

Appendix 8 – A Brief Tutorial on Using the winFDTD_draw Utility

In this example, we are going to model a simple printed circuit board consisting of a single transmission line with resistive voltage source driving a resistive load. The schematic of the system is shown in Figure A8.1.

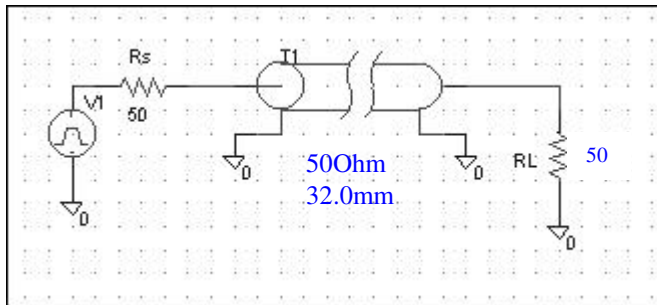


Figure A8.1 – The schematic of the example.

The physical model is shown in Figure A8.2, which consists of a double-sided printed circuit board (PCB), with the bottom side being the ground plane. The dielectric of the PCB is assumed to be FR4, with dielectric constant of 4.2.

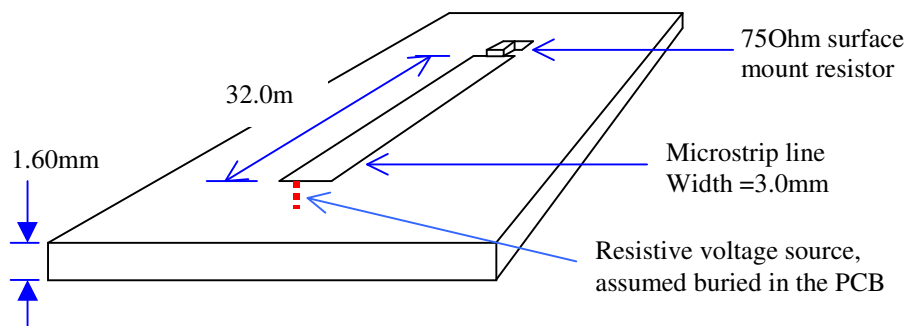


Figure A8.2 – Physical model corresponding to the schematic of Figure A8.1.

Step 1 – Determine the cell size to use

From the physical model of Figure A8.2, we choose the to divide the model into cells with dimensions shown in Figure A8.3.

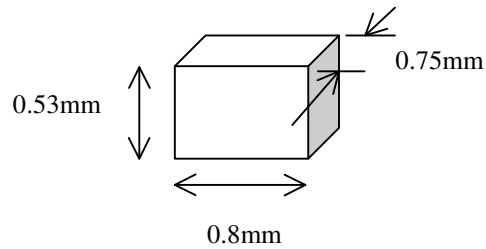


Figure A8.3 – Dimensions of cell.

Using the single cell of Figure A8.3, we could build up the physical model layer by layer. A cross-sectional view of how we could construct the physical model using layers of cells is depicted in Figure A8.4.

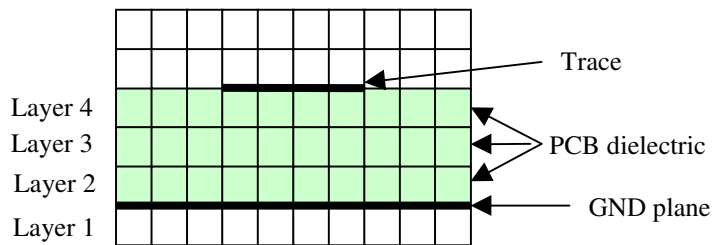



Figure A8.4 – Cross section view of the model.

Step 2 – Run the **winFDTD draw** program

Double-click on the following icon to run the winFDTD_draw utility.  Click the new button and give it a name, say “tline”.

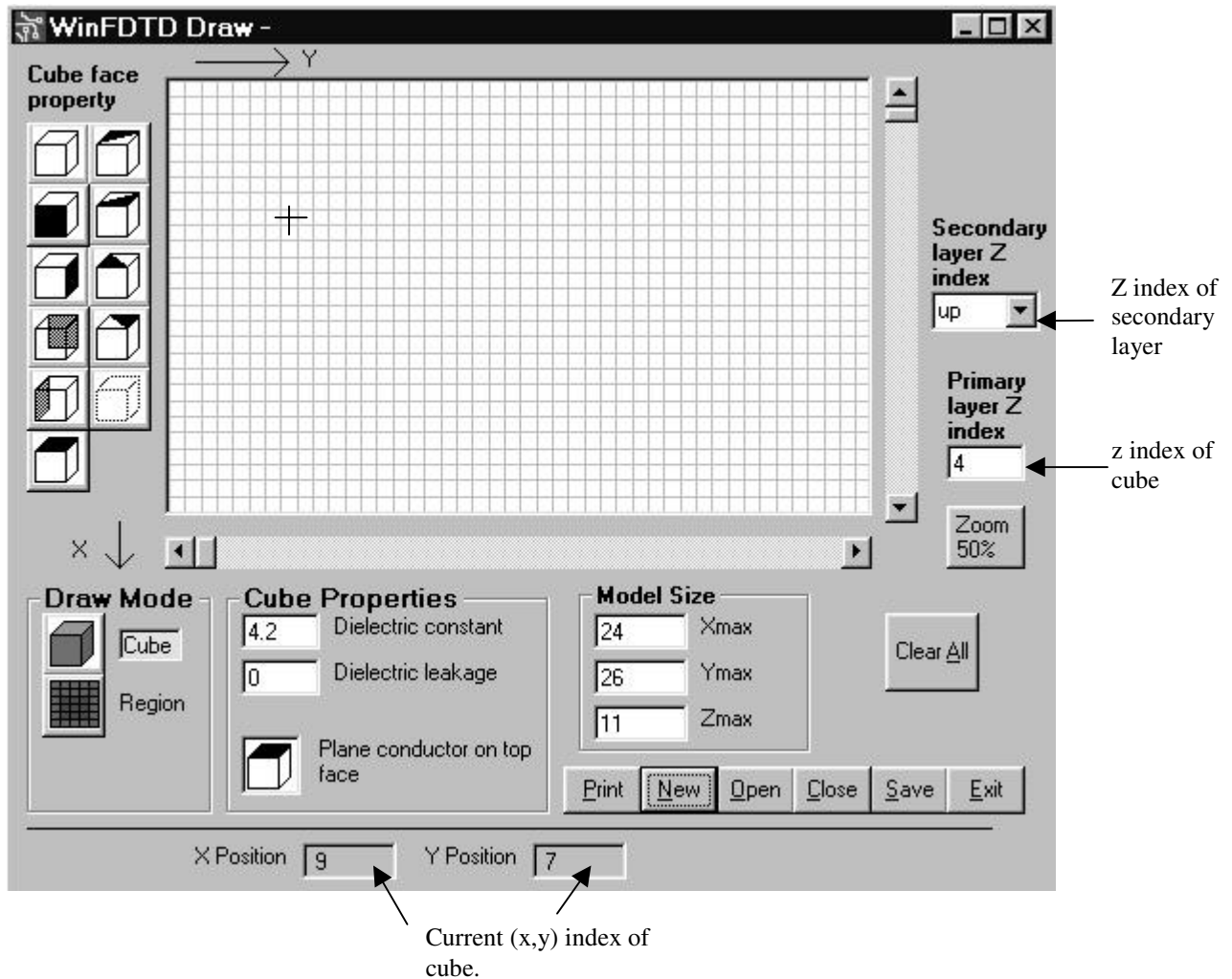


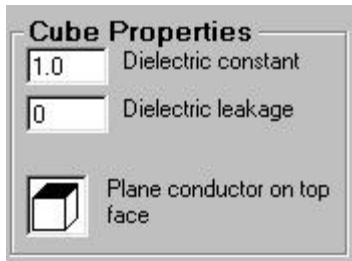


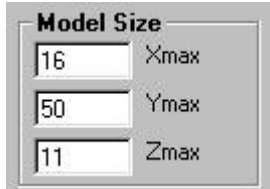
Figure A8.5 – The front panel of the winFDTD_draw utility.

Step 3 – Set the model size and draw the ground plane

- Set the draw mode to “region” by clicking on the button  .
- Select the cube characteristic with perfect conductor on the top face. 
- Set the cube properties, with dielectric constant of 1.0 for air and no leakage (dielectric leakage or conductivity = 0).



- Also set the size of the model, with 16 cubes along x -axis, 50 cubes along y -axis and 11 Cubes along z -axis.



Note1 – Determining the size of the model

The FDTD method employs a technique called absorbing boundary condition (ABC) to terminate a model space. An actual model space will be infinite in extent. To approximate an infinite space, ABC is employed on all six faces of the volume for the model.

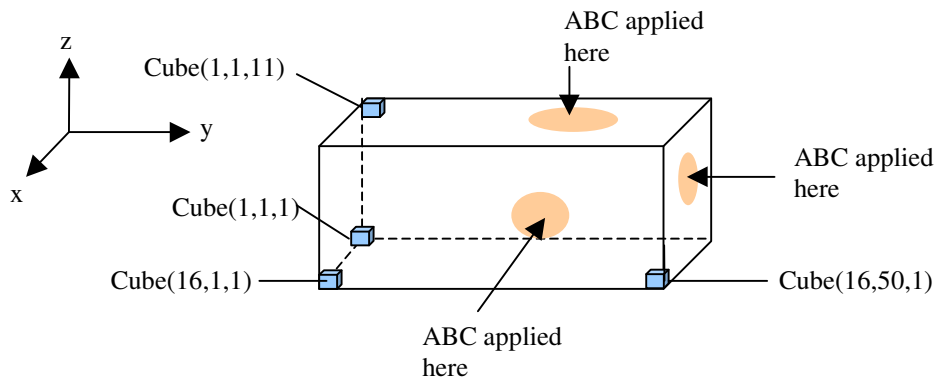


Figure A8.6 – Absorbing boundary condition applied to a finite space to simulate an infinite extent.

A rule of thumb for determining the model size is, always allows 5 to 10 buffer cubes before encountering the border of a model. An exception is the ground plane, it only

needs one cube from the boundary. As an example consider the current model along the z-axis. We allocate 6 buffer cubes from the top ABC and 1 buffer cube for the bottom ABC. The region of interest is the region where we place conducting traces and components. Similarly this rule is applied to the right, left, front and back ABCs. Figure A8.7 shows the cross section view of the model with the corresponding buffer layers.

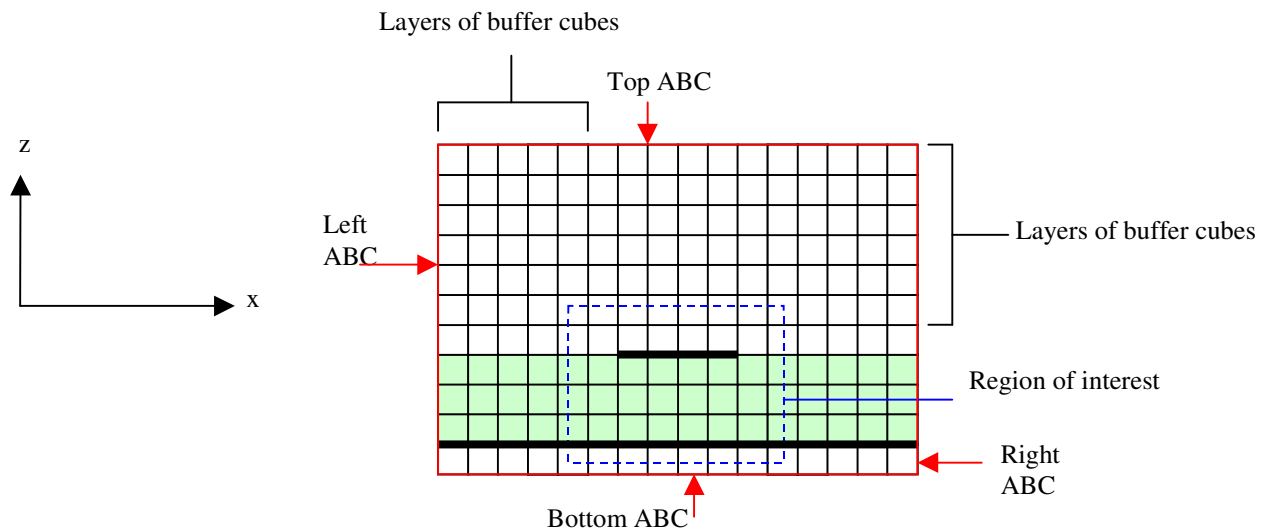


Figure A8.7 – Cross section view of the model.

- Now click on Cube (16,1,1) and Cube (1,50,1). All the cubes in layer $z = 1$ in between these two cubes will be changed to cubes with top face being perfect conductor, dielectric = 1.0 and conductivity = 0.0. This effect is shown in Figure A8.8.

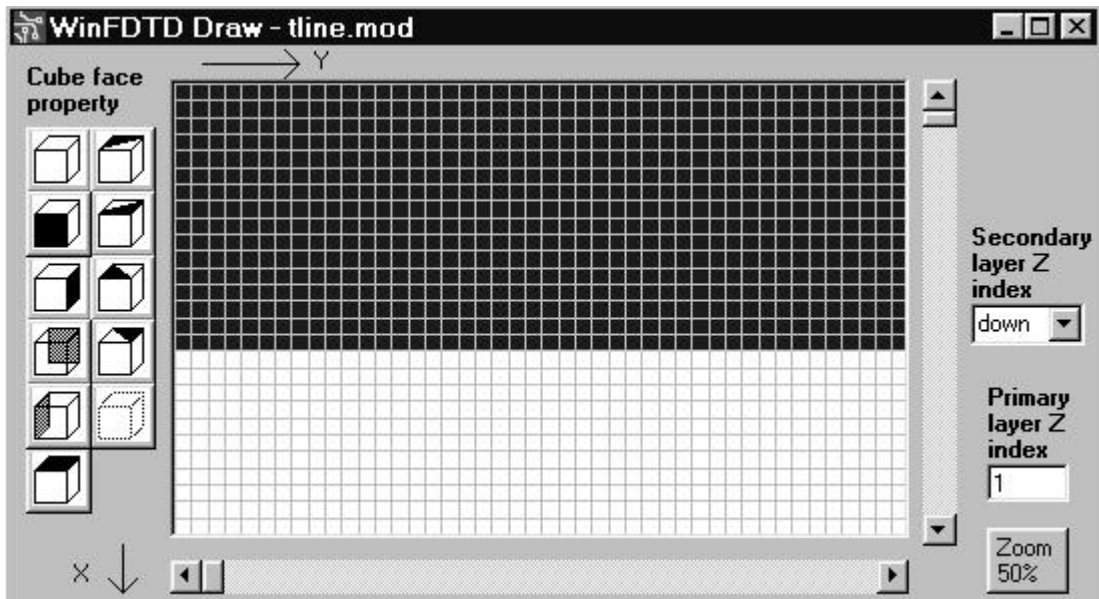
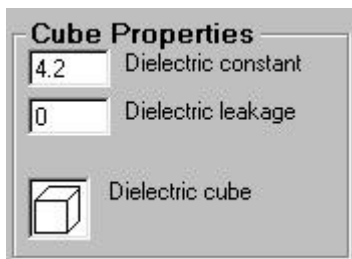


Figure A8.8 – After drawing the ground plane, layer 1.

Step 4 – Draw the dielectric layer

- Following the procedures in Step 3, draw the dielectric layers on Layer 2, Layer 3 and Layer 4. Remember to change the cube properties to dielectric constant = 4.2, dielectric leakage = 0.0 and the default cube face, no conducting faces.



- The result effect at Layer 4 ($z = 4$) is shown in Figure A8.9.

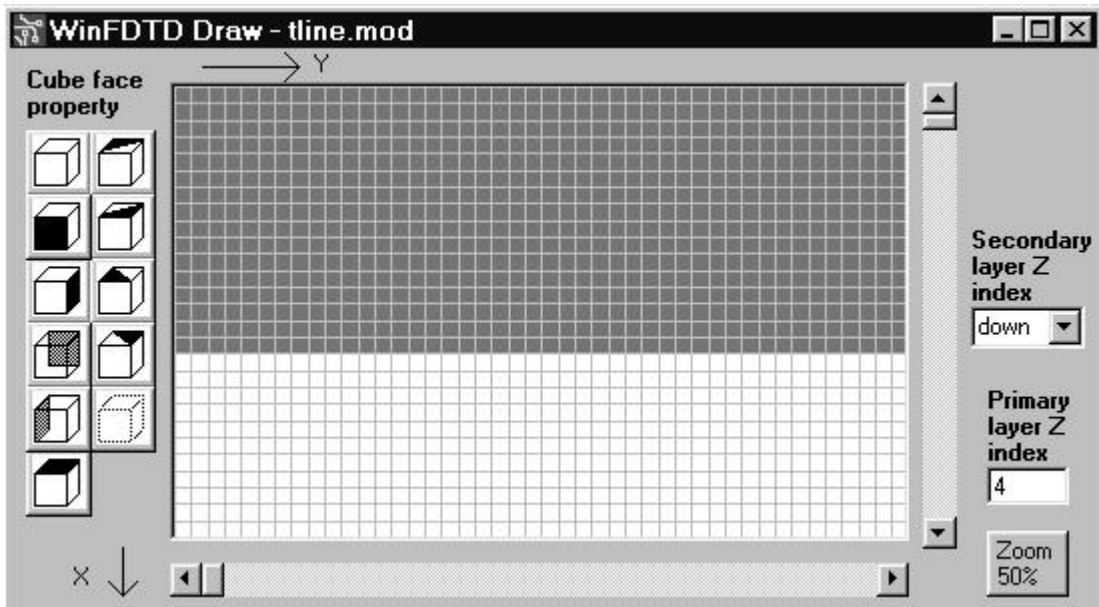
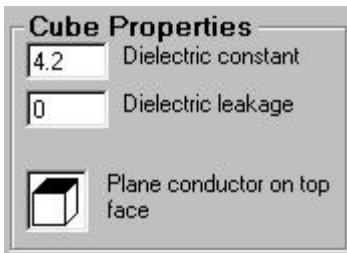


Figure A8.9 – After drawing the dielectric layers.

Step 5 – Draw the transmission line trace

- Setting the cube properties as follows:



- Draw a conducting trace for all cubes between Cube (7,6,4) and Cube (10,45,4). The result is depicted in Figure A8.10. In drawing the trace, you can use either the “Cube” or “Region” draw mode.

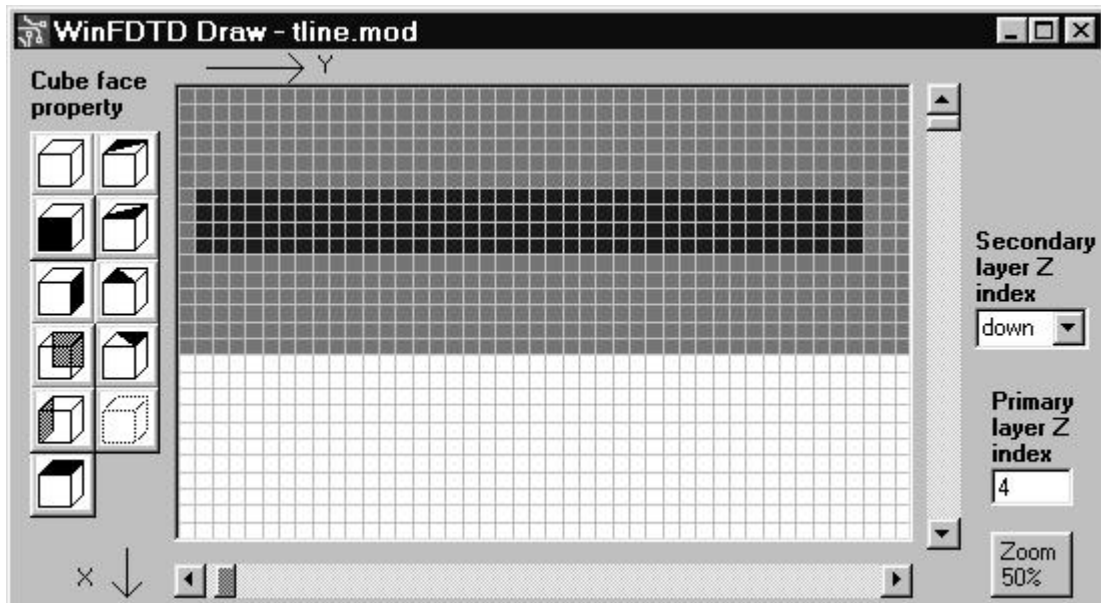


Figure A8.10 – After drawing the trace for transmission line.

Step 6 – Add resistive load

- Right click on Cube (9,45,4). A pop-up window will appear. The purpose of this pop-up window is to allow user to examine the E and H field components associated with this cube in detail. If a component is assigned to the E field, or if a probe instruction is assigned to either E or H field, this information will appear in the appropriate text boxes in the pop-up window.
- Click on the E field component button as illustrated in Figure A8.11.
- Select component 100, which represent lumped resistive component. Click the “parameter” button and set the parameter list to “50”. This sets the resistance to 50. Different component has different parameter list requirement. You are advised to refer to the online help file for a list of components and the parameter lists.
- Finally, click “Insert Component” button to assigned this to $E_y(9,46,5)$ field component. The final result should be as shown in Figure A8.13.

Note 2 – Deleting a component

To delete a component, right-click on the corresponding cube and access the E field component. Simply click the button ‘Remove Component’ and the component will be removed.

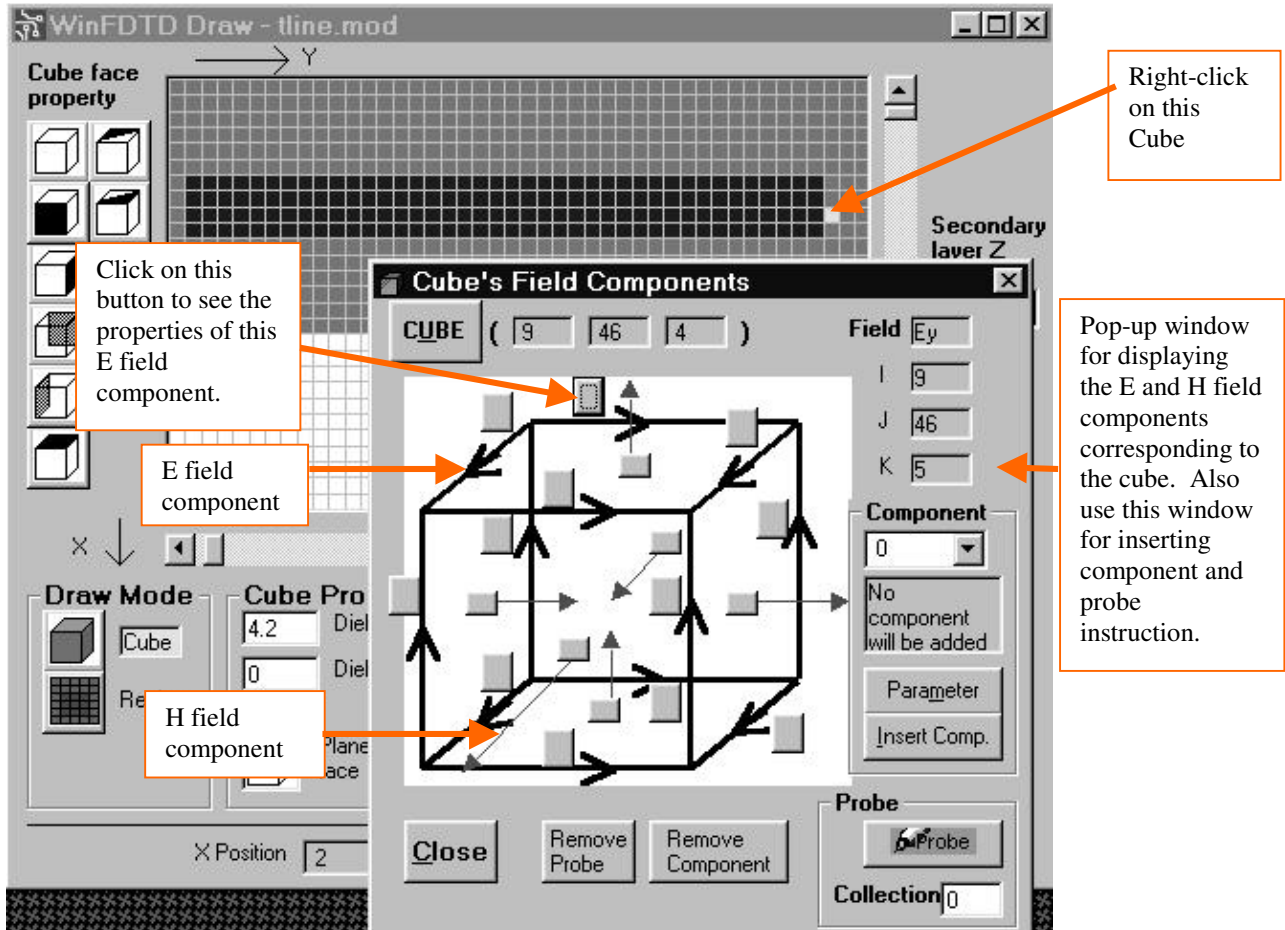


Figure A8.11 – Upon right clicking a cube, a pop-up windows will appear.

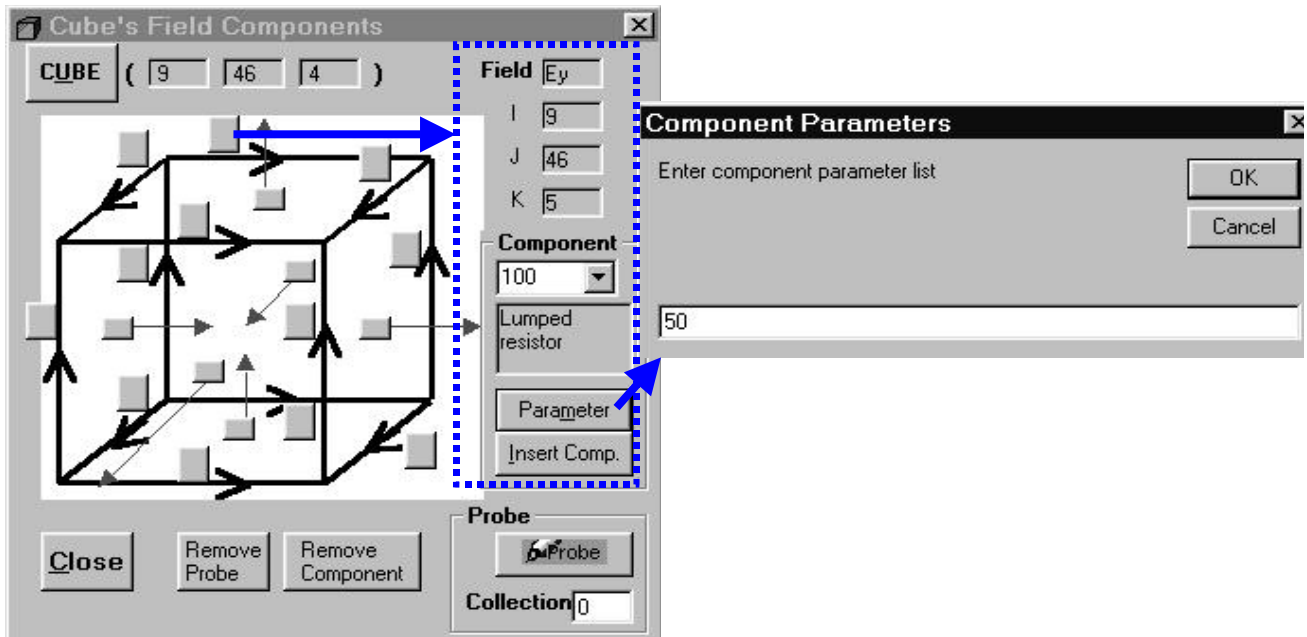


Figure A8.12 – Assign a resistive component to field component $E_y(9,46,5)$.

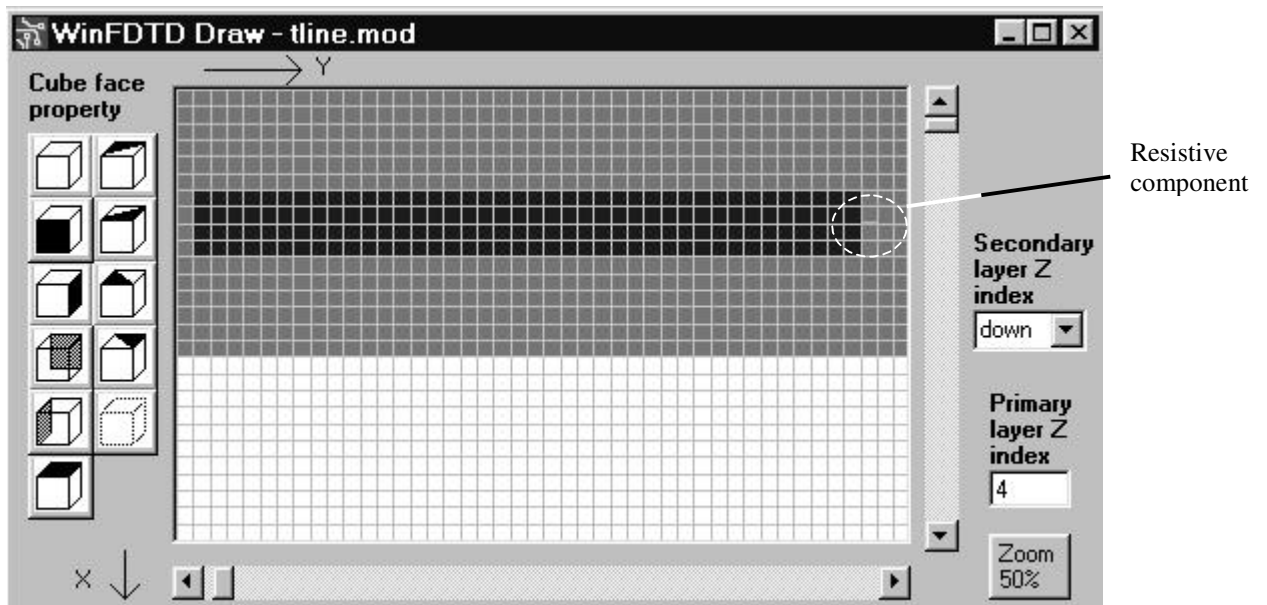


Figure A8.13 – After inserting a resistive component in y direction.

Step 7 – Add all through hole to connect resistive load to ground plane

- The procedure to add a through hole is similar to Step 6.

- The component to use is ‘1’, which represent a perfect conducting short wire. No parameter is required for this component. If you key in a parameter list, this will be ignored.
- You must insert the perfect conducting short wire on Layer 2, 3 and 4 so that they form a continuous connection from the resistive load to the ground plane. In doing so, you can use the secondary layer index to superimpose the objects on the secondary layer onto the current layer. This will assist in guiding you to place the component.
- An example is shown in Figure A8.14. A z-directed short wire is placed in layer 4. To place a z-directed short wire in layer 3, we go to layer 3. We can still view the objects on layer 4 by setting the secondary layer index to 4 or selecting the ‘up’ choice. This enables us to know where to place the short wire so that a continuous link is formed.

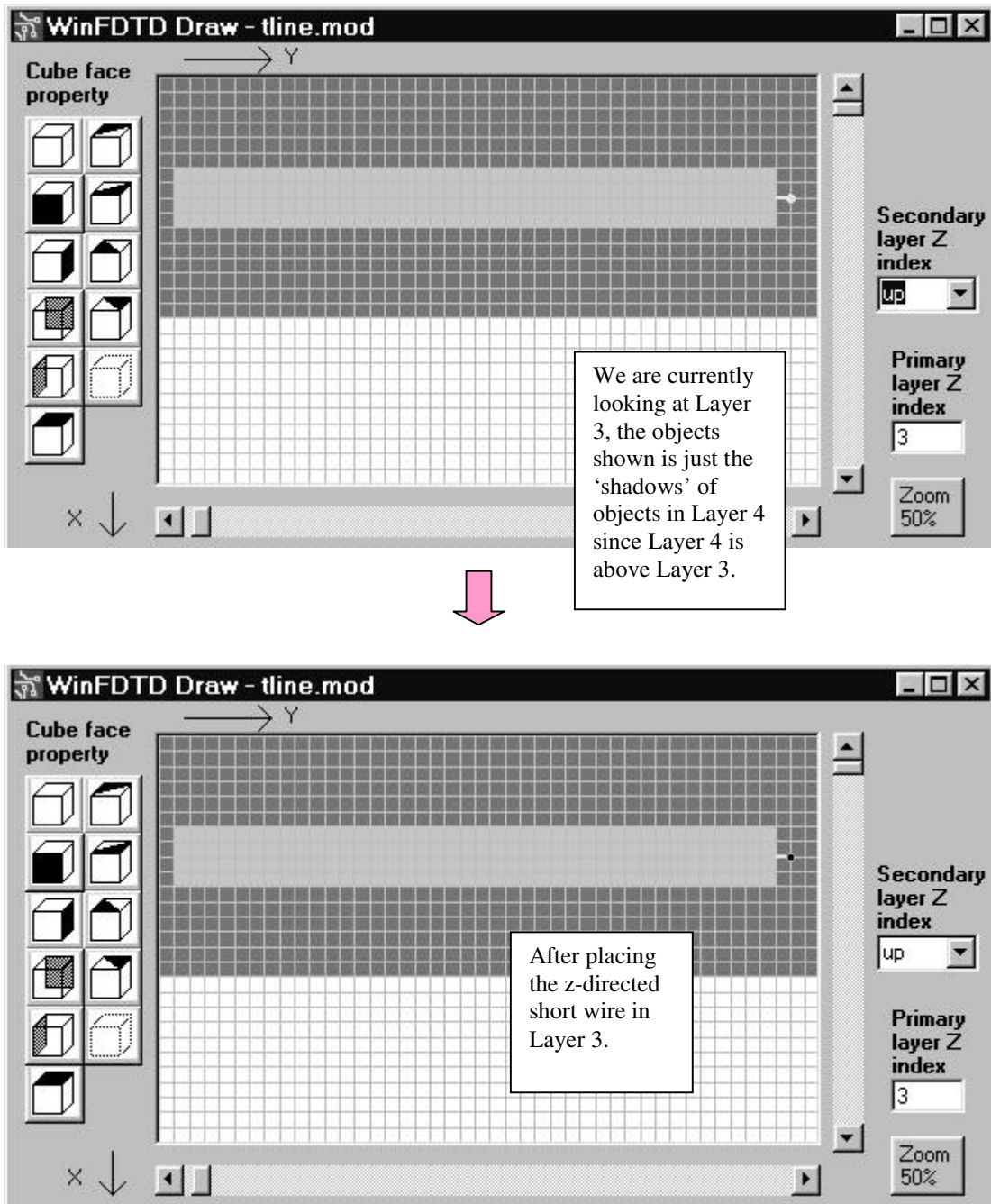


Figure A8.14 – Inserting the through hole using perfect conducting short wire (oriented in z direction). Using the secondary layer index to assist in placing the component.

Step 8 – Placing the source

- Placing the source uses similar procedures as in Step 6 and Step 7. The component used is “50”, representing the pulsed resistive voltage source.
- The parameter list is shown in Figure A8.15. This implies the following, source resistance = 50Ohm, low voltage = 0, high voltage = 3.3V, delay = 0ps, rise/fall time = 200ps, voltage high period=300ps, voltage low period = 700ps. The period of the pulse is given by:

$$T_{period} = 2 \times 200ps + 300ps + 700ps = 1400ps$$

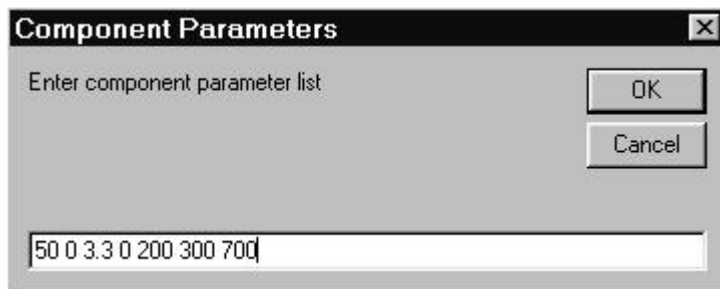


Figure A8.15 – Parameter list for the pulsed resistive voltage source component.

- The final layout of the model, as viewed from the top to Layer 4 is shown in Figure A8.16.

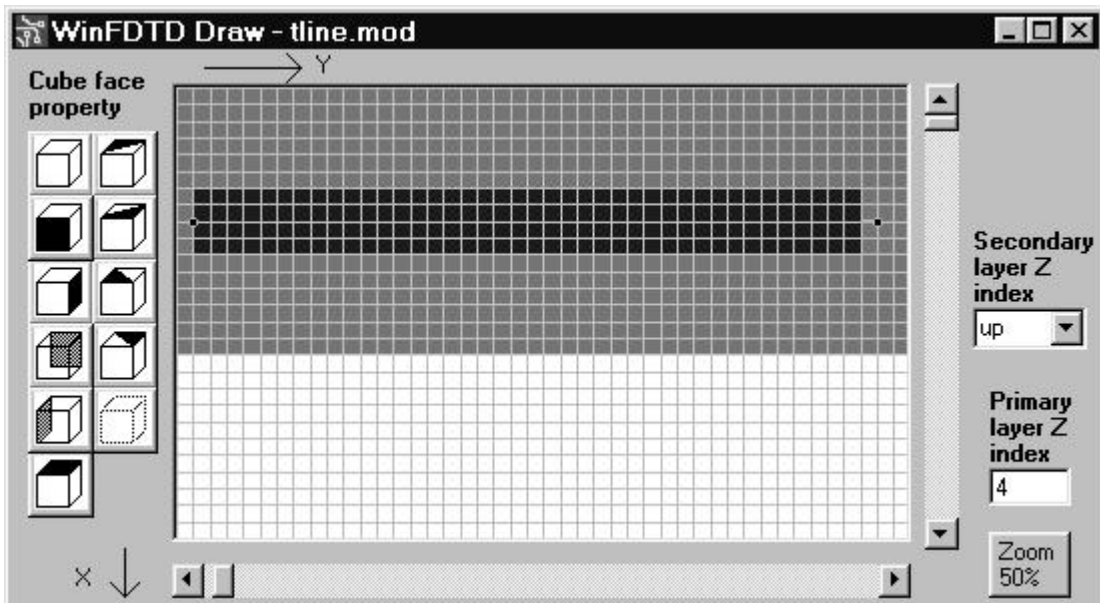


Figure A8.16 – The completed layout as seen on Layer 4, top view.

Step 9 – Placing the Probe instruction

- After completing the layout, you have to instruct the software to print out the parameters of interest to the output file so that you can view the result.
- Simply outputting every single field components to the output file will result in an immensely large file, comparable to a video file. Hence the need for a probe instruction.
- You can print out the field components, or a numerical line integral of the field component along a certain path.
- Here we are interested in knowing the voltage at the input and output ends of the transmission line. Voltage in this system can be approximated by:

$$V_{21} = - \int_{p_1}^{p_2} \vec{E} \cdot d\vec{l} \cong - \sum_i E_i \Delta d_i$$

- The procedure to instruct the software to print the voltage across the pulsed resistive voltage source is shown in Figure A8.17.
- After going through the steps in Figure A8.17, the following quantity will be saved in the output file at fixed intervals: $V = -E_z(9,6,4)\Delta z$

Note 3 – The collection number and line integration

The collection number must be a non-zero integer, if it is zero the software will merely save the field component value into the output file without performing the numerical line integration. The function of the collection number is to distinguish between different numerical line integration. For instance if you need to integrate more than one segment, then you can probe a few field components with the same collection number. All field components with similar collection number will be grouped together under one line integration. Suppose you want to do the following:

$$V = (-E_z(9,47,4)\Delta z) + (E_y(9,46,5)\Delta y)$$

The screen shots of the pop-up window illustrate how this can be achieved (Note the -2 collection number for $E_y(9,6,5)$).

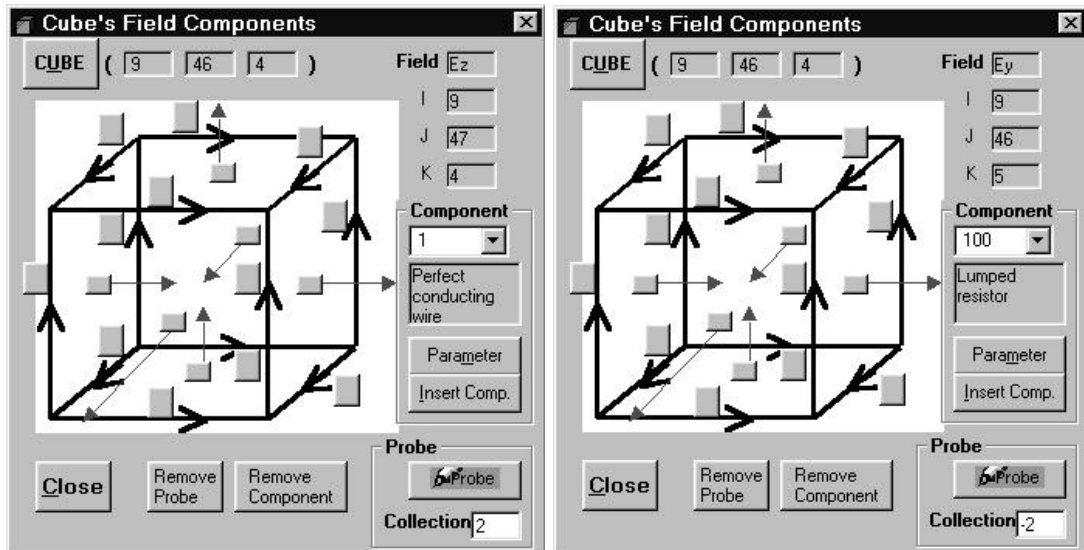


Figure A8.17 – Attaching 'probe' command to field components.

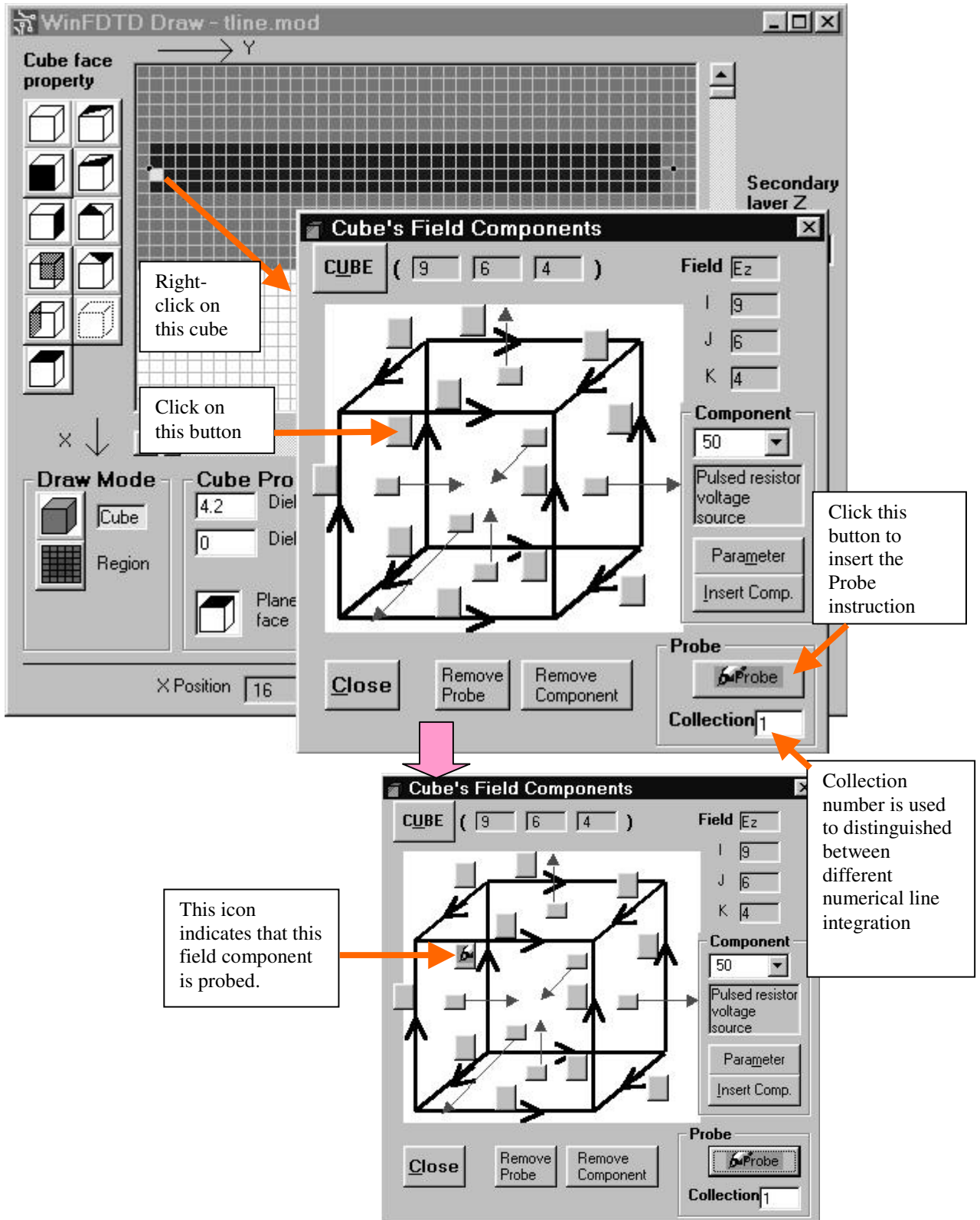
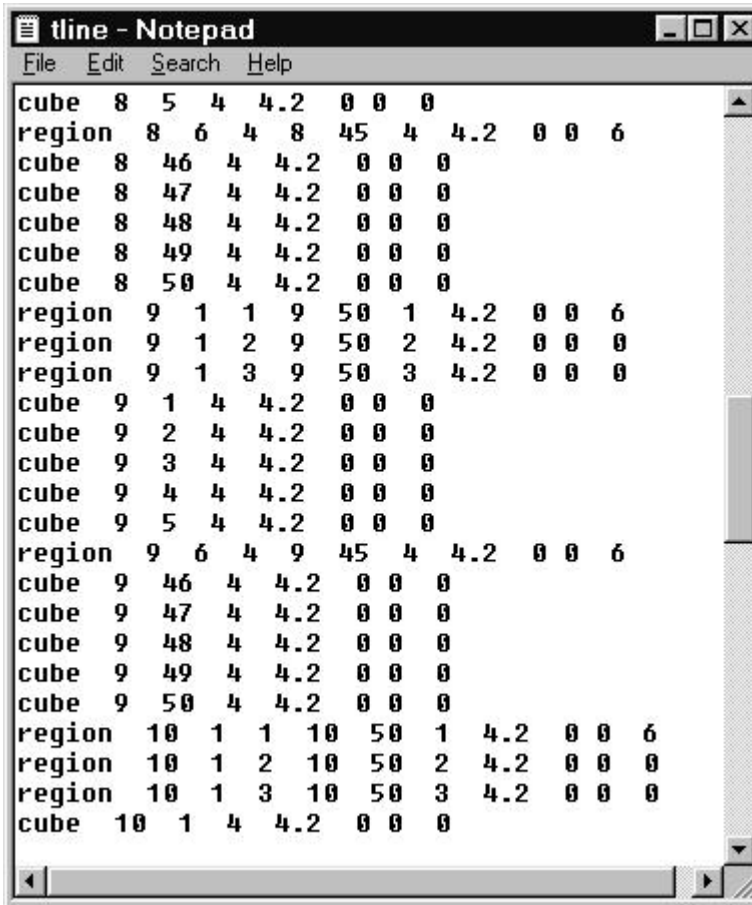


Figure A8.18 – Inserting the probe component at the pulsed resistive voltage source.

- Finally also place a probe instruction on $E_y(9,46,4)$, the resistive load. Thus we want to observe the voltage across the pulsed resistive voltage source and the resistive load. You can use a collection number of -2 . Negative because the orientation of positive voltage is opposite to the y-axis.

Step 10 – Save the file and Exit

Now you can save the file. The draw utility will generated a “tline.mod” file. You can open this file using Notepad and perform further editing. A screen shot of the file is shown in Figure A8.19.



```

tline - Notepad
File Edit Search Help
cube 8 5 4 4.2 0 0 0
region 8 6 4 8 45 4 4.2 0 0 6
cube 8 46 4 4.2 0 0 0
cube 8 47 4 4.2 0 0 0
cube 8 48 4 4.2 0 0 0
cube 8 49 4 4.2 0 0 0
cube 8 50 4 4.2 0 0 0
region 9 1 1 9 50 1 4.2 0 0 6
region 9 1 2 9 50 2 4.2 0 0 0
region 9 1 3 9 50 3 4.2 0 0 0
cube 9 1 4 4.2 0 0 0
cube 9 2 4 4.2 0 0 0
cube 9 3 4 4.2 0 0 0
cube 9 4 4 4.2 0 0 0
cube 9 5 4 4.2 0 0 0
region 9 6 4 9 45 4 4.2 0 0 6
cube 9 46 4 4.2 0 0 0
cube 9 47 4 4.2 0 0 0
cube 9 48 4 4.2 0 0 0
cube 9 49 4 4.2 0 0 0
cube 9 50 4 4.2 0 0 0
region 10 1 1 10 50 1 4.2 0 0 6
region 10 1 2 10 50 2 4.2 0 0 0
region 10 1 3 10 50 3 4.2 0 0 0
cube 10 1 4 4.2 0 0 0

```

Figure A8.19 – The “tline.mod” model definition file.

Step 11 – Simulate the model

- Run the **winFDTD** program. Access the ‘Simulation Setup’ command. Set $\Delta x = 0.75\text{mm}$, $\Delta y = 0.8\text{mm}$ and $\Delta z = 0.53\text{mm}$.
- Set maximum time step at 3500, the rest of the parameters leave as default. Notice the print interval is 10, this means after every time step the results as indicated by the Probe instructions will be saved in the output file. Our time step is set to 1ps, therefore the resolution is 10ps. You can change this parameter accordingly, however it will affect the output file size. Also note that with the current Δx , Δy , Δz and Δt setting, the software automatically calculates the CFL Stability factor (assuming free space dielectric). As a rough guide this value must be less than 1 to ensure stability of the model.
- Open the file “tline.mod”.
- Now run the simulation.

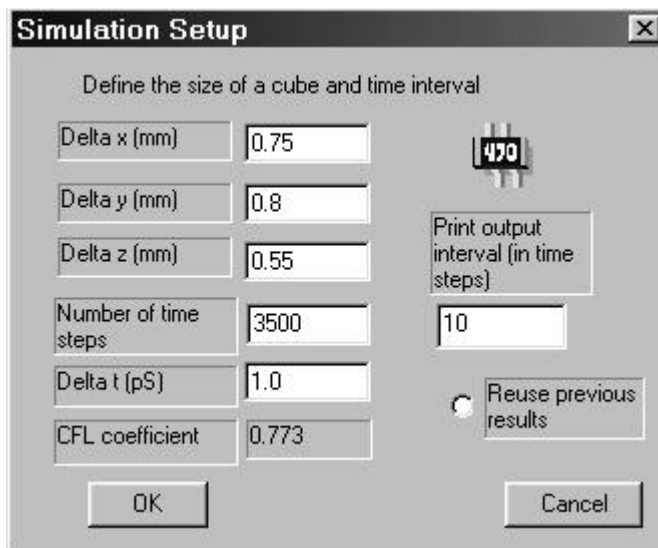


Figure A8.20 – Simulation setup for this example.

Step 12 – Plot the output

- Run the **plot** command from the **winFDTD** window.
- Click the Plot button to display the waveforms.

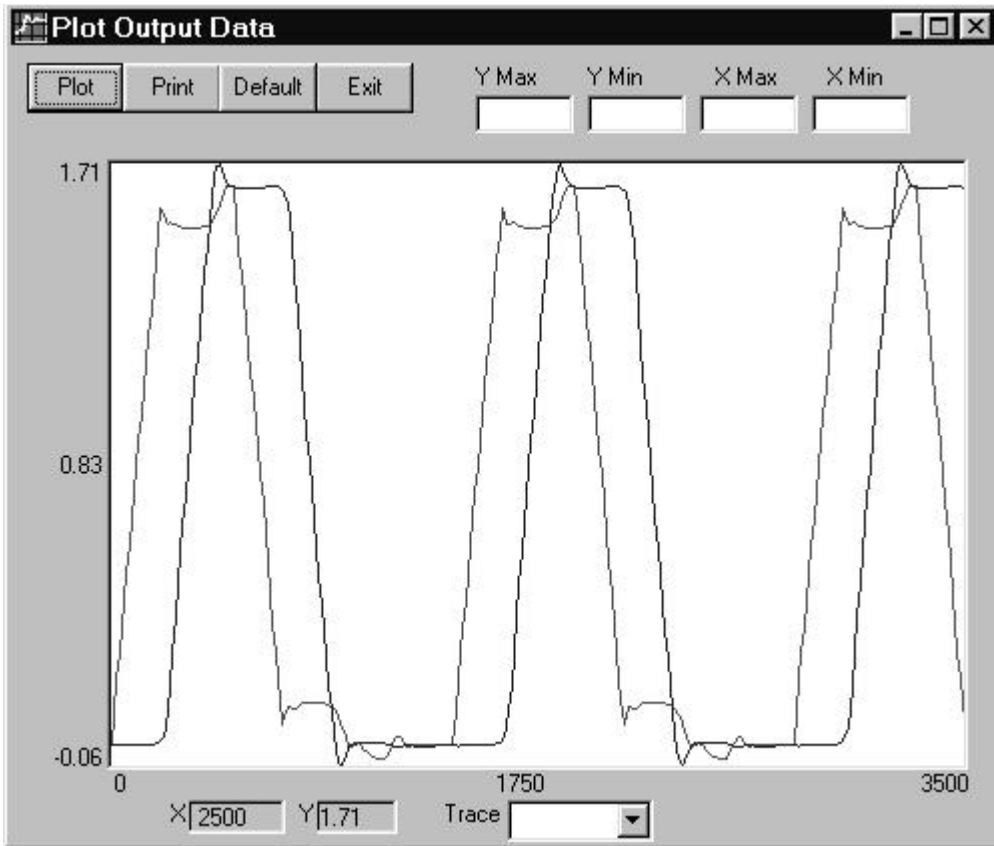


Figure A8.21 – The waveforms at the input and output ends of the transmission line.