AUTOMATIC

DYNAMIC

INCREMENTAL

NONLINEAR

ANALYSIS

Theory and Modeling Guide

Volume IV: EM

ADINA 9.6 April 2020

ADINA R & D, Inc.

ADINA
Theory and Modeling Guide

Volume IV: ADINA EM

April 2020

ADINA R&D, Inc.
71 Elton Avenue
Watertown, MA 02472 USA

tel. (617) 926-5199
telefax (617) 926-0238
www.adina.com

April 2020 Printing
Printed in the USA

Table of contents

This page is intentionally left blank

Nomenclature

Typical units used here are:

Time	= s, second
Length	= m, meter
Electrical potential	= V, volt
Electric current	= A, ampere

Scales used in nondimensional quanties are:

L_*	= length scale
μ_*	= permeability scale
ε_*	= permittivity scale
H_*	= magnetic field intensity scale

Notation	Explanation
Ω_e	electric domain
Ω_m	magnetic domain
Ω_f	fluid domain
Ω_{em}	electric and magnetic coupled domain
Ω_{ef}	electric and fluid coupled domain
Ω_{mf}	magnetic and fluid coupled domain
Ω_{emf}	electric, magnetic and fluid coupled domain
2DE	2D formulations in the E-plane
2DH	2D formulations in the H-plane

Notation	Explanation	Typical unit
$\mathbf{E}$	electric field intensity	$[\mathrm{V/m}]$
$\mathbf{H}$	magnetic field intensity	$[\mathrm{A/m}]$
ϕ	electric potential	$[\mathrm{V}]$
$\mathbf{A}$	magnetic potential	$[\mathrm{V\text{-}s/m}]$
$\mathbf{D}$	electric flux density	$[\mathrm{A\text{-}s/m^2}]$
$\mathbf{D}^*$	$=\varepsilon^*\mathbf{E}$	
$\mathbf{B}$	magnetic flux density	$[\mathrm{V\text{-}s/m^2}]$
$\mathbf{J}_c$	electric conduction current density	$[\mathrm{A/m^2}]$
$\mathbf{J}$	electric current density $(=\mathbf{J}_c+\partial\mathbf{D}/\partial t)$	$[\mathrm{A/m^2}]$
$\mathbf{K}$	magnetic current density	$[\mathrm{V/m^2}]$
μ	permeability	$[\mathrm{V\text{-}s/A\text{-}m}]$
ε	permittivity	$[\mathrm{A\text{-}s/V\text{-}m}]$
ε^*	$=\begin{cases}\varepsilon & \text{in static analysis}\\ \varepsilon-i\sigma\omega^{-1} & \text{in harmonic analysis}\end{cases}$	$[\mathrm{A\text{-}s/V\text{-}m}]$
σ	conductivity	$[\mathrm{A/V\text{-}m}]$
ρ_0	imposed source of electric charge density	$[\mathrm{A\text{-}s/m^3}]$
$\mathbf{J}_0$	imposed source of electric current density	$[\mathrm{A/m^2}]$
$\mathbf{K}_0$	imposed source of magnetic current density	$[\mathrm{V/m^2}]$
$\mathbf{I}_0$	imposed source of electric current	$[\mathrm{A}]$
ω	frequency	$[\mathrm{1/s}]$

Chapter 1 Introduction

ADINA-EM is a general finite-element/finite-volume code that can be used for analyzing electromagnetic (EM) problems. The solutions can be coupled with CFD solutions in the ADINA-CFD+EM package that includes both ADINA-EM and ADINA-CFD.

In this Theory and Modeling Guide, the theoretical bases and guidelines for the use of the ADINA-EM capabilities and the CFD-EM coupled solution procedure are presented.

This guide is organized as follows. A brief introduction is given in Chapter 1. It includes all key ingredients in ADINA-EM and the CFD-EM coupling. For experienced ADINA users, it is enough for them to start. More details on some important subjects are given in the remaining chapters.

The governing equations of first and second orders, in both *E-H* and A-ϕ mathematical formulations are presented in Chapter 2. The *modules* and *analysis* types are also introduced in ADINA-EM.

The concept of *path* is described in Chapter 3. The *path* is used to ease the input of space-varying directions and functions.

In Chapter 4, all the parameters required in element groups are described, including material data, electromagnetic sources and the control parameters.

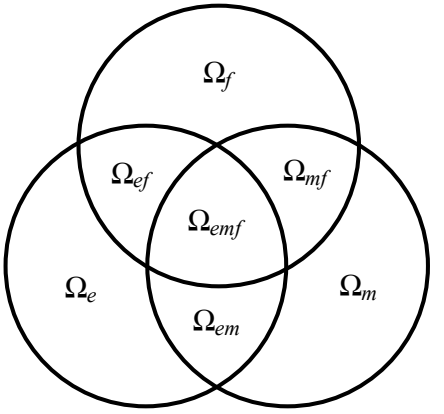
Various boundary conditions available in ADINA-EM are detailed in Chapter 5, including their mathematical representation and the required input parameters.

The discretized linear equations are solved using iterative solvers. In Chapter 6, we introduce the control parameters and explain how they work during the iteration procedure.

In Chapter 7, we explain how ADINA-EM and ADINA-CFD solutions are coupled, in terms of the Maxwell stress and the Joule heating rate. For CFD modeling, the ADINA-CFD theory and modeling guide is referenced.

1.1 Computational domain and coordinate system

The computational domain can be in 2 or 3 dimensional space. It may consist of an electric domain Ω_e and/or magnetic domain Ω_m. In CFD-EM coupled models, it also includes a fluid domain Ω_f. These sub-domains may be partially or fully coincident, illustrated in the figure below.



Ω_f = fluid domain
Ω_e = electric domain
Ω_m = magnetic domain
$\Omega_{ef} = \Omega_e \bigcap \Omega_f$
$\Omega_{mf} = \Omega_m \bigcap \Omega_f$
$\Omega_{em} = \Omega_e \bigcap \Omega_m$
$\Omega_{emf} = \Omega_e \bigcap \Omega_m \bigcap \Omega_f$

Figure 1.1 Physical sub-domains used in ADINA-CFD+EM

The Cartesian coordinate system (x, y, z) is used in 3 dimensional domains, while the system (y, z) is used in 2 dimensions.

1.2 Variable, source and material data

In ADINA-EM, the independent solution variables are electric field intensity $\mathbf{E}$ and magnetic field intensity $\mathbf{H}$, or electric potential ϕ and

magnetic potential $\mathbf{A}$. Other resultant variables are electric flux density $\mathbf{D}$, magnetic flux density $\mathbf{B}$, electric current density $\mathbf{J}$ and magnetic current density $\mathbf{K}$.

The material data required are permeability μ, permittivity ε, conductivity σ, and, in harmonic analysis, the frequency of the model ω.

The imposed sources are electric charge density ρ_0, electric current density $\mathbf{J}_0$, magnetic current density $\mathbf{K}_0$ and electric current $\mathbf{I_0}$.

The input of sources ρ_0, $\mathbf{J}_0$, $\mathbf{K}_0$ or $\mathbf{I}_0$ may need *path* data sets that ease the definition of space-varying directions and functions. The *path* option is described in detail in Chapter 3.

1.3 Analysis type

The static and harmonic *analysis* types are available in ADINA-EM.

In static analysis, all solution variables, boundary condition values, sources and material data are assumed to be independent of time.

In harmonic analysis, while the material data are still assumed constant in each element group, all solution variables, sources and boundary condition values are assumed to vary in the form

$$f = f_r \cos\omega t - f_i \sin\omega t = \mathrm{Re}\left(f^* e^{i\omega t}\right) \tag{1.1}$$

where $i = \sqrt{-1}$ and $f^* = f_r + if_i$. The real component f_r and the imaginary component f_i are required in the input.

A non-zero phase variable must be written in the form of Eq.(1.1) to obtain its real and complex components. Since $\mathrm{Re}\left(f^* e^{i(\omega t+\theta)}\right) = \mathrm{Re}\left(f^* e^{i\theta} e^{i\omega t}\right)$, we have

$$f^* e^{i\theta} = \underbrace{\left(f_r \cos\theta - f_i \sin\theta\right)}_{\text{real component}} + i\underbrace{\left(f_r \sin\theta + f_i \cos\theta\right)}_{\text{imaginary component}}$$

For example, the input values of the real and imaginary components of $2\cos\left(\omega t + \frac{\pi}{2}\right) + \sin\left(\omega t + \frac{\pi}{2}\right)$ are 1 $\left(= 2\cos\frac{\pi}{2} + \sin\frac{\pi}{2}\right)$ and 2 $\left(= 2\sin\frac{\pi}{2} - \cos\frac{\pi}{2}\right)$ respectively.

We need to note that the total number of equations or degrees of freedoms in harmonic analysis is double that in static analysis. The solutions have real and imaginary components.

In both static and harmonic analyses, the input data are assumed independent of time. However, in order to ease the solution execution procedure, "time-varying" inputs are allowed. They are not physically time varying, but simply represent data values at different computational steps. This feature is useful in solving a series of similar problems in one execution.

1.4 Mathematical formulation and module

There are two mathematical formulations available in ADINA-EM: the one that governs electric and magnetic field intensities (E-H formulation), and the one that governs magnetic and electric potentials (A-ϕ formulation).

Either the E-H or A-ϕ formulation can be used, but they cannot be used in the same model. Furthermore, in each formulation, one or both variables can be active.

In the 3D *E*-*H* formulation, both *E* and *H* are vectors in 3D space, and in the A-ϕ formulation, *A* is a vector in 3D space and ϕ is a scalar.

There are two types of 2D formulations, one with vector electric field intensity and one with vector magnetic field intensity. They are called here *E*-plane (2DE) and *H*-plane (2DH) formulations respectively. Based on

this definition, solution variables can be easily classified according to the governing equations.

For example, in 2DE E-H formulations, since electric filed intensity is a vector in the (y,z) plane, magnetic field intensity must be a scalar that contains only its *x*-component. In 2DE A-ϕ formulation, since magnetic field intensity is a scalar, its potential is then a vector in the (y,z) plane. Similarly, in 2DH formulations, the magnetic field intensity is a vector in the (y,z) plane, while the electric field intensity (or magnetic potential in the A-ϕ formulation) is a scalar (with *x*-component only).

ADINA-EM provides *modules* to identify those diversities. A *module* combines the mathematical formulation, the space dimension and the active/inactive variables. Only one module can be selected in a problem.

The following table lists the available modules in ADINA-EM.

MODULE	USED FOR	SOLUTION VARIABLES
EH3D	3D E-H model	$E_x, E_y, E_z, H_x, H_y, H_z$
E03D	3D E model	E_x, E_y, E_z
0H3D	3D H model	H_x, H_y, H_z
FA3D	3D A-ϕ model	A_x, A_y, A_z, ϕ
F03D	3D ϕ model	ϕ
0A3D	3D A model	A_x, A_y, A_z
EH2DE	2D E-H model in E-plane	E_y, E_z, H_x
E02DE	2D E model in E-plane	E_y, E_z
0H2DE	2D H model in E-plane	H_x
FA2DE	2D A-ϕ model in E-plane	A_y, A_z, ϕ
F02DE	2D ϕ model	ϕ
0A2DE	2D A model in E-plane	A_y, A_z
EH2DH	2D E-H model in H-plane	E_x, H_y, H_z
E02DH	2D E model in H-plane	E_x
0H2DH	2D H model in H-plane	H_y, H_z
0A2DH	2D A model in H-plane	A_x

Table 1.1 Electromagnetic modules in ADINA-EM

The module ID characters shown in the first column are self-explanatory. For example, F02DE indicates the 2D model in the electric plane (indicated by the last 3 characters "2DE") with only ϕ active (indicated by the first 2 characters "F0"), while FA2DE is the same module but having both ϕ and $\mathbf{A}$ active (indicated by characters "FA").

Table 1.1 contains three parts: the 3D modules are in the first part, and the 2DE and 2DH modules are in the second and third parts respectively.

The solution variables shown in the last column are for static analysis. For harmonic analysis, they represent both real and imaginary components. The number of unknowns is then doubled.

1.5 Elements and element groups

The sub-domains are discretized with *elements*, and elements are combined into one or more *element groups*. An element group normally represents a sub-domain that has the same material properties, sources and active variables.

In each element group, a set of material properties of the sub-domain must be defined. The data set includes permeability, permittivity, conductivity, etc.

Depending on the problem, multiple sources may be imposed. The type of source can be electric charge density (ρ_0), electric current density $(\mathbf{J}_0)$, magnetic current density $(\mathbf{K}_0)$ and electric current $(\mathbf{I}_0)$. Note that sources of the same type are added up in the program.

Electric/magnetic variables can be set active/inactive for an element group. Element groups with active electric variables form the electric domain, and those with active magnetic variables form the magnetic domain. One or both variables may be active in the same element group.

The table below details the possible choice in one element group.

	ELECTRIC VARIABLE	MAGNETIC VARIABLE	FLUID VARIABLE
Ω_e	active	inactive	inactive
Ω_m	inactive	active	inactive
Ω_{em}	active	active	inactive
Ω_f	inactive	inactive	active
Ω_{ef}	active	inactive	active
Ω_{mf}	inactive	active	active
Ω_{emf}	active	active	active

Table 1.2 Active variables in sub-domains

In ADINA-EM, the active/inactive electromagnetic variables are automatically set in all element groups according to the selected *module*. User may deactivate them later in element groups if needed. For example, 0H2DE indicates that active magnetic variable and inactive electric variable are initially set in all element groups.

In ADINA-CFD+EM, a default CFD model is assumed. Consequently, electromagnetic variables are initially set inactive. They may be activated later in two steps: (1) turn on EM option and (2) activate electromagnetic variables in the element groups where they are present.

2D/3D elements are used in 2D/3D models. Depending on the number of vertices, 2D elements are either 3-node (triangle) or 4-node (quadrilateral) elements, and 3D elements are 4-node (tetrahedron), 5-node (pyramid), 6-node (wedge) and 8-node (brick) elements.

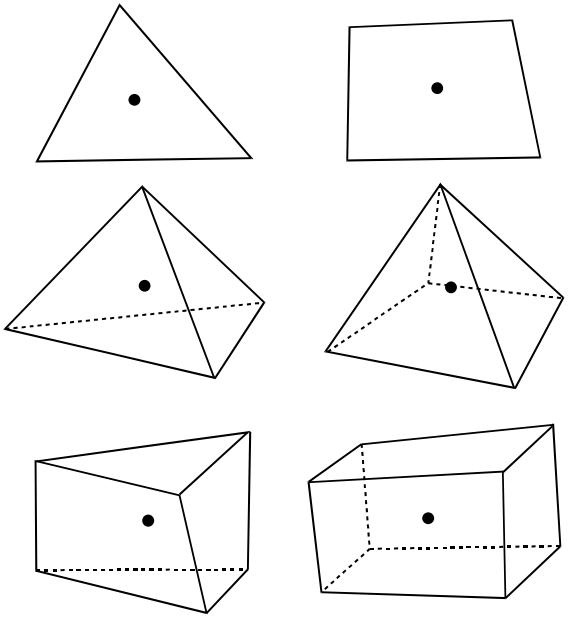


Figure 1.2 Elements and solution location in ADINA-EM

1.6 Boundary conditions

ADINA-EM provides some basic types of boundary conditions. The following table summaries these conditions, the corresponding solution variables that the conditions can be applied to, and conditions where these are applicable.

Condition	Variables	Covered sample conditions
Dirichlet	$\mathbf{E}, \mathbf{H}, \mathbf{A}, \phi$	Balloon
Normal	$\mathbf{E}, \mathbf{H}, \mathbf{A}, \phi$	Perfect symmetry planes
Parallel	$\mathbf{E}, \mathbf{H}, \mathbf{A}$	Perfect *E*; perfect *H;* perfect symmetry planes
Natural	$\mathbf{E}, \mathbf{H}$	
Impedance	$\mathbf{E}, \mathbf{H}, \mathbf{A}$	Finite conductivity; Lumped RLC; Imperfect conductor

A boundary condition is a data set that is applied to a part of the boundary. Each condition set is only for one variable. If a boundary condition is applied for the coupled variables, it must be defined twice, and applied to each of the variables separately.

If the electric and magnetic domains are not coincident, so are their boundaries. It is important to apply boundary conditions to the boundary of the corresponding sub-domains.

Every part of (electric of magnetic) boundary must have one and only one condition applied.

On coincident boundaries, the conditions for the two variables must be consistent.

For most types of boundary conditions, the boundary curvature is an important factor that affects the solution accuracy. A reasonably fine mesh is necessary on curved boundaries. ADINA-EM computes curvature within each boundary condition set. If a boundary consists of parts that have different curvatures, the curvature at the intersection might be incorrectly computed. Therefore, it is necessary that conditions be separately applied to each part of the boundary of different curvatures.

In case the parameters in a boundary condition set requires space-varying direction (e.g, non-zero *E-Parallel* condition) or values (e.g. *Balloon* condition), a *path* data set may be used. It eases the input of the direction and space-varying function. The *path* data input is described in detail in Chapter 3.

1.7 Units and non-dimensional analysis

The use of a proper unit system can help the solution convergence in certain problems. A general guidance on choosing units is to have solution variables ($\mathbf{E}/\mathbf{H}$ or $\mathbf{A}/\phi$) of order one.

It is our intention to leave the choice of unit system to users. Any unit system can be used, provided that units are consistent in the governing

equations, through all data input, including material data, boundary conditions, etc. The units of ADINA-EM solutions are consistent with the units of input.

In particular, a unit system with dimensionless input can be used. The solution is then in consistent dimensionless form.

ADINA-EM also provides an automatic procedure for using dimensionless variables after a model has been created with dimensional data. In this procedure, the user first prepares the model as usual — all input data are in a dimensional unit system. The user then specifies some independent variable scales in the final stage of model preparation. With those scales, ADINA-EM automatically transfers all input data into dimensionless form and uses these in the computations. However, the output solutions are seamlessly in the original dimensional unit.

The required variable scales in this procedure are the length (L_*), permeability (μ_*), permittivity (ε_*) and magnetic field intensity (H_*). All of them are required even in problems where only one variable is active. In this procedure, the following resultant scales are automatically determined and used in ADINA-EM:

$$
\begin{aligned}
E_* &= H_* \sqrt{\mu_* / \varepsilon_*} \\
\sigma_* &= L_*^{-1} \sqrt{\varepsilon_* / \mu_*} \\
\rho_{0*} &= E_* \varepsilon_* / L_* \\
J_{0*} &= H_* / L_* \\
K_{0*} &= E_* / L_* \\
\omega_* &= 1 \Big/ \left(L_* \sqrt{\mu_* \varepsilon_*} \right) \\
A_* &= \mu_* H_* L_* \\
\phi_* &= E_* L_*
\end{aligned}
$$

It is sometimes more efficient to choose σ instead of ε as an independent scale in problems with conductors present. In this case, input the permittivity scale as

$$\varepsilon_* = \mu_* \left(L_* \hat{\sigma} \right)^2$$

where $\hat{\sigma}$ is a typical conductivity in the model.

In ADINA-CFD+EM, the automatic nondimensinal procedure is also available, in which the length scale L_* is used for both the CFD and EM models.

1.8 Solution output

All computed independent solution variables in ADINA-EM are defined at the element center. The output solutions are defined at nodes. They are interpolated using a second-order algorithm using the element center solutions. The interpolation scheme also respects the specified boundary conditions on boundary nodes.

In the E-H mathematical formulation, the solution variables on material interfaces are specially interpolated, according to the interface conditions. Note that, since we do not introduce double nodes on an interface, the nodal solution represents an averaged value of neighboring elements, even if the accurate solution is discontinuous. This interpolated interface solution is for visualization purpose only. It is not, nor affects the computational solution that is defined at the element centers.

In the automatic nondimensional procedure, as mentioned previously, the solution output is presented in dimensional form that is consistent with the original input (default option). However, if required, the output solution can also be in dimensionless form according to the nondimensional system used in the computation.

1.9 Model preparation steps

An electromagnetic model is a FEM model. It consists of all steps and ingredients that a FEM model requires, such as geometry input, meshing,

etc. In addition to these standard steps in the FEM procedure, ADINA-EM requires a few more input data to generate an EM model. These steps are briefly described hereafter.

- Input global control parameters, including *module*, *analysis* type, and the *parameters* that govern the convergence;
- For harmonic analysis, input the *frequency*;
- If the A-ϕ formulation is selected, choose the *gauge* type;
- Generate element groups, including meshing, input of material data sets ($\varepsilon, \mu, \sigma, \ldots$), electromagnetic sources ($\rho_0, \mathbf{K}_0, \mathbf{J}_0, \mathbf{I}_0$), and set active variables (electric and/or magnetic);
- Apply boundary conditions;
- Apply other available conditions if needed.

1.10 EM and CFD coupled solutions

The electromagnetic solutions can be coupled with CFD solutions using ADINA-CFD+EM.

FCBI-C elements must be used in the CFD models.

The electromagnetic and CFD solutions are coupled in element groups where both the electromagnetic and CFD solution variables are active. The Maxwell stress and Joule heating rate are calculated using the EM solutions and added into the fluid stress and energy sources, respectively, in the CFD solutions.

The same mesh is used for the fluid and electromagnetic variables in the coupled element groups.

It is a good idea to always prepare the CFD and EM models separately and test them before running a coupled solution.

To activate the coupling algorithm, in addition to preparing the CFD and EM models, one must (1) turn on EM option in ADINA-CFD+EM; (2) choose the methods used in calculating the Maxwell stress and Joule heating rate; and (3) set both the fluid and electromagnetic variables active in the coupled element groups.

Note that the default element group active variables are different in ADINA-EM and ADINA-CFD+EM. So it is better to explicitly set the active/inactive variables, irrespective of the default option, in either separate or joined models.

There are three options for calculating Maxwell stress/Joule heating rate, and different types may be used for stress and heat source calculation.

(1) Zero stress/heat source (default);
(2) Averaged over one period in harmonic analysis; and
(3) Directly computed at a given time.

In CFD-EM coupled problems, the fluid analysis type is defined in the CFD model (see ADINA-CFD Theory and Modeling Guide for details), irrespective of the analysis type used in the EM model. For example, the CFD model can be transient while the EM model can be static or harmonic. However, one must be careful if the CFD solution is steady-state and the EM solution is harmonic. Usually, the averaged EM solution is used for computing the Maxwell stress and the heat source.

The following steps set up the coupling of the two models.

- Turn on the EM option (the default option is a pure CFD model in ADINA-CFD+EM);
- Choose the methods for calculating the Maxwell stress and Joule heating rate;
- Activate both the fluid and electromagnetic variables in the coupled element groups.

Chapter 2 Electromagnetic governing equations

2.1 Original Maxwell equations

The original first-order full Maxwell equations, in general time-varying fields, can be written as the Faraday law and the Maxwell-Ampere law

$$\nabla \times \mathbf{E} = -\mathbf{K} \qquad in\ \Omega_e \tag{2.1}$$

$$\nabla \times \mathbf{H} = \mathbf{J} \qquad in\ \Omega_m \tag{2.2}$$

together with the Gauss laws respectively for the electric and magnetic fields

$$\nabla \bullet \mathbf{D}^* = \rho_0 \qquad in\ \Omega_e \tag{2.3}$$

$$\nabla \bullet \mathbf{B} = 0 \qquad in\ \Omega_m \tag{2.4}$$

where,

$$\begin{aligned} \mathbf{D}^* &= \varepsilon^* \mathbf{E} \\ \mathbf{B} &= \mu \mathbf{H} \end{aligned} \tag{2.5}$$

and

$$\mathbf{J} = \mathbf{J}_0\big|_{\Omega_m} + \left(\sigma \mathbf{E} + \frac{\partial \mathbf{D}}{\partial t}\right)\Bigg|_{\Omega_m \cap \Omega_e}$$

$$\mathbf{K} = \mathbf{K}_0\big|_{\Omega_e} + \frac{\partial \mathbf{B}}{\partial t}\Bigg|_{\Omega_m \cap \Omega_e}$$

From now on, we will neglect the subscripts Ω_e, etc., that indicate the sub-domains.

In harmonic analysis, since the variables are expressed as $\mathrm{Re}\left(fe^{i\omega t}\right)$ as shown in Eq.(1.1), $\partial/\partial t \equiv i\omega$, we have

$$\begin{aligned} \mathbf{J} &= \mathbf{J}_0 + i\omega\mathbf{D}^* \\ \mathbf{K} &= \mathbf{K}_0 + i\omega\mathbf{B} \end{aligned} \tag{2.6}$$

In static analysis, Eq.(2.6) becomes

$$\begin{aligned} \mathbf{J} &= \mathbf{J}_0 + \sigma\mathbf{E} \\ \mathbf{K} &= \mathbf{K}_0 \end{aligned} \tag{2.7}$$

2.2 E-H formulation of second-order Maxwell equations

The second order equation system is obtained by applying the operator $\nabla\times$ to Eqs. (2.1-2.2),

$$\nabla\times\nabla\times\mathbf{E} = -\nabla\times\mathbf{K} \qquad in\ \Omega_e$$

$$\nabla\times\nabla\times\mathbf{H} = \nabla\times\mathbf{J} \qquad in\ \Omega_m$$

By introducing

$$\begin{aligned} p &= \nabla\bullet\mathbf{E} - \rho_0/\varepsilon^* \\ q &= \nabla\bullet\mathbf{H} \end{aligned}$$

the second order equations become

$$\nabla\bullet\left(\left(p + \rho_0/\varepsilon^*\right)\mathbf{I} - \nabla\mathbf{E} + \mathbf{I}\times\mathbf{K}\right) = 0 \quad in\ \Omega_e \tag{2.8}$$

$$\nabla\bullet\left(q\mathbf{I}-\nabla\mathbf{H}-\mathbf{I}\times\mathbf{J}\right)=0 \qquad in\ \Omega_m \tag{2.9}$$

Eqs. (2.3-2.9) form the E-H mathematical formulation used in ADINA-EM.

In electric-only *E-H* modules (E03D and all E02D), Eqs. (2.4) and (2.9) are omitted, while in magnetic-only modules (0H3D and all 0H2D), Eqs. (2.3) and (2.8) are omitted. The simplest E-H modules, in terms of number of equations, are the static E02DH and 0H2DE models. They are the Poisson equations governing the scalar variable E_x and H_x respectively.

The second-order equation system is equivalent to the first-order system, if proper boundary conditions are used.

2.3 A-ϕ potential formulation of electromagnetic equations

Introducing the electric and magnetic potentials

$$\begin{aligned}\mathbf{E}&=-\nabla\phi-\frac{\partial\mathbf{A}}{\partial t}\\ \mathbf{B}&=\nabla\times\mathbf{A}\end{aligned} \tag{2.10}$$

and assuming

$$\nabla\bullet\mathbf{A}=g_A \tag{2.11}$$

where

$$g_A=\begin{cases}0 & \textit{Coulomb gauge approximation}\\ -\mu\varepsilon^*\dfrac{\partial\phi}{\partial t} & \textit{Lorentz gauge approximation}\end{cases}$$

Eqs.(2.1) and (2.4) are automatically satisfied (assuming $\mathbf{K}_0 = \mathbf{0}$), and Eqs.(2.2-2.3) become, respectively,

$$\nabla \bullet \left(-\varepsilon^* \left(\nabla \phi + i\omega \mathbf{A} \right) \right) = \rho_0 \tag{2.12}$$

$$\nabla \times \left(\mu^{-1} \nabla \times \mathbf{A} \right) = \mathbf{J}_0 - i\omega \varepsilon^* \left(\nabla \phi + i\omega \mathbf{A} \right) \tag{2.13}$$

Eqs. (2.11-2.13) form the A-ϕ mathematical formulation for harmonic analysis used in ADINA-EM. For static analysis, Eqs.(2.12) and (2.13) become, respectively,

$$\nabla \bullet \left(-\varepsilon \nabla \phi \right) = \rho_0 \tag{2.12'}$$

and

$$\nabla \times \left(\mu^{-1} \nabla \times \mathbf{A} \right) = \mathbf{J}_0 - \sigma \nabla \phi \tag{2.13'}$$

In the electric-only A-ϕ modules (F03D and all F02D), Eqs. (2.11) and (2.13) are omitted, while in the magnetic-only modules (0A3D and all 0A2D), Eqs. (2.13) and (2.15) are omitted. The simplest A-ϕ modules, in terms of number of equations, are the static F02DE and 0A2DH models. They are Poisson equations governing the scalar variable ϕ and A_x respectively.

2.4 Three-dimensional models

All 3D models are associated with 3D meshes in the (x, y, z) Cartesian coordinate system.

In general, the above-described governing equations are for all models. In particular, they are valid for 3D models.

In the E-H and A-ϕ formulations, any one or both of the two variables

may be active. For example, we may want to solve the magnetic field intensity only, with a specified eddy-current. It is clear then that there are six 3D *modules*, namely EH3D (coupled *E* and *H*), E03D (*E*-only), 0H3D (*H*-only), FA3D (coupled *A* and ϕ), F03D (ϕ-only) and 0A3D (*A*-only). They are listed in the first part of Table (1.1).

In 3D models, all vector variables and vector sources are in 3D space.

2.5 Two-dimensional models

All 2D models are associated with 2D meshes in the (y,z) plane Cartesian coordinate system.

There are two different 2D models: in the electric plane (*E*-plane) and in the magnetic plane (*H*-plane).

In the *E*-plane models, the electric field intensity is a vector in the (y,z) plane, while the magnetic field intensity is a scalar along the *x*-axis. Similarly, 2D *H*-plane modules are defined if the magnetic field intensity is a vector and the electric field intensity is a scalar.

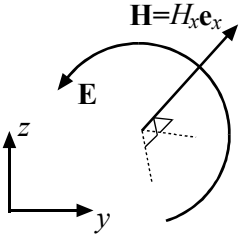
This definition is also valid for the A-ϕ formulation. Recalling the definition of these potentials, in the 2D *E*-plane modules, the magnetic potential is a vector in the (y,z) plane (since the magnetic filed intensity is a scalar), and in the 2D *H*-plane modules, the magnetic potential is a scalar along the *x*-axis (since the magnetic filed intensity is a vector). Note that the electric potential is always a scalar.

There are totally six 2D *E*-plane modules, namely EH2DE (coupled *E* and *H*), E02DE (*E*-only), 0H2DE (*H*-only), FA2DE (coupled *A* and ϕ), F02DE (ϕ-only) and 0A2DE (*A*-only). They are listed in the second part of Table (1.1).

However, there are only four 2D *H*-plane modules, namely EH2DH (coupled *E* and *H*), E02DH (*E*-only), 0H2DH (*H*-only) and 0A2DH (*A*-

only). It is understandable that modules FA2DH (coupled A and ϕ) and F02DH do not exist, since a scalar E indicates a known ϕ. They are listed in the last part of Table (1.1).

2.5.1 2D E-H formulation in the E-plane



In 2D E-plane modules, **E** is assumed vector in the y-z plane, while **H** is assumed a scalar along the x-axis. In E-H formulation, we solve for $\left(E_y, E_z, H_x\right)$.

Figure 2.1 Illustration of 2D E-plane modules

As an example, we present the first-order Maxwell equations in harmonic analysis

$$\begin{aligned} &\frac{\partial E_z}{\partial y} - \frac{\partial E_y}{\partial z} = -K_{0x} - i\omega\mu H_x \\ &\frac{\partial\left(\varepsilon^* E_y\right)}{\partial y} + \frac{\partial\left(\varepsilon^* E_z\right)}{\partial z} = \rho_0 \end{aligned} \tag{2.14}$$

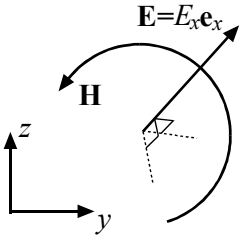
and

$$\begin{aligned} &\frac{\partial H_x}{\partial z} = J_{0y} + i\omega\varepsilon^* E_y \\ &-\frac{\partial H_x}{\partial y} = J_{0z} + i\omega\varepsilon^* E_z \end{aligned} \tag{2.15}$$

As we can see, the source K_{0x} is a scalar and $\left(J_{0y}, J_{0z}\right)$ is a vector, consistent with the solution variable assumptions.

In modules E02DE, Eq.(2.15) is omitted and in 0H2DE, Eq.(2.14) is omitted. In the two cases, the solution variables become $\left(E_y, E_z\right)$ and H_x respectively.

2.5.2 2D E-H formulation in the H-plane



In 2D *H*-plane modules, **E** is assumed a scalar along the *x*-axis, while **H** is assumed a vector in the *y*-*z* plane. In the *E*-*H* formulation, we solve for $\left(E_x, H_y, H_z\right)$.

Figure 2.2 Illustration of 2D *H*-plane modules

The equations in harmonic analysis are

$$\begin{aligned} \frac{\partial E_x}{\partial z} &= -K_{0y} - i\omega\mu H_y \\ -\frac{\partial E_x}{\partial y} &= -K_{0z} - i\omega\mu H_z \end{aligned} \tag{2.16}$$

and

$$\begin{aligned} \frac{\partial H_z}{\partial y} - \frac{\partial H_y}{\partial z} &= J_{0x} + i\omega\varepsilon^* E_x \\ \frac{\partial\left(\mu H_y\right)}{\partial y} + \frac{\partial\left(\mu H_z\right)}{\partial z} &= 0 \end{aligned} \tag{2.17}$$

Note that the source $\left(K_{0y}, K_{0z}\right)$ is a vector while J_{0x} is a scalar.

Similarly, Eq.(2.17) is omitted in module E02DH, and Eq.(2.16) is omitted in 0H2DH. In the two cases, the solution variables become E_x and $\left(H_y, H_z\right)$ respectively.

2.5.3 2D A-ϕ formulation in the E-plane

Since the electric field intensity is a vector while the magnetic filed intensity is a scalar, in this coupled model, we solve $\left(\phi, A_y, A_z\right)$. The governing equations are, for example, for static analysis,

$$-\frac{\partial}{\partial y}\left(\varepsilon\frac{\partial\phi}{\partial y}\right)-\frac{\partial}{\partial z}\left(\varepsilon\frac{\partial\phi}{\partial z}\right)=\rho_0 \tag{2.18}$$

and

$$\begin{aligned}
&\frac{\partial\left(\mu^{-1}g_A\right)}{\partial y}-\frac{\partial}{\partial y}\left(\mu^{-1}\frac{\partial A_y}{\partial y}\right)-\frac{\partial}{\partial z}\left(\mu^{-1}\frac{\partial A_y}{\partial z}\right)=J_{0y}-\sigma\frac{\partial\phi}{\partial y}\\
&\frac{\partial\left(\mu^{-1}g_A\right)}{\partial z}-\frac{\partial}{\partial y}\left(\mu^{-1}\frac{\partial A_z}{\partial y}\right)-\frac{\partial}{\partial z}\left(\mu^{-1}\frac{\partial A_z}{\partial z}\right)=J_{0z}-\sigma\frac{\partial\phi}{\partial z}\\
&\frac{\partial A_y}{\partial y}+\frac{\partial A_z}{\partial z}=g_A
\end{aligned} \tag{2.19}$$

Note that the source $\left(J_{0y}, J_{0z}\right)$ is a vector. In module F02DE, Eq.(2.19) is omitted and in 0A2DE, Eq.(2.18) is omitted. The solution variables are, respectively in the two modules, ϕ and $\left(A_y, A_z\right)$.

2.5.4 2D A-ϕ formulation in the H-plane

In this model, since *E* is a scalar and *H* is a vector, only a scalar A_x is the unknown solution variable. The governing equation is

$$-\frac{\partial}{\partial y}\left(\mu^{-1}\frac{\partial A_x}{\partial y}\right)-\frac{\partial}{\partial z}\left(\mu^{-1}\frac{\partial A_x}{\partial z}\right)=J_{0x}-i\omega\varepsilon^{*}\frac{\partial\phi}{\partial x} \tag{2.20}$$

In which, $\partial\phi/\partial x$ must be an imposed quantity. Furthermore, since all the terms in the right hand side of Eq.(2.20) must be specified, they can be combined into one variable. So the input of the value of J_{0x} physically represents $J_{0x}-i\omega\varepsilon^{*}\,\partial\phi/\partial x$.

2.6 Selection of mathematical model

The following global parameters control the selection of the mathematical model and the analysis process.

- *Module* module type, EH3D, EH2DE, etc. in ADINA-EM, this parameter must be input.
- *Analysis* analysis type, static or harmonic. The default option is static.

Module is a 4-5 character string, indicating the mathematical formulation, space dimensions, active/inactive solution variables and, for 2D cases, the vector plane. All available *modules* are given in the Table (1.1).

If the A-ϕ mathematical formulation is selected in *module*, the gauge type must be input

- *Gauge* gauge type, Coulomb or Lorentz approximation. The default type is Coulomb.

If harmonic is selected in *analysis*, the frequency and its associated time function must be input.

- *Frequency* ω, the value of frequency. In harmonic analysis, this parameter must be specified.

- *Nfrequency* time function for ω in harmonic analysis. The default is 0 (indicating a constant frequency).

Note that, in harmonic analysis, solution variables, sources and boundary condition values include both real and imaginary components. Accordingly, the total number of degrees of freedom (DOF) or number of equations is double that in static analysis.

Chapter 3 Path

Path is a tool in ADINA-EM to ease the input of space-dependent directions and space-varying data. They are referred to in the input of *sources* and *boundary conditions*. Currently there are three types of path available: *point*, *line* and *line-extruded*.

3.1 Definition and input of path

A *path* is a data set.

A *point-path* is a single point in space $\mathbf{r}_1^p$.

A *line-path* is a series of oriented points $\mathbf{r}_1^p \rightarrow \mathbf{r}_2^p \rightarrow \cdots \rightarrow \mathbf{r}_n^p$. The values of $\mathbf{r}_1^p$ and $\mathbf{r}_n^p$ may be coincident, forming a closed path. A line path is generated in the same way as a line mesh. Line-path should be long enough to cover the domain to which it is applied. The segment length should be small enough to represent the path's curvature, yet large enough to maintain computational efficiency. For example, two points are the best choice to define a straight line-path. For a curved line-path, on the other hand, the segment length should be about half the element length.

A *line-extruded-path* is the surface generated when a *line-path* is extruded infinitely in the specified directions $\pm\mathbf{d}_e$. Note that the size of the surface in the line direction is the same as the line size, and is of infinite length in both extruded directions.

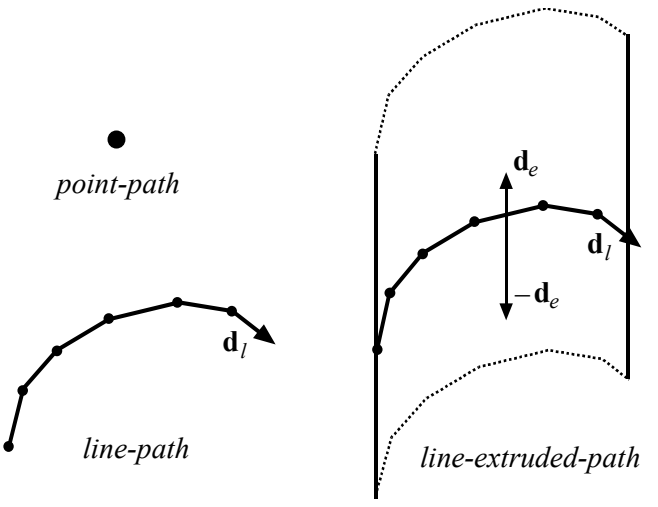


Figure 3.1 Illustration of paths

3.2 Direction definition

In this section, various direction types are described. These directions may require path data and be used in input of *source* and *boundary conditions*.

Consider a point **p** in space (as appearing in *source* set) or on a boundary (as appearing in *boundary condition* set). There are four basic directions at this point: (1) a fixed input direction $\mathbf{d}_0$; (2) the direction $\mathbf{d}_r$ from a path to the point **p** (in shortest distance); (3) the line direction $\mathbf{d}_l$ of a *line-path* or a *line-extruded-path*; and (4) the outward normal direction **n** on the boundary. Together with all the pairs of these directions, we define additional directions. They are types of directions available in ADINA-EM.

The simplest direction is given by the input one

$$\mathbf{d} = \mathbf{d}_0 \qquad type = D0$$

If the point **p** is on a boundary, **n** is available, so that two more direction types can be defined

$$\mathbf{d} = \begin{cases} \mathbf{n} & type = NR \\ \mathbf{d}_0 \times \mathbf{n} & type = D0XNR \end{cases}$$

The *type* characters are self-explanatory. For example, *D0XNR* represents *D0* "cross" *NR*. Note that the above three types of direction do not require *path*.

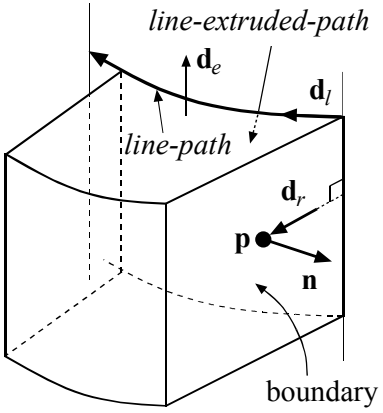
Associated with a *path*, we have another three directions

$$\mathbf{d} = \begin{cases} \mathbf{d}_r & type = DR \\ \mathbf{d}_0 \times \mathbf{d}_r & type = D0XDR \\ \mathbf{d}_r \times \mathbf{n} & type = DRXNR \end{cases}$$

In case the path is line or line extruded, we can also use the line direction $\mathbf{d}_l$ to have additional types. They are

$$\mathbf{d} = \begin{cases} \mathbf{d}_l & type = DL \\ \mathbf{d}_0 \times \mathbf{d}_l & type = D0XDL \\ \mathbf{d}_r \times \mathbf{d}_l & type = DRXDL \\ \mathbf{n} \times \mathbf{d}_l & type = NRXDL \end{cases}$$

All the direction ingredients are illustrated in the following figure.



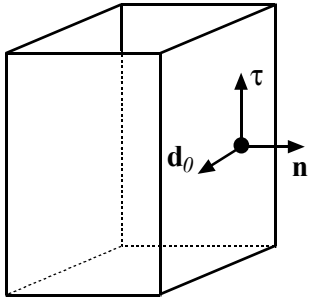
$\mathbf{d}_e$ = extruded direction
$\mathbf{d}_l$ = path direction
$\mathbf{p}$ = current point
$\mathbf{d}_r$ = direction from path to $\mathbf{p}$
$\mathbf{n}$ = boundary normal

Figure 3.2 Various ingredients associated with path

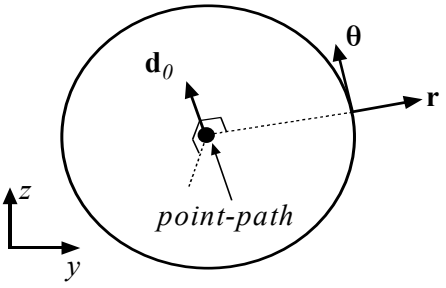
Here are some examples.

Example 3.1: A boundary normal direction is defined as *type=NR*.

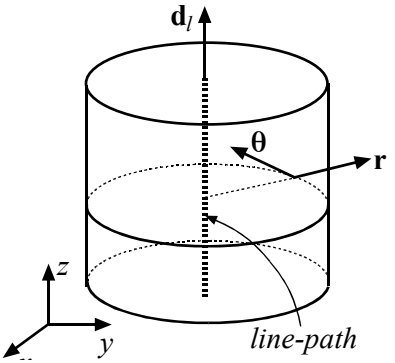
Example 3.2: A 2D boundary tangential direction is defined as $\boldsymbol{\tau} = \mathbf{d}_0 \times \mathbf{n}$ (*type=D0XNR*) with $\mathbf{d}_0 = (1,0,0)$.



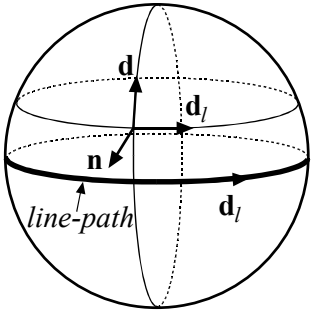
Example 3.3: A 3D boundary Cartesian coordinate system can be defined by an input $\mathbf{d}_0$ on the boundary surface, normal $\mathbf{n}$ (*type=NR*) and tangential $\boldsymbol{\tau} = \mathbf{d}_0 \times \mathbf{n}$ (*type=D0XNR*).



Example 3.4: Defining a path as the center point of a polar coordinate system (r,θ), *r*-direction is defined as *type=DR*, and θ-direction is defined as *type=D0XDR* with $\mathbf{d}_0 = (1,0,0)$.



Example 3.5: Defining a path as the *z*-axis in a cylindrical coordinate system (r,θ,z), *r*-direction is defined as *type=DR*, *z*-direction is the path direction $\mathbf{d}_l = (0,0,1)$, and θ-direction is $-\mathbf{n}\times\mathbf{d}_l$, defined as opposite direction of that given by *type=NRXDL*.



Example 3.6: Defining the Equator as a path on earth, then latitude and longitude directions are respectively $\mathbf{d}_l$ and $\mathbf{d} = \mathbf{n}\times\mathbf{d}_l$ (*type=NRXDL*). Together with the surface normal, they form a natural Cartesian coordinate system on earth.

3.3 Space-varying function

ADINA-EM allows certain types of space-varying data input. For an input parameter S, its space-varying data is defined as

$$S \leftarrow S\lambda(r)$$

where the function $\lambda(r)$ is associated with a *path* set and uniquely defined as

$$\lambda(r) = \begin{cases} \sum_{i=1}^{3} a_i r^{e_i} & if\ r \le R_{\max} \\ 0 & if\ r > R_{\max} \end{cases} \tag{3.4}$$

where, a_i, e_i and $R_{\max}$ are input constants, and r is the shortest distance from the path to the current point. Those constants are input in the data set where the *path* is referred. The default values of them are all zero, except $a_1 = 1$ and $R_{\max} = 10^{+21}$, so that $\lambda(r) \equiv 1$.

Note that this space function can represent some commonly used polynomial bases. For examples, $r, 1, r^{-1}, r^{-2}$, etc.

Chapter 4 Modeling data in element group

In the input for ADINA-EM, a parameter p is normally input via its multiplier p^m and the time function number n of its associated time function $f_n(t)$, so that its value is $p = p^m f_n(t)$. This does not mean the parameter is time-varying. The function is introduced for the purpose of ease of input for multiple step solutions. As default, $n = 0$ indicating a constant $p = p^m$. Usually, p^m has the same unit as p and the values of time function is dimensionless.

4.1 Input of material data

The material data set of ε, μ and σ are required in each element group. Different data sets may be assigned in different element group.

4.1.1 Constant material

To define a material set of constants data, input them directly

- *Epsilon* permittivity ε. No default value is assumed.
- *Mu* permeability μ. No default value is assumed.
- *Sigma* conductivity σ. No default value is assumed.

4.1.2 Time-dependent material

To define time-dependent material, in addition to the constants, input the numbers of their associated time functions as well.

- *Nepsilon* associated time function of permittivity.
- *Nmu* associated time function of permeability.
- *Nsigma* associated time function of conductivity.

Accordingly, the material parameters become

$$m \leftarrow mf_{Nm}(t) \qquad (m = \varepsilon, \mu, \sigma)$$

4.2 Input of sources

A *Source* is a data set, representing one of the charges, ρ_0, $\mathbf{K}_0$, $\mathbf{J}_0$ and $\mathbf{I}_0$ as defined in the governing equations. Source sets are applied to element groups. In harmonic analysis, they have both real and imaginary components. A source may be a scalar as ρ_0, or a vector.

4.2.1 Source in harmonic and static analyses

In harmonic analysis, the source has two components: real and imaginary. Recall the definition of them in Eq.(1.1), the coefficients of the basic functions $\cos\omega t$ and $-\sin\omega t$ are real and imaginary components respectively.

Example 4.1: The real and imaginary components of the electric charge density $\rho_0 = 0.9\times\cos\omega t + 0.1\times\sin\omega t$ are 0.9 and -0.1 respectively.

Example 4.2: The real and imaginary components of the electric charge density $\rho_0 = 0.1\times\sin\left(\omega t + \frac{1}{2}\pi\right) + 0.9\times\cos\left(\omega t + \frac{1}{2}\pi\right)$ are 0.1 and 0.9 respectively, since $\sin\left(\omega t + \frac{1}{2}\pi\right) = \cos\omega t$ and $\cos\left(\omega t + \frac{1}{2}\pi\right) = -\sin\omega t$.

In static analysis, only the real component is required.

4.2.2 Constant scalar source

To define a constant scalar source S, input the following parameters

- *Type* indicates type of source, ρ_0, $\mathbf{K}_0$, $\mathbf{J}_0$ or $\mathbf{I}_0$.
- *Component* indicates the component of the source, real or imaginary.
- *S* multiplier of the source component
- *Eg_i* the element groups to which the source is applied.

4.2.3 Time-dependent scalar source

A time-varying source is obtained by multiplying the value of a time function

$$S \leftarrow Sf_n(t)$$

The time function number defines the time function

- *Ns* source associated time function number. The default value is 0, meaning no time function associated with.

4.2.4 Vector source

A vector source $\mathbf{S}$ is a scalar source S, multiplied by a direction $\mathbf{d}$

$$\mathbf{S} = S\mathbf{d}$$

where the direction $\mathbf{d}$ is explained in *Path* section. The following parameters define the direction:

- *DirTyp* direction type
- *DX,DY,DZ* a constant direction $\mathbf{d}_0$
- *Path* path set number

Depending on the direction type, $\mathbf{d}_0$ and/or *path* may not be required. See details in Chapter 3 on the direction definition.

4.2.5 Space-varying source

Except the electric current $\mathbf{I}_0$, all sources can be varying in space. A *path* must be defined prior to this stage and referred to here, so that the space function $\lambda(r)$ in Eq.(3.4) can be defined (see details in Chapter 3 on the Space-varying function).

A space-varying source is obtained by multiplying the value of the function

$$S \leftarrow S\lambda(r)$$

The following parameters define the shape function $\lambda(r)$

- *Path* path number.
- *A1,A2,A3* constants a_1, a_2, a_3
- *EN1,EN2,EN3* constants e_1, e_2, e_3
- *Rmax* constant $R_{\max}$.

See detail explanation of these constants in Chapter 3.

4.2.6 Electric current source

An electric current source $\mathbf{I}_0$ specifies the current flow through a conductor cross-section. Unlike other types of sources that can be space-varying, it can only be constant or time-varying in the element group.

The electric current source is only available for the 2D magnetic plane modules EH2DH, 0H2DH and 0A2DH. In these modules, some element groups represent the modeled conductors, and the area is the conductor's cross-section. Furthermore, in these modules, $\mathbf{I}_0$ is always in the *x*-direction.

In addition to the governing equations, a constraint equation is imposed in each element group with a current present

$$\int_{A_i} \mathbf{J} dV = \mathbf{I}_{0i}$$

where the subscript *i* indicates the element group, and A_i is the surface occupied by element group *i*.

4.2.7 Notes on sources

Multiple sources can be input in one element group. The sources of the same type are added up to obtain the final source shown in the governing equations. To be more specific, we write a source in the general form

$$\mathbf{S} = \sum_j S_j f_{n_j}(t) \lambda_j (r_j) \mathbf{d}_j$$

where the subscript j refers to each source set. Of course, for scalar source, $\mathbf{d}$ is not required and for the electric current source, $\lambda_j (r_j)$ is 1.0.

4.3 Input of other parameters in element groups

The following parameters must also be input in element groups.

- *Electric* indicates whether the electric variable is active. In ADINA-EM, the default option follows the global parameter *module* selected. That is, it is active/inactive if the module indicates it is active/inactive. In ADINA-CFD+EM, the default option is inactive.
- *Magnetic* indicates whether the magnetic variable is active. In ADINA-EM, the default option follows the global parameter *module* selected. That is, it is active/inactive if the module indicates it is active/inactive. In ADINA-CFD+EM, the default option is inactive.

- *EM-Material* specifies the electromagnetic material set number. It must be set if either the electric or magnetic variable is active.
- *EM-Source* specifies the electromagnetic source set numbers. If omitted, zero sources are assumed. Multiple sources of various types can be input.

In ADINA-EM, the default option follows the global parameter *module* selected. That is, the electromagnetic variables are active/inactive if the module indicates they active/inactive.

In ADINA-CFD+EM, however, a pure CFD model is assumed as default. The electromagnetic variables are then, by default, inactive. The active fluid variable here means an incompressible or compressible fluid, porous medium or even a solid element group. For solid element groups, only the temperature is computed.

If it is necessary in modeling, the fluid variable can be turned inactive. Thus, the element group becomes a pure electromagnetic group and no CFD-EM coupling is presented in this group.

- *Fluid* indicates whether the fluid variable is active.

Chapter 5 Boundary conditions

5.1 Introduction

To ensure uniqueness and accuracy of electromagnetic field solutions, the application of proper boundary conditions is one of the most important steps. In many cases, improper conditions will result in solution divergence or an even wrong solution.

5.1.1 General rules of boundary conditions

Although boundary conditions of the coupled two variables are always related to each other in physics, it is a good idea to consider them separately at the stage of applying boundary conditions. It is also a good idea to separate the two sub-domains where the two variables are defined. In other words, apply the conditions of one variable at a time, to the boundary of the corresponding sub-domain. This statement is particularly important if the two domains are not coincident, since the boundaries S_e and S_m are at different locations. The following figure illustrates the sub-domains and their boundaries.

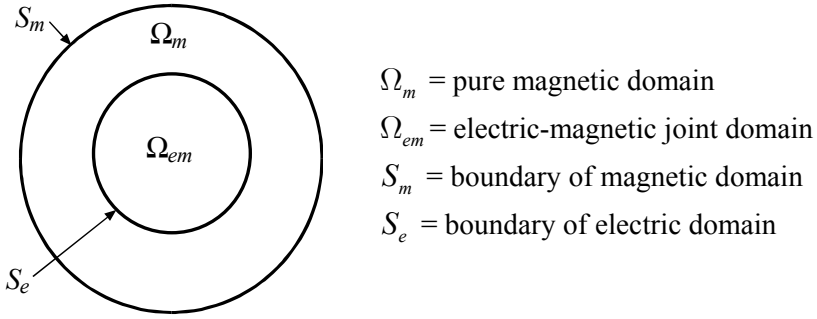


Figure 5.1 Illustration of electric and magnetic boundaries

Based on the above explanations, we can describe a general rule for one variable: all parts of the boundary must have one and only one condition applied. This statement is true for both electric and magnetic variables.

Next, the boundary conditions of the two variables must be compatible and consistent with their governing equations. This is particularly difficult yet important in E-H mathematical formulation. Some examples are presented here:

Example 5.1: *E-Normal* and *H-Parallel* conditions, and *H-Normal* and *E-Parallel* conditions are always consistent.

Example 5.2: In harmonic analysis, *H-Parallel* condition is consistent with *E-Dirichlet* or *E-Natural* condition. Similarly *E-Parallel* condition is consistent with *H-Dirichlet* or *H-Natural* condition.

Example 5.3: In static analysis, *E-Parallel* condition is consistent with *H-Dirichlet* or *H-Natural* condition on a conductor boundary $(\sigma > 0)$. However, *H-Parallel* condition is usually not consistent with *E-Natural* condition.

Example 5.4: *Natural* and *Normal* conditions are usually not consistent.

In ADINA-EM, a boundary condition set is only applicable for one variable $(\mathbf{E}, \mathbf{H}, \phi$ or $\mathbf{A})$. If a physical condition must be applied for both variables it must be applied separately for each of them. This condition may be modeled by the same or different boundary condition sets. Examples are given below:

Example 5.5: Impedance condition is for both *E* and *H*. The boundary condition must be defined twice, once for *E* and once for *H*, with input of the same Z_s, and applied separately on the same boundary.

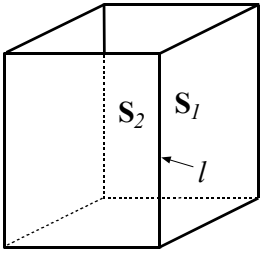
Example 5.6: Perfect *E*-symmetry condition consists of a zero *E-Parallel* condition set and a zero *H-normal* condition set.

Example 5.7: Perfect *H*-symmetry condition consists of a zero *H-Parallel* condition set and a zero *E-normal* condition set.

5.1.2 Boundary curvature

The curvature plays an important role in boundary conditions. The solution accuracy depends on it too.

In ADINA-EM, the curvature is approximately computed for each boundary where a condition set is applied. Therefore, it is necessary to apply the condition to boundary that has naturally similar curvature. If a boundary consists of different geometries, split it into a few boundary parts following the geometries, and then apply the same condition to each of them.



In the figure (left), boundary conditions must be applied separately onto the surfaces S_1 and S_2, even the physical conditions are the same. If $S_1 \bigcup S_2$ is in one condition set, the curvature on the joint line l will be incorrectly computed.

Figure 5.2 Curvatures of abutting boundary surfaces

5.1.3 Default boundary conditions

On a boundary, if no condition is assigned for a variable, a default one will be assigned for that variable. However, since these boundaries are not explicitly defined, the curvature cannot be computed correctly. ADINA-EM assumes therefore that the boundary is flat. It is important to be aware of that no default conditions should be assumed on curved boundaries. In other words, every curved boundary must be assigned conditions for all variables (even if the condition is the same as the default).

The default conditions are listed in the table below.

Variable	Default boundary condition
E	*Natural* condition on flat boundary
H	*Natural* condition on flat boundary
A	Zero *Parallel* condition on flat boundary
ϕ	Zero *Normal* condition on flat boundary

The details of the default conditions are described in separate sections in this Chapter.

5.1.4 Boundary conditions in harmonic and static analyses

In harmonic analysis, the input condition has two components: real and imaginary. Recall the definition of them in Eq.(1.1), the coefficients of the basic functions $\cos\omega t$ and $-\sin\omega t$ are real and imaginary components respectively.

Example 5.7: The real and imaginary components of the value $v = 0.1\times\sin\omega t + 0.9\times\cos\omega t$ are 0.9 and -0.1 respectively.

Example 5.8: The real and imaginary components of the value $v = 0.1\times\sin(\omega t+\pi) - 9\times\cos(\omega t+\pi)$ are 9 and 0.1 respectively, since $\sin(\omega t+\pi) = -\sin\omega t$ and $\cos(\omega t+\pi) = -\cos\omega t$.

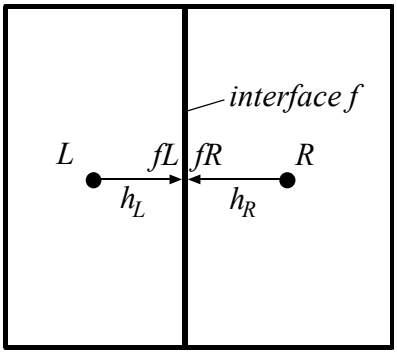
Example 5.9: In general case, the real and imaginary components of the value $v = a\sin(\omega t+\theta) + b\cos(\omega t+\theta)$ are $a\sin\theta + b\cos\theta$ and $b\sin\theta - a\cos\theta$ respectively.

In static analysis, only the real component is required.

5.2 Interface condition in E-H formulation

On a material interface, ADINA-EM applies the following interface condition in the E-H mathematical formulation. The user has no access to

alter this condition. The formulae presented here are for informational purpose.



$$\mathbf{n}\times\mathbf{E}_{fL}=\mathbf{n}\times\mathbf{E}_{fR}$$
$$\mathbf{n}\times\mathbf{H}_{fL}=\mathbf{n}\times\mathbf{H}_{fR}$$
$$\mathbf{n}\bullet\left(d\mathbf{E}\right)_{fL}=\mathbf{n}\bullet\left(d\mathbf{E}\right)_{fR}$$
$$\mathbf{n}\bullet\left(\mu\mathbf{H}\right)_{fL}=\mathbf{n}\bullet\left(\mu\mathbf{H}\right)_{fR}$$

Figure 5.3 Illustration of interface condition

where, $d=\varepsilon$ and ε^* for static and harmonic analyses respectively, the subscripts fL and fR indicate the face values of the two contiguous media abutting the interface.

Be aware of the difference of the element and nodal solution variables. The computed solution (defined at the element centers) is obtained directly from the solution algorithm, so it is accurate in terms of numerical solution. The nodal solution, on the other hand, is interpolated and output for visualization purpose only. It is not, nor affects the computed solution.

In theory, the nodal solution along material interface may be discontinuous. Since we do not introduce double nodes, the interface nodal solution represents an averaged value of variables on abutting elements. A special algorithm (described below) is used to interpolate for interface nodal solution.

The face solution is computed as, by integrating the corresponding governing equations properly,

$$\mathbf{R}_f=\mathbf{nn}\bullet\mathbf{R}_f-\mathbf{n}\times\mathbf{n}\times\mathbf{R}_f\qquad\left(\mathbf{R}=\mathbf{E},\mathbf{H}\right)$$

where

$$\mathbf{n}\bullet\mathbf{E}_f = \tfrac{1}{2}\frac{\varepsilon_L^*\mathbf{n}\bullet\mathbf{E}_L + \varepsilon_R^*\mathbf{n}\bullet\mathbf{E}_R - \Delta\left(h\rho_0\right)}{\varepsilon_L^* + \varepsilon_R^*}$$

$$\mathbf{n}\bullet\mathbf{H}_f = \tfrac{1}{2}\frac{\mu_L\mathbf{n}\bullet\mathbf{H}_L + \mu_R\mathbf{n}\bullet\mathbf{H}_R}{\mu_L + \mu_R}$$

$$\mathbf{n}\times\mathbf{n}\times\mathbf{E}_f = \left(1-ab\right)^{-1}\left(\mathbf{n}\times\mathbf{E}_{pf}^0 + a\mathbf{H}_{pf}^0\right)$$

$$\mathbf{n}\times\mathbf{n}\times\mathbf{H}_f = \left(1-ab\right)^{-1}\left(\mathbf{n}\times\mathbf{H}_{pf}^0 - b\mathbf{E}_{pf}^0\right)$$

and

$$\mathbf{E}_{pf}^0 = \tfrac{1}{2}\mathbf{n}\times\left(\mathbf{E}_L + \mathbf{E}_R\right) + \Delta\left[\tfrac{1}{2}h\left(\mathbf{K}_0 + \tfrac{1}{2}i\omega\mu\mathbf{H}\right)\right]$$

$$\mathbf{H}_{pf}^0 = \tfrac{1}{2}\mathbf{n}\times\left(\mathbf{H}_L + \mathbf{H}_R\right) - \Delta\left[\tfrac{1}{2}h\left(\mathbf{J}_0 + \tfrac{1}{2}i\omega\varepsilon^*\mathbf{E}\right)\right]$$

$$a = \tfrac{1}{4}i\omega\Delta\left(h\mu\right)$$

$$b = \tfrac{1}{4}i\omega\Delta\left(h\varepsilon^*\right)$$

$$\Delta\left(\bullet\right) = \left(\bullet\right)_R - \left(\bullet\right)_L$$

where, the subscripts f, and L and R indicate the values at the interface, and of the two contiguous media abutting the interface, and h is the distance from the element center to the interface.

5.3 Dirichlet conditions

This condition can be written as

$$v_b = v \qquad \left(v = \mathbf{E}, \mathbf{H}, \mathbf{A}, \phi\right)$$

It allows the user to directly prescribe solution variables on boundary. The variable can be a scalar as ϕ or a vector.

5.3.1 Constant condition for scalar variable

To input a constant scalar condition, specify the following parameters:

- *VAR* indicates the variable type the condition is applied for. It can be any active variable.
- *Vr, Vi* multipliers of the real and imaginary components of the variable v. The default values are all zero.

5.3.2 Condition for time-varying scalar variable

A time-varying value is obtained by multiplying a time function

$$v \leftarrow v f_n(t)$$

The following integers define the associated time functions

- *Nr, Ni* the function numbers associated with *Vr* and *Vi* respectively. The default values are all zero.

5.3.3 Condition for vector variable

A vector variable $\mathbf{v}$ is a scalar v, multiplied by a direction $\mathbf{d}$

$$\mathbf{v} = v\mathbf{d}$$

where the direction $\mathbf{d}$ is explained in the *path* section. The following parameters define the direction:

- *DirTyp* direction type
- *DX,DY,DZ* a constant direction $\mathbf{d}_0$
- *Path* path set number

Depending on the direction type, a *path* may not be required. See details in Chapter 3 on the direction definition.

5.3.4 Condition for space-varying variable

A *path* set must be defined prior to this stage. The set number will be referred here, so that the space function $\lambda(r)$ in Eq.(3.4) is defined (see details in Chapter 3 on the Space-varying function). The variable value is then multiplied by this factor

$$v \leftarrow v\lambda(r)$$

The following parameters define the shape function $\lambda(r)$

- *Path* path number
- *A1,A2,A3* constants a_1, a_2, a_3
- *EN1,EN2,EN3* constants e_1, e_2, e_3
- *Rmax* constant $R_{\max}$

See Chapter 3 for detail explanation of these constants.

5.3.5 Similar conditions

By proper defining a *path* and the constants a_i, e_i, in the definition of $\lambda(r)$ shown in Eq.(3.4), this condition can represent a large number of conditions. For example, a *balloon* condition where the solution decreases as r^{-k} can be realized by the input of a *point-path* (center point) and $a_1 = 1, e_1 = -k$.

5.4 Normal conditions

This condition is applicable to the solution variables $\mathbf{E}$, $\mathbf{H}$, $\mathbf{A}$ and ϕ. It allows the user to prescribe solution variables in the normal direction of the boundary.

5.4.1 Constant normal condition

A constant normal condition, when applied for a vector variable, is defined as

$$\mathbf{n}\bullet\mathbf{v} = v \qquad \left(\mathbf{v} = \mathbf{E}, \mathbf{H}, \mathbf{A}\right)$$

In the A-ϕ mathematical formulation, an *E-normal* condition applied to the electric potential naturally implies, for static analysis

$$-\nabla_n \phi - \frac{\partial \mathbf{A}}{\partial t} = v$$

To input this condition, specify the following parameters:

- *VAR* indicates the variable type the condition is applied for. It can be any active variable.
- *Vr, Vi* multipliers of the real and imaginary components of the variable v. The default values are all zero.

5.4.2 Time-varying normal condition

See section 5.3.2. for the input of time functions $f_n\left(t\right)$.

5.4.3 Space-varying normal condition

See section 5.3.4. for the input of space-varying function $\lambda\left(r\right)$.

5.4.4 Similar conditions

Together with the parallel condition, we can have symmetry conditions of coupled variables. For examples, a *perfect E-symmetry* condition is a combination of $\mathbf{n}\bullet\mathbf{H} = 0$ for *H* and $\mathbf{n}\times\mathbf{E} = \mathbf{0}$ for *E*, and a *perfect H-*

symmetry condition is a combination of $\mathbf{n} \cdot \mathbf{E} = 0$ for *E* and $\mathbf{n} \times \mathbf{H} = \mathbf{0}$ for *H*.

5.5 Parallel conditions

This condition is applicable to the solution variables $\mathbf{E}$, $\mathbf{H}$ and $\mathbf{A}$. It allows the user to prescribe solution variables in the tangential directions to the boundary.

5.5.1 Constant parallel condition

In a constant parallel condition, a boundary variable "parallel" component and an associated direction are specified

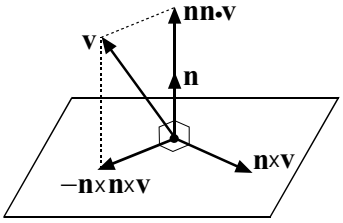
$$\mathbf{n} \times \mathbf{v} = v\mathbf{d} \qquad (\mathbf{v} = \mathbf{E}, \mathbf{H}, \mathbf{A})$$

To input this condition, specify the following parameters:

- *VAR* indicates the variable the condition is applied for. It can be any active variable.
- *Vr, Vi* multipliers of the real and imaginary components of the variable v. The default values are all zero.
- *DirTyp* direction type
- *DX,DY,DZ* a constant direction $\mathbf{d}_0$
- *Path* path set number

Depending on the direction type, a *path* may not be required. See details in Chapter 3 on the direction definition.

Note that the "parallel" component of $\mathbf{v}$ is not the tangential component of $\mathbf{v}$, the latter one being precisely defined as $-\mathbf{n} \times (\mathbf{n} \times \mathbf{v})$.



The figure (left) illustrates the mathematical and geometric ingredients associated with parallel boundary condition.

5.5.2 Time-varying parallel condition

See section 5.3.2. for the input of time functions $f_n(t)$.

5.5.3 Space-varying normal condition

See section 5.3.4. for the input of space-varying function $\lambda(r)$.

5.5.4 Similar conditions

Zero parallel *E/H* are also called perfect *E/H* plane condition.

Together with the parallel condition, we can have symmetry conditions of coupled variables. For examples, a *perfect E-symmetry* condition is a combination of $\mathbf{n}\bullet\mathbf{H} = 0$ for *H* and $\mathbf{n}\times\mathbf{E} = \mathbf{0}$ for *E*, and a *perfect H-symmetry* condition is a combination of $\mathbf{n}\bullet\mathbf{E} = 0$ for *E* and $\mathbf{n}\times\mathbf{H} = \mathbf{0}$ for *H*.

5.6 Natural conditions

This condition is applicable to the solution variables $\mathbf{E}$ and $\mathbf{H}$. In this condition, ADINA-EM solves the related first order equation system on the boundary. Specifically, the *E-Natural* condition is equivalent to Eqs. (2.1-2.2) and the *H-Natural* condition is equivalent to Eqs. (2.3-2.4).

No parameters are required in this condition.

The *Natural* condition is usually used together with the *Dirichlet* or *Parallel* condition in harmonic analysis. In static analysis, the *H-Natural* condition is usually used together with the *E-Dirichlet* or *E-Parallel* condition to a conductor boundary ($\sigma > 0$).

5.7 Impedance conditions

5.7.1 Condition definition

This condition is applicable to the solution variables $\mathbf{E}$, $\mathbf{H}$ and $\mathbf{A}$ in harmonic analysis. It allows the user to prescribe a known impedance to the solution variable.

In the *E-H* mathematical formulation, this condition is expressed as

$$\mathbf{n} \times \mathbf{E} = -Z_s \mathbf{n} \times \left(\mathbf{n} \times \mathbf{H} \right) \tag{5.1}$$

where Z_s is the surface impedance (a complex number). Eq.(5.1) is equivalent to (obtained by applying $Z_s^{-1}\mathbf{n}\times$ to it)

$$\mathbf{n} \times \mathbf{H} = Z_s^{-1} \mathbf{n} \times \left(\mathbf{n} \times \mathbf{E} \right) \tag{5.2}$$

Eqs.(5.1) and (5.2) are used respectively for *H*- and *E*-boundary equations.

In terms of physics, the impedance condition with the same Z_s is used for both *E* and *H*. In ADINA-EM, each boundary condition set is only for one variable. Therefore, if both *E* and *H* are active, the same impedance boundary conditions (meaning the same Z_s) must be applied twice, once for *E* and once for *H*.

By substituting Eq.(2.11) into Eq.(5.1), the impedance condition for the magnetic potential $\mathbf{A}$ is obtained

$$\mathbf{n}\times\left(\nabla\times\mathbf{A}\right)=-\mu Z_s^{-1}\mathbf{n}\times\left(\mathbf{n}\times\left(\nabla\phi+i\omega\mathbf{A}\right)\right)$$

The input parameters are

- *VAR* indicates the variable the condition is applied for. It can be *E*, *H* or *A*.
- *Vr, Vi* multipliers of the real and imaginary components of the impedance Z_s. Non-zero value must be input.
- *Nr, Ni* the function numbers associated with *Vr* and *Vi* respectively. The default values are all zero.

5.7.2 Similar conditions

The impedance condition originated from the assumption that the tangential component of the boundary solution variable decays as

$$\mathbf{v}=\mathbf{v}_B e^{-(1+i)\frac{\mathbf{x}\cdot\mathbf{n}}{\delta}} \qquad \left(\mathbf{v}=\mathbf{n}\times\left(\mathbf{n}\times\left(\mathbf{E},\mathbf{H},\mathbf{A}\right)\right)\right)$$

where δ is skin depth. Therefore, it also covers some other practical conditions.

One of them is the *Finite conductivity* condition that is used to model imperfect conductors. In this case, $\delta=\left(|\omega|\mu\sigma\right)^{-\frac{1}{2}}$ and the impedance is given by $Z_s=(1+i)(\delta\sigma)^{-1}=(1+i)\left(|\omega|\mu/\sigma\right)^{\frac{1}{2}}$.

Another one is the Lumped *RLC* boundary condition that is used to model a lumped resistor, inductor etc. The user should calculate the "impedance" based on the values of *R*, *L* and *C*. For a series device,

$$Z_s=R\left[1+i\left(b_L-b_C\right)\right]$$

and for a parallel device,

$$Z_s = R\frac{1+i\left(b_L^{-1}-b_C^{-1}\right)}{1+\left(b_L^{-1}-b_C^{-1}\right)^2}$$

where

$$b_L = \omega L/R$$
$$b_C = 1/(\omega CR)$$

Chapter 6 Solution of linear equation

6.1 Introduction

The linear electromagnetic equation system can be written as

$$AX = B \tag{6.1}$$

where x consists of all solution variables at the element centers, B consists of terms from sources and boundary conditions, and A is the coefficient matrix.

In order to obtain the solution efficiently, we apply to system Eq.(6.1), a general pre-conditioner A_* that is close to A but occupies much less memory. We then use an iterative solver for the modified equation. The final algorithm becomes

$$R_k = A_*^{-1}\left(B - AX_k\right) \quad \left(k = 0,1,2,...,k_{\max}\right) \tag{6.2}$$

where x_k and r_k are the approximate solution and residual, respectively, after the *k*-th iteration, and $k_{\max}$ is the maximum number of iterations allowed. The solution is obtained if $r_k \to 0$. The next sections describe on how the iteration algorithm is controlled in ADINA-EM.

6.2 Input of control parameters

In order to properly control the solution convergence, the following global control parameters must be input

- *Solver* the solver used for inversing the pre-conditioning matrix A_*. Either SPARSE solver or AMG solver can be selected. The default is SPARSE.
- *Tolerance* tolerance ε, used for control solution iteration. The default value is 10^{-6}.
- *MaxIte* allowed maximum iteration number $k_{\max}$.
- *Vcycle* maximum number of V-cycles used in AMG solver. The AMG solver returns A_*^{-1} either the iteration converged or maximum number of V-cycles has been reached.

The functioning of the tolerance ε can be roughly explained as the tolerance in convergence criterion

$$\frac{\|B - AX_k\|}{\|B\|} \leq \varepsilon$$

However, we have used some other criteria too, with additional tolerances that are all proportional to ε. The detail is given in the next section.

6.3 Convergence criteria

The convergence criteria are explained in a few steps.

Firstly, we define the criteria bounds (the superscript $f = e/v$, standing for equation/variable respectively),

minimum relative tolerance	$\varepsilon^f_{\min} = \varepsilon$
maximum relative tolerance	$\varepsilon^f_{\max} = \min\left\{10^2 \varepsilon^f_{\min}, 10^{-1}\right\}$
minimum absolute tolerance	$\mathcal{E}^f_{\min} = 10^{-16}$
maximum absolute tolerance	$\mathcal{E}^f_{\max} = \min\left\{\mathcal{E}^f_{\min} \frac{\varepsilon^f_{\max}}{\varepsilon^f_{\min}}, 10^{-8}\right\}$

Secondly, we define the residuals and scales for each variable (the subscript $g = e/m$, standing for electric/magnetic variables respectively),

absolute *e*-residue	$E_{kg} = \left\| R_{kg} \right\|_1$
absolute *v*-residue	$V_{kg} = \left\| X_{kg} - X_{(k-1)g} \right\|_1$
e-scale	$\bar{E}_{kg} = \max\left\{ \max_{i \le k} \left\{ E_{ig} \right\}, \mathcal{E}^e_{\max} \right\}$
v-scale	$\bar{V}_{kg} = \max\left\{ \max_{i \le k} \left\{ \left\| X_{ig} \right\|_1 \right\}, \mathcal{E}^v_{\max} \right\}$

The overall residuals are defined as

absolute *e*-residue	$E_k = \max\left\{ \lambda_e E_{ke}, \lambda_m E_{km} \right\}$
absolute *v*-residue	$V_k = \max\left\{ \lambda_e V_{ke}, \lambda_m V_{km} \right\}$
relative *e*-residue	$e_k = \max\left\{ E_{ke} / \bar{E}_{ke}, E_{km} / \bar{E}_{km} \right\}$
relative *v*-residue	$v_k = \max\left\{ V_{ke} / \bar{V}_{ke}, V_{km} / \bar{V}_{km} \right\}$
relative *e*-residue increment	$\Delta e_k = \max_{0 \le i < 4} \left\{ \left| e_{k-i} - e_{k-i-1} \right| \right\}$
relative *v*-residue increment	$\Delta v_k = \max_{0 \le i < 4} \left\{ \left| v_{k-i} - v_{k-i-1} \right| \right\}$

in which, the constant parameters λ_e and λ_m are automatically chosen parameters based on the model, so that the terms are unit-consistent.

Thirdly, we define the criteria as follows

$$[e_k]: \quad e_k \le \varepsilon^e_{\min}$$
$$[v_k]: \quad v_k \le \varepsilon^v_{\min}$$
$$[E_k]: \quad \left(E_k \le \mathcal{E}^e_{\min}\right) \& \left(e_k \le \varepsilon^e_{\max}\right)$$
$$[V_k]: \quad \left(V_k \le \mathcal{E}^v_{\min}\right) \& \left(v_k \le \varepsilon^v_{\max}\right)$$
$$[e_{k\min}]: e_k \le 0.01\varepsilon^e_{\min}$$
$$[v_{k\min}]: v_k \le 0.01\varepsilon^v_{\min}$$
$$[e_{k\max}]: e_k \le \max\left\{\varepsilon^e_{\max}, 10^{-4}\right\}$$
$$[v_{k\max}]: v_k \le \max\left\{\varepsilon^v_{\max}, 10^{-4}\right\}$$
$$[\Delta e_k]: \left(\Delta e_k \le \varepsilon^e_{\min}\right) \& \left(e_k \le 10^{-2}\right)$$
$$[\Delta v_k]: \left(\Delta v_k \le \varepsilon^e_{\min}\right) \& \left(v_k \le 10^{-2}\right)$$

Then, with the formulae defined, the linear equation system Eq.(6.2) is considered converged if any one of the following conditions is satisfied

(1) both criteria $[e_k]$ and $[v_k]$ are satisfied
(2) both criteria $[E_k]$ and $[V_k]$ are satisfied
(3) either criterion $[e_0]$ or $[E_0]$ is satisfied.
(4) either criterion $[v_1]$ or $[V_1]$ is satisfied.
(5) both criteria $[e_{k\min}]$ and $[v_{k\max}]$ are satisfied
(6) both criteria $[v_{k\min}]$ and $[e_{k\max}]$ are satisfied
(7) both criteria $[\Delta e_k]$ and $[\Delta v_k]$ are satisfied.

The iteration history is printed in the format

EM-IT ek vk dek dvk Eke Vke EBe VBe Ekm Vkm EBm VBm

where

EM-IT	$= k$, iteration number
ek, vk	$= e_k, v_k$
dek, dvk	$= \Delta e_k, \Delta v_k$
Eke, Vke, Ebe, VBe	$= E_{ke}, V_{ke}, \overline{E}_{ke}, \overline{V}_{ke}$
Ekm, Vkm, EBm, VBm	$= E_{km}, V_{km}, \overline{E}_{km}, \overline{V}_{km}$

6.4 Selection of solver

The most stable solver is sparse matrix solver. However, this solver is not suitable for very large problems that require large amounts of memory and CPU.

The second choice is to select AMG solver with default setting of V-cycle=1. This solver uses much less memory and converges faster for stable problems. It may have difficulty in convergence or even diverge for high-frequency problems or the problems with poor meshes.

Note that the high-frequency problems are characterized by the nondimensional number $\mu\varepsilon\left(\omega\Delta x\right)^2$. This number should always be less than 1. When it is close to 1, numerical instability may occur. In this case, the mesh should be refined.

An improvement to the default AMG solver is to increase the number of V-cycles. We refer this cycle as the inner iteration. A large V-cycle number (say 999) allows the inner iteration converge so that the outer iteration may behave like that using the sparse solver. However, attention should be made because too much inner iteration may slow down the overall convergence too.

This page is intentionally left blank

Chapter 7 EM solutions coupled with CFD solutions

In ADINA-CFD+EM, electromagnetic solutions can be coupled with CFD solutions.

The coupling occurs within the element groups for which both fluid and electromagnetic variables are active. In these groups, the Maxwell stress and Joule heating rate are calculated and added to the fluid stress and the energy source, respectively, in the CFD solution procedure.

In the EM-CFD coupled model, FCBI-C elements must be used in the ADINA-CFD models.

The EM solution can be applied to both transient and steady-state CFD models, irrespective of the analysis type in the electromagnetic model, as decided by the user. For example, if the EM model is static analysis, then the electromagnetic force and heat source are the same at every time step in the CFD model (if the sources and boundary conditions are constant). But one should be careful if the CFD model is steady and the EM model is harmonic. Usually the *averaged* EM force should be selected, unless the force at the phase represented at that particular time is the EM force used for the CFD solution.

7.1 Lorentz force and Maxwell stress

The Lorentz body force $\mathbf{f}_b^{em}$ per unit volume can be expressed in terms of the Maxwell stress

$$\mathbf{f}_b^{em} = \nabla \cdot \mathbf{T}^{em} + \mathbf{T}_b^{em}$$

where, the Maxwell stress $\mathbf{T}^{em}$ and $\mathbf{T}_b^{em}$ are

$$\begin{aligned}
\mathbf{T}^{em} &= \mathbf{T}^{e} + \mathbf{T}^{m} \\
\mathbf{T}^{e} &= \mathbf{DE} - \tfrac{1}{2}\left(\mathbf{D}\bullet\mathbf{E}\right)\mathbf{I} \\
\mathbf{T}^{m} &= \mathbf{BH} - \tfrac{1}{2}\left(\mathbf{B}\bullet\mathbf{H}\right)\mathbf{I} \\
\mathbf{T}_b^{em} &= -\frac{\partial \mathbf{D}\times\mathbf{B}}{\partial t}
\end{aligned} \tag{7.1}$$

In the *E-H* formulation, the variables are directly applicable in Eq.(7.1). In the A-ϕ formulation, the field intensities are calculated using Eq.(2.10) and then used in Eq.(7.1).

In the element groups with only active electric and CFD variables, $\mathbf{T}^m$ and $\mathbf{T}_b^{em}$ are omitted. Similarly, if only magnetic and CFD variables are coupled, $\mathbf{T}^e$ and $\mathbf{T}_b^{em}$ are omitted. In ADINA-CFD solid element groups, since no fluid velocity and pressure variables are active, the Maxwell stress is omitted.

7.2 Joule heating rate

The heat source generated by electromagnetic variables is calculated as, according to the Joule's law

$$q_b^{em} = \sigma^{-1}\mathbf{J}_c\bullet\mathbf{J}_c \tag{7.2}$$

In element groups with electric variable inactive, $\mathbf{J} = \mathbf{J}_0$ is used in Eq.(7.2).

7.3 Direct-computed and averaged stress and heat source

In harmonic analysis, the Maxwell stress and Joule heating rate are time varying, so they can be directly computed at any specified time. According to the definition in Eq.(1.1),

$$\mathbf{T}^e = \mathrm{Re}\left(\mathbf{D}e^{i\omega t}\right)\mathrm{Re}\left(\mathbf{E}e^{i\omega t}\right) - \tfrac{1}{2}\mathrm{Re}\left(\mathbf{D}e^{i\omega t}\right)\bullet\mathrm{Re}\left(\mathbf{E}e^{i\omega t}\right)\mathbf{I}$$
$$\mathbf{T}^m = \mathrm{Re}\left(\mathbf{B}e^{i\omega t}\right)\mathrm{Re}\left(\mathbf{H}e^{i\omega t}\right) - \tfrac{1}{2}\mathrm{Re}\left(\mathbf{B}e^{i\omega t}\right)\bullet\mathrm{Re}\left(\mathbf{H}e^{i\omega t}\right)\mathbf{I}$$
$$\mathbf{T}_b^{em} = \omega\left[\mathrm{Re}\left(\mathbf{D}e^{i\omega t}\right)\times\mathrm{Im}\left(\mathbf{B}e^{i\omega t}\right) + \mathrm{Im}\left(\mathbf{D}e^{i\omega t}\right)\times\mathrm{Re}\left(\mathbf{B}e^{i\omega t}\right)\right]$$

and

$$q_b^{em} = \sigma^{-1}\,\mathrm{Re}\left(\mathbf{J}_c e^{i\omega t}\right)\bullet\mathrm{Re}\left(\mathbf{J}_c e^{i\omega t}\right)$$

On the other hand, for steady or even transient CFD problems, the time-averaged values (over one period) can be used for many problems

$$\overline{f} = \tfrac{1}{2\pi}\int_0^{2\pi} f d\left(\omega t\right)$$

By use of the time-averaging method, Eqs.(7.1) and (7.2) become, respectively,

$$\overline{\mathbf{T}}^{em} = \overline{\mathbf{T}}^e + \overline{\mathbf{T}}^m$$
$$\overline{\mathbf{T}}^e = \tfrac{1}{2}\left(\mathbf{D}_r\mathbf{E}_r + \mathbf{D}_i\mathbf{E}_i\right) - \tfrac{1}{4}\left(\mathbf{D}_r\bullet\mathbf{E}_r + \mathbf{D}_i\bullet\mathbf{E}_i\right)\mathbf{I}$$
$$\overline{\mathbf{T}}^m = \tfrac{1}{2}\left(\mathbf{B}_r\mathbf{H}_r + \mathbf{B}_i\mathbf{H}_i\right) - \tfrac{1}{4}\left(\mathbf{B}_r\bullet\mathbf{H}_r + \mathbf{B}_i\bullet\mathbf{H}_i\right)\mathbf{I}$$
$$\overline{\mathbf{T}}_b^{em} = \mathbf{0}$$

and

$$\overline{q}_b^{em} = \tfrac{1}{2}\sigma^{-1}\left(\mathbf{J}_{cr}\bullet\mathbf{J}_{cr} + \mathbf{J}_{ci}\bullet\mathbf{J}_{ci}\right)$$

7.4 Input of coupling parameters

The coupled model must be analyzed using ADINA-CFD+EM that contains both CFD and EM programs.

To activate the coupling algorithm, the user must firstly turn on the EM option by selecting "EM-MODEL=EM" in the master command or its corresponding dialog box.

The user then select the methods for calculating the Maxwell stress and Joule heating rate, by input of the following parameters

- *Force* indicates how the Maxwell stress is computed. It can be no force, averaged or directed. The default is no force.
- *Energy* indicates how the Joule heating rate is computed. It can be no source, averaged or directed. The default is no source.

We note that, although unusual, different methods can be used for the stress and heat source computations.

Finally, the user has to set both the fluid and electromagnetic variables active in the coupled element groups. Note that pure CFD or EM element groups are allowed.