ParCFD'2019
31^{st} International Conference on Parallel Computational Fluid Dynamics
May-14-17 2019, Antalya TURKEY

PARALLEL CFD MODELING OF COLD FLOW IN GAS CIRCUIT BREAKERS

Saleh K. ABUHANIEH† **and Hasan U. AKAY***

†Atilim University
Modeling and Design of Engineering Systems Ph.D. Program
06830 Ankara, TURKEY
e-mail: abuhanieh.sksaleh@student.atilim.edu.tr, Web page: http://www.atilim.edu.tr/en

*Atilim University
Department of Mechanical Engineering
06830 Ankara, TURKEY
e-mail: hasan.akay@atilim.edu.tr, Web page: Web page: http://www.atilim.edu.tr/en

Key words: Gas Circuit Breaker, OpenFOAM, Dynamic Mesh, Compressible Flow, Parallel Computing

Abstract. A CFD model for cold flow of gas in an electrical circuit breaker has been created and validated using OpenFOAM. Static and dynamic mesh cases have been used and compared with the results obtained from the literature, where good matches have been observed. A parallel performance study has been performed to evaluate the scalability of the solver used.

1 INTRODUCTION

Circuit Breaker (CB) is an indispensable component for electrical power networks. It functions to open and close the electrical circuit mechanically at normal (load currents) for maintenance/operation and abnormal (short circuit currents) conditions during faults. Once the contacts start to separate, an arc plasma forms between the contacts, and the arc shall be extinguished (by cooling, lengthening or rotating the arc) within few power frequency cycles to declare a successful current interruption. Normally, CBs are classified according to the used medium for arc plasma extinguishing. For instance, Air Circuit Breaker (ACB), Vacuum Circuit Breaker (VCB), and Gas Circuit Breaker (GCB) where the commonly used gas for the last 40 years has been the SF6 gas. Due to the continuous development in numerical methods and computational resources over the decades, an accurate modeling for GCB is possible. Such a model can reduce the overall design and testing costs, and can enable the design optimizations. However, developing this model is difficult due to the multi-physics involved. Modeling the cold flow (without arc plasma) inside a GCB can be a good starting point towards the complete model since the switching performance is directly related to the pressure difference/elevation which is generated by a puffer system [1]. OpenFOAM which is an open source C++ FVM library for CFD [2] and multidisciplinary problems forms a suitable platform for this kind of modeling.

In this work, the foam-extend fork version 4.0 of OpenFOAM [3] has been used since the dynamic mesh library which supports the topological changes in the mesh is readily available. In this paper, the authors started from the thesis outlook of Wüthrich in [4], where a cold flow for GCB has been modeled using the sonicTurbFoam solver.

In this paper, first, a verification for the results has been obtained using static mesh. This is explained in section 2. In section 3, a dynamic mesh has been used instead of using different mesh for each plug position. The parallel performance has been explained in section 4. Finally, a conclusion has been presented in section 5.

2 USING STATIC MESH

As pointed out earlier, the solver in reference [4] is no longer in use. After testing several compressible flow solvers of OpenFOAM; the transient, turbulent and pressure-based solver **rhoPimpleFoam** has been selected.

Static, unstructured, hexahedrons-dominant and axi-symmetric meshes with 27k, 55k and 110k cells have been produced. The geometry outline is shown in Figure 1, where the plug has been positioned at x = 42 mm. The boundary and initial conditions used in [4] have been used. For instance, at the inlet, time varying boundary condition (from 1 bar to 3 bar in 25 ms) for pressure has been used and zero gradient for the velocity field is specified. The Mach number starts from zero everywhere and reaches 2.1 between the plug and the nozzle (x = 77 mm) after 10 ms for the most coarse mesh with 27k cells.

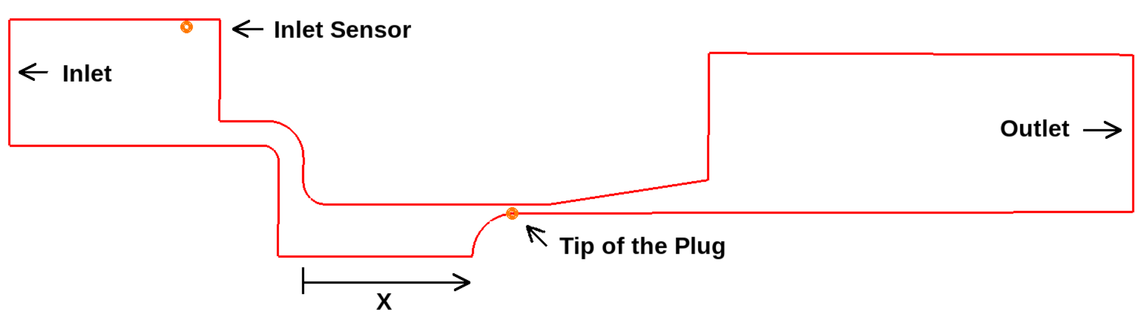

Figure 1: The geometry outline for the static mesh at plug position x = 42 mm

A comparison between the results in [4-5] and the ones presented in this work is shown in Figure 2, where the normalization of the pressure (measured at the tip of the plug and referred as sensor 3 in [4-5]) at plug position x = 42 mm is plotted as a function of the inlet pressure (measured at the inlet sensor location). The tip of the plug is normally where the maximum electric field appears. At the same time, the pressure and density are at their minimum values [5]. Thus, this position is a very probable place for dielectric breakdown. Changing the pressure at the inlet with respect to time is a simplification for the puffer system in GCB, where a piston-like system attached to the mechanical sliding shaft is responsible for changing the pressure dynamically.

Despite all the efforts exerted by the authors to reconstruct the original geometry from

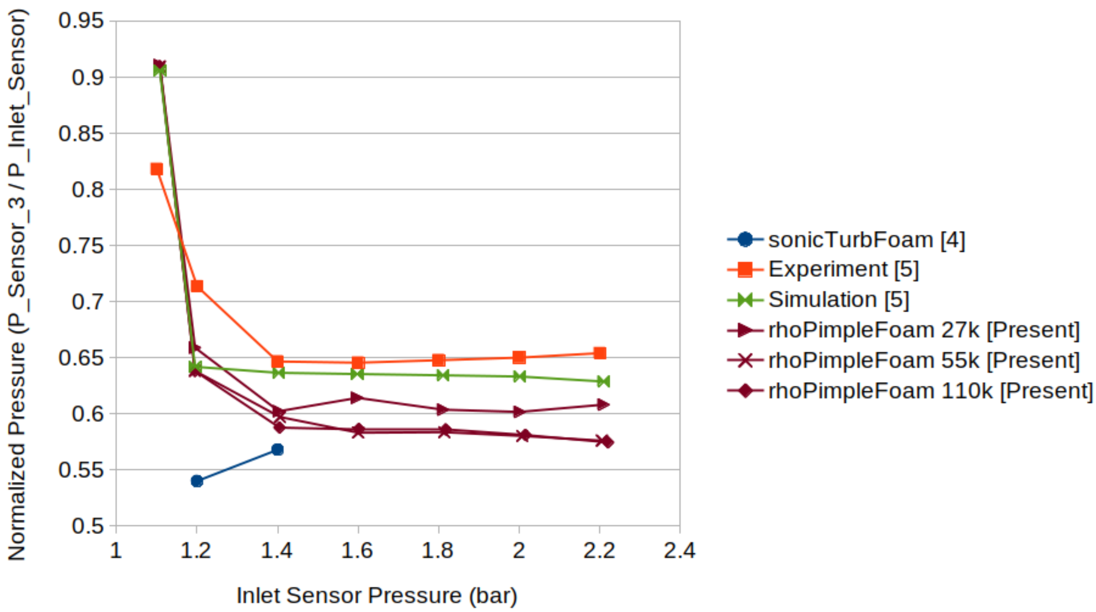

Figure 2: Comparison for the normalized pressure versus the inlet pressure at plug position x = 42 mm for different mesh sizes

the literature, it seems a deviation in geometries used still exists, due to curvatures and narrow passages occurring in the geometry. This may explain the observed shift in the results between the present study and [5].

3 USING DYNAMIC MESH

In the static mesh case, evaluating the pressure profile (or any other field) at each plug position requires preparing a new mesh. Starting from the outlook of [4], dynamic mesh which includes the whole range of plug positions (37 mm - 112 mm) has been prepared. The mesh at the first plug position (37 mm) has been divided into two meshes, the upper mesh which doesn't contain any moving cells, whereas the moving cells zone is located in the lower mesh. Left and right face zones bounding the moving cell zone have been assigned. The used geometry is explained in Figure 3. A complete column of cells are added to the left zone and removed from the right zone by the algorithm during the movement of the moving region. The interface between the two meshes has been considered as GGI (Generalized Grid Interface [6]) patch to couple the non-conformal two meshes during movement.

The used solver in the static case (rhoPimpleFoam) has been converted to a dynamic solver (rhoPimpleDyMFoam) considering the mesh motion and flux correction. Then **rhoPimpleDyMFoam** has been used to solve the flow. As in the static case, three different sizes of the mesh have been evaluated. However, the pressure at the inlet is fixed. The flow was evaluated at different inlet pressures (from 1.1 to 2.2 bar). A good match with both the experiment and the simulation results in [5] can be observed in Figure

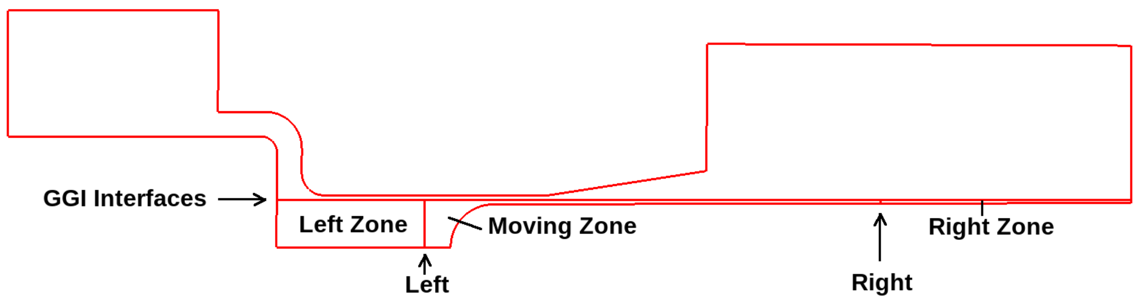

Figure 3: Dynamic mesh geometry

4. It's even better than the static case results, that may be due to keeping the pressure constant with time at the inlet in the dynamic case. It can be noticed that the presented solutions are getting closer as the element sizes decreases which indicates a convergence of the solver.

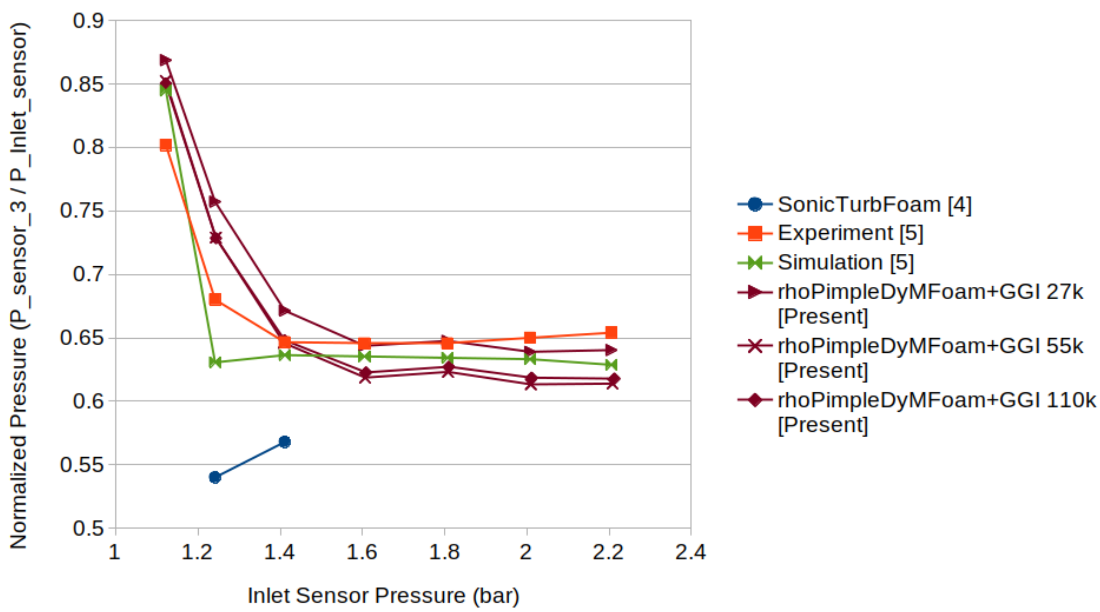

Figure 4: Comparison for the normalized pressure versus pressure at plug position x = 42 mm for different mesh sizes

4 PARALLEL PERFORMANCE

For the static and dynamic cases, a mesh with around 1 million elements has been considered, starting from 1 core up to 128 cores with maximum 16 cores per node for 1000 iterations. Speedup and efficiency values have been shown in Figure 5. All the runs have been executed at the same cluster of TRUBA infrastructure [7]. The general recommendation from the OpenFOAM community not to use the Hyper-Threading has been followed since it may drop the performance. The same has been observed by the authors. Thus, only physical cores have been used. For domain partitioning, the scotch method [8] which uses mainly the Dual Recursive Bi-partitioning algorithm has been selected since it minimizes the number of common element faces between the processors. This reduces the communication overhead and normally gives the best performance for the given conditions. The notable difference in scalability can be easily observed especially after the 16-processors run. The reason for that is the imposed constraints which need to be satisfied by the scotch method for the dynamic mesh. For instance, both GGI patches shall be on the same processor. This clearly increases the communication overhead and the load unbalance between the processors. For example, the maximum load unbalance is 16% for 8 processors and 54% at 16 processors.

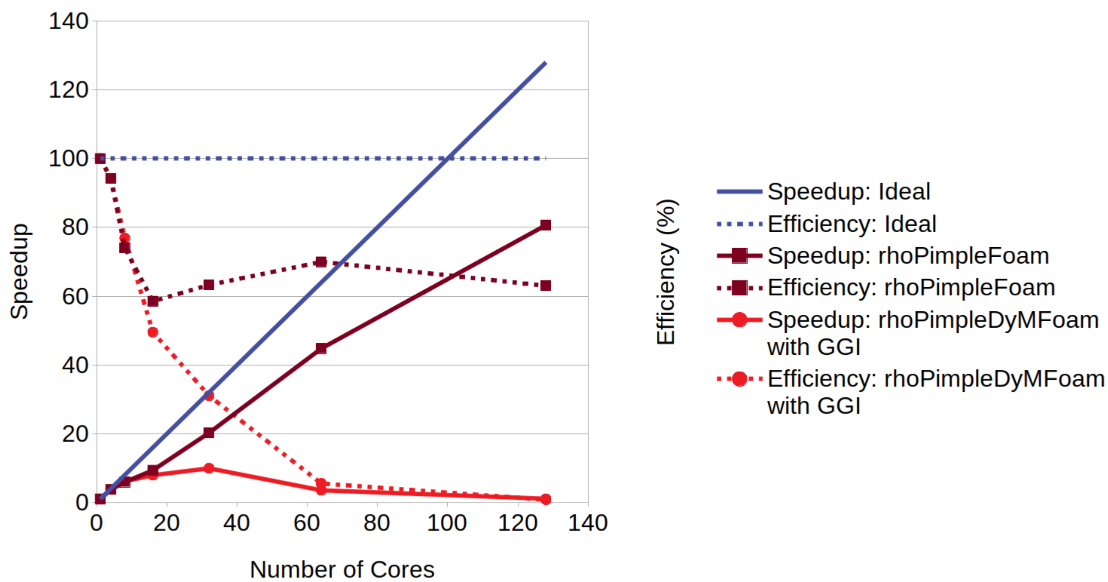

Figure 5: Speedup and efficiency plots for the static and dynamic meshes

5 CONCLUSION

In this work, a CFD model for a GCB using static and dynamic meshes has been presented using the rhoPimpleFoam solver of OpenFOAM. The obtained results showed a good match with the experiment in the literature. A parallel performance study has been performed using up to 128 cores. The results showed a reasonably good parallel scalability for a relatively small size mesh in the static mesh case. For the dynamic mesh case, the flow results were more accurate than the static mesh with comparison to the experiment, however, the scalability needs improvement which can be covered in a future work along with using piston to elevate the pressure instead of changing the pressure boundary condition value at the inlet.

ACKNOWLEDGMENT

The numerical calculations reported in this paper were partially performed on the resources of TUBITAK's ULAKBIM High Performance and Grid Computing Center.

REFERENCES

[1] N. S. Støa-Aanensen, M. Runde, E. Jonsson and A. D. Teigset. *Empirical Relationships Between Air-Load Break Switch Parameters and Interrupting Performance*. in IEEE Transactions on Power Delivery, vol. 31, no. 1, pp. 278-285, (Feb. 2016).

[2] https://www.openfoam.com

[3] https://sourceforge.net/p/foam-extend

[4] Benjamin Wüthrich. *Simulation and validation of compressible flow in nozzle geometries and validation of OpenFOAM for this application (Ph.D. thesis)*. ETH, Swiss Federal Institute of Technology Zurich, Institute of Fluid Dynamics, (2007).

[5] J. D. Mantilla and C. M. Franck and M. Seeger. *Measurements and Simulations of Cold Gas Flows in High Voltage Gas Circuit Breakers Geometries*. Conference Record of the 2008 IEEE International Symposium on Electrical Insulation, pp. 720-723, (June 2008).

[6] Martin Beaudoin and Hrvoje Jasak. *Development of a Generalized Grid Interface forTurbomachinery simulations with OpenFOAM*. Open Source CFD International Conference 2008, Berlin, Germany. (4th/5th December 2008).

[7] https://www.truba.gov.tr

[8] F. Pellegrini and J. Roman. *Scotch: A software package for static mapping by dual recursive bipartitioning of process and architecture graphs*. Proc. HPCN96, pp. 493-498, April. 1996.