

Department of Microelectronics and Computer Science  
Technical University of Lodz

# Scattering of a Plane EM Wave on a PEC Infinite Cylinder

M.Sc. Marta Zawieja  
[zawiejam@dmcs.p.lodz.pl](mailto:zawiejam@dmcs.p.lodz.pl)

Lodz, August 2002

## Problem description:

The interconnects and the devices formed by interconnects such as inductors, filters, resonant circuits, transformers and antennas are very often used in the high frequency electronic applications. The CFD-Maxwell module of the CFDRC-ACE environment specifically targets modeling of the electromagnetic phenomena in such devices and systems providing design tool aid.

In the following tutorial we will be analyzing a very simple model of a EM wave propagating around a perfectly electrically conducting (PEC) cylinder. This tutorial is divided into 4 parts:

1. Building a 3D model in the CFD-GEOM
2. Setting up the electromagnetic simulation parameters – CFD Maxwell Processing
3. Visualization of the results of the calculations
4. The project

## Part One

### Pre-Processing – Building a 3D model

#### Geometry

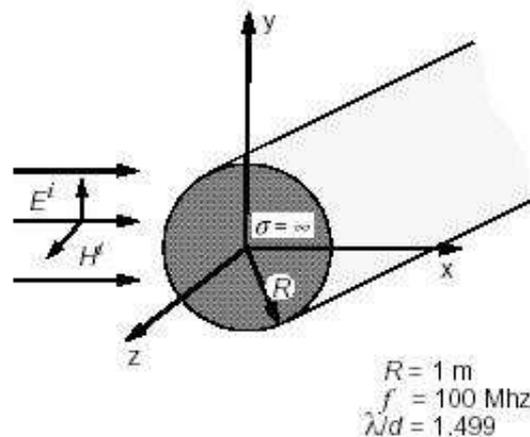
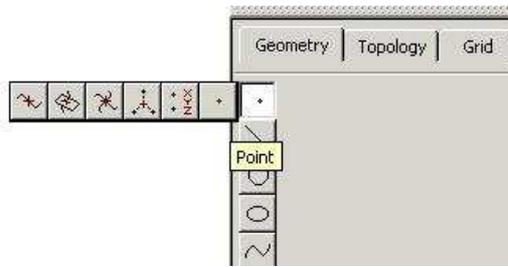


Fig. 1.1 The Geometry and Dimensions of the Analyzed Problem

#### 1. Create a top half of cylinder

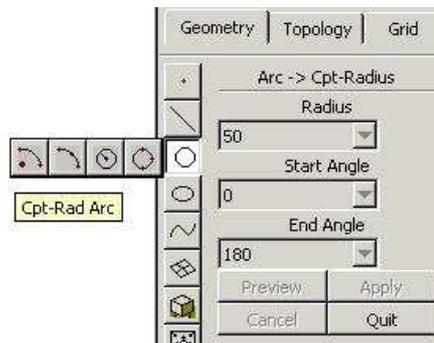
Start a CFD –GEOM. From the **Geometry** control panel, select **Point creation** -> **Point** (see fig. 1.2):



*Fig 1.2 The Point Creation tool under the Geometry tab*

Enter “0” in the X Value, Y Value and Z Value, next press **Preview** and **Apply**. In this way the point with specified coordinates is created.

Next from the Geometry tab, select Circle creation ->Cpt-Rad Arc (fig. 1.3).



*Fig. 1.3 The Arc Creation Tools*

In the **Point creation** select the **Pick Existing** and click with the left mouse button in the (0,0,0) point, which will be a central point of the created arc.

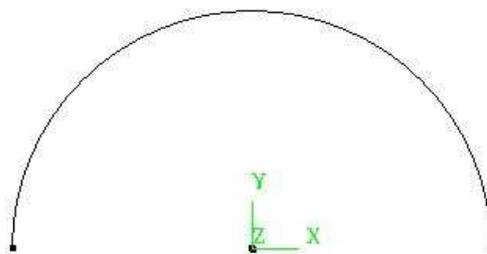
In the Arc->Cpt-Radius menu enter the following:

Radius: 1

Start Angle: 0

End Angle: 180

Press Preview and then Apply. Fig 1.3 shows results of our work.



*Fig 1.3 Half of the arc created using Arc Creation Tool*

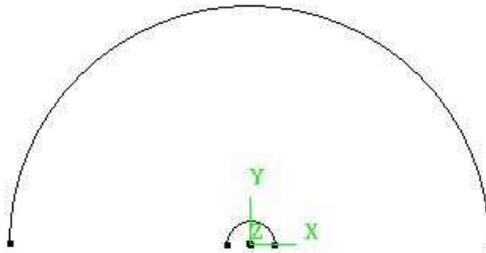
In the same way create another arc. Use the central point (0,0,0), and the following parameters:

Radius: 5

Start Angle: 0

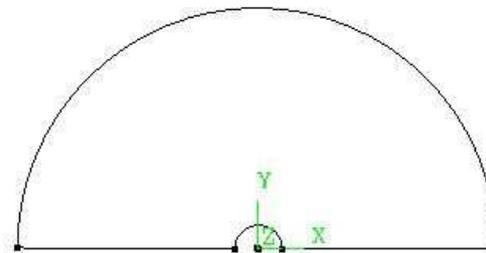
End Angle: 180

The project should look like in fig. 1.4



*Fig. 1.4 The two concentric arcs*

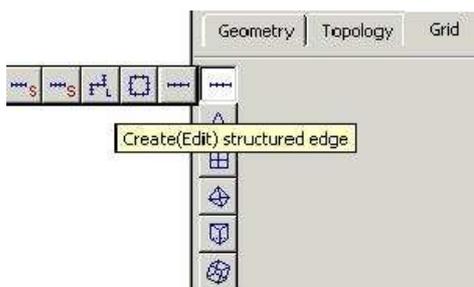
From the Geometry tab, choose Line creation -> Line.  
 Select the first point of the bigger arc and next, first point of the smaller one and press the middle mouse button or **Apply** to create the line. In the same way create the line between the final points of arcs (fig. 1.5).



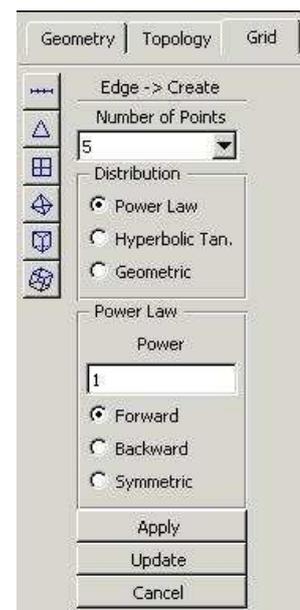
*Fig. 1.5 The basic geometry of the project.*

## 2. Creating Structured face

From the Grid tab, select Structured Edge options -> Create(Edit) structured edge (fig 2.1):



*Fig 2.1 The Create Structured Edge tool*

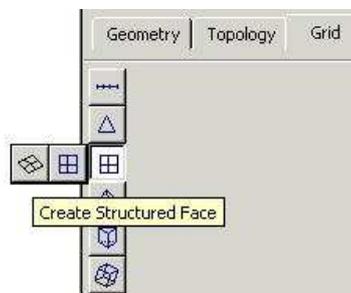


*Fig. 2.2 the Create edge control panel*

The Edge -> Create control panel appears (fig. 2.2).

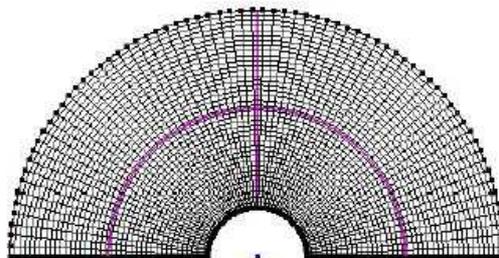
Enter "72" in the **Number of Points** field,  
Select the outer arc and press **Apply**,  
Select the inner arc and press **Apply**.  
In this way we created the edges on the outer and inner arcs.

Next,  
Enter "40" in the **Number of Points** field,  
Select the line that connects the left sides of the inner and outer arc and press **Apply**,  
Select the line that connects the right sides of the inner and outer arc and press **Apply**.  
In this way we created edges on the lines that connect the arcs.  
Using the 4 edges we have to create the structured face.  
Select Structured Face Options -> Create Structured Face (fig. 2.3).



*Fig. 2.3. The Create Structured Edge tool*

In the Face-> Create control panel select **Edge Sets**.  
Next, click all the edges and make sure that the middle mouse button was pressed after each edge selected. Click Apply. The structured face grid appears as shown in fig. 2.4:



*Fig. 2.4. The structured grid*

### *3. Rotating the model*

Under the **Geometry** tab, select Rotate -> Rotate Entity (fig. 3.1). Then select the face handler (see fig. 3.2) and press the middle mouse button.

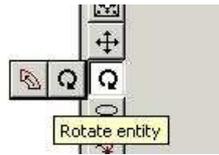


Fig 3.1 The Rotate Entity tool

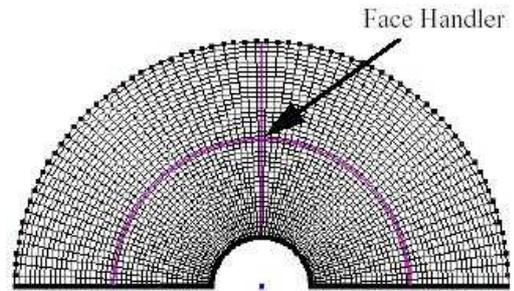


Fig 3.2 The Face Handler

From the Rotate -> Entities control panel set the following:

Rotation Mode: 0

Rotation Angle: 180

Repeat (time): 1

Options: **Duplicate**

Click **Apply**.

The model should appear as shown in the figure 3.2:

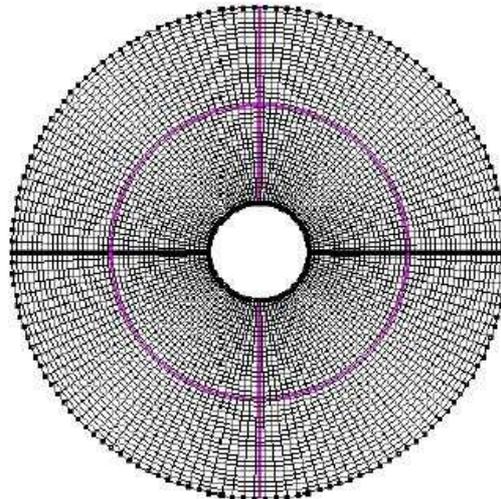


Fig. 3.2 The gridding system

#### 4. Creating structured 3D block

Under the Grid tab, select Structured Block options -> Create structured 3D Block (fig. 4.1).

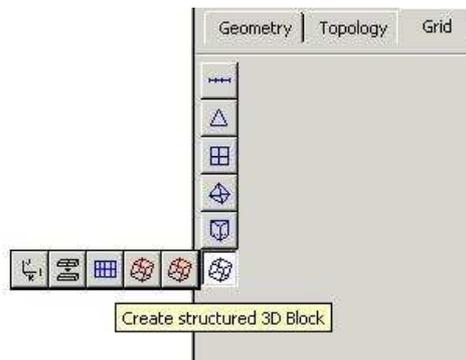


Fig. 4.1 The Create Structured 3D Block tool

In the Block -> Create panel choose **Extrusion**. Select each of the handlers and press the middle mouse button. From the Block -> Create control panel set the following:

Extrude options: Z

Distance: 0.1

Number of Points: 3

Create: Multiple Blocks

Click **Apply**.

### 5. Saving the model

Save the model as *your\_name.GGD* file in directory of your choice. This file allows you to edit this model later in CFD-GEOM. Next, save the model as *your\_name.DTF* file. This file contains the information about the mesh necessary to run simulations in CFD-Maxwell.

## Part Two

### Setting up the electromagnetic simulation parameters - CFD Maxwell Processing

#### 6. Opening the CFD-Maxwell

Launch the CFD-Maxwell and open the .dtf file using the menu command File -> Open. Click the PT (Problem Type) and verify that Maxwell checkbox is marked.

#### 7. Setting up a Model Options (MO)

In the Shared panel, in the Transient Time Step select:

Enter Number of Step = 2000

CFL Number = 0.3388036703

The CFL number is cause the time step size to be  $dt=5e-11$ . This value will ensure numerical stability.

In the Maxwell panel select:

Model: Tome Domain

Formulation: Scattered Wave

#### 8. Setting up the Volume conditions (VC)

Select the VC panel. The parameters connected with the material properties should be defined in this panel.

Select both entries in lower left window.

Change Properties to **Solid** under the VC Setting Mode area.

Leave all values on their defaults ( $\epsilon_r=1$ ,  $\mu_r=1$ ,  $\sigma = 0$  [S/m])

Click **Apply**.

#### 9. Setting up the Boundary Conditions (BC)

Select the BC panel.

Click on the “front –view” button or choose menu View -> Front.  
 Select the both half – circular faces by clicking on their center points and keeping the CTRL key pressed  
 Select **Extrapolation** as boundary Sub Type  
 Click **Apply**

Turn the model around by pressing the “Back View” button or picking menu View -> Back.  
 Select both half – circular faces and select **Extrapolation** as boundary Sub Type.  
 Click **Apply**

Turn the model until you see edge faces (fig. 9.1).

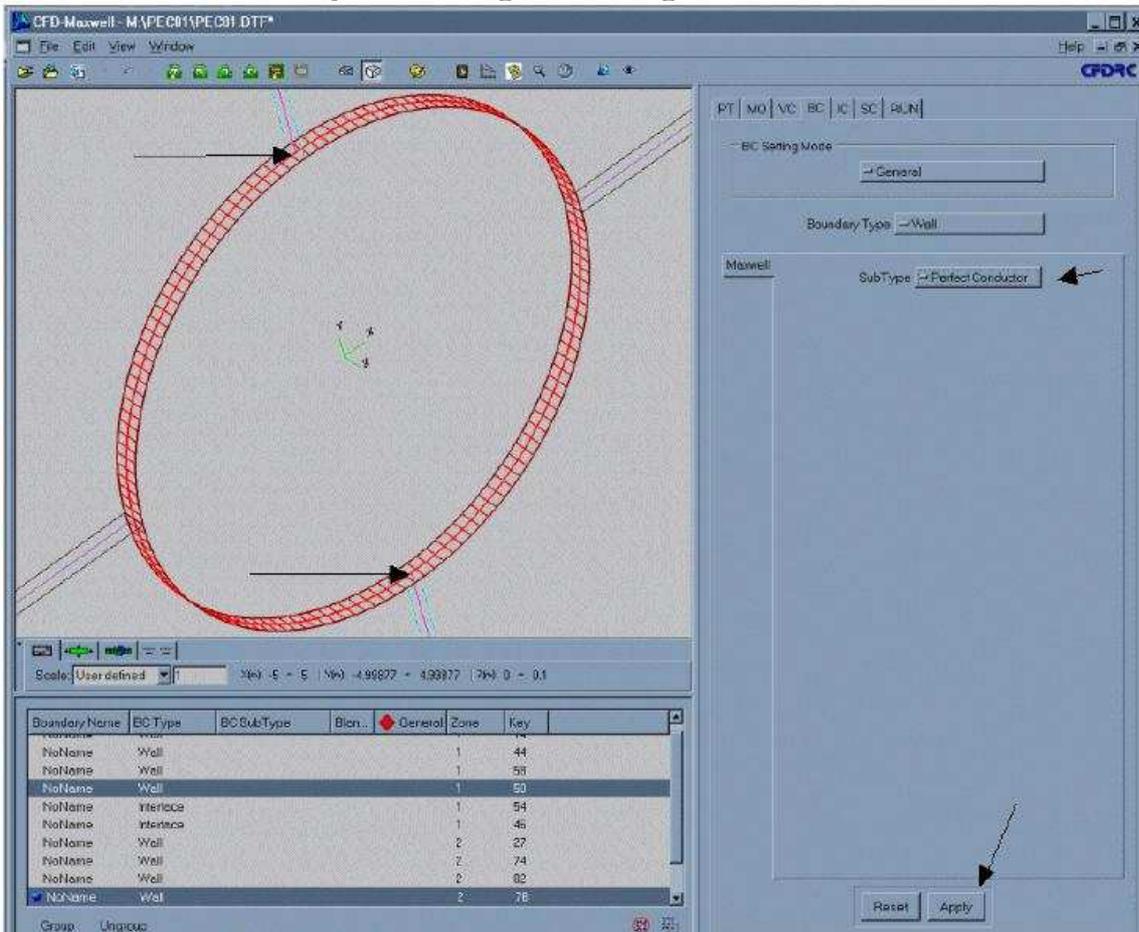


Fig. 9.1

Select the both faces by clicking on their center points and keeping the CTRL key pressed.  
 Select **Perfect Conductor** as boundary Sub Type.  
 Click **Apply**.

## 10. Setting up the Initial Conditions (IC)

Select the IC panel.  
 Under IC Global Settings choose: **For all Volumes**  
 Set the following values:

Electric field Propagation Vector: X=1  
Y=0  
Z=0  
Polarization Vector: X=0  
Y=1  
Z=0

Amplitude: 1 V/m

Under the Evaluation Method select: **Sinusoidal**  
Enter **100MHz** as frequency.  
Click **Apply**.

### *11. Setting up Solution Control parameters*

Select the SC panel.  
Choose the **Graphics** sub-panel.  
Pick all variables (which will be saved to the output file) containing the magnetic flux density B.

Select the **Output** sub-panel.  
Choose the "Unique Filename"  
and Enter 100 as Timestep Frequency.

### *12. Running the simulation*

Before running the simulation, save your file.  
Click on the **Run** panel and click on the **Submit** button to start the simulation process.  
You can press View Residuals or View Output buttons to see real-time displays of the residual history and output of the file contents.

## Part Three

### Visualization of the results of the calculations

#### *13. Simulation Post Processing in CFD – VIEW*

Go to CFD – VIEW window and open the DTF file. In menu **File -> Import DTF or Plot 3D** choose *your\_name.00200.DTF*.

In the Objects window in the bottom-right corner of the screen select Geom. From the toolbar select the following buttons:

Outline: on  
Contours: on  
Smooth Surface: on

Select the variable you want to display in Primary Variable pull-down menu, e.g.  $Bz_n$  for z component of magnetic flux density.

To **view results of many subsequent time steps**, saved into separate DTF files in your current directory, you may use the data-file animation feature in CFD – VIEW:

- from the menu, select **Gizmos – Animate Data File**
- in the *Animate Data File* window, type appropriate values in the *Data Files Range* fields:  
From: 200  
To: 2000  
Current: 200  
Increment: 200
- use *Run Animation* button to *play* the sequence of your transient simulation results
- you may also use the little up and down arrows next to *Current: step number*, to step through available DTF files with results

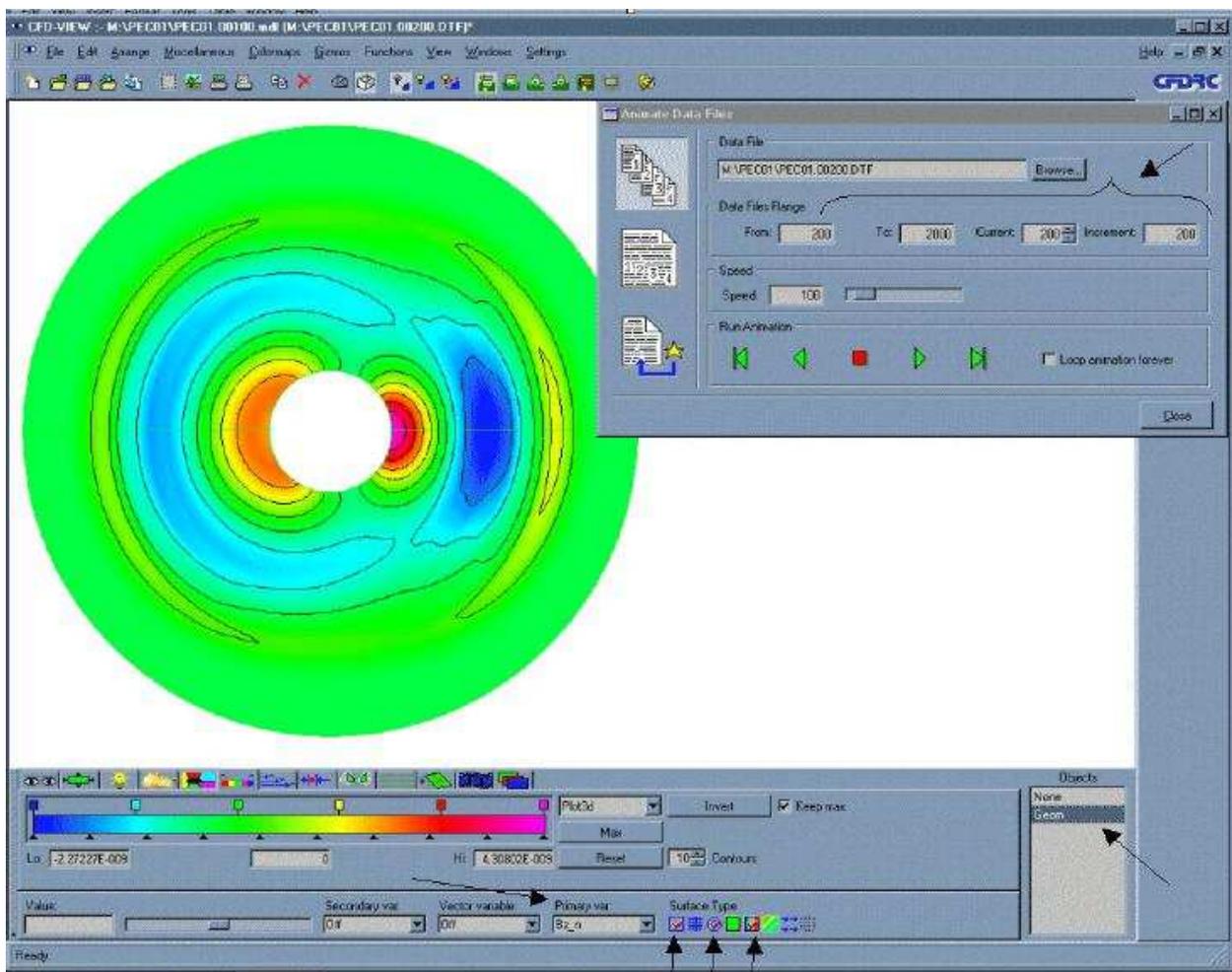


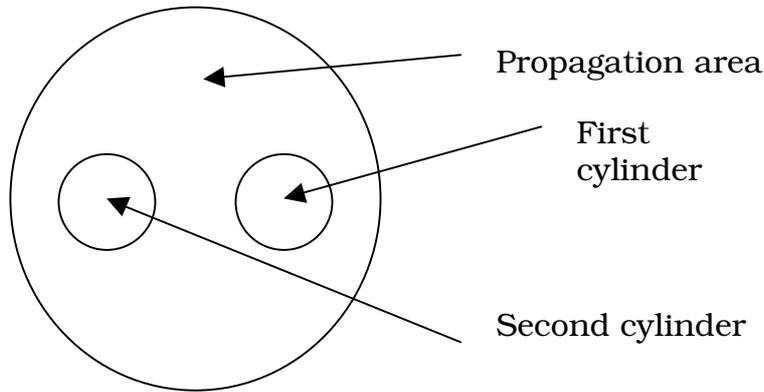
Fig 13.1 The CFD-VIEW gui with Animate Data Files window.

## Part Four

### The Project

#### 14.Project

Lets change geometrical model through addition of the second perfectly electrically conducting (PEC) cylinder (see fig.14.1).



*Fig. 14.1 The modified geometry*

In the first cylinder the electric field propagation vector, polarization vector and amplitude of electric field are left unchanged.

In the second cylinder set polarization vectors' directions as opposite:

- electric field propagation vector ( $X=-1, Y=0, Z=0$ )
- polarization vector ( $X=0, Y=-1, Z=0$ )

or change value of amplitude.

Compare the results with solution obtained in the tutorial. Write down your conclusions.