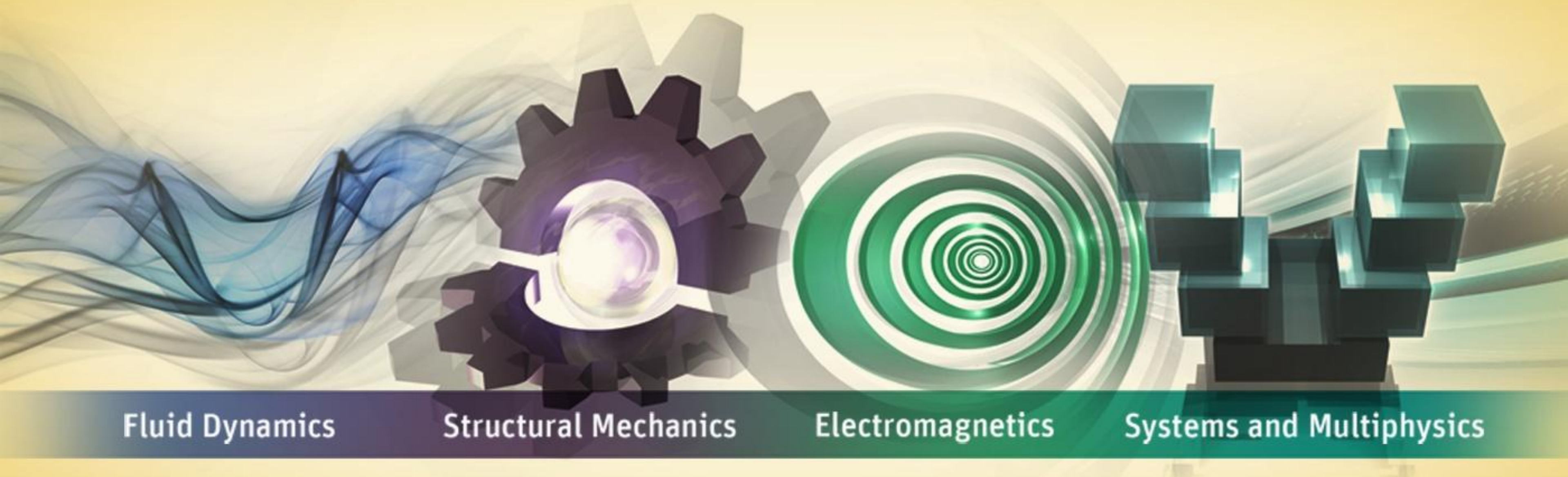
ANSYS

Realize Your Product Promise™

Lecture 3: Static Magnetic Solvers



ANSYS Maxwell V16 Training Manual

Content

A. Magnetostatic Solver
   a. Selecting the Magnetostatic Solver
   b. Material Definition
   c. Boundary Conditions
   d. Excitations
   e. Parameters
   f. Analysis Setup
   g. Solution Process

B. Eddy Current Solver
   a. Selecting the Eddy Current Solver
   b. Material Definition
   c. Boundary Conditions
   d. Excitations
   e. Parameters
   f. Analysis Setup
   g. Solution Process

A. Magnetostatic Solver

Magnetostatic Solver

- In the Magnetostatic Solver, a static magnetic field is solved resulting from a DC current flowing through a coil or due to a permanent magnet
- The Electric field inside the current carrying coil is completely decoupled from magnetic field
- Losses are only due to Ohmic losses in current carrying conductors
- The Magnetostatic solver utilizes an automatic adaptive mesh refinement technique to achieve an accurate and efficient mesh required to meet defined accuracy level (energy error).

Magnetostatic Equations

- Following two Maxwell's equations are solved with Magnetostatic solver

$$\nabla \times \vec{H} = \vec{J}$$

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

$$\vec{B} = \mu_0(\vec{H} + \vec{M}) = \mu_0 \cdot \mu_r \cdot \vec{H} + \mu_0 \cdot \vec{M}_p$$

Maxwell 3D

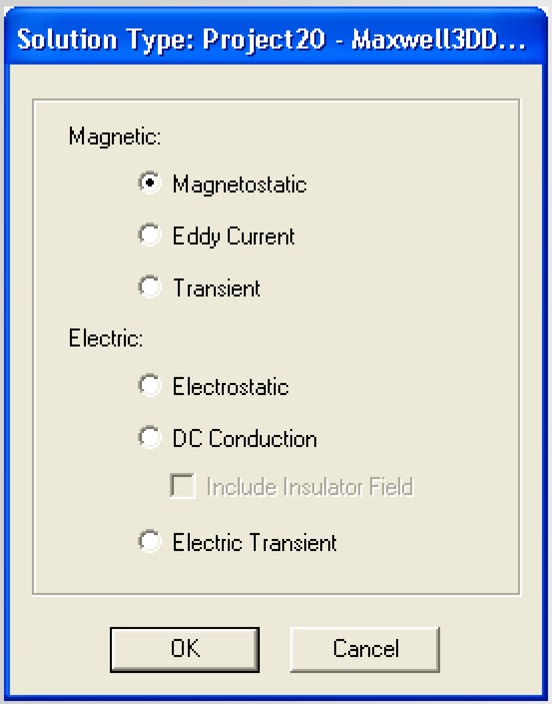
$$\boldsymbol{J_z}(\boldsymbol{x}, \boldsymbol{y}) = \nabla \times \left(\frac{1}{\boldsymbol{\mu}_0 \boldsymbol{\mu_r}} (\nabla \times \boldsymbol{A_z}(\boldsymbol{x}, \boldsymbol{y})) \right)$$

Maxwell 2D

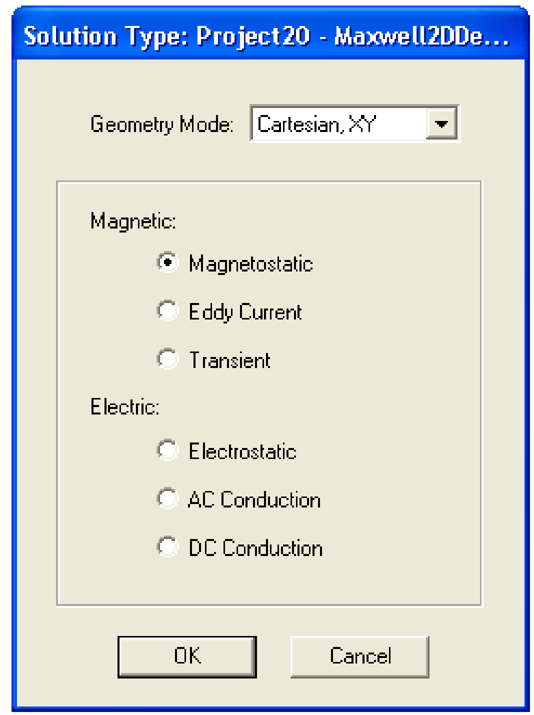
a. Selecting the Magnetostatic Problem

Selecting the Magnetostatic Solver

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



Maxwell 3D



Maxwell 2D

b. Material Definition

Magnetostatic Material Properties

– In a Magnetostatic simulation, the following parameters may be defined for a material (by clicking on the pull-down menu under **Type** and **Value**)

Relative Permeability:

- Permeability (μ) is defined as $\mu_0 * \mu_r$
- Relative permeability(μ_r) along with the Magnetic Coercivity determine the magnetic properties of the material.
- Relative permeability can be Simple(linear μ_r) or Nonlinear(BH Curve) or/and anisotropic

Bulk Conductivity:

- Used to determine the current distribution in current carrying conductors
- Does not have any impact on magnetic part of analysis
- Can be Simple or Anisotropic

Magnetic Coercivity:

- Used to define permanent magnetization of magnetic materials.
- Requires magnitude and direction specification.
- Direction specified is with respect to Orientation CS of bodies to which material is assigned

Composition:

- Can be Solid or Lamination
- Setting Composition to Lamination creates an anisotropic magnetization effect.

c. Boundary Conditions

Assigning Boundary Conditions in 3D

– Boundary conditions define behavior of the magnetic field at the interfaces or the edges of the problem region

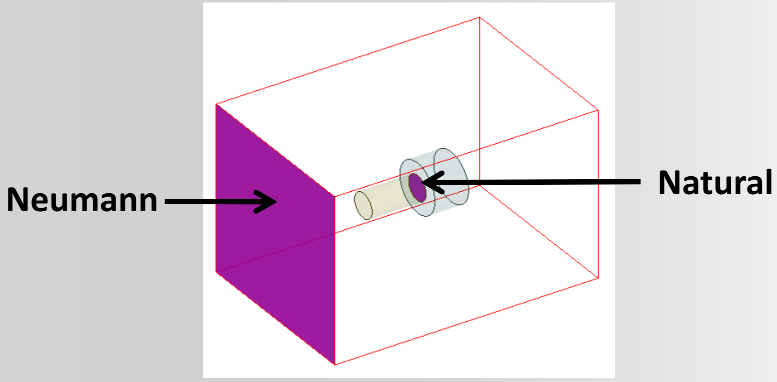
– A boundary can be assigned to a face from menu item ***Maxwell 3D → Boundaries → Assign*** and select the required boundary assignment

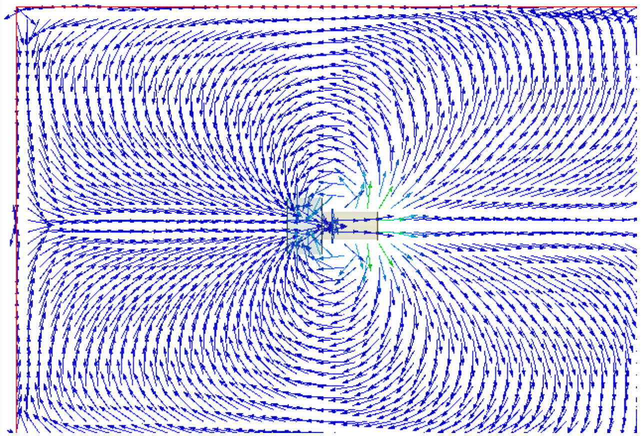
Boundary Types(3D)

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. H Field is continuous across the boundary.
- **Neumann:** For exterior boundaries of solution domain. H Field is tangential to the boundary and flux cannot cross it.



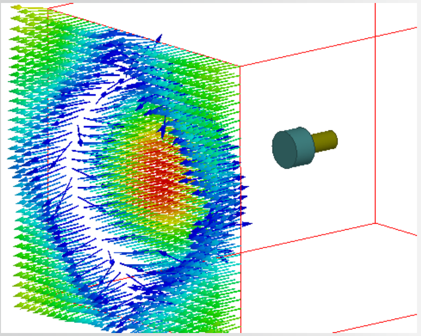


...Boundary Conditions

Boundary Types (3D)

Zero Tangential H-Field:

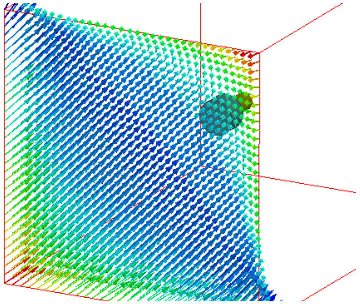
- Useful to assign external field.
- H field is normal to assigned surface
- Can be applied to external boundaries of the domain



H Vectors on Zero Tangential H field boundary

Tangential H-Field:

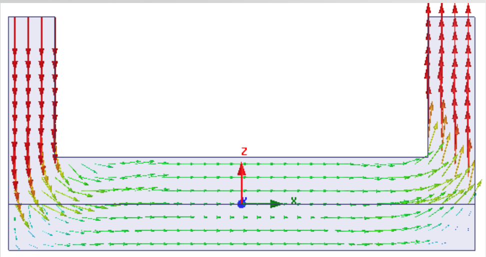
- Useful to assign external field.
- Tangential H field is applied using U and V components
- Can be applied to external boundaries of the domain



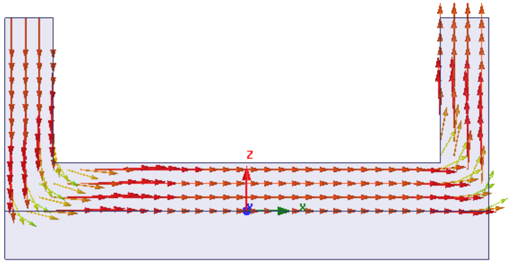
H Vectors on Tangential H field boundary

Insulating:

- Same as Neumann, except that current cannot cross the boundary.
- Can be used to insulate two conductors which are in contact with each other



J Vectors without insulating boundary between plates



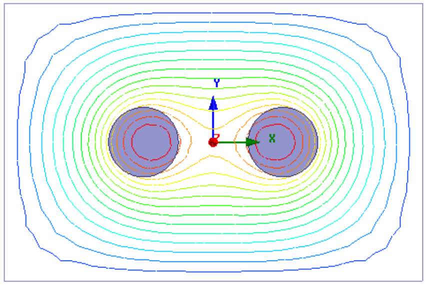
J Vectors with insulating boundary defined between plates

…Boundary Conditions

Boundary Types (2D)

Vector Potential:

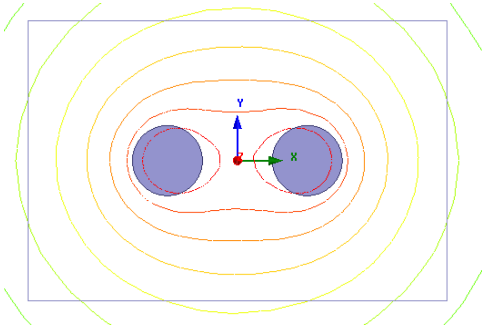
- Sets the specified value of magnetic vector potential on the boundary.
- Used to model Magnetically isolated structures.



Flux lines with zero vector potential on outer boundary

Balloon:

- Models the region outside drawing space as being infinitely large.
- Magnetic flux lines are neither tangential nor normal to the boundary



Flux lines with Balloon on outer boundary

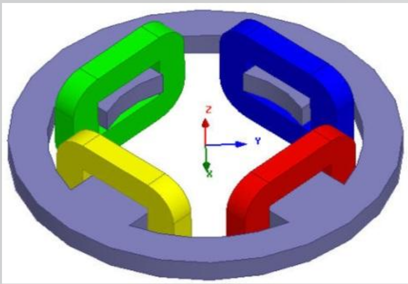
Note: In 2D, no default boundary is assigned to the boundaries of the simulation region. Users have to specify the behavior of simulation boundaries by assigning either Balloon or vector potential boundary.

…Boundary Conditions

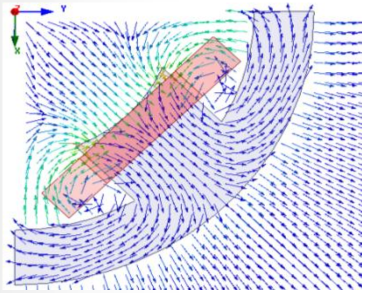
Boundary Types (2D & 3D):

Master/Slave :

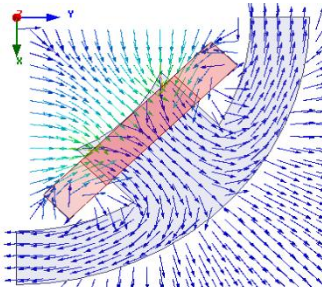
- Enable users to model only one period of a periodic structure, which reduces the design size
- This boundary condition matches the magnetic field at the slave boundary to the field at the master boundary based on U and V vectors defined.



1/4th Model



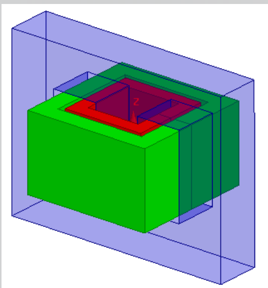
Master = Slave



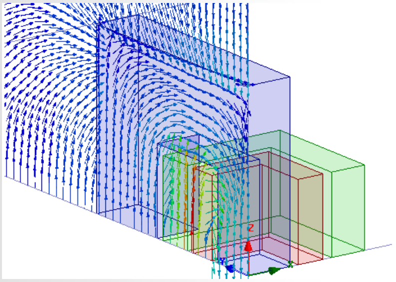
Master = -Slave

Symmetry Boundary:

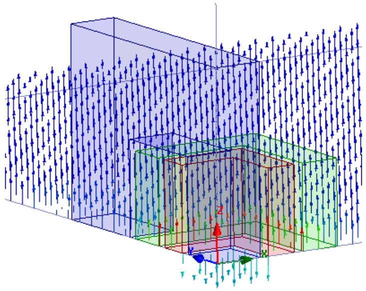
- Enable users to model only part of a structure, which reduces the size or complexity of design, thereby shortening the solution time.
- Applied to external boundaries of domain.



1/8th Model



Symmetry Odd: Same as default Boundary (Flux Tangential)

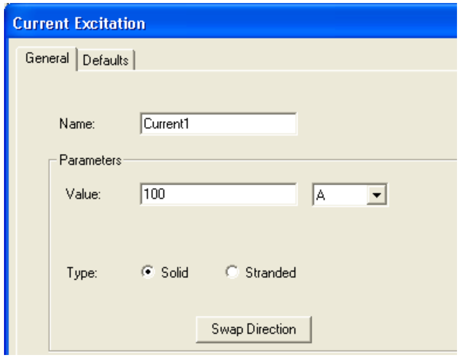


Symmetry Even: Same as Zero Tangential H-Field boundary (Flux Normal)

d. Excitations

Assigning Excitations

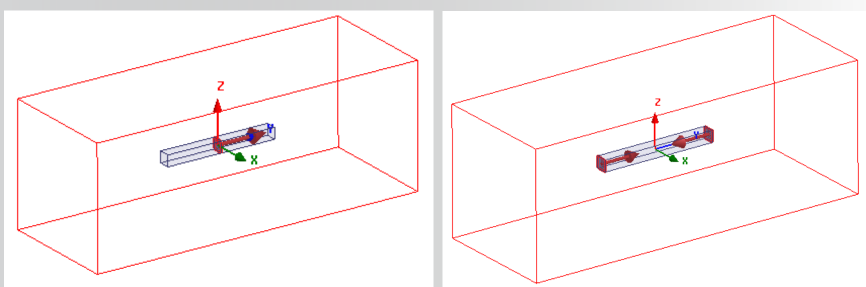
– Excitations can be assigned from the menu item ***Maxwell 2D/3D → Excitations → Assign***



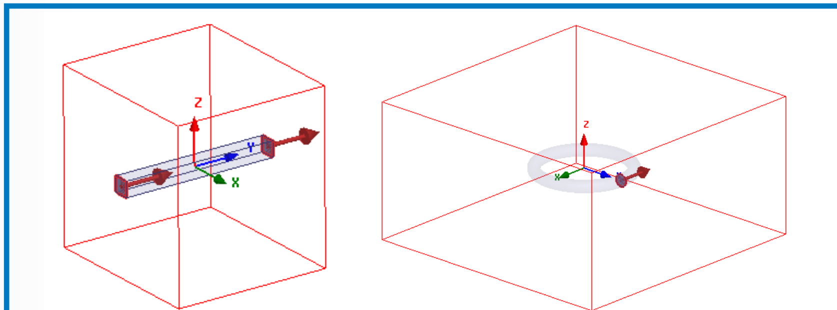
Excitation (2D & 3D)

Current :

- Defines total current in Amp-turns through the conductor
- Can be assigned to the conductor faces that lie on boundary of simulation domain or sheets that lie completely inside the conductor
- Conductor can be defined as Solid or Stranded



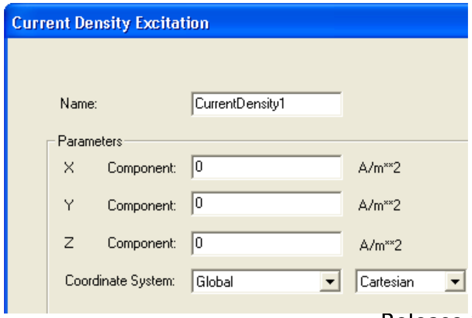
Incorrect Excitation definitions



Correctly defined Excitations

Current Density:

- Used to define a known current density throughout an object.
- In 3D, this definition should be accompanied with Current Density Terminal definition
- Current Density defined using X,Y and Z components of selected CS

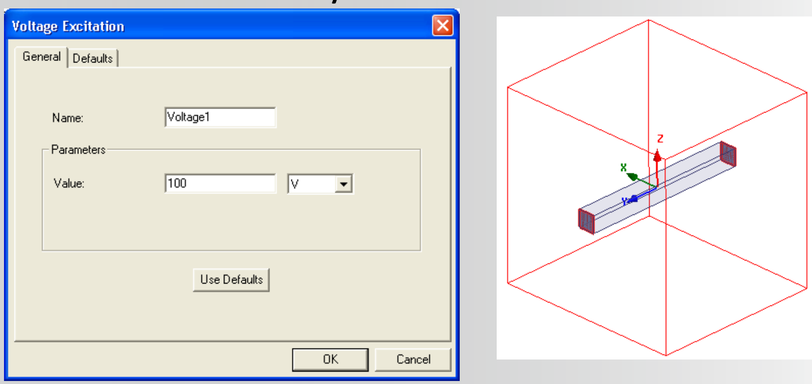


...Excitations

Excitation (3D)

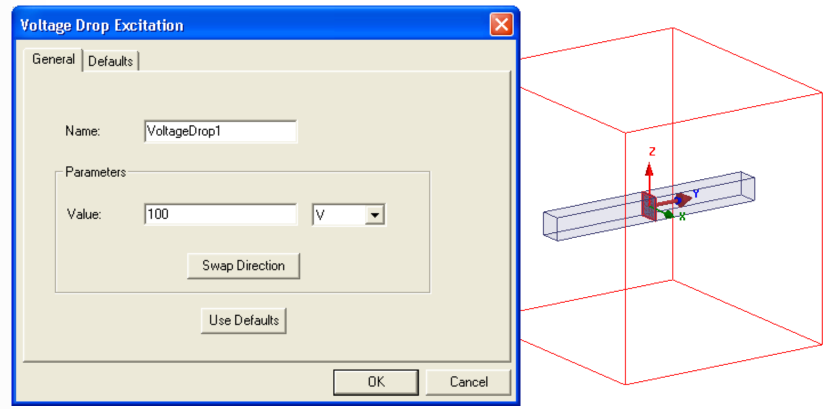
Voltage :

- Used in conjunction with material conductivity to define current through a solid conductor
- Can only be assigned to faces or sheets that lie on the boundary of simulation domain



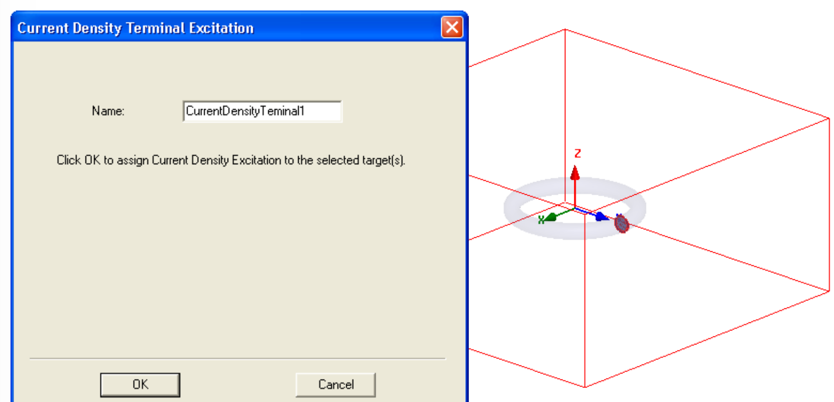
Voltage Drop:

- Similar to the voltage definition
- Can only be assigned to sheets which lie completely inside the conduction path



Current Density Terminal:

- Required to be defined if Current Density is defined
- Can be assigned to any 2D sheet which lies completely inside the conductor or Conductor faces that lie on the boundary of simulation domain



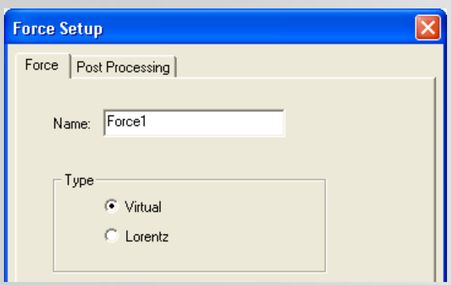
e. Parameters

Parameters

- Three calculation parameters can be assigned for magnetostatic solver which are computed using magnetic field solution
- A parameter can be added by selecting the required object and selecting menu item ***Maxwell 3D/2D → Parameters → Assign***

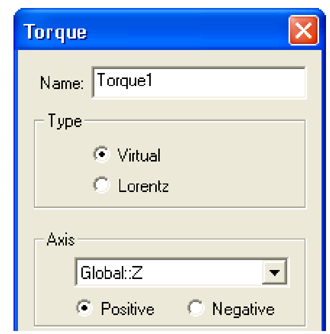
Force:

- Calculates force acting on assigned object
- Force can be Virtual or Lorentz
- Lorentz can not be used for magnetic material



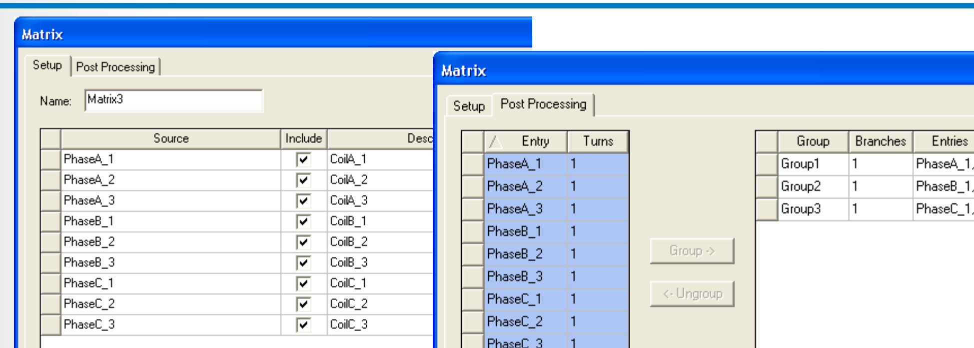
Torque:

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



Matrix:

- Calculates Inductance, resistance matrix
- Calculated matrix can be postprocessed based on defined groups



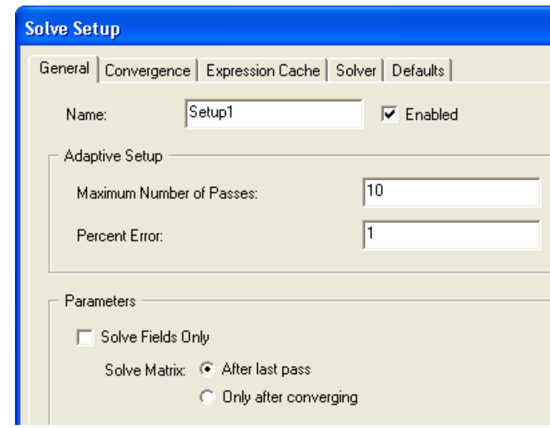
f. Analysis Setup

Solution Setup

- The solution setup defines the parameters used for solving the simulation
- A Solution Setup can be added from the menu item ***Maxwell 3D/2D → Analysis Setup → Add Solution Setup***

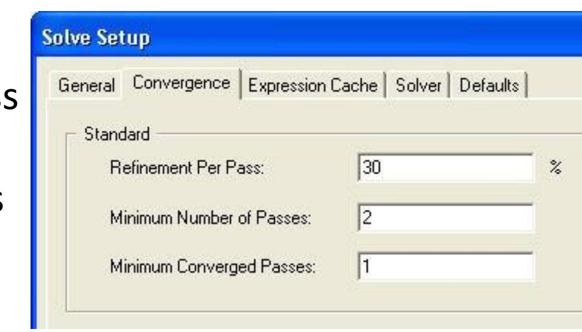
General Tab

- **Name:** sets the Name of the setup. Users can have multiple setups in the same design by repeating the procedure
- **Maximum Number of Passes:** Defines a limit to the number of adaptively refined passes that the solver performs
- **Percent Error:** Error goal for both Error Energy and Delta Energy.
- **Solve Fields Only:** Ignores any defined parameters if checked.
- **Solve Matrix:** Provides the options of calculating the matrix after the last solved pass or only if the solution converges.



Convergence Tab

- **Refinement Per Pass:** Defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass
- **Minimum Number of Passes:** Defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes
- **Minimum Converged Passes:** Defines the minimum number of converged adaptive passed before solution is stopped

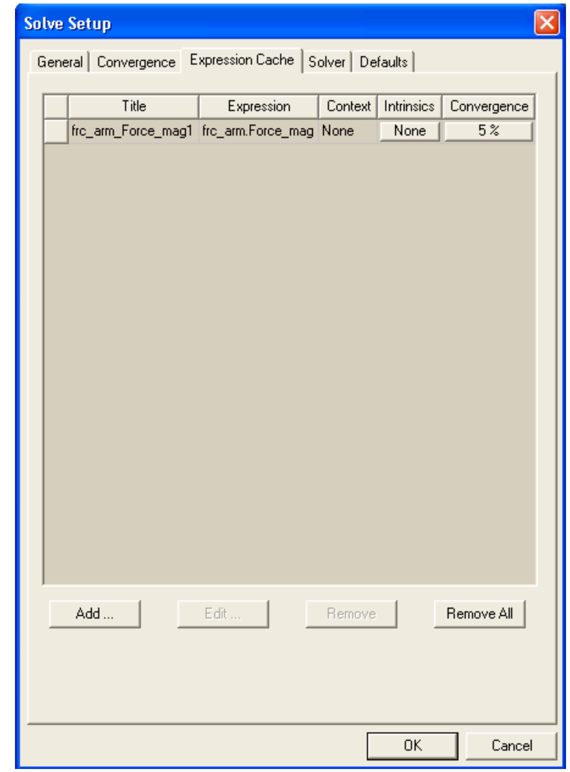


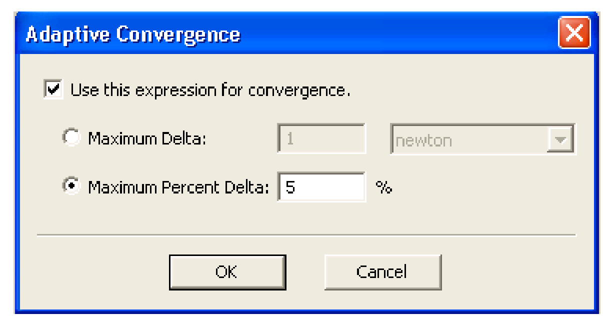
…Analysis Setup

Solution Setup

Expression Cache Tab:

- Enables users to define Output calculations at each adaptive pass or set an additional convergence criteria based on added parameters.
- Solution Setup should be completely defined first to enable adding Expression Cache variables
- Clicking on **Add** button enables users to define Output quantities which can be any defined parameters or derived quantities from Field Calculator
- Selecting the tab under Convergence will enable users to add the selected quantity as a convergence criteria
    - Users can define the permissible change in output quantity in percentile or absolute value
    - Output quantity will be evaluated at each Adaptive pass.
    - If the change in defined output is less than specified value, the solution is considered to be converged provided that energy is already converged.



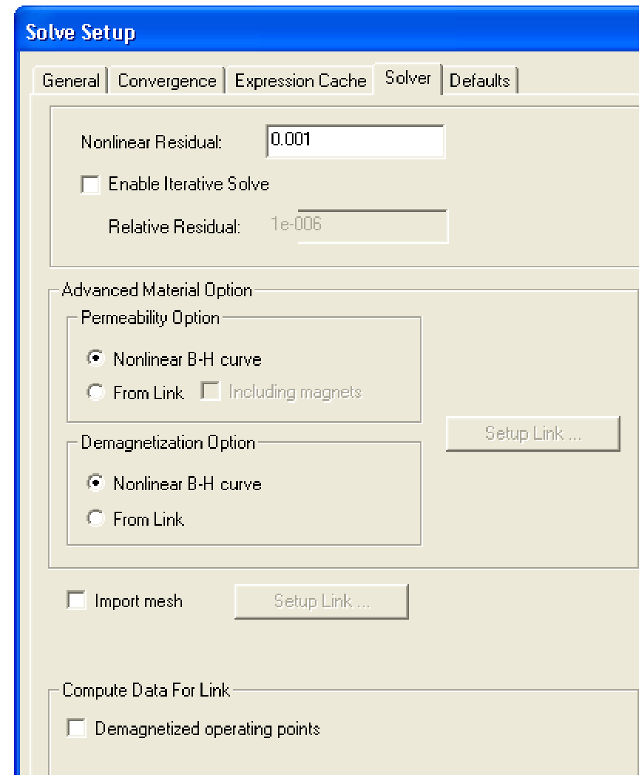


…Analysis Setup

Solution Setup

Solver Tab

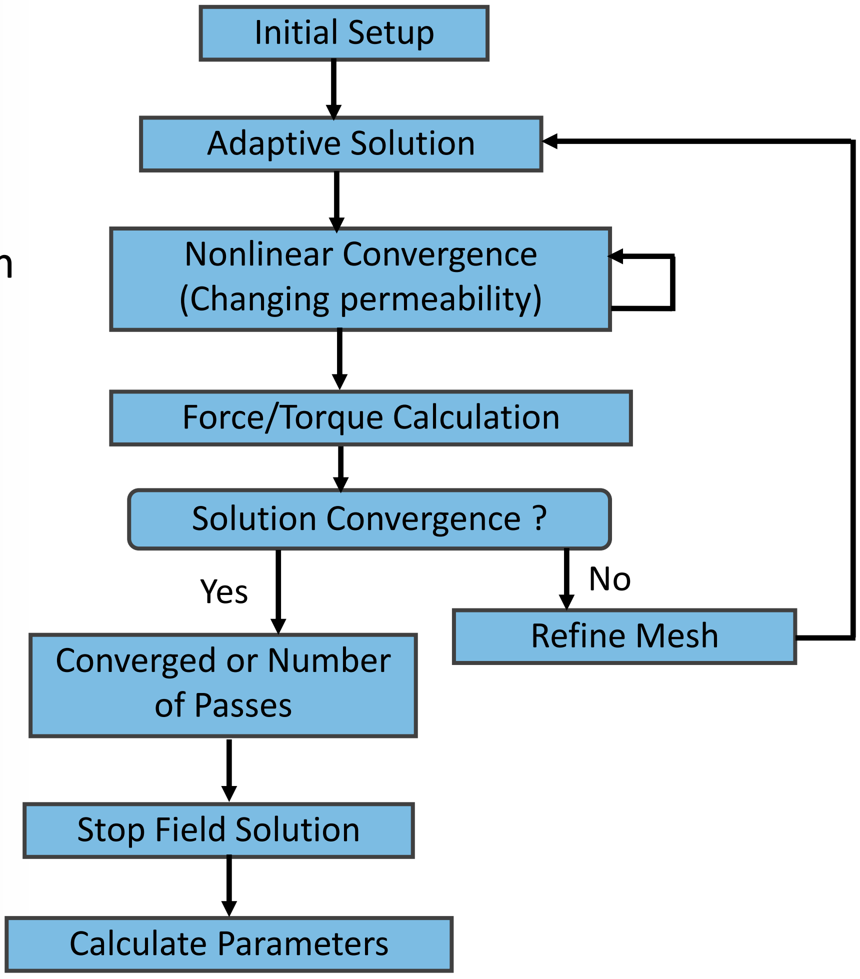
- **Nonlinear Residual:** Defines how precisely the nonlinear solution must define the B-H nonlinear operating points
- **Enable Iterative Solver:** Enables ICCG solvers (Direct is the default).
- **Permeability Option:** Allows nonlinear B-H operating points either to be calculated by the solver from Nonlinear B-H curve or to use frozen permeabilities From Link – the linked solution must have the exact same geometry as the current simulation
- **Demagnetization Option:** Allows the permanent demagnetization to be determined from the Nonlinear B-H curve or to use demagnetized values From Link - where the linked solution must the option "Compute Data for Link - Demagnetized operating points" checked and must have the exact same geometry
- **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 From Link or Import Mesh.



g. Solution Process

Magnetostatic Solution Process

- The solution process is very automated. Once problem is completely defined, Maxwell steps through several stages of solution process as shown in diagram
- A Solution process can be launched from the menu item ***Maxwell 3D/2D → Analyze All***



B. Eddy Current Solver

Eddy Current Solver

- Eddy current solver computes steady-state, time-varying (AC) magnetic fields at a given frequency
- This is a frequency domain solution and assumes frequency of the pulsating fields to be same throughout the domain
- 3D Eddy Current Solver is a full wave solver and solves for displacement currents
- The source of the AC magnetic field can be a Sinusoidal AC current in conductors or time-varying external magnetic fields represented by external boundary conditions.
- Eddy Current solver utilizes adaptive mesh refinement technique to achieve best mesh required to meet defined accuracy level

Eddy Current Equations

- Following equations are solved with Eddy Current solver

$$\nabla \times \left(\frac{1}{\sigma + j_{\omega \varepsilon}} \nabla \times H \right) = j_{\omega \mu} H$$

Maxwell 3D

$$\nabla \times \left(\frac{1}{\sigma + j_{\omega \varepsilon}} \nabla \times H \right) = j_{\omega \mu} H$$

Maxwell 2D

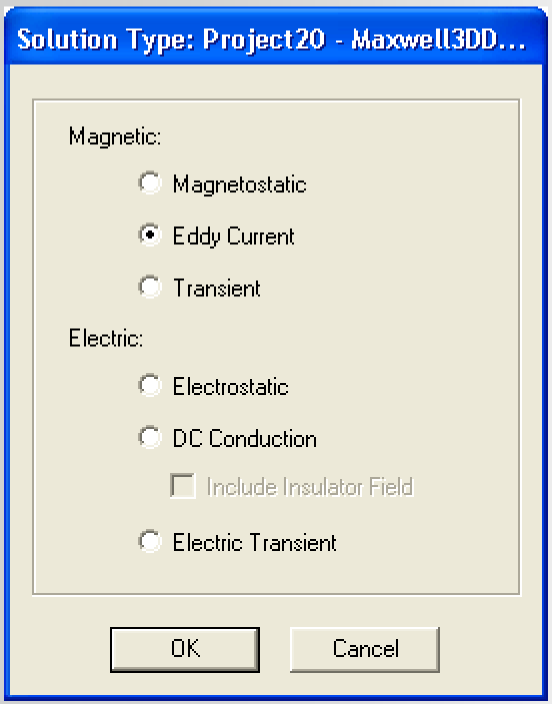
a. Selecting the Eddy Current Problem

Selecting the Eddy Current Solver

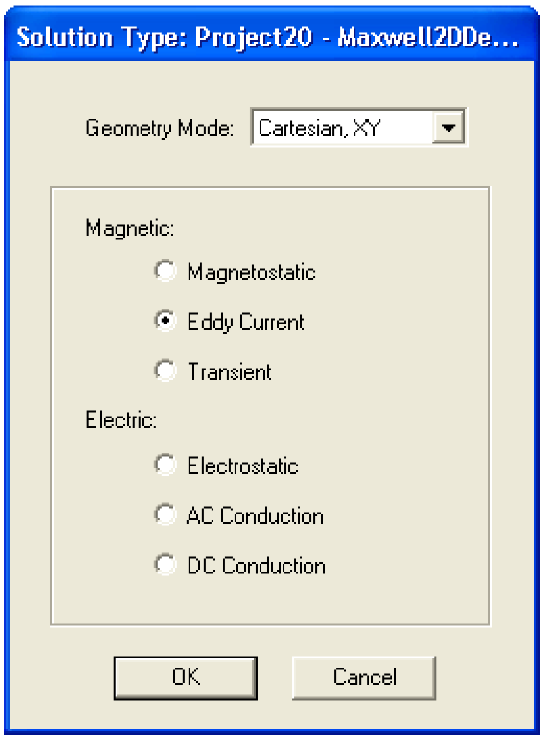
– By default, any newly created design will be set as a Magnetostatic problem

– 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

b. Material Definition

Eddy Current Material Properties

– In Eddy Current simulations, the following parameters may be defined for a material:

Relative Permittivity:

- Relative Permittivity effects solution when displacement currents are considered in an object.
- Relative Permittivity can be Simple or anisotropic

Relative Permeability :

- Relative Permeability along with the Bulk Conductivity determine the time-varying magnetic properties of the material.
- Relative Permeability can be Simple and Anisotropic. In Maxwell 2D, nonlinear permeability is supported by obtaining a linearized permeability for each element from non-linear curve. Solution is still assumed to be linear.

Bulk Conductivity:

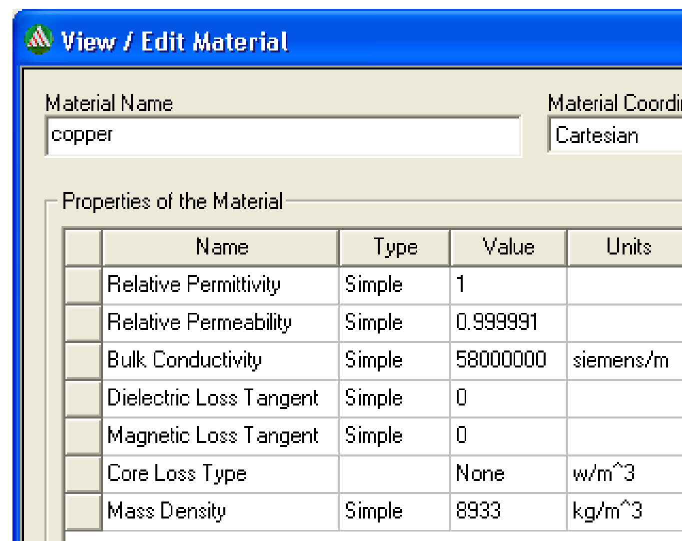
- Used in determining the current distribution in current carrying conductors and eddy currents from resulting magnetic field.
- Can be Simple or Anisotropic

Dielectric Loss Tangent:

- Defines the ratio of imaginary and real permittivities.
- Can be Simple or Anisotropic

Magnetic Loss Tangent:

- Defines the ratio of imaginary and real permeabilities.
- Can be Simple or Anisotropic



Name	Type	Value	Units
Relative Permittivity	Simple	1	
Relative Permeability	Simple	0.999991	
Bulk Conductivity	Simple	58000000	siemens/m
Dielectric Loss Tangent	Simple	0	
Magnetic Loss Tangent	Simple	0	
Core Loss Type		None	w/m^3
Mass Density	Simple	8933	kg/m^3

…Material Definition

Eddy Current Material Properties

Core Loss Type:

- Enables users to define Core Loss properties based on selected 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

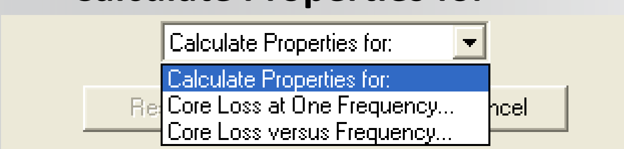
$$p_v = K_h f (B_m)^2 + K_c (fB_m)^2 + K_e (fB_m)^{1.5}$$

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

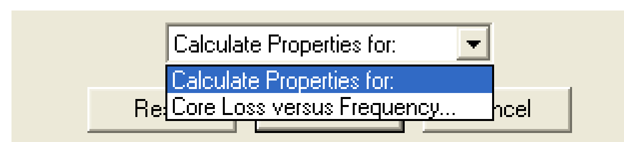
$$p_v = C_m f^x B_m^y$$

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 Definition

Eddy Current Material Properties

Core Loss at One Frequency:

- Selecting the option **Calculate Properties for "Core Loss at One Frequency"** enables users to input B-P Curve for a defined frequency point
- This option is available only for Electrical Steel
- Using specified B-P Curve, K_1 and K_2 are obtained by minimizing quadratic form

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

Where, i is the each entity in defined B-P Curve

- Eddy Current coefficient, k_c is evaluated as

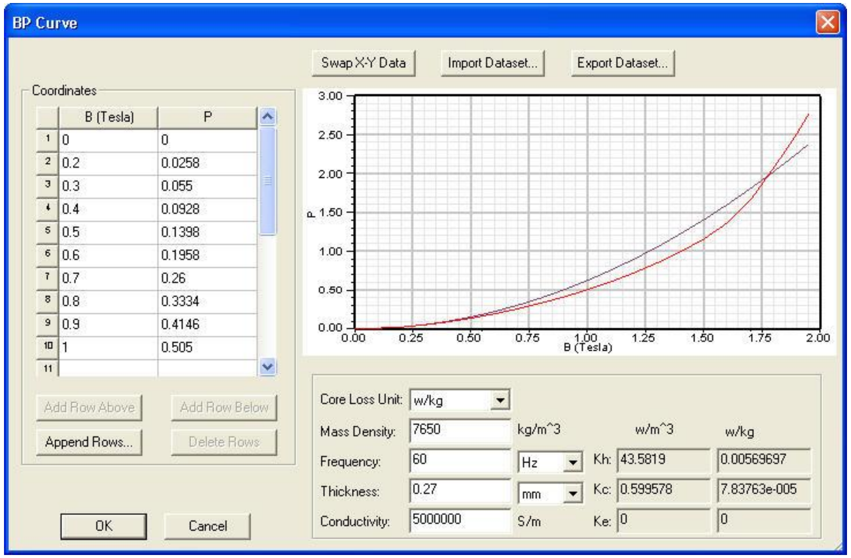
$$k_c = \pi^2 \sigma \frac{d^2}{\delta}$$

Where, σ is the conductivity and d is the thickness of one lamination sheets.

- Hysteresis Loss coefficient (k_h)and Excessive Loss Coefficients (k_e) are evaluated from K_1, K_2 and k_c

$$k_h = \frac{K_1 - k_c f_0^2}{f_0}$$

$$k_h = \frac{K_2}{f_0^{1.5}}$$



…Material Definition

Eddy Current Material Properties

Core Loss versus Frequency:

- Selecting the option **Calculate Properties for "Core Loss versus Frequency"** enables users to input B-P Curve for multiple frequency points
- **For Electrical Steel**
  - k_h,k_c and k_e are obtained by minimizing quadratic form

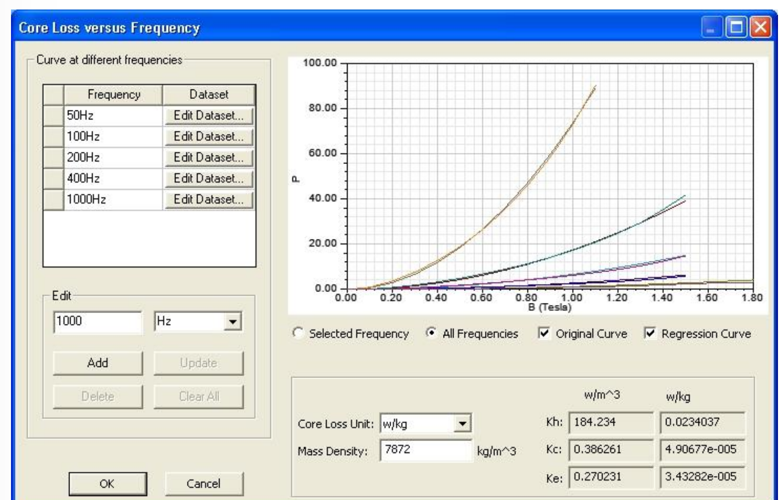
$$err(k_h, k_c, k_e) = \sum_{i=1}^{m} \sum_{j=1}^{n_i} [p_{vij} - (k_h f_i B_{mij}^2 + k_c f_i^2 B_{mij}^2 + k_e f_i^{1.5} B_{mij}^{1.5})]^2 = \min$$

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

- **For Power Ferrites**
  - Core loss coefficients C_m, *x* and *y* are obtained by minimizing the quadratic form

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

Where, $c = \log(C_m)$



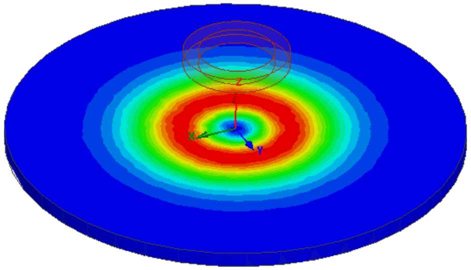
c. Boundary Conditions

Boundary Types

- All the boundaries which were discussed with the Magnetostatic solver are also applicable for Eddy Current Solver
- In addition, two other boundaries can be defined

Impedance Boundary (2D & 3D):

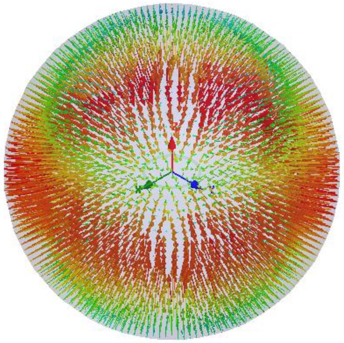
- Allows users to simulate effect of induced currents without explicitly solving and having to mesh for the skin depth
- Equivalent calculations are done on surface elements of the conductor without any flux computation for inside region of conductor
- Can be suitably used where skill depth of the conductor is two orders of magnitude smaller than its dimensions



Surface Loss Density on Impedance Boundary

Radiation Boundary (3D):

- To simulate problems that allow fields to radiate infinitely far into space
- The system absorbs the field at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure



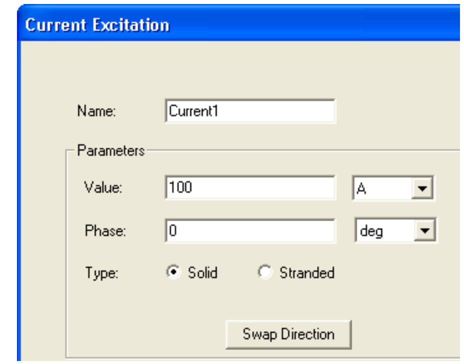
Poynting Vectors on Radiation Boundary

d. Excitations

Excitations

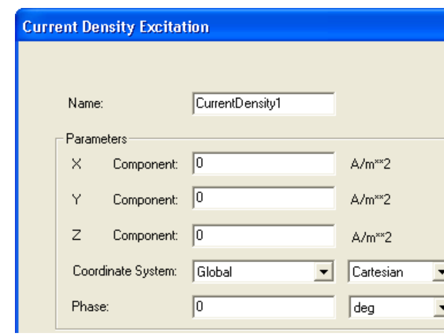
– Eddy Current solvers allows two type of excitations, Current and Current Density

Current :



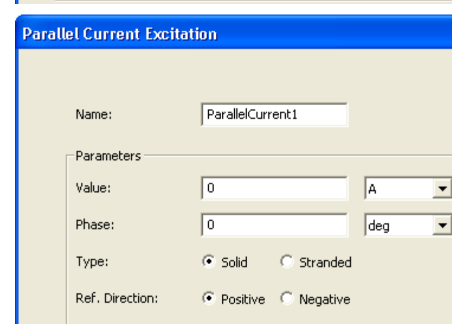
- Defines total peak current in Amp-turns through the conductor and phase
- Can be assigned to the conductor faces that lie on boundary of simulation domain or sheets that lie completely inside a conductor with a closed conduction path.
- Conductor can be defined as Solid or Stranded
- Induced eddy effects are not computed for stranded conductors

Current Density:



- Used to define known current density and phase throughout an object.
- In 3D, this definition should be accompanied with Current Density Terminal definition
- Current Density defined using X,Y and Z components of selected CS

Parallel Current (2D only):

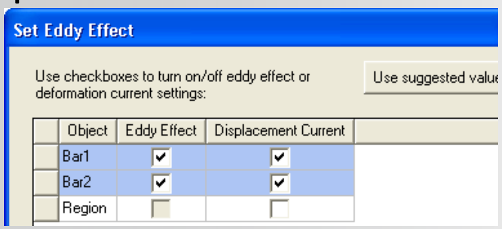


- Used to define total AC current in a parallel conduction path consisting of at least two conductors
- For Parallel Solid Conductors, total current split will be based on field solutions including eddy effects
- For Parallel Stranded Conductors, total current split will be based on relative areas of selected conductors

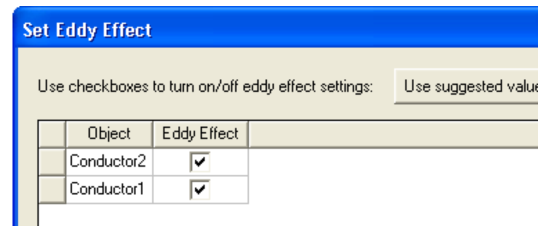
…Excitations

Setting Eddy Effects (Calculating Eddy Currents)

- Eddy Effects can be set from the menu item ***Maxwell 3D/2D → Excitations → Set Eddy Effects***
- Induced eddy current calculations can be enabled or disabled for an object
- Displacement current calculations can also be set for the 3D eddy current solver



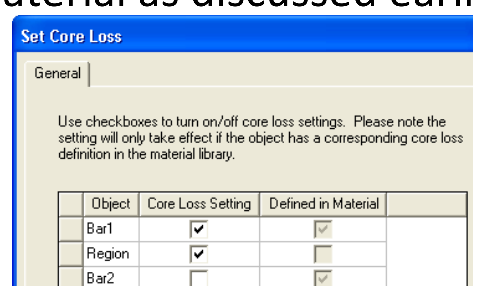
Maxwell 3D



Maxwell 2D

Setting Core Loss

- Core Loss calculations can be assigned from the menu item ***Maxwell 3D/2D → 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 includes Eddy losses
- Core Loss properties must be defined for the material as discussed earlier



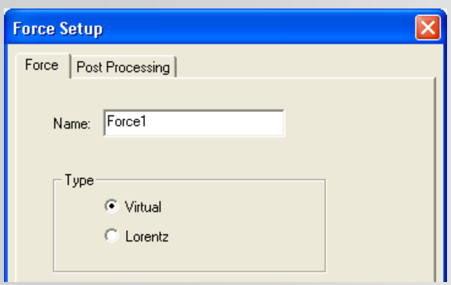
e. Parameters

Parameters

- All the parameters that are available with Magnetostatic Solver are also available with Eddy Current Solver
- A parameter cane added by selecting the required object and selecting menu item ***Maxwell 3D/2D → Parameters → Assign***

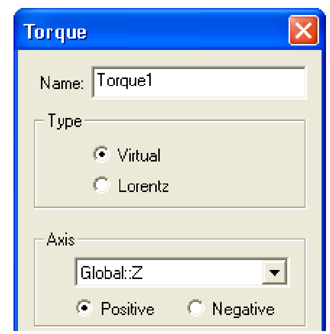
Force:

- Calculates force acting on assigned object
- Force can be Virtual or Lorentz
- Lorentz can not be used for magnetic material



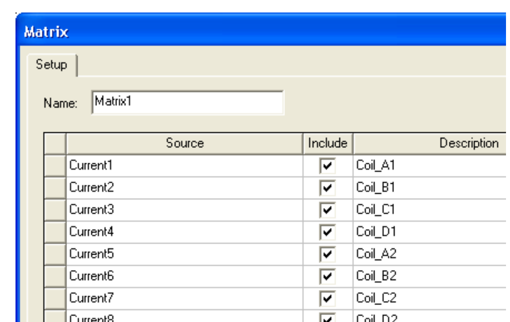
Torque:

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



Matrix:

- Calculates Inductance, resistance matrix
- Reported resistance values are AC resistances which include skin and proximity effects
- Postprocessing option is not available with Eddy Current matrix



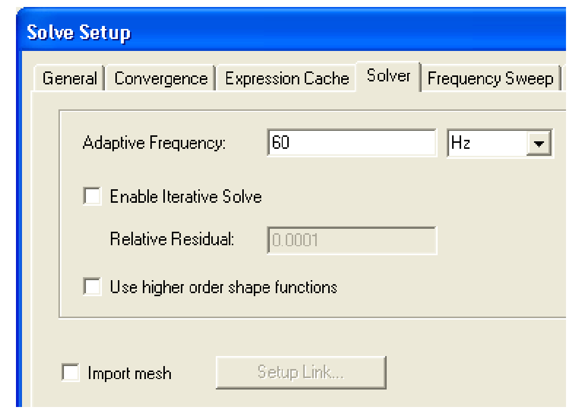
f. Analysis Setup

Solution Setup

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

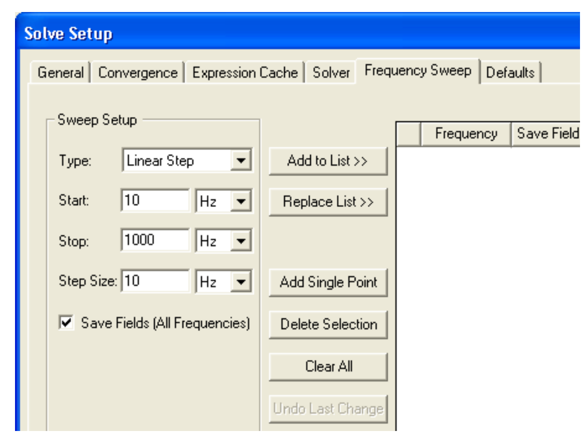
Solver Tab

- **Adaptive Frequency:** Defines the frequency at which the mesh is constructed and adapted, and at which solution is obtained
- **Enable Iterative Solve:** Enables ICCG solvers (Direct is the default).
- **Use higher order shape functions:** Enables higher order option gains better accuracy for eddy current regions.
- **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 or whether to save fields at specific frequencies in the list.



g. Solution Process

Eddy Current Solution Process

- Like the Magnetostatic Solver, the solution process in the Eddy Current solver is automated as shown in diagram below
- A Solution process can be launched from the menu item ***Maxwell 3D/2D → Analyze All***