

**SPARC-RF-05/001**  
**14 November 2005**

## **USE OF ANSYS FOR ELECTROMAGNETIC ANALYSIS OF RF CAVITIES**

M.Esposito and B.Spataro (*INFN/LNF*)

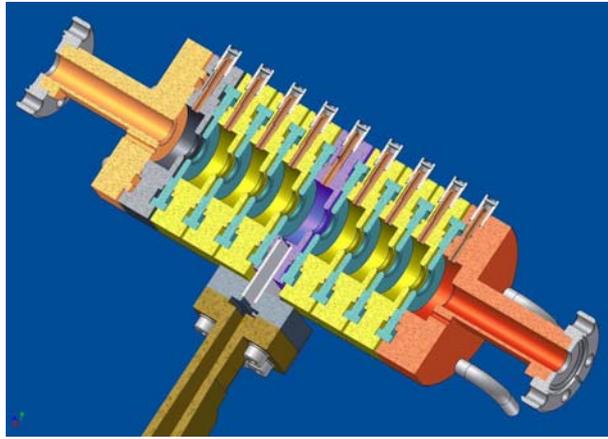
### **Abstract**

This paper describes the use of ANSYS [1] simulation software to perform high frequency electromagnetic analyses. A short description of the finite element methods is given, and an eigenmode analysis is applied to an RF cavity. Examples from this simulation are shown and some results are listed, plotted and discussed. A comparison with the values of the resonant frequencies obtained with a different code is presented.

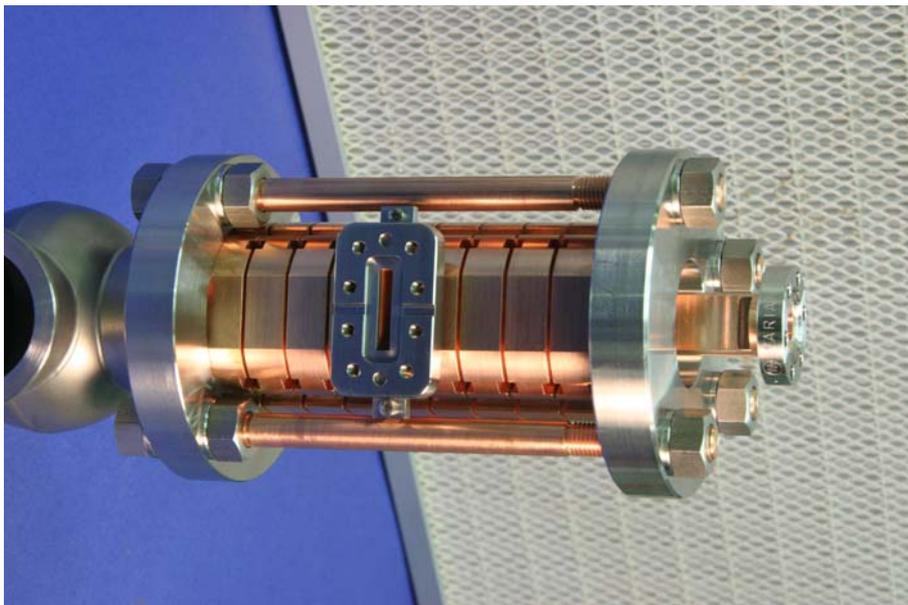


## 1 THE RF CAVITY

The cavity on which the simulation has been performed is a compact bi-periodic accelerating section, operating at the frequency of 11.424 GHz, for linearizing longitudinal phase space in the Frascati Linac Coherent Light Source (SPARC) [2]. The structure operates on a  $\pi/2$  standing wave mode with axial coupled cavities, and it is designed to obtain a 5 MV accelerating voltage. Figure 1 shows that the cavity is a bi-periodic structure with two sets of identical cells: 9 larger cells and 8 smaller cells. The 8 smaller cells have all exactly the same shape and dimensions (width  $l_s=1 \cdot 10^{-3}$  m , radius  $r_s=11.72 \cdot 10^{-3}$  m ), whilst, of the 9 large cells, the first and the last have a radius slightly smaller in order to take into account the effect of the beam tubes (width  $l_l=8.12 \cdot 10^{-3}$  m, radius  $r_{lc}=10.57 \cdot 10^{-3}$  m for the 7 central cells and  $r_{le}=10.55 \cdot 10^{-3}$  m for the first and the ninth one). Each cell is separated from the next by an iris wide  $l_i=2 \cdot 10^{-3}$  m and with a radius  $r_i=4 \cdot 10^{-3}$  m. The total length of a period is therefore  $L= l_l+ l_s+2 l_i=13.12 \cdot 10^{-3}$  m.



**FIG. 1:** Cross section of the cavity.



**FIG. 2:** Picture of the cavity realized at the LNF.

## **2 BASIC CONCEPTS OF THE FINITE ELEMENT METHOD**

There are several engineering problems for which we are not able to find exact solutions. This incapacity may derive either from the complex form of the differential equations that represent the problem that is being studied, or from the difficulty of imposing particular boundary and initial conditions that need to be respected.

To solve these problems, a solution might be the use of numerical approximations. Unlike analytical solutions, that represent the exact behaviour of the system in any point within the system itself, numerical solutions approximate the behaviour by considering a finite number of variables that correspond to the value of the function we are looking for in a discrete number of points, called nodes.

The first step in a numerical procedure, once the problem has been defined, is the discretization. This operation consists in dividing the medium, or structure, in a finite number of nodes and small sub-regions, that take the name of elements.

There are two main categories of numerical methods:

- 1) Finite difference method;
- 2) Finite element method;

With the finite difference method the differential equation is written for each node, and the derivatives are substituted by finite differences. This kind of approach gives as result a set of simultaneous linear equations. Although this method is quite simple to understand and to use for solving not too complex problems, its application becomes rather complicated for the solution of problems with more complex geometries and boundary conditions, as well as with problems in which the materials do not have isotropic properties.

The finite element method uses integral formulations, instead of finite differences, to create a system of algebraic equations. More specifically this method assumes a continuous function to represent the solution for each element. The complete solution is then generated by correcting or assembling the individual solutions, allowing for continuity at the interelemental boundaries. The finite element method is a numerical procedure that may be used to solve a great variety of engineering problems: static, dynamic, linear or non-linear problems of structural, thermal, fluid dynamics and electromagnetic analyses.

## **3 ANSYS**

ANSYS is a finite element numerical code with a great number of options, with which it is possible to perform structural, thermal, fluid dynamics and electromagnetic analyses. The recent versions of this software have many windows, among which a graphical user interface (GUI), dialogue boxes and tool bars. The commands may be given either with the GUI or by writing directly the command in the command line. The use of the graphical interface is easier to learn, more intuitive, and in some cases quicker. Anyway it is often preferable to use the command line for several reasons. First of all it gives the possibility to implement packets of functions to be executed in an iterative or sequential way, secondly it gives the possibility to create macros able to execute particular functions for which direct

commands do not exist, and finally it gives the possibility to store and to transfer a project or an analysis in the form of a simple text file.

The ANSYS simulator can also perform coupled-field analyses involving two or more different disciplines, i.e. it is able to perform a complete analysis of a structure studying different physical variables and eliminating the problem of transferring data from an analysis to the next.

Even though this direct analysis is very useful, easy to use and fast, it does not give the possibility to check the results of the simulation, and it might therefore lead to some perplexity on its reliability. For this reason it is preferable, whenever possible, to perform separately each single analysis referring to different fields of physics, because a procedure of this sort is definitely easier to keep under control.

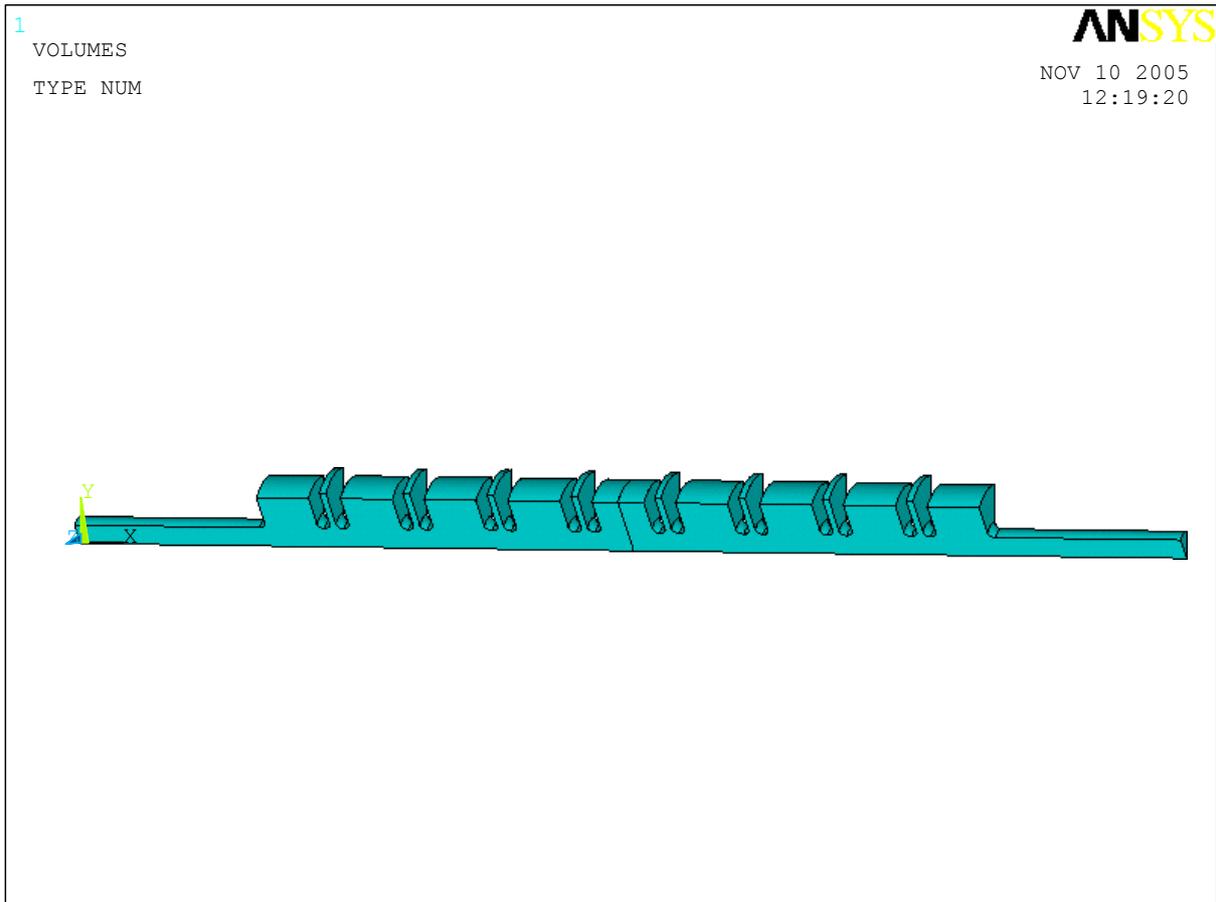
## **4 SIMULATIONS WITH ANSYS**

### **4.1 Creating the Geometry**

The first step in the process of simulation with ANSYS is obviously the creation of a model. Often a model of the desired geometry already exists, and it is usually faster to import this geometry into ANSYS, rather than creating the geometry internally. Importation may be accomplished using a transfer format, such as IGES, or by opening the native file directly into ANSYS, if the capability exists. If a model does not already exist, it can be created directly with the ANSYS preprocessor.

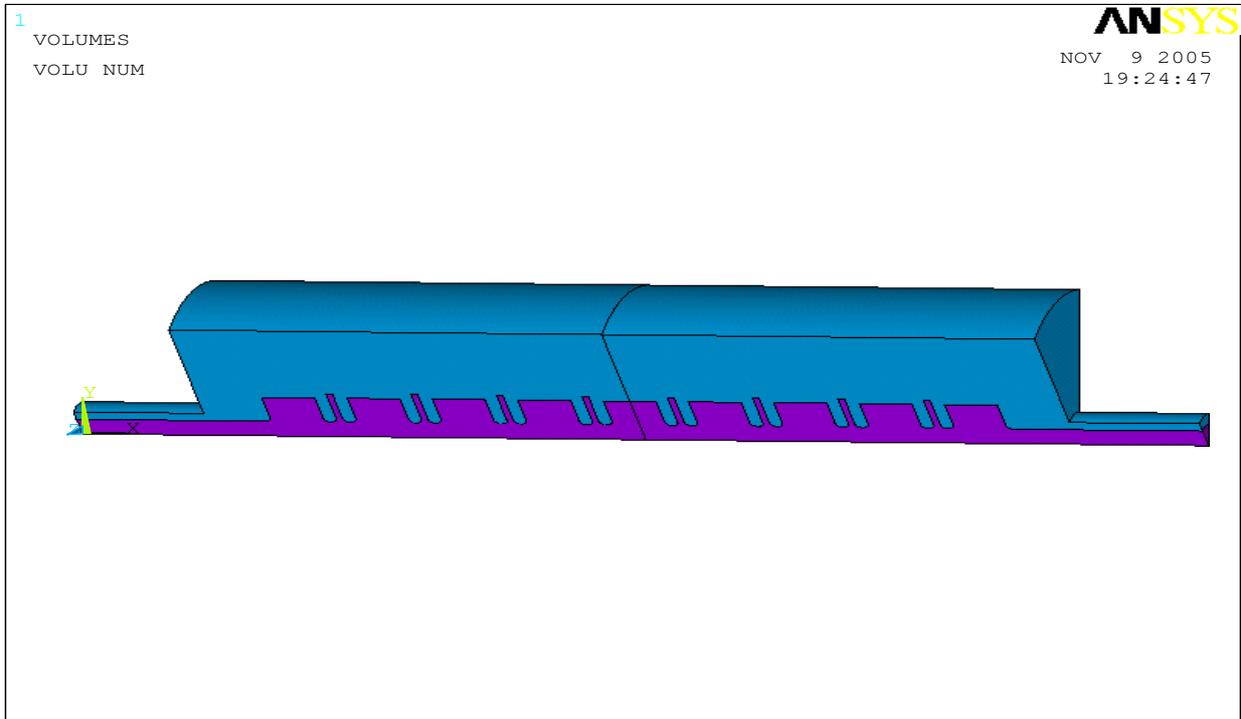
It is generally a good thing to simplify as much as possible the model in order to reduce the complexity, and most of all, the time necessary to calculate the solution. The simplifications may include suppressing extraneous features, removing components that are not needed in the analysis and taking advantage of the geometric symmetries of the model to cut it and reduce its size.

In our analysis concerning the bi-periodic SPARC cavity, as we can see in Fig.3, since our model has an axial symmetry, we have considered only an angular slice of  $45^\circ$  of the entire cavity. Being our cavity a periodic structure, as described in paragraph 1, we have created the central cells with a cyclic operation using a “dowhile” command. We have also considered the fact that the cavity can be symmetrically divided in two halves by a plane perpendicular to the axis and intersecting it in its midpoint. This consideration gave us the possibility to build first only half the cavity and then reflect it around the symmetry plane. It is important to glue the two half-volumes obtained with this procedure and subsequently to add them in order to have the cavity represented by only one volume.



**FIG. 3:** Model of the cavity created with the ANSYS preprocessor.

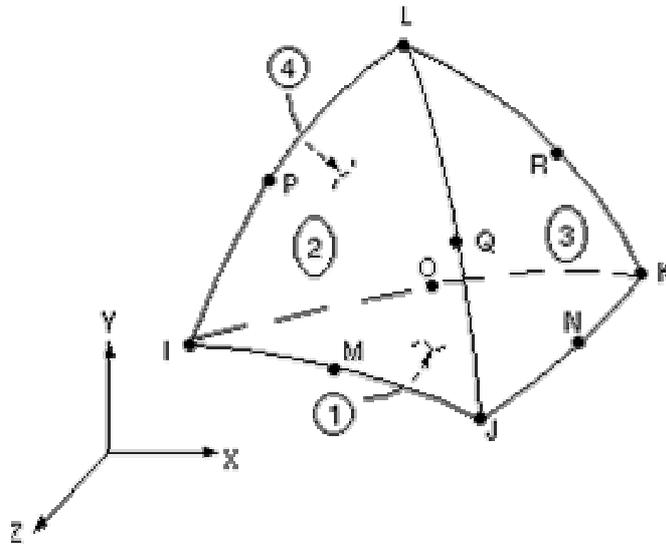
We may also consider the copper structure containing the cavity, which is important if we want to take into account the power loss due to the non-perfect conductivity of copper, in order to perform also a thermal analysis and to design a cooling system. The thermal analysis will not be discussed in this paper, it may however be important to prepare it as a possible, and generally necessary, future step. For this reason it may be useful to consider the copper structure, to specify its conductivity and to create a mesh with the condition that the nodes lying on the surface between the vacuum cavity and the copper structure belong both to elements of the cavity volume and of the copper volume, in other words it is important that the meshes are perfectly matched. By doing so it will be possible to evaluate the heat generated by the power losses and to consider it as a heat flux boundary condition for the thermal analysis. The entire model will then appear as shown in Fig.4.



**FIG. 4:** Model of the cavity with the copper (ANSYS preprocessor).

#### **4.2 Element Type and Material Definition and Meshing**

Several factors have to be considered in order to properly choose the element type, first of which is the analysis type (structural, thermal, fluid dynamics or electromagnetic). ANSYS has a certain number of different element types for each one of these fields and for coupled-field analysis. The model dimension is obviously important (2D or 3D). It is also important to establish which parameters or variables we need to measure or observe, and these will have to correspond to the degrees of freedom of the element type chosen. In our case we have opted for the element type HF119 that is a 3D tetrahedral element for high frequency electromagnetic analysis, because it is the only one that allows to obtain a mesh automatically. In fact the other element type for 3D high frequency electromagnetic simulations, the HF120, that has the shape of a brick, requires necessarily a manual meshing.



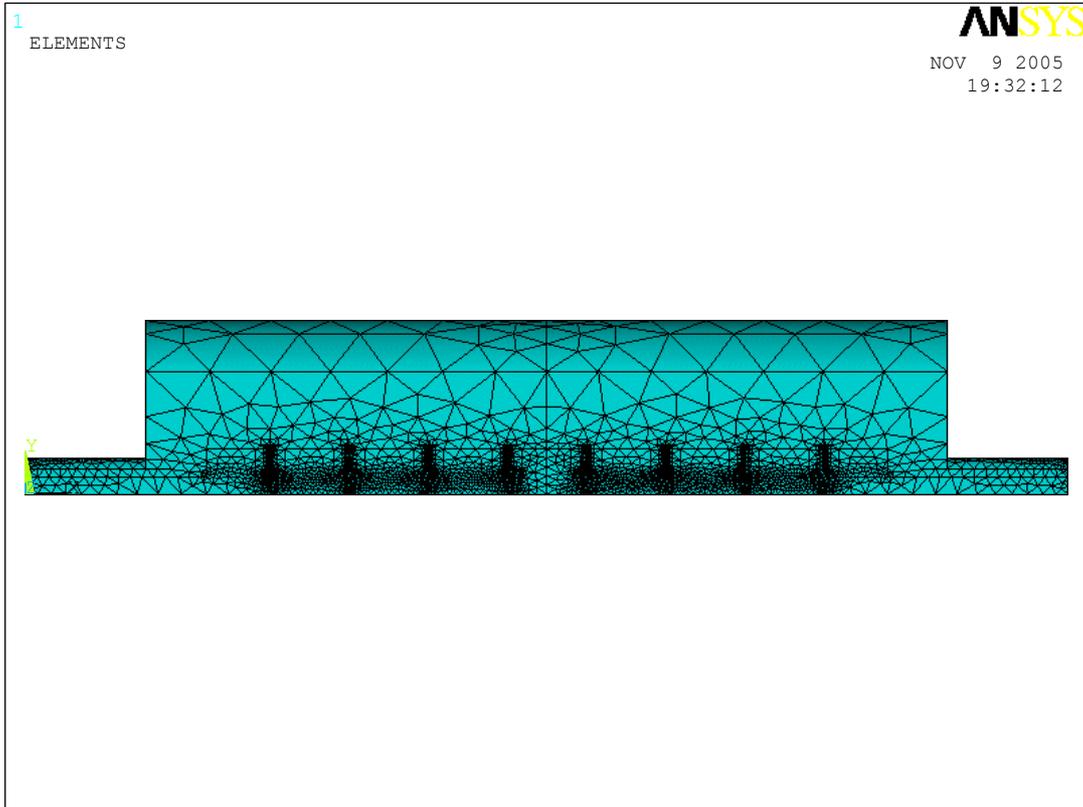
**FIG. 5:** Element Type HF119.

Once the element type has been chosen we have to set the material properties. In ANSYS, high frequency electromagnetic analyses default to MKS units, and the properties of the resonating material are given as ratios to free space permeability and permittivity. For the cavity the material is vacuum, so the relative permeability  $\mu_r$  and permittivity  $\epsilon_r$  are both 1. Also for copper we can approximately consider  $\mu_r = \epsilon_r = 1$ . We have given as input also the resistivity of copper ( $\rho = 1.724 \cdot 10^{-8} \Omega \cdot m$ ).

The next step is the division of the volume in elements, or meshing. Before issuing the mesh command, the element dimensions have to be established. This choice has to be made on the base of different factors, among which the size of the structure, the characteristic dimensions of the phenomenon being studied (as for example in our case the frequency, which is inversely proportional to the wavelength), and of course the level of accuracy desired. It is clear, as a matter of fact, that with a greater number of elements, i.e. with smaller dimensions of the elements, the analysis will generally be more accurate. But it is also true that very accurate analyses require quite long intervals of time for the solution to be done and may be prone to numerical errors. It is therefore necessary to find the right balance between the need for accuracy and reliability of results, and the need to find a correct solution in a short time. Usually the best way to obtain an acceptable mesh is to follow an iterative process. This means that you can start with a reasonably large mesh and carry out a first simulation. You may then perform some other subsequent analyses decreasing the size of the mesh step by step.

In many cases it is advantageous to create a fine mesh in critical areas. This is what happens for example when a certain variable requires a quite accurate evaluation in some areas, while in other areas it is not so important and we can use a larger mesh in order to reduce run time and memory usage. A simple way to achieve mesh variation is to divide the entire volume in two or more sub-volumes depending on the needed local mesh size.

We can see in Fig.6 that in our simulation we have created a finer mesh in the cavity volume and a larger mesh in the copper volume.



**FIG. 6:** Mesh tetrahedra inside the structure.

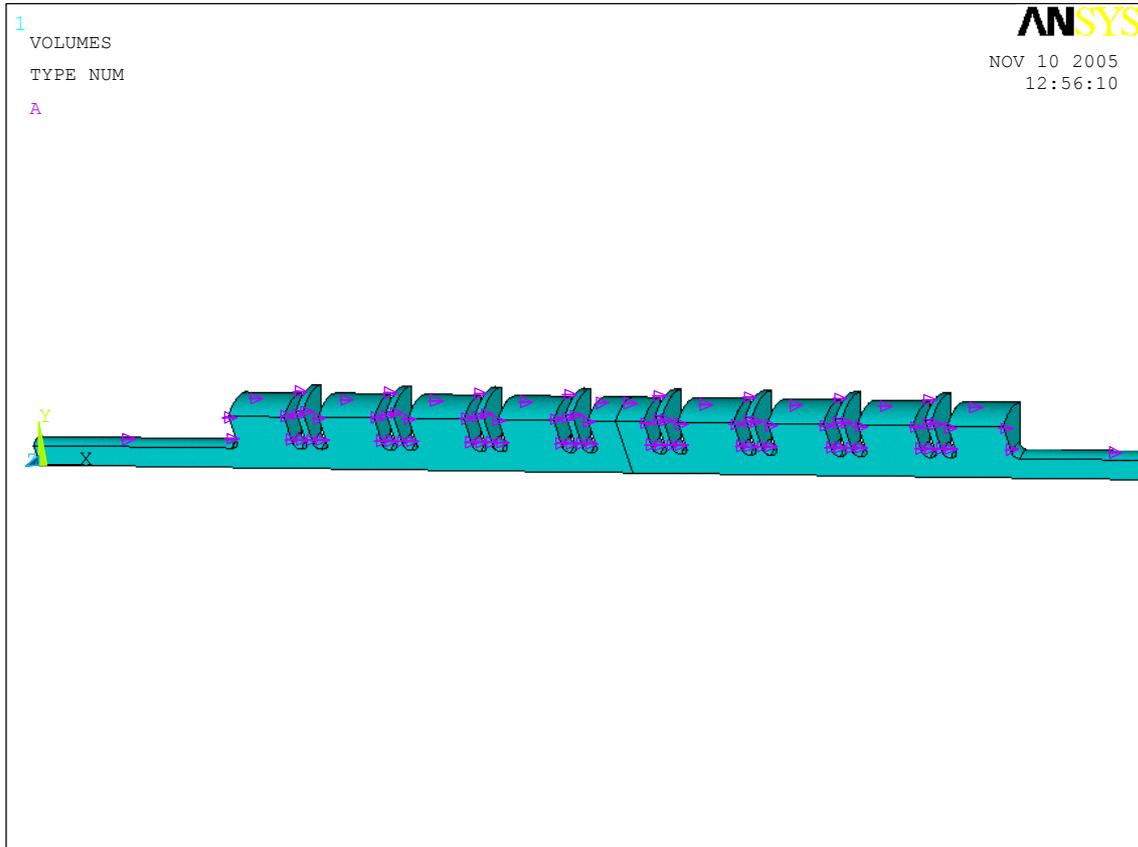
### 4.3 Boundary Conditions

In the case of a modal analysis, i.e. in the study of the eigenmodes of a structure, there are no external loads or excitations, and the only possible boundary conditions are:

- electric wall (the tangential component of the electric field  $E$  is 0);
- magnetic wall (the tangential component of the magnetic field  $H$  is 0).

ANSYS sets by default the condition of magnetic wall on all the surfaces that are not defined as electric walls.

In our simulation we have considered as an electric wall the surface of separation between the cavity and the copper structure (Fig.7), leaving the condition of magnetic walls on all the other surfaces that are symmetry planes.



**FIG. 7:** Boundary conditions, electric wall.

#### 4.4 Solution

After having set the boundary conditions, ANSYS is ready to solve the problem. Before issuing the solve command the analysis type must be set to modal and the frequency range must be defined. It is also important to specify that the modes need to be extracted. By doing so it is possible to have as result, not only the frequencies of the eigenmodes, but also the nodal and element solution for E and H fields.

In our cavity the target frequency for the  $\pi/2$  mode is 11.424 GHz so we have set a range of frequencies with a lower limit of 11GHz and an upper limit of 12GHz, and we have specified the extraction of 25 modes. Actually we expect to find 17 eigenmodes, since the complete cavity has 17 cells (9 longer cells and 8 shorter cells), and also because simulations with other software [3] on the cavity have proved that it has 17 normal modes in the range of frequencies that goes from 11.2 GHz to 11.6 GHz (Tab.1). But we have set these conditions for the solution to be sure that ANSYS does not find more than 17 modes, some of which could fall outside of the expected range.

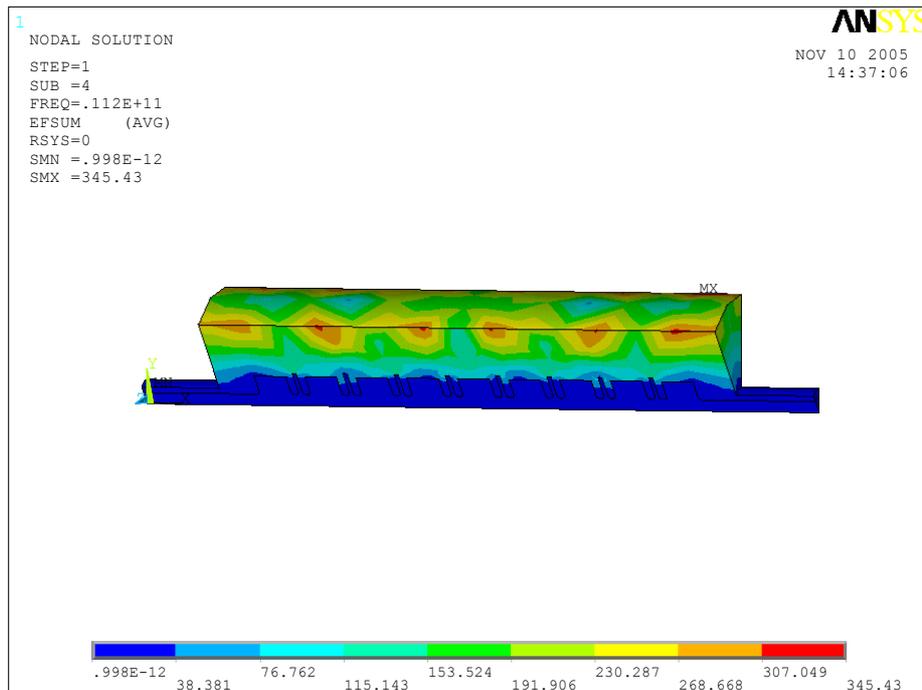
<b>MODE</b>	<b>F<sub>0</sub> (GHz)</b>
1	11.207
2	11.218
3	11.236
4	11.259
5	11.300
6	11.328
7	11.361
8	11.392
9	11.459
10	11.473
11	11.500
12	11.526
13	11.554
14	11.563
15	11.581
16	11.591
17	11.597

**TAB.1:** Resonance frequencies of the cavity calculated by a GdfidL analysis [4] with  $\sim 7 \cdot 10^6$  cubic elements

Unexpectedly our analysis has found 21 modes (Tab.2). By plotting the E field we can study these modes and try to understand the reason of this result that may seem a non-sense. Viewing these plots we have actually discovered that it was a conceptual non-sense, because 4 of the 21 modes have the electric field  $E \neq 0$  in the volume of the copper structure (Fig.8), where we know that the field should be negligible.

MODE	F <sub>0</sub> (GHz)
1	11.043
2	11.182
3	11.196
4	11.202
5	11.213
6	11.239
7	11.268
8	11.302
9	11.336
10	11.374
11	11.410
12	11.441
13	11.470
14	11.499
15	11.521
16	11.541
17	11.555
18	11.567
19	11.573
20	11.710
21	11.879

**TAB.2:** Resonance frequencies of a 45° slice of the cavity with the copper structure.



**FIG. 8:** E≠0 in the volume of the copper structure

These non-sense results are probably due to a too coarse mesh. A possible way to eliminate this problem would be to repeat an analysis with a finer mesh, i.e. increasing the number of elements by issuing the “refine” command after a first mesh operation. Unfortunately in our case this has not been possible because we were already close to the maximum number of nodes possible for ANSYS (120000).

Our alternative solution has been to reduce the size of the model. There were different possible solutions to achieve this. The first that we have considered was to eliminate the copper structure and carry out another analysis only on the cavity volume. The second was to reduce the angular sector of the model; as we said before, and as we can see in figures 3 and 4, we considered a slice of 45°, and this angle could be reduced to a lower value. A third possibility would have been to cut the model in half with a plane of symmetry perpendicular to the axis, and of course many other solutions were possible.

The analysis performed only on the cavity has confirmed that there are 17 normal modes and that the values of the resonating frequencies are pretty much as we expected (Tab.3)

<b>MODE</b>	<b>F<sub>0</sub> (GHz)</b>
1	11.183
2	11.195
3	11.213
4	11.240
5	11.267
6	11.302
7	11.337
8	11.374
9	11.411
10	11.441
11	11.470
12	11.498
13	11.520
14	11.542
15	11.555
16	11.567
17	11.591

**TAB.3:** Resonance frequencies of the cavity analyzed without the copper structure (45° slice)

The analysis performed on a slice of 15° of the cavity and the outer copper structure has given as result 19 modes (Tab.4), 2 of which (the first and the eighteenth one) are non physical, of the sort of the one shown in Fig.8. This further confirms our hypothesis that the “exceeding nonsensical modes” revealed may depend on a too coarse mesh.

MODE	F <sub>0</sub> (GHz)
1	11.148
2	11.176
3	11.190
4	11.208
5	11.233
6	11.262
7	11.294
8	11.327
9	11.365
10	11.412
11	11.432
12	11.462
13	11.491
14	11.513
15	11.533
16	11.546
17	11.555
18	11.569
19	11.799

**TAB.4:** Resonance frequencies of a 15° slice of the cavity with the copper structure

Once these “exceeding modes” have been identified and eliminated, the results appear to be quite in agreement among themselves and also if compared to other analyses (Tab.1).

The  $\pi/2$  mode, which is the mode we are interested in since it is the operating mode of the cavity in the Frascati Linac, is the 9<sup>th</sup> mode revealed in the analysis performed considering only the cavity (Tab.3), the 10<sup>th</sup> in the analysis considering a slice of cavity and copper structure of 15° (Tab.4) and the 11<sup>th</sup> considering a slice of 45° (Tab.2), and it has resonating frequencies of 11.411 GHz in the first case, 11.412 GHz in the second and 11.410 GHz in the third. It is possible to recognize the  $\pi/2$  mode not only by its expected frequency but also by plotting it. The plots (Fig.9) show clearly that the phase difference between the E field in a large cell and the E field in a contiguous small cell is  $\pi/2$ , whilst the phase difference of the field between two contiguous large cells is  $\pi$ .

For reader’s convenience we show the resonance frequencies of the four different analyses in only one table (Tab. 5), in order to allow an easier comparison.

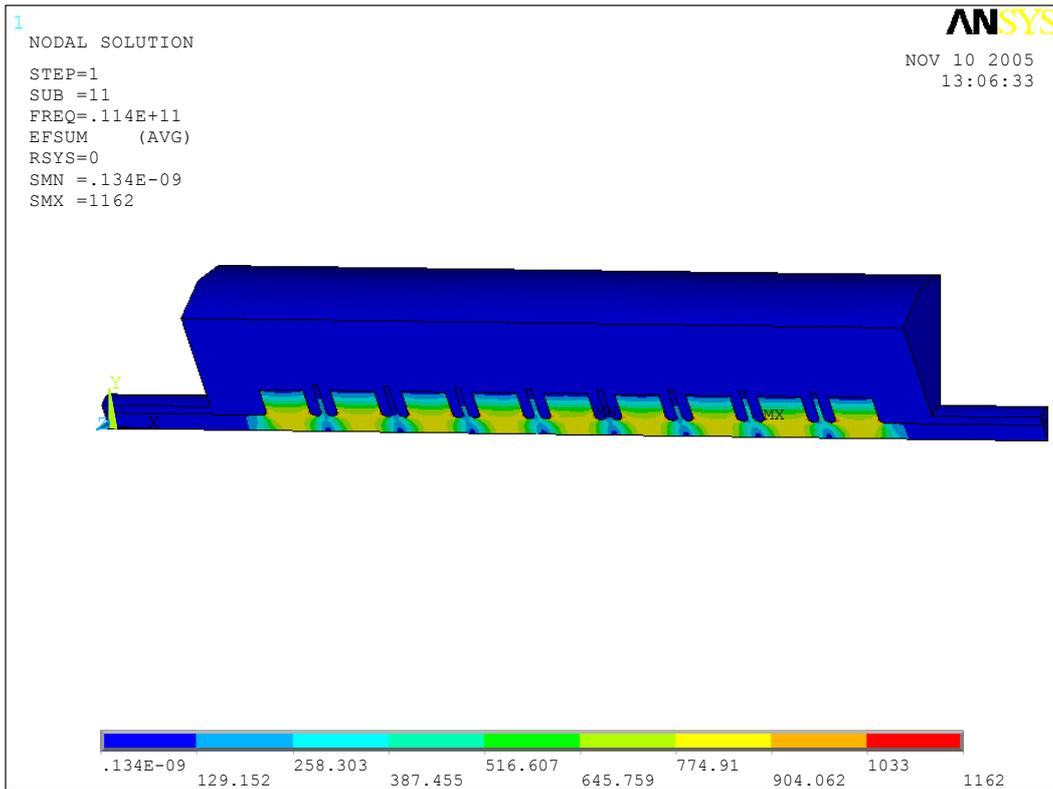
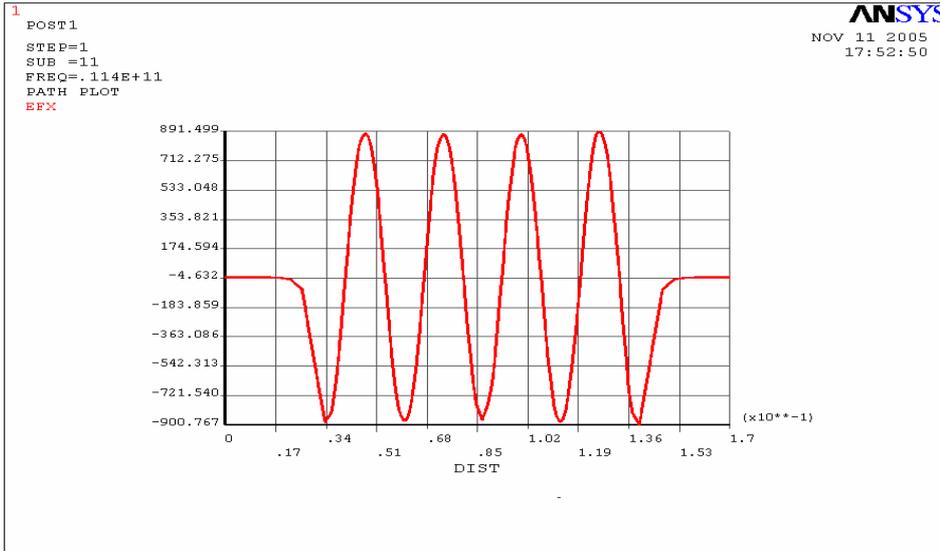


FIG. 9: E field in the  $\pi/2$  mode

MODE	$F_0$ (GHz) 45° slice with copper	$F_0$ (GHz) 15° slice with copper	$F_0$ (GHz) 45° slice without copper	$F_0$ (GHz) GdifiDL
1	11.043	11.148		
2	11.182	11.176	11.183	11.207
3	11.196	11.190	11.195	11.218
4	11.202			
5	11.213	11.208	11.213	11.236
6	11.239	11.233	11.240	11.259
7	11.268	11.262	11.267	11.300
8	11.302	11.294	11.302	11.328
9	11.336	11.327	11.337	11.361
10	11.374	11.365	11.374	11.392
11	11.410	11.412	11.411	11.459
12	11.441	11.432	11.441	11.473
13	11.470	11.462	11.470	11.500
14	11.499	11.491	11.498	11.526
15	11.521	11.513	11.520	11.554
16	11.541	11.533	11.542	11.563
17	11.555	11.546	11.555	11.581
18	11.567	11.555	11.567	11.591
19	11.573	11.799	11.591	11.597
20	11.710	11.569		
21	11.879			

TAB.5: Comparison of the resonant frequencies of the four different analyses

A very useful and nice way to study or observe the fields (in our case E or H) or other variables in the cavity, or more generally in any sort of model, is to plot the values that these variables assume in a particular location of interest by defining a path, mapping on the path and plotting the results for example on a graph. We have followed this procedure to plot the axial component of the E field on the cavity axis (Fig.10).



**FIG. 10:** axial component of E field on the cavity axis

## 5 CONCLUSIONS

In this paper the use of ANSYS finite element method software to perform high frequency electromagnetic analyses has been described. The principal steps to be followed have been explained and examples from analyses carried out on an X-band RF bi-periodic cavity operating at 11.424 GHz have been brought. Data from analyses performed with other simulating software have been considered for comparisons.

By the means of these examples we have obtained some interesting results that allowed to present some possible interpretations of data that may be in contrast with some expected values or with other analyses giving explanations of the possible reasons and useful suggestions on how to avoid or eliminate such problems.

At last, several possible ways to display or plot results have been presented.

## 6 ACKNOWLEDGEMENTS

Useful discussions with L. Ficcadenti, A. Mostacci and M. Zamperini are gratefully acknowledged.

## 7 REFERENCES

- [1] ANSYS is a trademark of SAS Inc. [www.ansys.com](http://www.ansys.com).
- [2] D. Alesini et al., “Studies on a bi-periodic x-band structure for SPARC”, SPARC-RF-03/002 LNF-03/013(R), August 2003.
- [3] L. Ficcadenti, private communication.
- [4] GdfidL Electromagnetic Field Simulator, [www.gdfidl.de](http://www.gdfidl.de).
- [5] N. Hartman, R. A. Rimmer, “Electromagnetic, thermal and structural analysis of RF cavities using ANSYS”, Lawrence Berkley National Laboratory, Berkley, CA, 2001.
- [6] J. W. Staples, S. P. Virostek, S. M. Lidia, “Engineering design of the LUX photoinjector”, Lawrence Berkley National Laboratory, Berkley, CA, 2004.
- [7] S. Moaveni, “Finite element analysis”, Prentice-Hall Inc., 1999.
- [8] R. Boyce, D. H. Dowell, J. Hodgson, J. F. Schmerge, N. Yu, “Design considerations for the LCLS RF gun”, Stanford Linear Accelerator Center, LCLS TN 04-4, April 2004.