Ansys Maxwell Getting Started

Module 01: Basics

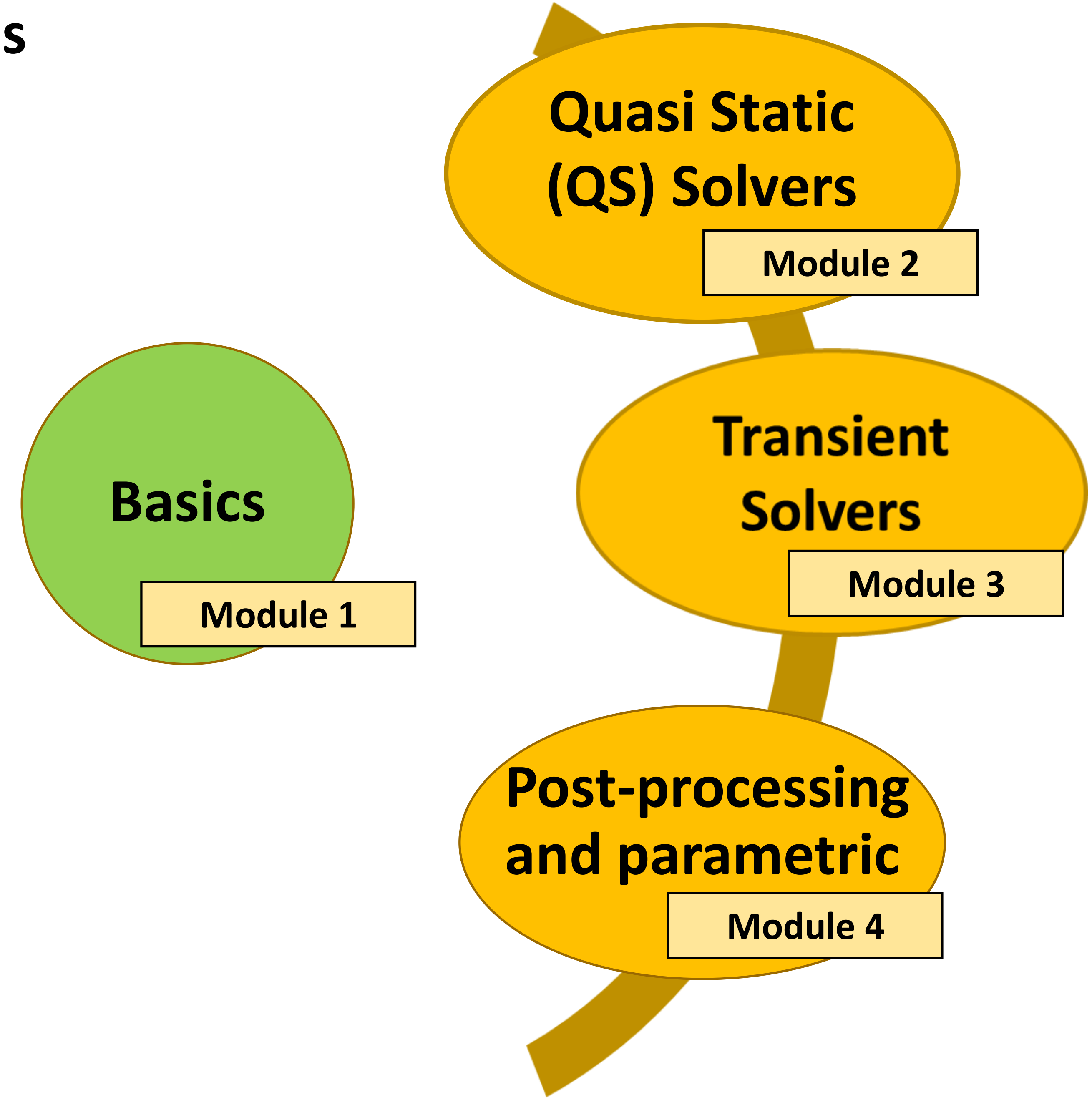
Release 2020 R2

Ansys

Overview

- Introduction to Ansys Maxwell
- Modeling
  - Geometry and Import
  - Materials, boundaries and sources
- Parameterization
- Adaptive meshing
- Inductance calculation
- Postprocessing 1
- Workshop 1: Magnetostatic Analysis

Overall Process



Ansys is the Simulation Leader

FOCUSED

This is all we do.
Leading product technologies in all physics areas
Largest development team focused on simulation

TRUSTED

96 of the top 100
FORTUNE 500 Industrials
ISO 9001 and NQA-1 certified

CAPABLE

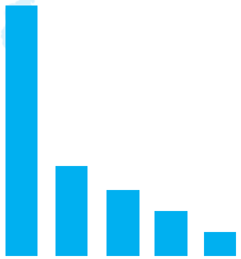
PROVEN

Recognized as one of the world's MOST INNOVATIVE AND FASTEST-GROWING COMPANIES*

INDEPENDENT

Long-term financial stability
CAD agnostic

LARGEST



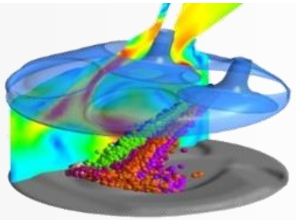
3x The size of our nearest competitor

*BusinessWeek, FORTUNE

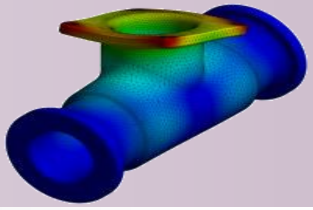
Breadth of Technologies

Fluid Mechanics:
From Single-Phase Flows

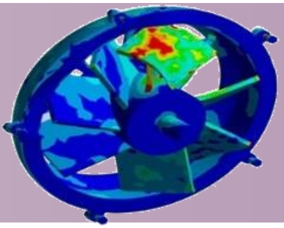
To Multiphase
Combustion



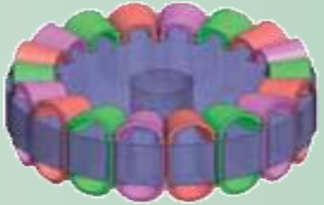
Structural Mechanics:
From Linear Statics



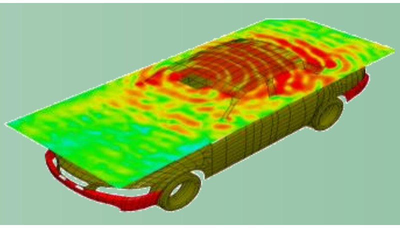
To High-Speed Impact



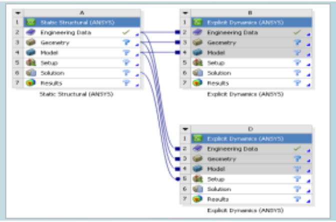
Electromagnetics: From
Low-Frequency Windings



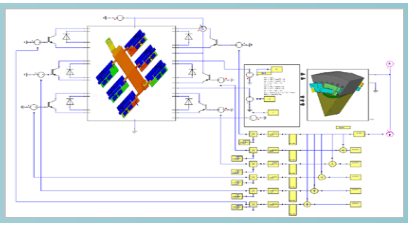
To High-Frequency
Field Analysis



Systems:
From Data Sharing



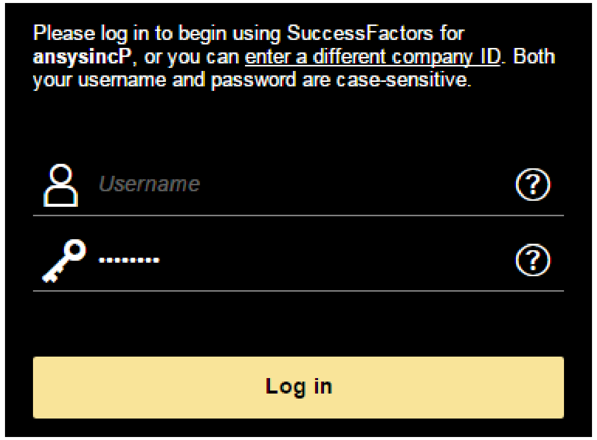
To Multi-Domain
System Analysis



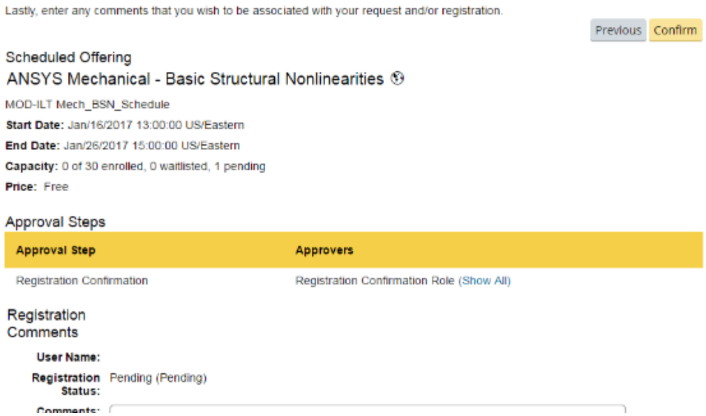
Ansys

Ansys Learning Hub Functions

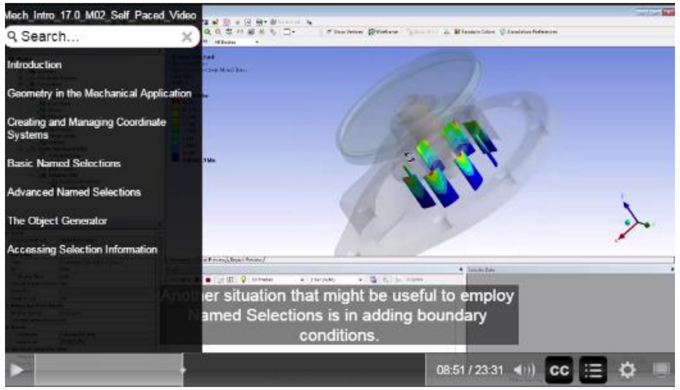
Search global course schedule and access materials



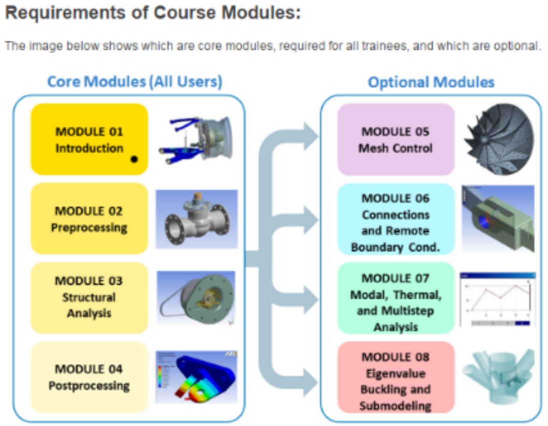
Learn whenever you like



Register for live and virtual classrooms



Self-paced videos - Go back and review with searchable captions



Mix and match – pick modules that you need

Ask questions in the interactive Learning Room

Introduction to Ansys Maxwell

- Maxwell is an electro-magnetic tool suitable to analyse low-frequency phenomena and devices
- Maxwell is integrated into the Electronics Desktop, together with all the other Ansys Electromagnetic tools
- Maxwell can be integrated in the Workbench Platform for Multiphysics analysis
- Geometries can be created either directly inside Maxwell or imported from external CAD tools
- Quasi-static Solvers offer an automatic meshing refinement algorithm
- Transient solver allows the analysis of large movements and mechanical transients

Introduction to Ansys Maxwell

- **Ansys Maxwell is a high-performance interactive software package that uses finite element analysis (FEA) to solve electric or magnetic problems.**
- **Maxwell solves the electromagnetic field problems by solving Maxwell's equations in a finite region of space with appropriate boundary conditions and user-specified initial conditions in order to obtain a solution with guaranteed uniqueness.**

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

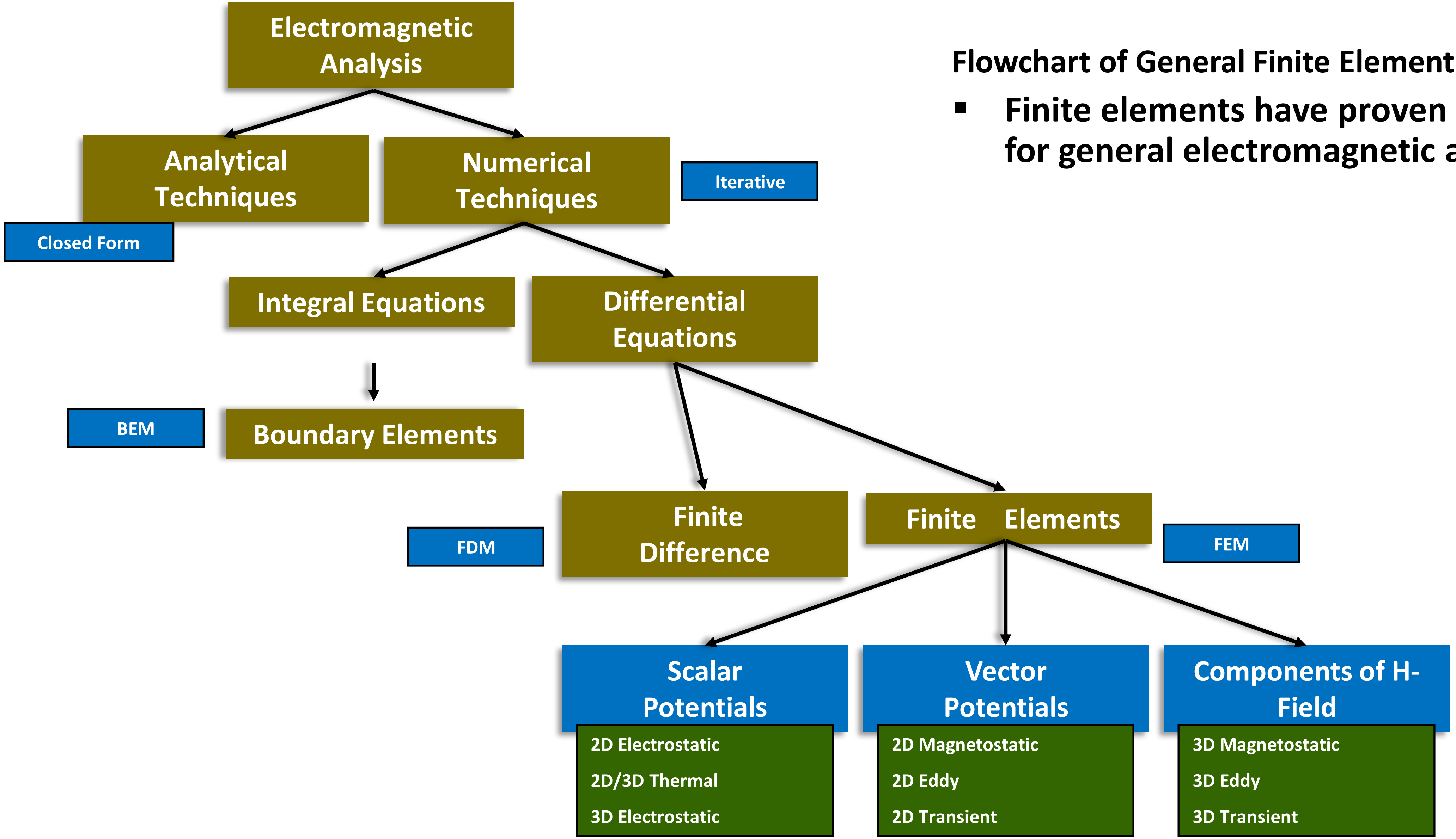
$$\nabla \cdot \boldsymbol{B} = 0$$

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

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

- **Appropriate set of equations and its terms are used based on the solver selected such as Electrostatic, Magnetostatic, Eddy Current and Magnetic Transient.**

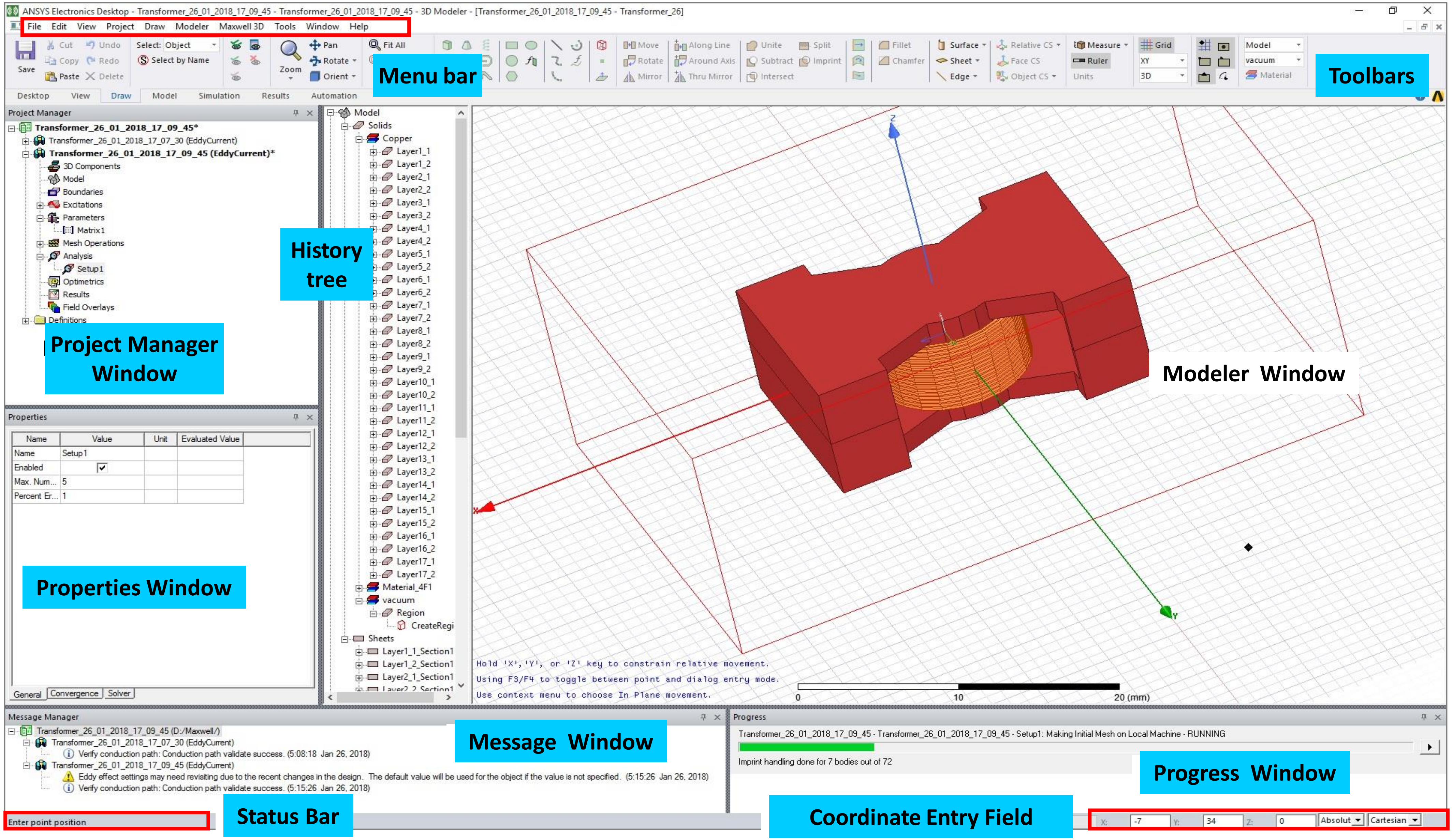
Introduction to Ansys Maxwell



Flowchart of General Finite Element Analysis Method

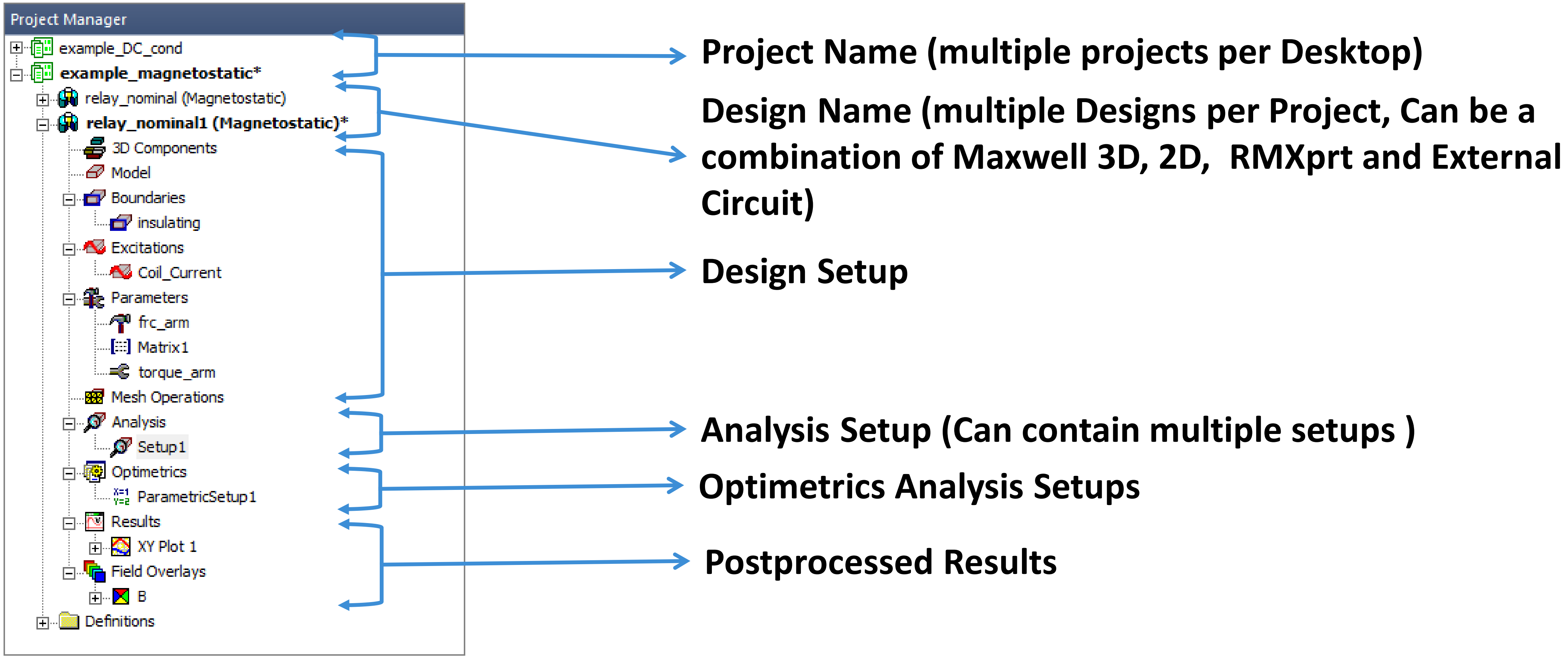
- **Finite elements have proven to be very robust for general electromagnetic analysis**

Graphical User Interface (GUI)



Graphical User Interface (GUI)

- **Project Manager window**
  - **Project Manager window contains all the details of Problem Setup done for any project**



Ansys

Adding a Maxwell design

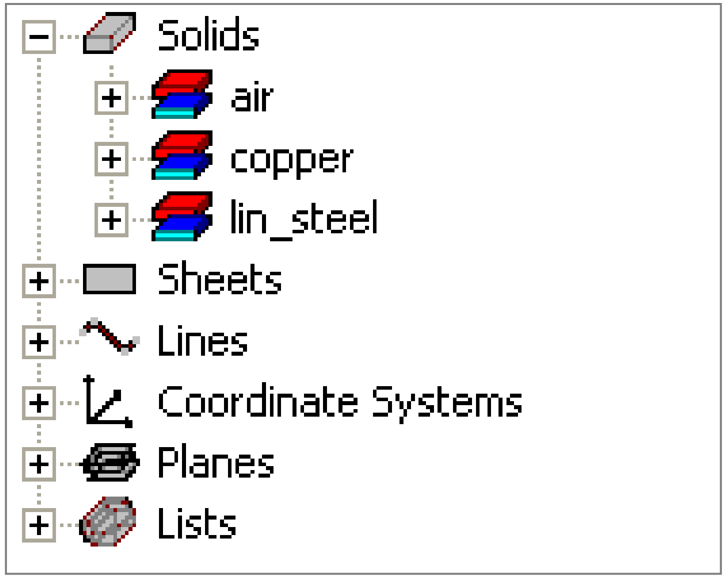
- Adding a Design to Maxwell
  - A design can be added to a Maxwell project from the Project menu bar or selecting icon from 

- Maxwell Design Types
  - RMxprt:
    - Rotating Machinery Expert is an interactive analytical tool used for designing and analyzing electrical machines
  - Maxwell 2D:
    - Maxwell 2D uses Finite Element Analysis to simulate and solve 2D electromagnetic fields in XY or RZ planes
  - Maxwell 3D:
    - Maxwell 3D uses Finite Element Analysis to simulate and solve three dimensional electromagnetic fields.

Note: Most of the lectures in this course are focused on Maxwell 2D and Maxwell 3D and do not cover RMxprt.

Graphical User Interface (GUI)

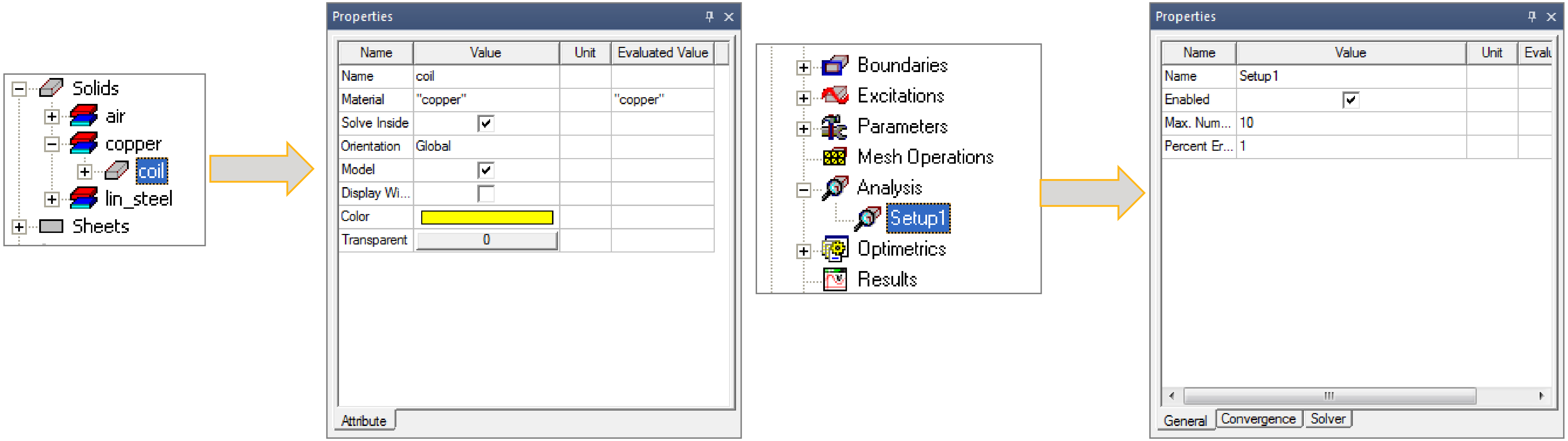
- **History Tree**
  - **History tree contains details of all geometrical operations performed in a Design**
  - **Any line, surface or solid objects, coordinate systems or its planes can be accessed from History tree**
  - **Solid objects are categorized based on assigned materials in 3D while in 2D sheet objects are categorized**



- **Toolbars**
  - **Most of the menu bar commands can be accessed from Toolbar as an icon**
  - **Toolbar can be customized to add or remove any options by right clicking on Toolbar or selecting the menu item Tools**

Graphical User Interface (GUI)

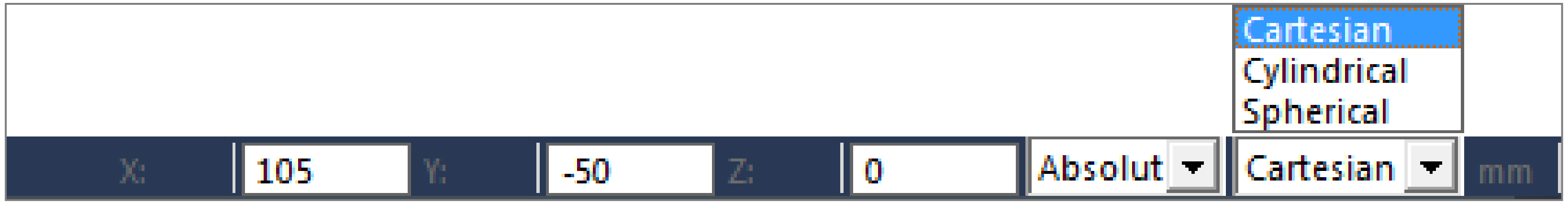
- **Properties Window**
  - **Shows all the properties of selected entities**
  - **Selected entity can be any geometrical object, performed geometrical operations from history tree or any added setup entities selected from Project Manager window**
  - **Display of Properties window will change based on selected entity**



Note: Double clicking on any selected entity will also initiate a dialog box for property display and editing

Graphical User Interface (GUI)

- Coordinate Entry Field
  - This window appears whenever an operation requires entry of coordinates
  - Coordinate values can be entered in Cartesian, Cylindrical or Spherical form

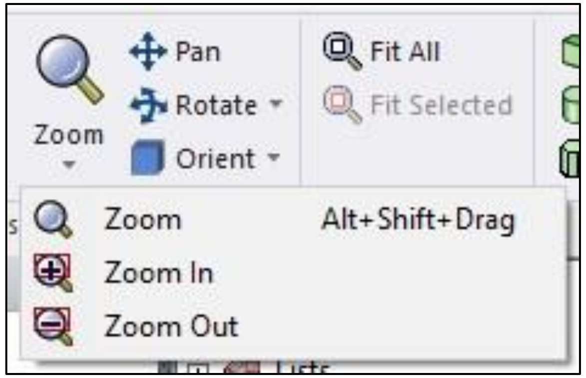


- Message Window
  - Displays Error, Warning or Information messages resulting from an operation
  - Messages can be Copy-Pasted to text file if required
- Status Bar
  - Shows status of Maxwell window, prompts next course of action for any geometrical operation or information about any command on which mouse is placed
- Progress Window
  - Shows progress of Analysis operation being carried out

Ansys

Graphical User Interface (GUI)

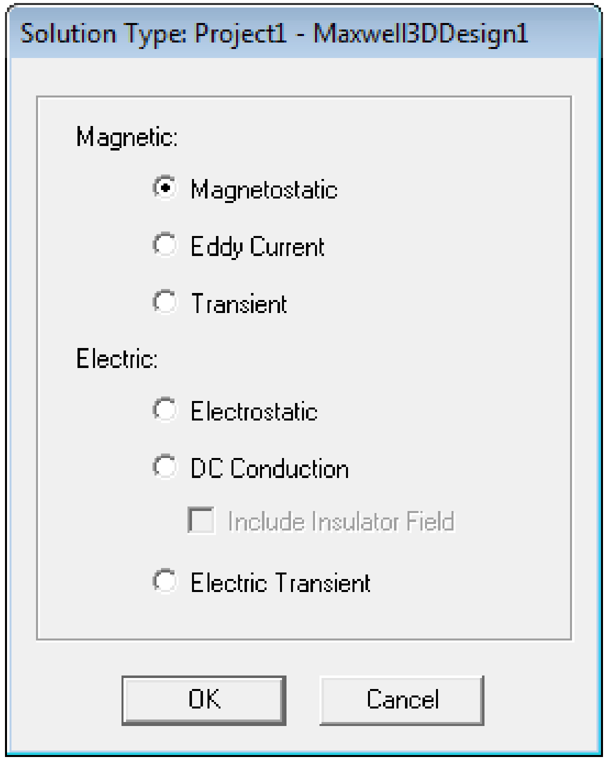
- **Modeler Window**
  - **Modeler window enables users to view and interact with geometry dynamically.**
  - **Geometry in Modeler window can be Manipulated either using Toolbar commands or Mouse-keyboard keys**
  - **Right Mouse click in Modeler window also gives access to various menu bar commands through context window**



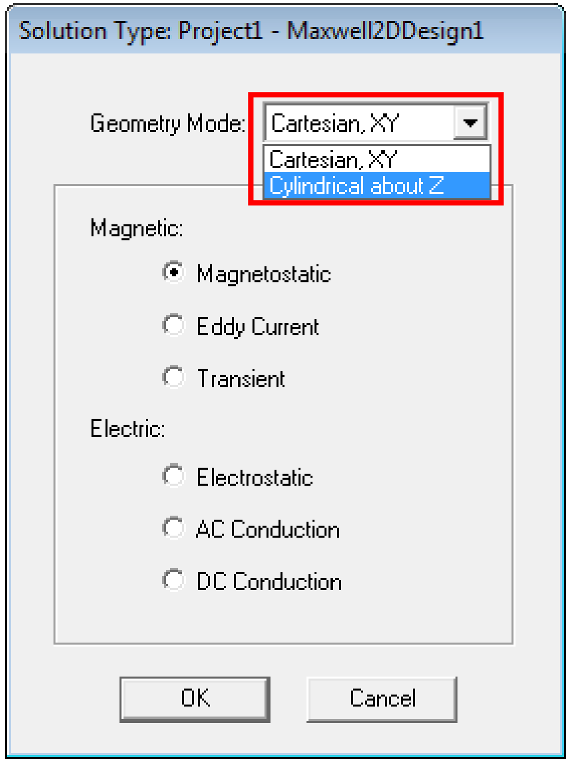
- **Mouse Commands for Geometry Manipulation**
  - ***Zoom*: Left click and drag the mouse button with *SHIFT+ALT* pressed**
  - ***Pan*: Left click and drag the mouse button with *SHIFT* pressed**
  - ***Rotate*: Left click and drag the mouse button with *ALT* pressed**

Solvers in Maxwell

- **Setting a Solver Type**
  - **Appropriate Maxwell solver can be selected based on the application being solved.**
  - **Solver type can be set by selecting the menu item *Maxwell 3D/2D → Solution Type***
  - **For 2D, users can also specify if the problem will be solved in XY or RZ plane**



Maxwell 3D



Maxwell 2D

Solvers in Maxwell

- **Magnetic Solvers**
  - **Magnetostatic Solver**
    - **Solves Static magnetic fields caused by DC currents and permanent magnets. Can solve both Linear and nonlinear materials**
  - **Eddy Current Solver**
    - **Solves sinusoidally-varying magnetic fields in frequency domain. Solves both Linear and nonlinear materials (creating energy-equivalent sinusoidal fields). Considers displacement currents. Induced fields such as skin and current proximity effects are also considered**
  - **Transient Magnetic**
    - **Solves Transient magnetic fields caused by time-varying or moving electrical sources and permanent magnets in Linear or Non-linear materials. Induced fields such as skin and current proximity effects are considered**

Note: Capacitive effects are not considered by Magnetic Solvers in Maxwell. These effects need to be accounted for separately

Solvers in Maxwell

- Electric Solvers
  - Electrostatic Solver
    - Solves Static Electric Fields in linear materials
- DC Conduction
  - Solves for voltage, electric field, and the current density from the potential. Can solve electric fields in insulators as well
- AC Conduction (2D only)
  - Solves sinusoidally-varying electric fields in Frequency domain. Available only with 2D solver
- Transient Electric
  - Transient electric fields caused by time-varying voltages, charge distributions, or current excitations in inhomogeneous materials. Electric potential is the solution quantity

Maxwell files structure

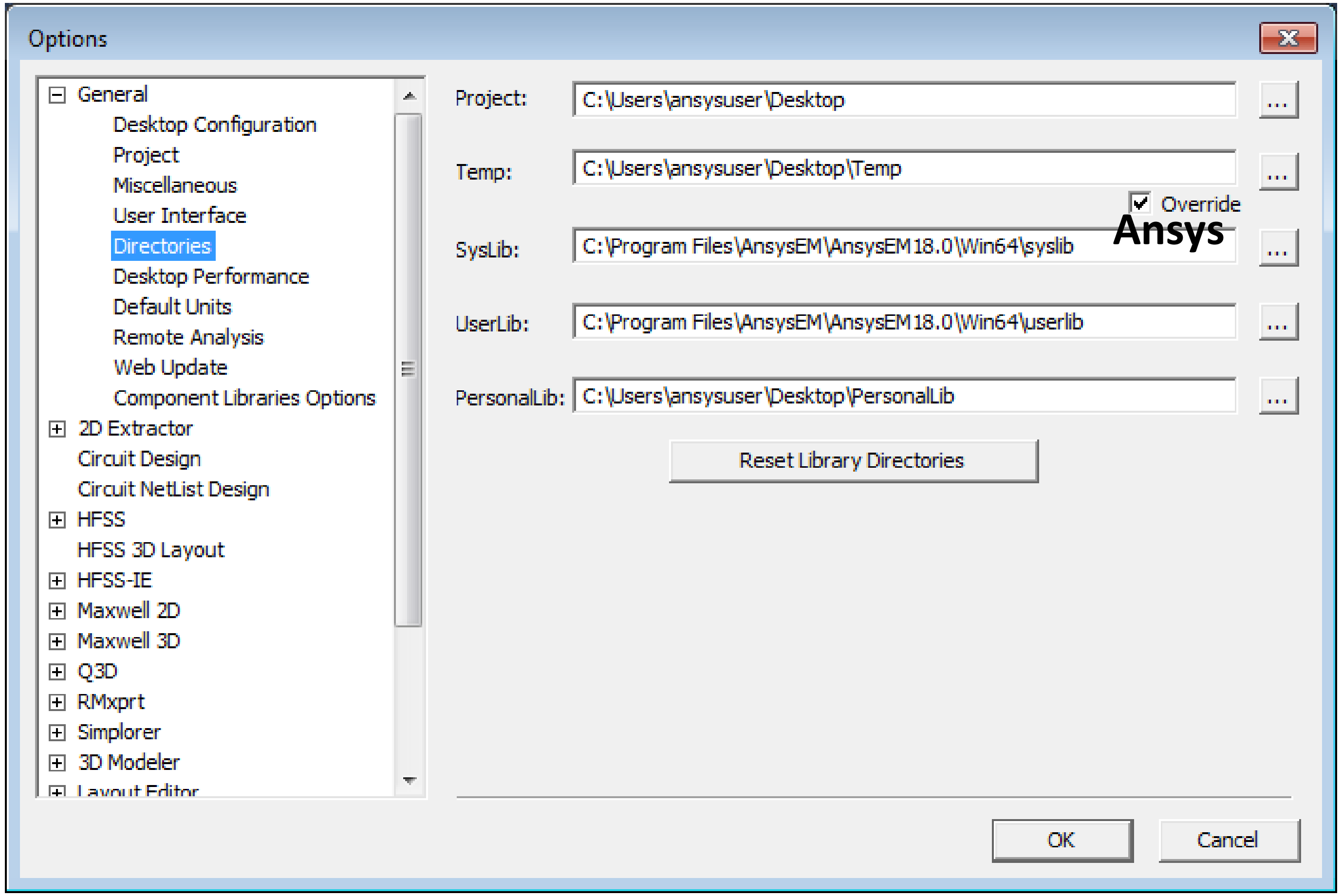
- **File Structure in Maxwell**
  - **FileName.aedt**
    - **This file contains all the information related to Maxwell project apart from results. The file is written in ASCII format.**
  - **FileName.aedtresults**
    - **This folder contains the mesh and solution related files. Users need to copy both .aedt file and this folder in order to transfer the project with results**
  - **FileName.aedt.lock**
    - **This is a lock file created when file is open to avoid overwriting of file from multiple sessions**
  - **FileName.aedt.auto**
    - **Autosave file in order to recover lost data in case of unusual file closure. Created only when Autosave is turned ON**

Name	Type	Size
example.aedtresults	File folder	
example.aedt	Ansoft Electronics...	613 KB
example.aedt.auto	AUTO File	572 KB
example.aedt.lock	LOCK File	1 KB

Ansys

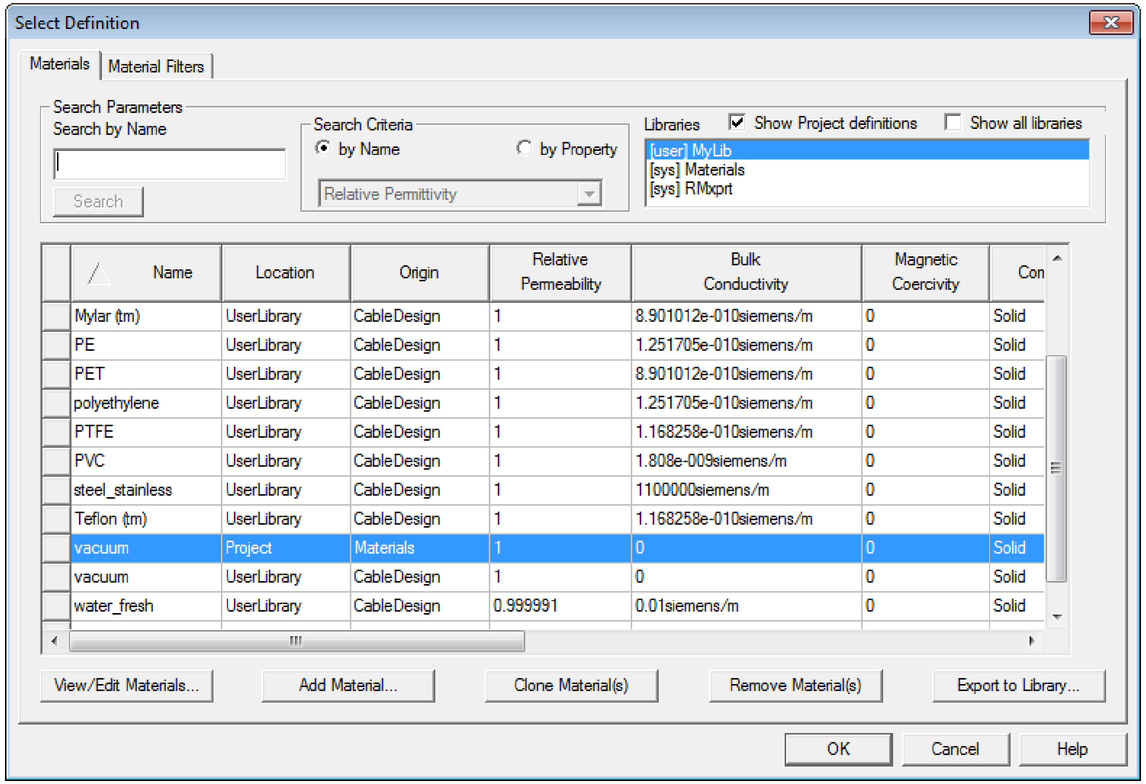
Maxwell Folders

- **Maxwell Folders**
  - **Maxwell Folder locations can be set from the menu item *Tools → Options → General → Directories***
    - **Project: The default location where Maxwell project file is saved**
    - **Temp: Location for saving Temporary results files. Files will be deleted once project is saved**
    - **SysLib: Global level directory predefined by and ships with new upgrades**
    - **UserLib: Can host user created material libraries or script files. Can be shared among all users at a company**
    - **PersonalLib: Can host user created material libraries or script files. Accessible only the user who creates it**



Maxwell Material Libraries

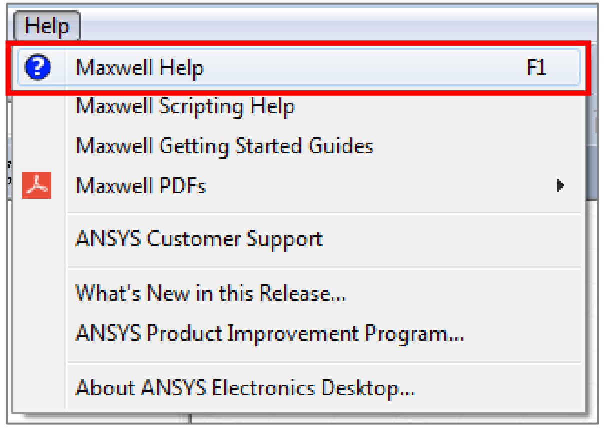
- **Maxwell Material Libraries**
  - **Maxwell default material library is loaded with Maxwell installation and is available under Syslib directory. This material database can not be modified by users**
  - **Users can add their own material database to Personal or User library directories**
  - **Once the file is added, users can select and assign material from the added library**

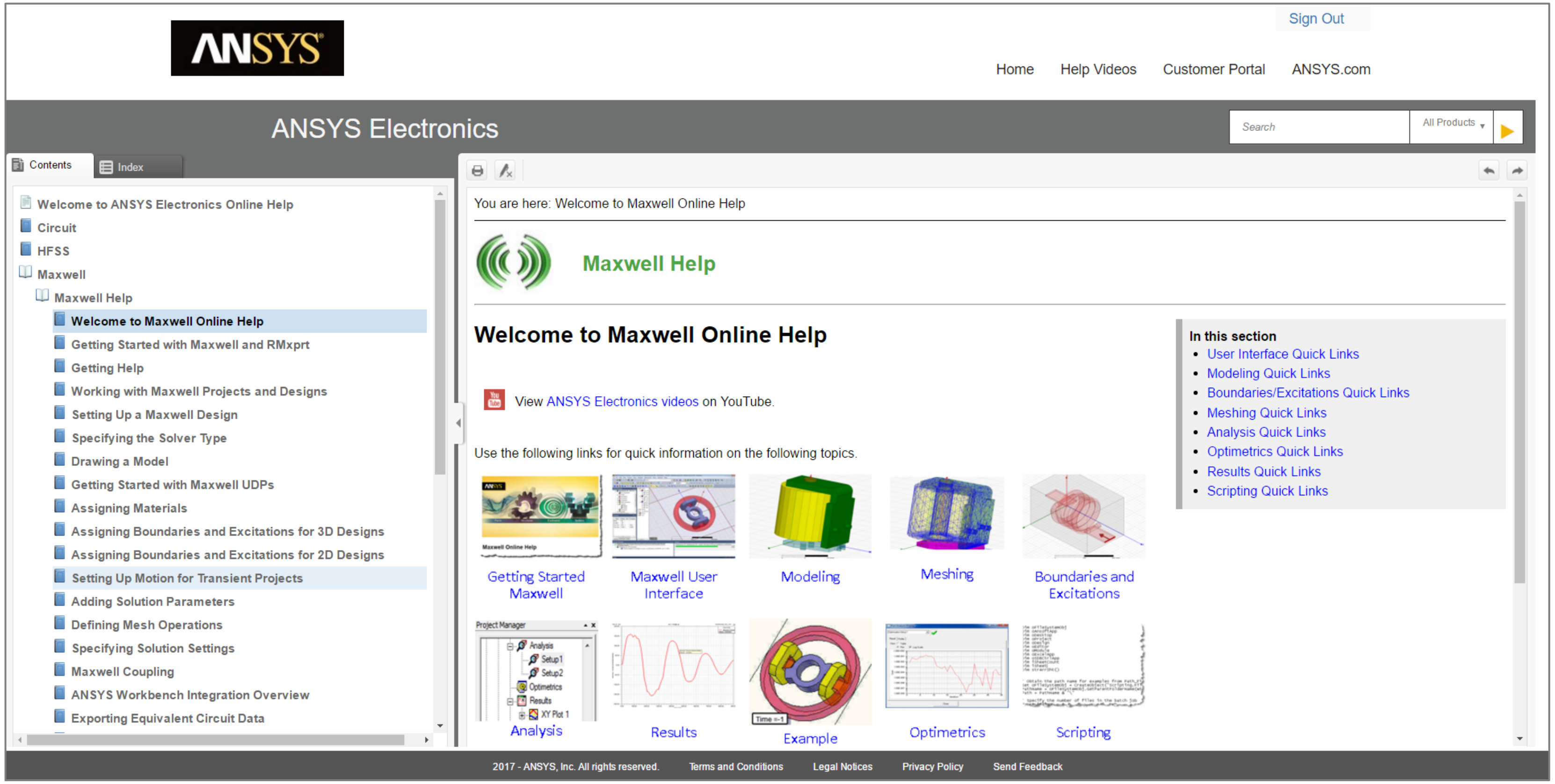


Maxwell Help

- **Maxwell Help manual can be accessed from following locations:**

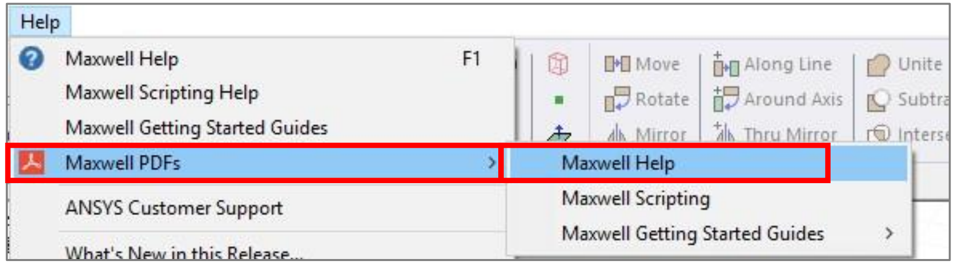
- **From menu item *Help → Maxwell Help* or using the shortcut F1:**
- **This will open the following web page**



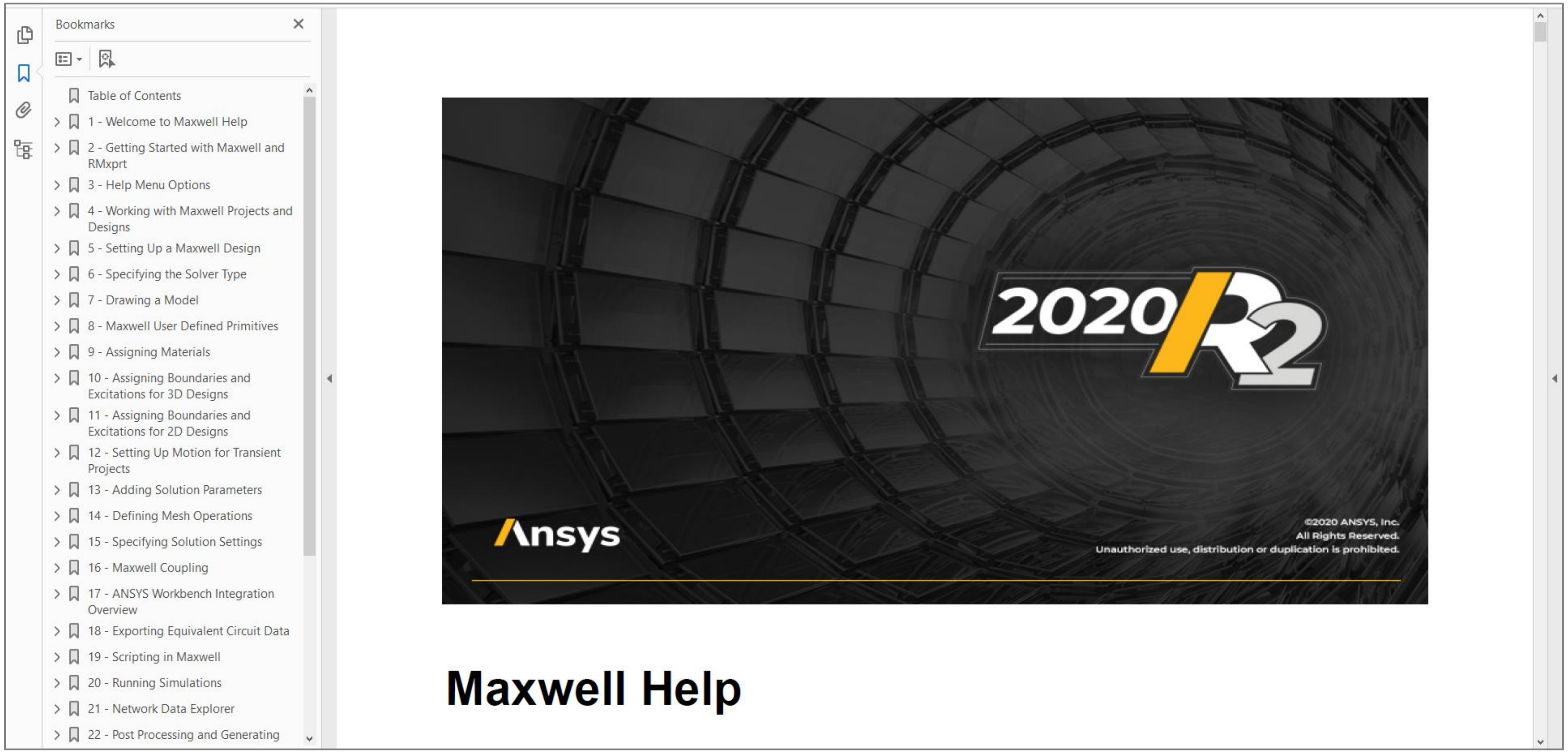


Maxwell Help

- **From menu item *Help → Maxwell PDFs → Maxwell Help:***



- **This will open the following file:**



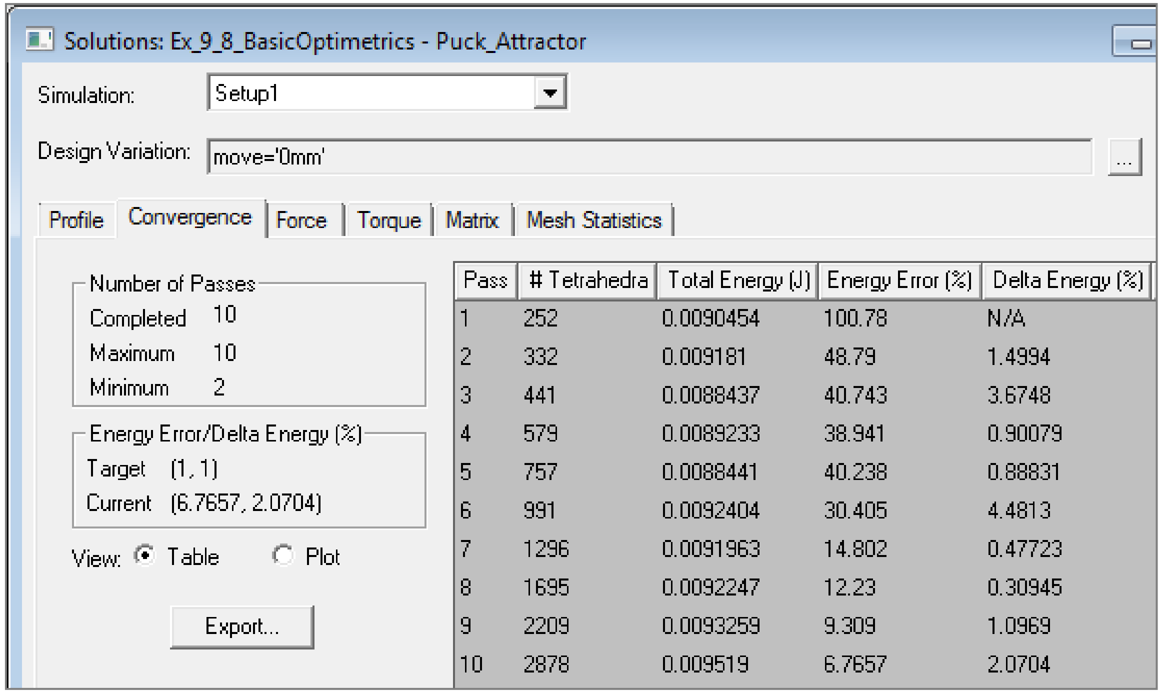
Ansys

Adaptive Meshing and Magnetostatic Solver

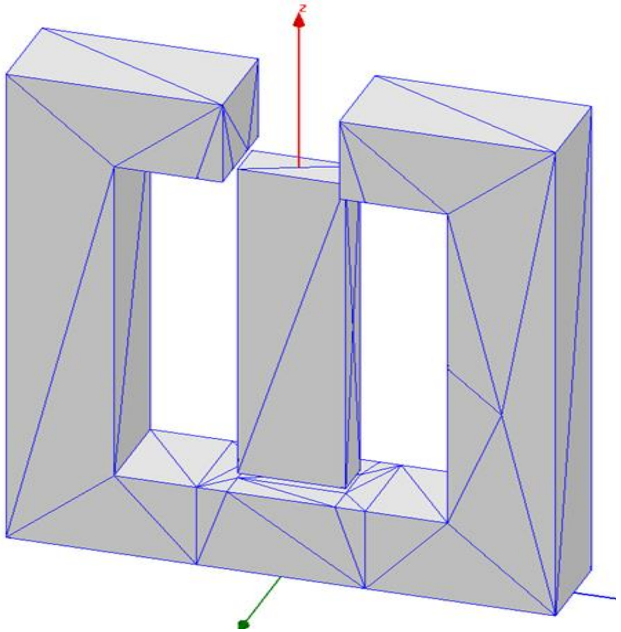
Ansys

Maxwell Adaptive Meshing

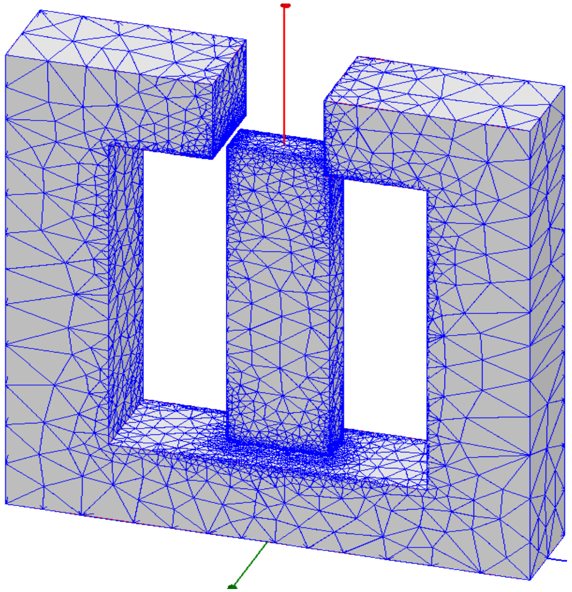
- **Adaptive Meshing**
  - For most of the cases, initial mesh is very coarse and more or less uniform in size throughout the region
  - To achieve required level of accuracy in results, this mesh needs to be refined in areas where fields are of interest or the field gradients are high
  - Adaptive meshing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



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



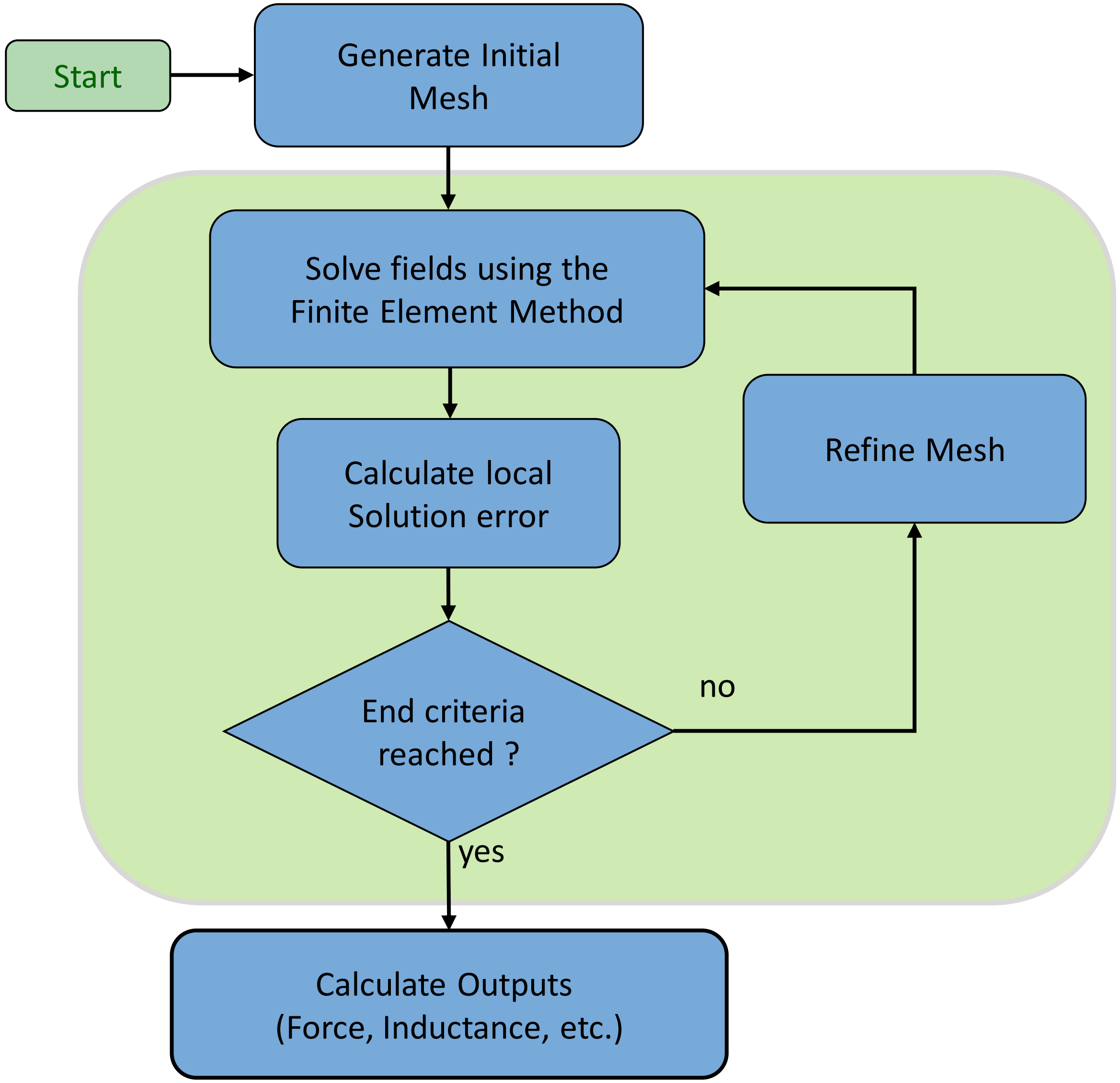
Initial Mesh



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**



Maxwell 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 the automatic adaptive mesh refinement technique
- Magnetostatic Equations
  - Following Maxwell’s equations are solved with Magnetostatic solver

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

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

$$\boldsymbol{B} = \mu_0 \mu_r(\boldsymbol{H}) \cdot \boldsymbol{H}$$

Maxwell 3D

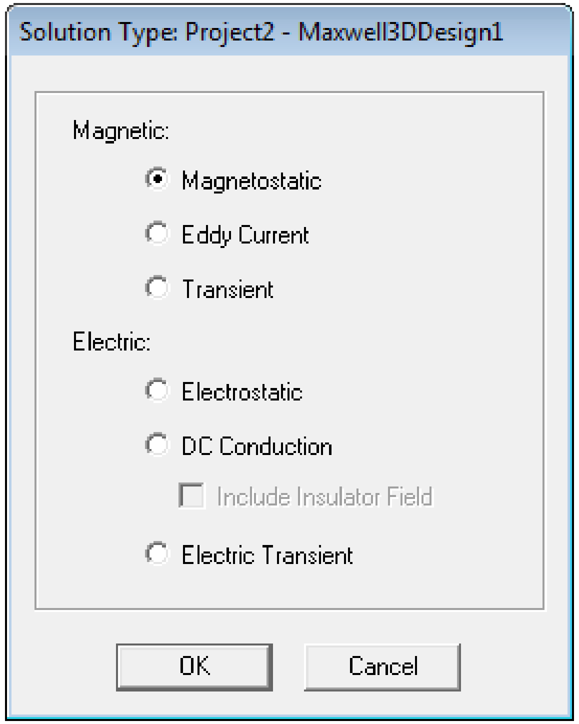
$$\boldsymbol{J_z}(x,y) = \nabla \times \left(\frac{1}{\mu_0 \mu_r} \cdot \left(\nabla \times \boldsymbol{A}_z(x,y) \right) \right)$$ **Cartesian XY**

$$\boldsymbol{J_\varphi}(r,z) = \nabla \times \left(\frac{1}{\mu_0 \mu_r} \cdot \left(\nabla \times \boldsymbol{A}_\varphi(r,z) \right) \right)$$ **Cylindrical about Z**

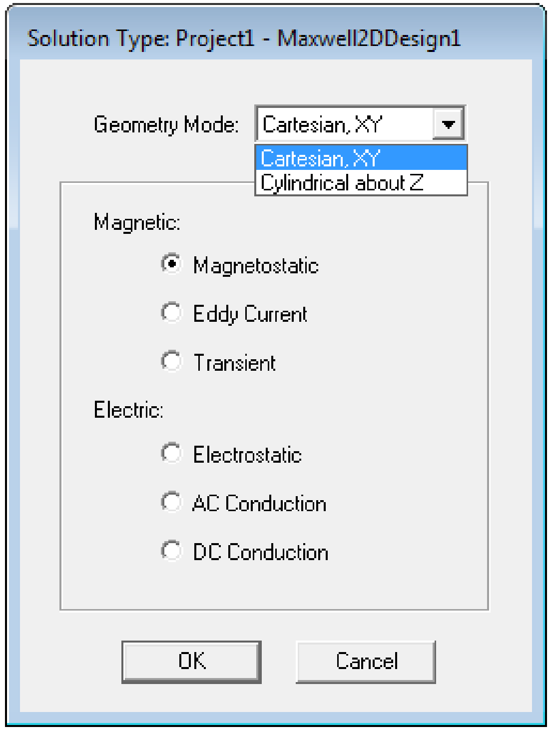
Maxwell 2D

Selecting the Magnetostatic Solver

- **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***
  - **In Maxwell 2D it is possible to choose between Cartesian XY and Cylindrical about Z symmetry**



Maxwell 3D

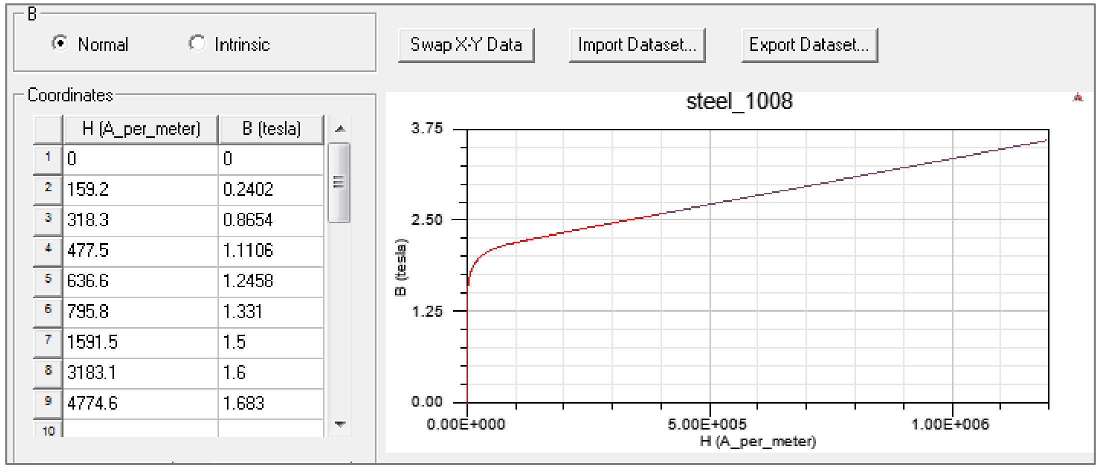


Maxwell 2D

Materials 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
    - Relative permeability can be either Simple (linear μr) or Nonlinear (BH Curve) or/and anisotropic
  - 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
  - Composition
    - Can be Solid or Lamination
    - Requires specification of direction normal to laminated pack and stacking factor

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	
Composition		Lamination	
- Stacking Factor	Simple	0.95	
- Stacking Direction		V(1)	

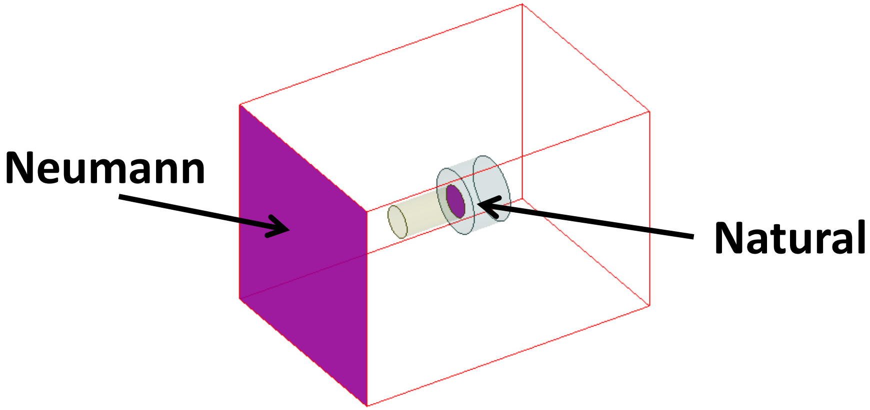


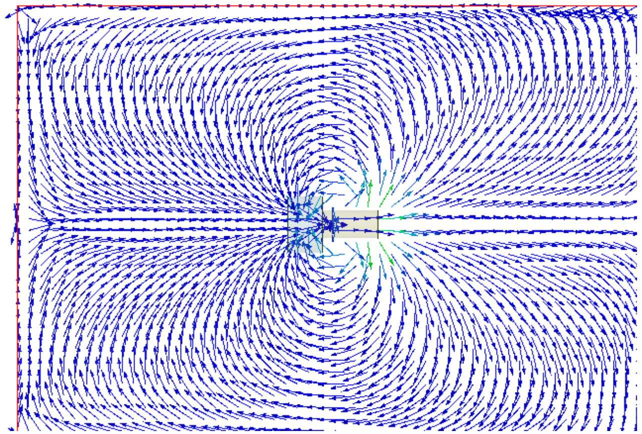
Ansys

Boundary Conditions

- Assigning Boundary Conditions
  - Boundary conditions define behavior of the magnetic field at the interfaces or the edges of the problem region
  - A boundary can be assigned to an edge/face from menu item ***Maxwell 2D/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. It describes the natural variation from one material to the next one, as defined by material properties
  - **Neumann**: For exterior boundaries of solution domain. H Field is tangential to the boundary and flux cannot cross it



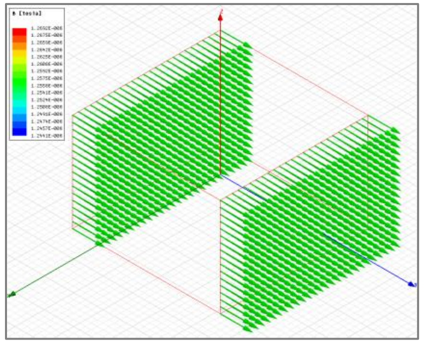


Boundary Conditions

- Boundary Types (3D)

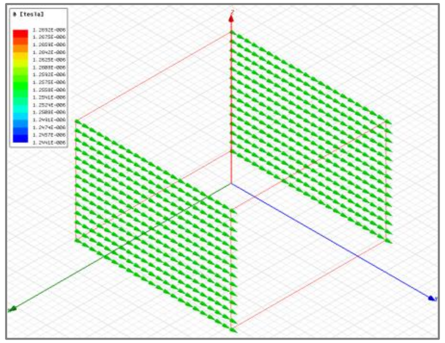
  – **Zero Tangential H-Field**
    - **Useful to assign external field.**
    - **H field is normal to assigned surface**
    - **Applied to external boundaries of the domain**

H field on Zero Tangential H field boundary

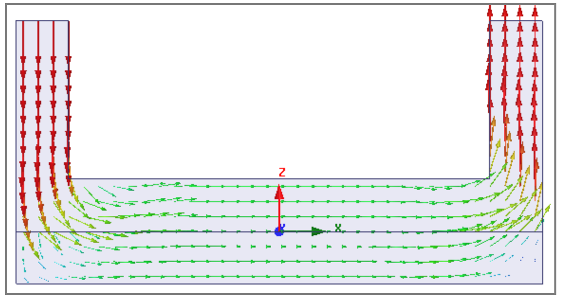


  – **Tangential H-Field**
    - **Useful to assign external field.**
    - **Tangential H field applied using U and V vectors**
    - **Applied to external boundaries of the domain**

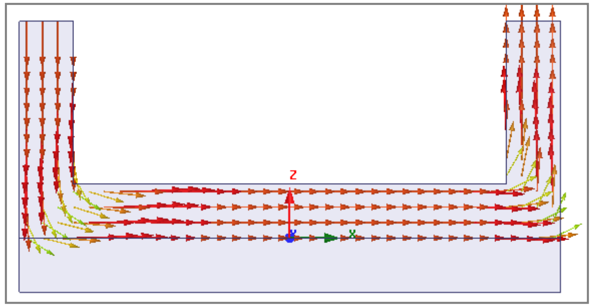
H field on Tangential H field boundary



  – **Insulating**
    - **Same as Neumann, except that current cannot cross the boundary.**
    - **Used to insulate two conductors which are in contact with each other**



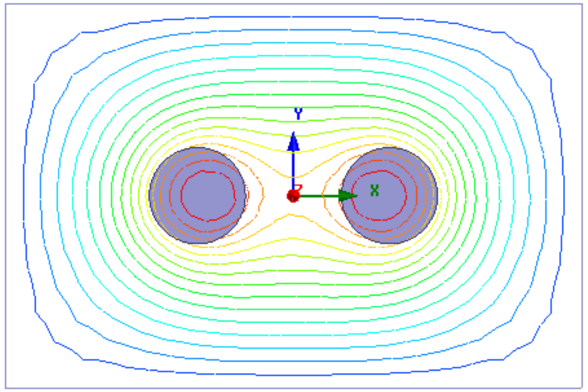
J Vectors without insulating boundary



J Vectors with insulating boundary

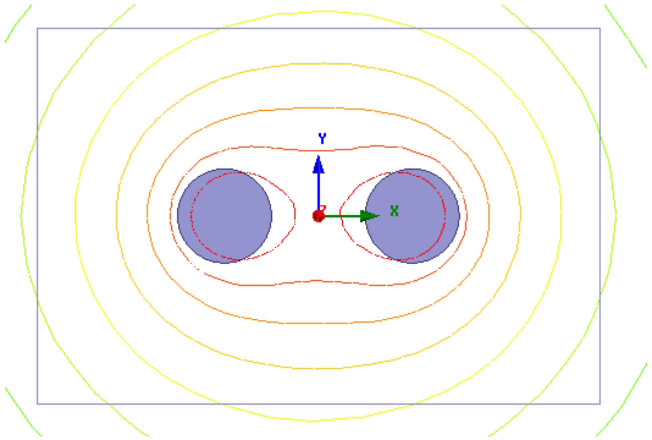
Boundary Conditions

- 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

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

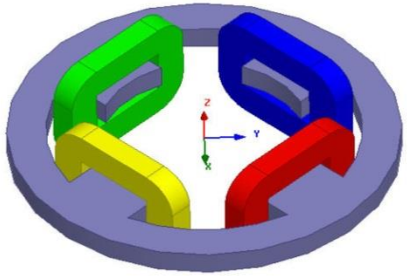


Flux lines with Balloon

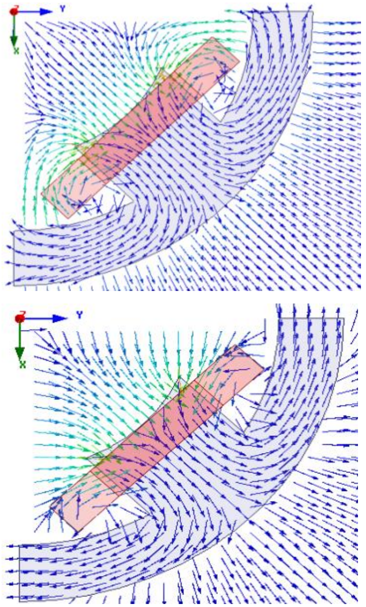
- **Note: In 2D, no default boundary condition is assigned to the simulation region edges. Users have to specify the behavior of simulation boundaries by assigning either Balloon or Vector Potential boundary.**

Boundary Conditions

- **Boundary Types (2D & 3D):**
  - **Independent/Dependent :**
    - **Enables to model only one period of a periodic structure, reducing design size**
    - **Magnetic fields at Independent and Dependent boundaries match each other**



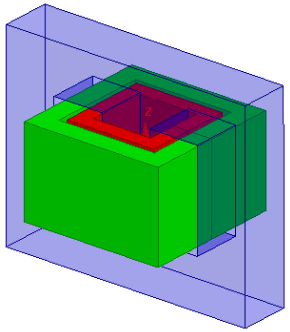
1/4th Model



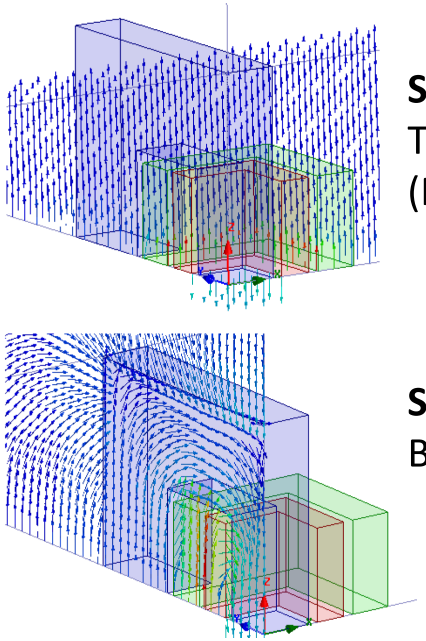
Dep = Indep

Dep = - Indep

  - **Symmetry Boundary:**
    - **Enables to model only part of a structure, reducing design size**



1/8th Model



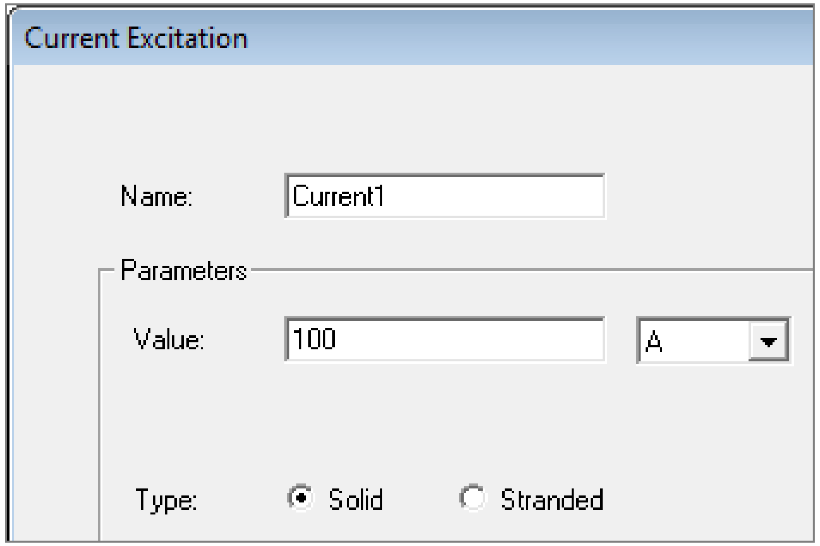
Symmetry Even: Same as Zero Tangential H-Field boundary (Flux Normal)

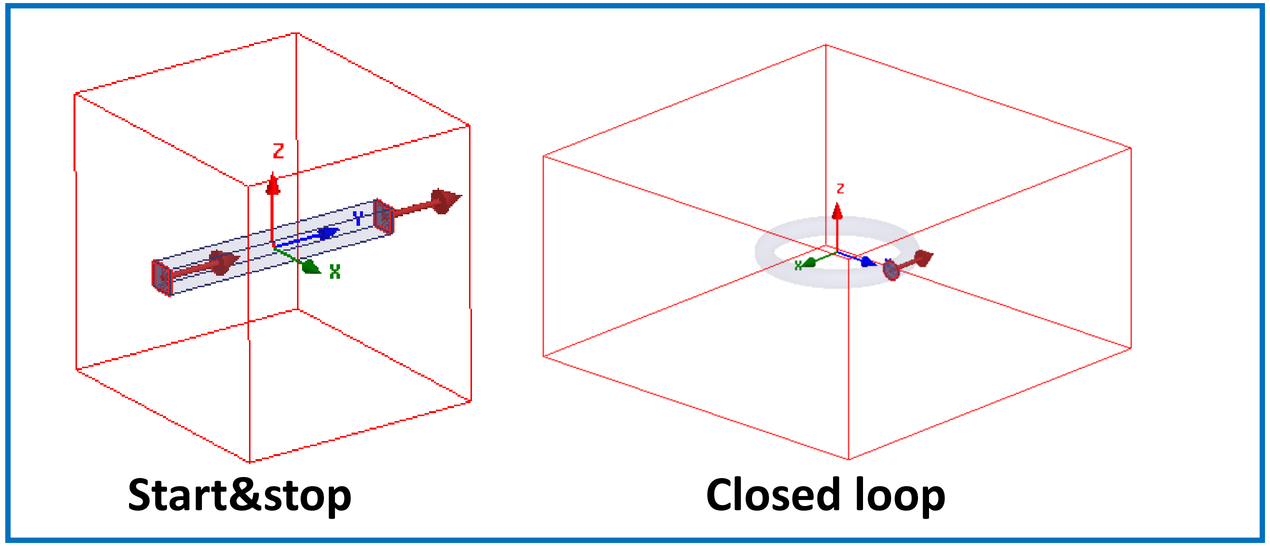
Symmetry Odd: Same as default Boundary (Flux Tangential)

Ansys

Excitations (energy sources)

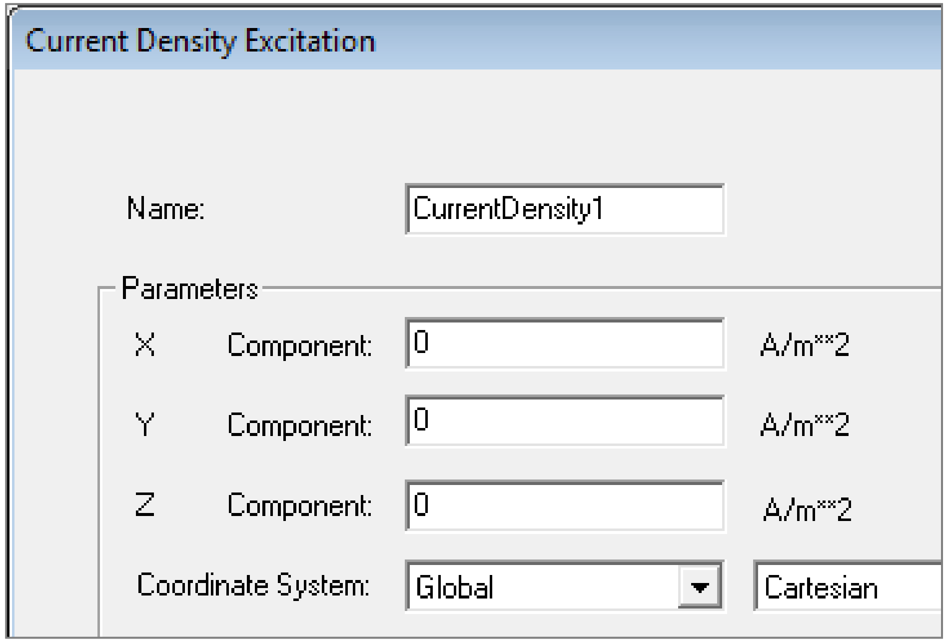
- **Excitations (2D & 3D)**
  - **Excitations can be assigned from the menu item *Maxwell 2D/3D → Excitations → Assign***
- **Current**
  - **Used in most magnetostatic analysis cases,**
  - **Assigned to 2D sheets (conductor sections) defining total current in Amp-turns through the conductors**
  - **The conducting path can be either a closed loop or start&stop on simulation domain outer boundary**
  - **Conductor defined as Solid or Stranded**



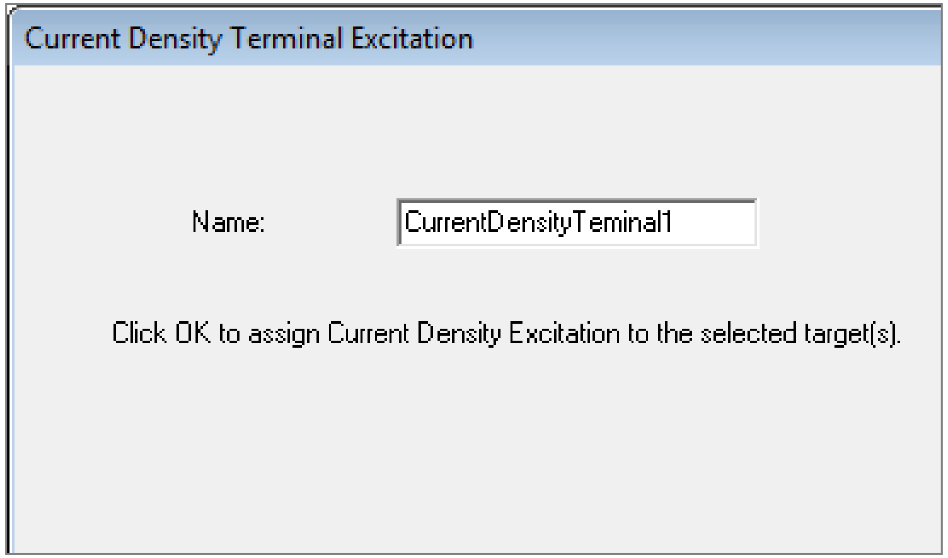


Excitations (energy sources)

- Excitations (2D and 3D)
  - Current Density
    - In practical applications this kind of source is seldom used
    - In 3D, this definition should be accompanied with Current Density Terminal definition
    - Defined using X,Y and Z components of selected CS



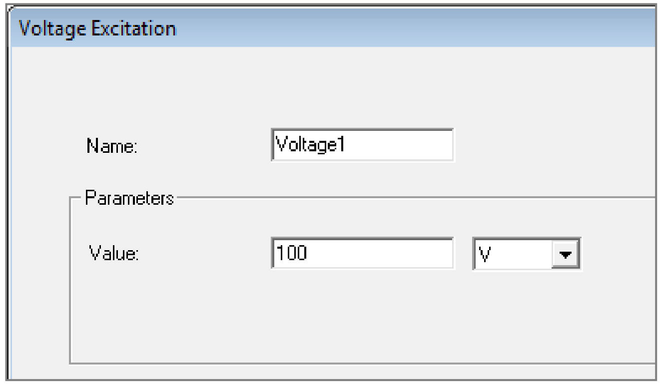
  - Current Density Terminal (3D only)
    - Required to be defined if Current Density is set
    - Can be assigned for either closed loop or start&stop conducting paths

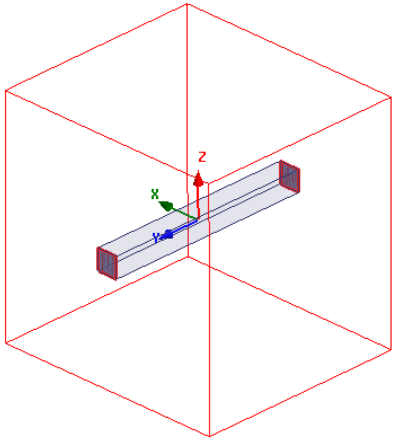


Excitations (energy sources)

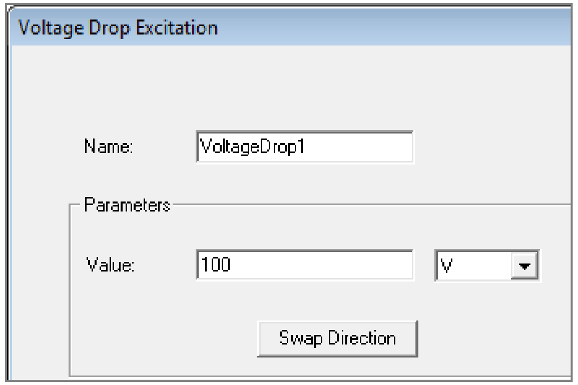
- Excitation (3D)

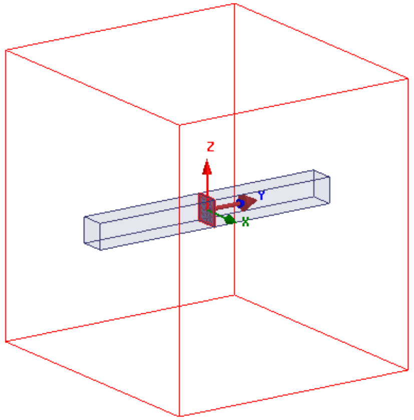
– **Voltage**
  - Used in conjunction with material conductivity to define current through a solid conductor
  - Can only be assigned to faces or sheets lying on simulation domain boundaries





– **Voltage Drop**
  - Similar to the voltage definition
  - Can only be assigned to conductor sections





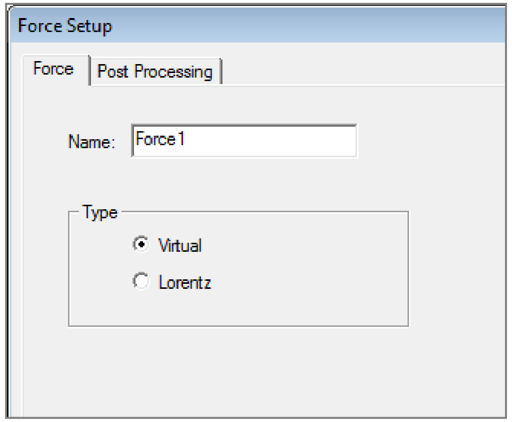
Excitations (energy sources)

- Further Energy Sources (2D and/or 3D)
  - Permanent Magnets (2D and 3D)
    - Defining objects as permanent magnets (through material properties) represents setting an energy source
  - Tangential H-Field (3D)
    - Defining the boundary Tangential H-Field is often used to create an external H-field surrounding all the devices under investigation and it is considered by the solver as an energy source
  - Vector Potential Gradients (2D)
    - Defining Vector Potential gradients on 2D boundaries is used to create external magnetic field surrounding all the devices under investigation and it is considered by the solver as an energy source

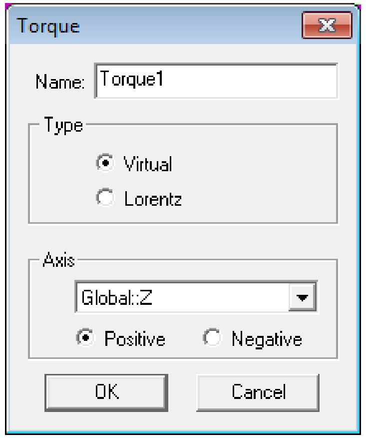
Parameters

- Parameters
  - Three parameters can be assigned for magnetostatic solver: Force, Torque, Matrix
  - 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

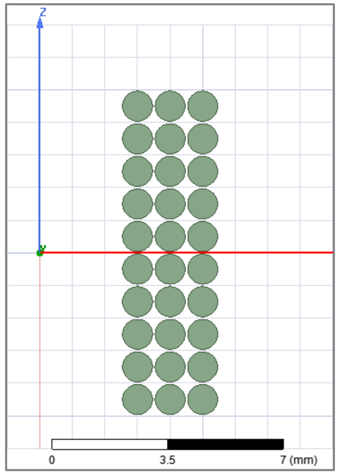


– **Torque:**
- Calculates torque on selected objects
- Can be Virtual or Lorentz
- The torque acting axis and direction must be set

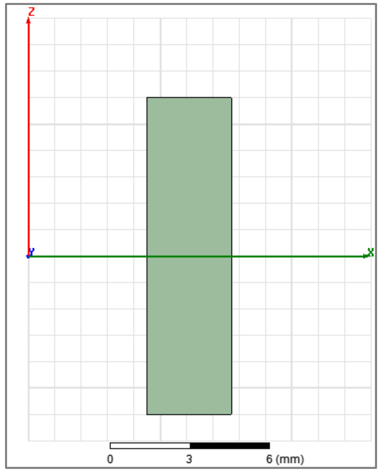


Parameters

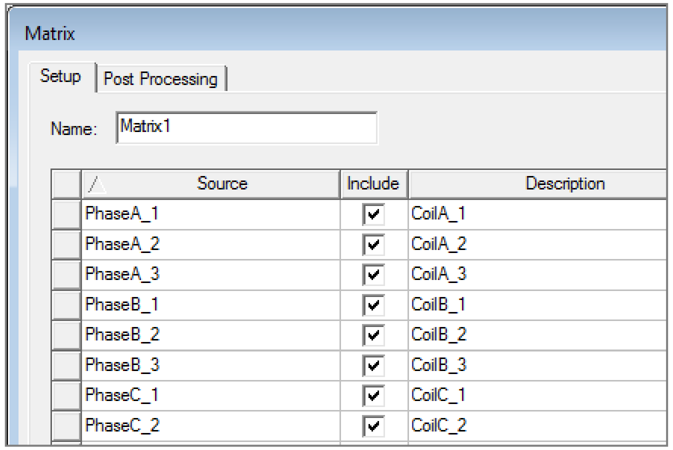
- **Matrix:**
  - Calculates Inductance matrix
  - It is often useful to model the winding turns with an equivalent cross section, in order to avoid mesh issues and to reduce simulation time
  - In that case Matrix allows postprocessing where it is possible to insert the number of turns and define eventual groups and branches
  - When grouping, the excitations are implicitly series connected each other

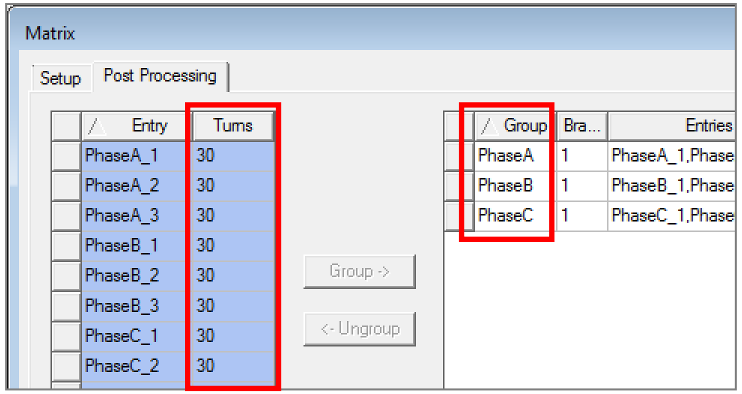


Individual turns modeling



Equivalent cross section

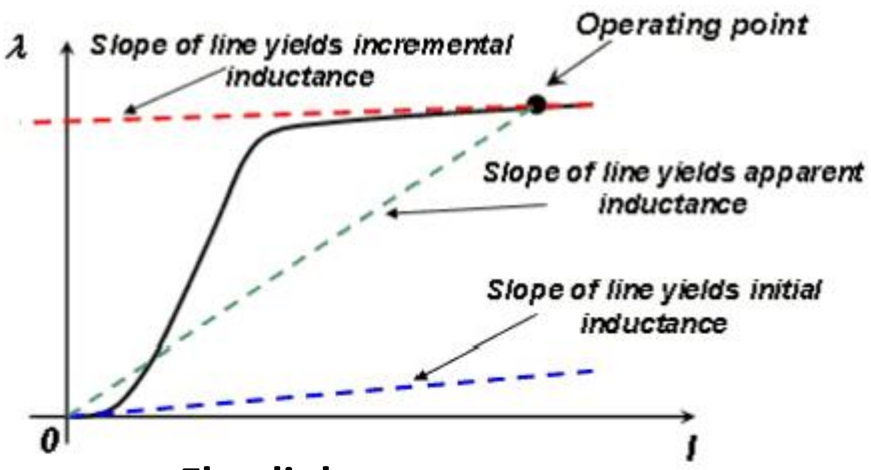




Ansys

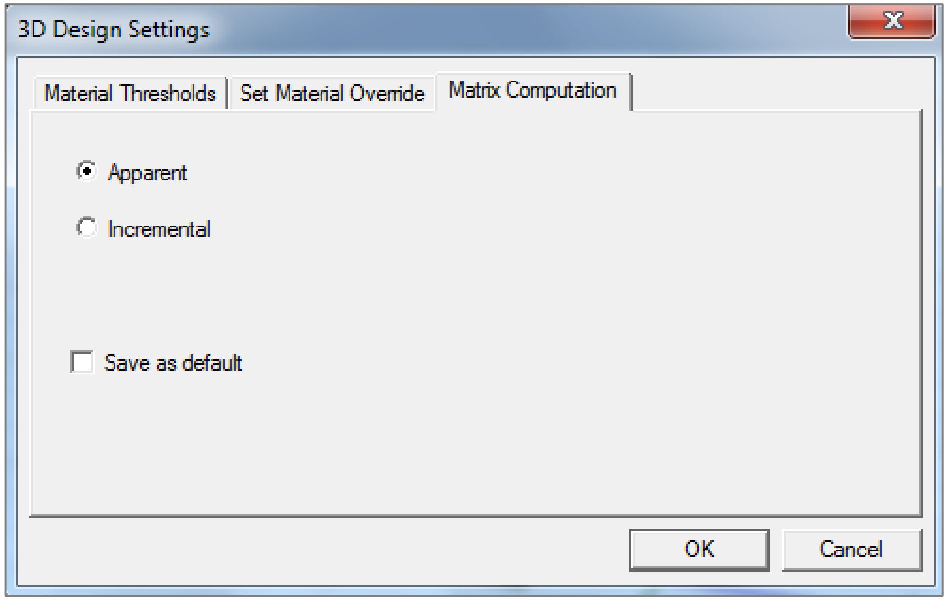
Inductance Matrix Calculation

- **Two types of inductance calculation:**
  - **Apparent Inductance**
    - **The slope of a line from the origin to the operating point. Apparent in function of current.**
  - **Incremental (differential) Inductance**
    - **The slope of the tangent to the curve at the operating point. Incremen definition.**



Flux linkage vs. current different inductances

$$L_{app} = \frac{\lambda}{I} \qquad L_{incr} = \frac{d\lambda}{dI}$$



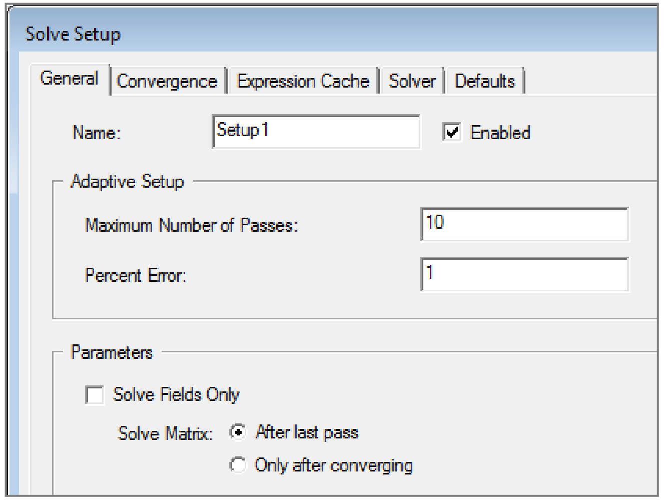
- **Inductance calculation method can be selected through menu item**

Maxwell 3D/2D → Design Settings → Matrix Computation

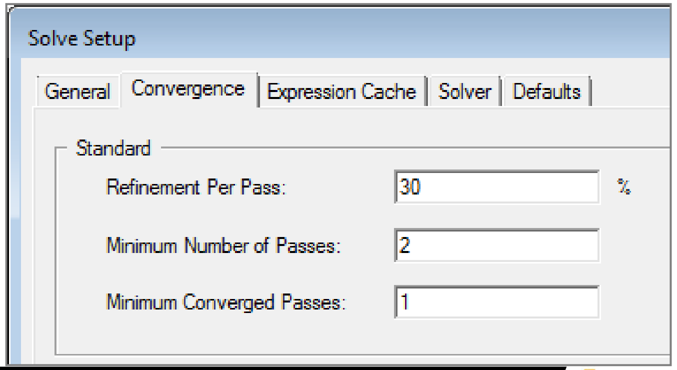
Analysis Setup

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

  - General Tab
    - Name: Setup name. Multiple setups can be present in the same design
    - Maximum Number of Passes: limit to number of adaptive passes
    - Percent Error: Error goal for both Error Energy and Delta Energy
    - Solve Fields Only: Ignores any defined parameters if checked
    - Solve Matrix: option to calculate the matrix after the last solved pass or only if the solution converges

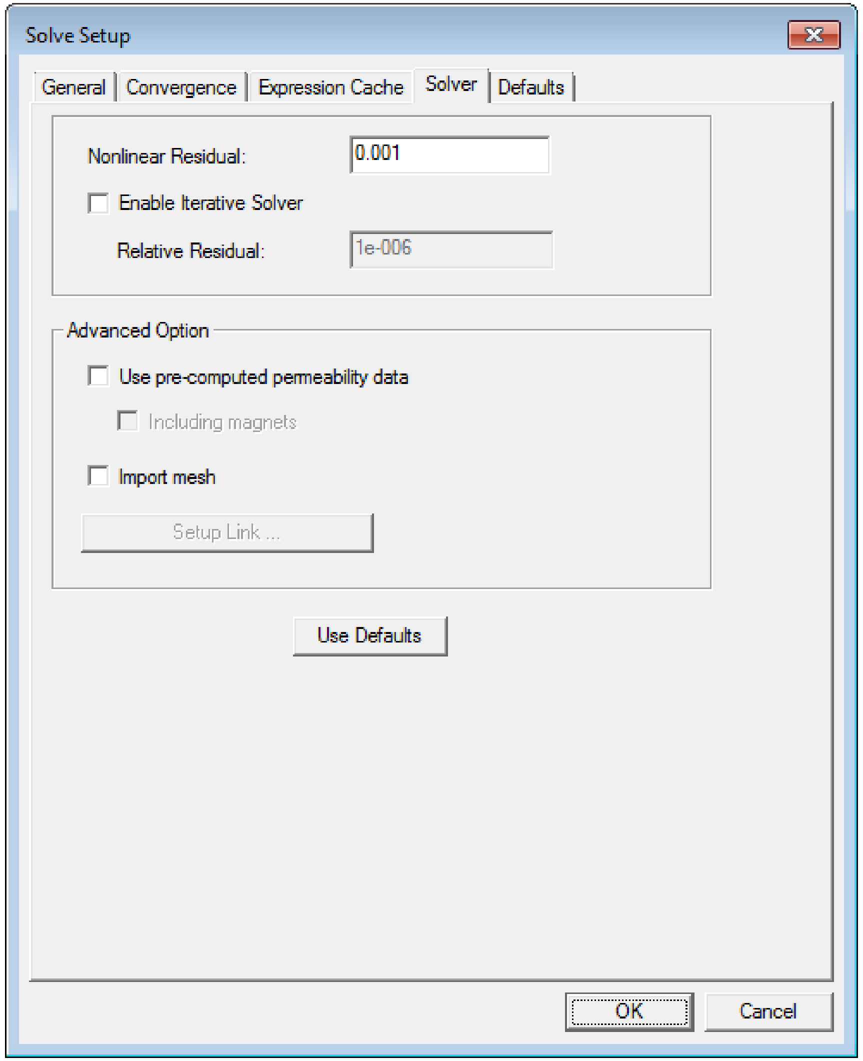


  - Convergence Tab
    - Refinement Per Pass: number of tetrahedral elements added at each pass as a percentage increment
    - Minimum Number of Passes: minimum number of adaptive passes before solution stops
    - Minimum Converged Passes: minimum number of converged adaptive passes before solution stops



Analysis Setup

- Solution Setup
  - Solver Tab
    - Nonlinear Residual: error in finding operating points along B-H curve
    - Enable Iterative Solver: Enables ICCG solvers (Direct is the default)
    - Permeability Option: allows operating points either to be calculated from Nonlinear B-H curve or to use frozen permeabilities From Link (linked model 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

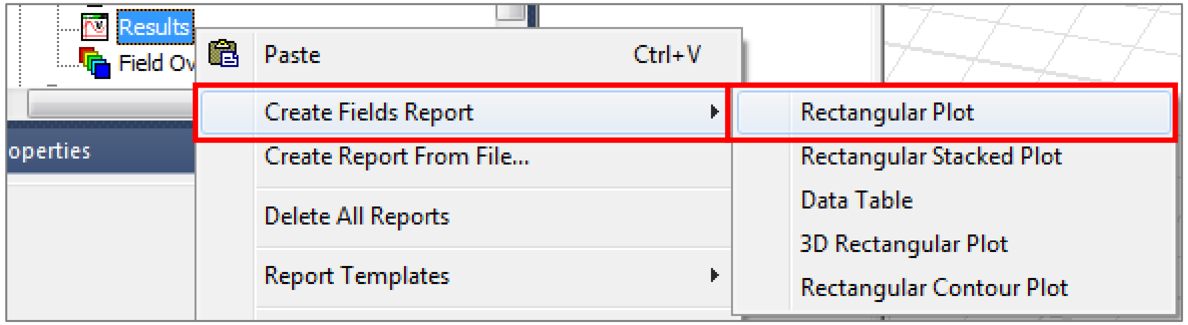


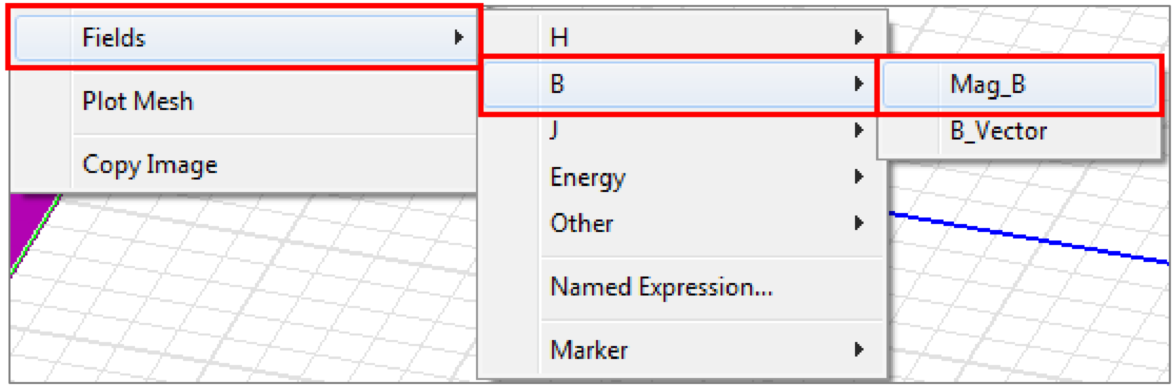
Variables and Parametric analysis

- Variables
  - All the geometry dimensions, sources and in general most of the settings can be parametrized through suitable variables
  - Variables can be either Local (Design) or Global (Project)
  - Local variables are valid only inside the Design where they have been defined
  - Global variables begin always with the $ symbol and are shared through all Designs belonging to the same Project
- Parametric analysis
  - The Optimetrics module offers the possibility to sweep variables values and run a set of (even numerous) different simulations
  - Parametric analysis is often used for model performances optimization or to study the influence of different parameters on devices performances.
  - Parametric analysis ca be added through RMB on *Optimetrics → Add → Parametric*

Postprocessing – Part 1

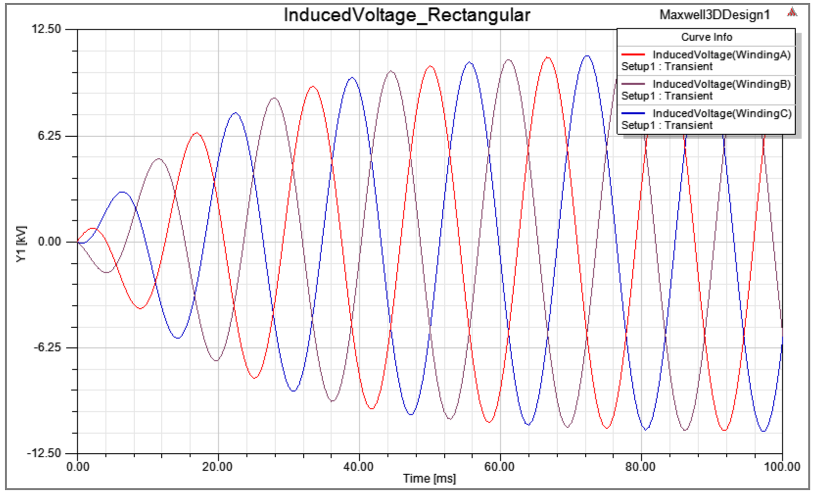
- Results Reports
  - Maxwell calculates the fields distribution and then other quantities, like, e.g. forces, torques, inductances, etc.
  - Those quantities can be accessed from Project manager through a *RMB on Results → Create Fields Report → (select one report from the list)*
  - Results are displayed on a XY graph where the quantities on the two axis can be selected at own convenience
- Fields Overlays
  - Selecting objects which are part of the model, it is possible to display the calculated field distribution, as well as the mesh
  - Fields can be displayed using *RMB on selected objects → Fields → (select one field from the list)*

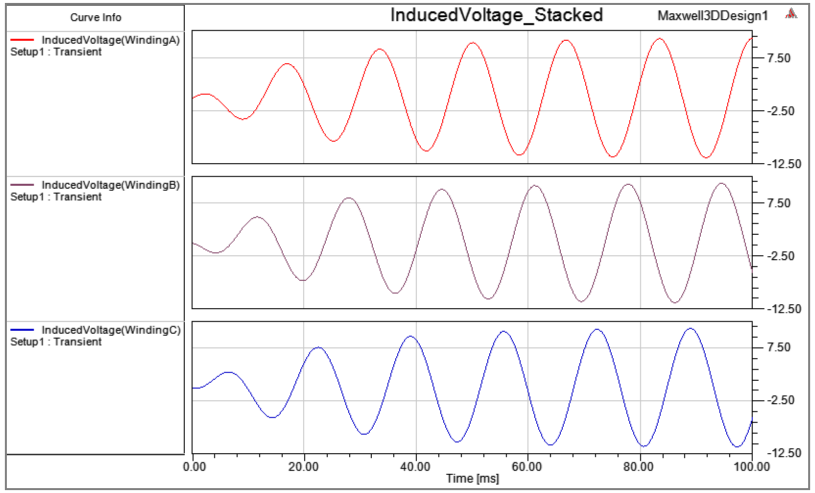




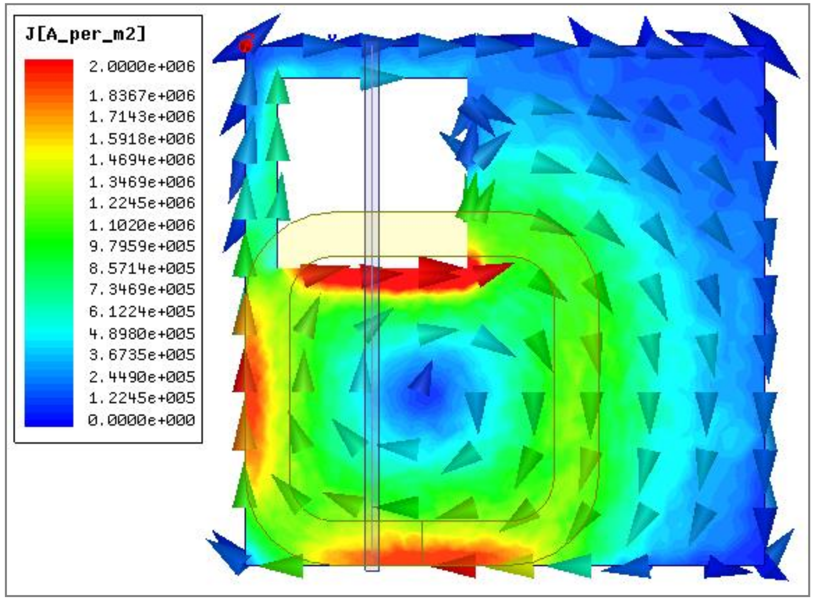
Postprocessing – Part 1

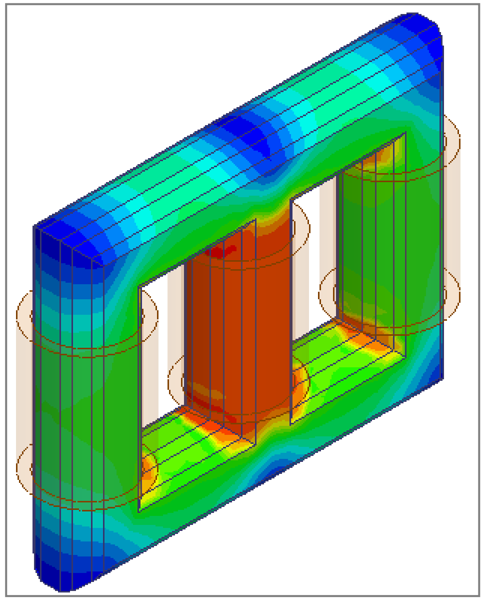
- **Results Reports examples**





- **Fields Overlays examples**





Ansys

Summary

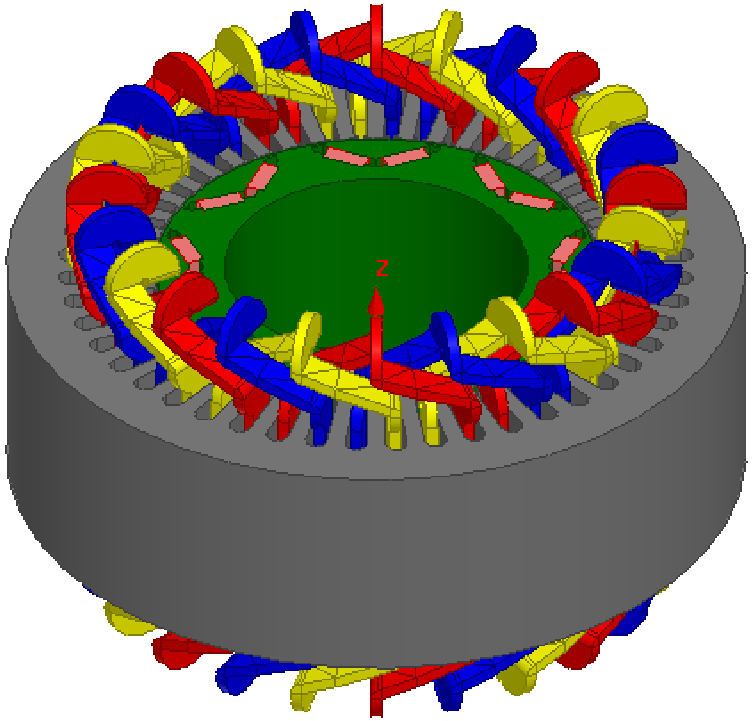
What have we learned in this lecture?

- **Maxwell settings and GUI**
- **Material Properties**
- **Boundary Conditions**
- **Excitations**
- **Adaptive meshing**
- **Magnetostatic solver**
- **Parametric analysis**
- **Basic of postprocessing**

APPENDIX: Geometries in Maxwell

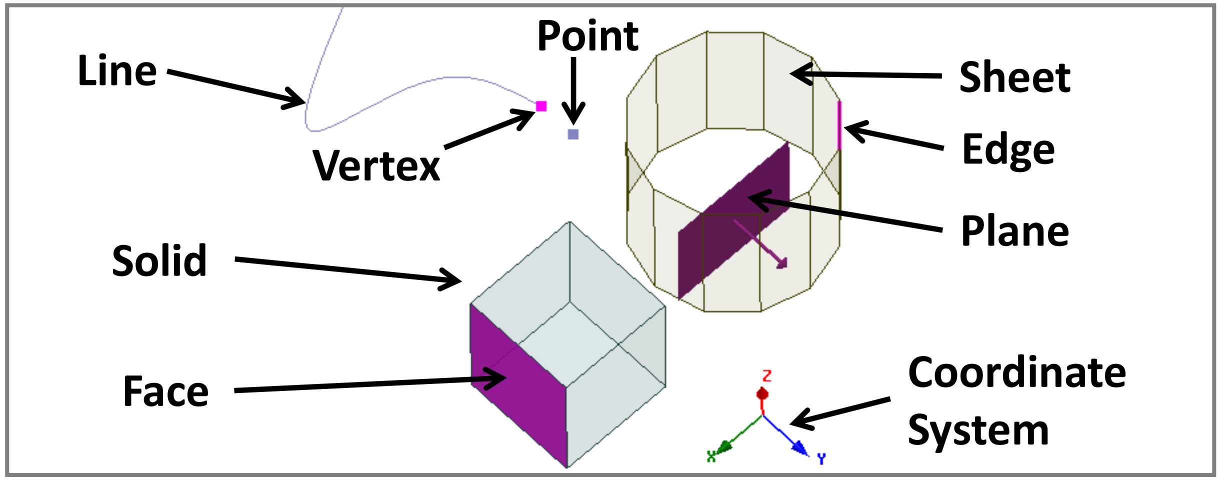
Maxwell Geometry

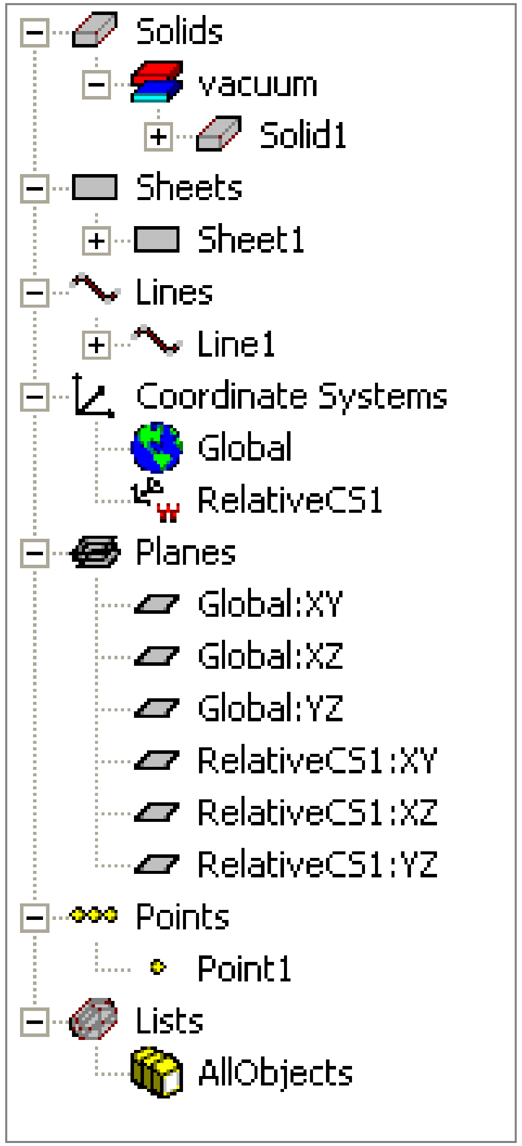
- **Maxwell Geometry**
  - **Ansys Maxwell uses primitives-based modeling technique where basic structure is created using Geometry Primitives and then Geometry operations are performed to achieve final object**
  - **The underlying solid modeling technology used by Maxwell products is provided by ACIS geometric modeler. ACIS version 27.0 is used in Maxwell R18.0**
  - **Geometry can also be created in Ansys Design Modeler and can be used in Maxwell through Ansys Workbench interface**



Geometry Terminology

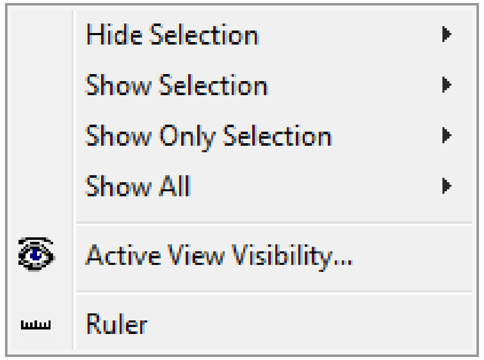
- **Geometry Terminology**
  - **Point:** Independent point entity
  - **Vertex:** Point entity that is a part of an Edge or Line object
  - **Line Object:** Independent line entity
  - **Edge:** Line entity which is part of either a sheet or a Face
  - **Sheet Objects:** Independent surface entity
  - **Face:** Surface entity that is part of a Sheet or a Solid
  - **Solid Objects:** Entities that have a definite volume
  - **Coordinate Systems:** Default “Global”
  - **Planes:** Default Global XY, YZ and XZ planes

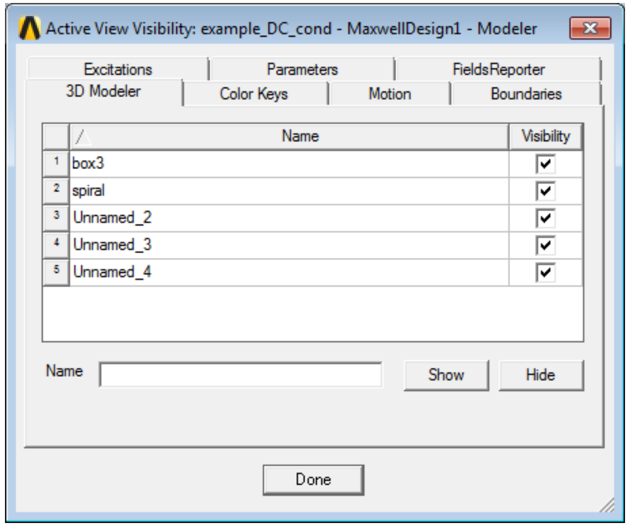




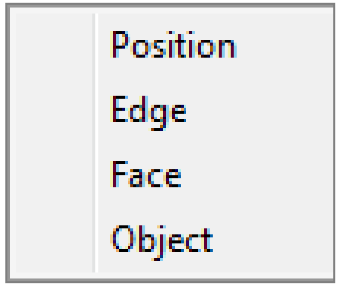
Geometry UI operations

- **Setting Visibility of the Object**
- **Visibility of the object in 3D Modeler window can be set from menu item *View → Visibility* or the toolbar**





- **Geometry Measurement**
- **Measurement command can be launched from the menu item *Modeler → Measure***



Measures position of cursor with reference to Current CS or last selected position
Measures length of an edge
Measures Area of a face
Measures Volume and Surface Area of an Object

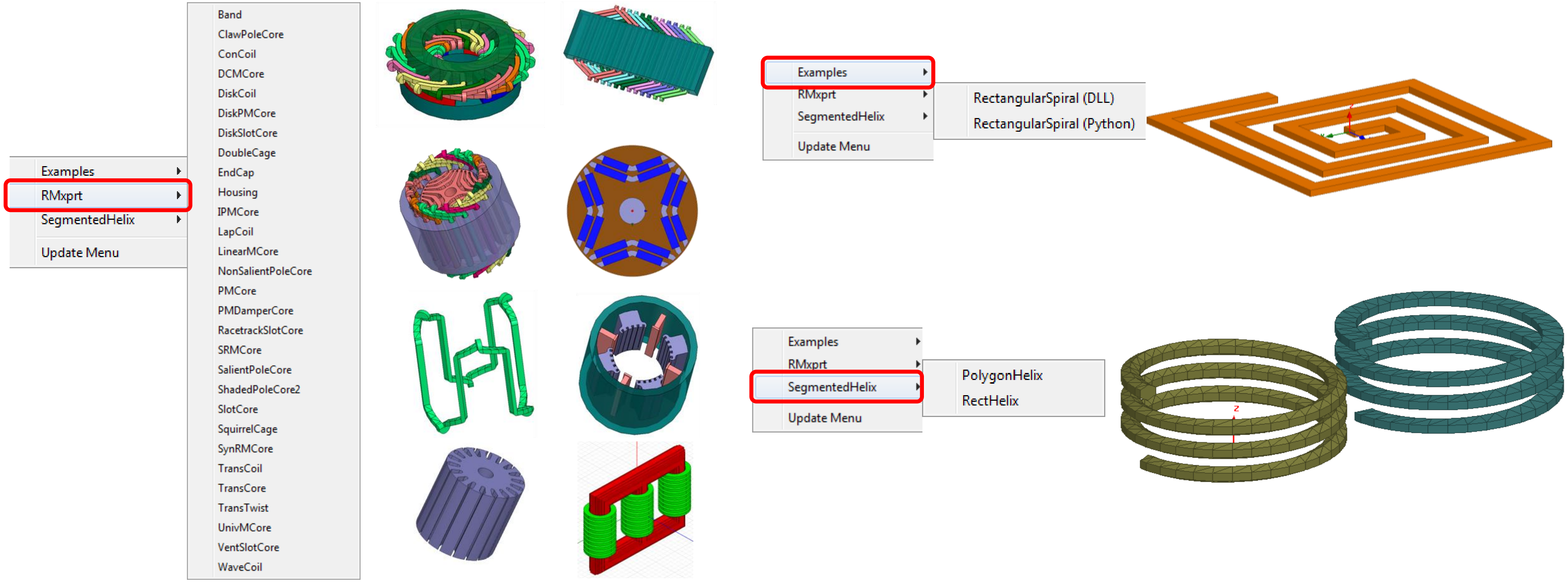
Geometry Creation

- **Geometry Creation from Primitives**
- **Geometrical entities can be created using primitives from the menu bar Draw**

Object type	Primitive	Inputs
Line Objects	Line	Start and End Points
	Spline	Start, intermediate and end points
	Arc ›	Coordinates of Three points of arc or Center and two end points
	Equation Based Curve	X, Y Z Coordinates as a function of the variable “_t”
Sheet Objects	Rectangle	Coordinates of two diagonal points
	Ellipse	Coordinates of center and major, minor radius
	Circle	Coordinates of center and Radius
	Regular Polygon	Coordinates of center, Radius and Number of sides
	Equation Based Surface	X, Y Z coordinates as a function of the variable “_u” and “_v”
Solid Objects	Box	Coordinates of two diagonal points of base and height
	Cylinder	Coordinates of center of base and radius
	Regular Polyhedron	Coordinates of center of base, radius and number of segments
	Cone	Coordinates of center of Base, lower and higher radius and height
	Sphere	Coordinates of center and radius
	Torus	Coordinates of Center, Inner and Outer radius
	Helix	Sheet defining cross section, helix vector, pitch and turns
	Spiral	Sheet defining cross section, spiral vector, radius change and turns
	Bondwire	Two points and parameters of the wire in parameter dialog

Geometry Creation

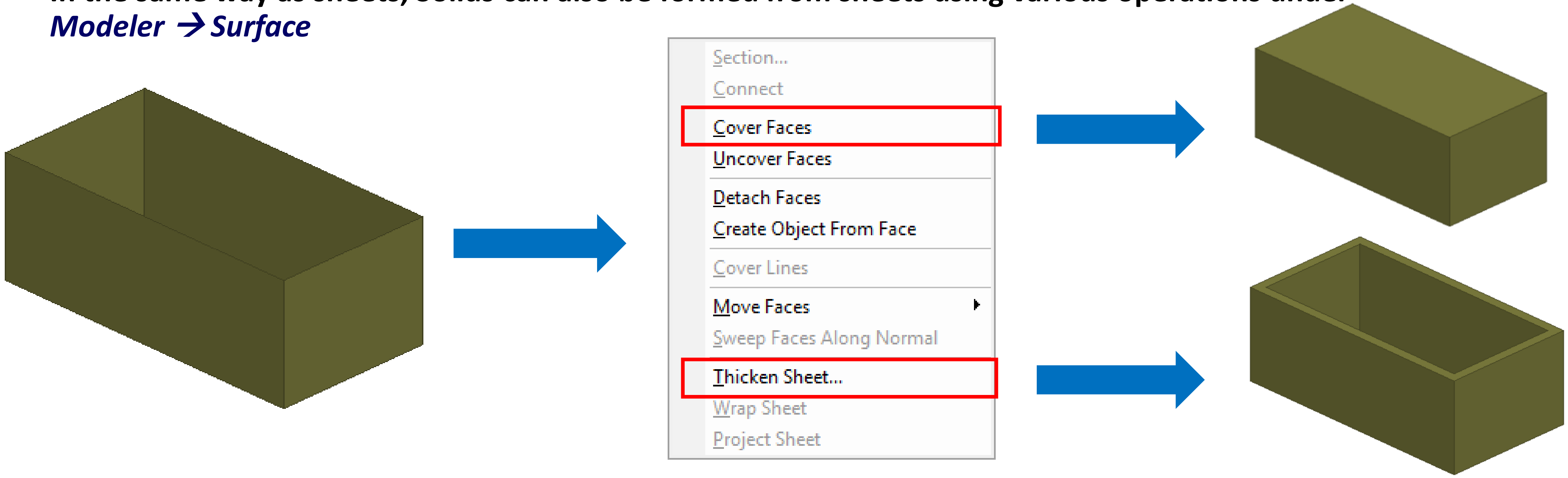
- **User Defined Primitives**
- **User defined primitives enable users to create and parameterize complex geometrical objects using inbuilt templates in Maxwell**
- **A User Defined Primitives can be added from the menu item *Draw → User Defined Primitive***



Note: Please refer Maxwell Help for more information on the User Defined Primitives

Geometry Operations

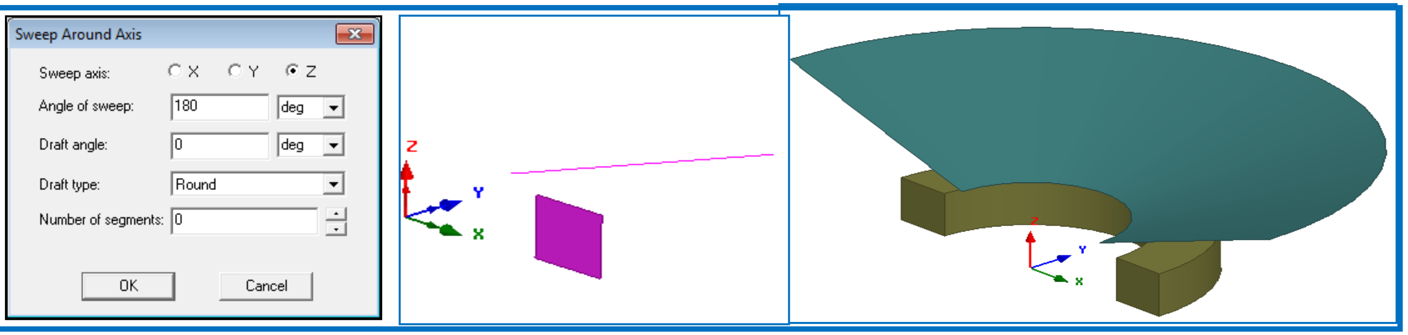
- Creating Sheets from Lines
  - Edge objects which form a closed loop can be used to create sheet object by selecting menu item *Modeler → Surface → Cover Lines*
  - This operations can be set as default execution on creation of closed polylines by setting the option under *Tools → Modeler Options → Automatically Cover Closed Polylines*
- Creating Solids from Sheets
  - In the same way as sheets, Solids can also be formed from sheets using various operations under *Modeler → Surface*



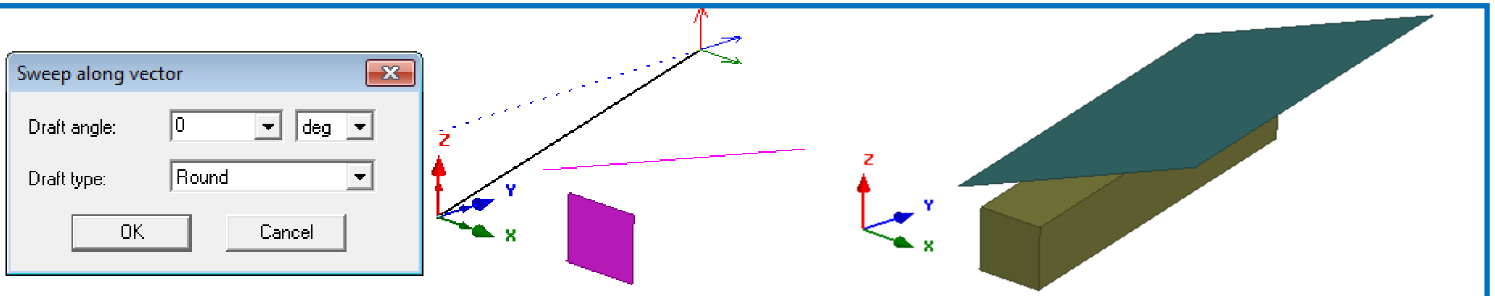
Geometry Operations

- Sweep Objects:
  - Sweep command can be accessed from menu item ***Draw → Sweep***

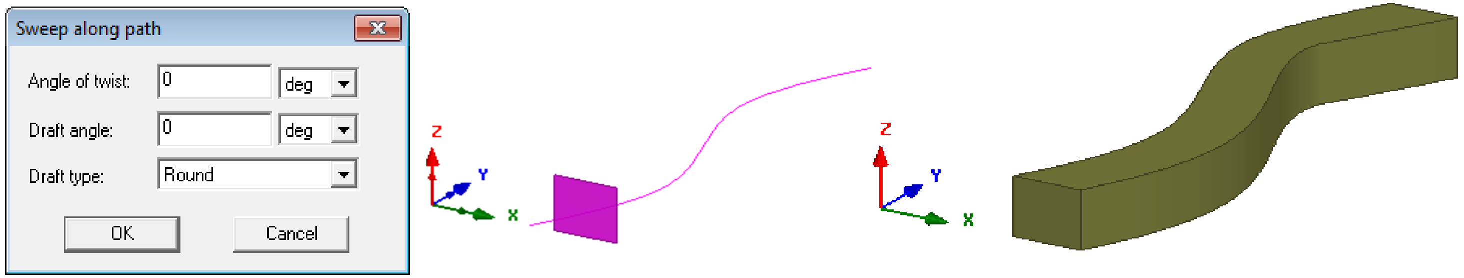
- **Sweep Around Axis**
  – Inputs: Profile Sheet or Line body, X,Y or Z axis of active CS, Angle of Sweep and Draft details if needed



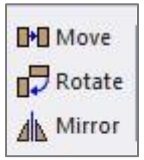
- **Sweep Along Vector**
  – Inputs: Profile Sheet or Line body, Sweep vector and Draft details if needed



- **Sweep Along Path**
  – Inputs: Profile Sheet or Line Body, Path Line Body, Twist and Draft details if needed

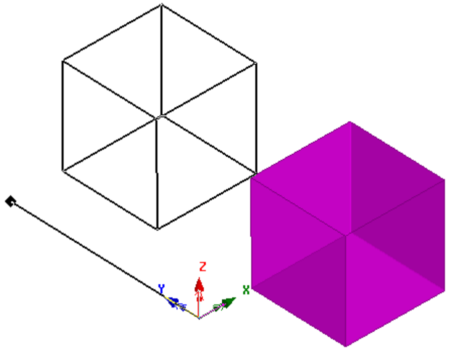


Geometry Manipulation

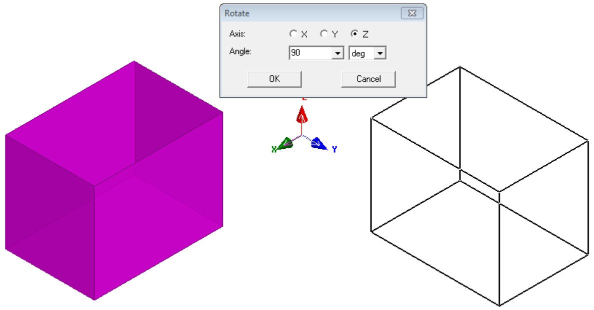


- **Arrange Geometry**
  - Arrange Geometry can be initiated from menu item ***Edit → Arrange***

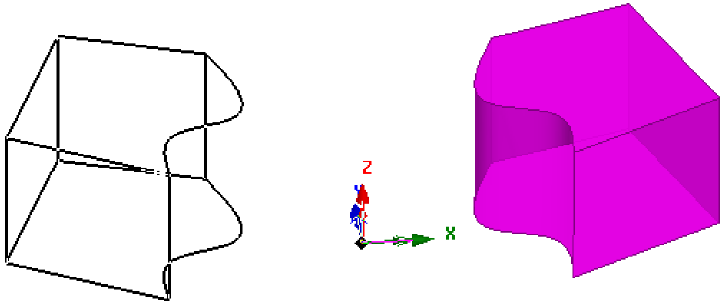
- **Move**
  - Translates geometry along the translation vector



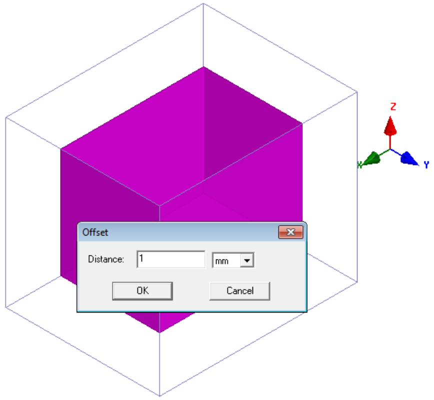
- **Rotate**
  - Rotates the geometry around selected axis of Work CS



- **Mirror**
  - Mirrors geometry about a plane defined using Plane Normal



- **Offset**
  - Scales the geometry abo its Centroid. Applicable only for Solids.

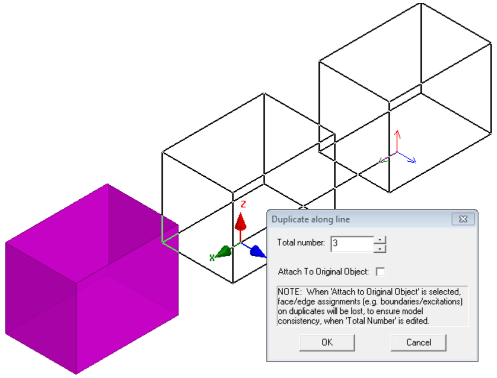


Ansys

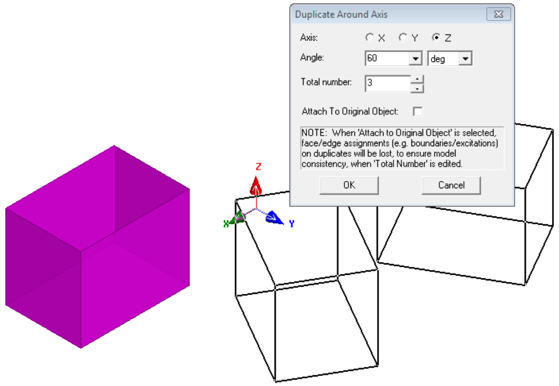
Geometry Manipulation

- Duplicate Geometry
  - Duplicate Geometry can be initiated from menu item *Edit → Duplicate*

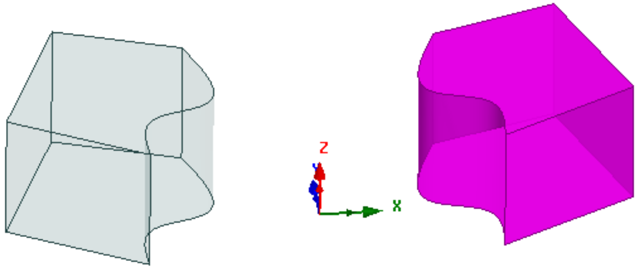
- **Along Line**
  - Duplicates geometry along specified vector

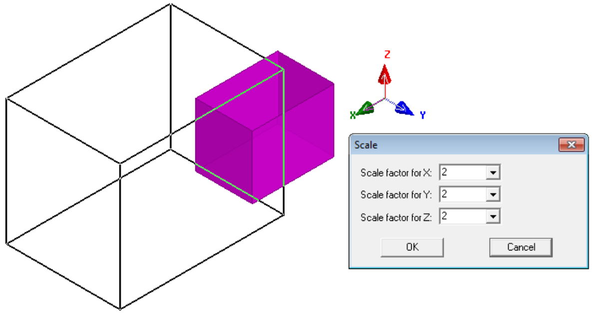


- **Around Axis**
  - Duplicates geometry around selected axis of Work CS

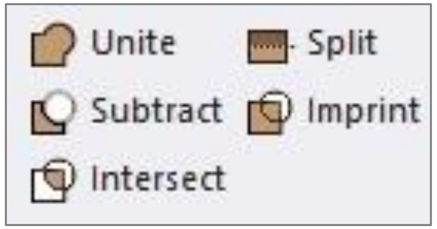


- **Mirror**
  - Duplicates geometry about a plane using Plane Normal

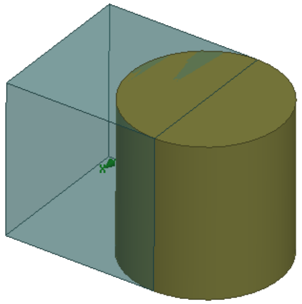




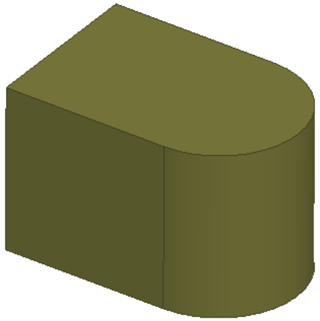
Boolean Operations



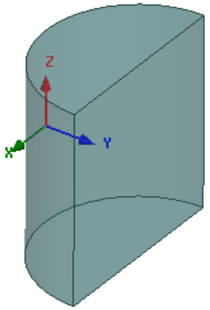
- **Booleans**
- **Boolean operations can be launched from menu item *Modeler → Boolean***



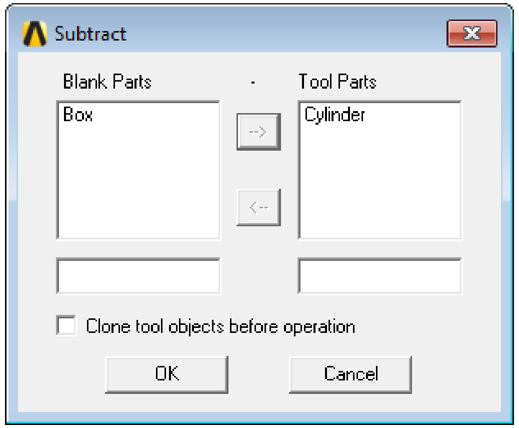
- **Unite**
  - **Unites all selected bodies into single Body**

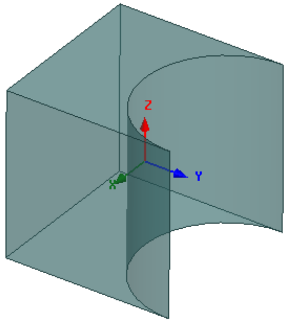


- **Intersect**
  - **Gives Intersection of Selected Bodies**

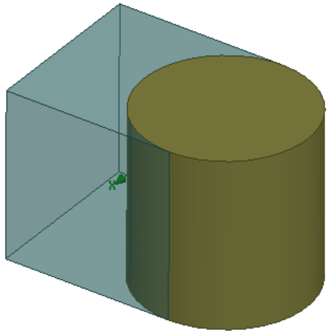


- **Subtract**
  - **Subtracts Tool Bodies from Blank Bodies**





Without Clone Tool Objects

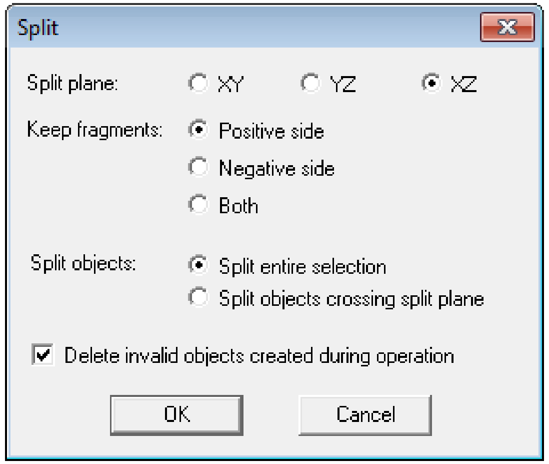


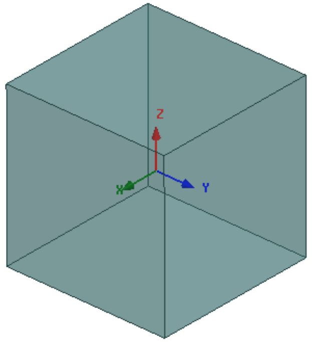
With Clone Tool Objects

Ansys

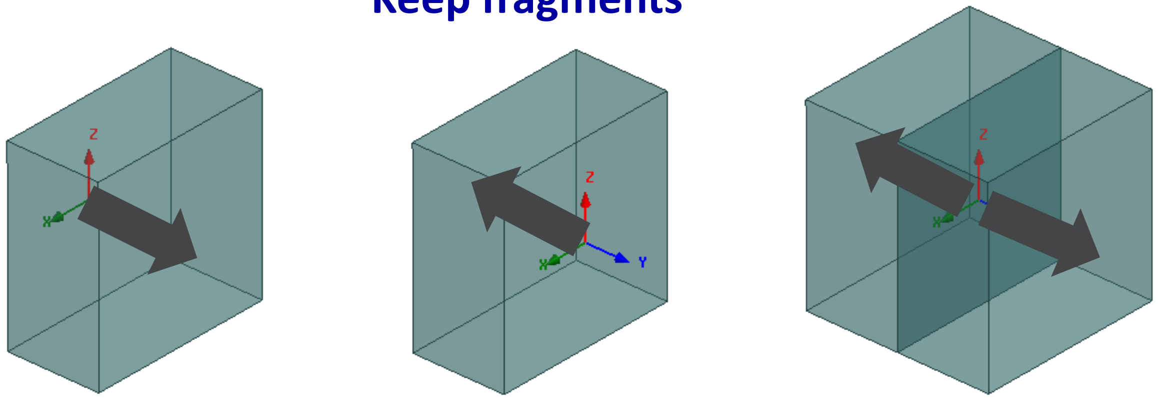
Boolean Operations

- **Split**
  - **Splits the Selected Geometry using XY, YZ or XZ plane of Active CS**

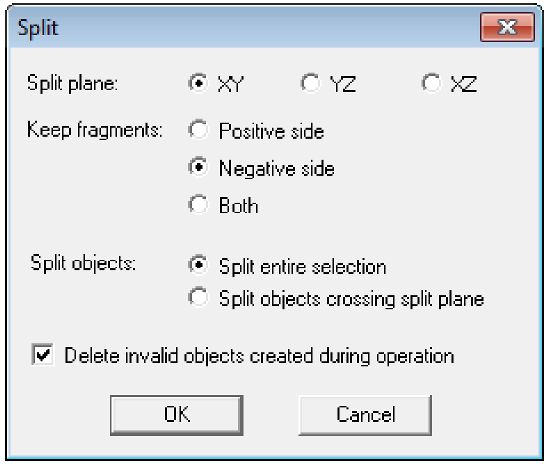


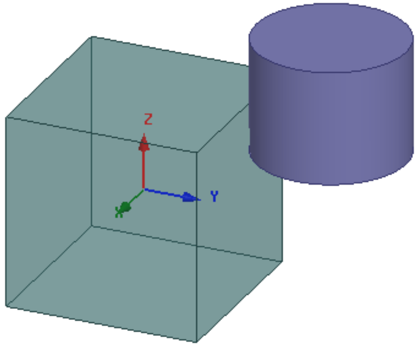


Keep fragments

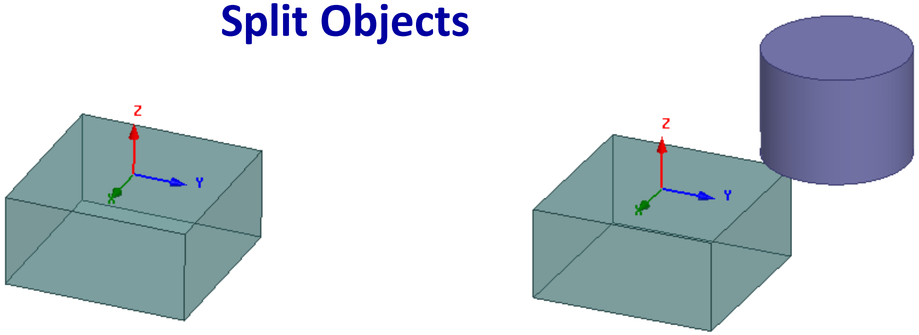


Positive Side **Negative Side** **Both**





Split Objects

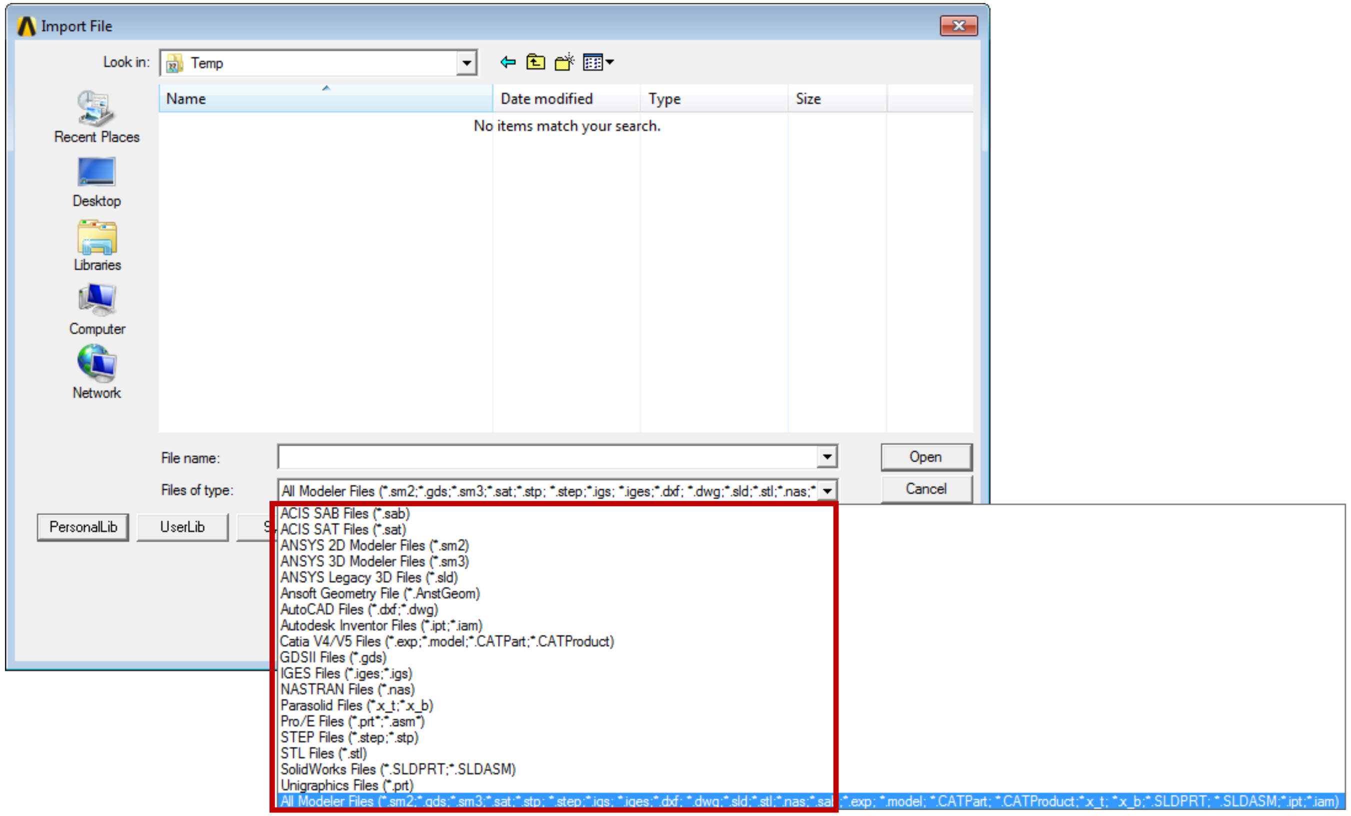


Split entire selection **Split objects crossing split plane**

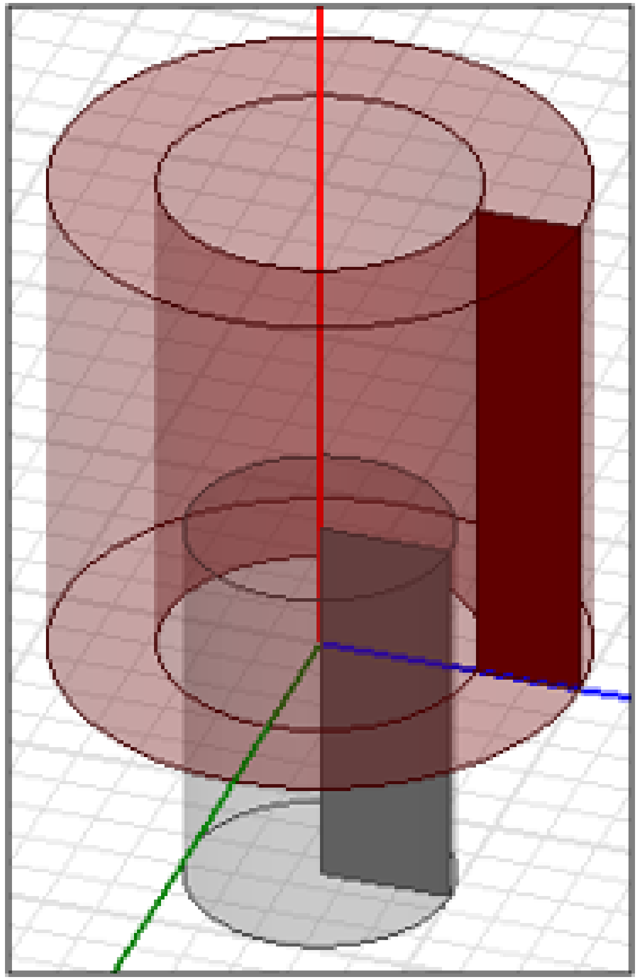
Ansys

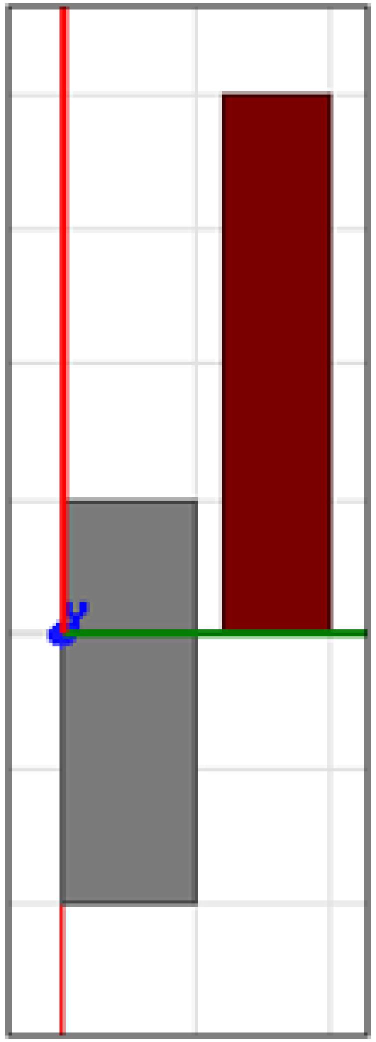
Geometry Import

- **Geometry Import in Maxwell**
  - **Users can import models in Maxwell using neutral formats such as STEP(*.step, *.stp), IGES(*.iges, *.igs), Parasolid (*.x_t, *.x_b). After importing into Maxwell, the files are translated into native ACIS kernel**
  - **Maxwell can also import CAD file formats such as AutoCAD(*.dwg, *.dxf), CATIA(*.model, *.CATPart, *.CATProduct), Creo (*.prt*, *.asm), Unigraphics(*.prt).**

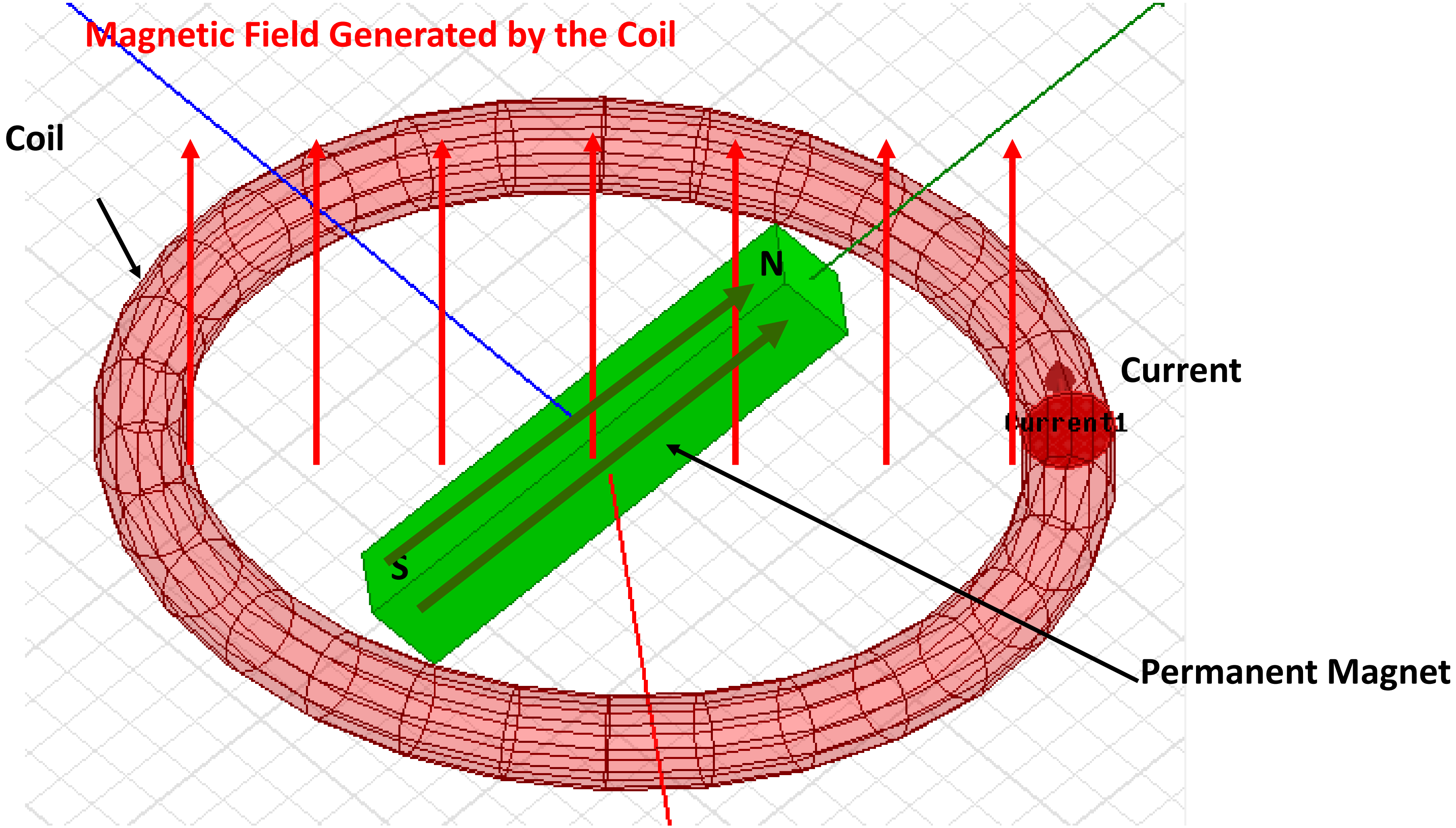


Workshop 1.1 – Magnetostatic 2D analysis





Workshop 1.2 – Magnetostatic 3D analysis



Ansys

End of Presentation