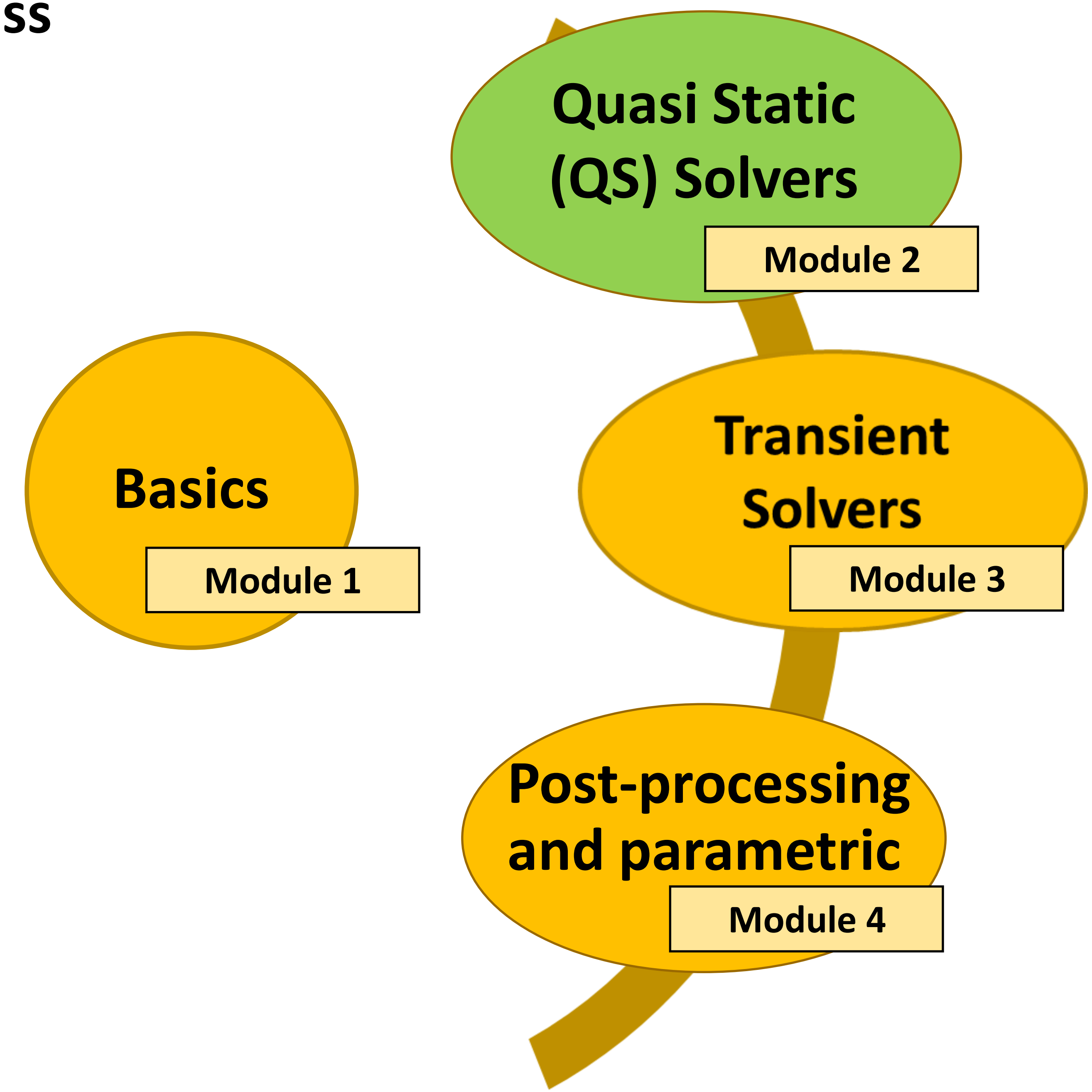
Module 02: Quasitatic Solvers

Release 2020R2

Overview

- Eddy current solver
- Electrostatic solver
- AC Conduction and DC conduction solvers
- Workshop 2.1: 2D Eddy Current Analysis
- Workshop 2.2: 2D Electrostatic Analysis
- Workshop 2.3: 3D Eddy Current Analysis
- Workshop 2.4: 3D Electrostatic Analysis

Overall Process



Eddy Current Solver

Ansys

Eddy Current Solver

- **Eddy Current Solver**
  - Eddy current solver computes steady-state, sinusoidal (AC) magnetic fields at a given frequency
  - The frequency domain solution assumes frequency to be same throughout the domain
  - 3D Eddy Current Solver solves for partial displacement currents
  - The AC source can be peak value of sinusoidal AC currents and voltages, time-varying AC external magnetic fields represented by external boundary conditions as well as an external circuit
  - Eddy Current solver utilizes the adaptive mesh refinement technique
- **Eddy Current Equations**
  - Following equations are solved with Eddy Current solver

$$\nabla \times \left(\frac{1}{\sigma + j\omega\varepsilon} \cdot (\nabla \times \bar{\boldsymbol{H}}) \right) = j\omega\mu_0\bar{\mu}_r\bar{\boldsymbol{H}}$$

Maxwell 3D

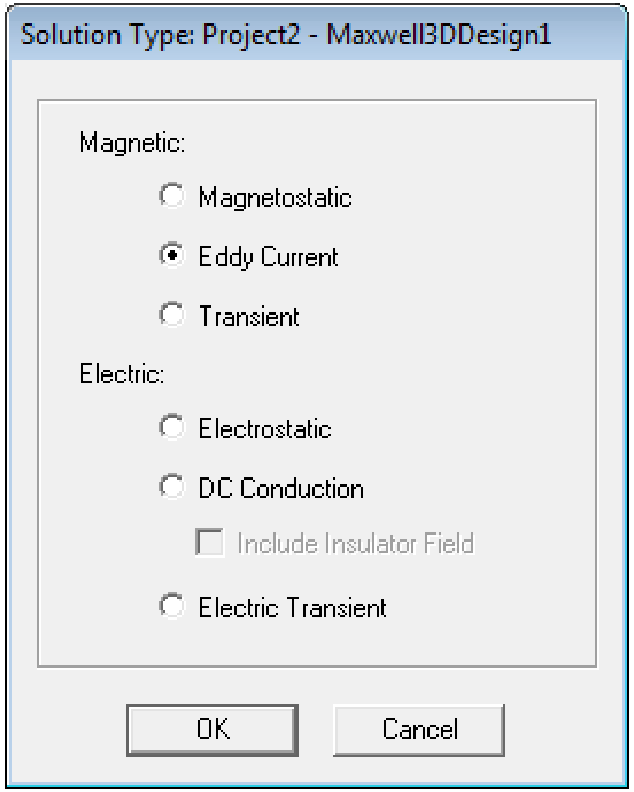
$$\nabla \times \left(\frac{1}{\mu_0\bar{\mu}_r} \cdot \left(\nabla \times \bar{\boldsymbol{A}}_z(x,y)\right) \right) = (\sigma + j\omega\varepsilon)(-j\omega\bar{\boldsymbol{A}}_z(x,y) - \nabla\Phi)$$

$$\nabla \times \left(\frac{1}{\mu_0\bar{\mu}_r} \cdot \left(\nabla \times \bar{\boldsymbol{A}}_\varphi(r,z)\right) \right) = (\sigma + j\omega\varepsilon)\left(-j\omega\bar{\boldsymbol{A}}_\varphi(r,z) - \nabla\Phi\right)$$

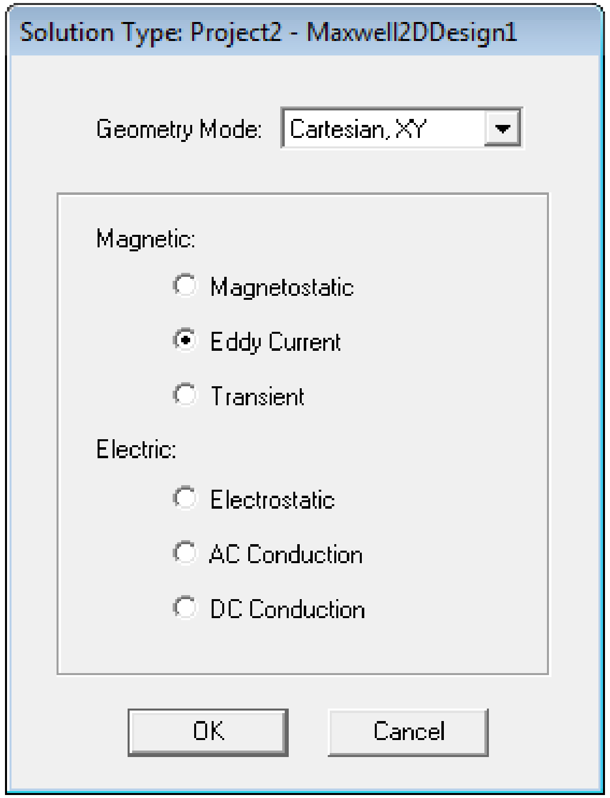
Maxwell 2D

Selecting the Eddy Current Solver

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



Maxwell 3D

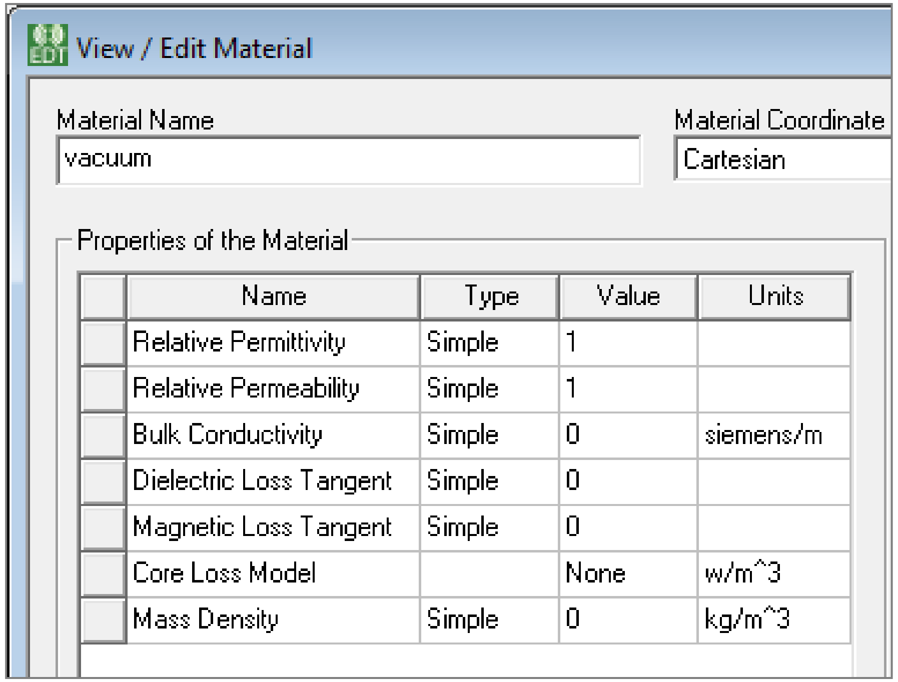


Maxwell 2D

Ansys

Material Definitions

- Eddy Current Material Properties
  - **Relative Permittivity:**
    - Relative Permittivity mostly affects calculation of dielectric losses when dielectric loss tangent is set. Simple or Anisotropic
  - **Relative Permeability :**
    - Relative Permeability can be Simple or Anisotropic. Nonlinear permeability is supported by obtaining a linearized permeability from non-linear B-H curve. Solution is still assumed to be sinusoidal
  - **Bulk Conductivity:**
    - Used in determining the current distribution and eddy currents in conductor materials. Simple or Anisotropic
  - **Dielectric Loss Tangent:**
    - Defines the ratio of imaginary and real permittivities. Simple or Anisotropic
  - **Magnetic Loss Tangent:**
    - Defines the ratio of imaginary and real permeabilities. Simple or Anisotropic



Name	Type	Value	Units
Relative Permittivity	Simple	1	
Relative Permeability	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
Dielectric Loss Tangent	Simple	0	
Magnetic Loss Tangent	Simple	0	
Core Loss Model		None	w/m^3
Mass Density	Simple	0	kg/m^3

Ansys

Material Definitions

- **Eddy Current Material Properties**
  - **Core Loss Type**
    - **Core Loss Type can be either Electrical Steel or Power ferrite**
    - **Core Loss Coefficients will change according to selected Core Loss type**

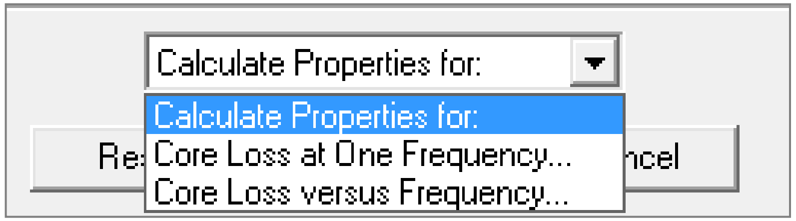
Core Loss Type		Electrical Steel	w/m^3
- Kh	Simple	0	
- Kc	Simple	0	
- Ke	Simple	0	
Mass Density	Simple	8055	kg/m^3

$$p_v = K_h f(B_m^2) + K_c(fB_m)^2 + K_e(fB_m)^{1.5}$$

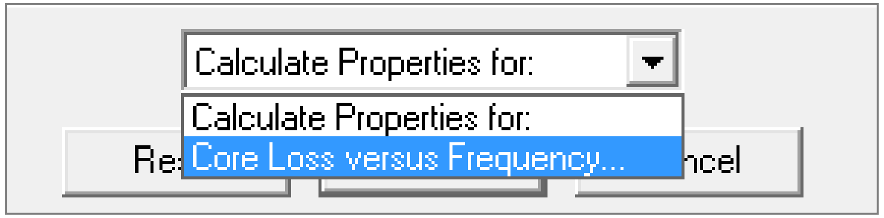
Core Loss Type		Power Ferrite	w/m^3
- Cm	Simple	0	
- X	Simple	0	
- Y	Simple	0	
Mass Density	Simple	8055	kg/m^3

$$p_v = C_m f^x B_m^y$$

  - **Core Loss Coefficient Calculations**
    - **Maxwell provides tools to evaluate core loss coefficients based on core loss data provided by users**
    - **Users can select tab at the bottom of View/Edit Material window and select the option “Calculate Properties for”**



For Electrical Steel



For Power Ferrites

Material Definitions

- **Eddy Current Material Properties**
  - **Core Loss at One Frequency:**
    - **The option Calculate Properties for "Core Loss at One Frequency" enables to input B-P Curve for a defined frequency. This option is available only for Electrical Steel**
    - **Using specified B-P Curve, K_1 and K_2 coefficients are obtained by minimizing quadratic form**

$$err(K_1, K_2) = \sum_i \left[P_{vi} - \left(K_1 B_{mi}^2 + K_2 B_{mi}^{1.5}\right)\right]^2 = \min$$

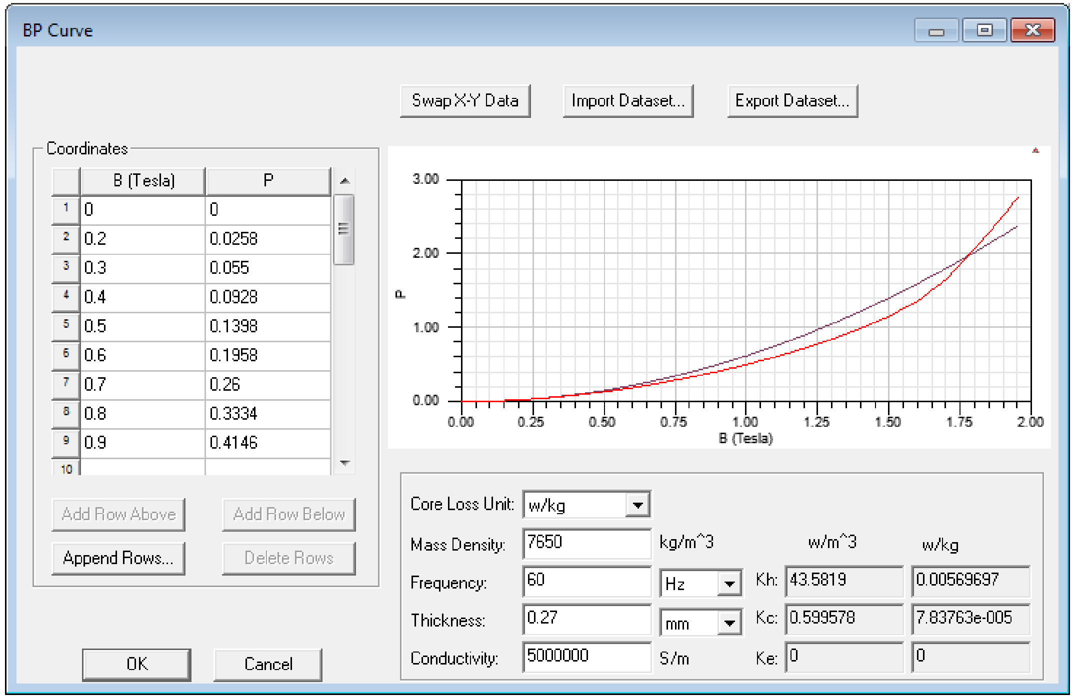
Where *i* represents each point of defined B-P Curve

- **Eddy Current coefficient, K_c is evaluated as**

$$K_c = \pi^2 \sigma \frac{d^2}{6}$$

- **Where σ is the conductivity and d is the thickness of one lamination sheet**
- **Hysteresis Loss coefficient (K_h) and Excessive Loss Coefficient (K_e) are evaluated from K_1, K_2 and K_c**

$$K_h = \frac{K_1 - K_c f_0^2}{f_0} \qquad K_e = \frac{K_2}{f_0^{1.5}}$$



Ansys

Material Definitions

- **Eddy Current Material Properties**
  - **Core Loss versus Frequency:**
    - **Selecting the option Calculate Properties for "Core Loss versus Frequency" enables to input B-P Curve for multiple frequency values**
    - **For Electrical Steel *Kh, Kc* and *Ke* are obtained by minimizing quadratic form**

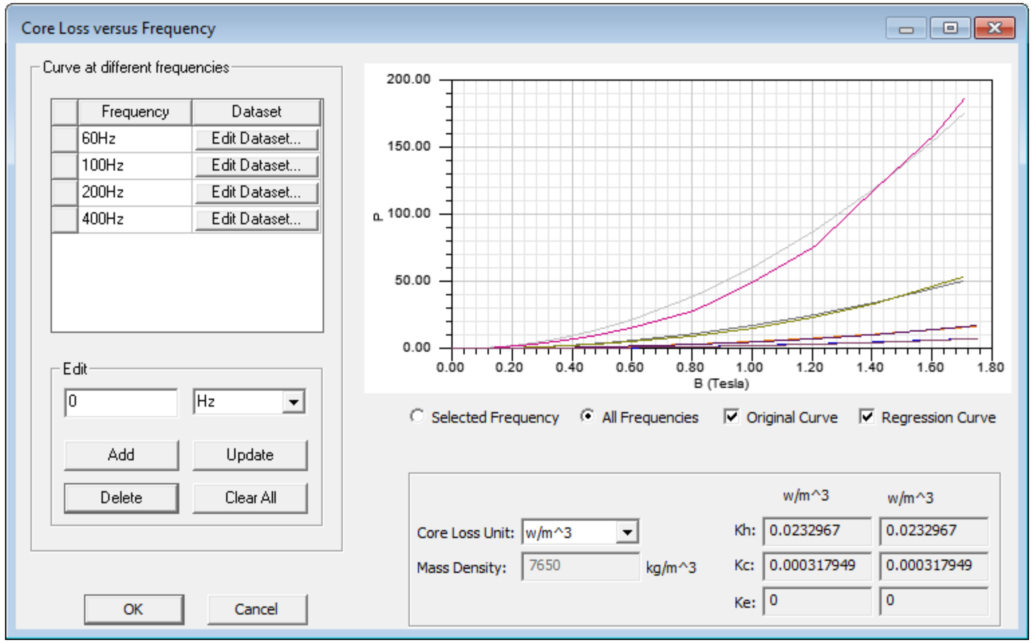
$$err(K_h, K_c, K_e) = \sum_{i=1}^{m}\sum_{j=1}^{n_i}\left[P_{vij} - \left(K_h f_i B_{mij}^2 + K_c f_i^2 B_{mij}^2 + K_e f_i^{1.5} B_{mij}^{1.5}\right)\right]^2 = \min$$

Where, *m* is number of curves added and n_i is number of points defined in i^{th} curve

- **For Power Ferrites *Cm, x* and *y* are obtained by minimizing quadratic form**

$$err(c, x, y) = \sum_{i=1}^{m}\sum_{j=1}^{n_i}\left[\log(P_{vij}) - \left(c + x\log(f_i) + y\log(B_{mij})\right)\right]^2 = \min$$

$$c = \log(C_m)$$

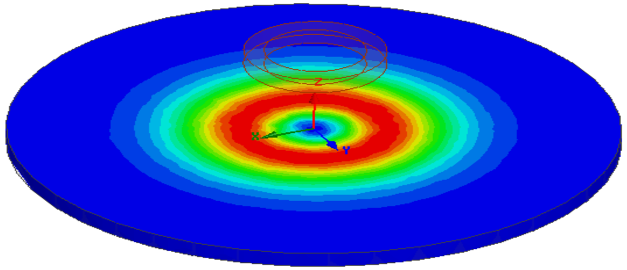


Boundary Conditions

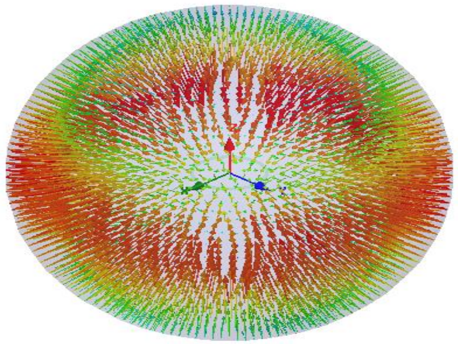
- Boundary Types
  - All BCs present in Magnetostatic are also valid for Eddy Current Solver
  - In addition, two further boundaries can be defined

  - **Impedance Boundary (2D & 3D):**
    - Allows to simulate eddy effects without explicitly solving and meshing for the skin depth
    - Equivalent calculations are done on surface elements of the conductor without any computation for internal conductor region
    - Recommended to be used when skin depth is two orders of magnitude smaller than conductor thickness

  - **Radiation Boundary (3D):**
    - To simulate fields radiating infinitely far into space
    - Only used for antenna simulations with Displacement Current calculation on the Region
    - The system absorbs the field at the radiation boundary, essentially ballooning the boundary infinitely far away from the source



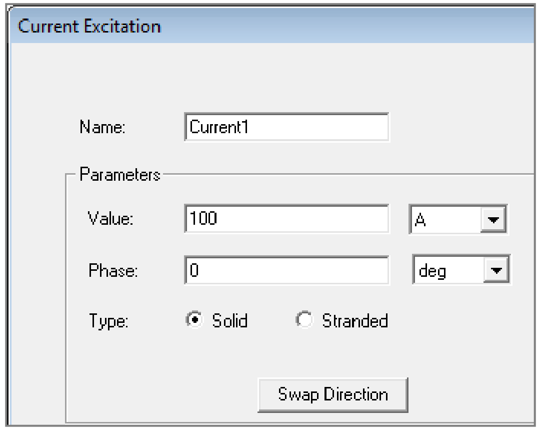
Surface Loss Density on Impedance Boundary

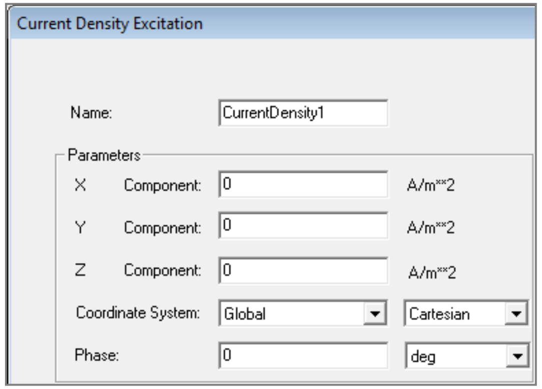


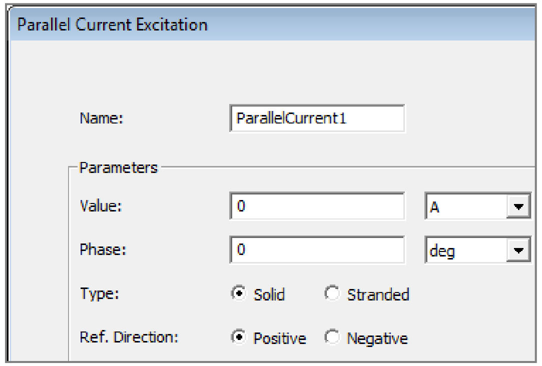
Poynting Vectors on Radiation Boundary

Excitations

- Excitations
  - Eddy Current solver allows four types of excitations:
  - Current
    - Defines total peak current (Amp-turns) and phase
    - Can be assigned to the conductor faces lying on simulation domain boundaries or conductor sections with a closed conduction path.
    - Conductor can be defined as Solid or Stranded
    - Eddy effects are not computed for stranded conductors
  - Current Density
    - Current density and phase throughout a conductor, defined using X,Y and Z components of selected CS
    - In 3D should be accompanied with Current Density Terminal definition
  - Parallel Current (2D only)
    - Used to define total AC current in a parallel conduction path consisting of at least two conductors
    - For Solid conductors, current split includes also eddy effects
    - For Stranded conductors, current split is based on relative areas of selected conductors







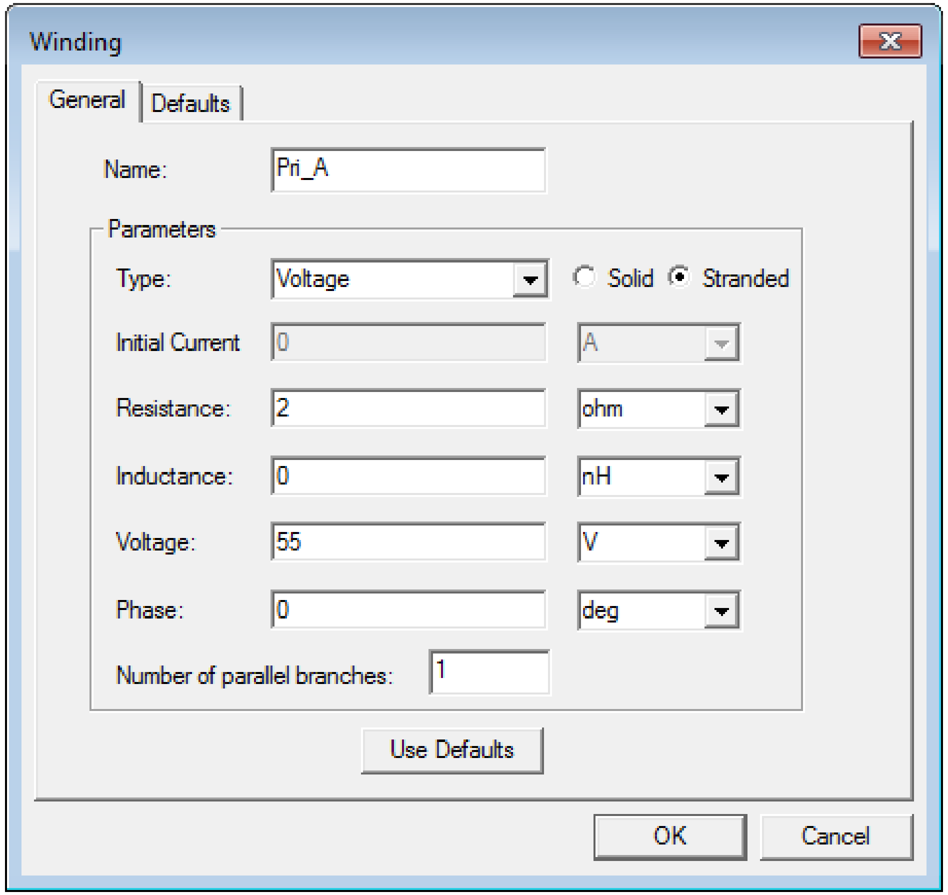
Ansys

Excitations

- **Winding**
  - Winding definitions determine how the coils are connected together and the type of source (Voltage, Current, External Circuit)
  - Winding can be added from menu item **Maxwell 2D/3D → Excitations → Add Winding**
  - A winding can be Solid or Stranded: Stranded winding does not compute Eddy Current in conduction path
  - All the coils belonging to one winding are in-series connected
  - Maxwell allows users to define three types of windings

- **Current Type**
  - Defines specified current through the conduction path

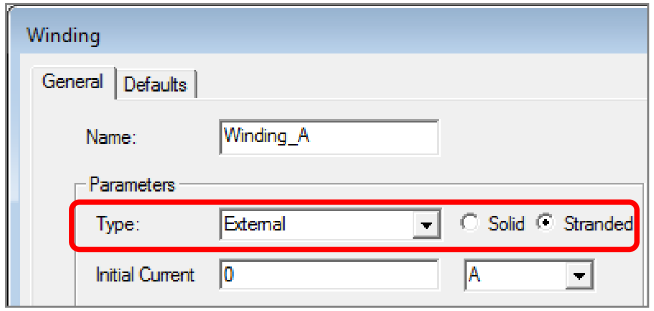
- **Voltage Type**
  - Defines Specified voltage across the coil terminals
  - Voltage definition along with Resistance and Inductance definition is used to evaluate current flowing through the winding

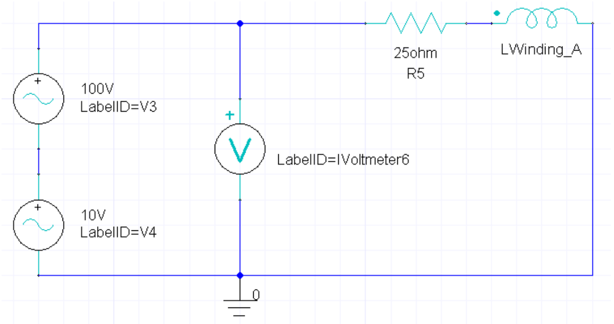


Ansys

Excitations

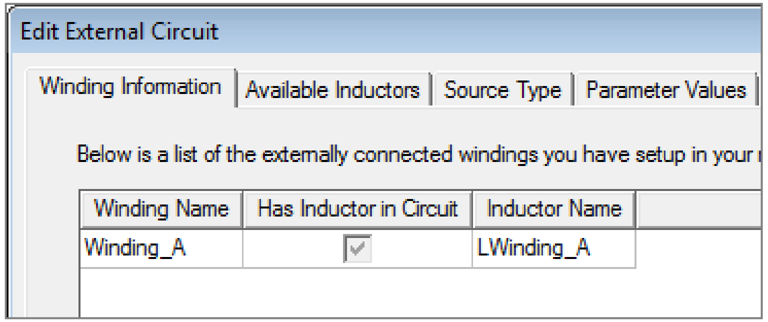
- **External**
  - External winding enables to assign excitations from either the Maxwell Circuit Editor, or Simplorer (see ***Maxwell 2D/3D → Design Settings***)
  - Select menu ***Maxwell 2D/3D → Excitations → External Circuit → Edit External Circuit***, click on Edit Circuit to launch the Maxwell Circuit Editor
  - Name of the winding representing Conduction path in circuit schematic should be same as Name defined in Maxwell





Maxwell Circuit Editor

Export Netlist



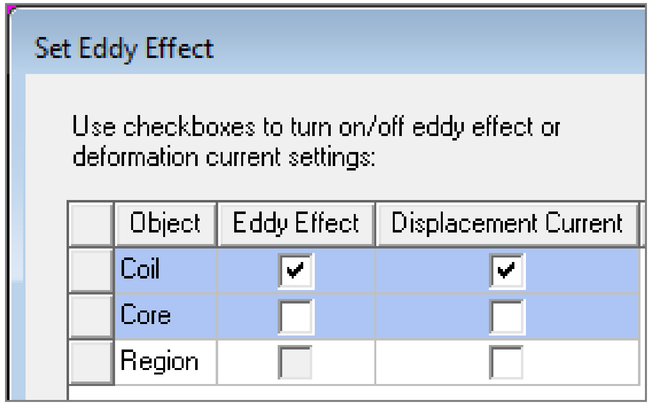
Import Netlist from *Maxwell 3D/2D → External Circuit → Edit External Circuit*

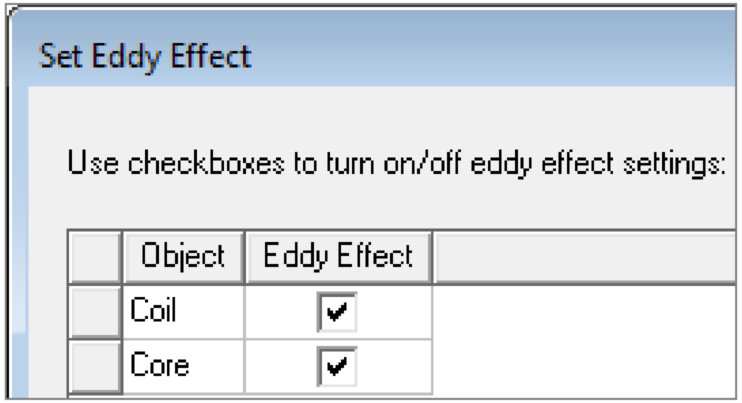
- **Add Coil Terminals to Winding**
  - To completely define the excitations, Coil terminals are added to the Winding definition
  - Right Click on Windings in Project Manager window and select ***Assign Coil Terminals***

Excitations

- **Setting Eddy Effects (Calculating Eddy Currents)**
  - **Eddy Effects can be set from Project Manager *RMB on Excitations → Set Eddy Effects***
  - **Induced eddy (2D/3D) and displacement (only 3D) current calculations can be enabled or disabled for an object**

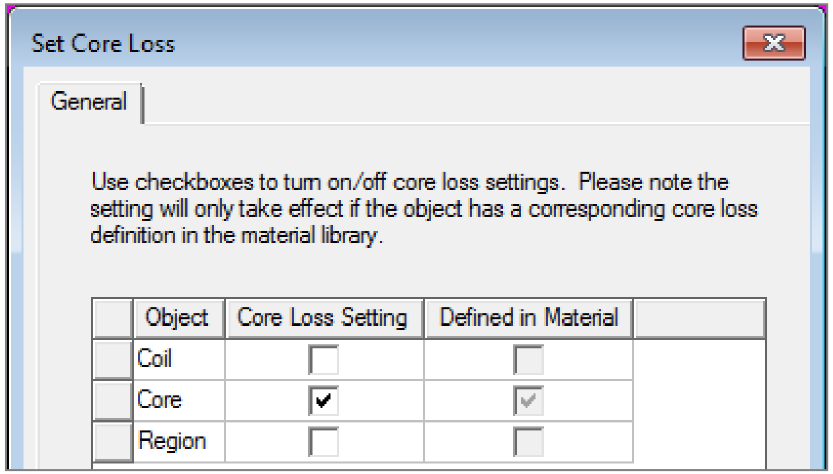
Maxwell 3D





Maxwell 2D

- **Setting Core Loss (only 3D)**
  - **Core Loss calculations can be set from Project Manager *RMB on Excitations → Set Core Loss***
  - **If Core Loss is enabled for an object, Eddy effects should be disabled for that object since the Core loss calculation already includes Eddy losses**
  - **Core Loss properties must be defined for the material as discussed earlier**

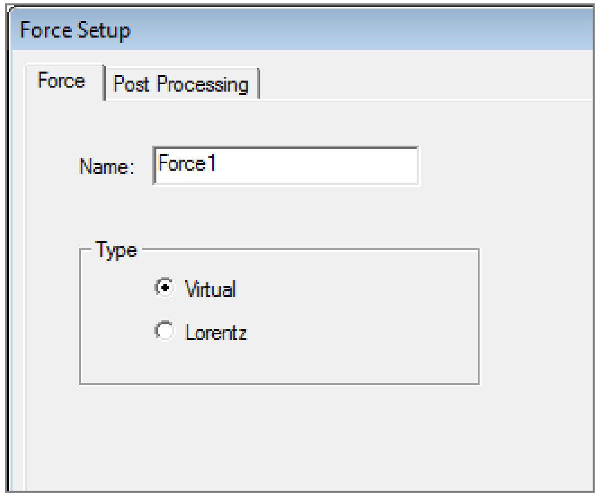


Ansys

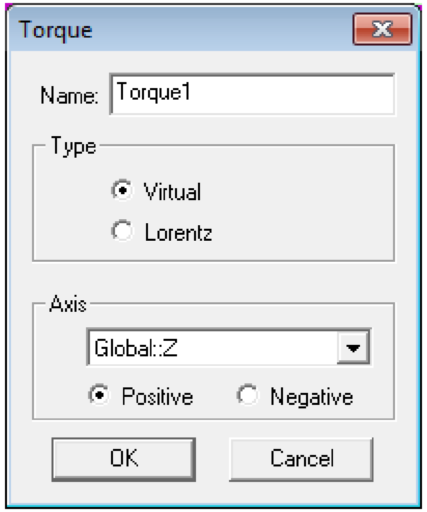
Parameters

- **Parameters**
  - **All the parameters available in Magnetostatic Solver are also available in Eddy Current Solver**
  - **Parameters can be added through menu item *Maxwell 3D/2D → Parameters → Assign***

– **Force:**
- **Calculates force acting on selected objects**
- **Can be Virtual or Lorentz**
- **Lorentz can not be used for magnetic materials**

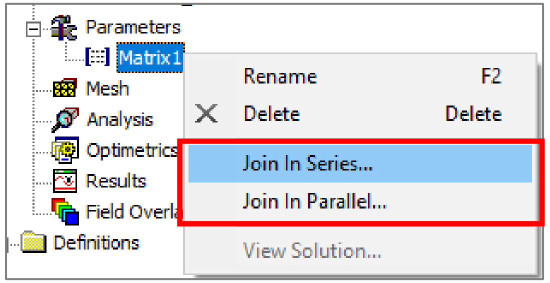


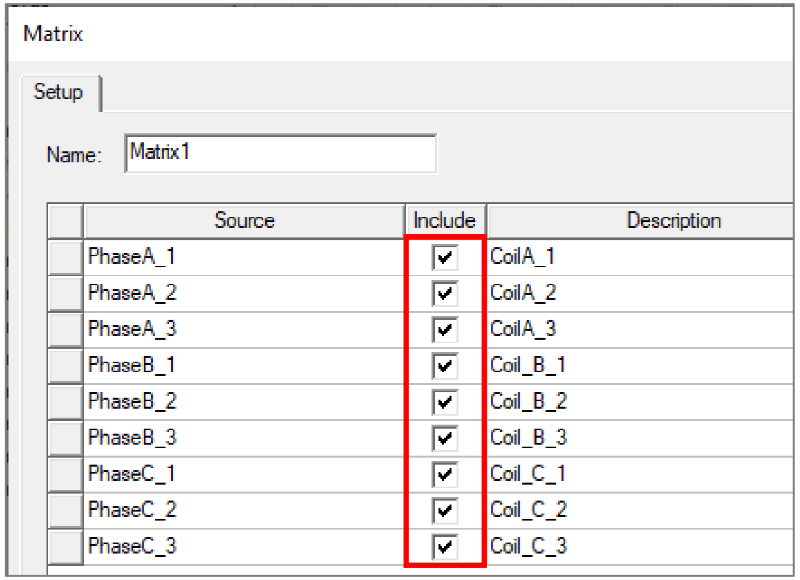
– **Torque:**
- **Calculates torque on selected objects**
- **Can be Virtual or Lorentz**
- **The torque acting axis and direction must be set**

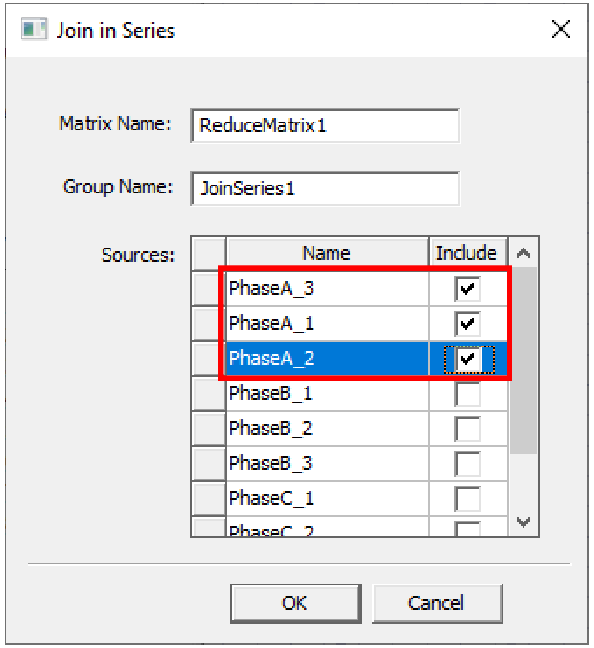


Parameters

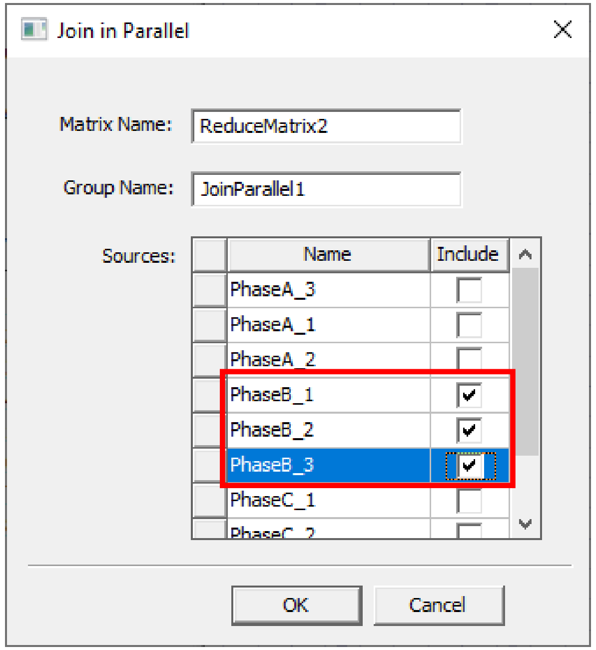
- **Matrix:**
  - **Calculates Inductance and Resistance matrix**
  - **Once assigned by using the Include checkboxes, Matrix allows additional connections, both “in Series” and “in Parallel” for further post-processing**



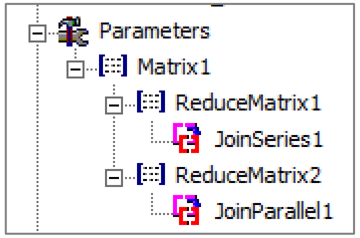




Join in Series example



Join in Parallel example

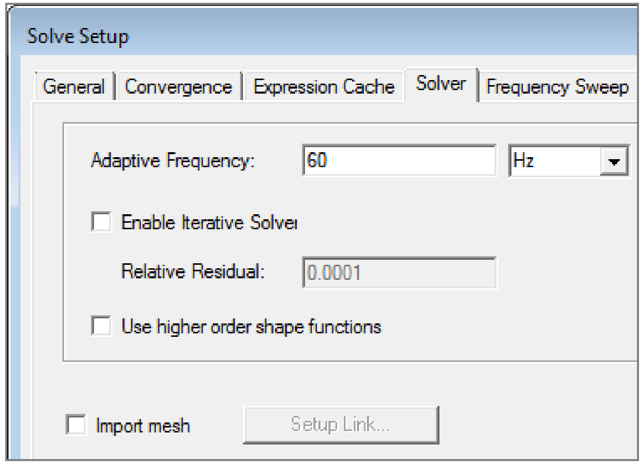


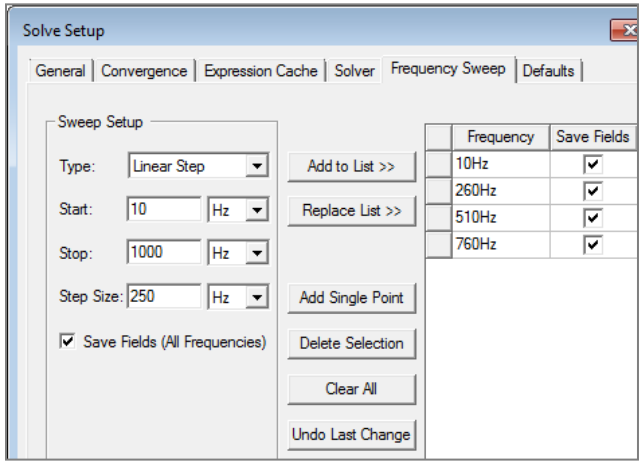
Final configuration

Ansys

Analysis Setup

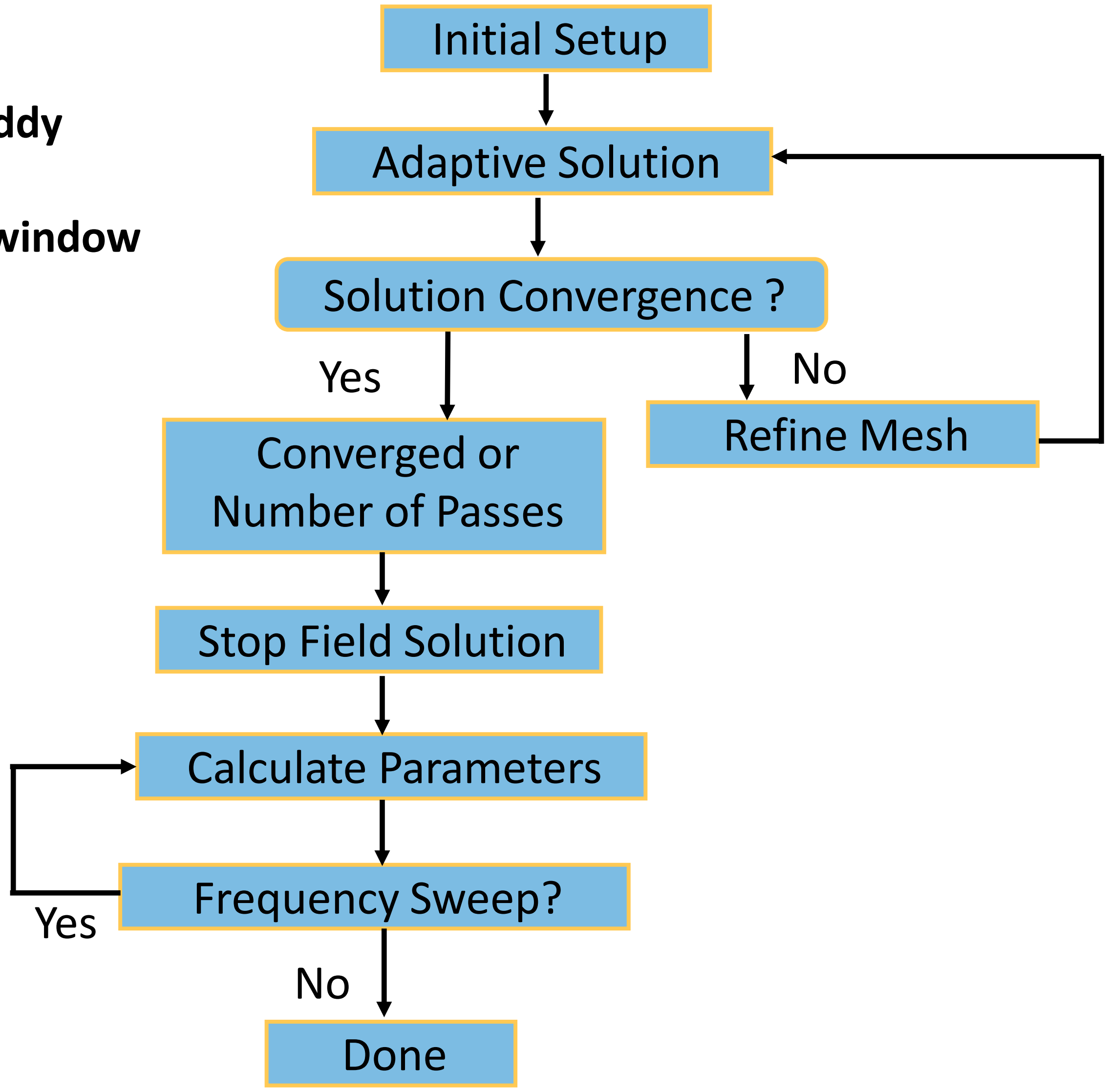
- Solution Setup
  - A Solution Setup can be added from Project Manager ***RMB on Analysis → Add Solution Setup***
  - Options on the **General** and **Convergence** tab are the same as for Magnetostatic solver
  - **Solver Tab**
    - Adaptive Frequency: sets the frequency at which the mesh is constructed and adapted, and at which solution is obtained
    - Enable Iterative Solve (only for 3D designs): Enables ICCG solvers (Direct is the default)
    - Use higher order shape functions (only for 3D designs): enables higher order option to gain better accuracy
    - Import Mesh: initial mesh is imported from another solution – that solution must have the exact same geometry as the current one
  - **Frequency Sweep Tab**
    - Sweep Setup (Type, Start, Stop, Step): Enables to define frequency sweep range and values
    - Save Fields: Saves the fields for defined swept frequencies
    - Add to List >>: activate frequency sweep





Solution Process

- **Eddy Current Solution Process**
  - **Like the Magnetostatic Solver, the solution process in the Eddy Current solver is automated as shown in diagram**
  - **A Solution process can be launched from Project Manager window** ***RMB on Setup1 → Analyze***



Electrostatic Solver

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 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 the automatic adaptive mesh refinement technique
- Electrostatic Equations
  - Following equations are solved with Electrostatic solver

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

Maxwell 3D

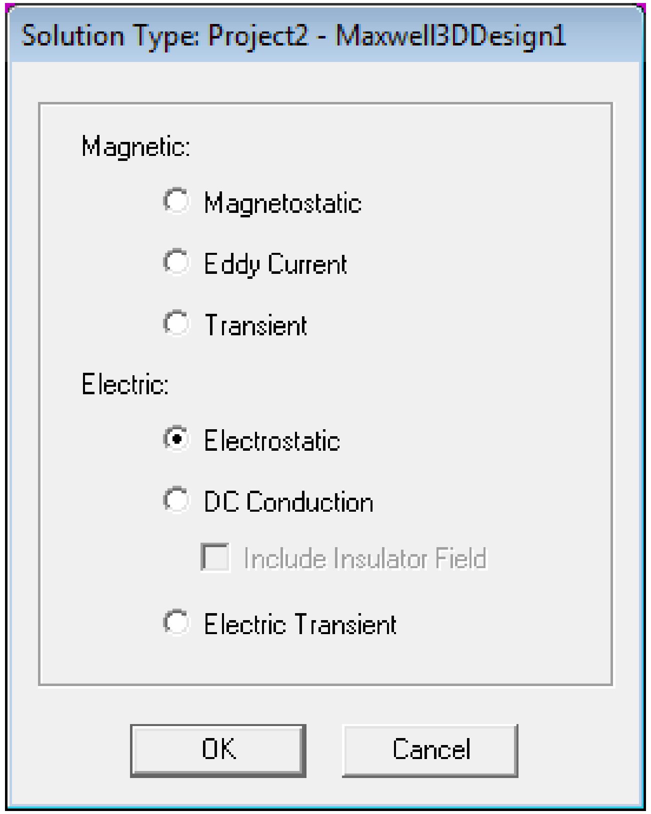
$$\nabla \cdot \left(\varepsilon_r \varepsilon_0 \nabla \Phi(x, y)\right) = -\rho$$ Cartesian XY

$$\nabla \cdot \left(\varepsilon_r \varepsilon_0 \nabla \Phi(r, z)\right) = -\rho$$ Cylindrical about Z

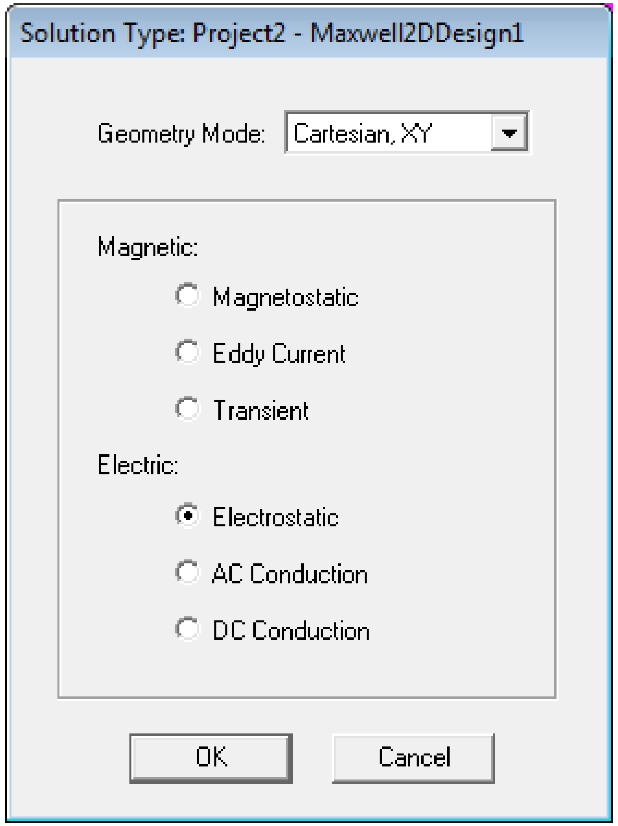
Maxwell 2D

Selecting the Electrostatic Solver

- **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**



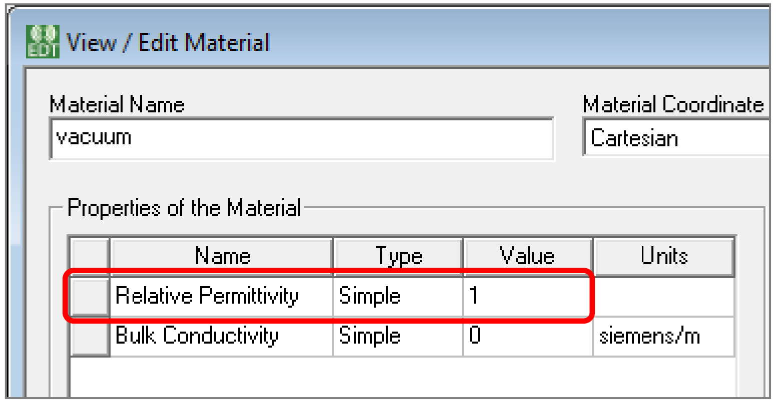
Maxwell 3D

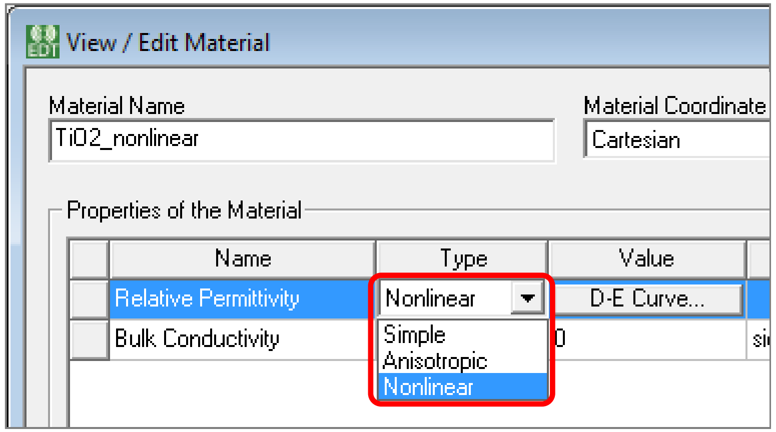


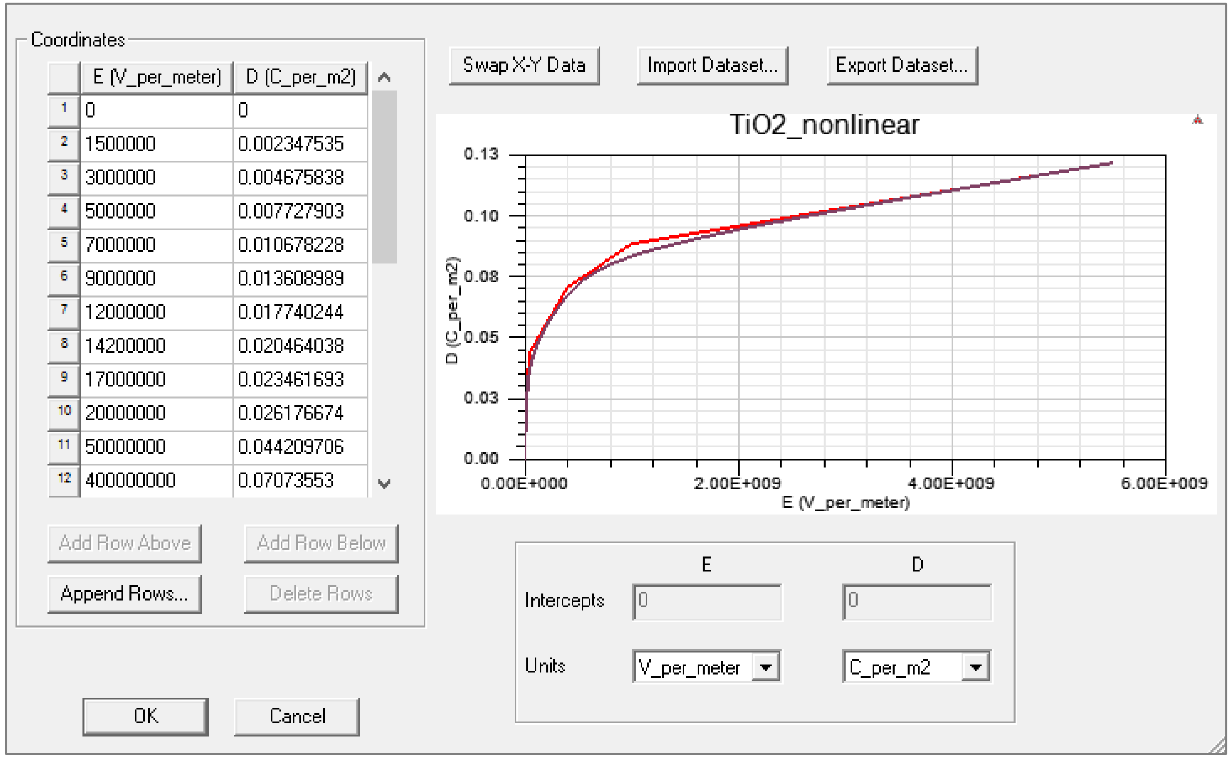
Maxwell 2D

Material Definition

- **Electrostatic Material Properties**
  - **In aN Electrostatic simulation, the following parameters may be defined for a material:**
  - **Relative Permittivity**
    - **Relative permittivity εr determines the electric field solution and can be Simple, Nonlinear or Anisotropic**

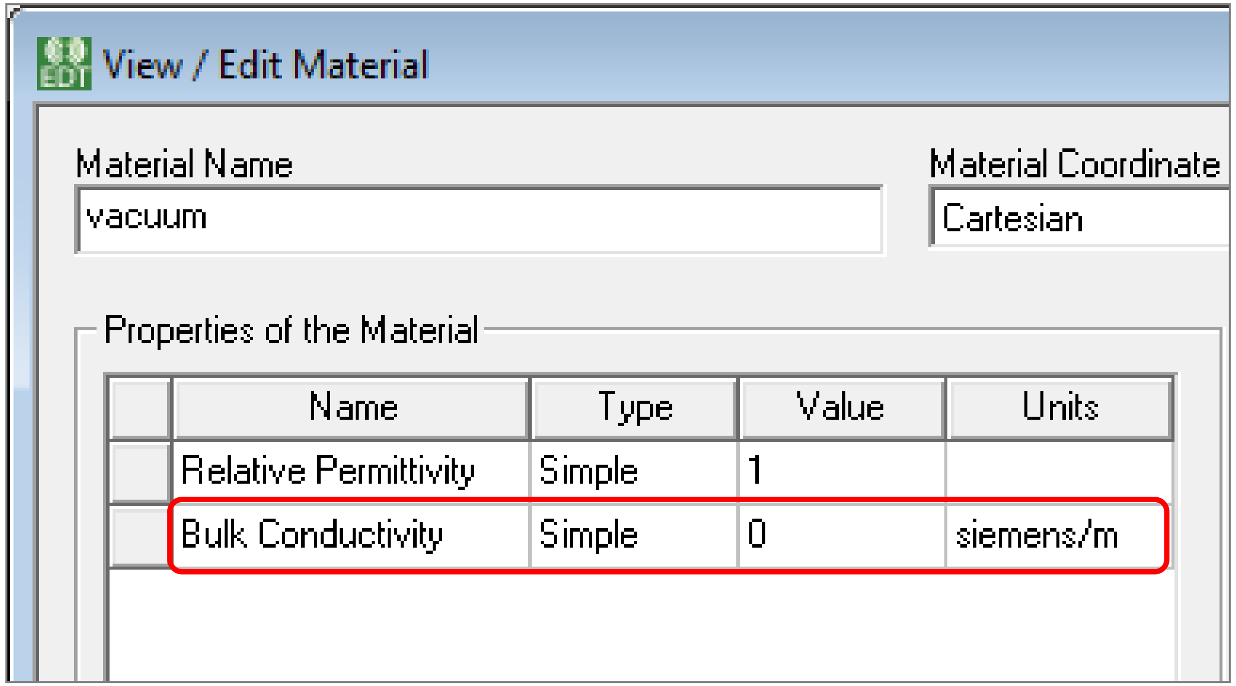


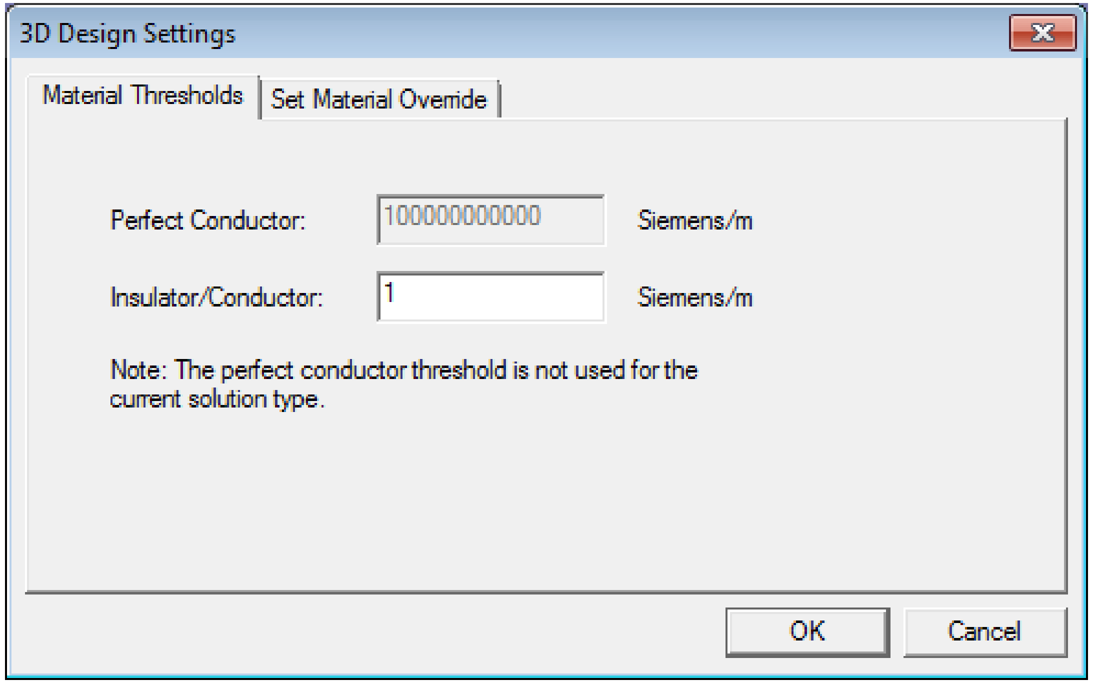




Material Definition

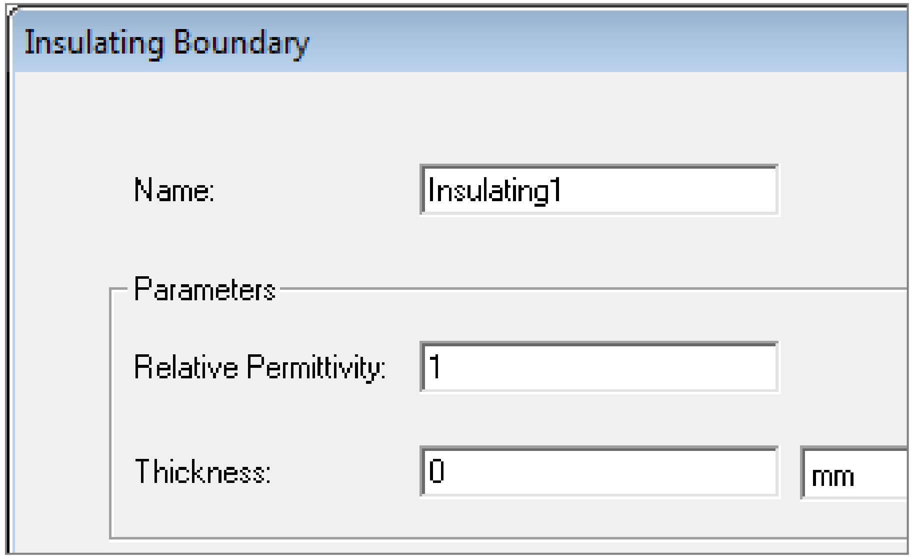
- **Electrostatic Material Properties (continued)**
  - **Bulk Conductivity**
    - **Defines whether an object is a conductor (treated as a perfect conductor) or an insulator.**
    - **This classification is determined by the insulator/conductor material threshold setting defined under Maxwell 3D/2D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





Boundary Conditions (BC)

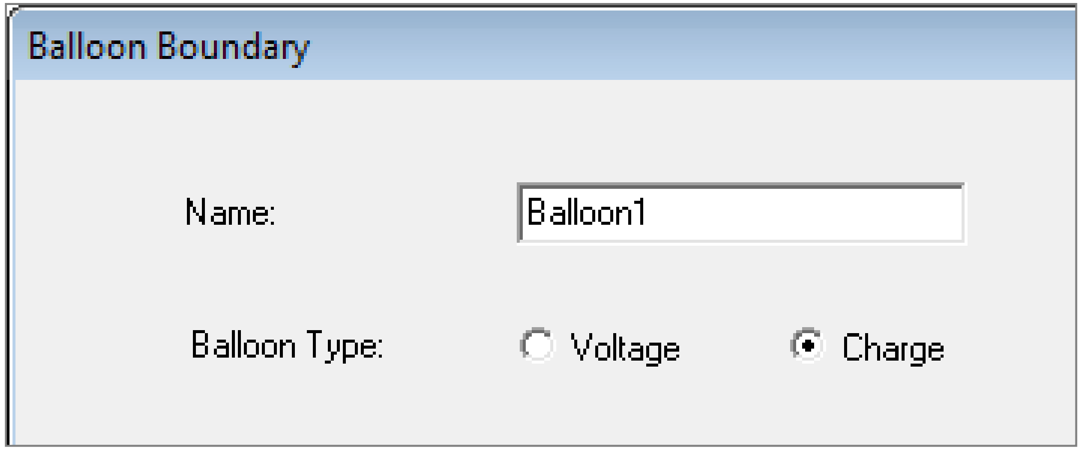
- Assigning Boundary Conditions in 3D
  - Boundary Conditions define behavior of electric field at the interfaces or the edges of the problem region
  - A BC can be assigned to a face from menu item ***Maxwell 3D → Boundaries → Assign***
- Boundary Types (3D)
  - **Default (No Boundary Assigned):** When no boundary is specified for a surface following two treatments are assigned based on the surface position
    - Natural: on interfaces between objects. D-Field Normal component at the interface 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 discontinuous across the insulating boundary
    - Can be used to model thin layer of insulation by specifying Permittivity for the layer



Boundary Conditions (BC)

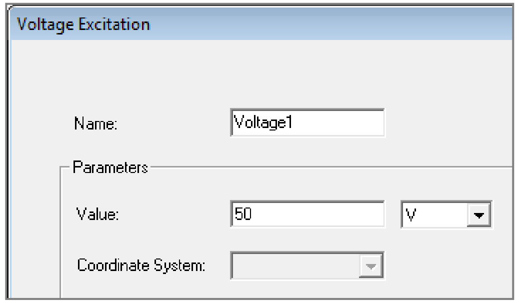
- Boundary Types (2D & 3D):
  - Master/Slave
    - Enables to model only one period of a periodic structure, reducing design size.
    - Magnetic fields at Master and Slave boundaries match each other
  - Symmetry Boundary
    - Enables to model only part of a structure, reducing design size and solution time.
    - Applied to external boundaries of domain.

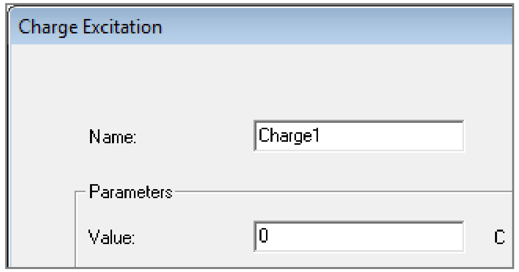
- Boundary Types(2D)
  - Balloon (two types):
    - Voltage: voltage is zero at infinity
    - Charge: the charge at infinity matches the charge in solution region, forcing net charge to be zero

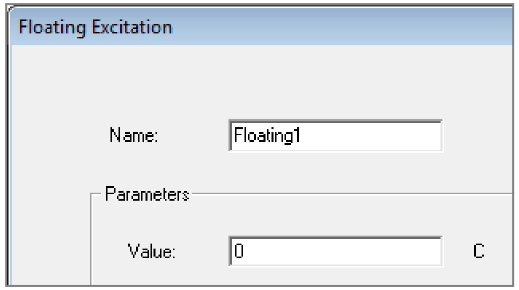


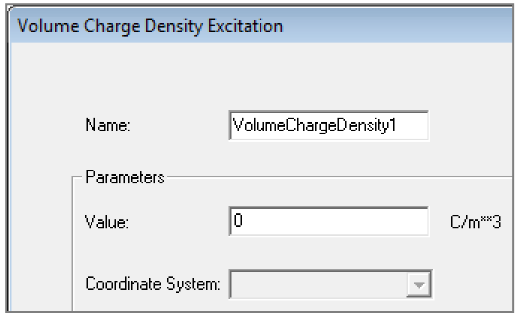
Excitations

- Assigning Excitations
  - Excitations can be assigned from *RMB on Excitations → Assign*
    - **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 objects) or line charge density (assigned to edges)
      - In 3D, only volume charge density can be assigned









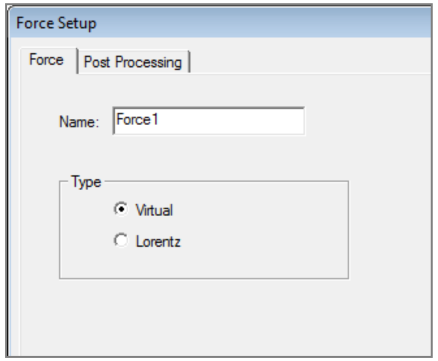
Ansys

Parameters

- **Parameters**
  - Three parameters can be assigned for magnetostatic solver: Force, Torque, Capacitance Matrix
  - Parameters can be added through ***Maxwell 3D/2D → Parameters → Assign***

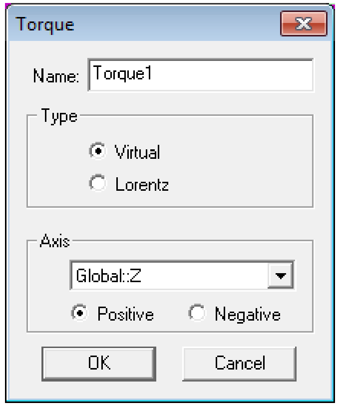
– Force:

- Calculates force acting on selected objects
- Can be Virtual or Lorentz
- Lorentz can not be used for magnetic materials



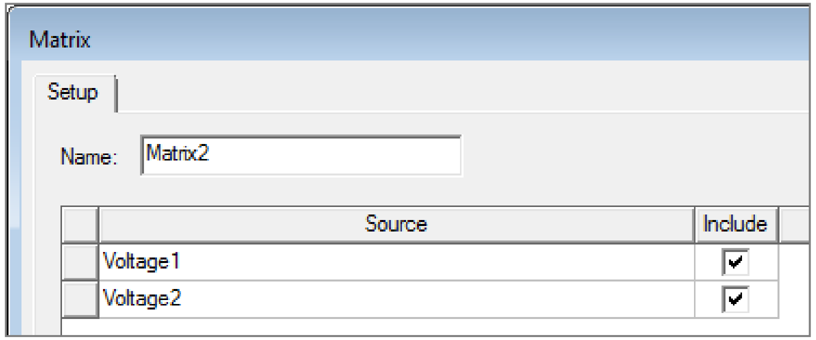
– Torque:

- Calculates torque on selected objects
- Can be Virtual or Lorentz
- The torque acting axis and direction must be set



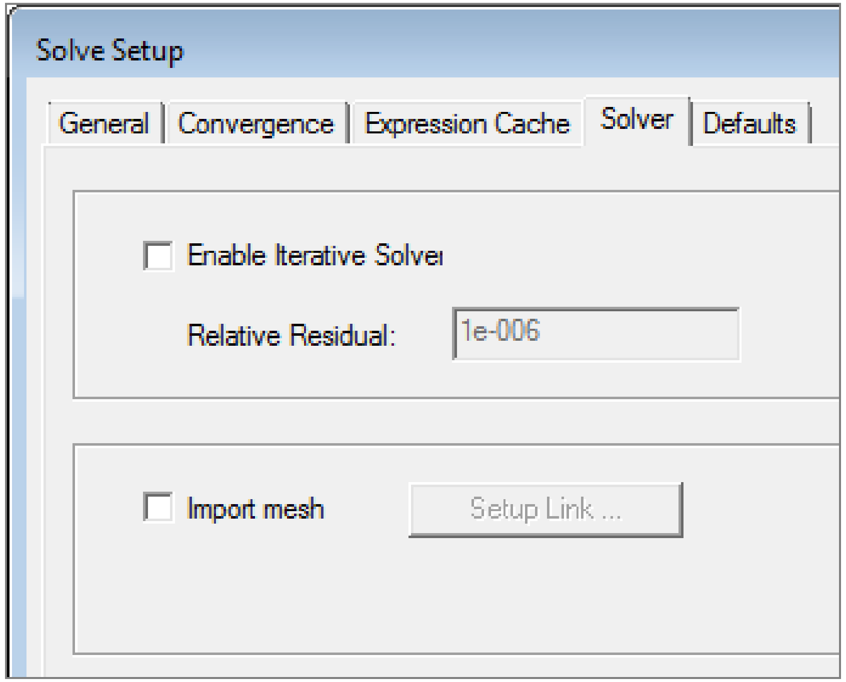
– Matrix:

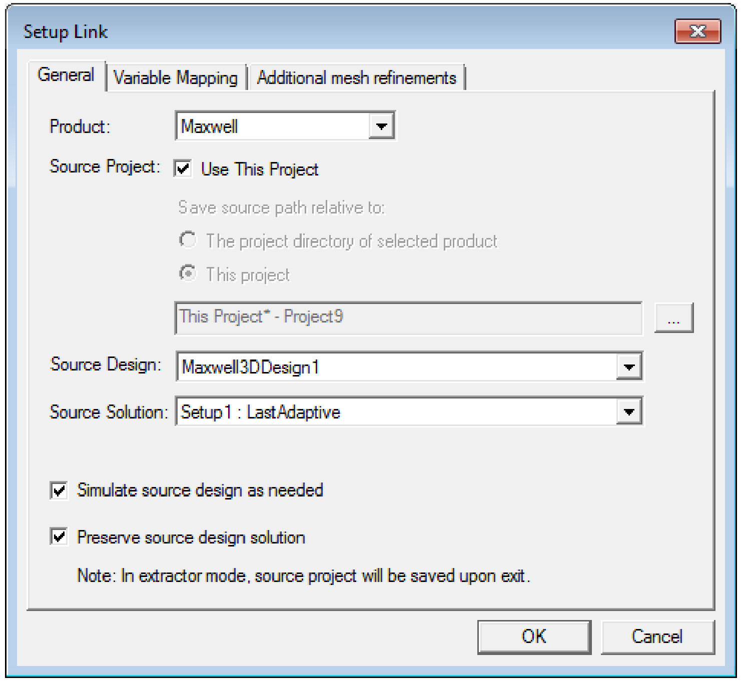
- Calculates Capacitance matrix
- Matrix results can be seen under Results → Solution Data



Analysis Setup

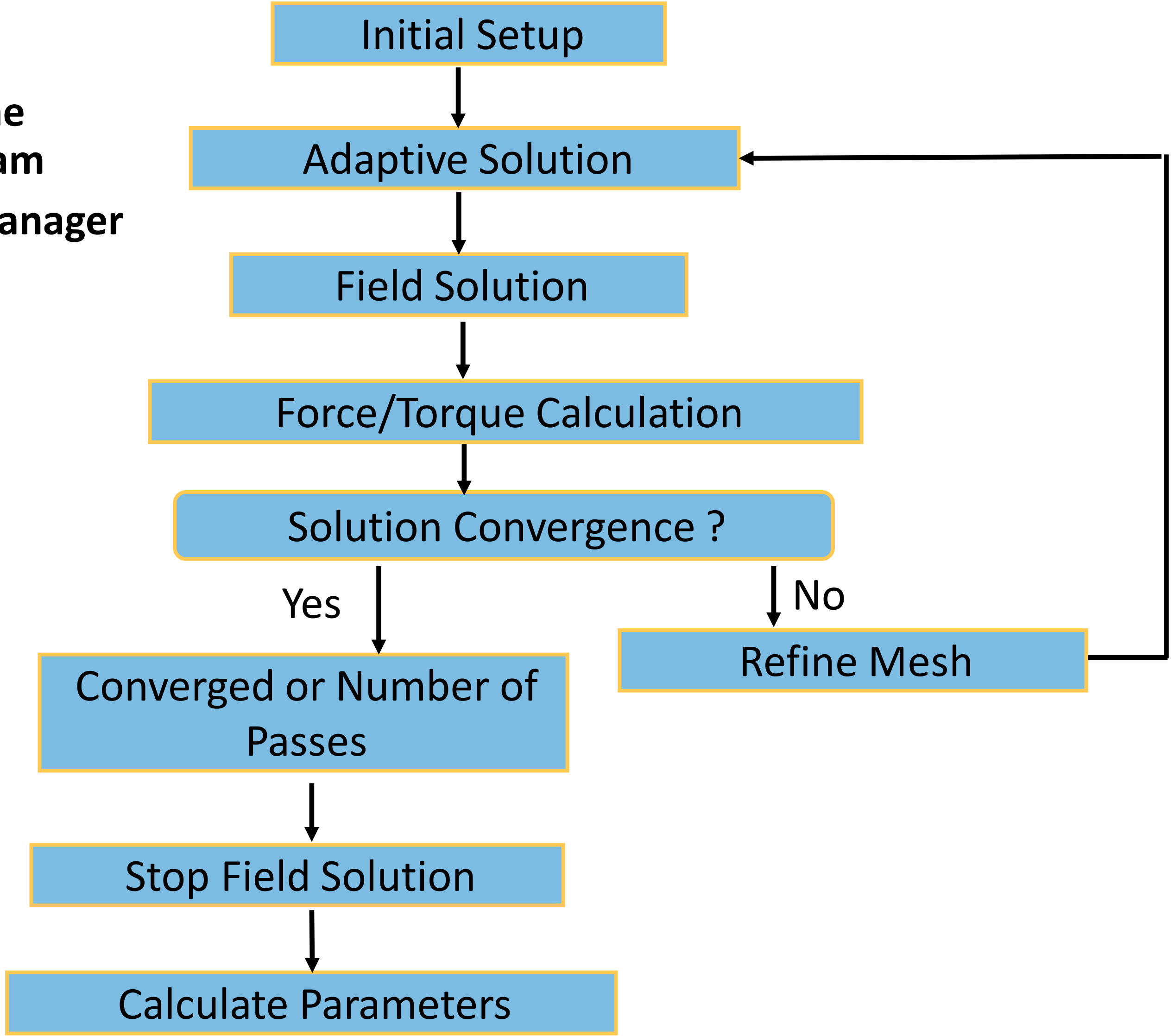
- Solution Setup
  - A Solution Setup can be added from the menu *Maxwell 3D/2D → Analysis Setup → Add Solution Setup*
  - Options on General and Convergence tab are the same as for Magnetostatic solver
  - Solver Tab
    - Enable Iterative Solve (only for 3D designs): Enables ICCG solvers (Direct is 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 set





Solution Process

- **Electrostatic Solution Process**
  - **Like the other QS Solvers, the solution process in the Electrostatic solver is automated as shown in diagram**
  - **A Solution process can be launched from Project Manager window *RMB on Setup1 → Analyze***



DC Conduction Solver

DC Conduction Solver

- **DC Conduction Solver**
  - The DC Conduction solver solves for the DC Currents resulting in conductors
  - The quantity solved is the electric scalar potential (F)
  - Current density (J) and Electric Field (E) are automatically calculated from the electric scalar potential (F)
  - All fields outside the conductors are not calculated – 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

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

Maxwell 3D

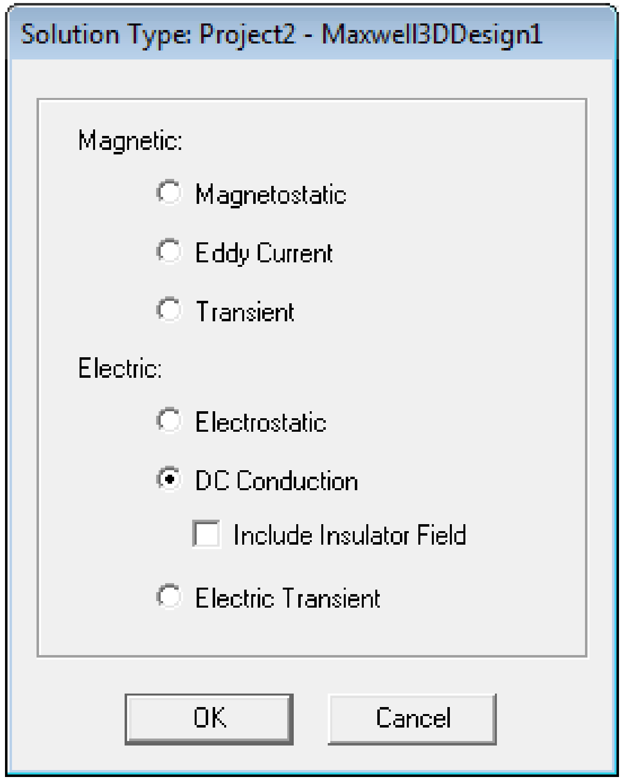
$$J(x,y) = \sigma E(x,y) = -\sigma \nabla \Phi(x,y)$$ **Cartesian XY**

$$J(r,z) = \sigma E(r,z) = -\sigma \nabla \Phi(r,z)$$ **Cylindrical about Z**

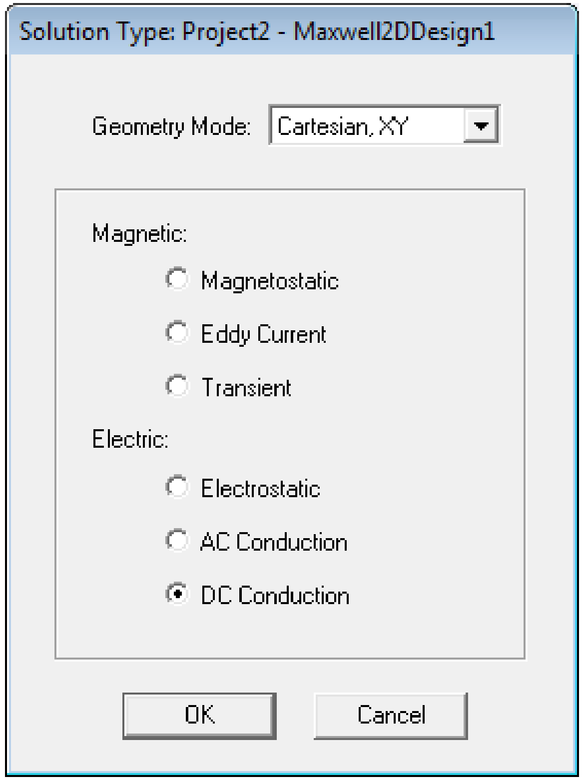
Maxwell 2D

Selecting DC Conduction solver

- **Selecting 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**



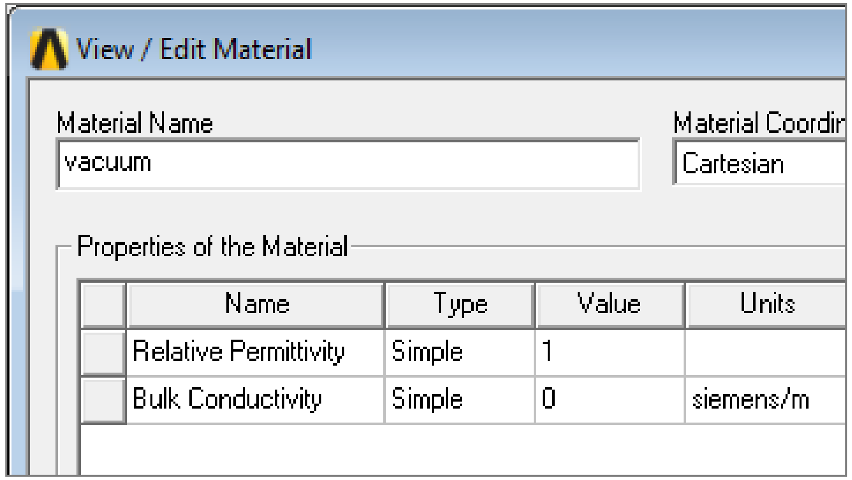
Maxwell 3D

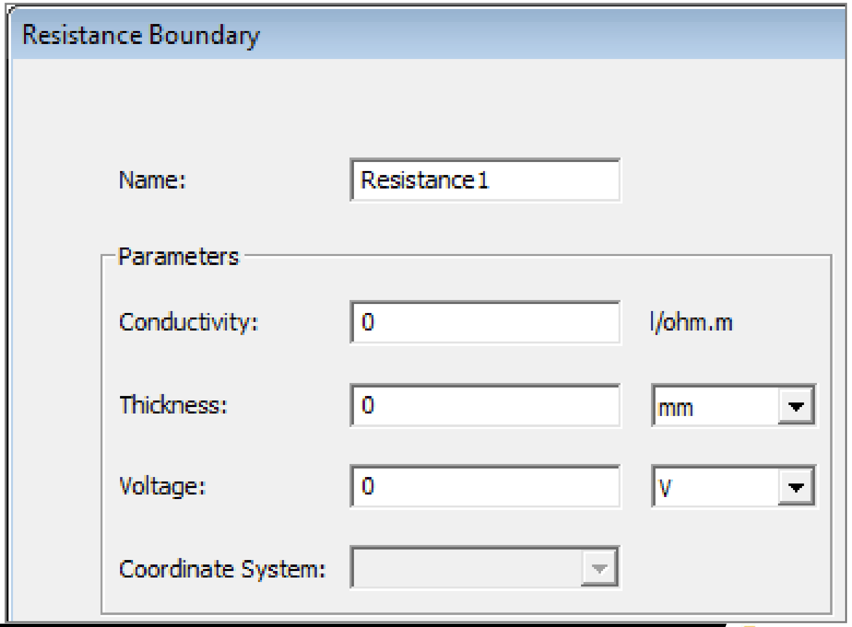


Maxwell 2D

DC Conduction Setup

- Material Properties
  - Material properties for the DC Conduction solver are the same as for the Electrostatic solver
  - It is worth to highlight that in DC Conduction the conductivity is tied directly to the Conductivity Threshold, determining what objects are conductors (current carrying) vs. Insulators (not current carrying)
  - Relative permittivity does not affect DC conduction results but is required if insulator fields are included
- Boundary Conditions
  - All the boundary conditions available in the Electrostatic Solver are also valid for the DC Conduction Solver
  - In 2D, a Resistance Boundary can also be defined:
    - It models a very thin layer of resistive material on a conductor at known potential
    - Can be assigned only to the boundary edges of solution domain

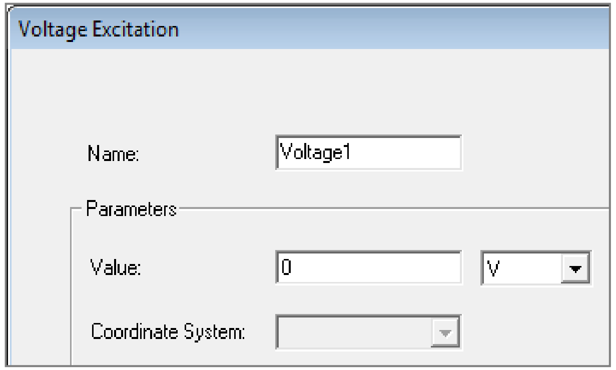


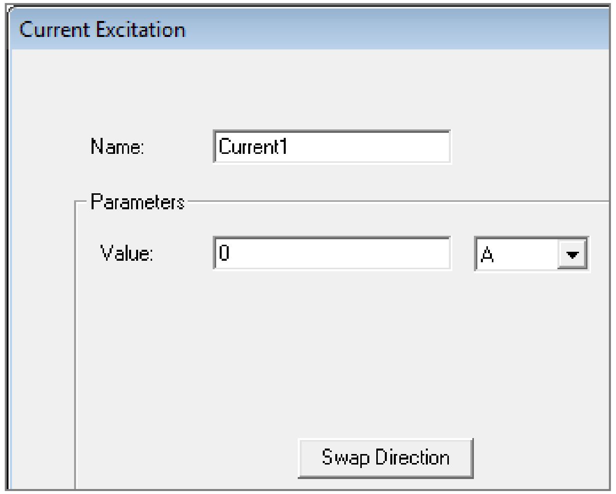


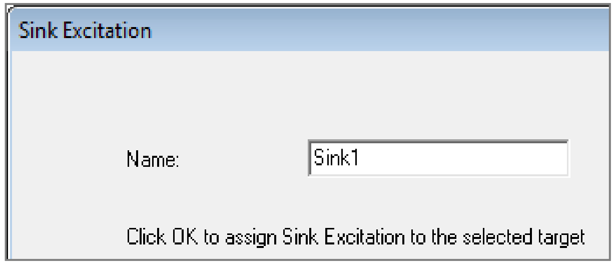
Excitations

- **Excitations**
  - **Excitations can be assigned from *RMB on Excitations → Assign***
    - **Voltage (2D &3D)**
      - **Assigns potential at different terminals (Faces in 3D, Edges in 2D) of a conductor**
      - **If assigned to an Object, that object will be considered equipotential and current will flow in/out of the object's boundaries**
  - **Current (3D Only)**
    - **Assigns total current through conductor cross section**
    - **Can be assigned to a Face of an Object**
    - **Sink should be defined along with Current Excitation**
  - **Sink (3D Only)**
    - **Requires to be defined along with Current Excitation**
    - **Ensures total current flowing "into" and "out of" the model is 0**

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

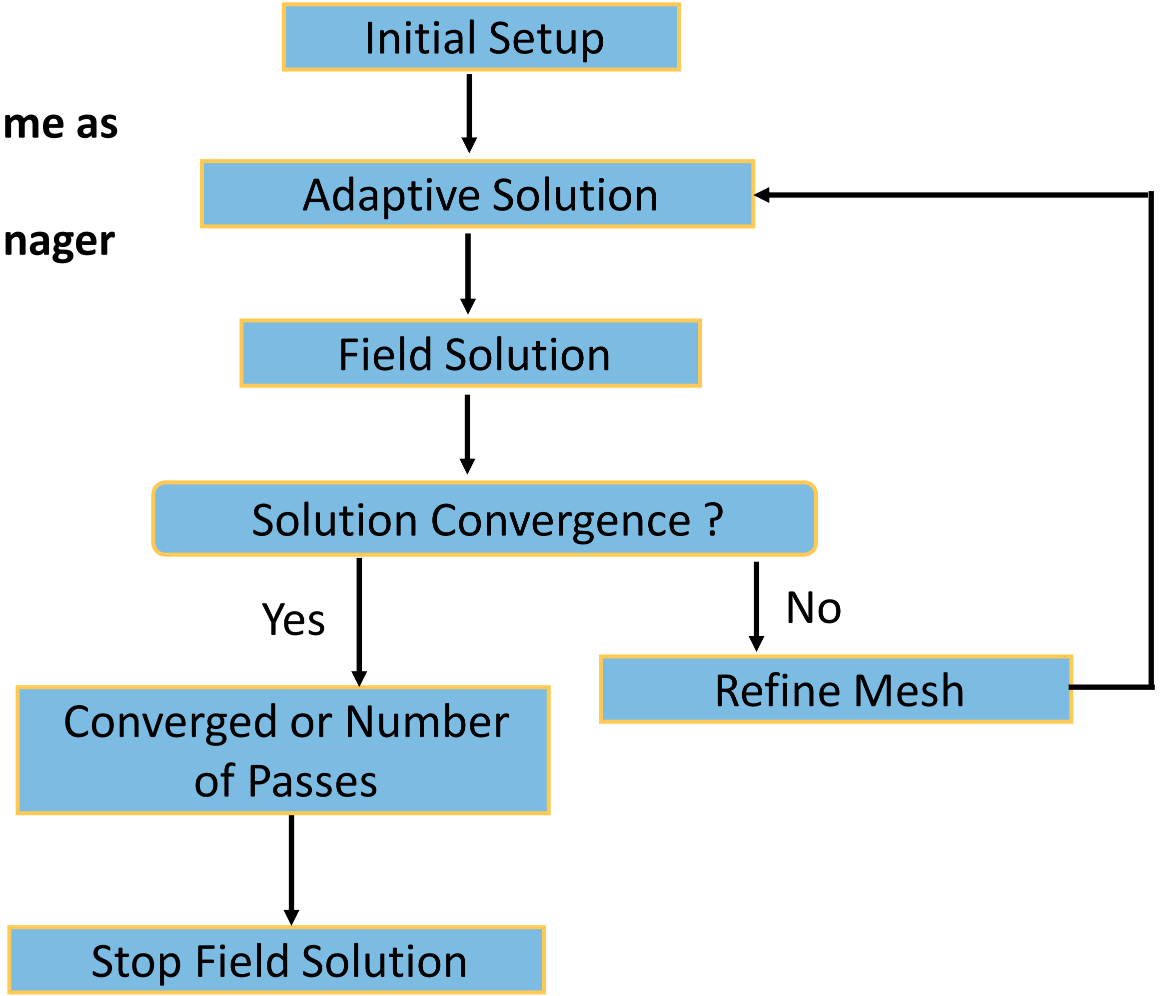






Solution Process

- **Analysis Setup**
  - **All the options in the Solve Setup window are the same as in the Electrostatic Solver**
  - **A Solution process can be launched from Project Manager window *RMB on Setup1 → Analyze***



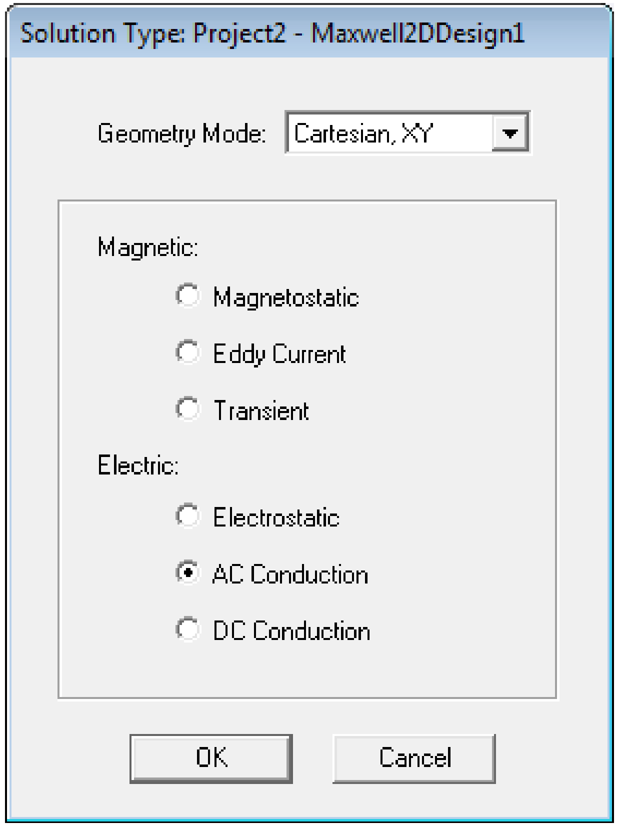
AC Conduction Solver

AC Conduction Solver

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

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

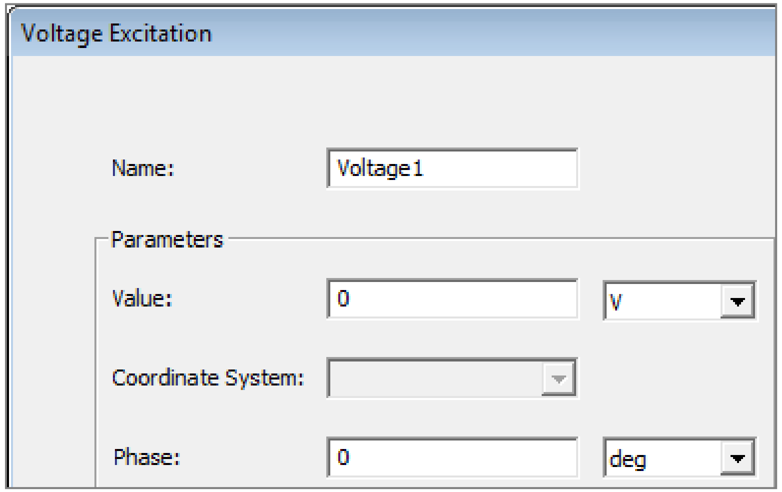
- **Selecting the AC Conduction Solver**
  - **Specify the AC Conduction Solver through *Maxwell 2D → Solution Type***
  - **In the Solution type window, select *Electric → AC Conduction* and press OK**

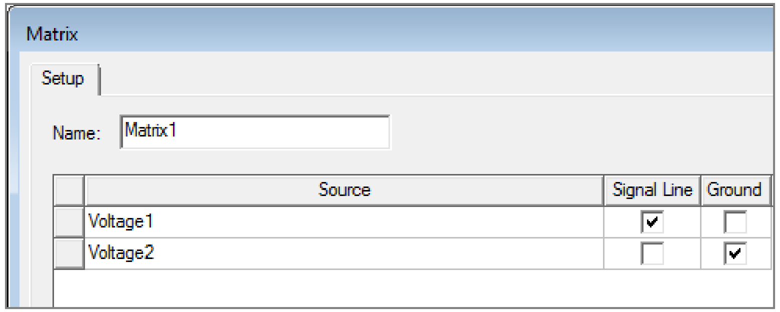


Ansys

AC Conduction Setup

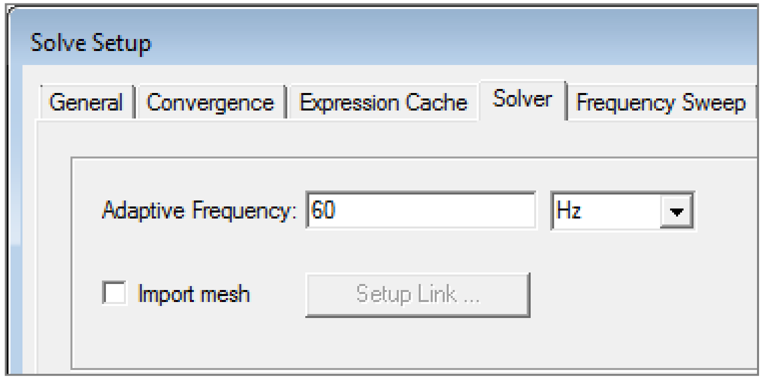
- Material Properties
  - Material properties required for AC Conduction Solver are the same as for Electrostatic Solver.
- Boundary Conditions
  - 3 types of boundaries - Master/Slave, Symmetry and Balloon - same as for Electrostatic Solver
- Excitations
  - **Voltage:**
    - Assigns potential at different terminals (edges) of a conductor
    - If assigned to an Object, that object will be considered equi-potential and current will flow in/out of the object's boundaries to other conductors.
    - AC voltage is specified using Magnitude and Phase
- Parameters
  - Matrix:
    - Calculates Admittance and Capacitance, Conductance matrix
    - Matrix values can be seen in the *Results → Solution Data*

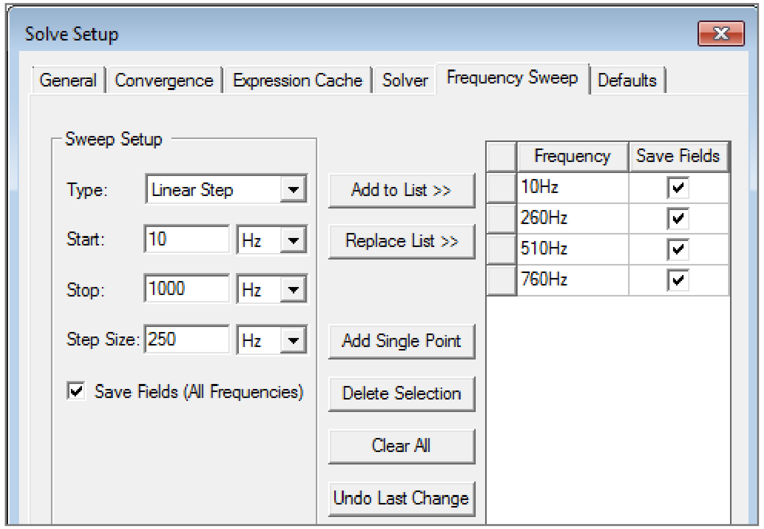




Analysis Setup

- Solution Setup
  - A Solution Setup can be added from the menu *Maxwell 2D → Analysis Setup → Add Solution Setup*
  - Options on General and Convergence tab are the same as for Electrostatic Solver
  - Solver Tab
    - Adaptive Frequency: sets the frequency at which the mesh is constructed and adapted, and at which solution is obtained
    - Import Mesh: initial mesh is imported from another solution – that solution must have the exact same geometry as the current one
  - Frequency Sweep Tab
    - Sweep Setup (Type, Start, Stop, Step): Enables to define frequency sweep range and values
    - Save Fields: Saves the fields for defined swept frequencies
    - Add to List >>: activate frequency sweep



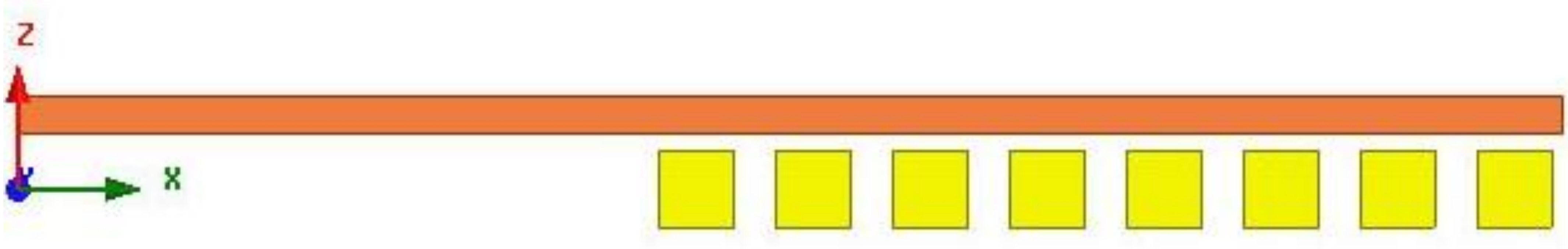


Summary

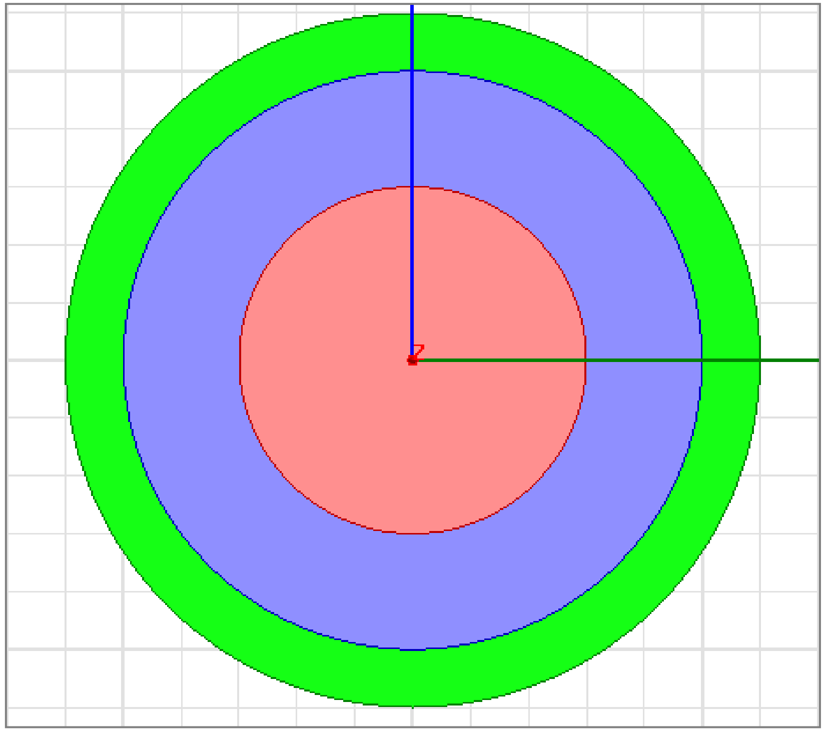
What have we learned in this lecture?

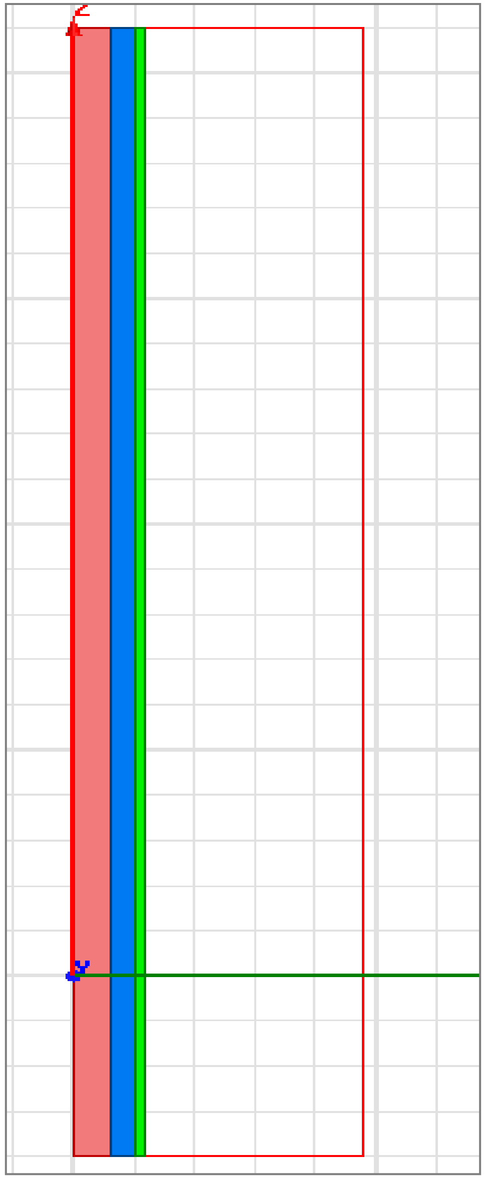
- **Eddy current solver**
- **Material Properties**
- **Electrostatic solver**
- **DC Conduction Solver**
- **AC Conduction Solver**

Workshop 2.1 – 2D Eddy Current analysis

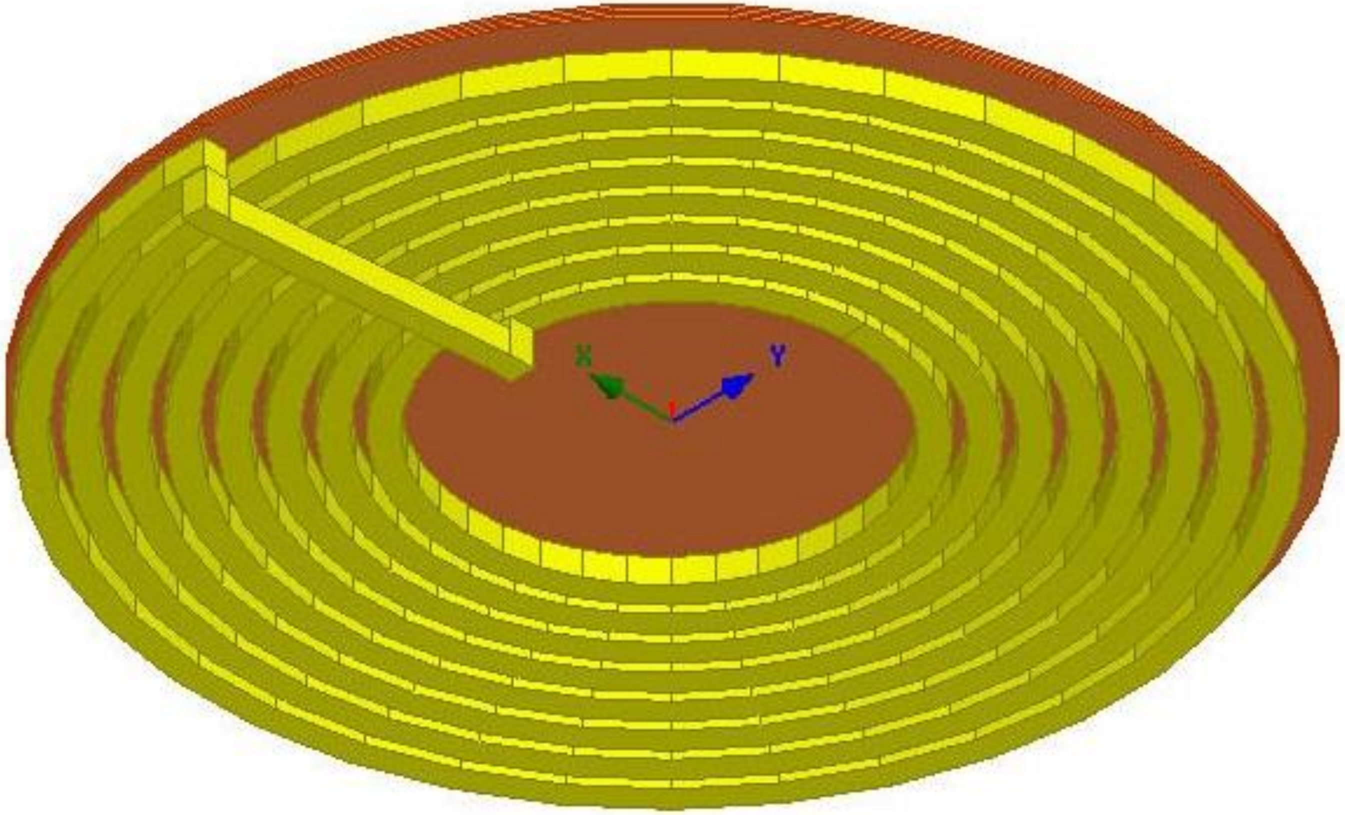


Workshop 2.2 – 2D Electrostatic analysis

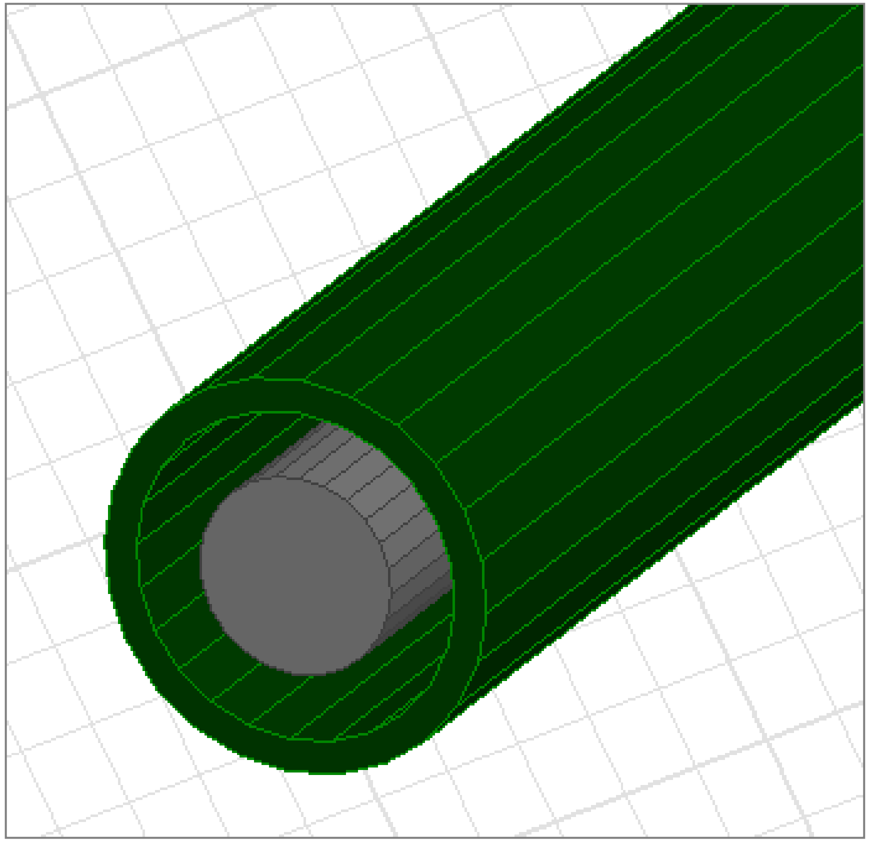


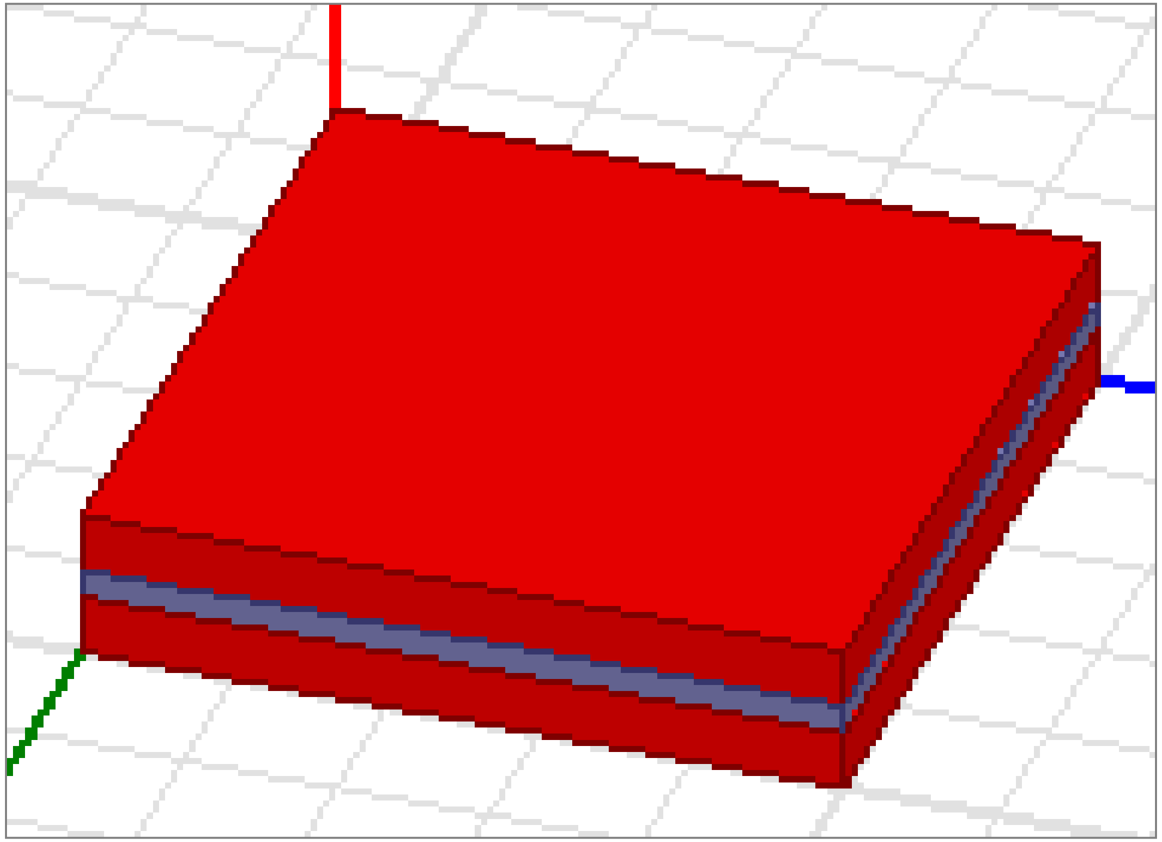


Workshop 2.3 – 3D Eddy Current analysis



Workshop 2.4 – 3D Electrostatic analysis





Ansys

End of Presentation