Fluid simulations to optimize building ventilation systems

CFD simulation of ventilation components

***K.Tawackolian, T. Huang, M. Kriegel**, Hermann-Rietschel-Institut, Energy, Comfort & Health in Buildings, TU Berlin*

In Short

- Design and optimization of ventilation systems
- Pressure losses
- Part load conditions
- Turbulence modelling

Modern buildings use controlled mechanical or hybrid ventilation systems to ensure a healthy and comfortable indoor environment. Ventilation units are amongst the biggest consumers of indoor electricity, after heating, cooling, and lighting. An example ventilation network that supplies multiple office rooms is shown in Fig.1. The design of ventilation systems is presently carried out with field-specific software.

The calculation approach for the sizing and component selection is based on simplified one-dimensional methods that depend on technical data in the form of tabulated pressure loss coefficients. Correct system sizing is important for the energy efficiency and cost of the whole system. In practice, required pressure loss coefficients are often not available and rough guesses are made. What makes the situation more difficult is that in partial load operation of ventilation systems, the pressure losses of the duct system change and this affects various aspects of the design. It is already known from the earlier EnEff:Luft project (BMWi 03ET1223) that pressure loss coefficients of duct network components depend on the velocity or Reynolds number.

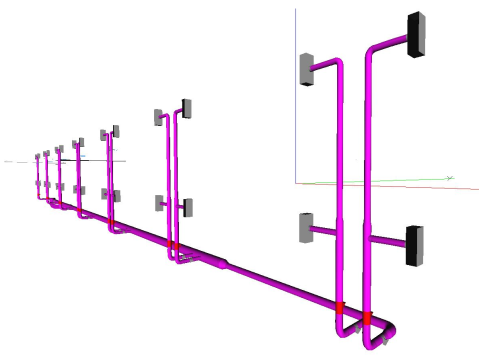

***Figure 1:** Example of an office building ventilation duct network.*

For some components, they can change by a factor of two. In practice, this influence can presently not be considered because technical design data for the full operation range of ventilation components is mostly unavailable.

When looking at the past immense increase of available computing power it can be expected that CFD simulations will be feasible in the near future for obtaining pressure loss coefficients of ventilation components. A formidable task for most CFD simulations is that of turbulence modelling. Reynolds averaged Navier Stokes turbulence modelling (RANS) is the main approach that is used for engineering problems although it suffers from well-known limitations. It is a simplified approach that requires validation and calibration based on adequate reference data. This approach is widely used because its computational cost is often at least one magnitude smaller than that of other common more detailed methods.

This project looks to find a suitable RANS turbulence model for CFD simulation and calculating pressure loss coefficients ζ of ventilation components. In building ventilation, the most relevant Reynolds number range is between 3×10^4 and 6×10^4, depending on the duct dimensions and airflow rates. Pressure loss coefficients can increase considerably for some components at Reynolds numbers below $Re < 2 \times 10^5$.

An initial survey of popular turbulence models was conducted for a selected test case of a bend with such a strong Reynolds number dependence[1]. Most of the turbulence models failed in reproducing this dependence and predicted curve progressions that were too flat and only applicable for higher Reynolds numbers.

The SST turbulence model can be considered as the current main industry turbulence model for internal flows. A trend in RANS turbulence can presently be seen towards new elliptic blending models. The elliptic blending approach promises a better prediction of the near-wall regions and is therefore also

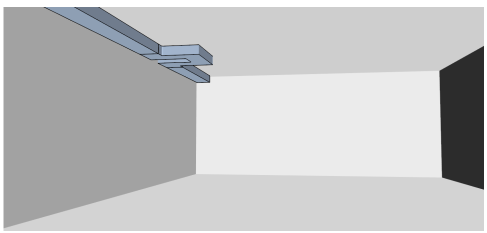

***Figure 2:** Example of a custom component in a duct network: diversion around an obstacle.*

interesting for internal and separated flows that depend strongly on near-wall models. The near-wall region is also often paramount for engineering questions as it affects the heat transfer and the energy (pressure) losses. High fidelity time-resolved simulations of near-wall flows at practical Reynolds numbers are notoriously expensive. This is why simplified RANS turbulence models are immensely useful for internal flows where wall effects are omnipresent.

The present study assesses the performance of RANS turbulence models for several test cases relevant for ventilation systems and strives to give a guidance on turbulence modelling for this application. This will support a new CFD based calculation basis for ventilation design that currently depends heavily on tabulated design data.

The necessity of large computing resources arises from the Reynolds number range that needs to be considered for each component. Furthermore, every case is simulated with at least three turbulence models. Therefore at least 45 simulations are performed for every component. Thanks to the support of HLRN more validation cases can be considered during the project duration and additional sensitivity and optimization studies can be carried out.

WWW

https://www.hri.tu-berlin.de/menue/research/research_projects/luftkonverter/

More Information

[1] Tawackolian K, Kriegel M, *Building Simulation*. Accepted for publication (2021).

Project Partners

RWTH Aachen, TROX, Lavair

Funding

German Federal Ministry for Economic Affairs and Energy, research program LuftKonVerTeR/BMWi 03ET1606A.