Simulation of Supersonic Flow in High-voltage Circuit Breakers

Y. Qiao[1], S.Y. Matharage[1] and Z.D. Wang[1]*
[1]Centre for Smart Grid, Department of Engineering, University of Exeter, Exeter, EX4 4PY, UK
*E-mail: zhongdong.wang@exeter.ac.uk

***Abstract--* High-voltage circuit breakers (HVCB) are key protection equipment in the power system. Modelling and simulation are key parts of the development process of the HVCB, which can be used to study design improvements and also the impact of alternative gases. The flow pattern in the nozzle area is an important part to investigate the arc quenching process.**

The first stage of typical simulation includes the 'cold flow simulation' during which flow is simulated without simulating the arc. This paper investigates the circuit breaker modelling strategies and implementation of the cold flow simulation through computational fluid dynamics simulation.

Background & Main Results

Gas-insulated high-voltage circuit breakers are used in the power network to protect the equipment and to isolate the system. SF6 is the most commonly used gas insulation due to its excellent dielectric strength and arc extinguishing capabilities [1]. However, due to the global warming potential, the power industry is in search of alternative environmentally friendly gases. Modelling and simulation are key parts of the circuit breaker development process which helps not only to improve the structure of the circuit breaker but also to study the impact of alternative gases [2].

In this paper, a circuit breaker model was first built based on an experimental setup available in the literature [1, 3]. The nozzle is designed to obtain Mach 1 at the throat, and hence a supersonic flow occurs in the expanding region after the smallest cross-section of the outflow path [4]. The developed model was used to simulate the cold flow scenarios at different inlet pressure values. The simulation showed the occurrence of supersonic flow in the nozzle area and other related effects such as the wake effect and the presence of shocks in the nozzle area.

Furthermore, the velocity increase with the inlet pressure and the position of maximum velocity moves downwards to the nozzle outlet. Because of the existence of the shock, the velocity drops in front of the downstream electrodes and then increases slightly. The increasing extent is related to the position of the downstream electrode and inlet pressure.

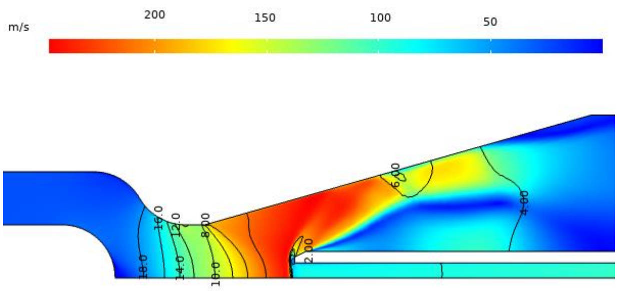

Figure 1: Velocity distribution with pressure contour, SF6, inlet pressure (P0) is 21.4 atm, outlet pressure is 0.25*P0

Conclusions

A circuit breaker model developed based on an experimental setup was used to simulate the supersonic flow patterns. Simulation of the cold flow indicated the shock formation in front of the downstream hollow electrode and a wake near the tip of the upstream electrode. The results indicated that inlet pressure and electrode distance are key impact parameters that influence flow patterns.

References

[1] G. F. D. M. Benenson, R. E. Kinsinger, H. T. Nagamatsu, H. O. Noeske, R. E. Sheer. Jr, "Fundamental investigation of arc interruption in gas flows," General Electric Company, New York, EPRI-EL-1455, Jul 1980.

[2] M. Eves, "A literature review on SF6 gas alternatives for use on the distribution network," Western Power Distribution, Jan 2018.

[3] J. A. R. G. Frind, R. E. Kinsinger, R. D. Miller, H. T. Nagamatsu, H. O. Noeske, "Fundamental investigation of arc interruption in gas flows," General Electric Company, New York, EPRI-EL284, Jan 1977

[4] M. T. C. Fang, S. Kwan and W. Hall, "Arc-shock interaction inside a supersonic nozzle," in IEEE Transactions on Plasma Science, vol. 24, no. 1, pp. 85-86, Feb 1996.