GAS FLOW ANALYSIS IN LOW ENERGY ARC PUFFER INTERRUPTERS

Elham ATTAR
ABB – Norway
elham.attar@no.abb.com

Patrick STOLLER
ABB CRC – Switzerland
patrick.stoller@ch.abb.com

Michael SCHWINNE
ABB CRC – Switzerland
michael.schwinne@ch.abb.com

Pål SKRYTEN
ABB – Norway
pal.skryten@no.abb.com

Nitesh RANJAN
ABB CRC – Switzerland
nitesh.ranjan@ch.abb.com

Benjamin WUETHRICH
ABB CRC – Switzerland
benjamin.wuethrich@ch.abb.com

Tom Rune BJØRTUFT
ABB – Norway
tom-rune.bjortuft@no.abb.com

Ole GRANHAUG
ABB – Norway
ole.granhaug@no.abb.com

ABSTRACT

Puffer interrupters, which use the gas blast produced by relative movement of a piston and a cylinder, are widely used due to their compact size, simple structure and excellent interruption properties.

For puffer technology, the interruption capability depends on the interaction between arc, gas flow, pressure build up, nozzle material and geometry. The interrupter is divided into two main compartments with the same initial pressure. During arc interruption, the gas in the arcing zone is heated resulting in higher pressure. The pressurized gas from the other compartment blasts into the arcing zone and helps to extinguish the arc. Puffer design plays an important role in determining puffer pressure build up.
In this paper computational fluid dynamics (CFD) simulations are used in order to understand the complex phenomena inside puffer interrupters during switching of medium voltage load currents.

Two different nozzle geometries have been investigated using CFD modeling. In order to validate the simulation results, experiments have been conducted in a high power laboratory and compared with simulation.

Understanding the gas flow and pressure build up during current interruption helps to develop a puffer interrupter that provides higher performance with better reliability in a more compact size. Optimizing the nozzle geometry and controlling the leakage rate inside the puffer breaker result in higher pressure build up and better interruption performance.

INTRODUCTION

A puffer interrupter uses compressed gas to build up pressure to generate gas flow that extinguishes the arc. After contact separation, current is carried through an arc and this arc has to be controlled and cooled down by a gas blast with sufficient intensity. The gas is used both as an arc interrupting and a dielectric medium. The arc is extinguished at the first current zero crossing after sufficient contact distance to withstand the transient recovery voltage is achieved. The process is improved by pushing cold gas through the nozzle and the arcing zone, replacing hot gas with cold gas.

The puffer interrupter, which is shown in figure 1, consists of a compression chamber, an interrupting chamber, a piston and a refill valve which is fitted on the piston. The principle of operation is to "blow out" the arc by using the overpressure generated by compression of the gas obtained by the piston movement.

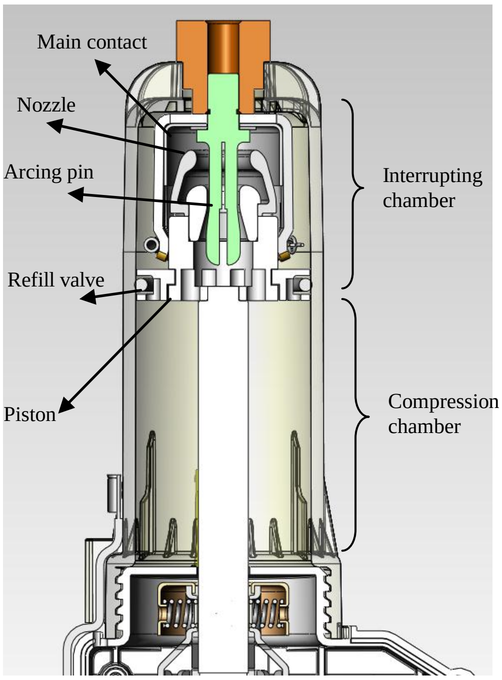

Figure 1: Schematic of the gas puffer interrupter

This paper presents a numerical model used to simulate the gas flow inside puffer interrupters during breaking operation. Although the current interruption in a puffer interrupter is rather complex and involves many different physical phenomena, better understanding of the gas flow and associated pressure build up helps to predict its performance.

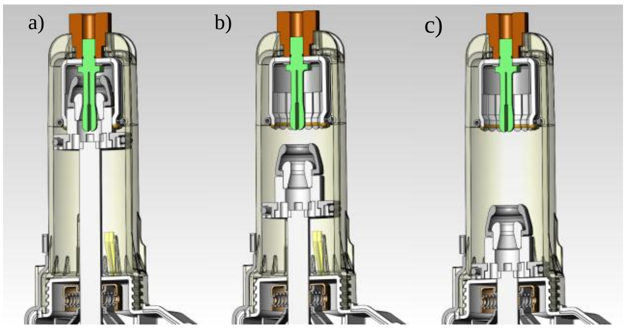

Figure 2: Gas puffer interrupter during the opening operation

During the opening operation, the piston is displaced from the closed position (figure 2, a) to the open position (figure 2, c). As the piston moves through the displacement stroke, the gas is compressed and closing of the refill valve leads to isolation of the interrupting chamber from the compression chamber and hence results in an overpressure in the compression chamber (figure 2, b). The arc extinction should preferably occur at the first zero crossing, and enough gas pressure is needed to blow out the arc. The puffer design influences the gas flow and pressure build up in the system and will result in different interruption capabilities.

In puffer interrupters, the overpressure necessary for breaking can be improved through optimal design of the nozzle and by minimizing the leakage. For this purpose two nozzles, with different nozzle diameters (the narrow nozzle and the wide nozzle), have been investigated and modeled using computational fluid dynamics (CFD) techniques. The numerically predicted results are compared with experimental data obtained at the NEFI high power laboratory.

COLD GAS SIMULATION

Figure 2 illustrates the opening process of the investigated puffer interrupter. The complex geometry and moving parts make numerical modeling time consuming. Therefore, for the gas flow analysis, a 2D model is solved by using Fluent CFD software.

For the purpose of numerical simplicity, a rotationally symmetric simulation geometry is assumed (figure 3) and piston motion is modeled by using the layering functionality of Fluent. The piston speed is extracted from the measured travel curve.

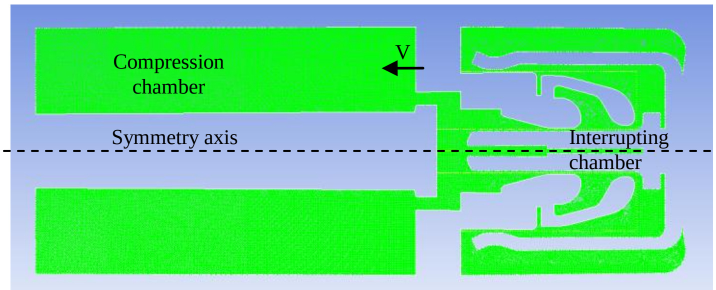

Figure 3: 2D geometry for CFD simulation

During the opening operation, the gas is compressed by a relative movement of the piston and the cylinder.
Figure 4 shows an example of pressure distribution in the puffer interrupter during the opening operation for the narrow nozzle.

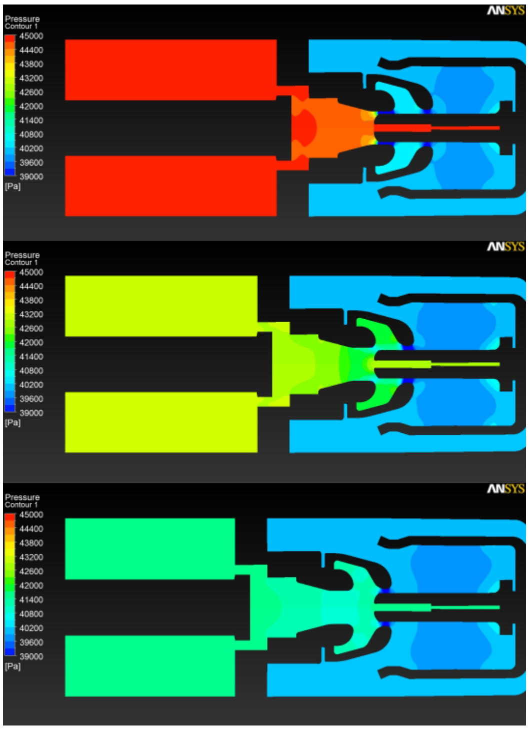

Figure 4: Pressure build up in puffer interrupter during the opening operation for the narrow nozzle

For a realistic pressure build up simulation the leakage in the compression chamber has been considered. For the narrow nozzle case, different gap sizes between the piston and the hole through which it passes are simulated and the results are shown in figure 5.

According to these graphs, simulation results for an 0.7 mm gap, which corresponds to a leakage area of 42.22 mm^2, show similar pressure build up in the compression chamber as the experiments.

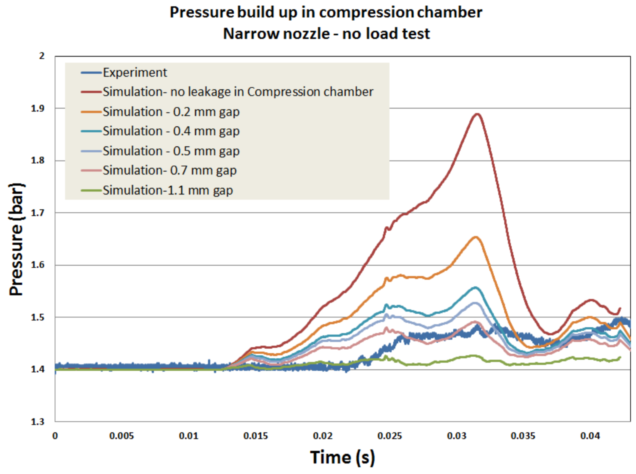

Figure 5: Pressure build up with different gap sizes for narrow nozzle

Comparing the pressure build up of the experiment and simulation in figure 4 shows that there is a time lag between the first rise in pressure, which may be due to the design of the refill valve. The refill valve is designed so that an O-ring is used to stop the gas flow between the compression chamber and the interrupting chamber during the opening operation just after 2mm stroke.

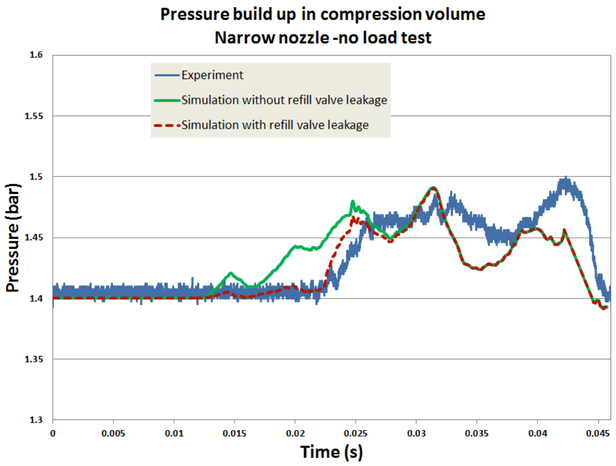

Figure 6: Simulated pressure build up with different refill valve leakage settings compared to experiment for the narrow nozzle case

In order to investigate the delayed pressure build up in the experiment compared to the simulation, it is assumed that the O-ring cannot stop the gas flow for the first 11-13mm of piston movement. To test this assumption a leakage area at the refill valve is considered. This leakage results in a delay in pressure build up in the simulations as shown in figure 6. Therefore, the performance of the O-ring seal can be considered as a possible reason for the delayed pressure build up in the experiments.

The same approach is applied for the wide nozzle case and the simulation results are presented in figure 7. In this figure the simulated pressure build up with no leakage in the compression chamber is also shown. Comparison of Figure 6 and figure 7 shows that the wide nozzle causes lower pressure build up in the compression chamber.

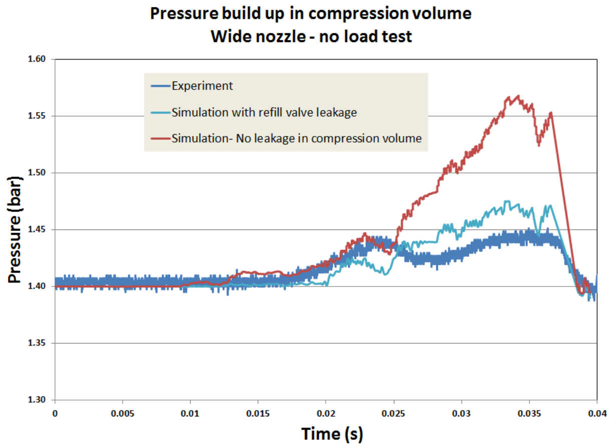

Figure 7: Simulated pressure build up with different refill valve leakage settings compared to experiment for the wide nozzle case

As the simulated pressure build up indicates, controlling the leakage rate inside the puffer breaker results in higher pressure build up during the opening operation, which can lead to better interruption performance.

LOAD BREAK SIMULATION

For design of a puffer interrupter with higher breaking capacity, the interaction between the arc and the gas flow must be investigated accurately. Many numerical techniques have been developed for this purpose; they combine CFD tools and arc models [1].

In this paper the arc model is implemented by using user defined functions in the Fluent software to simulate the arc physics for a gas circuit breaker with moving boundaries. During arc formation, gas conductivity changes from an insulator to an excellent conductor at high temperature across the contact gap. In this model the arc is heated up by Ohmic heat sources and cooled down by radiation, convection and conduction. The mass ablation rate at the surface of the nozzle is a function of the incident radiation [1, 2].

The tests with the narrow nozzle puffer interrupter were performed with opening operations under load current (with short and long arcing times). The measured pressures in the compression chamber are presented in figure 8, together with numerical simulation results.

As figure 8 makes clear, simulation predicts much higher pressure build up in the compression chamber compared to the measured values. It has to be kept in mind that this could be due to the arc model, developed for high voltage circuit breakers, which takes into account the ablation of the nozzle, a phenomenon probably absent in the load current puffer interrupter arc. The differences become more pronounced for longer arcing times.

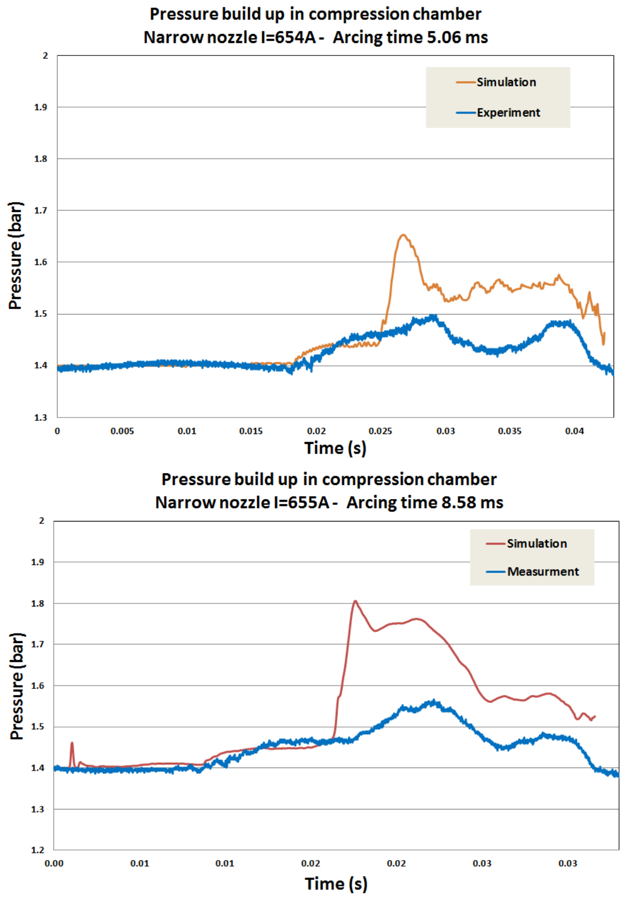

Figure 8: Pressure build up for load break tests with different arcing time

CONCLUSION

CFD models are valuable tools for investigating physical phenomena in puffer interrupters that are difficult to measure. In this study a systematic analysis for simulating the gas flow in puffer interrupters has been presented. The focus is on understanding the flow picture and pressure build up in a puffer interrupter during the opening operation. The cold gas simulation gives valuable indications for the optimization of such interrupting device. It was shown that optimizing the nozzle geometry and controlling the leakage rate inside the puffer breaker results in higher pressure build up and therefore, better interruption performance of the breaker.

An arc model developed for high voltage circuit breakers was used to simulate the interaction between the arc and the gas flow in the puffer interrupter. Simulation results show much higher pressure build up compared to experiments; the model has to be modified to correctly simulate load current puffer interrupter arcs. Developing a model to accurately simulate the physical processes in load current puffer interrupters is work that remains to be done.

REFERENCES

[1] R. Bini, B. Galletti, A. Iordanidis, M. Schwinne, Th. Werder Schläpfer, 2011, "CFD in Circuit Breaker Research & Development", Proceedings of the 1st International Conference on Electrical Power Equipment -Switching Technologies, 375–378.

[2] T. Christen, 2011, "Radiation and nozzle ablation models for CFD simulations of gas circuit breakers", Proceedings of the 1st International Conference on Electrical Power Equipment -Switching Technologies, 471–474.