DEVELOPMENT OF CFD MODEL FOR A COOLED BEAM – IMPORTANCE OF PRODUCT SPECIFIC BOUNDARY CONDITIONS

Panu Mustakallio, Tuomas Moilanen, Mika Ruponen, Risto Kosonen, Kim Hagström
Halton Oy
Paper reference code: 140s

ABSTRACT

It has been found in earlier studies that boundary conditions of a supply air inlet are a critical element in room airflow simulation. The behaviour of the induction device in the computational fluid dynamics (CFD) simulation can vary greatly depending on the modelling of velocity profile, momentum and heat exchange of the supply air inlet. The object of this work was to develop CFD models of a cooled beam against full-scale laboratory test. In this study characteristics of throw pattern were analysed in laboratory measurements. CFD simulations of a typical office space ventilated with cooled beam system was conducted with two different sets of product boundary conditions, to compare a generic model without detailed knowledge of the device and a product specific model with supply air inlet velocity profile. This cooled beam CFD model is implemented in product selection tool that generates realistic boundary conditions for several types of cooled beams at any operating point. A one-person office room was studied in summer conditions. The use of the generic boundary conditions resulted in an unrealistic throw pattern from the cooled beam as a consequence it changed the flow picture in the room. The validation results pointed the importance of using product specific boundary conditions for air terminal devices in indoor air CFD modelling. This study depicts that only with product specific boundary conditions, it is possible to simulate the ventilated beam performance correctly.

KEYWORDS: CFD, Cooled beam, Boundary condition

INTRODUCTION

CFD (Computational Fluid Dynamics) simulations have been increasingly used in design and analysis of indoor climate. Beside conventional laboratory measurement computer based flow simulation helps to ensure that the ventilation system creates the proper indoor climate conditions. For the simulation one of the greatest challenge has been proper boundary condition for the products.

Boundary conditions for HVAC devices are a very critical element in room airflow simulation. If for instance supply air jet behaviour is not modelled correctly the whole flow picture in the room might fail. The wrong air jet diffusion affects greatly also on prediction of local comfort parameters, such as velocity and temperature conditions.

Ventilated cooled beams have been problematic for CFD modelling. Using wrong boundary data could lead totally misleading results. This depicts the importance of the validated velocity profile in cooled beam CFD simulation. The right input data is only possible to get from manufacturer’s product data. Generic velocity profile could lead totally wrong results.

In this paper, CFD model for a cooled beam is developed against full-scale laboratory test. This cooled beam model is implemented in a product selection tool that generates realistic
boundary conditions for cooled beams. As case study a typical office space ventilated with cooled beam system was simulated with two different sets of product boundary conditions, to compare a generic model without detailed knowledge of the device and a product specific model with realistic supply air velocity profile.

METHODS

A ventilated cooled beam (Halton CBC) was simulated with commercial CFD tool (Ansys CFX-5, tested also with Fluent Airpak) using main principles for reliable CFD modelling of supply air jet from diffuser that was studied earlier in the paper “CFD Modelling of an Air Jet from a Supply Air Nozzle” [1] and also taking into account the principles from the paper “A Procedure for Verification, Validation, and Reporting of Indoor Environment CFD Analyses” [2]. Correct jet velocity decay was achieved by using following methods:

- Generating fine grid in the front of supply air inlet and also on the trajectory of the jet
- Using k-ε turbulence model (also SST turbulence model tested)
- Solving the case with 2nd order discretization

The operating principle of the ventilated cooled beam is shown in the Figure 1. The cross section of the beam is shown in the figure 1, the width of the device is 0.6 meters and typical length 1 – 4 meters. Ventilated cooled beam can be used for both cooling and heating of spaces. The target in the development of CFD model for the cooled beam was to develop a simplified model of the beam with realistic functioning for the CFD simulation of indoor climate conditions.

![Figure 1. Ventilated cooled beam operating principle.](image)

CFD models of cooled beams were simulated in the similar fluid domain where the measurements conducted. Measurement room was 10 meters long, 6 wide and 3 high. It was conforms to standards ISO 5219 and ISO 5167 for measuring characteristics of throw pattern from supply air terminals, and equipped with computer controlled measuring robot. Measurements were conducted in stable, isothermal conditions with hot-wire anemometers.

Ventilated cooled beam was modelled using outlet boundary condition for induction air opening (2) (see Fig. 1.). Air leaving the cooled beam (2) was modelled with inlet boundary conditions by defining velocity profile for the inlet air flow. Geometry of the CFD model of the cooled beam and boundary condition values were determined from the measurements, manufacturer’s product data and turbulent jet theory definitions. Correct cooling or heating rate of the air leaving the beam was modelled with manufacturer’s calculation model for cooled beam operating point by defining inlet temperature for air leaving the unit as function of induction air temperature.
When the realistic velocity profile was worked out for the supply air jet from the cooled beam, it was applied to the office room CFD simulation. For comparison the same office case was modelled using “generic” CFD model of the cooled beam, by using also carefully defined CFD model data but with assumed uniform velocity profile for the inlet, which was the same size as the slot for the air leaving the beam.

RESULTS

CFD simulation visualisation of the cooled beam in the room, where the throw pattern measurements were conducted, is shown in the figure 2. This shows the velocity distribution from the side. In the figure 3, the measured and simulated velocity decay of the supply air jet is compared. There are results with the realistic and with generic velocity profile. It can be seen that the velocity decay with the generic model of the beam, is about 30% shorter than the real throw pattern.

![Figure 2. Simulated throw pattern of the realistic CFD model of ventilated cooled beam](image)

![Figure 3. Comparison of velocity decay with product specific or generic CFD model](image)
The generated CFD model for the cooled beam is shown in the figure 4. This shows also implementation in the manufacturer's product selection tool (Halton CFD Product Modeller) that generates product specific CFD model for several cooled beam type at the user selected operation point. The cooled beam type and the operating point in the figure 4 are the otherwise quite same as in the measurements shown in the figures 2-3, except cooling output from the beam.

As case study a typical office space ventilated with cooled beam system was simulated in summer conditions. Velocity streamline visualisations of the CFD results can be seen in the figures 5 and 6. Office space was simulated with two different sets of boundary conditions for cooled beams, to compare a generic model without detailed knowledge of the device (Figure 5) and a product specific model with realistic supply air velocity profile (Figure 6). In the simulation with generic cooled beam model, the supply air jet gets turned because of the plume from the warm window on the right side. This results in an unrealistic throw pattern from the cooled beam as a consequence it changed the flow picture in the room.
Figure 5. Office room simulation with generic CFD model of the ventilated cooled beam

Figure 6. Office room simulation with product specific CFD model of the ventilated cooled beam
DISCUSSION

CFD model of a ventilated cooled beam has been developed and compared against full-scale laboratory test. For comparison, the beam was simulated using generic model without detailed knowledge of the device and with unrealistic velocity profile for supply air inlet.

The validation results pointed the importance of using product specific boundary conditions for air terminal devices in indoor air CFD modelling. This study depicts that only with product specific boundary conditions, it is possible to simulate the ventilated beam performance correctly. The right input data is only possible to get from manufacturer’s product data. Generic velocity profile could lead totally wrong results when predicting local comfort parameters, such as velocity and temperature conditions in the room space.

CFD simulations are increasingly used in design and analysis of indoor climate to ensure that the ventilation system creates the proper indoor climate conditions. This creates obvious need for product specific boundary conditions. The developed CFD model for cooled beam has been implemented in the manufacturer’s CFD product model tool, that it includes also low velocity units in the current version. It can be used as a stand-alone tool with any CFD simulation software. This tool has been very useful in different indoor climate simulations. By implementing the CFD models in the HVAC device-dimensioning tool, they can be easily utilized by anyone requiring reliable models for the design and simulation with the manufacturer’s products.

ACKNOWLEDGEMENTS

The study is supported by Technology Agency of Finland (TEKES).

REFERENCES

