DEVELOPMENT OF CUSTOMIZED INDOOR AIR SIMULATOR USING OPEN SOURCE LIBRARIES

ILYOUP SOHN[1], HYUNSUK ROH[2], JAESUNG KIM[3]
[1]*Korea Institute of Science and Technology Information, ilyoupsohn@gmail.com*
[2] *NextFOAM Inc., hsroh@nextfoam.co.kr*
[3] *Korea Institute of Science and Technology Information, jaesungkim@kisti.re.kr*

Keywords: *CFD, OpenFOAM, Indoor Air, FEMM*

A customized indoor air flow simulator to perform computational thermo-fluid dynamic simulations in a closed indoor space is presented. The purpose of this system enables engineers in architectural, HVAC mechanical systems used in housings and buildings to perform CFD simulation for the initially designed space, therefore, all functions and options in this simulator are developed, especially for non-CFD experts or engineers. CFD mesh generation has been performed by using a triangular mesh generation module provided by an open source code for electro-magnetic computation tool, FEMM (Finite Element Method Magnetics) as shown in Fig. 1. One of major characteristics of the presented simulator is that users can directly import a floor plan 2D-CAD data and the 2-D triangular mesh can be automatically generated with a variable size. Figure 2 shows that a full 3-D mesh is constructed along the vertical direction by extruding 2-D mesh into 3-D with a ceiling height which users put in through OpenFOAM mesh generation utilities. Three types of boundary conditions such as air supply, exhaust and wall can be specified. After setting up the boundary conditions and mesh generation, aerodynamic calculations using OpenFOAM solver is performed. As shown in Fig, 3, the simple flow results such as velocity, temperature and pressure of indoor air are obtained. In addition to basic flow properties, an air-age is calculated. In this paper, the whole development processes are presented and simple flow results for sample problems using the developed system are also discussed.

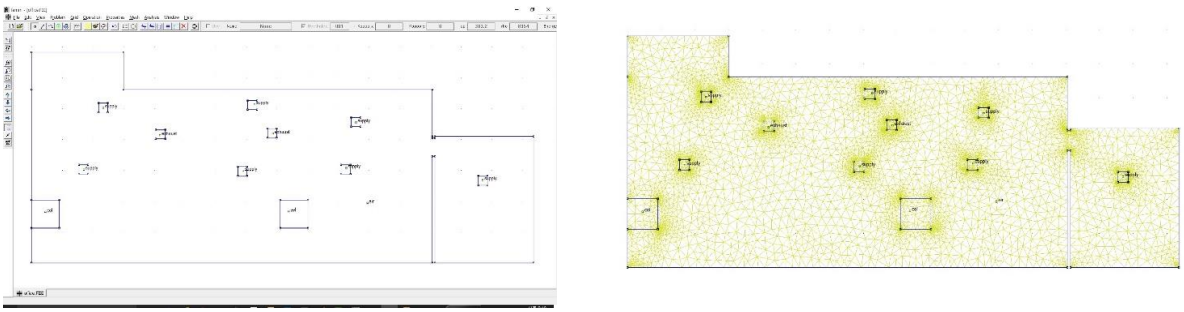

Figure 1: Setting up 2D floor plan (left) and 2-D triangular mesh (right)

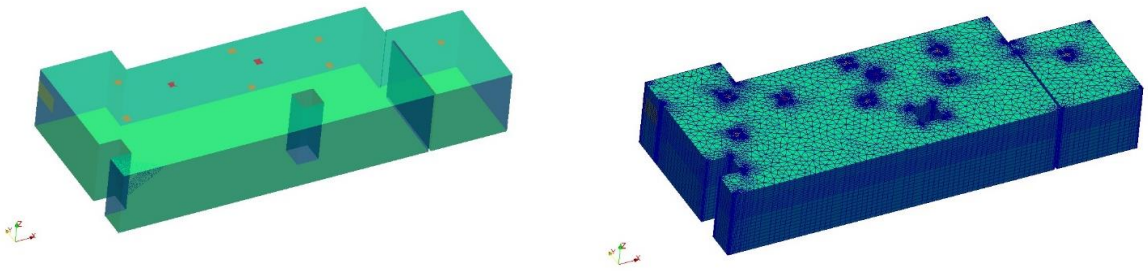

Figure 2: Extruding of 2D floor plan (left) to 3-D and 3D mesh generation (right)

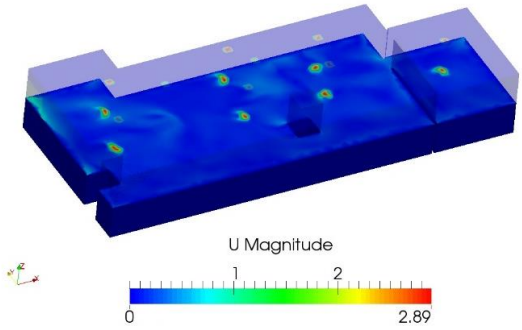

Figure 3: Velocity magnitude distribution computed by OpenFOAM for a sample floor plan

Acknowledgements
The authors are thankful for financial support from Korea Institute of Science and Technology Information (KISTI).

References

[1] Park, J. K. and Kang, K. H., 2010, "Review on OpenFOAM - An Open Source Software", *Journal of Computational Fluids Engineering*, Vol 15. No. 3, pp. 46~53.
[2] Kim, T., Oh, S. and Yee, K., 2011, "Verification of the Open Source Code, OpenFOAM to the External Flows", *Journal of the Korean Society for Aeronautical and Space Sciences*, Vol. 39, No 8.
[3] Sohn, I. and Hwang, J., 2015, "Trend of Computational Thermo-Fluids Analysis and Applications of Supercomputing CFD", *CAD/CAM Review*, Vol.21, No. 1, pp. 53~59.