Project 7 HOU Zhihao

This project aimed to simulate the wind field inside a fuel cell, specifically within its microchannels. It also required analysing the fuel cell's performance. Based on the project's requirements, I chose to use Fluent for the wind field simulation. The overall project process and content are summarized in the document **Wind field and performance analysis of air-cooled fuel cell system.pdf**, which is my undergraduate thesis and provides a detailed description of the experimental process and results, as well as unresolved issues and potential solutions.

1.Project Introduction

Simulating the wind field within the microchannels of a fuel cell is extremely challenging. This is because the fuel cell's products contain water, which can clog the microchannels and affect the fuel cell's further efficiency. Therefore, this issue requires careful and in-depth research. Due to the limitations of time and simulation technology at the time, the first step of this project was to simulate the internal wind field when the fuel cell was not operating. If conditions permit, we can then simulate the multi-physics field coupling of the thermal and wet chemical reactions during fuel cell operation, which will more closely simulate the actual operating conditions of the fuel cell. However, time was limited at the time, and I was the first person in my undergraduate supervisor's research group to perform this simulation. Due to time constraints, I only achieved the first step of the simulation.

2. Simulation Ideas

Since the computational complexity of this simulation was manageable, we used Fluent to perform a 3D modelling analysis, setting up pressure inlets and outlets for simulation. The simulation results were then analysed in longitudinal sections to analyse the wind field variations within the microchannels. However, fuel cells use sealed fans for suction, so the simulation results differ somewhat from the actual results. Since we only analysed the wind field simulation, the complexity was manageable.

3. Current Optimization Ideas

If conditions permit, I think the wind field simulation can be optimized according to the following steps: First, the fan can be physically modeled at the fan outlet, rather than simply setting the outlet surface as the fan action surface. Secondly, the inability to directly choose to set the suction fan function in Fluent is an inherent problem of the software. In fact, we can customize it through UDF files. In addition, the mechanical properties of the entire fuel cell can be increased throughout the process, such as thermal deformation, stress deformation and other factors. Finally, the water generated by the fuel cell reaction is a random process, and this process can also be studied and simulated in depth. The experimental process can be recorded by experimental equipment such as high-definition high-speed cameras, which can be used to verify the simulation results.

4. Summary and enthusiasm for the UNFoLD lab

This is the product of my undergraduate studies. Although immature, it marks a significant milestone for me and has been very beneficial. I was initially drawn to joining the UNFoLD lab by the dynamic video of vortex formation featured on the lab's homepage. The intuitive experimental conditions were captivating (which is exactly what I strive for: clearly presenting project results through visual videos). I continue to learn, constantly challenge myself, and truly enjoy these fascinating experiments. Of course, my years of scientific experience tell me that behind these videos lie countless nights of dedicated research. I hope to hone my theoretical application and problem-solving skills through practical projects. If I am fortunate enough to join the lab, I believe UNFoLD will inspire new ideas and enable me to achieve fruitful research results.