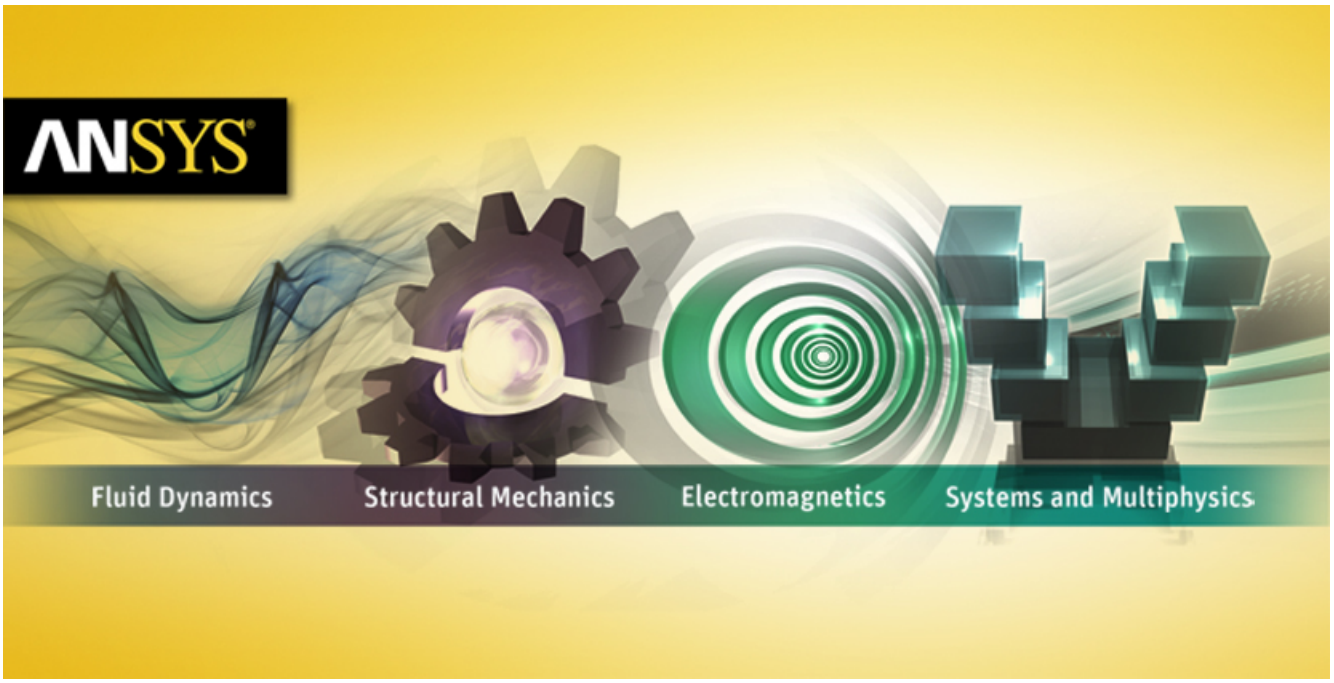


ANSYS FLUENT Battery Module Manual

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 14.0
November 2011

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual

The ANSYS FLUENT Battery Module Manual provides information about using the Battery Model with ANSYS FLUENT. In this manual, you will find a theoretical discussion of the model used in ANSYS FLUENT, and a description of using the model with your CFD simulations.

2. The Contents of the FLUENT Manuals

The manuals listed below form the FLUENT product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use FLUENT products.

FLUENT Getting Started Guide contains general information about getting started with using FLUENT.
FLUENT User's Guide contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.
FLUENT in Workbench User's Guide contains information about getting started with and using FLUENT within the Workbench environment.
FLUENT Theory Guide contains reference information for how the physical models are implemented in FLUENT.
FLUENT UDF Manual contains information about writing and using user-defined functions (UDFs).
FLUENT Tutorial Guide contains a number of example problems with detailed instructions, commentary, and postprocessing of results.
FLUENT Text Command List contains a brief description of each of the commands in FLUENT's text interface.
FLUENT Adjoint Solver Module Manual contains information about the background and usage of FLUENT's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.
FLUENT Battery Module Manual contains information about the background and usage of FLUENT's Battery Module that allows you to analyze the behavior of electric batteries.
FLUENT Continuous Fiber Module Manual contains information about the background and usage of FLUENT's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
FLUENT Fuel Cell Modules Manual contains information about the background and the usage of two separate add-on fuel cell models for FLUENT that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with FLUENT.
FLUENT Magnetohydrodynamics (MHD) Module Manual contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.

FLUENT Migration Manual contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.
FLUENT Population Balance Module Manual contains information about the background and usage of FLUENT's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.
Running FLUENT Under LSF contains information about the using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.
Running FLUENT Under PBS Professional contains information about the using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.
Running FLUENT Under SGE contains information about the using FLUENT with Sun Grid Engine (SGE) software, a distributed computing resource management tool.

3. Typographical Conventions

Several typographical conventions are used in this manual's text to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (for example, **Iso-Surface** dialog box, `surface/iso-surface` command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a dialog box. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.
- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,

  Models → **Multiphase** → **Edit...**

  indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Multiphase** is selected from the list. Clicking the **Edit...** button opens the **Multiphase** dialog box.

  Also, a mini flow chart is used to indicate the menu selections that lead you to a specific command or dialog box. For example,

  Define → **Injections...**

  indicates that the **Injections...** menu item can be selected from the **Define** pull-down menu, and

  `display` → `mesh`

  indicates that the `mesh` command is available in the `display` text menu.

  In this manual, mini flow charts usually precede a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They allow you to look up information about a command or dialog box and quickly determine how to access it without having to search the preceding material.
- The menu selections that will lead you to a particular dialog box or task page are also indicated (usually within a paragraph) using a "**/**". For example, **Define/Materials...** tells you to choose the **Materials...** menu item from the **Define** pull-down menu.

4. Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, I).
- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{\imath}+\frac{\partial}{\partial y}\vec{j}+\frac{\partial}{\partial z}\vec{k} \tag{1}$$

  ∇ appears in several ways:

  – The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p=\frac{\partial p}{\partial x}\vec{\imath}+\frac{\partial p}{\partial y}\vec{j}+\frac{\partial p}{\partial z}\vec{k} \tag{2}$$

  – The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v})=\left(\frac{\partial}{\partial x}\vec{\imath}+\frac{\partial}{\partial y}\vec{j}+\frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{\imath}+v_j\vec{j}+v_z\vec{k}\right) \tag{3}$$

  This tensor is usually written as

$$\begin{pmatrix}\frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z}\\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z}\\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z}\end{pmatrix} \tag{4}$$

  – The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla\cdot\vec{v}=\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}+\frac{\partial v_z}{\partial z} \tag{5}$$

  – The operator $\nabla\cdot\nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \tag{6}$$

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2 \tag{7}$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in "Modeling Turbulence" in the User's Guide, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

5. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS FLUENT.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **About ANSYS> Contacts and Locations**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: `Systems/Installation`. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Telephone: +81-3-5324-7305

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225

Email: support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Chapter 1: Battery Model Theory

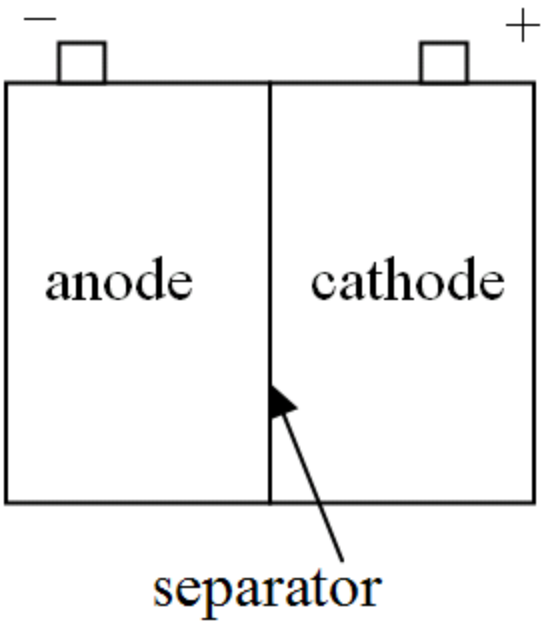
The ANSYS FLUENT Battery Model is provided as an add-on module with the standard ANSYS FLUENT licensed software. This document describes the model theory that includes mathematical equations and physical interpretations of independent and dependent variables used in the model. The procedure for setting up and solving battery modeling is described in detail in *Using the Battery Model* (p. 5).

1.1. Introduction

Given the important role of a battery in electric and/or hybrid electric vehicles there have been a number of models proposed in the literature to simulate the transient behavior of a rechargeable battery. These models vary in complexity from a zero-dimension resistor-capacitor circuit to a multi-dimension potential-current distribution, and, all the way to detailed electrochemistry modeling inside active separator layers. Computing resources (CPU time and memory) increase considerably with model complexity. Combining the need for model accuracy and the requirement for model usability, ANSYS FLUENT has adopted a modeling approach that was based upon the 1D model initially proposed by Tiedemann and Newman [*1*] (p. 15), later used by Gu [*2*] (p. 15), and, more recently used by Kim et al [*3*] (p. 15) in their 2D study. ANSYS FLUENT has extended the model formulation for use in 3D computations.

1.2. Computation of the Electric Potential and Current Density

The computational domain includes only the anode and cathode electrodes and their current collectors. The separator layer is modeled as an infinitely thin interface between the two electrodes across which there is a potential jump due to loss. A cross-section of the anode-cathode assembly is schematically shown in the figure below.



The integral form of the electric potential equation reads,

$$\int_V \nabla \cdot (\sigma \nabla \phi)\, dV = \int_A j dA \qquad \textbf{(1–1)}$$

In the above equation the source term *j,* also called the apparent current density, has non-zero value only at the anode-cathode interface (separator interface); *A* is the local surface area of the interface, and σ is the electric conductivity.

In their independent studies of small electrodes, Tiedemann and Newman [*1*] (p. 15), and Gu [*2*] (p. 15) observed that the apparent current density *j* (*A/m2*) varies *linearly* with cell voltage,

$$j = Y\,(\phi_c - \phi_a - U\,) \qquad \textbf{(1–2)}$$

where the cell voltage is the difference between the cathode- and anode-side electric potentials at the separator interface ($\phi_c - \phi_a$). From *Equation 1–2* (p. 2) it is clear that on an experimentally measured polarization curve, namely the voltage-current (V-I) curve of a battery cell, *U* would be the intercept of V at I=0; and, *Y* would be the inverse of the slope of the V-I curve. Moreover, Gu's experimental data [*2*] (p. 15) indicate that both *U* and *Y* vary with respect to the depth of discharge (DoD) defined relative to the theoretical battery capacity (*Qt*),

$$U = a_0 + a_1 \cdot DoD + a_2 \cdot DoD^2 + a_3 DoD^4 + a_4 DoD^4 + a_5 DoD^5 \qquad \textbf{(1–3)}$$

$$Y = b_0 + b_1 \cdot DoD + b_2 \cdot DoD^2 + b_3 DoD^4 + b_4 DoD^4 + b_5 DoD^5 \qquad \textbf{(1–4)}$$

where the coefficients a_i and b_i $(i = 0, \ldots 5)$ are constants; and the local value of the depth of discharge is computed as follows,

$$DoD = \frac{A}{Q_t}\int_0^t j dt \quad \textbf{(1–5)}$$

1.3. Thermal and Electrical Coupling

While solution of *Equation 1–1* (p. 2) to *Equation 1–5* (p. 3) provides potential and the current density distributions in anode and cathode, the coupling between thermal and the electrical fields considers the following,

1. Temperature dependent electric conductivity $\sigma = \sigma\,(T)$
2. Temperature dependent apparent current density $(1/\Omega m^2)$ by modifying *Equation 1–4* (p. 2),

$$Y = \left(\sum_{n=0}^{5} b_n\,(DoD)^n\right) e^{C_1\,(\frac{1}{T_{ref}} - \frac{1}{T})} \quad \textbf{(1–6)}$$

3. Temperature dependent equilibrium voltage (V) by modifying *Equation 1–3* (p. 2)

$$U = \left(\sum_{n=0}^{5} a_n\,(DoD)^n\right) - C_2\,(T - T_{ref}) \quad \textbf{(1–7)}$$

4. Joule heating as a volumetric source term for thermal energy equation

$$S_{joule} = \frac{i^2}{\sigma} = \frac{\sigma^2 \nabla^2 \phi}{\sigma} = \sigma \nabla^2 \phi \quad \textbf{(1–8)}$$

5. Reaction heating as a volumetric source term for thermal energy equation,

$$S_{echem} = j \cdot \left[U - (\phi_c - \phi_a)\right] \frac{A}{Vol} \quad \textbf{(1–9)}$$

where Vol is the volume of the computing cell. Since the potential jump takes place only at the separator interface, the heat generated, *Equation 1–9* (p. 3), is split equally between the two computing cells on either side of the interface. C_1 and C_2 are two additional model coefficients.

Chapter 2: Using the Battery Model

The ANSYS FLUENT Battery Model is provided as an addon module with the standard ANSYS FLUENT licensed software. The procedure for setting up and solving battery modeling is described in detail in this chapter. Only the steps related to battery modeling are shown here. Refer to *Battery Model Theory* (p. 1) for information about the theory.

For information about inputs related to other models used in conjunction with the battery model, see the appropriate sections for those models in the ANSYS FLUENT User's Guide.

2.1. Geometry Definition for the Battery Model

Due to the fact that there are a number of different physical zones associated with the battery, the following regions must be present in the battery mesh:

- Anode
- Cathode
- Separator ('zero' thickness wall/wall-shadow interface)

Note

For electro-chemical types of simulation, 3D double-precision is recommended.

2.2. Installing the Battery Module

The battery addon module is installed with the standard installation of ANSYS FLUENT in a directory called `addons/battery` in your installation area. The battery module consists of a UDF library and a pre-compiled scheme library, which need to be loaded and activated before calculations can be performed.

2.3. Loading the Battery Module

The battery module is loaded into ANSYS FLUENT through the text user interface (TUI). The module can be loaded only when a valid ANSYS FLUENT case file has been set or read. The text command to load the module is

`define` → `models` → `addon-module`

A list of ANSYS FLUENT addon modules is displayed:

```
  FLUENT Addon Modules:
  0. none
  1. MHD Model
  2. Fiber Model
  3. Fuel Cell and Electrolysis Model
  4. SOFC Model with Unresolved Electrolyte
  5. Population Balance Model
  6. Adjoint Solver
  7. Battery Model
 Enter Module Number: [0] 7
```

Select the battery model by entering the module number 7. During the loading process, a scheme library containing the graphical and text user interface, and a UDF library containing a set of user-defined functions (UDFs) for the battery module are loaded into ANSYS FLUENT. This is reported to the console. The UDF library also becomes visible as a new entry in the **UDF Library Manager** dialog box. The basic setup of the battery model is performed automatically when the battery module is successfully loaded.

2.4. Setting Up the Battery Module

The following describes an overview of the procedure required in order to use the Battery Model in ANSYS FLUENT.

1. Start ANSYS FLUENT.
2. Read the case file.
3. Scale the grid, if necessary.
4. Load the module and use the **Battery Model** dialog box to define the battery model parameters.
5. Define material properties.
6. Set the operating conditions.
7. Set the boundary conditions.
8. Start the calculations.
9. Save the case and data files.
10. Process your results.

Important

Note that the majority of this chapter describes how to set up the ANSYS FLUENT Battery Model using the graphical user interface. You can also perform various tasks using the text user interface. For more information, see *Using the Battery Model Text User Interface* (p. 13).

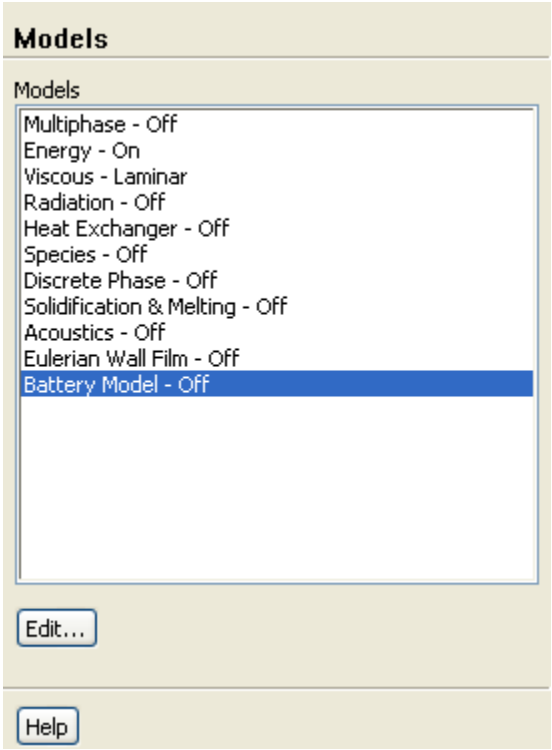
2.5. Getting Started With the Battery Model

The battery model is implemented by user-defined functions (UDFs) and scheme routines in ANSYS FLUENT. A number of UDFs are used to solve the battery equations. When you loaded the battery addon module in the previous step (*Loading the Battery Module* (p. 5)), UDF and scheme libraries that are required by the battery model were *automatically* loaded. Before you can begin the process of defining your battery model, however, you will need to perform some additional setup tasks that involve allocating user-defined memory for the UDFs and hooking an adjust UDF to ANSYS FLUENT. Follow the procedure below.

Once the module has been loaded, in order to set battery model parameters and assign properties to the relevant regions in your battery, you need to access the battery graphical user interface (the **Battery Model** dialog box).

To open the **Battery Model** dialog box, select **Models** under **Problem Setup** in the navigation pane to display the **Models** task page.

Figure 2.1 The Battery Model Option in the Models Task Page



In the **Models** task page, select the **Battery Model** option in the **Models** list and click the **Edit...** button.

Models → **Battery Model** → **Edit...**

*This opens the **Battery Model** dialog box.*

Once you open the **Battery Model** dialog box, you can select the **Enable Battery Model** check box to enable the model so that you can use it in your simulation. Enabling the model expands the dialog box to reveal additional model options and solution controls.

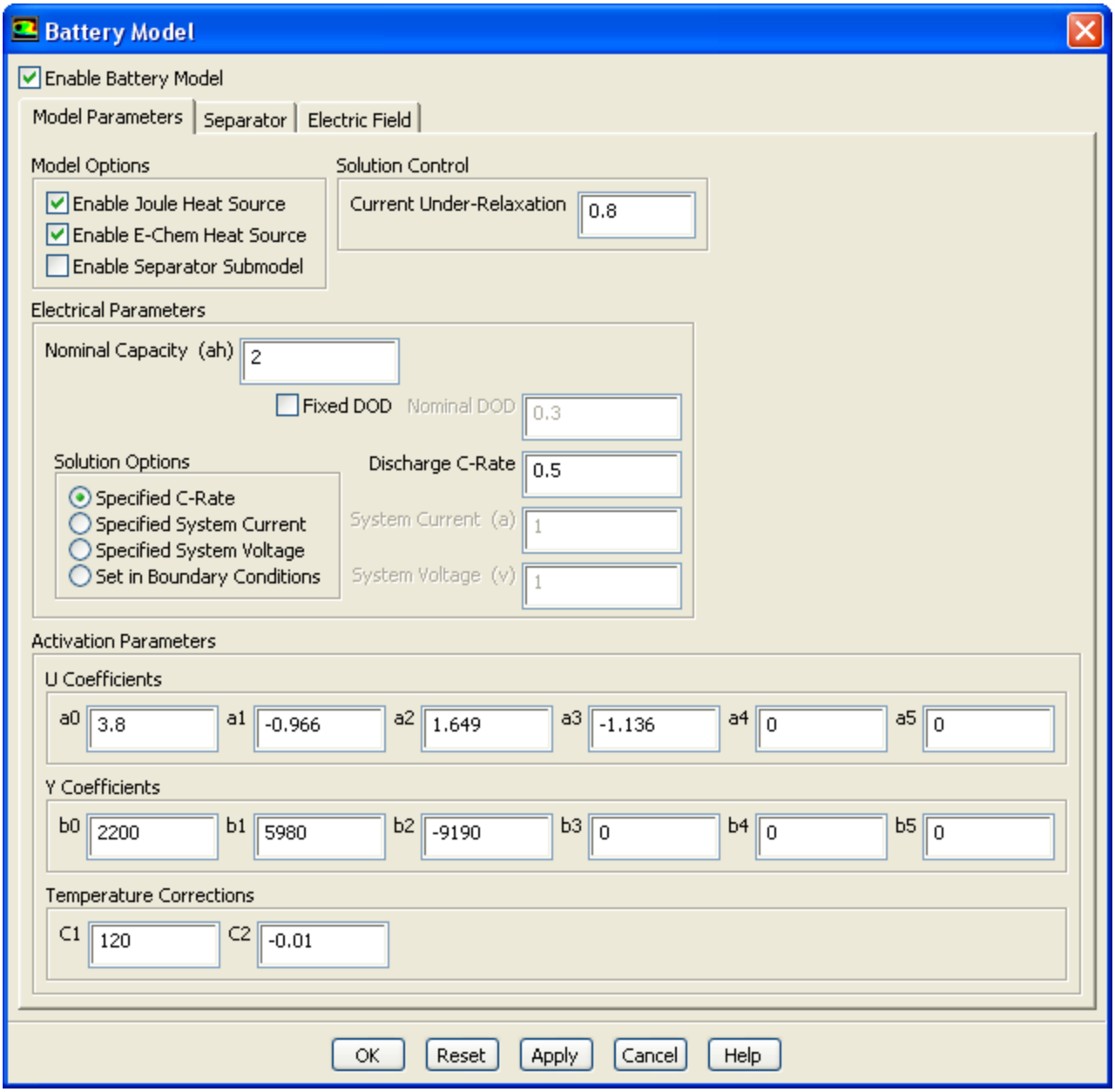
The **Model Parameters** tab of the **Battery Model** dialog box allows you to access general model settings when solving a battery problem. Likewise, the **Separator** tab allows you to set options for the battery separator. Finally, the **Electric Field** tab allows you to set parameters for the electric field.

2.5.3. Specifying Electric Field Parameters

2.5.1. Specifying Battery Model Parameters

The **Model Parameters** tab of the **Battery Model** dialog box allows you to turn on or off various options when solving a battery problem.

Figure 2.2 The Battery Model Dialog Box (Model Parameters Tab)



In the **Model Parameters** tab, you can set various model options, solution controls, electrical parameters, as well as activation parameters.

For **Model Options**, you can:

- **Enable Joule Heat Source** in order to include the Joule Heating source in the thermal energy equation (*Equation 1–8* (p. 3)). (enabled by default)

- **Enable E-Chem Heat Source** in order to include the heat source due to electrochemistry in the thermal energy equation (*Equation 1–9* (p. 3)). (enabled by default)
- **Enable Separator Submodel** in order to specify your own values for the **Separator Thickness** and **Resistivity**, instead of the **Y Coefficients**, for modeling the separator. Note that when this option is selected, the inputs for the **Y Coefficients** are grayed out, and the **Separator Property** fields are active in the **Separator** tab.

Note

The separator submodel is provided as an alternative method to calculate the separator resistance. When the option is enabled, the separator resistance is computed as *R = (separator resistivity)*(separator thickness)*, where *separator resistivity* and *separator thickness* are supplied by you. When the option is disabled (the default), the separator resistance is computed by $R = 1 / \mathrm{Y}$.

For **Solution Controls**, you can set the **Current Under- Relaxation Factor**.

For **Electrical Parameters**, you can set the **Nominal Capacity** (the capacity of the battery cell). If you select **Fixed DoD**, then you can specify a **Nominal DoD** value (depth of discharge). For the **Solution Options**, if you select:

- **Specified C-Rate**, you can set a value for the **Discharge C-Rate** (the hourly rate at which a battery is discharged). In this case, the total current at the cathode tabs are fixed as the product of C-Rate and Nominal Capacity, while the electrical potential is anchored at zero on the anode tabs.
- **Specified System Current**, you can set a value for the total current (applied to the anode tabs). In this case, the electrical potential is set to zero at the anode tabs.
- **Specified System Voltage**, you can set a value for the **System Voltage** (applied to the cathode tab; the anode tab has a voltage of 0 V).
- **Set in Boundary Conditions**, you can set the UDS boundary conditions directly, e.g., the voltage value or the current value (specified flux), using the **Boundary Conditions** task page in ANSYS FLUENT for the specific face zone.

Note

When the steady state solver is used, the **Fixed DoD** option has to be selected. Alternatively, the transient solver can be used to analyze variable DoD problems.

For **Activation Parameters**, you can specify the **U Coefficients** for *Equation 1–3* (p. 2) and the **Y Coefficients** for *Equation 1–4* (p. 2) (if the **Enable Separator Submodel** option is disabled).

Note

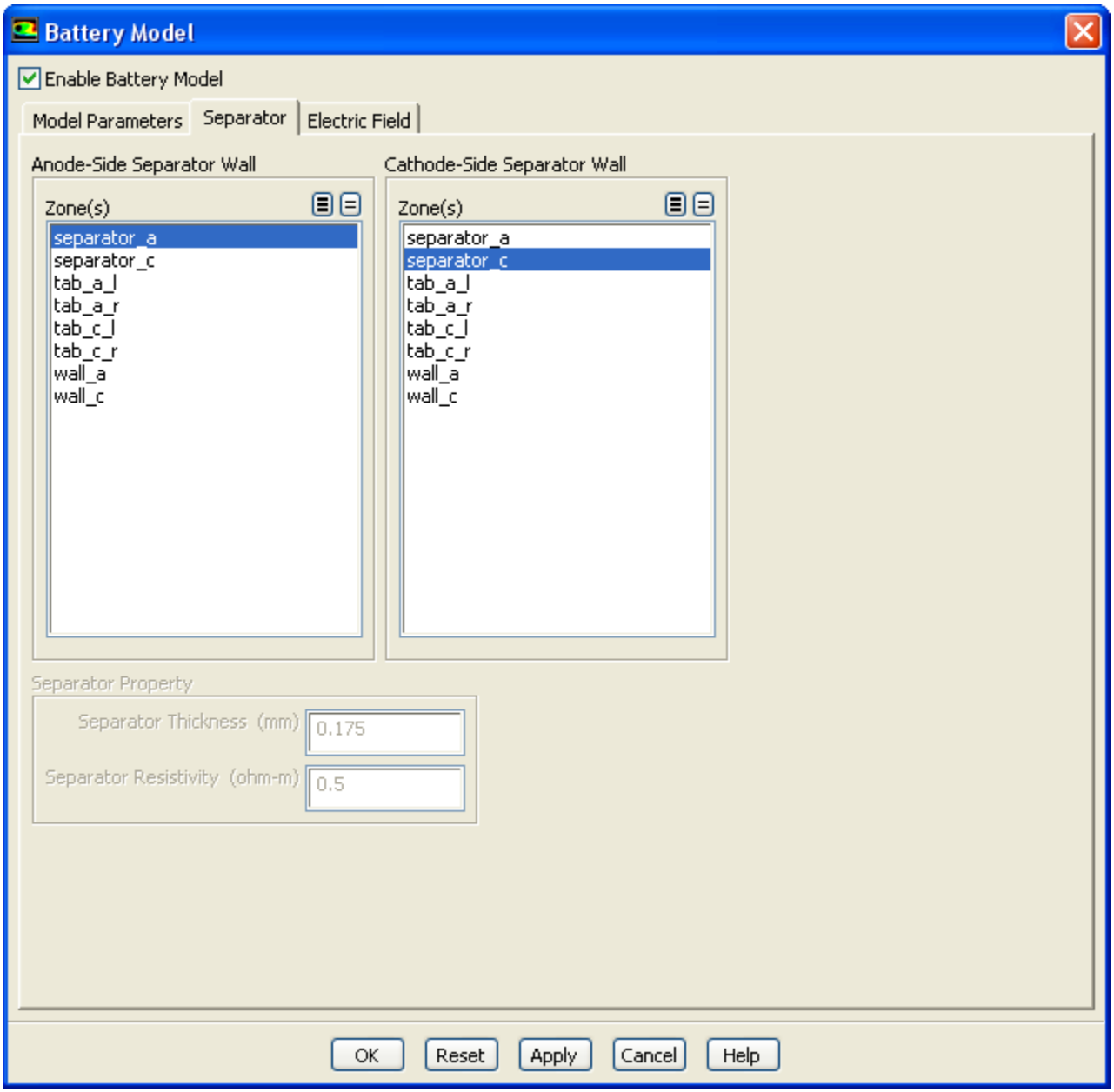
The coefficient values for the **Activation Parameters** are based on battery cell polarization test curves. Obtaining coefficient values (other than the default values) can be dependant on your battery configuration and material properties. For more information about coefficient values, refer to the work performed by Gu [*2*] (p. 15). You will likely need to make adjustments (e.g., if you are modeling lithium ion batteries) when using your own experimental data.

You can also specify the **Temperature Corrections**, if needed, though the default values are suitable in most cases. The temperature corrections provide additional accuracy to account for local temperature effects, and correspond to the temperature terms in *Equation 1–6* (p. 3) and *Equation 1–7* (p. 3).

2.5.2. Specifying Separator Parameters

The **Separator** tab of the **Battery Model** dialog box allows you to select interfaces as the **Anode Separator**, the **Cathode Separator**, as well as the **Separator Properties**, if appropriate.

Figure 2.3 The Battery Model Dialog Box (Separator Tab)

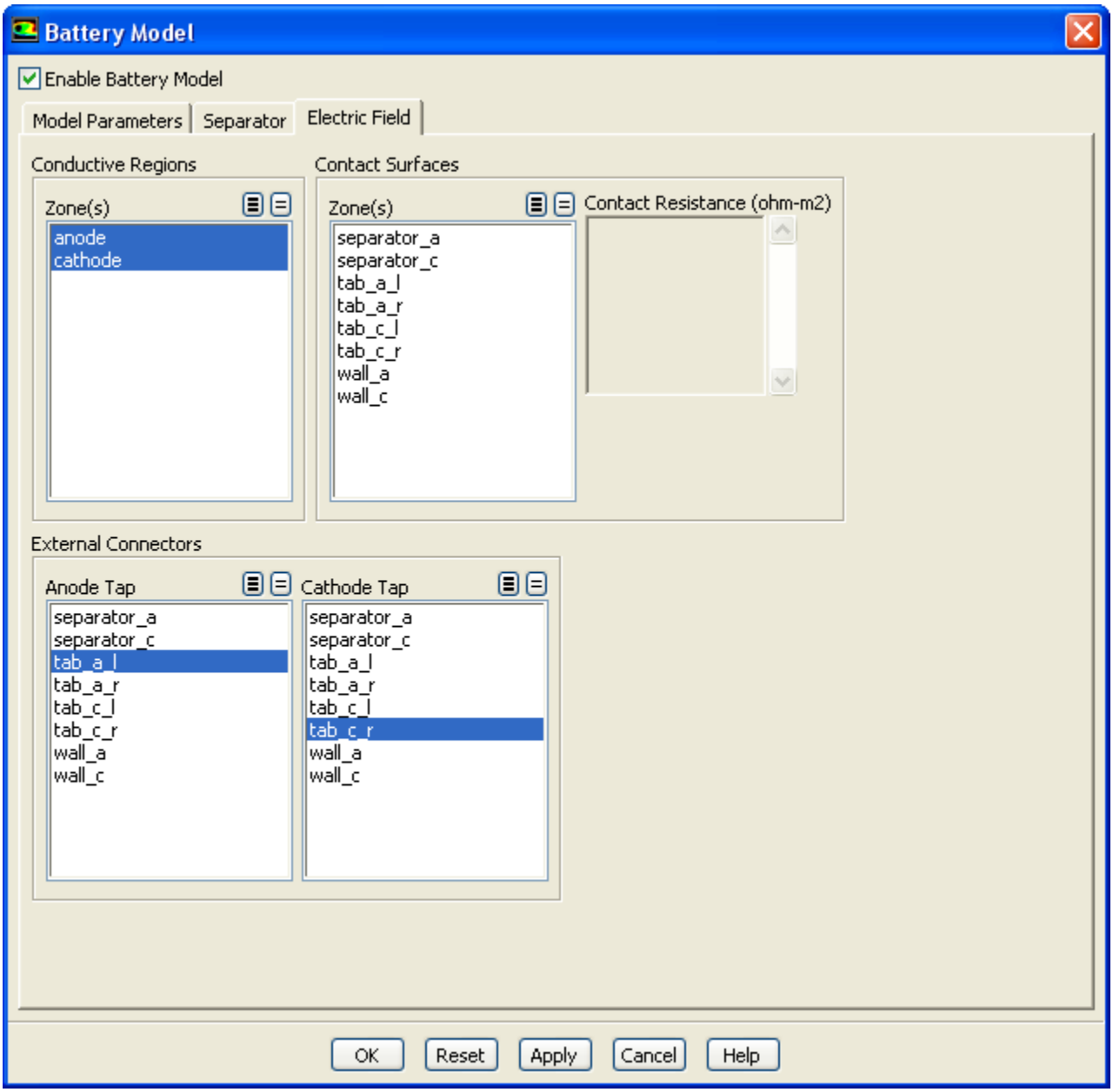


In the **Separator** tab, specify the zones for the **Anode Separator** and the **Cathode Separator**. If the **Enable Separator Submodel** option is selected in the **Model Parameters** tab, you can specify the **Separator Properties** such as the **Separator Thickness** and the **Separator Resistivity**.

2.5.3. Specifying Electric Field Parameters

The **Electric Field** tab of the **Battery Model** dialog box allows you to set the properties of the **Conductive Regions**, **Contact Surfaces**, and the **External Connectors**.

Figure 2.4 The Battery Model Dialog Box (Electric Field Tab)



In the **Electric Field** tab, specify the zones for the **Conductive Regions** and the **Contact Surfaces** (as well as the **Contact Resistance**) for any selected contact surface. In addition, you can specify the anode and cathode tap surfaces for the External Connectors of the battery.

2.6. Solution Controls for the Battery Model

When you use the Battery model, the electric potential equation is solved in addition to other fluid dynamic equations, depending on the type of simulation. The electric potential equation is listed as

one of the solved equations by ANSYS FLUENT in the **Equation** dialog box, where it can be enabled/disabled in the solution process.

Solution Controls → Equations...

Also, keep in mind the **Advanced Solution Controls** dialog box, where you can set the multigrid cycle type for the electric potential equation, if required.

Solution Controls → Advanced...

Note

When choosing a solution method for your simulation, the Least Squares Cell Based gradient spatial discretization method is recommended because of its greater accuracy. The Green-Gauss Cell Based gradient spatial discretization method is adequate if the mesh is evenly spaced in the system current direction, and if there are not large differences in the electrical conductivities in the materials used in the simulation.

Note

In transient simulations, it is recommended to start the calculation with a smaller time step (1 ~ 2 seconds) initially. The time step can be increased to a large value (e.g., 30 seconds), however, you will likely need to search for a suitable value to make sure reasonable convergence is achieved within each time step.

2.7. Postprocessing the Battery Model

You can perform post-processing using standard ANSYS FLUENT quantities and by using user-defined scalars and user-defined memory allocations. When using the Battery model, the following additional variables will be available for postprocessing:

- Under **User-Defined Scalars...**
  - **Electric Potential**
  - **Diff Coef of Electric Potential** (the electrical conductivity of the conductive field)
- Under **User-Defined Memory...**
  - **Interface Current Density** (the separator current density, i.e. J (A/m2))
  - **X Current Density**
  - **Y Current Density**
  - **Z Current Density**
  - **Magnitude of Current Density**
  - **Volumetric Ohmic Source** (the energy source due to the electric Joule heating)
  - **Electrochemistry Source**
  - **Activation Over-Potential** (the net electrode potential change across the anode and cathode of the system when there is a current flowing through the system, i.e., $\phi_c - \phi_a - U$ (Volts)
  - **Depth of Discharge**

- **U Function**
- **Y Function**
- **Separator Voltage Jump** (the net potential difference across the separator)
- **Effective Electric Resistance** (the effective resistance of the separator used in the potential field calculation)
- **Other**

By default, the ANSYS FLUENT Battery Model defines several user-defined scalars and user-defined memory allocations, described in *Table 2.1: User-Defined Scalar Allocations* (p. 13) and *Table 2.2: User-Defined Memory Allocations* (p. 13).

Table 2.1 User-Defined Scalar Allocations

UDS 0	Electric Potential (Volts)
UDS 1	Diff Coef of Electric Potential

Table 2.2 User-Defined Memory Allocations

UDM 0	Interface Current Density
UDM 1	X Current Density
UDM 2	Y Current Density
UDM 3	Z Current Density
UDM 4	Magnitude of Current Density
UDM 5	Volumetric Ohmic Source
UDM 6	Electrochemistry Source
UDM 7	Activation Over-Potential
UDM 8	Depth of Discharge
UDM 9	U Function
UDM 10	Y Function
UDM 11	Separator Voltage Jump
UDM 12	Effective Electric Resistance
UDM 13	Other

2.8. Using the Battery Model Text User Interface

All of the features for the Battery Model that are available through the graphical user interface are also available through text user interface (TUI) commands. The TUI allows text commands to be typed directly in the ANSYS FLUENT console window where additional information can be extracted and processed for more advanced analysis.

Once the battery module is loaded, you can access the text user interface through the Console Window under `battery-model`. A listing of the various text commands is as follows:

`battery-model/`
Battery model menu

activation-parameters/
Activation parameter setup.

t-coefficients
Specify the temperature coefficients in *Equation 1–6* (p. 3) and *Equation 1–7* (p. 3).

u-coefficients
Specify the U coefficients for *Equation 1–3* (p. 2) .

y-coefficients
Specify the Y coefficients for *Equation 1–4* (p. 2).

anode-interface
Anode interface options.

cathode-interface
Cathode interface options.

electric-field-model/
Electric field setup.

conductive-regions
List zone names and IDs.

contact-resistance-regions
List zone names and IDs.

current-tap
List zone names and IDs.

voltage-tap
List zone names and IDs.

electrochemistry
Electrochemistry parameters.

enable-battery-model?
Enable/disable battery model.

model-parameters
Model parameters.

Bibliography

[1] Tiedemann and Newman, . *"Battery Design and Optimization"*. *Electrochemical Soc. Proceedings*. Princeton, NJ. 79-1. 39. 1979.

[2] H. Gu, . *"Mathematical Analysis of a Zn/NiOOH Cell"*. *J. Electrochemical Soc.*. Princeton, NJ. 1459-1. 464. July 1983.

[3] U. S. Kim, C. B. Shin, and C. –S. Kim, . *"Effect of Electrode Configuration on the Thermal Behavior of a Lithium-Polymer Battery"*. *Journal of Power Sources*. Princeton, NJ. 180. 909–916. 2008.

Index