IOP Publishing
2042 (2021) 012069 doi:10.1088/1742-6596/2042/1/012069

Development of a freely distributable CFD tool for approximate and detailed simulations of the flow around a complex of buildings

J Decaix[1*], P Jaboyedoff[2], G. Duthé[3], E. El Sergany[4], L. Aiulfi[5]

[1]Institut Énergie et Environnement, Haute Ecole d'Ingénierie, HES-SO Valais-Wallis, Rawil 47, 1950 Sion, Switzerland.
[2]Effinart, Chemin de Pré-Fleuri 6, CH-1006 Lausanne, Switzerland.
[3]Department of Civil, Environmental and Geomatic Engineering, ETH Zurich, Stefano-Franscini-Platz 5, Zurich, 8093, Switzerland.
[4]Physics Master student at Ecole Polytechnique Fédérale de Lausanne, Switzerland.
[5]3D Geometry Engineer at ESRI, Zurich, Switzerland.

[*]jean.decaix@hevs.ch

Abstract. Indo-Swiss Building Energy Efficiency Project (BEEP) is a cooperation project between the Ministry of Power, Government of India, and the Federal Department of Foreign Affairs of the Swiss Confederation. Started in 2011, the project's central focus is to help India mainstream Energy-Efficient and Thermally Comfortable (EETC) Building Design. BEEP works with building industry, policy makers, and building owners to catalyse adoption of EETC building design and technologies. India wants to avoid or reduce the use of air conditioning by improving natural ventilation at night, which requires numerical simulations to compute the flow around the buildings. However, the simulations of fluid flows are time consuming and are not used at the beginning of a project when the locations of the buildings are set. To improve the situation, a freely distributable environment based on the OpenFOAM toolbox has been developed providing two levels of resolution: an approximate level computing the flow in few minutes and a RANS level of simulation. The user inputs are limited to the geometry and the velocity direction and magnitude. The mesh and the numerical set up are automated. The accuracy of the two levels of resolution have been checked by computing test cases from the CEDVAL database.

1. Introduction

The twentieth century has been marked by the increase in the number of human beings living in an urban area [1]. Urban areas are known to be responsible for heat island effect that causes several problems and challenges. One of them is the lack of temperature drop during the nights, which prevents the cooling of air inside the building except if electrical air-cooling systems are used. However, such systems are known to promote the urban heat island (UHI) effect by releasing hot air outdoor. Furthermore, these systems require to increase the production of electricity, which is today achieved using mainly fossil energy sources [2] responsible for the global warming effect.

To avoid turning to electrical air-cooling systems, one solution is to improve the natural ventilation of the urban areas. Today most urban dwellers in India do not have any active cooling. Therefore, the location of the buildings must be defined in such a way that the air can circulate between the buildings and is not blocked or trapped by the layout of the buildings. Computational Fluids Dynamics (CFD) can be a useful tool to investigate the air flow inside urban areas [3]. However, CFD often requires computational resources and specific knowledge. Therefore, it is not used at the beginning of the projects when the location and orientation of the buildings is decided. The night cooling potential is rarely optimized.

Indo-Swiss Building Energy Efficiency Project (BEEP) is a bilateral cooperation project between the Ministry of Power, Government of India, and the Federal Department of Foreign Affairs of the Swiss Confederation (FDFA). Started in 2011, the project's central focus is to help India mainstream Energy-Efficient and Thermally Comfortable (EETC) Building Design for both commercial and residential buildings. The Bureau of Energy Efficiency is BEEP's implementing agency for the Ministry of Power, while the Swiss Agency for Development and Cooperation oversees the project for FDFA. In the framework of BEEP project, one task is dedicated to the development of a freely available CFD tool allowing the simulation of the air flow in an urban area at an acceptable level of resolution requiring only a low computational cost and no expert knowledges in CFD.

To carry out this task, the free OpenFOAM toolbox is packed in a friendly user interface coded in Python and embedded in a Docker allowing the use of the software on various operating systems such as Windows 10. A methodology has been developed for two different levels of resolution: an approximate level and more detailed one, which is close to a standard RANS simulation. For both levels, the mesh and the numerical set up have been automated using the recommendation available for instance in [4] and [5]. The methodology has been validated by computing free test cases from the CEDVAL [6] database.

The paper is organized as follows: first the software with the implemented methodology is described, then validation test cases are summarized and finally few examples of urban area that can be handled by the software are shown.

2. Description of the software and the methodology

The software is based on the free OpenFOAM toolbox managed by the OpenFOAM Foundation and used by a large community of CFD users both in academics and industries. This toolbox provides meshing tools, fluid flow solvers, pre-defined boundary conditions and post-processing tools. Consequently, most of the works can be carried out using specific templates dedicated to the simulation of air flows around buildings. Two drawbacks of the toolbox are the lack of a graphical user interface (GUI) and its limitations to run only on Linux operating systems. The first point has been tackled by developing a dedicated GUI in Python, whereas for the second, the OpenFOAM toolbox has been embedded in a Docker image.

The three main tabs of the GUI are shown in figure 1. They allow the user to load the 2D layout of the building area stored in a dxf file format. Then, the user must specify the height of each building, the direction of the wind and the reference velocity used to calculate the inlet boundary conditions. Other features are available such as the possibility to change the orientation of the buildings. Once the previous mandatory inputs are provided, the user must choose between the approximate or the detailed level of resolution and the number of cores to be used for the mesh generation and the simulations. After the simulation ends, the results can be viewed in a dedicated window allowing for contours, streamlines and vector plots. Furthermore, a PDF report can be created automatically.

For the mesh generation, the blockMesh and the snappyHexMesh utilities are used. The first one creates a cartesian mesh that allows defining the computational domain surrounding the building area, whereas the second one allows refining the initial cartesian mesh. The computational domain is defined in such a way that the inlet, the sides and the top boundaries are located at $5H$ away from the building area, with H the height of the highest building. The outlet is located $15H$ downstream the urban area as suggested in [4]. For the approximate level of resolution, the smallest mesh size is based on distance between buildings. Compared to the detailed level of resolution, the approximate level does not perform any refinement close to the walls as shown in figure 2, which allows decreasing the number of cells.

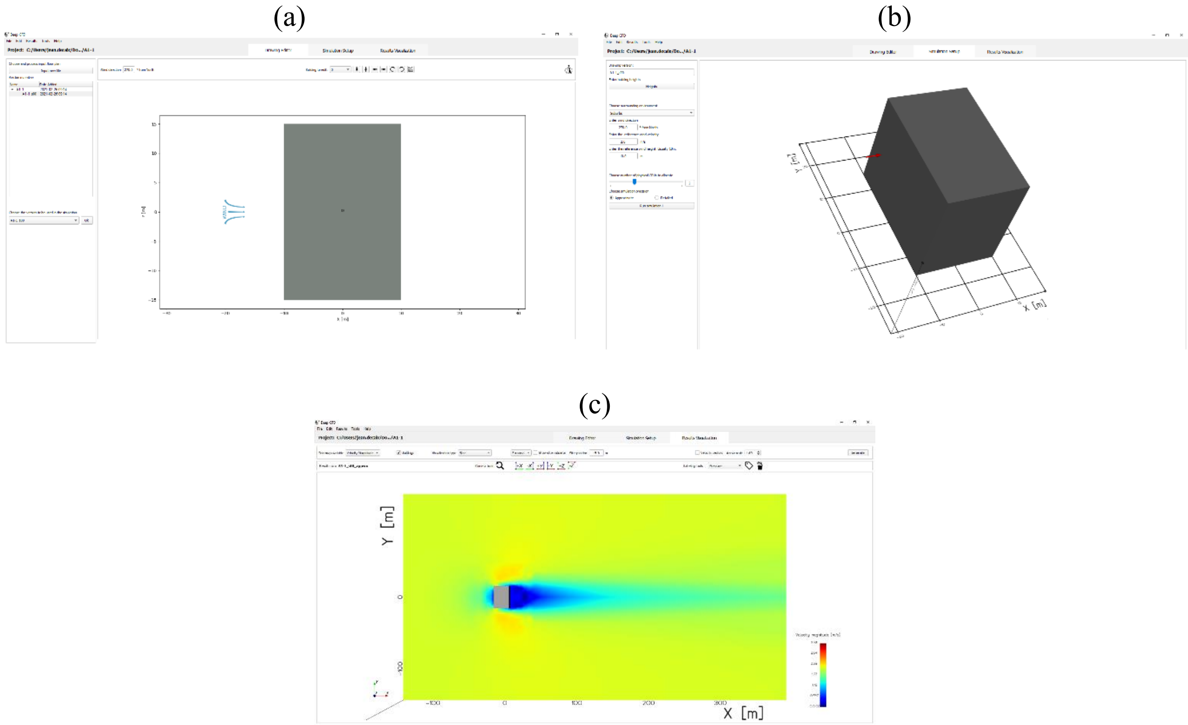

Figure 1. (a) Drawing editor tab, (b) Simulation set up tab, (c) Results visualization tab of the GUI developed for the software.

The boundary conditions are the same for both levels of resolution except for the solid walls, which are considered as free slip wall for the approximate level of resolution instead of no-slip wall. The inlet velocity profile is modelled by the standard logarithmic atmospheric boundary layer formulation [5]. At the top and the sides of the domain, a free slip condition is specified.

The main difference between the two levels of resolution concerns the fluid modelling. For the detailed level of resolution, the set of equations solved is the standard Reynolds-Averaged Navier Stokes equations coupled with the two-equations realizable k-ε turbulence model. For the approximate level of resolution, a dummy laminar turbulence model is used with an artificially high fluid viscosity. This modelling considered that the inlet flow is only convicted through the urban area and distorted by the buildings. It allows a fast convergence of the flow with maximum number of iterations set to 200.

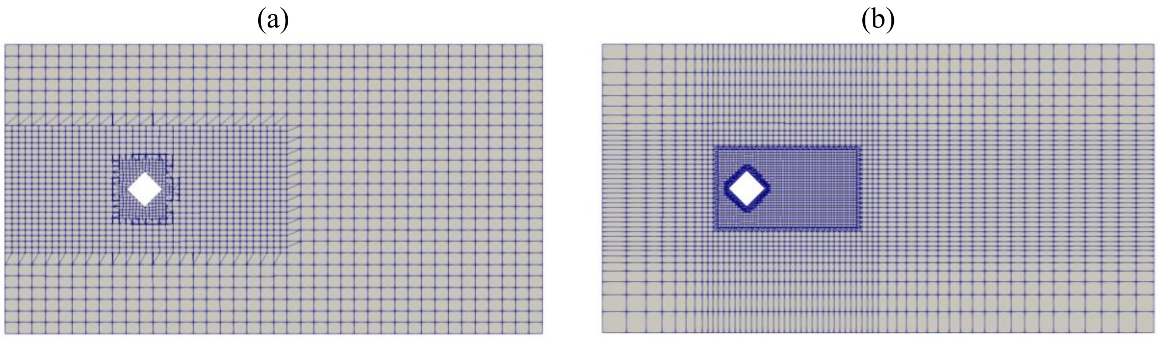

Figure 2. Horizontal plane view of the mesh for the A1-6 CEDVAL test case (flow coming from the left): (a) approximate level of resolution, (b) detailed level of resolution.

3. Validation

The validation of the methodology has been carried out by computing documented test cases such as the cases A1-6 and B1-1 available in the CEDVAL database [6]. The A1-6 test case consists of a single cube mounted at 45° compared to the incoming wind (see figure 2). The B1-1 test case is an array of 21 buildings equally space in 3 lines of 7 rows (see figure 3). The number of cells for each mesh are respectively of 25 000 (A1-6 approximate level), 280 000 (A1-6 detailed level), 36 500 (B1-1 approximate level) and 640 000 (B1-1 detailed level).

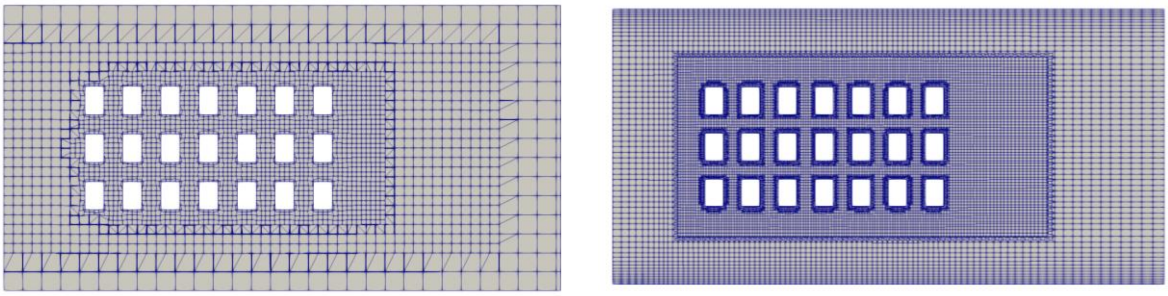

Figure 3. Horizontal plane view of the mesh for the B1-1 CEDVAL test case (flow coming from the left): (a) approximate level of resolution, (b) detailed level of resolution.

For both cases, the validation is measured using the metric hit rate q_i defined by equations (1) and also used in [7].

$$q_i = \frac{1}{N} \sum_{n=1}^{N} i_n \tag{1}$$

$$i_n = \begin{cases} 1, & \left|\frac{u_{cfd} - u_{exp}}{u_{exp}}\right| \leq \Delta_r \text{ or } \left|u_{cfd} - u_{exp}\right| \leq \Delta_a \\ 0, & \text{otherwise} \end{cases}$$

With u_{cfd} the dimensionless velocity from the CFD simulation (i.e., the velocity is first divided by the inlet velocity at the height of the building), u_{exp} the dimensionless measured velocity, Δ_r the allowed relative error and Δ_a the allowed absolute error. Δ_r is set to 0.25 for both test cases whereas Δ_a is set to 0.06 for the A1-6 test case and to 0.1 for the B1-1 test case.

The values of the metric hit rate for each velocity components are given in table 1 for both cases and level of resolution. In overall, the hit rate values are close between the two levels of resolution. Moreover, except for the hit rate q_v for the test case A1-1, the hit rates are above 0.66, which is considered as a threshold for considering the simulation as accurate [7]. This feature could be surprising at a first stage, but it is explained by the fact that most of the experimental probes are located outside the wake developing downstream the buildings. The wake region is the most critical region for the simulation accuracy. In figure 4, for both cases, the magnitude of the dimensionless velocity in a horizontal plane located at the altitude $z = 0.8H$ are compared between the two levels of resolution. Compared to the detail level of resolution, the approximate simulation is not able to predict a symmetric flow downstream the urban area. Moreover, the approximate simulation tends to predict a higher magnitude of the velocity inside the building area. However, the flow pattern is still well predicted in accordance with the detailed simulations.

The CPU time required by the simulations at the approximate level of resolution is 200 times smaller than the one for the simulation at the detailed level of resolution.

Table 1. Hit rate value for each velocity component and for three different meshes.

	A1-6			B1-1		
Mesh	q_u	q_v	q_w	q_u	q_v	q_w
Detailed level	0.70	0.59	0.73	0.76	0.93	0.87
Approximate level	0.68	0.52	0.81	0.91	0.64	0.79

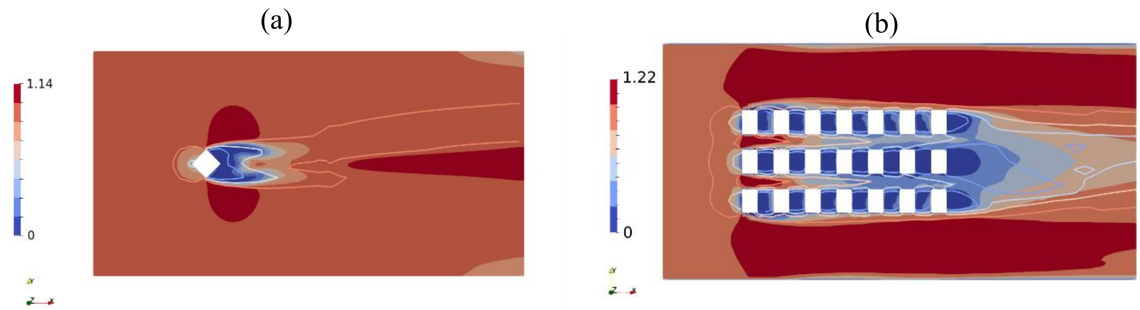

Figure 4. Comparison of the magnitude of the dimensionless velocity between the approximate level of resolution (lines) and the detailed level of resolution (filled contours) in plane located at altitude $z = 0.8H$. (a) A1-6 test case. (b) B1-1 test case.

4. Example of application

The beta release of the software is used to compute the Rajkot project in India that consists of twelve buildings with different heights, spacing between them and orientations. The test case is computed using both levels of resolution, which requires 10 minutes on a laptop Dell Latitude 7400 using two cores for the approximate level and 80 minutes using four cores for the detailed level of resolution. Figure 5 compares the streamwise velocity contours and the wall pressure between the two levels of resolution. In overall, the approximate level of resolution predicts the same velocity and pressure pattern than the detailed simulation. Compared to the approximate level of resolution, the detailed simulation shows a higher velocity outside of the wake and the presence of flow instabilities in the wakes of the buildings that have also a slightly different length and width.

5. Conclusion

A free distributable CFD tools has been developed in the framework of the Indo-Swiss BEEP project aiming at computing the air flow in an urban area. This tool based on the OpenFOAM toolbox and embedded in Docker image allows through a dedicate GUI performing simulations at two levels of resolution: an approximate level and a detailed level. No expert CFD knowledge is required to use the software since the mesh and the numerical set up are entirely automated. The two levels of resolution solve the standard RANS equations coupled with the realizable k-ε turbulence model for the detailed level of resolution and with a dummy laminar model including an artificial high fluid viscosity for the approximate level of resolution. In addition, for the approximate level of resolution, a free slip boundary condition is imposed to the solid walls instead of a no-slip boundary condition.

The comparisons carried out for two test cases available in the CEDVAL database show that the approximate level of resolution gives results like the detailed level of resolution at a CPU cost that can be at least 20 times lower.

This software should be used in the first steps of a building project to avoid setting buildings at locations that prevent natural ventilation and consequently reduce the cooling of urban areas during the nights. It should be also used to improve the natural ventilation in built areas by testing solutions or to determine the impact of new buildings.

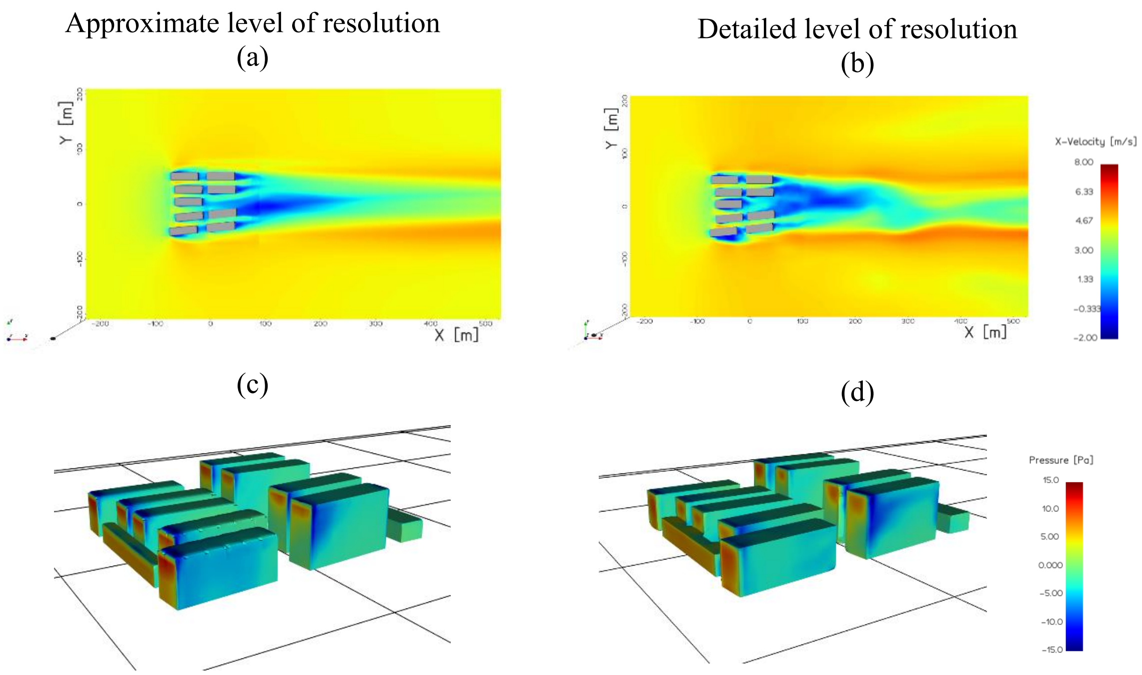

Figure 5. Rajkot project. (a) and (b) Contours of the streamwise velocity component, (c) and (d) Pressure on the building walls.

Funding
The BEEP project is supported the Department of Foreign Affairs (FDFA) of the Swiss Confederation.

Acknowledgments
The BEEP project team thanks the Swiss Agency for Development and Cooperation (SDC) for its continuous support during the development of this CFD software. The support of the Bureau of Energy Efficiency has also helped in framing the objectives and taking care of its future launch and distribution to a wide audience in India.

References

[1] United Nations Department of Economic and Social Affairs Population Division 2014 World urbanization prospects: the 2014 revision, highlights (ST/ESA/SER.A/352)
[2] IEA 2020 *World Energy Balances:* Overview (IEA, Paris https://www.iea.org/reports/world-energy-balances-overview)
[3] Toparlar Y., Blocken B, Maiheu, B, van Heijst G J F 2017 A review on the CFD analysis of urban microclimate *Renewable and Sustainable Energy Reviews* vol 80 pp 1613–40
[4] Franke J 2006 Recommendations of the COST action C14 on the use of CFD in predicting pedestrian wind environment *Proc CWE symposium*
[5] Blocken B 2015 Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations *Building and Environment* vol 91 pp 219–45
[6] CEDVAL test cases https://mi-pub.cen.uni-hamburg.de/index.php?id=433
[7] Franke J Sturm M Kalmbach C 2012 Validation of OpenFOAM 1.6.x with the German VDI guideline for obstacle resolving micro-scale models *J of Wind Engineering and Industrial Aerodynamics* pp 350–9