Agilent Technologies

EMPro 2010
May 2010
EMPro FEM Simulation

Acknowledgments

Errata The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Using the Finite Element Method (FEM)

The Finite Element Method (FEM) Simulator within EMPro provides a complete solution for electromagnetic simulation of arbitrarily-shaped and passive three-dimensional structures. It provides a complete 3D EM simulation for designers working with RF circuits, MMICs, PC boards, modules, and Signal Integrity applications. The FEM simulation provides the best price/performance, 3D EM simulator on the market, with a full 3D electromagnetic field solver, and fully automated meshing and convergence capabilities for modeling arbitrary 3D shapes such as bond wires and finite dielectric substrates.

Developed with the designer of high-frequency/high-speed circuits in mind, FEM Simulator offers a powerful finite-element EM simulator that solves a wide array of applications with impressive accuracy and speed.

By combining fast solution times, efficient memory usage, and powerful display capabilities, FEM Simulator delivers leading price/performance solution to complex high-frequency problems. FEM Simulator users need very little background in electromagnetic field theory in order to operate and achieve accurate, meaningful solutions.

Major Features and Benefits

FEM in EMPro provides the following key technological features, which demonstrate the advantages of full 3D EM design and verification:

- Conductors, isotropic dielectrics, isotropic linear magnetic material modeling that allow a wide range of application coverage.
- An unlimited number of ports, which enables simulating multi-I/O design applications such as packages.
- Electric and magnetic fields modeling, enabling visualization of EM fields in a design.
- Absorbing boundary condition (free space), enabling antenna modeling.
- Full-wave, EM-accuracy for first-pass design success.
- Antenna parameters (gain, directivity, polarization, and so on), to enable better insight into antenna design.
- FEM Simulator/ADS integration providing an integrated approach to EM/Circuit design.

Application Areas

EM modeling tools are known for their great accuracy. FEM Simulator redefines this term with broad application coverage, including the following:

- Microstrip, stripline, CPW elements (filters, couplers, spiral inductors, via holes, air bridges, meander lines...)
- Multilayer structures
- Ceramic filters
- Adapters/transitions
- Antennas
- Couplers
- Power splitters/combiners

Visualization and Display of Results

The visualization and animation capabilities in FEM Simulator enable you to evaluate simulation results thoroughly. For analyzing designs, you can perform the EM field animation and dynamic rotation of structures simultaneously. You can choose from shaded plots or vectors. 3D far-field plots illustrate beam shapes in both azimuth and elevation on a single plot. To aid in analyzing your designs, EM field animation and dynamic rotation of structures can be performed simultaneously. You can choose from shaded plots, contour lines, or vectors. 3D far-field plots illustrate beam shapes in both azimuth and elevation on a single plot.

FEM Simulator Overview

The FEM Simulator commands are available from the EMPro GUI. The following steps describe a typical process for creating and simulating a design with the FEM Simulator:

1. **Create a physical design**: Create a physical design for your FEM simulation in EMPro.
2. **Set Simulation Options**: A mesh is a pattern of tetrahedra that is applied to a design in order to break down (discretize) the design into small cells. A mesh is required in order to simulate the design effectively. You can specify a variety of mesh parameters to customize the mesh to your design, or use default values and let FEM generate an optimal mesh automatically.
3. **Simulate the circuit**: You set up a simulation by specifying the parameters of a frequency plan, such as the frequency range of the simulation and the sweep type. When the setup is complete, you run the simulation. The simulation process uses the mesh pattern, and the electric fields in the design are calculated. S-parameters are then computed based on the electric fields. If the Adaptive Frequency Sample sweep type is chosen, a fast, accurate simulation is generated, based on a rational fit model.
4. **View the results**: The data from an FEM simulation is saved as S-parameters or as fields.
5. **FEM Advance Visualization**: The FEM Advance Visualization tool enables you to view and analyze, S-parameters, far-fields, and antenna parameters. Data can be analyzed in a variety of 2D and 3D plot formats. Some types of data are displayed in tabular form.
6. **Radiation patterns**: Once the electric fields on the circuit are known, the electromagnetic fields can be computed. They can be expressed in the spherical coordinate system attached to your circuit.

Understanding the Process of FEM Simulation

The simulation technique used to calculate the full three-dimensional electromagnetic field inside a structure is based on the finite element method. This section provides an overview of the FEM simulation process, its implementation in EMPro, and a description of how S-parameters are computed from the simulated electric and magnetic fields.

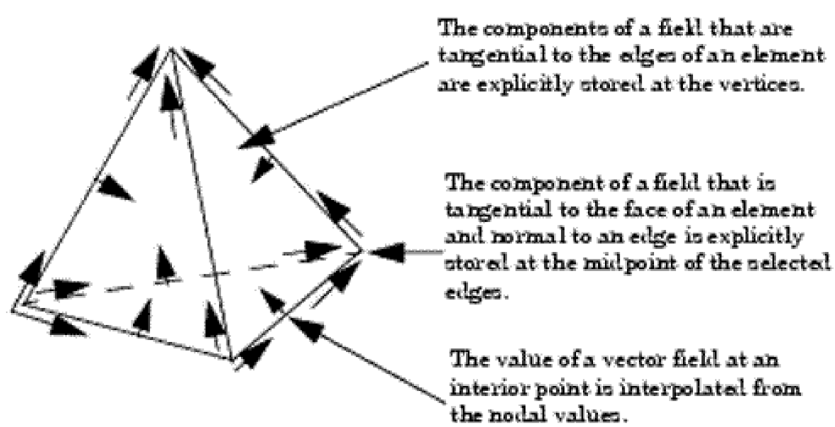
The Finite Element Method

The finite element method divides the full problem space into many smaller regions (elements) and represents the field in each element by a local function. The FEM implementation in EMPro uses tetrahedral elements.

Representation of a Field Quantity

The value of a vector field quantity (such as the H-field or the E-field) at points inside each tetrahedron is interpolated from the vertices of the tetrahedron. At each vertex, FEM Simulator stores the components of the field that are tangential to the three edges of the tetrahedron. In addition, the component of the vector field at the midpoint of selected edges that is tangential to a face and normal to the edge can also be stored. The field inside each tetrahedron is interpolated from these nodal values.

Field Quantities are Interpolated from Nodal Values



The components of a field that are tangential to the edges of an element are explicitly stored at the vertices.
The component of a field that is tangential to the face of an element and normal to an edge is explicitly stored at the midpoint of the selected edges.

The value of a vector field at an interior point is interpolated from the nodal values.
By representing field quantities in this way, Maxwell's equations can be transformed into matrix equations that are solved using traditional numerical methods.

Basis Functions

A first-order tangential element basis function interpolates field values from both nodal values at vertices and on edges. First-order tangential elements have 20 unknowns per tetrahedra.

Size of Mesh Versus Accuracy

There is a trade-off between the size of the mesh, the desired level of accuracy, and the amount of available computing resources.

On one hand, the accuracy of the solution depends on how small each of the individual elements (tetrahedra) are. Solutions based on meshes that use a large number of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each tetrahedron must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

On the other hand, generating a field solution for meshes with a large number of elements requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh that is fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.

To produce the optimal mesh, FEM Simulator uses an iterative process in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh based on suitable error criteria and generates a new solution. When selected S-parameters converge to within a desired limit, the iteration process ends.

Field Solutions

During the iterative solution process, the S-parameters typically stabilize before the full field solution. Therefore, when you are interested in analyzing the field solution associated with a structure, it may be desirable to use convergence criteria that is tighter than usual.

In addition, for any given number of adaptive iterations, the magnetic field (H-field) is less accurate than the solution for the electric field (E-field) because the H-field is computed from the E-field using the following relationship:

$$\boldsymbol{H} = \frac{\nabla \times \boldsymbol{E}}{-j\omega\mu}$$

thus making the polynomial interpolation function an order lower than those used for the electric field.

Implementation Overview

To calculate the S-matrix associated with a structure, the following steps are performed:

1. The structure is divided into a finite element mesh.
2. The waves on each port of the structure that are supported by a transmission line having the same cross section as the port are computed.
3. The full electromagnetic field pattern inside the structure is computed, assuming that each of the ports is excited by one of the waves.
4. The generalized S-matrix is computed from the amount of reflection and transmission that occurs.

The final result is an S-matrix that allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals, reducing the full three-dimensional electromagnetic behavior of a structure to a set of high frequency circuit values.

The Solution Process

There are three variations to the solution process:

- Adaptive solution
- Non-adaptive discrete frequency sweep
- Non-adaptive fast frequency sweep

Adaptive Solution

An adaptive solution is one in which a finite element mesh is created and automatically refined to increase the accuracy of succeeding adaptive solutions. The adaptive solution is performed at a single frequency. (Often, this is the first step in generating a non-adaptive frequency sweep or a fast frequency sweep).

Non-adaptive Discrete Frequency Sweep

To perform this type of solution, an existing mesh is used to generate a solution over a range of frequencies. You specify the starting and ending frequency, and the interval at which new solutions are generated. The same mesh is used for each solution, regardless of the frequency.

Non-adaptive Fast Frequency Sweep

This type of solution is similar to a discrete frequency sweep, except that a single field solution is performed at a specified center frequency. From this initial solution, the system employes asymptotic waveform evaluation (AWE) to extrapolate an entire bandwidth of solution information. While solutions can be computed and viewed at any frequency, the solution at the center frequency is the most accurate.

The Mesher

A mesh is the basis from which a simulation begins. Initially, the structure's geometry is divided into a number of relatively coarse tetrahedra, with each tetrahedron having four triangular faces. The mesher uses the vertices of objects as the initial set of tetrahedra vertices. Other points are added to serve as the vertices of tetrahedra only as needed to create a robust mesh. Adding points is referred to as seeding the mesh.

After the initial field solution has been created, if adaptive refinement is enabled, the mesh is refined further.

2D Mesh Refinement

For 2D objects or ports, the mesher treats its computation of the excitation field pattern as a two-dimensional finite element problem. The mesh associated with each port is simply the 2D mesh of triangles corresponding to the face of tetrahedra that lie on the port surface.

The mesher performs an iterative refinement of this 2D mesh as follows:

1. Using the triangular mesh formed by the tetrahedra faces of the initial mesh, solutions for the electric field, **E**, are calculated.
2. The 2D solution is verified for accuracy.
3. If the computed error falls within a pre-specified tolerance, the solution is accepted. Otherwise, the 2D mesh on the port face is refined and another iteration is performed.
   Any mesh points that have been added to the face of a port are incorporated into the full 3D mesh.

Modes

For a waveguide or transmission line with a given cross section, there is a series of basic field patterns (modes) that satisfy Maxwell's equations at a specific frequency. Any linear combination of these modes can exist in the waveguide.

Modes, Reflections, and Propagation

It is also possible for a 3D field solution generated by an excitation signal of one specific mode to contain reflections of higher-order modes which arise due to discontinuities in a high frequency structure. If these higher-order modes are reflected back to the excitation port or transmitted onto another port, the S-parameters associated with these modes should be calculated.

If the higher-order mode decays before reaching any port-either because of attenuation due to losses or because it is a non-propagating evanescent mode-there is no need to obtain the S-parameters for that mode. Therefore, one way to avoid the need for computing the S-parameters for a higher-order mode is to include a length of waveguide in the geometric model that is long enough for the higher-order mode to decay.

For example, if the mode 2 wave associated with a certain port decays to near zero in 0.5 mm, then the "constant cross section" portion of the geometric model leading up to the port should be at least 0.5 mm long. Otherwise, for accurate S-parameters, the mode 2 S-parameters must be included in the S-matrix.

The length of the constant cross section segment to be included in the model depends on the value of the mode's attenuation constant, α.

Modes and Frequency

The field patterns associated with each mode generally vary with frequency. However, the propagation constants and impedance always vary with frequency. Therefore, when a frequency sweep has been requested, a solution is calculated for each frequency point of interest.

When performing frequency sweeps, be aware that as the frequency increases, the likelihood of higher-order modes propagating also increases.

Modes and Multiple Ports on a Face

Visualize a port face on a microstrip that contains two conducting strips side by side as two separate ports. If the two ports are defined as being separate, they are treated as two ports are connected to uncoupled transmission structures. It is as if a conductive wall separates the excitation waves.

However, in actuality, there will be electromagnetic coupling between the two strips. The accurate way to model this coupling is to analyze the two ports as a single port with multiple modes.

The 3D Solver

To calculate the full 3D field solution, the following wave equation is solved:

$$\nabla \times \left(\frac{1}{\mu_r}\nabla \times \boldsymbol{E}(x,y,z)\right) - k_0^2 \varepsilon_r \boldsymbol{E}(x,y,z) = 0$$

where:

- *E(x,y,z)* is a complex vector representing an oscillating electric field.
- μ_r(*x, y*) is the complex relative permeability.
- k_0 is the free space phase constant, $\omega\sqrt{\mu_0 \varepsilon_0}$,
- ω is the angular frequency, 2πf.
- ε_r(*x, y*) is the complex relative permittivity.

This is the same equation that the 2D solver solves for in calculating the 2D field pattern at each port. The difference is that the 3D solver does not assume that the electric field is a traveling wave propagating in a single direction. It assumes that the vector E is a function of x, y, and z. The physical electric field, *E(x,y,z,t)* , is the real part of the product of the phasor, *E(x,y,z)* , and ε^{j} ω *t* :

$$\boldsymbol{E}(x,y,z,t) = Re[\boldsymbol{E}(x,y,z)e^{j\omega t}]$$

Boundary Conditions

FEM Simulator imposes boundary conditions at all surfaces exposed to the edge of the meshed problem region. This includes all outer surfaces and all surfaces exposed to voids and surface discontinuities within the structure.
The following types of boundary conditions are recognized by the 3D solver:

- Port
- Absorbing (Radiation)
- PEC (Perfect Electric Conductor)
- PMC (Perfect Magnetic Conductor)
- ESymmetry (Electric Symmetry condition - odd symmetry of tangential E-fields)

- MSymmetry (Magnetic Symmetry condition - even symmetry of tangential E-fields)

Port Boundaries

The 2D field solutions generated by the 2D solver for each port serve as boundary conditions at those ports. The final field solution that is computed for the structure must match the 2D field pattern at each port.

FEM Simulator solves several problems in parallel. Consider the case of analyzing modes 1 and 2 in a two-port device. To compute how much of a mode 1 excitation at port 1 is transmitted as a mode 2 wave at port 2, the 3D mesher uses the following as boundary conditions:

- A "mode 1" field pattern at port 1.
- A "mode 2" field pattern at port 2.

To compute the full set of S-parameters, solutions involving other boundary conditions must also be solved. Because the S-matrix is symmetric for reciprocal structures (that is, S_{12} is the same as S_{21}), only half of the S-parameters need to be explicitly computed.

Absorbing (Radiation)

Radiation boundaries model surfaces that represent open space. Energy is allowed to radiate from these boundaries instead of being contained within them. At these surfaces, the second order radiation boundary condition is employed:

$$(\nabla \times \boldsymbol{E})_{tan} = jk_0\boldsymbol{E}_{tan} - \frac{j}{k_0}\nabla \times \hat{n}(\nabla \times \boldsymbol{E})_n + \frac{j}{k_0}\nabla_{tan}(\nabla_{tan} \bullet \boldsymbol{E}_{tan})$$

where:

- E tan is the component of the E-field that is tangential to the surface.
- $\hat{n}$ is the unit vector normal to the radiation surface.
- k_0 is the free space phase constant, $\omega\sqrt{\mu_0\varepsilon_0}$.
- j is equal to $\sqrt{-1}$.

To ensure accurate results, radiation boundaries should be applied at least one quarter of a wavelength away from the source of the signal. However, they do not have to be spherical. The only restriction regarding their shape is that they be convex with regard to the radiation source.

Computing Radiated Fields

Electromagnetic Design System maps the E-field computed by the 3D solver on the radiation surfaces to plane registers and then calculates the radiated E-field using the following equation:

$$\boldsymbol{E}(x,y,z) = \int_s ((j\omega\mu_0\boldsymbol{H}_{tan})G + (\boldsymbol{E}_{tan} \times \nabla G) + (\boldsymbol{E}_{normal}\nabla G))ds$$

where:

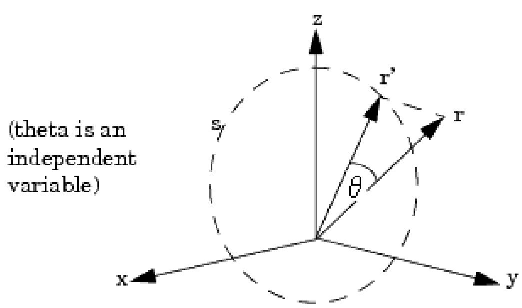
- *s* represents the radiation surfaces.
- *j* is the imaginary unit, $\sqrt{-1}$.
- ω is the angular frequency, 2πf.
- μ_0 is the relative permeability of the free space.
- *H* tan is the component of the magnetic field that is tangential to the surface.
- *H* normal is the component of the magnetic field that is normal to the surface.
- *E* tan is the component of the electric field that is tangential to the surface.
- *G* is the free space Green's Function, given by:

$$G = \frac{e^{-jk_0|\bar{r} - \bar{r}'|}}{|\bar{r} - \bar{r}'|}$$

where:

- k $_0$ is the free space wave number, $\omega\sqrt{\mu_0\varepsilon_0}$.
- *r* and $\bar{r}'$ represent, respectively.

Implementing Green's Function When Computing Radiated Fields



Displaying Field Solutions

The 3D solver is also used to manipulate field quantities for display. The system enables you to display or manipulate the field associated with any excitation wave at any port-for example, the field inside the structure due to a discrete mode 2 excitation wave at port 3. Waves excited on different modes can also be superimposed, even if they have different magnitudes and phases-for example, the waves excited on mode 1 at port 1 and mode 2 at port 2. In addition, far-field radiation in structures with radiation boundaries can be displayed.

The available fields depend on the type of solution that was performed:

- For adaptive solutions, the fields associated with the solution frequency are available.
- For frequency sweeps, the fields at each solved frequency point are available.
- For fast frequency sweeps, the fields associated with the center frequency point are initially available.

Ports-only Solutions and Impedance Computations

This section addresses how impedances are computed for multi-conductor transmission line ports. Some examples of such structures are:

- Two coupled microstrip lines
- Coplanar waveguide modeled with 3 separate strips
- Shielded twin-wire leads

For structures with one or two conductors, you will need to define a single line segment, called an "impedance line", for each mode. Some examples of such structures include:

- microstrip transmission line (two-conductor structure)
- grounded CPW (two-conductor structure) where the CPW ground fins are also attached to the *OUTER* ground

For these structures, the port solver will compute the voltage V along the impedance line which is used to calculate $Z_{pv} = V^2/(2*\text{Power})$. The power is always normalized to 1 Watt.

If one models N-conductor structures where N>2, then FEM Simulator uses a different algorithm for computing Z_{pv} and Z_{pi}. The user must define an impedance line for each interior conductor. The impedance lines should go from the center of each interior conductor to the outer conductor.

The port solver computes a voltage along the first N line segments for each of the first N modes when the port solver detects that there are N+1 conductors. This becomes a "voltage vector" $\bar{V}$
(of length N) for each of the N quasi-TEM modes. Then, when computing Z_{pv}, the square of the scalar voltage is now replaced by the dot product of the voltage vectors, for example:

$$Z_{pv} = \bar{V} \bullet \bar{V} / (2 \cdot Power)$$

For a more detailed explanation as to why this is done, refer to reference "1" at the end of this section.
For Z_{pi}, the current is generally computed by adding the currents flowing into and out of the port and taking the average of the two. (If the simulator computed currents to perfect accuracy, the inward and outward currents would be identical.) For all mode numbers >= N, where N = number of conductors, the currents are calculated in this way.
For the first (N-1) quasi-TEM modes, the currents are computed on the N-1 interior conductors producing a current eigenvector. Then the impedance:

$$Z_{pi} = (2 \cdot Power) / (\bar{I} \bullet \bar{I})$$

The result is that the FEM Simulator impedance computations for such structures as coupled microstrip lines match the published equations for even- and odd-mode impedances.

As an example, take a CPW modeled as three interior strips surrounded by an enclosure. The ground strips do not touch the enclosure. Such a model is in the examples directory of FEM Simulator and is called *cpwtaper* . The port solver shows us that the desired CPW mode is not the dominant mode, but is actually mode 3. To identify the modes, one can use the arrow plots in the Port Calibration menu or the Arrow display of the E-field in the post processor.

Each port consists of a 4 conductor system, the outer (ground) conductor, the inner strip, and the two "ground" strips. This results in 3 quasi-TEM modes. Mode 1 has E-field lines predominately in the substrate, all pointing in the same direction. This is the common mode (+V, +V, +V). Mode 2 also has E-field lines predominately in the substrate, but in opposite directions under the two "ground" strips. This is the slot mode (-V, 0, +V). Mode 3 has nearly zero E-fields everywhere because the fields are predominately between the inner strip and the "ground" strips. This is the CPW mode (0, +V, 0).

For further help in identifying modes in such a structure, one can look at the distributions with the "full" scale. One will notice that modes 1 and 3 obviously have the same "even" symmetry in the E fields, while mode 2 has an odd symmetry. The CPW mode has an "even" symmetry, so it has to be mode 1 or 3.

Modes 1 and 2 have significant E-field strengths in the substrate, especially under the "ground" strips. So, there is a potential difference between the "ground" strips and the outer ground for these two modes. However, for mode 3, the "ground" strips are at the same potential as the outer ground, which is consistent with the CPW mode.

Thus one can identify the modes. For mode 3, the "ground" strips are at 0 volts with respect to the outer ground, and the signal line has +V. The Z_{pv} impedance computed for this mode using a dot product of the voltage vector for mode 3 gives the same Z_{pv} as by computing the simple voltage between center strip and either "ground" strip. That is because the voltage vector for mode 3 along the three impedance lines is $\bar{V}$ =[0,V,0] and

$$\bar{V} \cdot \bar{V} = [0, V, 0] \cdot [0, V, 0] = 0 \cdot 0 + V^2 + 0 \cdot 0 = V^2$$

However, the impedances for the other modes now match accepted impedance definitions found in the literature for multi-conductor transmission lines.

1. G.G. Gentili and M. Salazar-Palma, "The definition and computation of modal characteristic impedance in quasi-TEM coupled transmission lines," IEEE Trans. Microwave Theory Tech., Feb. 1995, pp. 338-343.

Calculating S-Parameters in FEM Simulation

A generalized S-matrix describes what fraction of power associated with a given field excitation is transmitted or reflected at each port.

The S-matrix for a three port structure is shown below:

$$\begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

where:

- All quantities are complex numbers.
- The magnitudes of a and b are normalized to a field carrying one watt of power.
- $|a_i|^2$ represents the excitation power at port i.
- $|b_i|^2$ represents the power of the transmitted or reflected field at port i.
- The full field pattern at a port is the sum of the port's excitation field and all reflected/transmitted fields.
- The phase of a_i and b_i represent the phase of the incident and reflected/transmitted field at t=0, $\angle a_i$
- Represents the phase angle of the excitation field on port i at t=0. (By default, it is zero.)
$\angle b_i$
- Represents the phase angle of the reflected or transmitted field with respect to the excitation field.
- S_{ij} is the S-parameter describing how much of the excitation field at port j is reflected back or transmitted to port i.

For example, S_{31} is used to compute the amount of power from the port 1 excitation field that is transmitted to port 3. The phase of S_{31} specifies the phase shift that occurs as the field travels from port 1 to port 3.

> **Note**
> When the 2D solver computes the excitation field for a given port, it has no information indicating which way is *up* or *down*. Therefore, if ports have not been calibrated, it is possible to obtain solutions in which the S-parameters are out of phase with the expected solution.

Frequency Points

The S-parameters associated with a structure are a function of frequency. Therefore, separate field solutions and S-matrices are generated for each frequency point of interest. FEM Simulator supports two types of frequency sweeps:

- Discrete frequency sweeps, in which a solution is generated for the structure at each frequency point you specify.
- Fast frequency sweeps, in which asymptotic waveform evaluation is used to extrapolate solutions for a range of frequencies from a single solution at a center

frequency.

Fast frequency sweeps are useful for analyzing the behavior of high Q structures. For wide bands of information, they are much faster than solving the problem at individual frequencies.

> **Note**
> Within a fast frequency solution, there is a bandwidth where the solution results are most accurate. This range is indicated by an error criterion using a matrix residue that measures the accuracy of the solution. For complex frequency spectra that have many peaks and valleys, a fast sweep may not be able to accurately model the entire frequency range. In this case, additional fast sweeps with different expansion frequencies will automatically be computed and combined into a single frequency response.

Z- and Y-Matrices

Calculating and displaying the unique impedance matrices (Z) associated with a structure is performed in the post processor.

Characteristic Impedances

FEM Simulator calculates the characteristic impedance of each port in order to compute a renormalized S-matrix, Z-matrix, or Y-matrix. The system computes the characteristic impedance of each port in three ways-as Z_{pi}, Z_{pv}, and Z_{vi} impedances.

You have the option of specifying which impedance is to be used in the renormalization calculations.

PI Impedance

The Z_{pi} impedance is the impedance calculated from values of power (P) and current (I):

$$Z_{pi} = \frac{2P}{\boldsymbol{I} \bullet \boldsymbol{I}}$$

The power and current are computed directly from the simulated fields. The power passing through a port is equal to the following:

$$P = \frac{1}{2}\oint_s E \times H ds$$

where the surface integral is over the surface of the port.

The current is computed by applying Ampere's Law to a path around the port:

$$I = \oint_l H \bullet dl$$

While the net current computed in this way will be near zero, the current of interest is that flowing into the structure, I^- or that flowing out of the structure, I^+. In integrating around the port, the system keeps a running total of the contributions to each and uses the average of the two in the computation of impedances.

PV Impedance

The Z $_{pv}$ impedance is the impedance calculated from values of power (P) and voltage (V):

$$Z_{pv} = \frac{V \bullet V}{2P}$$

where the power and voltage are computed directly from the simulated fields. The power is computed in the same way as for the Z $_{pi}$ impedance. The voltage is computed as follows:

$$V = \oint_l E \bullet dl$$

The path over which the system integrates is referred to as the impedance line, which is defined when setting up the ports. To define the impedance line for a port, select the two points across which the maximum voltage difference occurs. FEM Simulator cannot determine where the maximum voltage difference will be unless you define an impedance line.

VI Impedance

The Z $_{vi}$ impedance is given by:

$$Z_{vi} = \sqrt{Z_{pi} Z_{pv}}$$

For TEM waves, the Z $_{pi}$ and Z $_{pv}$ impedances form upper and lower boundaries to a port's actual characteristic impedance. Therefore, the value of Z $_{vi}$ approaches a port's actual impedance for TEM waves.

Choice of Impedance

- When the system is instructed to renormalize the generalized S-matrix or compute a Y- or Z-matrix, you must specify which value to use in the computations, Z $_{pi}$, Z $_{pv}$, or Z $_{vi}$.
- For TEM waves, the Z $_{vi}$ impedance converges on the port's actual impedance and should be used.
- When modeling microstrips, it is sometimes more appropriate to use the Z $_{pi}$ impedance.
- For slot-type structures (such as finline or coplanar waveguides), Z $_{pv}$ impedance is the most appropriate.

De-embedding

If a uniform length of transmission line is added to (or removed from) a port, the S-matrix of the modified structure can be calculated using the following relationship:

$$[S'] = [e^{\gamma l}][S][e^{\gamma l}]$$

Where, ε^{γ^l} is a diagonal matrix with the following entries:

$$\begin{bmatrix} e^{\gamma_1 l_1} & 0 & 0 \\ 0 & e^{\gamma_2 l_2} & 0 \\ 0 & 0 & e^{\gamma_3 l_3} \end{bmatrix}$$

$\gamma = \alpha + j\beta$ is the complex propagation constant, where:

- αi is the attenuation constant of the wave of port *i*.
- βi is the propagation constant associated with the uniform transmission line at port *i*.
- *l* i is the length of the uniform transmission line that has been added to or removed from the structure at port *i*. A positive value indicates that a length of transmission line has been removed from the structure.

The value of γ for the dominant mode of each port is automatically calculated by the 2D solver.

Equations

The sections below describe some of the equations that are solved in a simulation or used to define elements of a structure.

Derivation of Wave Equation

The solution to the following wave equation is found during a simulation:

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}\right) - k_0^2 \varepsilon_r \boldsymbol{E} = 0$$

where:

- E(*x,y,z*) is a phasor representing an oscillating electric field
- k_0 is the free space wave number, $\omega\sqrt{\mu_0\varepsilon_0}$.
- ω is the angular frequency, $2\pi f$.
- μ_r(*x,y,z*) is the complex relative permeability.
- ε_r(*x,y,z*) is the complex relative permittivity.

The difference between the 2D and 3D solvers is that the 2D solver assumes that the electric field is a traveling wave with this form:

$$\boldsymbol{E}(x,y,z,t) = Re[\boldsymbol{E}(x,y)e^{(j\omega t - \gamma z)}]$$

while the 3D solver assumes that the phasor E is a function of x, y, and z:

$$\boldsymbol{E}(x,y,z,t) = \Re[\boldsymbol{E}(x,y,z)e^{j\omega t}]$$

Maxwell's Equations

The field equation solved during a simulation is derived from Maxwell's Equations, which in their time-domain form are:

$$\nabla \times \boldsymbol{H}(t) = \boldsymbol{J}(t) + \frac{\partial}{\partial t}\boldsymbol{D}(t)$$

$$\nabla \times \boldsymbol{E}(t) = \frac{\partial}{\partial t}\boldsymbol{B}(t)$$

$$\nabla \bullet \boldsymbol{D}(t) = \rho$$

$$\nabla \bullet \boldsymbol{B}(t) = 0$$

Where:

- **E** (t) is the electric field intensity.
- **D** (t) is the electric flux density, ε **E** (t), and ε is the complex permittivity.
- **H** (t) is the magnetic field intensity.
- **B** (t) is the magnetic flux density, μ **H** (t), and μ is the complex permeability.
- **J** (t) is the current density, σ **E** (t).
- ρ is the charge density.

Phasor Notation

Because all time-varying electromagnetic quantities are oscillating at the same frequency, they can be treated as phasors multiplied by $e^{j\omega t}$ (in the 3D solver) or by $e^{j\omega t - \gamma z}$ (in the 2D solver).
In the general case with the 3D solver, the equations become:

$$\nabla \times \boldsymbol{H}e^{j\omega t} = \boldsymbol{J}e^{j\omega t} + \frac{\partial}{\partial t}\boldsymbol{D}e^{j\omega t}$$

$$\nabla \times \boldsymbol{E}e^{j\omega t} = \frac{\partial}{\partial t}\boldsymbol{B}e^{j\omega t}$$

$$\nabla \bullet \boldsymbol{D}e^{j\omega t} = \rho e^{j\omega t}$$

$$\nabla \bullet \boldsymbol{B} e^{j\omega t} = 0$$

By factoring out the quantity $e^{j \omega t}$ and using the following relationships:

$$\frac{\partial}{\partial t} \boldsymbol{E} e^{j\omega t} = j\omega \boldsymbol{E} e^{j\omega t}$$

$$\frac{\partial \boldsymbol{H}}{\partial t} = j\omega \boldsymbol{H}$$

Maxwell's Equations in phasor form reduce to:

$$\nabla \times \boldsymbol{H} = \boldsymbol{J} + j\omega \boldsymbol{E}$$

$$\nabla \times \boldsymbol{E} = -j\omega \boldsymbol{B}$$

$$\nabla \bullet \boldsymbol{D} = \rho$$

$$\nabla \bullet \boldsymbol{B} = 0$$

where **B** , **H** , **E** , and **D** are phasors in the frequency domain. Now, using the relationships **B** = μ **H** , **D** = ε **E** , and **J** = σ **E** , Maxwell's Equations in phasor form become:

$$\nabla \times \boldsymbol{H} = (j\omega\varepsilon + \sigma)\boldsymbol{E} = j(\omega\varepsilon)\boldsymbol{E}$$

for $\sigma = 0$

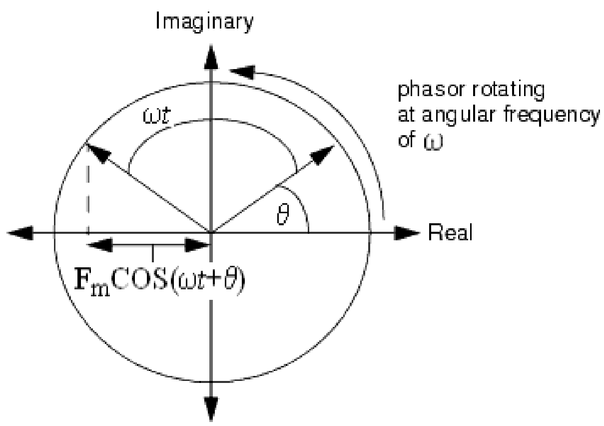
$$\nabla \times \boldsymbol{H} = j(\omega\varepsilon)\boldsymbol{E}$$

$$\nabla \bullet \varepsilon \boldsymbol{E} = \rho$$

$$\nabla \bullet \mu \boldsymbol{H} = 0$$

Where:

- **H** and **E** are phasors in the frequency domain, μ is the complex permeability, and ε is the complex permittivity.
- **H** and **E** are stored as phasors, can be visualized as a magnitude and phase or as a complex quantity.



Electric and Magnetic Fields Can Be Represented as Phasors

Assumptions

To generate the final field equation, place **H** in the equation in terms of **E** to obtain:

$$\boldsymbol{H} = \frac{1}{j\omega\mu}\nabla \times \boldsymbol{E}$$

Then, substitute this expression for **H** in the $\nabla \times H$ equation to produce:

$$\nabla \times \left(-\frac{1}{j\omega\mu}\nabla \times \boldsymbol{E}\right) = j\omega\varepsilon\boldsymbol{E}$$

Conductivity

Although good conductors can be included in a model, the system does not solve for any fields inside these materials. Because fields penetrate lossy conductors only to one skin depth (which is a very small distance in good conductors), the behavior of a field can be represented with an equivalent impedance boundary.

For perfect conductors, the skin depth is zero and no fields exist inside the conductor. Perfect conductors are assumed to be surrounded with Perfect E boundaries.

Dielectric Loss Tangent

Dielectric losses can be modeled by assuming that the relative permittivity, $\hat{\varepsilon}_r$, is complex:

$$\hat{\varepsilon} = \varepsilon' - \varepsilon''$$

Expressed in terms of the dielectric (electric) loss tangent, $\tan\delta_\varepsilon = \varepsilon''_r / (\varepsilon'_r)$, the complex relative permittivity, $\hat{\varepsilon}_r$ becomes:

$$\hat{\varepsilon}_r = \varepsilon_r' - j\varepsilon_r'\tan\delta_e$$

Magnetic Loss Tangent

Losses in magnetic materials can be modeled by assuming that μ_r is complex.

$$\hat{\mu}_r = \mu_r' - j\mu_r''$$

Expressed in terms of the magnetic loss tangent, $(\mu''_r)/(\mu'_r)$, the complex relative permeability becomes:

$$\hat{\mu}_r = \mu_r'\left(1 - \frac{j\mu_r''}{\mu_r'}\right) = \mu_r'(1 - j\tan\delta_m)$$

Definition of Freespace Phase Constant

Using the relationships $\varepsilon = \varepsilon_0\varepsilon_r$ and $\mu = \mu_0\mu_r$, the wave equation being solved can be placed in this form:

$$\nabla \times \left(\frac{1}{\mu_r}\nabla \times \boldsymbol{E}\right) - \omega^2\mu_0\varepsilon_0\varepsilon_r\boldsymbol{E} = 0$$

Now, if the freespace phase constant (or wave number) is defined as, $k_0^2 = \omega^2\mu_0\varepsilon_0$, the above reduces to:

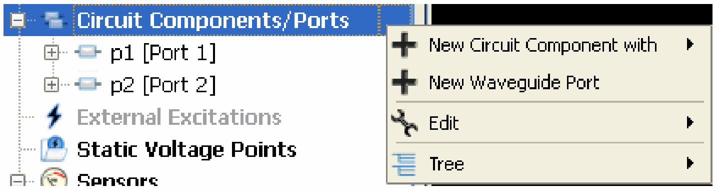
$$\nabla \times \left(\frac{1}{\mu_r}\nabla \times \boldsymbol{E}\right) - k_0^2\varepsilon_r\boldsymbol{E} = 0$$

which is the equation that the 2D and 3D engines solve.

Using Ports in EMPro FEM Simulation

EMPro FEM supports two kinds of ports: **Circuit Component Port** and **Waveguide Port**.

Circuit Component Port and Waveguide Port

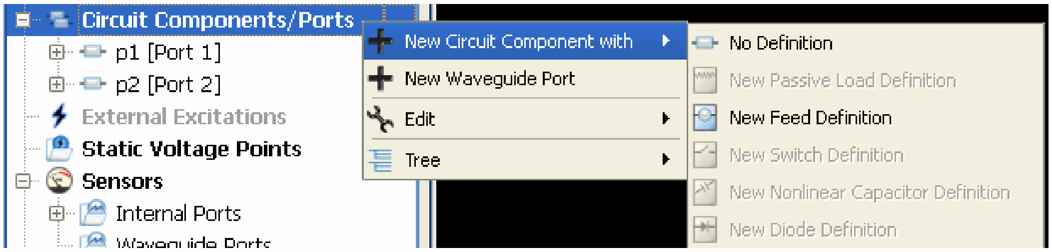


Circuit Component Port

EMPro supports the **Component Feed** port in the FEM simulation setup.

The Component Feed ports are direct point feeds having a negative and positive connection point to the metallization of the design.

Component Feed in Circuit Component Port



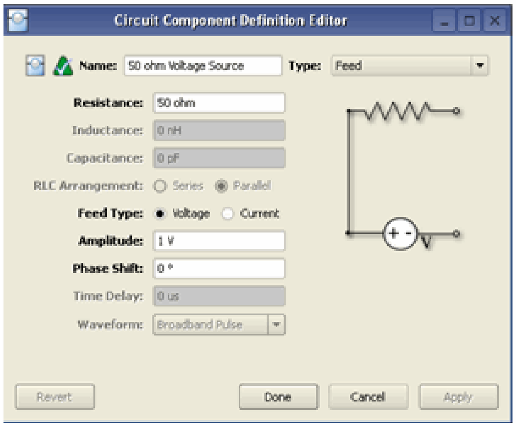
EMPro FEM only supports **New feed** definition. The Passive load, Switch, Non Linear Capacitor and Diode ports are not supported in FEM.
For more information about Component ports at Defining Circuit Components and Excitations, refer to the *Defining Circuit Components and Excitations* (using) section.

Component Feeds

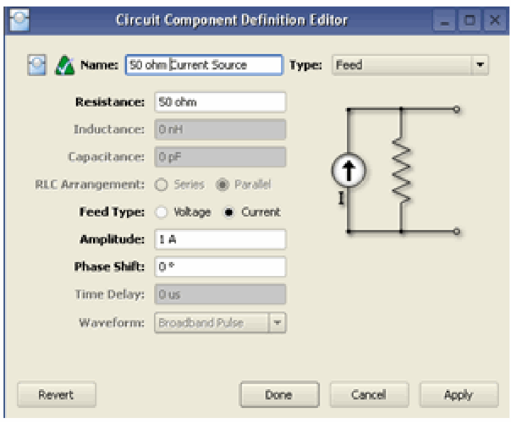
EMPro supports the **Component Feed** port in the FEM simulation setup. The Component Feed ports are direct point feeds having a negative and positive connection point to the metallization of the design.
For more information about Component Feeds, refer to the *Defining Circuit Components and Excitations* (using) section.

For FEM simulations, only the following component feed definitions are allowed, which specify the voltage sources with a termination impedance.

Feed Definition as Voltage



Feed Definition as Current

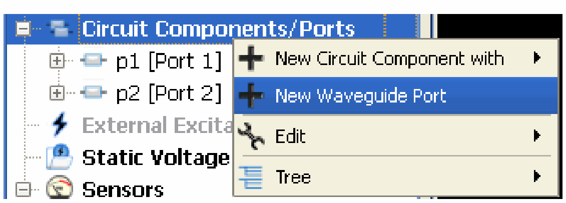


The termination resistance can be of any value, but Amplitude can be maximum of 1 V or 1 A. If while defining component port **No definition** option is used, then after defining port, appropriate definition of the Voltage source or Current source must be applied.

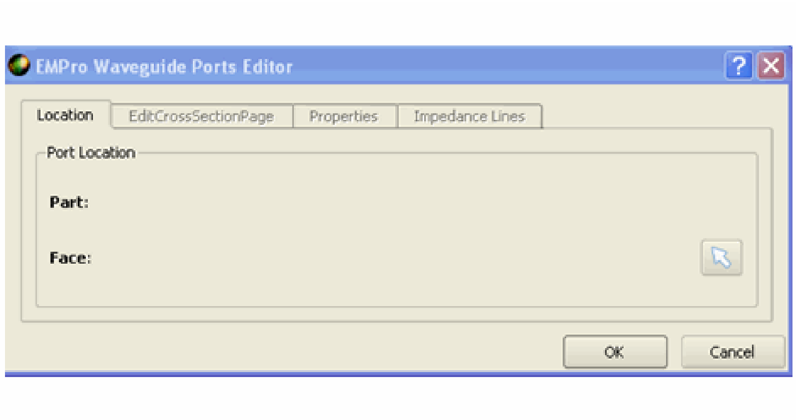
Waveguide Port

EMPro FEM also supports **Waveguide Port** option.

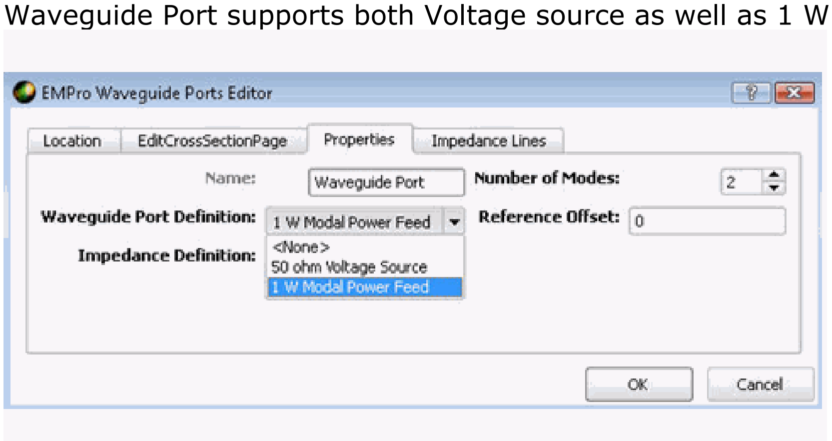
Waveguide Port in Circuit Component Port



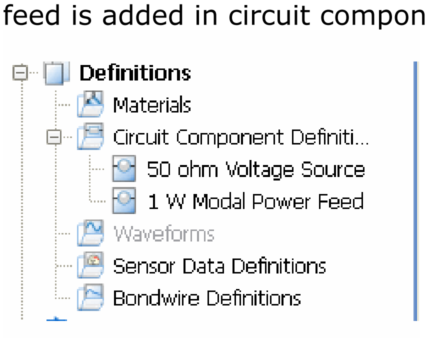
Waveguide Port is supported in FEM only. It is not supported in FDTD simulations. To activate the Waveguide Port, the application should be in FEM mode. After a new waveguide port option is chosen, the *Waveguide Port Editor* window opens.



Waveguide Port supports both Voltage source as well as 1 W modal power feed.



When waveguide port is chosen automatically, both Voltage source and 1 W Modal power feed is added in circuit component definition.



If there is a voltage source already present, Waveguide port will only add 1 W modal power source.

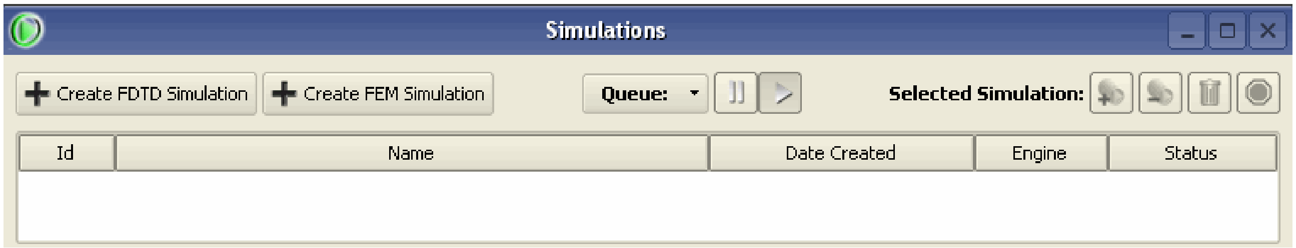
To set up a waveguide port with voltage source, refer to *Creating a Microstrip Line for FEM Simulation* (fem).

To set up a waveguide port with 1 W modal power source, refer to *Performing Multimode Analysis on Rectangular Waveguide* (fem). Modal power feed is useful if modal analysis of the structure needs to be carried.

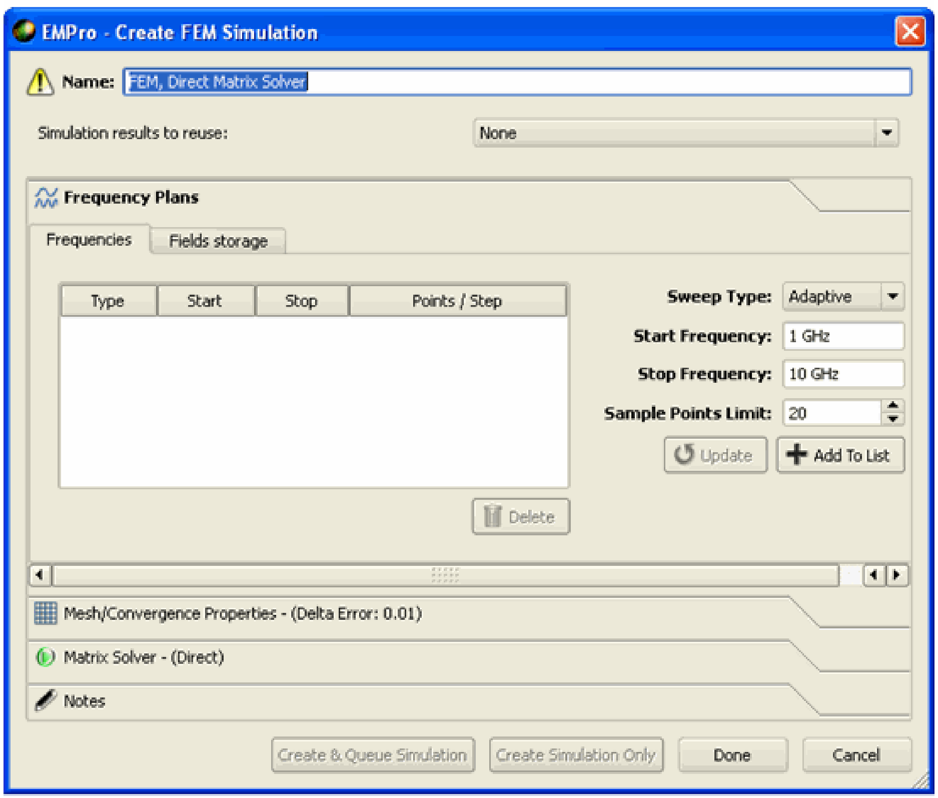
Specifying FEM Simulation Setup in EMPro

You can specify the simulation options that are specific to the FEM Simulator in the *Create FEM Simulation* dialog box. This dialog box enables you to control how the FEM mesh is generated and solution is achieved.

In the *Simulations* workspace window, click **New FEM Simulation**.



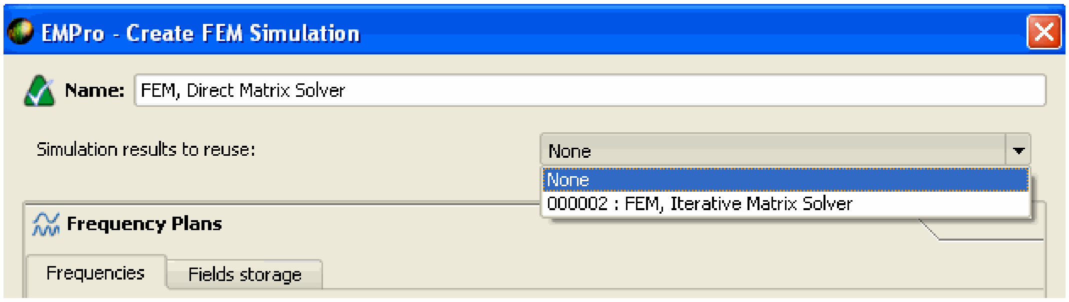
This will open a *Create FEM Simulation* dialog box where all the set up requirement for FEM simulation can be entered.



In the **Name** field, you can enter the FEM simulation name. Name suggested by default can be used or you can enter the simulation name as per your choice. The invalid symbol on the window refers that you have not entered any frequency plan.

Simulation Results to Reuse

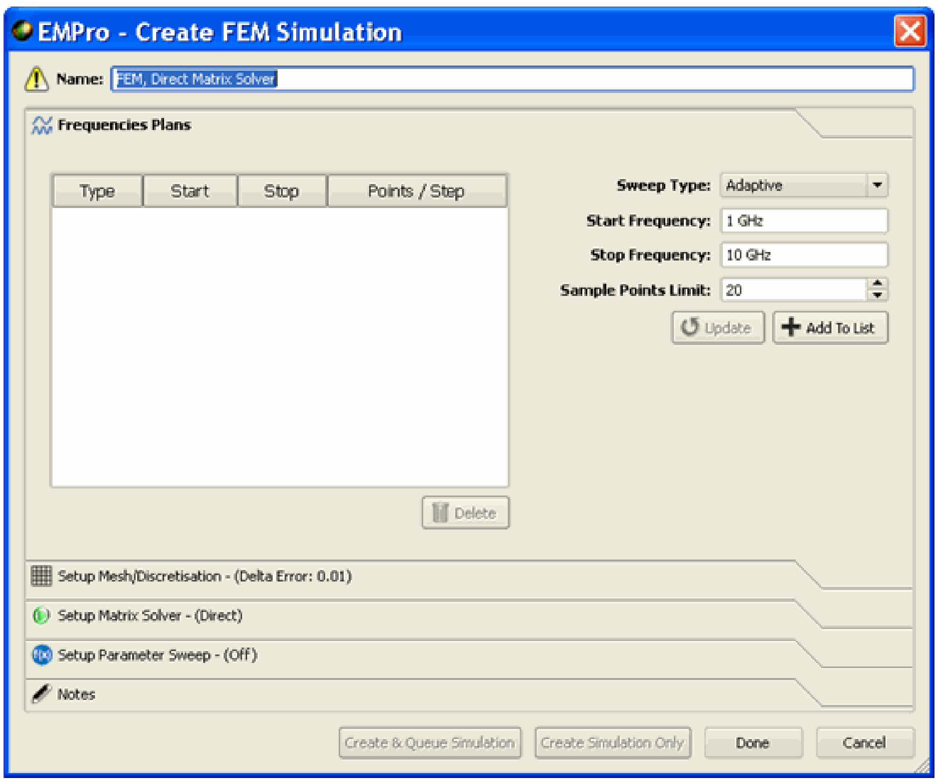
EMPro 2010 provides new feature of reusing any existing FEM simulation. For a new FEM simulation, simulation setup in terms of mesh refinement, AFS points, solver can be reused from existing simulation. In *Creating FEM Simulation* window, the **Simulation Results to Reuse** drop-down displays the list of already solved simulations. You can choose the FEM simulation to be reused using the **Simulation Results to Reuse** drop-down list.



To know more about the instances which are useful to reuse, refer to *Reuse of Mesh and Frequency Points* (fem) section.

Specifying Frequency Plans

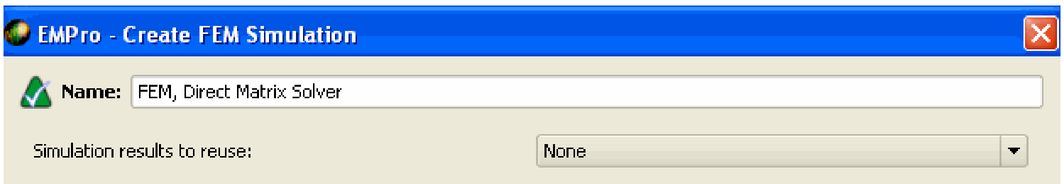
Using the *Create FEM Simulation* dialog box, you can specify the frequency settings for your FEM simulation.



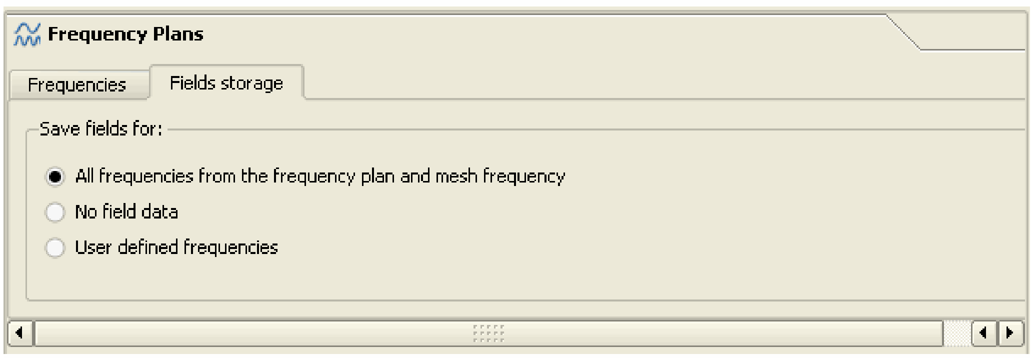
In the *Frequencies Plans* screen, specify the options as listed in the following table:

Option	Description
Sweep Type	Specify the type of sweep: Adaptive, Logarithmic, Linear, and Single.
Start Frequency	Specify the start frequency value in GHz.
Stop Frequency	Specify the stop frequency value in GHz.
Sample Points Limit	Specify the value of sample points limit.
Add To List	Adds the specified values to the pane present on the Frequencies Plans screen.
Update	Updates the values in the pane present on the Frequencies Plans screen.
Delete	Deletes the values specified in the pane present on the Frequencies Plans screen.

Once any frequency plan is added, the invalid symbol on the top of *Create FEM simulation* dialog box will turn to valid.



In the **Frequencies Plans > Field Storage** dialog box, specify the options as listed in the table following the below screenshot:

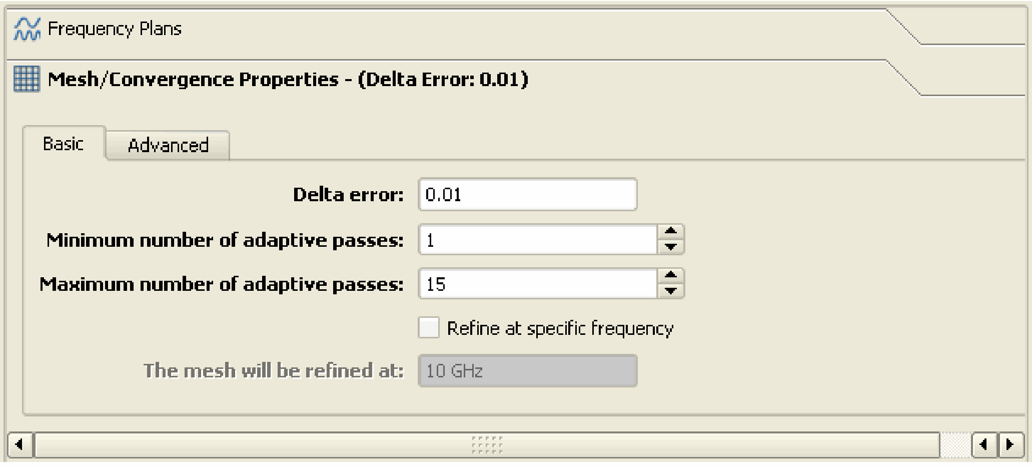


Option	Description
All frequencies from the frequency plan and mesh frequency	Field data will be stored for the frequencies for which either mesh is refined or solution is calculated.
No field data	Field data will not stored for any of the frequencies used in calculation. This will save space on hard disk. If the intention is just to see S parameter, this option can be used.
User defined frequencies	Field data will be stored for only specified frequencies. The specified frequencies are the same sued in frequency plan. It will be start and end frequency of AFS, any single frequency, all the frequencies of the linear & logarithmic frequencies.

Specifying the Mesh Refinement Settings

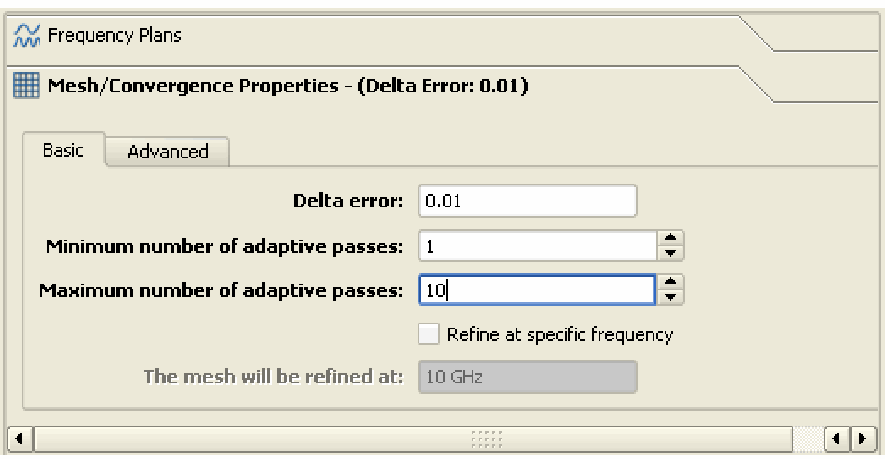
An FEM simulation mesh is a sub-division of the entire 3D problem domain into a set of tetrahedra (or cells). This pattern of cells is based on the geometry of a circuit and optionally, user-defined parameters, so each circuit will have a unique mesh calculated for it. The mesh is then applied to the circuit in order to compute the electric within each cell and identify any coupling effects in the circuit during simulation. From these calculations, S-parameters are then calculated for the circuit.

The FEM Simulator implements an adaptive mesh algorithm, where an initial mesh is generated and the electric fields (and S-parameters) are computed on that initial mesh for a single frequency. An error estimate is generated for each tetrahedron. The tetrahedra with the largest estimated error are refined to create a new mesh on which the electric fields (and S-parameters) are computed. The S-parameters from consecutive meshes are compared. If the S-parameters do not change significantly, then electric fields (and S-parameters) are computed for all the requested frequencies. If the S-parameters do change significantly, then new error estimates are computed, a new mesh is generated and new electric fields (and S-parameters) are computed.



Click **Mesh/Convergence Properties- (Delta Error: 0.01)** in the *Create FEM Simulation* window. This displays the **Mesh/Convergence Properties- (Delta Error: 0.01)** screen, which consists of two tabs: **Basic** and **Advanced**.

Specifying Basic Tab Settings

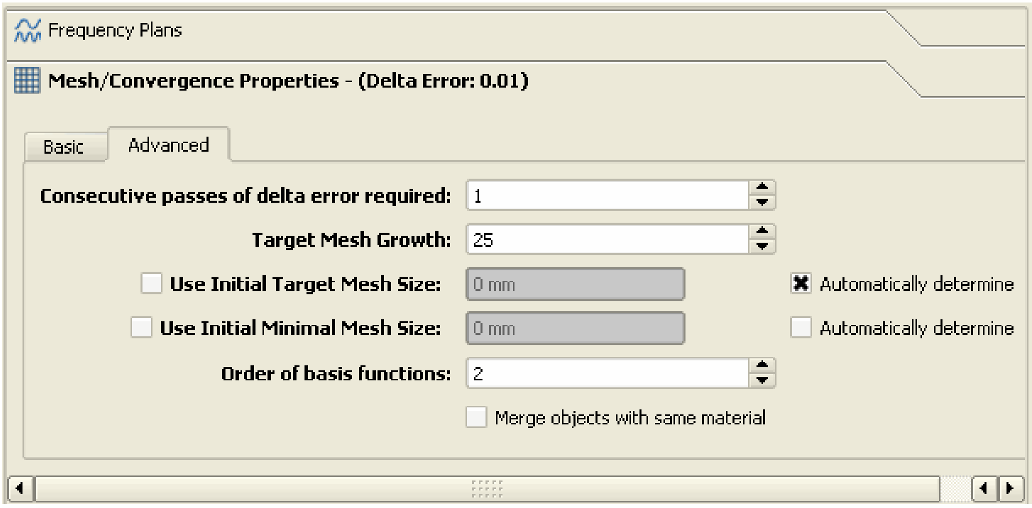


The following table describes the Basic tab settings:

Option	Description
Delta Error	Global Delta S-parameter sets a value that is applied to all S-parameters in the solution. Enter a value in the Delta Error field. This is the allowable change in the magnitude of the vector difference for all S-parameters for at least two consecutive refinement passes.
Minimum number of adaptive passes	Enter the number of consecutive passes that must meet or exceed the delta error. If you have concerns about convergence, you can increase this value, otherwise, use the default value of 5.
Maximum number of adaptive passes	Enter the maximum number of passes to be attempted. If the number of refinement passes entered is reached before the delta error criteria is met, the refinement process will end, based upon this limit. The number of passes from all prior simulations is also displayed. Typically, a value between 10 and 20 is recommended.
Refine at specific frequency	Select this option to enable the **The mesh will be refined at** text box. By default, the mesh refinement is performed at the highest frequency specified in the simulation. To change the frequency at which the mesh refinement is performed, enable **Refine at Specified Frequency** and enter a value in **The mesh will be refined at** field.
The mesh will be refined at	Enable **Refine at Specified Frequency** and enter a value in **The mesh will be refined at** field, in GHz.

Specifying Advanced tab Settings

Using the Advanced tab Settings, you can specify the consecutive passes of delta error, maximum refinement for target mesh growth, and merge objects with the same material.

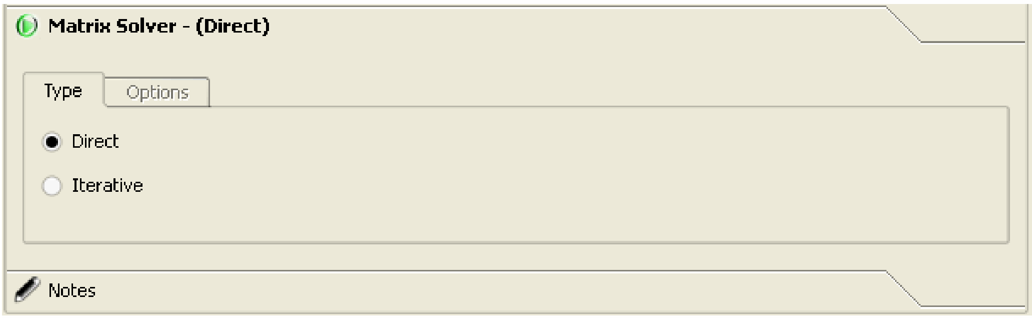


The following table describes the Advanced tab settings:

Option	Description
Consecutive passes of delta error required	Specify the required consecutive passes of delta error.
Maximum Refinement target mesh growth and value is in percentage.	Max Refinement Points enables you to specify the maximum number of points added at each mesh refinement iteration. You can set this limit between 400 and 30,000. The locations of the refinement points are determined by FEM Simulator.
Use Initial Target mesh size	An initial target mesh size can be provided by checking this option. If **Automatically determine** is checked then EMPro automatically determines the initial target mesh size. To know more, refer to New Features in EMPro 2010 > Pre-seeding of electrically large structures
Use Initial minimum mesh size	This option can be used to set minimum mesh size to begin mesh refinement process.
Merge objects with same material	Select this checkbox to merge objects with same material.

Specifying the Solver Type Settings

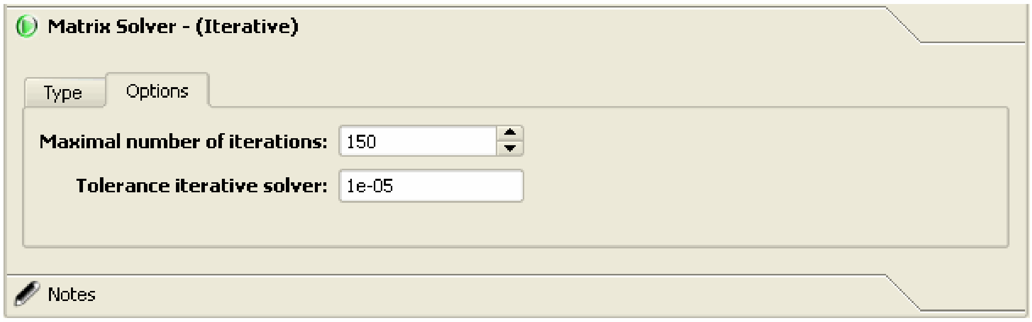
Using the *Create FEM Simulation* dialog box, you can specify the solver type settings. Click **Matrix Solver- (Direct)** to display the **Matrix Solver- (Direct)** screen, as shown in the following figure:



You can select the following solver types to be used by FEM during a simulation.

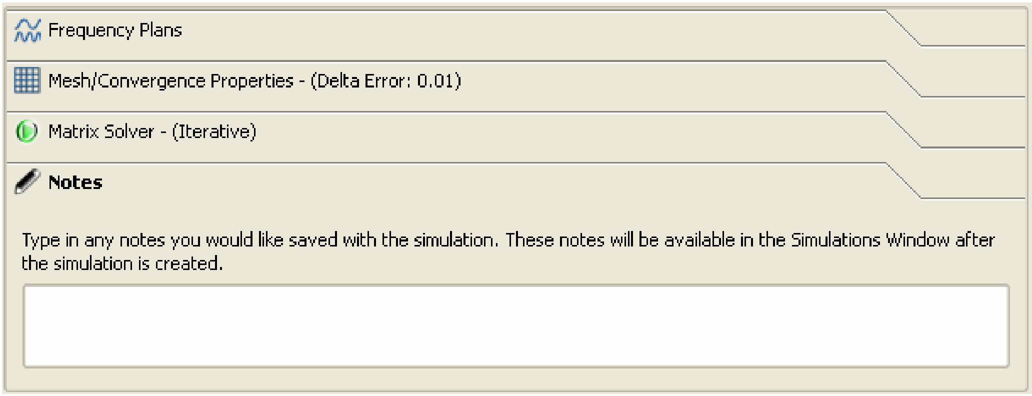
- **Direct**: When selecting **Direct**, the FEM Simulator will use a multithreaded sparse direct solver. The memory requirements and computing time of this solver typically scale quadratically to cubically respectively, as a function of the matrix size. This solver does not suffer from potential convergence problems, and is guaranteed to yield a solution if sufficient memory is available.

- **Iterative**: When selecting **Iterative**, FEM Simulator will use the iterative matrix solver. The memory requirements and computing time of this solver scale linearly to quadratically as a function of the matrix size. In other words, the computing resources of the iterative solver are typically one order of magnitude lower then the computing resources of the direct solver. Especially, the iterative solver requires significantly less memory then the direct solver. However, contrary to the direct solver, the iterative solver is not guaranteed to converge.
  Contrary to the direct solver, the accuracy of the iterative solver can be chosen, by means of the **Tolerance** option under **Iterative solver > Options**. If this option is decreased, the results will be more accurate. However, the number of iterations will increase, resulting in an increased solve time. The maximum number of iterations can be set by the **Maximal number of iterations option**.



Using the Notes Section

If you want to add any notes or observation with your simulation, you can specify it in the Notes text box. Click **Notes** in the *Create FEM Simulation* dialog box to display the **Notes** screen, as shown in the following figure:



After you have completed entering your FEM Simulation options, click the **Done** button to apply the current settings in the *Create FEM Simulation* dialog box, or click **Create Simulation Only** to accept the settings. You can also click **Create and Queue Simulation** to create and queue the simulation. Alternatively, you can click the **Cancel** button to abort the changes and dismiss the dialog box.

Viewing FEM Simulation Results

After running the calculation, you can view the results from the *Results* workspace window. Some results are displayed in the form of numerical values, while other results are displayed in the form of plots. There are several types of plots available to view results based on whether they are time-dependent, frequency-dependent, or angle-dependent:

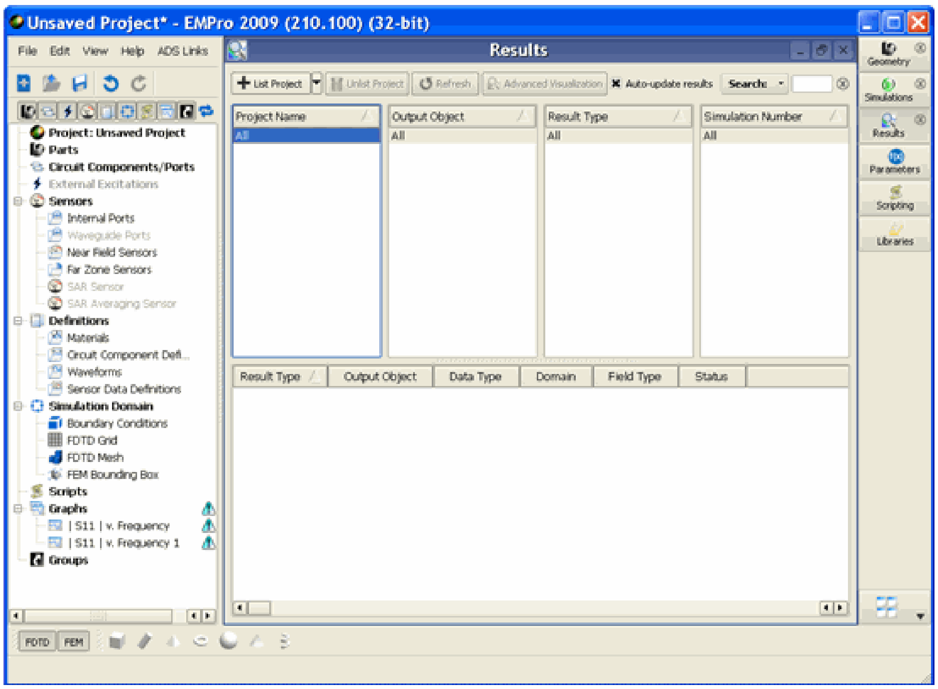
- S-parameter
- Sensors
- Advanced Visualization

In this section, you will learn about how to display FEM Simulator results.

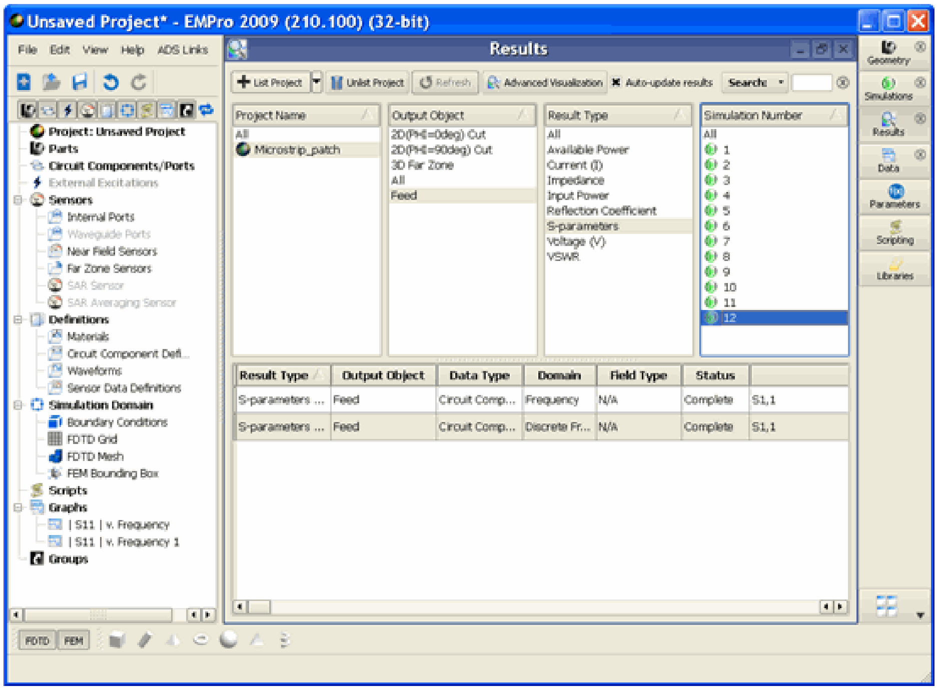
Viewing the Default Output of FEM Simulation

To view the default results of a specific project:

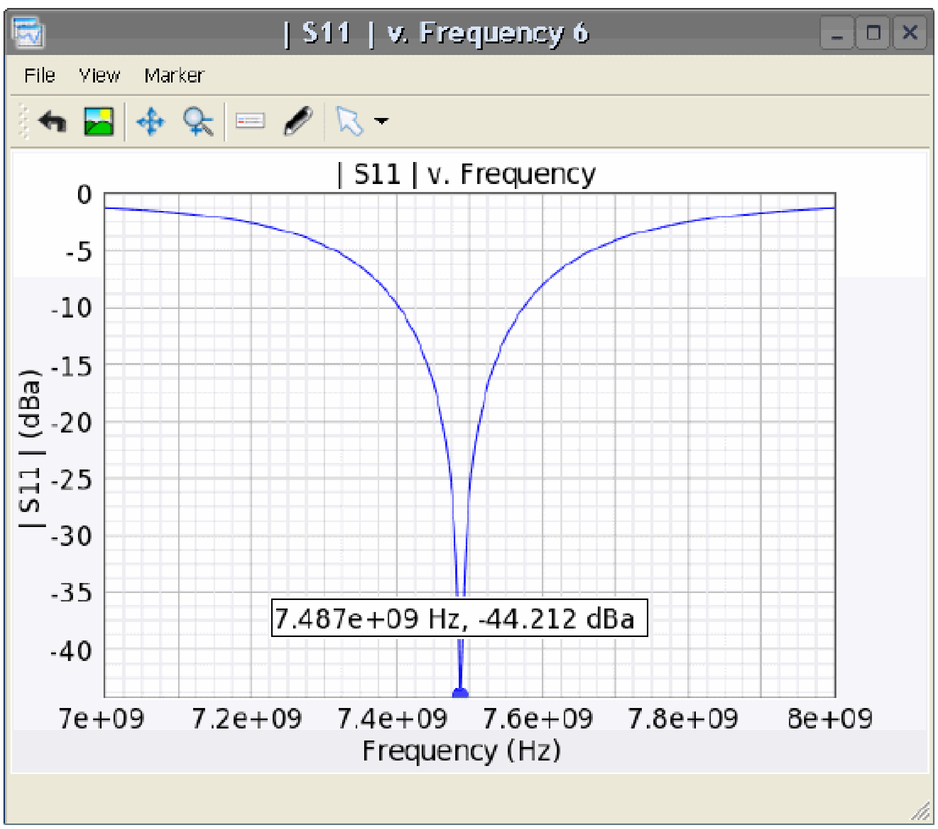
1. Open the *Results* workspace window by clicking **Results** in EMPro.



2. Click **List Project** to open the required project. This opens the *EMPro- List Project Results* dialog box.
3. Select the required project and click **Choose**.
4. In the Results window, you can select the required **Output Object**, **Result Type**, and **Simulation Number**.



5. Right-click the required result type in the *Results* pane, which is present at the bottom.
6. Select **View (default)** to display the default output, as shown in the following figure:

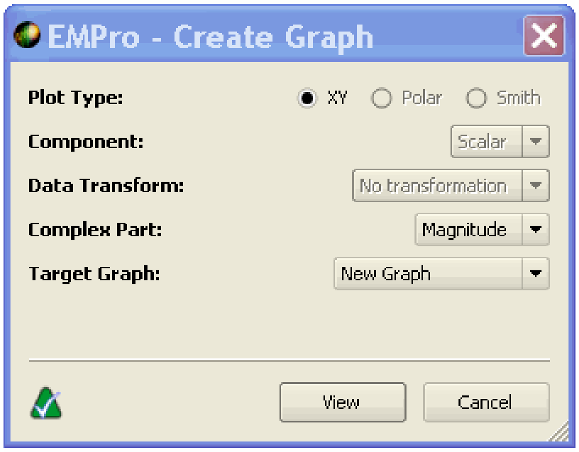


Creating a Line Graph

1. Right-click the required result type in the *Results* pane, which is present at the

bottom.

2. Select the **Create Line Graph** option. This opens the *Create Line Graph* dialog box:



3. Select the plot type by selecting the required option: **XY**, **Polar**, or **Smith**.
4. Select the required component from the **Component** drop-down list.
5. Specify **Data Transformation** option from the drop-down list.
6. From the **Complex Part** drop-down list select the required option for displaying results: **Magnitude**, **Real**, **Phase**, and **Imaginary**.
7. In the **Target Graph**, select **New graph**.
8. Click **View** to view the data.

Viewing S-parameters

Standard and AFS Datasets

If the *Adaptive* sweep type is used for a simulation, both the adaptive and discrete S-parameter results are available in the Results for plotting. The adaptive results contain the S-parameters from the rational fitting model resamples with a very dense frequency distribution. The discrete frequency S-parameter results are indicated with 'Discrete Frequencies' in the Domain column. The adaptive S-parameter results are indicated with 'Frequencies' in the Domain column.

Typically, when viewing data adaptive S-parameter results, you should see very smooth results, which reflect the behavior of the circuit. Viewing the discrete S-parameter results on the same plot will show the frequency points for which the simulator was invoked, superimposed on the smooth curve. Those frequency points will be scattered over the whole range, with relatively more points grouped in areas where the S-parameters show more variation. Viewing both datasets can help you determine the quality of the AFS process on your simulation.

Viewing S-parameters in graphical format

You can view the S-parameter results retrieved with the port sensor placed at the location of the Feed. To filter the list accordingly, select the following options in the columns in the top pane of the Results window (You may need to change your column headings first).

- Output Object: Feed
- Data Type: Circuit Component
- Result Type: S-Parameters

You can view the following formats for displaying S-parameter values for each port:

- Magnitude

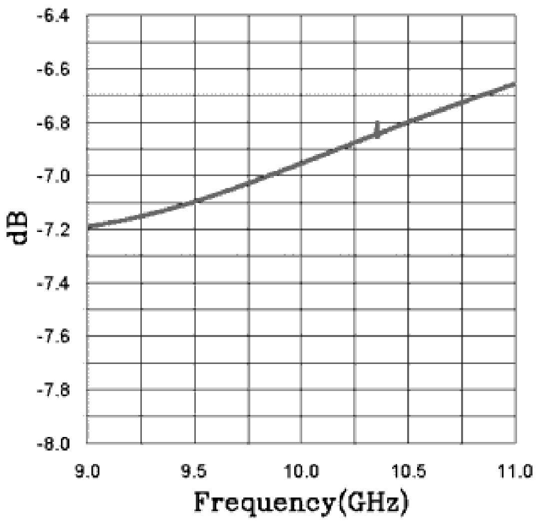
- Real
- Imaginary
- Phase

To view S-parameters:

1. Right-click the result with a Domain value of Frequency to view transient S-parameter results.
2. Select the **Create Line Graph** option.
3. Select the data format from the **Complex Part** drop-down list:
    - **Magnitude** Displays the magnitude of the S-parameters
    - **Real** Displays the S-parameters as complex numbers
    - **Phase** Displays the magnitude and phase of the S-parameters
    - **Imaginary** Displays the S-parameters as complex numbers
4. Click **View** to display the result.

Plotting S-parameter Magnitude

The S-parameter magnitude plot displays magnitude in dB with respect to frequency.



To display an S-parameter magnitude plot:

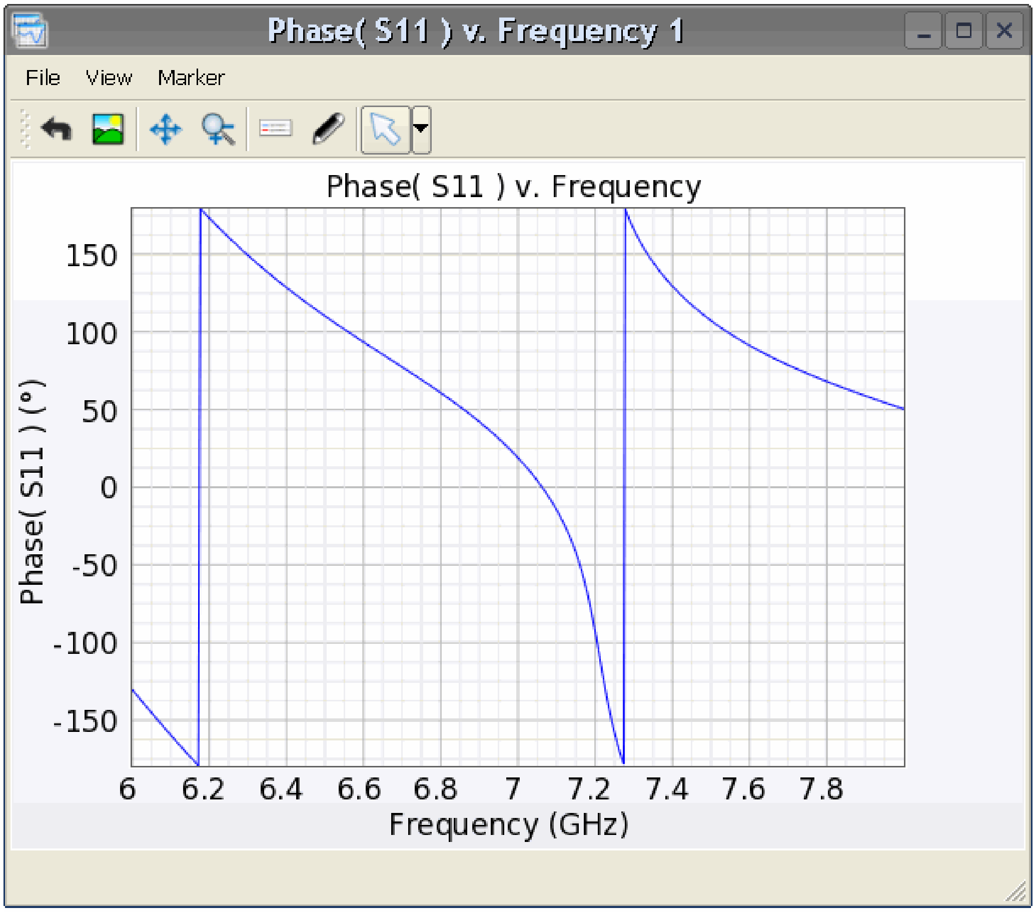
1. Select the data you want to view from the S Parameters list.
2. Open the **Create Line Graph** dialog box.
3. From the **Complex Part** drop-down list, select **Magnitude**.
4. Click **View**.

Plotting S-parameter Phase

The S-parameter phase plot displays phase in degrees with respect to frequency. Note that when viewing S-parameter phase, in some cases the phase delay is plotted. For example, a layout of an electrical length with a phase of 60 degrees, S_{21} represents a phase delay, and appears as -60 degrees. For phases greater than 180 degrees, it may appear that the phase predicted from Gamma compared to S_{21} is incorrect, but this is not so if delay is taken into account.

To display an S-parameter phase plot:

1. Select the data you want to view from the S Parameters list.
2. Open the *Create Line Graph* dialog box.
3. From the **Complex Part** drop-down list, select **Phase**.
4. Click **View**.

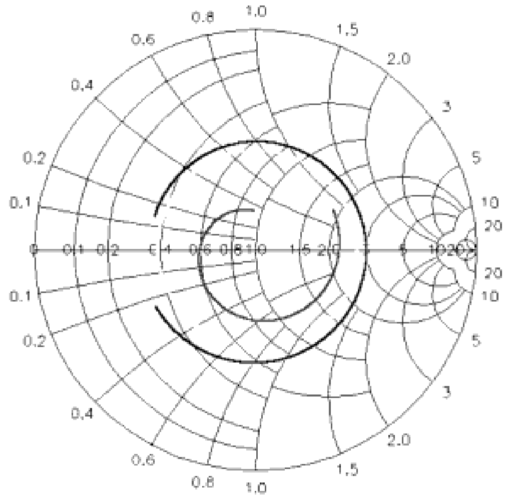


Plotting S-parameters on a Smith Chart

The Smith chart displays real and imaginary components of S-parameters.

To display a Smith chart:

1. Right-click the data you want to view from the S Parameters list.
2. Choose the **View Smith Chart** option.
3. The Smith chart is displayed:



Exporting S-parameters

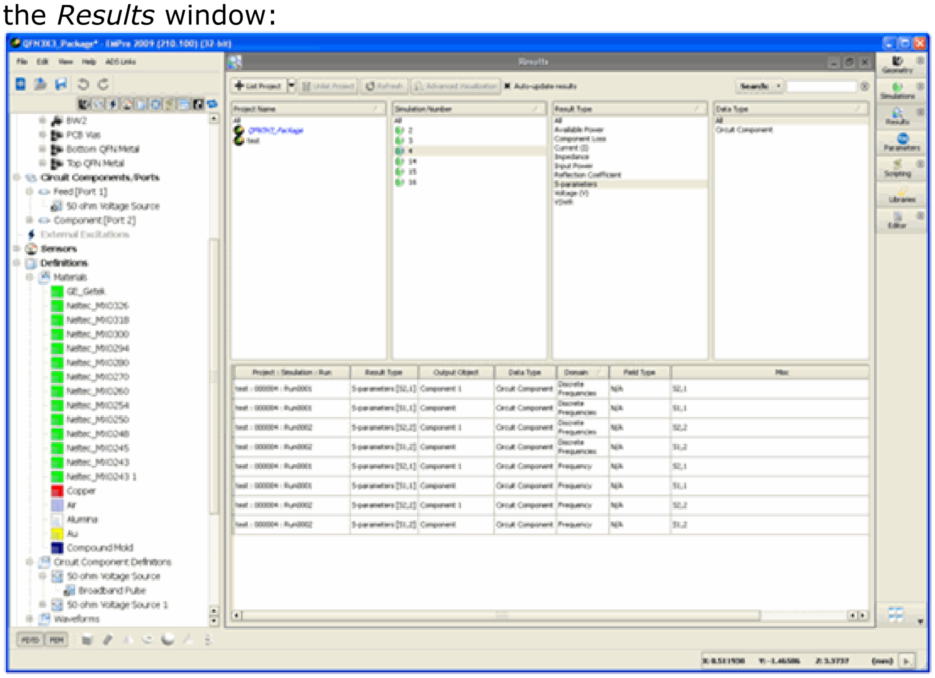
S-parameters can be exported to a Citi file by selecting the S-paramater data in the result browser, and after right-clicking select **Export of CITI file ...**

EMPro FEM-Specific Sensors and Output

From within the results browser, the following sensors can be used for FEM simulations:

- Planar sensors
- Far field sensors

More post processing is possible by using the **Advanced Visualization** tool available in the *Results* window:



Using the Advanced Visualization Feature

The **Advanced Visualization** feature enables you to view and analyze the following types of simulation data:

- E Fields
- Far-fields
- Antenna parameters

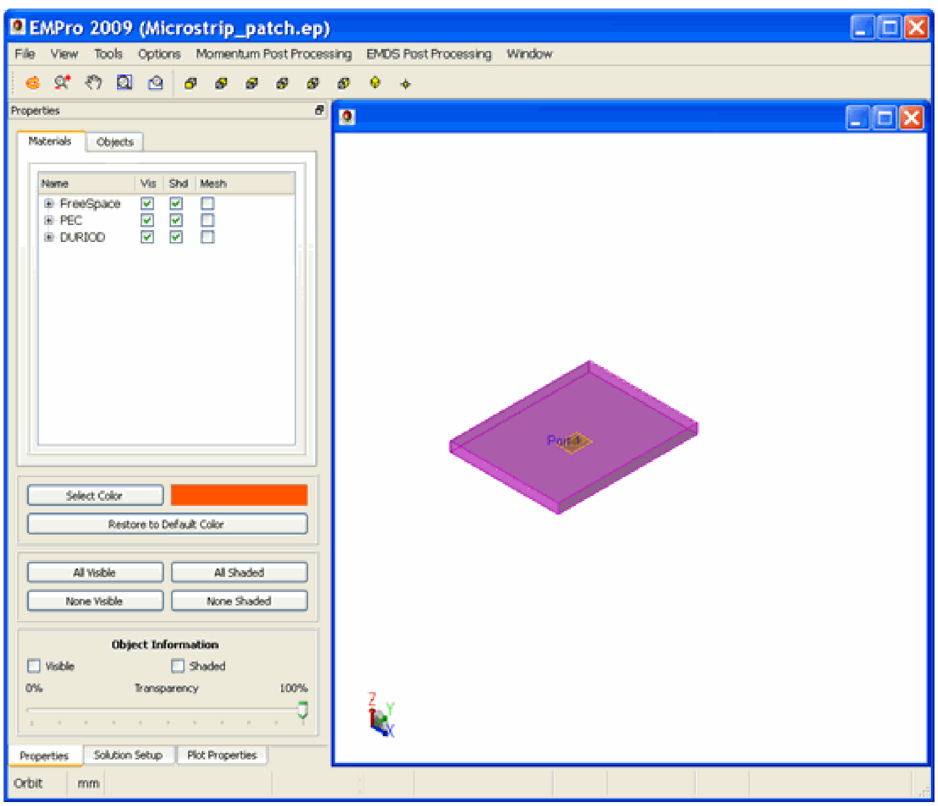
Data can be analyzed in a variety of 3D plot formats. This section gives an overview on how to use the Advanced Visualization feature. Viewing FEM Simulation Results describes in detail how to use FEM Simulator Visualization for viewing specific types of data, such as S-parameters or currents.

Starting the Advanced Visualizer

You can invoke the Advanced Visualization feature by clicking the **Advanced Visualization** button in the **Results** window. The current project must run through a complete simulation use Visualization to view data. If simulation has been previously completed for a project, you can start the visualizer directly to view the existing data.

Using the Advanced Visualizer on PC and Linux Machines

On the PC and Linux platforms, FEM Simulator Visualization is based on the Advanced Visualizer. The following figure highlights the basic window which comes up when Advanced Visualizer is invoked.



Upon startup, the following window appears having:

- All the object shading removed.
- Two additional tabs for controlling the view are added to the window.

The left side of the Advanced Visualizer contains the basic controls for the view in the docking widget.

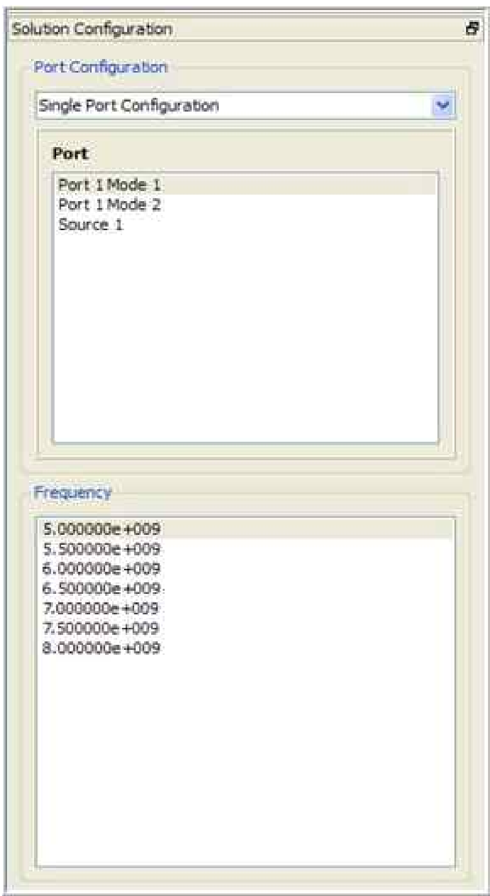
Types of Plots

The Advanced Visualization supports three basic types of plots:

- **Shaded Plot -** This plot displays the E and H fields on the selected planar surfaces.
- **Arrow Plot** - This plot displays a vector representation of the E and H fields on the selected planes.
- **Mesh Plot** - This plot displays the FEM Simulator Mesh which is used to compute the field.

Only one shaded plot and one arrow plot can be displayed. The arrow and shaded plot will always correspond to the selected excitation. Once an excitation is changed, the plot will automatically update using the new excitation values at the current phase.

Solution Configuration Tab



The solution configuration tab is used to select the current excitation for the plots. All of the plots will automatically reflect the current solution configuration once it has been selected. By simply selecting either a port or frequency, the excitation is changed and the plots are automatically updated.

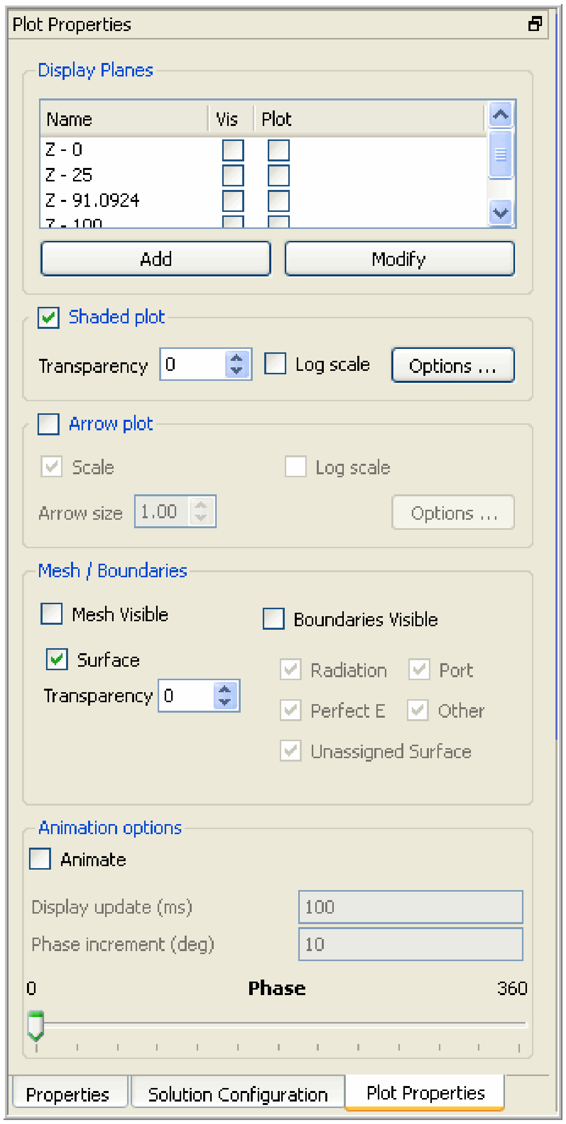
The excitations in the visualization are modal. This means that the solutions are represented using ports and modes. Excitations are based on these modal solutions.

Maximum Value

All of the plot properties for a given plot are retained when the excitation is changed with the exception of the maximum value. This value is reset when the plot is regenerated.

The Plot Properties Tab

The Plot Properties tab enables you to control the three basic plots, as well as, the animation settings.



Plane Entry

FEM Simulator Visualization automatically creates z planes at all the surfaces where there is horizontal xy plane at a z coordinate. These are displayed in the plane list. Each plane has a separate button, allowing you to control the individual display on the plane.

- **Vis** - Display the two dimensional triangular mesh which is the intersection of the 3D mesh with the 2D plane.
- **Plot** - Plots the selected plots within the plane. The plots are determined by the arrow and shade check boxes.

Besides the default planes, you can enter a new plane using the **Add** button. The **Add** button lets you select three points which define the plane. Once entered, you can plot on this plane. A plane can be added by selecting a plan directly also.

Planes can also be moved using the **Modify** button. When a plane is moved, it is moved perpendicular to its surface. Anything that is plotted on the place is automatically updated during this move.

Shaded Plot

Displaying the shaded current plot is controlled by using the check box next to the plot name. When it is selected, the plot is visible. Within the plot there are some basic controls:

- **Log Scale** - This controls whether the scaling and color representation uses a logarithmic scale or a linear one.
- **Transparency** - This controls the transparency of the shaded plot.

Arrow Plot

Displaying the arrow plot is controlled by using the check box next to the plot name. When it is selected, the plot is visible. Within the plot there are some basic controls:

- **Scale -** This controls whether the arrows are scaled, based on the relative magnitude of the current density through out the design. When it is selected, the arrows will be scaled, making the lower current density areas have smaller arrows. If it is not selected all of the arrows will be the same size. However, their size can still be changed by changing the **Arrow Size**.
- **Log Scale** - This controls whether the scaling and color representation use a logarithmic scale or a linear one. If scaling is not enabled, only the color weighting is affected.
- **Arrow Size** - This controls the relative size of the arrow on the screen. Keep in mind that if the arrows are not scaled, the default size of the arrow will often appear to be larger than when the arrows are scaled.

Mesh Plot

When this plot is enabled the FEM Simulator mesh which is used to compute the field is displayed. If the surface box is checked, the surface of the 3D mesh is displayed. Otherwise a 3 dimension wire representation of the mesh is shown.

Boundary Plot

When this plot is enabled the FEM Simulator boundary surface meshes are displayed. These meshes are used by the solution process to represent specific boundary conditions, such as port surfaces and radiation boundaries. By individually selecting the various boundary types, you can see what surfaces were used for each type of boundary condition.

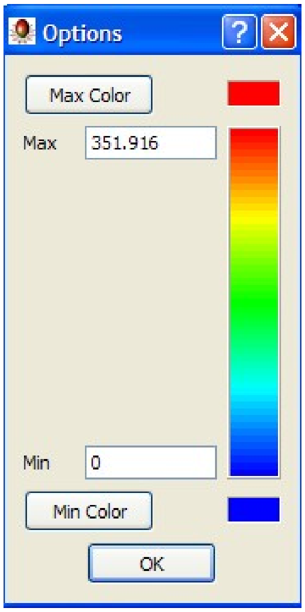
Unassigned surfaces can also be seen. These surfaces are those which could be assigned, but were not assigned during the solution process. If you are not getting the expected solution, this view is an excellent way to determine if the problem is set up correctly.

Advanced Options

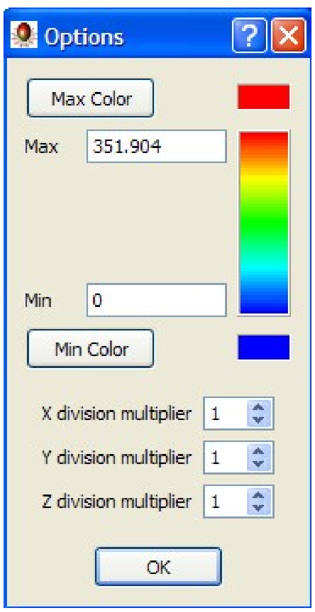
The advanced options dialogs, selectable from the *Shaded* and *Arrow* plot areas enable you to control the color ranges for the plots and the maximum and minimum values for the respective plots.

Shaded Advanced Options

Shaded Advanced Options



Arrow Advanced Options



The arrow advanced options also enables you to change the sampling for the arrow plot when you change from the default multiplier of 1.

Arrow Sampling
When the arrow sampling is changed, it may take a few seconds for the system to update the display.

Animation

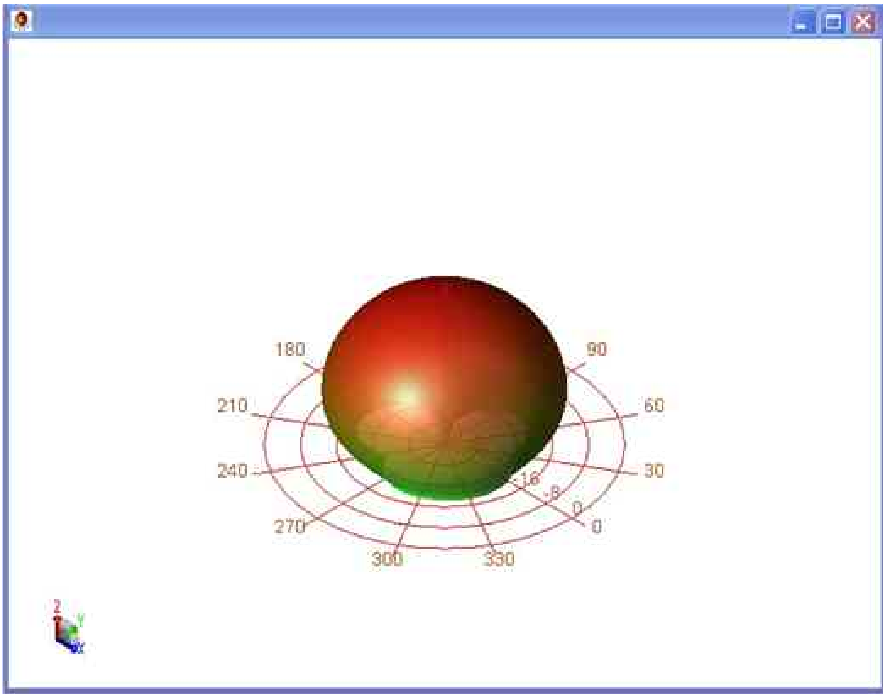
Animation of arrow and shaded plots is controlled by the animation section. There are two controls within this area.

- **Display Update -** This option determines the minimal time between display updates in milliseconds. Since some updates may take longer than this setting, it is a minimum number and not an absolute one.

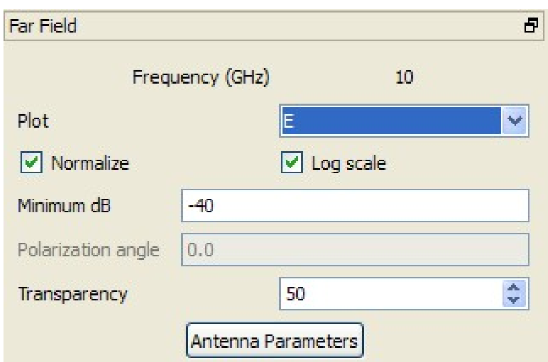
- **Phase Increment**- This controls the number of degrees added to the current phase when an update occurs.

Far Field Plots

When a problem contains far field data, a second window will be enabled within FEM Visualization. This window is used to display the far field plots in 3D. By default, the window is initialized displaying the **E** field as shown below:



A new docking widget is also added to this widget, which controls the far field options.



These options are:

- **Plot** - Controls the type of plot which will be displayed. Once the plot type is selected, the window is updated with the plot.
- **Normalize** - This control is used to normalize the data.
- **Log Scale** - This control is used to enable the scaling of the data logarithmically.
- **Minimum dB** - This is the minimum dB value which is used when plotting the data using the *Log Scale*.
- **Polarization Angle** - This is the polarization angle in degrees.
- **Transparency** - Controls the transparency of the plot.

The antenna parameters for the far field can be displayed by selecting the **Antenna**

Parameters button.

Setting Up the Advanced Visualization on External X Window Displays

On Linux Systems, it is common practice to use a local machine as a display and do the actual processing on another machine. The display from the original machine is typically mapped back to the local display using the following command:

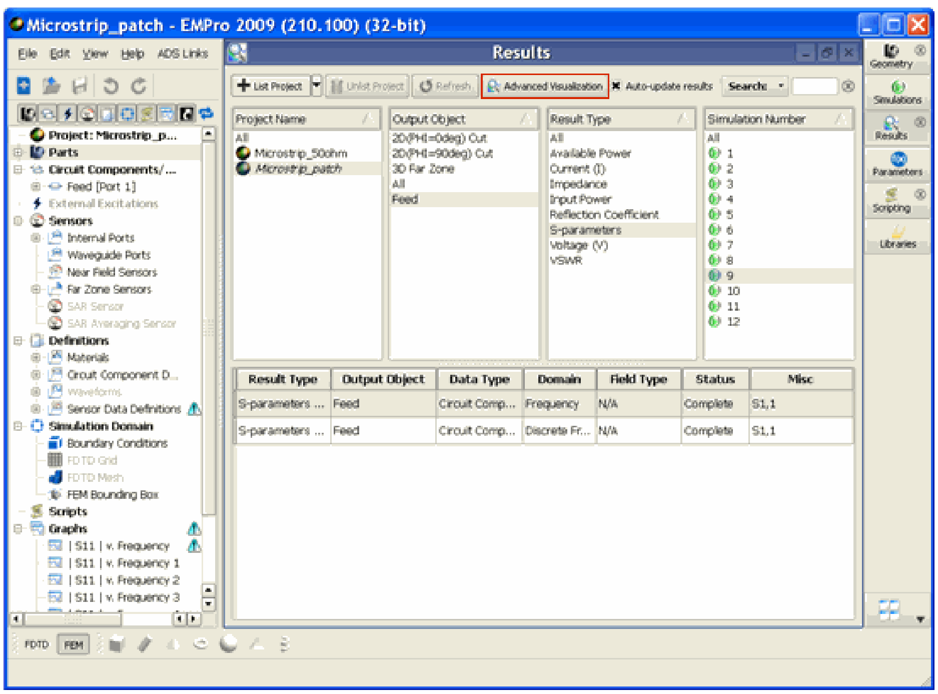
export DISPLAY=machine:0.0

For using the Advanced Visualizer feature, you need to need to set an additional environment variable, **HOOPS_PICTURE.** This environment variable is set to the same value as the display variable with the addition of **X11/** prior to the machine name. Using the above example, the HOOPS_PICTURE variable would be set to

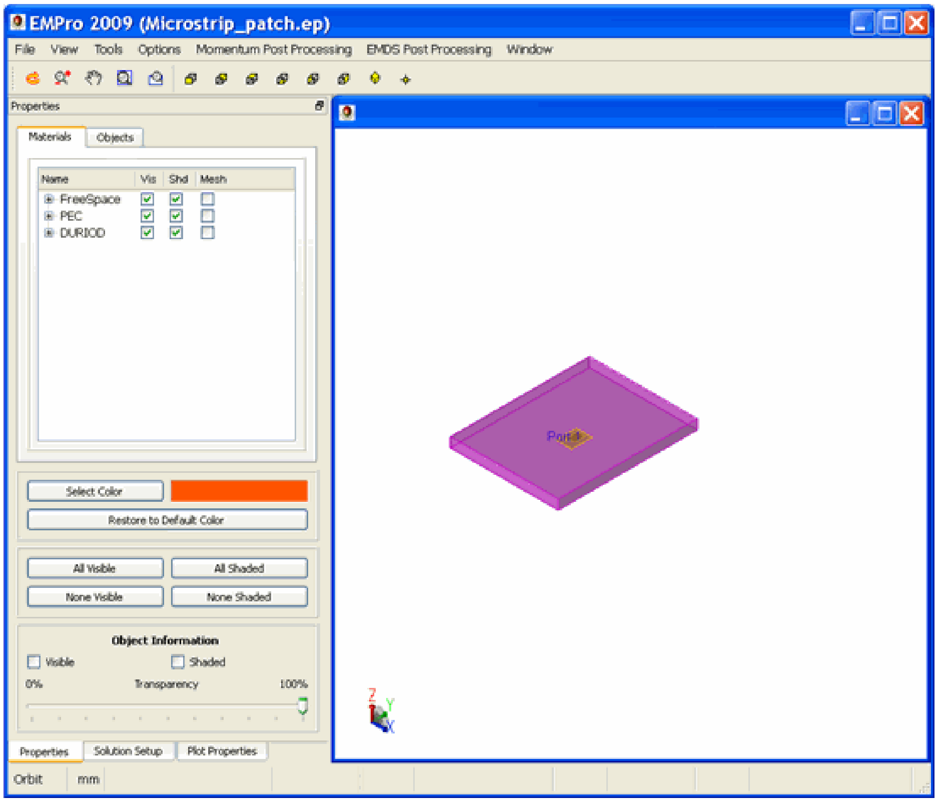
export HOOPS_PICTURE=X11/machine:0.0

Validating Your Geometry Visually

You can invoke the Advanced Visualization feature by clicking the **Advanced Visualization** button in the Results window. You must select a project name before launching the Advanced Visualization tool. The following figure highlights the Advanced Visualization button:



The Advanced Visualizer needs to be started each and every time that you want to review your design as it does not remain synchronized with the current design. It is recommended that you do not keep one instance of the visualizer open while an older instance is present. It will not interfere with the data; however, it might be confusing and cause an unnecessary error.



Identifying and Highlighting Individual Objects

Individual objects can be selected and highlighted in two ways:

- **Picking on the Screen** - Objects can be selected graphically using the mouse. If the visualizer is in Query mode, an object can be selected by clicking on any line or vertex of the object. Once selected, the object's lines are highlighted, the object is selected in the material and object list and the coordinates of the selection point are displayed in the lower right area of the status bar.
- **Selection from the Material or Object List** - Objects can be selected from either the material or object list box. Once selected, the object's lines become highlighted.

Identifying and Highlighting Materials

The material selection is located in the docking widget. It has the same controls for individual objects as the Object Tab. In addition, each material's visibility and shading can be controlled using check boxes associated with the material itself.

Controlling the Visibility and Translucency of Selected Objects and Materials

Once an object has been selected, its visibility and translucency can be controlled using the *Object Information* control located at the bottom of the docking widget. This option enables you to control the translucency and visibility of an object.

It is also possible to control the visibility and shading for the substrate and mask layers. Within the *Material* portion of the docking widget, each material and object has separate toggles for visibility and shading. By setting these controls appropriately, you can control the visibility and shading for all the objects that share this substrate or mask. However, the translucency can only be controlled at the object level.

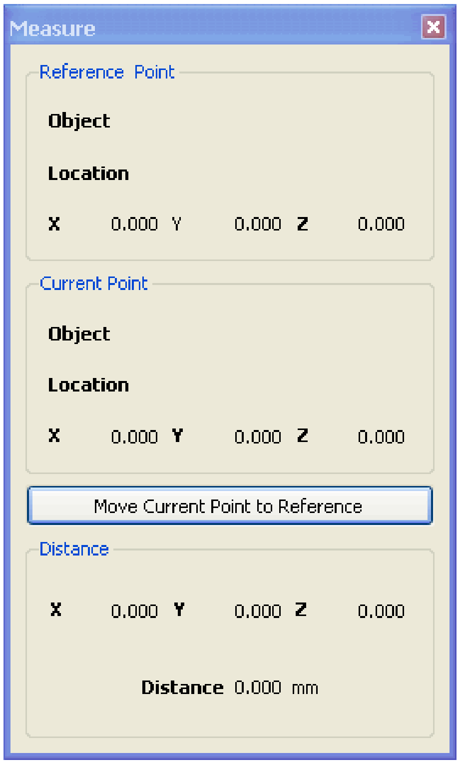
Navigating in the 3D Environment

The Advanced Visualizer operates in several different modes. The current mode is displayed in the lower left portion of the status bar. Regardless of the mode, the mouse wheel will enable you to zoom in and out of the design at the location of the mouse cursor on the screen.

- **Zoom Box** - You can select a region on the design by selecting a rectangle around the area of interest. Once selected, the visualizer will zoom into this area and update the local rotation origin.
- **Zoom** - You can zoom in or out on the design by moving the mouse up or down on the image respectively.
- **Rotate** - The rotation mode is the default start up mode for the visualizer. You can rotate the design around its current origin by holding down the main mouse button and moving it around the screen.
- **Pan** - You can move the design around on the screen by holding down the main mouse button and moving it around the screen.
- **Query** - You can query your model using this command. When you *click* on an object's edge or vertex, the location and object name are displayed in the status bar. In addition, the object is highlighted and the object is automatically selected in the Object and Material tabs in the Docking widget. Visually, a solid dot is placed if the selected location is on a *vertex* and a hollow dot is placed if the selected location is on an edge.

Measuring Distances in the Advanced Visualizer

The measure dialog is activated from the *Window/Measure* command. Once activated, the measure dialog is displayed.



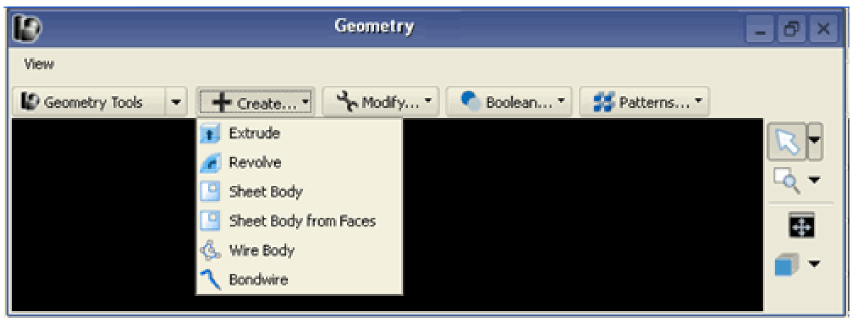
Measurement is done between a *reference point* and the current *query point* . The query point is updated after every mouse selection using the query command. The reference point remains fixed until it is explicitly updated using the *Move Current Point to Reference*.

Creating Bondwire Geometry

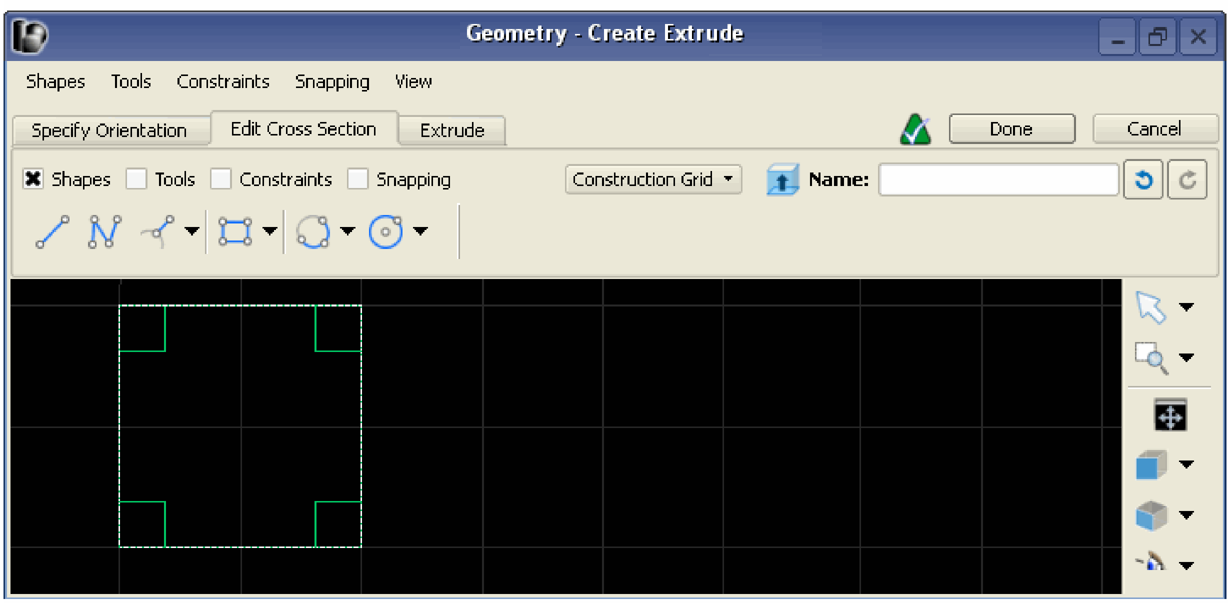
Bondwire allows you to make interconnections between an integrated circuit (IC) and a printed circuit board (PCB) during semiconductor device fabrication.
It can also be used to connect an IC to other electronics or to connect from one PCB to another.

Creating Bondwire Geometry Shapes

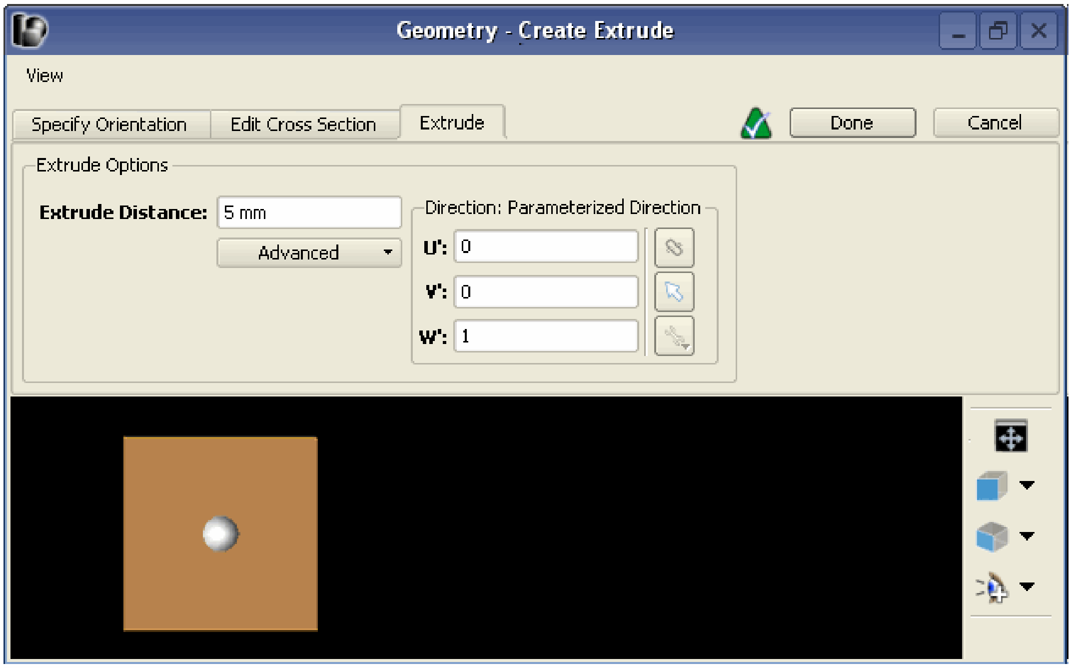
1. In *Geometry* workspace, chose **Extrude** from Create drop-down list. The *Geometry-Create Extrude* window appears.



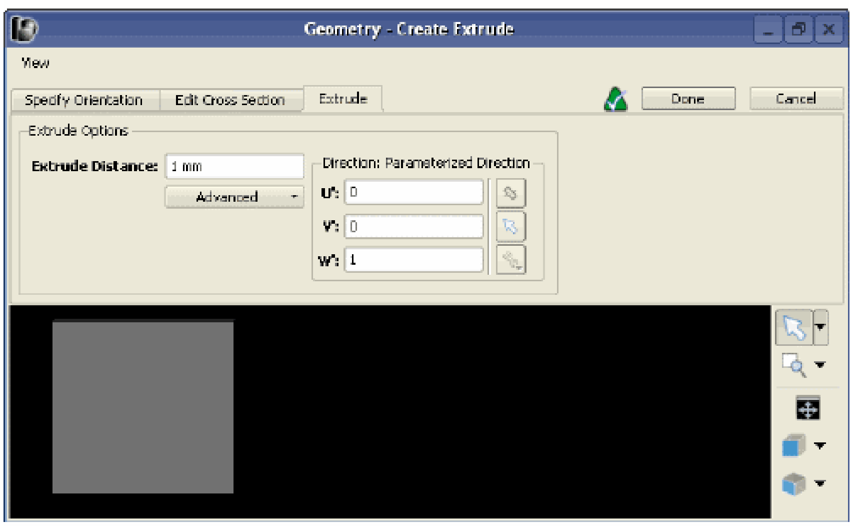
2. In the *Geometry-Create Extrude* window, select **Rectangle** tool  and draw a rectangular shaped object.



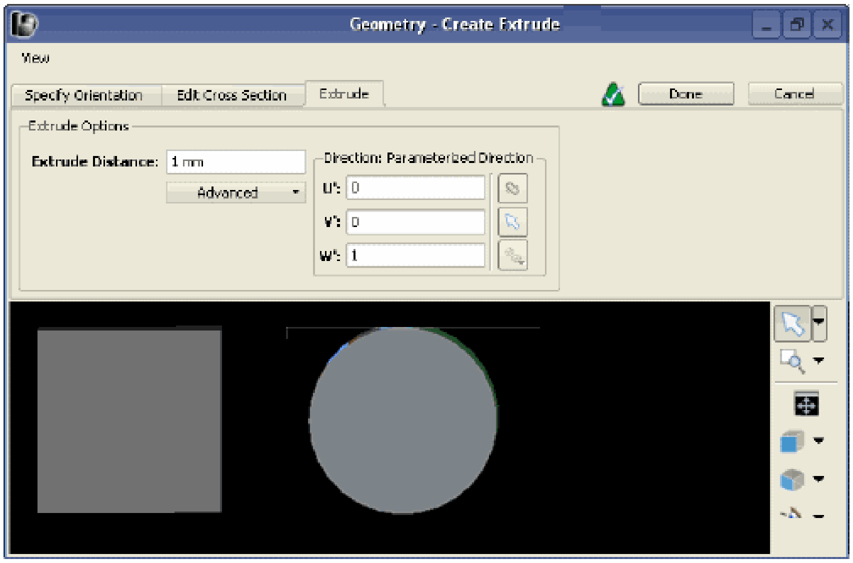
3. Choose **Extrude** tab and modify the parameters through Extrude Option (if required).



4. Click **Done**. The rectangular shape is visible in the *Geometry* window.

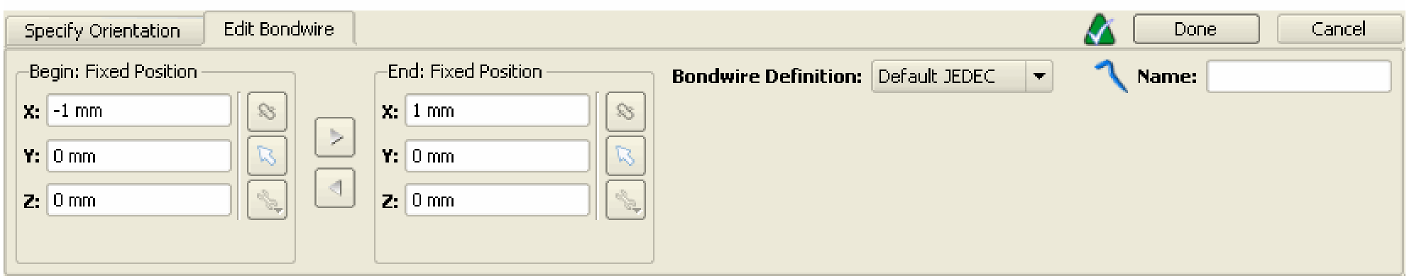


Similarly, create a circular shape as another geometry shape. The final geometry will appear as shown below:

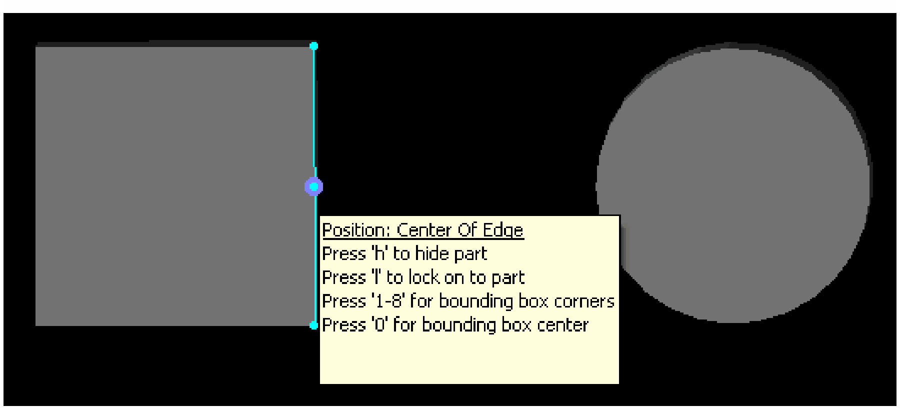


Creating Bondwire

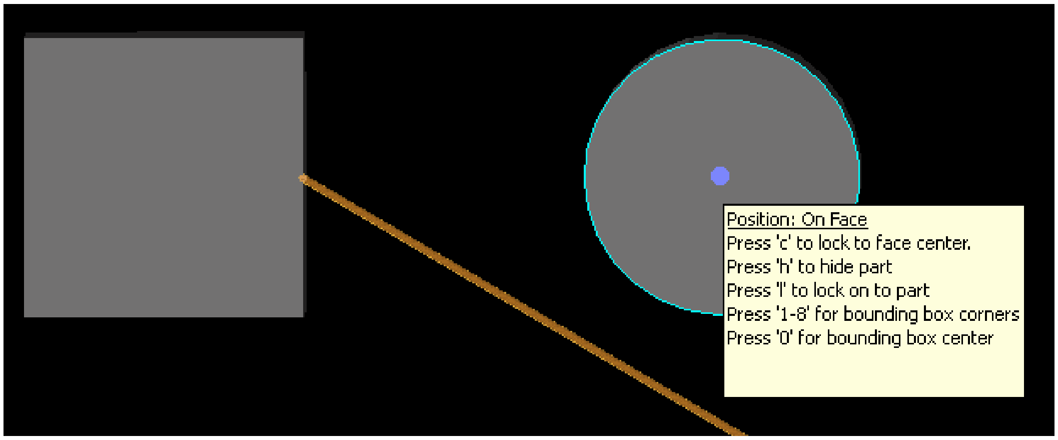
1. To create a Bondwire, select **Bondwire** from Create drop-down list. The *Geometry-Create Bondwire* window appears.



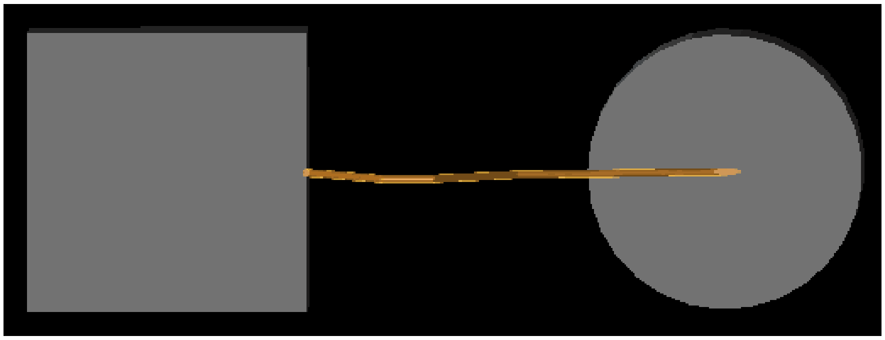
2. In **Begin: Fixed Position** pane, click  icon in **Y** field and select a start position on the first rectangular object to start the Bondwire connection.

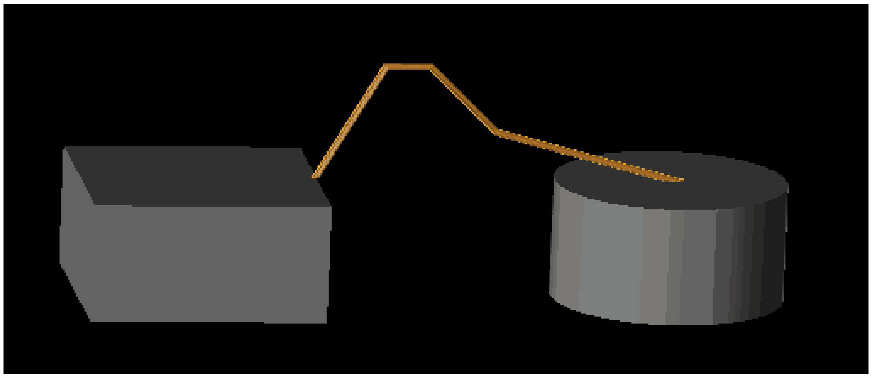


3. Then, in **End: Fixed Position** pane, click  icon in **Y** field and select another position on the second circular object to end the Bondwire connection.

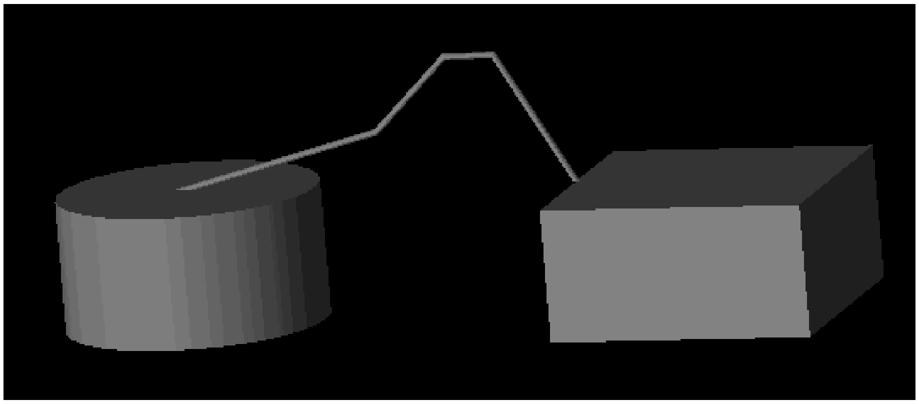


After selecting the start and end positions, the Bondwire will be created as shown in the screenshots below.





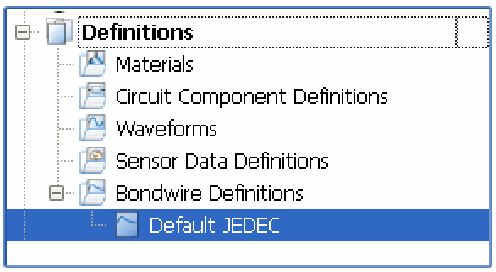
4. Click **Done** to complete the Bondwire creation. The Bondwire is displayed in the Geometry window.



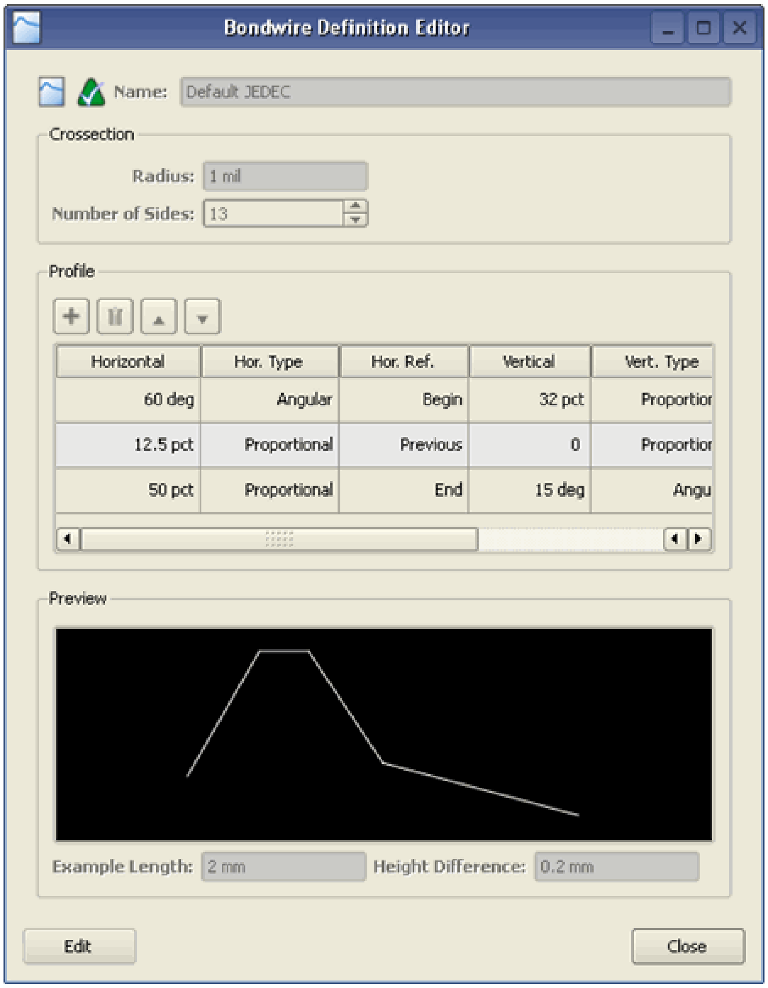
Editing Bondwire Definition

To edit the Bondwire definition:

1. Choose **Definitions > Bondwire Definition > Default JEDEC** in the project tree. JEDEC is a standard for Bondwire profiles. In EMPro, the Bondwire definitions are more general and do not have to implement JEDEC profiles. The default Bondwire definition is a JEDEC profile as an example.

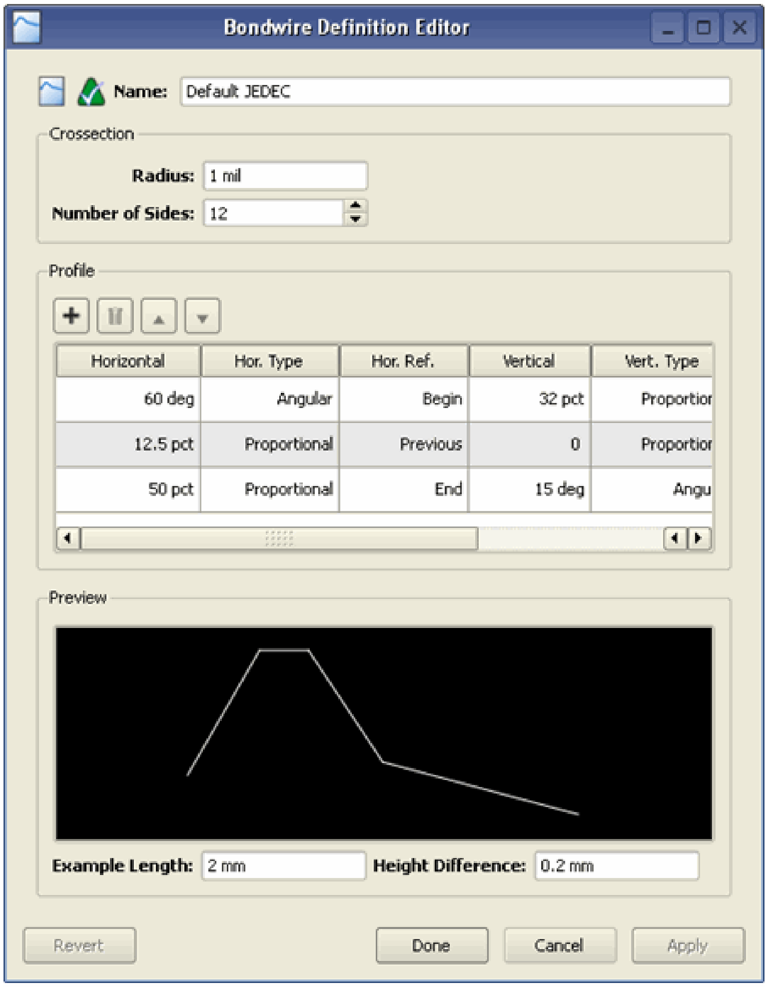


The *Bondwire Definition Editor* window is displayed.



The definitions of Default JEDEC Bondwire may appear in Read only mode.

2. To edit the Bondwire definition, click **Edit** in the *Bondwire Definition Editor* window.



The *Bondwire Definition Editor* consists of three parts: Crossection, profile and preview.

Crossection:

Bondwires have a polygonal cross section. You can choose the radius and the number of sides.

Preview:

Here you can view a preview of the profile. There are two parameters to steer this, the preview length which is the horizontal distance between beginning and the end point. The height difference which is the vertical distance between begin and end point. These values are not properties of the Bondwire definition. These are just two values necessary for a preview.

Profile:

Each profile consists of a number of vertices between a beginning and endpoint. The beginning and endpoints will be choosen when a Bondwire is created so they are not part of the definition.

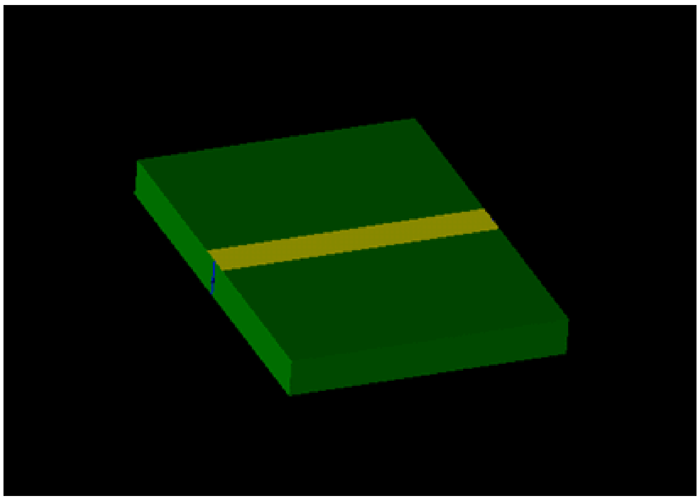
The vertices are defined by horizontal and vertical offsets in the vertical plane passing through beginning and endpoint. Each offset value is accompanied by a type and a reference:

- Reference: Offsets are added to the horizontal or vertical position of the Previous vertex, Begin or End point (horizontal offsets are subtracted from the End point instead). When selecting a vertex in the editor, green markers will highlight the chosen references.
- Type: Each offset is either an absolute Length, Proportional to the horizontal distance between begin- and end point, or specifies an Angular constraint.
- Length: The offset is an absolute length, and will be identical for all bondwires that use this definition.
- Proportional: The offset is proportional to the horizontal distance between begin- and end point, and the exact offset can differ between different bondwires. Use this for definitions that need to stretch depending on the location of the bondwire.
- Angular: For each vertex, at most one of the offsets can be of the Angular type. It will put a constraint on the angle between the horizontal line and the line passing through the vertex and reference point. If both offsets of a single vertex are Angular, the vertex is invalid.

Creating a Microstrip Line for FEM Simulation

In this project, you will learn how to:

- Create a Microstrip Line geometry
- Assign a Waveguide port
- Setup an FEM simulation



Getting Started

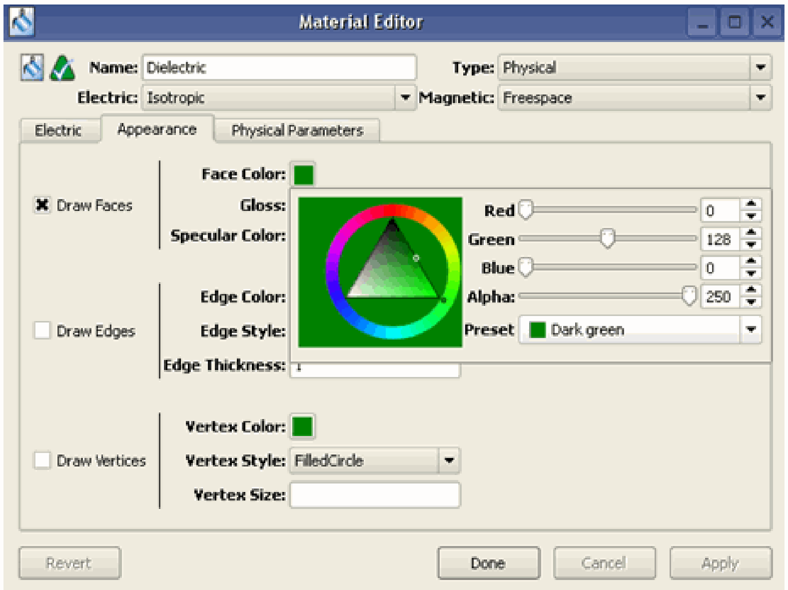
Open a new project and activate the FEM tab, as shown in the following figure:

In the *Project Properties Editor* window (from **Edit** menu), navigate to the **Display Units** tab:

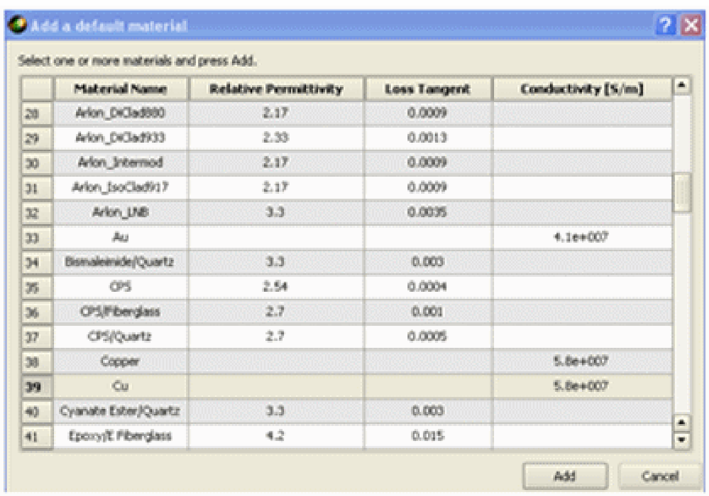
1. Select **SI Metric** in the Unit Set drop-down list.
2. Change **Length** to millimeters (mm). This changes the Unit Set value to **Custom**.
3. Change **Frequency Unit** to **GHz**.
4. Click **Done**.

Creating Materials

1. Right-click **Definitions:Materials** branch of the project tree and select **New Material Definition**.
2. Double-click the new material to edit its properties. Specify the following properties for the electric conductor material:
    - **Name**: Dielectric
    - **Electric**: Isotropic
    - **Magnetic**: Freespace
    - **Relative Permittivity** = 9.9
    - **Conductivity** = 0 S/m
      Depending on your requirements, you can also set the display color of the PEC material in the **Appearance** tab, as shown in the following figure.



3. Right-click **Definitions:Materials** branch of the project tree and select the **Default material** library. Select the material **Cu (copper)** and click the **Add** button. This material will be added in the list.



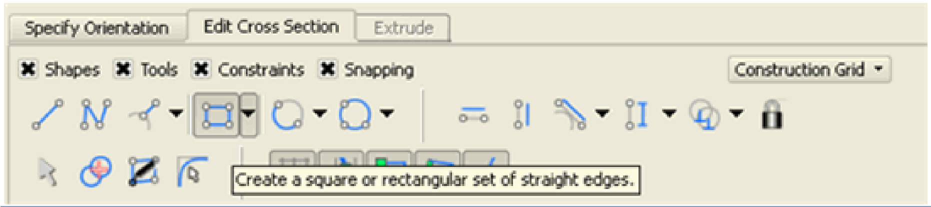
Creating the Microstrip line Geometry

You will create the Microstrip line geometry from 2 simple components: a rectangular substrate and a Microstrip line. For this example, you will use the Geometry Tools interface to create a rectangular Extrusion and rectangular Sheet Bodies.

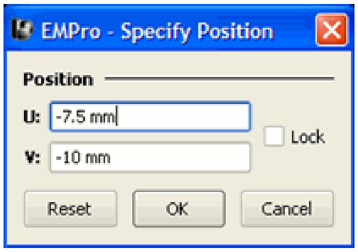
Modeling the Substrate

First, you will create the rectangular substrate named Substrate. This object will stretch from (-7.5, -10, 0) to (7.5, 10, 2) and have a 2mm extrusion in the +Z direction.

1. Right-click the **Parts** branch of the Project Tree. Choose **Create New > Extrude** from the context menu.
2. Name the substrate by typing **Substrate** in the **Name** text box in the upper-right corner of the window.
3. Choose the Rectangle tool from the **Shapes** toolbar.

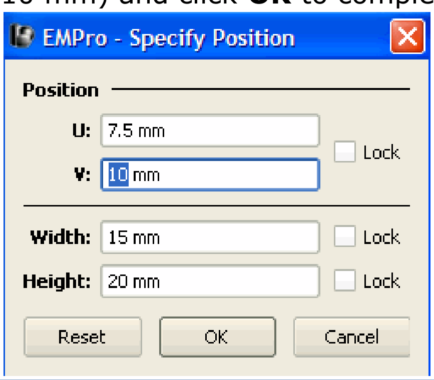


4. The *Specify Position* dialog box allows exact entry of coordinates. Press the Tab key in the geometry space to activate the *Specify Position* dialog box. Specify the position of first point.



> **Note**
> If the Specify Position dialog box does not appear, right-click in the geometry space to activate the window.

5. Press the **Tab** key to display the creation dialog for the second point. Enter (7.5 mm, 10 mm) and click **OK** to complete the rectangle.

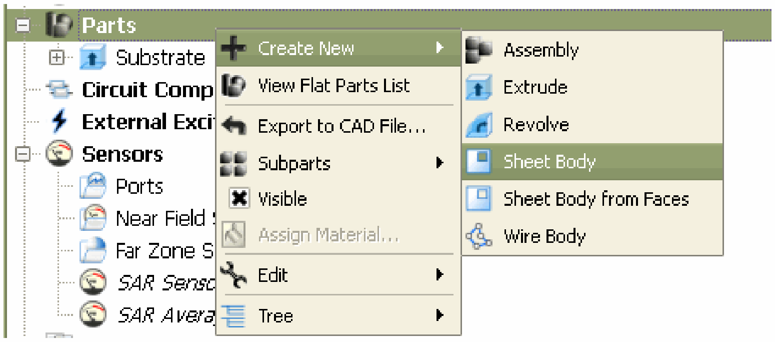


6. Navigate to the **Extrude** tab to extrude the rectangular region. Enter a distance of 2 mm.
7. Click **Done** to finish the Substrate geometry.

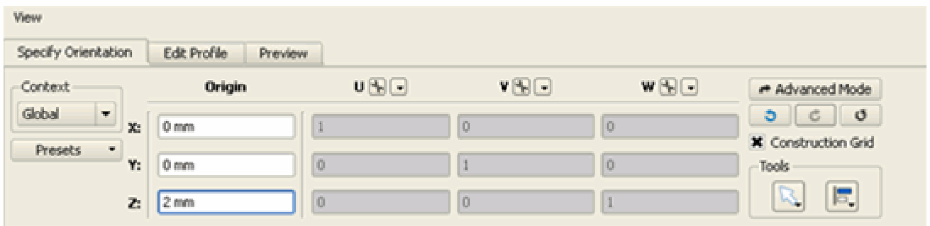
Modeling the Microstrip Line

The Microstrip Line will be created with a Sheet Body object that rests on top of the Substrate. This shape comprises of one rectangle. The line will stretch from (-7.5, -1, 2) to (7.5, 1, 2).

1. Right-click the **Parts** branch of the Project Tree. Choose **Create New > Sheet Body** from the context menu.



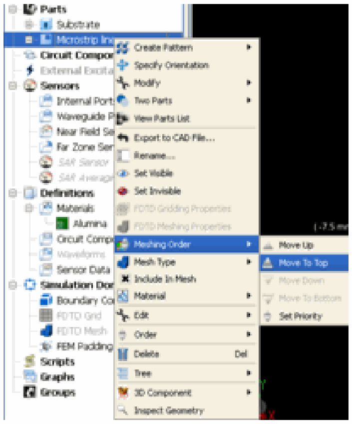
2. Navigate to the **Specify Orientation** tab. Set the origin to (0, 0, 2 mm) to place the Sheet Body on top of the Substrate.



3. Navigate to the **Edit Profile** tab. Type **Microstrip Line** into the Name text box.
4. To draw the Microstrip line, select the **Rectangle** tool. Use the *Specify Position* dialog box to enter the corners of the microstrip rectangle:
   o Endpoint 1: (-7.5 mm, -1 mm)
   o Endpoint 2: (7.5mm, 1 mm)

Meshing Priority

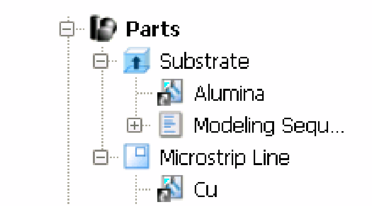
Ensure that the meshing priority of the Microstrip Line is greater than the Substrate for an accurate calculation.
Right-click the **Microstrip** in the Project Tree. Under **Meshing Order**, select **Move to Top** if it is an available option.



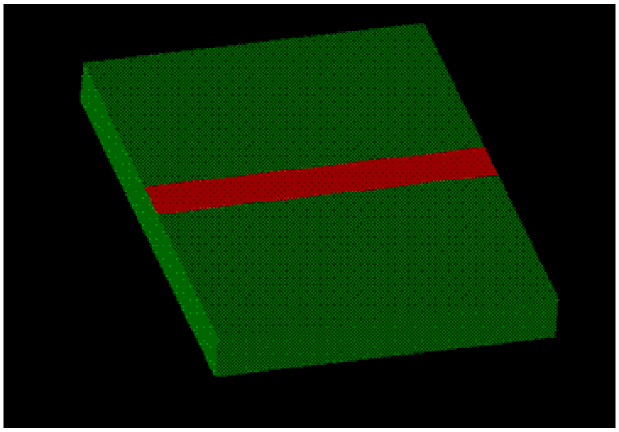
Assigning Materials

1. Click and drag the **CU** material object located in the project tree and drop it on top of the Microstrip Line objects in the **Parts** branch of the tree.
2. Assign the **Alumina** material to the Substrate object using the same procedure.

The following figure displays the project tree after material objects have been dropped on their respective parts.

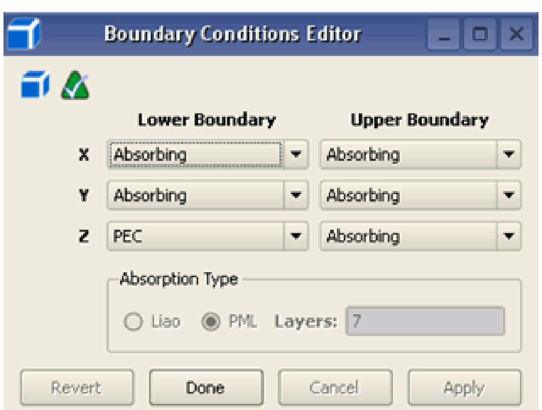


The following image shows the Microstrip Line geometry with materials applied and colors set for each.



Defining the Outer Boundary

1. Double-click the **Simulation Domain> Boundary Conditions** branch of the project tree to open the **Boundary Conditions Editor**.
2. Set the outer boundary properties as follows:
   - Boundary: Select **Absorbing** for all boundaries except Lower Boundary Z, which should be **PEC**.



3. Click **Done** to apply the outer boundary settings.
4. Double-click the **Simulation Domain> FEM Padding** branch of the project tree. Set the **Upper** and **Lower** limits as shown below:

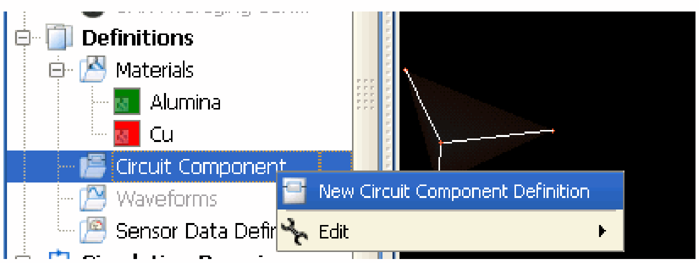
5. Click **Done**.

Adding a Waveguide Feed

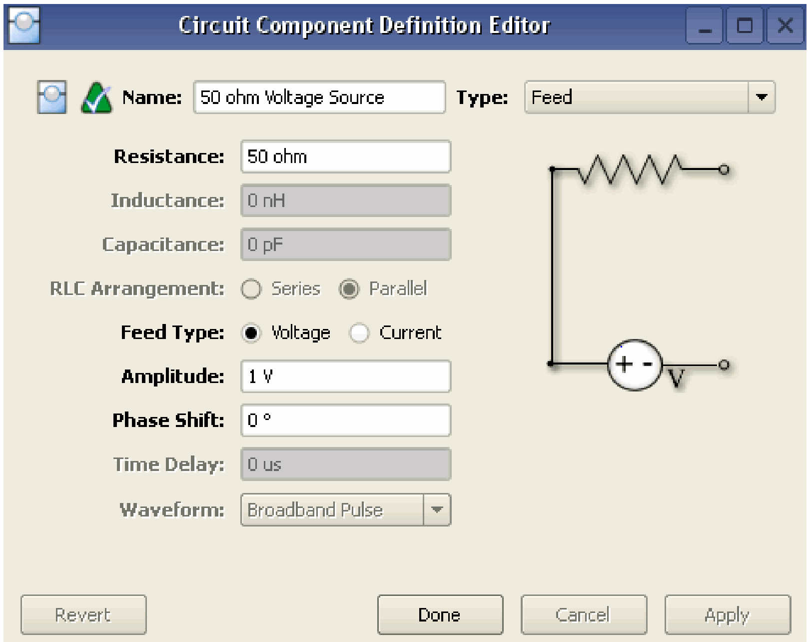
You can now add a Waveguide Feed to the Microstrip Line Geometry. It will consist of a voltage source and series 50Ω resistor connected between Microstrip Line and the ground plane.

> **Note**
> Waveguide Port with Modal power needs to be used for WG or Coaxial structures.
>
> - For TEM -type waveguides (e.g. microstrips), you need to use a **feed** circuit component definition.
> - For TE/TM-type wavegues (e.g. hollow waveguides), you need to use a **modal** circuit component definition.

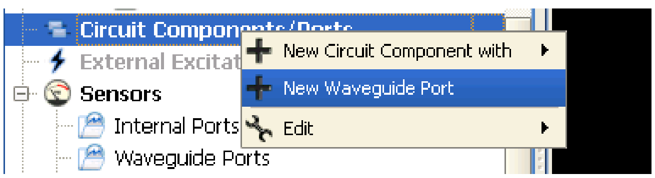
1. Define a new circuit component definition from **Definition** under the project tree , as shown below:



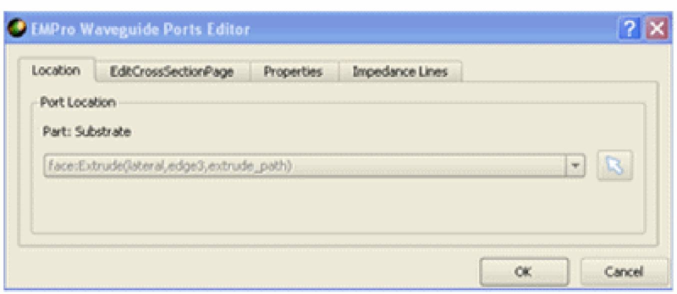
2. The **Circuit Component Definition Editor** dialog box is displayed:



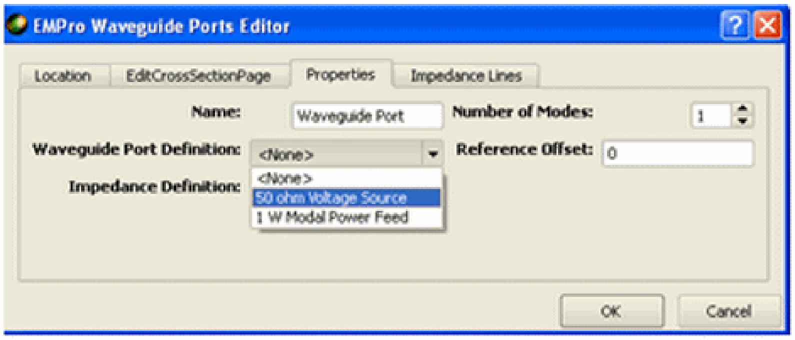
3. Define **Waveguide Ports** from **Project Tree > Circuit Components/Ports** as shown below:



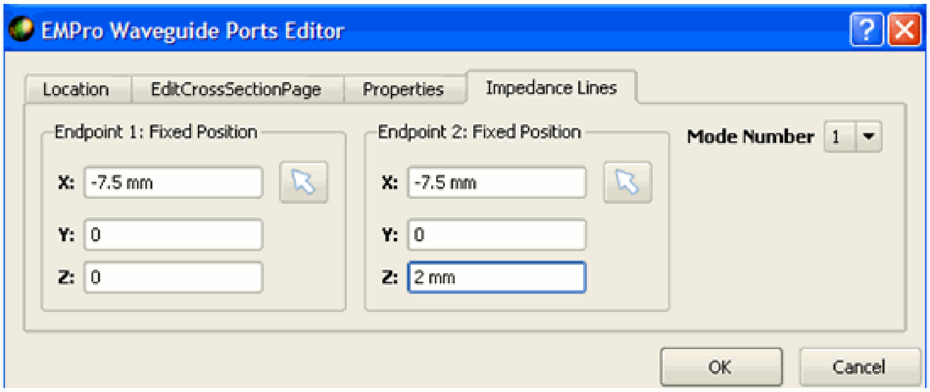
4. From the **Waveguide Port Editor** dialog box in location tab use  to select edge of Substrate coinciding X=-7.5 mm plane as shown below:



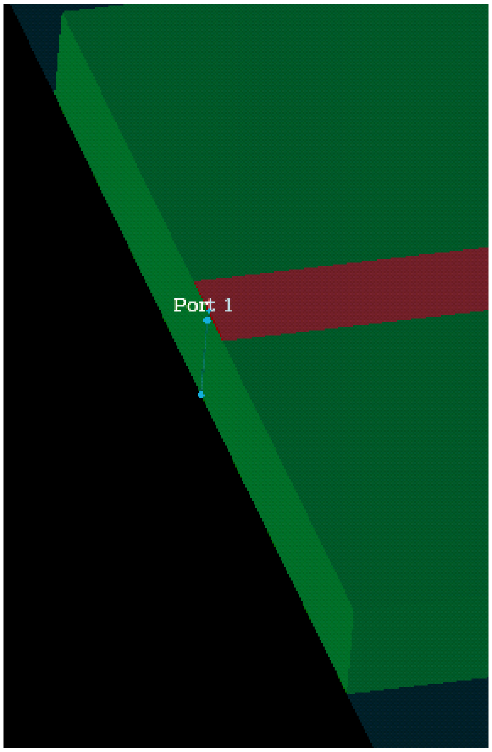
5. Under *Properties* tab chose 50 ohm Voltage source in *Waveguide Port Definition* dialog box as shown below:



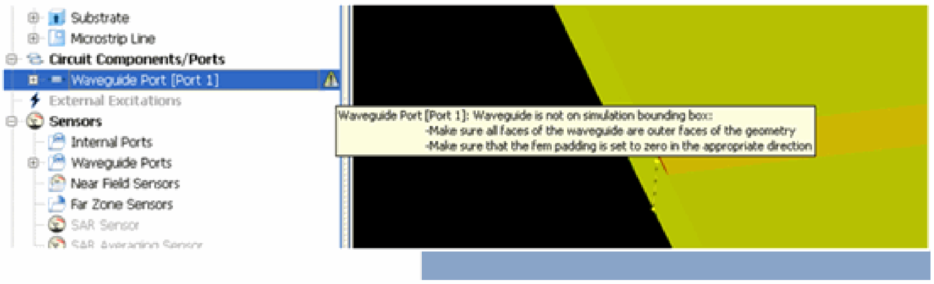
6. Under **Impedance Line** define **Endpoint1** and **Endpoint2** as shown below:



7. Click **OK**.
   The impedance line will appear on the waveguide port as shown below:

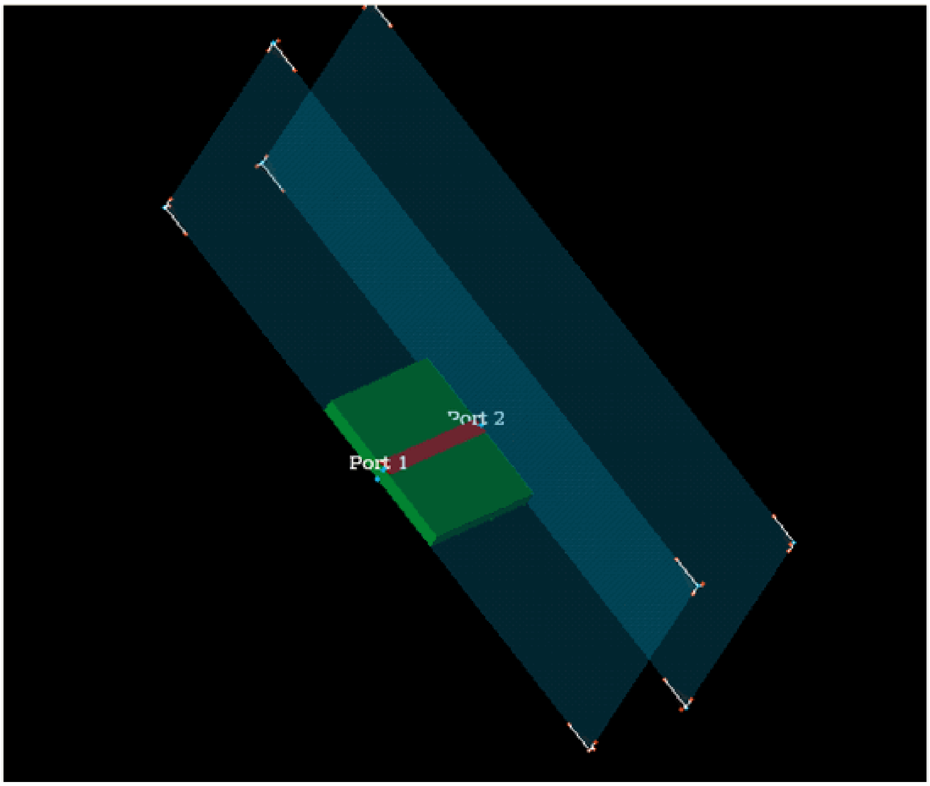


Waveguide port so defined still show invalid symbol:



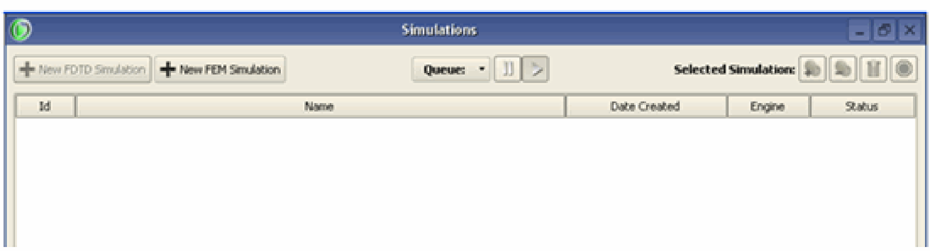
As per the message waveguide port should lie on the faces of the geometry. To accommodate this we need to change padding in x direction. Both lower and upper padding in X direction should be 0mm as waveguide port lie on X plane as shown in figure:

8. Click **Done**. This will make waveguide port valid.
Similarly define another waveguide port at X=7.5 mm plane. Once both waveguide port is defined the waveguide ports on the structure look like as shown in following figure:

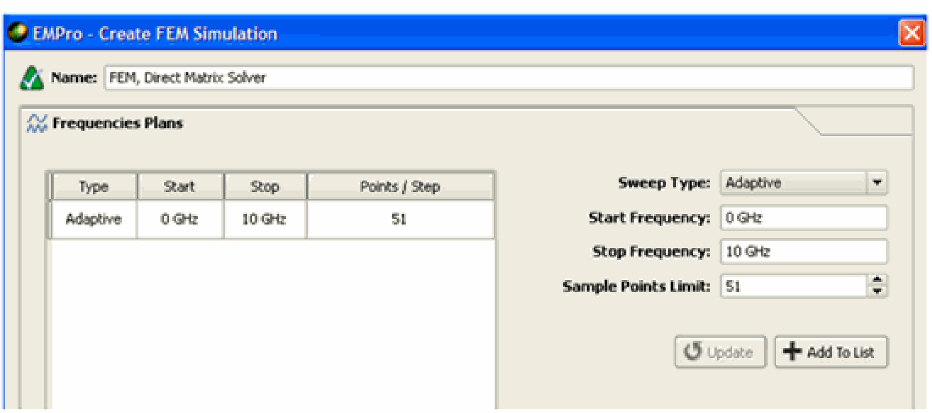


Setting up new FEM simulation

1. Click the **Simulation** tab on right side of GUI. This opens the *Simulation* window, as shown in following figure.

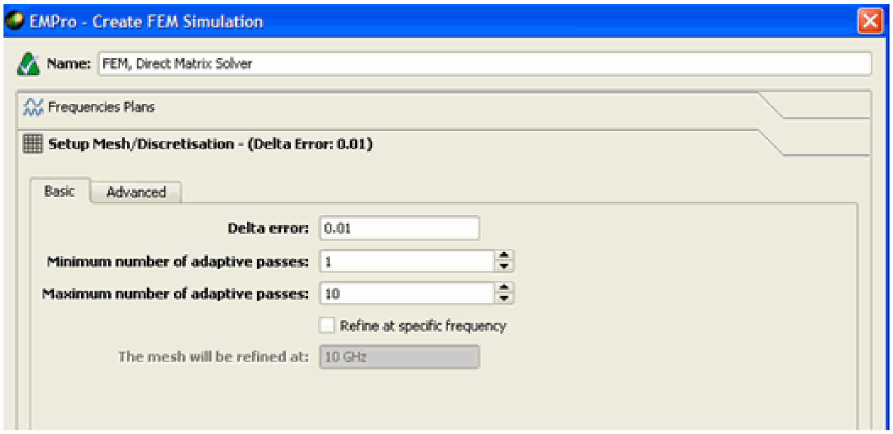


2. Click **FEM Simulations** following window for FEM simulation appears.
3. Add the following **Frequency Plans**:

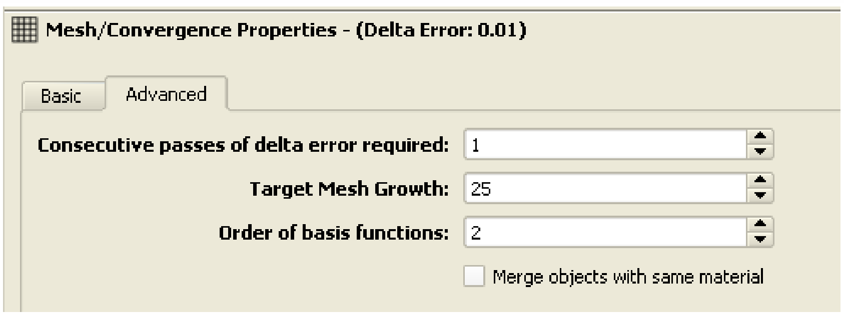


4. Under **Mesh/Convergence Properties**, use following settings in Basic and Advance

tabs:



5. Under **Setup matrix** solver, choose Direct solver:



6. Click **Create and Queue Simulation** on *Create FEM Simulation* window to start FEM simulation.

Simulating a Microstrip Line with Symmetric Plane

In this project, you will learn how to:

- Create a Microstrip Line geometry
- Apply Symmetric plane Boundary condition
- Assign Port
- Set up an FEM simulation

Getting Started

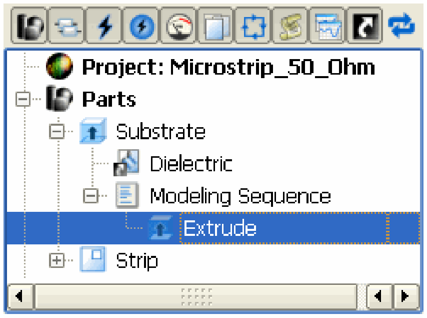
To demonstrate how to set up symmetric plane, we will use Microstrip 50 Ohm line project from example set:

1. Unarchieve the Microstrip 50 Ohm line project from **Help > Example**.
2. Save the example as *Microstrip_50_Ohm_Symm_M.ep*.

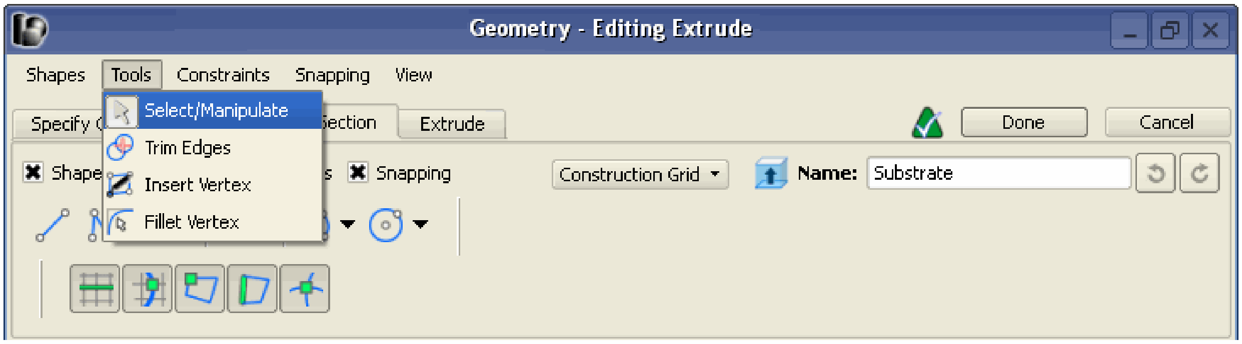
Modify Microstrip Geometry

To modify the Microstrip Geometry:

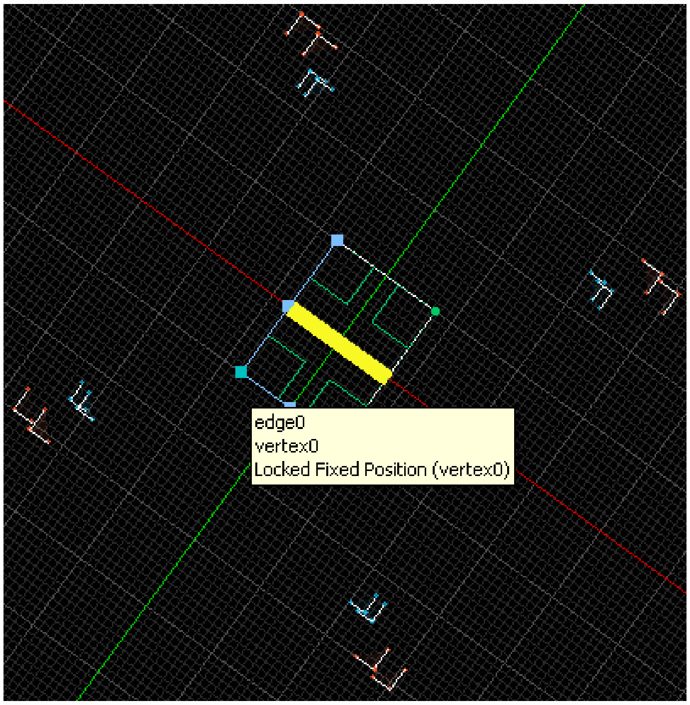
1. Choose and expand **Substrate** under **Parts** list.



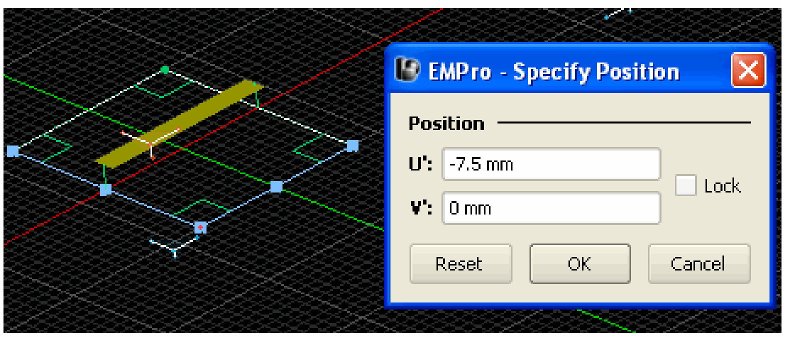
2. Double-click **Extrude** to display the part geometry in the *Geometry-Editing Extrude* window.
3. In Geometry-Editing Extrude window, choose **Tools > Select/Manipulate**.



4. Choose **Vertex 0** of the substrate for editing.



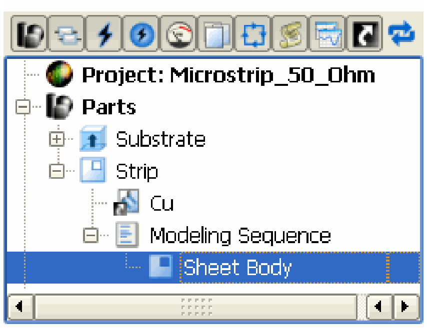
5. Right-click on Vertex 0 and select **Edit Position**.



6. Under *Position* pane, change **V** coordinate to 0 and uncheck the **Lock** check box. This will reduce the width of the Substrate to half.
7. Click **Ok**.
8. Then, click **Done** on the *Geometry–Editing Extrude* window.

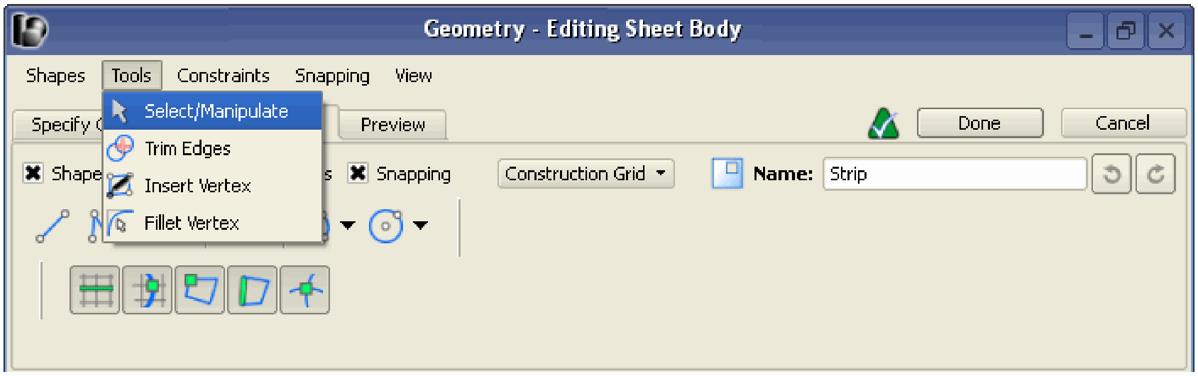
Similarly, to reduce the Strip width to half:

1. Choose and expand **Strip** under **Parts** list.

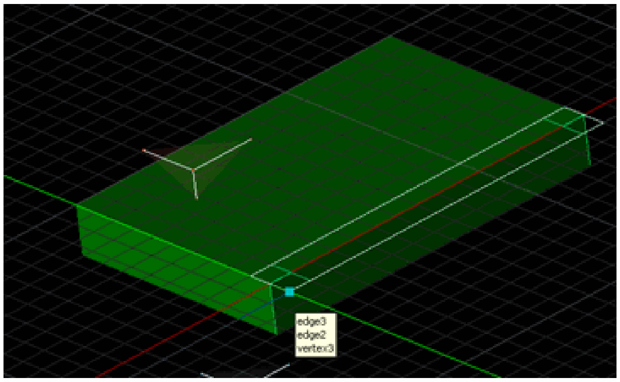


2. Double-click **Strip** to display the part geometry in the *Geometry-Editing Sheet Body* window.

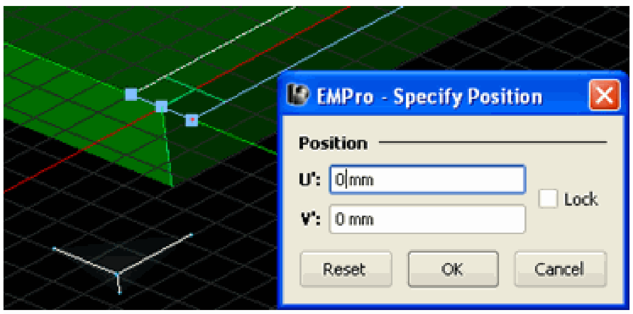
3. In *Geometry-Editing Sheet Body* window, choose **Tools > Select/Manipulate**.



4. Choose **Vertex 2** of the Strip for editing.

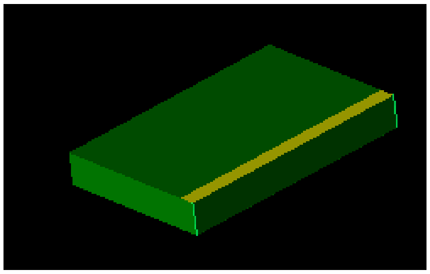


5. Right-click on Vertex 2 and select **Edit Position**.



6. Under *Position* pane, change **V** coordinate to 0 and uncheck the **Lock** check box. This will reduce the width of the Strip to half.
7. Click **Ok**.
8. Then, click **Done** on the *Geometry–Editing Sheet Body* window.

Final geometry is displayed as shown below:



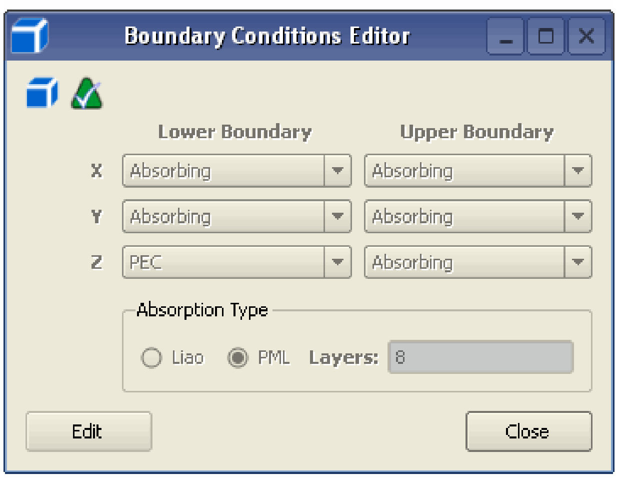
The above figure displays that the Ports are at edges of the Substrate.

Applying Symmetry Boundary Condition

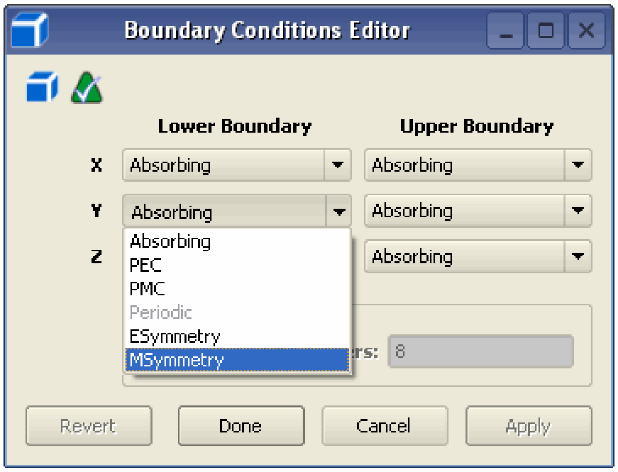
To apply the symmetry boundary condition:

1. Switch off the **FDTD** simulation option. The Symmetry boundary condition is applicable only for FEM simulator.

2. Choose **Simulation Domain > Boundary Conditions** from the project tree. The *Boundary Condition Editor* is displayed.

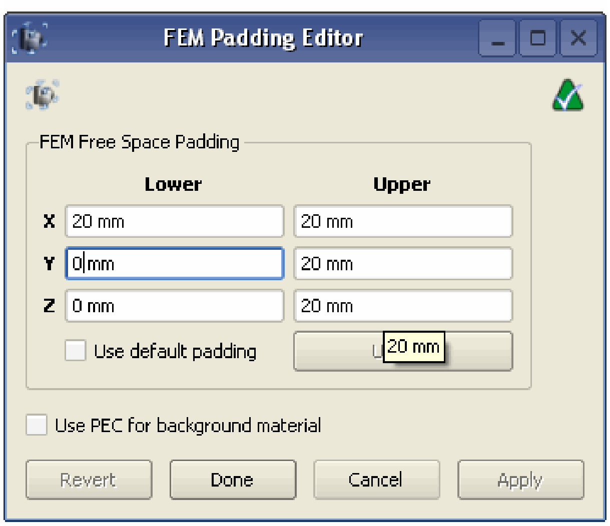


3. Change **Y** parameter of **Lower Boundary** to **MSymmetry** and click **Done**.



4. Now, choose **Simulation Domain > FEM Padding** from the project tree. The *FEM Padding Editor* is displayed.

5. Change **Y** parameter of **Lower** from **0 mm** from **20 mm** and click **Done**.

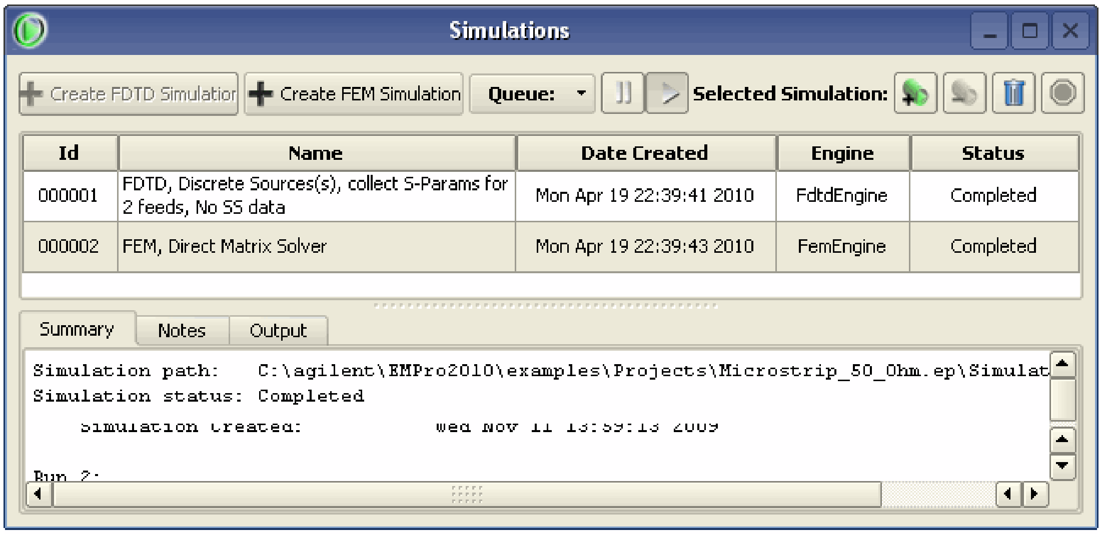


Setting up New FEM Simulation

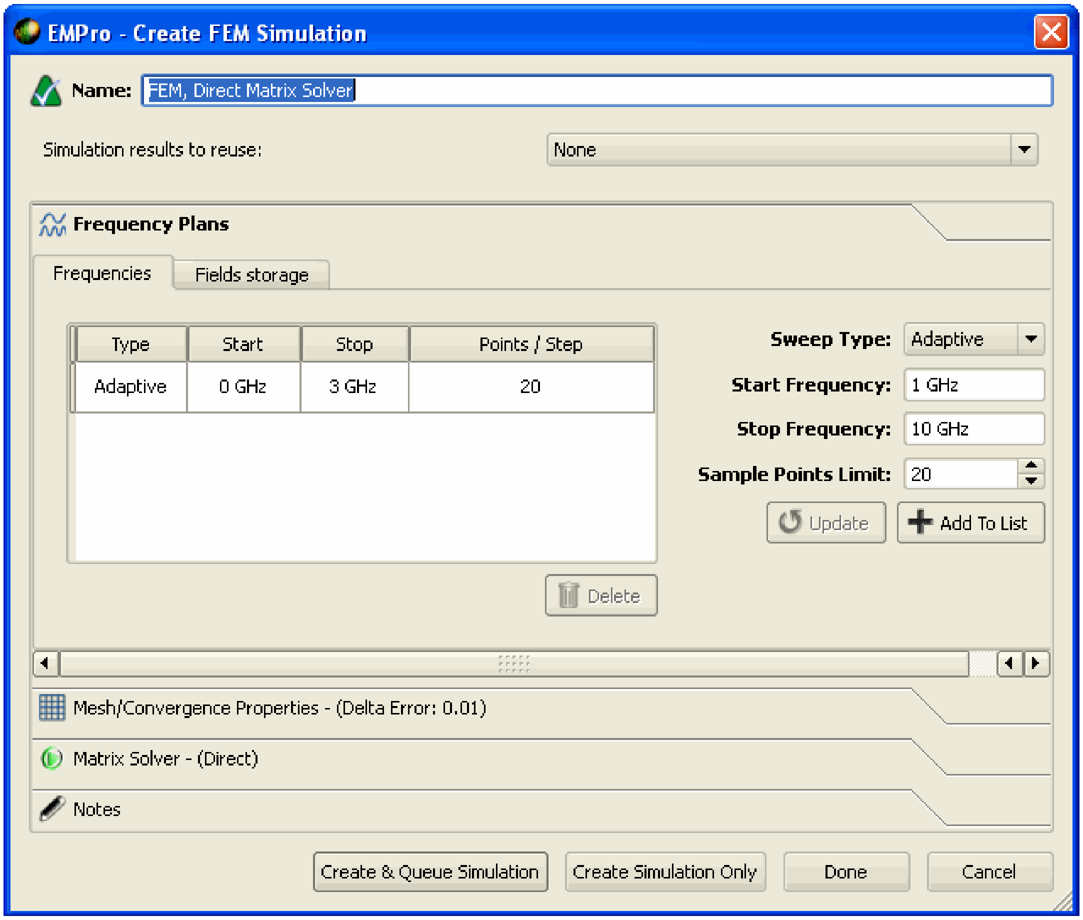
To set up a new FEM simulation:

1. Click the **Simulation** tab 

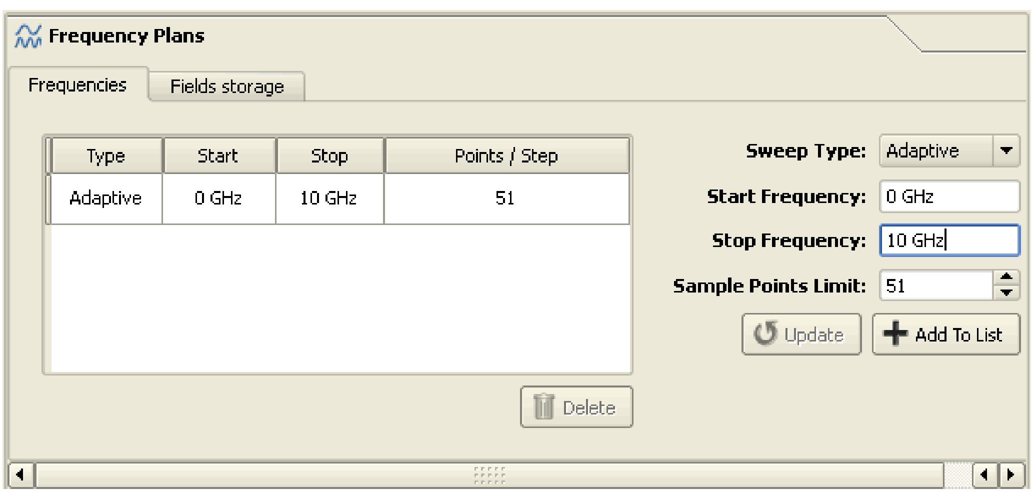
on right-side of the EMPro GUI. The *Simulation* window is displayed.



2. Click **Create FEM Simulation**. The *EMPro-Create FEM Simulation* window for FEM simulation appears.

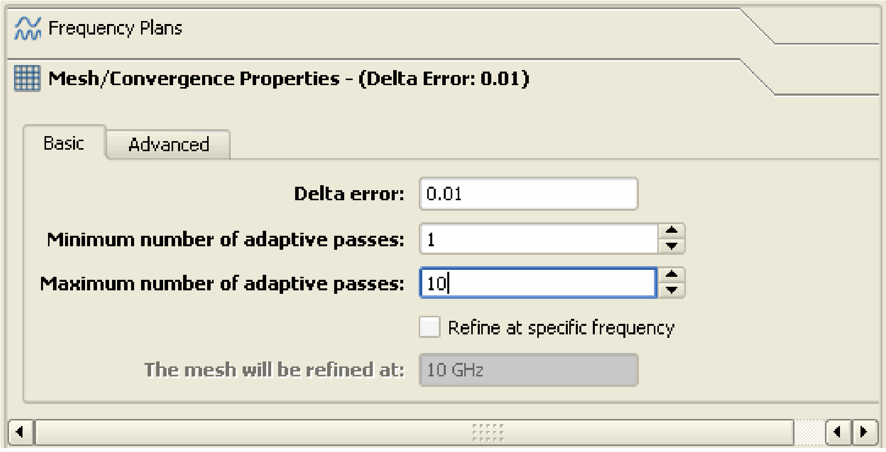


3. Add the *Frequency Plans* as shown in the *EMPro-Create FEM Simulation* window below.

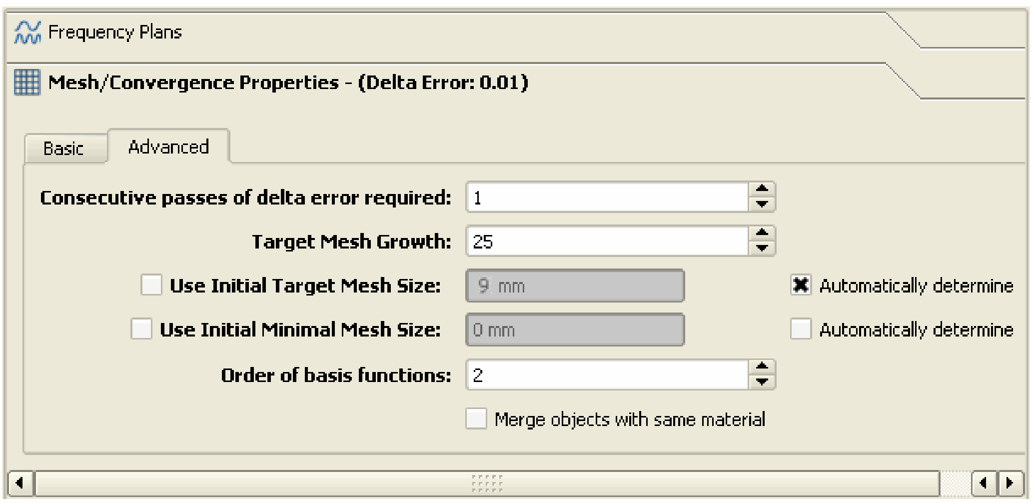


4. Under *Mesh/Convergence Properties* pane, perform the settings as displayed in the below screenshots:

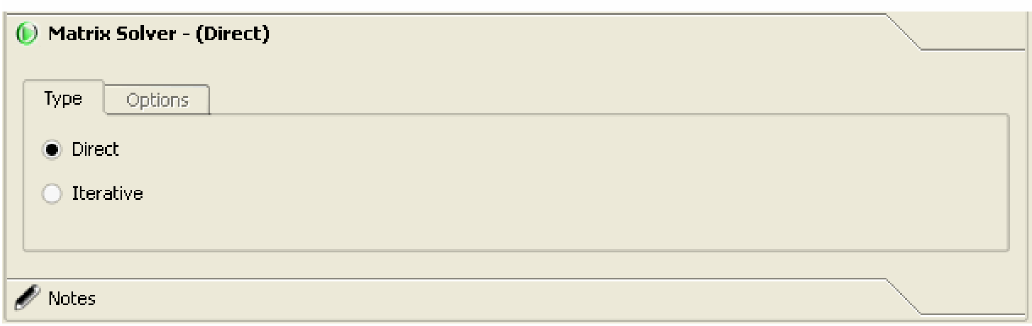
Basic Properties



Advanced Properties



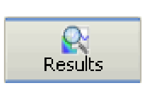
5. Choose **Direct** solver in *Matrix Solver* window.



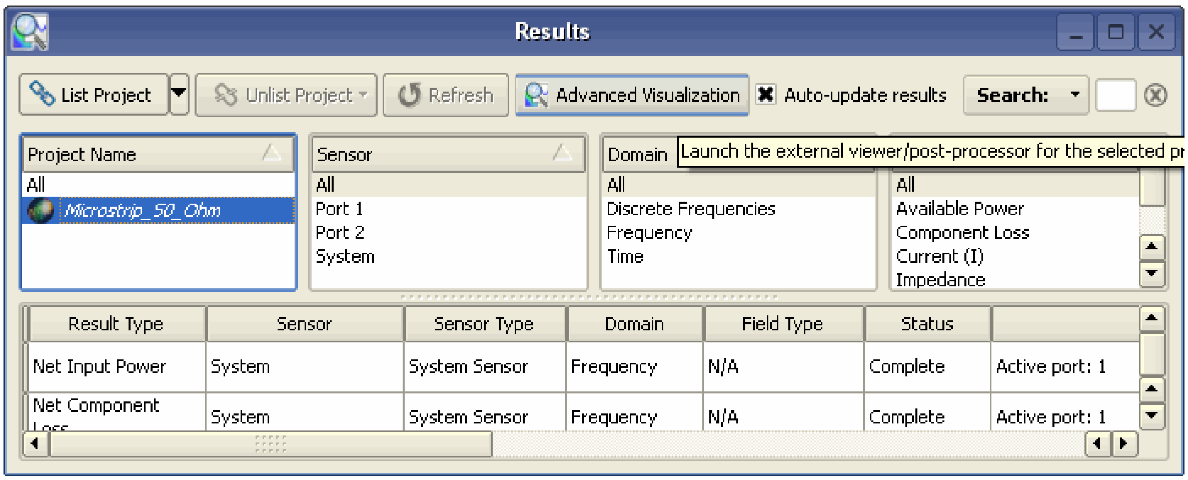
6. Click **Create and Queue Simulation** to start the FEM simulation.

Visualizing Symmetric Plane

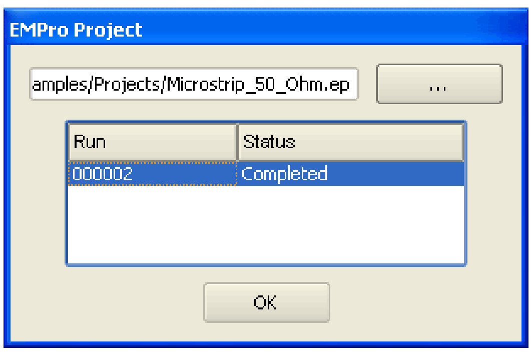
To visualize the symmetric plane:

1. Click the **Results** tab 

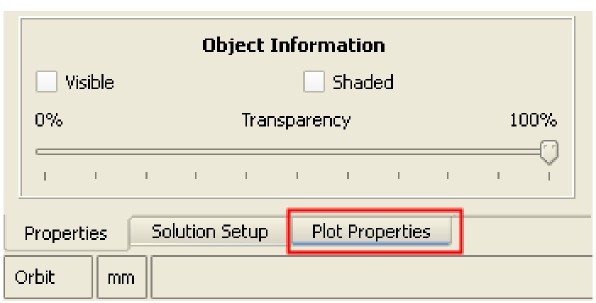
on right-side of the EMPro GUI. The *Results* window is displayed.



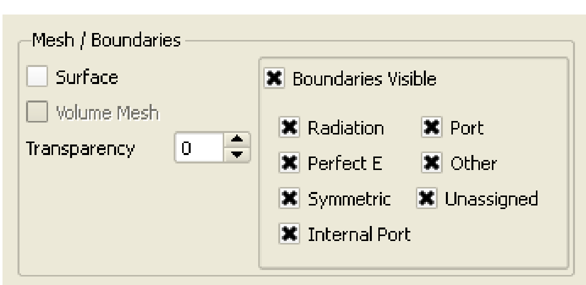
2. In *Results* window, select the project and click **Advance Visualization** tab.
3. After you click the Advance Visualization tab, the EMPro Project window is displayed.
4. In the EMPro Project window, select a project **Status** and click **OK**.



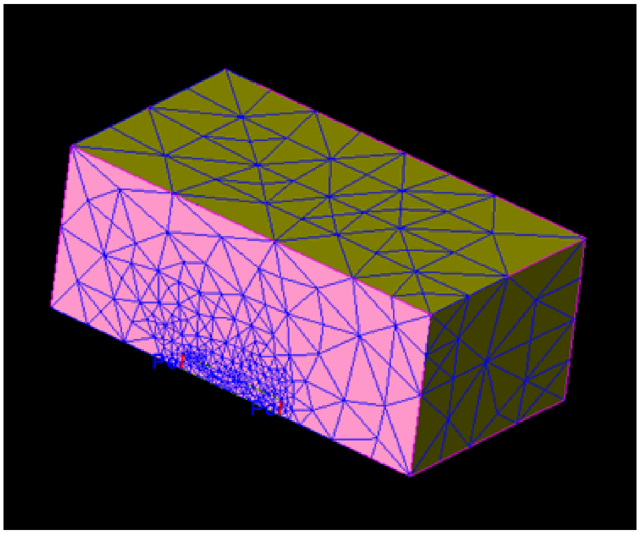
5. In the Advance Visualization, click **Plot Properties** tab.



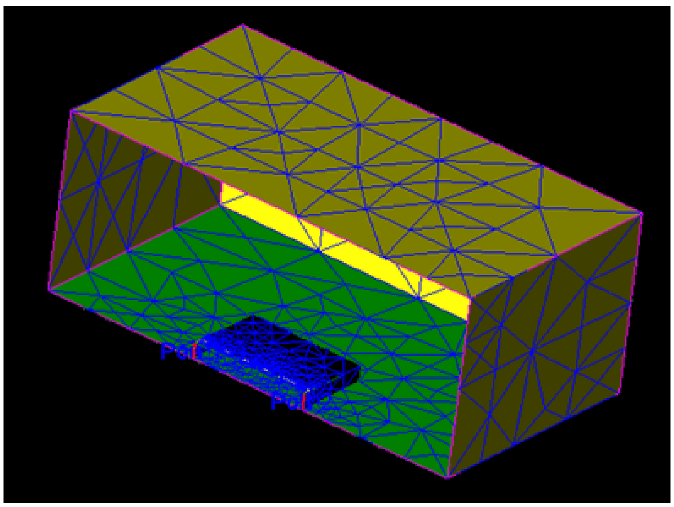
6. Under **Mesh/Boundaries** pane, select the **Boundaries Visible** check box.



Boundaries are visible on the object now. The pink color represents the symmetric plane.



To view the internal structure of the object, uncheck the **Symmetric** check box in the *Mesh/Boundaries* pane.



Simulating a Microstrip Line with Sheet Port

In this project, you will learn how to:

- Define a sheet current source

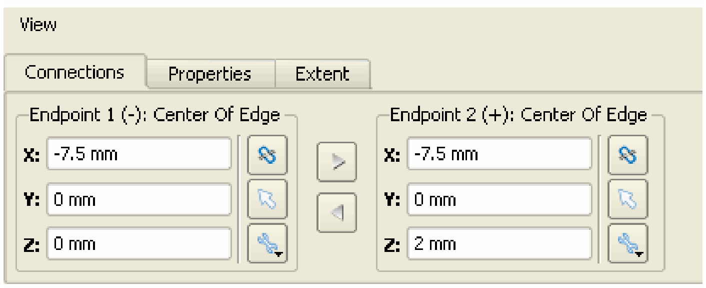
Getting Started

To demonstrate how to set up a sheet current source, we will use the Microstrip 50 Ohm line project from the example set:

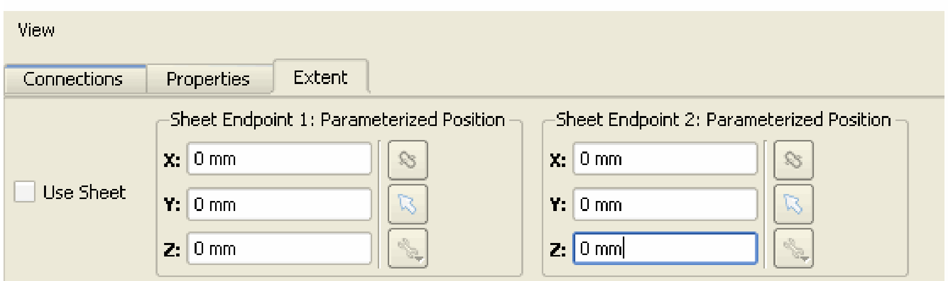
1. Unarchieve the **Microstrip 50 Ohm** project from **Help > Example**.
2. Save as *Microstrip_50_Ohm_Sheet.ep*.

Adding Sheet to Port

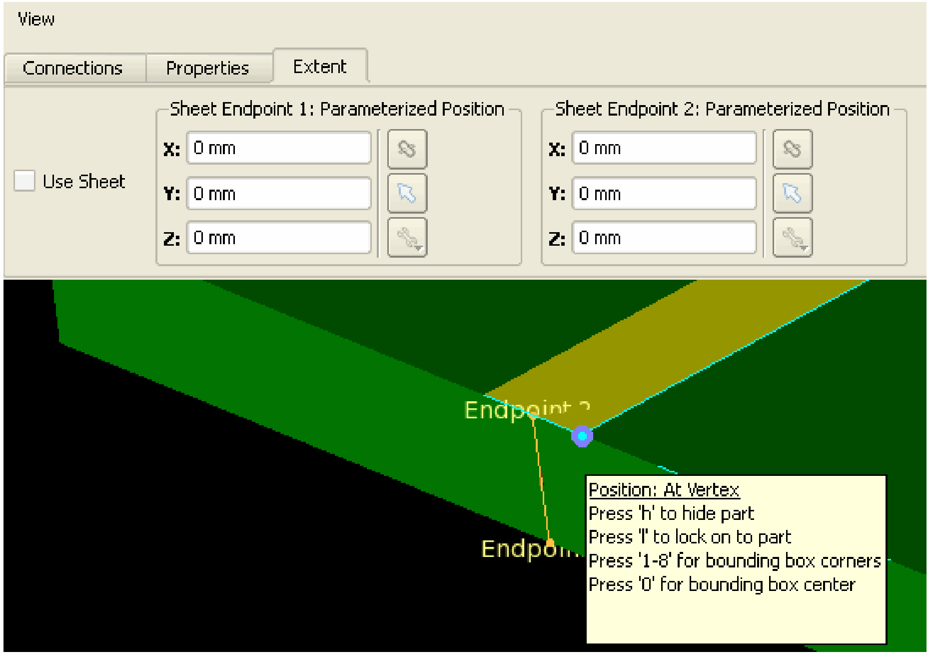
1. Double-click **Port 1** under **Circuit Components/Ports** in the project tree. The *Geometry-Editing Circuit Component* windows appears.



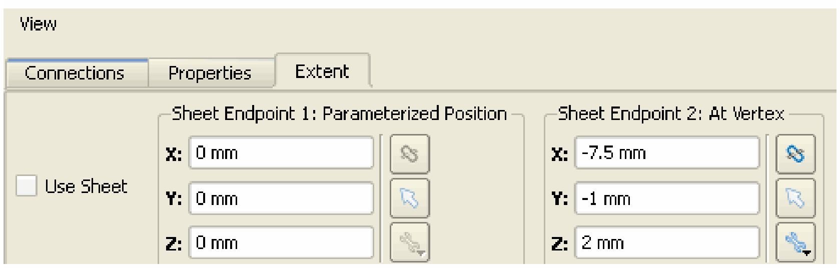
2. Click **Extent** tab. The sheet parameters are displayed.



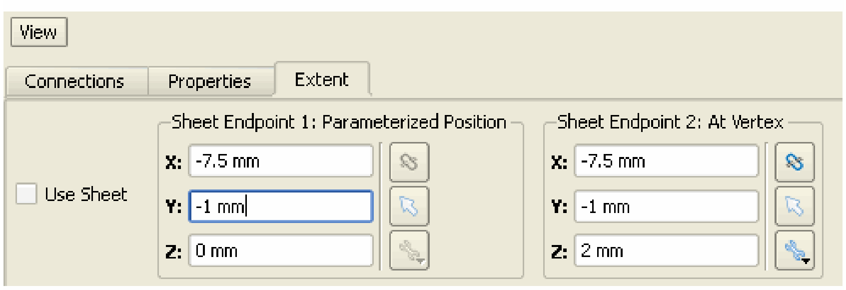
3. For Endpoint 2, Click  to specify the position and select one corner of Microstrip line as shown below:



Endpoint 2 will display the coordinates of one corner of the Microstrip line.

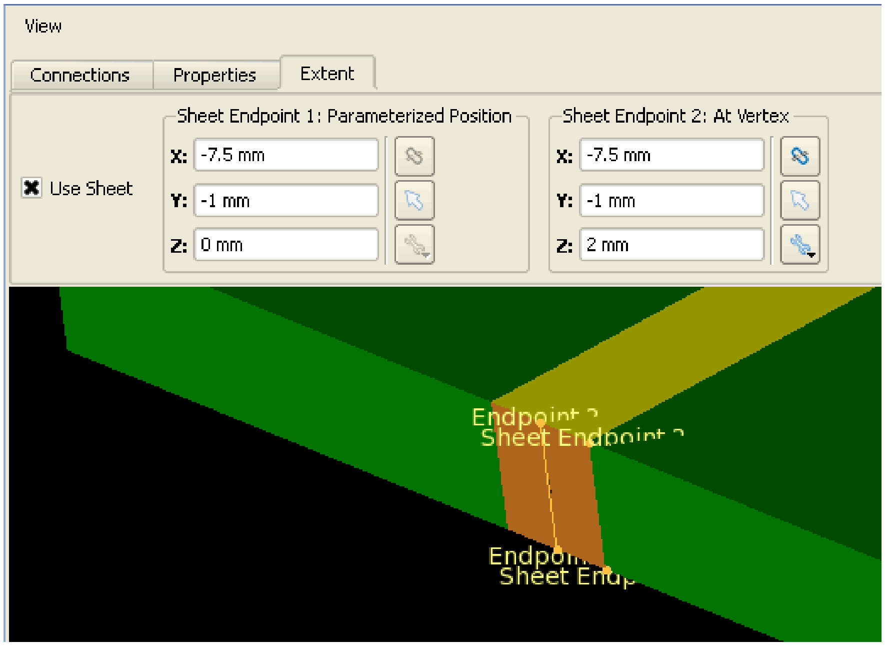


4. Copy X and Y coordinates from Endpoint 2 and paste to Endpoint 1.



5. Select **Use Sheet** check box and click **Done**.

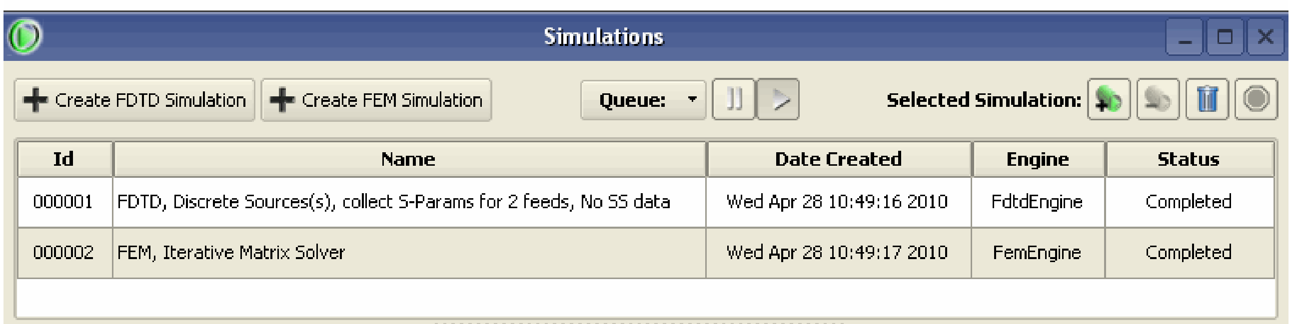
A sheet will appear of width equal to the width of the Microstrip line starting from the base of the substrate to the Microstrip line.



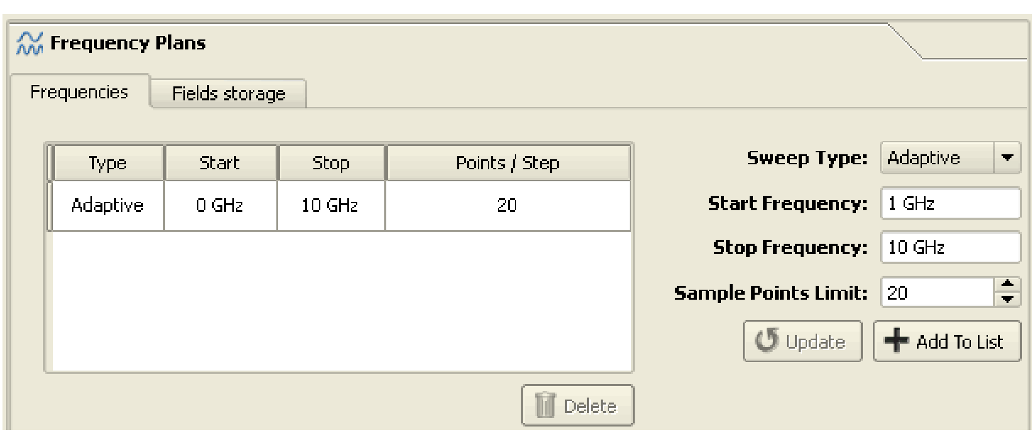
Following the above demonstrated process, add a sheet to Port 2 also.

Setting up New FEM Simulation

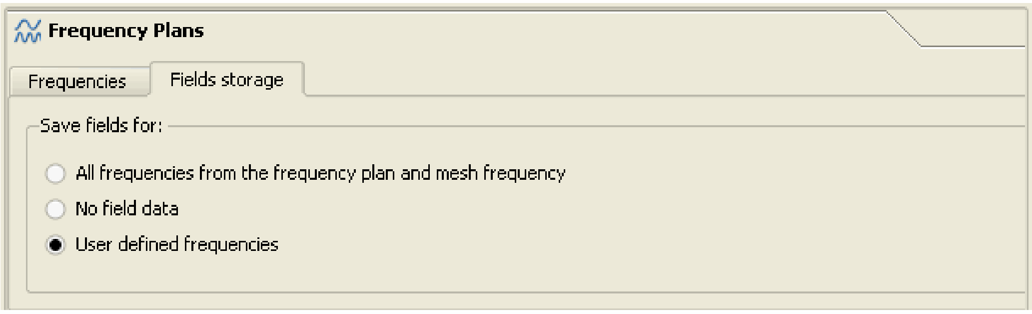
1. Click the **Simulation** tab on right-side of EMPro GUI. The *Simulation* window appears.



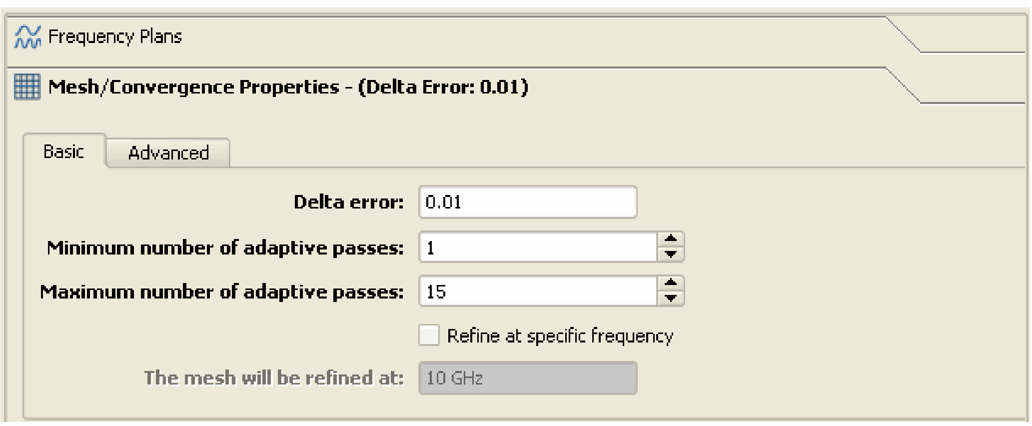
2. Click **Create FEM Simulation** on the *Simulation* window. The *EMPro-Create FEM Simulation* window appears.
3. Add the **Frequency Plans** as shown below.



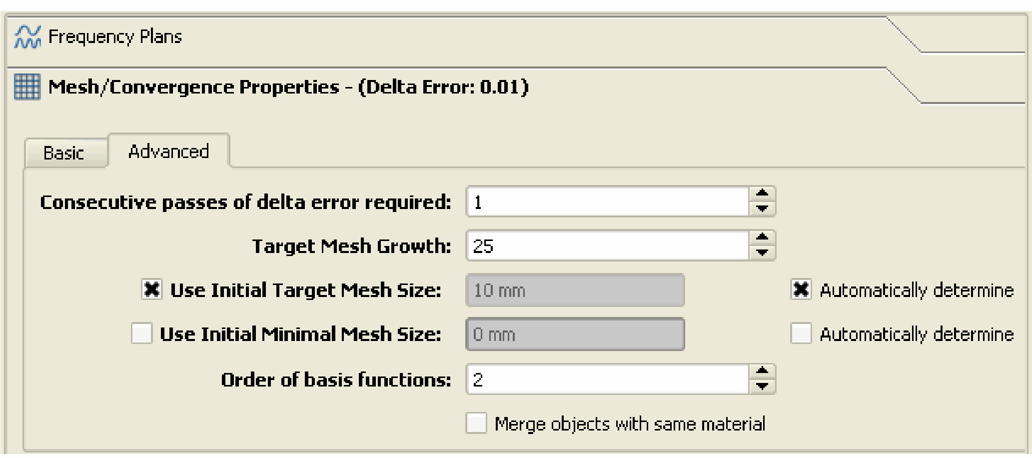
4. Click **Field Storage** tab and select **User Specified frequencies** radio button. This will save field data only at the starting and end frequency of the Adaptive frequency sweep.



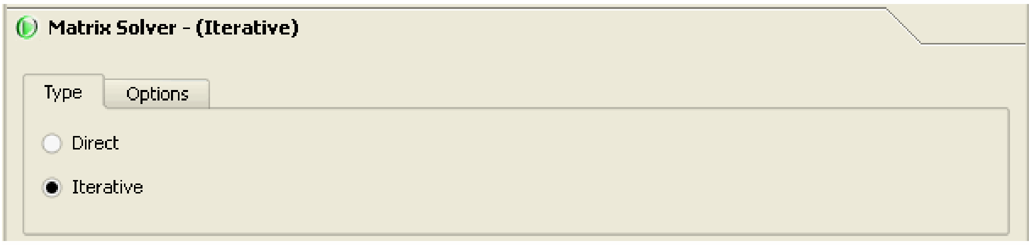
5. Click **Setup Mesh/Convergence Properties** tab and perform the Basic and Advance settings as shown below.
Basic Settings



Advance Settings
In Advanced tab, choose **Use Initial Target Mesh Size** check box.



6. Click **Matrix Solver** tab and select **Iterative** radio button.

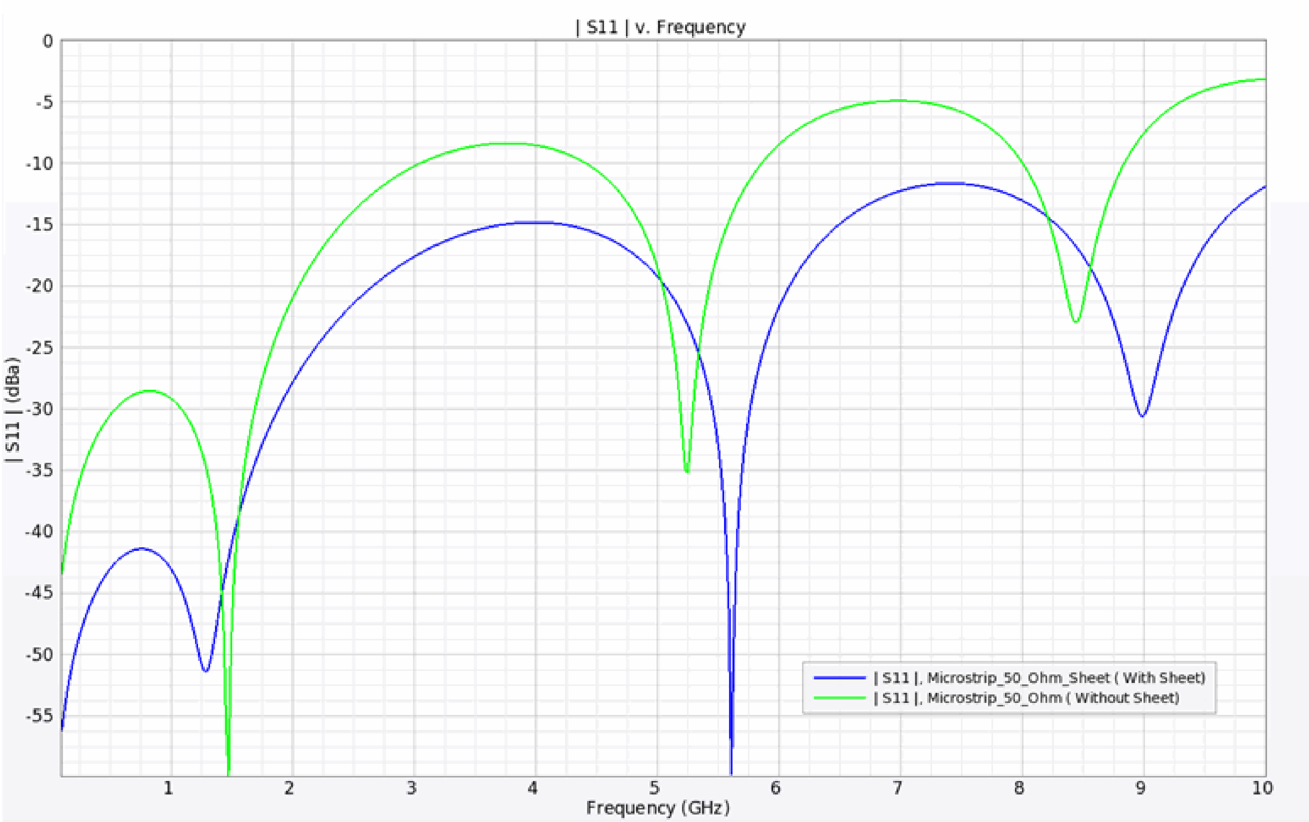


7. Click **Create and Queue Simulation** on *EMPro-Create FEM Simulation* window to start FEM simulation.

Result Comparison

An important advantage of sheet current sources is that they have a smaller parasitic inductance than line current sources.

This is highlighted in the following plot, showing S11 with and without sheet ports. If sheet ports are used, the resonance frequencies are shifted to the left. This shift is caused by the reduction of the parasitic inductance of the current sources.



Performing Multimode Analysis on Rectangular Waveguide

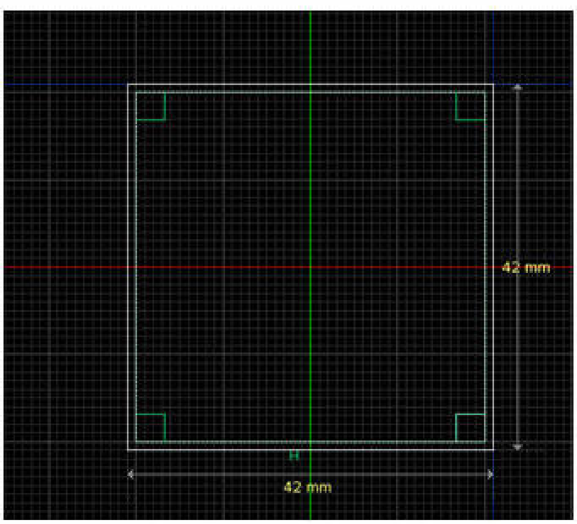
In this section, you will learn how to create a multimode analysis on any structure in EMPro. To demonstrate the design flow we have used a square waveguide. Perform the following steps to analyze a rectangular waveguide:

- Set up Waveguide Geometry Model
- Set up Waveguide Ports
- Set up Simulation
- View Results

Setting up Waveguide Geometry Model

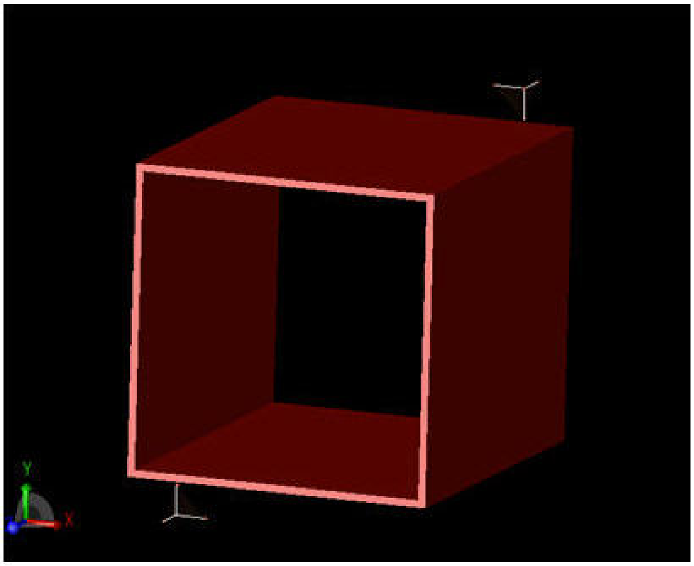
You can set up a waveguide geometry model by performing the following steps:

1. Select **Create > Extrude** to draw a square waveguide.
2. Choose **Rectangle** and specify the inner dimensions as (-20 mm, -20 mm) and (20 mm,20 mm). Also, specify the outer dimensions as (-21mm, -21 mm) and (21 mm,21mm).
3. Extrude the waveguide to 50 mm length. This will draw a square waveguide of 40 mm x 40 mm cross section and length 50 mm.



> **Note**
> This waveguide will support TE10 and TE01 mode at C band.

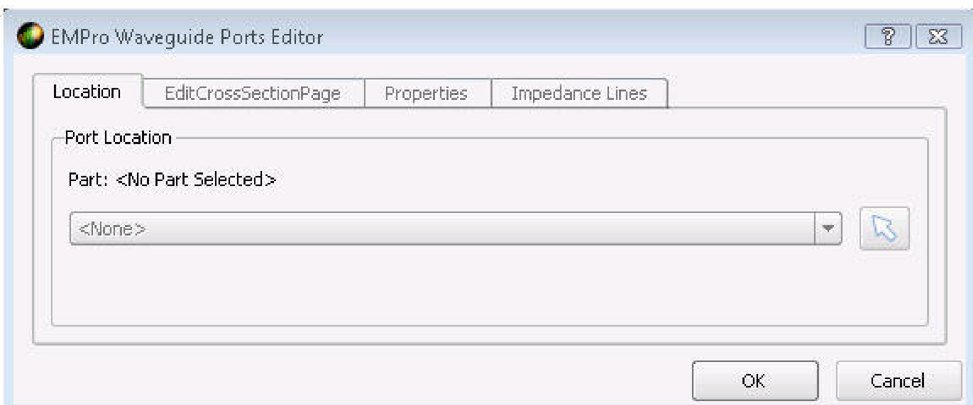
4. Specify the name the of the waveguide as **Square Waveguide** and assign the material **Aluminium**. The resulting waveguide appears as follows:



Setting up Waveguide Ports

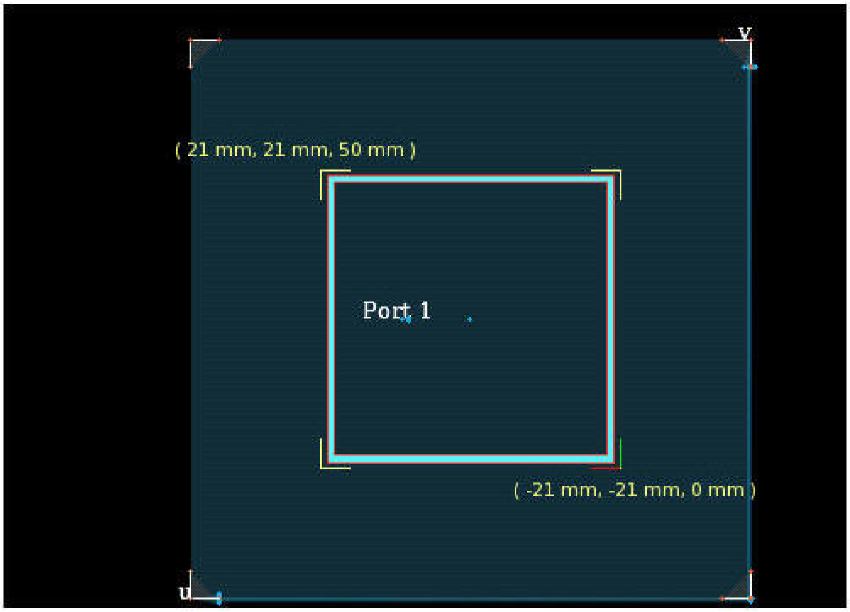
You can set up a waveguide port by performing the following steps:

1. Right-click **Circuit Components/Ports**> and select **New Waveguide Port** to a new waveguide port. This opens the **EMPro Waveguide Port Editor**, as shown in the following figure:

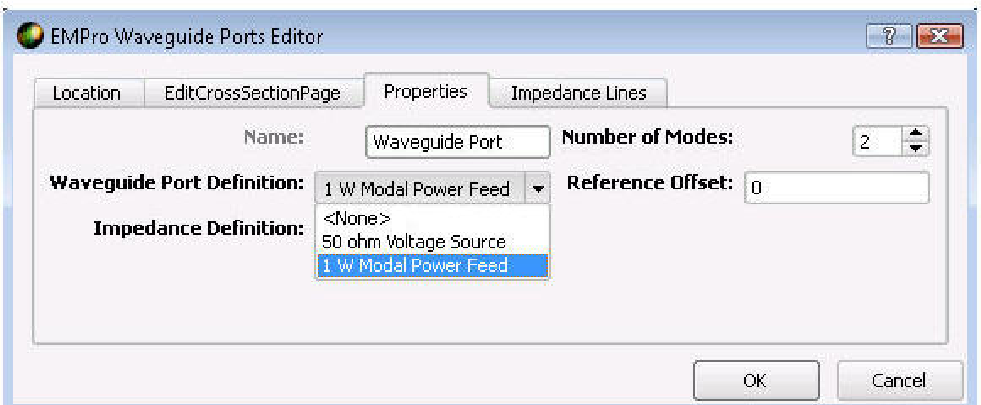


2. Choose view **Bottom**(_z)to define the Waveguide port at Z equals to 0mm value.

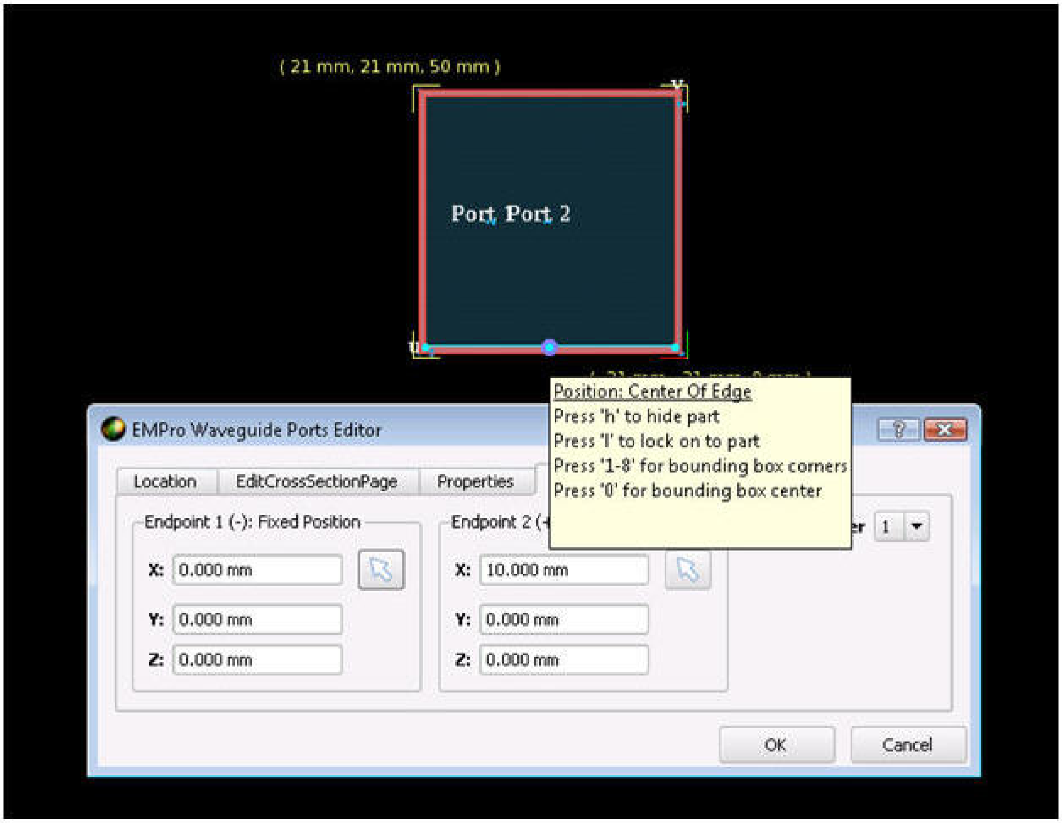
   From **Waveguide Port Editor** > **Location**, choose  to select the face:



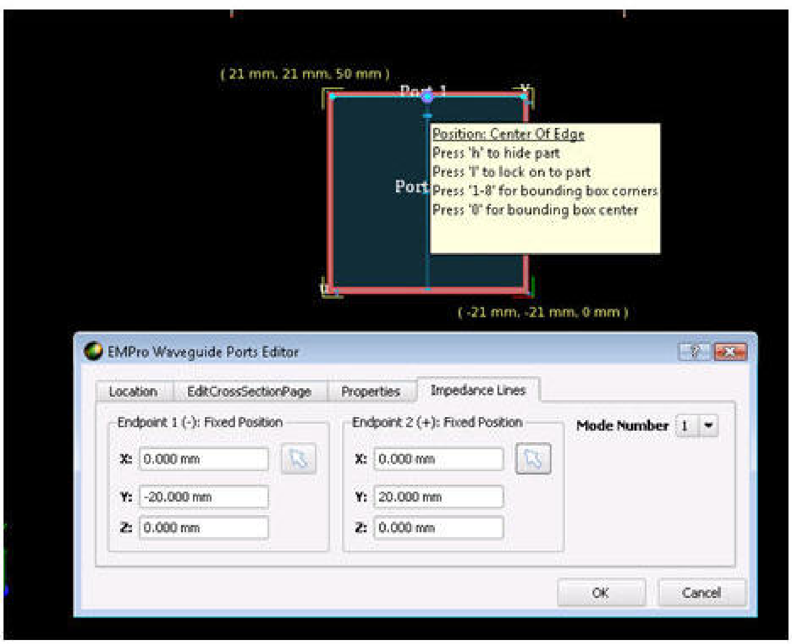
3. Under the **Edit** cross section, remove the selection of **Auto Extend to simulation domain boundaries**. This will restrict waveguide port to waveguide cross section only.
4. Under **Properties**, choose **Waveguide Port Definition** and choose the number of modes equals to 2. Also, for Waveguide port definition choose 1 W Modal power feed, as shown in the following figure:



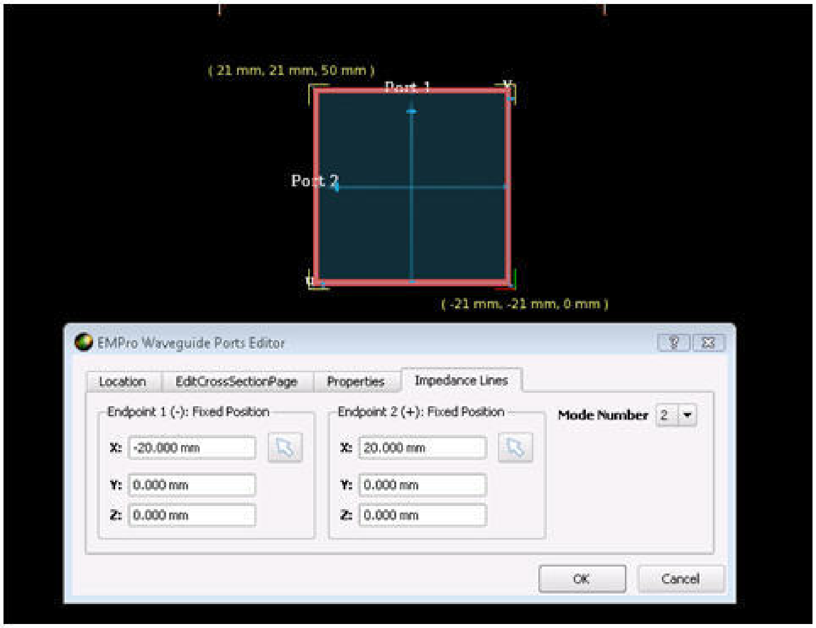
5. Under **Impedance for Endpoint 1**, choose **center of the lower edge**.



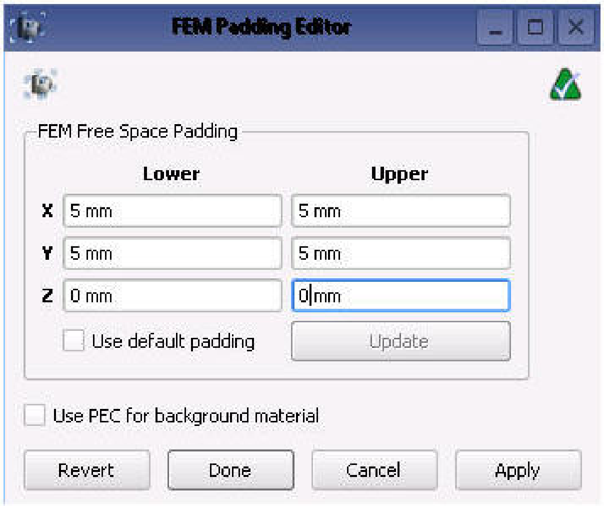
6. For endpoint 2, choose **center point of the upper edge**.



This defines mode 1 for port 1. Similarly define mode 2 for port 1.



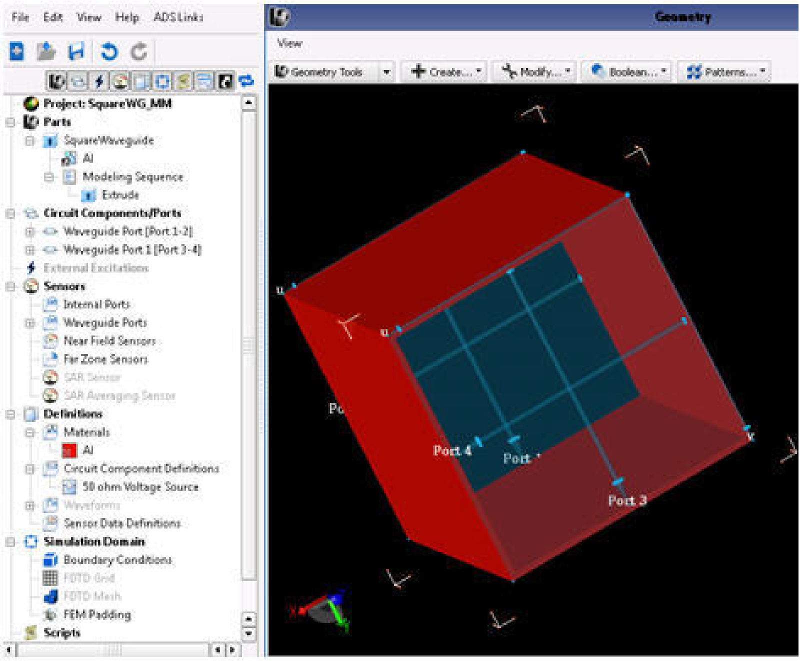
If Waveguide port 1 still show invalid please change the padding as shown below:



While doing the FEM padding of waveguide structure, consider the following guidelines:

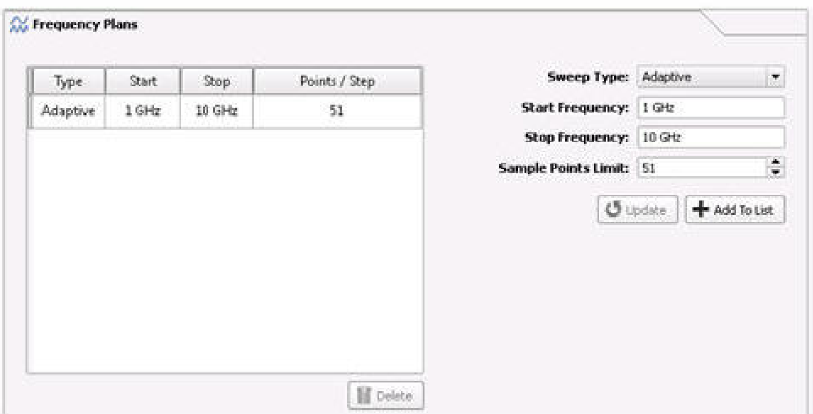
- The +/- Z padding MUST be zero for a valid waveguide port definition. The padding in the orthogonal direction of waveguide plane should be 0 mm. In this case, the Z axis is orthogonal to waveguide planes. After placing these paddings, the waveguide port will become valid.
- The +/- X and Y padding COULD be set to zero for a more efficient simulation Choose Top(-z) view to define second waveguide port at z=50 mm. Repeat the same steps as defined above to create second waveguide port and two modes.

A Waveguide with waveguide ports having two excited modes appears as follows:

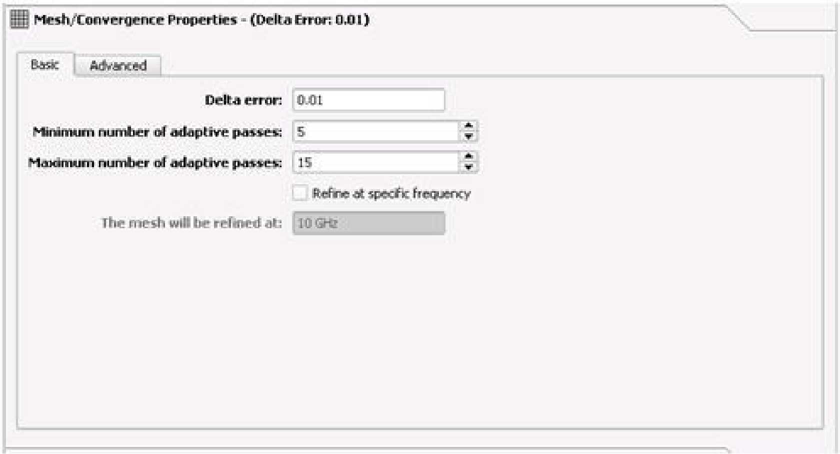


Setting up Simulation

1. Select simulation and New FEM simulation. Define the frequency plan as shown below:



2. Choose following Mesh/Convergence properties.

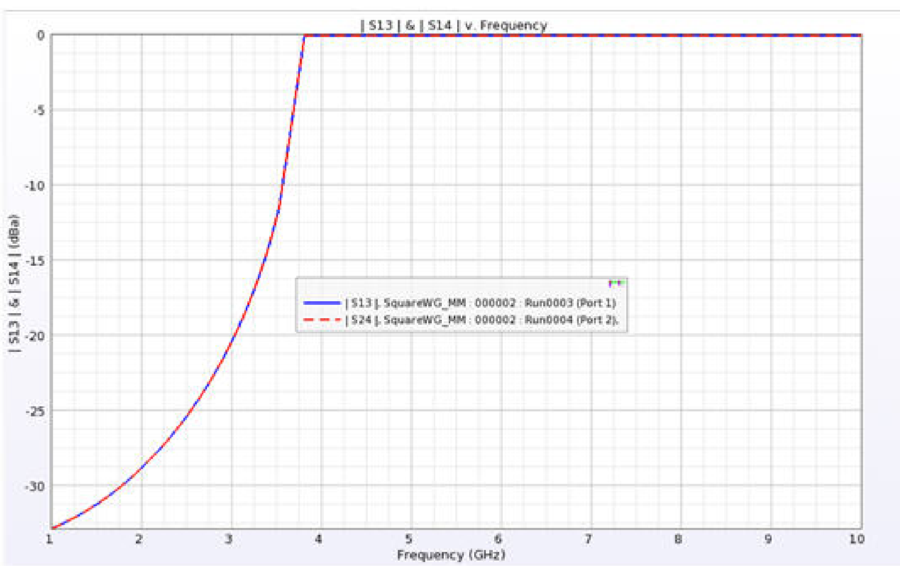


3. Choose direct solver and press Create and Queue simulation.

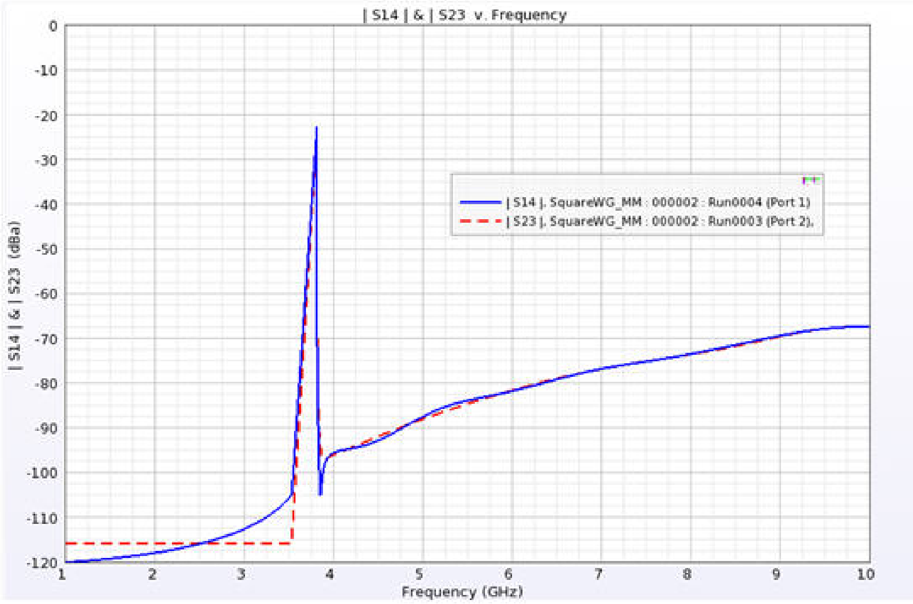
Viewing Results

Viewing S Parameter

For waveguide port 1, port 1 and port 2 are defined for 2 modes. And for waveguide port 2, port3 and port4 are defined for 2 modes. So port 1 of waveguide port 1 will couple to port 3 of waveguide port2 since they are for same mode. Similar coupling will take place for port 2 and port 4. The S13 and S24 plot is:



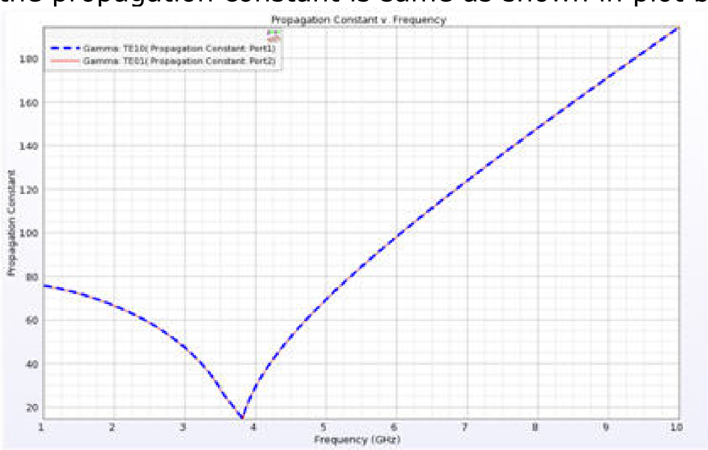
Cross mode coupling S14 and S23 is shown below:



The set of equations solved by FEM become ill-conditioned at waveguide mode cut-off frequencies. This leads to the spike in S14 and S23 seen around 3.75 GHz

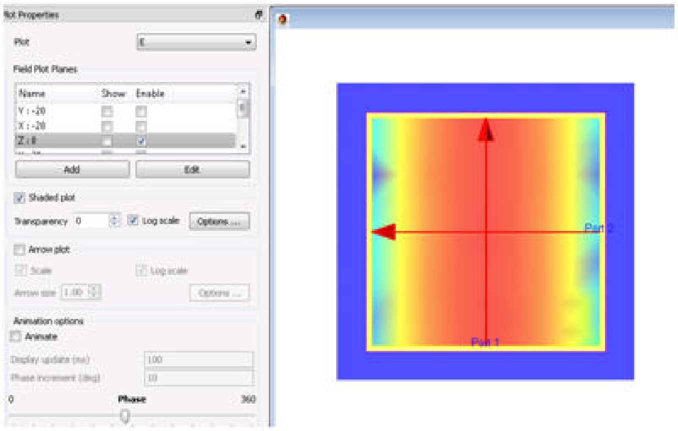
Viewing Propagation Constant

TE10 and TE01 mode is degenerate mode in square waveguide. For degenerate mode’s the propagation constant is same as shown in plot below:

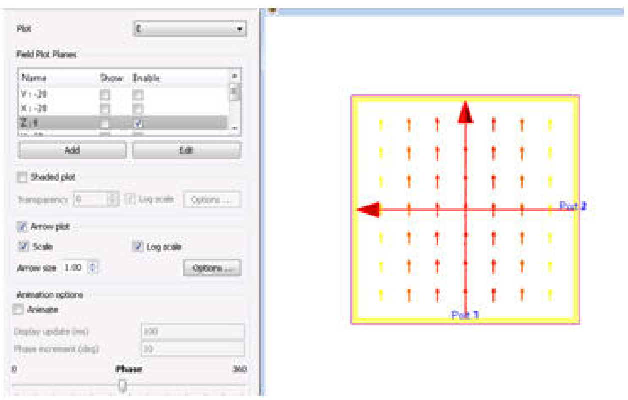


Viewing Field

Viewing field for different modes. From Result window> Choose project and then Advance Visualization. In advance visualization choose solution setup and select one of the frequency in pass band. For following plots frequency of 6.9 GHz is selected. Also choose Port 1 mode 1 to see TE10 mode in the waveguide.
In Plot properties, choose Z=0 in field plot planes. Choose Shaded plot and log scale. The plot will appear as

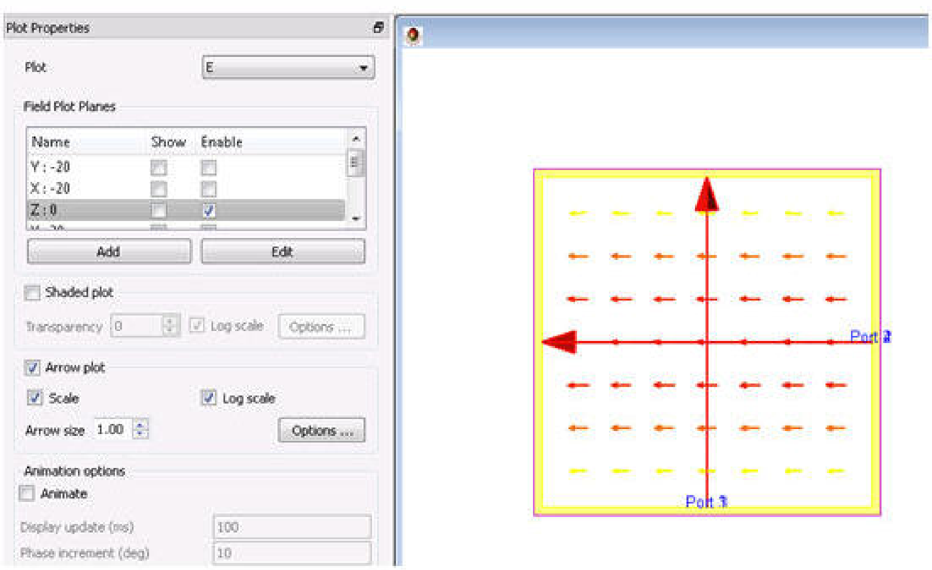


Select animate to see the progress of the field with phase. Remove the selection from **Shaded plot** and select **Arrow plot**. This is TE01 mode, as shown below:

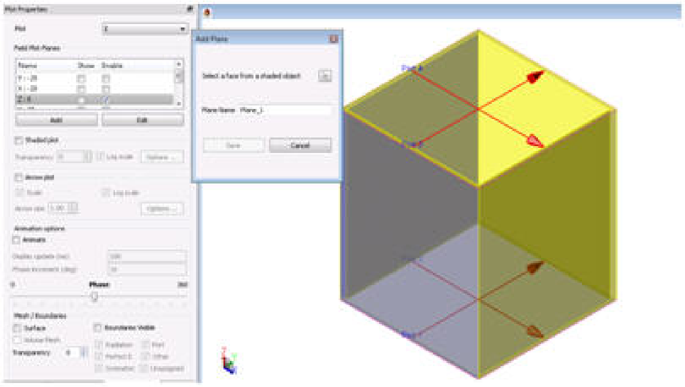


Go to Solution setup and select Port 1 mode2. Choose shaded plot in solution setup.

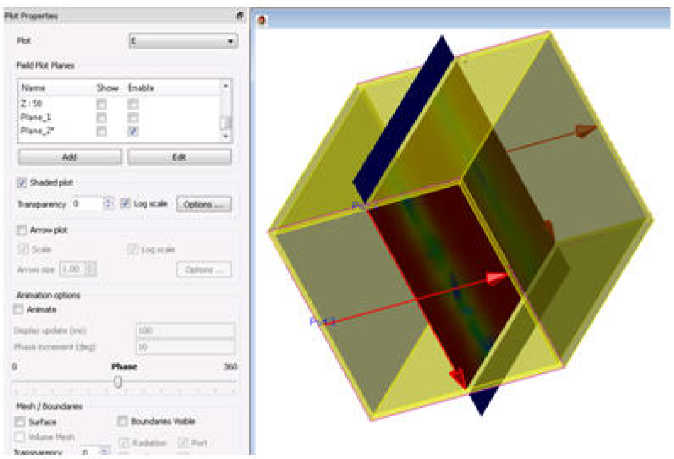
Uncheck shaded plot and check arrow plot. This is TE01 mode:



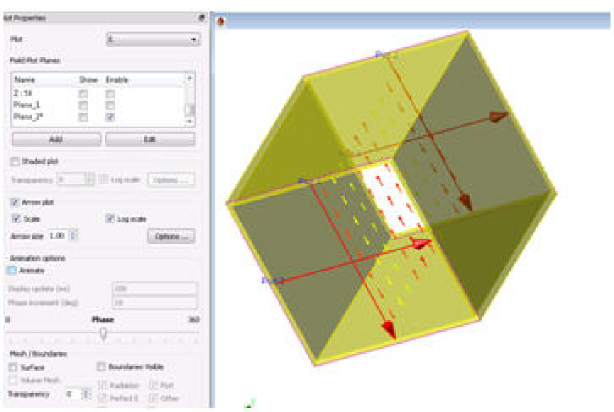
Add a plane by choosing Add in field plot planes to see the field along the length of the waveguide section.



Move this plane on the centre of the waveguide by using edit option. Choose Shade plot to see fields on this plane.



Similarly arrow plot on the plane is shown below:



Troubleshooting Mesh Failures

If the mesh generation fails in the surface meshing a list of failed faces assembled in an SAT file will be generated in the design directory. The exact location of this file will be shown in the output window of the simulation.

Best practices

- Simplify the design by removing unnecessary (for the EM-modeling) detail. Various geometry modification tools are available within the EMPro GUI to achieve this.
- Avoid shelled objects where the thickness of the shell is much smaller than the total dimensions of the object. In this case, replace the thick shell with a thin sheet.
- Visually inspect the design for small gaps between two 3D objects. Consider removing the gaps by moving one of the objects to create overlap between objects.
- Avoid using complex Boolean subtractions: use the mesh priority to set the correct object precedence for meshing. For example, a metal object sticking through a dielectric object will be meshed as such if the priority of the metal object is set to a higher value than the dielectric object.

To isolate specific objects that are causing mesh failures, you can selectively include or exclude objects from a simulation by using the Include In Mesh option in the context menu of objects.

Known Issue

Using English units may create small gaps or slivers in some designs, which can lead to mesh failures

EMPro uses SI (metric) units as the default system units. When a design is created and modified using English units, an internal unit conversion is made during each operation. Each internal conversion introduces small rounding errors. As more operations are completed, the rounding errors can accumulate to the point where small gaps or slivers may appear in the design. The FEM mesher has difficulty in meshing geometries containing these small gaps or slivers. This problem can be minimized either by finding and eliminating the small gaps or slivers, by reducing the number of operations on a design (reducing the number of internal unit conversions), by using metric units during the creation and modification of the design (eliminating internal unit conversions), or by using “Match Points” alignment tool. This issue will be addressed in the next release of EMPro.

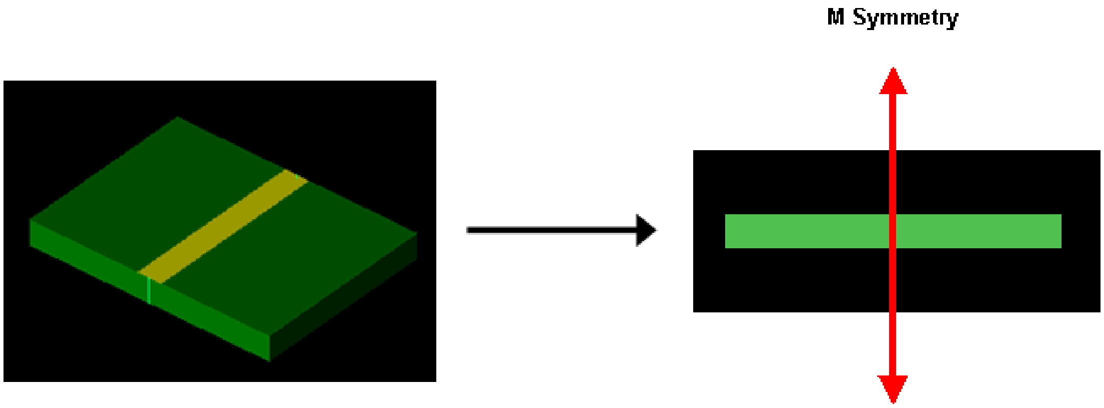
Tips for avoiding this problem

- Import designs using metric units. This will reduce the occurrence of errors caused by the conversion between metric units and English units.
- When designs are imported in English units, move designs as a group, and minimize unnecessary drawing operations to avoid the creation of small gaps or slivers.
- When performing operations, such as moving, on designs containing imported objects, using the “Match Points” alignment tool may eliminate any small geometrical offsets created.
- Create parametric equations in metric units. The conversion between metric and English units has limited precision, so mixing parametric equations and direct coordinate entry may introduce small gaps or slivers.

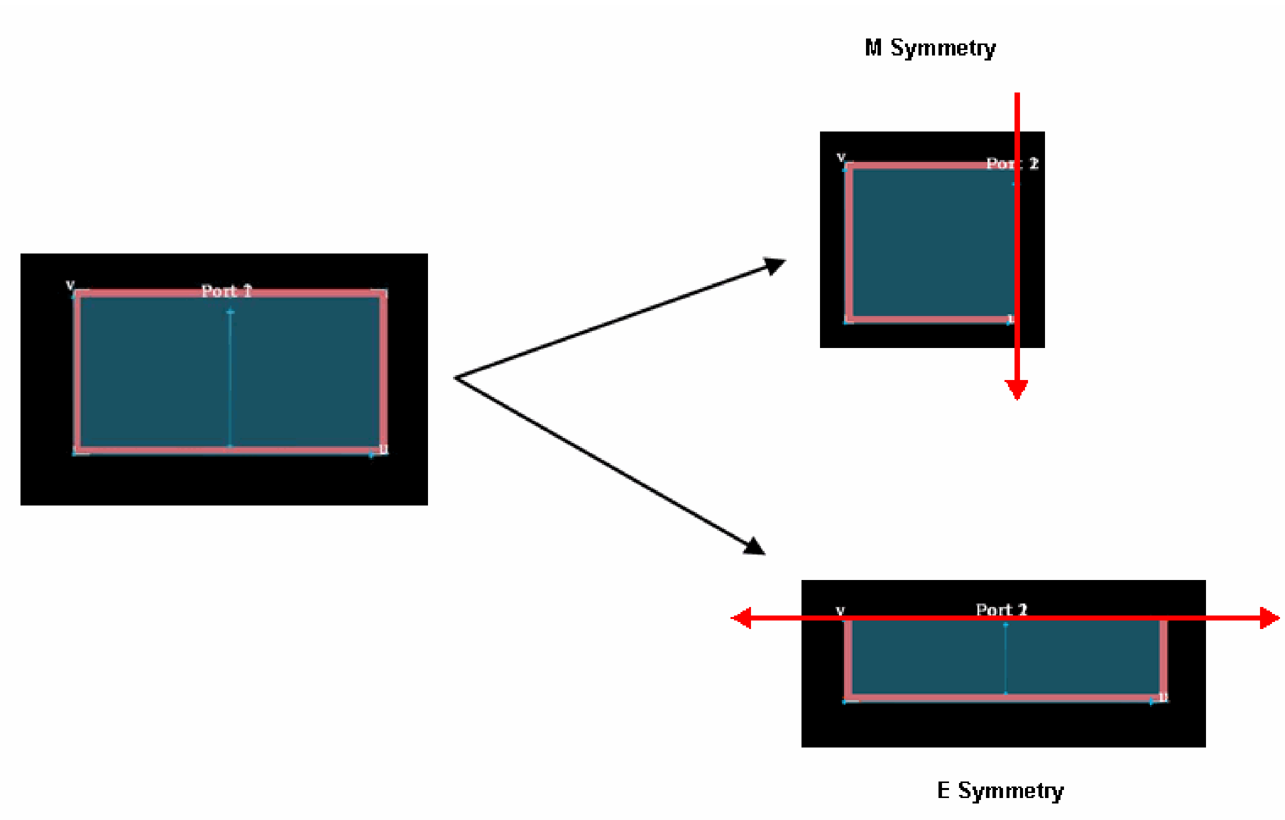
Symmetry Plane Boundary Condition

The Symmetry Plane Boundary Condition is supported by FEM simulator option only. It can be applied to structures which have electric (E) and magnetic fields (M) symmetry. This can be understood by simple structure like microstrip line and rectangular waveguide operating in dominant mode.

Microstrip line

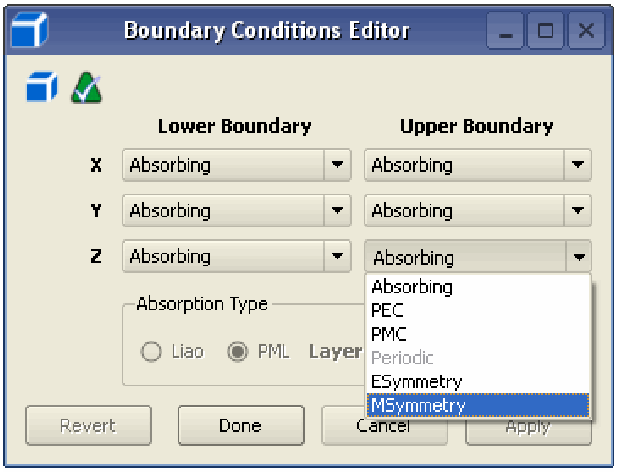


Rectangular Waveguide



If a structure has any symmetry (E or M), the structure physical size can be made half and symmetric plane boundary condition can be applied. Symmetric plane will take care of other half. This way the size of the problem subjected to simulation is reduced to half which now can be simulated in less amount of memory and lesser time. If a structure has both kind of symmetry (like rectangular waveguide)then it can be reduced to quarter, thus giving further advantage on memory and time.

Symmetric plane can be accessed under **Simulation Domain > Boundary Conditions** in the Project Tree.



Following are few guidelines to follow when specifying circuit sources with symmetry planes:

- Circuit sources should be parallel to MSymmetry planes.
- Circuit sources should be normal to ESymmetry planes.
- If there are symmetry planes, then circuit sources should 'touch' all symmetry planes.

Note
For more details on how to create a symmetric plane and apply the symmetric plane boundary condition, see *Simulating a Microstrip Line with Symmetric Plane* (fem) .

Internal Sheet Ports with Reduced Parasitics

The primary non-compensated (or parasitic) effect of the internal ports implemented in EMPro FEM is a self-inductance generated by the current impressed locally by the source.

In EMPro 2009, this locally impressed current has a distribution that is localized to the impedance line which is the path along which the voltage of the source is computed (displayed in the 3D EM Preview window). The parasitic inductance of this source is approximately the inductance of a wire that has the length of the *impedance line* and a small *effective radius*. The effective radius is impacted by both the local mesh density and mesh orientation.

In EMPro 2010, the impressed current of a current source can be distributed uniformly along a user-defined sheet. As such, the parasitic inductance of this source is essentially the inductance of a sheet of uniformly distributed current with an *effective thickness*. This thickness is of the same order as the *effective radius* of the *non-distributed* source implementation.

Sheet current sources have the following advantages in comparison to non-distributed current sources:

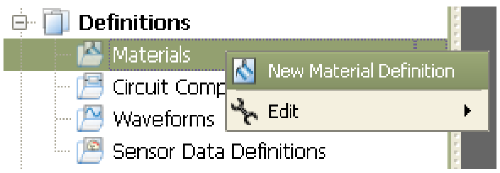
- Lower parasitic inductance
- Decrease in the current fan-out as the current spreads to be the current distribution of the signal line

Note
For details on how to implement sheet ports with reduced parasitics, see *Simulating a Microstrip Line with Sheet Port* (fem) .

Debye and Lorentz Materials

EMPro 2010 adds support for Debye and Lorentz material properties. This allows modelling of frequency dependent materials including FR4 and plasmas.

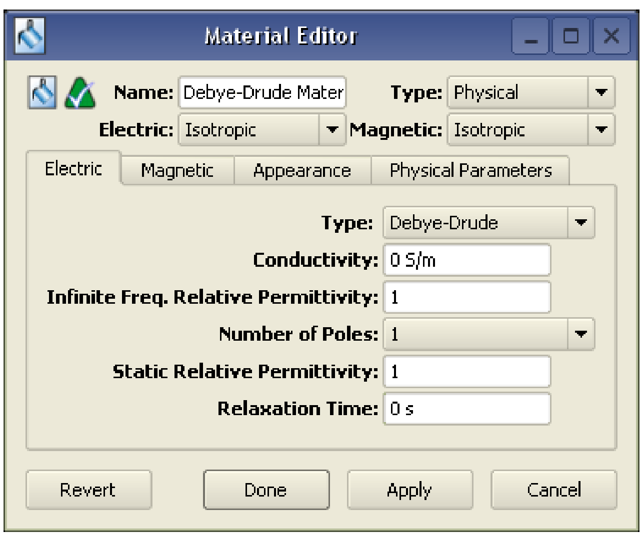
To define a new material, right-click **Definitions: Materials** branch of the *Project Tree* and select *New Material Definition*. A *Material* object will be added to this branch. Depending on the project preferences, the *Material Editor* window will appear automatically. If not, simply double-click on this object to bring up the editor. Similarly, double-click on any existing *Material* icon to edit an existing material within the *Material Editor*.



Debye/Drude

For a *Debye/Drude* material, the electrical *Conductivity* (σ) in siemens per meter, Infinite Frequency Relative Permittivity (ϵ_∞), *Number Of Poles*, *Static Relative Permittivity* (ϵ_r), and *Relaxation Time* (τ) in seconds must be specified. For a *Debye* material, σ must equal zero. A non-zero conductivity value results in a *Drude* material.

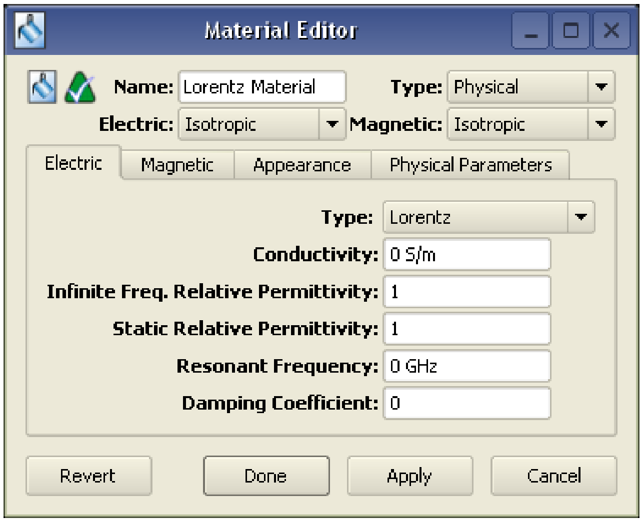
Defining a Debye/Drude material



Lorentz

For a *Lorentz* material, the electrical *Conductivity* (σ) in siemens per meter, Infinite Frequency Relative Permittivity (ϵ_∞), *Static Relative Permittivity* (ϵ_r), *Resonant Frequency* and *Damping Coefficient* must be specified.

Defining a Lorentz material



Note
For more details on how to specify Debye and Lorentz materials, see *Creating Materials* (using) .

Reuse of Mesh and Frequency Points

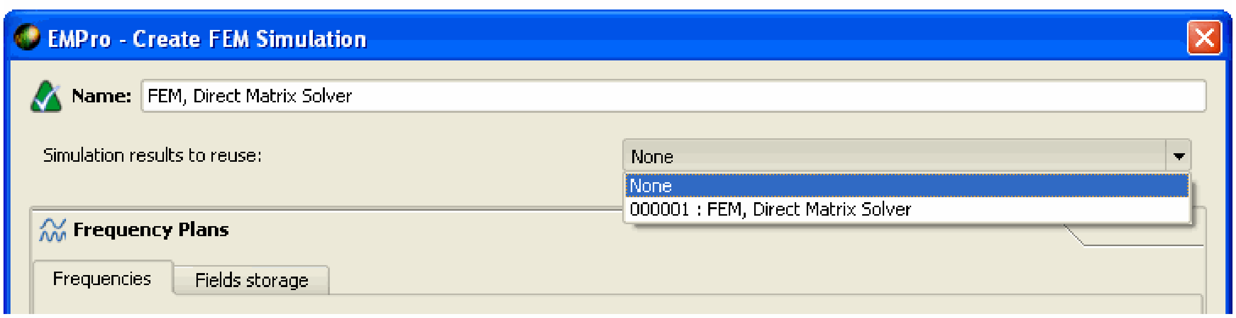
EMPro 2010 provides a new feature for reusing any existing FEM simulation.

For a new FEM simulation, simulation setup in terms of mesh refinement, AFS points, solver can be reused from existing simulation. The results from an existing FEM simulation can be reused in a new FEM simulation. In this case, the results from the existing simulation will be the starting point for the new simulation.

To demonstrate how to reuse the simulation setup, we will use the QFN Package project from the example set:

1. Unarchieve the **QFN Package** project from **Help > Example**.
2. Open the *QFN Package.ep*.
3. Click the *Simulation* tab on right-side of EMPro GUI. The *Simulation* window appears.
4. Click **Create FEM Simulation** on the **Simulation** window. The *EMPro-Create FEM Simulation* window appears.
5. Add the Frequency Plans relevant for your project.
6. Once the simulation seetings are complete, click **Create and Queue Simulation** on *EMPro-Create FEM Simulation* window to start FEM simulation.

You can reuse the above created simulation in another simulation by selecting a simulation from list of already solved simulations from the **Simulation Results to Reuse** drop-down list in the *EMPro-Create FEM Simulation* window. Change the pre-existing simulation settings according to your new requirements, and start the new simulation. In the output, it will be shown which results are reused.



Following are few scenarios where reuse of already solved simulations option can be useful:

1. **Increasing the maximum number of refinement steps** - This can be useful if the mesh refinement did not converge.
2. **Increasing the accuracy** - This can be done by decreasing target delta S or by increasing the number of consecutive passes the target delta S criterium must be satisfied. This will result in additional refinement steps.
3. **Changing the refinement frequency** - Additional refinement steps will be done for the new refinement frequency. This can be useful if the field solutions are fundamentally different for different frequencies. It also can be used to achieve a broadband refinement.
4. **Increasing the maximum number of frequency points for AFS** - This is useful if AFS did not converge.
5. **Using a different setting for the linear solver**:
   1. If the direct solver aborted because of an out of memory problem, the simulation can be continued by reusing it with the iterative solver selected instead.
   2. If the iterative solver aborted because it did not converge, the simulation can be

continued by reusing it and:

   1. either allowing more iterations for the iterative solver or,
   2. decreasing the tolerance for the iterative solver or,
   3. with the direct solver selected instead.

6. **Changing the frequency plan:**
   1. Extending the frequency range.
   2. Adding discrete frequencies: this can be used to force AFS to zoom in into a specific frequency range.

Any of the above use scenario’s can be combined.