



Model created in COMSOL Multiphysics 6.4

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.

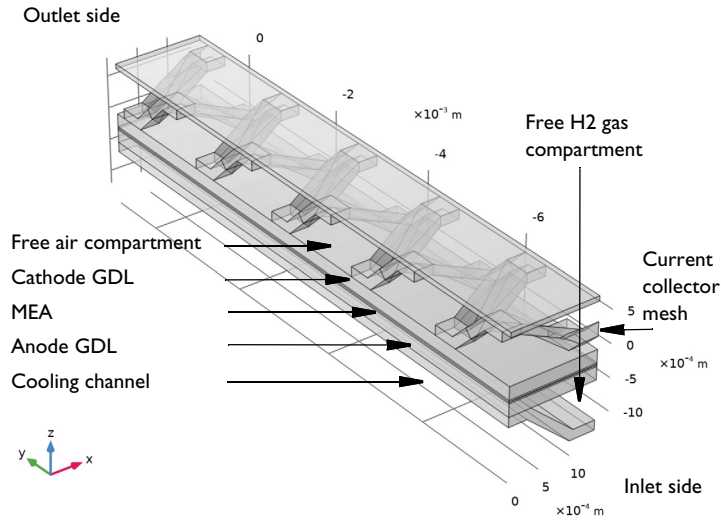


Figure 1: Model geometry.

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 toward 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 nonmatching 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 toward the outlet side.

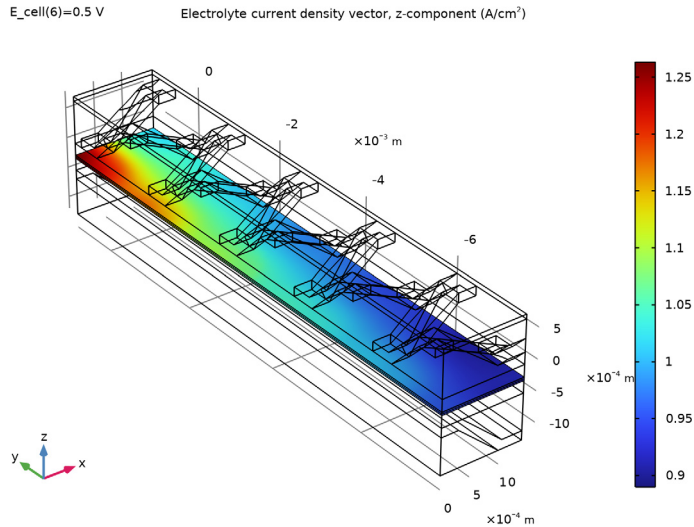


Figure 2: Cross-membrane electrolyte current density.

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 toward 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 than 10°C , compared to the inlet temperature.

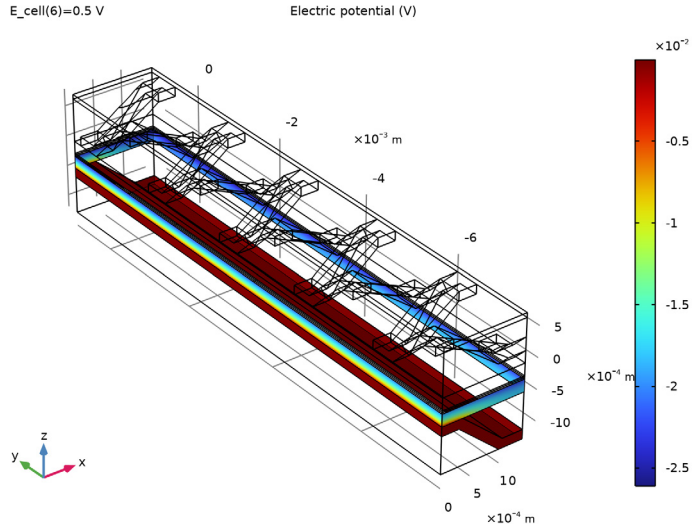


Figure 3: Electric potential in the metal conductor and GDL at the anode side of the cell.

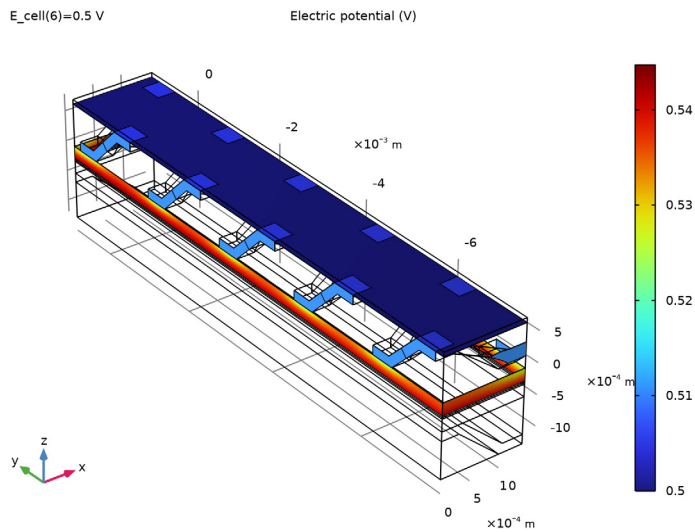


Figure 4: Electric potential in the metal conductor and GDL at the cathode side of the cell.

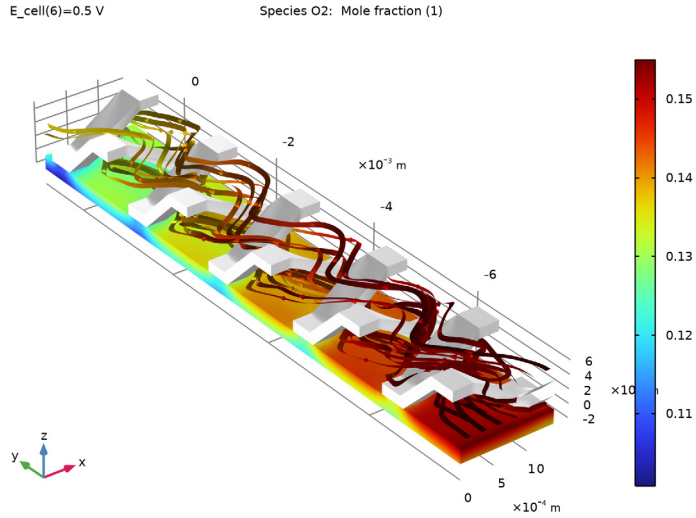


Figure 5: Oxygen molar fraction.

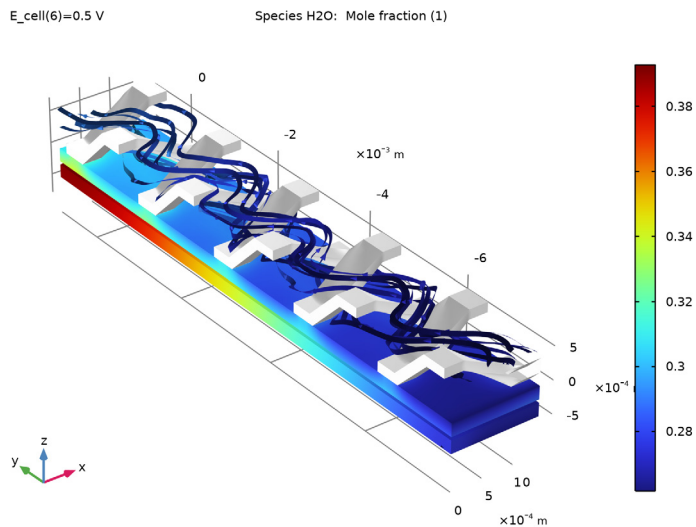


Figure 6: Water vapor molar fraction.

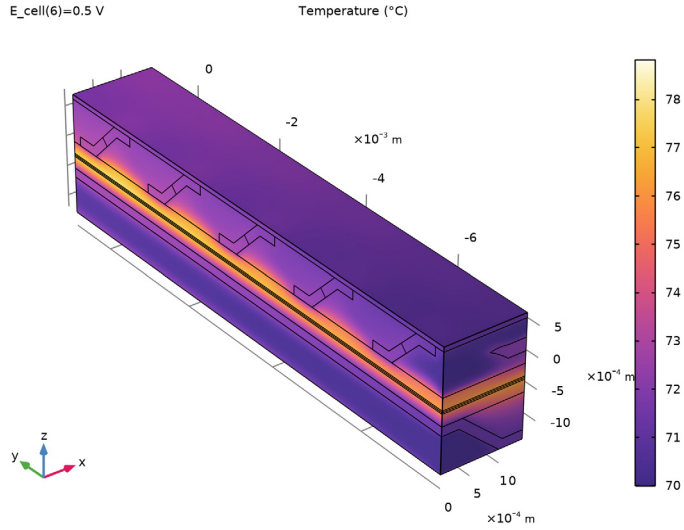


Figure 7: Temperature.

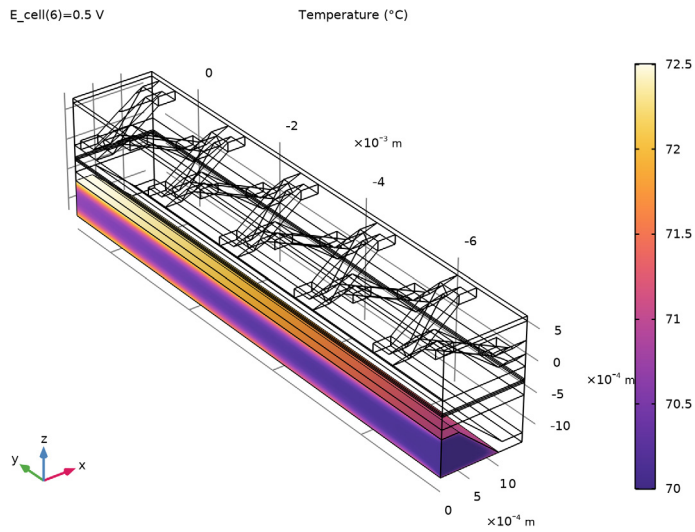


Figure 8: Temperature in the cooling channel.

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 toward 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 toward the outlet that were seen in Figure 2.

The highest relative humidities are seen toward 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.

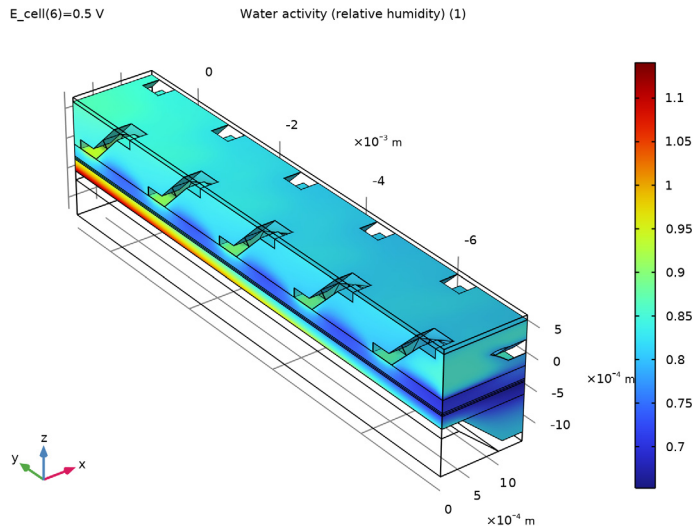


Figure 9: Water activity in the gas phase and in the membrane.

Reference


I. 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/Fuel_Cells/
nonisothermal_pem_fuel_cell


Modeling Instructions

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 Membrane (fc)**.

3 Click **Add**.

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

5 Click **Add**.

6 In the **Velocity field (m/s)** text field, type ua.

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

ua

va

wa

8 In the **Pressure (Pa)** text field, type pa.

9 Click **Add**.

10 In the **Velocity field (m/s)** text field, type uc.

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

uc

vc

wc

12 In the **Pressure (Pa)** 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 (m/s)** 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 (Pa)** text field, type p_cool.

20 Click  **Study**.

21 Click  **Done**.

GEOMETRY I


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

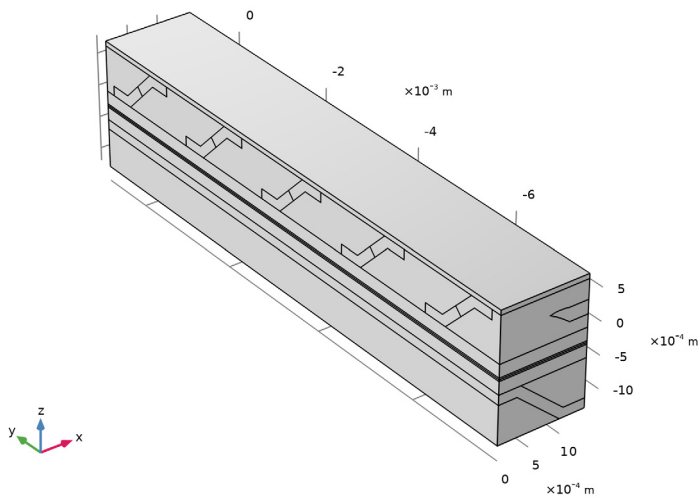
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.

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 I**.
- 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 **Electrolyte and Membrane Transport** section. Select the **Electroosmotic water drag** checkbox.

ADD MATERIAL

- 1 In the **Materials** 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 **Materials** 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 Specify the σ_s matrix as


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 Specify the σ_s matrix as

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 I

- 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 I

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


Electric Ground I

- 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 I

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


Electric Potential I

- 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**.

Electronic Conducting Phase I

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

Internal Electrode Contact Resistance I

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Internal Electrode Contact Resistance**.
- 2 In the **Settings** window for **Internal Electrode Contact Resistance**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Electronic Conducting Phase Internal Boundaries**.

4 Locate the **Contact Resistance** section. In the R_c text field, type Rc.


Initial Values I

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Hydrogen Fuel Cell (fc) > H2 Gas Phase 1** node, then click **Initial Values I**.
- 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 RH_{hum} text field, type RH_an.
- 5 In the T_{hum} text field, type T_in.

H2 Gas Phase I

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


H2 Inlet I

- 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**.
- 4 Locate the **Inlet Flow Type** section. From the **Inlet flow type** list, choose **Mixture composition constraint**.

H2 Gas Phase I

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

H2 Outlet I

- 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 I

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Hydrogen Fuel Cell (fc) > O2 Gas Phase 1** node, then click **Initial Values I**.
- 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 RH_{hum} text field, type RH_cath.
- 5 In the T_{hum} text field, type T_in.

O2 Gas Phase I

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


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**.
- 4 Locate the **Inlet Flow Type** section. From the **Inlet flow type** list, choose **Mixture composition constraint**.


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_02_ref`.
- 4 In the α_a text field, type `alphaa_02`.
- 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, Brinkman (fp)**.
- 2 In the **Settings** window for **Free and Porous Media Flow, Brinkman**, 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 **Physics** toolbar, click  **Boundaries** and choose **Wall**.
- 2 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
- 3 From the **Wall condition** list, choose **Slip**.
- 4 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, Brinkman 2 (fp2)**.
- 2 In the **Settings** window for **Free and Porous Media Flow, Brinkman**, 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 **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 **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 Specify the k matrix as


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, 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, 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, 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, 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, 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, select **Sweep Domains** in the **Selections to invert** list.
- 5 Click **OK**.

MESH I


Size I

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh I** 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** checkbox. In the associated text field, type $m_th/4$.

Corner Refinement I

- 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**.

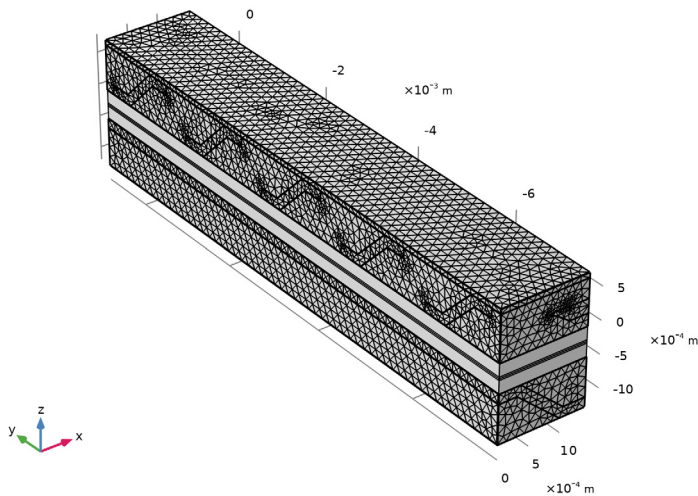
Free Tetrahedral I

- 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**.



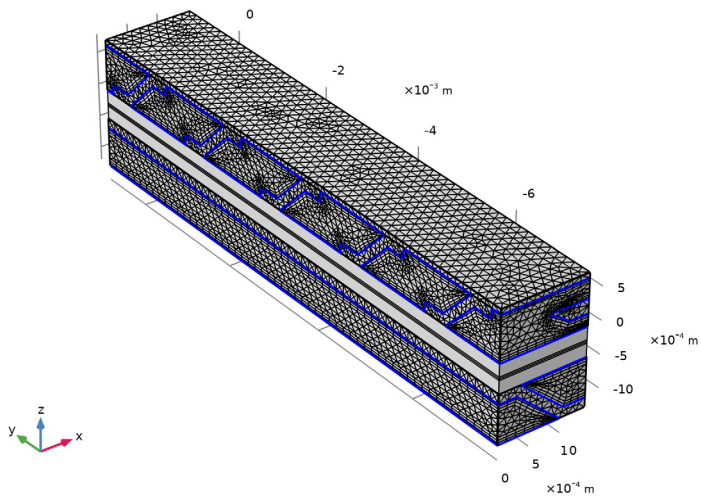
Boundary Layers 1

- 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.**



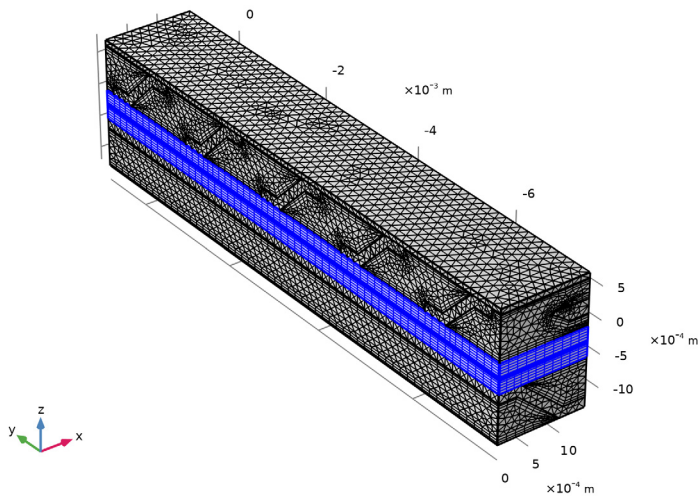
Swept 1

- 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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Hydrogen Fuel Cell (fc)**, **Free and Porous Media Flow - Cathode (fp2)**, **Heat Transfer in Solids and Fluids (ht)**, and **Laminar Flow (spf)**.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkbox for **Reacting Flow, O2 Gas Phase 1 (rfo1)**.


Stationary - Cathode Flow

- 1 In the **Study** toolbar, click  **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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Hydrogen Fuel Cell (fc)**, **Free and Porous Media Flow - Anode (fp)**, **Heat Transfer in Solids and Fluids (ht)**, and **Laminar Flow (spf)**.
- 4 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkbox for **Reacting Flow, H2 Gas Phase 1 (rfh1)**.

Stationary - Cooling Flow



- 1 In the **Study** toolbar, click  **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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Hydrogen Fuel Cell (fc)**, **Free and Porous Media Flow - Anode (fp)**, **Free and Porous Media Flow - Cathode (fp2)**, and **Heat Transfer in Solids and Fluids (ht)**.
- 4 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkboxes for **Reacting Flow, H2 Gas Phase 1 (rfh1)** and **Reacting Flow, O2 Gas Phase 1 (rfo1)**.

Stationary - All Physics Except Laminar Flow

- 1 In the **Study** toolbar, click  **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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Laminar Flow (spf)**.
- 4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 5 Click **+ Add**.
- 6 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** checkbox.
- 4 In the table, enter the following settings:



Plot group	Plot window
Temperature (ht)	Graphics

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


RESULTS

Set default units for result presentation.


Preferred Units I

- 1 In the **Results** toolbar, click  **Configurations** and choose **Preferred Units**.
- 2 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 3 Click  **Add Physical Quantity**.
- 4 In the **Physical Quantity** dialog, select **General > Temperature (K)** in the tree.
- 5 Click **OK**.
- 6 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

- 8 In the **Model Builder** window, click **Preferred Units I**.
- 9 Click  **Apply**.

Membrane Current Density

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Membrane Current Density in the **Label** text field.

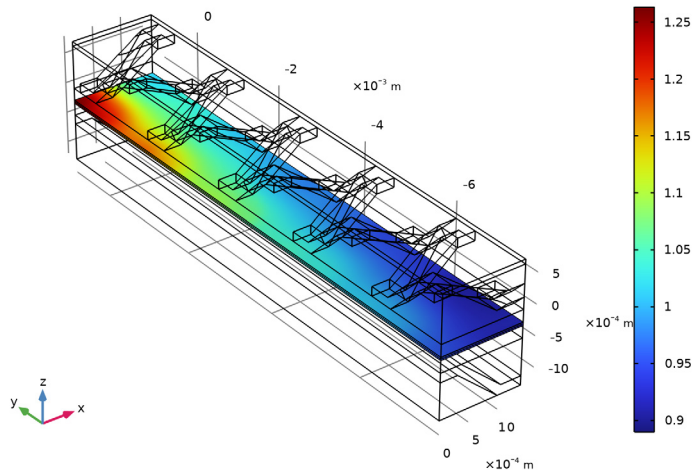
Surface I

- 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 I (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/cm².


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

$E_{\text{cell}(6)}=0.5 \text{ V}$

Electrolyte current density vector, z-component (A/cm^2)



Electrode Phase Potential, Anode Side

1 In the **Results** toolbar, click  **3D Plot Group**.

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


3 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

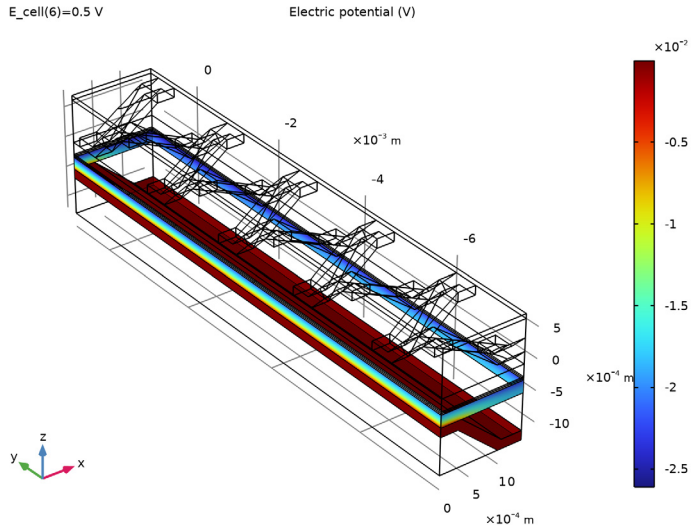
4 From the **Selection** list, choose **Electrode Phase Potential, Anode Side**.

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**.

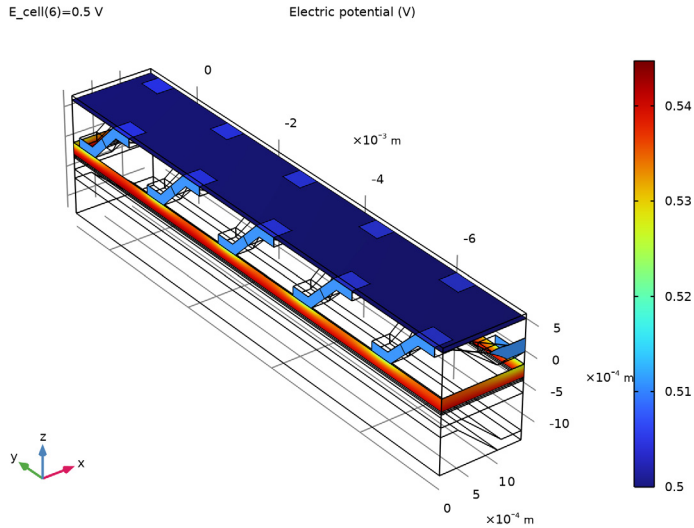
3 In the **Electrode Phase Potential, Anode Side** toolbar, click  **Plot**.



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 Locate the **Selection** section. From the **Selection** list, choose **Electrode Phase Potential, Cathode**.

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



Mole Fraction, O₂, Streamline (fc)

- 1 In the **Model Builder** window, 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** checkbox.

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 entities**.
- 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 - 1**.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.
- 4 Find the **Type and data** subsection. Clear the **Type** checkbox.
- 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**.

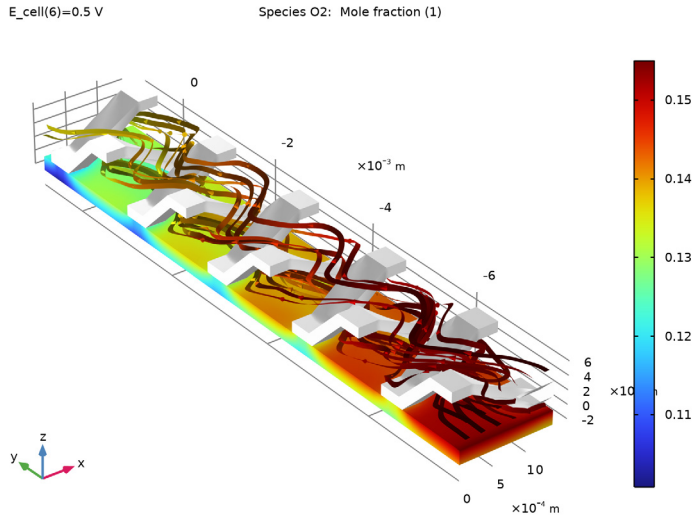
Selection 1

- 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** checkbox.

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 - 1**.
- 3 Locate the **Title** section. From the **Title type** list, choose **Custom**.
- 4 Find the **Type and data** subsection. Clear the **Type** checkbox.
- 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**.

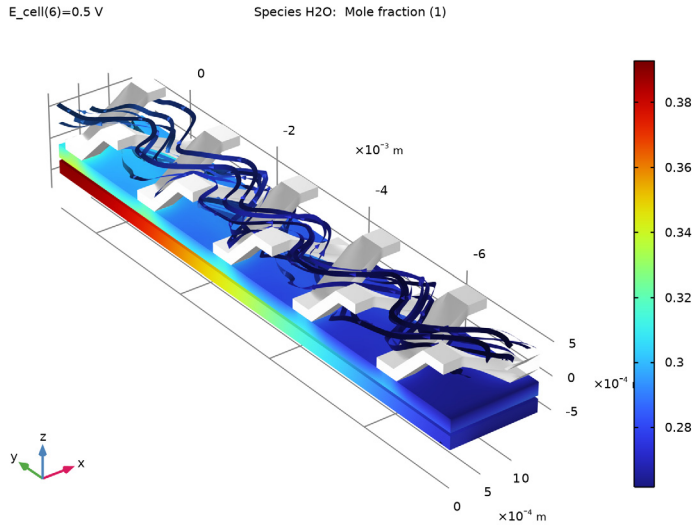
Surface 2

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**.

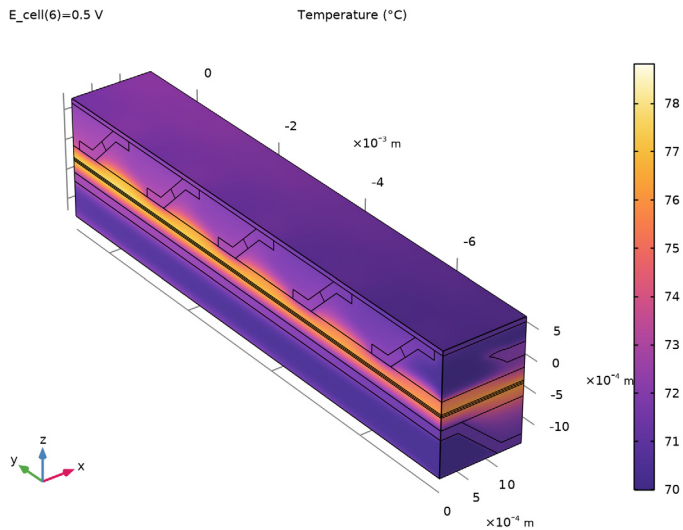
3 In the **Model Builder** window, click **Surface 1**.




Volume 1

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


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

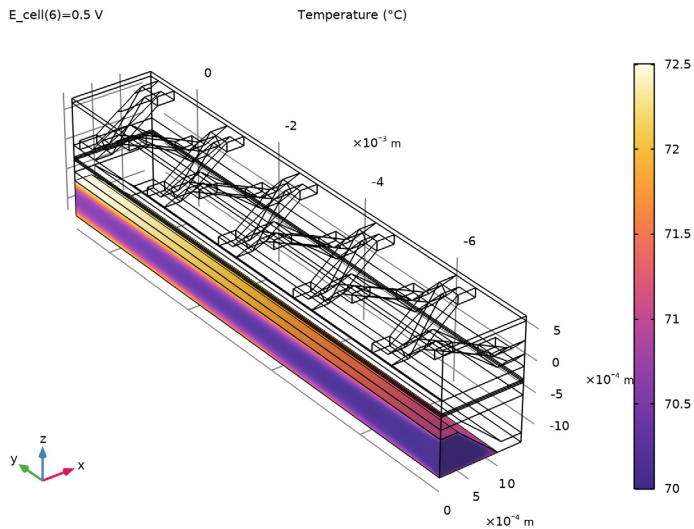


Cooling Channel Temperature


- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Cooling Channel Temperature in the **Label** text field.
- 3 Locate the **Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Cooling Channels**.

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 **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.
- 4 In the **Cooling Channel Temperature** toolbar, click  **Plot**.




Water Activity

- 1 In the **Results** toolbar, click  **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 I (comp1) > Hydrogen Fuel Cell > fc.aw - Water activity (relative humidity) - I**.
- 3 In the **Water Activity** toolbar, click  **Plot**.

