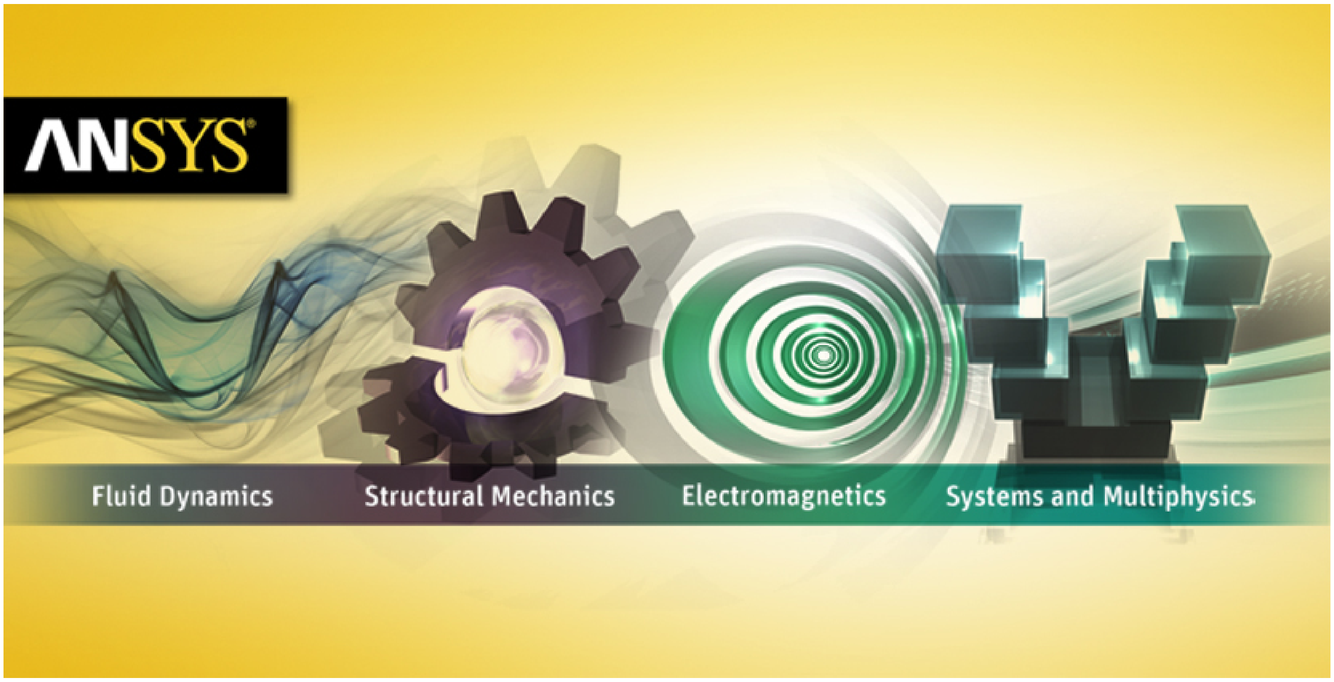


ANSYS FLUENT Magnetohydrodynamics (MHD) Module Manual

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

ANSYS, Inc. is certified to ISO 9001:2008.

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 Magnetohydrodynamics (MHD) Module Manual tells you what you need to know to model magnetohydrodynamics with ANSYS FLUENT. In this manual, you will find background information pertaining to the model, a theoretical discussion of the model used in ANSYS FLUENT, and a description of using the model for 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

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

∇ appears in several ways:

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

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

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

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

  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,

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \qquad \textbf{(5)}$$

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

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \qquad \textbf{(6)}$$

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

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

- 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: Introduction

The Magnetohydrodynamics (MHD) module is provided as an add-on module with the standard ANSYS FLUENT licensed software.

Magnetohydrodynamics refers to the interaction between an applied electromagnetic field and a flowing, electrically-conductive fluid. The ANSYS FLUENT MHD model enables you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields. The externally-imposed magnetic field may be generated either by selecting simple built-in functions or by importing a user-supplied data file. For multiphase flows, the MHD model is compatible with both the discrete phase model (DPM), the volume-of-fluid (VOF) and Eulerian mixture approaches in ANSYS FLUENT, including the effects of a discrete phase on the electrical conductivity of the mixture.

This document describes the ANSYS FLUENT MHD model. *Magnetohydrodynamic Model Theory* (p. 3) provides theoretical background information. *Implementation* (p. 9) summarizes the UDF-based software implementation. Instructions for getting started with the model are provided in *Using the ANSYS FLUENT MHD Module* (p. 11). *Appendix A* (p. 23) provides a condensed overview on how to use the MHD model, while *Appendix B* (p. 27) contains definitions for the magnetic field, *Appendix C* (p. 29) describes the external magnetic field data format, and *Appendix D* (p. 31) lists the text commands in the MHD model.

Chapter 2: Magnetohydrodynamic Model Theory

This chapter presents an overview of the theory and the governing equations for the mathematical models used in ANSYS FLUENT to predict flow in an electromagnetic field.

2.1. Introduction

The coupling between the fluid flow field and the magnetic field can be understood on the basis of two fundamental effects: the induction of electric current due to the movement of conducting material in a magnetic field, and the effect of Lorentz force as the result of electric current and magnetic field interaction. In general, the induced electric current and the Lorentz force tend to oppose the mechanisms that create them. Movements that lead to electromagnetic induction are therefore systematically braked by the resulting Lorentz force. Electric induction can also occur in the presence of a time-varying magnetic field. The effect is the stirring of fluid movement by the Lorentz force.

Electromagnetic fields are described by Maxwell's equations:

$$\nabla \cdot \vec{B} = 0 \tag{2–1}$$

$$\nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t} \tag{2–2}$$

$$\nabla \cdot \vec{D} = q \tag{2–3}$$

$$\nabla \times \vec{H} = \vec{j} + \frac{\partial \vec{j}}{\partial t} \tag{2–4}$$

where $\vec{B}$ (Tesla) and $\vec{E}$ (V/m) are the magnetic and electric fields, respectively, and $\vec{H}$ and $\vec{D}$ are the induction fields for the magnetic and electric fields, respectively. q $(\mathrm{C} / \mathrm{m}^3)$ is the electric charge density, and $\vec{j}$ $(\mathrm{A} / \mathrm{m}^2)$ is the electric current density vector.

The induction fields $\vec{H}$ and $\vec{D}$ are defined as:

$$\vec{H} = \frac{1}{\mu}\vec{B} \tag{2–5}$$

$$\vec{D} = \varepsilon\vec{E} \tag{2–6}$$

where μ and ε are the magnetic permeability and the electric permittivity, respectively. For sufficiently conducting media such as liquid metals, the electric charge density q and the displacement current $\frac{\partial \vec{D}}{\partial t}$ are customarily neglected [1] (p. 33).

In studying the interaction between flow field and electromagnetic field, it is critical to know the current density $\vec{j}$ due to induction. Generally, two approaches may be used to evaluate the current density. One is through the solution of a magnetic induction equation; the other is through solving an electric potential equation.

2.2. Magnetic Induction Method

In the first approach, the magnetic induction equation is derived from Ohm's law and Maxwell's equation. The equation provides the coupling between the flow field and the magnetic field.

In general, Ohm's law that defines the current density is given by:

$$\vec{j} = \sigma\vec{E} \tag{2–7}$$

where σ is the electrical conductivity of the media. For fluid velocity field $\vec{U}$ in a magnetic field $\vec{B}$, Ohm's law takes the form:

$$\vec{j} = \sigma\left(\vec{E} + \vec{U} \times \vec{B}\right) \tag{2–8}$$

From Ohm's law and Maxwell's equation, the induction equation can be derived as:

$$\frac{\partial \vec{B}}{\partial t} + \left(\vec{U} \cdot \nabla\right)\vec{B} = \frac{1}{\mu\sigma}\nabla^2\vec{B} + \left(\vec{B} \cdot \nabla\right)\vec{U} \tag{2–9}$$

From the solved magnetic field $\vec{B}$, the current density $\vec{j}$ can be calculated using Ampere's relation as:

$$\vec{j} = \frac{1}{\mu}\nabla \times \vec{B} \tag{2–10}$$

Generally, the magnetic field $\vec{B}$ in a MHD problem can be decomposed into the externally imposed field $\vec{B_0}$ and the induced field $\vec{b}$ due to fluid motion. Only the induced field $\vec{b}$ must be solved.

From Maxwell's equations, the imposed field $\vec{B_0}$ satisfies the following equation:

$$\nabla^2 \vec{B_0} - \mu\sigma' \frac{\partial \vec{B_0}}{\partial t} = 0 \quad (2–11)$$

where σ' is the electrical conductivity of the media in which field $\vec{B_0}$ is generated. Two cases need to be considered.

For more information, please see the following sections:

2.2.1. Case 1: Externally Imposed Magnetic Field Generated in Non-conducting Media

In this case the imposed field $\vec{B_0}$ satisfies the following conditions:

$$\nabla \times \vec{B_0} = 0 \quad (2–12)$$

$$\nabla^2 \vec{B_0} = 0 \quad (2–13)$$

With $\vec{B} = \vec{B_0} + \vec{b}$, the induction equation (*Equation 2–9* (p. 4)) can be written as:

$$\frac{\partial \vec{b}}{\partial t} + (\vec{U} \cdot \nabla)\vec{b} = \frac{1}{\mu\sigma}\nabla^2 \vec{b} + ((\vec{B_0} + \vec{b}) \cdot \nabla)\vec{U} - (\vec{U} \cdot \nabla)\vec{B_0} - \frac{\partial \vec{B_0}}{\partial t} \quad (2–14)$$

The current density is given by:

$$\vec{j} = \frac{1}{\mu}\nabla \times \vec{b} \quad (2–15)$$

2.2.2. Case 2: Externally Imposed Magnetic Field Generated in Conducting Media

In this case the conditions given in *Equation 2–12* (p. 5) and *Equation 2–13* (p. 5) are not true. Assuming that the electrical conductivity of the media in which field $\vec{B_0}$ is generated is the same as that of the flow, that is $\sigma' = \sigma$, from *Equation 2–9* (p. 4) and *Equation 2–11* (p. 5) the induction equation can be written as:

$$\frac{\partial \vec{b}}{\partial t} + (\vec{U} \cdot \nabla)\vec{b} = \frac{1}{\mu\sigma}\nabla^2 \vec{b} + ((\vec{B_0} + \vec{b}) \cdot \nabla)\vec{U} - (\vec{U} \cdot \nabla)\vec{B_0} \quad (2–16)$$

and the current density is given by:

$$\vec{j} = \frac{1}{\mu}\nabla \times (\vec{B_0} + \vec{b}) \tag{2–17}$$

For the induction equation Equation 2–14 (p. 5) or Equation 2–16 (p. 5), the boundary conditions for the induced field are given by:

$$\vec{b} = \{b_n \quad b_{t1} \quad b_{t2}\}^T = \vec{b^*} \tag{2–18}$$

where the subscripts denote the normal and tangential components of the field and $\vec{b^*}$ is specified by you. For an electrically insulating boundary, as $j_n = 0$ at the boundary, from Ampere's relation one has $b_{t1} = b_{t2} = 0$ at the boundary.

2.3. Electric Potential Method

The second approach for the current density is to solve the electric potential equation and calculate the current density using Ohm's law. In general, the electric field $\vec{E}$ can be expressed as:

$$\vec{E} = -\nabla\varphi - \frac{\partial \vec{A}}{\partial t} \tag{2–19}$$

where φ and $\vec{A}$ are the scalar potential and the vector potential, respectively. For a static field and assuming $\vec{b} << \vec{B_0}$, Ohm's law given in Equation 2–8 (p. 4) can be written as:

$$\vec{j} = \sigma(-\nabla\varphi + (\vec{U} \times \vec{B_0})) \tag{2–20}$$

For sufficiently conducting media, the principle of conservation of electric charge gives:

$$\nabla \cdot \vec{j} = 0 \tag{2–21}$$

The electric potential equation is therefore given by:

$$\nabla^2\varphi = \nabla \cdot (\vec{U} \times \vec{B_0}) \tag{2–22}$$

The boundary condition for the electric potential φ is given by:

$$\frac{\partial\varphi}{\partial n} = (\vec{U} \times \vec{B_0})_{boundary} \cdot \vec{n} \tag{2–23}$$

for an insulating boundary, where $\vec{n}$ is the unit vector normal to the boundary, and

$$\varphi = \varphi_0 \tag{2–24}$$

for a conducting boundary, where φ_0 is the specified potential at the boundary. The current density can then be calculated from *Equation 2–20* (p. 6).

With the knowledge of the induced electric current, the MHD coupling is achieved by introducing additional source terms to the fluid momentum equation and energy equation. For the fluid momentum equation, the additional source term is the Lorentz force given by:

$$\vec{F} = \vec{j} \times \vec{B} \tag{2–25}$$

which has units of N/m $\mathrm{N} / \mathrm{m}^3$ in the SI system. For the energy equation, the additional source term is the Joule heating rate given by:

$$Q = \frac{1}{\sigma} \vec{j} \cdot \vec{j} \tag{2–26}$$

which has units of $\mathrm{W} / \mathrm{m}^3$.

For charged particles in an electromagnetic field, the Lorentz force acting on the particle is given by:

$$\vec{F}_p = q \left(\vec{E} + \vec{v}_p \times \vec{B} \right) \tag{2–27}$$

where q is the particle charge density ($\mathrm{Coulomb} / \mathrm{m}^3$) and v_p is the particle velocity. The force $\vec{F}_p$ has units of $\mathrm{N} / \mathrm{m}^3$.

For multiphase flows, assuming that the electric surface current at the interface between phases can be ignored, the electric conductivity for the mixture is given by:

$$\sigma_m = \sum_i \sigma_i v_i \tag{2–28}$$

where σ_i and v_i are respectively the electric conductivity and volume fraction of phase $i. \sigma_m$ is used in solving the induction equations.

Chapter 3: Implementation

The MHD model is implemented using user-defined functions (UDF) as an ANSYS FLUENT add-on module, which is loaded into ANSYS FLUENT at run-time. The model is accessed through a number of UDF schemes. The magnetic induction equation given by *Equation 2–14* (p. 5) or *Equation 2–16* (p. 5) and the electric potential equation given by *Equation 2–22* (p. 6) are solved through user-defined scalar (UDS) transport equations. Other model-related variables such as the external magnetic field data, current density, Lorentz force and Joule heat are stored as user-defined memory (UDM) variables. The MHD model setup and parameters are input using the **MHD Model** graphical user interface (GUI) dialog box and a set of text user interface (TUI) commands described in *Using the ANSYS FLUENT MHD Module* (p. 11). Detailed information can be found in the following sections:

3.1. Solving Magnetic Induction and Electric Potential Equations

The magnetic induction equation and the electric potential equations are solved through user-defined scalar transport equations. For the magnetic induction equation a set of 2 or 3 scalar equations are solved, each representing a Cartesian component of the induced magnetic field vector in a 2D or 3D case. For the electric potential equation a single scalar equation is solved.

The convection and the diffusion terms of the scalar equations are defined using user functions `DEFINE_UDS_FLUX(mhd_flux, ..., ns)` and `DEFINE_DIFFUSIVITY (mhd_ magnetic_diffusivity, ..., ns)` respectively. The user-defined scalar equation is identified by the scalar index `ns`.

The source terms to the induction equations and the potential equation are implemented using user function `DEFINE_SOURCE(mhd_mag_source, ..., eqn)` and `DEFINE_SOURCE (mhd_phi_source, ..., eqn)` respectively, where `eqn` identifies the scalar equations.

For transient cases, the additional unsteady source term is introduced through the user function `DEFINE_UDS_UNSTEADY(mhd_unsteady_source, ..., ns)`, where `ns` identifies the scalar being solved.

The induction and potential equations can also be solved in solid zones, in which case the fluid velocity terms in the equations are not considered. For multiphase flows, the MHD equations are solved in the mixture domain only.

The wall boundary conditions are implemented through user profile functions (`DEFINE_PROFILE(mhd_bc_...`), and are applied to the Cartesian components of the induced magnetic field vector or to the electric potential. For external wall boundaries, three types of boundary conditions (electrically insulating, conducting, and "thin wall"), can be applied. The 'thin wall' type boundary refers to an external wall where a 1D magnetic or electric potential diffusion normal to the boundary is assumed, and the wall material and the thickness are specified for the boundary. For internal wall boundaries, that is the boundaries between fluid/solid or solid/solid zones, a coupled boundary condition is applied.

3.2. Calculation of MHD Variables

Apart from the Cartesian components of the magnetic field vectors and the electric potential function, which are stored as user-defined scalars, other MHD-related variables include the induced electric current density vector, induced electric field vector, the Lorentz force vector and Joule heat. These variables are stored in user-defined memory locations. Updating of MHD variables is accessed through the user function `DEFINE_ADJUST(mhd_adjust,...)`. The variables are updated at the start of each iteration using the solved induced magnetic field from the previous iteration.

3.3. MHD Interaction with Fluid Flows

Additional source terms due to the magnetic induction are added to the flow momentum and energy equations as user defined source terms. For the momentum equation, user function `DEFINE_SOURCE(mhd_mom_source, ..., eqn)` is used to introduce the Lorentz force to the equation, where `eqn` identifies the Cartesian component of the fluid momentum. For the energy equation, the additional source due to Joule heating is added through user function `DEFINE_SOURCE(mhd_energy_source,..., eqn)`, where `eqn` is the energy equation index.

3.4. MHD Interaction with Discrete Phase Model

In discrete phase modelling, the Lorentz force acting on charged particles is introduced through the user function `DEFINE_DPM_BODY_FORCE(mhd_dpm_force, ...)`. User function `DEFINE_DPM_SOURCE(mhd_dpm_source, ...)` is used to update the volume fraction of the discrete phase inside a fluid cell and the volume-weighted electric conductivity of the discrete phase.

3.5. General User-Defined Functions

Several general UDFs are used as part of the MHD model implementation.

- `DEFINE_INIT(mhd_init,...)` is an initialization function called during the general case initialization to set up the external magnetic field and initialize MHD model parameters and variables.
- `DEFINE_ADJUST(mhd_adjust,...)` is called at the start of each iteration. It is used to adjust the magnetic boundary conditions and update MHD related variables and properties.

Chapter 4: Using the ANSYS FLUENT MHD Module

This chapter provides basic instructions to install the magnetohydrodynamics (MHD) module and solve MHD problems in ANSYS FLUENT. It assumes that you are already familiar with standard ANSYS FLUENT features, including the user-defined function procedures described in the ANSYS FLUENT UDF Manual. *Appendix A* (p. 23) also outlines the general procedure for using the MHD model. This chapter describes the following:

4.1. MHD Module Installation

The MHD module is provided as an add-on module with the standard ANSYS FLUENT licensed software. A special license is required to use the MHD module. The module is installed with the standard installation of ANSYS FLUENT in a directory called `addons/mhd` in your installation area. The MHD module consists of a UDF library and a pre-compiled scheme library, which needs to be loaded and activated before calculations can be performed.

4.2. Loading the MHD Module

The MHD module is loaded into ANSYS FLUENT through the text user interface (TUI). The module can only be loaded 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 add-on modules is displayed:

```
> /define/models/addon-module
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 Module
Enter Module Number: [0] 1
```

Select the MHD model by entering the module number 1. 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 are loaded into ANSYS FLUENT. A message **Addon Module: mhd...loaded!** is displayed at the end of the loading process.

The basic setup of the MHD model is performed automatically when the MHD module is loaded successfully. The setup includes:

- Selecting the default MHD method
- Allocating the required number of user-defined scalars and memory locations naming of:
  - User-defined scalars and memory locations
  - All UDF Hooks for MHD initialization and adjustment
  - MHD equation flux and unsteady terms
  - Source terms for the MHD equations
  - Additional source terms for the fluid momentum and energy equations
  - Default MHD boundary conditions for external and internal boundaries
  - A default set of model parameters
- DPM related functions are also set, if the DPM option has been selected in the ANSYS FLUENT case setup.

The MHD module setup is saved with the ANSYS FLUENT case file. The module is loaded automatically when the case file is subsequently read into ANSYS FLUENT. Note that in the saved case file, the MHD module is saved with the absolute path. Therefore, if the locations of the MHD module installation or the saved case file are changed, ANSYS FLUENT will not be able to load the module when the case file is subsequently read.

To unload the previously saved MHD module library, use the

Define → **User-Defined** → **Functions** → **Manage...**

menu and reload the module as described above. Note that the previously saved MHD model setup and parameters are preserved.

4.3. MHD Model Setup

Following the loading of the MHD module, you can access the **MHD Model** dialog box using

Models → **MHD Model** → **Edit...**

or using the text command

`define` → `models` → `mhd-model`

Both the **MHD Model** dialog box and TUI commands are designed for the following tasks:

- Enable/disable the MHD model.
- Select the MHD method.
- Apply an external magnetic field.
- Set boundary conditions.
- Set solution control parameters.

Operations of these tasks through the **MHD Model** dialog box are described in the following sections. The set of MHD text commands are listed in *Appendix D* (p. 31).

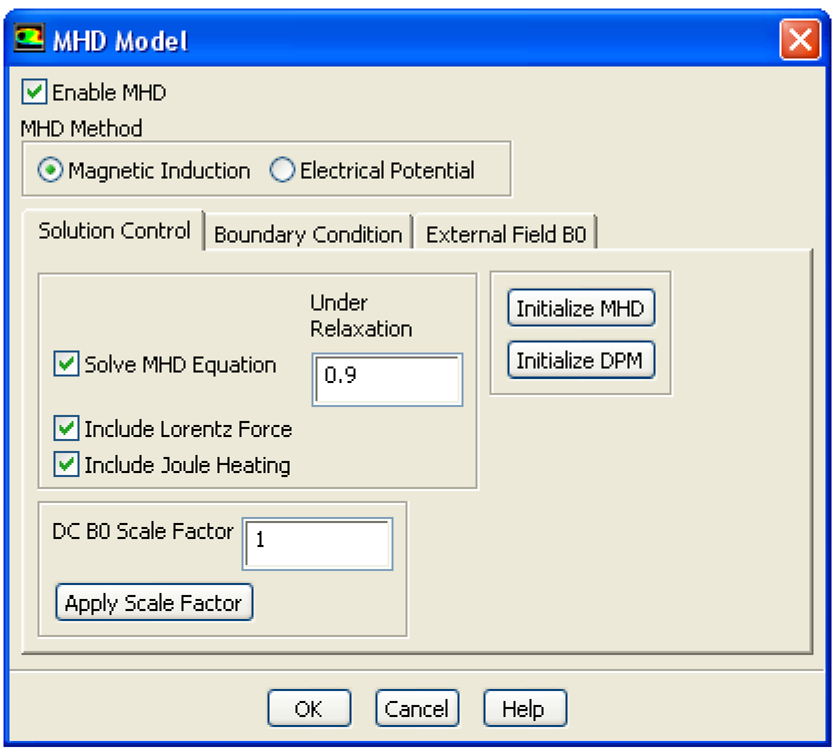
For more information, please see the following sections:

4.3.1. Enabling the MHD Model

If the MHD model is not enabled after the MHD module is loaded for the first time, you can enable it by selecting **Enable MHD** in the **MHD Model** dialog box, shown in *Figure 4.1* (p. 13). The dialog box expands to its full size when the model is enabled, as shown in *Figure 4.2* (p. 13).

Figure 4.1 Enabling the MHD Model Dialog Box

Figure 4.2 The MHD Model Dialog Box



4.3.2. Selecting an MHD Method

The method used for MHD calculation can be selected under **MHD Method** in the **MHD Model** dialog box. The two methods, **Magnetic Induction** and **Electrical Potential**, are described in *Magnetic Induction Method* (p. 4) and *Electric Potential Method* (p. 6),

For the Magnetic Induction method, 2 or 3 user-defined scalars are allocated for the solution of the induced magnetic field in 2-D or 3-D cases. The scalars are listed as B_x, B_y and B_z representing the Cartesian components of the induced magnetic field vector. The unit for the scalar is Tesla.

For the Electrical Potential method, 1 user-defined scalar is solved for the electric potential field. The scalar is listed as φ and has the unit of Volt.

Table 4.1: User-Defined Scalars in MHD Model (p. 14) lists the user-defined scalars used by the two methods.

Table 4.1 User-Defined Scalars in MHD Model

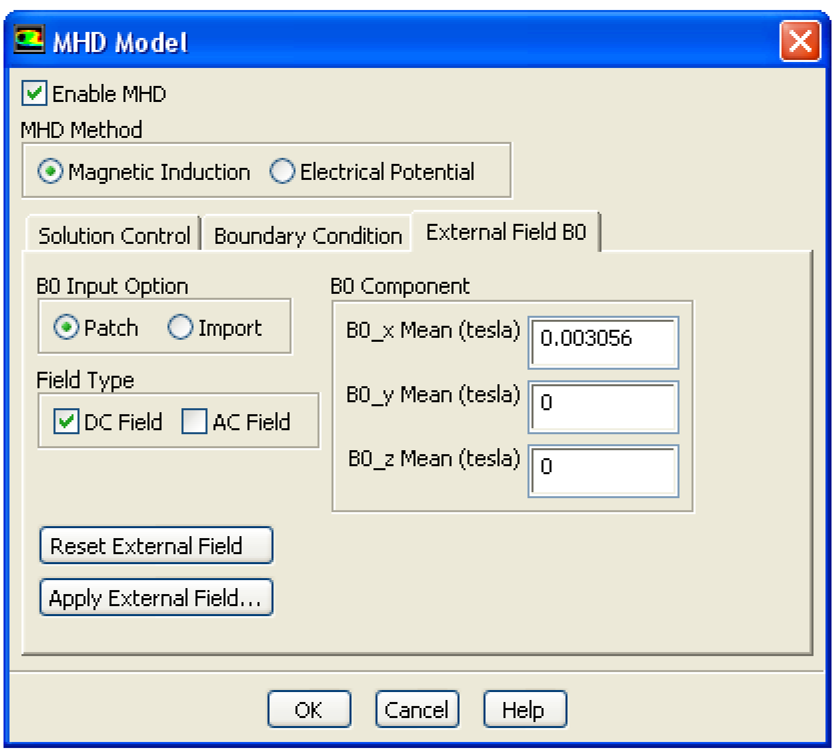
Method	Scalar	Name	Unit	Description
Induction	Scalar-0	B_x	Tesla	X component of induced magnetic field (b_x)
	Scalar-1	B_y	Tesla	Y component of induced magnetic field (b_y)
	Scalar-2 (3-D)	B_z	Tesla	Z component of induced magnetic field (b_z)
Potential	Scalar-0	Phi	Volt	Electric potential (φ)

4.3.3. Applying an External Magnetic Field

Application of an external magnetic field to the computation domain is done under the **External Field B0** tab in the **MHD Model** dialog box, as shown in *Figure 4.3* (p. 14). Two **B0 Input Option**s are available for setting up the external magnetic field. One option is to **Patch** the computational domain with a constant (**DC Field**) and/or varying (**AC Field**) type. The other option is to **Import** the field data from a magnetic data file that you provide.

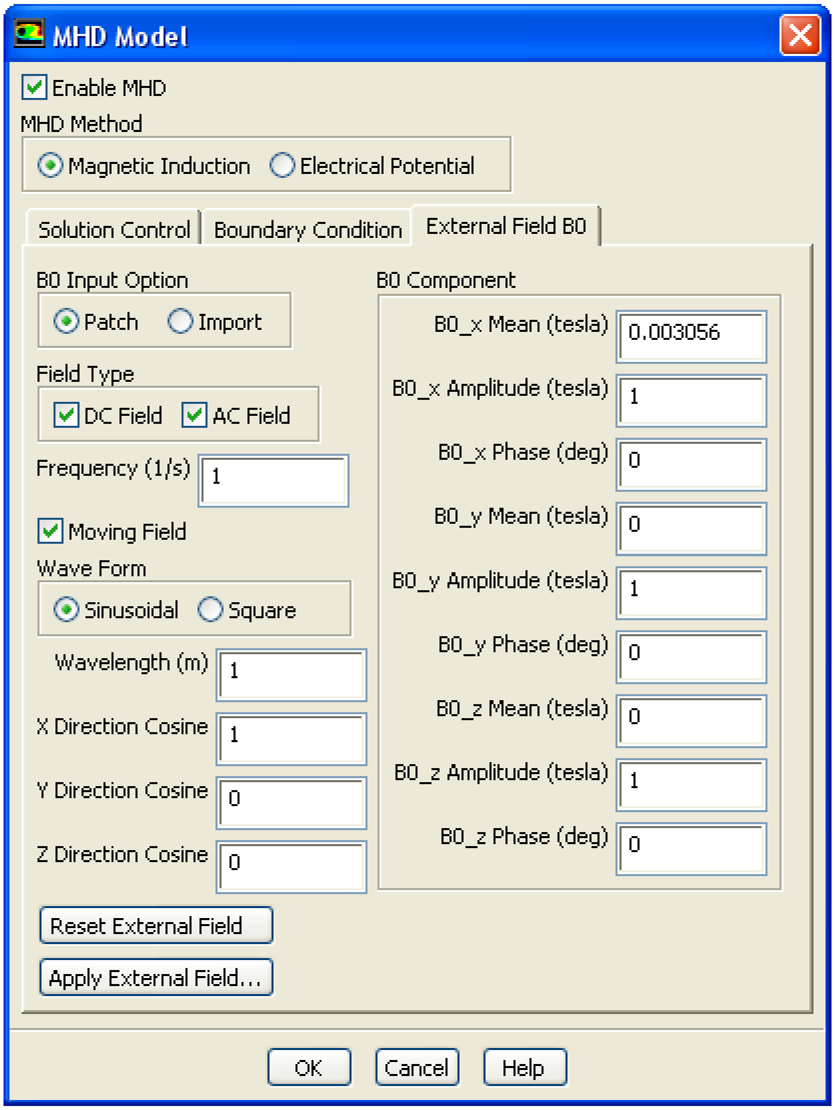
With the **Patch** option enabled, the AC field can be expressed as a function of time (specified by **Frequency**), and space (specified by wavelength, propagation direction and initial phase offset). The space components are set under the **B0 Component**, as in *Figure 4.3* (p. 14).

Figure 4.3 The MHD Model Dialog Box for Patching an External Magnetic Field



You can also specify a **Moving Field** with a wave form that is either a sinusoidal or a square wave function (*Figure 4.4* (p. 15)). Definitions for the sinusoidal and square wave forms of patched magnetic fields are provided in *Appendix B* (p. 27).

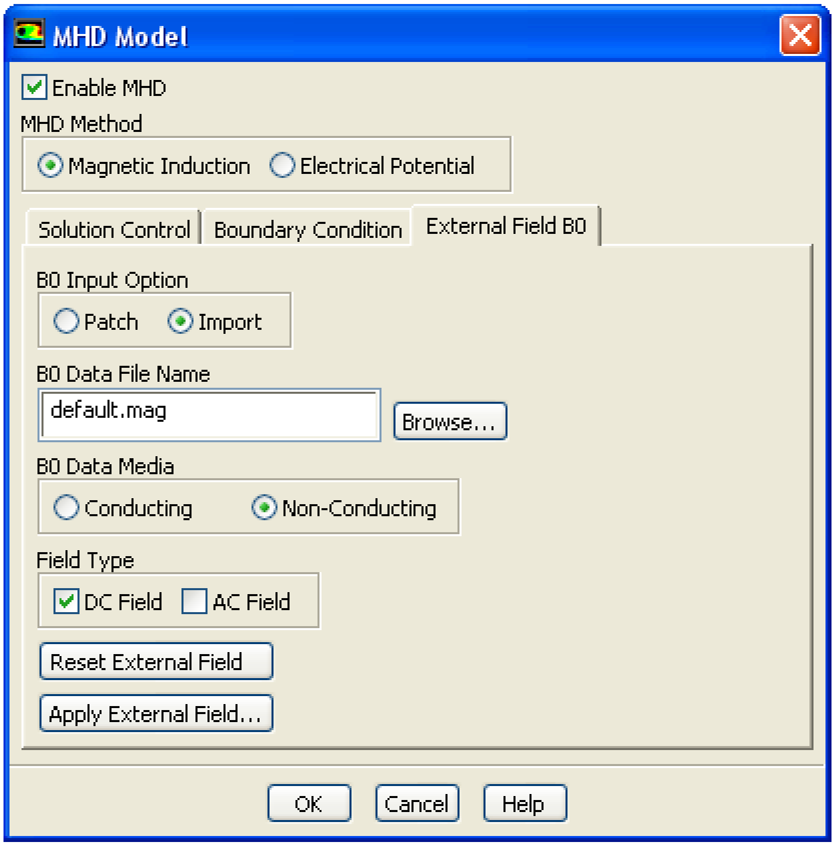
Figure 4.4 The MHD Model Dialog Box for Specifying a Moving Field



Selecting **Import** under the **B0 Input Option** in the **MHD Model** dialog box, as seen in *Figure 4.5* (p. 16), will result in the import of magnetic field data. The data file name can be entered in the **B0 Data File Name** field, or selected from your computer file system using the **Browse...** button. Magnetic data can also be generated using a third-party program such as MAGNA. The required format of the magnetic field data file is given in *Appendix C* (p. 29).

When using the **Import** option, the **B0 Data Media** is either set to **Non-Conducting** or **Conducting**, depending on the assumptions used in generating the magnetic field data. (These choices correspond to "Case 1" and "Case 2", respectively, as discussed in *Magnetic Induction Method* (p. 4).)

Figure 4.5 The MHD Model Dialog Box for Importing an External Magnetic Field

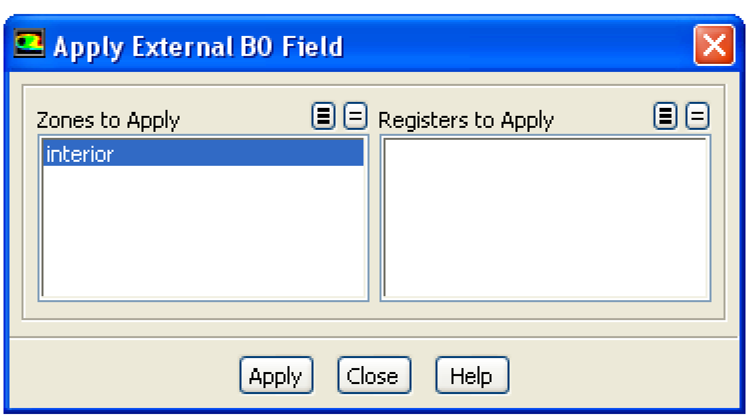


The **Field Type** is determined by the field data from the data file. The choice of either the **DC Field** or the **AC Field** option in the dialog box is irrelevant if the import data is either DC or AC. However, selection of both options indicates that data of both field types are to be imported from the data file, and superimposed together to provide the final external field data. Make sure that the data file contains two sections for the required data. See *Appendix C* (p. 29) for details on data file with two data sections.

The **Apply External Field...** button opens the **Apply External B0 Field** dialog box as shown in *Figure 4.6* (p. 16). To apply the external field data to zones or regions in the computational domain, select the zone names or register names of marked regions from the dialog box and click the **Apply** button.

The **Reset External Field** button sets the external magnetic field variable to zero.

Figure 4.6 Apply External B0 Field Dialog Box

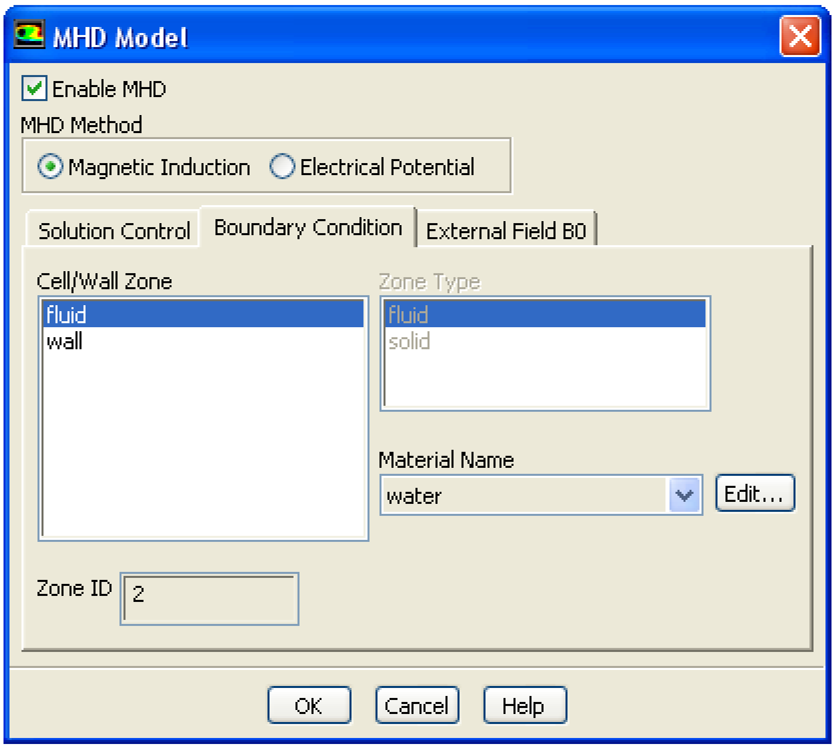


4.3.4. Setting Up Boundary Conditions

Boundary conditions related to MHD calculations are set under the **Boundary Condition** tab in the **MHD Model** dialog box. Boundary conditions can be set to cell zones and wall boundaries.

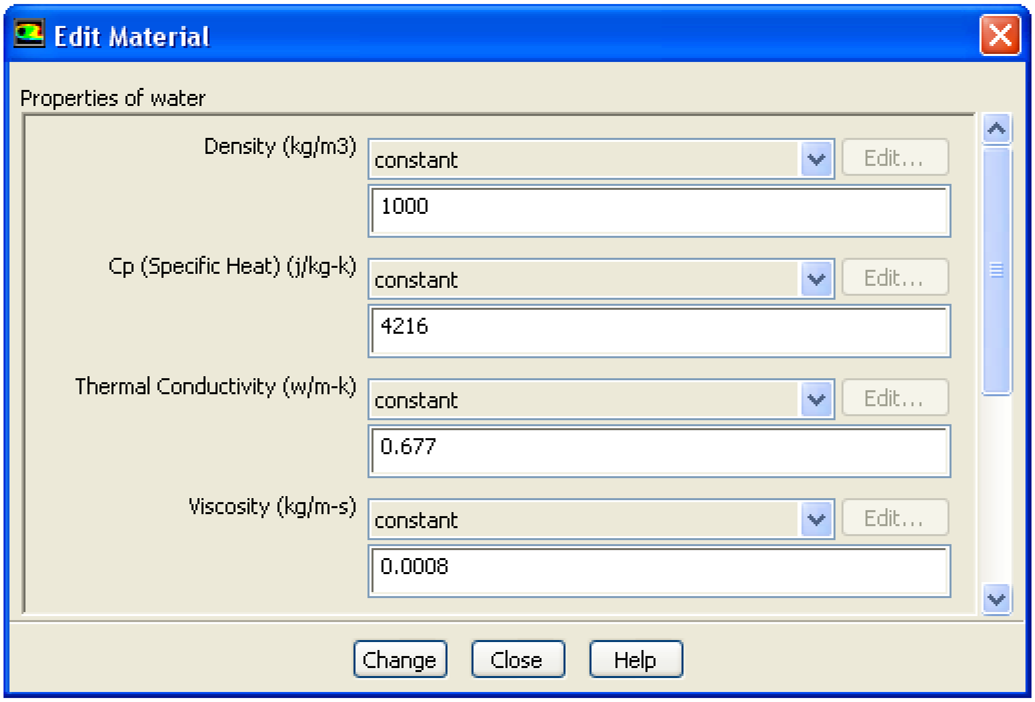
For cell zones, only the associated material can be changed and its properties modified. *Figure 4.7* (p. 17) shows the dialog box for cell zone boundary condition setup. The cell zone material can be selected from the **Material Name** drop-down list.

Figure 4.7 Cell Boundary Condition Setup



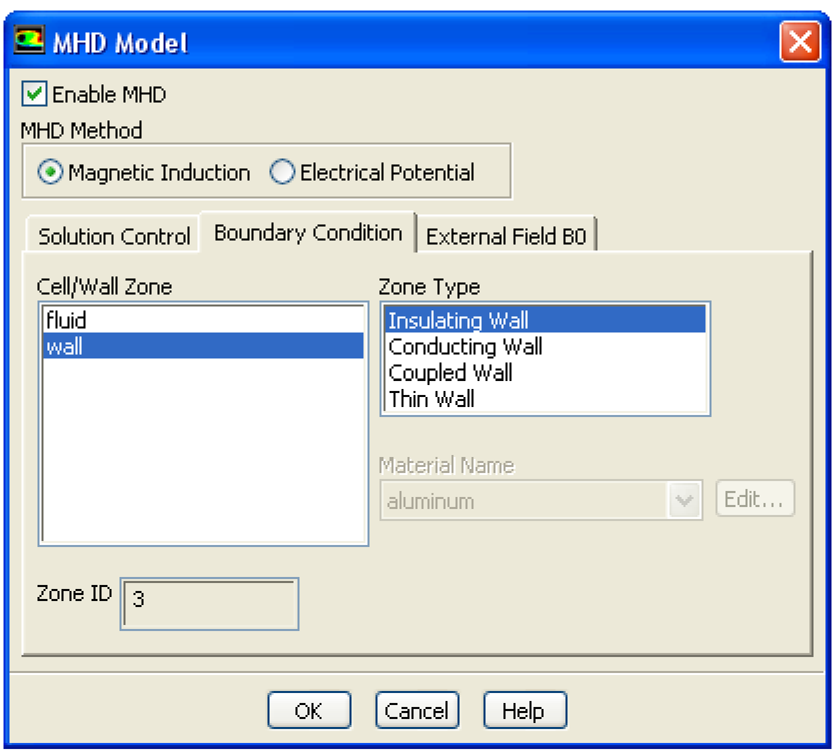
Note that the materials available in the list are set in the general ANSYS FLUENT case setup. Please refer to the ANSYS FLUENT User Guide for details on adding materials to an ANSYS FLUENT case. The properties of the selected material can be modified in the **Boundary Condition** tab by clicking **Edit...** to the right of the material name. This opens the **Edit Material** dialog box, as shown in *Figure 4.8* (p. 18). The material properties that may be modified include the electrical conductivity and magnetic permeability. The material electrical conductivity can be set as constant, a function of temperature in forms of piecewise-linear, piecewise-polynomial or polynomial, or as a user-defined function. The material magnetic permeability can only be set as a constant.

Figure 4.8 Editing Material Properties within Boundary Condition Setup



For wall boundaries, the boundary condition can be set as an **Insulating Wall**, **Conducting Wall**, **Coupled Wall** or **Thin Wall**. The dialog box for wall boundary condition setup is shown in *Figure 4.9* (p. 18).

Figure 4.9 Wall Boundary Condition Setup



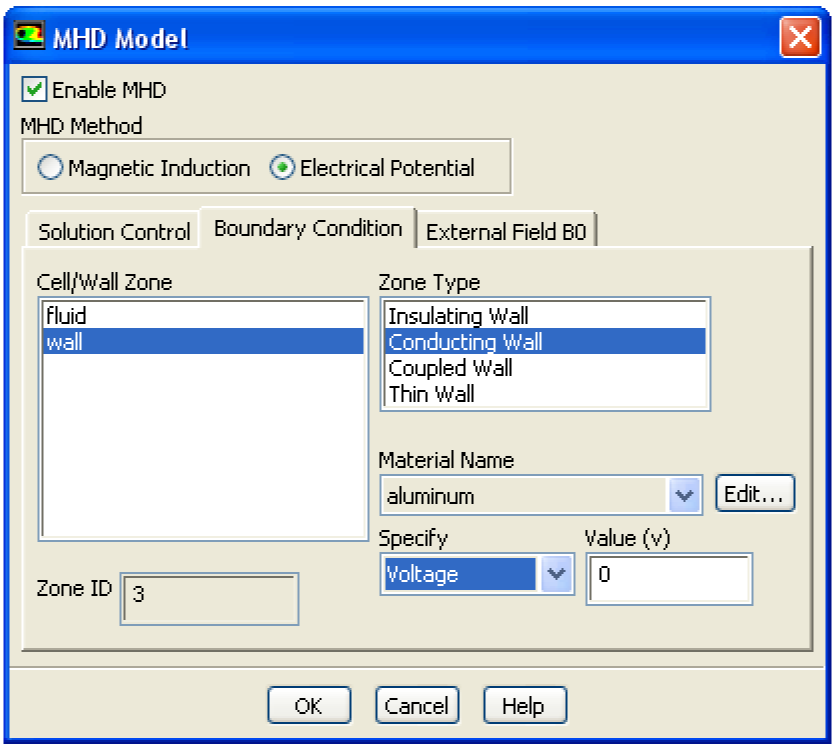
- The insulating wall is used for boundaries where there is no electric current going through the boundary.

- The conducting wall is used for boundaries that are perfect conductors.
- The coupled wall should be used for wall boundaries between solid/solid or solid/fluid zones where the MHD equations are solved.
- The thin wall type boundary can be used for external wall that has a finite electrical conductivity.

For conducting walls and thin wall boundaries, the wall material can be selected from the **Material Name** drop-down list, and its properties modified through the **Edit Material** dialog box. A wall thickness needs to be specified for thin wall type boundaries.

If the **Electric Potential** method is selected, the conducting wall boundary is specified by either of **Voltage** or **Current Density** at the boundary, as shown in *Figure 4.10* (p. 19).

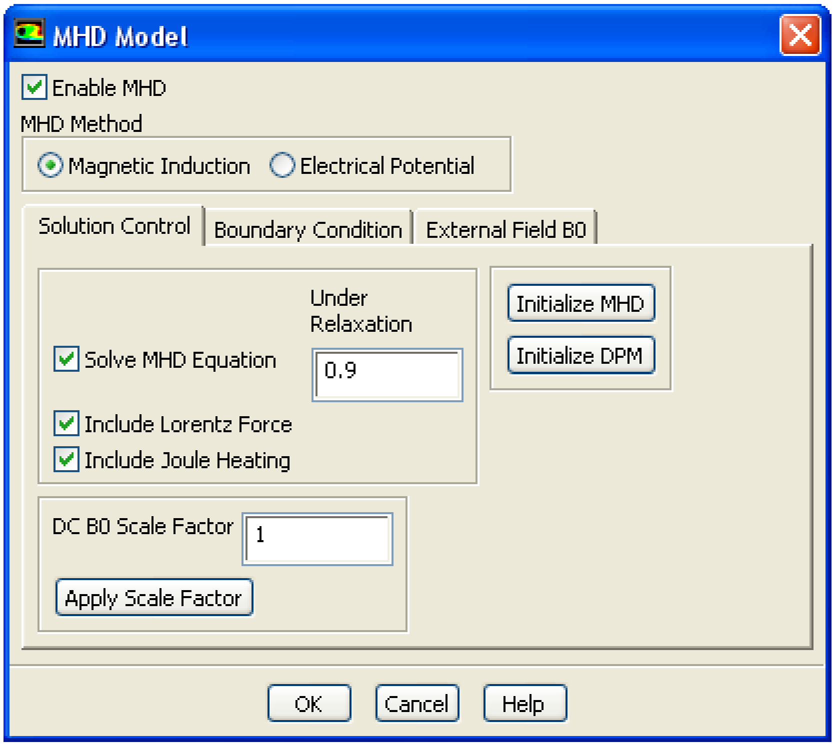
Figure 4.10 Conducting Wall Boundary Conditions in Electrical Potential Method



4.3.5. Solution Controls

Under the **Solution Control** tab in the **MHD Model** dialog box, *Figure 4.11* (p. 20), a number of parameters can be set that control the solution process in an MHD calculation. The MHD model can be initialized using the **Initialize MHD** button. When the DPM model is enabled in the ANSYS FLUENT case setup, the related variables used in the MHD model can be initialized using the **Initialize DPM** button.

Figure 4.11 Solution Control Tab in the MHD Model Dialog Box



You have the option to enable or disable the **Solve MHD Equation**. When the **Solve MHD Equation** is enabled, you have the choice to **Include Lorentz Force** and/or **Include Joule Heating** in the solution of flow momentum and energy equations. The underrelaxation factor for the MHD equations can also be set.

The strength of the imposed external magnetic field can be adjusted by specifying and applying scale factors to the external DC and/or AC magnetic field data.

4.4. MHD Solution and Postprocessing

The following sections describe the solution and postprocessing steps for the MHD model:

4.4.1. MHD Model Initialization

Initialization of the MHD model involves setting the externally-imposed magnetic field and initializing all MHD related user-defined scalars and memory variables.

When an ANSYS FLUENT case is initialized, all user-defined scalar and memory variables are set to zero. The external magnetic field data is set from the **External Field B0** tab in the **MHD Model** dialog box. The **Initialize MHD** button under the **Solution Control** tab can be used to initialize the model during an ANSYS FLUENT solution process. It is used when MHD effects are added to a fully or partially solved flow field, or when the model parameters are changed during an MHD calculation. It only clears the scalar variables and most of the memory locations used in the MHD model, the memory variables for the external magnetic field data are preserved.

4.4.2. Iteration

It is often an effective strategy to begin your MHD calculations using a previously-converged flow field solution. With this approach, the induction equations themselves are generally easy to converge. The underrelaxation factors for these equations can be set to 0.8 ~ 0.9, although for very strong magnetic fields, smaller values may be needed. For the electric potential equation, the convergence is generally slow. However, the underrelaxation value for this equation should not be set to 1. As additional source terms are added to the momentum and energy equations, the underrelaxation factors for these equations should generally be reduced to improve the rate of convergence. In case of convergence difficulties, another helpful strategy is to use the **B0 Scale Factor** in the **Solution Control** tab (*Figure 4.2* (p. 13)). This will gradually increase the MHD effect to its actual magnitude through a series of restarts. When the strength of the externally imposed magnetic field is strong, it is advisable to start the calculation with a reduced strength external field by applying a small scale factor. When the calculation is approaching convergence the scale factor can be increased gradually until the required external field strength is reached.

4.4.3. Postprocessing

You can use the standard postprocessing facilities of ANSYS FLUENT to display the MHD calculation results.

Contours of MHD variables can be displayed using

Graphics and Animations → **Contours** → **Set Up...**

The MHD variables can be selected from the variable list.

Vectors of MHD variables, such as the magnetic field vector and current density vector, can be displayed using

Graphics and Animations → **Vectors** → **Set Up...**

The vector fields of the MHD variables are listed in the **Vectors of** drop-down list in the **Vectors** dialog box. *Table 4.2: MHD Vectors* (p. 21) lists the MHD related vector fields.

Table 4.2 MHD Vectors

Name	Unit	Description
Induced- $\vec{B}$ -Field	Tesla	Induced magnetic field vector
External- $\vec{B}$ -Field	Tesla	Applied external magnetic field vector
Current-Density $\vec{J}$	A / m^2	Induced current density field vector
Electric-Field $\vec{E}$	V/m	Electric field density vector
Lorentz-Force $\vec{F}$	N / m^3	Lorentz force vector

4.5. Limitations

Many MHD applications involve the simultaneous use of other advanced ANSYS FLUENT capabilities such as solidification, free surface modeling with the volume of fluid (VOF) approach, DPM, Eulerian multiphase, and so on. You should consult the latest ANSYS FLUENT documentation for the limitations that apply to those features. In addition, you should be aware of the following limitations of the MHD capability.

- As explained in *Magnetohydrodynamic Model Theory* (p. 3), the MHD module assumes a sufficiently conductive fluid so that the charge density and displacement current terms in Maxwell's equations can be neglected. For marginally conductive fluids, this assumption may not be valid. More information about this simplification is available in the bibliography.
- For electromagnetic material properties, only constant isotropic models are available. Multiphase volume fractions are not dependent on temperature, species concentration, or field strength. However, sufficiently strong magnetic fields can cause the constant-permeability assumption to become invalid.
- You must specify the applied magnetic field directly. The alternative specification of an imposed electrical current is not supported.
- In the case of alternating-current (AC) magnetic fields, the capability has been designed for relatively low frequencies; explicit temporal resolution of each cycle is required. Although not a fundamental limitation, the computational expense of simulating high-frequency effects may become excessive due to small required time step size. Time-averaging methods to incorporate high-frequency MHD effects have not been implemented.

Appendix A. Guidelines For Using the ANSYS FLUENT MHD Model

This appendix provides a basic outline for installing the magnetohydrodynamics (MHD) module and solving MHD problems in ANSYS FLUENT.

Important

While *Using the ANSYS FLUENT MHD Module* (p. 11) covers much of the same material in greater detail, this appendix presents a set of guidelines for solving typical MHD problems with ANSYS FLUENT, with occasional references to *Using the ANSYS FLUENT MHD Module* (p. 11) where more information can be found.

A.1. Installing the MHD Module

Before using the MHD module, you first need to install the necessary files onto your computer. These files are provided with your standard installation of ANSYS FLUENT. They can be found in your installation area in a directory called `addons/mhd`. The MHD module is loaded into ANSYS FLUENT through the text user interface (TUI)

`define` → `models` → `addon-module`

only after a valid ANSYS FLUENT case file has been set or read.

Once the MHD model is installed, beneath the `mhd` directory there are two subdirectories: a `lib` directory, and a directory corresponding to your specific architecture, `ntx86` for example. The `lib` directory holds a Scheme code called `addon.bin` that contains the MHD module graphical interface. The specific architecture directory, `ntx86` for example, contains the following subdirectories that hold various ANSYS FLUENT files:

```
2d   2ddp   3d   3ddp
2d_host  2ddp_host  3d_host  3ddp_host
2d_node  2ddp_node  3d_node  3ddp_node
```

A.2. An Overview of Using the MHD Module

To use the MHD module in an ANSYS FLUENT simulation, follow the general guidelines:

1. Start ANSYS FLUENT.

   To begin modeling your MHD simulation, you need to start an appropriate ANSYS FLUENT session. Choose from either the 2D, 3D, **Double Precision**, or the parallel version of ANSYS FLUENT.

2. Read in a mesh file or a case file.

   You can have ANSYS FLUENT read in your mesh file, a previously saved non-MHD case file, or a previously saved MHD case file.

Important

Note that if you read in a new mesh file, you need to perform the appropriate mesh check and mesh scale procedures.

3. Load the MHD module.

   The MHD module is loaded into ANSYS FLUENT using the text command

   `define → models → addon-module`

   and entering the corresponding module number (*Loading the MHD Module* (p. 11)).

4. Set up the MHD model.

   The **MHD Model** dialog box is accessed using the graphical user interface (GUI):

   Models → **MHD Model** → **Edit...**

   If the MHD model is not enabled after the MHD module is loaded for the first time, you can enable it by clicking the **Enable MHD** button, which will display the expanded dialog box (*Enabling the MHD Model* (p. 13)).

5. Select an MHD method.

   The method used for the MHD calculation can be selected under **MHD Method**. The two methods are

   - **Magnetic Induction** (*Magnetic Induction Method* (p. 4))
   - **Electrical Potential** (*Electric Potential Method* (p. 6))

6. Apply an external magnetic field.

   This is done by entering values for the B0 components in the **External Field B0** tab. B0 input options can either be

   - **Patch**ed, or
   - **Import**ed

   The **Field Type** will either be the **DC Field** or the **AC Field**. The **Field Type** is determined by the field data from the data file. Refer to *Applying an External Magnetic Field* (p. 14) for details on applying an external magnetic field.

7. Set up the boundary conditions.

   Under the **Boundary Condition** tab, cell zones and wall boundaries can be selected as well as the corresponding zone type.

   Cell zone materials are selected from the **Material Name** drop-down list. The properties of the selected material can be modified by clicking on the **Edit...** button to the right of the material name. Note that the materials available in the list are set in the general ANSYS FLUENT case setup.

   Materials

The material properties that may be modified include the electrical conductivity and magnetic permeability.

Wall boundary conditions can be set as an **Insulating Wall**, **Conducting Wall**, **Coupled Wall** or **Thin Wall** (see *Setting Up Boundary Conditions* (p. 17)).

8. Set solution controls.

   Under the **Solution Control** tab:

   - The MHD equation is enabled or disabled.
   - Lorentz force and Joule heat sources are enabled or disabled.
   - Underrelaxation factors are set (reasonable underrelaxation factors for the MHD equations are 0.8 ~ 0.9).
   - Scale factors can be used to adjust the strength of the imposed external magnetic field. As the calculation approaches convergence, the scale factor in the **Solution Control** tab can be increased gradually until the required external field strength is reached (*Solution Controls* (p. 19)).
   - The MHD model is initialized (*MHD Model Initialization* (p. 20)).

9. Run the ANSYS FLUENT MHD simulation.

   Run Calculation

   Set the number of iterations. It is often an effective strategy to begin your MHD calculations using a previously-converged flow field solution. With this approach, the induction equations themselves are generally easy to converge. For more information, see *Iteration* (p. 21).

10. Process the solution data.

    You can use the standard postprocessing facilities of ANSYS FLUENT to display the results of an MHD calculation. Contours of MHD variables can be displayed.

    Graphics and Animations → **Contours** → **Set Up...**

    The MHD variables can be selected from the variable list. Vectors of MHD variables, such as the magnetic field vector and current density vector, can be displayed using custom vectors.

    Graphics and Animations → **Vectors** → **Set Up...**

    For more information, see *Postprocessing* (p. 21).

Appendix B. Definitions of the Magnetic Field

The sinusoidal form of the magnetic field is defined as:

$$B_0 = \overline{B}_0 + A_0 \cos\,(2\pi ft - K \cdot R + \phi)$$
$$K = \frac{1}{\lambda}\left\{\frac{1}{\cos\alpha}i + \frac{1}{\cos\beta}j + \frac{1}{\cos\gamma}k\right\} \tag{B–1}$$

where $\overline{B}_0$ is the mean vector, A_0 is the amplitude vector, K is defined as the propagation vector, R is the position vector of an arbitrary point. $\cos\alpha$, $\cos\beta$ and $\cos\gamma$ are the x, y and z direction cosines respectively. The quantities f, λ, and ϕ are the frequency, wavelength, and phase offset, respectively. For a non-moving field the propagation vector is zero. For a static field only applies.

The square form of the magnetic field is defined as:

$$B_0 = \overline{B}_0 + A_0 \frac{\cos\,(2\pi ft - K \cdot R + \phi)}{|\cos\,(2\pi ft - K \cdot R + \phi)\,|} \tag{B–2}$$

The definition of the propagation vector is the same as for the sinusoidal form.

Appendix C. External Magnetic Field Data Format

The external magnetic field data file is in text format and of the following structure:

$$
\begin{array}{llllll}
MAG\text{-}DATA & & & & & \\
nX & nY & nZ & & & \\
X1 & Xn & & & & \\
Y1 & Yn & & & & \\
Z1 & Zn & & & & \\
nAC & Freq & & & & \\
BX_{re-1} & BY_{re-1} & BZ_{re-1} & BX_{im-1} & BY_{im-1} & BZ_{im-1} \\
\ldots & & & & & \\
BX_{re-n} & BY_{re-n} & BZ_{re-n} & BX_{im-n} & BY_{im-n} & BZ_{im-n}
\end{array}
$$

The first line is an identification tag for the data file. The second line defines the number of data points in the x, y and z directions. The next three lines define the ranges in x, y and z directions. The data points are assumed to be evenly distributed along each direction. Line 6 defines the AC field flag and frequency. When `nAC = 0`, the magnetic field is static. For AC field, `nAC = 1` and `Freq` is the frequency in Hz.

The rest of the data file contains the magnetic field data points. Each line defines the components of the real and imaginary parts of the magnetic field vector on one data point. The data points are indexed as:

$$
\begin{aligned}
& l = i + nX\left(\left(j-1\right) + nY\left(k-1\right)\right) \\
& i = 1,\ldots,nX;\ j = 1,\ldots,nY;\ k = 1,\ldots,nZ
\end{aligned}
\tag{C–1}
$$

The data is listed in the ascending order from 1 to n, where n is the total number of data points given by $n = nXnYnZ$.

For magnetic fields comprised of both DC and AC fields, the entire file structure described above is repeated for the DC and AC parts. These two sections within the same file will be imported into ANSYS FLUENT and stored separately. The order of the DC and AC sections of the file is not important.

The imported data is interpreted as a snapshot of the applied magnetic field at an instant in time. Complex form is used to accommodate oscillating/moving fields. Thus, using complex numbers, and with reference to the quantities defined in Appendix B (p. 27),

$$\begin{aligned} \overrightarrow{B_0} &\equiv \begin{Bmatrix} BX_{re} \\ BY_{re} \\ BZ_{re} \end{Bmatrix} + i \begin{Bmatrix} BX_{im} \\ BY_{im} \\ BZ_{im} \end{Bmatrix} \\ &= \overline{B}_0 + A_0 exp\ [i\ (2\pi ft + \phi)\ ] \end{aligned} \quad \textbf{(C–2)}$$

For a DC field,

$$\overline{B}_{o,i} = B_{re,i} \qquad i = x, y, z \quad \textbf{(C–3)}$$

For an AC field,

$$A_{o,i} = \sqrt{B_{re,i}^{\ 2} + B_{im,i}^{\ 2}} \qquad i = x, y, z \quad \textbf{(C–4)}$$

and

$$\phi_i = tg^{-1}\left(\frac{B_{im,i}}{B_{re,i}}\right) \qquad i = x, y, z \quad \textbf{(C–5)}$$

Note that when the external magnetic field import option is used, the frequency, f, read from this file supersedes the value specified in the GUI.

Appendix D. MHD Module Text Commands

`mhd-models/`
Define solver configuration.

  `enable-mhd?`
  Enable/disable MHD model.

  `mhd-method`
  Select MHD method.

  `boundary-conditions/`
  Define MHD boundary conditions

    `list-zones`
    List ANSYS FLUENT zone information.

    `fluid`
    Set fluid zone boundary condition.

    `solid`
    Set solid zone boundary condition.

    `wall`
    Set wall boundary condition.

`b0-scale-factor`
Set and apply external magnetic field scale factor.

`external-b0-field`
Set and apply external magnetic field data.

`initialize-mhd`
Initialize MHD model.

`initialize-dpm`
Initialize DPM related MHD variables.

`solution-control`
Set MHD solution control parameters.

Bibliography

[1] R. Moreau. *Magnetohydrodynamics*. Kluwer Academic Publishers. 1990.

Index