The Development of a Benchmark for Room Air Distribution

the backward facing step flow

Nielsen, Peter V.; Zhang, Chen; Bugenings, Laura Annabella; Schaffer, Markus

Published in:
15th ROOMVENT Virtual Conference, 2021

Publication date:
2021

Document Version
Publisher's PDF, also known as Version of record

THE DEVELOPMENT OF A BENCHMARK FOR ROOM AIR DISTRIBUTION: THE BACKWARD FACING STEP FLOW

Peter V. Nielsen, Chen Zhang, Laura Annabella Bugenings, Markus Schaffer
Aalborg University, Aalborg, Denmark

Abstract

A CFD workshop on a geometry that can be typical for deep buildings and tunnels took place in 2015 at the ISHVAC-COBEE 2015 conference. The fluid problem investigated in the workshop is known as the backward facing step flow. Members of the workshop made Computational Fluid dynamics (CFD) predictions of the air distribution without having measurements for comparison. Their results show a significant diversity, among other things due to the chosen turbulence models.

A benchmark for the backward-facing step flow is introduced to validate the numerical predictions as well as to assess the influence of different turbulence models, wall functions and other important elements of the software.

Keywords: CFD, room air distribution, benchmark, laminar flow, transitional flow, turbulent flow

Introduction

For many years, the indoor environment community has used CFD as a tool for the prediction of air movement in ventilated spaces; see e.g. Nielsen (2015). In CFD the airflow is described by a set of coupled differential equations known as the Navier-Stokes equations. To solve the room air flow in a practical economical way it is necessary to extend these equations by extra differential equations expressing the turbulence in the flow, the so-called turbulence models. Turbulence models are optimized for the different types of flow, which are present in a room. These are for example two or three-dimensional jets, free jets or wall jets, impingement flows, boundary layer flows, buoyant flows, separations, transitional and fully developed turbulent flows, transient flows and potential flows; see e.g. Nielsen et al. (2019).

The airflow in a room can therefore be a combination of many different types of flows or flow elements. As a result, it is obviously difficult to select a specific turbulence model for a general valid solution of the flow in a room. A possible procedure, therefore, is to test different turbulence models in geometries similar to the actual room, which is to be studied. In this paper, we work with a geometry, which has been used earlier in two different workshops, on the ISHVAC-COBEE conference in July 2015; see Peng et al (2016) and on the Indoor Air conference 2016; see van Hoof et al. (2018). This flow scenario is different from flows in ventilated rooms of regular size with a short section, but it is relevant for elongated industrial buildings and deep tunnels

.

The benchmark

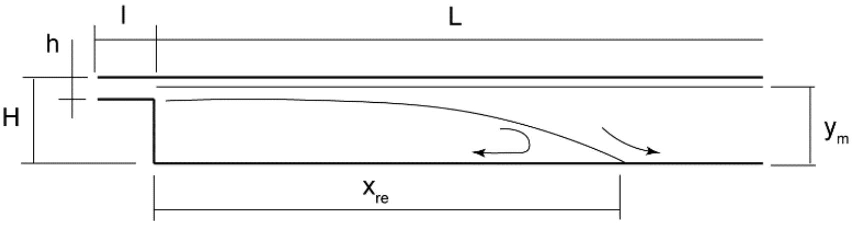

Figure 1. *The backward-facing step flow used in the ISHVAC-COBEE workshop.*

Figure 1 shows the model of the backward-facing step flow. The model has the following dimensions: h/H = 0.2, l/H = 4, width W = $2H$. The *Re* number is based on the inlet velocity and slot height h.

The separation length x_{re} is the distance from the vertical left wall to the location where the reattachment flow is separated into a flow back to entrainment into the wall jet and a forward flow towards the exit (i.e. reattachment point). Another variable used in the benchmark is the velocity u along a horizontal line in the height of y_m above the floor, see Figure 1. The experiments are more detailed described by Zhang et al. (2020).

Laminar, transitional and fully developed isothermal turbulent flow

The benchmark covers fully developed turbulent flow, transitional flow and laminar flow. Fully developed flow and transitional flow are important in room-air distribution while laminar flow only take place in near-wall flow, and not in the main part of the room air flow.

Figure 2 shows the prediction of laminar flow based on Navier-Stokes equations and transitional turbulent flow based on RANS equations with a *Realizable k-ε* model and with a *SST k-ω* model respectively.

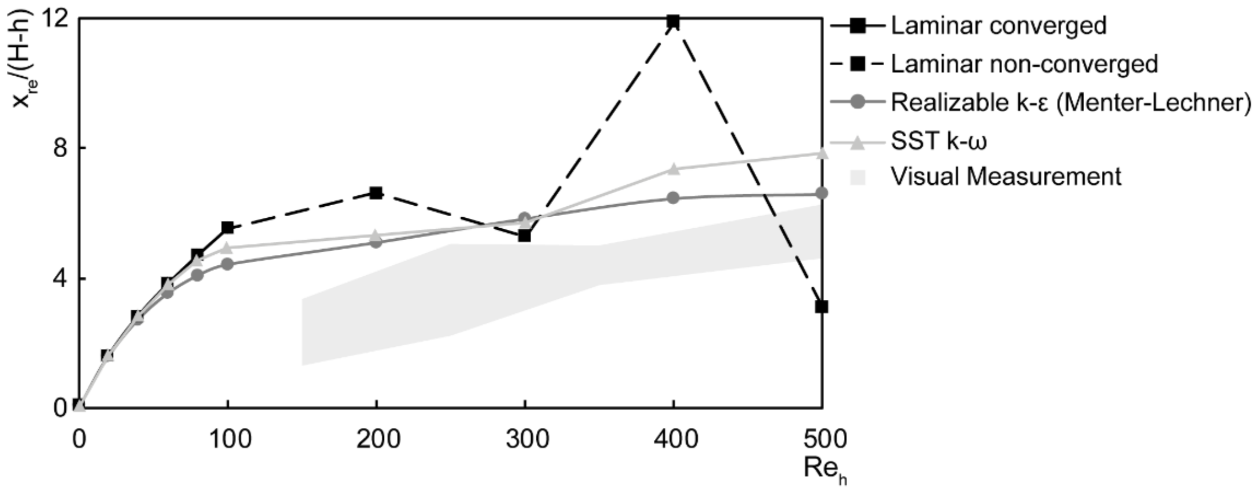

Figure 2. *Separation length versus Reynolds number. Laminar prediction and RANS predictions with SST and k-ε turbulence models. Comparison with smoke experiments.*

The laminar predictions up to a Reynolds number of 100 show converged solutions. The results for higher Reynolds numbers are non-converged with too high residuals (~1e-3) which indicate that the used direct numerical simulation is not feasible for solving higher Reynolds number flow. This however is not connected to the position of the critical Reynolds number (Re_c) where the transitions from laminar to turbulent flow takes place. The Re_c depends on details in the boundary conditions such as irregularities in the design of the air supply, presence of turbulence in the supply flow, difficulties with small temperature differences, vibrations etc.

Other measurements show a characteristic peak at the position of the transition, see e.g. Restivo (1979). A comparison with the smoke experiments made by Zhang et al. (2020) show that the flow in this model does not have a typical laminar flow even down to Re = 150. The smoke experiments show lower separation lengths than the laminar predictions in those areas, as it can be seen in Figure 2, probably due to irregularities in design of the air-supply and the presence of small temperature differences.

It is typical that both the *SST k-ω* and the *Realizable k-ε* model show solutions close to the laminar flow for small *Re* numbers (*Re* < 40) as the models generate a small turbulent viscosity μ_t compared to physical viscosity μ in the RANS equations, at the low *Re* number, see Weng et al. (2012).

The most RANS equations with low Reynolds number models represent both transitional flow and fully developed turbulent flow at high Reynolds number (5000 – 10000). The models in the low Reynolds number area are often just a modification of the turbulence model for fully developed turbulent flow.

CFD predictions and comparison with measurements

The measured velocity profile and the predicted ones are compared in the height of $y_m/(H-h)$ = 1.17, see Figure 3. Three CFD predictions are made with the *Realisable k-ε* model, the *SST k-ω* model and the Reynolds Stress Model (*RSM)*. Other setups such as mesh, boundary conditions and solving algorithms are unchanged. All the turbulence models utilized the near wall model approach, where the *SST* model is sufficient by it-selves toward the edges and walls, while *k-ε* and *RSM* models need modification near the wall. The Enhanced Wall Treatment is used since the *y*+-value is below 5 near all solid surfaces.

Figure 3 shows that *RSM* and the *Realizable k-ε model* are the relevant turbulence models to be used at the high Reynolds number of 4000 while other predictions show that the *SST k-ω* model is the most relevant at a Reynolds number of 400, see Nielsen et al (2019)

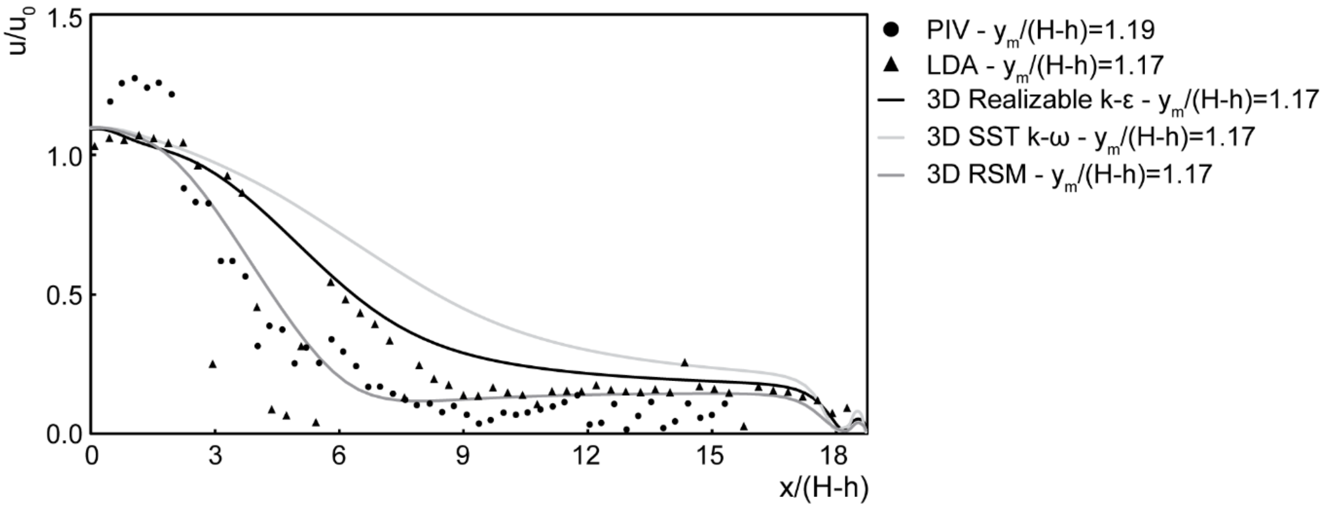

Figure 3. *Comparison between measurements and predictions in the height $y_m/(H-h)$. The predictions are made as three dimensional flow and Re = 4000.*

The *RSM* is an anisotropic turbulence model while *Realizable k-ε* and *SST k-ω* are isotropic turbulence models. It is known that *RSM* predictions with a wall-reflection term are preferable for flows in elongated ventilated sections with supply openings of small width because the anisotropic behavior of wall jet flow is predicted with correct growth rates (Schälin and Nielsen 2004). Figure 4A shows that the wall-reflection term will also have an effect in the backward-facing step flow. Predictions indicates that the wall jet velocity increases in the upper corners and move down the sidewalls, but it is discussed in literature that the wall-reflection term will not improve the prediction when the flow involves several surfaces, ANSYS (2019), Schälin and Nielsen (2004). Figure 4B show that a *RSM* without wall-reflection term show a solution similar to the predictions with isotropic turbulence models. This effect-has not been studied by measurements.

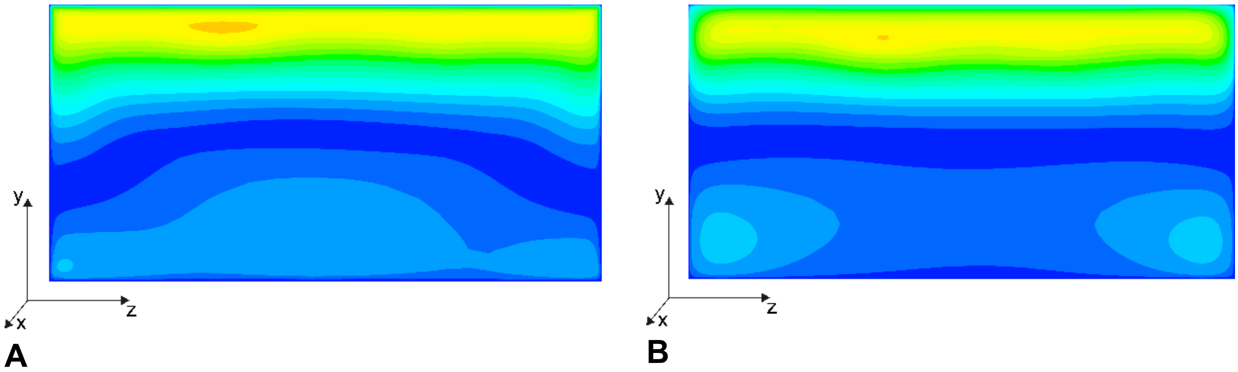

Figure 4. *Cross-section (H x W) of the flow at x/H = 1.5. A) RSM prediction with wall-reflection term modifying the growth rate of the wall jet. B) RSM prediction without wall reflection term.*

Conclusions

A benchmark for the backward-facing step flow is described in this study. The flow regime ranges from laminar to fully turbulent flow. CFD predictions with different turbulence models are validated by LDA, PIV and smoke test measurements. The validation indicates that the *Realizable k-ε* model is a good option as turbulence model for a fully developed turbulence flow (Re=4000). The results discussed in this paper will be presented on www.cfd-benchmarks.com.

References

ANSYS INC., 2019-last update, Ansys Fluent 19.1 *Theory Guide* [Homepage of ANSYS Inc.], [Online]. Available: https://ansyshelp.ansys.com, Retrieved: 20.02.2020

van Hooff, T, Nielsen, PV & Li, Y 2018, *Computational fluid dynamics predictions of non-isothermal ventilation flow—How can the user factor be minimized?*, Indoor Air, bind 28, nr. 6, s. 866-880. https://doi.org/10.1111/ina.12492

Nielsen, PV, 2015, *Fifty years of CFD for room air distribution*, Building and Environment, bind 91, no. September, s. 78–90. https://doi.org/10.1016/j.buildenv.2015.02.035

Nielsen, PV, Zhang, C, Kjær, C, Leiria, D, Nørholm, H, Ramstad, T & Rovithakis, A 2019, *A benchmark test for room air distribution: The backward facing step flow.* in Proceedings of IAQVEC2019 Conference: IOP Conference Series: Materials Science and Engineering. Band 609, Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings, Bari, Italy, 05/09/2019. https://doi.org/10.1088/1757-899X/609/3/032017

Peng, L, Nielsen, PV, Wang, X, Sadrizadeh, S, Liu, L & Li, Y, 2016, *Possible User-Dependent CFD Predictions of Transitional Flow in Building Ventilation*, Building and Environment, bind 99, nr. April, s. 130–141. https://doi.org/10.1016/j.buildenv.2016.01.014

Restivo, AMO, 1979, *Turbulent flow in ventilated rooms*. PhD thesis, University of London (Imperial College of Science and Technology)

Schälin, A & Nielsen, PV, 2004, *Impact of turbulence anisotropy near walls in room airflow*, Indoor Air, volume 14, nr. 3, s. 159-168.

Weng, M, Nielsen, PV & Liu, L, 2012, *A Discussion of Low Reynolds Number Flow for the Two-Dimensional Benchmark Test Case.* DCE Technical reports, nr. 116, Department of Civil Engineering, Aalborg University, Aalborg. See also www.cfd-benchmarks.com

Zhang, C, Nielsen PV, Bugenings, LA, Schaffer, M, Jensen, RL, 2020, *A benchmark test for the isothermal backward-facing step flow*, To be published at Indoor Air 2020.