

ME-474 Numerical Flow Simulation

Case study: Flow and thermal regulation in a solid-oxide fuel cell

Fall 2021

A. Problem description

The fuel cell stack of interest consists of 72 identical modules (figure 1). Each module is comprised of a **cell** (with **cathode** and **anode** sides) sealed between **metallic interconnector plates** (“MIC” on figure 1). Each cell is oblong in shape, with 2 circular ends and 2 circular holes. Air and fuel are circulated across the cell **reactive area** on the cathode and anode side, respectively. The role of the airflow is two-fold: 1) provide oxygen for the electrochemical reaction with the fuel, and 2) evacuate the heat dissipated by the cell. It is proposed to undertake a preliminary study to improve the air distribution and thermoregulation pattern of the solid-oxide fuel cell stack based on simplified CFD analysis.

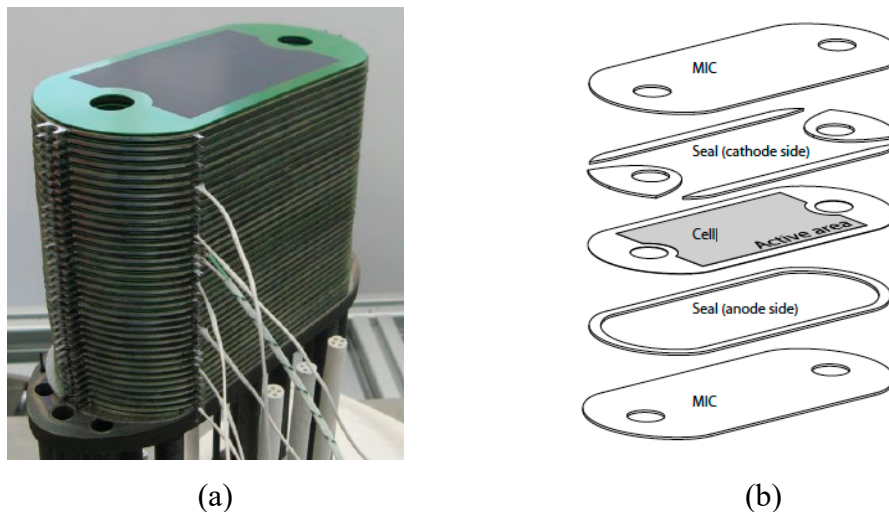


Figure 1: (a) A solid-oxide fuel cell stack and (b) expanded view of one cell module.

NOTE: The present preliminary study concerns only the cooling of the fuel cell by the airflow; the generation of electrical power through the reaction of fuel and air within the fuel cell is not treated, and knowledge of this process is not required.

For this preliminary study, it is suggested that you consider a steady flow simulation for a simplified 2D cross-section of the thin (1 mm) pattern on the cathode side of a cell, as shown in figure 2. The pattern is 24 cm long and 8 cm wide. The cooling air enters via the left inlet hole and exits from the right outlet hole (both of 2 cm diameter). The complex design of the cell reactive surface can be approximated for the present study by 3 identical rectangular inserts (of length 10 cm and width 1 cm) separating 4 airflow channels (shown in dark grey). For this simplified 2D geometry, it is proposed to model the heat generated at the reactive surface by a heat source in the airflow channels themselves.

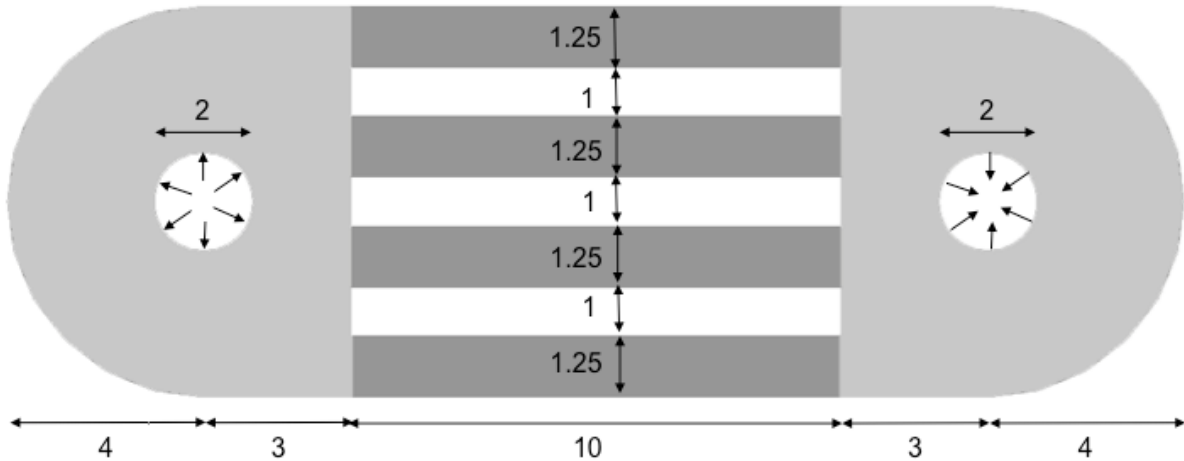


Figure 2: Schematic diagram of the simplified 2D section of the fuel cell. Grey areas are airflow regions, white areas are walls and inlet/outlet holes. Heat generation is in the dark grey areas. All measurements are in centimetres.

For the present study, the following high-temperature properties are to be considered:

- air density, $\rho = 0.34 \text{ kg/m}^3$,
- dynamic viscosity, $\mu = 4.30 \times 10^{-5} \text{ Pa s}$,
- specific heat capacity at constant pressure, $C_p = 1145 \text{ J/kg K}$,
- thermal conductivity, $k = 0.068 \text{ W/m K}$.

Boundary conditions are summarized below. For the standard operation of the fuel cell, a uniform airflow of 4 m/s (with a turbulence intensity of 10%) enters through the inlet in a normal direction. The temperature of the inlet air is 1000 K. A uniform pressure is assumed at the outlet. Due to the power dissipated in the fuel cell, the airflow channels are assumed to have a uniform heat generation rate of 1 MW/m^3 . The outer wall of the cell and the sidewalls of the 3 rectangular inserts are assumed to be adiabatic.

- uniform inflow velocity $V = 4 \text{ m/s}$,
- inflow turbulence intensity 10%,
- inflow temperature $T = 1000 \text{ K}$,
- uniform heat generation rate in the 4 flow channels 1 MW/m^3 ,
- outer cell wall and sidewalls of 3 rectangular inserts assumed adiabatic.

Gravitation and associated buoyancy effects, as well as radiation, are to be neglected.

A good choice of pattern design should provide a uniform airflow distribution and a suitable thermoregulation (temperature gradients as smooth as possible across the cell). The objective of this preliminary study is to assess the current design regarding these criteria and to suggest possible improvements.

For the present study, you are required to perform a numerical simulation of suitable accuracy, to extract from the solution physical insights of both qualitative and quantitative nature, and to propose improvements in the system design.

Follow the numerical simulation procedure described in Section B, and provide answers to the questions listed in Section C.

B. Numerical simulation procedure

B.1 Pre-processing

The computational mesh to be used to perform the flow simulation is provided in the file `Fuel_cell_mesh.zip` (available on the Moodle web site).

B.2 Flow computation

Import the mesh into Fluent, and perform any necessary initial checks and/or scaling.

Choose the appropriate physical modelling and a suitable numerical method for the problem.

Apply the boundary conditions described in Section A.

Monitor (plot and print) during the iteration procedure:

- the global residuals,
- the average temperature on the surface of the outlet.

Initialize the flow field and perform a suitable number of iterations until convergence is obtained.

Hint: in Fluent, heat generation can be specified as an energy source term as follows:

- In Cell Zone Conditions menu, select and edit the zone containing the channels called “fluid-channels”,
- Check the Source Terms option, and select the Source Terms tab,
- Edit the source term for Energy (W/m^3) at the bottom of the list (the energy equation must be activated first),
- Add 1 source, and set the given constant value for the heat generation rate.

B.3 Post-processing

Visualize contour plots of the computed flow fields (e.g. pressure, velocity, temperature) as well as the velocity vectors throughout the computational domain.

Plot streamlines indicating the flow throughout the fuel cell.

Plot the computed velocity profile along a suitably chosen line to provide additional indication of the flow behaviour.

Determine the average and maximum temperature on the surface of the outlet.

Calculate the temperature profile and average temperature along each of the 3 rectangular inserts.

B.4 Validation

Numerical flow solutions should be verified to gauge if they are mathematically and physically realistic. A detailed validation study will not be undertaken for the present preliminary study. Nevertheless, an estimation of the temperature difference ΔT between the inlet and outlet surfaces can be determined from the following equation:

$$Q = M C_p \Delta T, \quad (1)$$

where Q is the total heat input (J/s), M is the total mass flow rate (kg/s), and C_p is the specific heat capacity at constant pressure.

Hint: fluxes for a 2D geometry are determined from the corresponding 3D geometry that has the same cross-section and a suitably chosen depth. (The default reference value for depth in Fluent is 1 m.)

Determine the values of Q and M , and calculate from Equation (1) the difference in temperature between the inlet and outlet. Compare this value with that obtained from the numerical simulation.

C. Flow analysis

Based on the numerical simulation results, describe the behaviour of the flow in the fuel cell.

What do the values and distributions of temperature indicate about the effectiveness of the thermoregulation pattern? Use the computed values and profiles of temperature to support your comments regarding the design of the thermoregulation pattern.

What are the determined values of Q , M and ΔT ? How do the analytical and numerical simulation results compare? Discuss possible causes for any significant differences. More generally, provide details of how verification and validation procedures could be applied to the present problem. Discuss how the numerical simulation could be improved to increase the physical reality of the results.

How can the operation of the fuel cell system be improved? Based on the numerical simulation results, describe possible improvements to the pattern geometry, and the reasons for these modifications.