INDOOR AIR 2016 Workshop: CFD prediction of non-isothermal flow – How to minimize the user factor?

Peter V. Nielsen, Twan van Hooff and Yuguo Li

Background

We know that non-isothermal flow is difficult to predict by CFD, especially in the high Archimedes number area where the flow is mainly driven by the buoyancy forces. This may be due to limitations of available turbulence models in connections with RANS equations, limitations of computer capacity and limitations in other types of methods as e.g. the use of LES. Errors due to the choice of numerical methods, choice of the right physical model of the engineering problem you solve, choice of relevant boundary conditions and even errors due to user’s experience are also a part of relevant problems in performing non-isothermal CFD predictions.

Significant development and improvement has taken place in the CFD community in the last 30 years and, at least, some of the above mentioned problems have been addressed. It is therefore interesting to see how far we are today collectively with the relevant software and hardware.

The workshop is a continuation of the ISHVAC – COBEE 2015 workshop on the prediction of isothermal low Reynolds number flow.

Invitation

We invite you to participate in a workshop which will be finalized at the Indoor Air conference in July 2016.

The purpose of this new workshop is to focus on errors existing in CFD simulations, especially in connection with large Archimedes number flow, and discuss what we can do, using a simple method.

We present a simple CFD problem (with no benchmark or experimental data for comparison) for all participants of the workshop to simulate by using CFD software. We ask you to send your CFD results to us, and we shall consolidate all the results, and present in the workshop for discussion. You are also welcome to submit your CFD simulations even you do not plan to physically attend the workshop.

Consider this problem was a request from your client – your client is interested to have an accurate prediction of the penetration length for the whole Archimedes number range from zero to 8.

All participants who have submitted the simulations are invited to join, and we will discuss together on the following topics:

1. How large are the differences in CFD results? How significant are the differences? Why are there differences? What can we do to minimize potential CFD errors?
2. How are the different schemes able to handle the high Archimedes number flow (buoyancy driven flow) in the selected case? How can we find/define the limits for use of different turbulence models?

3. What are the challenges in applying CFD to a problem? How to build confidence in CFD simulations?

The CFD problem

We propose a simple CFD problem that is easy for all to simulate even on your PC. The flow is incompressible and two-dimensional at low Archimedes number flow, but not necessarily in the whole regime. It may be considered as a simple building ventilation problem, and the lay out has a certain connection to the isothermal ISHVAC – COBEE case.

![Diagram of the CFD problem](image)

**Figure 1.** The geometry to use in the test case. The red vertical line indicates the heated wall.

**Figure 1** shows the geometry of our proposed case. The flow indicated is typical for isothermal two-dimensional room air flow in deep rooms but non-isothermal flow will change the flow pattern. $H$, $L$, $h$ and $l$ are room height, room length, supply slot height and length of supply opening. $l$ and $h$ are also the dimensions of the outlet opening. The vertical wall below the supply opening is the heated wall in the non-isothermal predictions. All other walls than the heated wall are adiabatic ($\partial T/\partial n = 0.0$).

$L$ is the length of the model/room. This length is large and therefore the left side of the model can, at low Archimedes numbers, be compared to the situation studied in the ISHVAC – COBEE 2015 workshop. The small return opening ensures that downstream reverse flow is impossible in the case of large temperature differences. $x_{re}$ is the length from the end wall to the location where the reattached flow is separated in a flow back to entrainment into the wall jet and a forward flow towards the exit. The length $x_{re}$ is referred to as the penetration length of the supply jet.

We will also consider the maximum velocity in the “occupied zone”, $u_{rm}/u_o$, which is the maximum velocity in the return flow where the streamlines are close (below the center of the circulation in the large recirculation bubble), see **Figure 2**. The position of this velocity may be difficult to locate but this aspect is probably not so important because it only has a small variation in the direction of the flow.
Figure 2. Indication of the location of the maximum velocity $u_{rm}$ of the “occupied” zone.

The distance from the heated end wall to the location of the velocity $u_{rm}$ is called $x_{rm}$.

The following dimensions should be used:

\[
\frac{h}{H} = \frac{1}{5} \\
\frac{l}{h} = 4 \\
L = 10H
\]

The flow might be unsteady and three-dimensional in a certain range of Archimedes numbers (we cannot exclude the possibility). We will therefore also define the test case as a 3D geometry with a width of:

\[
W = 2H
\]

We consider the velocity and the penetration depth in the 3D case in the vertical median plane at $y = 0.5W$.

The inlet flow is a Top Hat profile with a constant velocity $u_o$ everywhere in the profile. The turbulence variables are specified as 10% turbulent intensity and a viscosity ratio $\nu_t/\nu = 10$ at inlet.

The Reynolds number is:

\[
Re = \frac{h \cdot u_o}{\nu}
\]

Where $\nu$ is kinematic viscosity. $Re$ should have a value of 10,000 which corresponds to the highest $Re$ number in the COBEE workshop giving fully developed turbulent flow in this work.

The Archmedes number is:

\[
Ar = \frac{\beta g H \Delta T_o}{u_o^3}
\]

Where $\beta$ is the volume expansion coefficient, $g$ the gravitational acceleration and $\Delta T_o$ the temperature difference between the average temperature at the return below the right wall (see red line and red surface in Figure 3 below) and the temperature at the supply opening. Please make sure to check that you only have an outflow at the return “line/surface” (i.e. only flow directed towards outlet opening). If not, please mention this in your report.
Figure 3. Schematic representation of the domain (not on correct scale) with indication of the locations at which the average temperature at the return should be determined. For 2D simulations the average return temperature as input for the determination of the $Ar$ number should be taken at the red dashed line; while the average return temperature should be taken on the red surface for 3D simulations. Please make sure the flow at the return is only directed towards the outlet opening.

One possible physical dimension of the problem is as follows.

$H = 5 \text{ m}$, $h = 1 \text{ m}$, $l = 4 \text{ m}$, $L = 50 \text{ m}$, $u_o = 0.16 \text{ m/s}$, the heat input provided by the user will give you the corresponding $Ar$, based on $\Delta T_o$. It is suggested to study a range of heat fluxes between 0 and 100 W/m$^2$ to obtain appropriate $Ar$ numbers ($0 < Ar < 8$).

The workshop

You are invited to participate in a workshop where you predict the flow in the geometry from very small Archimedes numbers to high numbers with strongly buoyancy driven flow. The range should be from $Ar = 0$ to $Ar = 8$. It is especially interesting to see if it is possible to handle the high Archimedes number regime or at least a part of it.

We ask you to select one or several CFD code(s) (commercial code, open source code (openFOAM), or your own development) and we ask you to select an appropriate turbulence model. Alternatively, you could also resort to use LES. You are free to select the boundary conditions, as long as the requirements in the previous section are fulfilled.

We will ask you to submit a short report by June 1 2016 by email to Twan van Hooff, email twan.vanhooff@bwk.kuleuven.be. Your report should contain a very brief description of your simulations including CFD modeling strategies, the commercial codes, the grid type (hexahedral, tetrahedral, etc.), grid numbers, convergence criteria, number of iterations, turbulence model if used, and numerical schemes.
(convection schemes), and solution methods (SIMPLE, Multigrid) etc. Note that you do not need to spend a lot of time on this report, and list of factual items will be fine. A template will be provided by the workshop coordinators. Please send an email to Twan van Hooff to inform the workshop coordinators about your intended participation and to receive this template.

You also need to submit an Excel file with predicted recirculation lengths for a minimum of 7 Archimedes numbers (between 0 and 8) with four columns (Archimedes number $Ar$, recirculation length $x_{re}/(H-h)$, length to maximum velocity $x_{rm}/(H-h)$ and maximum velocity $u_{rm}/u_o$).

We recommend you to follow well-documented CFD guidelines in performing such simulations. You should select the turbulence model and other conditions, you think are the most appropriate to use considering the type of flow taking place. You may even consider recommending different models and procedures in different Archimedes number regimes.

You are reminded to take a special care for the high Archimedes number region. If your code or software tells you that your chosen turbulence model may not work for a particular region, please report it in your results. Do also tell us if your code gives any other warning.

**Please note that no one knows the exact results including us as coordinators.**

Each of you will be assigned a team code, C1, C2,...C10... and all results will be presented anonymously using the team number code, and only you and the coordinators know the code assignment...so you will not feel embarrassed in any way. Be brave, just submit your results. The code will be assigned randomly, and your team code will be sent to you after we have received your results.

We will present and compare your results on the Indoor Air 2016 conference workshop. In the workshop we will focus on:

1. How to minimize the user factor?
2. The influence of turbulence models
3. The commercial codes
4. The influence of boundary conditions
5. Type of grid, grid numbers and number of iterations
6. Effective strategies in the high Archimedes number flow area

Part of the results will be published on the home page [www.cfd-benchmarks.com](http://www.cfd-benchmarks.com) and it could probably also be published in a peer reviewed journal.