Problem description

In this problem, we analyze microwave heating in a cavity. In part 1, we analyze the harmonic electromagnetic field resonance in a two-dimensional cavity and demonstrate the following topics:

- Setting up a harmonic electromagnetic model in ADINA EM
- Defining an electromagnetic material
- Applying electromagnetic boundary conditions

In part 2, we demonstrate the following additional topics:

- Setting up a three dimensional electromagnetic model in ADINA EM
- Setting up a thermal model in ADINA CFD coupled with electromagnetic fields
- Plotting the Joule heating effect caused by the harmonic electromagnetic fields

We assume that you have worked through problem 1 to 53, 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.

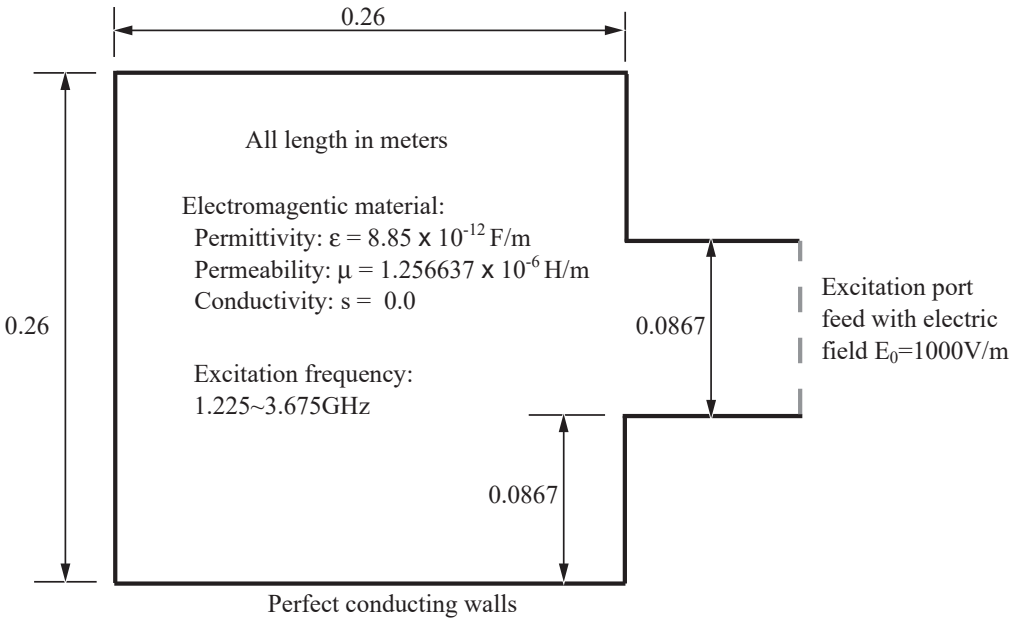
You need to allocate 400 MB to ADINA EM in the first part of the analysis, and at least 1000 MB to ADINA CFD+EM in the second part of the analysis.

Much of the input for this problem is stored in the following files: `prob54a_1.in`, `prob54b_1.in`, `prob54b_2.in`. You need to copy these files from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Part 1: Resonance in a 2D cavity

Defining the model

In this part, the problem of a 2D cavity excited by an electric field is demonstrated. The model geometry and main physical properties are shown in the figure on the next page.



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 54a: Electromagnetic fields in 2D resonant cavity" and click OK.

Analysis type: Set the Analysis Type drop-down list to Harmonic.

EM analysis setting: Choose Model→Analysis Settings, set the Model Type to "2D E-H model on magnetic plane", verify that the analysis type is set to Harmonic, set the Frequency Value to 1.53938E10, and the Frequency Time Function to 1, and also set the Tolerance for Residuals to 1E-9. Click OK to close the dialog box.

Frequency sweep time function: Choose Control→Time Function, edit the table to read as follows, then click OK.

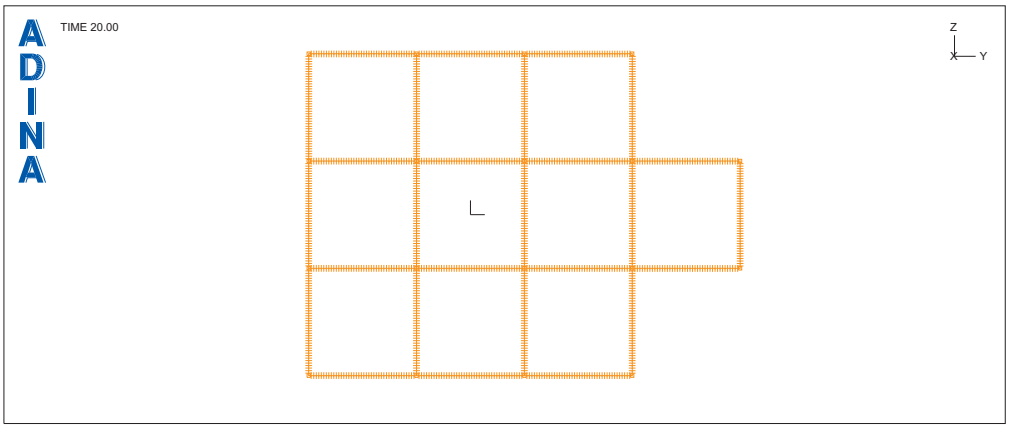
Time	Value
0.0	0.5
20.0	1.5

Time steps: Choose Control→Time Step, edit the table to read as follows, then click OK.

Number of Steps	Magnitude
20	1.0

Defining the model geometry

We have put the model geometry definitions into file `prob54a_1.in`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob54a_1.in` and click Open. The graphics window should look something like this:



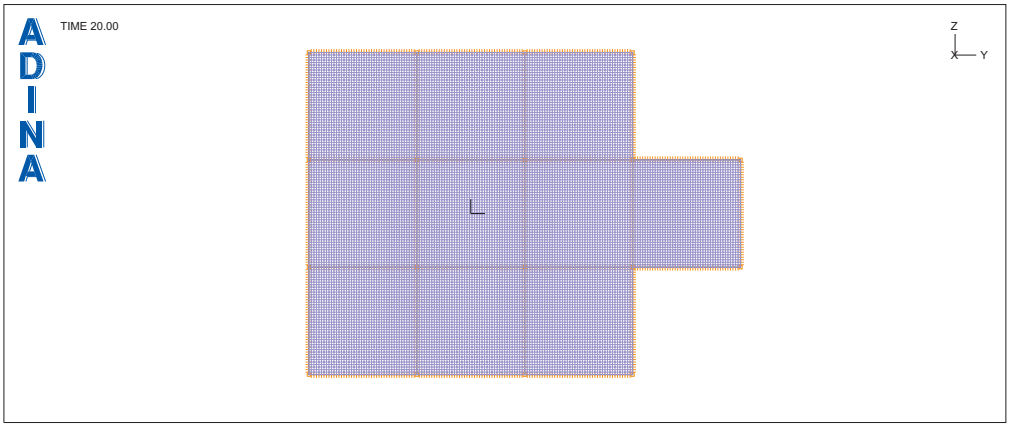
Defining material properties

Click the Manage Materials icon , add Material 1, set the Permittivity (Epsilon) to 8.85E-12, the Permeability (Mu) to 1.256637E-6, make sure that the Conductivity (Sigma) is 0.0, and click OK.

Defining the elements

Element group: Click the Element Groups icon , add Element Group 1, verify that the Type is 2-D Electromagnetic, the material is 1, and both Electric Effects and Magnetic Effects are checked. Then click OK to close the dialog box.

Now click the Mesh Surfaces icon  , verify that the Element Group is 1, the Meshing Type is Rule-Based, and the Nodes per Element is 4. Enter 1 to 10 in the first ten rows of the table, then click OK to close the dialog box. The graphics window should look something like this:



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 Field Intensity Dirichlet boundary conditions: Choose Model→Boundary Conditions..., add boundary condition 1 and verify that the Type is Dirichlet. Make sure that the Variable Type is Electric Field Intensity and set the Real part to 1000. Make sure that the Direction Type is "VECTOR, D0", and set DX to 1. Then make sure the boundary condition is applied to Lines, and enter 19 in the first row of the table. Click Save.

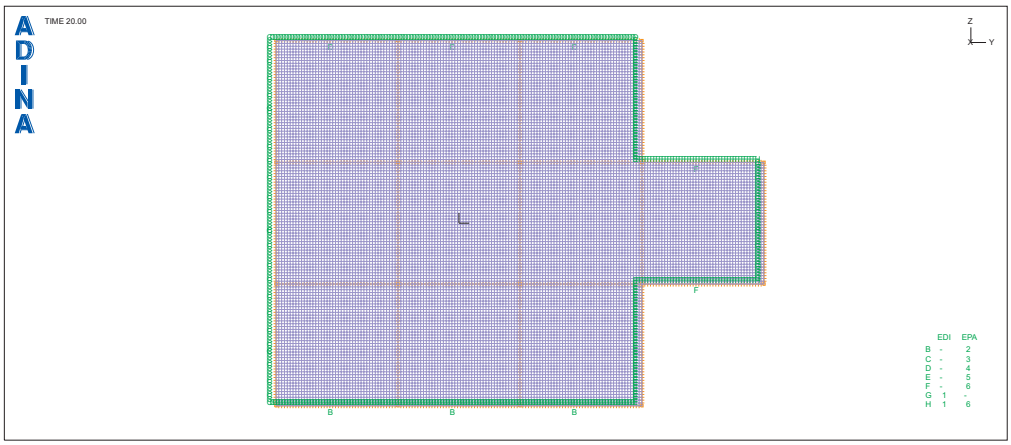
EM Parallel boundary condition on electric field intensity: We will now add the EM Parallel boundary conditions. Add the boundary conditions in the table below, with the Type set to Parallel, the Variable Type set to Electric Field Intensity and the Real Part set to 0.0.

Boundary condition number	Lines
2	1, 5, 8
3	4, 13, 23
4	22, 25, 27
5	9, 26
6	18, 20

Click OK to close the dialog box.

(Note that each of these boundary conditions is the same physical boundary condition, applied to different groups of lines.)

When you click the Boundary Plot icon , the graphics window should look something like this:



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

Click the Save icon and save the database to file `prob54a`. Click the Data File/Solution icon , set the file name to `prob54a`, make sure that the Run Solution button is checked, set the Maximum Memory for Solution to at least 400 MB, and click Save.

ADINA EM runs for 20 steps.

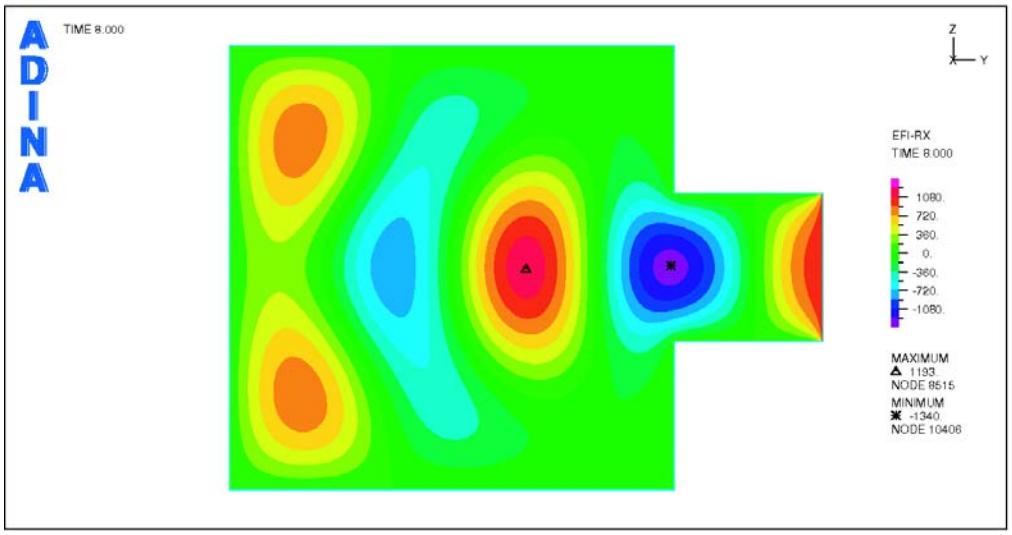
When ADINA EM is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob54a`.

Examining the solution

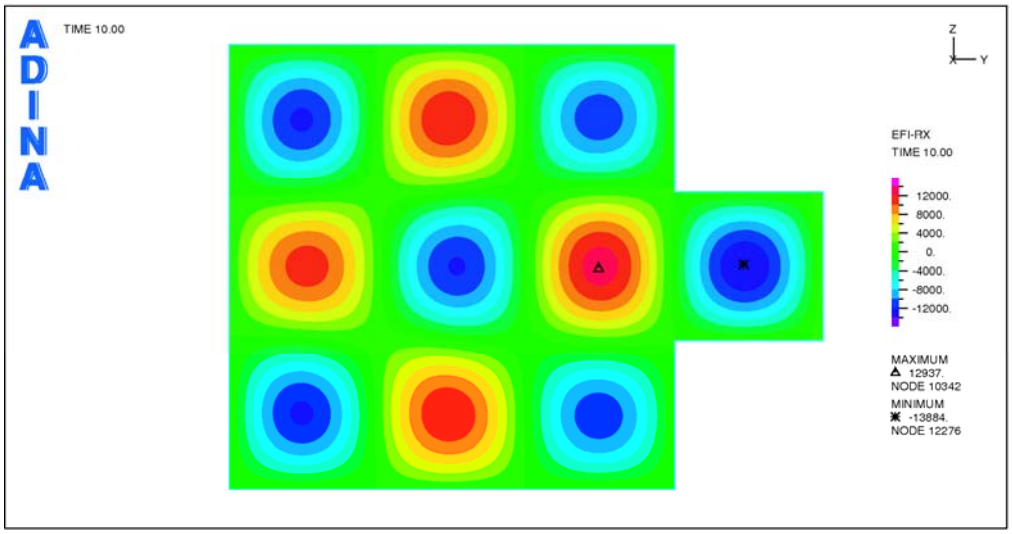
Save mesh plot defaults: We will create plots of the results within the enclosure. As the underlying mesh plots will all have the same appearance, we set the appearance of the first mesh plot, then set the defaults to that appearance. Click the Model Outline icon , then click the Save Mesh Plot Style icon .

First we would like to check the electric and magnetic field intensities at three frequencies inside the resonant cavity: $f = 2.2, 2.45, 2.7$ GHz , which correspond to time steps 8, 10 and 12 respectively (note that the angular frequency is entered in the input to this model, which equals 2π times the frequency we used here).

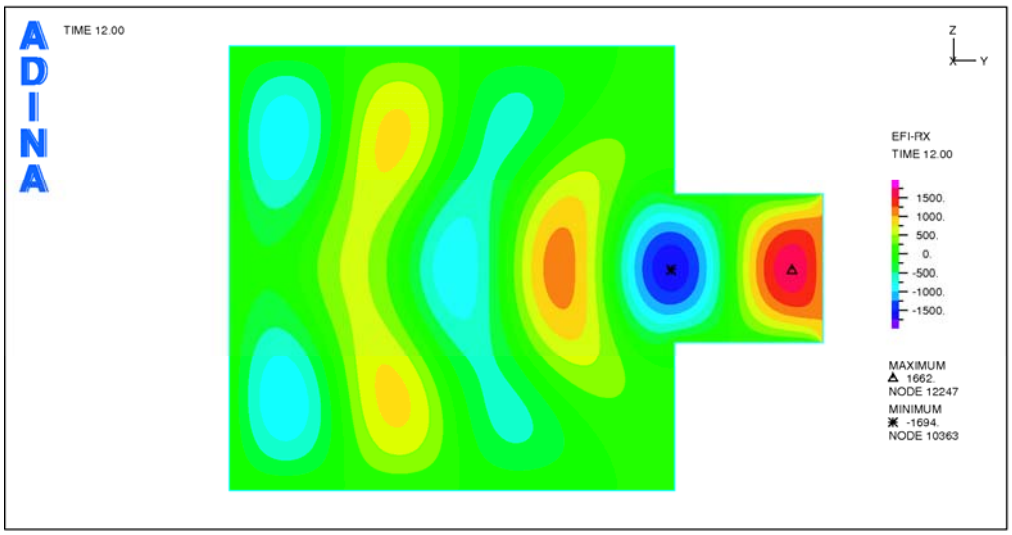
Electric field intensity: Use the Previous Solution and Next Solution icons (and) to change the solution time to 8.0 (corresponding to frequency 2.2 GHz). Click the Create Band Plot icon , set the Band Plot Variable to (Electromagnetic: EFI-RX) (real part of electric field intensity in x-direction) and click OK. (In the present EM harmonic model, the imaginary part of electric field intensity is trivial.) The graphics window should look something like this:



Then click the Next Solution icon twice to change the solution time to 10.0 (corresponding to frequency 2.45 GHz). Notice that the band table scaling doesn't change. To rescale the band table, click the Modify Band Plot icon , click the Band Table... button, and in the Value Range box, set Maximum and Minimum to Automatic, then click OK twice to close both dialog boxes. The graphics window should look something like the figure on the next page.

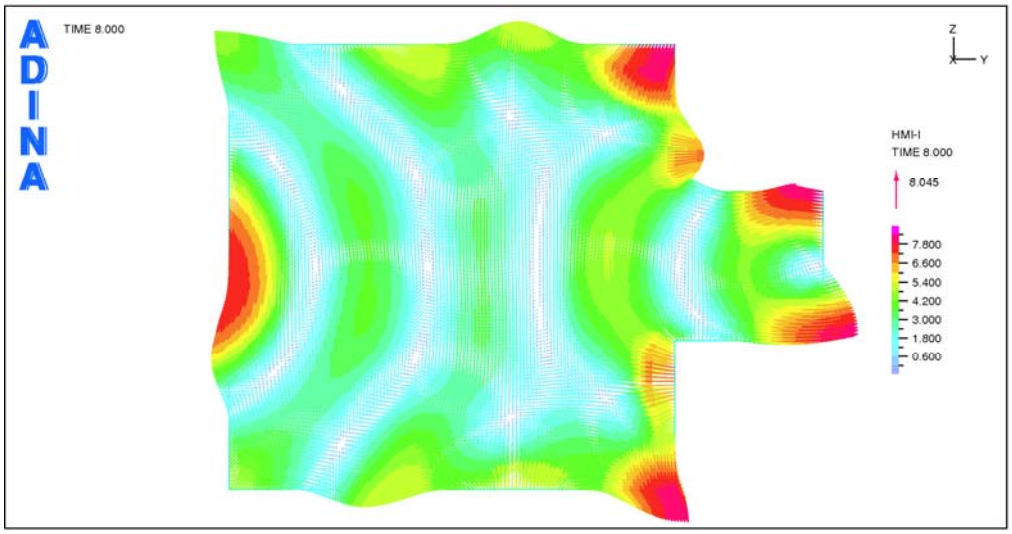


Then click the Next Solution icon ▶ twice to change the solution time to 12.0 (corresponding to frequency 2.7 GHz). Repeat the above instructions to rescale the band table. The graphics window should look something like this:

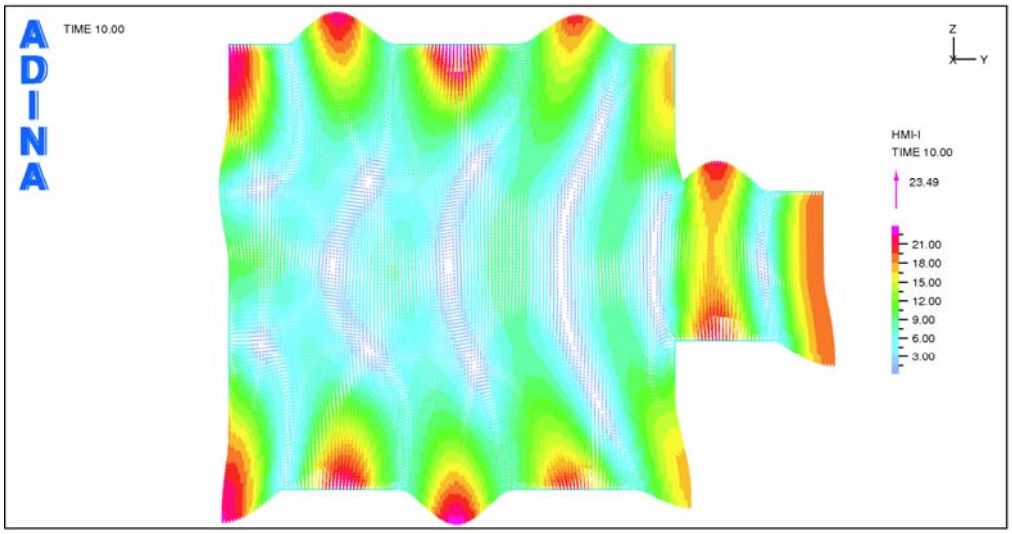


We can also list out the instantaneous electric field at a specific phase angle. Choose List→Value List→Zone, set the first variable to (Electromagnetic: EFI-X) and click the ... button to the right of the Result Control field. In the Define Result Control Depiction dialog box, set the Phase Angle (degrees) field to 45 and click OK. In the List Zone Values dialog box, set the Response Option to Single Response and click the ... button to the right of the Response field. In the Define Response dialog box, set the Solution time to Latest and click OK. In the List Zone Values dialog box, click Apply. The listing displays the electric field intensity in the x-direction over the whole domain for frequency 3.675 GHz (corresponds to solution at time 20). Click Close to close the dialog box.

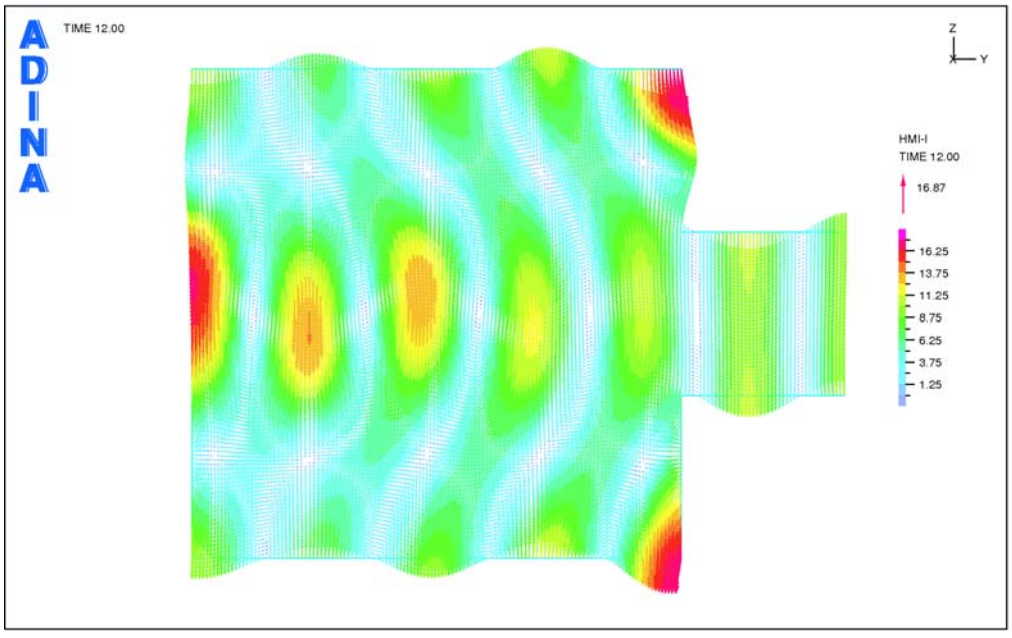
Magnetic field intensity: Click the Clear icon and the Mesh Plot icon, then use the Previous Solution and Next Solution icons (and) to change the solution time to 8.0. Click the Create Vector Plot icon, set the Vector Quantity to HMI-I (imaginary part of magnetic field intensity vector) and click OK. (In the present EM harmonic model, the real part of magnetic field intensity is trivial.) The graphics window should look something like this:



Then click the Next Solution icon twice to change the solution time to 10.0 (corresponding to frequency 2.45 GHz). Notice that the vector scaling doesn't change. To replot the vectors, click the Modify Vector Plot icon, click the Rendering... button, set the Scale Option to Automatic and click OK twice to close both dialog boxes. The graphics window should look something like the figure on the next page.



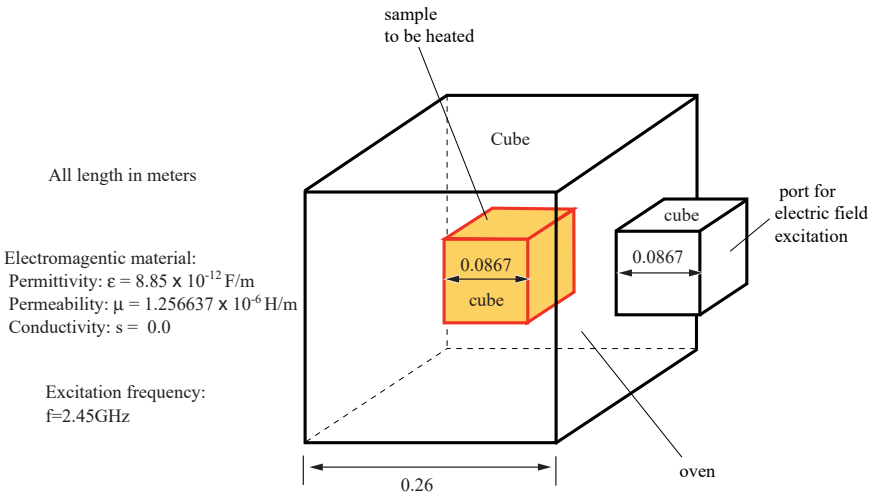
Follow the same procedure to create the vector plot for HMI-I at time 12.0. The graphics window should look something like this:



Frequency sweep movie show: It is very convenient to show the electromagnetic responses at different frequencies in a single animation. Click the Clear icon and the Mesh Plot icon , then use the Previous Solution and Next Solution icons (and) to change the solution time to 1.0. Click the Create Band Plot icon , set the Band Plot Variable to (Electromagnetic: EFI-RX) and click OK. Click the Modify Band Plot icon , and the Band Table... button, uncheck the Freeze Range field, then click OK twice to close both dialog boxes. Click the Movie Load Step icon to create a movie showing the real part of the electric field intensity at different times and hence at different frequencies. To play back the movie, click the Animate icon or choose Display→Animate. Click the Refresh icon to clear the animation.

Part 2: Microwave heating in a 3D cavity

In this part, we will extend the 2D cavity analysis in the previous part to 3D and include the Joule heating effect. The sketch of this 3D model is as follows:



Choosing the finite element program

Set the Program Module drop-down list to ADINA EM (you can discard all changes).

Defining model control data

Problem heading: Choose Control→Heading, enter “Problem 54b: 3D microwave heating” and click OK.

Analysis type: Set the Analysis Type drop-down list to "Harmonic", and set the Multiphysics Coupling drop-down list to "with CFD".

We also need to change the analysis type for ADINA CFD. Set the Program Module drop-down list to ADINA CFD and set the Analysis Type drop-down list to "Transient". Make sure that the Multiphysics Coupling drop-down list is set to "with EM". Now set the Program Module drop-down list back to ADINA EM.

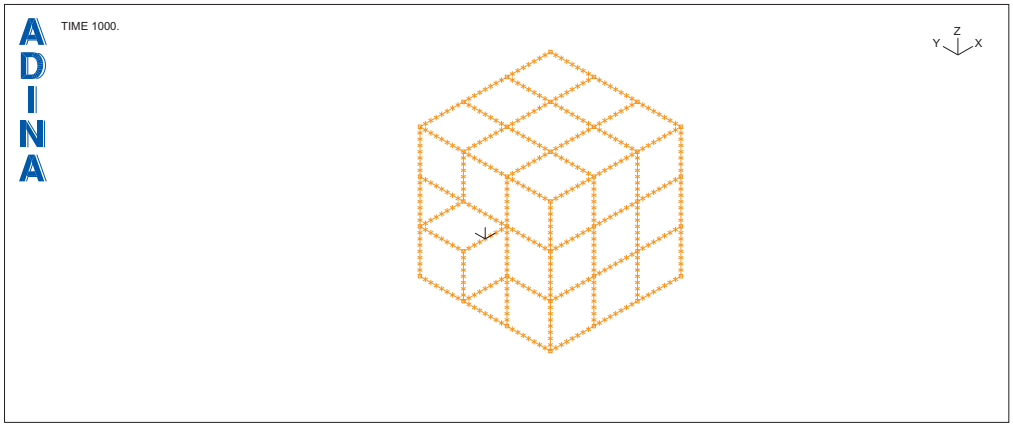
EM analysis settings: Choose Model→Electromagnetic→Settings, set the Model Type to "3D E-H model", verify the analysis type is set to Harmonic, set the Frequency Value to 1.53938E10, set the Tolerance for Residuals to 1E-9, set the 'Electromagnetic Force Applied' to None, and then click OK to close the dialog box. Choose Control→Solution Process, click the Outer Iteration... button, then click the Advanced Settings... button. Set the Equation Residual to All and the two Tolerance fields to 1E-6. Click OK three times to close all dialog boxes.

Time Steps: Choose Control→Time Step, edit the table to read as follows, then click OK.

Number of Steps	Magnitude
1000	1.0

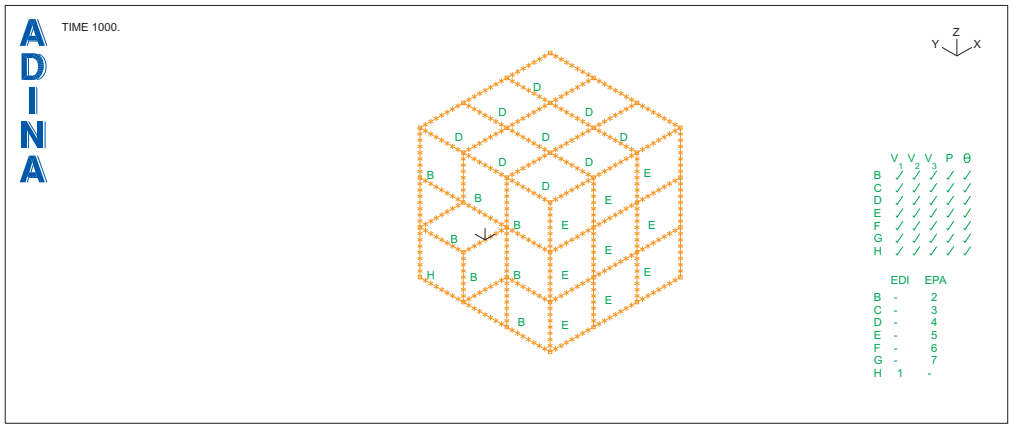
Defining the model geometry

We have put the model geometry definitions into file `prob54b_1.in`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob54b_1.in` and click Open. The graphics window should look something like the figure on the next page.



Defining the electromagnetic boundary conditions

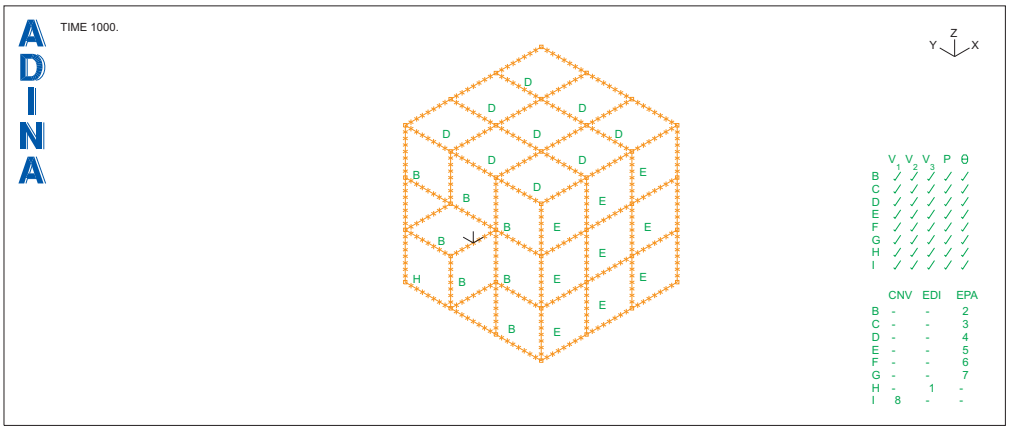
We have put the electromagnetic boundary condition definitions into file `prob54b_2.in`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob54b_2.in` and click Open. When you click the Boundary Plot icon , the graphics window should look something like this:



Defining the boundary conditions on the fluid

Temperature boundary condition: Click the Special Boundary Conditions icon SBC, add boundary condition 8 and set the Type to Heat Transfer Convection. Set the Convection Coefficient Function Multiplier to 5.0, and set the Environment Temperature Function Multiplier to 293.0. Then make sure that the "Apply to" field is set to Faces/Surfaces, enter 29,

50, 56, 60, 61, 62 in the first column of the table, then click OK. When you click the Redraw icon  , the graphics window should look something like this:



Defining initial conditions on the fluid

Initial temperature over heating sample: Choose Model→Initial Conditions→ Define, add condition TINIT, and, in the table, set the first row to TEMPERATURE, 293. Click Save, then Apply... . In the Apply Initial Conditions dialog box, set the 'Apply to' field to Volumes, then set the first row of the table to 14. Click OK twice to close both dialog boxes.

Defining material properties

Fluid: Click the Manage Materials icon  , click the Laminar button and add material 1. Set the Viscosity to 0.1, the Density to 1120, the Thermal Conductivity to 1.381 and the Specific Heat at Constant Pressure to 153.1. Click OK, then Close, to close both dialog boxes.

Electromagnetic: Choose Model→Electromagnetic→Material Constant, add material 1, set the Permittivity (Epsilon) to 7.08E-11, the Permeability (Mu) to 1.256637E-6, the Conductivity (Sigma) to 1.3, and click Save. Then add material 2, set the Permittivity(Epsilon) to 8.85E-12, the Permeability(Mu) to 1.256637E-6, the Conductivity(Sigma) to 0.0, and click OK.

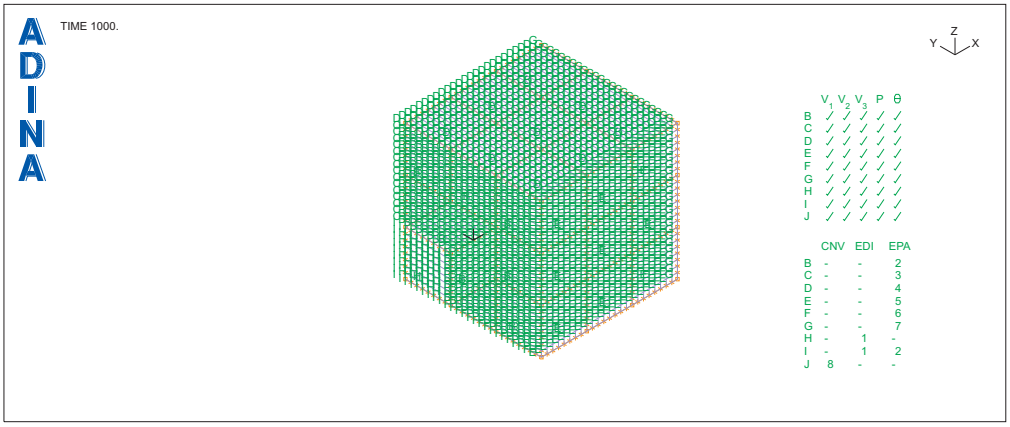
Defining the elements

Element groups: Click the Element Groups icon  , add Element Group 1, set the Type to 3-D Fluid/EM, make sure that the default material is 1, and check both the Electric Effects and Magnetic Effects fields. Set the Element Option to Solid, and set the Electromagnetic Material to 1. Click Save.

Now add element group 2, verify that the Type is 3-D Fluid/EM, uncheck the Fluid Element field and check both the Electric Effects and Magnetic Effects fields. Set the Electromagnetic Material to 2, then click OK to close the dialog box.

Meshing: Click the Mesh Volumes icon  , set the Element Group to 1, enter 14 in the first row of the table and click Apply. Now set the Element Group to 2, input Volumes 1 to 28 except 14 into the Volumes to be Meshed table, and click OK.

The graphics window should look something like this:



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

Click the Save icon  and save the database to file `prob54b`. Click the Data File/Solution icon  , set the file name to `prob54b`, make sure that the Run Solution button is checked, set the Maximum Memory for Solution to at least 1000 MB (and preferably to at least 4500 MB), and click Save.

ADINA CFD+EM runs for 1000 steps. The first step takes most of the solution time.

When ADINA CFD+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 `prob54b`.

Examining the solution

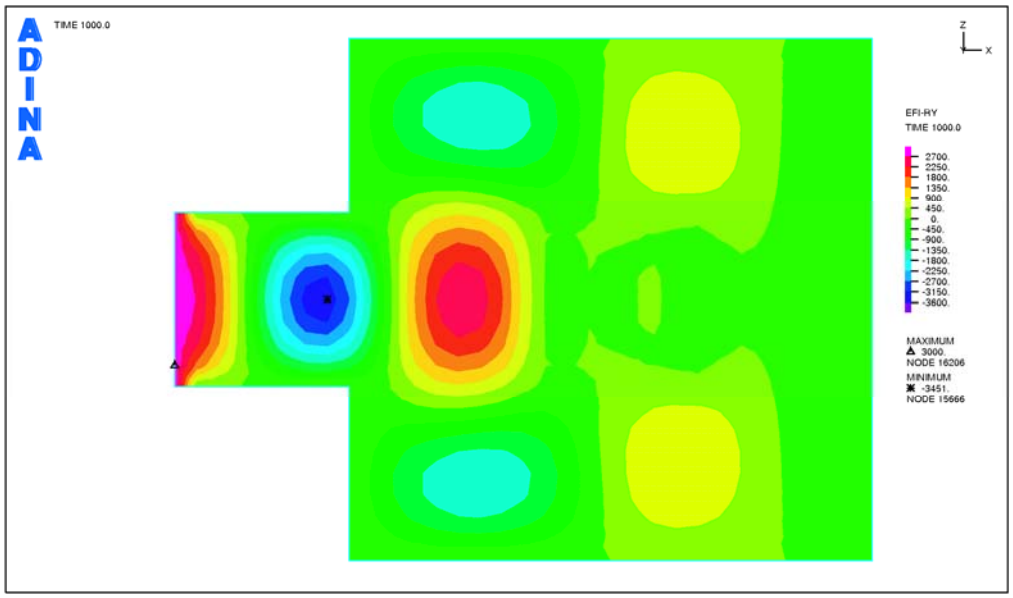
We want to create plots of the results within the cavity. So we will define a cutting plane, then change the view and remove all of the cutplane interior lines.

Cutting plane: Click the Cut Surface icon  . Set the Type to Cutting Plane, set "Defined by" to Y-Plane and uncheck the Display the Plane(s) field. Then set 'Below the Cutplane' and 'Above the Cutplane' to 'Do Not Display'. Click OK to close the dialog box.

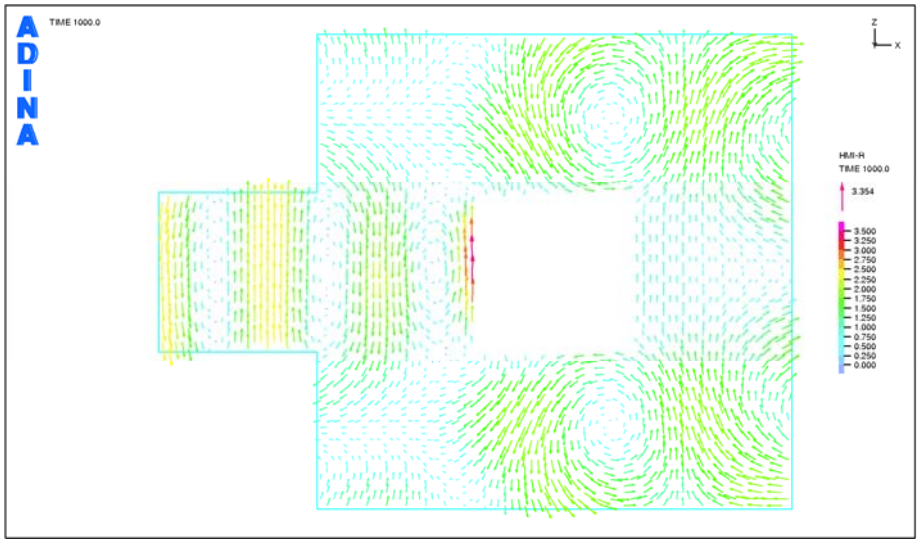
View and cutting plane interior lines: Click the XZ View icon  and the Model Outline icon  .

Save mesh plot defaults: We will use this mesh plot appearance for several plots, so we save the mesh plot defaults so that we don't have to repeat the above steps for each plot. Click the Save Mesh Plot Style icon  .

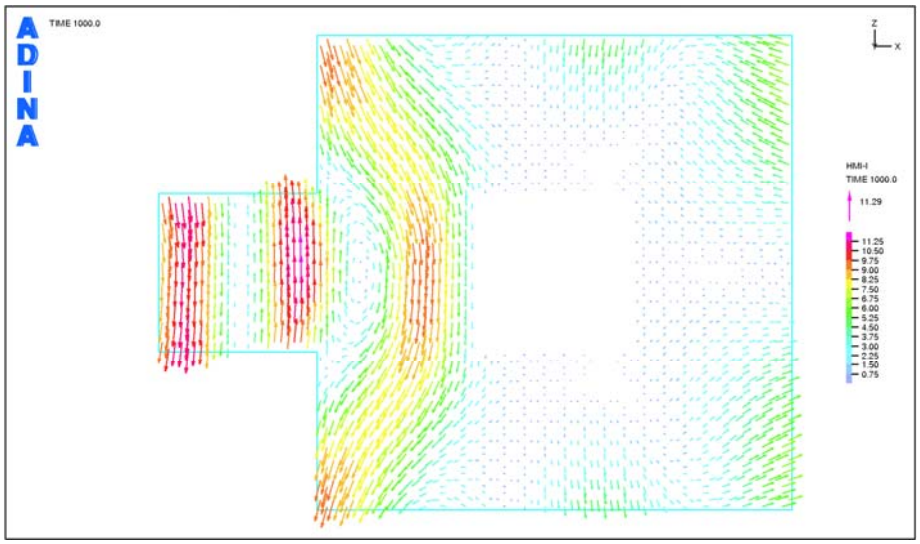
Electric field intensity: Click the Create Band Plot icon  , set the Band Plot Variable to (Electromagnetic: EFI-RY) (Real part of electric field intensity in y-direction) and click OK. The graphics window should look something like this:



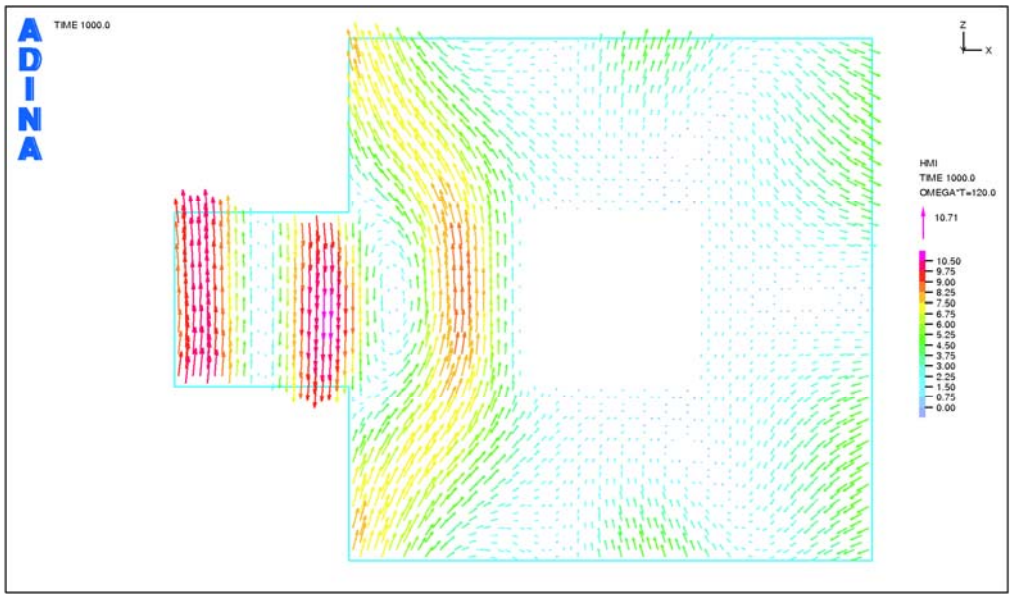
Magnetic field intensity: Click the Clear Band Plot icon  , then click the Create Vector Plot icon  , set the Vector Quantity to HMI-R (real part of magnetic field intensity vector) and click OK. The graphics window should look something like this:



Click the Modify Vector Plot icon  , set the Vector Quantity to HMI-I (imaginary part of magnetic field intensity vector) and click OK. The graphics window should look something like this:

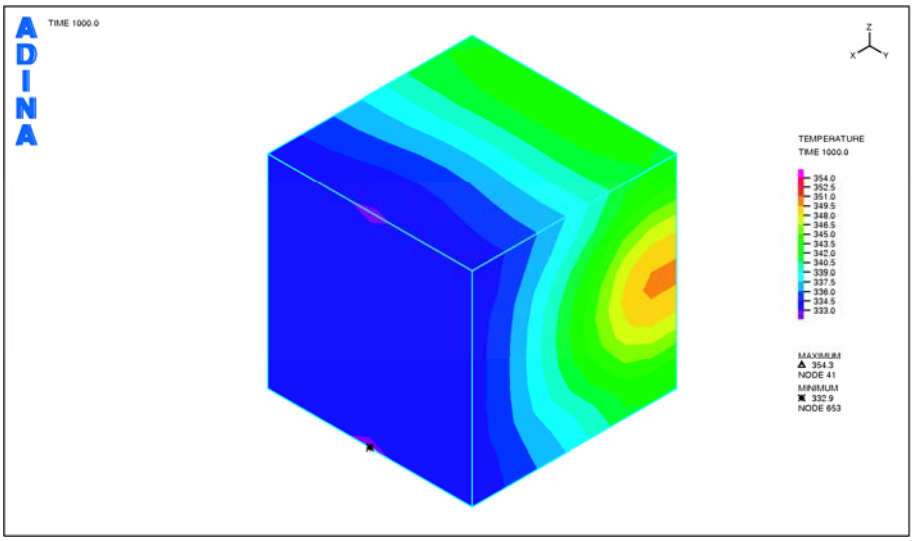


You can also show the magnetic field intensity at a specific phase angle. Click the Modify Vector Plot icon  , set the Vector Quantity to HMI (magnetic field intensity vector) and click the Result Control... button. In the Define Result Control dialog box, set the Phase Angle (degrees) to 120 and click OK twice to close both dialog boxes. The graphics window should look something like this:



Temperature distribution: Now click the Reset Mesh Plot Style icon  , then click the Clear icon  and the Mesh Plot icon  . In the Model Tree, expand the Zone entry, right-click 2. EG1 and choose Display. We will show the temperature distribution of the heated sample. Click the Model Outline icon  , then click the Create Band Plot icon  , set the Band Plot Variable to (Temperature: TEMPERATURE) and click OK. The graphics window should look something like the figure on the next page.

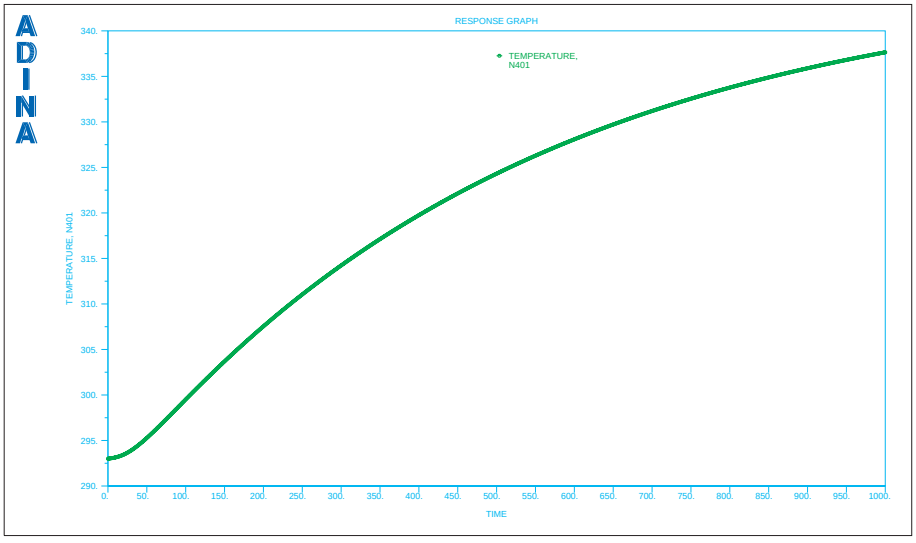
You can see the temperature gradient over the surface of the heated sample. This is due to the non-uniformity of the electric field distribution. You can also generate a movie for temperature distribution over time, by clicking the Movie Load Step icon  , and then the Animate icon  . Click the Refresh icon to  clear the animation.



Solution at node 401: First we need to define a model point at a node. Choose Definitions→Model Point→Node, add point N401, set the Node # to 401 and click OK.

Temperature vs. time graph: We can plot the temperature at node 401 as a function of time. Click the Clear icon 

, then choose Graph→Response Curve (Model Point), make sure that the Y Coordinate Variable is TEMPERATURE and the Y Coordinate Model Point is N401, then click OK. The graphics window should look something like this:



List solution at node 401: We want to list the electromagnetic and temperature solutions at node 401. Choose List→Value List→Model Point, set the Variables to List to (Temperature: TEMPERATURE), (Electromagnetic: EFI_MAGNITUDE), (Electromagnetic: HMI_MAGNITUDE), then click Apply. Click Close to close the dialog box.

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

This page intentionally left blank.