Tuyển tập Công trình Hội nghị khoa học
Cơ học Thủy khí toàn quốc lần thứ 22

Comparing the performance of Ansys Fluent and OpenFOAM in terms of accuracy and computational expense.

***Do Van Dai*[1], *Khieu Huu Loc*[2]**
[1] *Faculty of Automotive Engineering, Cao Thang Technical College, Ho Chi Minh City*
[2] *Faculty of Engineering, Vietnamese German University, Thu Dau Mot City, Binh Duong*

Abstract: *Natural ventilation is always an attractive topic in civil engineering. The attractiveness of natural ventilation comes from the potential reduction in energy consumption for the purposes of keeping the building atmosphere habitable. Nowadays with the development of super-computers, scientists have been applying theory of Computational Fluid Dynamics (CFD) in calculating and simulating flow fields in order to assess the ventilation characteristics of a given building. For simulation engineers, it is important that those CFD simulations generate reliable results at reasonable computational costs. The main purpose of this work is to investigate the performance of Ansys Fluent and OpenFOAM in terms of accuracy and computational expense for simulations of natural ventilation. The results in this paper shows that the outcomes of two softwares are similar and it should be noted that OpenFOAM is completely free distribution and the flexibility it offers allow the development of specific solvers by the user, which can be integrated with already existing tools.*

1. Introduction

Natural ventilation plays a vital role in the development of healthy indoor envionments. It is driven by wind or buoyancy or combined. In the recent years, a lot of researchers made effort to investigate the natural ventilation perfomance of the buildings [1].

By several ways, venitlation performance can be assessed by experiments, theoretical analyses and simulations using Computational Fluid Dynamics (CFD). CFD has a number of advantages compared with the other approaches: (1) CFD provides whole flow-field data, i.e., values of flow variables (velocity, pressure, temperature, etc.) at every point in the computational domain; (2) CFD advoids the sometimes incompatible similarity requirements in reduced-scales testing because simulations can be performed at "full scale;" and (3) CFD allows full control over the boundary conditions [2].

In the simulation field, the accuracy, reliability and the computational cost of CFD calculations are main concerns. Therefore, the duty of CFD engineers is to try to get the most accurate solutions in the shortest time with the cheapest cost. For sure, the accuracy is always the first priority and the task of this part is to ruplicate the republished results that were validated with experimental data.

The aim of next work is the development of procedures for the simulation of atmospheric flows over simple building, using OpenFOAM v5, thereby contributing for the use of open source CFD (Computational Fluid Dynamics) codes in this specific field of engineering.

The goals are procedures for the case pre-processing, like the generation of computational mesh, definition of boundary conditions, roughness mapping and set up of turbulence models, and data extraction of simulation results.

2. Theoretical background of CFD

2.1. Governing Equation of Fluid Mechanics

The governing equations of Computational Fluid Dynamics are based on the conservation laws of physical properties of fluid.

Mass conservation equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j)=0 \tag{1}$$

Momentum conservation equation

$$\frac{\partial \rho}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_j u_i)=\rho g_i-\frac{\partial p}{\partial x_i}-\frac{\partial \tau_{\mathrm{ij}}}{\partial x_j} \tag{2}$$

Energy conservation equation

$$\frac{\partial \rho E}{\partial t}+\frac{\partial(\rho E u_i)}{\partial x_i}=-\frac{\partial q_i}{\partial x_j}+\rho g_i u_i-\frac{\partial p}{\partial x_i}(p u_j)-\frac{\partial}{\partial x_j}(\tau_{\mathrm{ij}} i_j) \tag{3}$$

2.2. Turbulence Modelling

Flow behaviour is charactierized by the dimensionless Reynolds number.

$$\mathrm{Re}=\frac{\rho v L}{\mu} \tag{4}$$

At low Reynolds numbers flows are laminar; at high values they become turbulent. For a pipe flow, increasing the inflow velocity i.e., the characteristic velocity, leads to the transition of the flow. Direct Numerical Solution (DNS) and Large-Eddy Simulation (LES) approaches have long been believed to have great potential for the accurate prediction of difficult turbulent flows, but the associated computational cost has been prohibitive.

The Reynolds averaged Navier-Stokes equations, shortly RANS, are obtained applying the Reynolds decomposition to the unknowns appearing in the Navier-Stokes equations and time averaging. The RANS contain further unknowns called Reynolds stresses which are due to all scales turbulence and need to be modelled in order to close the set of equations. The biggest advantage of using the RANS in simulating turbulence flows is that they allow to treat the turbulence as a steady phenomena. That is a great means for saving of computational. The RANS based turbulence models are usually classified by the number of additional differential equations needed to close the original set of PDE.

RANS models offer the most economic approach for computing complex turbulent industrial flows. Typical examples of such models are the *k - epsilon* or the *k - omega* models in their different forms. These models simplify the problem to the solution of two additional transport equations and introduce an Eddy-Viscosity (turbulent viscosity) to compute the Reynolds Stresses. RANS models are suitable for many engineering applications and typically provide the level of accuracy required. Since none of the models is universal, you have to decide which model is the most suitable for a given applications [3].

k − ε Model

The first transported variable is *the turbulence kinetic energy (k)*

$$\frac{\partial \rho k}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\rho}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\rho k}{\rho k_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon \tag{5}$$

The second transported variable is the *rate of dissipation of turbulence energy (ε).*

$$\frac{\partial \rho\varepsilon}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\rho\varepsilon}{\rho x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

k − ω Model

The first transported variable is *the turbulence kinetic energy (k)*

$$\frac{\partial \rho k}{\partial t}+\frac{\partial(\rho k u_j)}{\partial x_j}=\rho P+\beta^*\rho\omega k+\frac{\partial}{\partial x_j}\left[\left(\mu+\rho_k\frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right] \tag{7}$$

The first transported variable is the *specific rate of disipation* (ω)

$$\frac{\partial \rho\omega}{\partial t}+\frac{\partial(\rho\omega u_j)}{\partial x_j}=\frac{\gamma\omega}{k}P-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_\omega\frac{\rho k}{\omega}\right)\frac{\partial\omega}{\partial x_j}\right]+\frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \tag{8}$$

k − ω model is a mix of the $k-\varepsilon$ and $k-\omega$ models. In the near-wall region $k-\omega$ is used and further away from wall in the fully turbulent regions the $k-\varepsilon$ method is used. These are available in ANSYS Fluent as an option [3].

3. Simulation Implementation in Ansys Fluent and OpenFOAM

3.1. Physical Problem

In this study, the outdoor wind flow and indoor air flow are solved within the same computational domain and the interaction (coupling) to observe the ventilation performance of a cubed house with dimensions W×D×H = 20×20×16 m^3 [2].

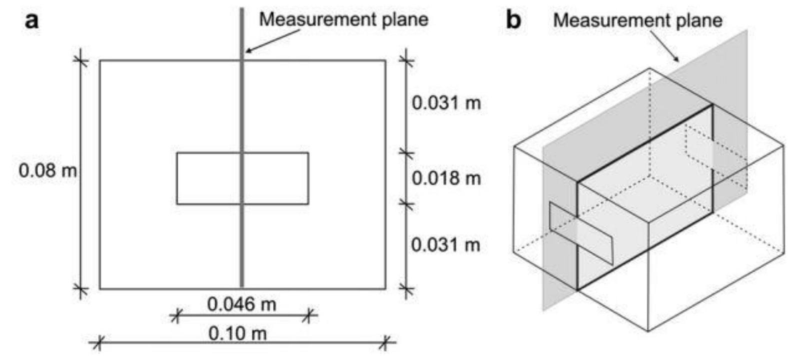


Figure 1. The building section [2]

However, a reduced scale equivalent geometry building with the dimensions: W×D×H = 100×100×80 mm^3 corresponding to the above full-scale is used to simulate. The advantage of this scale reduction is to have an alike result with reasonable computational cost and calculation time.

Generally, these problems are usually treated by 2 ways of approaching:

- **Coupled approach:** investigate the impact of external flow on internal flow in a computational domain.
- **Decouple approach:** devide the computational domain into 2 sub-domains and assume that the openings are closed and calculate them one by one. The result of the outter one will be used to calculate for the inner one as the boundary conditions. This means can develop errors as these ventilation openings are enough large [2].

In this study, the outdoor wind flow and indoor air flow are solved within the same computational domain and the interaction (coupling) to observe the ventilation performance and compare to the experimental data of Ramponi and Blocken [2].

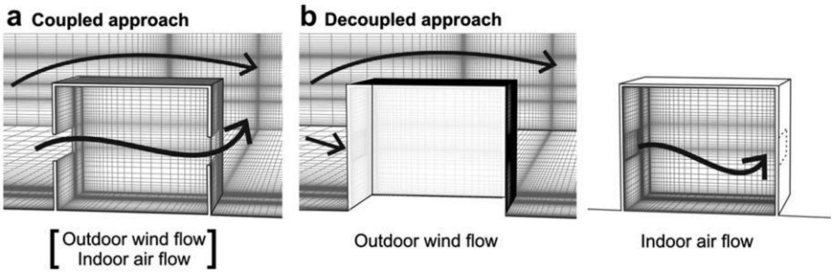


Figure 2. Coupled and (b) decoupled approach for analysis of wind-induced crossventilation of buildings [2]

Moreover, The Atmospheric Boundary Layer (ABL) can be described like the "lowest 1-2 km of the atmosphere, the region most directly influenced by the exchange of momentum, heat, and watervapour at the earth's surface", Kaimal and Finnigan (1994) [4].

According to Richards and Hoxey in their pioneering 1993 paper on "Appropriate boundary conditions for computational wind engineering models using the $k-\varepsilon$ turbulence model" [5], the most profiles used for RANS CFD simulations in urban physics and wind engineering are:

$$U(z)=\frac{u^{*}_{ABL}}{\kappa}\ln\left(\frac{z+z_0}{z_0}\right) \tag{9}$$

$$k(z)=\frac{u^{*2}_{ABL}}{\sqrt{C_\mu}} \tag{10}$$

$$\varepsilon(z)=\frac{u^{*3}_{ABL}}{\kappa(z+z_0)} \tag{11}$$

Where: u^{*}_{ABL} is *friction velocity*, κ is Karman constant (0.42) and C_μ constant, generally taken equal to 0.09.

3.2. Simulation Setting in Ansys Fluent and OpenFOAM

The computational domain size is choosen based on the "Best Guideline for CFD simulation of flows in the urban environment" (2007) and Tominaga that results in the reduced scale with dimensions: $0.9 \times 1.54 \times 0.48 m^3$ corresponding to the reduced scale building as discussed above.

The biogeometric mesh law inside Ansys ICEM is used to generate structured meshes in order to make sure that the corners surrounding of the buiding models are meshed finely enough with the transition of stretching ration 1.2 is from the building wall (2 mm thickness) to the larger scales in the domain in order to optimize the computational cost.

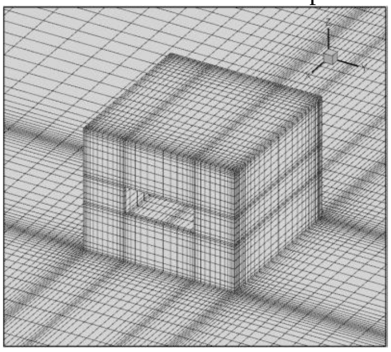


Figure 3. 577049-cell structured mesh.

At the building height z = 0.08 m, the reference velocity Uref = 6.97 m/s. Therefore, the ABL friction velocity is 0.363 m/s.

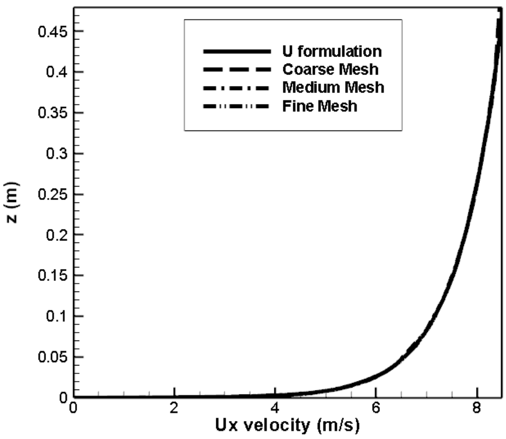
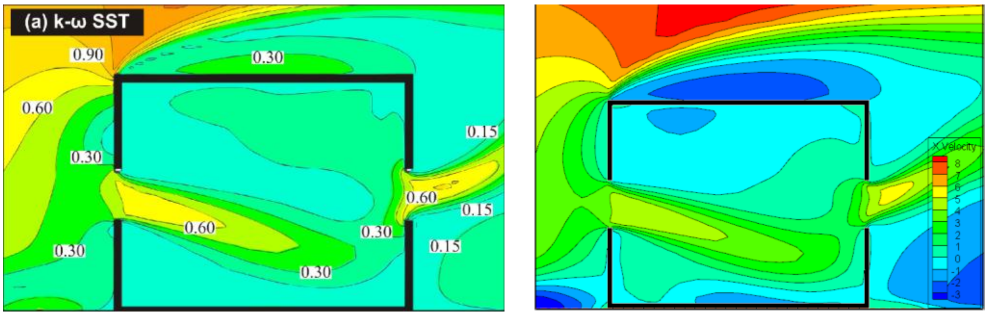


Figure 4. Inlet velocity profile

4. Simulation Results and Comparisions:

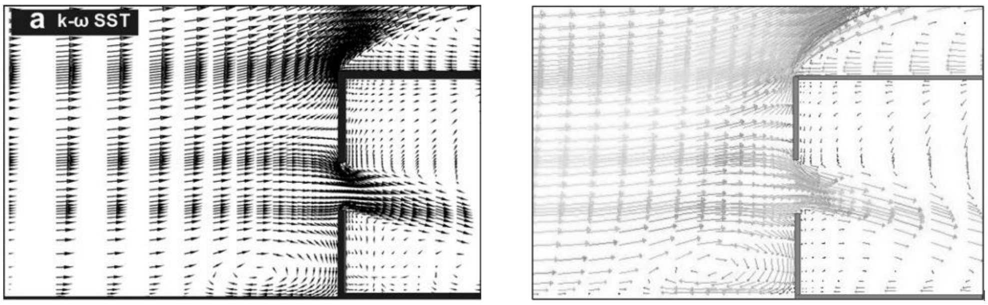
4.1. Preliminary Result

The results in this study were compared to the experimental data obtained from PIV measurements for the streamwise wind speed ratio U/Uref along the centerline at h = 0.04 m with the reference case by Ramponi and Blocken [2].



(a) The reference case [2] (b) Ansys Fluent

Figure 5. Velocity contour



(a) The reference case [2] (b) Ansys Fluent

Figure 6. Velocity vector

The figures showed the agreement of flow trend along the numerical solutions, both in the outdoor wind flow and the indoor air flow.

From the solution data, it is shown that a good agreement is obtained in the front face and behind face of the building. When the wind comes inside the building, the results from current work were slightly higher than the PIV data. Among them, the FINE mesh leads to the best on with the same parameter settings.

In the reference case [2], author showed that there is an extensive generic sensitivity of setting parameters that contribute to improved accuracy, reliability and evaluation of coupled CFD simulations for cross-ventilation assessment.

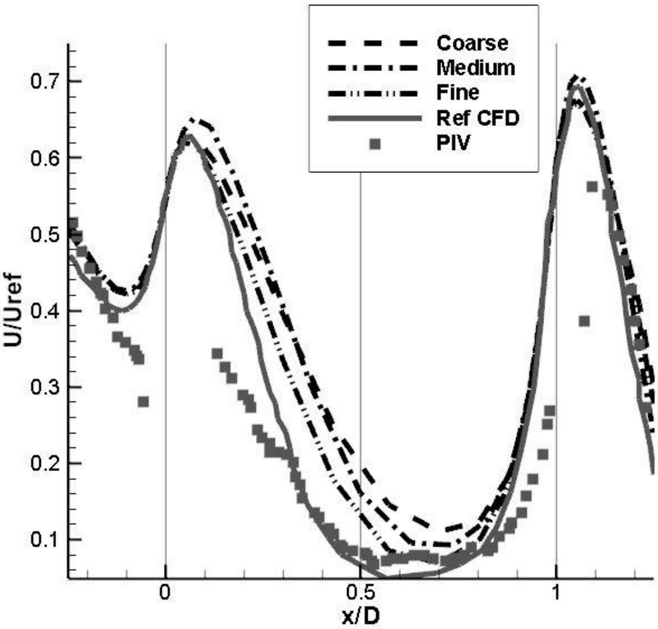
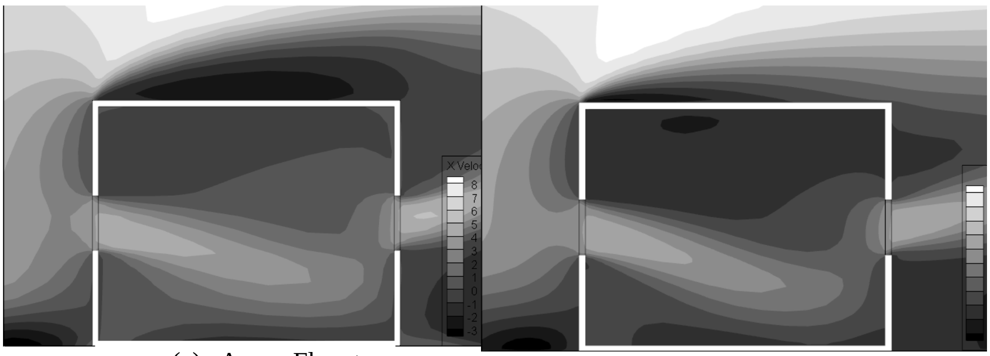


Figure 7. Comparision of experimental (PIV) and numerical results

4.2. Comparion of results in Ansys Fluent and OpenFOAM



(a) Ansys Fluent (b) OpenFOAM

Figure 8. Velocity contour comparision

The figures show a agreement between 2 CFD softwares Fluent and OpenFOAM in terms of flow trend and wind distribution around the building and inside the building also.

However, to illustrate how different they are; Let’s investigate the solution of velocity variable in the *x-direction* over the centerline of the opening throung the house.

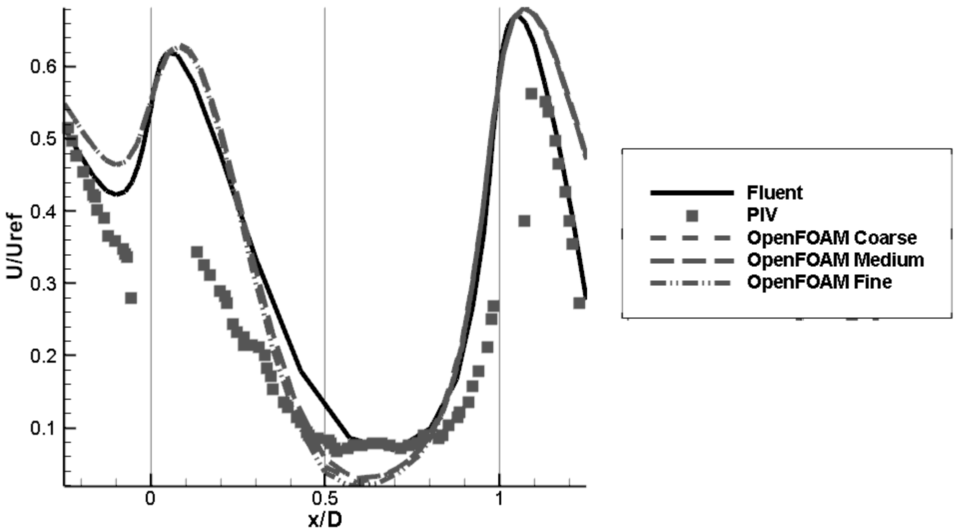


Figure 9. U/U_{ref} ratio at the centerline

Overall, the discrepancies by various models is in-considerable in OpenFOAM.

In the other hand, the Fluent result in the previous subsection indicated the discrepancies by $k-\varepsilon$ models are very large inside the house and the parameters in setting for a simulation are very sensitive while they are quite stable in the open source platform.

All simulations ran on an Intel Xeon (R) @ 2.4 GHz CPU - desktop without parallelization with calculation time:

Table 1. Computation Time

Mesh	**Ansys Fluent**	**OpenFOAM**
Coarse (549853 cell)	9 hours 45 mins	11 hours 50 mins
Medium (563451 cell)	13 hours	12 hours 10 mins
Fine (577049 cell)	15 hours 10 mins	12 hours 10 mins

It is obvious that OpenFOAM takes the same time for 3 levels of refinement mesh to calculate. The above table shows that Fluent solver takes more 3 hours than OpenFOAM to run fully 10000 iterations for the fine mesh case.

However, it should be mentioned again that Open Source CFD OpenFOAM is completely free of charge. Therefore, it shoud be prioritized in terms of computational cost.

5. Conclusion

In conclusion, both Ansys Fluent and OpenFOAM solve this problem with good solutions.

With equivalent setting, OpenFOAM calculates these equations faster about 3 hours/15 hours (20%) for the fine mesh with fully 10000 iterations. This shows that OpenFOAM is more effective in terms of computing time. On the other hand, these solutions from Ansys are converged after under 7000 iterations for the coarse and medium mesh in the meantime OpenFOAM does not. Therefore, the sensitivity in mesh refinement of Fluent is higher.

Furthermore, setting parameters are very sensitive in Fluent. As we can see from [2], a small change in parameter setting can lead to a much different solution while the open source flatform OpenFOAM gives stable solutions with different turbulence models.

It should be noted again that OpenFOAM is distributed for free. Users base on existing cases and modify to get their own issues. In different circumstance, CFD engineers have to pay for commercial license to use the Fluent Module in Ansys Software.

In academic environment, students can run CFD simulations on Ansys Fluent with "ACADEMIC" version that is limited the number of cells for a computational domain.

References

[1] A. Prieto, U. Knaack, T. Klein, and T. Auer, "25 Years of cooling research in office buildings: Review for the integration of cooling strategies into the building façade (1990–2014)," *Renewable and Sustainable Energy Reviews*. 2017.

[2] R. Ramponi and B.Blocken, "CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters," *Build. Environ.*, vol. 53, pp. 34–48, 2012.

[3] W. Frei, "Which Turbulence Model Should I Choose for My CFD Application?, https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/, 2017.

[4] J. J. F. J.C Kaimal, "Atmospheric boundary layer flows: Their structure and measurement," *Oxford Univ. Press. New York, NY.*, 1994.

[5] H. Richards, "Appropriate boundary conditions for computational wind engineering models using the k-turbulence model.," *J Wind Eng Ind Aerodyn*, pp. 145–153, 1993.