AMFIA V1.0 USER'S MANUAL

BAE SYSTEMS
95 Canal Street NCA1-6268
P.O. Box 868
Nashua, NH 03061-0868

Erdem Topsakal
John Volakis

October 2001
Integral-equation methods and to a lesser degree, differential equation methods are commonly used for electromagnetic scattering and radiation computations. Integral equation formulations are more attractive for perfectly conducting, impedance, resistive and layered-material structures [1-4] whereas volume formulations are particularly suited for modeling inhomogeneous caterers[5-8]. Hybrid technique that combine Finite Element (volume formulation) and Boundary Integral (surface-equation formulation) have been found particularly attractive for antenna analysis [9]. Earlier applications of the hybrid methods are restricted to the antennas residing in cavities[10]. A recently developed formulation has the advantage of analyzing finite antenna arrays and includes edge effects [11].

A very important step in FE-BI analysis is the choice of the finite element geometry. The More commonly used element shapes are bricks, prisms and tetrahedrons[12]. The first code developed under AMRF S was ARRAY_TETRA and employs tetrahedral elements to analyse finite/infinite antenna arrays.

ARRAY_TETRA code was shown to be highly robust for simulating printed arrays (patches, slots). However its convergence is highly effected by the mesh. We experienced that different meshes may give convergent or divergent results for the same antenna element. Difficulties have been encountered when using tetrahedral to model Tapered slot antennas (TSAs) due to the mix of small and large elements within the same domain. We ported different solvers and preconditioners into ARRAY_TETRA to improve convergence (QMR, CGS, BICG). However, although preconditioning improved the condition of the mass matrix, in the case of irregular meshes, convergence still remained an unresolved issue. Thus, we examined different elements for TSA modeling.

Unlike ARRAY_TETRA, our new AMFIA code employs hexahedral elements and is therefore more suitable for modeling thin layers without a need to introduce elongated elements (i.e. elements with large aspect ratios). AMFIA is therefore expectable to be highly stable for simulating TSAs. AMFIA uses the exact FE-BI formulation mentioned above. Thus formulation is valid for finite arrays and thus edge effects are also included in the solution. This is an important feature when analyzing coupling between antenna elements or sub-arrays.

Another feature of the new code is the Mutilevel Fast Multiple Method for BI acceleration [13]. Memory is reduced dramatically when MLFMM is employed.
CODE DESCRIPTION

At this time AMFIA has the following unique features

- Hexahedral Elements for FEM (Distorted hexas can be used to model arbitrarily shaped structures)
- MLFMM (Multilevel Fast Multiple Method) for Boundary Integral acceleration
- Choice of iterative solvers
- Latest, most efficient preconditioners
- Antenna Radiation and Scattering calculations
- Antenna input impedance, gain, far field, near field calculations
- Mutual Coupling between elements in an array
- Mutual Coupling between Sub Array to Sub Array
- Near Field Visualization in PATRAN

This manual has 3 sections,

1. Mesh Generation
   1.1. Creating a database
   1.2. Creating points, lines, surfaces, volumes
   1.3. Assigning different materials to volumes
   1.4. Assigning metallic boundaries to surfaces
   1.5. Generating the mesh
   1.6. Assigning feeds and loads
   1.7. Exporting the mesh

2. AMFIA I/O description
   2.1. Input files (Amfia.in)
   2.2. Output files
   2.3. Plots

3. Post processing
   3.1. Importing near field data into PATRAN
   3.2. Visualization
1. MESH GENERATION

To show the mesh generation process we use a slot antenna as an example (see Fig.1). The antenna cavity is 10cm x 1cm x 3cm in -xyz directions, respectively. A feed with a single probe is located at x=7, y ε (-0.5, 0.5), z=3, assuming that the origin is located at point P (Fig.1.).

![Fig. 1 Slot Antenna Geometry](image)

1.1. Creating a database

In the following sections we demonstrate the mesh creation process from beginning to the end. At each step, a pictorial PATRAN window is given. In this report we used PATRAN V9.0. Though there are several versions of PATRAN, they are all similar. In this window, Fig. 2 shows the PATRAN window when PATRAN is first run. Enter a database name that you wanted to create. It is always convenient to use a name which represents the antenna geometry at hand.

![OK](image)

Another window will pop up (Fig.3). In this window enter the following setups for your newly created database.

- **Tolerance**: Based on Model
- **Model Dimensions (default is 10.0 )**: Choose a number depending your geometry
- **Analysis Code**: MSC/PATRAN
- **Analysis Type**: Structural

![OK](image)
Fig 2. Entering Database name

Fig 3. Setup configuration for database
1.2. Creating points, lines, surfaces, volumes

When the setup configuration complete an empty default_viewport is will appear. This is the place to create our geometry. Almost all mesh generation software packages use a hierarchical structure when creating geometries.

- Creating points
- Creating lines
- Creating surfaces
- Creating volumes

PATRAN also follows this order. To start creating points, click on top left corner of the screen (see Fig. 5). The geometry window will pop up on the left hand side. Make the following selections from the menu.

- Action (Create)
- Object (Point)
- Method (XYZ)

Note that the mesh generation procedure considered here is not unique. There are other ways of generating points, curves, surfaces and volumes in PATRAN. We choose the simplest way in this manual. Once the user gets more familiar with PATRAN software she/he can decide on their own.

We are going to start with point P(0,0,0) (see Fig. 1) and create all 8 corner points.

Press [Apply] after entering each of the following points (see Fig. 5).

- point 1 – [ 0 0 0 ]
- point 2 – [10 0 0 ]
- point 3 – [ 0 1 0 ]
- point 4 – [10 1 0 ]
- point 5 – [ 0 0 3 ]
- point 6 – [10 0 3 ]
- point 7 – [ 0 1 3 ]
- point 8 – [10 1 3 ]

To see the points created, click on DISPLAY and select Geometry. A “Geometric Attributes’ window will pop up. Point size can be increased to the desired size (see Fig. 6). Note that sometimes those extra windows might block the “default_viewport” and make the geometry invisible even after they have been moved to the side. It is recommended that the user “refresh” the viewport often (see Fig.6).
Fig. 7 shows the 8 corner points of the slot antenna. After the point creation is completed, we proceed to the next step to create the curves from those points. From the menu choose,

- Action (Create)
- Object (Curve)
- Method (Point)

Click on 2 points on the viewport to make the edges. If you turn on the “Auto execute” option you do not need to click the “Apply” button after each curve creation. When all curves are completed, the geometry should look like the one shown in Fig.8. The next step is to make surfaces from curves. To do so make the following selections.

- Action (Create)
- Object (Surface)
- Method (Curve)

Click on the “Starting Curve List” on the menu first. Then click on the desired curved on the viewport. Subsequently, click on the second curve which should be the parallel to the first curve. Both curves should be on the same plane in order to make a surface. Fig. 9 shows 6 faces (surfaces) created using 12 edges. To see the surface labels as in Fig.9 you should click the surface label icon in “Geometrical Attributes” window (See fig. 6).

Next and the last step in creating a geometry is to create solids from surfaces. To do so, activate the following comments from the menu.

- Action (Create)
- Object (Solid)
- Method (Face)
- Option (6-Faces)

Choose all 6 surfaces, and then the solid will be created automatically if “Auto Execute” is activated prior to the selection. If not, click on the APPLY button to create the solid. Note that we assumed that this antenna is filled with an homogenous dielectric. If different materials will be used in the antenna region, several solids should be created for this purpose. Different regions will correspond to different solids. After the solid has been created, the next step will be assigning boundary conditions to the surfaces.
Fig. 7. Creating Curves

Fig. 8 Creating surface - I
Fig. 9 Creating surface-II

Fig. 10 Creating Solids
1.3. Assigning materials to volumes

Note that the material assignment is only necessary if there are more than one materials in the FEM region. When the temperature value described below is was kept at default(0.) or set to 1, the program will assume that all FEM regions in the geometry are filled with the same material (material values will be read from the AMFIA.in file to be discussed in the following sections).

From the “Loads/BCs” menu choose the following options

- Action (Create)
- Object (Temperature)
- Type (Element Uniform)
- Current Load Case (Default)
- Target Element Type (3D)

Type a name (eg. Dielectric1) to the “New Set Name” section (see Fig. 10). Click on “Input Data”. A new window will pop up as shown in Figure 10. Keep the scale factor as default (1). Default temperature value is 0. Setting it to 1, 2, ..., N will define the N different region. After the temperature value is set, hit O.K. then from the main menu (Load/Boundary Conditions) click on “Select Application Region”. A third window will appear (see Fig. 11). Choose “Geometry” then click on the solids on the default viewport for the volumes which you want to assign as Dielectric1. Then click “add”. You will see the solid you had chosen in the application region. Finally hit “O.K.”. Follow the same steps for all other volumes (viz. Dielectric2, 3, 4 ... N).
Fig. 10 Assigning different materials
Fig. 11 Choosing application region
1.4. Assigning metallic boundaries to surfaces

Make the following setups from the “Loads and Boundary Conditions” menu.

- Action (Create)
- Object (Pressure)
- Type (Element Uniform)
- Current Load Case (Default)
- Target Element Type (3D)

Type a name (e.g. PEC) to the “New Set Name” section (see Fig. 12). Click on “Input Data”. A new window will pop up as shown in Figure 13. Set pressure to “0” and hit “O.K.” then select application region (Fig. 14). Choose “geometry” and select the solid faces (surfaces) on the default viewport in order to assign PEC boundary condition. If you choose

- Action (Plot Contours)
- Object (Pressure)
- Existing set (PEC)
- Data Variable (Pressure)
- Group Filter (Current Viewport)
- Select Groups (Default group)

and hit apply you can see the surfaces that are assigned as PEC boundaries (Fig. 15). **Note that in order to use this feature, a valid mesh should be created first. Refer to the next section to learn how to create the mesh.**
Fig. 12 Choosing application region
Fig. 13 Entering input data
Fig. 14 Selecting application region
Fig. 15 Viewing the PEC boundaries
1.5 Generating the mesh

After the geometry has been created and the boundary conditions are assigned, a mesh can be created. To do so click on the “Finite Elements’ icon on the viewport (see Fig.16) and make the necessary activations.

- Action (Create)
- Object (Mesh)
- Type (Solid)
- Global Edge Length (This number should be choosen according to the dimensions of your geometry, note that for an accurate FEM analysis usually this number should be \(<\lambda/15\).)
- Mesher (Isomesh)
- Element Topology (Hex27-This indicates that hexahedral elements will be used and each hexa will be defined with 27 nodes)
- Solid List (Click on the solid that you want to mesh)

After clicking on the “Apply” the mesh shown in Figure 16 will be generated. You can see the Node and element numbers in the “Node” and “Element ID list”.

Note that if your geometry has different sub-solid sections (if the FEM region has been represented with more than one solid), after creating the mesh, the “Equivalence” feature in the “Action” menu should be used (see Fig.17) to remove double nodes which are created at the junction of two neighboring solids. Also the nodes should be renumbered using “Renumber” option in the same menu.
Fig. 16 Creating the mesh
Fig. 17 Using Equivalence Feature
1.6 Assigning feeds and loads

In this section we demonstrate how “Feeds and Loads” are assigned in PATRAN for AMFIA.

To assign “Feeds and Loads” activate the following features in the “Loads and Boundary Conditions menu”:

- Action (Create)
- Object (Force)
- Type (Nodal)
- Current Load Case (Default)
- New Set Name (Enter a name ex. Feed1)

Click on “Input Data” and enter “**Force** as \(<1,0,0>\) (see Fig.18)”. Enter O.K., and then in the Loads and Boundary Conditions menu click on “Select Application Region icon”. A new menu will pop up (see Fig. 19). Choose FEM for Geometry Filtering. Click on the middle node of the edge on the “default _viewport” where you want your feed to be located, then the “add” and “O.K.” button then click on “Apply” in “Loads and BCs menu”. Note that each feed is assumed to be a probe defined on an edge (see Fig.20). If there are more than one edge at the feed location (see Fig.21), several feeds should be assigned.

For the loads, the only change that should be made is the “Force” which has to be \(<2,0,0>\).
Fig. 18 Assigning Loads
Fig. 19 Selecting Application Region
Fig. 20 Assigned feed view
Fig. 21 Multi Feeding
1.7 Exporting the mesh

To export the mesh file, click on the File menu located left upper corner of the viewport (see Fig. 22). When you select “export”, export menu will appear. Choose format as “Neutral” click on “Select All FEM” and “Select All LBC” and hit O.K. Enter a file name under the “Create new file name” section and press “Apply” (see Fig. 23). All FEM and LBC data will be written in a file named “the name of the file you entered”.out

Note that geometry data can also be saved in the IGS format. An IGS copy of the geometry is recommended before mesh generation and LBC assignments for future modifications to the geometry if needed. IGS format can be read by many other meshing programs. This file contains point, line, surface and solid data and their connectivity tables.
Fig. 23 Selecting Entities to be exported
2. AMFIA I/O description

In this section we describe AMFIA input and output files. The input and the output files are as follows

- Amfia.in
- Amfia.log
- Zin.txt
- Solver.txt
- Farfield.txt
- Unk.txt
- Efield.txt.

The contents of these files is given below for the slot antenna problem.

2.1 Input Files (Amfia.in)

This is the input file to the code. Note that there are some parameters used for the MLFMM calculations and some others used for solvers. Refer to the recommended values for those parameters.

```plaintext
! ==================================================================
! AMFIA INPUT FILE
! ==================================================================

gometry_file, "slotantenna.out" ! Name of the mesh file

gometry_file, "ltsa_mini.out"

gometry_file, "35ghztest.out"

gometry_file, "8x8.out"

gometry_file, "tower.out"

gometry_file, "halftower.out"

gometry_file, "baretower.out"

gometry_file, "ltsa_mini_mstrip.out"

gometry_file, "ltsa_realshort.out"

gometry_file, "ltsa_dual.out"

gometry_file, "ltsa10x10.out"

gometry_file, "ltsa_abs2.out"

gometry_file, "ltsa_mini_xpol.out"

gometry_file, "ltsa_a.out"

gometry_file, "cylinder.out"

gometry_file, "ltsa_mini4X3.out"

gometry_file, "slotpatch4.out"

drawing_scale, "cm" ! PATRAN drawing_scale can be cm,m,mm,mil,inch
```
analysis_type, 1  ! Problem (1-Finite Antenna, 2-Infinite Periodic Antenna-metal backed, 3-Infinite Periodic Antenna) - THIS SWITCH IS NOT YET IMPLEMENTED
Excitation_flag, 1  ! Excitation flag (1-feed, 2-plane wave)
RCS_flag, 1  ! RCS flag-(Ignored unless excitation flag is equal to 2)-(1-Bistatic/2-Monostatic)
FMM_flag, 1  ! Set FMM flag to 1 = on, 0 = off (0 gives complete fill in of BI matrix = takes more memory)
alphaL, 2.0  ! FMM parameter (2.0 for standard/accurate, 1.0 for greater memory savings/less accuracy)
deltaL, 1.732  ! FMM parameter (1.732 optimum)
start_frequency, 10e9  ! Start of frequency sweep (value in Hz)
end_frequency, 10e9  ! End of frequency sweep (value in Hz)
frequency_points, 1  ! Number of frequency points in sweep

Iterative_solver, 3  ! 1=cgs,2=bicg,3=bicgstabl,4=qmrft (3 recommended)
ell_param, 8  ! BICGSTAB(ELL) PARAMETER (range 1 to 8 recommended)
nfft, 1  ! PARAMETER FOR # OF TIMES TO FIT NEARFIELD MATRIX BEFORE FARFIELD (1 or 2 recommended, 0 turns off)
err_param, 10  ! QMR PARAMETER - CHECKS REAL RESIDUAL EVERY ERX ITERATIONS (VS THE UPPER BOUND) (10 recommended)
preconditioner, 0  ! 0=no preconditioner, 1=ilu(0), 2=diag (1 recommended for smaller problems but requires extra memory, 2 recommended for larger problems)
near_only, 0  ! (CAUTION) 1 = assume contribution outside FMM radius not significant and throw it away (0 RECOMMENDED)
Tolerance, 0.06  ! iterative solver stopping criterion ( tol <= ||b - Ax|| ) 0.02 recommended
Max_Iterations, 10000  ! max iterations for attempted convergence
save_unknowns, 1  ! 1 = save unknowns to disk
load_last_solution, 0  ! 1 = load last solution from disk as a starting point (if available)
use_previous_solution, 1  ! 1 = start from last - solved solution for next frequency/angle (1 recommended)
save_fields, 1  ! 1 = save near fields to disk (element excitations)

save_pattern_cuts, 1  ! 1 = save farfield pattern cuts to disk
theta_start, -180  ! start point of theta pattern cut
theta_end, 180  ! end point of theta pattern cut
theta_points, 361  ! number of points in theta cut
phi_start, 0  ! start point in phi pattern cut
phi_end, 90  ! end point in phi pattern cut
phi_points, 2  ! number of points in phi pattern cut

! FEM MATERIAL PARAMETERS
diel_1, (2.2,0.0d0)  ! if dielectric unassigned in patran, it defaults to this value)
diel_2, (1.0,0.0d0)  ! specified with element uniform tempureture boundary condition
do 2 in patran
diel_3, (1.0d0,0.0)
diel_4, (1.0d0,0.0)
diel_5, (1.0d0,0.0)
diel_6, (1.0d0,0.0)
mu_1, (1.0d0,0.0d0)  ! relative permeability
mu_2, (1.0d0,0.0d0)
mu_3, (1.0d0,0.0d0)
mu_4, (1.0d0,0.0d0)
mu_5, (1.0d0,0.0d0)
mu_6, (1.0d0,0.0d0)
! plane wave scattering parameters (only valid for excitation_flag = 2)
incident_phi_start, 90
incident_phi_end, 90
incident_phi_points, 1
incident_theta_start, 90
incident_theta_end, 90
incident_theta_points, 1

2.2 Output Files

AMFIA produces several output files used to calculate the necessary parameters for
radiation and coupling analysis and post processing. Following are the files and their
descriptions.

2.2.1 Amfia.log

This file creates a database of the problems that have been run. Geometry data, input data
and some further data regarding MLFMM, CPU, memory usage can be found in this file.

meshfile = slotantenna.out
drawing scale = cm
problem flag = 1
exciteflag = 1
resflag = 1
fmmflag = 0
solver flag = 3
loading last solution: 0
use previous solution: 1
tolerance = 5.999999865889549E-002
max iterations = 10000
preconditioner = 0
bicgstab(ell) param = 8
nfit = 1
qmr erx parameter = 10
alphaL parameter = 2.000000
FEM Unkowns = 27
BI Unkowns = 172
TOTAL Unkowns = 199
naux = 306
number of nonzeros in FE matrix = 175
Memory required for FE Matrix = 2.0026702E-03MBytes
21 Frequency points, from 1.0000 to 20.0000GHz with 0.9500GHz spacing.
Frequency Iteration: 1 Frequency: 1.000000 GHz
Clustering is done for 1 levels.
Number of clusters = 8
Level: 1 has 8 clusters, Max radius: 4.330127
Multipoles: 6
number of nonzeros in zbib2 matrix: 36
Memory required for zbib Matrix = 4.1197788E-04MBytes
FULL MATRIX SIZE = 199
Solution completed in = 0.1562500 sec.
Frequency Iteration: 2 Frequency: 1.950000 GHz
Clustering is done for 2 levels.
Number of clusters = 16
Level: 2 has 16 clusters, Max radius: 2.165064
Multipoles: 6
Level: 1 has 8 clusters, Max radius: 4.330127
Multipoles: 8
Number of non-zeros in M near-field matrix 29584
Memory required for M Matrix = 0.3385543 MBytes
Number of non-zeros in J near-field matrix 1548
Memory required for J Matrix = 1.7715048E-02MBytes
Total Memory for Matrix = 0.3562693 MBytes - memory saved: 0.0000000E+00%
number of nonzeros in zbib2 matrix: 36
Memory required for zbib Matrix = 4.1197788E-04MBytes
FULL MATRIX SIZE = 199
Solution completed in = 0.2031250 sec.

...
Multipoles: 25 25
Level: 1 has 8 clusters, Max radius: 4.330127
Multipoles: 44 44
Number of non-zeros in M near-field matrix 29584
Memory required for M Matrix = 0.3385543 MBytes
Number of non-zeros in J near-field matrix 1548
Memory required for J Matrix = 1.7715048E-02MBytes
Total Memory for Matrix = 0.3562693 MBytes - memory saved: 0.0000000E+00%
number of nonzeros in zbib2 matrix: 36
Memory required for zbib Matrix = 4.1197788E-04MBytes
FULL MATRIX SIZE = 199
Solution completed in = 1.250000 sec.

2.2.2 Zin.txt file

This file has the input impedance data for antenna applications. The first column is the node number for the feed and the second column is the frequency in Hz. The third column is the real part of the impedance and the last column is the imaginary part of the impedance.

<table>
<thead>
<tr>
<th>Node</th>
<th>Real Part</th>
<th>Imaginary Part</th>
</tr>
</thead>
<tbody>
<tr>
<td>23</td>
<td>1000000000.0000</td>
<td>2.5928</td>
</tr>
<tr>
<td>23</td>
<td>1950000000.0000</td>
<td>281.6845</td>
</tr>
<tr>
<td>23</td>
<td>2900000000.0000</td>
<td>23.9289</td>
</tr>
<tr>
<td>23</td>
<td>3849999872.0000</td>
<td>65.7498</td>
</tr>
<tr>
<td>23</td>
<td>4800000000.0000</td>
<td>73.6660</td>
</tr>
<tr>
<td>23</td>
<td>5750000128.0000</td>
<td>452.4126</td>
</tr>
<tr>
<td>23</td>
<td>6700000256.0000</td>
<td>98.2463</td>
</tr>
<tr>
<td>23</td>
<td>7649999872.0000</td>
<td>392.7809</td>
</tr>
<tr>
<td>23</td>
<td>8600000512.0000</td>
<td>1267.5032</td>
</tr>
<tr>
<td>23</td>
<td>9550000128.0000</td>
<td>269.3477</td>
</tr>
<tr>
<td>23</td>
<td>10499999744.0000</td>
<td>431.3055</td>
</tr>
<tr>
<td>23</td>
<td>11450000384.0000</td>
<td>323.2536</td>
</tr>
<tr>
<td>23</td>
<td>12400000000.0000</td>
<td>423.1139</td>
</tr>
<tr>
<td>23</td>
<td>13349999616.0000</td>
<td>481.1049</td>
</tr>
<tr>
<td>23</td>
<td>14300000256.0000</td>
<td>253.0912</td>
</tr>
<tr>
<td>23</td>
<td>15249999872.0000</td>
<td>338.6689</td>
</tr>
<tr>
<td>23</td>
<td>16200000512.0000</td>
<td>124.3804</td>
</tr>
<tr>
<td>23</td>
<td>17150000128.0000</td>
<td>87.8632</td>
</tr>
<tr>
<td>23</td>
<td>18100000768.0000</td>
<td>64.2989</td>
</tr>
<tr>
<td>23</td>
<td>19050000384.0000</td>
<td>51.8515</td>
</tr>
<tr>
<td>23</td>
<td>20000000000.0000</td>
<td>41.8530</td>
</tr>
</tbody>
</table>
### 2.2.3 Solver.txt file

Convergence data is kept in this file. The first column shows the number of iteration steps, the second column gives the total solution time after each iteration. The last column is the residual error.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.4375</td>
<td>0.0291</td>
</tr>
<tr>
<td>2</td>
<td>1.4844</td>
<td>0.0048</td>
</tr>
<tr>
<td>3</td>
<td>1.5156</td>
<td>0.0020</td>
</tr>
<tr>
<td>4</td>
<td>1.5469</td>
<td>0.0007</td>
</tr>
<tr>
<td>5</td>
<td>1.5938</td>
<td>0.0001</td>
</tr>
<tr>
<td>1</td>
<td>3.5781</td>
<td>0.0564</td>
</tr>
<tr>
<td>2</td>
<td>3.6250</td>
<td>0.0031</td>
</tr>
<tr>
<td>3</td>
<td>3.6562</td>
<td>0.0015</td>
</tr>
<tr>
<td>4</td>
<td>3.6875</td>
<td>0.0002</td>
</tr>
<tr>
<td>5</td>
<td>3.7344</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

....

....

....

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>53.8438</td>
<td>0.1357</td>
</tr>
<tr>
<td>2</td>
<td>53.8750</td>
<td>0.1370</td>
</tr>
<tr>
<td>3</td>
<td>53.9062</td>
<td>0.0970</td>
</tr>
<tr>
<td>4</td>
<td>53.9375</td>
<td>0.0995</td>
</tr>
<tr>
<td>5</td>
<td>53.9688</td>
<td>0.0965</td>
</tr>
<tr>
<td>6</td>
<td>54.0000</td>
<td>0.1026</td>
</tr>
<tr>
<td>7</td>
<td>54.0312</td>
<td>0.0935</td>
</tr>
<tr>
<td>8</td>
<td>54.0625</td>
<td>0.0767</td>
</tr>
<tr>
<td>9</td>
<td>54.0938</td>
<td>0.1018</td>
</tr>
<tr>
<td>10</td>
<td>54.1250</td>
<td>0.1399</td>
</tr>
<tr>
<td>11</td>
<td>54.1562</td>
<td>0.1448</td>
</tr>
<tr>
<td>12</td>
<td>54.1875</td>
<td>0.1205</td>
</tr>
<tr>
<td>13</td>
<td>54.2031</td>
<td>0.1101</td>
</tr>
<tr>
<td>14</td>
<td>54.2344</td>
<td>0.4003</td>
</tr>
<tr>
<td>15</td>
<td>54.2656</td>
<td>0.0865</td>
</tr>
<tr>
<td>16</td>
<td>54.2969</td>
<td>0.0530</td>
</tr>
</tbody>
</table>
2.2.4 Farfield.txt file

This file contains the farfield and antenna gain data. The first column is the observation angle, the second column is the angle phi, the third column is the gain in dB, the fourth column is the $|E_{\theta}|$ in dB, the fifth column is the $E_{\theta}$ (phase), the sixth column is $|E_{\phi}|$ in dB, the seventh column is $E_{\phi}$ (phase) and the last column is the free space wave number.

| Observation Angle | Angle Phi | Gain (dB) | $|E_{\theta}|$ (dB) | $E_{\theta}$ (Phase) | $|E_{\phi}|$ (dB) | $E_{\phi}$ (Phase) | Free Space Wave Number |
|-------------------|-----------|-----------|-------------------|---------------------|-----------------|-------------------|------------------------|
| -180.00000        | 0.00000   | 1.47751   | -269.82554        | 154.43714           | -32.36208       | 139.33551         | 0.20944                |
| -179.00000        | 0.00000   | 1.47590   | -269.79033        | 156.03487           | -32.36368       | 140.37765         | 0.20944                |
| -178.00000        | 0.00000   | 1.47110   | -269.75294        | 157.62317           | -32.36849       | 141.40947         | 0.20944                |
| -177.00000        | 0.00000   | 1.46308   | -269.71372        | 159.20093           | -32.37650       | 142.43065         | 0.20944                |
| -176.00000        | 0.00000   | 1.45186   | -269.67268        | 160.76887           | -32.38773       | 143.44090         | 0.20944                |
| -175.00000        | 0.00000   | 1.43742   | -269.63027        | 162.32351           | -32.40216       | 144.43993         | 0.20944                |
| -174.00000        | 0.00000   | 1.41977   | -269.58665        | 163.86681           | -32.41982       | 145.42744         | 0.20944                |
| -173.00000        | 0.00000   | 1.39888   | -269.54202        | 165.39741           | -32.44070       | 146.40316         | 0.20944                |
| -172.00000        | 0.00000   | 1.37476   | -269.49624        | 166.91604           | -32.46482       | 147.36682         | 0.20944                |
| -171.00000        | 0.00000   | 1.34740   | -269.44975        | 168.42117           | -32.49219       | 148.31815         | 0.20944                |
| -170.00000        | 0.00000   | 1.31678   | -269.40276        | 169.91251           | -32.52281       | 149.25692         | 0.20944                |
| -169.00000        | 0.00000   | 1.28289   | -269.35551        | 171.39113           | -32.55670       | 150.18285         | 0.20944                |
| -168.00000        | 0.00000   | 1.24572   | -269.30820        | 172.85503           | -32.59387       | 151.09573         | 0.20944                |
| -167.00000        | 0.00000   | 1.20524   | -269.26089        | 174.30439           | -32.63435       | 151.99532         | 0.20944                |
| -166.00000        | 0.00000   | 1.16145   | -269.21399        | 175.74066           | -32.67814       | 152.88141         | 0.20944                |
|                   |           |           |                   |                     |                 |                   |                        |
| 169.00000         | 90.00000  | -4.17444  | -14.52292         | 84.21229            | -46.00386       | 97.29349          | 4.18879                |
| 170.00000         | 90.00000  | -4.23721  | -14.58576         | 87.76211            | -45.96957       | 101.11928         | 4.18879                |
| 171.00000         | 90.00000  | -4.29767  | -14.64628         | 91.19469            | -45.94393       | 104.81904         | 4.18879                |
| 172.00000         | 90.00000  | -4.35557  | -14.70424         | 94.50432            | -45.92633       | 108.39607         | 4.18879                |
| 173.00000         | 90.00000  | -4.41069  | -14.75941         | 97.68561            | -45.91624       | 111.85298         | 4.18879                |
| 174.00000         | 90.00000  | -4.46281  | -14.81157         | 100.73358           | -45.91318       | 115.19227         | 4.18879                |
| 175.00000         | 90.00000  | -4.51175  | -14.86054         | 103.64362           | -45.91673       | 118.41626         | 4.18879                |
| 176.00000         | 90.00000  | -4.55732  | -14.90614         | 106.41165           | -45.92655       | 121.52712         | 4.18879                |
| 178.00000         | 90.00000  | -4.63777  | -14.98663         | 111.50775           | -45.96404       | 127.41763         | 4.18879                |
| 179.00000         | 90.00000  | -4.67241  | -15.02127         | 113.83032           | -45.99142       | 130.20122         | 4.18879                |
| 180.00000         | 90.00000  | -4.70319  | -15.05205         | 115.99988           | -46.02454       | 132.87960         | 4.18879                |
### 2.2.5 Unk.txt file

This file contains the solution of the matrix system. The first row has 3 numbers which identify to the number of nodes, number of elements and number of edges of the solved problem, respectively. Starting from the second row, the first column is the real part and the second column is the imaginary part of the edge unknowns.

```
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>441</td>
<td>30</td>
<td>199</td>
</tr>
<tr>
<td>1.1402947E-03</td>
<td>-1.0822310E-02</td>
<td></td>
</tr>
<tr>
<td>-3.1745189E-03</td>
<td>2.1807922E-03</td>
<td></td>
</tr>
<tr>
<td>-1.6768403E-02</td>
<td>4.7321372E-02</td>
<td></td>
</tr>
<tr>
<td>7.7323928E-03</td>
<td>-1.5241545E-02</td>
<td></td>
</tr>
<tr>
<td>2.8104907E-02</td>
<td>-0.1343380</td>
<td></td>
</tr>
<tr>
<td>-1.1974756E-02</td>
<td>4.7691219E-02</td>
<td></td>
</tr>
<tr>
<td>-4.6135327E-03</td>
<td>0.3158260</td>
<td></td>
</tr>
<tr>
<td>-1.3691365E-03</td>
<td>-0.1109501</td>
<td></td>
</tr>
<tr>
<td>-7.6575123E-02</td>
<td>-0.6964256</td>
<td></td>
</tr>
<tr>
<td>2.7324101E-02</td>
<td>0.2306932</td>
<td></td>
</tr>
<tr>
<td>-4.9592401E-03</td>
<td>0.3136652</td>
<td></td>
</tr>
<tr>
<td>-3.4611262E-03</td>
<td>-0.1103154</td>
<td></td>
</tr>
<tr>
<td>2.3482572E-02</td>
<td>-0.1298027</td>
<td></td>
</tr>
<tr>
<td>-9.7154286E-03</td>
<td>4.8963424E-02</td>
<td></td>
</tr>
<tr>
<td>-1.1000624E-02</td>
<td>4.4037141E-02</td>
<td></td>
</tr>
<tr>
<td>5.7663918E-03</td>
<td>-1.5348383E-02</td>
<td></td>
</tr>
<tr>
<td>2.5958717E-03</td>
<td>-8.9949882E-03</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-8.3478196E-03</td>
<td>1.0589331E-02</td>
<td></td>
</tr>
<tr>
<td>6.9731385E-02</td>
<td>-3.7788223E-02</td>
<td></td>
</tr>
<tr>
<td>3.9119616E-02</td>
<td>-2.1997938E-02</td>
<td></td>
</tr>
<tr>
<td>8.2472537E-04</td>
<td>-1.6914666E-02</td>
<td></td>
</tr>
<tr>
<td>-2.6502043E-02</td>
<td>1.0057846E-03</td>
<td></td>
</tr>
<tr>
<td>-1.4488138E-03</td>
<td>-8.5019311E-03</td>
<td></td>
</tr>
<tr>
<td>-2.5751784E-02</td>
<td>-3.9097685E-03</td>
<td></td>
</tr>
<tr>
<td>-1.0411549E-02</td>
<td>1.5343880E-02</td>
<td></td>
</tr>
</tbody>
</table>
```
2.2.6 Efield.txt file

This file is used for post processing. It is formatted such that the near field visualization can be made in PATRAN.

bistatic
  1 1 1 0 441 30 1 0 0 0
bistatic
  1 0.00000 0.00000
  0 441 1 1 1 0 0 0 0 0
bistatic
  1 1
0.000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
  2 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
  3 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
  4 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
  5 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00

......

......

438 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
439 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
440 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
441 1
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
2.3 Plots

The following we give some results from the sample slot antenna. Fig. 24 and 25 show the input impedance versus frequency. Two different meshes are used for the calculations. Coarse mesh is the one with edge length=1cm and the dense mesh is for 0.5cm. Note that coarse and dense meshes agree up to ~7.3GHz which is an expected value since the edge length for that frequency in the case of coarse mesh is ~λ/5 which is far more than λ/15 (a recommended value for FEM analysis). For frequencies <2.5GHz, the edge length is <λ/15 and that is the reason for almost identical results in 1-2.5GHz for 2 different meshes. Fig. 26 and 27 show the directivity in dB for E and H-planes as a function of the observation angle at 3 different frequencies.

![Graph showing input impedance as a function of frequency](image)

*Fig. 24 Input Impedance as a function of frequency (real part)*
Fig. 25 Input Impedance as a function of frequency (Imaginary Part)
Fig. 26 Directivity as a function of observation angle (E-plane)
Fig. 27 Directivity as a function of observation angle (H-plane)
3. Post-Processing

In this section we are going to show how to use “Efield.txt” file to visualize fields in PATRAN. PATRAN has an “restxt.exe” file for converting “*.txt” files to “*.res” files. This executable can be found in the \bin directory. Using this file Efield.txt file should be converted to Efield.res file. Note that if you are running the code for several different frequencies the last frequency data will be kept in Efield.txt file. If needed all frequency data can be written into this file with a minor code modification but in order to visualize the results several *.txt files should be created for each frequency before using “restxt.exe”.

3.1 Importing near field data into PATRAN

After the “*.res” files have been created, they can be ported into PATRAN. Fig. 28 shows the “import window”. In the import window choose

- Object (Results)
- Format (P/FEA2.res)
- Current Analysis Code (MSCNASTRAN).

Click on the file to be imported and press the “Apply” button. To see the results click on the results icon on the viewport (see Fig. 29) and set

- Action (Create)
- Object (Fringe)
- Select Result (bistatic, MAX DEFLECTION)
- Select Fringe Result (DISPLACEMENT, ROTATION)

and “Apply”. Note that the terminology is different since PATRAN was originally written for mechanical engineering analysis.

Finally, Fig. 30 and 31 show the efield variation for 3 and 5Ghz, respectively. These are the postscript (*.ps) files exported from PATRAN using the “print to a file” option.
Fig. 28 Importing results data
3.2 Visualization

Fig. 29 Viewing the results
Fig. 30  E-field variation at 3-GHz
Fig. 31  E-field variation at 5-GHz
References