Tutorial of Electro Magnetica Numerica 1

The First Edition

Hiroshi ABE

Three Wells

Contents

Chapter 1

Introduction

The present document describes how to use the software Electro Magnetica Numerica (hereafter EM) using several examples.

1.1   What's EM

EM is a time-domain simulation software based on Finite Difference-Time Domain method using unstructured grid, named adaptive Cartesian method, which has been developed in Computational Fluid Dynamics domain with a great success.

The characteristics are,

1. the grid mesh of EM is UNSTRUCTURED grids refined only on the necessary area so that the computation costs, such as memory and the amounts of computation, can be small comparing with the conventional FDTD on a structured grids. This means you can simulate SERVER compatible simulation using your desktop PC.

2. the scheme is based on normal FDTD scheme, Leap-Frog method, so that many reliable algorithms for FDTD are available and applicable to EM.

3. the results are visualized in REAL-TIME manner using brand-new visualization tools so that the user can see what is going on in the simulation world.

1.2   Simulation Flow

Because of FDTD, EM generates the mesh grids first of all. Then you integrate the time-marching solution. EM is a Real-Time Visualizing application so the calculating results are visualized.

The unstructured grids are automatically generated adaptively with respect to the geometries of model objects. You can control the fineness of the generating cells by the level of the blocked cell.

Chapter 2

Scattering by a PEC sphere

In this chapter, a simulation model will be described showing procedure to model a scattering simulation by a perfect electric conducting sphere.
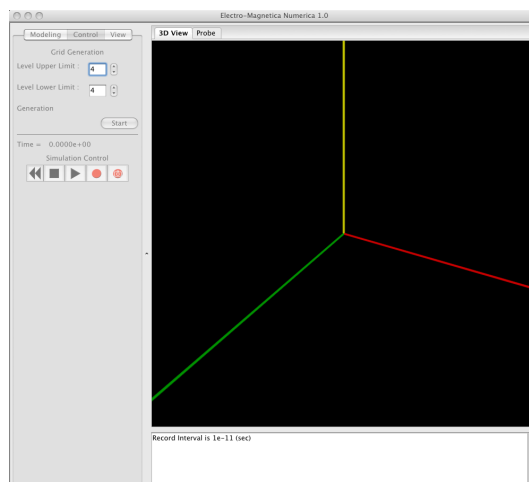

Figure 2.1: Initial View of EM.

Figure 2.1 shows the default view of EM. The graphical user interface (GUI) is divided into three as is shown in the Fig.2.1.

The left side of EM view is for simulation settings. The right side is for the 3D visualization and 2D visualization to show the results of your simulation. And the lower of the right is for logging window to show the response from EM.

At the left side panel, there are Modeling, Control and View tab panels, they have their own setting panels inside. The details of these tab panels are described in the following sections with explaining how and what to be set for the example.
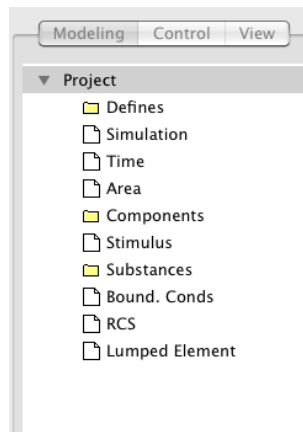
2.1 Work at Modeling tab

Figure 2.2: Modeling setting panel.

You can specify your model here. To set your model you select tree nodes then enter your parameters. The tree nodes view is shown in Fig.2.2.

First of all, make a PEC sphere. You can define PEC model with **Components** node in the tree menu. Right click the node then select **New** from the pop-up menu. Then you have **Components** dialog (Fig.2.3). Confirm the default values and then click **OK** button to close the dialog. Where the unit of the length is meter. Click the **3D View** window to activate the view area then you can see a red sphere in the view. This is your PEC sphere.

Next, in order to set the total simulation time, right-click the **Time** node in the tree and select **Edit** to appear a dialog (Fig.2.4). Enter numeric value (1.E-8) as shown in Fig.2.4 and click **OK** button. The unit of time is *second*.

Then plane wave is to be set as the incident wave of stimulation. Right click **Stimulus** node, which is for the signal model, then you can see a dialog as Fig.2.5. You can specify the spatial model and the temporal model of stimulation independently.

You select **Plane Wave** from the spatial model and select **Impulsive** from temporal model for the example. The numeric value is left to the default values. The procedures means you define a plane wave which incidents from Z negative side and which has y-polarized electric field. The amplitude of the wave varies as an Gaussian pulse as well.

To see the response results of the example, radar cross section (RCS) is to be set to see the far-field by scattering. By the restriction of the algorithm adopted by EM, you must specify the frequency values for RCS before the simulation starts.

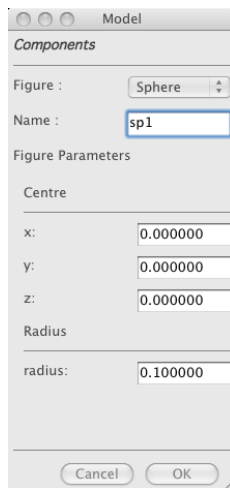Right-click **RCS** node then you get a dialog as Fig.2.6.

Figure 2.3: Setting panel for components.

You specify the frequency values you want to obtain in the list. Enter numeric value (1E8), which means 0.1GHz. The unit is Hz here. When you finish click **OK** to close.

The other nodes uses the default values if you not touch. For instance, the boundary conditions are predefined to absorbing boundary condition (CPML) as default.

This is all you specify with the **Model** tab.

2.2   Work at Control tab

Click the **Control** tab then you see Fig.2.7. In this panel, you can control the mesh generation. Also you can start the simulation and stop, etc.

You specify **Upper Limit** value to 3 and **Lower Limit** to 2 to control the mesh fineness. Click **Start** button to generate mesh grids. Wait until the **Start** button back click the 3D View window to redraw then you can see the generated blocked cells (Fig.2.8). The bounding boxes are corresponding to the blocked cells to be generated. Each blocked cell contains 8 mesh cells in x,y and z directions. That means one blocked cell contains 64 mesh cells inside.

The mesh cell control parameter **Level** is corresponding to the nested level of the octet-tree of the unstructured grid. The larger the level value is, the finer the blocked cells are generated. If the level is 6 then it means the corresponding Yee's cell is 256 mesh cells in one direction.

At the first, EM generates blocked cell at the **Lower Limit**. Then next EM refines the blocked cells up to **Upper Limit** to satisfy a certain threshold with regard to the components.
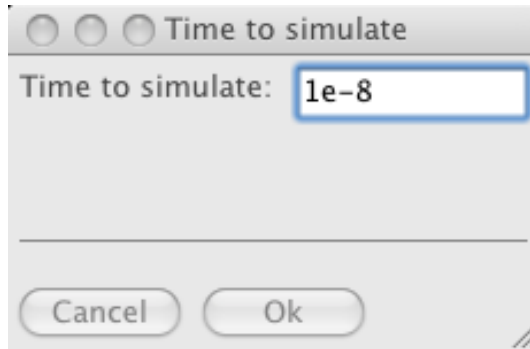
Figure 2.4: Time Setting.

Name	Value	Description
Vector	$\vec{E}, \vec{B}$	3D Arrow
Isosurface	$|E|, |B|$	Isosurface of values
Contour	$|E|, |B|$	Contour on a Plane
RCS	RCS	RCS value at a frequency
Probe	E,B	Time Series of values at points.

Table 2.1: Visualizable data.

You finish all the data necessary for simulation.

Before click the **Start** button in **Simulation Control**, you specify several settings for visualization.

2.3   Work at View tab

Click **View** tab then a dialog appears (Fig.2.9). You can establish the visualization in **3D View**. In the Fig.2.9 **Vector** is visualized as a default setting.

The **Vector** visualizes a certain vector filed on a certain plane. To establish the plane you click 3D View to activate and enter **i** key to appear an interactor as is shown in Fig.2.10.

You can change the plane by dragging the **interactor** as you want, however, you leave it as default by entering **i** again. The **interactor** disappears.

Possible physical values to visualize are listed in Tab.2.1.

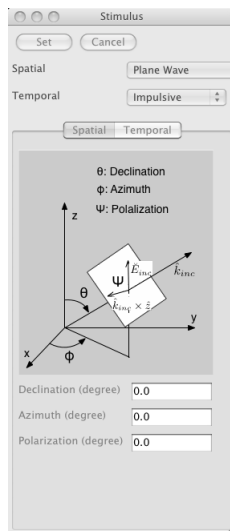Now all the setting for your simulation are done.

Figure 2.5: Stimulus Setting.

2.4 Starting your simulation

Click **Control** tab to see the display as Fig.2.7. Click the right-directed triangle button to start your simulation. If you want to stop the simulation, click a solid square button to stop. (Note that the response time depends upon the specification of your computer) If you want to resume click the triangle button to start again.

You can see the time-varying electric field in **3D View**. Figure2.11 is a snap shot.

After the scattering wave is disappeared from the view, you can see RCS view as is shown in Fig:2.12 by selecting **RCS** from a selection menu in the **View** tab window panel. The figure shows the force of RCS vector as the distance from the origin of the coordinates.
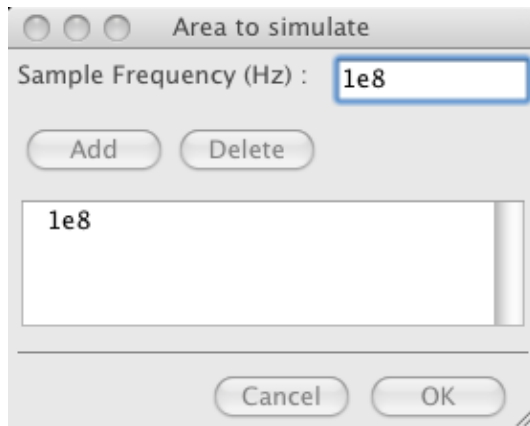
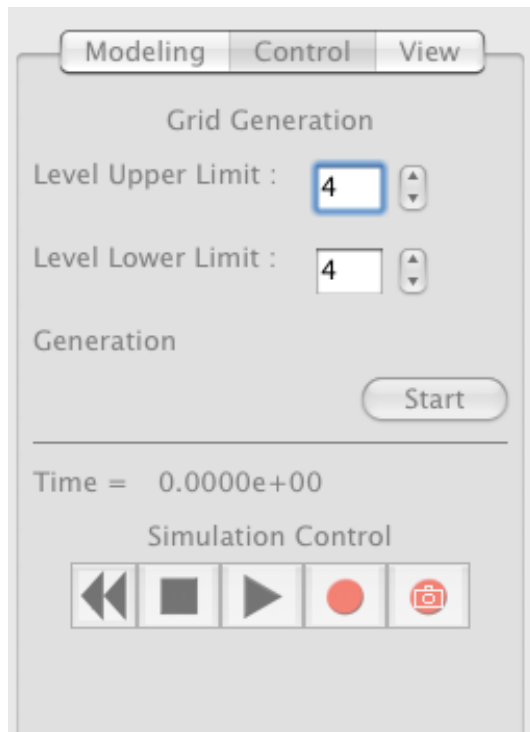Figure 2.6: Setting RCS frequency.

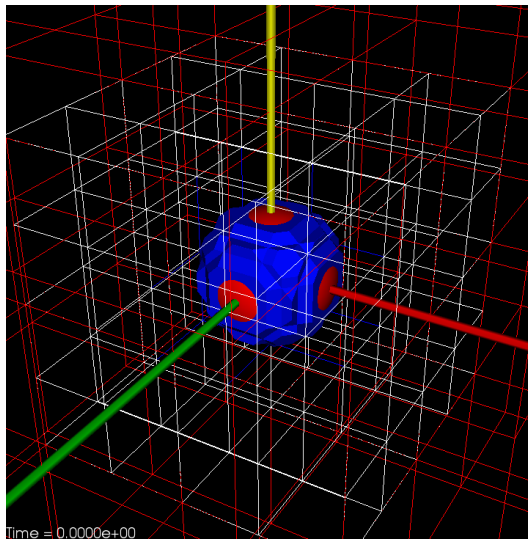Figure 2.7: Control setting panel.
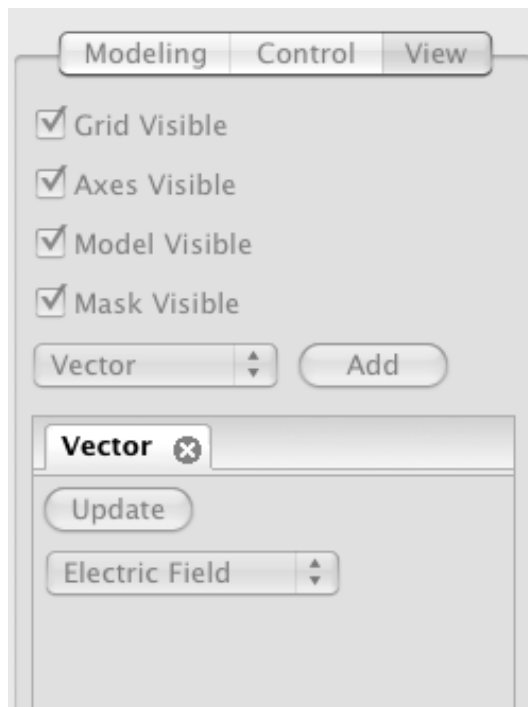
Figure 2.8: Generated blocked cells.
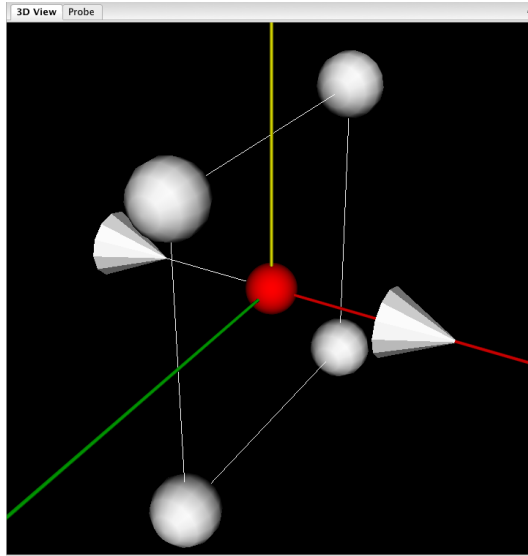

Figure 2.9: View setting panel.
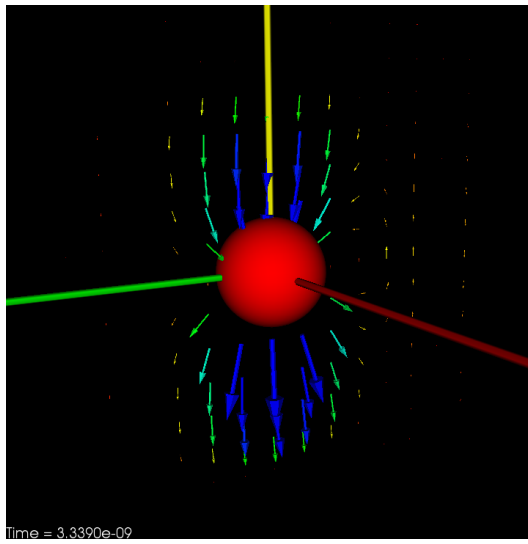
10

Figure 2.10: Interactor.

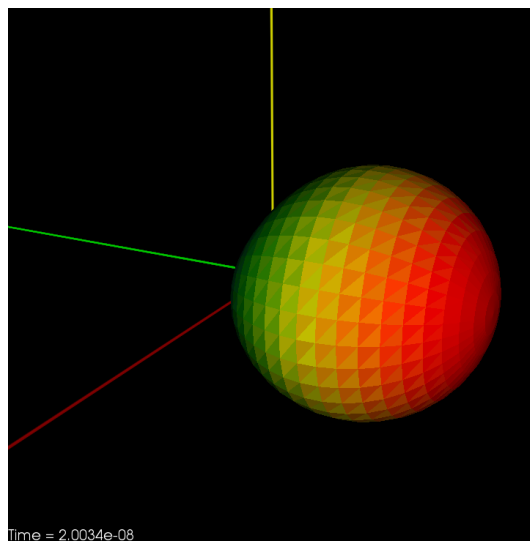Figure 2.11: Electric field distribution around the PEC sphere.

Figure 2.12: Scattering by a PEC sphere at 0.1GHz.

Chapter 3

Emit from dipole antenna

The purpose of the chapter is to obtain a radiation pattern from a dipole antenna, which consists of two elements and a lumped element between the elements.

3.1 Modeling tab

You specify **Time、 Components、 Stimulus** and **RCS** here. You set **Time** node to the same as the previous example. The **Bound. Cond.** node is set to the default, **CPML**.

Figure 3.1: Cylinder Component of dipole element.

Right click **Component** node and select **New** from the pop-up menu. Then **Component** dialog appears. Select **Cylinder** from **Figure** selection menu and set parameters same as Fig.3.1. Confirm the values and then click **OK** button to close the dialog.

Just the same way, select **Cylinder** and set parameters to the same as Fig.3.2.

Figure 3.2: The another cylinder component of dipole element.

Next, right-click **Stimulus** node to appear the setting dialog as Fig.3.3. **Temporal** is set to **Impulsive** and leave the data as they are.

As for **Spatial**, you select **Lumped Element**. Note that the lumped element model has no spatial volume so that you must specify wire to connect the lumped element with the dipole elements. **Wiring Begin/End** are the both ends of the wire. After setting the parameters as same as Fig.3.3, click **Set** button to close the dialog.

3.2 Control tab

Just the same way as the previous example, set the Levels to 4-3, click **Grid Generation** button to generate the mesh cells. After a while, the mesh cells are generated automatically to fit the dipole antenna.

Click 3D View panel to activate and enter **i** to show **Interactor**. After setting the vector plane, enter **i** to disappear **Interactor**. When you are ready click the start button to start your simulation.

3.3 View tab

EM stops automatically when the simulation time reaches the total time which is set in **Time** node in **Modeling** tab.

Figure 3.4 shows a snap shot of the electric field distribution on a plane at a certain time.

The default **Interactor** indicates $x = 0$ plane so all the vector field is directed to x direction. Enter **i** key to display **Interactor**. You can change the visualizing plane as you like by dragging **Interactor**. This allows you to see the 3D distribution of the vector field.
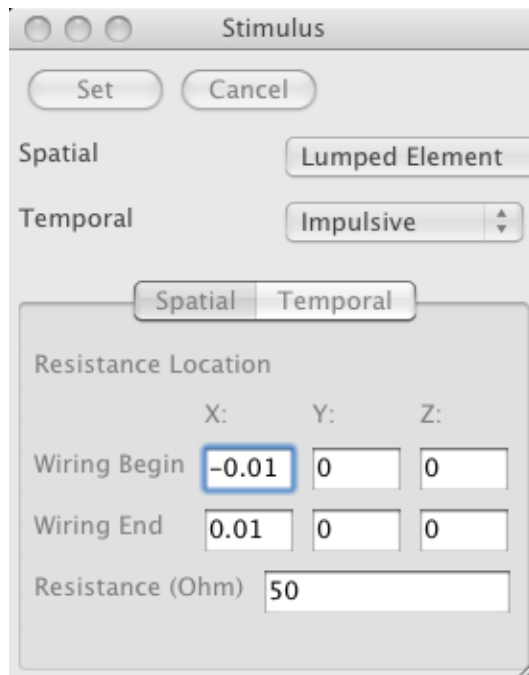
Figure 3.3: Stimulus model of a lumped element.

Next, click the close box of **Vector** tab in **View** panel to eliminate the vector from **3D View** panel. Select **RCS** from the selection menu to appear a **RCS** tab. Click **Update** button to display RCS pattern of your dipole antenna as Fig.3.5
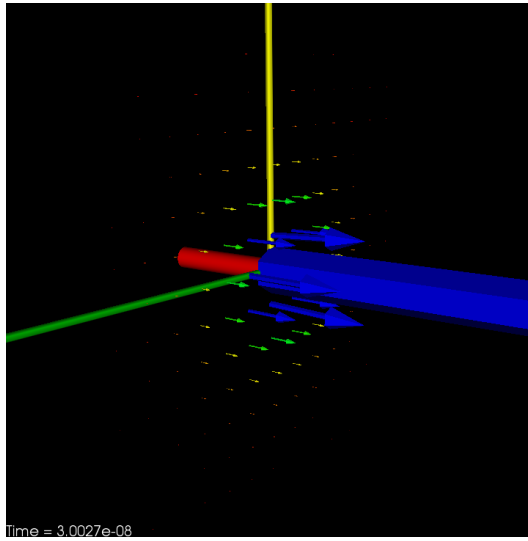
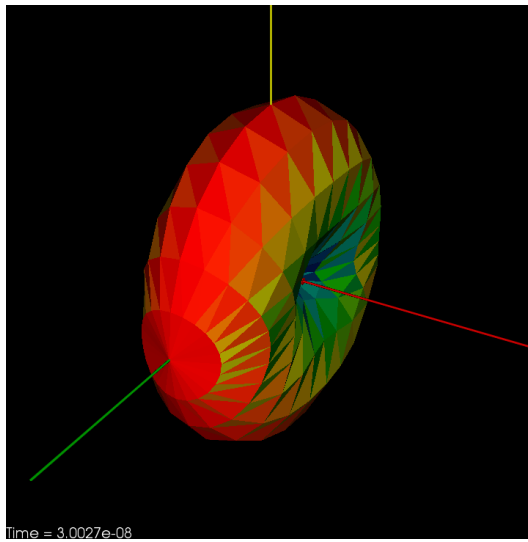Figure 3.4: Electric field distribution around the dipole element.

Figure 3.5: RCS pattern emitted from a dipole antenna.

Chapter 4

Scattering by a 3D DXF model

In this chapter, you learn how to load DXF data file, which is one of the popular 3D format, to get RCS data.

EM can load DXF/STL 3D formatted file which consists of triangulated facets. A case of DXF file is explained in the present chapter. You can load STL file in the same way.

You specify the parameters of nodes in **Modeling** tab to values as listed in the following.

Time 3.0e-8

Stimulus Plane Wave / Impulsive

RCS 1e8

Select **Shape** from **Figure** selection menu in **Components** node to get a display as Fig.4.1.

Click **Load Shape** button to load the data file. Just after clicking button a file selection dialog appears to specify the DXF file. Select **penguin.dxf** file in the Tutorial folder. The actual penguin.dxf file's scale is too big so set **Scale** to **0.01** (Fig.4.1.

If the file is loaded properly, you can see the shape in **3D View** as Fig.4.2.

Next, click **Control** tab and set the Level to 3-2. And click **Generate** button to get mesh cells. Finally, click the start button to start your simulation.

Figure 4.3 show a snap shot of the electric field around the shape object. Incident plane wave propagates from z negative side then scattered by the shape. Figure 4.4 shows the RCS data by adding **RCS** in **View** tab.

You can save the RCS data by clicking **Save File** located in the lower of the setting panel.

For you information, only the **Axes, Mask** check button are selected to get a view of Fig.4.5.

The mask in Figure4.5 shows the **Shape** data actually resolved by the mesh cells. If you feel the resolution is not enough, then you increase **Upper Level**. Although the computation time becomes longer, the accuracy of the model becomes higher.
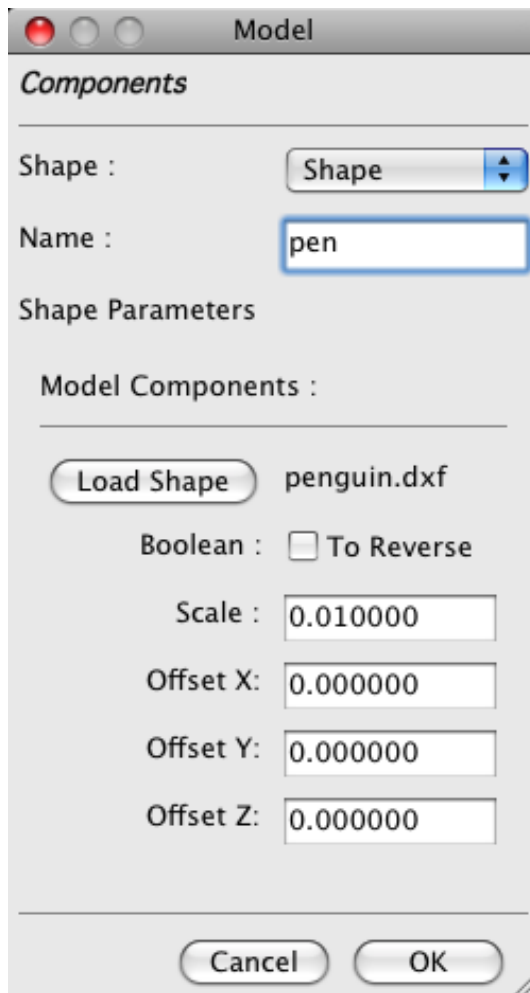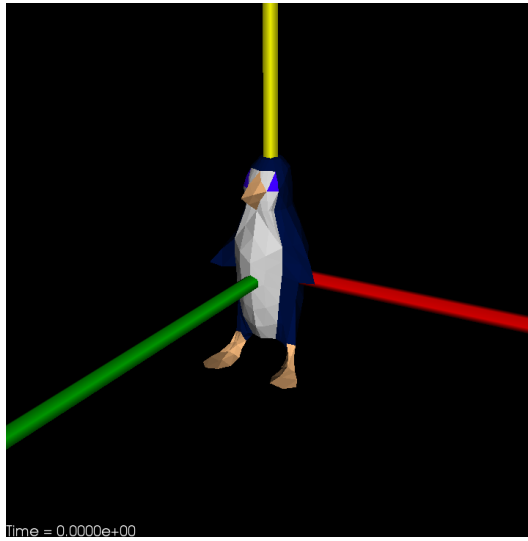
Figure 4.1:

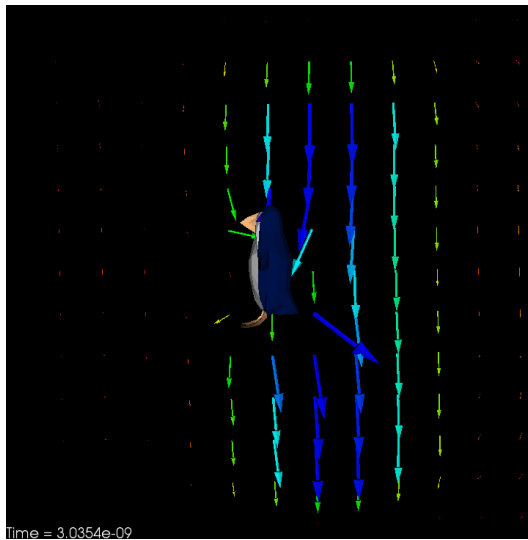Figure 4.2: The loaded DXF file, which is provided with wxWidgets library.

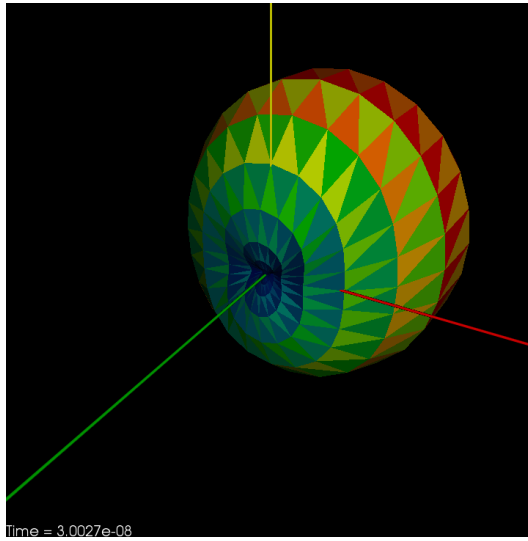Figure 4.3: Snapshot of the electric field at three nanosecond elapsed.
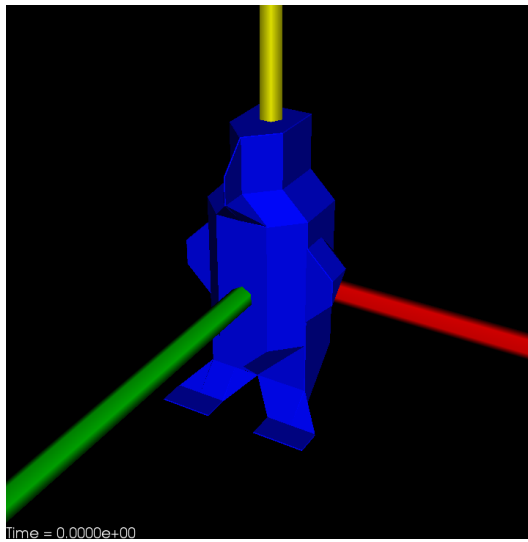
Figure 4.4: The distribution of RCS at 0.1GHz.

Figure 4.5: The actual shape recognized in the simulator.

Chapter 5

Scattering by a Dielectric Sphere

A scattering by a dielectric sphere is described in the present chapter.

5.1 Preference Dialog

You can build up dielectric objects composed of predefined primitive figures. You must specify the electromagnetic constants of the materials for the objects. The electromagnetic values, such as permittivity, of materials can be set in the **Preference** dialog.
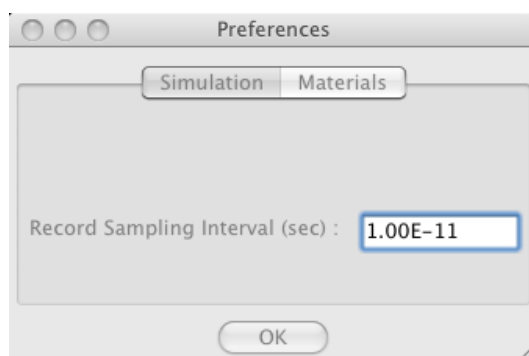

Figure 5.1: Preferences dialog.

Command/Alt-, or select **Preference** from application menu bar to appear a preference dialog as is shown in Fig.5.1 Click **Materials** tab to flip the window to see a view as is shown in Fig.5.2
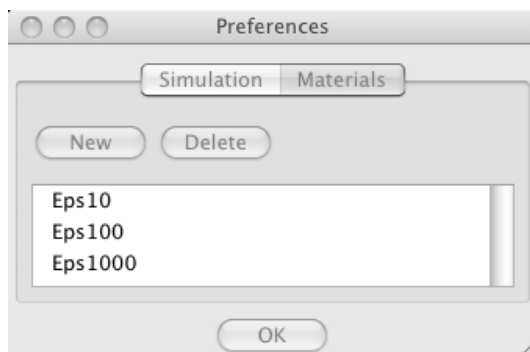Click **New** to register new material's parameters and its name. A dialog

Figure 5.2: Materials tab view.

appears as is shown in Fig.5.3. You can specify a material with a certain permittivity, permeability and conductivity. Each value allows to be different with respect to each direction.

The values you specify here are:

Material Name Eps10

Permittivity (x,y,z) 10.0

Permeability (x,y,z) 1.0

Conductivity (x,y,z) 0.0

After filling all the data, click **OK** to close and register the material's values. Click **OK** button of Preference dialog to close the dialog.

5.2 Modeling tab

Back to the main view of EM. Click **Modeling** tab in the left panel to see a tree node view.

Select **Substances** node and Right-Click and select **New** from the pop-up menu to see **Substance** setting dialog (Fig.5.4).

You can set whatever you want in the **Substance Name**. **Substance 1** or something like that is already filled in the text input area as default. Replace it with **Sphere**.

Select **Eps10**, which is registered in the previous section, from **Material** choice menu. Select **Sphere** from **Figure** choice menu and you can see a parameter setting menu for **Sphere** in the bottom panel. Leave the **Centre** coordinate of the sphere unchanged and set radius to **0.1**. If you fill the all click the **OK** button to close the **Substance** dialog.

Click the **3D View** area to see an opaque blue sphere is at the centre of the area. You have placed the dielectric sphere in your model.
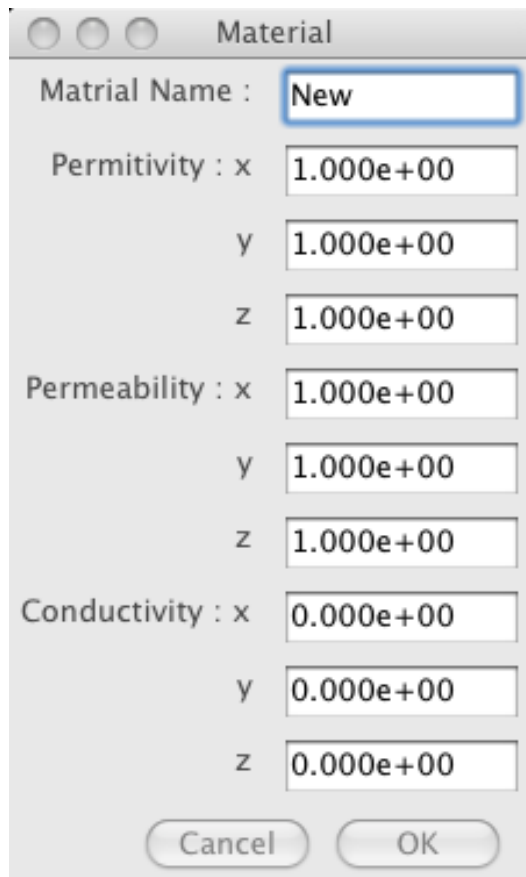
Figure 5.3: Material setting dialog.

Set **Time** as is indicated in the following.

Time 1.0E-8

and **Stimulus** is set as;

Spatial Plane Wave

Temporal Impulsive

The values for **Stimulus** are left unchanged as defaults.
All the work in **Modeling** tab is done.

5.3 Control tab

Click **Control** tab of the left panel to see the **Control** panel.

Set the **Level Limit** to **3** - **2**. If you want to increase the spatial resolution, increase the **Level Upper Limit** value. Then click **Start** button to generate the adaptive grid mesh.

Before you start the simulation, you may want to set probing points in the simulation. In order set the probing points, you may go into **View** tab.

5.4   View tab

Click **View** tab to see **View** setting panel. Select **Probe** from choice menu which is labeled as **Vector** in the first appear and click **Add** button to add **Probe** setting panel in the bottom of **View** panel.

Type the coordinates to probe in the **Probe Location**. You may specify as is shown in Table5.4. You fill a probe location then click **Add Probe** button to add the data in the list below. After all the locations are set click **Update** button to appear the probe points in the **3D View** as small yellow spheres as is shown in Fig.5.5.

5.5   Viewing the probe result

Wait for a while to get EM stop automatically, you can see a 3D view image as is shown in Fig.5.6.

You can get the time series of the probed field data in the **Probe** tab next to the **3D View** tab (Fig.2.1).

Click **Probe** tab to flip the **3D View** to 2 dimensional plot view. You may see a blank view in the panel. To display the plot, Right-Click anywhere in the panel to get a pop-up menu as is shown in Fig.5.7. Select **Fit** first and next **Zoom out** to display the whole plot graph in the panel.

You can see three plots which are registered in the **Probe** setting panel in **View** tab. The ID numbers (1,2 and 3) are corresponding to the order in the list in **Probe** setting panel.

If you want to save the data to a file, Right-Click to get the pop-up menu and select **Save to File** from the pop-up menu. A file saving dialog will appear. You can get a tab-separated file of the probed data.

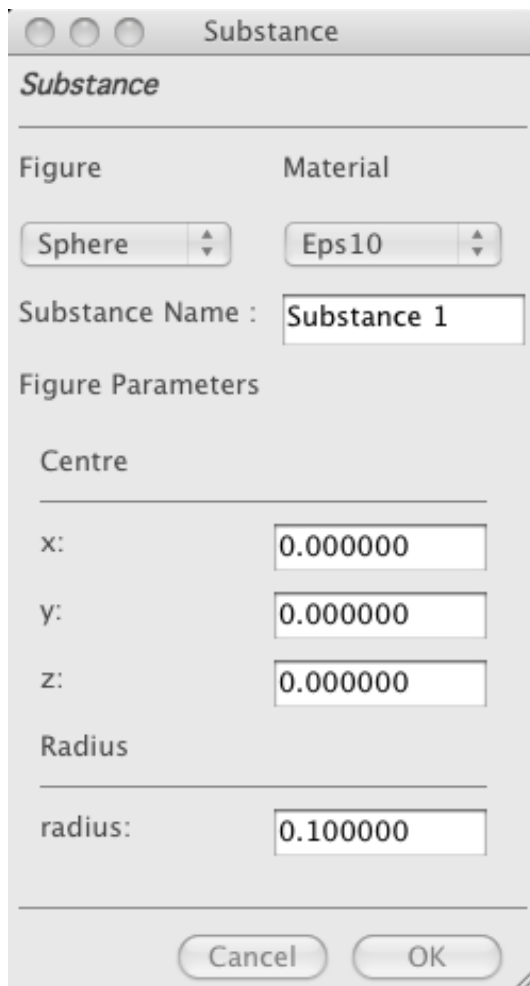| Probe Location	x	y	z
1	0	0	0
2	0	0	-0.2
3	0	0	0.2

Table 5.1: Probe Locations.
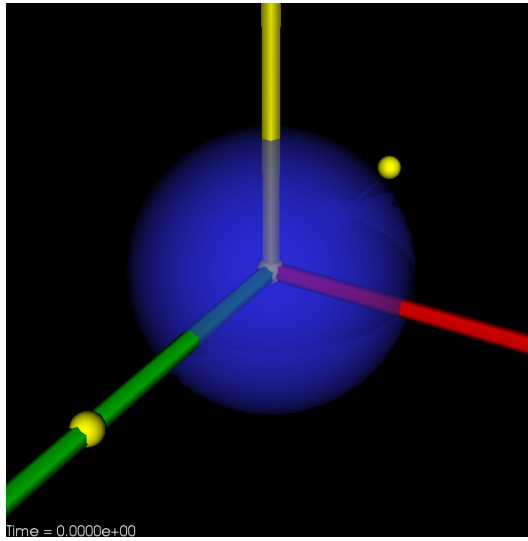
Figure 5.4: Substances setting dialog.

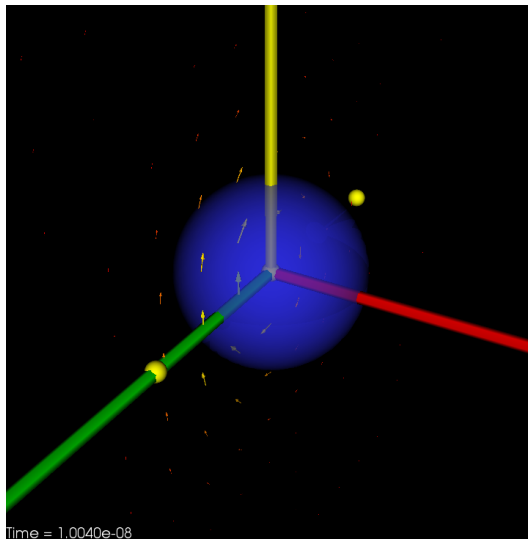Figure 5.5: Probe markers which specify their locations.

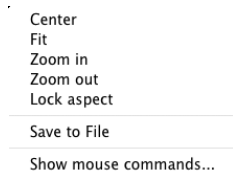Figure 5.6: The 3D View at $Time = 1.0 \times 10^{-8}$.
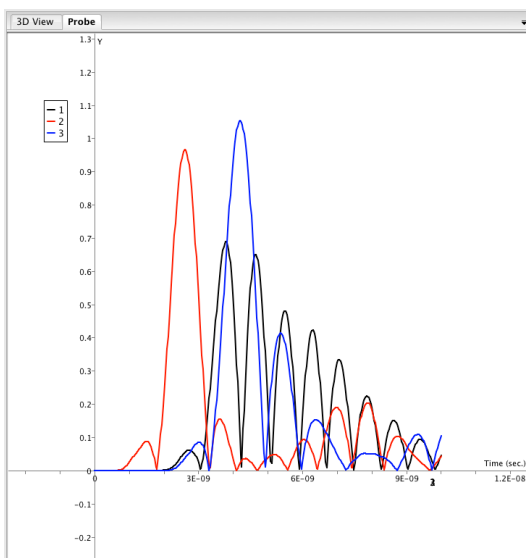
Figure 5.7: Probe markers which specify their locations.

Figure 5.8: Probed data of electric field force.