10th OpenFOAM Conference

Simulations of a centrifugal fan at different flow conditions using OpenFoam and comparison with commercial packages

Mohammad Moshfeghi[a], Reza Chamkalani[b], Mark Crasto[a], Gavin Tabor[c], David Moxey[d]

a. Torin-Sifan Ltd

b. Computational Thermo-Fluid Dynamics Lab, University of Tehran, Iran

c. CEMPS, University of Exeter

d. King's College London.

OpenFoam is extensively used not only by academic researchers but also in industry, due to its open-source nature and zero licensing cost. However it is often thought that OpenFoam is less robust than commercial CFD codes, and that the user must of necessity be an expert to be able to drive the code. There are also questions about the effects of mesh quality on the results obtained by OpenFoam, in particular on lower quality meshes which might be used for rapid turn-around industrial simulation. Finally, a crucial question is how comparable OpenFoam results are with those obtained from commercial packages. The present research aims at shedding light on these questions for a centrifugal fan.

The computational case selected for the comparison here is that of a centrifugal air conditioning fan for which we have experimental data for comparison. Moderate size meshes were developed for this case of a size and quality likely to be used for an industrial application aiming at fast performance analysis of a fan. Two meshes, one generated by the FLUENT tetrahedral mesh generator and one with T-Rex in Pointwise, were solved by the OpenFoam using steady state numerics and the SST-k-ω turbulence model. Additionally, the FLUENT tetrahedral mesh and STAR-CCM polyhedral meshes were solved by those packages using the same turbulence model. Some other factors such as using periodic boundaries versus a complete model, and the length of the inlet duct were also studied. In all of the meshes, each mesh used approximately 200k cells.

The outcomes demonstrate that despite all the differences, the numerical results are very consistent between the different codes. All the CFD results showed a systematic under-prediction of the experimental results but this was at maximum an under-prediction of only 10% lower than the test results. The differences between the different CFD results were less than 5%, which is a promising conclusion for using OpenFoam for industrial use.