035675-1-T

K. Sertel
J. L. Volakis

Users Manual for FMM-SWITCH

May 1999

35675-1-T = RL-2495

PROJECT INFORMATION

PROJECT TITLE: Parallel Computational Tools

REPORT TITLE: Manual for FMM SWITCH

U-M REPORT No.: 035675-1-T

CONTRACT
START DATE:
END DATE: November 1999

DATE: April 1999

SPONSOR:

Dr. Kuiechien Hill
AFRL
Wright Laboratory
WL/XPN Bldg. 254
2591 K. Street
WPAFB, OH 45433-7602
Phone: (937) 255-0277
hillkc@deneb.ws.afrl.af.mil

Dennis Andersh
DEMACO Inc/SAIC Inc
100 Trade Centre Dr., suite 303
Champaign, IL 61280-7237
Phone: (217) 355-4748
Fax: (217) 355-4749
dandersh@demaco.com

SPONSOR
CONTRACT No.: P.O. SW01073

U-M PRINCIPAL
INVESTIGATOR: John L. Volakis

EECS Dept.
University of Michigan
1301 Beal Ave
Ann Arbor, MI 48109-2122
Phone: (313) 764-0500 FAX: (313) 647-2106
volakis@umich.edu
http://www-personal.engin.umich.edu/~volakis/

PROJECT PEOPLE: K. Sertel, M. Smelyaskiy and J.L. Volakis

User's Manual for FMM-SWITCH Version 1.0

Kubilay Sertel and John L. Volakis
Radiation Laboratory
Department of Electrical Engineering and Computer Science
University of Michigan
Ann Arbor, MI 48109-2212, USA

May 10, 1999

Contents

1 Introduction

This document is intended to describe the compilation and execution of the code FMM-SWITCH developed at the Radiation Laboratory of the University of Michigan Electrical Engineering and Computer Science Department.

FMM-SWITCH is a Method of Moments (MoM) based Computational Electromagnetics (CEM) tool to simulate electromagnetic (EM) scattering scenerios involving arbitrarily curved perfect electric conductor (PEC) targets. It is implemented in Fortran 77. The MoM formulation used in the implementation of the code is based on the Electric–Field–Integral–Equation (EFIE), the Magnetic–Field–Integral–Equation (MFIE), and the Combined–Field–Integral–Equation (CFIE) formulations of EM scattering from PEC targets. To overcome the computation time length and memory requirements of the convensional MoM implementations ($O(N^3)$ and $O(N^2)$ respectively), the Fast Multipole Method (FMM) is implemented. The FMM reduces the computation complexity to $O(N^{1.5})$ per iteration (in iterative solution of the resulting linear system) and the memory to $O(N^{1.5})$. The reader is referred to the references [1, 2, 3] for further information on the formulations used in the implementation of the code.

The target geometry is represented in the code using curved quadrilateral surface patches which are defined by 9 points in space located on a topologically rectangular 3×3 grid [1]. The generation of target geometry is discussed in Section 2.2.

The input parameters related to the simulation should be given in the input file as outlined in Section 2.3.

Section 2.4 describes the output files generated by the code. RCS and induced surface current density data produced at the output can be visualized using the MATLAB and PATRAN visualization files as described in Section 4.

There are two versions of the code. The "non-symmetric" version located in **SWITCH/Non_Symmetric/** uses the whole target geometry, whereas the "symmetric" version in **SWITCH/Symmetric/** makes use of symmetry to model half of the target. This manual is based on the "non–symmetric" version. The difference between the two versions are mensioned whenever necessary.

2 Input and Output Files

The following subsections outline the I/O and visualization files used by the solver FMM-SWITCH.

2.1 Dimension File

Before compiling the solver, the dimensions of the data arrays used in the code must be approximately specified in the dimension file **main.dim** located in **SWITCH/Non_Symmetric/src** directory. The structure of the file is

```
parameter (

& NNodes = 5600,
& Nntri = 3600,
& Nunknowns = 4500,
& Ncluster = 60,
& NL = 15,
& Nnzbimat = 1500000
&
& )

C NNodes :  Number of NODES in target geometry
C Nntri :  Number of ELEMENTS in target geometry
C Nunknowns :  Number of UNKNOWNS in target geometry
C Ncluster :  Number of CLUSTERS in target geometry
C (Ncluster=sqrt(Nunknowns))
C NL : Number of MULTIPOLES in target geometry
C Nnzbimat :  Number of NONZEROES+1 in the near-field matrix
```

The number of NODES and ELEMENTS can be found in the first few lines of the geometry files. Number of UNKNOWNS is equal to the number of edges shared by two quad elements. It is equal to twice the number of ELEMENTS for a closed geometry. Number of CLUSTERS should be set to be Ncluster=sqrt(Nunknowns). Number of MULTIPOLES is computed by the code and it varies between 6 and 20. Number of NONZEROES in the near-field matrix is also problem dependent and an estimate to this number can be found as Nnzbimat=(10 to 20)*Ncluster*Nunknowns.

The whole code needs to be recompiled whenever this file is modified. The compilation is machine dependent and must be done using proper "make-files".

The dimension file for the "symmetric" version is located in **SWITCH/-Symmetric/src** and has the same structure as that of the "non–symmetric" version as explained above.

2.2 Geometry Files

The geometry files used in the code are the neutral PATRAN output files and the original SWITCH geometry files. Two seperate geometry files are required. One file contains the coordinates of the nodes of the quad elements forming the mesh, and the other contains the nodes of the elements. The geometry file containing the quad element nodes must be placed in the directory **SWITCH/Non_Symmetric /geom/elements** and the other geometry file must be placed in the directory **SWITCH/Non_Symmetric /geom/nodes**. These two geometry files must have the same name. It is crucial that the element normals be pointing outward (necessary when the MFIE or CFIE option is used). This can easily be checked in commercial mesh generation packages such as PATRAN. Example element and node files for PATRAN and IDEAS outputs are given below for the "non–symmetric" version.

For the "symmetric" version, the same structure is used. However, the geometry files contain only half of the symmetric geometry. The symmetry plane is specified in the input file. The assumption used is that the symmetric geometry lies on one side of the symmetry plane. The latter is assumed to be on one of the principle planes in cartesian coordinate system.

2.2.1 PATRAN Geometry Output

After the target mesh is generated using PATRAN, it is then exported in Neutral File format to two seperate files (one for nodes and the other for node connectivity as noted above). The format of the output node file from PATRAN is like the one below.

```
25 0 0 1 0 0 0 0 0
P3/PATRAN Neutral File from:  /tmp/PATRAN/dart2.db
26 0 0 1 8906 0 0 0 -1
05-Mar-99 16:01:00 3.0
1 1 0 2 0 0 0 0 0
-1.000000000E+0 -7.814555225E-9 3.184821606E-1
1G 6 0 0 000000
1 2 0 2 0 0 0 0 0
-1.000000000E+0 5.538074672E-2 3.129804432E-1
1G 6 0 0 000000
1 3 0 2 0 0 0 0 0
-1.000000477E+0 1.105192080E-1 3.056425154E-1
1G 6 0 0 000000
⋮
1 8906 0 2 0 0 0 0 0
-4.162685776E+1 -5.847183704E+0 1.057478547E+0
1G 6 0 0 000000
99 0 0 1 0 0 0 0 0
```

Not all of the lines above are needed by the FMM–SWITCH code. Clearly the line including the 3 decimal numbers refer to the node coordinates. The actual node number is given as the second number in the line above the coordinates. The format of the element connectivity file exported from PATRAN is

```
25 0 0 1 0 0 0 0 0
P3/PATRAN Neutral File from:  /tmp/PATRAN/dart2.db
26 0 0 1 8906 2226 0 0 -1
05-Mar-99 16:01:12 3.0
2 1 4 2 0 0 0 0 0
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
219 216 185 189 217 218 188 221 220
2 2 4 2 0 0 0 0 0
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
178 184 189 185 182 187 188 183 186
⋮
```

```
2 2226 4 2 0 0 0 0 0
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
1710 6229 6231 1712 6221 6230 6223 1711 6222
99 0 0 1 0 0 0 0 0
```

Again, not all the lines are used by FMM–SWICTH. Here, the lines containing the 9 integers gives the node numbers of the element. As noted above, the node file should be placed in the directory **SWITCH/Non_Symmetric /geom/nodes** and the element file should be placed in the directory **SWITCH/Non_Symmetric/geom/elements** with the same name for proper operation of the code. Also the geometry type in the main input file should be specified as "patran". This file should be placed in the directory **SWITCH /Non_Symmetric/input**. For the "symmetric" version, the node file should be placed in the directory **SWITCH/Symmetric /geom/nodes** and the element file should be placed in the directory **SWITCH/Symmetric/geom/elements** with the same name. Also the geometry type in the input file in **SWITCH /Symmetric/input** directory should be specified as "PATRAN".

2.2.2 SWITCH Geometry Output

The geometry files that are in the format of the original Northrop SWITCH code can also be read into the FMM–SWITCH solver. In this caes, the node file should have the format

```
962
1 0.4999994E+01 0.2838807E-13 -0.6494433E-06
2 0.1192099E-05 0.9999973E+00 0.0000000E+00
3 0.4566187E+01 0.1716829E+00 0.0000000E+00
⋮
962 -0.3427966E+01 -0.2119954E+00 -0.4965332E+00
```

Clearly, the first column in the above listing is the node number followed by its coordinates. The first line gives the total number of nodes (962 here). The element file should have the format

```
P3/PATRAN Neutral File /ogive480.db small 240 quad EMCC test
```

```
case, 11/24/97
240 0 240 240 0 0
0 480 480
0 0
0 0
1 9 7 0
3 72 27 45 118 71 1 70 26
0 0 1 2 0 3 4 0 0 0 0 0
2 9 7 0
4 74 28 46 119 73 3 72 27
0 0 2 5 0 6 7 0 0 0 0 0
⋮
240 9 7 0
754 783 755 932 962 931 870 892 871
0 0 -440 -386 0 480 479 0 0 0 0 0
```

The node file should be placed in the directory **SWITCH/Non_Symmetric /geom/nodes** and the element file should be placed in the directory **SWITCH/Non_Symmetric/geom/elements** with the same name for proper operation of the code. Also the geometry type in the input file in **SWITCH /Non_Symmetric/input** directory should be specified as "switch".

For the "symmetric" version, the node file should be placed in the directory **SWITCH/Symmetric /geom/nodes** and the element file should be placed in the directory **SWITCH/Symmetric/geom/elements** with the same name. Also the geometry type in the input file to be placed in the directory **SWITCH /Symmetric/input** should be specified as "switch".

2.3 Input File

The parameters (frequency, pattern cuts, etc) related to the specific simulation runs must be specified in **filename.input**. As noted earlier, this should be placed in **SWITCH/Non_Symmetric/input**. The file structure is

```
"switch"
"inches"
5.91
1.0
```

```
"bistatic"
2
90.0 90.0 0.0
0.0 180.0 0.5
0.001
500
```

The first line in this file specifies the format of the geometry file. Valid options are "switch" and "PATRAN". Others can be added in the future.

The second line gives the units of the geometry. Valid options are "inches" and "meters". When the unit is specified as "inches" the solver automatically scales the frequency so that the read geometry data are treated in units of "meters". The RCS results are unaffected.

The solution frequency is entered in GHz and is specified in the third line. This version of the solver only works at a single frequency.

The α parameter of the CFIE is specified in the fourth line as a real number. Valid values are $0.0 \leq \alpha \leq 1.0$ for closed targets. For open structures α must be set to 1.0.

The RCS pattern type is specified in line 5. Valid entries are "bistatic" and "monostatic".

The polarization of the incident plane electromagnetic wave is specified in line 6. Valid entries are 1 and 2 (1 for vertical polarization, 2 for horizontal polarization).

Line 7 specifies the range of observation angle θ measured from the vertical axis. The first value is the beginning value and the second is the ending value; the third value is the angle increment.

Line 8 specifies the range of observation angles ϕ with reference to the x-axis. Again, the first value is the beginning value and the second is the ending value; the third value is the angle increment.

Line 9 specifies the convergence tolerance for the iterative solver. The Conjugate Gradient Squared (CGS) is implemented this time but others can be used.

Line 10 specifies the maximum number of iterations prior to terminating th eiterative solver. If the solver does not converge in the specified number of iterations, it will stop. Output data will be generated, but depending on the error in the last step it may or may not reflect a correct solution.

For the "symmetric" version, the input file must contain the symmetry plane information, so its structure is slightly different from the above. It should be placed in the directory **SWITCH/Symmetric/input** and has the structure

```
"PATRAN"
"x-z"
"inches"
5.91
1.0
"bistatic"
2
90.0 90.0 0.0
0.0 180.0 0.5
0.001
500
```

The symmetry–plane is specified in the second line, the other information is the same as in the "non–symmetric" version. Valid entries for the symmetry–plane specification are

```
"x-y"
"y-x"
"x-z"
"z-x"
"y-z"
"z-y"
```

2.4 Output Files

The output files for the "symmetric" and "non–symmetric" versions are the same. The following output files are generated.

2.4.1 Farfield Radar-Cross-Section (RCS) Data

After solving the problem, the code computes the scattered field by the target and generates the RCS data in the following format

```
90.00000 0.00000 -52.49841 120.04202 -65.76239 108.26443
```

```
90.00000 0.50000 -52.52096 119.97476 -65.92950 108.37727
90.00000 1.00000 -52.56872 119.91295 -66.09913 108.49729
90.00000 1.50000 -52.64211 119.85621 -66.27121 108.62270
⋮
90.00000 179.50000 -19.82681 -30.93680 -63.67008 84.47012
90.00000 180.00000 -19.82989 -30.90826 -63.61202 83.64088
```

This is the specific output for the **ogive480** geometry for VV pol. The first column gives the θ–angle value (in deg.) and the second column gives the ϕ–angle value (in deg.) corresponding to the computed RCS values given in the remaining colmuns to the right. The third and fifth colums give the RCS results for vertical and horizontal field components in dBs. The forth and sixth colums are the corresponding phases of the vertical and horizontal field components in degrees.

2.4.2 Induced Surface Currents

For visualization purposes, the values of the induced surface current are also output. The file structure is

```
bistatic
1 1 1 0 962 240 1 0 0 0
bistatic
1 0.00000 0.00000 0 962 1 1 1 1 0 0 0 0 0 0
bistatic
1 1
6.0863093E-072.7060833E-035.1328743E-016.0863093E-07 ...
2 1
9.7520167E-035.8606099E-101.1953557E-019.7520167E-03 ...
⋮
962 1
1.4312185E+009.0374618E-023.6556860E-011.4312185E+00 ...
```

This is an auxiliary file to be used as an input to the converter **restxt**. The file contains the magnitudes of the induced surface current on the target at each node of the mesh. It can be converted into binary format using an auxiliary PATRAN file **restxt**. It can then be imported into PATRAN to visualize the surface currents induced on the target. This will be adressed in Section 4.

2.4.3 Solution Coefficients

This file is generated for completeness. It contains the solution vector (the unknown coefficient in the basis function expansion of the induced surface current) for the problem being analyzed. The structure is

```
1 1.58523E-03 5.14229E-04
2 1.55354E-02 5.81232E-02
3 -0.257247 -0.200015
4 5.44715E-02 0.349153
⋮
479 -0.523232 -0.763163
480 -0.623706 0.479390
```

The values in the first column are the associated unknown-number of the basis functions used (here they refer to the edges shared by two quad elements in the mesh). Second and the third columns are the real and imaginary parts of the coefficient. It should be noted that the basis functions are not numbered as they appear in the element file.

2.4.4 Clustering Information

To check the quality of the clustering for FMM implementation, the file **ccenters.xyz.filename** is output after clustering is done. The file structure is

```
1.874762 0.318430 0.675829 0.800198
3.048203 -0.124327 0.438653 0.733757
0.400361 0.680715 0.552857 0.829164
⋮
-0.462368 -0.557853 -0.695336 0.794405
-2.523491 -0.546767 -0.092770 0.851855
```

Each line refers to one of the clusters. The first column is the values of the maximum radii of the cluster; the second, third and forth columns are simply the x, y, and z coordinates of the cluster centers.

3 Executing the Solver

The following directories must be available/created prior to compiling the code
SWITCH /Non_Symmetric/geom/nodes
SWITCH /Non_Symmetric/geom/elements
SWITCH /Non_Symmetric/input
SWITCH /Non_Symmetric/src
SWITCH /Non_Symmetric/workspace
SWITCH /Non_Symmetric/bin
SWITCH /Non_Symmetric/datavis
SWITCH /Non_Symmetric/output

The explanation to execute the solver given below is based on the "non–symmetric" version of the code. The same instructions also apply to the "symmetric" version if the steps are carried out in the directory of the "symmetric" version (**SWITCH /Symmetric/**).

Once the geometry and input files are ready for the simulation, they must be put in the proper directories. The source files are located in the directory **SWITCH /Non_Symmetric/src**. Before compiling the code, the sparse matrix solver library needs to be compiled. The source files for this library are located in the directory **SWITCH/sparse**. When compiled, the file **sparse.a** is generated. This file is an assembly file and its compilation is machine dependent, hence it should be recompiled when the operation platform is changed. The generated library file is linked by the compiler of FMM-SWITCH. Hence, **sparse.a** should be copied in the directory **SWITCH/Non_Symmetric /src**. Once the dimension file **main.dim** is also ready, the user can proceed with its compilation. The code is compiled using the **make** utility with a proper **makefile** depending on the machine being used. After compilation the executable is automatically placed in the directory **SWITCH/Non_Symmetric/bin**. This directory must therefore be available prior to compilation.

To run the code, the user must be in the directory **SWITCH/Non_Symmetric**. Executing the available batch file runsw *filename*

will execute the solver.

If the dimensions in the file **main.dim** are less than needed, the code will prompt the information and stop. The dimensions should be fixed, the code should be recompiled and rerun. The sparse solver library file need not be recompiled unless the operating platform is changed.

The code uses the directory **SWITCH/Non_Symmetric/workspace** to write auxiliary files while it is running. The output files are written in this directory, and after the run is complete those files are moved to **SWITCH/Non_Symmetric/output**.

4 Data Visualization

The output files can be imported in e.g. Matlab for visualization. The following subsections outline the visualization tools for Matlab and PATRAN.

4.0.5 RCS Data Visualization

The Matlab file **ffphi.m** can be used to generate RCS plots on constant θ cuts using the data in the RCS output file generated and similarly **fftheta.m** can be used to generate RCS plots on constant ϕ cuts. Both files are located in the directory **SWITCH/Non_Symmetric/datavis**. The RCS data file should be copied in **SWITCH/Non_Symmetric/datavis** under the name **farfield.output**.

4.0.6 Surface Currents Visualization

The output file **current.txt.filename** can be converted into PATRAN binary results file using an auxiliary PATRAN file **PATRAN/bin/restxt**. Extensive information about the auxiliary files of PATRAN can be found in PATRAN's user's manual. The output file can then be imported into PATRAN and the magnitude of the induced surface currents can be plotted in PATRAN. To do so, the output file, e.g. **current.txt.dart2** should first be copied into **current.txt**. When the auxiliary file **restxt** is executed, it prompts for the type of conversion. The "txt–to–res" conversion option should be selected and the name of the file should be input as *current* (i.e.

without the .txt extension). The output will create the file **curent.res**. In the PATRAN Analysis menu, this file should be read into PATRAN. In the Results menu in PATRAN the surface currents can then be plotted.

4.0.7 Cluster Visualization

The quality of clustering can be checked using the Matlab file **plcntr.m** located in **SWITCH/Non_Symmetric/datavis**. The clustering file (e.g. **ccenters.ogive**) should be copied in **SWITCH/Non_Symmetric/datavis** with the name **ccenters.xyz**. The Matlab script **plcntr.m** can be used to plot the circles representing the clusters. This version only plots 2–D circles representing the clusters.

5 Example Runs

This section is intended to demostrate typical runs and outputs of the solver FMM-SWITCH.

5.1 The Ogive

In this example a PEC ogive is analyzed at 5.91 GHz. The geometry is the same as that used in the original SWITCH manual. The two geometry files are located in the directory **SWITCH/Non_Symmetric/geom** with the name **ogive480**. The dimension file for this problem given in **main.dim** is

```
parameter (

& NNodes = 962,
& Nntri = 240,
& Nunknowns = 480,
& Ncluster = 21,
& NL = 8,
& Nnzbimat = 113923
&
& )
```

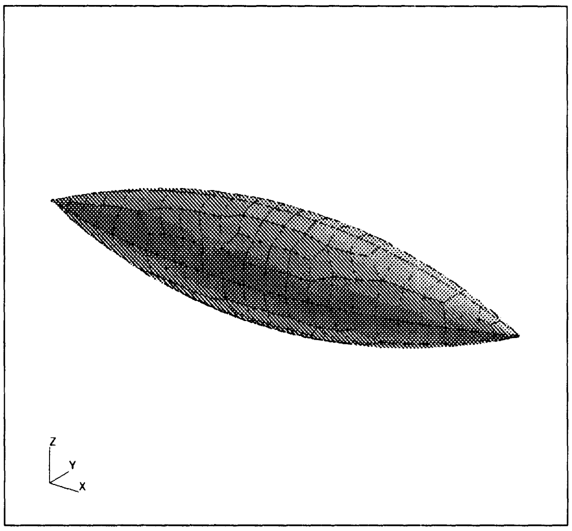

Figure 1: The Ogive Mesh

The input file for the ogive depicted in Fig 1 is

```
"switch"
"inches"
5.91
0.5
"bistatic"
2
90.0 90.0 0.0
0.0 180.0 0.5
0.001
500
```

Fig 2 depicts the RCS result for the ogive480 geometry.

5.2 Sphere

In this example a PEC sphere of radius 1m. is analyzed at 0.3 GHz. The geometry is generated in PATRAN. The two geometry files are located in the

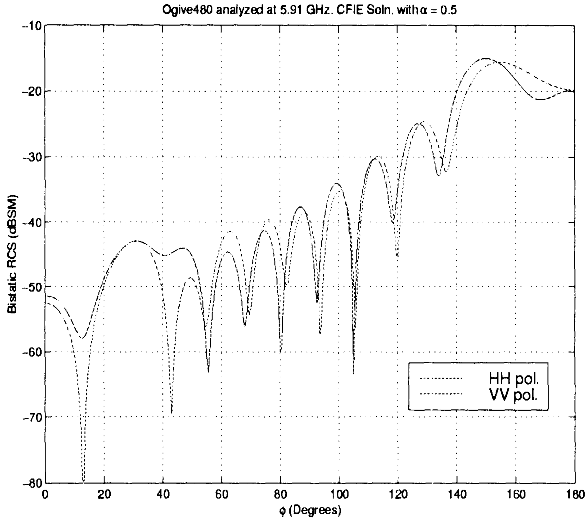

Figure 2: The RCS of the ogive at 5.91 GHz

directory **SWITCH/Non_Symmetric/geom** with the name **sphere05**. The dimension file for this problem given in **main.dim** is

```
parameter (

& NNodes = 5494,
& Nntri = 1373,
& Nunknowns = 2746,
& Ncluster = 52,
& NL = 7,
& Nnzbimat = 2448423
&
& )
```

The input file of the sphere depicted in Fig 3 is

```
"PATRAN"
"meters"
0.3
```

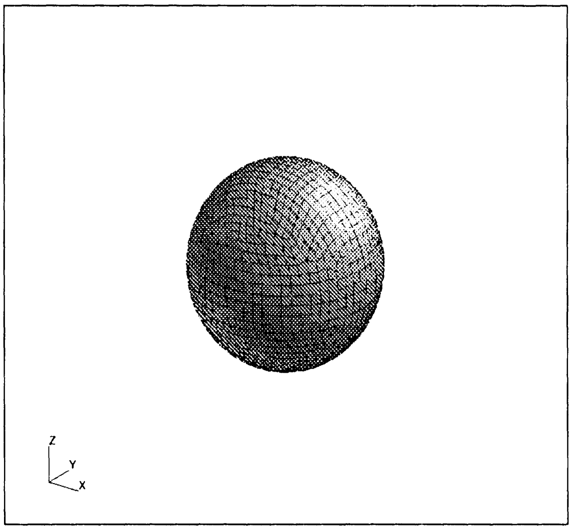

Figure 3: The Sphere Mesh

```
0.5
"bistatic"
2
90.0 90.0 0.0
0.0 180.0 0.5
0.001
500
```

Fig 4 depicts the RCS result for the sphere05 geometry and Fig 5 depicts the magnitude plot of the induced surface currents.

5.3 Dart

In this example the PEC dart is analyzed at 2 GHz. The geometry is generated using PATRAN. The two geometry files are located in the directory **geom** with the name **dart2**. The dimension file for this problem is given in **main.dim**. The input file of the dart depicted in Fig 6 is

```
parameter (
```

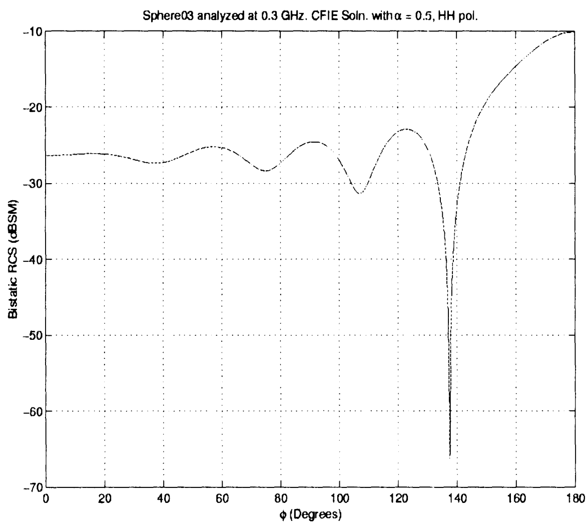

Figure 4: Bistatic RCS of the sphere at 0.3 GHz

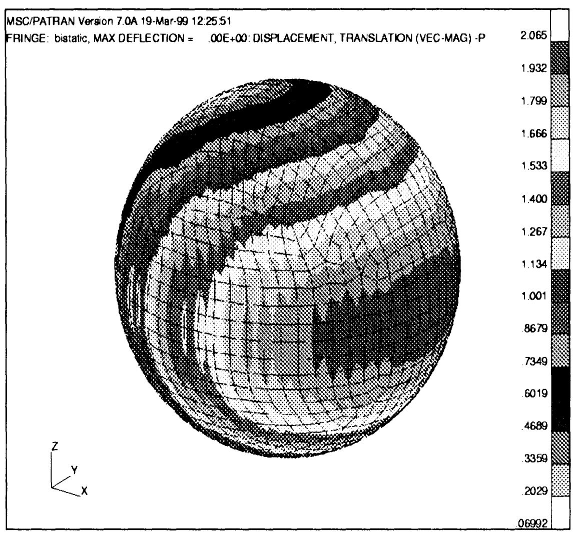

Figure 5: Induced surface currents on the sphere at 0.3 GHz

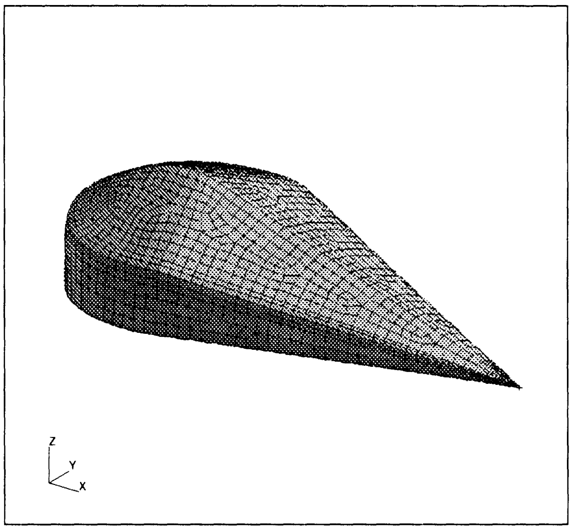

Figure 6: The Dart Mesh

```
& NNodes = 8906,
& Nntri = 2226,
& Nunknowns = 4452,
& Ncluster = 66,
& NL = 11,
& Nnzbimat = 4248015
&
& )
```

The input file of the dart depicted in Fig 6 is

```
"PATRAN"
"inches"
2.0
0.5
"bistatic"
2
90.0 90.0 0.0
0.0 180.0 0.5
```

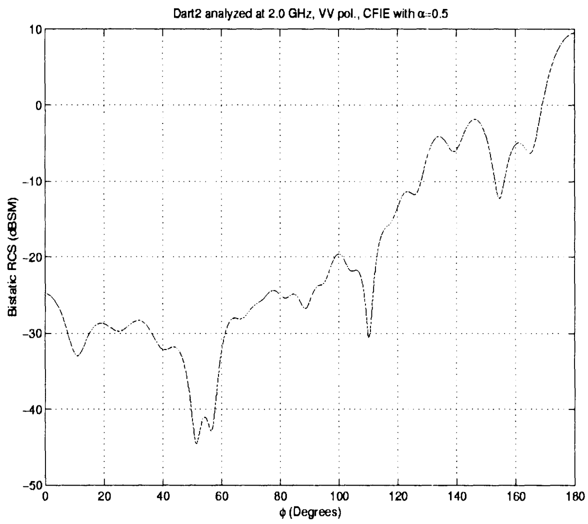

Figure 7: Bistatic RCS of the dart at 2.0 GHz

```
0.001
500
```

Fig 7 depicts the RCS result for the dart2 and Fig 8 depicts the magnitude plot of the induced surface currents.

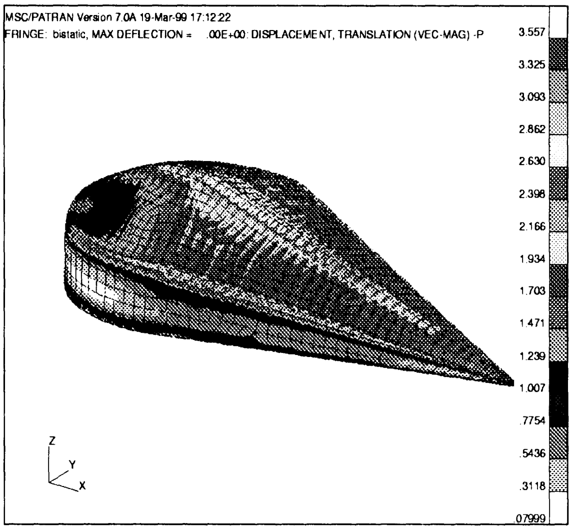

Figure 8: Induced surface currents the dart at 2.0 GHz, VV pol.

6 Example: Sphere

Given in this section is an example run to demonstrate the usage of the code. The sphere geometry is generated using PATRAN. The radius of the sphere is 1m. Two files are generated one of which containes the element nodes and the other contains the node coordinates. The file containing the element nodes reads as follows:

```
25 0 0 1 0 0 0 0 0
P3/PATRAN Neutral File from:  /tmp/PATRAN/sphere.db
26 0 0 1 5494 1373 0 0 -1
18-Mar-99 20:05:39 3.0
2 1 4 2 0 0 0 0 0
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
3 63 62 1 443 442 441 380 1105
2 2 4 2 0 0 0 0 0
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
4 64 63 3 445 444 443 381 1106
⋮
2 1373 4 2 0 0 0 0 0
```

```
9 0 0 0 .000000000E+00 .000000000E+00 .000000000E+00
4478 4477 4508 4479 5077 5155 5154 5079 5494
99 0 0 1 0 0 0 0 0
```

This file is copied to **SWITCH/Non_Symmetric/geom/elements/**. The file containing the node coordinates reads as follows:

```
25 0 0 1 0 0 0 0 0
P3/PATRAN Neutral File from:  /tmp/PATRAN/sphere.db
26 0 0 1 5494 0 0 0 -1
18-Mar-99 20:05:25 3.0
1 1 0 2 0 0 0 0 0
-1.000000000E+0-3.797475040E-20 8.687610093E-13
1G 6 0 0 000000
1 2 0 2 0 0 0 0 0
9.999999404E-1 -8.741933044E-8 0.000000000E+0
1G 6 0 0 000000
⋮
1 5494 0 2 0 0 0 0 0
-5.068947077E-1 7.210024595E-1 -4.724544585E-1
1G 6 0 0 000000
99 0 0 1 0 0 0 0 0
```

This file is copied to **SWITCH/Non_Symmetric/geom/nodes/**. For this problem the bistatic RCS of the sphere is computed at 0.3 GHz. for VV polarization. So the input file is as follows:

```
"PATRAN"
"meters"
0.3
0.5
"bistatic"
1
90.0 90.0 0.0
0.0 180.0 0.5
```

```
0.001
500
```

This file is copied to **SWITCH/Non_Symmetric/input/**. The dimension file **main.dim** in **SWITCH/Non_Symmetric/src/** reads as:

```
parameter (

& NNodes = 5494,
& Nntri = 1373,
& Nunknowns = 2746,
& Ncluster = 52,
& NL = 7,
& Nnzbimat = 2448423
&
& )

C NNodes :  Number of NODES in target geometry
C Nntri :  Number of ELEMENTS in target geometry
C Nunknowns :  Number of UNKNOWNS in target geometry
C Ncluster :  Number of CLUSTERS in target geometry
C (Ncluster=sqrt(Nunknowns))
C NL : Number of MULTIPOLES in target geometry
C Nnzbimat :  Number of NONZEROES in the near-field matrix
```

Then the sparse solver is compiled in **SWITCH/sparse/**. To compile the sparse solver, use

```
make f makefile_sun
```

on a SUN machine. The compiler generates a library file **sparse.a**. This file is then copied to **SWITCH/Non_Symmetric/src/**.

The solver is then compiled in **SWITCH/Non_Symmetric/src/** using

```
make f makefile_sun
```

Now the solver is ready to run. User must be in **SWITCH/Non_Symmetric/**. Type

```
runsw sphere05
```

to execute the solver. The following is output on the screen on a single processor SUN Ultra:

```
wavelength= 1.0000000000000 m.
nnode,nquad= 5494 , 1373
Geometry file read in = 0.710684
Elements paired in = 3.40323
nbasis= 2746
nclus= 52
Clusters formed in = 0.183546
Clusters paired in = 5.19966E-02
Translations computed in = 2.04550
Aggregations computed in = 39.1651
alpha= 0.50000000000000
Sparse matrix structure formed in = 0.493427
10% done.
20% done.
30% done.
40% done.
50% done.
60% done.
70% done.
80% done.
90% done.
100% done.
Near-field matrix filled in = 1318.00
Non-zeros in preconditioner= 80354
Near-field matrix factored in = 204.301
1 of 1 is being processed.
ITER= 1 ERROR= 0.29340707366055
ITER= 2 ERROR= 8.9859981758242D-02
ITER= 3 ERROR= 8.4873548373478D-02
ITER= 4 ERROR= 5.5020817083269D-03
ITER= 5 ERROR= 3.2430174817679D-04
ITER= 5 ERROR= 3.2430174817679D-04
Solution computed in = 15.1542
```

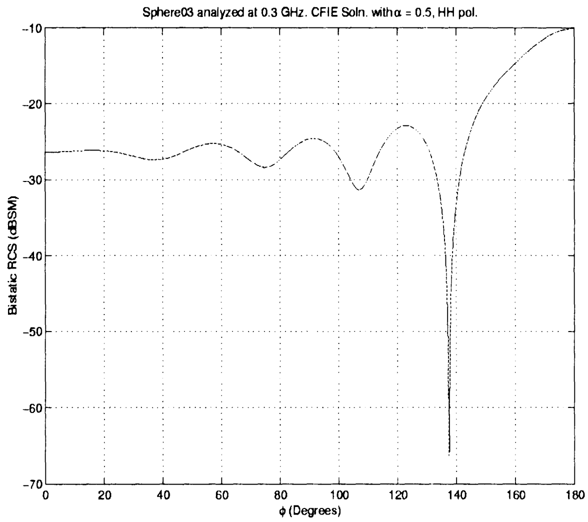

Figure 9: Bistatic RCS of the sphere at 0.3 GHz

```
Time per Iteration = 3.03085
```

After a successfull run, the output files are moved in the directory **SWITCH/Non_Symmetric/output**. Fig 9 is generated using the visualizetion tool **ffphi.m** in **SWITCH/Non_Symmetric/datavis/**.

7 Quick Reference

This section is a step-by-step reference to execute the code FMM-SWITCH. It is assumed that the directory structure of the code is already created on user's account. To use the "symmetric" version of the solver, users should execute the steps below in the directory **SWITCH/Symmetric/** as opposed to executing them in **SWITCH/Non_Symmetric/**. It sould be kept in mind that the geometry and input files of the "symmetric" version are different than those of the "non-symmetric" version.

7.1 Compilation of the Solver

- Go to directory **SWITCH/sparse**
- Compile the sparse solver by executing *make*
- Copy the assembly file **sparse.a** to **SWITCH/Non_Symmetric/src**
- Go to directory **SWITCH/Non_Symmetric/src**
- Design the parameter file **main.dim** subject to the geometry size.
- Type *touch*.f* so that each file includes the updated **main.dim**
- Compile the code by executing *make makefile_XXX* using the proper *makefile_XXX* where "*XXX*" refers to the machine type such as "sun", "hp", or "sgi".
- Now the executable **sw** is generated and moved in the directory **SWITCH/Non_Symmetric/bin**

7.2 Executing the Solver

- Put the geometry files in **SWITCH/Non_Symmetric/geom**. See Section 2.2 for more information.
- Select the run parameters and put the input file in **SWITCH/Non_Symmetric/input**
- Go to directory **SWITCH/Non_Symmetric**

- Execute the solver by typing *runsw filename*. The name of the geometry files and the input file should all be the same.

7.3 Checking the Output

- Go to directory **SWITCH/Non_Symmetric/output**
- Check that all four output files are generated after the simulation is completed. These are
  1. ccenters.filename
  2. coeff.filename
  3. current.txt.filename
  4. farfield.filename

7.4 Visualizing the Outputs

- Copy **farfield.filename** to **SWITCH/Non_Symmetric /datavis/farfield.output**.
- Copy **ccenters.filename** to **SWITCH/Non_Symmetric /datavis/ccenters.xyz**.
- Copy **current.txt.filename** to **SWITCH/Non_Symmetric /datavis/current.txt**.
- Go to **SWITCH/Non_Symmetric/datavis/.**
- Start up Matlab by typing *matlab*.
- In Matlab run the RCS visualization files to plot the RCS data. See Section 4.0.5 for more information.
- In Matlab run the clustering visualization files to plot the clustering data. See Section 4.0.7 for more information.
- Convert the file **current.txt** into **current.res** using **restxt** uitlity of PATRAN. See Section 4.0.6 for more information.

- Start up PATRAN by typing *PATRAN*.
- Import the geometry files into PATRAN.
- Import **current.res** into PATRAN using the Analysis menu.
- Choose the PATRAN "Results" menu to display the surface currents.

References

[1] G. E. Antilla and N. G. Alexopoulos, "Scattering from Complex 3D Geometries by a Curvilinear Hybrid Finite Element-Integral Equation Approach", *J. Opt. Soc. Am. A*, Vol. 11, No. 4, pp. 1445-1457, 1994

[2] J. M. Song and W. C. Chew, "Multilevel Fast Multipole Algorithm for solving Combined Field Integral Equations of Electromagnetic Scattering", Microwave and Opt. Tech. Letters, Vol. 10, No. 1, pp. 14-19, September 1995

[3] R. Coifman, V. Rokhlin, and S. Wandzura, "The Fast Multipole Method for the Wave Equation: A Pedestrian Prescription", IEEE Antennas and Propagation Magazine, Vol. 35, pp. 7-12, June 1993