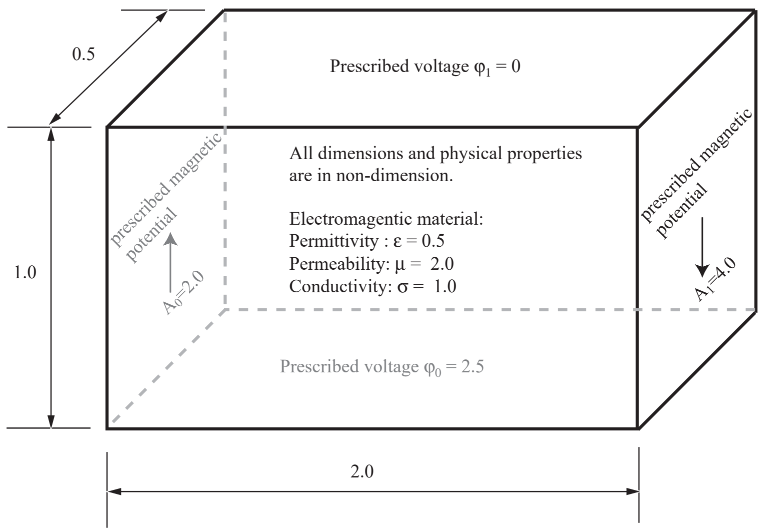
Problem description

We determine the three dimensional static electromagnetic fields inside a conducting block, as shown:



We will demonstrate the following topics:

- Setting up an electromagnetic model using potential formulation in ADINA EM
- Defining an electromagnetic material
- Applying potential based electromagnetic boundary conditions in ADINA EM

We assume that you have worked through problem 1 to 52, or have equivalent experience with the AUI.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This model cannot be solved with the 900 nodes version of the ADINA System because there are too many nodes in the model.

Some of the input for this problem is stored in the following file: `prob53_1.plo`. You need to copy this file from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA EM.

Defining model control data

Problem heading: Choose Control→Heading, enter “Problem 53: static 3D EM fields in a conducting block” and click OK.

Analysis type: Make sure that the Analysis Type drop-down list is set to "Static".

EM analysis setting: Choose Model→Analysis Settings, set the model type to "3D A-f model", verify that the Analysis Type is set to STATIC, set the "Tolerance for Residuals" to 1.0E-9 and click OK.

Defining the model geometry

The key geometry of the conducting block is as shown in the above sketch. It is, of course, possible to use ADINA-M to define the geometry, but we use the native AUI geometry in this problem.

Click the Define Points icon, enter the following points into the table (you can leave the X1 column blank) and click OK:

Point #	X1	X2	X3
1		-1.0	-0.5
2		1.0	-0.5
3		1.0	0.5
4		-1.0	0.5

Now click the Define Surfaces icon, add surface 1, set the Type to Vertex, set the Points to 1 ,2, 3, 4 respectively, and click OK.

Now click the Define Volumes icon, add volume 1, set the Type to Extruded, set the Initial Surface to 1, set the components of the Vector to 0.5, 0.0, 0.0, and click OK.

Defining material properties

Click the Manage Materials icon, add material 1, set the Permittivity (Epsilon) to 0.5, the Permeability (Mu) to 2.0, the Conductivity (Sigma) to 1.0, and click OK. These values correspond to a non-dimensionalized material.

Defining the boundary conditions

Note that every EM boundary condition must be applied onto boundary geometry with continuous curvature, except the Dirichlet condition (see detailed explanation in Section 5.1.2 of the ADINA EM Theory and Modeling Guide).

Electric potential boundary conditions: Choose Model→Boundary Conditions, add boundary condition 1 and verify that the Type is Dirichlet. Set the Variable Type to Electric Potential, set the Real part to 2.5, verify the boundary condition is applied to Surfaces, then apply this boundary condition 1 to surface 2. Add boundary condition 2, verify that the type is Dirichlet, set the Variable Type to Electric Potential, set the Real part to 0, then verify the boundary condition is applied to Surfaces, and apply this boundary condition to surface 4. Click Save (do not close the dialog box yet.)

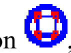
Boundary conditions on all other surfaces for electric potential are natural boundary conditions by default.

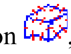
Magnetic potential boundary conditions: Add boundary condition 3 and verify that the Type is Dirichlet. Set the Variable Type to Magnetic Potential, then set the Real part to 2.0, set the Direction Type to D0 x NR, and set DX to -1. Then verify that the boundary condition is applied to Surfaces, and apply this boundary condition to surface 5. Add boundary condition 4, verify that the type is Dirichlet, and set the Variable Type to Magnetic Potential, set the Real part to 4.0, set the Direction Type to D0 x NR, and set DX to -1. Then verify that the boundary condition is applied to Surfaces, and apply this boundary condition to surface 3. Click Save (do not close the dialog box yet.)

EM Parallel boundary condition on magnetic potential: Add boundary condition 5 and set the Type to Parallel. Set the Variable Type to Magnetic Potential, and make sure that the Real part is 0.0. Then verify that this boundary condition is applied to Surfaces, and apply this boundary condition to surfaces 2 and 4. Click Save (do not close the dialog box yet.)

EM Normal boundary condition on magnetic potential: Add boundary condition 6 and set the Type to Normal. Set the Variable Type to Magnetic Potential, and make sure that the Real part is 0.0. Then verify that this boundary condition is applied to Surfaces, and apply this boundary condition to surfaces 1 and 6. Click OK to close the dialog box.

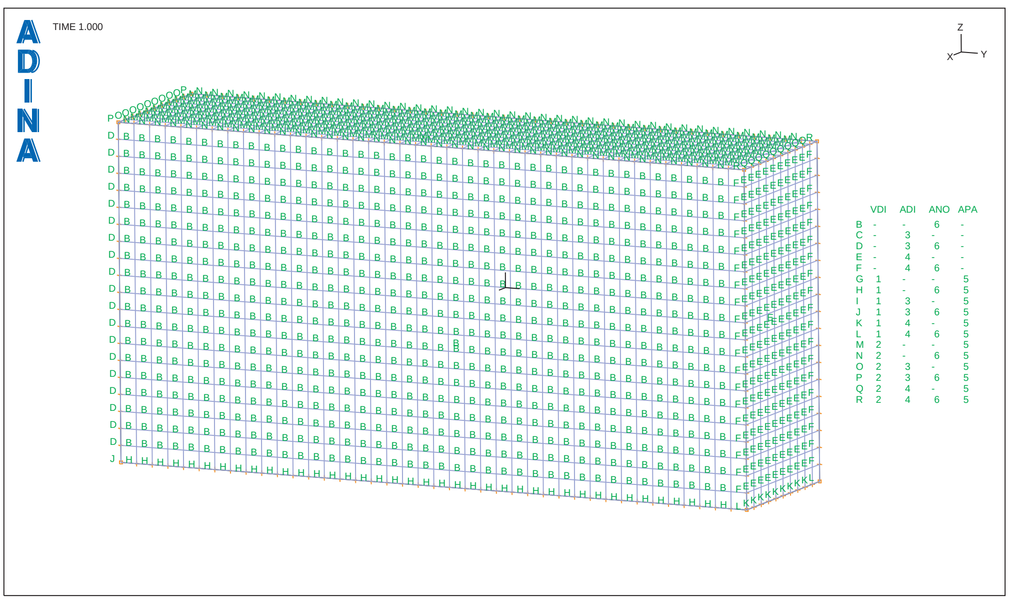
Defining the elements

Element group: Click the Element Groups icon , add element group 1, set the Type to 3-D Electromagnetic, verify that the material is 1, and make sure that both Electric Effects and Magnetic Effects are checked. Then click OK to close the dialog box.

Subdivision data: Choose the Subdivide Volumes icon , set the "Method" to Use Length, set the "Element Edge Length" to 0.05 and click OK.

Meshing the volume: Now click the Mesh Volumes icon, make sure that the Element Group is 1 and that Nodes per Element is 8, then enter Volume 1 in the first row of the table and click OK.

Click the Boundary Plot icon and use the mouse to rearrange the graphics window until it looks something like this:



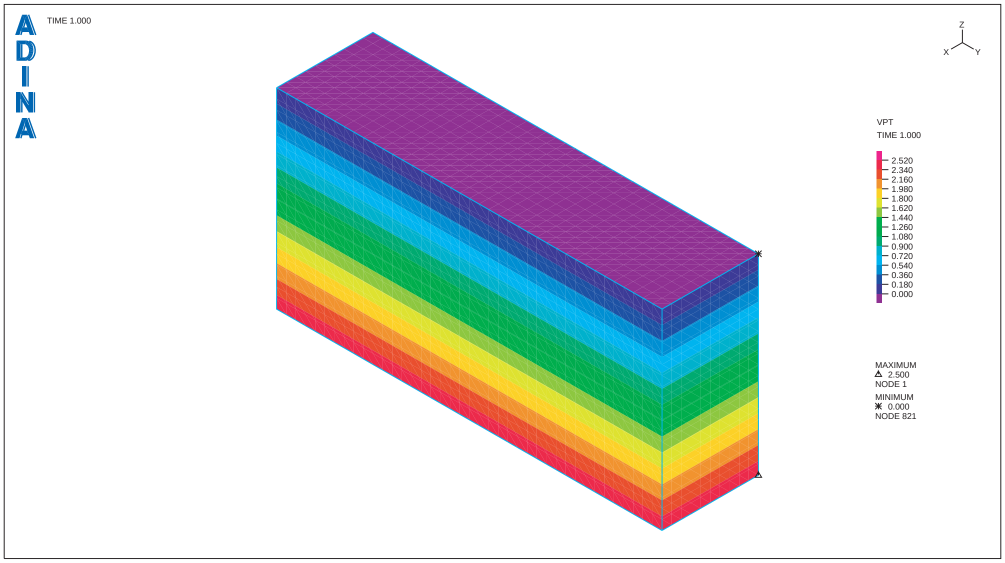
Generating the data file, running ADINA EM, loading the porthole file

Click the Save icon and save the database to file `prob53`. Click the Data File/Solution icon, set the file name to `prob53`, make sure that the Run Solution button is checked and that the Max Memory for Solution is at least 30 MB, then click Save. When ADINA EM is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob53`.

Examining the solution

Click the Model Outline icon, then click the Save Mesh Plot Style icon. (We save the mesh plot style so that we don't have to repeat the above step for each plot.)

Electric potential plot: Click the Create Band Plot icon, set the Band Plot Variable to (Electromagnetic:VPT) and click OK. The graphics window should look something like this:

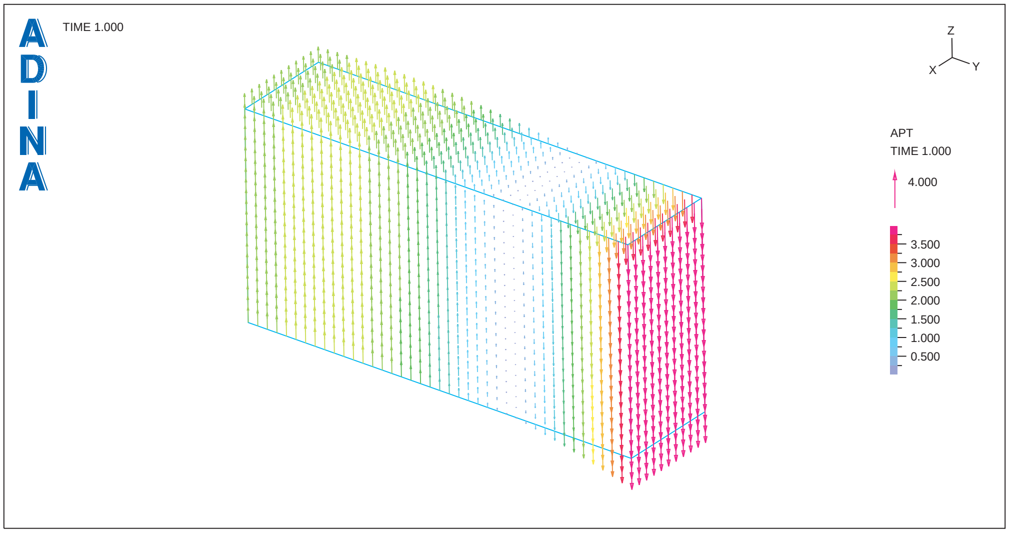


Magnetic vector potential plot: Click the Clear icon and the Mesh Plot icon to plot the mesh outline. Click the Create Vector Plot icon, set the Vector Quantity to APT, and click OK. The graphics window should look something like the figure on the next page.

Comparison with analytical solutions

We want to compare the electric potential and magnetic vector potential with analytic solutions along given vertical and horizontal lines. Since the mesh is mapped, it is straightforward to define node lines corresponding to these lines. It is also straightforward to determine the analytical solutions at points along these lines.

Thus, for the vertical line, we will define a node line VL for the nodes along the vertical line, and we will define a user data V for the analytical electric potential along the vertical line. For the horizontal line, we will define a node line AL for the nodes along the horizontal line, and we will define a user data A for the magnetic potential along the horizontal line.

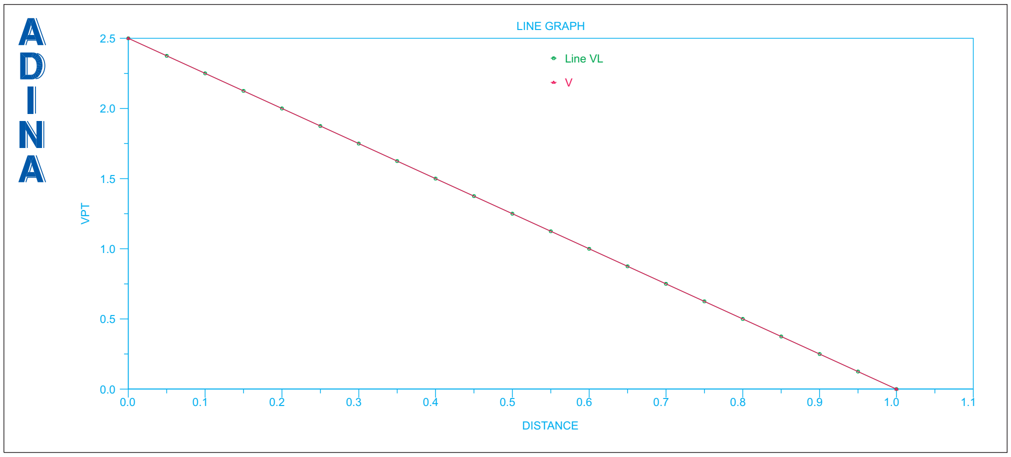


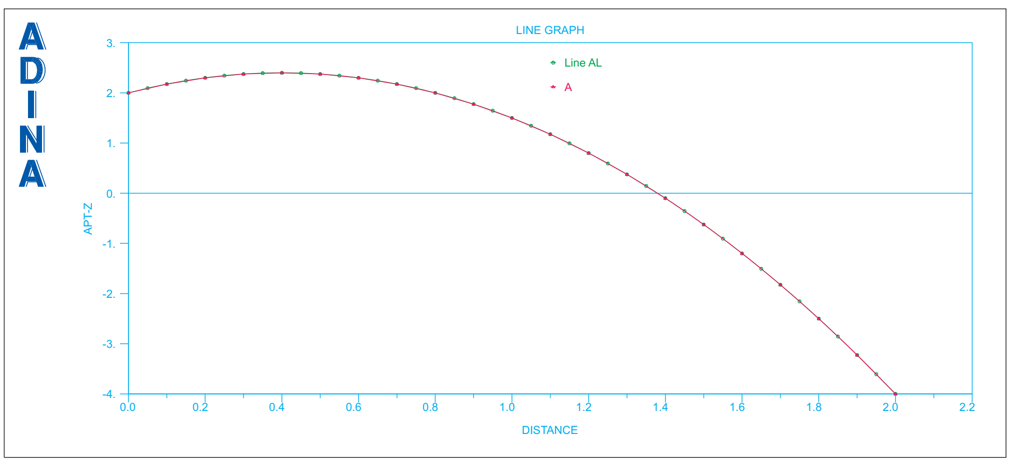
We have put the definitions of VL, AL, V and A into batch file `prob53_1.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob53_1.plo` and click Open. You can examine the definitions of lines VL and AL by choosing Definitions→Model Line→Node, and you can examine the definitions of V and A by choosing Graph→Define User Data.

Analytical solution comparison for electric potential: Click the Clear icon , choose Graph→Response Curve (Model Line), set the Model Line Name to VL, set the Y Coordinate Variable to (Electromagnetic: VPT) and click OK. Now choose Graph→Plot User Data, set the Data Name to V and the Plot Name to PREVIOUS, then click OK. The graphics window should look something like the top figure on the next page.

It is of course possible to customize the curve legends and axes labels as shown in primer problem 2.

Analytical solution comparison for magnetic vector potential: Click the Clear icon , choose Graph→Response Curve (Model Line), set the Model Line Name to AL, set the Y Coordinate Variable to (Electromagnetic: APT-Z) and click OK. Now choose Graph→Plot User Data, set the Data Name to A and the Plot Name to PREVIOUS, then click OK. The graphics window should look something like the bottom figure on the next page.





Listing of results: Choose List→Value List→Model Line, set the Model Line Name to VL, set the Variables to List to (Electromagnetic: VPT) and click Apply. The electric potentials on the node line are displayed. Similarly, set the Model Line Name to AL, set the first variable to (Electromagnetic: APT-Y) and the second variable to (Electromagnetic: APT-Z), then click Apply. The components of magnetic potential are displayed.

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

This page intentionally left blank.