Peter V. Nielsen, Li Rong and Inés Olmedo

The IEA Annex 20 Two-Dimensional Benchmark Test for CFD Predictions

ISBN 978-975-6907-14-6

Clima 2010, 10th REHVA World Congress

The IEA Annex 20 Two-Dimensional Benchmark Test for CFD Predictions

Peter V. Nielsen[1], Li Rong[1] and Inés Olmedo[2]

[1]Aalborg University, Denmark
[2]Córdoba University, Spain

Corresponding email: pvn@civil.aau.dk

SUMMARY

This paper describes a benchmark test which can be used for tests of CFD predictions of room air distribution. The benchmark describes a box-like room with a supply slot along the side wall. Laser-Doppler measurements and hot-wire measurements are given for comparison with the obtained CFD predictions both for isothermal flow and for nonisothermal flow. The benchmark is defined on a web page, which also shows about 50 different benchmark tests with studies of e.g. grid dependence, numerical schemes, different source codes, different turbulence models, RANS or LES, different turbulence levels in a supply opening, study of local emission and study of airborne chemical reactions. Therefore the web page is also a collection of information which describes the importance of the different elements of a CFD procedure.

The benchmark is originally developed for test of two-dimensional flow, but the paper also shows results for three-dimensional steady flow.

INTRODUCTION

The IEA Annex 20 two-dimensional test case was developed by the participants of the Annex 20 work in 1990 as a benchmark [1]. Figure 1 shows the dimensions of the geometry.

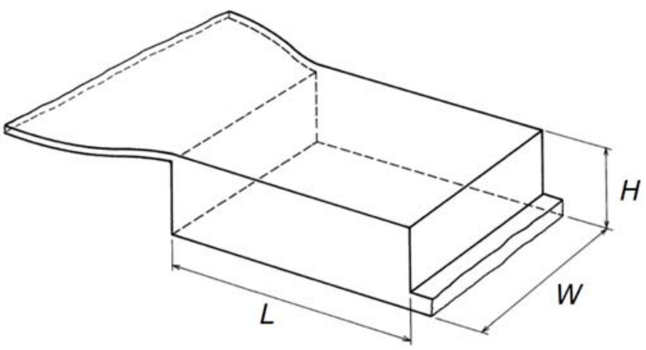


Figure 1. The two-dimensional benchmark test.

The air distribution in the benchmark corresponds to the situation in a room with a slot inlet. The dimensions and flow conditions are the following: L/H = 3.0, h/H = 0.056 and Re = 5000, where *L, H, h, Re* are length, height, slot height and Reynolds number, respectively. Velocities are measured in a model with $W/H = 1.0$ [2], and streaklines are photographed in a model with $W/H = 4.7$, see figure 2, [3].

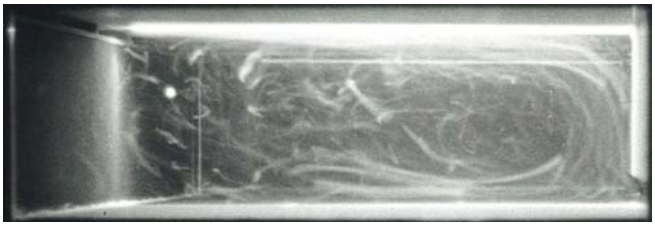


Figure 2. Streaklines obtained by photographing metaldehyde particle flow. The particles are illuminated in the symmetry plane of the room.

The velocity profiles for the inlet slot are also given in [1], and the following conditions should be used in most benchmark tests

$$k_o = 1.5\,(0.04 \cdot u_o)^2, \qquad \varepsilon_o = k_o^{1.5}/l_o \qquad \text{and} \qquad l_o = h/10 \qquad (1)$$

where k_o is turbulent kinetic energy in the supply opening, ε_o is the dissipation, and l_o is the length scale.

The benchmark test is divided into an isothermal case, 2D1, as described above, and a nonisothermal case, 2D2, where the temperature distribution is measured along a horizontal plane in the room with heat released along the whole floor, figure 3 [3]. A small Archimedes number and a high velocity ensure an almost isothermal flow, and the temperature distribution corresponds to a concentration distribution from an even distributed source along the floor. A nonisothermal test with high Archimedes number and restricted penetration of the plane wall jet into the room is also introduced [1]. Those measurements are based on Scwenke's experiments [4].

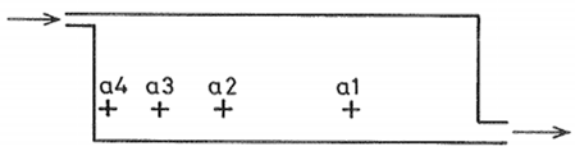


Figure 3. Position of measurement points for temperature distribution in the benchmark test.

APPLICATION OF THE BENCHMARK

The benchmark is located together with three other benchmarks on the web page:

www.cfd-benchmarks.com

The other benchmarks are about computer simulated manikins. They deal with mixing ventilation and displacement ventilation, thermal comfort and personalized ventilation.

About 50 different applications of the IEA Annex 20 two-dimensional benchmark (made between 1974 and now) can be found on the web page. This gives the possibility to compare a prediction with other authors' results (other turbulence models, other numerical schemes, different inlet conditions, etc.). The basic knowledge behind the benchmark test has thus been expanded during the years and made the benchmark test more useful. A few examples are discussed in the following sections.

Early predictions

Figure 4 shows some early two-dimensional predictions based on the vorticity-stream function equations, ω-ψ, and on the RANS equations, [3] and [5] as well as predictions based on RANS equations with a one equation turbulence model, [6].

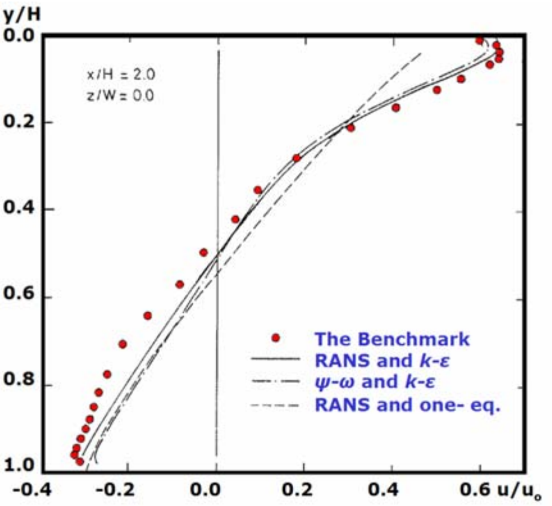


Figure 4. Early predictions of the velocity profile at x/H = 2.0 with different sets of transport equations, and different turbulence models.

Different numerical schemes

Figure 5 shows a set of predictions based on a low Reynolds number k-ε model (the LRN model), [7]. The results are compared at two different vertical sections. The recirculating flow is well predicted in the simulations, although the magnitude of the counter flow is slightly underestimated. The maximum velocity in the occupied zone is accurate with respect to both location and magnitude.

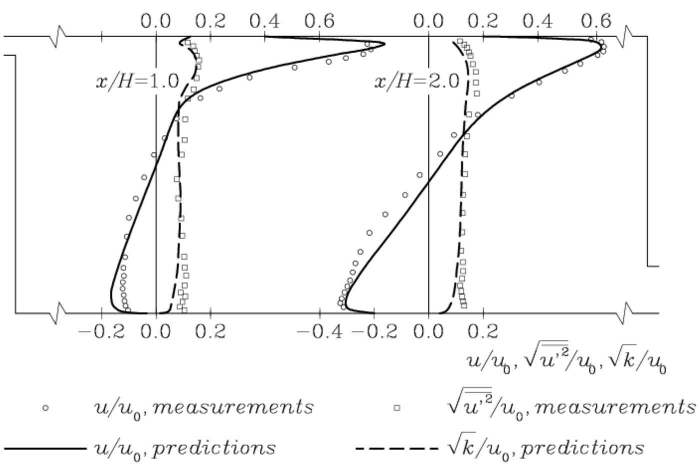


Figure 5. Velocity profiles and turbulent kinetic energy predicted by a LRN model.

It is possible to compare the measured RMS value $\sqrt{u'^2}$ with the square root of the turbulent kinetic energy k because the flow can be considered as a flow with wall jet profiles ($\sqrt{k}$ ~ 1.1 $\sqrt{u'^2}$). The figure shows that the predicted level of turbulence is in good agreement with the measurements.

Large Eddy Simulations, LES, can be used to obtain detailed information on turbulence, which can not be achieved by the traditional time-averaged turbulence models. The turbulence is predicted in details, and it is possible to make a direct prediction of $\sqrt{u'^2}$. Figure 6 shows some early predictions by Davidson and Nielsen [8]. It was difficult to get sufficient integration time for the calculation of u and $\sqrt{u'^2}$.

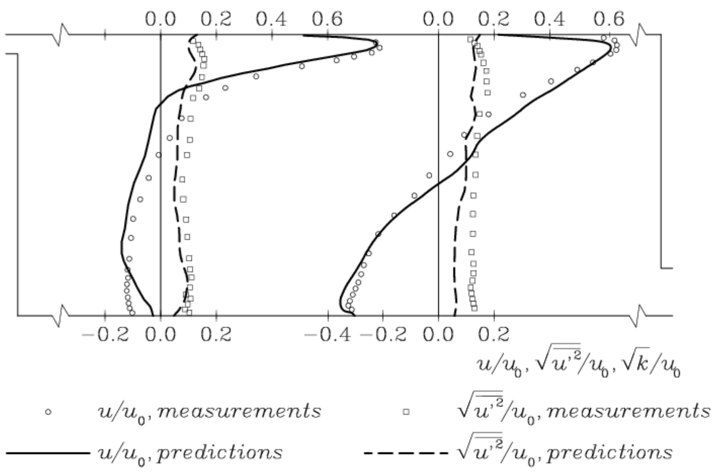


Figure 6. Time-averaged velocity and turbulence intensity in the symmetry plane predicted by LES.

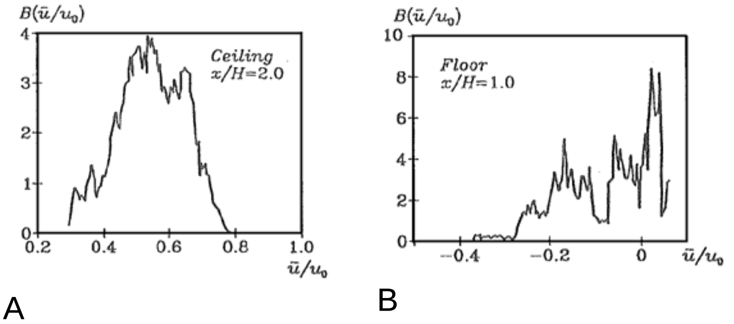


Figure 7. Probability density function of $\overline{u}/u_o$ close to the ceiling at x/H = 2.0, and close to the floor at x/H = 1.0. See [8].

The probability density function of $\overline{u}/u_o$ close to the ceiling at x/H = 2.0 has a well-defined mean velocity, and the velocity fluctuates around the mean velocity in a symmetrical distribution, figure 7A, while the flow close to the floor at x/H = 1.0 is asymmetric (has a skewness) with a probability of both a negative and a positive velocity, figure 7B.

Different turbulence models

Two-dimensional steady state predictions with different turbulence models indicate some differences, especially in two of the corners as shown in figure 8, [9]. The grid is identical in all four cases, which means that the difference in the flow can be ascribed to the turbulence models used to predict the two-dimensional flow. The predictions with the SST model show especially a large recirculating flow in the occupied zone below the supply slot $0 < x/H < 1.5$. Similar results have been obtained by Voigt [10]. It is not possible to fully exclude this flow pattern from the observation of the streaklines shown by photos as in figure 2, but it does not corresponds to the Laser-Doppler measurements in the benchmark.

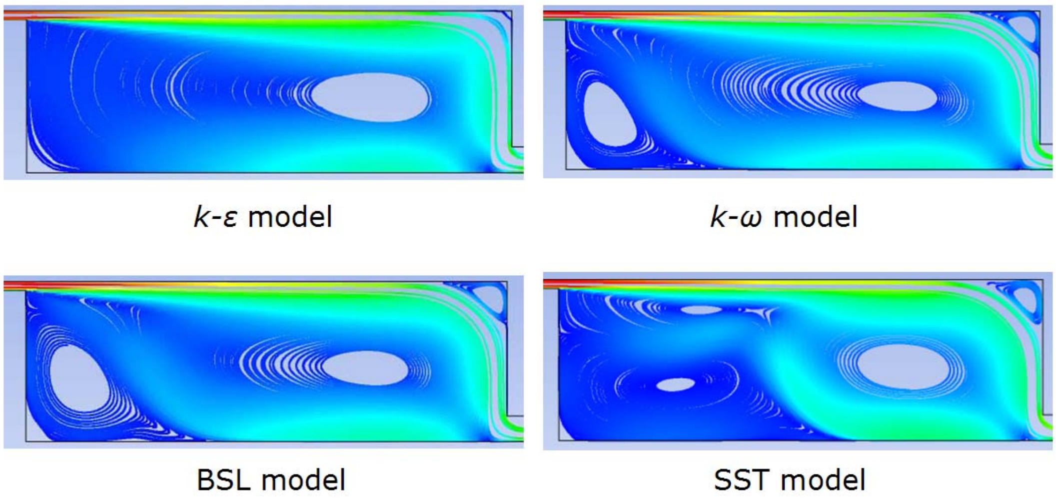
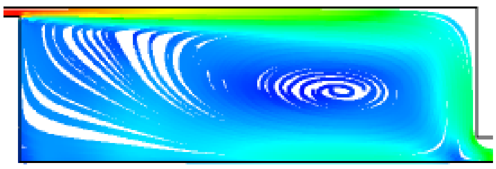


Figure 8. Two-dimensional isothermal and steady state simulations of the Annex 20 2D benchmark test. Four different turbulence models are used together with the RANS equations.

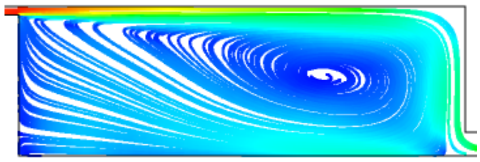
THREE-DIMENSIONAL STEADY STATE PREDICTIONS

The benchmark was originally selected for the simulation of steady two-dimensional flows for the IEA Annex 20 work back in 1990. The flow in the benchmark was considered to be two-dimensional and steady, but three-dimensional unsteady flows could, at least in principle, occur. LES predictions indicate an asymmetric probability density function at the floor at x/H = 1.0 with a probability of both negative and positive velocities, figure 7B, and measurements in rooms with $L/H \geq 4.0$ and $W/H = 4.7$ do also show unsteady flow at the floor for $0 < x/H < 1.0$, [3].

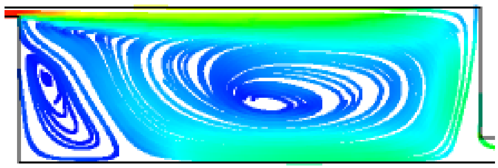
The predictions have been repeated as steady three-dimensional flow in the benchmark with W/H = 1.0 to examine if this geometry gives results which are more identical to each other when different turbulence models are used. Figure 9 shows the flow in the symmetry plane of the room for the *k-ε* model and the SST model. The three-dimensional prediction with the *k-ε* model looks similar to the two-dimensional predictions in figure 8, while the three-dimensional prediction with the SST model is slightly different from the two-dimensional version in figure 8. The large recirculating flow in the occupied zone below the supply slot is reduced compared to the flow in the two-dimensional case. This change in the flow towards the flow predicted by the other turbulence models could be a result of having three-dimensional flow, but it could not be fully excluded that the use of a different grid might have some influence. (The two-dimensional predictions, figure 8, are made in a structured grid, while the three-dimensional predictions in figure 9 and 10 are made in an unstructured grid).



k-ε model



k-ω model

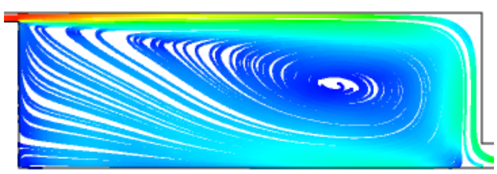


SST model

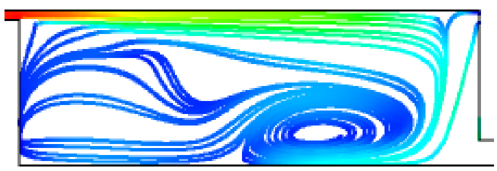
Figure 9. Three-dimensional isothermal and steady state simulations of the Annex 20 2D benchmark test. Three different turbulence models are used together with the RANS equations.

The difference between a two-dimensional *k-ω* prediction and a three-dimensional *k-ω* prediction of the flow can be explained by looking at the flow in the occupied zone. Figure 10 show the three-dimensional steady flow predicted by the *k-ω* model. The flow in the middle plane is very different from the flow close to the side wall. A horizontal section in the occupied zone shows how the recirculating flow in the middle plane has a higher velocity than the flow close to the side walls. The flow in the middle of the room therefore creates two horizontal circulations in the region below the supply opening, which ensures a fully vertical recirculation in the middle plane. The flow is unsymmetric in the room, although the room is symmetric.

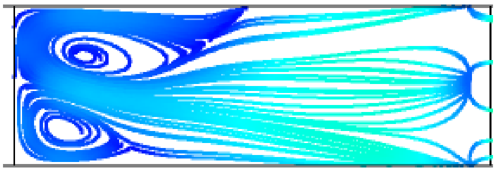
The measured flow in the benchmark, [1], does not show this large difference between the velocity distribution in the middle plane and the velocity distribution at a distance of 0.1*W* from the side wall.



Flow in middle plane



Flow close to side wall (0.1*W* from side wall)



Flow in horizontal plane in the occupied zone (0.5·h above floor)

Figure 10. Three-dimensional isothermal and steady state simulations of the Annex 20 2D benchmark test with a k-ω turbulence model solved together with the RANS equations.

A study by Susin et al. [11] using three-dimensional flow concludes that the k-ε model performed best compared to the RNG k-ε model and the k-ω model, and that the most significant differences between calculated and experimental velocity profiles were also found along the floor.

CONCLUSIONS

The two-dimensional benchmark test, called the "IEA 2D test case" was defined in 1990 to be used in the IEA Annex 20 work.

The benchmark is defined on a web page, so it can also be used in other CFD work. There have been a very large number of papers that use this geometry, and about 50 works are reported on the web page. This gives the possibility to compare a prediction with other authors' results (other turbulence models, other numerical schemes, different inlet conditions, etc.). The basic knowledge behind the benchmark test has thus been expanded during the years and made the benchmark test more useful.

Predictions by different turbulence models show some variation in the obtained streamline pattern. Especially the k-ω SST model gives a flow different from the flow obtained by the other models.

The benchmark was originally selected for the simulation of steady two-dimensional flow, and the flow in the benchmark was considered to be two-dimensional and steady, but three-dimensional unsteady flow could, at least in principle, occur.

Predictions by a steady state three-dimensional flow show streamline pattern similar to the two-dimensional predictions, but the SST model introduces a change in the flow which is slightly towards the flow predicted by the other turbulence models.

An analysis of the three-dimensional flow predicted by the k-ω model shows a large difference of the flow in the middle plane compared to the flow in a plane close to the side wall. This difference could not be confirmed by the measurements in the benchmark.

REFERENCES

1. Nielsen, P V. 1990. Specification of a Two-Dimensional Test Case, Aalborg University, IEA Annex 20: Air Flow Patterns within Buildings.

2. Restivo, A M. 1979. Turbulent Flow in Ventilated Rooms, PhD thesis, Imperial College, London.

3. Nielsen, P V. 1974. Strømningsforhold i luftkonditionerede lokaler, PhD thesis, Technical University of Denmark. (English translation: Flow in Air Conditioned Rooms, 1976).

4. Schwenke, H. 1975. Über das Verhalten ebener horizontaler Zuluftstrahlen im begrenzten Raum, Luft- und Kältetechnik, Nr. 5, Dresden.

5. Nielsen, P V, Restivo, A, and Whitelaw, J H. 1978. The Velocity Characteristic of Ventilated Rooms, Journal of Fluid Engineering, Vol. 100, pp. 291-298.

6. Davidson, L and Olsson, E. 1987. Calculation of Some Parabolic and Elliptic Flows Using a New One-Equation Turbulence Model, Proc. 5th. International Conference on Numerical Methods in Laminar and Turbulent Flow, Montreal.

7. Skovgaard, M and Nielsen, P V. 1991. Simulation of Simple Test Case, Case 2D1, Internal report for the International Energy Agency, Annex 20, Aalborg University, ISSN 0902-7513 R9131.

8. Davidson, L and Nielsen, P V. 1996. Large Eddy Simulation of the Flow in a Three-Dimensional Ventilated Room, Proc. of Roomvent'96, Yokohama.

9. Rong, L and Nielsen, P V. 2008. Simulation with Different Turbulence Models in an Annex 20 Room Benchmark Test Using Ansys CFX 11.0, DCE Technical Report No. 46, Aalborg University.

10. Voigt, L K. 2000. Comparison of Turbulence Models for Numerical Calculation of Airflow in an Annex 20 Room, Department of Energy Engineering, Technical University of Denmark, ET-AFM 2000-01, ISBN 87-7475-225-1.

11. Susin, R M, Lindner, G A, Mariani, V C, and Mendonca, K C. 2009. Evaluating the Influence of the Width of Inlet Slot on the Prediction of Indoor Airflow: Comparison with Experimental Data, Building and Environment, 44, pp. 971-986.