Simulation of Buoyant Convection in Buildings via Computational Fluid Dynamics (CFD)

Wolfram Haupt

University of Kassel, Fachgebiet Bauphysik,
Gottschalkstr. 28, D-34109 Kassel

Abstract. As detailed measurements of buoyant flows in buildings are very rarely found in literature, a box with a heated plate on bottom and a cooled plate on top of the box was measured by Particle Image Velocimetry (PIV) and smoke experiments [1]. The results show asymmetric and unsteady behaviour of the buoyant flow inside the box.

While CFD-simulations in 2D (both steady-state and transient) were incapable of calculating the asymmetric status of the flow, even coarse grids on 3D were able to predict the observed asymmetry as well as certain aspects of the measured fluid flow. The transient simulations showed much better convergence if carried out with rather short time-steps of less than one second. However not all aspects of the fluid flow could be predicted by the calculations.

1 Preface

CFD-Software has the potential of being a valuable tool in the area of building physics. Not only the effects of heat exchange (conduction, convection and radiation) as well as thermal comfort can be modelled by this software-tool, but also moisture can be taken into account.

In areas as aviation or turbo-machinery the use of CFD-software is very wide-spread and many validations origin from these areas. In the area of building physics the tool was first used in simulations of forced convection, often in 2D-steady-state-models, due to the large numbers of grid-cells needed to model whole buildings, atria or even single rooms.

In 1993 a summary report [2] of the International Energy Agency (IEA) Annex 20 (Air Flow Patterns within Buildings) states that "Experimental and numerical results suggest unsteady air motion under certain conditions at high Rayleigh number. However, this must still be verified. If in fact, steady solutions do not exist under some circumstances, time dependent simulation would be appropriate". Furthermore it is stated that "The method to compute only a half room under symmetrical boundary conditions, is not always valid. The benchmark exercises indicate that geometric symmetry may not result in flow symmetry. This should be investigated further".

Fig. 1. Measurement setup.

2 Measurements

In order to validate the CFD-Simulations, a box of 1.1 x 0.74 x 0.49 m (length, width, height) containing a plate (0.5 x 0.5 m) heated to 40 degrees centigrade on the bottom and a plate (0.52 x 0.51 m) cooled to 10 degrees centigrade on top of the box was measured via PIV. Two coupled Neodym-YAG lasers with an energy pulse of 25 mJ and a digital camera with a resolution of 1280 x 1024 pixel were used. Via a cylindrical lens a light sheet of an angle of 20-30 degrees and (depending on the contrast) approximately 60 cm depth can be reached, which made it possible to measure the right half of the box.

When 20 laser shots, taken with 1 second time interleave are averaged, the measurements show a big vortex with the maximum vertical velocity up to 0.12 m/s reached close to the right hand edge of the heated plate. The shots itself show a very transient behaviour with velocities up to 0.23 m/s followed by 2-4 seconds of rather low velocity as can be seen in fig. 3 and 4.

In addition to the PIV measurements, smoke experiments were carried out, which showed clearly the unsteady nature of the flow. It could bee seen, that the flow was not only asymmetric along the longitudinal axis of the box but also along the narrow side and that fluctuations happened constantly, carrying the flow back and forth parallel to the narrow side of the box.

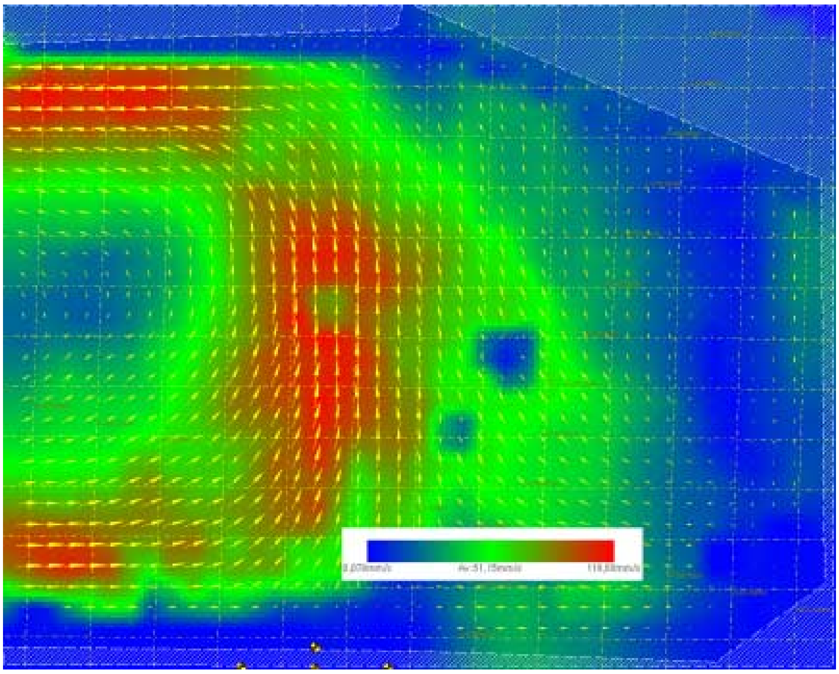

Fig. 2. Averaged velocity.

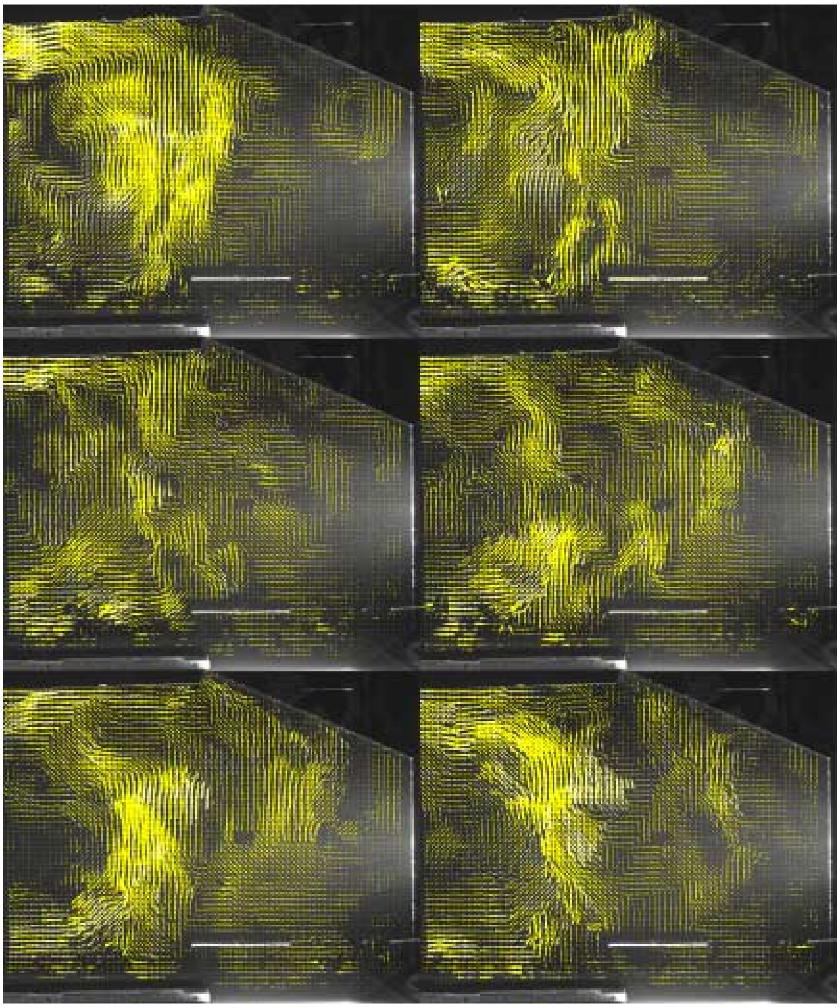

Fig. 3. Velocity timesteps 1-6 seconds (1 second interleave).

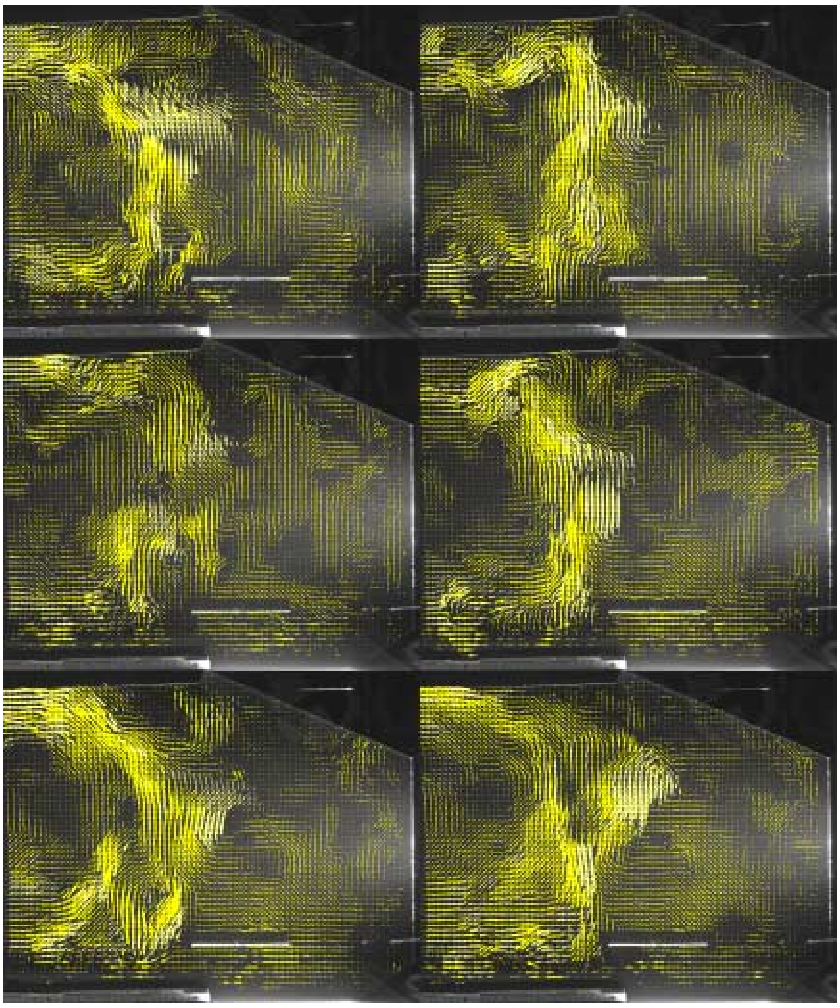

Fig. 4. Velocity timesteps 7-12 seconds (1 second interleave).

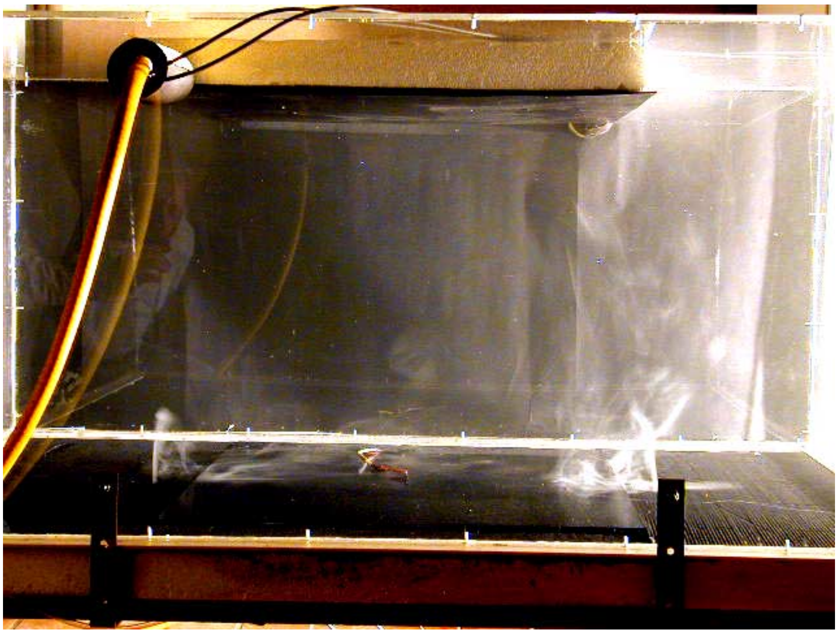

Fig. 5. Smoke experiments side view.

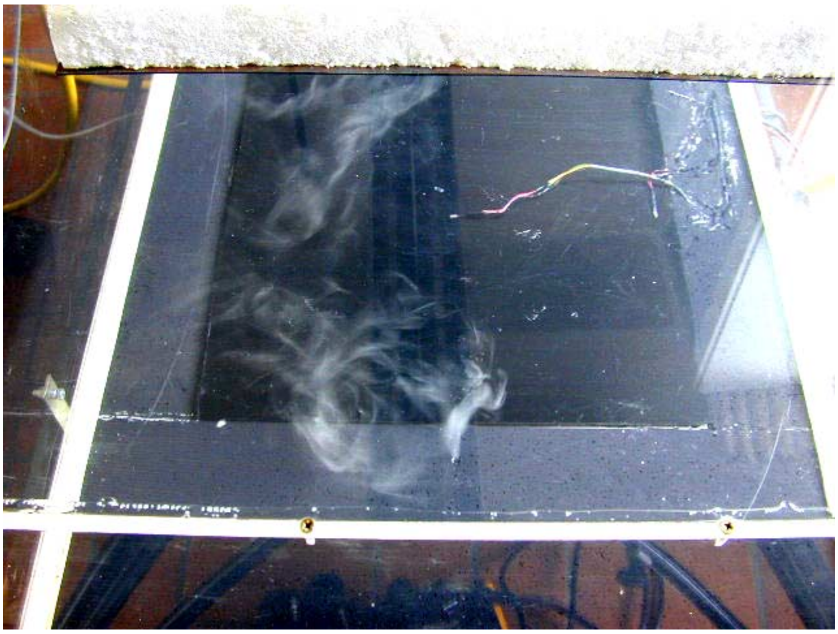

Fig. 6. Smoke experiments top view.

3 CFD-Simulations

The CFD-simulations included 2D as well as 3D cases with both steady-state and transient calculations. The grid was varied from rather coarse (363 gridcells on a 2D case) to very fine (393556 gridcells on a 3D case). Also different topology of the grids was tested: triangular and quadrilateral cells on 2D, tetrahedral as well as hexahedral cells on 3D with and without grid refinement close to the walls in order to make use of advanced wall treatment.

The simulations were run on a Cray T3E with 512 Processors (MPP) and mainly on two AMD-Athlon800-based PCs with 1Gb RAM each. The software used was FLUENT 5.1 on the Cray and FLUENT 5.4 on the PCs.

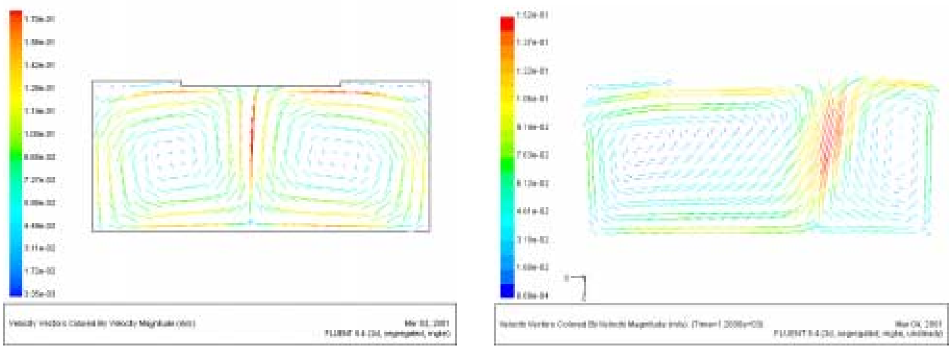

Fig. 7. 2D results vs. 3D results.

As long as the numerical model could be stored in the PCs' RAM, the velocity advantage of the Cray was only marginal. For a small model with 58968 cells a calculation-step on the Cray takes 7 seconds while on the Athlon a step is calculated in 8 seconds. The biggest model that would fit into the PCs RAM with 393556 cells takes 54 seconds per calculation-step on the Cray compared to 90 seconds on the Athlon.

The budget on the Cray allowed only 42 hours of calculation on 16 CPUs, so it was only used for one 3D transient case. The remaining cases were calculated on the PCs. The result of the Cray case shows after 75 calculated seconds a symmetrical buoyant flow centred over the heated plate.

The other cases on coarser or 2D grids were carried out for up to 600 seconds. The 3D cases all show the measured asymmetric flow. When observing the development of the flow, it can be seen that all cases show the centred flow with two vortices until the simulated time has reached 2 to 2.5 minutes (see fig. 8 and fig. 9). After this time all of the 3D cases but the case with tetrahedral mesh change their behaviour to the asymmetric flow with only one vortex.

As the case simulated on the Cray had to be stopped before reaching this margin, it was obvious that the correct flow pattern could not be predicted by this simulation. Based on the result of this simulation the case was simulated further on the PC. After another 50 seconds of simulated time-steps (calculation time on the PC about 3 days), the results show the pattern of the asymmetric single vortex.

The 2D cases however show merely the flow pattern of an axisymmetric flow with two vortices (see fig. 7), which was not observed in the PIV measurements. Even when in the transient cases the calculation time is extended to more than 60 minutes there is no change in the flow pattern. The same wrong flow pattern was calculated when, for the sake of saving computational resources, only one half of the box was modelled.

In contrast to the 2D steady-state cases, the convergence of the 3D steady-state calculations was over all very poor and could only be achieved by using under-relaxation of the computed equations, which means less coupling between the variables on successive steps of the solution. Changing the solution method to transient calculations with sufficiently short time-steps of less than a second enhanced the convergence of these cases distinctly. In addition the hovering of the flow back and forth along the narrow side of the box, as seen in the smoke experiments could be observed clearly in the numerical results.

Even rather coarse grids (6215 gridcells) gave results close to the measured velocities. Only the prediction of the location of the maximum vertical velocity was better on fine grids.

4 Conclusions

In order to achieve numerical results close to the experimental results in the case of the buoyant flow in a box as described above, it was essential to use 3D simulations without making use of the symmetrical geometry and boundary conditions. The results could improved further by using transient simulation.

As concerns this examined case, 2D and 3D simulation with symmetry plane was incapable to predict the mere flow pattern.

The outcome of the experiments suggest a very judicious examination of CFD results concerning buoyant flow, if produced by 2D simulation.

References

1. W. Haupt. *Zur numerischen Simulation auftriebserregter Innenraumströmungen.* PhD thesis, Universität Kassel, FB Architektur, D-34109 Kassel, April 2001. Also available via http://www.dr-cfd.de.
2. A.D. Lemaire, editor. *Room Air and Contaminant Flow, Evaluation of Computational Methods.* Annex 20-Air Flow Patterns within Buildings; TNO Building and Construction Research/Delft, 1993.

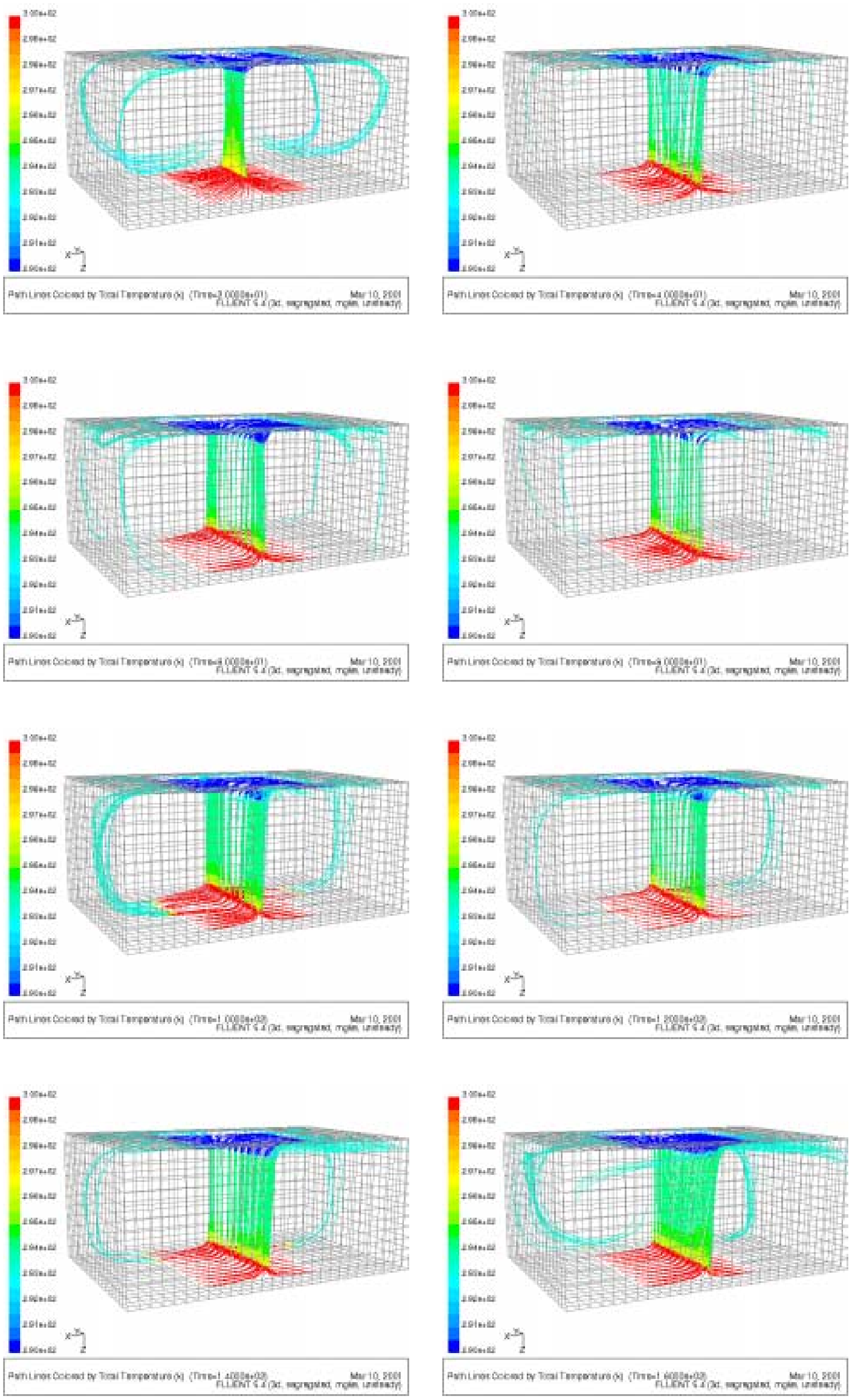

Fig. 8. Simulation results 20-160 seconds (20 seconds interleave).

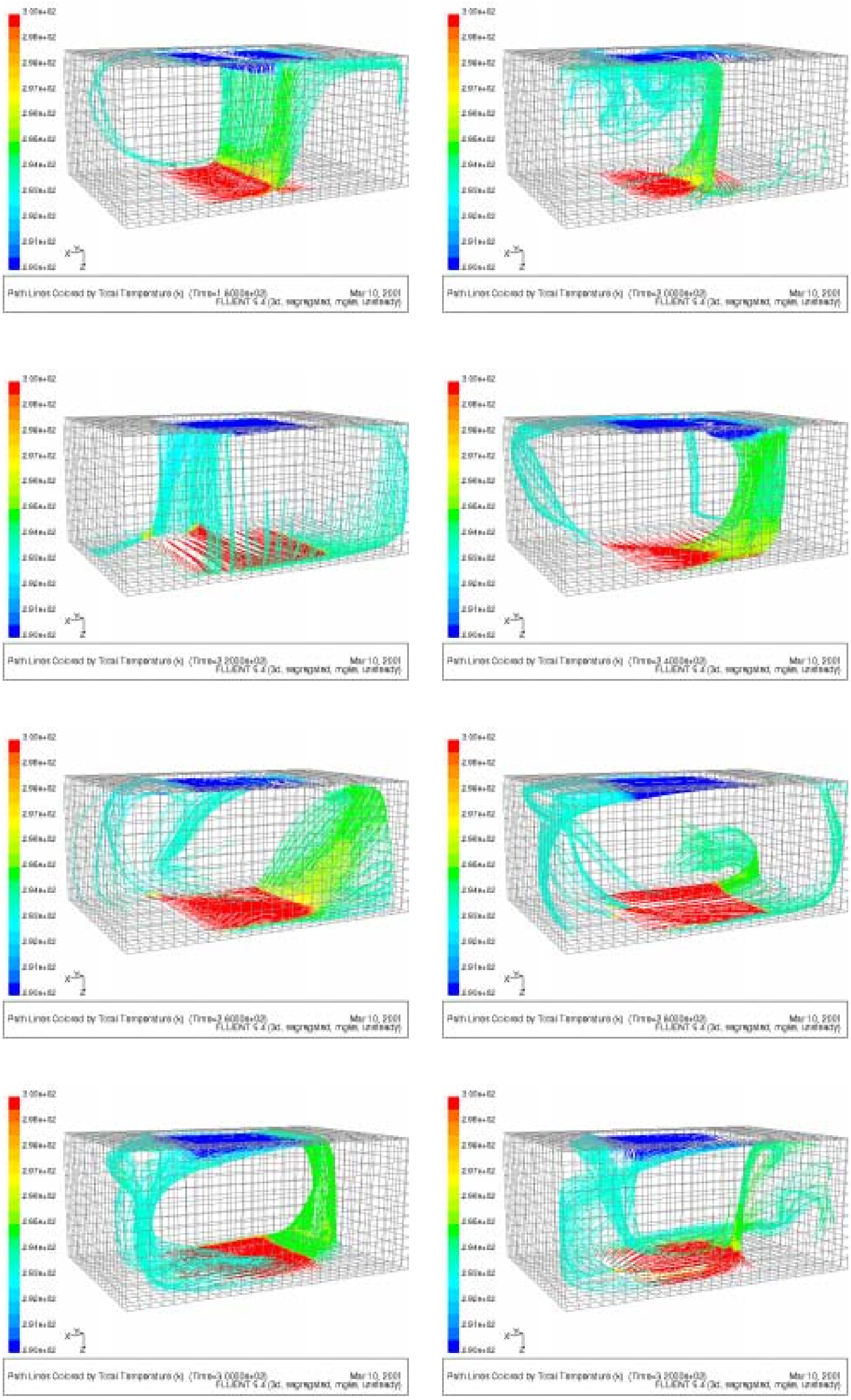

Fig. 9. Simulation results 180-320 seconds (20 seconds interleave).