

Appendix A

Instruction for Running the PEMFC and SOFC Models and Their Distributed Generation Applications

The CD-ROM included with this book contains the following files:

- A physically-based dynamic model for a 500-W PEM Fuel Cell (PEMFC) stack developed in Chapter 3,
- A physically-based dynamic model for a 5-kW Solid Oxide Fuel Cell (SOFC) stack, developed in Chapter 4,
- A PEMFC distributed generation (PEMFCDG) system model, discussed in Chapter 7. The PEMFCDG system model is based on the 500-W PEMFC stack model.
- A SOFC distributed generation (SOFCDG) system model, discussed in Chapter 7. The SOFCDG system model is based on the 5-kW SOFC stack model.

The following sections give a brief instruction for running the above models. Basic knowledge of MATLAB/Simulink or PSpice is required to run the models. The reader is referred to the *MATLAB User's Guide* and *PSpice User's Guide* for help on how to run simulations using MATLAB/Simulink and PSpice.

I. PEMFC Model

The dynamic PEMFC model is an autonomous module operated under constant channel pressure with no control on the input fuel flow into the fuel cell. The fuel cell stack will adjust the input fuel flow according to its load current to keep the channel pressure constant. In addition to the model developed in MATLAB/Simulink, an equivalent electrical circuit model for the same PEMFC stack has also been developed using PSPICE. Both models have been validated through experimental data obtained on a 500-W Avista (now Relion) PEMFC stack. The models can be used in studies related to PEMFC performance evaluation, controller design, fuel cell vehicle studies, and for developing models of high power PEMFC power plants, to name a few.

A. PEMFC MATLAB/SIMULINK Model

◆ Steps to run the model

1. Look for the folder “PEMFC_Model\PEMFC_Simulink” and open the file of the model named “PEMFC500W.mdl” in MATLAB/ SIMULINK. The model was built using MATLAB Version 7.0.4.365 (R14) Service Pack 2.
2. Set proper input quantities for the model. The fuel cell (load) current can either be set explicitly, or be measured from an actual load and then be fed back to the model.
3. Choose a proper solver and set the simulation time. The recommended solver for this model is “ode23tb”. Select the solver by clicking “Simulation” in the menu and then select “Simulation parameters.”
4. Click “Start simulation” to run the model.

◆ Model Input Quantities:

Fuel cell load current, I (A). Maximum load current is 25A.

Anode channel pressure, P_{anode} (atm)

Cathode channel pressure at, $P_{cathode}$ (atm)

Room temperature, T_{room} (K)

Initial temperature of FC, $T_{initial}$ (K)

◆ Model Output Quantities

Output terminal voltage, V (V)

FC temperature, T_{out} (K)

◆ Example files

1. Before running the MATLAB/SIMULINK model, three example load current files need to be loaded into Workspace. To do this, one can just run the file “firstload.m” under the same directory. It will load the three example files into the MATLAB Workspace.
2. Any load current data can be used for simulation studies. However, the load current is limited to 25A in the model. Figure A.1 shows the SIMULINK diagram for an example, where a resistor R is connected to the PEMFC model and the load current is fed back to the model as input.

3. Proper simulation time needs to be set before simulation. Simulation time is determined by the length of simulation study desired and/or the length of the actual input data. Simulation time for the following three examples are:

InputI_stdy_ideal: ideal fuel cell load current in steady state, simulation time: 4900s;

InputI_stdy: real fuel cell load current in steady state, simulation time: 3900s;

InputI-dynshrt4: real fuel cell load current in transient state, simulation time: 2040s.

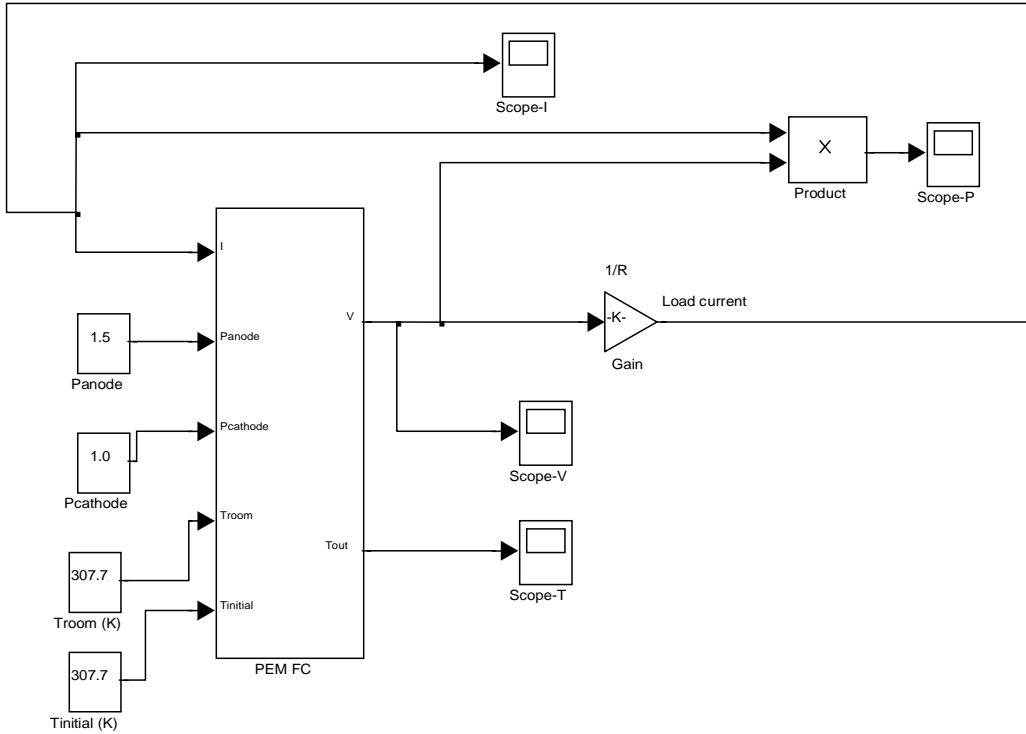


Figure A.1 Example for using the PEMFC model in MATLAB/Simulink.

B. PEMFC PSpice Model

◆ Steps to run the model

1. Look for the folder of “PEMFC_Model\PEMFC_Pspice\Version 10.1.0.p001.”
Open the model file “ETModel.opj” in Capture CIS (Version 10.0.0.p001).
2. Look for a block named “PEMFC500”. Connect the model in your simulation system properly. The fuel cell model block can be used as a regular PSpice component. An example showing how to run the model is given below.

◆ Input Terminals:

Room temperature: Troom (degrees Kelvin presented in volts)
Ground (see Figure A.2)

◆ Output Terminals

Output terminal, Vout

FC temperature, Tout (degrees Kelvin presented in volts)

◆ Example

Figure A.2 shows an example system consisting of the above PSPICE PEMFC model and a super-capacitor (the series combination of a large capacitor and a small resistor) connected across the PEMFC model output terminals. The load current is defined in a file named “iload_c”, which is saved under of the model file folder. Load current for a desired simulation study can be defined as needed, as long as the current does not exceed 25 A. Make sure the load definition file is clearly specified in the system. Otherwise, one may get an error for finding the load file. It is not necessary to model the load only as a current source, which is pre-defined by a file in the example. For example, the load could be just as simple as a resistor.

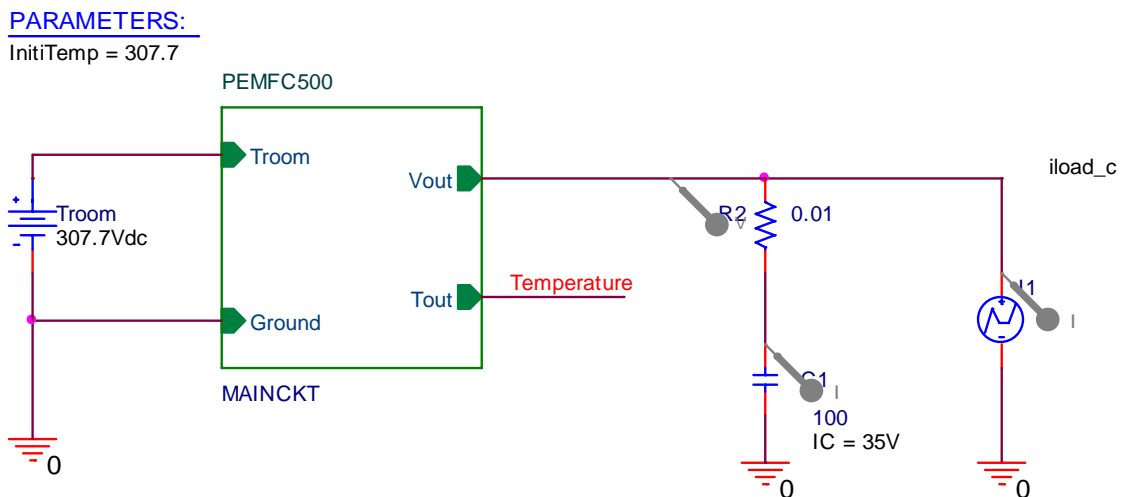


Figure A.2 Example for using the PEMFC model in PSpice.

II. SOFC Model

The SOFC model has 9 input quantities and two main outputs, as listed in the table A.1. The “Scope” block in MATLAB/Simulink can be used to measure any quantities of interest. Some scope blocks have already been used in the model to measure the following quantities: Load current, FC output power, FC output voltage, FC temperature, activation voltage drop, ohmic voltage drop, concentration voltage drop, fuel utilization and effective partial pressure of H₂, H₂O and O₂.

Table A.1 SOFC model input and output quantities

◆ Input quantities

I	Fuel cell load current (A). Maximum current is 160A.
Pa	Anode channel pressure (atm)
MH ₂	Mole flow rate of H ₂ input at the anode (mol/s)
MH ₂ O	Mole flow rate of H ₂ O input at the anode (mol/s)
Pc	Cathode channel pressure (atm)
Tairinlet	Air temperature at air inlet (K)
Mair	Mole flow rate of air (mol/s)
Tfuelinlet	Fuel temperature at fuel inlet (K)
TSteadyState	A Boolean value, 0/1 Option ‘1’ is used for steady-state study only, where, there is no temperature dynamics. The fuel cell temperature is set by Tairinlet, meaning that Tout will not change. .

Option ‘0’ is used for dynamic analysis, where thermal dynamics are included in the model.

◆ Output quantities

Vout	Output voltage of the SOFC model (V)
Tout	Temperature of the SOFC model (K)

◆ Steps to run the model

1. Start MATLAB/ Simulink (Version 7.0.4.365 (R14) Service Pack 2).
2. In the folder “\SOFC_Model,” open the model file “SOFC_5kW.mdl” in MATLAB/Simulink.
3. Set the proper model input quantities.
4. Choose a proper solver and set the simulation time. The recommended solver for this model is “ode23tb”. Select the solver by clicking on “Simulation” in the menu and then select “Simulation parameters.”
5. Click “Start simulation” to run the model.
6. Observe any output by double-clicking on its corresponding scope block.

◆ Examples

1. Steady-state characteristics of the SOFC model

Set $T_{SteadyState} = 1$. Set the proper T_{airlet} , which in this case is taken as the fuel cell operating temperature. For example, set $T_{airlet} = 1173K$. Use the “Ramp” block for the load current input. Slope rate = 0.01. Simulation time = 15800 sec. Under this scenario, the load current will increase from 0 to 158A at the rate 0.01A/s. Other parameter values (for the parameters given in Table A.1) are as set in the model. The parameter values can be changed by the user. Run the model and check the results.

2. Dynamic response of the SOFC model

Set $T_{SteadyState} = 0$. Set $T_{airinlet}$ and $T_{fuelinlet}$ accordingly. For example, set them both to 1173K, as shown in Figure A.3. Use “I_step” block given in the model file to simulate load transients. Simulation time = 15800s. Default values are used for other input quantities. Run the model and check its dynamic response.

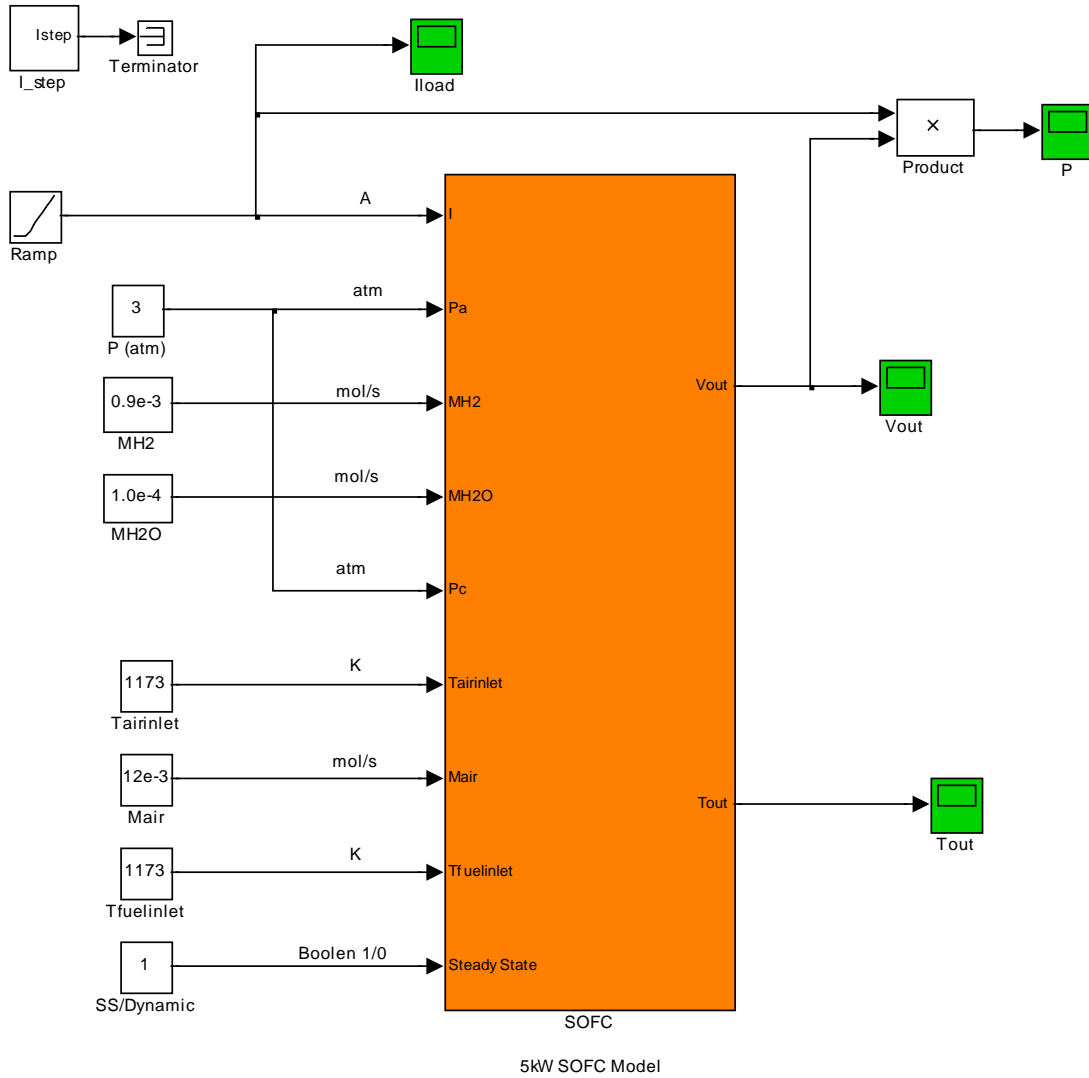


Figure A. 3 Example for using the SOFC model in MATLAB/Simulink.

III. FCDG Systems

The example FCDG systems based on both PEMFC and SOFC stack models are given in the CD. The FCDG system configurations have been discussed and shown in Figure 7.1 of Chapter 7. The principle of operation and design of the power electronic circuits (the DC/DC converters and the inverter) in the systems are addressed in Chapters 6, and controller design for the inverter and DC/DC converters is discussed in Chapter 7. A brief introduction for running the SOFC DG system simulation is given below. A similar procedure can be followed to run the PEMFC DG simulation system file.

◆ Steps to run the model

1. Start MATLAB/ Simulink (Version 7.0.4.365 (R14) Service Pack 2). SimPowerSystems toolbox (version 4.0.1) under Simulink was used to model the power electronic components in the system. Running the system on a different version of MATLAB/Simulink may cause incompatibility problems. Assistance from MATLAB technical support professionals may be required.
2. Look for the folder of “\FCDG” and open the model file with name of “SOFC_DG.mdl”. User should see the subsystem blocks of fuel cell power plant, DC/DC converters and their controllers, DC/AC inverter and its controller, and the utility grid. The details of each subsystem can be explored by either double-clicking on it or by right-clicking on it and choosing “Look Under Mask” from the pop-up menu. A simple Delay Block with a small time constant has been used in the system to avoid any algebraic loop.
3. The default sampling time for the pulse width modulation (PWM) blocks used for the power switching devices is set to $T_s = 2 \mu\text{s}$. This parameter can be changed as needed. However, any value larger than $10 \mu\text{s}$ is not recommended.
4. Choose the proper solver and set the simulation time. The recommended solver for this model is “ode23tb”. Select the solver by clicking on “Simulation” in the menu and then select “Simulation parameters.”

5. In order to increase simulation speed, the use of Simulink Accelerator is recommended. Even with the Accelerator, *be aware that it may take quite some time (several hours) to finish the simulation.*
6. Click “Start simulation” to run the model.
7. When the simulation is done, the outputs can be observed by loading the output data files, saved as MAT type in the same folder, to MATLAB workspace. Note that the desired outputs are saved as their corresponding MAT files. For example, “P.mat” is the data file for real power data, P. The command “load P.mat” is used to load the real power data when the simulation is finished. Then, the power curve can be plotted by using the command “plot(P(1,:), P(2,:))”.