PURPOSE-LED
PUBLISHING™

PAPER • OPEN ACCESS

A benchmark test for room air distribution: the backward facing step flow

View the article online for updates and enhancements.

A benchmark test for room air distribution: the backward facing step flow

P V Nielsen[1], C Zhang[1], C T Kjær[1], D Leiria[1], H Nørholm[1], T Ramstad[1], A Rovithakis[1] and R L Jensen[1]

[1] Aalborg University, Thomas Manns Vej 23, DK-9220 Aalborg Ø, Denmark
* pvn@civil.aau.dk

Abstract. This paper concerns the use of Computational Fluid Dynamics (CFD) for the prediction of room air movement and provide a guide on selecting of the proper turbulence model. The benchmark which developed here for the first time is a backward-facing step flow problem. The measurements is performed in a small-scale model and the velocity filed is measured by Particle Image Velocimetry (PIV). The measurements focus on transitional flow and fully developed turbulent flow at isothermal condition. The PIV measurements and different CFD predictions are compared. The CFD predictions are generally slightly diverse, and this is, among other things, the result of using different turbulence models as well as using different software codes, grid distribution and boundary values etc. The k-ε family of turbulence models is an option for fully developed flow, and the k-ω family does work for transitional low turbulent flow in this backward-facing step flow.

1. Introduction

The indoor environment community has adopted computational fluid dynamics (CFD) as a useful tool for the prediction of air movement in ventilated spaces. Researchers have used the method for many years as a research tool, see e.g. Nielsen [1] and [2]. Now, it is used routinely in civil engineering when designing a large or complicated air distribution system, Nielsen [3].

The airflow is described mathematically by a set of coupled differential equations, known as the Navier-Stokes equations. These equations are reformulated into a high number of ordinary equations and solved by a numerical method. It is necessary to add additional (partly empirical) equations for the description of the turbulence in the flow, and different turbulence models are introduced for different types of flow elements, see e.g. Zhang et al. [4]. Therefore, a CFD prediction is dependent on many parameters such as the selected turbulence model, the software scheme, the grid number and grid distribution, the selected boundary conditions etc.

A way to select the right CFD procedure is to test the turbulence model and all other elements in geometry with a measured air distribution, which has some similarities to the room geometry for which we want to solve the flow. In this paper, we will make measurements in a geometry, which has been used earlier in two different workshops, namely in a workshop on the ISHVAC-COBEE conference in July 2015, see Peng et al. [5] and in a workshop on the Indoor Air Conference 2016, see van Hooff et al. [6]. It is a simple supply air flow scenario in building ventilation, similar to the backward-facing step flow. The turbulence models did very strongly influence the results of the work. Both workshops were designed as blind tests in which the measurements were unknown for the participants so it was not possible to decide the quality level of the different predictions. The aim of this paper is, therefore to make measurements (a benchmark tests) in the above-mentioned geometry.

2. Turbulence models for room air distribution

The flow in a room with e.g. mixing ventilation is a combination of many different types of flow and flow elements.
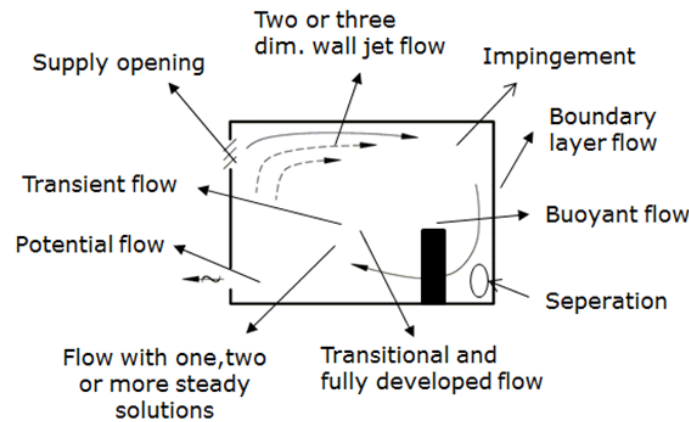
Figure 1. Examples of different types of flow in a room with mixing ventilation.

Some of the flow or flow elements in figure 1 can be solved separately by CFD. Different turbulence models are in fact optimized so as to solve the single flow elements, and it is therefore not obvious which turbulence model should be used in the case of a combined flow like the one in figure 1. Table 1 shows the turbulence models optimal for some of the flow elements.

Table 1. Flow elements and turbulence models

Flow Element	Supply opening [7]	Two or three dim. wall jet flow [4], [7], [8], [9], [10]	Buoyant flow [4]	Transitional and fully developed recirculating flow [4], [7], [8]
Turbulence Models	LRN	k-ε, k-ω, BSL, v^2-f, RSM	SST	k-ε family LRN, v^2-f

The air movement in the **supply opening** can be a flow with a low turbulence level due to design details as contraction etc. although it enters into a room with a high turbulence level, [7]. The low Reynolds number models (LRN) are an option for the direct study of the flow in the diffuser. The k-ε family, k-ω and BSL models work well for **two-dimensional wall jet flow**, [7] and [8]. For **three-dimensional flow** in a wall jet, there is a difference in growth rates parallel to ceiling and perpendicular to ceiling. This effect is handled by the Reynolds stress model (RSM) and partly by a v^2-f model, [9], [10]. The effect is especially important in elongated rooms (and tunnels) but not so important in normal short rooms, [9]. The SST k-ω worked well for strong **buoyant flow** in, for example, smoke management and rooms with thermal loads, [4]. Finally, the k-ε model and especially the RNG k-ε and the v^2-f models have the best overall performance compared to other models in **fully developed recirculating flow** [4]. Predictions with standard k-ε, realizable k-ε and RNG k-ε model in a livestock building room with a complicated geometry including slatted floor show that all three models in this situation produced acceptably, but slightly different solutions at the isothermal flow, Rong et al. [11]. **In transitional flow**, the low Reynolds number k-ε model (LRN) is a possibility, but the workshops at the ISHVAC-COBEE conference, [5] and the workshop at the Indoor air Conference 2016, [6] show that it is a difficult situation to handle with CFD predictions.

It is obviously a difficult situation to decide a single turbulence model for a general solution of the flow in a room like the one in figure 1. A possible procedure therefore is to test the different models in geometry similar to the room. The ISHVAC-COBEE workshop geometry is an elongated

room with a ventilation opening in one end wall (backward facing step flow), and the flow in this model is addressed the following.

3. The backward-facing step flow – measurements
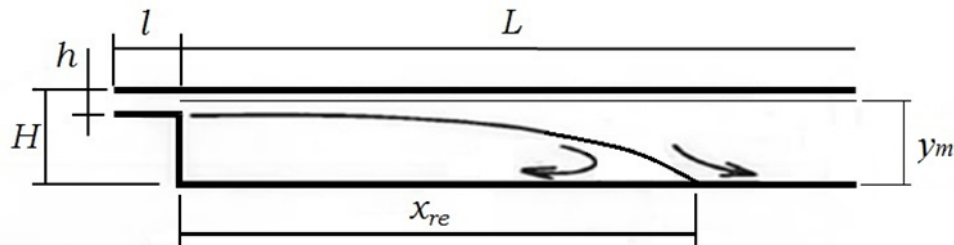
Figure 2. ISHVAC-COBEE model

Figure 2 shows the model of the backward-facing step flow. The model has the following dimensions: $h/H = 1/5 = 0.2$, $l/H = 4$, width $W = 2H$. Predictions are carried out in isothermal conditions with the Reynolds numbers, $0 < Re \leq 10,000$. The Re number is based on the inlet velocity and slot dimension. The length x_{re} is from the end wall to the location where the reattached flow is separated into a flow back to entrainment into the wall jet and a forward flow towards the exit (i.e. reattachment point). The length x_{re} is referred to as the penetration length of the supply jet, and it was selected as one of the parameters in the ISHVAC-COBEE workshop.
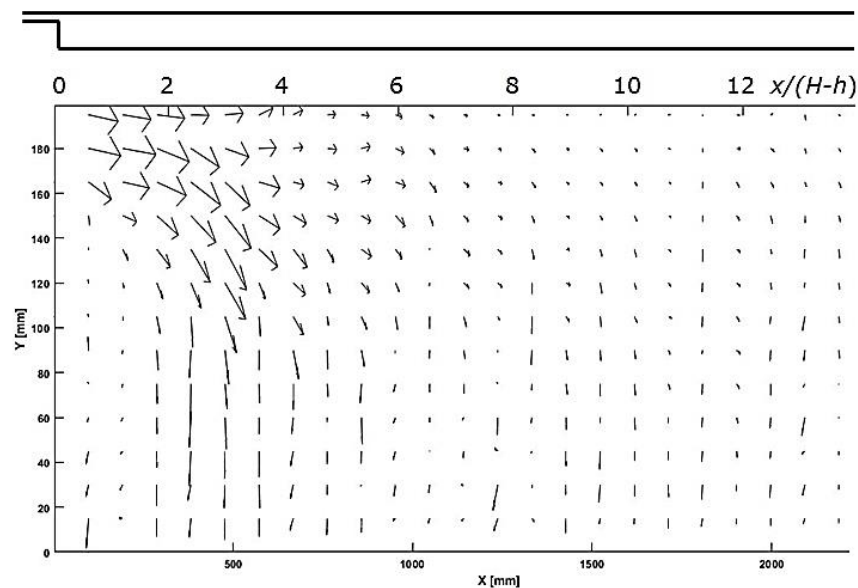

Figure 3. PIV measurements of the velocity distribution in the centre plane of the flow. $Re = 4000$.

A small scaled model made by acrylic is used in the measurement, with h=0.04 m and L=3 m, and the other dimensions are based on geometrical similarity as Figure 2. In order to validate isothermal condition assumption, six thermocouples are used to measure temperatures at model inlet, exhaust, surfaces and surrounding air. The velocity distribution in the model is measured by Particle Image Velocimetry (PIV). The 2D-PIV system consists of a double cavity Nd:YAG laser with a wavelength of 532 nm, and a CCD camera with a resolution of 2048*2048 pixels.

Figure 3 is an example of the velocity distribution in the centre plane of the flow with Re number of 4000 (the y-axis is extended compared to the x-axis). The measurements do not give clear indication on the separation flow at the floor region and it is very difficult to locate the distance x_{re} in the model. It is more efficient to work with the measurements in the high velocity area of the flow. Therefore, we select the velocity distribution along a horizontal line in the height y_m as datasets for

comparison with CFD predictions when the measurements are used as a benchmark, see figures 2 and 4.
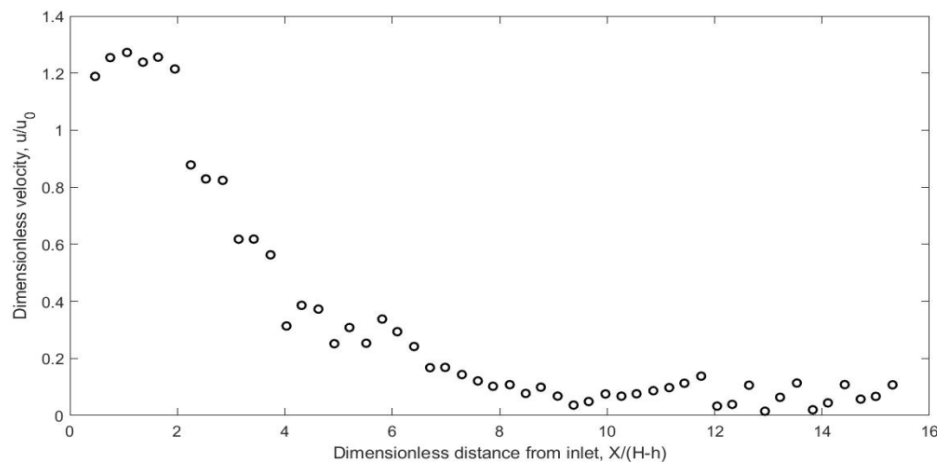
**Figure 4.** Velocity distribution in the height of $y_m/(H-h)$ = 1.19. Re = 4000.

Measurements are made for $y_m/(H-h)$ equal to 0.34, 0.81 and 1.19 and for the Reynolds numbers Re = 500, 1000, 4000, and 10,000.

4. The backward-facing step flow, CFD predictions and comparison with measurements

Four different sets of CFD predictions are made for the backward-facing step flow where only the turbulence model is changed between the *RNG k-ε* model, the realisable *k-ε* model, the *BSL k-ω* model and the *SST k-ω* model. Other setup as mesh, boundary conditions and solving algorithms are unchanged. All the turbulence models utilized the near wall model approach, where the *k-ω* models are sufficient by themselves toward the edges and walls, while *k-ε* models needs modification near the wall. The Enhanced Wall Treatment is used since the y^+-value is below 5 near the floor, Predictions are made in both a two-dimensional and a three-dimensional model. There are differences in the flow, so the conclusion is that the flow is three-dimensional.

Figure 5 and figure 6 show that it could be relevant to change turbulence model according to the supply velocity. The *k-ω* family seems to be an option for low turbulent flow (*Re* = 500) while the *k-ε* family looks as the best option for fully developed turbulent flow (*Re* = 4,000).

Several turbulence models can be relevant according to the different flow elements present in a room (see figure 1), and now it is also seen that the flow rate to the room can have an influence on the selection of the turbulence model.

The ISHVAC-COBEE workshop [5] and the Indoor Air 2016 workshop [6] show that CFD prediction is dependent on more parameters than the selected turbulence model. The software scheme, the order of accuracy, the grid number and grid distribution, the selected boundary conditions, the experience of the user, etc. and all the combinations are all parameters that can generate spreading of the results. We can illustrate this problem by comparing predictions from the ISHVAC-COBEE workshop with the predictions in this paper and earlier predictions at AAU.
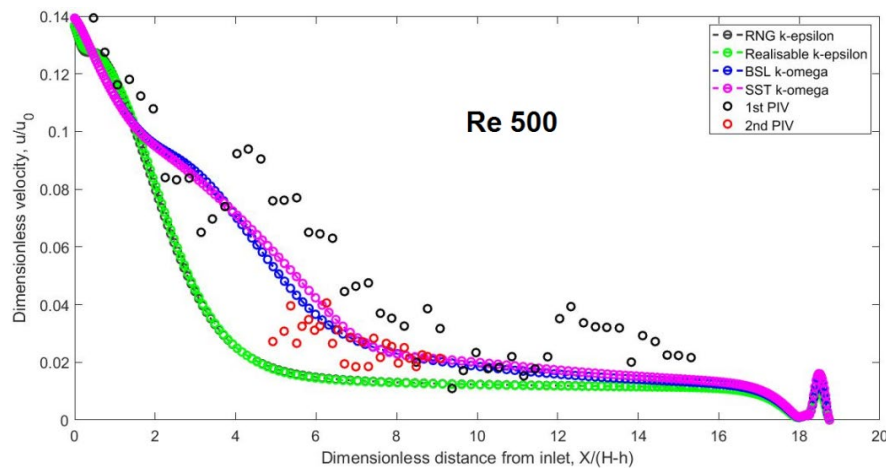
Figure 5. Comparison between measurements and predictions in the height $y_m/(H-h)$ = 1.19. 3D flow and Reynolds number equal to 500.
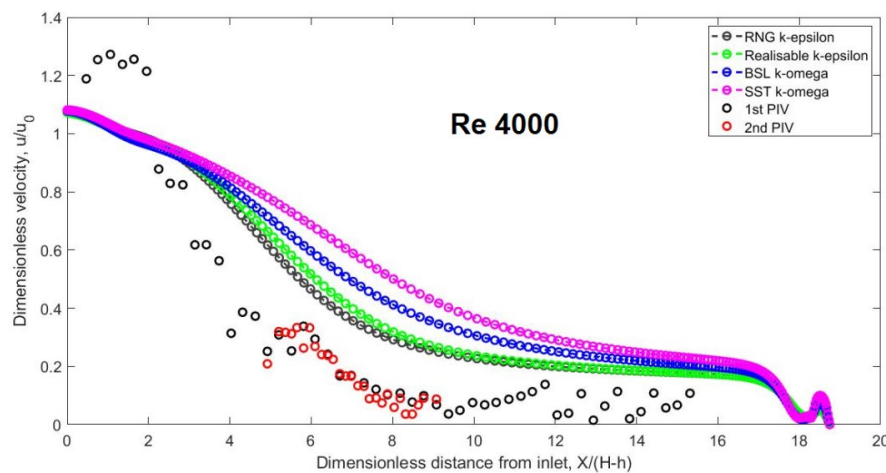
Figure 6. Comparison between measurements and predictions in the height $y_m/(H-h)$ = 1.19. 3D flow and Reynolds number equal to 4000.
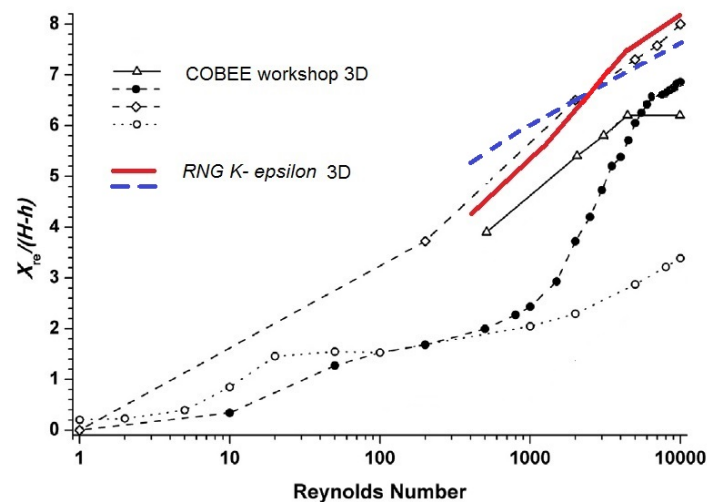
Figure 7. CFD predictions of the 3 D flow in the backward facing step model. All predictions are made with the *RNG k-ε* model. The present prediction is illustrated by a thick (red) curve and an earlier AAU prediction is shown by a dotted thick (blue) curve, [12].

All the predictions in figure 7 are made by the *RNG k-ε* turbulence model in a three-dimensional flow. They do not arrive at the same solution and they are influenced by some or all the different parameters mentioned above.

5. Conclusions
A benchmark is developed by making PIV measurements on the ISHVAC-COBEE model.

It is discussed that different turbulence models can solve the local flow elements in room air distribution optimally. It is therefore not so obvious which turbulence model should be used in the case of the combined flow in a room, but a number of promising models are introduced.

Results from benchmark test indicate that the *k-ω* family seems to be an option for low turbulent flow while the *k-ε* family looks as the best option for fully developed turbulent flow in the ISHVAC-COBEE model.

References

[1]   Nielsen PV 1973 Berechnung der Luftbewegung in einem zwangsbelüfteten Raum, *Gesundheits-Ingenieur*, 94, pp. 299-302.
[2]   Nielsen PV 1975 Prediction of Air Flow and Comfort in Air Conditioned Spaces. *ASHRAE Transactions 1975*, Vol. 81, Part II.
[3]   Nielsen PV 2015 Fifty years of CFD for room air distribution, *Building and Environment 91* pp78-90
[4]   Zhang Z, Zhang W, Zhai Z and Chen Q (2007) Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-2: comparison with experimental data from literature, *HVAC&R Research*, 13(6).
[5]   Peng L, Nielsen PV, Wang X, Sadrizadeh S, Liu L, Li Y. Possible user-dependent CFD predictions of transitional flow in building ventilation. *Build Environ*. 2016 99:130-141.
[6]   van Hooff T, Nielsen PV, Li Y. Computational fluid dynamics predictions of non-isothermal ventilation flow—How can the user factor be minimized? *Indoor Air*. 2018;28(6):866-880. https://doi.org/10.1111/ina.12492
[7]   Le Dreau J Heiselberg P, Nielsen PV, Simulation with Different Turbulence Models in an Annex 20 Benchmark Test using Star-CCM+. Aalborg : Department of Civil Engineering, Aalborg University, 2012. 22 s. (DCE Technical reports; Nr. 147).
[8]   Rong L, Nielsen PV, Simulation with Different Turbulence Models in an Annex 20 Room Benchmark Test Using Ansys CFX 11.0. Aalborg : Department of Civil Engineering, Aalborg University, 2008. 16 s. (DCE Technical reports; Nr. 46).
[9]   Schälin A, Nielsen PV, Impact of turbulence anisotropy near walls in room airflow. I: *Indoor Air*. 2004 ; Bind 14, Nr. 3. s. 159-168.
[10]   Davidson L, Nielsen PV, Sveningsson A. Modifications of the V2 Model for Computing the Flow in a 3D Wall Jet. I Proceedings of the International Symposium on Turbulence, *Heat and Mass Transfer*, October 12 - 17, 2003, Antalya, Turkey. 2003
[11]   Rong L, Nielsen PV, Bjerg B and Zhang G, Summary of best guidelines and validation of CFD modeling in livestock buildings to ensure prediction quality, *Computers and Electronics in Agriculture*, 121, 2016, 180–190
[12]   Andersen KH, Knudsen SS, Mikalainis M and Nikolaisson IT, Private communication, 2018, Aalborg University.