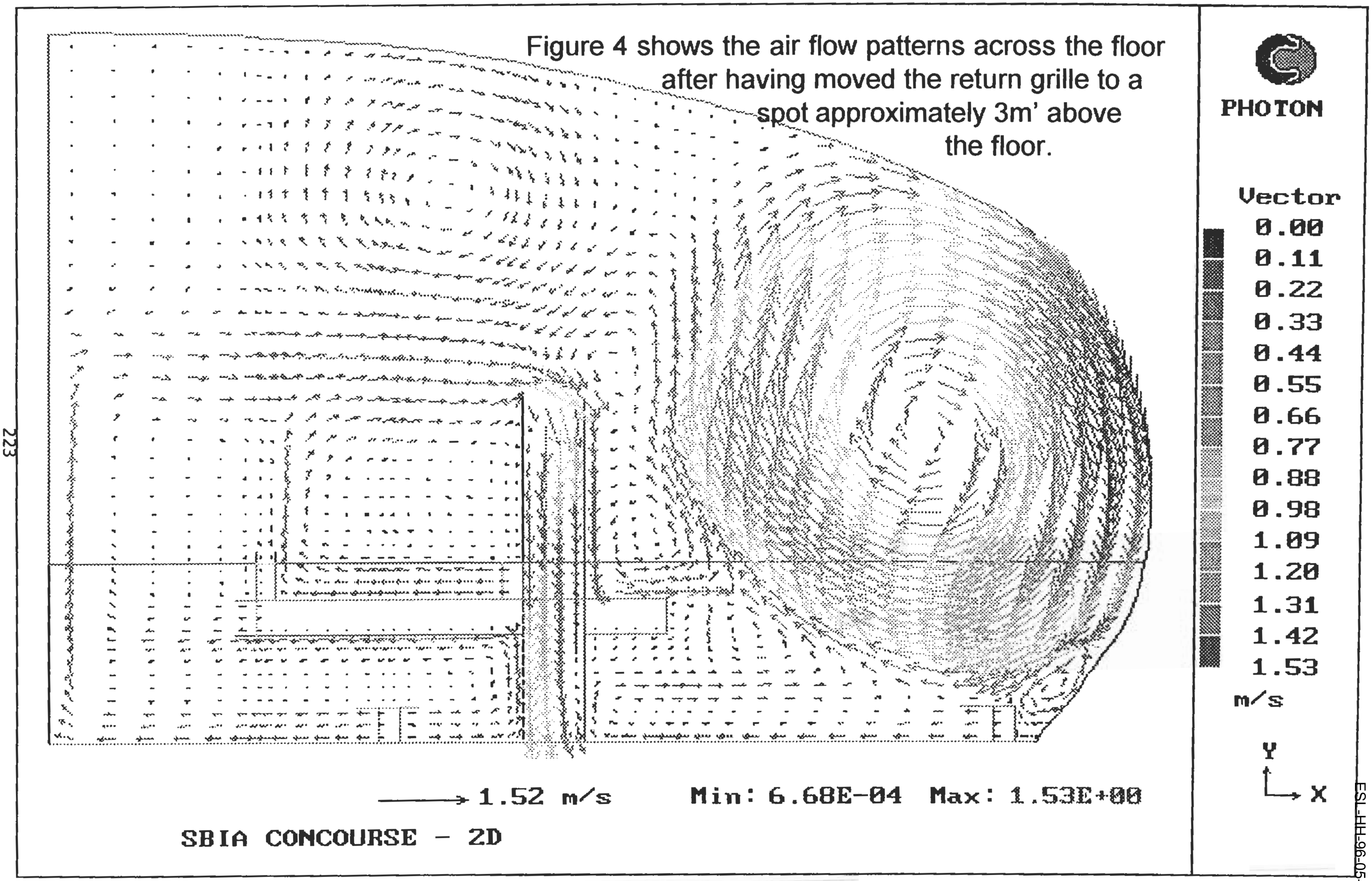


Figure 4 shows the air flow patterns across the floor after having moved the return grille to a spot approximately 3m' above the floor.

Figure 5 shows the section through the concourse, the temperature stratification can be easily seen. The occupied spaces are conditioned with cool fresh air, while the in the unoccupied areas the temperatures are allowed to rise which reduces the temperature difference across the envelope, which decreases the heat gain to the space.

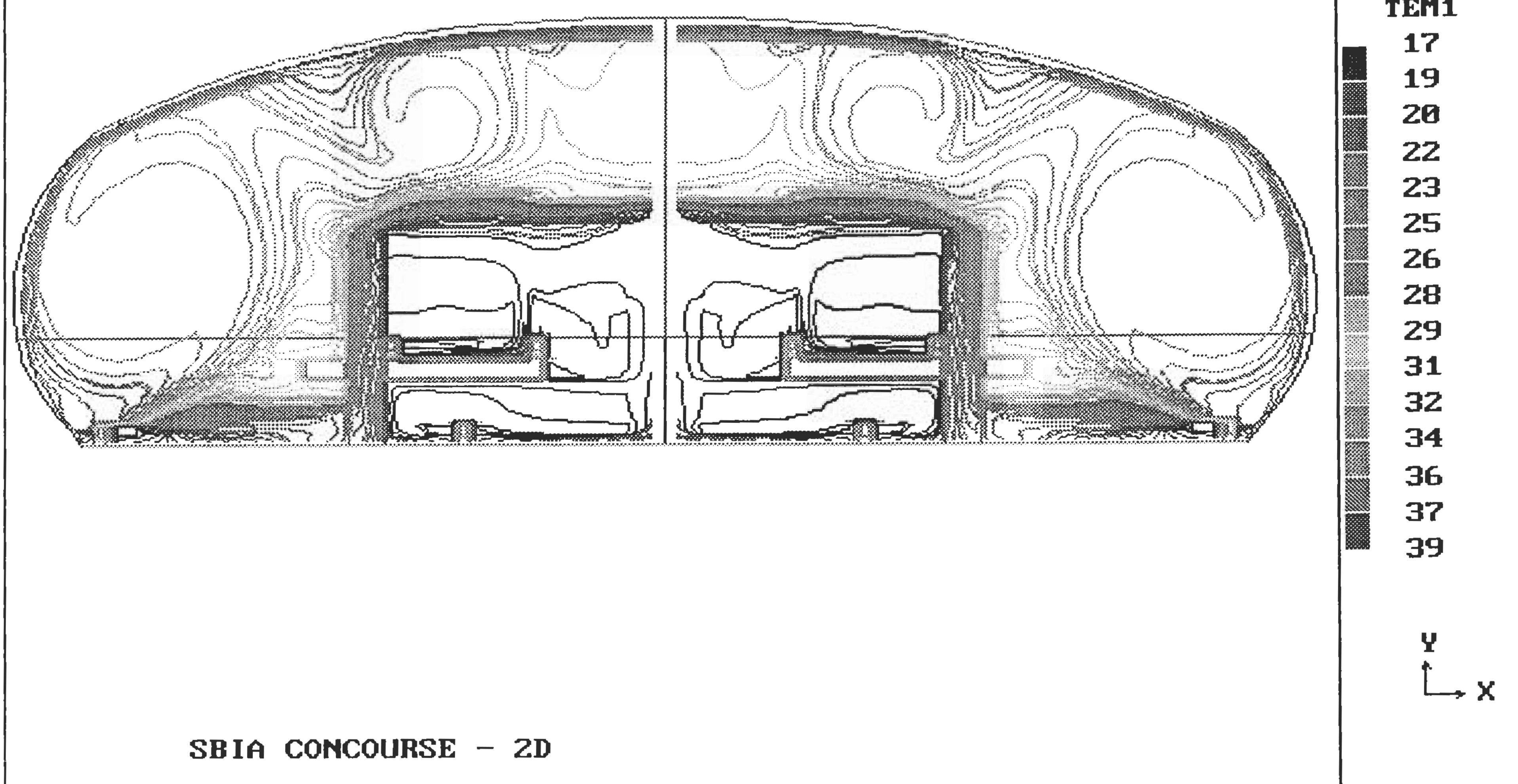


Figure 6 shows the air velocities in the space, the occupied zone is conditioned by a laminar flowof ventilation air pouring across the floor.

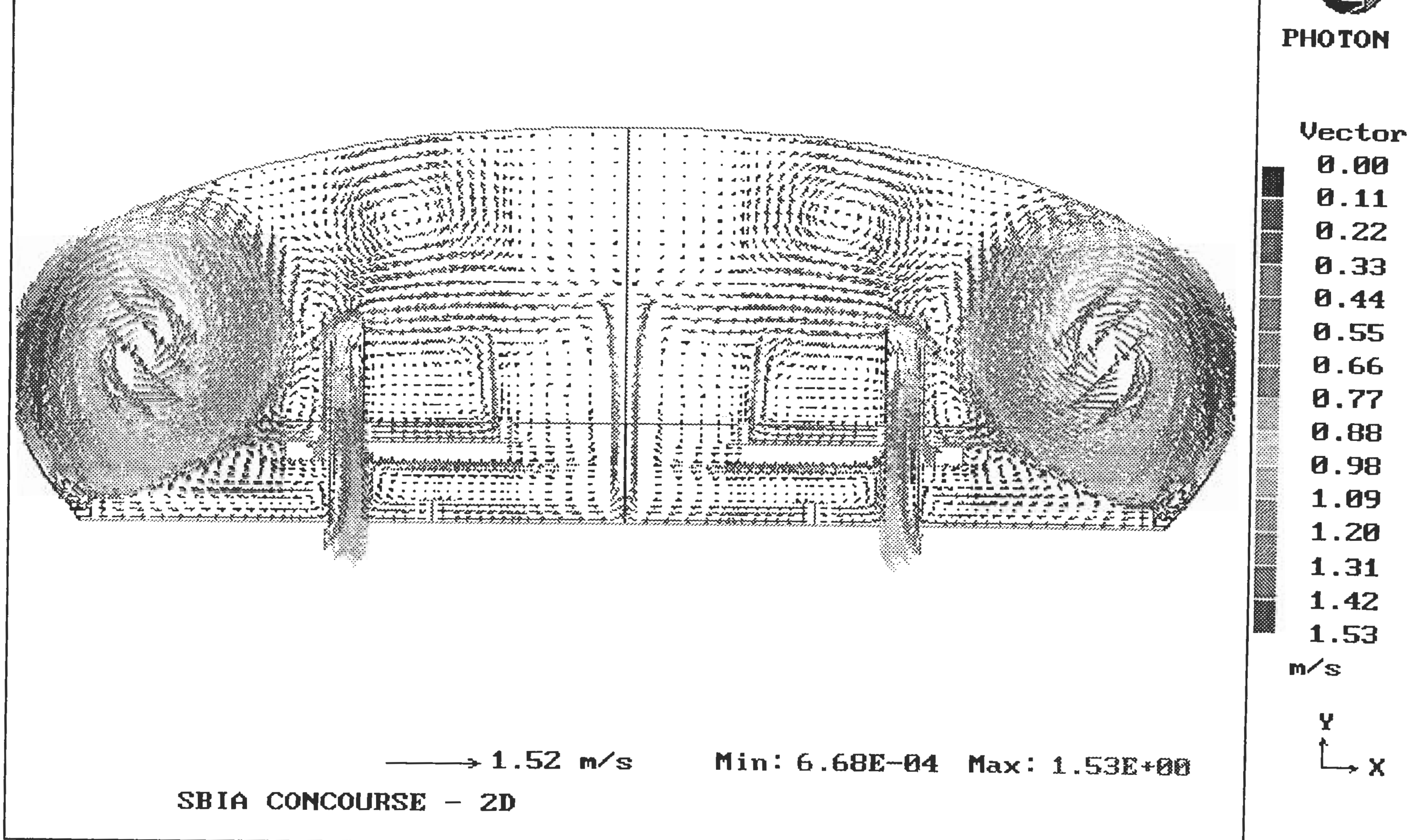


Figure 7 shows aview across the terminal, the floor diffusers are supplying too much ventilation air and there is insufficient stratification.

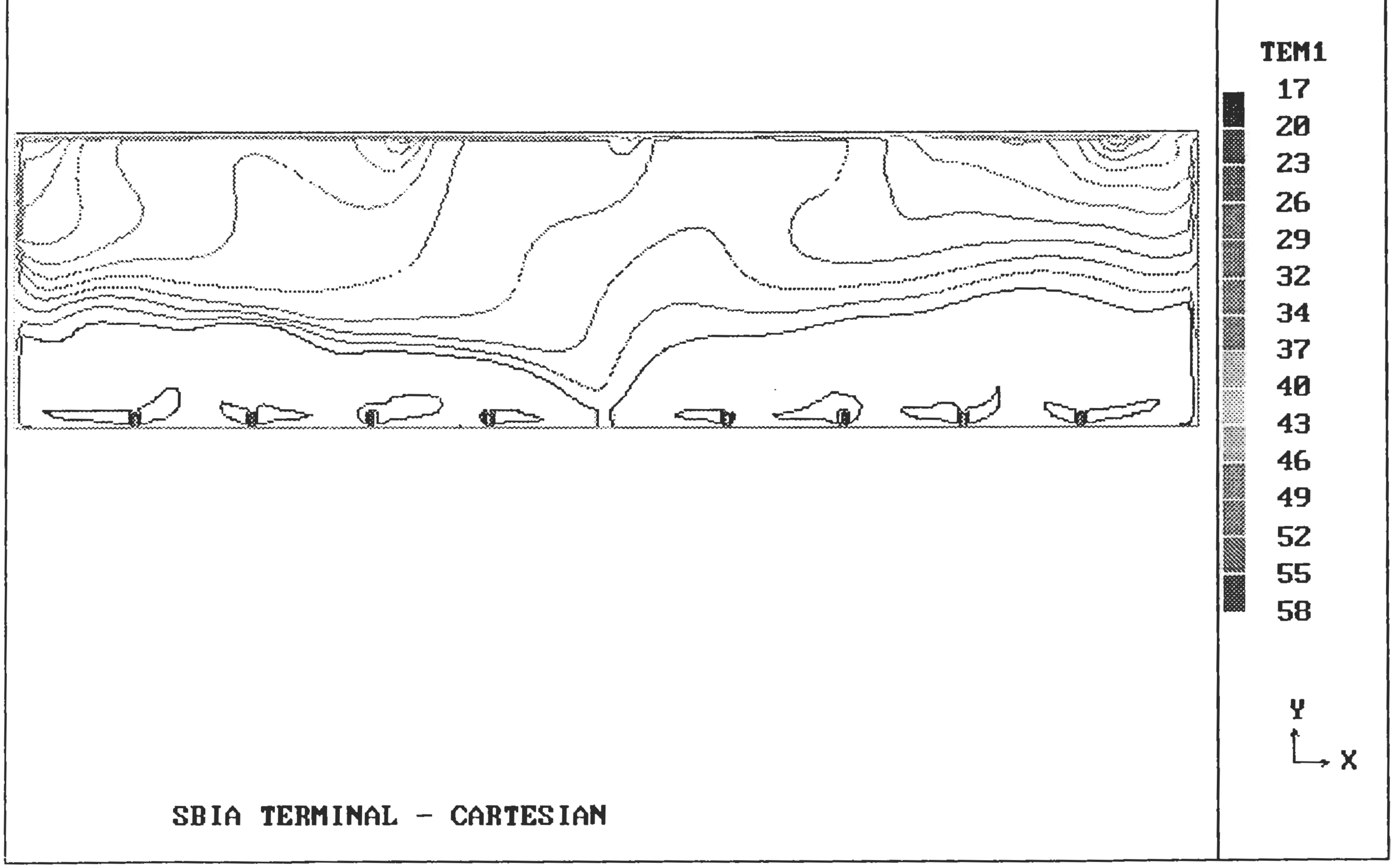
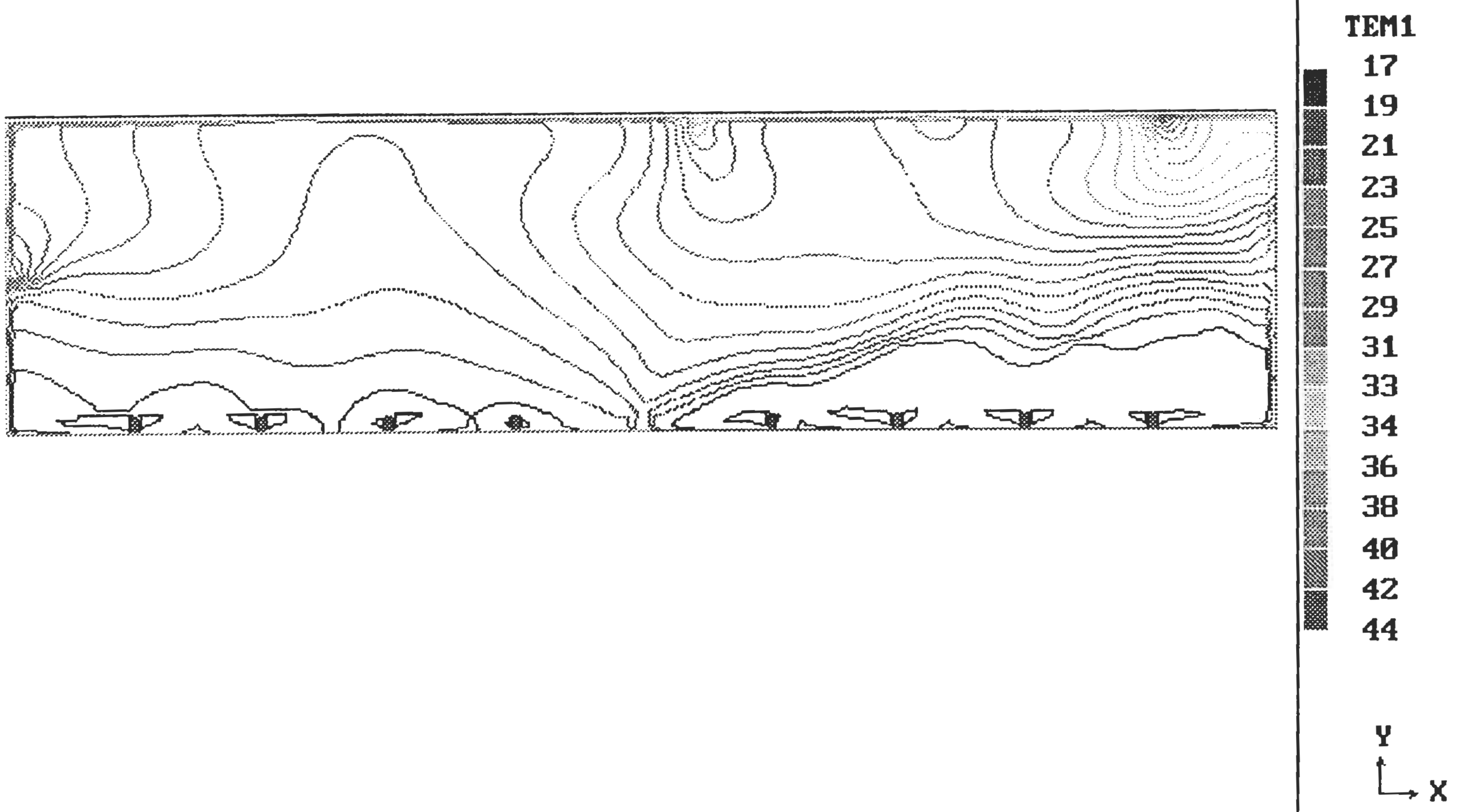


Figure 8 shows the results obtained by decreasing the air flow to induce stratifcation.

PHOTON



TERMINAL - low-velocity