

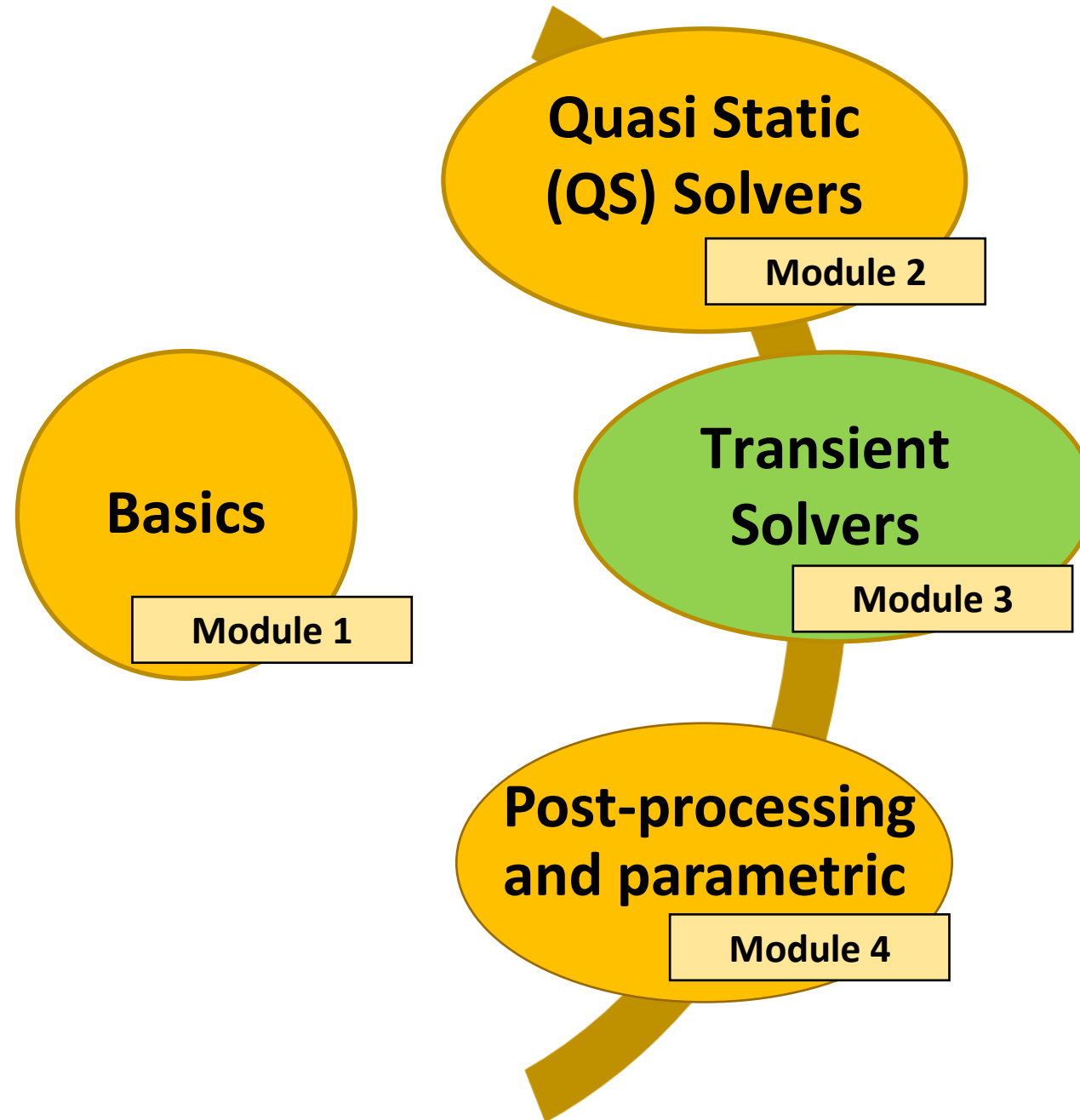
## Module 03: Transient Solvers

Release 2020R2

# Overview

- **Magneto- and Electrotransient Solvers**
- **Eddy effects and losses**
- **Sources, windings and external circuit**
- **Mesh operations and linking**
- **Workshop 3.1: 2D Magnetotransient Analysis including mesh operations**
- **Workshop 3.2: 3D Magnetotransient Analysis including mesh operations**

# Overall Process



# Magnetic Transient Solver

- **Magnetic Transient Solver:**

- The Magnetic Transient Solver computes time varying magnetic fields in time domain and solves for instantaneous magnetic fields at each time step
- The source of the time varying magnetic field can be:
  - Arbitrary time-varying voltage or current sources, eventually placed in an external circuit fully coupled with Field quantities
  - Moving sources (e.g. permanent magnets)
- Magnetic Transient solver does not use Adaptive mesh refinement and thus mesh operations or a Linked Mesh is required to be defined to obtain a sufficient mesh.

- **Magnetic Transient Equations**

- Following equations are solved by the Transient solver:

$$\nabla \times \frac{1}{\sigma} \nabla \times \mathbf{H} = - \frac{\partial \mathbf{B}}{\partial t}$$

$$\nabla \cdot \mathbf{B} = 0$$

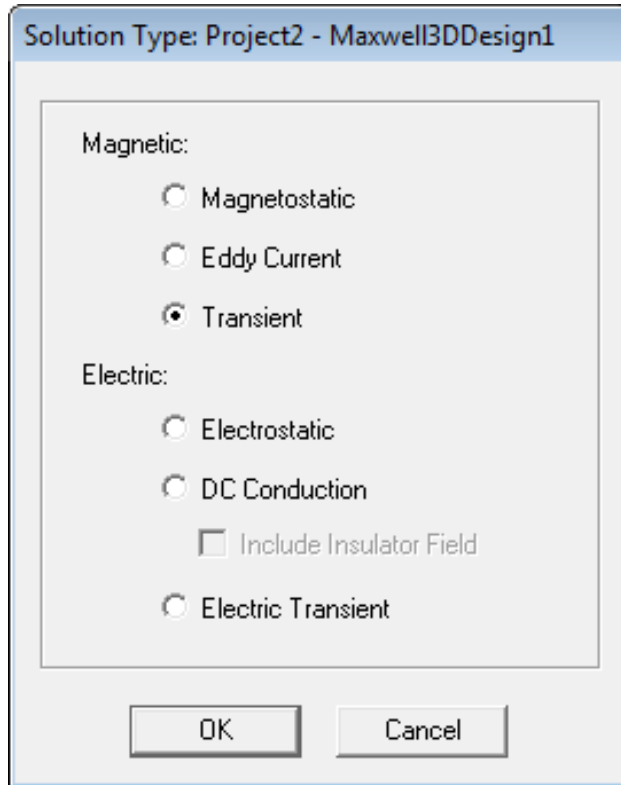
Maxwell 3D

$$\nabla \times \frac{1}{\mu_0 \mu_r} \nabla \times \mathbf{A} = \mathbf{J}_S - \sigma \frac{\partial \mathbf{A}}{\partial t} - \sigma \nabla \Phi + \nabla \times \mathbf{H}_c + \sigma \mathbf{v} \times \nabla \times \mathbf{A}$$

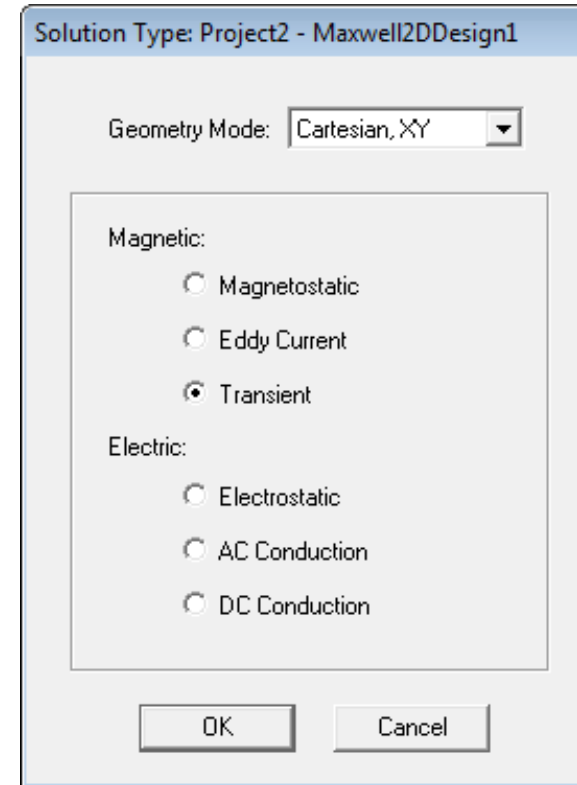
Maxwell 2D

# Selecting the Magnetic Transient Solver

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



Maxwell 3D



Maxwell 2D

# Material Definition

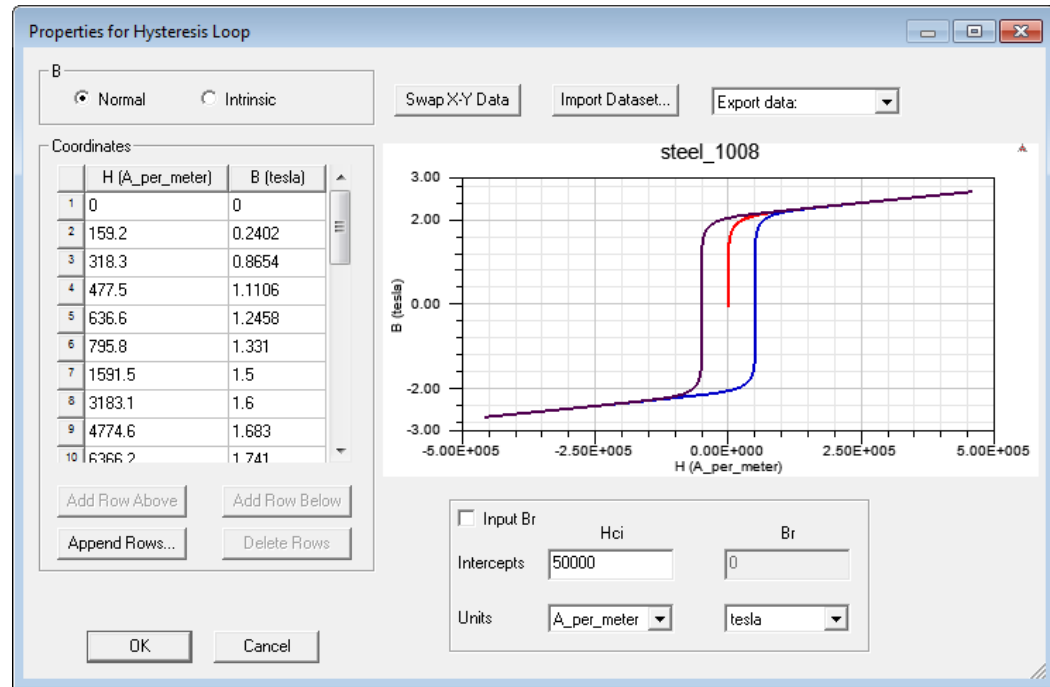
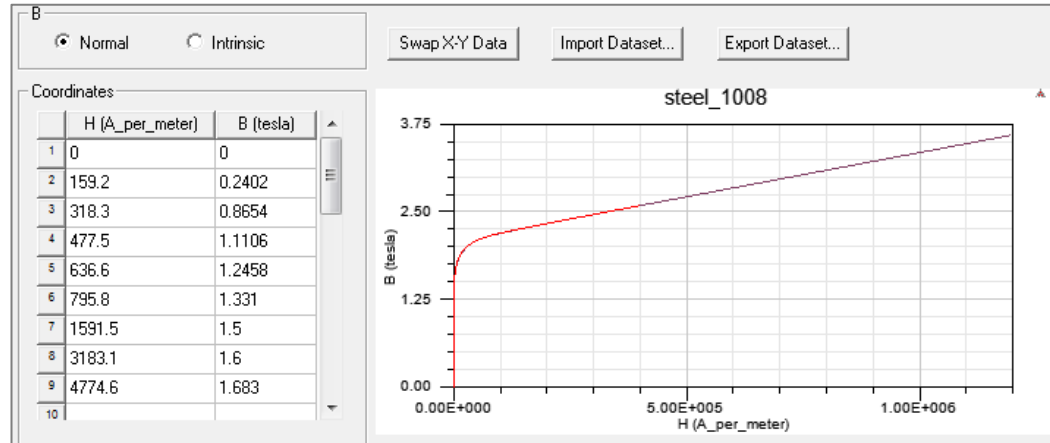
- **Transient Material Properties**
  - In a Magnetostatic simulation, the following parameters may be defined for a material
  - **Relative Permeability:**
    - Relative permeability can be either Simple (linear  $\mu_r$ ) or Nonlinear (BH Curve) or/and anisotropic. The hysteresis loop can also be represented
  - **Bulk Conductivity:**
    - Used to determine the current distribution in conductors
    - Can be Simple or Anisotropic
  - **Magnetic Coercivity:**
    - Used to define permanent magnets properties.
    - Requires magnitude and direction specification.
    - Direction specified is with respect to Orientation CS of bodies to which material is assigned
  - **Core Loss Type:**
    - Can be Electrical Steel, Power Ferrite, Hysteresis Model or None
    - Allows to define Core Loss coefficients for loss calculation
  - **Composition:**
    - Can be Solid or Lamination
    - Requires specification of direction normal to laminated pack and stacking factor

# Material Definition

- Transient Material Properties

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Nonlinear	B-H Curve...	
Bulk Conductivity	Simple	2000000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Core Loss Model		Electrical Steel	w/m^3
- Kh	Simple	190	
- Kc	Simple	2	
- Ke	Simple	0.1	
- Kdc	Simple	0	
Mass Density	Simple	7872	kg/m^3
Composition		Lamination	
- Stacking Factor	Simple	0.95	
- Stacking Direction		V(1)	



# Material Definitions

- Transient Material Properties

- Core Loss Type

- Core Loss Type can be either Electrical Steel, Power ferrite or Hysteresis Model
- Core Loss Coefficients will change according to selected Core Loss type

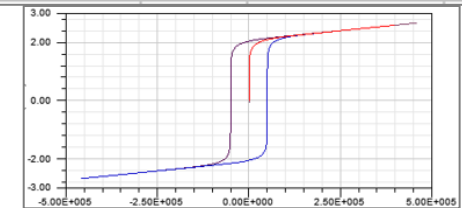
Core Loss Type		Electrical Steel	w/m <sup>3</sup>
- Kh	Simple	0	
- Kc	Simple	0	
- Ke	Simple	0	
Mass Density	Simple	8055	kg/m <sup>3</sup>

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

Core Loss Model		Hysteresis Model	w/m <sup>3</sup>
- Intrinsic Coercivity Hci	Simple	0	A_per_meter
- Remanence Br	Simple	0	tesla
- Kc	Simple	0	

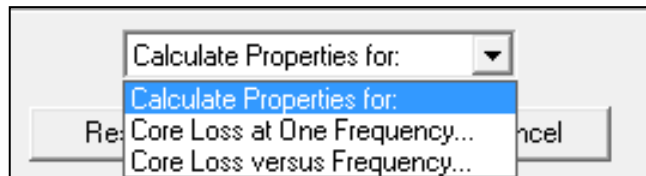
$$p_v = K_h f (B_m^2) + K_c (f B_m)^2 + K_e (f B_m)^{1.5}$$

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

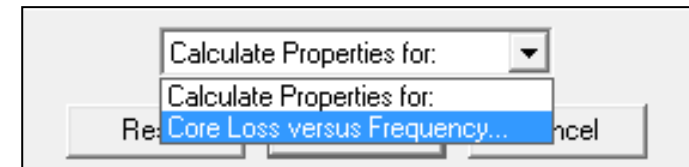


- Core Loss Coefficient Calculations

- 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

$$err(K_1, K_2) = \sum_i [P_{vi} - (K_1 B_{mi}^2 + K_2 B_{mi}^{1.5})]^2 = \min \quad \text{Where } i \text{ represents each point of defined B-P Curve}$$

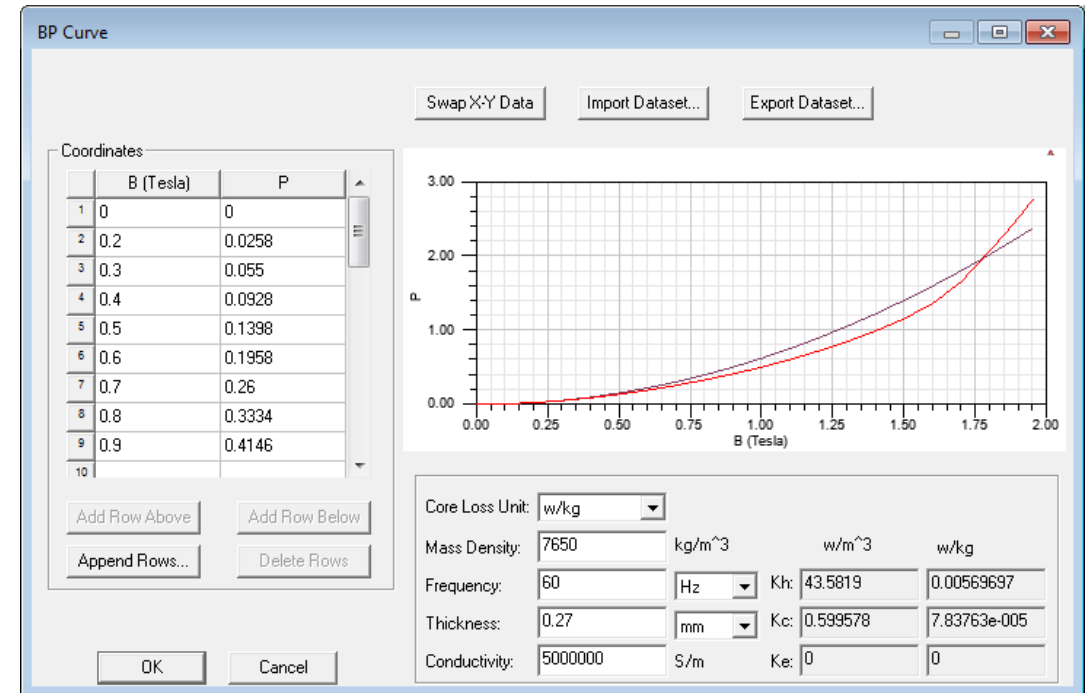
- Eddy Current coefficient,  $K_c$  is evaluated as

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

- Where  $\sigma$  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  $K_c$

$$K_h = \frac{K_1 - K_c f_0^2}{f_0}$$

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



# Material Definitions

- Transient 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  $K_h$ ,  $K_c$  and  $K_e$  are obtained by minimizing quadratic form

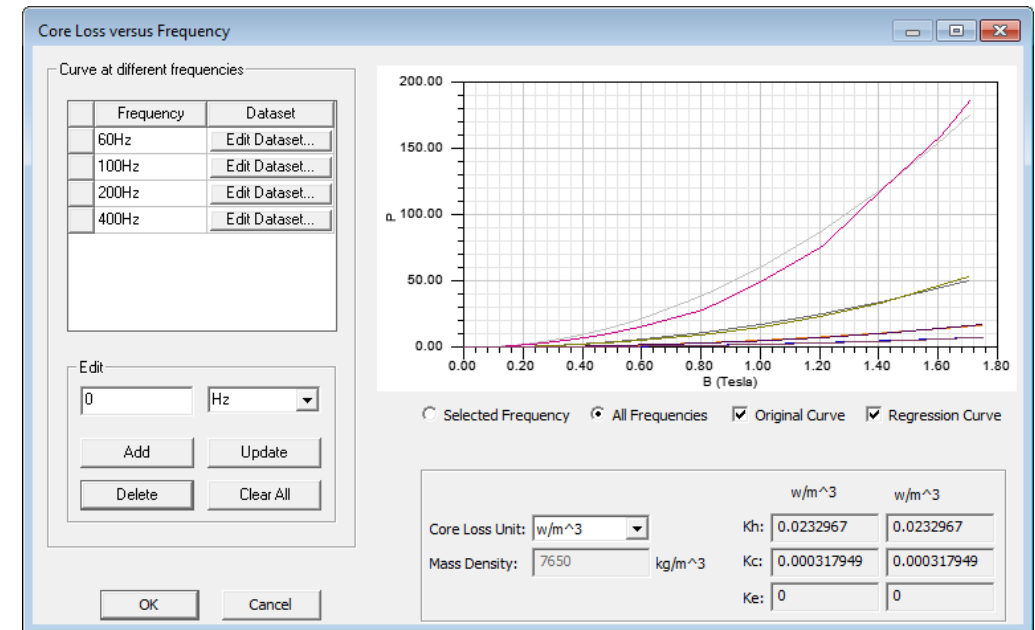
$$err(K_h, K_c, K_e) = \sum_{i=1}^m \sum_{j=1}^{n_i} [P_{vij} - (K_h f_i B_{mij}^2 + K_c f_i^2 B_{mij}^2 + K_e f_i^{1.5} B_{mij}^{1.5})]^2 = \min$$

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

- For Power Ferrites  $C_m$ ,  $x$  and  $y$  are obtained by
- minimizing quadratic form

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

$$c = \log(C_m)$$



# Boundary Conditions (BC)

- Assigning Boundary Conditions in 3D
  - Boundary conditions define behavior of the magnetic field at problem region limits
  - A boundary can be assigned to a face (Edge in 2D) from menu item *Maxwell 3D* → *Boundaries* → *Assign* and select the required boundary assignment
- Boundary Types
  - Following Boundary types are available for Transient Solvers

## – Maxwell 3D:

- Default: Natural or Neumann
- Zero Tangential H-Field
- Insulating
- Symmetry
- Master/Slave

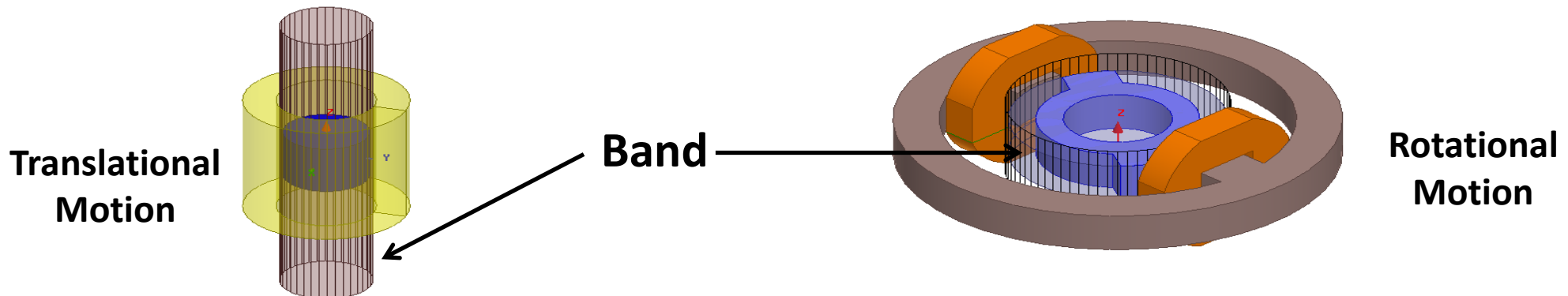
## – Maxwell 2D:

- Vector Potential
- Balloon
- Symmetry
- Master/Slave

- The behavior of each boundary remains the same as discussed for Magnetostatic Solver

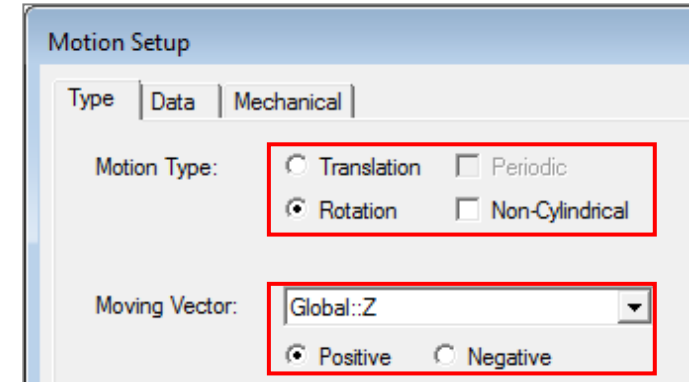
# Motion Setup

- Motion Setup
  - Time varying magnetic fields solved by the Transient solver can also result from moving excitations or permanent magnets
  - Maxwell allows users to define motion types as **translation, rotation, or non-cylindrical rotation**
- Requirements for Motion Setup
  - All moving objects should be physically (geometrically) separated from stationary objects and not overlapping them during the movement
  - Moving object should be enclosed by a so-called **Band** object which separates moving parts and stationary part of the geometry. The band object should be segmented and not a true surface
  - In 3D, it is recommended to add a second “dummy” object enclosing moving parts if those are more than 1. That “dummy” object (often referred as **Inner Band**) lets the Band “see” only one rigid moving object. It must be smaller than the Band.

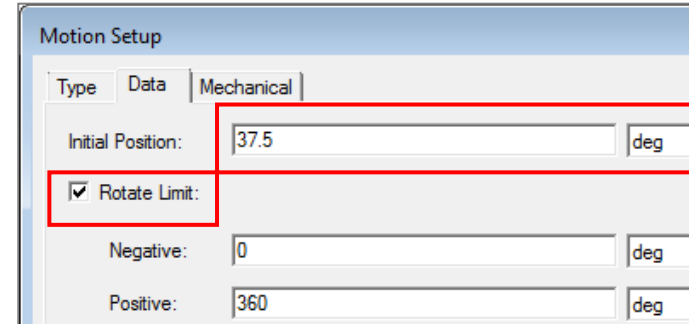


# Motion Setup

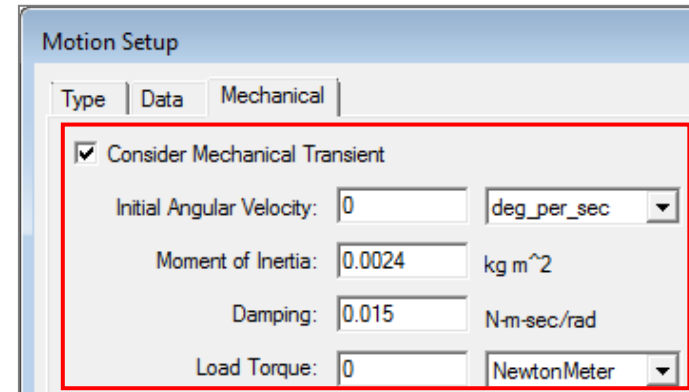
- Assign Band
  - Select the object enclosing all moving parts then *Maxwell 2D/3D* → *Model* → *Motion Setup* → *Assign Band*
  - The object is set as a Band and all enclosed objects are set as Moving
- Motion Assignment
  - **Type Tab**
    - Motion Type: Defines motion – Translation or Rotation
    - Rotational Axis for Rotational or Moving Vector for translational set direction of motion
  - **Data Tab**
    - Initial Position: Sets initial position at t=0
    - Motion Limit: Sets rotational or translational limits of motion
  - **Mechanical Tab**
    - Velocity: Specifies translational or rotational velocity. If Consider Mechanical Transient is unchecked, constant velocity is assigned
    - Consider Mechanical Transient: Maxwell computed forces along with input mechanical forces are used to determine object motion



Motion Setup dialog, Type Tab. The 'Mechanical' tab is selected. The 'Motion Type' section has 'Rotation' selected with a radio button, and 'Translation', 'Periodic', and 'Non-Cylindrical' are unselected. The 'Moving Vector' dropdown is set to 'Global::Z', and the 'Positive' radio button is selected.



Motion Setup dialog, Data Tab. The 'Initial Position' is set to 37.5 deg. The 'Rotate Limit' checkbox is checked. The 'Negative' limit is 0 deg and the 'Positive' limit is 360 deg.



Motion Setup dialog, Mechanical Tab. The 'Consider Mechanical Transient' checkbox is checked. The 'Initial Angular Velocity' is 0 deg\_per\_sec. The 'Moment of Inertia' is 0.0024 kg m^2. The 'Damping' is 0.015 N-m-sec/rad. The 'Load Torque' is 0 NewtonMeter.

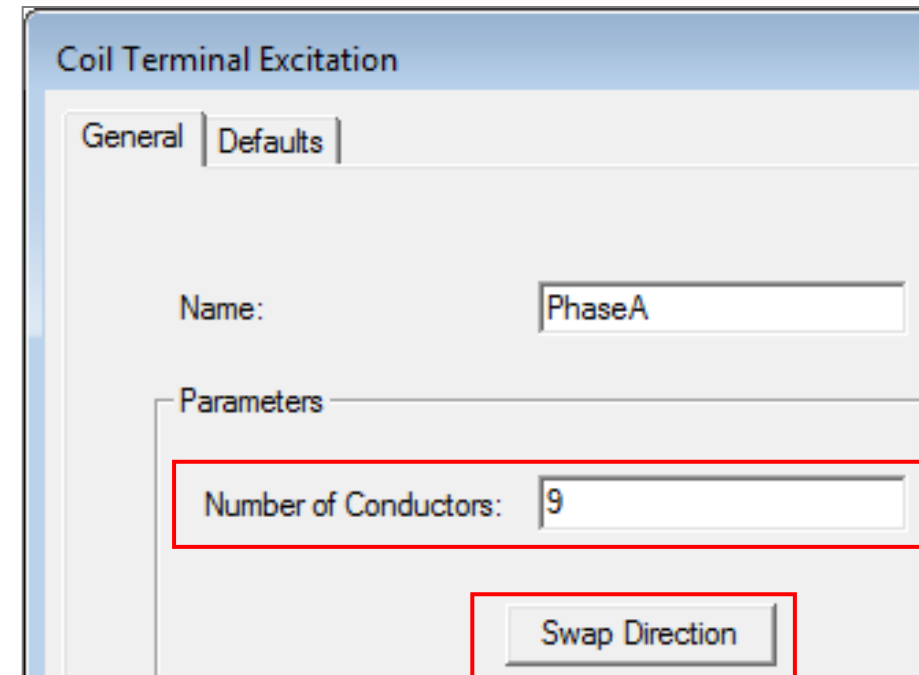
# Excitations

- Excitation

- Excitation assignments in the Transient solver are different from the Static Solvers
- The transient solver requires specific information about the coil (such as the number of turns, parallel paths, and resistance) and the type of Excitation (Voltage, Current, or External)
- In general, Excitations require a definition of a Coil Terminal assignment, which is then assigned to Winding definition.
- Permanent Magnets also qualify as excitations, and allow for a transient solution (e.g. with motion)

- Coil Terminal (“Coil” in 2D)

- Coil terminals designate the cross-sectional faces of the 3D conductors.
- Terminal assignment can be internally located 2D sheet for closed loop or the boundary faces of an open Conduction path
- The Coil Terminal defines the direction of current and number of conductors in a Conduction path
- Coil Terminal can be added by selecting the 2D sheet object and menu *Maxwell 2D/3D → Excitation → Assign → Coil Terminal*



# 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
- Current can be constant or a function of time

- 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
- Voltage can be constant or function of time

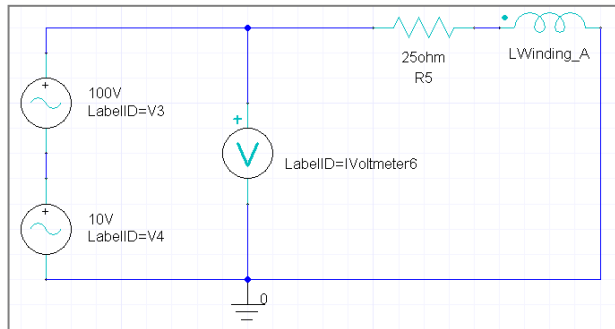
The screenshot shows the 'Winding' dialog box with the 'General' tab selected. The 'Name' field contains 'Phase\_A'. The 'Parameters' section is highlighted with a red box and contains the following settings: 'Type' is set to 'Current', 'Solid' is unselected and 'Stranded' is selected, and the 'Current' field contains the mathematical expression  $100 \cdot \sin(2 \cdot \pi \cdot 50 \cdot \text{Time} + \text{Thet})$ .

The screenshot shows the 'Winding' dialog box with the 'General' tab selected. The 'Name' field contains 'WindingA'. The 'Parameters' section is highlighted with a red box and contains the following settings: 'Type' is set to 'Voltage', 'Solid' is unselected and 'Stranded' is selected, 'Initial Current' is 0 with units 'A', 'Resistance' is 1 with units 'mOhm', 'Inductance' is 0 with units 'mH', 'Voltage' is  $100 \cdot \cos(2 \cdot \pi \cdot 60 \cdot \text{time})$ , and 'Number of parallel branches' is 1.

# Excitations

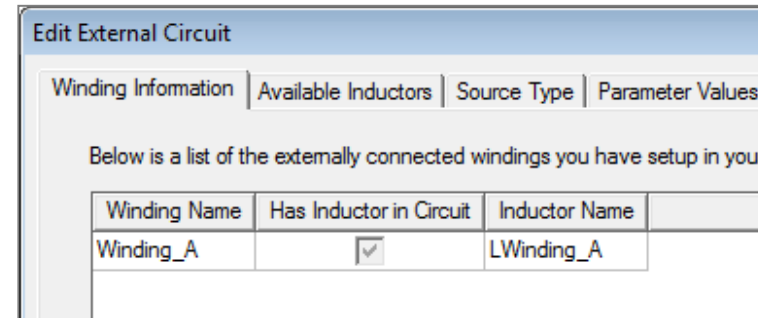
## - External

- 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

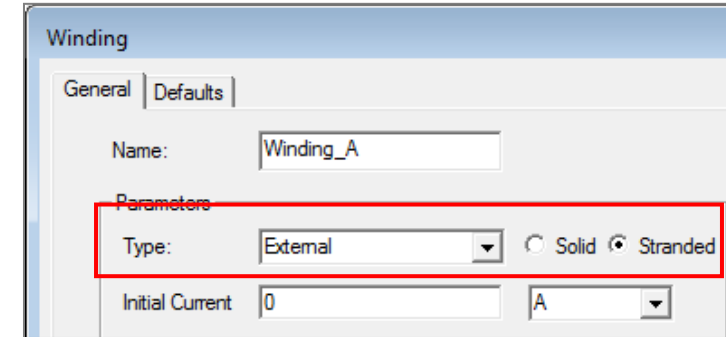


Maxwell Circuit Editor

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*



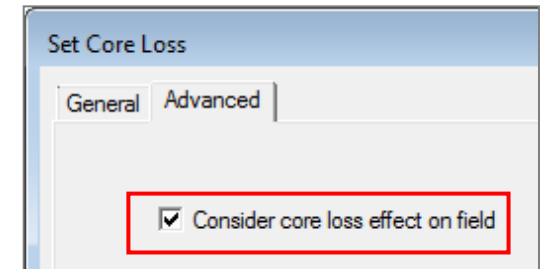
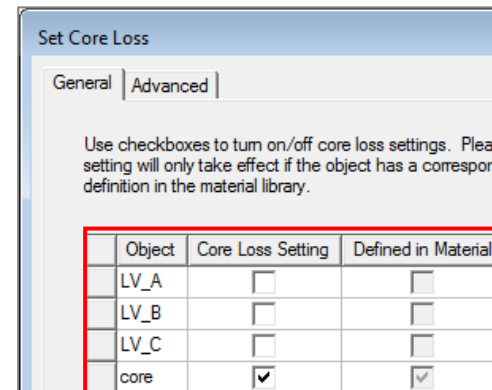
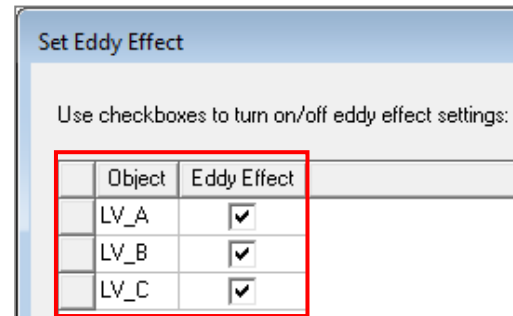
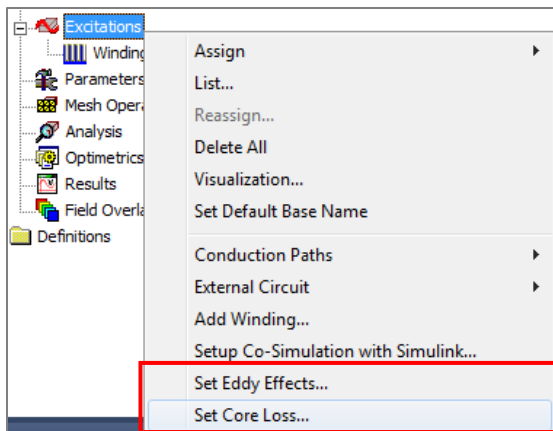
# Excitations

- **Setting Eddy Effects**

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

- **Setting Core Loss**

- Core Loss calculations can be set from menu item *Maxwell 2D/3D* → *Excitations* → *Set Core Loss*
- If Core Loss is enabled for an object, Eddy effects should be disabled for it as Core loss already includes Eddy losses
- Core Loss properties must be defined in material definition
- Advanced tab enables to consider impact of Core loss on magnetic fields

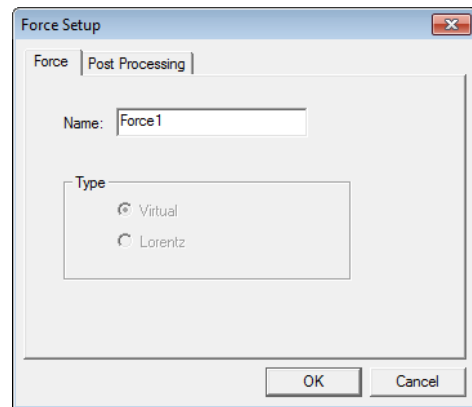


# Parameters

- Parameters
  - Two parameters can be assigned for Transient solver: Force and Torque
  - Parameters can be added from menu item *Maxwell 3D/2D* → *Parameters* → *Assign*

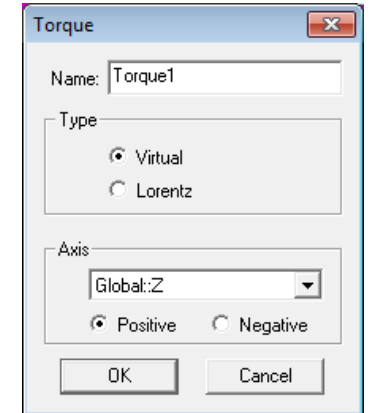
## – Force

- Calculates force acting on selected objects
- Can be only Virtual



## – Torque

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



**Note:** *If motion setup is assigned, force (translational motions) and torque (rotational motions) calculations are assigned by default for the moving objects and computed at all time steps. User can plot these values using XY Plots*

# Design Settings

- Design Settings

- Design Settings can be accessed from menu item *Maxwell 3D/2D* → *Design Settings*

- Symmetry Multiplier

- When partial geometry is modeled using Symmetry or Master/ Slave boundary, Symmetry Multiplier is required to scale results
- All specified Input parameters and reported output parameters (except field calculator quantities) are scaled based on assigned Symmetry Multiplier

- Model Depth (2D XY Designs)

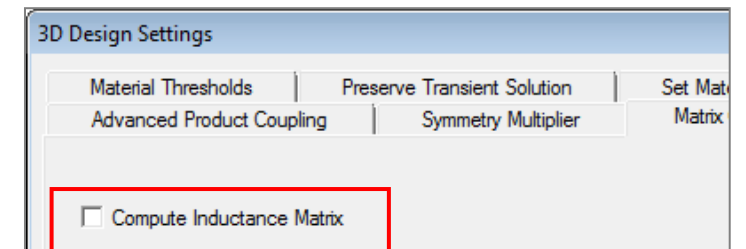
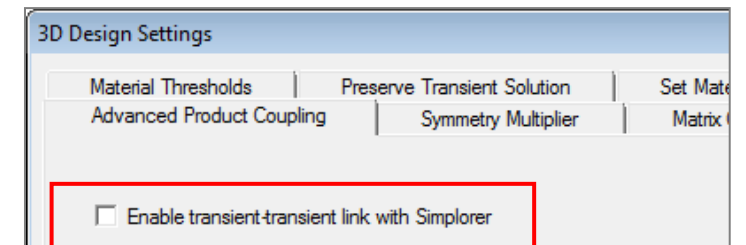
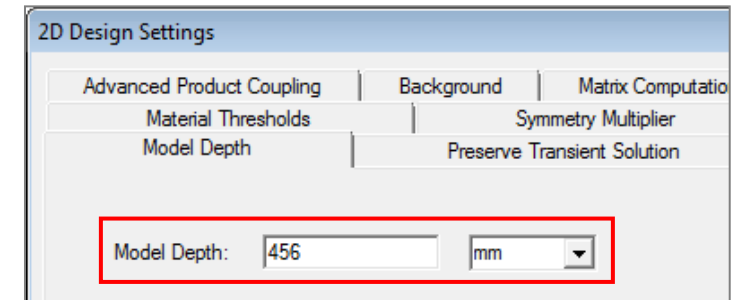
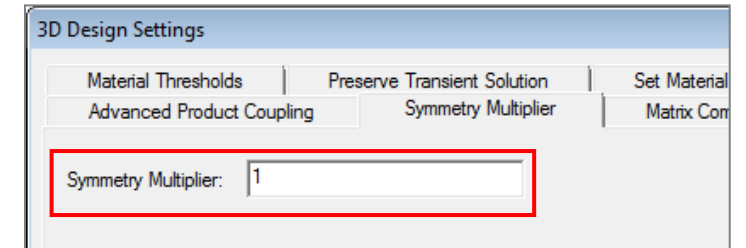
- Allows to set the length of the model perpendicular to XY plane
- Outputs are scaled based on specified length

- Advanced Product Coupling Tab

- Enables Maxwell-Simplorer Transient cosimulation

- Matrix Computation Tab

- Enables Inductance matrix computation for Transient Solver



# Analysis Setup

- Solution Setup

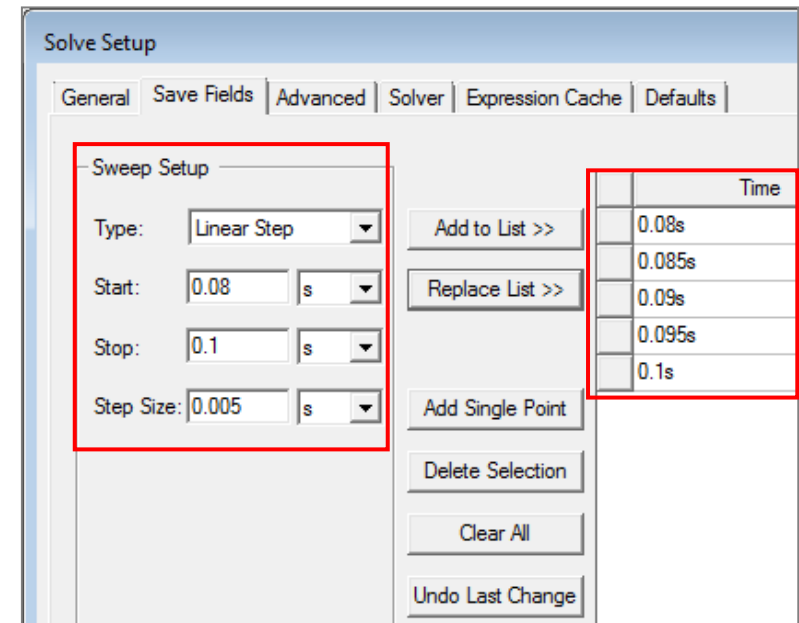
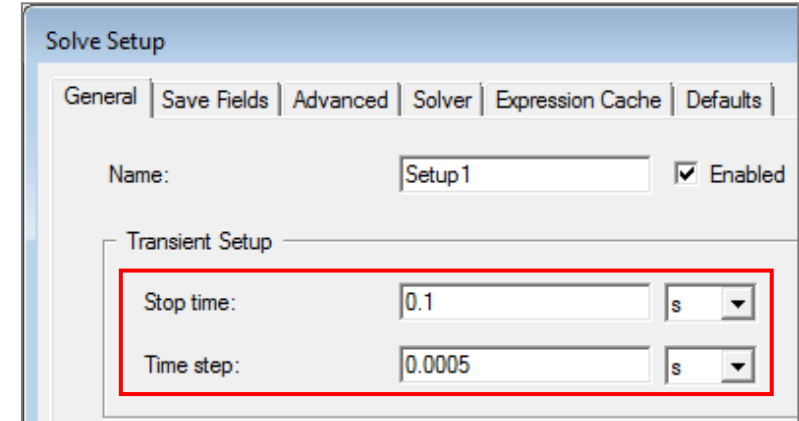
- The solution setup defines the parameters used for solving the simulation
- Can be added from menu item *Maxwell2D/3D* → *Analysis Setup* → *Add Solution Setup*

- General Tab

- Name: sets the Name of the setup. Multiple setups can be present in the same design
- Stop Time: Defines time at which simulation will stop
- Time step: Discrete steps of time used in transient simulation

- Save Fields Tab

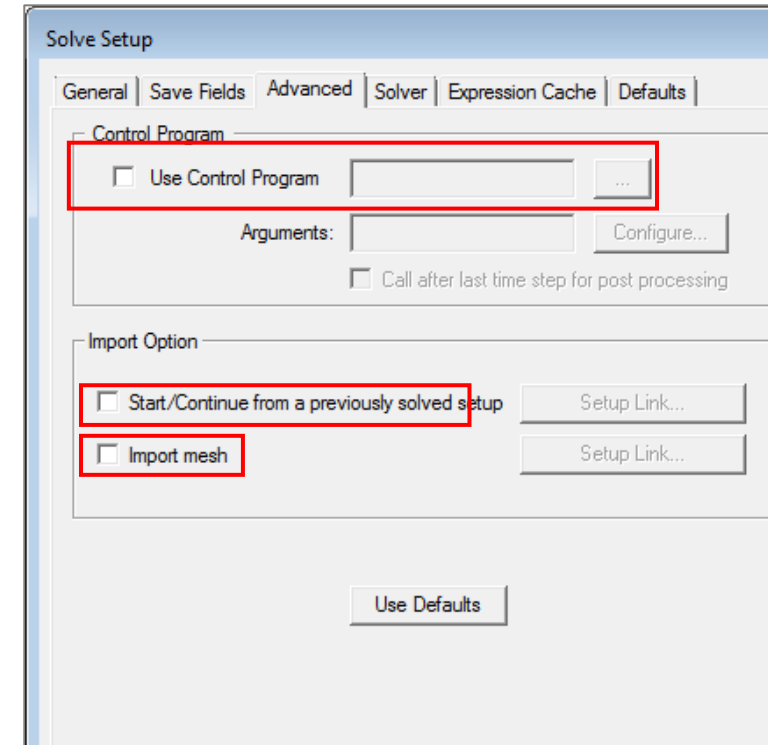
- Sweep Setup (Type, Start, Stop, Step): Defines time steps at which fields are saved
- Add to List >>: activates the previous sweep settings fulfilling the list



# Analysis Setup

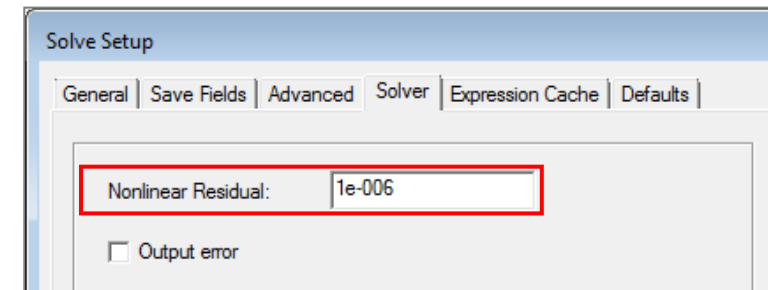
- **Advanced Tab**

- **Control Program:** Used to dynamically adjust parameters and control the simulation. It requires an external code.
- **Continue from a previously solved setup:** Enables to continue the solution from a linked Maxwell design setup
- **Import Mesh:** Allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry



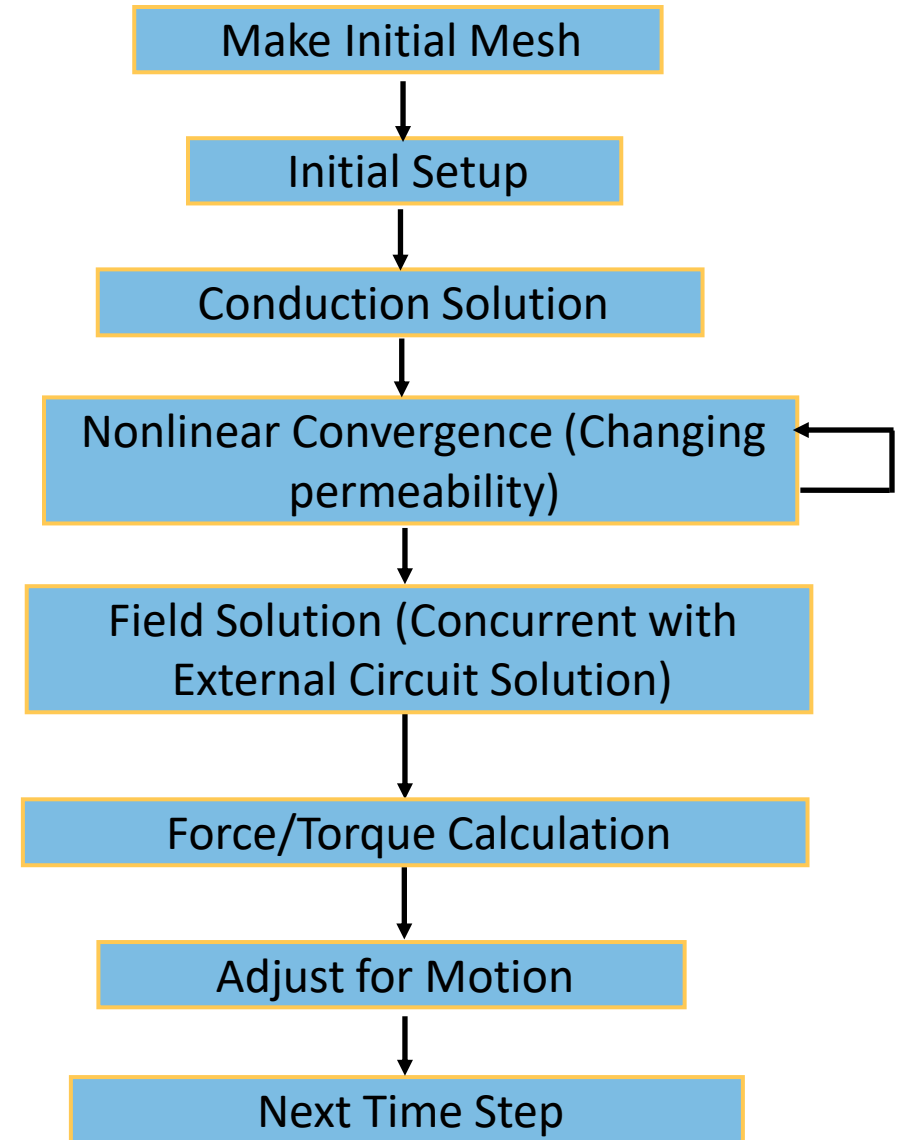
- **Solver Tab**

- **Nonlinear Residual:** Defines the error tolerance in finding the operating points along the nonlinear B-H curve



# Solution Process

- Transient Solution Process
  - A Solution process can be launched from Project Manager window  
*RMB on Setup1 → Analyze*



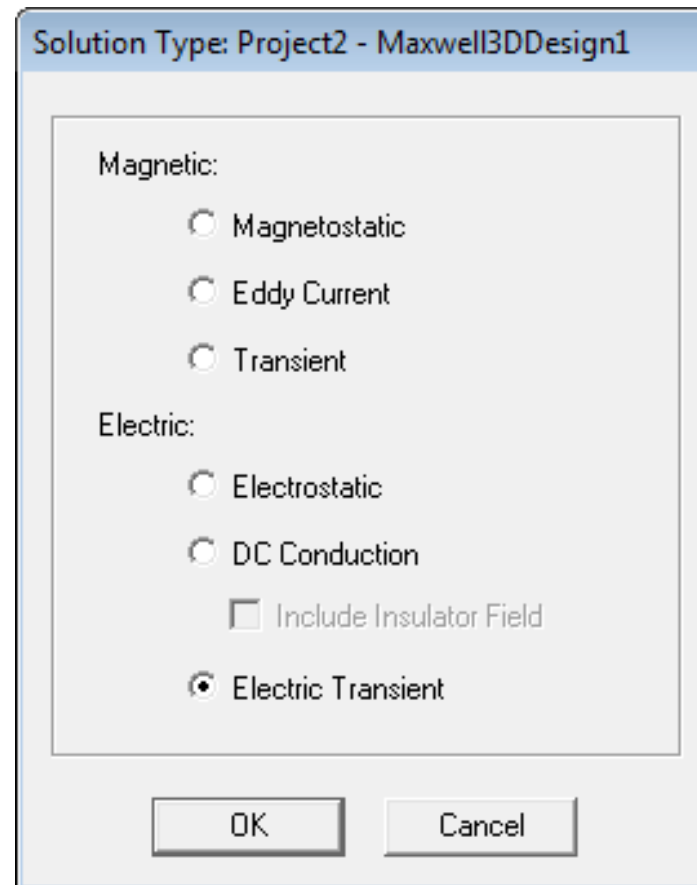
# Electric Transient Solver

- **Electric Transient Solver:**
  - **Electric Transient Solver computes time varying electric fields within lossy conductors and insulators**
  - **The source of the Time varying electric field can be a time-varying potential, charge distribution or applied currents**
  - **Electric Transient solver is available only with Maxwell 3D**
  - **Electric Transient solver uses adaptive time stepping approach where time step size changes based on the solution convergence**
  - **Electric Transient solver does not use Adaptive mesh refinement and thus mesh operations are required to obtain a better mesh**
- **Electric Transient Equations**
  - **Following equations are solved with Electric Transient solver**

$$-\nabla \cdot \left( \epsilon \nabla \frac{\partial \Phi}{\partial t} \right) - \nabla \cdot (\sigma \nabla \Phi) = 0$$

# Overview

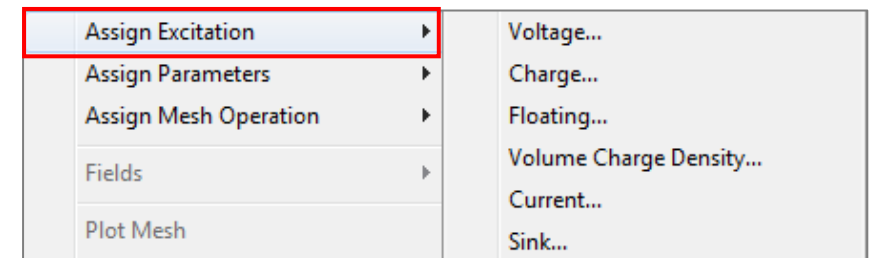
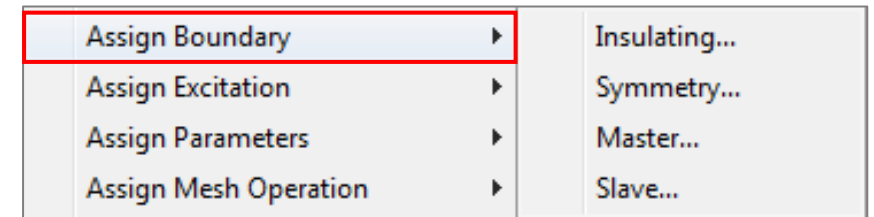
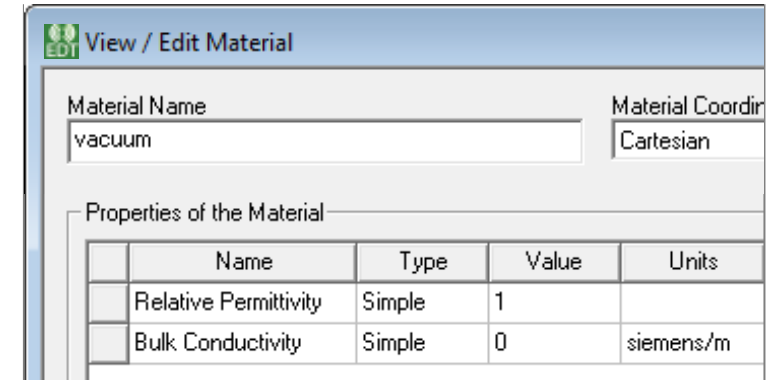
- Defining Electric Transient Solver
- Specify the Electric Transient Solver by selecting the menu item *Maxwell 3D* → *Solution Type*
- In Solution type window, select *Electric* → *Electric Transient* and press OK





# Material Definition, Boundary Conditions and Excitations

- **Material Properties**
  - **Material properties for Electric Transient Solver are the same as for Electrostatic Solver (but fields are solved in lossy conductors as well as dielectrics)**
- **Boundary Types**
  - **Following Boundary types which were discussed with Electrostatic solver are also available for Electric Transient Solvers:**
    - **Default: Natural or Neumann**
    - **Insulating**
    - **Symmetry**
    - **Master/Slave**
- **Excitations**
  - **All the excitations valid in Electrostatic and DC conduction solver can be assigned with Electric Transient Solver as a function of time**
    - **Voltage**
    - **Charge**
    - **Floating**
    - **Volume Charge Density**
    - **Current**
    - **Sink**



# Analysis Setup

- Solution Setup

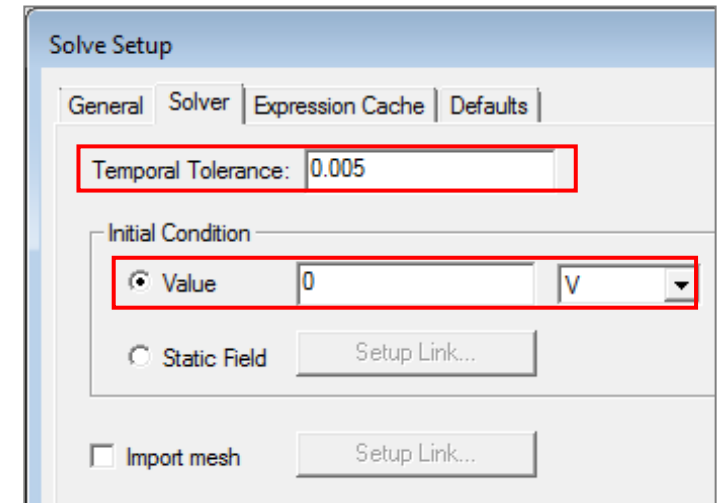
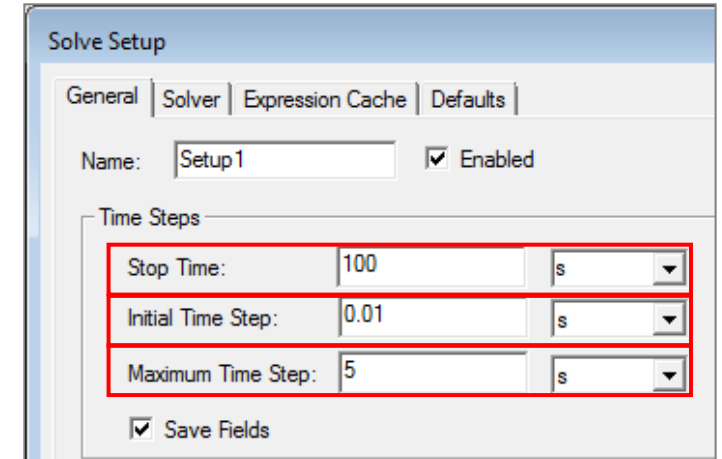
- A Solution Setup can be added from menu item *Maxwell 3D* → *Analysis Setup* → *Add Solution Setup*

- **General Tab**

- **Name:** sets the Name of the setup. Multiple setups can be present in the same design
- **Stop Time:** Defines time at which simulation will stop
- **Initial Time Step:** Defines time step size used at solution start
- **Maximum Time Step:** Defines maximum size of the time step used by solver
- **Save Fields:** Enables to save fields at each time step

- **Solver Tab**

- **Temporal Tolerance:** Defines tolerance in time sizes
- **Initial Condition:** Can be a preset Value or the solution of a Static Field. Setup Link tab can be used to link a static field
- **Import Mesh:** Allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry



# Meshing

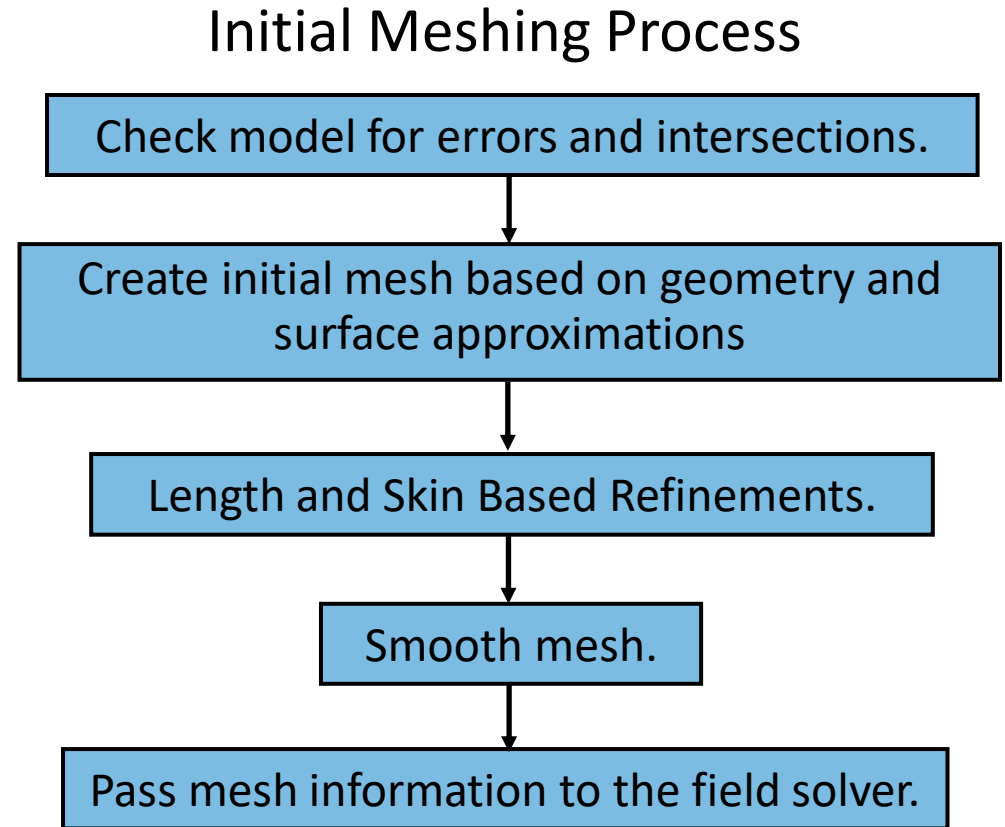
**Ansys**

# Maxwell Meshing

- **About Mesh**
  - Maxwell uses the Finite Element Method (FEM) to solve Maxwell's equations.
  - In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into basic building blocks (triangles in 2D, tetrahedra in 3D)
  - The assembly of all triangles or tetrahedra is referred to as the finite element mesh or simply the **mesh**
  - Mesh plays important role in accuracy of the computed results and thus requires higher mesh resolution in regions where field fields are of interest
- **Meshing in Maxwell**
  - Maxwell meshes all model Objects in the geometry automatically before solution process starts
  - In Maxwell's Static Solvers, the mesh is automatically refined to achieve the required level of accuracy in field computation. This is referred as **Adaptive Mesh Refinement**
  - Maxwell also offers wide range of mesh operations which can be used to achieve the required accuracy

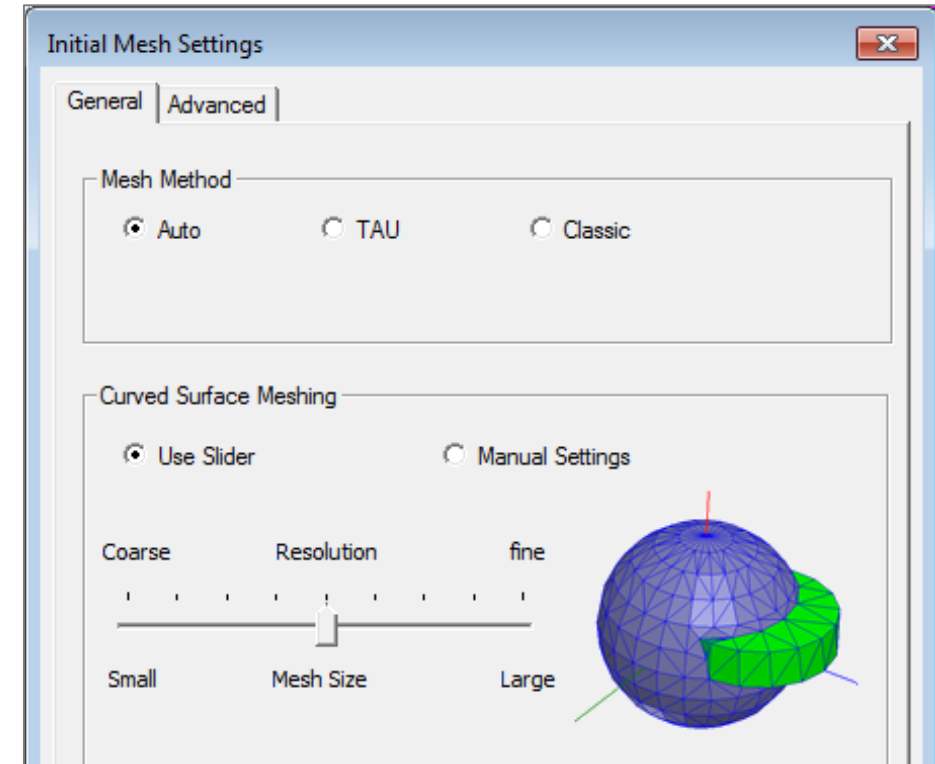
# Initial Mesh

- Initial Mesh
  - When the Solution process is initiated, Maxwell uses an initial mesh to perform field calculations
  - Initial mesh is automatically created by Maxwell without any instructions from users prior to performing field calculations



# Initial Mesh Settings

- Initial Mesh Settings
  - Default Initial Mesh Settings are appropriate for most geometries
  - Initial Mesh settings can be accessed from menu item **Maxwell 3D** → **Mesh Operations** → **Initial Mesh Settings**
- Meshing Methods
  - **Auto (3D Only):**
    - Default meshing method in 3D, allowing Maxwell to automatically select the appropriate mesher based on geometry
  - **Tau Mesh:**
    - Well suited for curved surfaces and transient with movement analysis. The mesher starts meshing the Band and expands to the surrounding geometries
    - It creates a fine mesh at the beginning relaxing it when smoothing
  - **Classic Mesh:**
    - Might not be suitable for Curved surfaces and requires geometry segmentation but works better for thin, flat objects
    - It creates a coarse mesh at the beginning, refining it while smoothing



# Maxwell Adaptive Meshing

- Adaptive Meshing

- For most of the cases, initial mesh is very coarse and more or less uniform in size throughout the region
- To achieve required level of accuracy in results, this mesh needs to be refined in areas where fields are of interest or the field gradients are high
- Adaptive meshing provides automated mesh refinement capability based on reported energy error in simulation
- Adaptive meshing is available only with static solvers

Simulation: Setup1

Design Variation: move='0mm'

Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

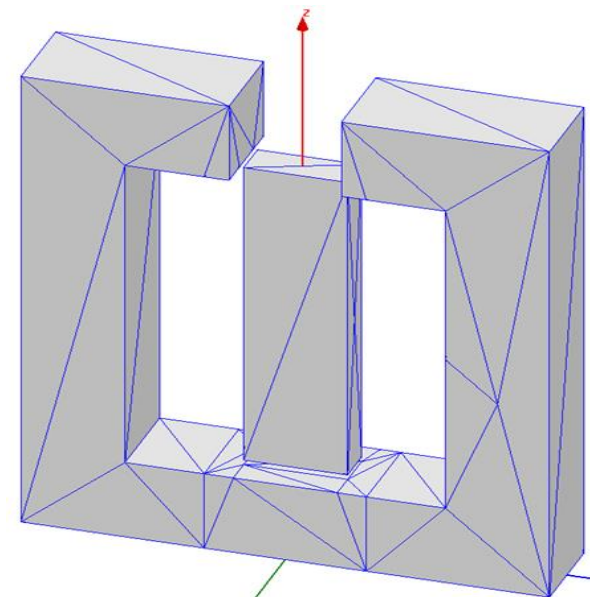
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	252	0.0090454	100.78	N/A
2	332	0.009181	48.79	1.4994
3	441	0.0088437	40.743	3.6748
4	579	0.0089233	38.941	0.90079
5	757	0.0088441	40.238	0.88831
6	991	0.0092404	30.405	4.4813
7	1296	0.0091963	14.802	0.47723
8	1695	0.0092247	12.23	0.30945
9	2209	0.0093259	9.309	1.0969
10	2878	0.009519	6.7657	2.0704

Number of Passes: Completed 10, Maximum 10, Minimum 2

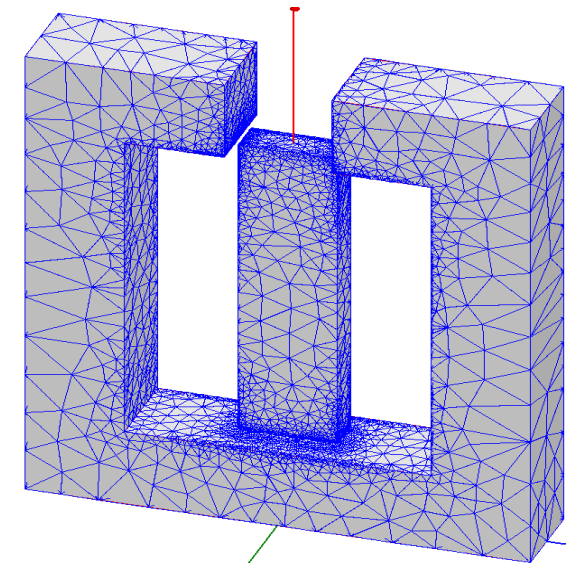
Energy Error/Delta Energy (%): Target (1, 1), Current (6.7657, 2.0704)

View:  Table  Plot

Export...



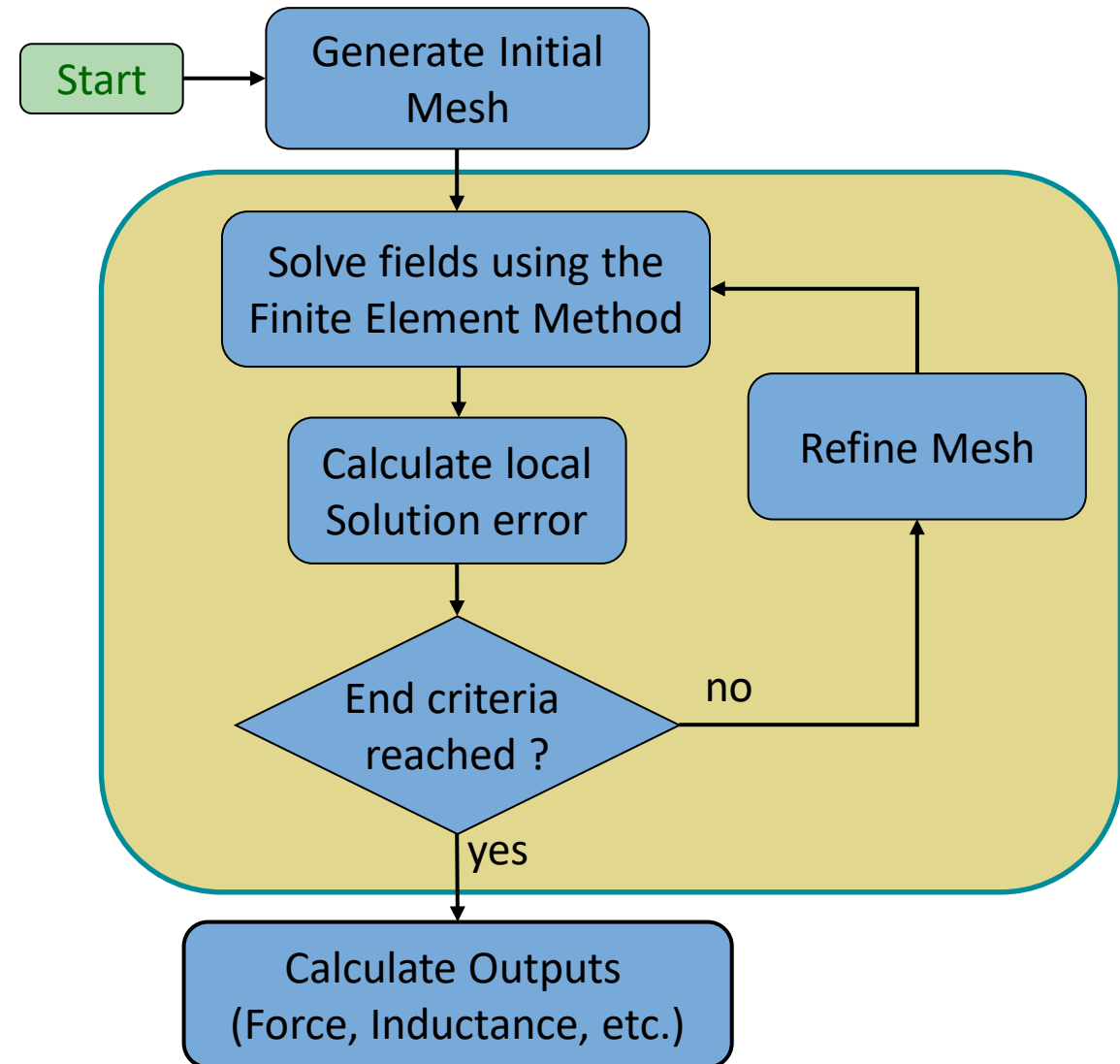
Initial Mesh



Adaptively Refined Mesh

# Maxwell Adaptive Meshing

- Adaptive Meshing Workflow
  - Adaptive meshing technique starts with initial mesh and refines it until required accuracy (Energy % error) is met or Maximum number of passes is reached



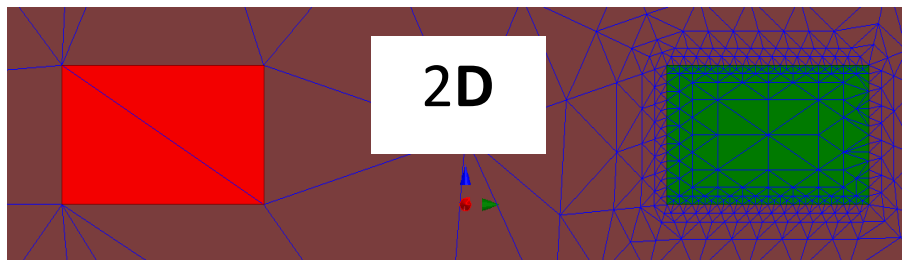


# Mesh Operations

- **Mesh Operations**
  - **Maxwell's Adaptive mesh refinement feature can be effectively used to achieve an optimized mesh**
  - **Transient Solvers do not have this capability to refine the initial mesh. Thus Transient Solvers require either Mesh Operations to be specified or use the Link Mesh option to an adaptively refined mesh from a static solver.**
  - **In complex Static problems, it is also recommended to use Mesh Operations**
    - **To reduce number of passes required to achieve desired accuracy**
    - **To increase mesh density in areas of interest before the adaptive mesh refinement begins**
  - **Maxwell offers following mesh operation specifications**
    - **On Selection/ Length Based;**
    - **On Selection / Skin Depth Based**
    - **Inside Selection / Length Based**
    - **Surface Approximation**
    - **Model Resolution**
    - **Cylindrical Gap Treatment**

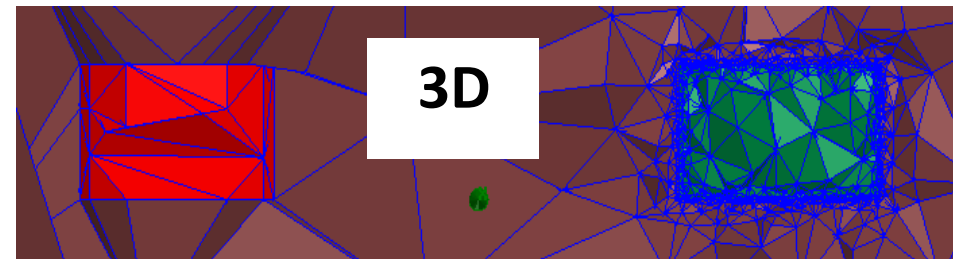
# On Selection

- **Mesh Operation: On Selection/Length Based**
  - The Length-based On-selection refinement will limit the edge length of all triangles formed on the surface of a selected object or any selected faces.
  - Can be added selecting the Object and menu item *Maxwell 2D/3D* → *Mesh Operations* → *Assign* → *On Selection* → *Length Based*
  - **Restrict Length of Elements:**
    - Refines the mesh by controlling maximum elements size on the boundary of assigned object
  - **Restrict the Number of Elements:**
    - Refines the mesh by controlling maximum element count on the boundary of assigned object



Without Mesh  
Operation

With Mesh  
Operation



Without Mesh  
Operation

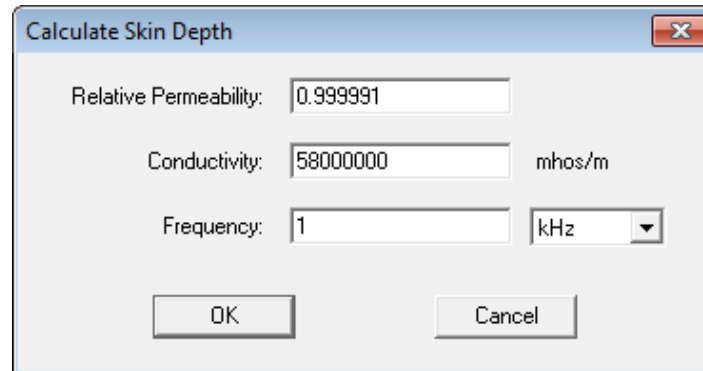
With Mesh  
Operation

**Note: When Restrict Length of Elements and Restrict Number of Elements are both selected, mesh refinement will stop when any of the conditions is met**

# On Selection

- Mesh Operation: On Selection/Skin Depth Based
  - Skin Depth Based mesh operations are assigned to resolve induced eddy current near conductor surfaces
  - This refinement method creates mesh layers within the selected surfaces of objects
  - Can be added selecting the Object and menu item *Maxwell 2D/3D* → *Mesh Operations* → *Assign* → *On Selection* → *Skin Depth Based*
  - Skin Depth:
    - Skin Depth field allows users to enter known value of the skin depth and number of layers of mesh to be created
  - Calculate Skin Depth:
    - Calculate Skin Depth tab allows user to compute resulting skin depth value based on Permeability, Conductivity and Frequency
    - Computed value is automatically assigned in Skin Depth field

$$\delta = \sqrt{\frac{2}{\omega \mu_0 \mu_r \sigma}} = \frac{1}{\sqrt{\pi f \mu_0 \mu_r \sigma}}$$



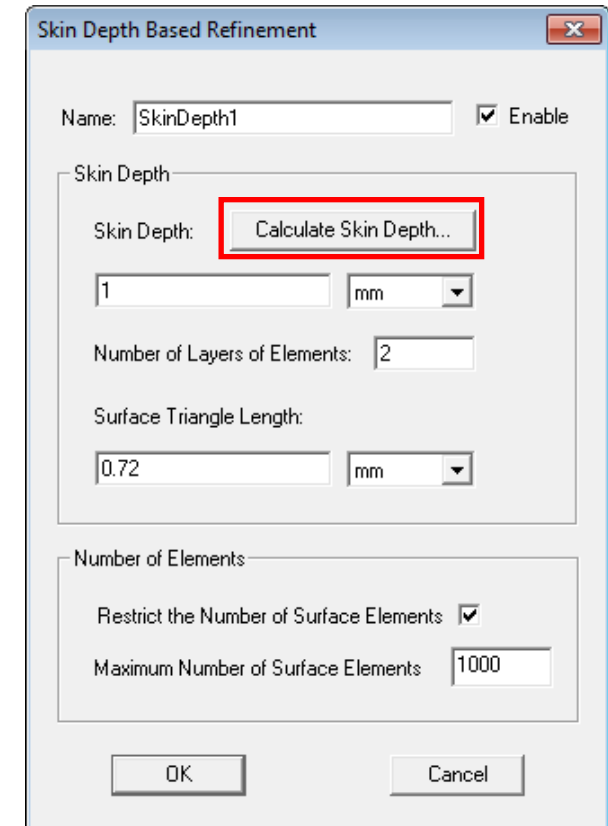
Calculate Skin Depth

Relative Permeability: 0.999991

Conductivity: 58000000 mhos/m

Frequency: 1 kHz

OK Cancel



Skin Depth Based Refinement

Name: SkinDepth1  Enable

Skin Depth

Skin Depth: Calculate Skin Depth...

1 mm

Number of Layers of Elements: 2

Surface Triangle Length: 0.72 mm

Number of Elements

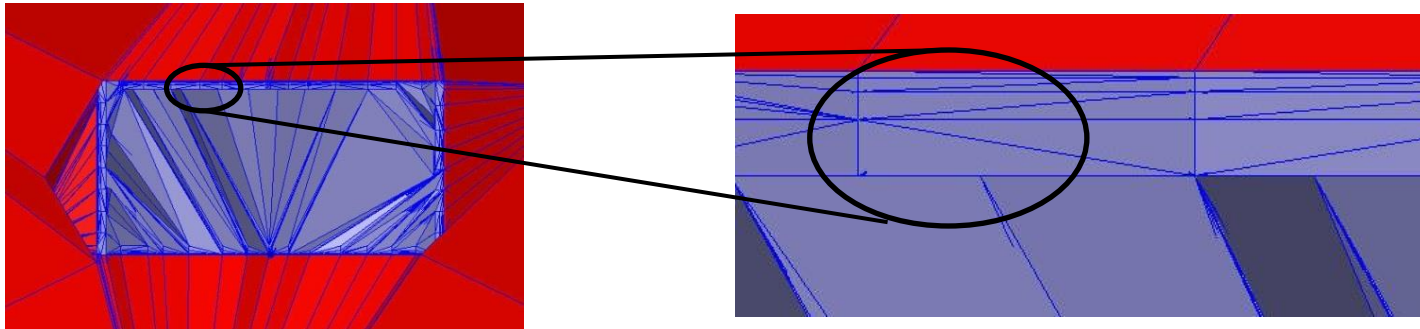
Restrict the Number of Surface Elements

Maximum Number of Surface Elements: 1000

OK Cancel

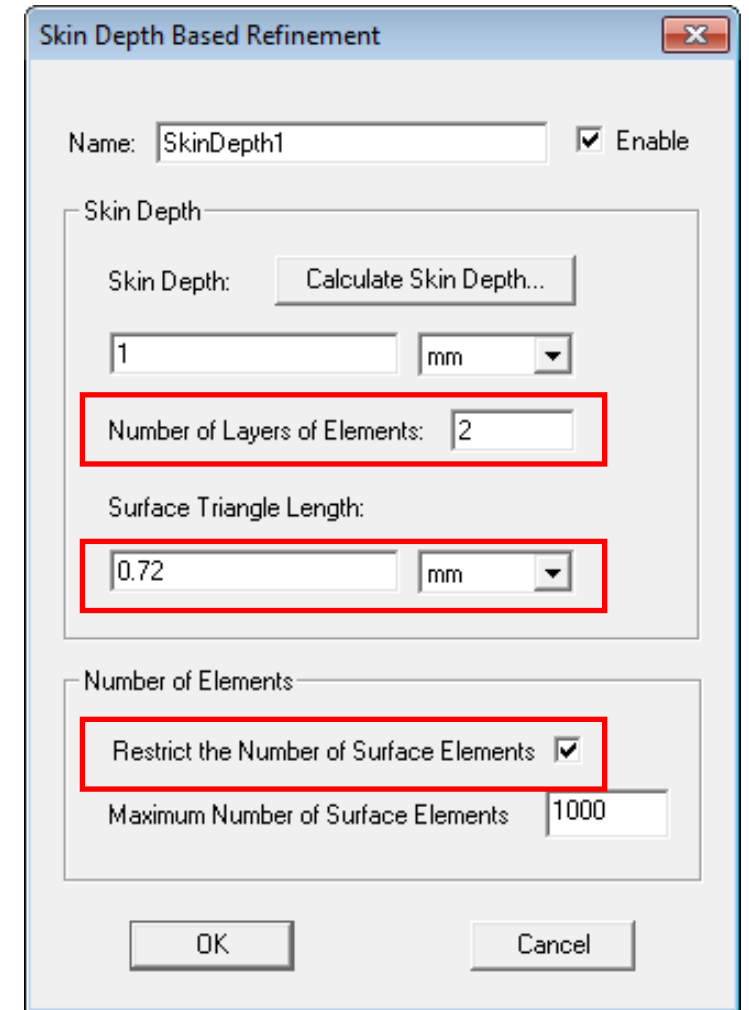
# On Selection

- Mesh Operation: On Selection/Skin Depth Based
  - Number of Layers of Elements:
    - Sets maximum number of mesh layers created in skin region
  - Surface Triangle Length:
    - Sets the maximum size of elements on the assigned objects
    - Surface Triangle Length controls the aspect ratio of the elements
  - Restrict the Number of Surface Elements:
    - Restricts the count of elements to a specified value



Four Layers of Skin Depth Mesh

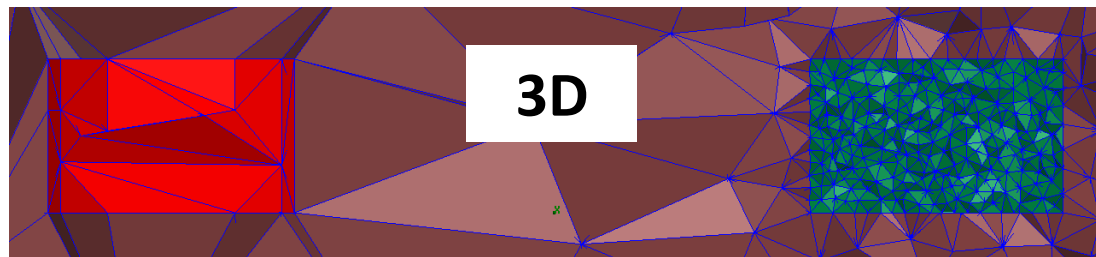
**Note: Skin Depth Based mesh operation may result in high aspect ratio tetrahedra, thus it should be used very carefully**



# Inside Selection

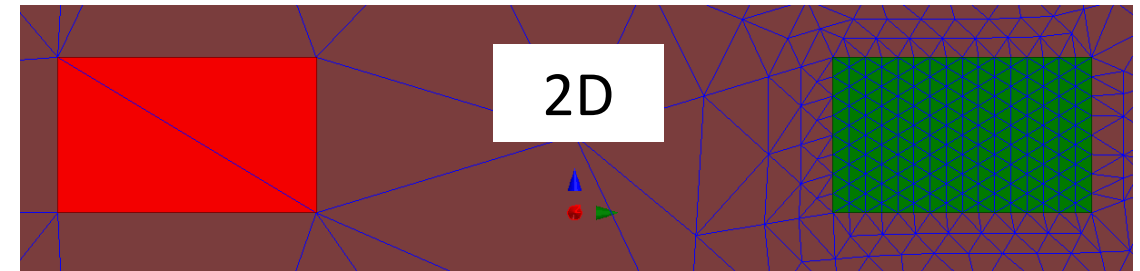
- **Mesh Operation: Inside Selection/Length Based**

- The Length-based Inside-selection refinement will limit the edge length of all tetrahedrons (or triangles) formed inside a selected solid or sheet object
- Can be added selecting the Object and menu item *Maxwell 2D/3D* → *Mesh Operations* → *Assign* → *Inside Selection* → *Length Based*
- All the options in the Element Length Based Refinement window are the same as for On Selection mesh operation except that the inside selection refinement will control size or number of elements inside the selected object, forming an homogeneous mesh.



Without Mesh Operation

With Mesh Operation

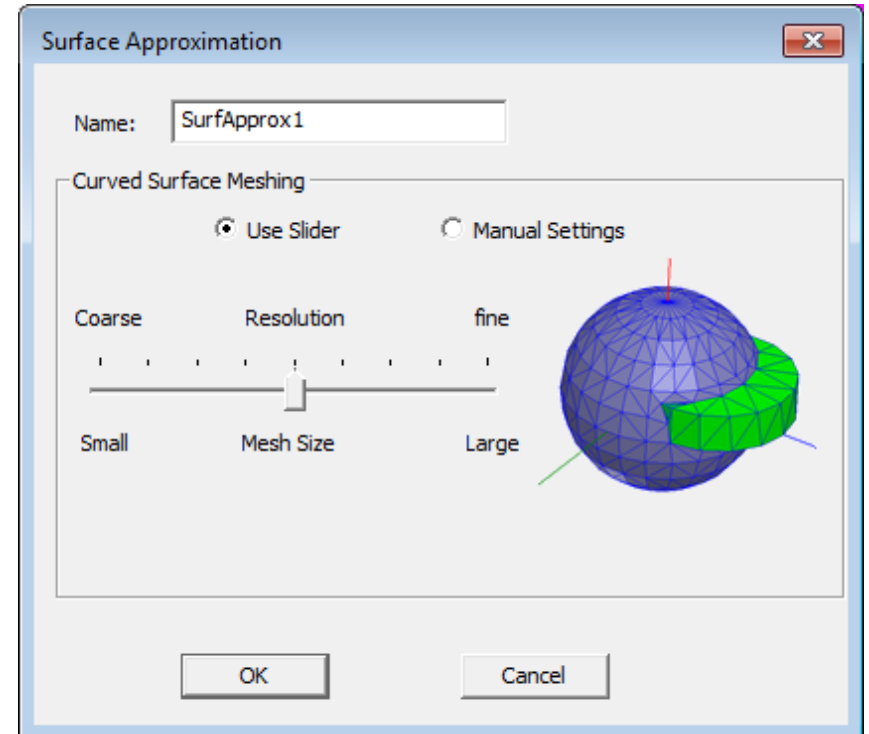


Without Mesh Operation

With Mesh Operation

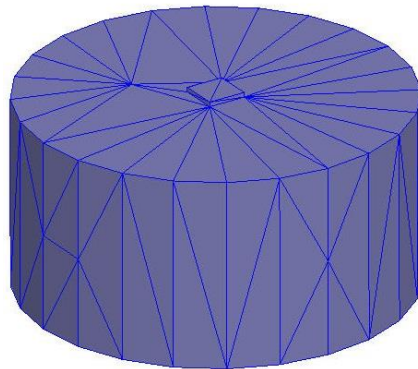
# Surface Approximation

- Surface Approximation
  - Surface Approximation Mesh Operations are helpful to resolve curved surfaces with a good quality mesh and can be used to both increase or decrease mesh density on curved surfaces
  - By default, Surface Approximation mesh operation is performed while creating initial mesh using Initial Mesh Settings
  - Can be assigned selecting the Object and menu item *Maxwell 2D/3D* → *Mesh Operations* → *Assign* → *Surface Approximation*
  - Use the Slider to modify the Surface approximations settings

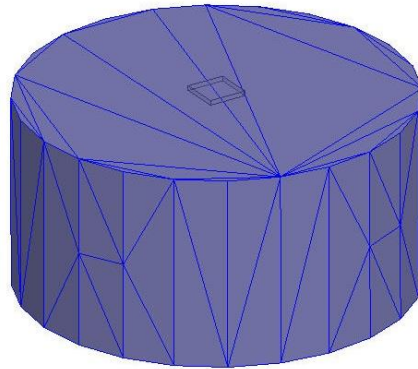


# Model Resolution

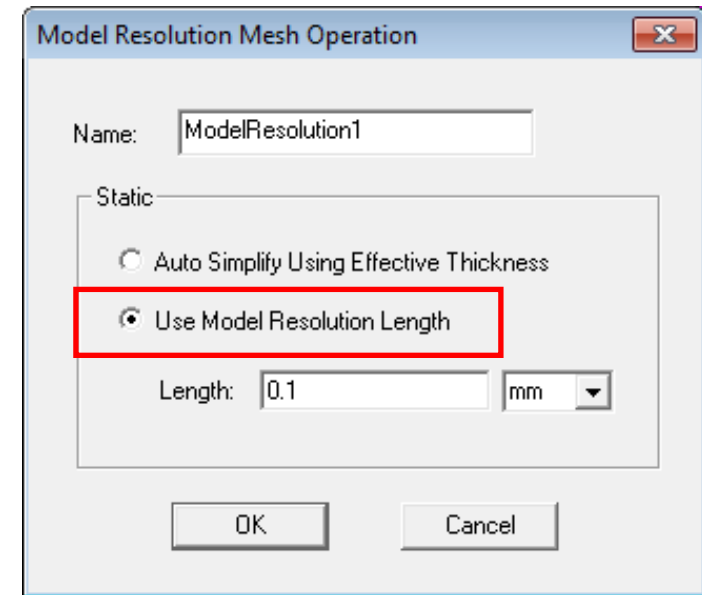
- Model Resolution
  - Model Resolution enables users to ignore small features of geometry which might not be important for simulation
  - Users can specify the maximum length of geometry features which will be ignored by mesh
  - Default Option is set to Auto Simplify, which automatically calculates the minimum feature length
  - Can be assigned from menu *Maxwell 2D/3D* → *Mesh Operations* → *Assign* → *Model Resolution*



Without Model Resolution



With Model Resolution



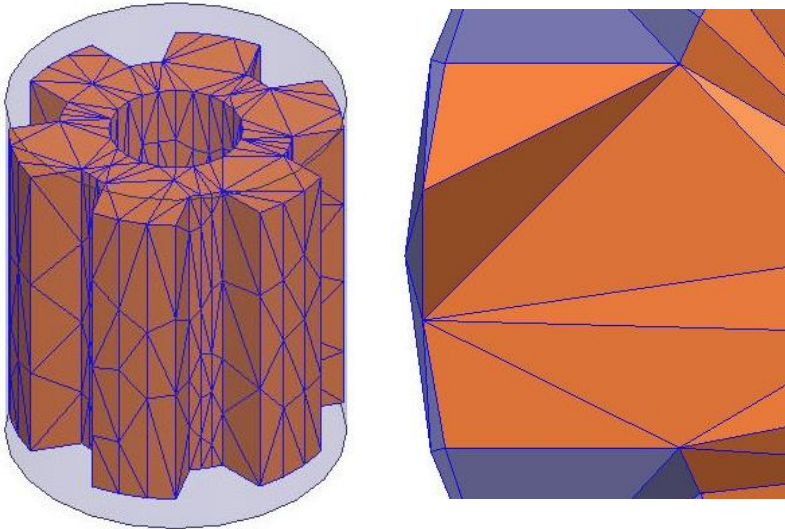
**Note: Model resolution must be used with caution as sometimes mesh might not be able to represent geometry correctly**



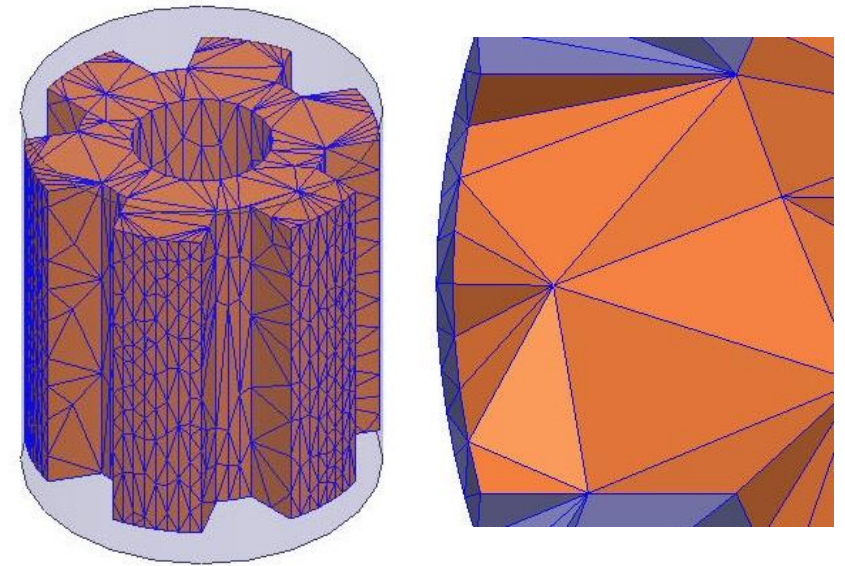
# Cylindrical Gap Treatment

- Cylindrical Gap Treatment

- Cylindrical Gap Treatment mesh operation is a 3D-only proximity based mesh refinement and usually assigned to Band objects for rotational motion
- Refinement is done on the applied objects based on the closeness of the geometry lying inside it
- For Transient Solver involving rotational motion, this mesh operation is automatically created once the rotational motion is defined in order to resolve air gap between Stator and Rotor parts
- Mesh Operation can be assigned from menu item *Maxwell 3D* → *Mesh Operations* → *Assign* → *Cylindrical gap Treatment*



Without Cyl. Gap Treatment



With Cyl. Gap Treatment



# Applying Mesh Operations

- Apply Mesh Operations

- When Analysis Process is started mesh operations are automatically applied on initial mesh
- It is advisable to verify mesh quality and element count before starting the solution process by inspecting both the Mesh Statistics, and visual inspection of Mesh plots.
- Mesh Operations can be assigned from menu item *Maxwell 2D/3D* → *Analysis Setup* → *Apply Mesh Operations* or *RMB on Analysis Setup* → *Apply Mesh Operations*
- If the mesh is not satisfactory, it is possible to come back to initial mesh using *Maxwell 2D/3D* → *Analysis Setup* → *Revert to Initial Mesh* or *RMB on Analysis Setup* → *Revert to Initial Mesh* and then applying again modified mesh operations

- Mesh Statistics

- Once Mesh Operations are applied, mesh quality and element count can be verified from the *Maxwell 2D/3D* → *Results* → *Solution Data*
- In Solutions window, select *Mesh Statistics* tab

Simulation: Setup1

Design Variation: Vpeak='11268V'

Profile | Force | Torque | **Mesh Statistics**

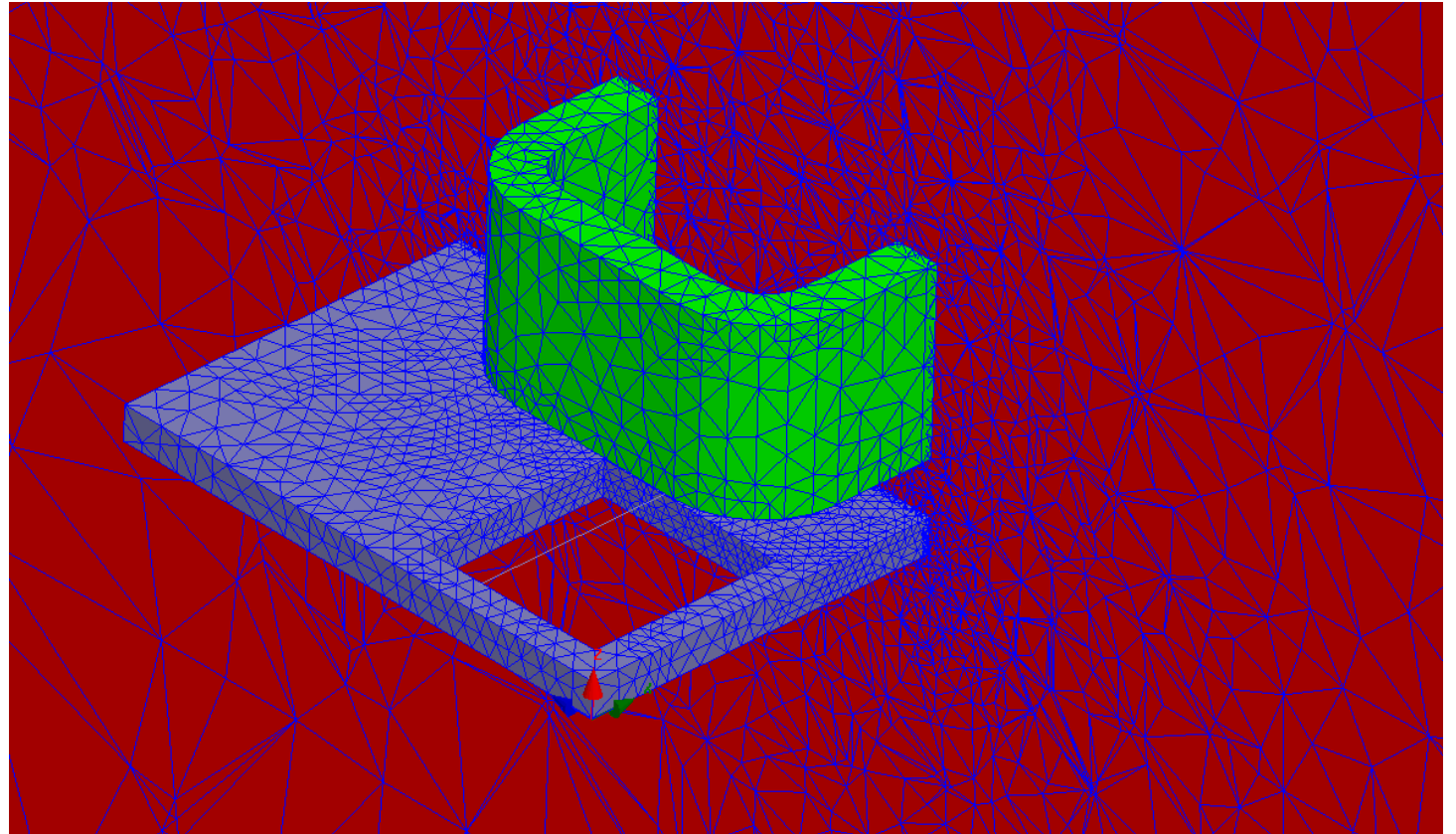
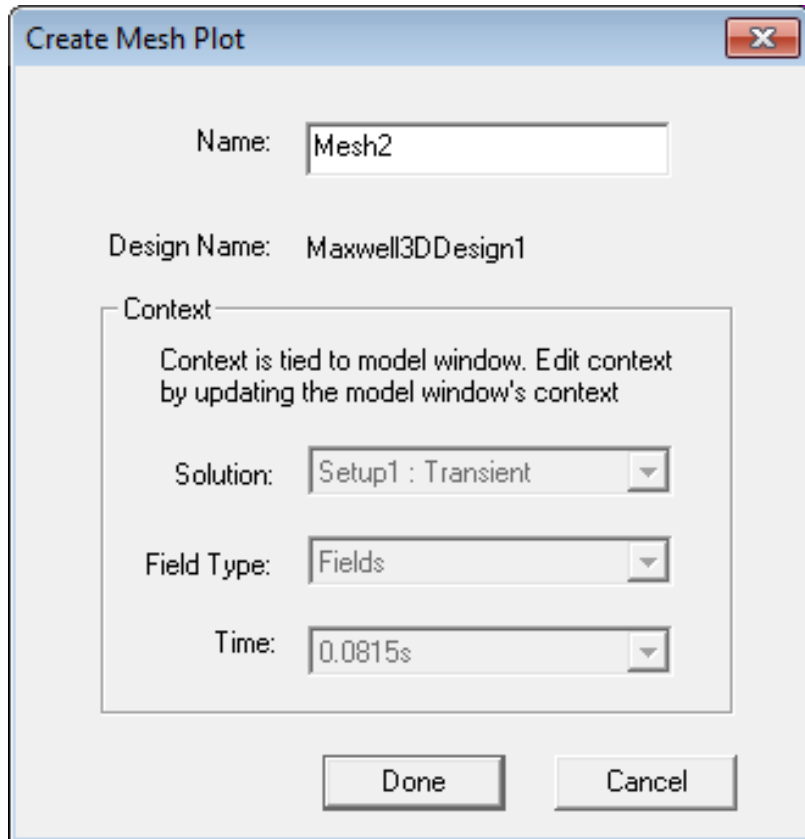
Total number of elements: 26347

	Num Tet...	Min edge len...	Max edge len...	RMS edge len...	Min tet vo...	Max tet vol...	Mean tet vo...
core	7939...	101.845	260.159	188.752	40091.3...	1.05369e+0...	320975...
LV_A	2521...	54.2234	189.271	129.857	8026.52...	262525...	71732.9...
LV_B	2421...	57.1159	215.262	129.882	7554.98...	254243...	74621.8...
LV_C	2526...	49.0802	190.281	128.824	2605.18...	252806...	71651.6...
Region	10940...	68.1971	4117.11	541.098	318.737...	1.42996e+0...	6.62512e+0...

# Applying Mesh Operations

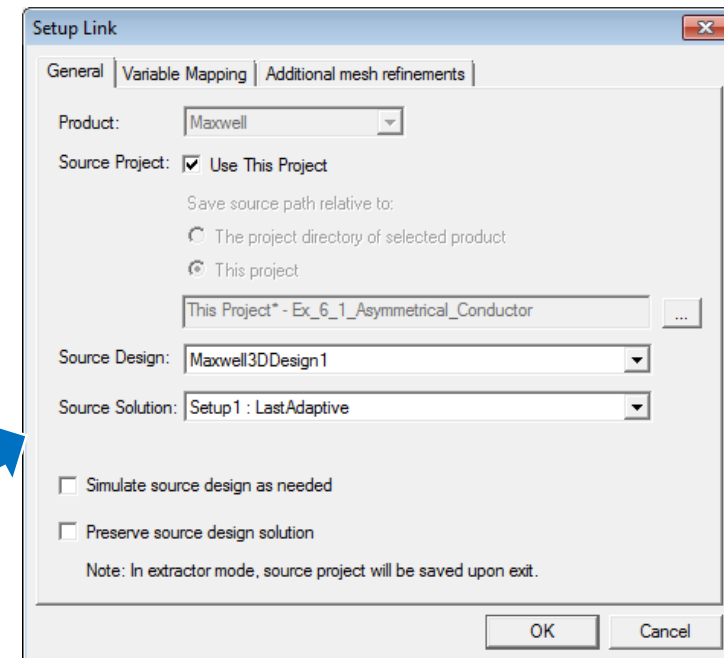
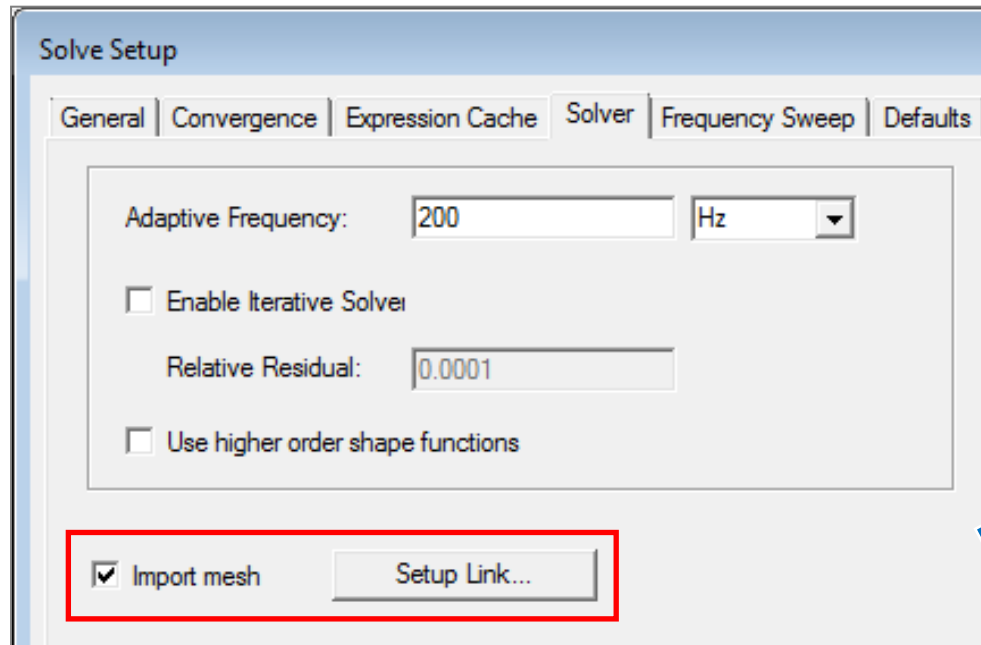
- Mesh Plots

- Mesh plots enables to inspect the mesh on objects or the sections of mesh to verify its validity
- A Mesh plot can be created on objects, sheets or planes
- To create the mesh plot, select the required entities and *Maxwell 2D/3D* → *Fields* → *Plot Mesh*



# Mesh Linking

- **Linking Mesh to Other Design**
  - In some static cases as well it is beneficial to link mesh across designs to achieve optimum results
  - A Transient Design can also be linked to an adaptively refined mesh coming from a Static Solution
  - Mesh can be linked from Analysis Setup window
  - Import Mesh option is available under Solver tab for Static solvers and Advanced tab for Transient Solver
  - Source and Target design should have exactly same geometry



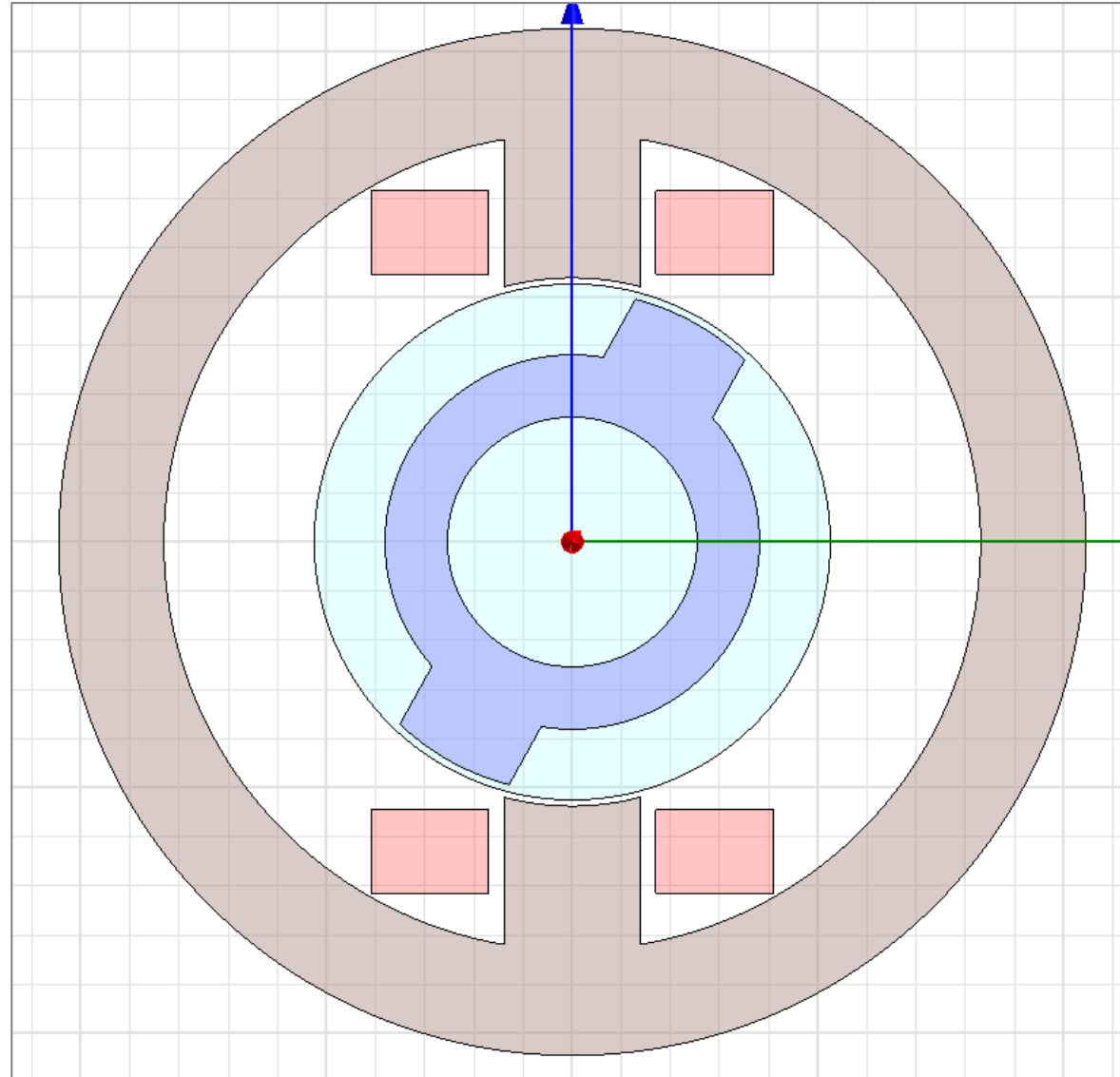
# Troubleshooting

- **Mesh Failure Troubleshooting**
  - Mesh generation might fail due to various reasons related to geometry
  - If mesh failure occurs, users are advised to follow below steps
    - Select the menu item *Modeler* → *Model Analysis* > *Show Analysis Dialog* > *Last Simulation Mesh* to identify reason for mesh failure
    - Use the command *Modeler* → *Model Analysis* → *Analyze Object* to analyze geometry errors and perform healing
    - Use the command *Modeler* → *Model Analysis* → *Analyze InterObject Misalignments* to analyze and correct misalignments
    - Turn some parts of the geometry to Non-Model and perform meshing to identify exactly where is located the issue
    - Remove or simplify unnecessary complex features which are causing problem in meshing by redrawing
    - Use Surface Approximation for higher curvature objects to resolve curved faces
    - Use Model resolution cautiously to neglect unimportant small features

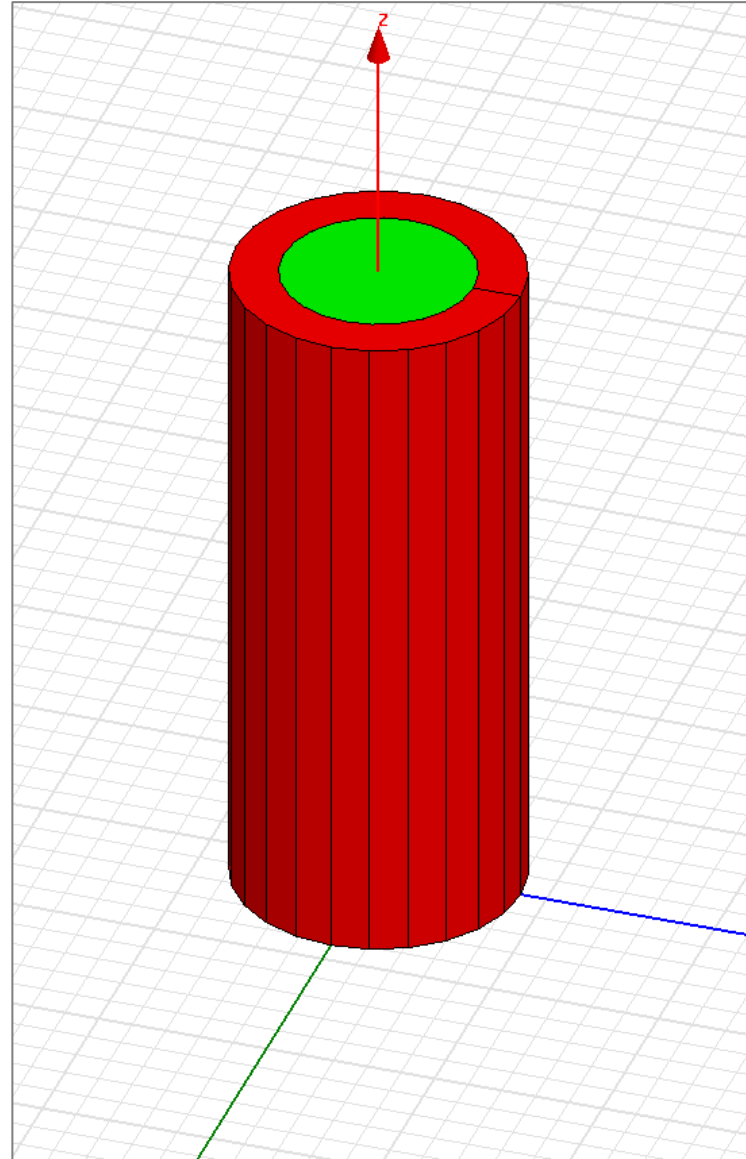
# Summary

- **What have we learned in this session?**
  - **Magnetic Transient solver**
  - **Material Properties**
  - **Electric Transient solver**
  - **Mesh Operations**
  - **Mesh Linking**

# Workshop 3.1 – 2D Magnetic Transient analysis



# Workshop 3.2 – 3D Magnetic Transient analysis





**End of Presentation**

