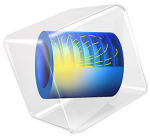


Nonisothermal PEM Fuel Cell

Introduction

This tutorial models the intercoupled electrochemical reactions, charge and species transport, and heat transfer in a polymer electrolyte membrane (PEM) fuel cell. For the gas flow fields, straight channels are used on the hydrogen anode side, whereas a mesh structure is used on the air cathode side. The cell is cooled by a cooling fluid, flowing in a separate channel. Periodic temperature boundary conditions are used for the top and bottom boundaries, thereby emulating a stacked cell configuration. Electroosmotic transport (drag) and permeation of water through the membrane is also included in the model.

Note: A Design Module license is required to construct the model geometry and to run the model.

The tutorial assumes that the reader is already fairly well acquainted with fuel-cell modeling in COMSOL Multiphysics. For a general introduction to fuel-cell modeling, see the Mass Transport and Electrochemical Reaction in a Fuel Cell Cathode tutorial, and for a detailed discussion on modeling of the membrane-electrode-assembly (MEA) of a PEM fuel cell, see the Transport Phenomena in a Polymer Electrolyte Fuel Cell Membrane-Electrode Assembly tutorial.

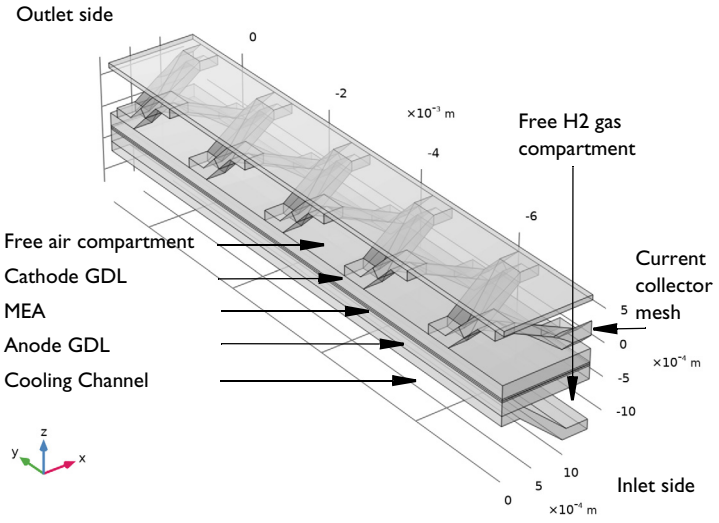


Figure 1: Model geometry.

MODEL GEOMETRY

Figure 1 shows the model geometry. The inlets for the humidified air and hydrogen gas streams, as well as the liquid cooling fluid is located towards the bottom right in the figure. The metal current collector on the cathode side is constructed from an extruded mesh, whereas a molded metal plate is used to form the straight gas and liquid cooling channels on the anode (hydrogen) side.

PHYSICS INTERFACES AND COUPLINGS

The model is defined using a number of different physics interfaces:

- The charge and species balances, reaction and gas phase thermodynamics and the electrochemical reactions are all defined by the use of a **Hydrogen Fuel Cell** (`fc`) interface. This interface also includes water permeation and electroosmotic drag through the membrane.
- The convective flow and pressure of the gas phases in the free gas compartments and the gas diffusion layers (GDLs) are defined by two **Free and Porous Media** (`fp`) interfaces, one for each gas mixture. These physics interfaces solve for the Navier-Stokes equations in the free gas domains, and the Brinkman equations in the GDLs.

- The convective flow and pressure of the liquid cooling fluid is defined using a **Laminar Flow** (`spf`) interface. This interface solves for the Navier–Stokes equations.
- The heat transfer and temperature of the cell is defined and solved for by the use of a **Heat Transfer** (`ht`) interface

The interfaces are intercoupled in a multitude of ways. The following **Multiphysics** nodes are used in the model to define various couplings:

- The **Reacting Flow** nodes, one for each gas phase, applies the convective velocities and pressures of the `fp` interfaces into the species transport equations and electrochemical reaction kinetics expressions of the `fc` interface. These coupling nodes also set the density and dynamic viscosity in the `fp` interfaces to the variables calculated by the `fc` interface.
- The **Electrochemical Heating** node applies the heat sources stemming from the electrochemical reactions and ohmic (joule) heating calculated by the `fc` interface into the `ht` interface. The node also sets the temperature in all `fc` domains to that of the `ht` interface
- The **Nonisothermal Flow** nodes, one for each fluid flow interface, couples the velocity field in the `fp` and `spf` interfaces to the fluid domains of the ht interfaces. The node also sets the temperature in all `fp` and `spf` domains to that of the `ht` interface

In addition, the fluid heat capacities and thermal conductivities of the gas domains in the `ht` fluid domain are set manually to the corresponding built-in domain variables calculated by the `fc` interface.

SYMMETRY AND PERIODIC BOUNDARY CONDITIONS

Symmetry is assumed in the *x* direction, with **Symmetry** (or **Insulation/No flux**) conditions used in all physics interfaces on the corresponding outer *yz*-planes of the model geometry.

For the ht interface, a periodic condition is used to intercouple the heat flux and temperature of the top and bottom *xy*-planes. In this way, a stacked cell configuration is modeled.

MATERIAL PROPERTIES AND OPERATING CONDITIONS

The inlet hydrogen and air gas streams are humidified to 85% at a temperature of 70°C. The gas phase properties were calculated using the built-in thermodynamic functions of the `fc` interface.

The cooling fluid is using the properties of the **Water** in the Built-In COMSOL Multiphysics material library. The inlet cooling temperature is 70°C.

The conductivity and membrane water transport properties are taken from **Nafion, EW 1100, Vapor Equilibrated, Protonated** in the Fuel Cell and Electrolyzer Material library.

The current collector and feeder domains are using the properties of the **Steel AISI 4340** material in the Built-In COMSOL Multiphysics material library.

Anisotropic thermal and electrical conductivity values are used for the GDLs. The anisotropic thermal conductivities were taken from Ref. 1. Due to the much higher thermal conductivity of the solid matrix of the GDLs, compared to the gas phase, the GDL domains are modeled as solids in the `ht` interface.

The remaining parameter values were arbitrarily chosen for tutorial purposes.

MESHING

A user-defined mesh is used in the model. A free tetrahedral mesh is used for all domains except the GDLs and the membrane, which are swept in the z direction. By constructing the model geometry as an assembly of two parts, with a resulting continuity boundary placed at the xy-plane in the middle of the membrane, the sweeping operation allows for non-matching meshes on each side of the membrane.

Boundary layer meshes are added in the free flow domain in order to resolve the steep gradients in velocity.

STUDY SEQUENCE

The model is solved in a study sequence consisting of multiple steps. Each step uses the solution of the previous step as initial values for the dependent variables. All study steps are solved using a stationary solver.

- Step 1: Current Distribution Initialization. This study step solves for the potential variables of the `fc` interface only, for a cell potential of 1 V.
- Step 2: Stationary - Anode Flow. This study step solves for the velocity field and pressure of the anode (hydrogen) gas mixture only.
- Step 3: Stationary - Cathode Flow. This study step solves for the velocity field and pressure of the cathode (air) gas mixture only.
- Step 4: Stationary - Cooling Flow. This study step solves for the velocity field and pressure of the cooling flow (`spf` interface) only.
- Step 5: Stationary - All Physics Except Laminar Flow. This study step starts solving for the full problem at a cell potential of 1 V, ramping it down to 0.5 V by the use of an **Auxiliary Sweep**. Since the properties of the cooling fluid are not assumed to be affected by changes in temperature, the `spf` interface is excluded from solving in this study step.

The usage of the above stepped approach, in combination with the cell potential sweep in the last sweeps, results in a more robust solver setup for this highly coupled problem.

Results and Discussion

Figure 2 shows the through-plane current density of the membrane. The current densities increase towards the outlet side.

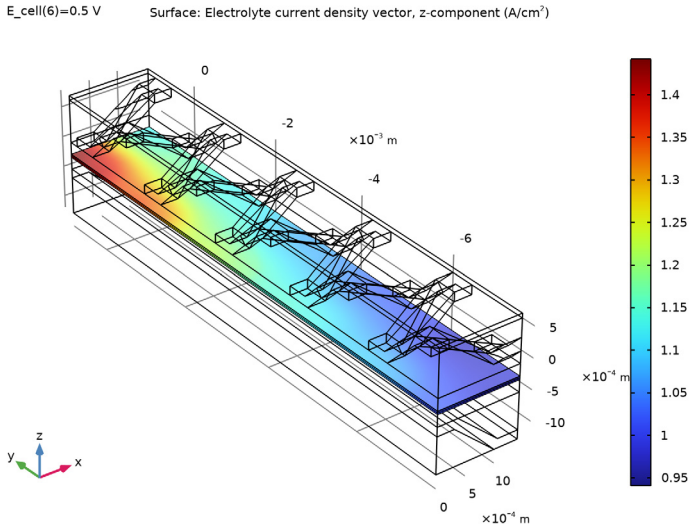


Figure 2: Cross-membrane electrolyte current density.

Figure 3 and Figure 4 show the potential drops in the anode current feeder and cathode current collector, respectively, and the GDLs. The potential drops are approximately 200 mV on each side, and stem mainly from losses in the GDLs.

Figure 5 and Figure 6 show the oxygen and water vapor molar fraction in the cathode gas mixture, respectively. The oxygen levels decrease whereas the water levels increase towards the outlet. Under the "feet" of the current collector mesh, the oxygen levels close to the outlet are about half of the inlet levels.

Figures Figure 7 and Figure 8 show the temperature in the whole cell, and in the cooling channel only, respectively. The highest temperatures are found in the MEA, with a temperature increase of more that 10°C, compared to the inlet temperature.

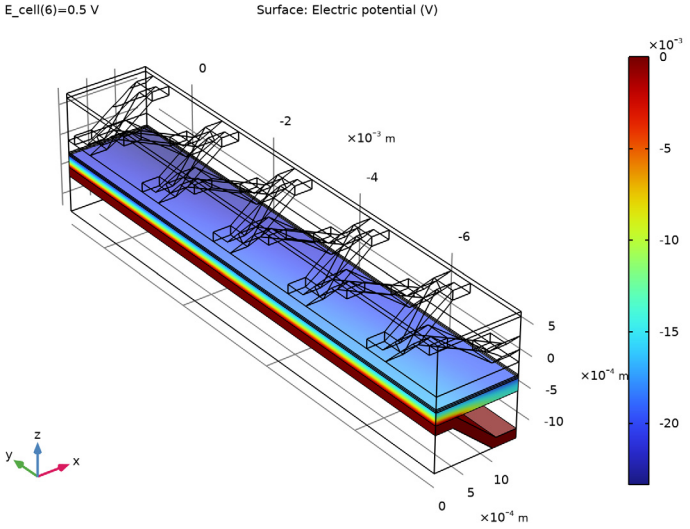


Figure 3: Electric potential in the metal conductor and GDL at the anode side of the cell.

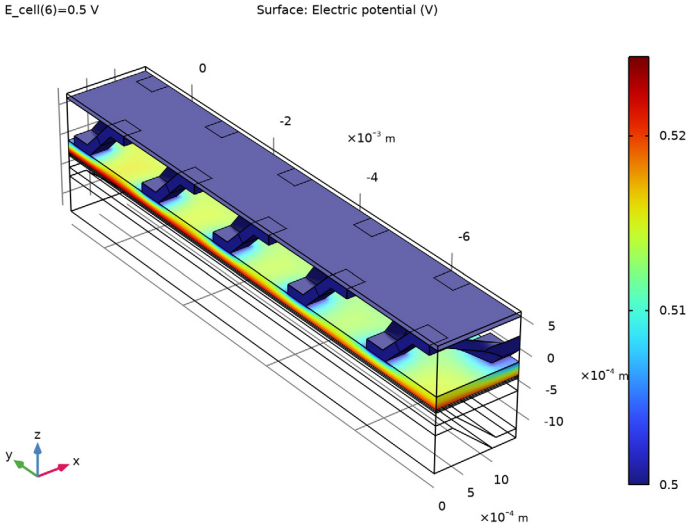


Figure 4: Electric potential in the metal conductor and GDL at the cathode side of the cell.

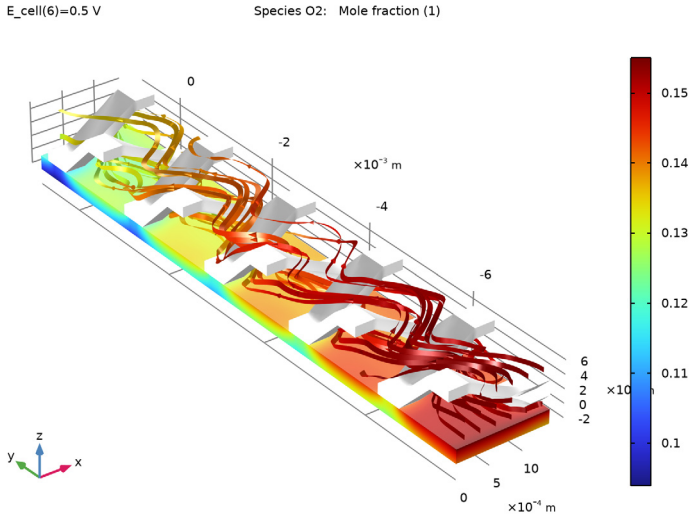


Figure 5: Oxygen molar fraction.

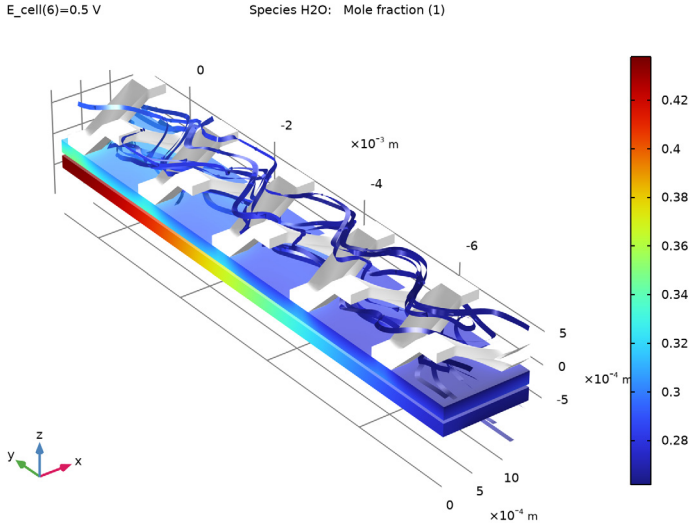


Figure 6: Water vapor molar fraction.

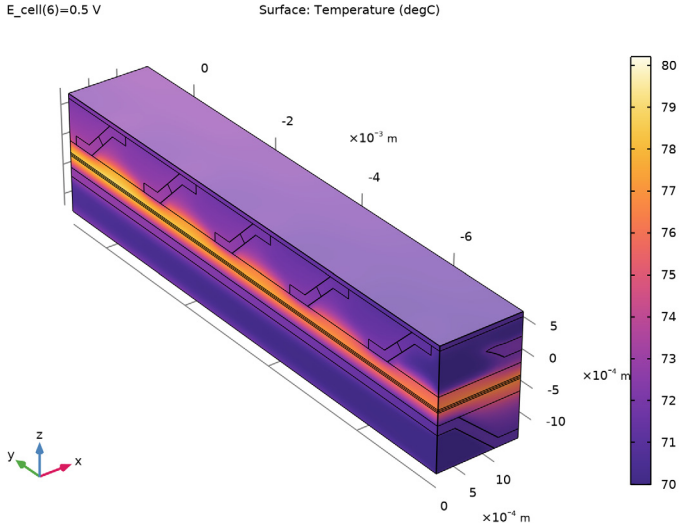


Figure 7: Temperature.

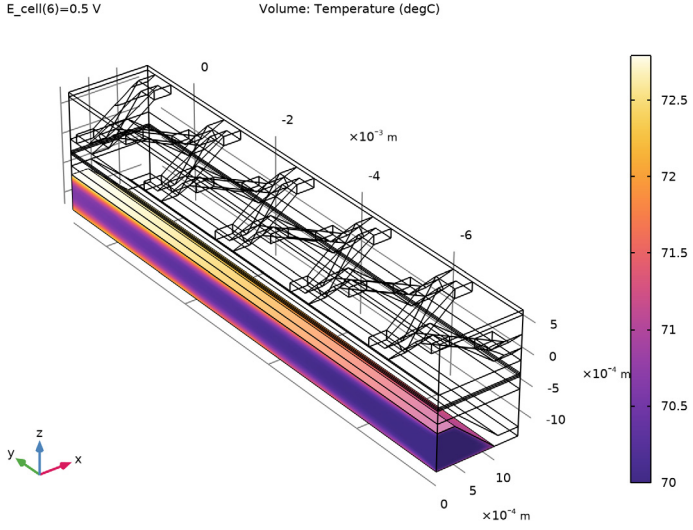


Figure 8: Temperature in the cooling channel.

Figure 9 shows the relative humidity of the gas mixtures, and the corresponding water activity in the membrane electrolyte phase. As can be seen, the relative humidity increases towards the outlet due to the production of water. The higher relative humidity results in a higher membrane conductivity, and explains the locally higher current densities towards the outlet that were seen in Figure 2.

The highest relative humidities are seen towards the outlet in the anode gas stream at the boundary facing the cooling channel. Water condensation and droplet formation would be thermodynamically most favored in this part of the cell.

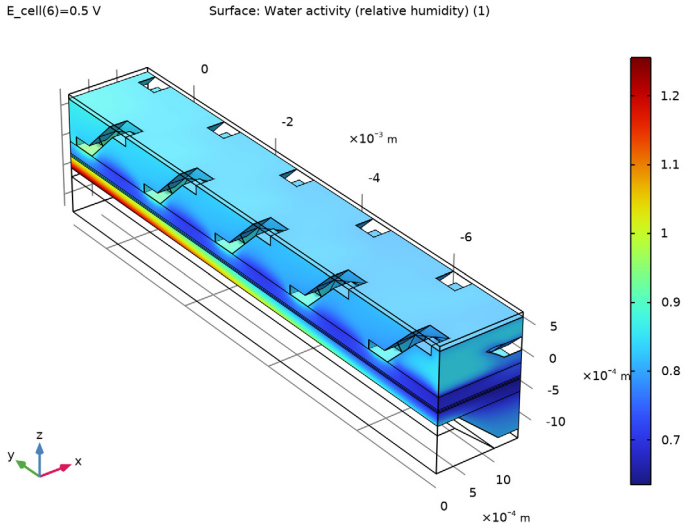


Figure 9: Water activity in the gas phase and in the membrane.

Reference

1. R. Bock, H. Karoliussen, B.G. Pollet, M. Secanell, F. Seland, D. Stanier, and O.S. Burheim, "The influence of graphitization on the thermal conductivity of catalyst layers and temperature gradients in proton exchange membrane fuel cells," *Int. J. Hydrog. Energy*, vol. 45, no. 2, pp. 1335–1342, 2020.

Application Library path: `Fuel_Cell_and_Electrolyzer_Module/Thermal_Management/nonisothermal_pem_fuel_cell`

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Electrochemistry>Hydrogen Fuel Cells>Proton Exchange (fc)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Free and Porous Media Flow (fp)**.

5 Click **Add**.

6 In the **Velocity field** text field, type `ua`.

7 In the **Velocity field components** table, enter the following settings:

ua
va
wa

8 In the **Pressure** text field, type `pa`.

9 Click **Add**.

10 In the **Velocity field** text field, type `uc`.

11 In the **Velocity field components** table, enter the following settings:

uc
vc
wc

12 In the **Pressure** text field, type `pc`.

13 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids and Fluids (ht)**.

14 Click **Add**.

15 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

16 Click **Add**.

17 In the **Velocity field** text field, type `u_cool`.

18 In the **Velocity field components** table, enter the following settings:

u_cool
v_cool
w_cool

19 In the **Pressure** text field, type p_cool.

20 Click **Study**.

21 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

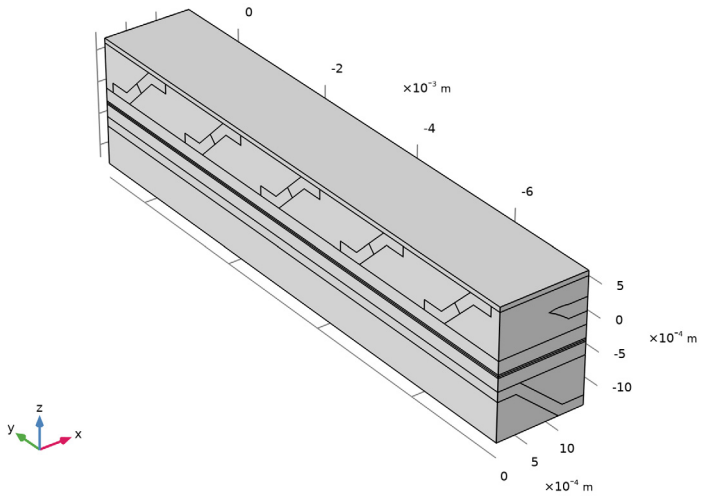
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

4 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

5 Browse to the model's Application Libraries folder and double-click the file nonisothermal_pem_fuel_cell_geom_sequence.mph.

6 In the **Geometry** toolbar, click **Build All**.



GLOBAL DEFINITIONS

Geometry Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Physics Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Physics Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `nonisothermal_pem_fuel_cell_physics_parameters.txt`.

HYDROGEN FUEL CELL (FC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Hydrogen Fuel Cell (fc)**.

2 In the **Settings** window for **Hydrogen Fuel Cell**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Fuel Cell Physics Domains**.

4 Click to expand the **Membrane Transport** section. Select the **Electroosmotic water drag** check box.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Steel AISI 4340**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the tree, select **Built-in>Water, liquid**.

6 Right-click and choose **Add to Component 1 (comp1)**.

7 In the tree, select **Fuel Cell and Electrolyzer>Polymer Electrolytes>Nafion, EW 1100, Vapor Equilibrated, Protonated**.

8 Right-click and choose **Add to Component 1 (comp1)**.

9 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Steel AISI 4340 (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Metal Conductors**.

Cooling Fluid

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

2 In the **Settings** window for **Material**, type `Cooling Fluid` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Cooling Channels**.

Nafion, EW 1100, Vapor Equilibrated, Protonated (mat3)

1 In the **Model Builder** window, click **Nafion, EW 1100, Vapor Equilibrated, Protonated (mat3)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Membrane**.

HYDROGEN FUEL CELL (FC)

Membrane 1

1 In the **Physics** toolbar, click **Domains** and choose **Membrane**.

2 In the **Settings** window for **Membrane**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Membrane**.

Initial Values 1

1 In the **Model Builder** window, expand the **Membrane 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T_0 text field, type `T_in`.

Water Absorption-Desorption, H2 Side 1

1 In the **Model Builder** window, click **Water Absorption-Desorption, H2 Side 1**.

2 In the **Settings** window for **Water Absorption-Desorption, H2 Side**, locate the **Absorption-Desorption Condition** section.

3 From the **Electrolyte material** list, choose **Nafion, EW 1100, Vapor Equilibrated, Protonated (mat3)**.

Water Absorption-Desorption, O2 Side 1

1 In the **Model Builder** window, click **Water Absorption-Desorption, O2 Side 1**.

2 In the **Settings** window for **Water Absorption-Desorption, O2 Side**, locate the **Absorption-Desorption Condition** section.

3 From the **Electrolyte material** list, choose **Nafion, EW 1100, Vapor Equilibrated, Protonated (mat3)**.

H2 Gas Flow Channel 1

1 In the **Physics** toolbar, click **Domains** and choose **H2 Gas Flow Channel**.

2 In the **Settings** window for **H2 Gas Flow Channel**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Anode Free Gas Compartment**.

H2 Gas Diffusion Layer 1

1 In the **Physics** toolbar, click **Domains** and choose **H2 Gas Diffusion Layer**.

2 In the **Settings** window for **H2 Gas Diffusion Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Anode GDL**.

4 Locate the **Electrode Charge Transport** section. From the list, choose **Diagonal**.

5 In the σ_s table, enter the following settings:

sigmas_GDL_IP	0	0
0	sigmas_GDL_IP	0
0	0	sigmas_GDL_TP

6 Locate the **Gas Transport** section. In the ε_g text field, type `epsg_GDL`.

O2 Gas Flow Channel 1

1 In the **Physics** toolbar, click **Domains** and choose **O2 Gas Flow Channel**.

2 In the **Settings** window for **O2 Gas Flow Channel**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cathode Free Gas Compartment**.

O2 Gas Diffusion Layer 1

1 In the **Physics** toolbar, click **Domains** and choose **O2 Gas Diffusion Layer**.

2 In the **Settings** window for **O2 Gas Diffusion Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cathode GDL**.

4 Locate the **Electrode Charge Transport** section. From the list, choose **Diagonal**.

5 In the σ_s table, enter the following settings:

sigmas_GDL_IP	0	0
0	sigmas_GDL_IP	0
0	0	sigmas_GDL_TP

6 Locate the **Gas Transport** section. In the ε_g text field, type epsg_GDL.

Current Collector 1

1 In the **Physics** toolbar, click **Domains** and choose **Current Collector**.

2 In the **Settings** window for **Current Collector**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Metal Conductors**.

4 Locate the **Electrode Charge Transport** section. From the σ_s list, choose **From material**.

Electronic Conducting Phase 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Hydrogen Fuel Cell (fc)> Electronic Conducting Phase 1** node, then click **Electronic Conducting Phase 1**.

Electric Ground 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Ground**.

2 In the **Settings** window for **Electric Ground**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode Current Feeder Contact**.

Electronic Conducting Phase 1

In the **Model Builder** window, click **Electronic Conducting Phase 1**.

Electric Potential 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the $\phi_{s,bnd}$ text field, type E_cell.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Top Plate Surface**.

Initial Values 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Hydrogen Fuel Cell (fc)> H2 Gas Phase 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Composition** section.

3 From the **Mixture specification** list, choose **Humidified mixture**.

4 In the $\mathrm{RH}_{\mathrm{hum}}$ text field, type RH_an.

5 In the T_{hum} text field, type T_in.

H2 Gas Phase 1

In the **Model Builder** window, click **H2 Gas Phase 1**.

H2 Inlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **H2 Inlet**.

2 In the **Settings** window for **H2 Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode Gas Inlet**.

H2 Gas Phase 1

In the **Model Builder** window, click **H2 Gas Phase 1**.

H2 Outlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **H2 Outlet**.

2 In the **Settings** window for **H2 Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode Gas Outlet**.

Initial Values 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Hydrogen Fuel Cell (fc)> O2 Gas Phase 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Composition** section.

3 From the **Mixture specification** list, choose **Humidified air**.

4 In the $\mathrm{RH}_{\mathrm{hum}}$ text field, type `RH_cath`.

5 In the T_{hum} text field, type `T_in`.

O2 Gas Phase 1

In the **Model Builder** window, click **O2 Gas Phase 1**.

O2 Inlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **O2 Inlet**.

2 In the **Settings** window for **O2 Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode Gas Inlet**.

O2 Gas Phase 1

In the **Model Builder** window, click **O2 Gas Phase 1**.

O2 Outlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **O2 Outlet**.

2 In the **Settings** window for **O2 Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode Gas Outlet**.

Thin H2 Gas Diffusion Electrode 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin H2 Gas Diffusion Electrode**.

2 In the **Settings** window for **Thin H2 Gas Diffusion Electrode**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode GDE**.

4 Locate the **Electrode Thickness** section. In the d_{gde} text field, type `L_CL`.

Thin H2 Gas Diffusion Electrode Reaction 1

1 In the **Model Builder** window, click **Thin H2 Gas Diffusion Electrode Reaction 1**.

2 In the **Settings** window for **Thin H2 Gas Diffusion Electrode Reaction**, locate the **Electrode Kinetics** section.

3 In the $i_{0,\text{ref}}(T)$ text field, type `i0_H2_ref`.

4 Locate the **Active Specific Surface Area** section. In the a_{v} text field, type `a_CL` .

Thin O2 Gas Diffusion Electrode 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin O2 Gas Diffusion Electrode**.

2 In the **Settings** window for **Thin O2 Gas Diffusion Electrode**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode GDE**.

4 Locate the **Electrode Thickness** section. In the d_{gde} text field, type `L_CL`.

Thin O2 Gas Diffusion Electrode Reaction 1

1 In the **Model Builder** window, click **Thin O2 Gas Diffusion Electrode Reaction 1**.

2 In the **Settings** window for **Thin O2 Gas Diffusion Electrode Reaction**, locate the **Electrode Kinetics** section.

3 In the $i_{0,\text{ref}}(T)$ text field, type `i0_O2_ref`.

4 In the α_{a} text field, type `alphaa_O2`.

5 Locate the **Active Specific Surface Area** section. In the a_{v} text field, type `a_CL`.

FREE AND POROUS MEDIA FLOW - ANODE

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow (fp)**.

2 In the **Settings** window for **Free and Porous Media Flow**, type `Free and Porous Media Flow - Anode` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Anode Free Gas and GDL**.

4 Locate the **Physical Model** section. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

MULTIPHYSICS

Reacting Flow, H2 Gas Phase 1 (rfh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Reacting Flow, H2 Gas Phase**.

FREE AND POROUS MEDIA FLOW - ANODE (FP)

Porous Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 In the **Settings** window for **Porous Medium**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Anode GDL**.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_p list, choose **User defined**. In the associated text field, type `epsg_GDL`.

4 From the κ list, choose **User defined**. In the associated text field, type `kappag_GDL`.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

3 From the list, choose **Fully developed flow**.

4 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `v_in_an`.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **Anode Gas Inlet**.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode Gas Outlet**.

Wall 2

1 In the **Model Builder** window, expand the **Free and Porous Media Flow - Anode (fp)** node.

2 Right-click **Free and Porous Media Flow - Anode (fp)** and choose **Wall**.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **GDL Boundaries**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Right and Left Symmetry Boundaries**.

FREE AND POROUS MEDIA FLOW - CATHODE

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow 2 (fp2)**.

2 In the **Settings** window for **Free and Porous Media Flow**, type `Free and Porous Media Flow - Cathode` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Cathode Free Gas and GDL**.

4 Locate the **Physical Model** section. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

MULTIPHYSICS

Reacting Flow, O2 Gas Phase 1 (rfo1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Reacting Flow, O2 Gas Phase**.

2 In the **Settings** window for **Reacting Flow, O2 Gas Phase**, locate the **Coupled Interfaces** section.

3 From the **Fluid flow** list, choose **Free and Porous Media Flow - Cathode (fp2)**.

FREE AND POROUS MEDIA FLOW - CATHODE (FP2)

Porous Medium 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Free and Porous Media Flow - Cathode (fp2)** node.

2 Right-click **Free and Porous Media Flow - Cathode (fp2)** and choose **Porous Medium**.

3 In the **Settings** window for **Porous Medium**, locate the **Domain Selection** section.

4 From the **Selection** list, choose **Cathode GDL**.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_p list, choose **User defined**. In the associated text field, type `epsg_GDL`.

4 From the κ list, choose **User defined**. In the associated text field, type `kappag_GDL`.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode Gas Inlet**.

4 Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `v_in_cath`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode Gas Outlet**.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **GDL Boundaries**.

4 Locate the **Boundary Condition** section. From the **Wall condition** list, choose **Slip**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Right and Left Symmetry Boundaries**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid - Anode Gas

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, type `Fluid - Anode Gas` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Anode Free Gas Compartment**.

4 Locate the **Heat Conduction, Fluid** section. From the k list, choose **Thermal conductivity, gas phase (fc)**.

5 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Gas/Liquid**.

6 From the ρ list, choose **Density of gas phase (fc)**.

7 From the C_p list, choose **Heat capacity at constant pressure, gas phase (fc)**.

8 From the γ list, choose **User defined**.

Fluid - Cathode Gas

1 In the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 In the **Settings** window for **Fluid**, type `Fluid - Cathode Gas` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Cathode Free Gas Compartment**.

4 Locate the **Heat Conduction, Fluid** section. From the k list, choose **Thermal conductivity, gas phase (fc)**.

5 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Gas/Liquid**.

6 From the ρ list, choose **Density of gas phase (fc)**.

7 From the C_p list, choose **Heat capacity at constant pressure, gas phase (fc)**.

8 From the γ list, choose **User defined**.

Fluid - Cooling

1 In the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 In the **Settings** window for **Fluid**, type `Fluid - Cooling` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Cooling Channels**.

Solid - GDLs

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for **Solid**, type `Solid - GDLs` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **GDLs**.

4 Locate the **Heat Conduction, Solid** section. From the k list, choose **User defined**. From the list, choose **Diagonal**.

5 In the k table, enter the following settings:

kappa_GDL_IP	0	0
0	kappa_GDL_IP	0
0	0	kappa_GDL_TP

6 Locate the **Thermodynamics, Solid** section. From the ρ list, choose **User defined**. From the C_p list, choose **User defined**.

Solid - Membrane

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Membrane**.

4 In the **Label** text field, type `Solid - Membrane`.

5 Locate the **Thermodynamics, Solid** section. From the ρ list, choose **User defined**. From the C_p list, choose **User defined**.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlets**.

4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_in`.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlets**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Right and Left Symmetry Boundaries**.

Periodic Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 In the **Settings** window for **Periodic Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Top and Bottom Boundaries**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cooling Channels**.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cooling Inlet**.

4 Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `v_in_cool`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cooling Outlet**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Right and Left Symmetry Boundaries**.

MULTIPHYSICS

Electrochemical Heating 1 (ech1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Electrochemical Heating**.

Nonisothermal Flow - Anode Gas

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Nonisothermal Flow**.

2 In the **Settings** window for **Nonisothermal Flow**, type `Nonisothermal Flow - Anode Gas` in the **Label** text field.

Nonisothermal Flow - Cathode Gas

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Nonisothermal Flow**.

2 In the **Settings** window for **Nonisothermal Flow**, type `Nonisothermal Flow - Cathode Gas` in the **Label** text field.

3 Locate the **Coupled Interfaces** section. From the **Fluid flow** list, choose **Free and Porous Media Flow - Cathode (fp2)**.

Nonisothermal Flow - Cooling Fluid

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Nonisothermal Flow**.

2 In the **Settings** window for **Nonisothermal Flow**, type `Nonisothermal Flow - Cooling Fluid` in the **Label** text field.

3 Locate the **Coupled Interfaces** section. From the **Fluid flow** list, choose **Laminar Flow (spf)**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids and Fluids**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `T_in`.

DEFINITIONS

Free Flow Domains

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Selections>Union**.

3 In the **Settings** window for **Union**, locate the **Input Entities** section.

4 Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Cooling Channels** and **Free Gas Domains**.

6 Click **OK**.

7 In the **Settings** window for **Union**, type `Free Flow Domains` in the **Label** text field.

Free Flow Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Free Flow Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Free Flow Domains** in the **Input selections** list.

5 Click **OK**.

Boundary Layer Boundaries

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Boundary Layer Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Free Flow Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click + **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Cathode Gas Inlet**, **Cathode Gas Outlet**, **Anode Gas Inlet**, **Anode Gas Outlet**, **Cooling Inlet**, **Cooling Outlet**, and **Right and Left Symmetry Boundaries**.

10 Click **OK**.

Sweep Domains

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Sweep Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Membrane**, **Cathode GDL**, and **Anode GDL**.

5 Click **OK**.

Free Tet Domains

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Free Tet Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click + **Add**.

4 In the **Add** dialog box, select **Sweep Domains** in the **Selections to invert** list.

5 Click **OK**.

MESH 1

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Cathode Free Gas Compartment**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Minimum element size** check box. In the associated text field, type `m_th/4`.

Corner Refinement 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Corner Refinement**.

2 In the **Settings** window for **Corner Refinement**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Free Flow Domains**.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **Boundary Layer Boundaries**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Free Tet Domains**.

Size

1 In the **Model Builder** window, click **Size**.

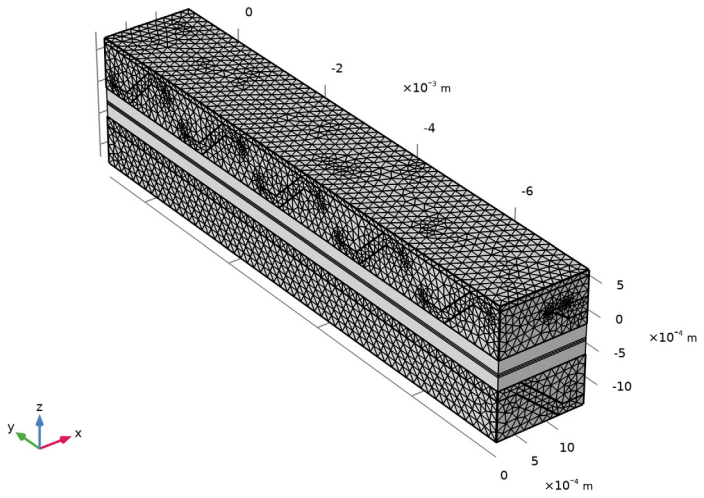
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `m_th`.

5 In the **Minimum element size** text field, type `m_th/2`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Free Flow Domains**.

5 Click to expand the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Trimming**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

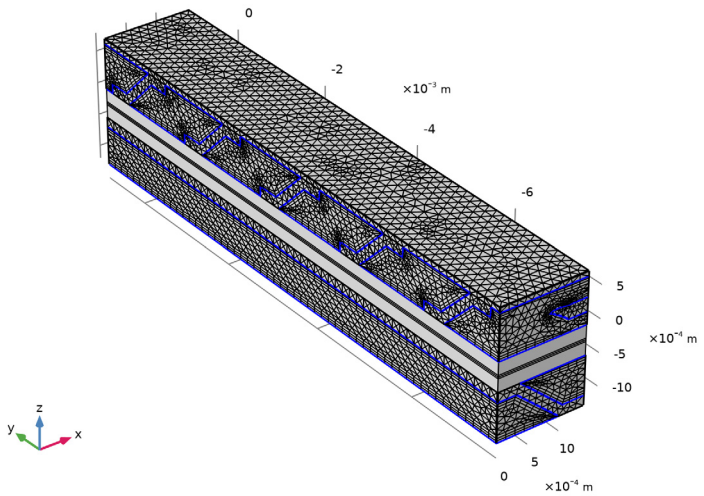
3 From the **Selection** list, choose **Boundary Layer Boundaries**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `3`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `m_th/5`.

7 Click **Build Selected**.



Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

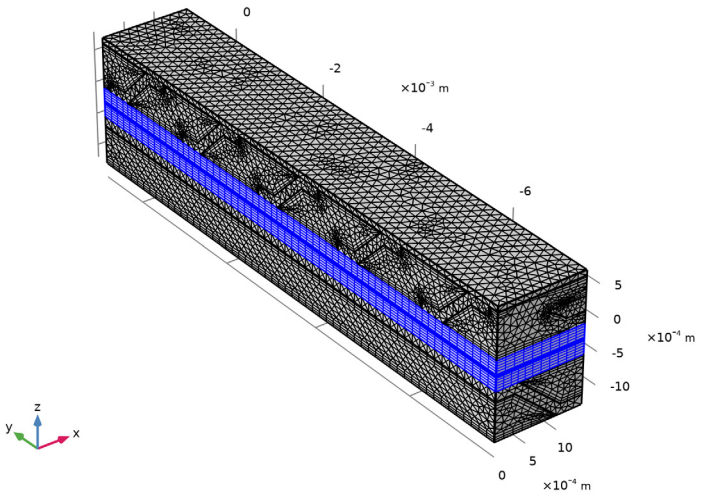
3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Sweep Domains**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 Right-click **Distribution 1** and choose **Build All**.



ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Hydrogen Fuel Cell> Stationary with Initialization**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Stationary - Anode Flow

1 In the **Settings** window for **Stationary**, type `Stationary - Anode Flow` in the **Label** text field.

2 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Hydrogen Fuel Cell (fc)		Automatic (Current distribution initialization)
Free and Porous Media Flow - Cathode (fp2)		Automatic (Stationary)
Heat Transfer in Solids and Fluids (ht)		Automatic (Stationary)
Laminar Flow (spf)		Automatic (Stationary)

3 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Reacting Flow, O2 Gas Phase 1 (rfo1)		Automatic (Current distribution initialization)

Stationary - Cathode Flow

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, type `Stationary - Cathode Flow` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Hydrogen Fuel Cell (fc)		Automatic (Current distribution initialization)
Free and Porous Media Flow - Anode (fp)		Automatic (Stationary)
Heat Transfer in Solids and Fluids (ht)		Automatic (Stationary)
Laminar Flow (spf)		Automatic (Stationary)

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Reacting Flow, H2 Gas Phase 1 (rfh1)		Automatic (Stationary)

Stationary - Cooling Flow

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, type `Stationary - Cooling Flow` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Hydrogen Fuel Cell (fc)		Automatic (Current distribution initialization)
Free and Porous Media Flow - Anode (fp)		Automatic (Stationary)
Free and Porous Media Flow - Cathode (fp2)		Automatic (Stationary)
Heat Transfer in Solids and Fluids (ht)		Automatic (Stationary)

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Reacting Flow, H2 Gas Phase 1 (rfh1)		Automatic (Stationary)
Reacting Flow, O2 Gas Phase 1 (rfo1)		Automatic (Stationary)

Stationary - All Physics Except Laminar Flow

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, type `Stationary - All Physics Except Laminar Flow` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Laminar Flow (spf)		Automatic (Stationary)

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, click to select the cell at row number 1 and column number 3.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
E_cell (Cell voltage (changed in Auxiliary Sweep))	`range(1,-0.1,0.5)`	V

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 5** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 From the **Stabilization and acceleration** list, choose **None**.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 5>Segregated 1** node, then click **Velocity ua, Pressure pa**.

7 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

8 In the **Damping factor** text field, type 1.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 5>Segregated 1** click **Temperature**.

10 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

11 In the **Damping factor** text field, type 1.

12 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 5>Segregated 1** click **Velocity uc, Pressure pc**.

13 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

14 In the **Damping factor** text field, type 1.

15 In the **Study** toolbar, click **Show Default Plots**.

Step 5: Stationary - All Physics Except Laminar Flow

1 In the **Model Builder** window, under **Study 1** click **Step 5: Stationary - All Physics Except Laminar Flow**.

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Temperature (ht)**.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Membrane Current Density

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Membrane Current Density` in the **Label** text field.

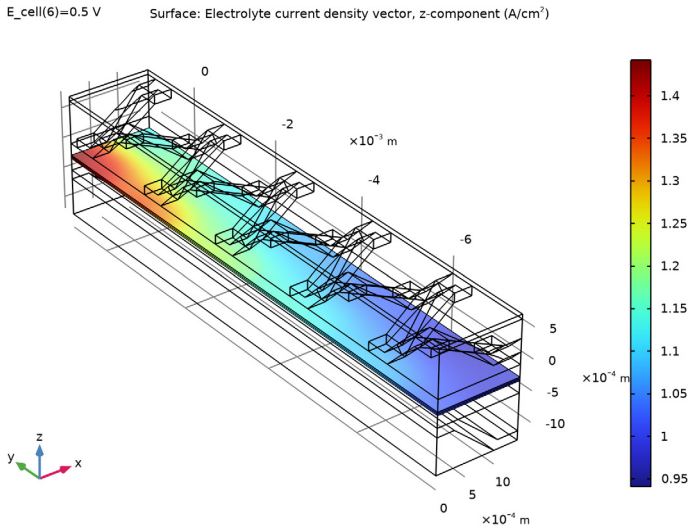
Surface 1

1 Right-click **Membrane Current Density** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Hydrogen Fuel Cell>Electrolyte current density vector - A/m²>fc.IIz - Electrolyte current density vector, z-component**.

3 Locate the **Expression** section. In the **Unit** field, type `A/cm^2`.

4 In the **Membrane Current Density** toolbar, click **Plot**.



Electrode Phase Potential, Anode Side

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Electrode Phase Potential, Anode Side` in the **Label** text field.

Surface 1

1 Right-click **Electrode Phase Potential, Anode Side** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Hydrogen Fuel Cell>fc.phis - Electric potential - V**.

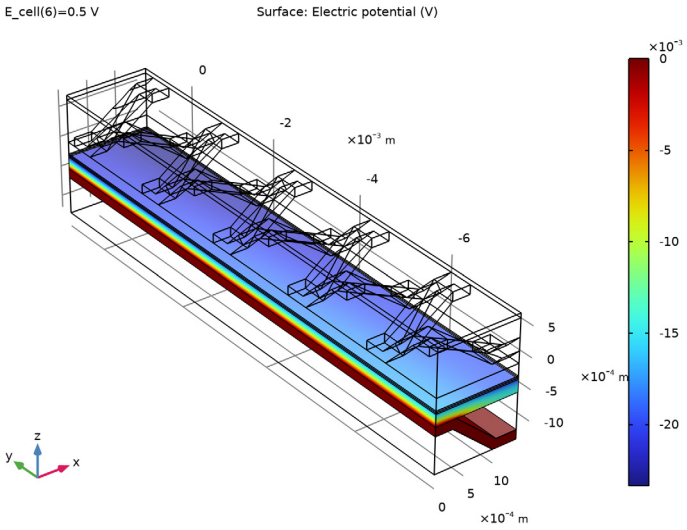
Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Anode Side Domains**.

4 In the **Electrode Phase Potential, Anode Side** toolbar, click **Plot**.



Electrode Phase Potential, Cathode

1 In the **Model Builder** window, right-click **Electrode Phase Potential, Anode Side** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Electrode Phase Potential, Cathode` in the **Label** text field.

3 In the **Model Builder** window, expand the **Electrode Phase Potential, Cathode** node.

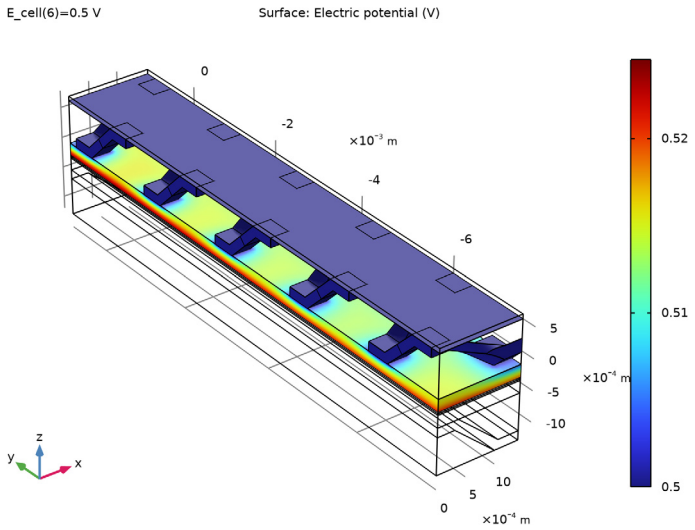
Selection 1

1 In the **Model Builder** window, expand the **Results>Electrode Phase Potential, Cathode> Surface 1** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Cathode Side**.

4 In the **Electrode Phase Potential, Cathode** toolbar, click **Plot**.



Mole Fraction, O2, Streamline (fc)

1 In the **Model Builder** window, under **Results** click **Mole Fraction, O2, Streamline (fc)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Streamline 1

1 In the **Model Builder** window, expand the **Mole Fraction, O2, Streamline (fc)** node, then click **Streamline 1**.

2 In the **Settings** window for **Streamline**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected boundaries**.

5 In the **Number** text field, type 15.

6 Locate the **Selection** section. From the **Selection** list, choose **Cathode Gas Inlet**.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

Color Expression

1 In the **Model Builder** window, expand the **Streamline 1** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Surface 1

1 In the **Model Builder** window, right-click **Mole Fraction, O2, Streamline (fc)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Hydrogen Fuel Cell>Species O2>fc.xO2 - Mole fraction**.

3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

4 Find the **Type and data** subsection. Clear the **Type** check box.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **GDL Boundaries**.

Surface 2

1 In the **Model Builder** window, right-click **Mole Fraction, O2, Streamline (fc)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type 1.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **White**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

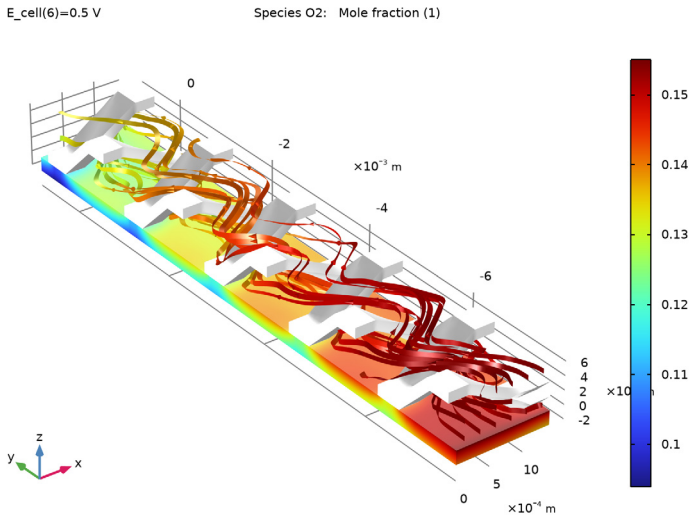
3 From the **Selection** list, choose **Mesh**.

Surface 1

1 Click the  **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Model Builder** window, under **Results>Mole Fraction, O2, Streamline (fc)** click **Surface 1**.

3 In the **Mole Fraction, O2, Streamline (fc)** toolbar, click **Plot**.



Mole Fraction, H2O, Streamline (fc)

1 In the **Model Builder** window, expand the **Results>Mole Fraction, H2O, Streamline (fc)** node, then click **Mole Fraction, H2O, Streamline (fc)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Streamline 1

1 In the **Model Builder** window, click **Streamline 1**.

2 In the **Settings** window for **Streamline**, locate the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

Color Expression

1 In the **Model Builder** window, expand the **Streamline 1** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Title** section.

3 From the **Title type** list, choose **None**.

Surface 1

1 In the **Model Builder** window, right-click **Mole Fraction, H2O, Streamline (fc)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Hydrogen Fuel Cell>Species H2O>fc.xH2O - Mole fraction**.

3 Locate the **Title** section. From the **Title type** list, choose **Custom**.

4 Find the **Type and data** subsection. Clear the **Type** check box.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **GDL Boundaries**.

Surface 2

In the **Model Builder** window, under **Results>Mole Fraction, O2, Streamline (fc)** right-click **Surface 2** and choose **Copy**.

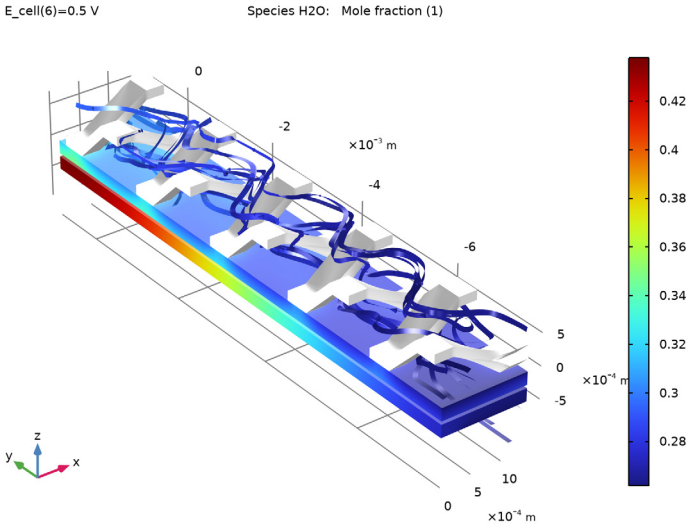
Surface 2

In the **Model Builder** window, right-click **Mole Fraction, H2O, Streamline (fc)** and choose **Paste Surface**.

Surface 1

1 Click the  **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Mole Fraction, H2O, Streamline (fc)** toolbar, click **Plot**.



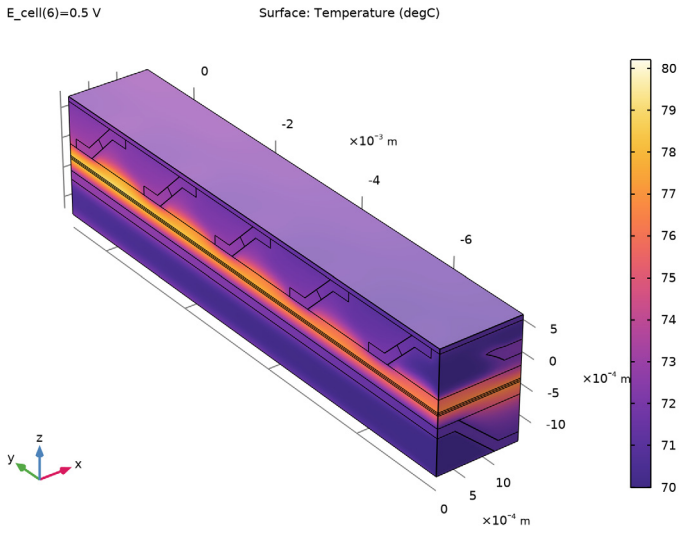
Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 In the **Temperature (ht)** toolbar, click **Plot**.



Cooling Channel Temperature

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Cooling Channel Temperature` in the **Label** text field.

Volume 1

1 Right-click **Cooling Channel Temperature** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Solids and Fluids>Temperature>T - Temperature - K**.

3 Locate the **Expression** section. From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

6 Click **OK**.

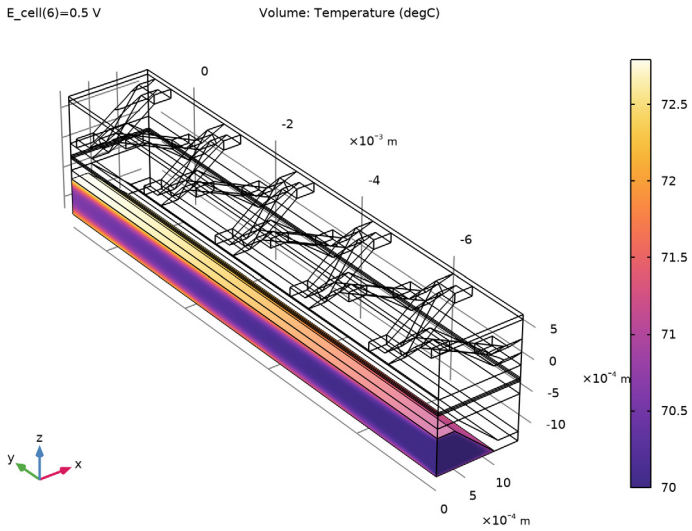
Selection 1

1 Right-click **Volume 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Cooling Channels**.

4 In the **Cooling Channel Temperature** toolbar, click **Plot**.



Water Activity

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Water Activity` in the **Label** text field.

Surface 1

1 Right-click **Water Activity** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Hydrogen Fuel Cell>fc.aw - Water activity (relative humidity)**.

3 In the **Water Activity** toolbar, click **Plot**.