
Tutorial: Modeling a Single-Channel, Counter-Flow Polymer Electrolyte Membrane (PEM) Fuel Cell

Introduction

The purpose of this tutorial is to provide guidelines and recommendations for using the PEM Fuel Cell add-on module available in FLUENT.

This tutorial demonstrates how to do the following:

- Build and generate a mesh for a single-channel PEM fuel cell.
- Assign zone names and types that are required in the FLUENT PEM fuel cell add-on module.
- Import the mesh into FLUENT, set up the case, and run the calculations.
- Postprocess the results.

Prerequisites

This tutorial assumes you are familiar with both the GAMBIT and FLUENT interfaces and that you have a good understanding of the basic setup and solution procedures. Details of most steps will not be shown explicitly.

If you have not used FLUENT before, it would be helpful to first refer to the FLUENT 6.3 User's Guide and the FLUENT 6.3 Tutorial Guide. You can also refer to the GAMBIT User's Guide, GAMBIT Tutorial Guide, and GAMBIT Modeling Guide.

Problem Description

A schematic description of a single, straight-channel, counter-flow PEM fuel cell is shown in Figure 1. The dimensions of the fuel cell are $2.4 \times 2.88 \times 125$ mm in the x , y , and z directions, respectively. This model represents a repeating channel of a larger counter-flow PEM fuel cell. The cross-sectional area of the membrane electrode assembly is $2.4 \times 125 = 300$ mm².

All the zones as marked in Figure 1 (with exception of the optional cooling channel) *must* be defined as separate zones within GAMBIT. Note that there can be multiples of each zone. For example, there can be multiple cathode gas diffusion layers, each with different material properties.

In FLUENT 6.3, all continuum zones must be of the fluid type except for the current collectors which may be either solid or fluid. For simplicity, it is recommended that you make the current collectors solid continuum zones.

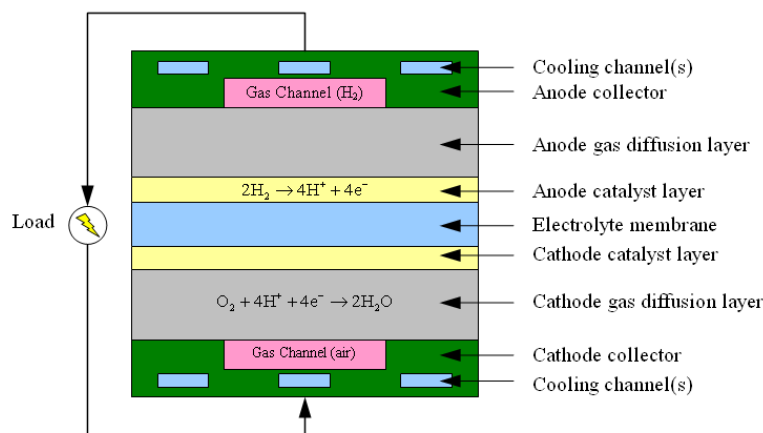


Figure 1: PEM Fuel Cell Zones

For details about the FLUENT PEMFC model, refer to [FLUENT 6.3 Fuel Cell Modules Manual](#).

Note: *If you would like to bypass all GAMBIT steps and proceed to the FLUENT portion of the tutorial, skip to Step .*

Preparation in GAMBIT




1. Create a folder called `pem-single-channel` to contain the files generated in this tutorial.
2. Start GAMBIT and specify this folder as the working folder.

To simplify the geometry construction and meshing steps, a journal file, `pem-single-channel.jou` is provided. This file contains GAMBIT instructions which will create the geometry and generate the mesh. It is recommended that you step through the journal file to understand each of the steps and recognize the assigned zone boundary types pertinent to fuel cell problems.

Geometry Creation and Mesh Generation

Step 1: Geometry Creation

1. Create a rectangular face primitive in the xy plane.




Operation → Geometry  → Face  → Create Real Rectangular Face 

- (a) Enter 2.4 for Width and 1.2 for Height.

(b) Select +X +Y for Direction,

(c) Click Apply and close the Create Real Rectangular Face panel.

2. Create another face by sweeping the uppermost edge by 0.21 units in the y -direction.

Operation → Geometry  → Face  → Sweep Edges 

3. Create another face by sweeping the uppermost edge by 0.012 units in the y -direction.

4. Create another face by sweeping the uppermost edge by 0.036 units in the y -direction.




5. Create another face by sweeping the uppermost edge by 0.012 units in the y -direction.

6. Create another face by sweeping the uppermost edge by 0.21 units in the y -direction.

7. Create another face by sweeping the uppermost edge by 1.2 units in the y -direction.




8. Create a rectangular face with dimensions $(x, y) = (0.8, 0.6)$. Use +x+y for the direction (so that the global coordinate system origin is in the lower left corner of the face).

9. Move the newly created face using a vector of $(0.8, 0.6, 0)$.

Operation → Geometry  → Face  → Move/Copy Faces 

10. Copy the face by 1.08 units in the y -direction.

11. Split the lowermost face with the face you just created.

Operation → Geometry  → Face  → Split Face 




12. Similarly, split the uppermost face with its internal face.

The resulting mesh should appear as shown in Figure 2.




Step 2: Mesh Generation (Manual)

1. Mesh the edges as shown in Figure 2.




The numbers of cells along each edge are indicated below. The geometry and mesh are symmetric about its horizontal midplane.

Operation → Mesh  → Edge  → Mesh Edges 

2. Mesh the nine faces using the Quad Submap scheme.

Operation → Mesh  → Face  → Mesh Faces 

3. Create an edge by sweeping any one of the vertices by 125 units in the positive z direction.

Operation → Geometry  → Edge  → Sweep Vertices 

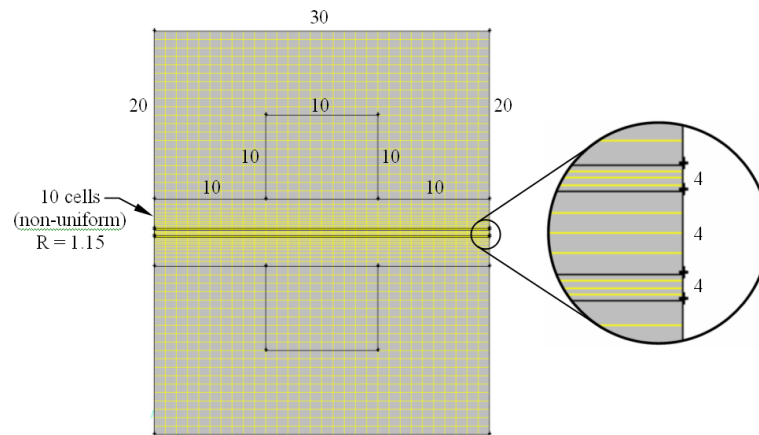





Figure 2: Edge and Face Mesh

- (a) Select Vector for Path and click the Define button.
 - (b) Enable Magnitude and enter 125.
 - (c) Select Positive for Z from the Direction list.
 - (d) Click Apply in the Vector Definition form.
 - (e) Click Apply and close the Sweep Vertices form.
4. Mesh the newly created edge using a double-sided graded edge mesh that consists of 60 elements.
 - (a) Enable Double sided.
 - (b) Enter 1.1 for Ratio 1 and Ratio 2.
 - (c) Click Apply and close the Mesh Edges form.
 5. Create volumes by sweeping the nine faces along the newly created edge.

Operation → Geometry  → Volume  → Sweep Faces 

- (a) Enable With Mesh.
- (b) Click Apply and close the Sweep Faces form.

The volume mesh is as shown in Figure 3.

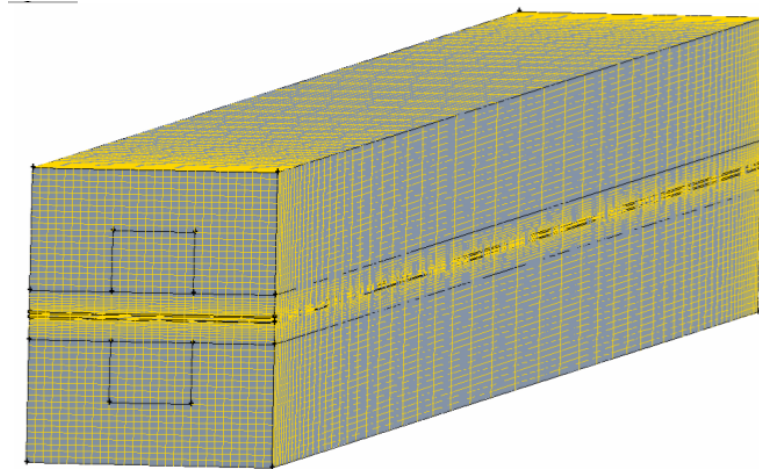


Figure 3: Volume Mesh

Step 3: Zone Assignments and Mesh Export

FLUENT's PEM fuel cell add-on module requires that boundary and continuum zones be rigorously defined. Care should be taken in order to have a logical system of naming each zone to represent each of the regions shown in Figure 1.

At a minimum, the boundary zones that are required include the following:

- Inlet and outlet zones for the anode gas channel.
- Inlet and outlet zones for the cathode gas channel.
- Surfaces representing anode and cathode terminals.

Optional boundary zones that could be defined include any voltage jump surfaces, interior flow surfaces, or non-conformal interfaces that are required.

The following continuum zones are also required:

- Flow channels for anode- and cathode-side fluids.
- Anode and cathode current collectors.
- Anode and cathode gas diffusion layers.
- Anode and cathode catalyst layers.
- Electrolyte membrane.

The inlets should all be assigned the boundary zone type **MASS FLOW INLET** and outlets should be assigned the **PRESSURE OUTLET** type. The terminals are the regions where the voltage (or current flux density) is known. Normally, the anode is grounded ($V = 0$) and

the cathode terminal is at a fixed potential that is less than the open-circuit potential. Both of the terminals should be assigned the WALL boundary type.

Voltage jump zones can optionally be placed between the various components (such as between the gas diffusion layer and the current collector). Faces which represent fluid/solid interfaces must be of type WALL.

Additional interior zones may be defined for purposes of post-processing. If such interior zones are defined, they should have no effect on the solution.

FLUENT’s PEM add-on module supports the use of non-conformal grid interfaces. In such cases, it is recommended that the INTERFACE boundary type be used halfway between the membrane continuum zone, and that the mesh on opposite sides of the interface have similar size, aspect ratio, and orientation. In such cases, the membrane will consist of two fluid zones.

1. Assign boundary zones according to the definitions listed in Table 1.

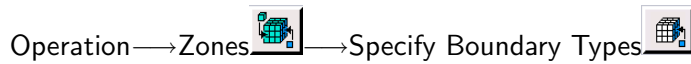


Table 1: Boundary Zone Assignments

Anode-side inlet ($z = 125$, upper)	inlet-a	MASS FLOW INLET
Cathode-side inlet ($z = 0$, lower)	inlet-c	MASS FLOW INLET
Anode-side outlet ($z = 0$, upper)	outlet-a	PRESSURE OUTLET
Cathode-side outlet ($z = 125$, lower)	outlet-c	PRESSURE OUTLET
Anode terminal ($y = 2.88$)	wall-terminal-a	WALL
Cathode terminal ($y = 0$)	wall-terminal-c	WALL
Anode-side flow channel walls	wall-ch-a	WALL
Cathode-side flow channel walls	wall-ch-c	WALL
Fuel cell ends	wall-ends	WALL
Anode-side diffusion layer walls	wall-gdl-a	WALL
Cathode-side diffusion layer boundaries	wall-gdl-c	WALL
Lateral boundaries of the fuel cell	wall-sides	WALL

2. Assign continuum zones according to the definitions listed in Table 2. Refer to Figure 1 when selecting which volumes to assign to each zone.

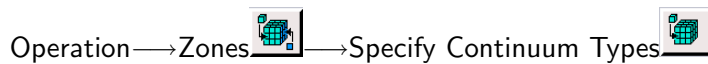


Table 2: Continuum Zone Assignments

Anode-side catalyst layer	catalyst-a	FLUID
Cathode-side catalyst layer	catalyst-c	FLUID
Anode-side flow channel	channel-a	FLUID
Cathode-side flow channel	channel-c	FLUID
Anode-side gas diffusion layer	gdl-a	FLUID
Cathode-side gas diffusion layer	gdl-c	FLUID
Electrolyte membrane	membrane	FLUID
Anode current collector	current-a	SOLID
Cathode current collector	current-c	SOLID

3. Export the mesh file as `pem-single-channel.msh`.

Setup and Solution in FLUENT

FLUENT's PEM Fuel Cell (PEMFC) model is provided as an add-on module with the standard FLUENT licensed software. A special license is required in order to use this model. The module is installed as part of the standard FLUENT installation in the folder `/addons/fuelcells2.2` within the FLUENT installation folder. The PEMFC model consists of a user-defined function (UDF) library and a compiled Scheme library which can be loaded using a text user interface (TUI) command.

trtitlePreparation

1. Copy the mesh file `pem-single-channel.msh.gz` to your working folder.

If you worked through the GAMBIT portion of this tutorial, an uncompressed version of this file will already be in place.

2. Start the 3DDP (3ddp) version of FLUENT.



If you wish to solve this case in parallel, you will need to set up and save the case file in serial mode first. Once this is done, you can start a parallel FLUENT session and proceed with the calculations.

Step 1: Grid

1. Read the mesh file, `pem-single-channel.msh.gz`.

`File` → `Read` → `Case...`

FLUENT will perform various checks on the mesh and will report the progress in the console window. Make sure that the reported minimum volume is a positive number.

2. Check the grid.

`Grid` → `Check`

A grid check should always be performed in order to verify the integrity of the mesh file. Specifically, you should verify that the minimum cell volume is a positive value.

3. Since the mesh was created in units of millimeters, it must be scaled.

Grid → Scale...

- (a) Select mm from the Grid Was Created In drop-down list.
- (b) Click the Change Length Units button.
- (c) Click Scale.
- (d) Verify that the value of Zmax is 125 mm and close the Scale Grid panel.

Step 2: Models

1. To load the PEMFC model using the text user interface, enter the following command in the console:

```
/define/models/addon-module 3
```

This command will load a Scheme library which contains the PEM model GUI and a UDF library. Upon successful execution, the following message will be displayed in the console:

```
Addon Module: fuelcells2.2...loaded!
```

2. Calculate the surface area of the membrane for post-processing.

In this case, the membrane area is equal to the surface area of the cathode terminal. This surface is named wall-terminal-c.

Report → Projected Areas...

- (a) Select Y from the Projection Direction group box.
- (b) Select wall-terminal-c from the Surfaces list.
- (c) Click Compute.

The projected area reported is 0.0003 m².

- (d) Close the Projected Surface Areas panel.

3. Change the Solution Zones for user-defined scalars 2 and 3 to all zones.

Define → **User-Defined** → Scalars...

This is an optional step which allows UDS-2 and UDS-3 to be postprocessed on both fluid and solid zones.

4. Configure the PEM model.

Define → **Models** → PEMFC...

- (a) Click the Anode tab.

- i. Select Current Collector in the Anode Zone Type group box and select current-a from the Zone(s) selection list.
 - ii. Select Flow Channel in the Anode Zone Type group box and select channel-a from the Zone(s) selection list.
 - iii. Select Diffusion Layer in the Anode Zone Type group box and select gdl-a from the Zone(s) selection list.
 - iv. Select Catalyst Layer in the Anode Zone Type group box and select catalyst-a from the Zone(s) selection list.
- (b) Click the Membrane tab.
- i. Select membrane from the Membrane Zone(s) selection list.
- (c) Click the Cathode tab.
- i. Select Current Collector in the Cathode Zone Type group box and select current-c from the Zone(s) selection list.
 - ii. Select Flow Channel in the Cathode Zone Type and select channel-c from the Zone(s) selection list.
 - iii. Select Diffusion Layer in the Cathode Zone Type and select gdl-c from the Zone(s) selection list.
 - iv. Select Catalyst Layer in the Cathode Zone Type and select catalyst-c from the Zone(s) selection list.
- (d) Click the Reports tab.
- i. Specify the value of Membrane-Electrode-Assembly Projected Area to 0.0003 m^2 .
Recall that this value was obtained earlier in the tutorial.
 - ii. Select wall-terminal-a from the Anode selection list and select wall-terminal-c from the Cathode selection list.
- (e) Click OK to close the PEM panel.

FLUENT reports in the console that the energy equation has been enabled automatically. For convenience, the PEMFC model also automatically enables species transport and creates default materials.

Step 3: Materials

Retain the default settings for the materials.

Step 4: Operating Conditions

Define → Operating Conditions...

1. Set the Operating Pressure to 200000 pascal.
2. Click OK to close the Operating Conditions panel.

Step 5: Boundary Conditions

Define → Boundary Conditions...

There are several zones which must be specified in the boundary conditions panel. These are the anode and cathode voltage terminals, as well as the inlets and outlets.

1. Set boundary conditions for the anode voltage terminal, wall-terminal-a.

At this surface, the voltage is grounded and the temperature is constant.

- (a) Click the Thermal tab and enter 353 K for Temperature.
- (b) Click the UDS tab.
 - i. Select Specified Value from the Electric Potential drop-down list User-Defined Scalar Boundary Condition group box.
 - ii. Enter 0 for Electric Potential in the User-Defined Scalar Boundary Value group box.

This boundary condition represents a grounded terminal.

- (c) Click OK to close the Wall panel.

2. Set boundary conditions for the cathode voltage terminal, wall-terminal-c.

At this surface, the voltage is maintained at a constant, positive value.

- (a) Click the Thermal tab and enter 353 K for Temperature.
- (b) Click the UDS tab.
 - i. Select Specified Value from the Electric Potential drop-down list User-Defined Scalar Boundary Condition group box.
 - ii. Enter 0.75 for Electric Potential in the User-Defined Scalar Boundary Value group box.

This boundary condition represents a terminal operating at 0.75 Volts.

- iii. Click OK to close the Wall panel.

To calculate an IV polarization curve, you should vary the Electric Potential for the cathode, starting from a voltage near the open circuit voltage and gradually decreasing it, converging the solution each time you change the value.

3. Set boundary conditions for the anode gas flow inlet, inlet-a.

At this inlet, a humidified hydrogen stream enters the fuel cell. No liquid enters the channel.

- (a) Enter $6.0\text{e-}7$ kg/s for Mass Flow Rate and 0 for Supersonic/Initial Gauge Pressure.
- (b) Click the Thermal tab and enter 353 K for Temperature.
- (c) Click the Species tab and set the mass fractions of h2, o2 and h2o to 0.8, 0.0, and 0.2, respectively.
- (d) Click the UDS tab and select Specified Value from the Water Saturation drop-down list in the User-Defined Scalar Boundary Condition group box.

- (e) Enter 0 for Water Saturation in the User-Defined Scalar Boundary Value group box.
 - (f) Click OK to close the Mass-Flow Inlet panel.
4. Set boundary conditions for the cathode gas flow inlet, inlet-c.
At this inlet, a humidified air stream enters the fuel cell. No liquid enters the channel.
 - (a) Enter $5.0\text{e-}6$ kg/s for Mass Flow Rate.
 - (b) Click the Thermal tab and enter 353 K for Total Temperature.
 - (c) Click the Species tab and set the mass fractions of h2, o2 and h2o to 0.0, 0.2, and 0.1, respectively.
 - (d) Click the UDS tab and select Specified Value from the Water Saturation drop-down list in the User Defined Scalar Boundary Condition group box.
 - (e) Enter 0 for Water Saturation in the User-Defined Scalar Boundary Value group box.
 - (f) Click OK to close the Mass-Flow Inlet panel.
 5. Set boundary conditions for the anode gas flow outlet, outlet-a.
 - (a) Click the Thermal tab and enter 353 K for Backflow Total Temperature.
 - (b) Click OK to close the Pressure Outlet panel.
 6. To set boundary conditions for the cathode gas flow outlet, copy the boundary conditions from outlet-a to outlet-c
 7. Close the Boundary Conditions panel.

Step 6: Solution Controls

The default solver settings are not sufficient to obtain a converged solution. Therefore, the following modifications must be made.

1. Set the under-relaxation factor for Pressure to 0.7, Momentum to 0.3, Protonic Potential to 0.95, and Water Content to 0.95.

→ → Solution...

2. Modify the multigrid settings.

→ → Multigrid...

- (a) Select F-Cycle from the Cycle Type drop-down lists for all equations.
You will need to scroll down to set all equations.
- (b) Enter 0.001 for Termination Restriction for h2, o2, h2o, and Water Saturation.
- (c) Select BCGSTAB from the Stabilization Method drop-down list for h2, o2, h2o, Water Saturation, Electric Potential and Protonic Potential.
- (d) Enter 0.0001 for Termination Restriction for Electric Potential and Protonic Potential.
- (e) Increase the value of Max Cycles to 50 in the Algebraic Multigrid Controls group box.

(f) Click OK to close the Multigrid Controls panel.

3. Enable the plotting of residuals.

Solve → **Monitors** → Residual...

Note: *The PEMFC model automatically disables convergence checking for all equations.*

4. Initialize the solution.

Solve → **Initialize** → Initialize...

(a) Set Temperature to 353 K.

(b) Click Apply.

(c) Click Init and close the Solution Initialization panel.

5. Save the case and data files as `pem-single-channel.cas.gz` and `pem-single-channel.dat.gz`.

File → **Write** → Case & Data...



If you want to run the calculations in parallel, exit FLUENT and start a parallel session at this point. Open the case and data files you saved in the previous step and proceed.

6. Request 200 iterations.

The solution residuals will drop to acceptable values.

Solve → Iterate...

*The solution residual plot should resemble that shown in Figure 4. The average current density is displayed in the console at the end of each iteration. At the end of the calculations, the current density is reported as approximately **0.324 A/cm²**.*

Step 7: Postprocessing

1. Create surfaces for postprocessing.

Surface → Iso-Surface...

(a) Select Grid... and Z-Coordinate from the Surface of Constant drop-down lists.

(b) Click Compute.

(c) Enter 62.5 for Iso-Values (mm).

(d) Enter `plane-xy` for New Surface Name.

(e) Click Create.

(f) Similarly, create another surface along the length of the fuel cell. This surface should be a surface of constant X-Coordinate, with a value of 1.2. Name this surface `plane-yz`.

(g) Close the Iso-Surface panel.

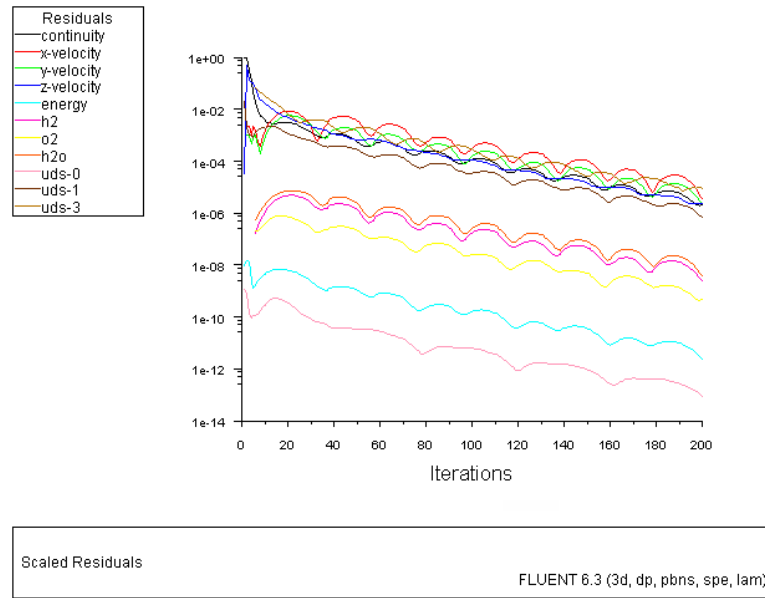


Figure 4: Residual Plot

2. Create custom vectors for display.

→ Vectors...

- (a) Click the Custom Vectors... button to open the Custom Vectors panel.
 - i. Enter current-flux-density for Vector Name.
 - ii. Select User Defined Memory... and X Current Flux Density from the X Component drop-down lists.
 - iii. Select User Defined Memory... and Y Current Flux Density from the Y Component drop-down lists.
 - iv. Select User Defined Memory... and Z Current Flux Density from the Z Component drop-down lists.
 - v. Click Define and close the Custom Vectors panel.
- (b) Select current-flux-density from the Vectors of drop-down list.
- (c) Select filled-arrow from the Style drop-down list.
- (d) Click the Vector Options... button to open the Vector Options panel.
 - i. Enter 0.5 for Scale Head.
 - ii. Click Apply and close the Vector Options panel.
- (e) Select User-Defined Memory and Current Flux Density Magnitude from the Color by drop-down lists.
- (f) Enable Draw Grid from the Options list to open the Grid Display panel.
 - i. Deselect all surfaces except plane-xy from the Surfaces selection list.

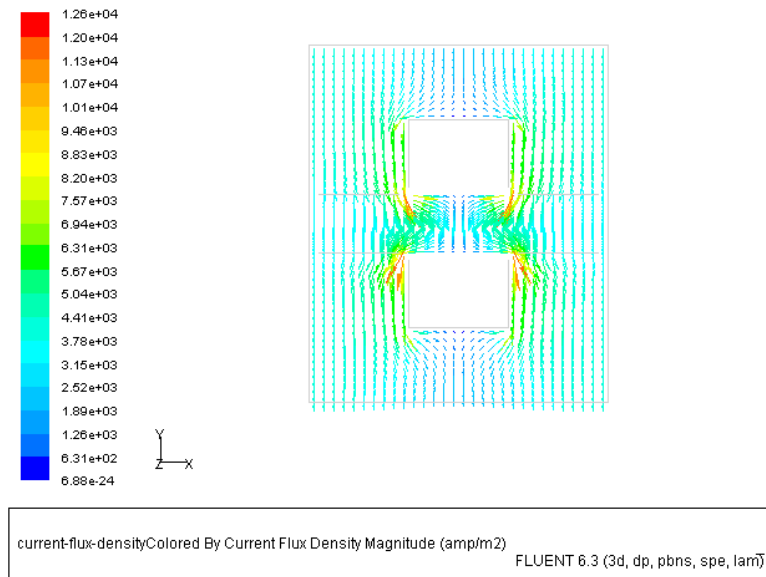


Figure 5: Current Flux at a Cross-Section Midway Down the Length of the PEM Channel

- ii. Ensure that Edges is enabled from the Options list and Feature is selected from the Edge Type list.
 - iii. Click Display and close the Grid Display panel.
 - (g) Select plane-xy in the Surfaces selection list.
 - (h) Click Display and close the Vectors panel.
3. Auto-fit the display to the graphics window by pressing **Ctrl-A** on the keyboard.
 4. Compare your results with those shown in Figure 5.

Note: *The maximum current density occurs in the regions between the channels and also that the plot units are A/m^2 .*

5. Plot contours of hydrogen mass fraction along the surface plane-yz

Display → Contours...

- (a) Enable Filled and Draw Grid from the Options list.
- (b) Select plane-yz from the Surfaces selection list in the Grid Display panel.
- (c) Click Display and close the Grid Display panel.
- (d) Restore the right view.

Display → View...

- i. Select right from the Views selection list.
- ii. Click Apply and close the Views panel.

The display updates to the right-hand view.

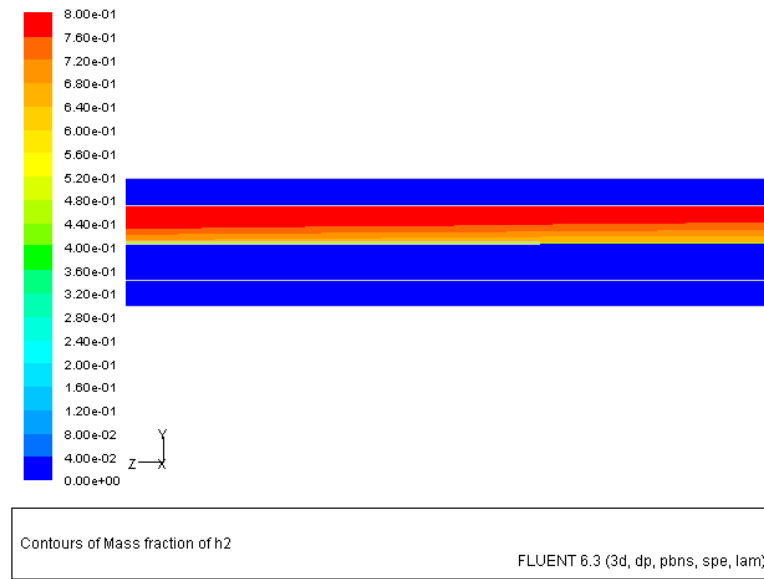


Figure 6: Contours of Hydrogen Mass Fraction Along the Channel Length

- (e) Select plane-yz from the Surfaces selection list.
- (f) Select Species... and Mass fraction of h2 from the Contours of drop-down lists.
- (g) Click Display.

The resulting display is difficult to visualize since the aspect ratio of the channel is large. You can change the way FLUENT displays data using the following steps:

→ Scene...

- i. Select all entries from the Names selection list.
- ii. Click the Transform... button to open the Transformations panel.
- iii. Set Z to 0.1 in the Scale group box.
- iv. Click Apply and close the Transformations panel.

The graphics display will be scaled accordingly.

- (h) Auto-fit the image to the window by pressing **Ctrl-A** in the graphics window.
- (i) Compare your result with that shown in Figure 6.

The flow in the anode (upper) channel is from left to right. Note that the hydrogen mass fraction decreases in the direction of flow. This is due to water being pulled through the membrane along with hydrogen as it is consumed in the fuel cell.

6. Plot contours of oxygen mass fraction along the surface plane-yz

→ Contours...

- (a) Follow the same procedure to generate the oxygen mass fraction contour plot that you used to generate the hydrogen mass fraction contours.

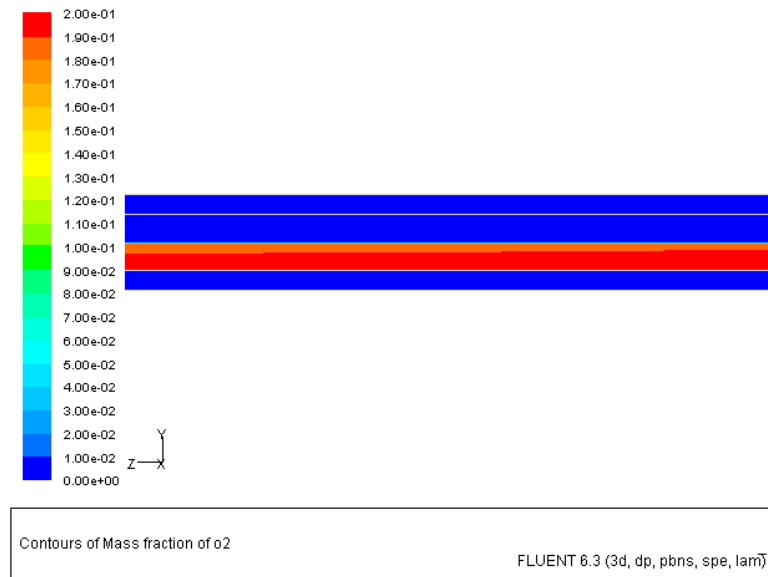


Figure 7: Contours of Oxygen Mass Fraction Along the Channel Length

- (b) Compare your result with that shown in Figure 7. *The flow in the cathode (lower) channel is from right to left. As expected, the oxygen mass fraction decreases in the direction of flow.*
7. Verify that global conservation of mass is observed. This will be done using a few text user interface (TUI) commands and basic electrochemistry concepts.
- (a) Compute the net oxygen consumption.

Enter the TUI command:

```
/report/species-mass-flow
```

If you have read a data file instead of performing iterations, you must perform at least one iteration in order to populate this data from the solver.

The output is as follows:

```
zone 22 (inlet-a): (4.8e-07 0 1.2e-07)
zone 21 (inlet-c): (0 1e-06 5e-07)
zone 20 (outlet-a): (-4.6958153e-07 -8.9805489e-11 -2.8157297e-07)
zone 19 (outlet-c): (-2.6251551e-10 -9.1930928e-07 -4.2904232e-07)
zone 53 (wall-ch-a-shadow): (0 0 0)
zone 15 (wall-ch-c): (0 0 0)
zone 29 (wall-ends:029): (0 0 0)
zone 30 (wall-ends:030): (0 0 0)
zone 31 (wall-ends:031): (0 0 0)
zone 32 (wall-ends:032): (0 0 0)
zone 33 (wall-ends:033): (0 0 0)
```

```

zone 51 (wall-gdl-a-shadow): (0 0 0)
zone 11 (wall-gdl-c): (0 0 0)
zone 1 (wall-sides:001): (0 0 0)
zone 23 (wall-sides:023): (0 0 0)
zone 25 (wall-sides:025): (0 0 0)
zone 26 (wall-sides:026): (0 0 0)
zone 27 (wall-sides:027): (0 0 0)

net species-mass-flow: (1.0155954e-08 8.0600915e-08 -9.061529e-08)

```

To interpret the output, each line can be read as

`zone num (zone-name): ($\dot{m}_1 \dot{m}_2 \dots \dot{m}_n$)` where the subscripts 1, 2, ..., n refer to each species being calculated. Here, we are considering three species, namely h_2 , o_2 and h_2o . Therefore, the *second* value in each list is the calculated oxygen mass flow rate in kilograms per second. In addition, a negative number indicates flow out of the domain from that boundary. From the last line, the net oxygen consumed is 8.06×10^{-8} kg/s. The molecular weight of oxygen is 31.9988 kg/kmol. Also, since the valence of a diatomic oxygen molecule is 4, there are 4 kmol of electrons released per kmol of oxygen. Finally Faraday's constant is 9.6485×10^7 C/kmol-electrons. Thus, the total release of electrons (which is equivalent to the current in Amperes), is

$$I = \frac{\dot{m}v\mathcal{F}}{M} = \frac{(8.06 \times 10^{-8})(4.0)(9.6485 \times 10^7)}{31.9988} = 0.972 \text{ A} \quad (1)$$

The total current is obtained by integrating the current density over the surface of the terminal. This integral value can be calculated in several ways. One way is to multiply the membrane area by the reported current density. This gives

$$I = (0.0003)(0.3241)(100)^2 = 0.972 \text{ A} \quad (2)$$

Alternatively, you can integrate the user memory Y Current Flux Density (the y -component of current density) over the terminal surface. This integration yields an accurate result since the y -direction is normal to the terminal. To do this, use the Surface Integrals panel.

Report → Surface Integrals...

- i. Select Report Type from the Integral drop-down list.
- ii. Select User Defined Memory... and Y Current Flux Density from the Field Variable drop-down lists.
- iii. Select wall-terminal-a from the Surfaces selection list and click Compute.

The absolute value of the number reported is approximately 0.972 A. We have electrochemical balance in the calculations.

- iv. Close the Surface Integrals panel.

Summary

In this tutorial, you learned how to set up and model a single-channel PEM fuel cell. The model provides detailed information on the distribution of current and voltage on all the electrically conducting regions, along with species and current flux density distribution throughout the fuel cell.