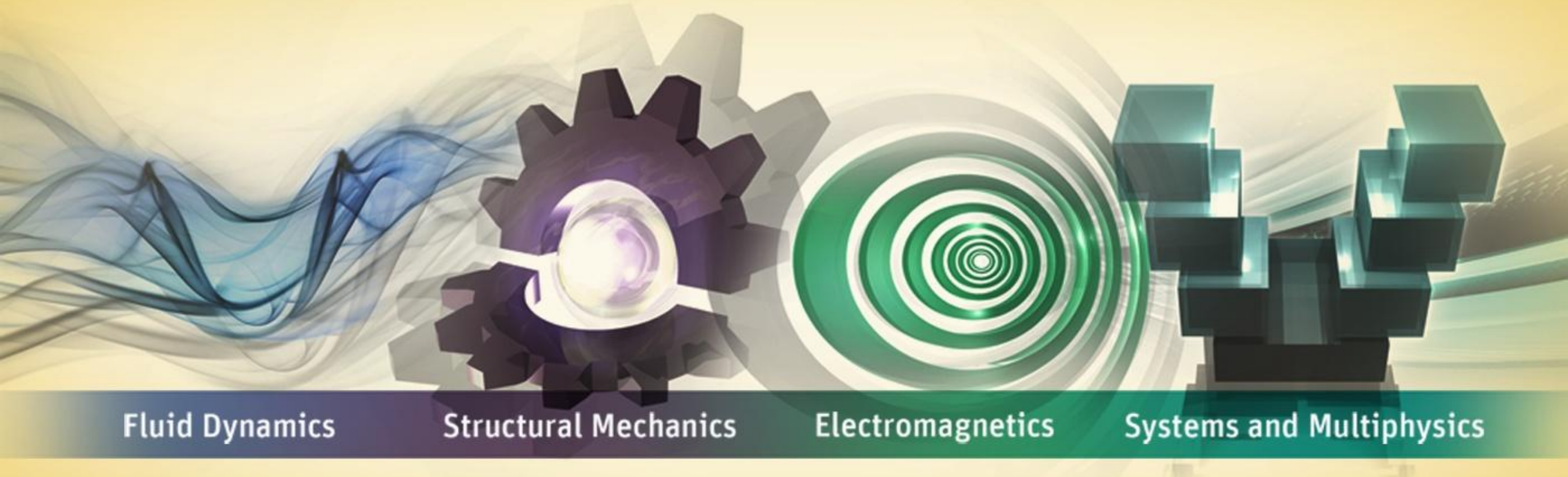
Realize Your Product Promise™

Lecture 4:
Static Electric Solvers



ANSYS Maxwell V16
Training Manual

Content

A. Electrostatic Solver

Electrostatic Solver

- The Electrostatic solver solves for the static electric fields resulting from stationary charge distribution or applied potentials
- Electric Field (E) and Electric Flux Density (D) are automatically calculated from the scalar potential (ø)
- All fields inside conductors are assumed to be perfect and equipotential in an electrostatic equilibrium (no current flow), therefore Joule losses are zero everywhere
- The Electrostatic solver utilizes an automatic adaptive mesh refinement technique to achieve an accurate and efficient mesh required to meet defined accuracy level (energy error)

Electrostatic Equations

- Following equations are solved with Electrostatic solver

$$\nabla \bullet (\varepsilon_r \varepsilon_0 \nabla \Phi) = -\rho_v$$

Maxwell 3D

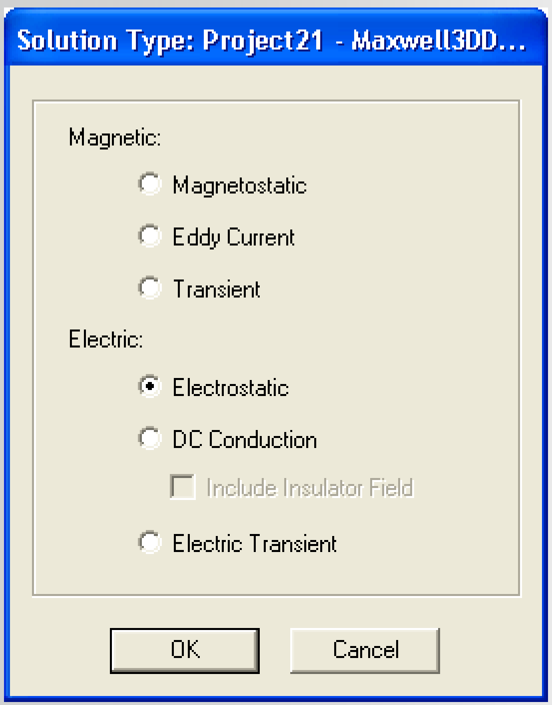
$$\nabla \bullet (\varepsilon_r \varepsilon_0 \nabla \phi(x, y)) = -\rho$$

Maxwell 2D

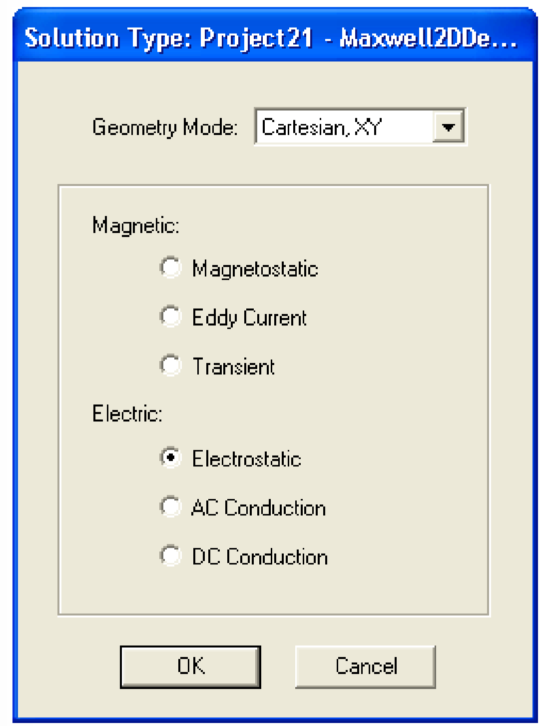
a. Selecting the Electrostatic Problem

Selecting the Electrostatic Solver

- By default, any newly created design will be set as a Magnetostatic problem
- Specify the Electrostatic solver by selecting the menu item ***Maxwell 2D/3D → Solution Type***
- In Solution type window, select **Electric > Electrostatic** and press **OK**



Maxwell 3D



Maxwell 2D

b. Material Definition

Electrostatic Material Properties

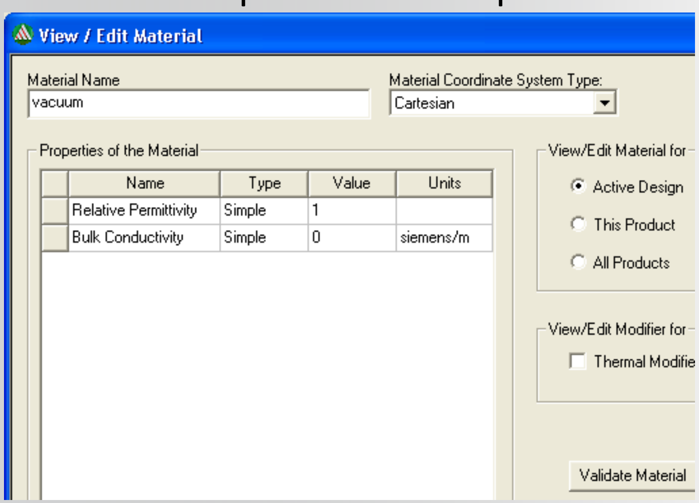
– In a Electrostatic simulation, the following parameters may be defined for a material:

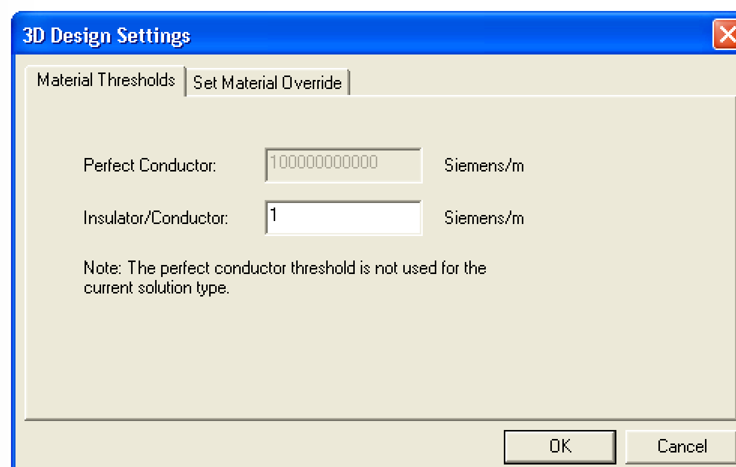
Relative Permittivity:

- Permittivity (ε) is give as $\boldsymbol{\varepsilon_0 * \varepsilon_r}$
- Relative permittivity(ε_0) determines the electric field solution in the insulators.
- Relative permittivity can be Simple or anisotropic

Bulk Conductivity:

- Defines whether an object is a conductor (treated as a perfect conductor in the Electrostatic solver) or an insulator.
- This classification is determined by the insulator/conductor material threshold setting defined under ***Maxwell 3D/2D → Design Settings → Material Thresholds***
- Can be Simple or Anisotropic





c. Boundary Conditions

Assigning Boundary Conditions in 3D

- Boundary conditions define behavior of the electric field at the interfaces or the edges of the problem region
- A boundary can be assigned to a face from menu item ***Maxwell 3D → Boundaries → Assign*** and select the required boundary assignment

Boundary Types(3D)

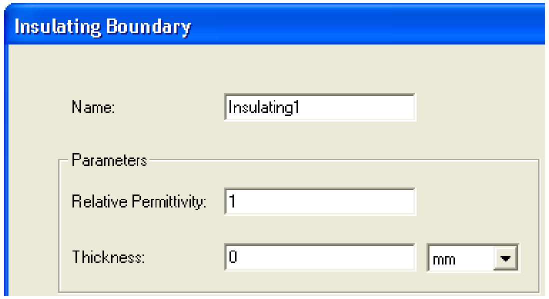
Default (No Boundary Assigned):

When no boundary is specified for a surface following two treatments are assigned based on the surface position

- **Natural**: For the boundaries on the interface between objects. Normal component of the D Field at the boundary changes by the amount of surface charge density on the boundary.
- **Neumann:** For exterior boundaries of solution domain. E Field is tangential to the boundary and flux cannot cross it.

Insulating:

- E Field can be discontinues across the insulating boundary
- Can be used to model thin layer of insulation by specifying Permittivity for the layer



…Boundary Conditions

Boundary Types (2D & 3D):

Master/Slave :

- Enable users to model only one period of a periodic structure, which will reduce the size of a design.
- This boundary condition matches the electric field at the slave boundary to the field at the master boundary based on U and V vectors defined.

Symmetry Boundary:

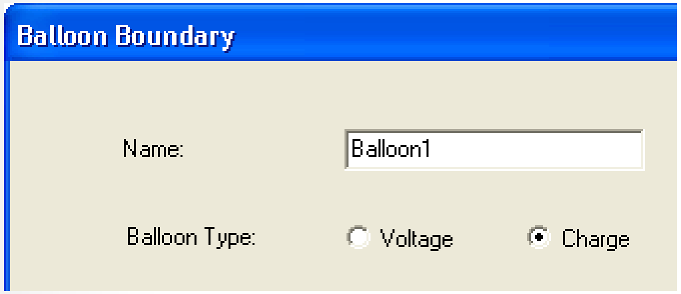
- Enable users to model only part of a structure, which reduces the size or complexity of design, thereby shortening the solution time.
- Applied to external boundaries of domain.

Boundary Types(2D)

Balloon:

Balloon boundary can be of two types

- **Voltage**: Models the case where the voltage at infinity is zero
- **Charge:** Models the case where the charge at infinity matches the charge in solution region forcing net charge to be zero



d. Excitations

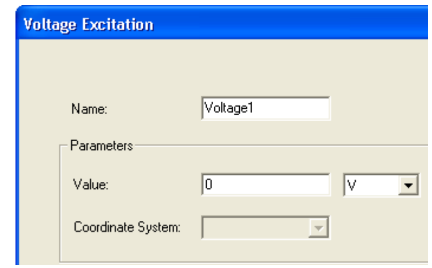
Assigning Excitations

– Excitations can be assigned from the menu item ***Maxwell 2D/3D → Excitations → Assign***

Excitation

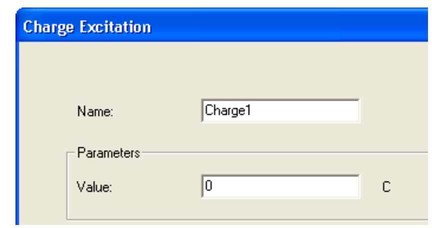
Voltage:

- Assigns DC voltage on selected entity
- Can be assigned to an Object or a Face (Edge in 2D) of an Object



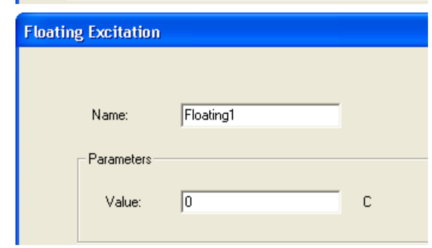
Charge:

- Assigns total Charge on selected entity
- Can be assigned to an Object or a Face (Edge in 2D) of an Object



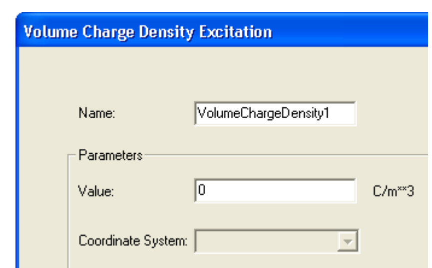
Floating:

- Used to model conductors of unknown potential
- Can be assigned to an Object or a Face (Edge in 2D) of an Object



Charge Density:

- Assigns Charge Density on Selected object
- In 2D, charge density can be surface charge density (assigned to object) or line charge density (assigned to edge)
- In 3D, only volume Charge density can be assigned



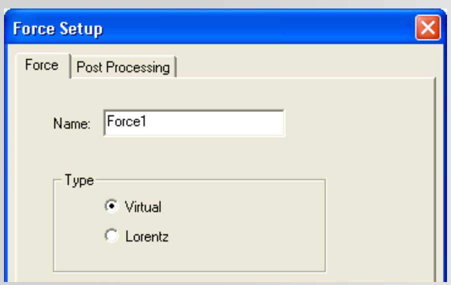
e. Parameters

Parameters

- Three calculation parameters can be assigned for electrostatic solver (Force, Torque and Capacitance Matrix)
- A parameter cane added by selecting the required object and selecting menu item ***Maxwell 3D/2D → Parameters → Assign***

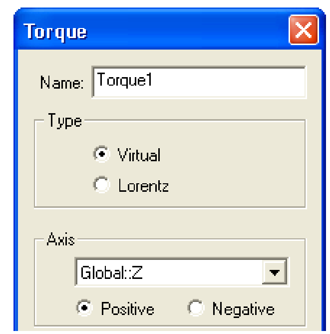
Force:

- Calculates force acting on assigned object
- Force can be Virtual or Lorentz



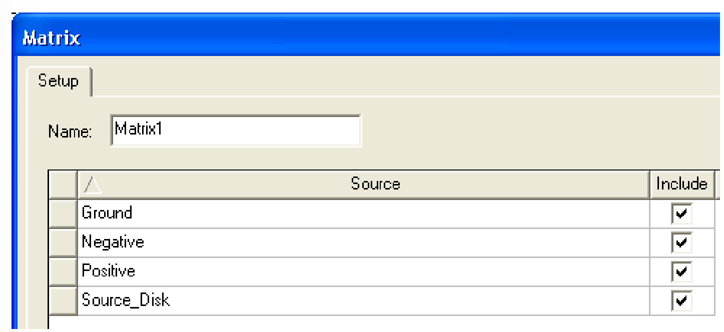
Torque:

- Calculates torque on assigned object
- Torque can be Virtual or Lorentz



Matrix:

- Calculates Capacitance Matrix for selected excitations
- Capacitance matrix values can be seen in the Solution Data after completion of solution process



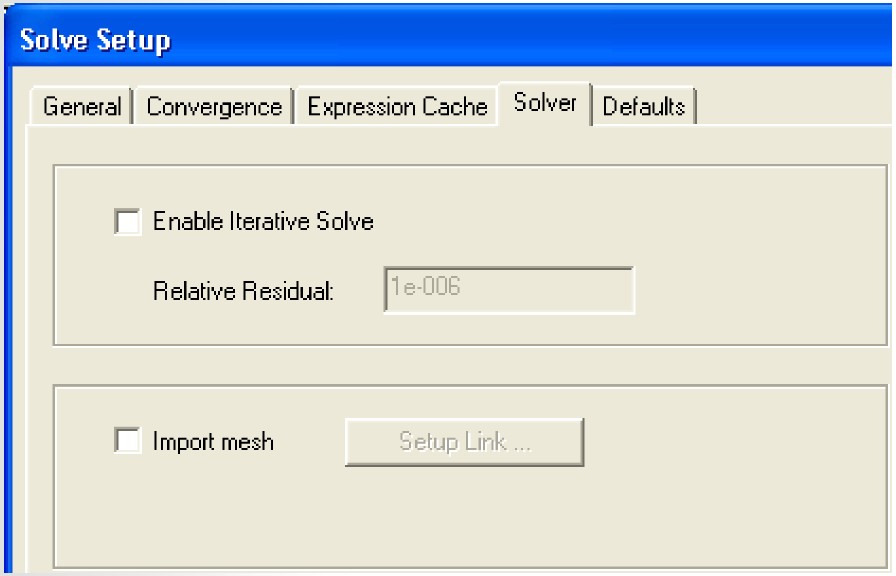
f. Analysis Setup

Solution Setup

- A Solution Setup can be added from the menu item ***Maxwell 3D/2D → Analysis Setup → Add Solution Setup***
- Options on General and Convergence tab of Solve Setup window are same as option with Magnetostatic solver

Solver Tab

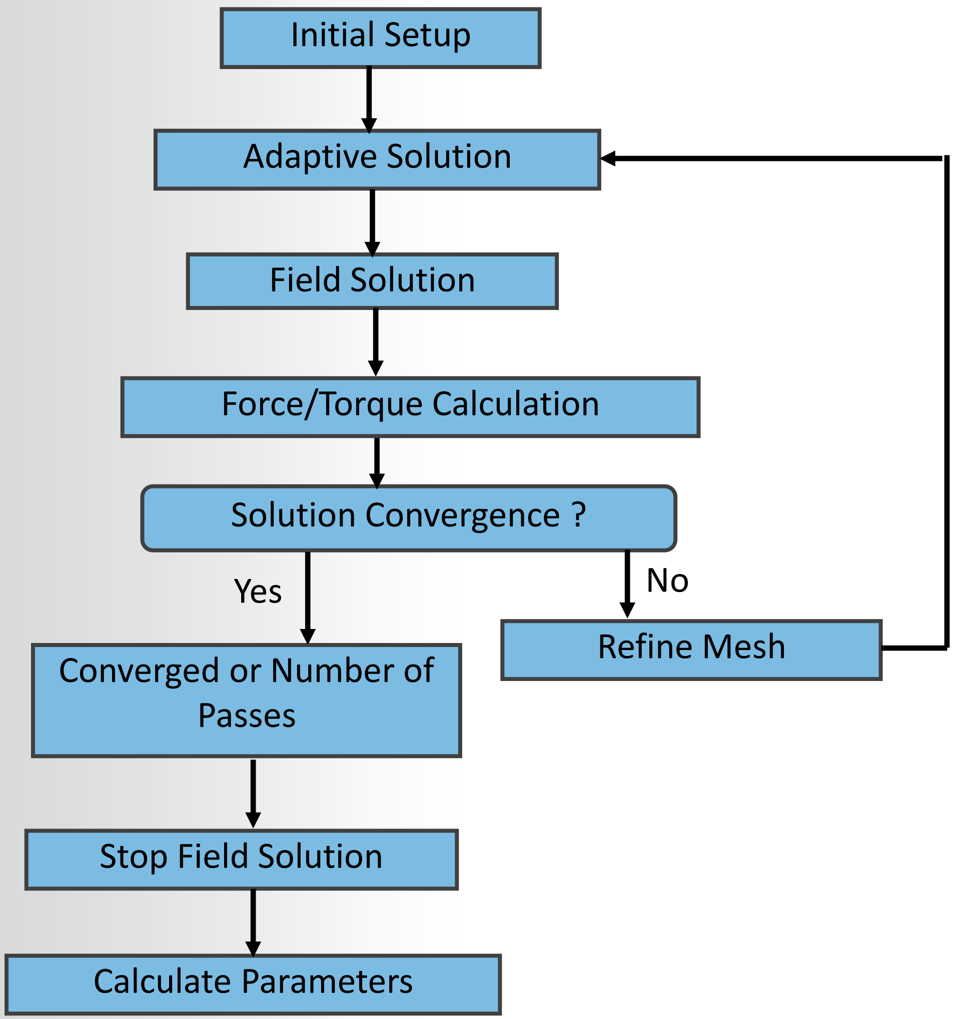
- **Enable Iterative Solve:** Enables ICCG solvers (Direct is the default).
- **Import Mesh:** Allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **Import Mesh** to define the Project and Design from where the mesh will be imported



g. Solution Process

Electrostatic Solution Process

– A Solution process can be launched form the menu item ***Maxwell 3D/2D → Analyze All***



B. DC Conduction Solver

DC Conduction Solver

- The DC Conduction solver solves for the DC Currents resulting in a conductor
- The quantity solved is the electric scalar potential (ø)
- Current density (J) and Electric Field (E) are automatically calculated from the electric scalar potential (ø)
- All fields outside of the conductors are not calculated and totally decoupled from the electric field distribution in the conductors – permittivity is irrelevant in this calculation.
- The DC Conduction solver can be coupled with Electrostatic solver to solve for electric field in Insulators

DC Conduction Equations

- Following equations are solved with DC Conduction solver

$$\nabla \bullet (\sigma \nabla \Phi) = 0$$

Maxwell 3D

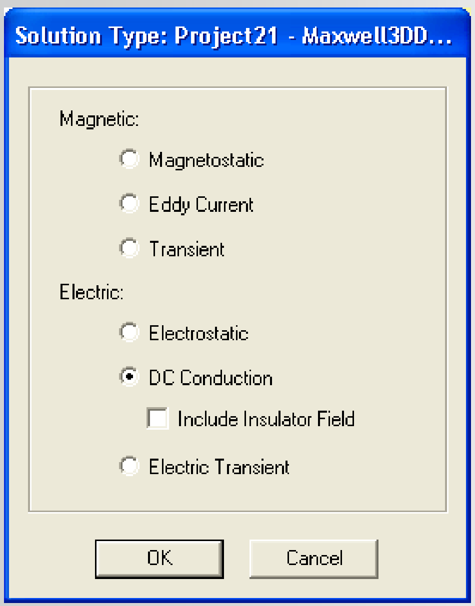
$$J(x,y) = \sigma E(x,y) = -\sigma \nabla \phi(x,y)$$

Maxwell 2D

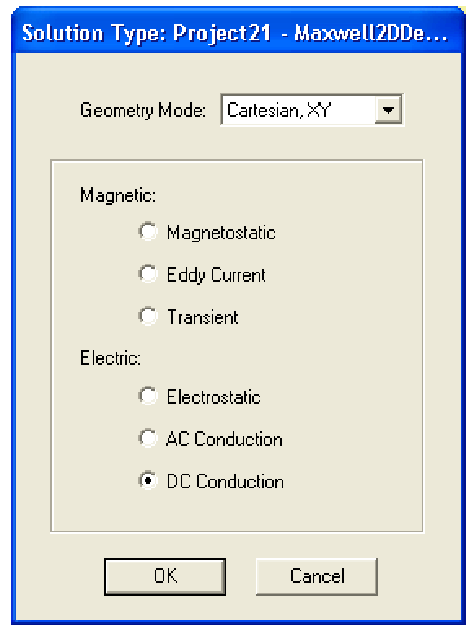
a. Selecting the DC Conduction Problem

Selecting the DC Conduction Solver

- Specify the DC Conduction Solver by selecting the menu item ***Maxwell 2D/3D → Solution Type***
- In Solution Type window, select **Electric > DC Conduction** and press **OK**
- Enabling the option **“Include Insulator Field”** will couple the DC Conduction solver with the Electrostatic solver



Maxwell 3D

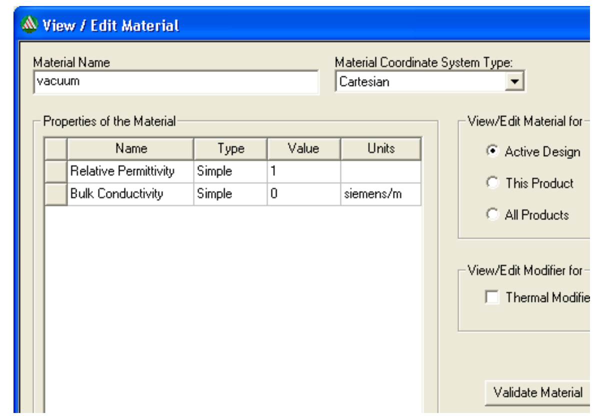


Maxwell 2D

b. DC Conduction Setup

Material Properties

- Material properties required for the DC conduction solver are the same as the Electrostatic solver
- Relative permittivity will not affect the DC conduction results but will be required when insulator fields are included

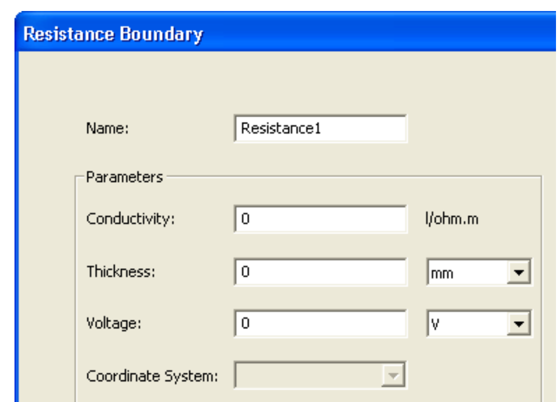


Boundary Conditions

- All the boundary conditions that are available with the Electrostatic solver are also applicable for the DC conduction solver
- In 2D, a Resistance Boundary can also be defined

Resistance Boundary (2D):

- Models a very thin layer of resistive material on a conductor at know potential
- Can be assigned only to the boundary edges of Solution space



c. Excitations

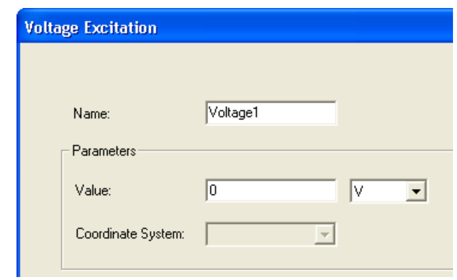
Assigning Excitations

– Excitations can be assigned from the menu item ***Maxwell 2D/3D → Excitations → Assign***

Excitation

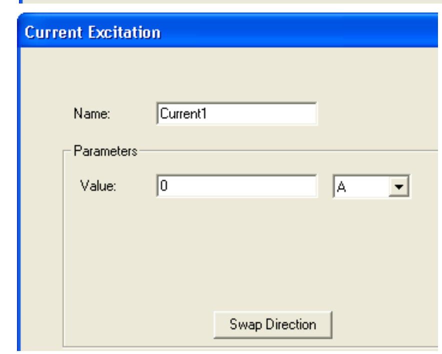
Voltage (2D &3D):

- Assigns DC voltage on the selected entity
- Can be assigned to an Object or a Face (Edge in 2D) of an Object
- Can be defined along with a current excitation to define a voltage reference for the electric field solution



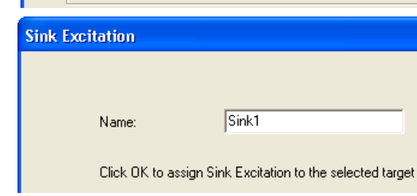
Current (3D Only):

- Assigns total current through the cross section of conductor
- Can be assigned to a Face of an Object
- Sink should be defined along with Current Excitation



Sink (3D Only) :

- Required to be defined along with Current Excitation
- Ensures the total current flowing "into" and "out of" the model is exactly zero



Note: When Insulator Field is included, all excitations discussed in Electrostatic solver are available with DC conduction solver as well

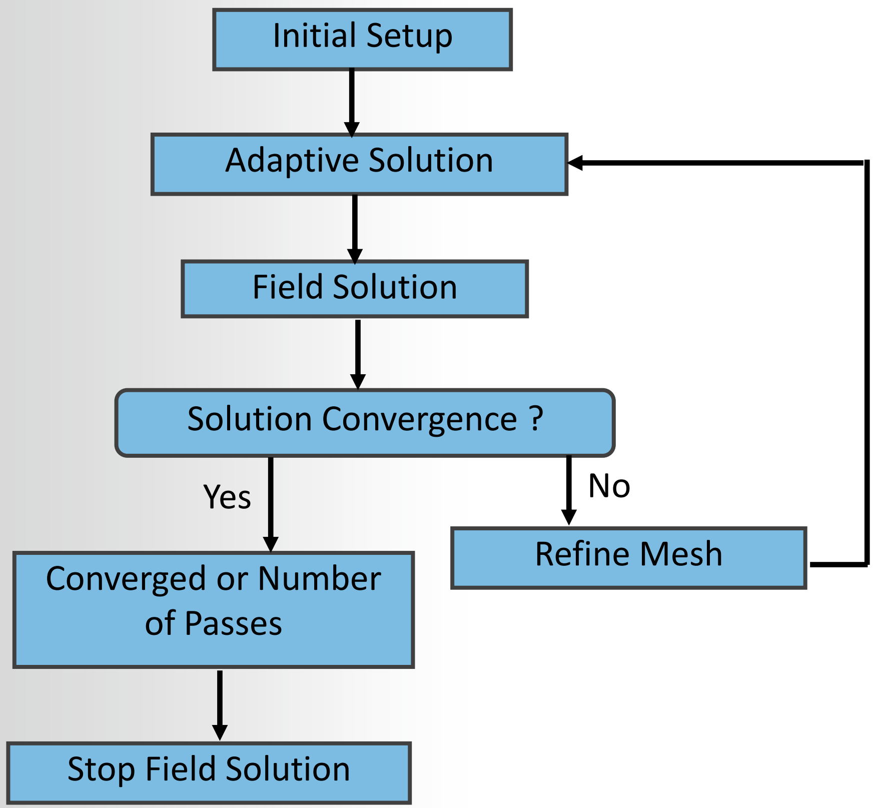
d. Solution Process

Analysis Setup

– All the options in the Solve Setup window are the same as in the Electrostatic Solver

Electrostatic Solution Process

– A Solution process can be launched form the menu item ***Maxwell 3D/2D → Analyze All***



C. AC Conduction Solver

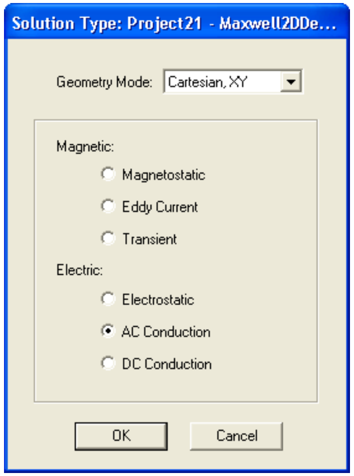
AC Conduction Solver

- The AC Conduction Solver simulates conduction currents due to time-varying periodic electric fields in conductors and dielectrics
- This is a frequency domain solver and assumes all sources are sinusoids oscillating at the same frequency
- This Solver is available only with the Maxwell 2D design type
- Following equation is solved with AC conduction solver

$$\nabla \bullet \left(\sigma E + j\omega\varepsilon\nabla\phi(x,y)\right) = 0$$

Selecting the AC Conduction Solver

- Specify the AC Conduction Solver by selecting the menu item ***Maxwell 2D → Solution Type***
- In the Solution type window, select **Electric > AC Conduction** and press **OK**



ANSYS

a. AC Conduction Setup

Material Properties

– Material properties required for AC conduction solver are same as the Electrostatic Solver. Please refer Material Definition selection for the Electrostatic Solver

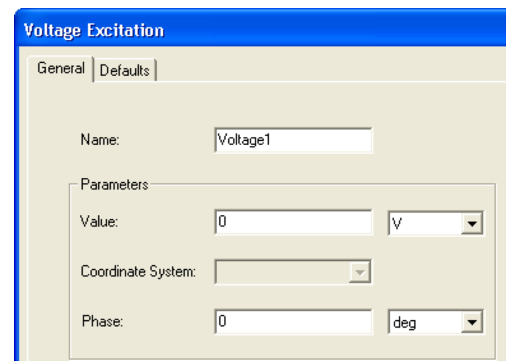
Boundary Conditions

– AC Conduction Solver offers 3 type of boundary setups (Master/Slave, Symmetry and Balloon) which are same as Electrostatic Solver

Excitations

Voltage:

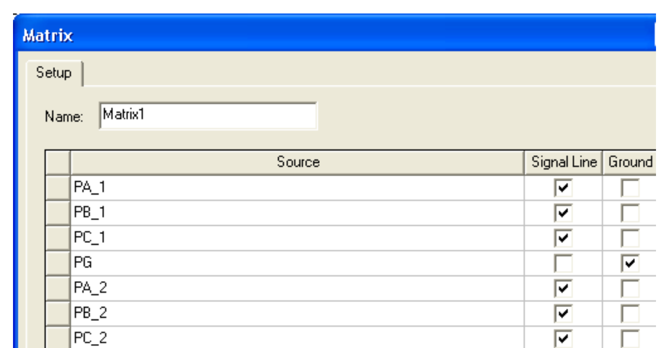
- Assigns AC voltage on selected entity using Magnitude and Phase
- Can be assigned to an Object or Edge of an object



Parameters

Matrix:

- Calculates Admittance and Capacitance, Conductance matrix in per meter values
- Matrix values can be seen in the Solution Data after completion of solution process

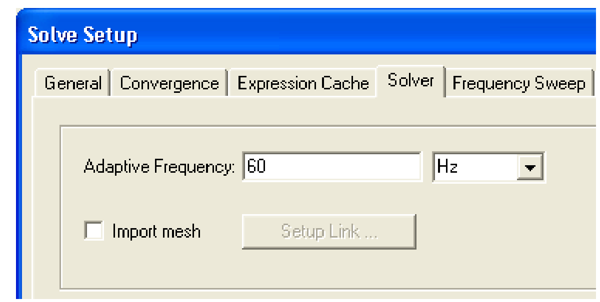


b. Analysis Setup

Solution Setup

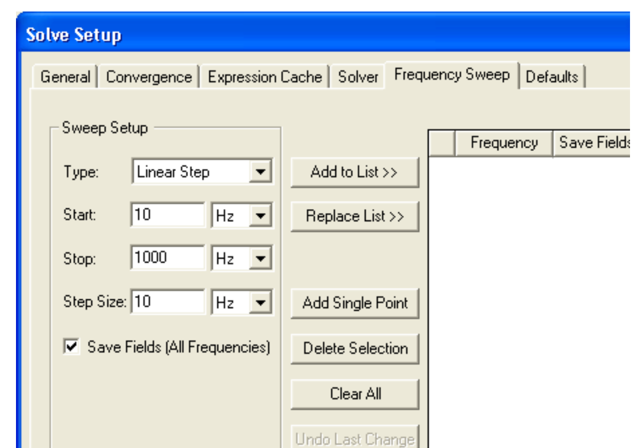
- A Solution Setup can be added from the menu item ***Maxwell 2D → Analysis Setup → Add Solution Setup***
- Options on General and Convergence tab of the Solve Setup window are same as option with the Electrostatic solver. Please refer to Analysis Setup section for the Electrostatic Solver

Solver Tab



- **Adaptive Frequency:** Defines the frequency at which the mesh is constructed and adapted, and at which solution is obtained and used for subsequent frequency sweeps
- **Import Mesh:** Allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation

Frequency Sweep Tab



- **Sweep Setup** (Type, Start, Stop, Step): Enables to define frequency sweep range and points
- **Save Fields:** Saves the fields for defined frequency Sweep
- **Add to List >>:** Places sweep definition in the Sweep List (the Sweep List is displayed in the right panel).
- Edit any entries in the Sweep List to adjust solution frequencies and click **Save Fields** to save field at frequencies in the list