

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.

E_cell(6)=0.5 V Surface: Electrolyte current density vector, z component (A/cm²)



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/Fuel_Cells/ nonisothermal_pem_fuel_cell

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 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.

IO In the **Velocity field** text field, type uc.

II In the Velocity field components table, enter the following settings:

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).

I4 Click Add.

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

I6 Click Add.

I7 In the **Velocity field** text field, type u_cool.

uc

vc

I8 In the **Velocity field components** table, enter the following settings:

u_cool

v cool

w_cool

I9 In the **Pressure** text field, type p_cool.

20 Click \bigcirc Study.

21 Click M Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 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

- I 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

- I In the Home toolbar, click Pi 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)

- I In the Model Builder window, under Component I (compl) 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.

MATERIALS

In the Home toolbar, click 📑 Windows and choose Add Material from Library.

ADD MATERIAL

- I Go to the Add Material window.
- 2 In the tree, select Built-in>Steel AISI 4340.
- **3** Right-click and choose **Add to Component I (compl)**.
- 4 In the tree, select Built-in>Water, liquid.
- 5 Right-click and choose Add to Component I (compl).
- 6 In the tree, select Fuel Cell and Electrolyzer>Polymer Electrolytes>Nafion, EW 1100, Vapor Equilibrated, Protonated.
- 7 Right-click and choose Add to Component I (compl).
- 8 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Steel AISI 4340 (mat1)

- I In the Settings window for Material, locate the Geometric Entity Selection section.
- 2 From the Selection list, choose Metal Conductors.

Cooling Fluid

- I In the Model Builder window, under Component I (compl)>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)

- I 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 I

- I 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

- I In the Model Builder window, expand the Membrane I node, then click Initial Values I.
- 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 I

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

- I In the Model Builder window, click Water Absorption-Desorption, 02 Side I.
- 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 I

- I 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 I

- I 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 I

- I 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 I

- I 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 I

- I 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, expand the Component I (comp1)>Hydrogen Fuel Cell (fc)> Electronic Conducting Phase I node, then click Electronic Conducting Phase I.

Electric Ground 1

- I 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 I.

Electric Potential 1

- I In the Physics toolbar, click 层 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

- In the Model Builder window, expand the Component I (comp1)>Hydrogen Fuel Cell (fc)>
 H2 Gas Phase I 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

- I In the Physics toolbar, click 层 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 I

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

H2 Outlet I

- I 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

- In the Model Builder window, expand the Component I (comp1)>Hydrogen Fuel Cell (fc)>
 O2 Gas Phase I 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 I

- I In the Physics toolbar, click 📃 Attributes and choose **02 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 I

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

O2 Outlet I

- I In the Physics toolbar, click 📃 Attributes and choose **02 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 I

- I In the Physics toolbar, click 间 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 I

- I In the Model Builder window, click Thin H2 Gas Diffusion Electrode Reaction I.
- 2 In the Settings window for Thin H2 Gas Diffusion Electrode Reaction, locate the Electrode Kinetics section.
- **3** In the $i_{0,ref}(T)$ 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

- I In the Physics toolbar, click 📁 Boundaries and choose Thin 02 Gas Diffusion Electrode.
- 2 In the Settings window for Thin 02 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 I

- I In the Model Builder window, click Thin O2 Gas Diffusion Electrode Reaction I.
- **2** In the Settings window for Thin **02** Gas Diffusion Electrode Reaction, locate the Electrode Kinetics section.
- **3** In the $i_{0,ref}(T)$ 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

- I In the Model Builder window, under Component I (compl) 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 I (rfh1)

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

FREE AND POROUS MEDIA FLOW - ANODE (FP)

Porous Medium I

- I 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 I

- I In the Model Builder window, click Porous Matrix I.
- 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

- I 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 I

- I 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

- I 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 I

- I 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

- I In the Model Builder window, under Component I (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).</p>

MULTIPHYSICS

Reacting Flow, O2 Gas Phase 1 (rfo1)

- I In the Physics toolbar, click An Multiphysics Couplings and choose Domain> Reacting Flow, O2 Gas Phase.
- **2** In the Settings window for Reacting Flow, **02** 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 I

- I In the Model Builder window, expand the Component I (compl)> 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 I

I In the Model Builder window, click Porous Matrix I.

- 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

- I 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 I

- I 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

- I 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 I

- I 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

- I In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Fluid I.
- 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

- I 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

- I 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

- I 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

- I 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 I

- I 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 I

- I 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 I

- I 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

- I 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)

- I In the Model Builder window, under Component I (compl) 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 I

- I 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 I

- I 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 I

- I 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 An Multiphysics Couplings and choose Domain> Electrochemical Heating.

Nonisothermal Flow - Anode Gas

- I In the Physics toolbar, click An 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

- I In the Physics toolbar, click A 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

- I In the Physics toolbar, click A 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)

- I In the Model Builder window, under Component I (compl) 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

- I In the Model Builder window, expand the Component I (compl)>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

- I In the **Definitions** toolbar, click **heat** 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

- I 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.

IO Click OK.

Sweep Domains

- I 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

- I 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 I

Size 1

- I In the Model Builder window, under Component I (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. Select the Minimum element size check box.
- 7 In the associated text field, type m_th/4.

Corner Refinement I

I In the Mesh toolbar, click A 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

- I In the Mesh toolbar, click \land 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

- I 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 I and choose Build All.



Boundary Layers 1

- I 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

- I 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

- I In the Mesh toolbar, click A 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 I

- I Right-click Swept I and choose Distribution.
- 2 Right-click Distribution I and choose Build All.



ADD STUDY

- I In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ 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 $\stackrel{\sim}{\longrightarrow}$ Add Study to close the Add Study window.

STUDY I

Stationary - Anode Flow

I 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 I		Automatic (Current distribution
(rfol)		initialization)

Stationary - Cathode Flow

- I 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 I		Automatic (Stationary)
(rfhl)		

Stationary - Cooling Flow

- I 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 I (rfh1)		Automatic (Stationary)
Reacting Flow, O2 Gas Phase 1 (rfo1)		Automatic (Stationary)

Stationary - All Physics Except Laminar Flow

- I 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 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver 5 node, then click Segregated I.
- 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 I>Solver Configurations> Solution I (soll)>Stationary Solver 5>Segregated I 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 I>Solver Configurations>Solution I (soll)> Stationary Solver 5>Segregated I click Temperature.
- 10 In the Settings window for Segregated Step, locate the Method and Termination section.
- II In the **Damping factor** text field, type 1.
- 12 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Stationary Solver 5>Segregated I click Velocity uc, Pressure pc.
- **13** In the Settings window for Segregated Step, locate the Method and Termination section.
- **I4** 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

- I In the Model Builder window, under Study I 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

- I 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

- I Right-click Membrane Current Density and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Hydrogen Fuel Cell>Electrolyte current density vector A/m²>fc.llz 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 **I** Plot.

E_cell(6)=0.5 V Surface: Electrolyte current density vector, z component (A/cm²)



Electrode Phase Potential, Anode Side

- I 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

- I Right-click Electrode Phase Potential, Anode Side and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Hydrogen Fuel Cell>fc.phis Electric potential V.

Selection 1

- I Right-click Surface I 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 **O Plot**.



Electrode Phase Potential, Cathode

- I 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 I

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

- 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 **O Plot**.



Mole Fraction, O2, Streamline (fc)

- I In the Model Builder window, under Results click Mole Fraction, 02, 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

- I In the Model Builder window, expand the Mole Fraction, 02, Streamline (fc) node, then click Streamline I.
- 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

- I In the Model Builder window, expand the Streamline I 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

- I In the Model Builder window, right-click Mole Fraction, O2, Streamline (fc) and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Hydrogen Fuel Cell>Species 02>fc.x02 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 I.

Selection I

- I Right-click Surface I and choose Selection.
- 2 In the Settings window for Selection, locate the Selection section.
- **3** From the Selection list, choose GDL Boundaries.

Surface 2

- I 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 I

- I 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

I Click the **Graphics** toolbar.

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



3 In the Mole Fraction, 02, Streamline (fc) toolbar, click **O** Plot.

Mole Fraction, H2O, Streamline (fc)

- I 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

- I In the Model Builder window, click Streamline I.
- 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

- I In the Model Builder window, expand the Streamline I 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

- I In the Model Builder window, right-click Mole Fraction, H2O, Streamline (fc) and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 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 I

- I Right-click Surface I 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

I Click the 🕂 Zoom Extents button in the Graphics toolbar.

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



Surface

- I 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 **I** Plot.



Cooling Channel Temperature

- I 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

- I Right-click Cooling Channel Temperature and choose Volume.
- In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 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. From the Color table list, choose HeatCameraLight.

Selection I

- I Right-click Volume I 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 **I** Plot.



Water Activity

- I 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

- I Right-click Water Activity and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Hydrogen Fuel Cell>fc.aw Water activity (relative humidity).

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



Surface: Water activity (relative humidity) (1)