Sanjivani Rural Education Society's

Sanjivani College of Engineering, Kopargaon,

Kopargaon 423 603 [MS]

(An Autonomous Institute Affiliated to SPPU, Pune)

A Report Based On

"Introduction To Ansys Electronics Desktop And Motor-Cad"

1] Designing parametric Rotor in ANSYS Maxwell.

2] Transformer Core Loss Calculation in Maxwell 2D and 3D.

Submitted By

Name: Prem Manoj Mule

PRN No.: UEE22M1044

Roll Number: 42

Class: SY Electrical

Guided By

DR. DIPESH B. PARDESHI

(HOD, DEE)

Introduction to ANSYS Software:

ANSYS is a powerful engineering simulation software suite widely used across industries for **Finite Element Analysis** (FEA), **Computational Fluid Dynamics** (CFD), Electromagnetics, etc. Ansys has been founded in 1970 and incorporated in 1994. Ansys is a leading provider of engineering simulation software used across various industries like aerospace, automotive, and biomedical. Their software helps engineers solve complex problems by simulating how things behave in different situations, from how a car crash to how heat moves through a building.

Ansys is well-known for its Finite Element Analysis (FEA) capabilities, which have become essential for designing and testing everything from bridges to smartphones. With powerful tools for simulating physics like fluid dynamics, heat transfer, and structural mechanics, Ansys provides a comprehensive platform for product development, from initial design concepts to final testing and validation, making it a go-to choose for both academia and industry professionals.

Mechanical Computer-Aided Design (MCAD) Software:
ANSYS SpaceClaim: SpaceClaim is a 3D modeling software that facilitates rapid concept modeling and geometry editing. It enables engineers and designers to create, edit, and repair 3D models quickly and intuitively, making it an ideal tool for pre-processing tasks in simulation workflows. SpaceClaim's direct modeling approach allows users to manipulate geometry without the constraints of history-based parametric modeling.

Electronic Computer-Aided Design (ECAD) Software:
ANSYS Electronics Desktop: This software suite provides a comprehensive set of tools for simulating and analyzing electronic systems and components. Electronics Desktop enables engineers to optimize the performance and reliability of electronic designs across a wide range of industries, including consumer electronics, telecommunications, and automotive.

Structural Analysis Software: ANSYS offers several tools for structural analysis, including ANSYS Mechanical, ANSYS Structural. These tools enable engineers to predict the behavior of structures under various loading conditions, including static, dynamic, and nonlinear analyses. They are used extensively in industries such as aerospace, automotive, civil engineering, and manufacturing.

Fluid Dynamics Software: ANSYS Fluent and ANSYS CFX are leading tools for computational fluid dynamics (CFD) simulations. They allow engineers to model and analyze fluid flow, heat transfer, and other related phenomena in diverse applications such as aerospace, automotive, energy, and chemical processing industries.

Electromagnetics Software: ANSYS Maxwell and ANSYS HFSS are prominent tools for electromagnetic simulation. Engineers use these tools to analyze electromagnetic fields, antenna designs, RF/microwave devices, motors, transformers, and more. ANSYS Electromagnetics Suite provides a comprehensive set of capabilities for electromagnetic analysis.

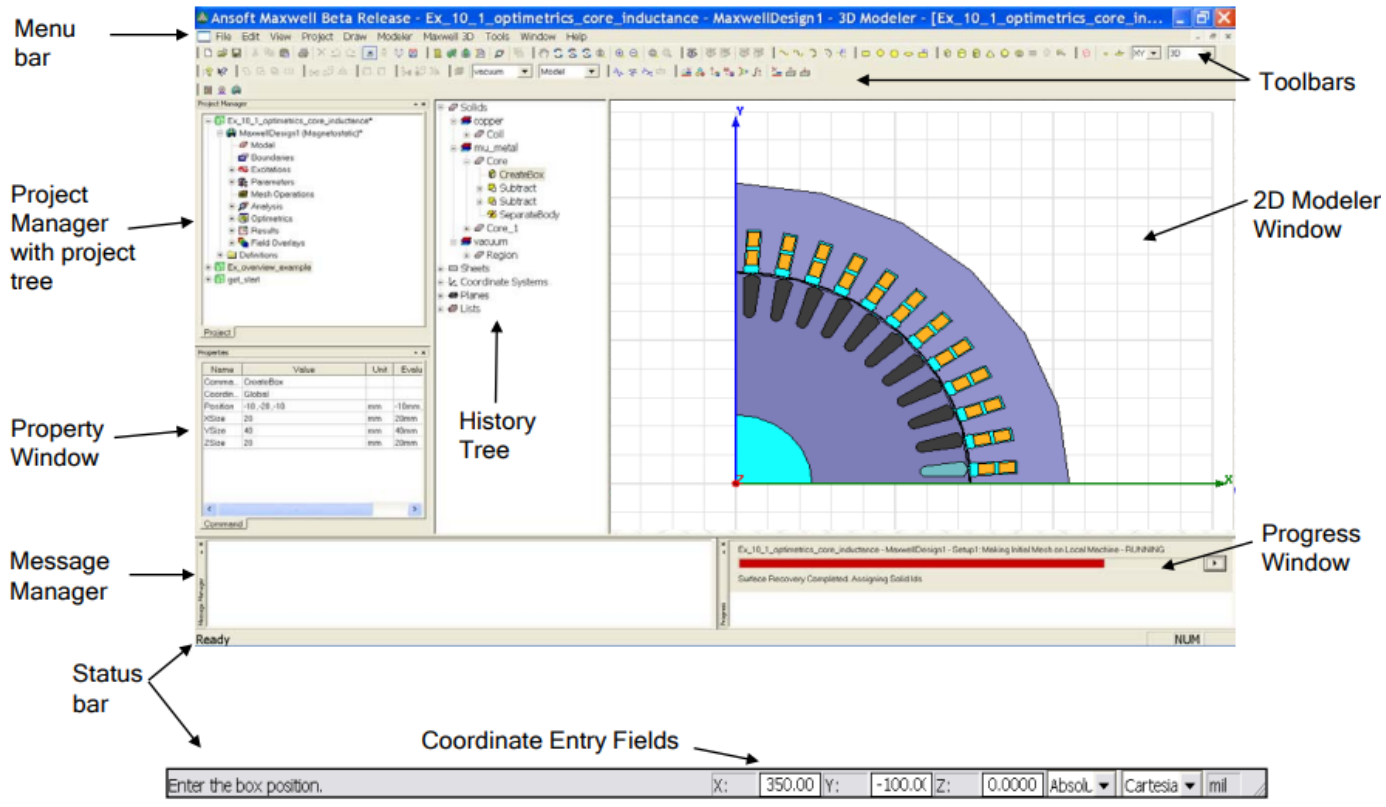
Multiphysics Software: ANSYS Multiphysics solutions enable engineers to simulate interactions between different physical phenomena, such as fluid-structure interaction (FSI), thermal-electrical coupling, and fluid-thermal coupling. These tools allow for more realistic simulations by considering the effects of multiple physics simultaneously.

Explicit Dynamics Software: ANSYS Explicit Dynamics, including ANSYS Autodyn, specializes in simulating highly transient, dynamic events such as impact, crash, and explosion simulations. These tools are used in automotive safety analysis, aerospace, defense, and other industries where dynamic events are critical to design considerations.

Embedded Software Development Tools: ANSYS SCADE Suite offers a comprehensive solution for model-based development of embedded software systems, particularly in safety-critical industries such as aerospace, automotive, and railway. It enables engineers to design, simulate, and verify complex control systems and embedded software.

System Simulation Software: ANSYS Simplorer is a powerful tool for system simulation and modeling of multi-domain systems. It allows engineers to model and simulate complex systems consisting of electrical, mechanical, hydraulic, and thermal components, facilitating the analysis of system-level performance and behavior.

The Maxwell Desktop:



Introduction to Maxwell Desktop

Maxwell is an interactive software package that uses Finite Element Analysis (FEA) to simulate (solve) electromagnetic field problems. Maxwell integrates with other Ansys software packages to perform complex tasks while remaining simple to use. Maxwell incorporates both a set of 2D solvers and 3D solvers in an integrated user interface. This guide will focus on 3D capabilities. 2D problems examples are cover in a separate 2D Getting Started Guide. The following six types of stand-alone solutions are supported by Maxwell 3D. Magnetostatic linear and nonlinear 3D fields caused by a user-specified distribution of DC current density and permanent or externally applied magnetic fields. Materials can be non-linear and anisotropic. Additional quantities that can be computed include torque, force, and self and mutual inductances.

Harmonic (sinusoidal variation in time) steady-state magnetic fields with pulsation induced eddy currents in massive solid conductors caused by A user-specified distribution of AC currents (all with the same frequency but with possibly different initial phase angles). Externally applied magnetic fields. This solution includes displacement currents for calculating near field electromagnetic wave radiation. Transient (time domain) magnetic fields caused by permanent magnets, conductors, and windings supplied by voltage and current sources with arbitrary variation as functions of time.

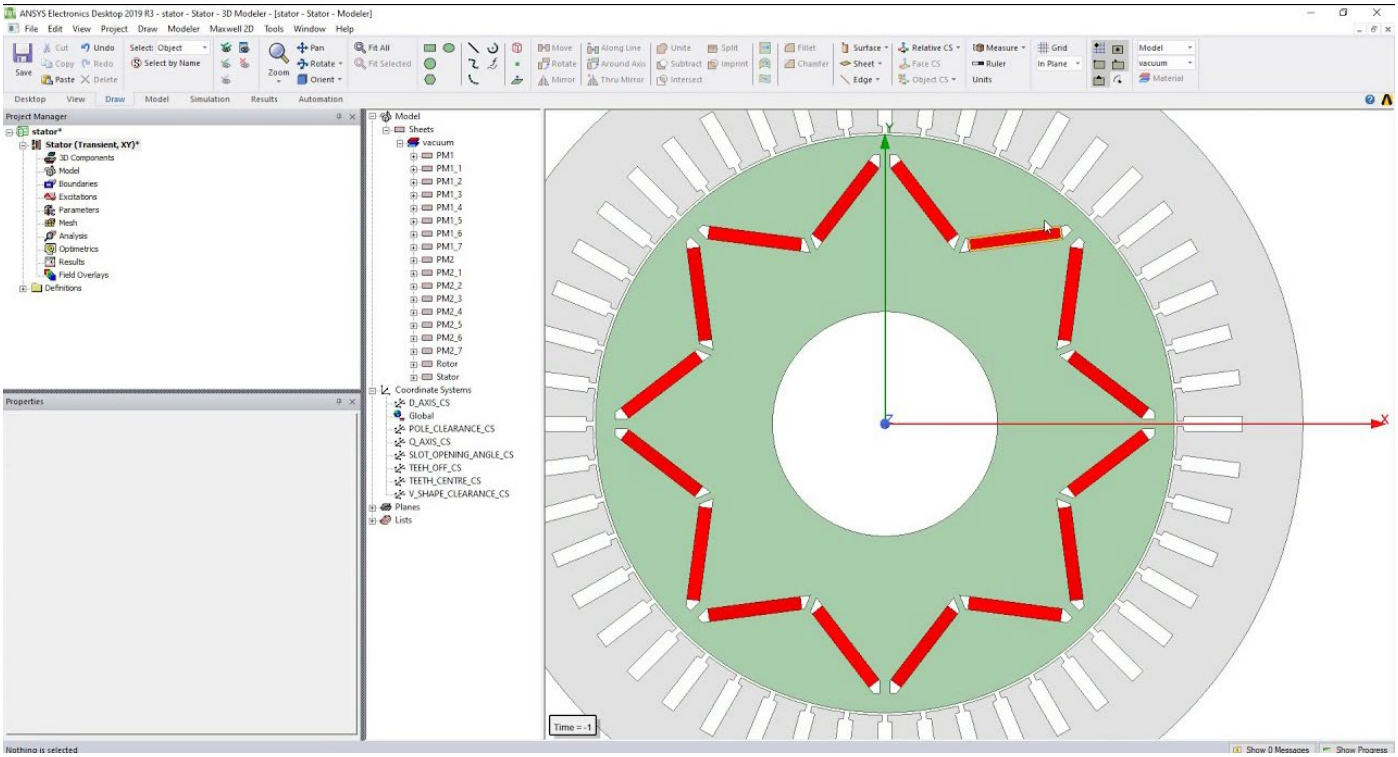
Rotational or translational motion effects can be included in the simulation. Electrostatic 3D fields caused by a user-specified distribution of voltages and charges in non-conducting regions. Additional quantities that can be computed include torque, force, and capacitances. Electric DC Conduction 3D fields in conductors characterized by a spatial distribution of voltage, electric field, and current density. Power loss can also be computed. In addition, optional simulation of fields in insulating materials is supported. Transient (time domain) 3D Electric fields caused by time dependent voltage, current and charge distributions. All sources are arbitrary functions of time. In addition, Maxwell may be coupled with other simulators to provide a greater range of solution capability.

General Procedure for Setting Up Maxwell Designs

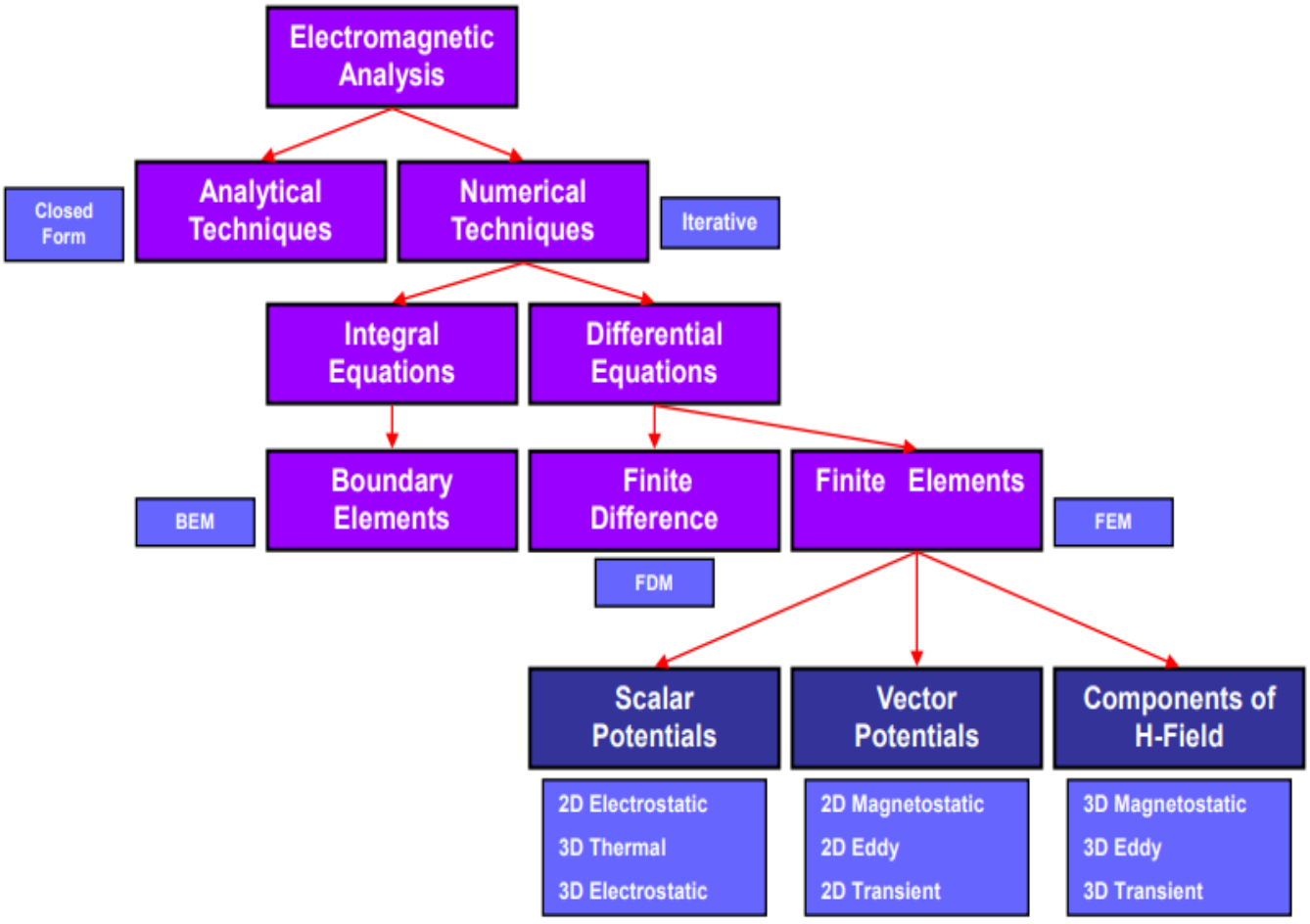
You are not required to follow a specific order when setting up your Maxwell design. However, the following order is recommended, particularly for new users:

1) Open Ansys Electronics Desktop by double-clicking the desktop icon or by clicking **Start** > **Programs** > **Ansys EM Suite** [version] > **Ansys Electronics Desktop** [version] from the Windows taskbar.
2) Add a Maxwell 3D design and save the new project.
3) Draw the geometry of the model.
4) Optionally, modify the model's design parameters.
5) Assign variables to design parameters.
6) Assign excitations and boundary conditions.
7) Specify solution settings.
8) Run a Maxwell simulation.
9) Create post-processing plots.
10) Create a parametric analysis.
11) Create a field animation of the parametric analysis results.

1] Designing parametric Rotor in ANSYS Maxwell

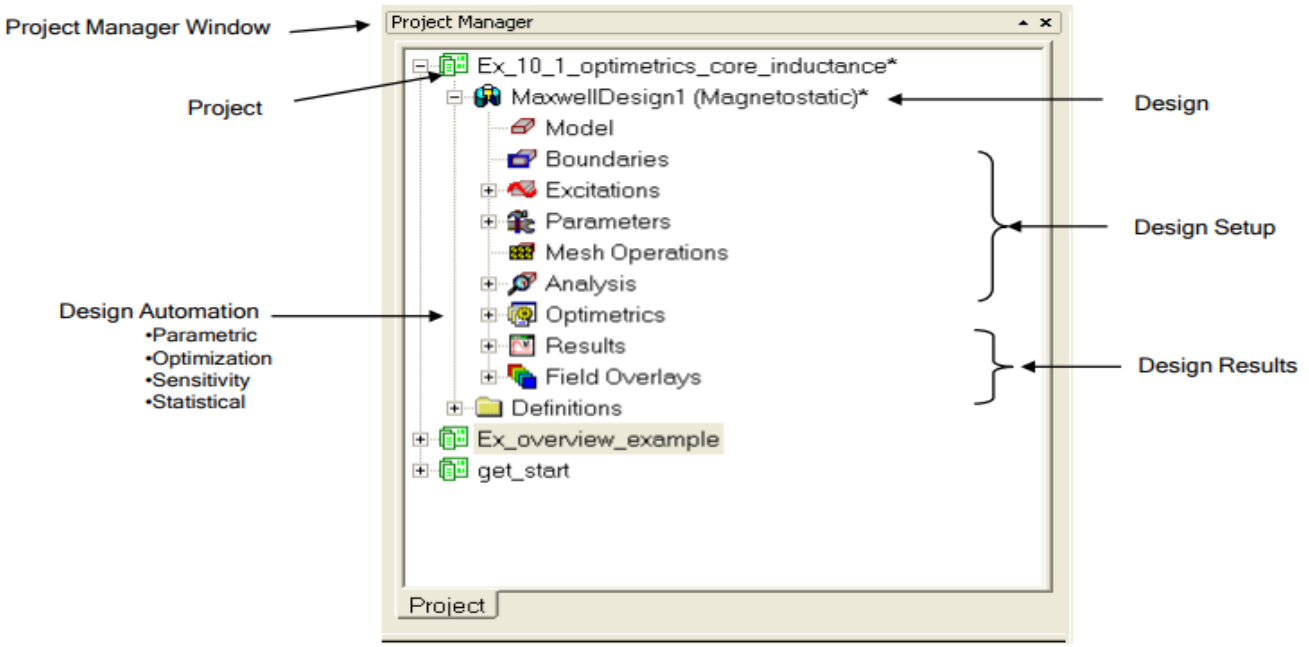


Different Methods of Electromagnetic Analysis

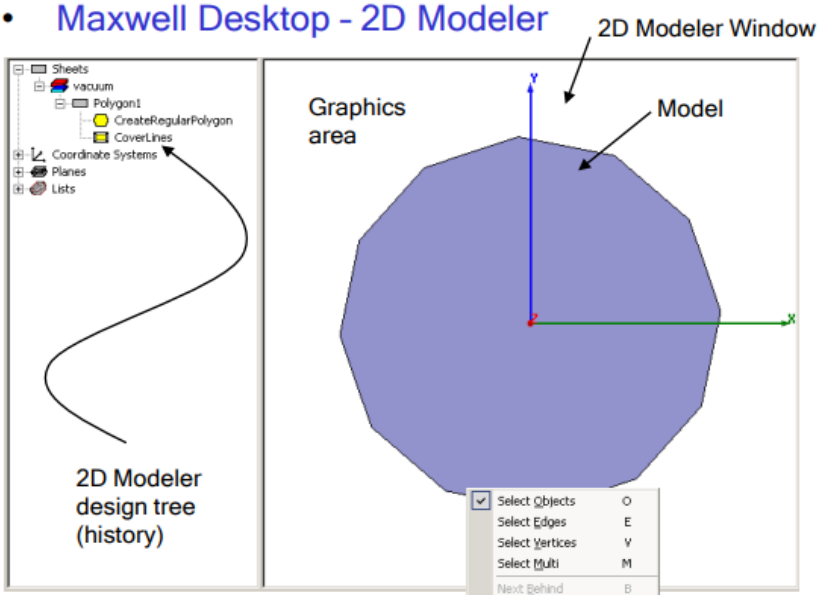


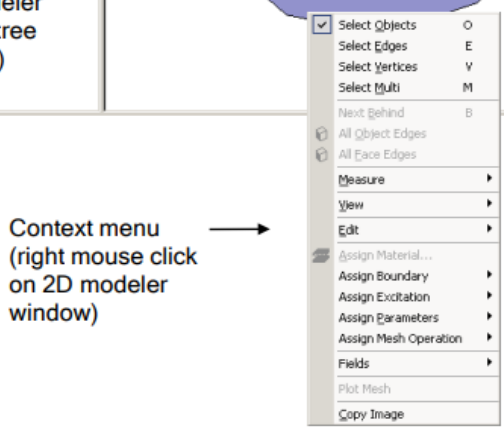
Maxwell Desktop - Project Manager

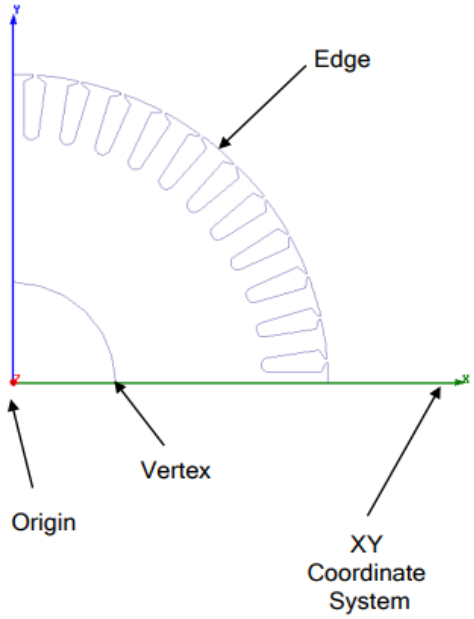
- Multiple Designs per Project
- Multiple Projects per Desktop
- Integrated Optimetrics Setup (requires license for analysis)



- Maxwell Desktop - 2D Modeler







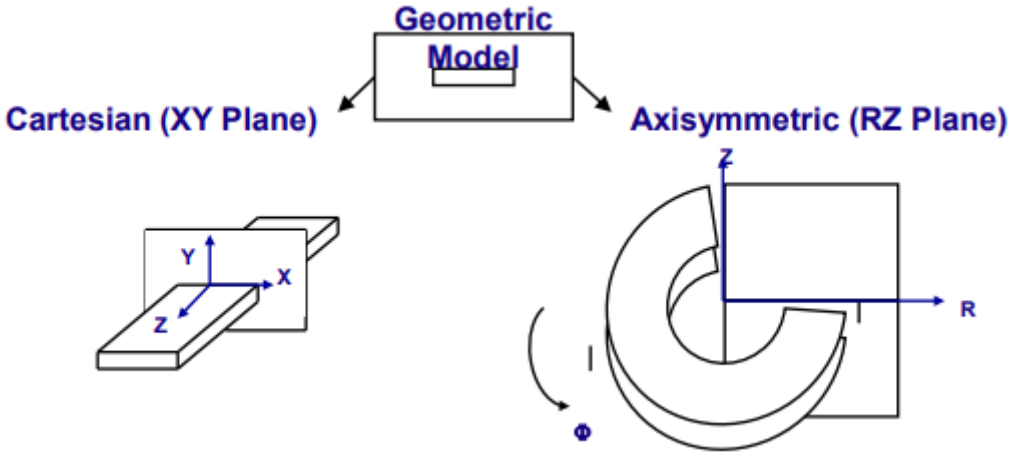
Geometry Mode

To set the geometry mode:

- Select the menu item **Maxwell 2D > Solution Type**
- Solution Type Window: Choose Geometry Mode: **Cartesian XY**

Maxwell – Geometry Modes

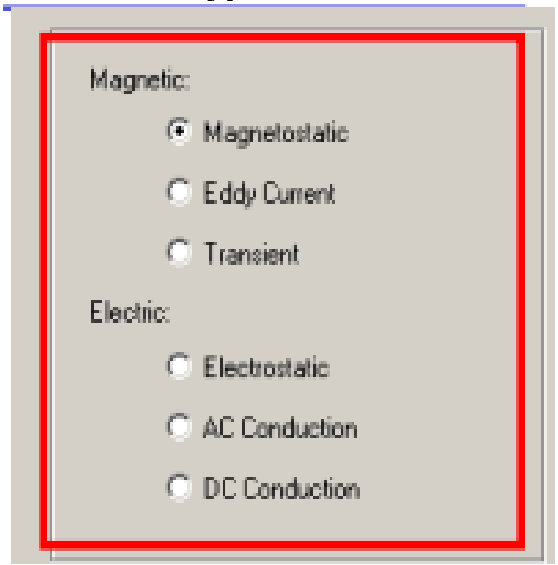
- A **Cartesian (XY)** model represents a cross-section of a device that extends in the z-direction. Visualize the geometric model as extending perpendicular to the plane being modeled.
- An **Axis Symmetric (RZ)** model represents a cross-section of a device that is revolved 360° around an axis of symmetry (the z-axis). Visualize the geometric model as being revolved around the z-axis.



Set Solution Type

To set the solution type:

select the menu item **Maxwell 2D > Solution Type.**



Set Model Units

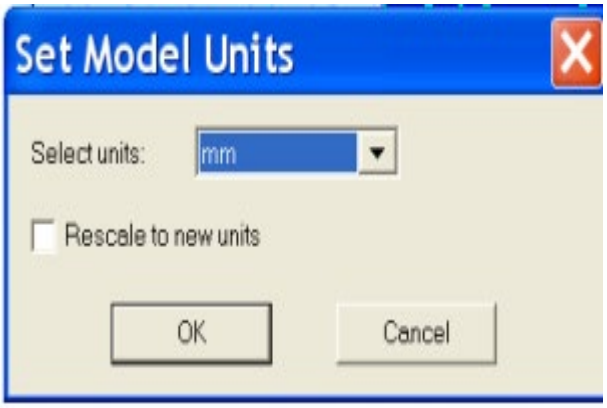
To set the units:

Select the menu item Modeler > Units

Set Model Units:

Select Units: mm

Click the OK button

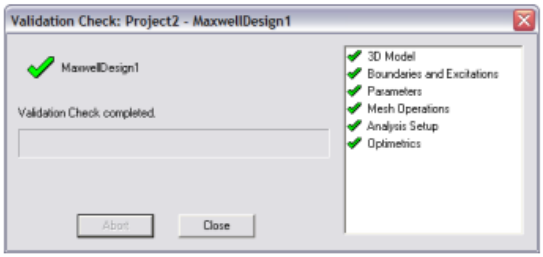


Save Project

- **To save the project:**
  1. In a Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **2D_simple_example**
  3. Click the **Save** button

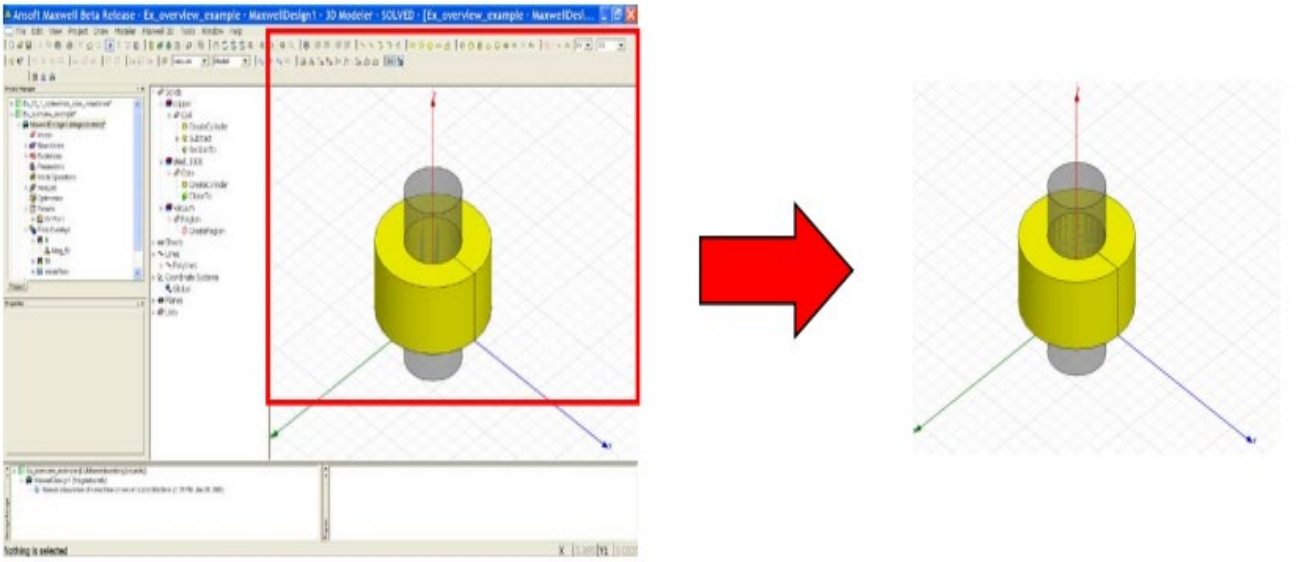
Model Validation

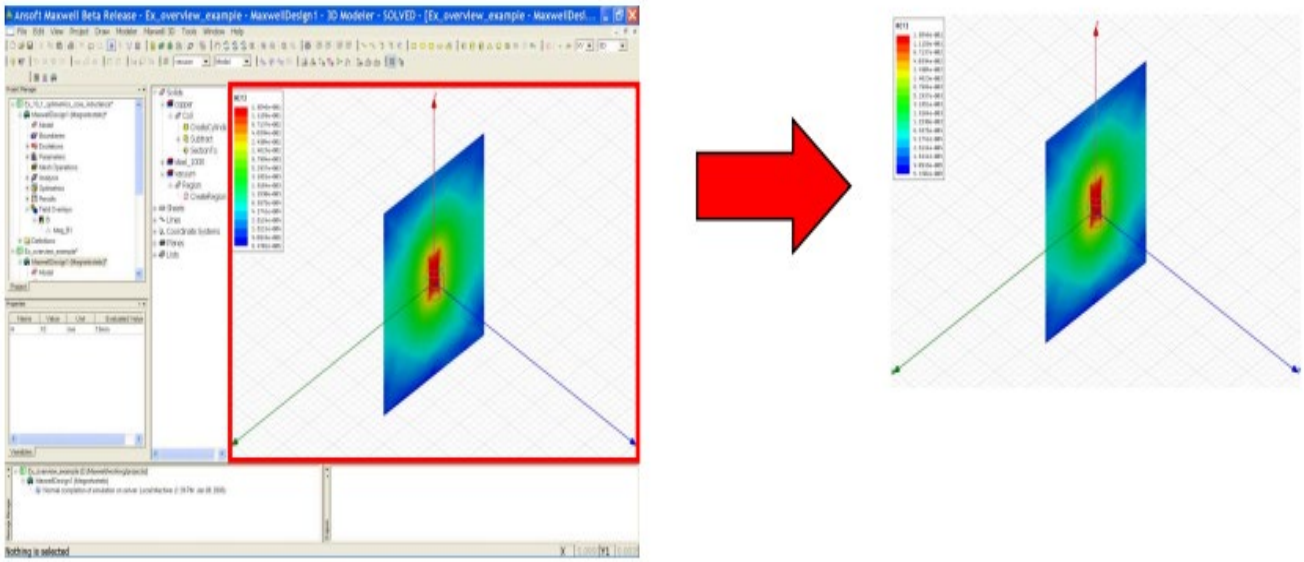
- **To validate the model:**
  1. Select the menu item ***Maxwell 3D> Validation Check***
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

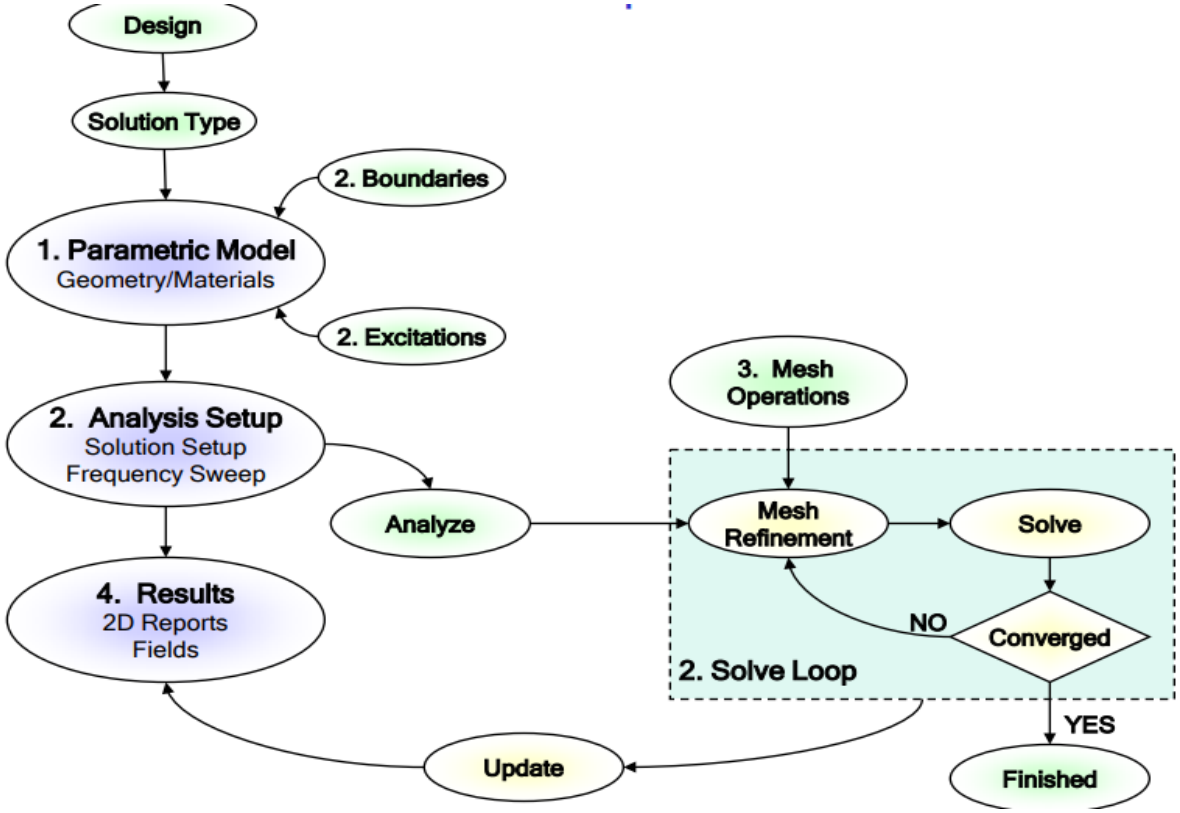


Analyze

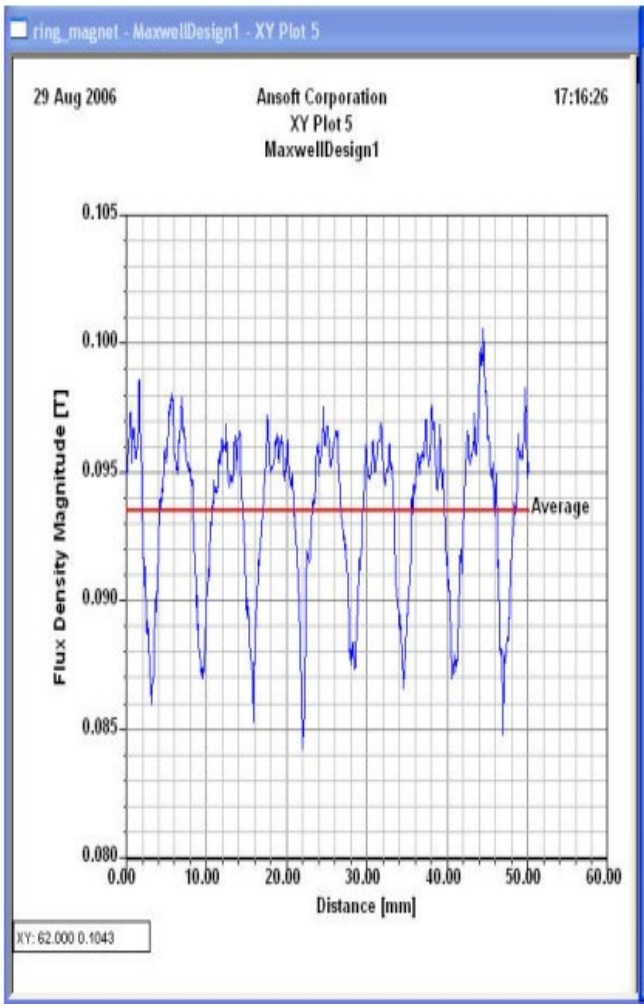
- **To start the solution process:**
  1. Select the menu item ***Maxwell 2D> Analyze All***

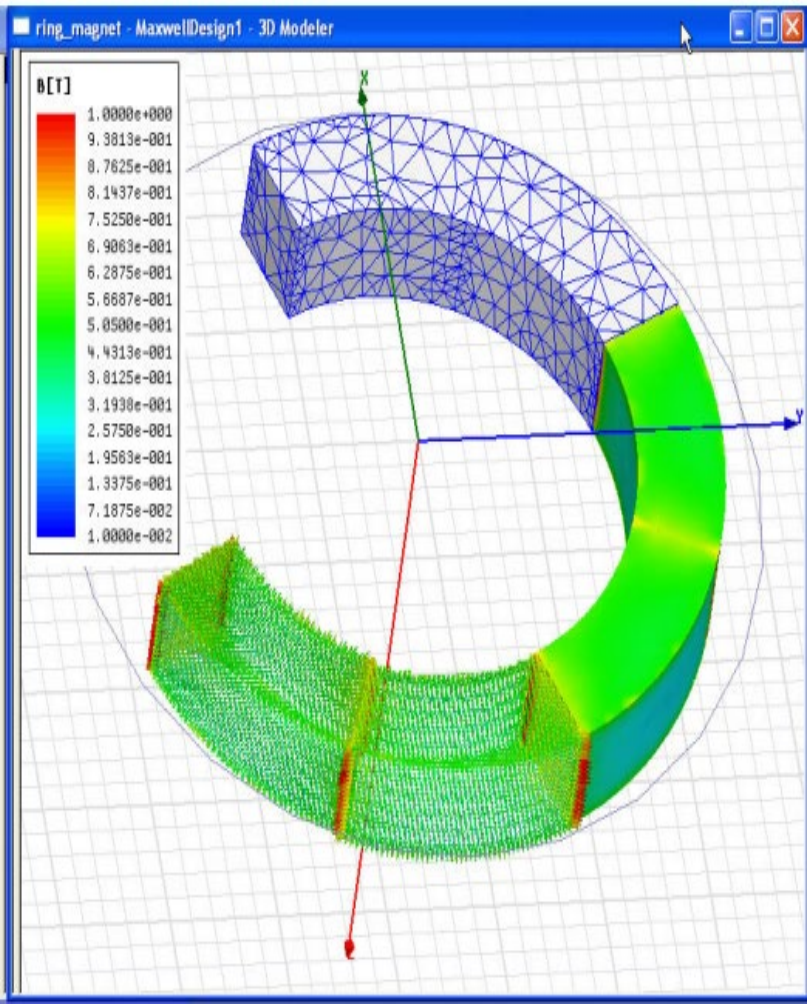


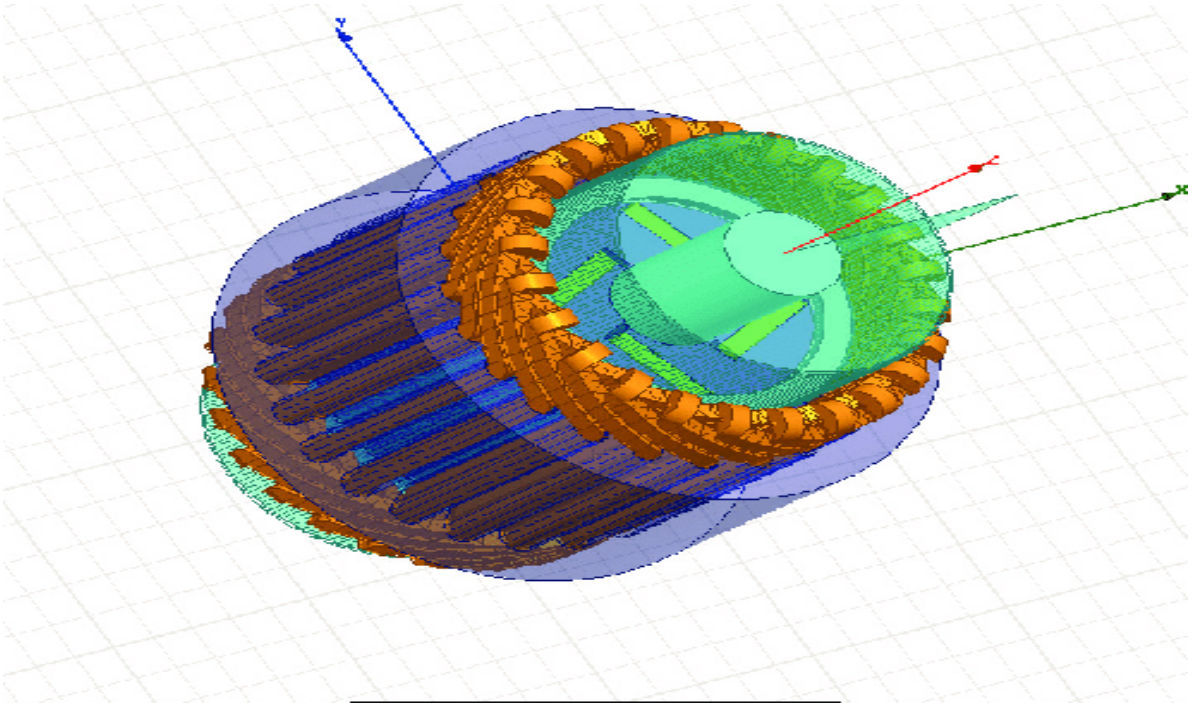



Design
Solution Type
2. Boundaries
1. Parametric Model
Geometry/Materials
2. Excitations
2. Analysis Setup
Solution Setup
Frequency Sweep
Analyze
3. Mesh Operations
Mesh Refinement
Solve
NO
Converged
2. Solve Loop
YES
4. Results
2D Reports
Fields
Update
Finished

Model:


ring_magnet - MaxwellDesign1 - XY Plot 5
29 Aug 2006
Ansoft Corporation
XY Plot 5
MaxwellDesign1
17:16:26
Flux Density Magnitude [T]
Distance [mm]
Average
0.105
0.100
0.095
0.090
0.085
0.080
0.00
10.00
20.00
30.00
40.00
50.00
60.00
XY: 62.000 0.1043


ring_magnet - MaxwellDesign1 - 3D Modeler
B[T]
1.0000e+000
9.3813e-001
8.7625e-001
8.1437e-001
7.5250e-001
6.9063e-001
6.2875e-001
5.6687e-001
5.0500e-001
4.4313e-001
3.8125e-001
3.1938e-001
2.5750e-001
1.9563e-001
1.3375e-001
7.1875e-002
1.0000e-002



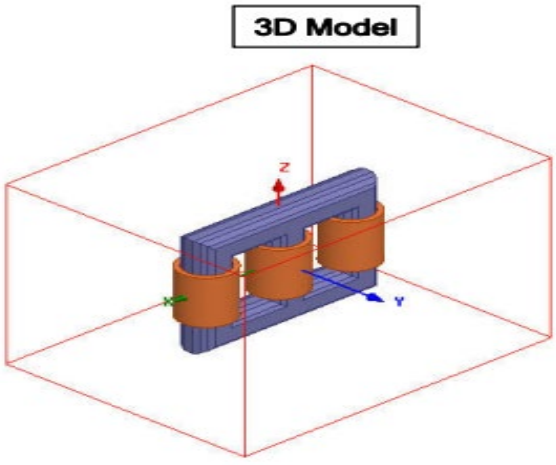
2] **Example (2D/3D Transient) – Core Loss.**

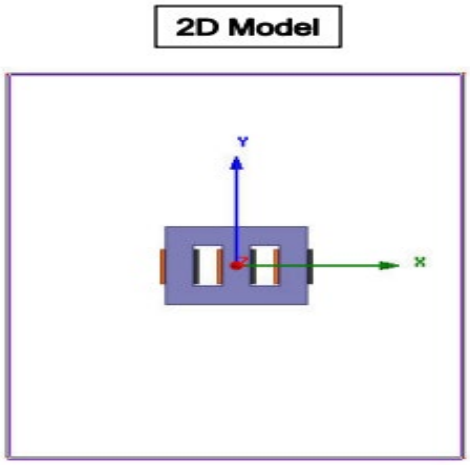
Transformer Core Loss Calculation in Maxwell 2D and 3D.

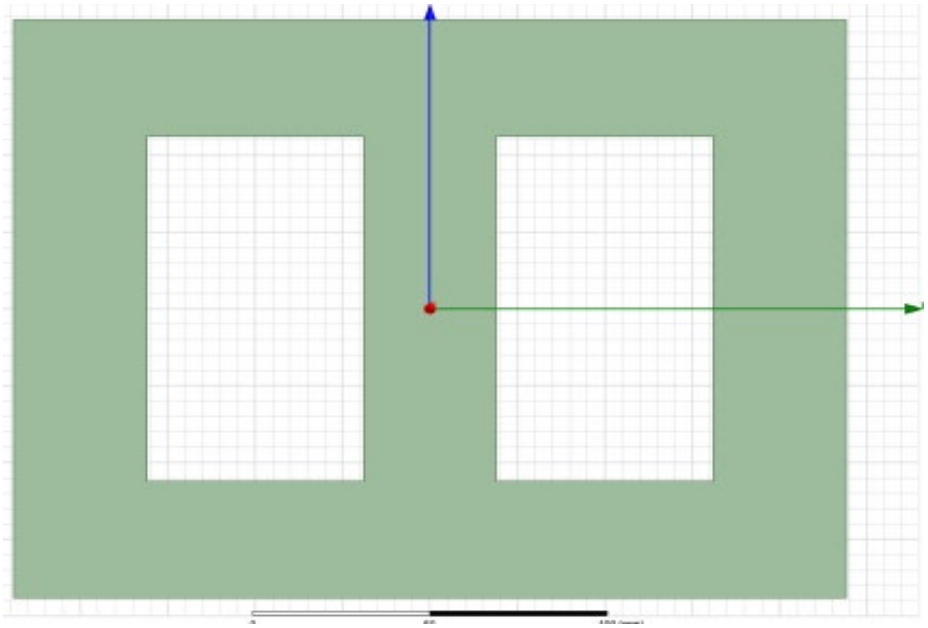
This example analyzes cores losses for a 3ph power transformer having a laminated steel core using Maxwell 2D and 3D. The transformer is rated 115-13.8kV, 60Hz and 30MVA. The tested power losses are 23,710W. It is important to realize that a finite element model cannot consider all of the physical and manufacturing core loss effects in a laminated core.

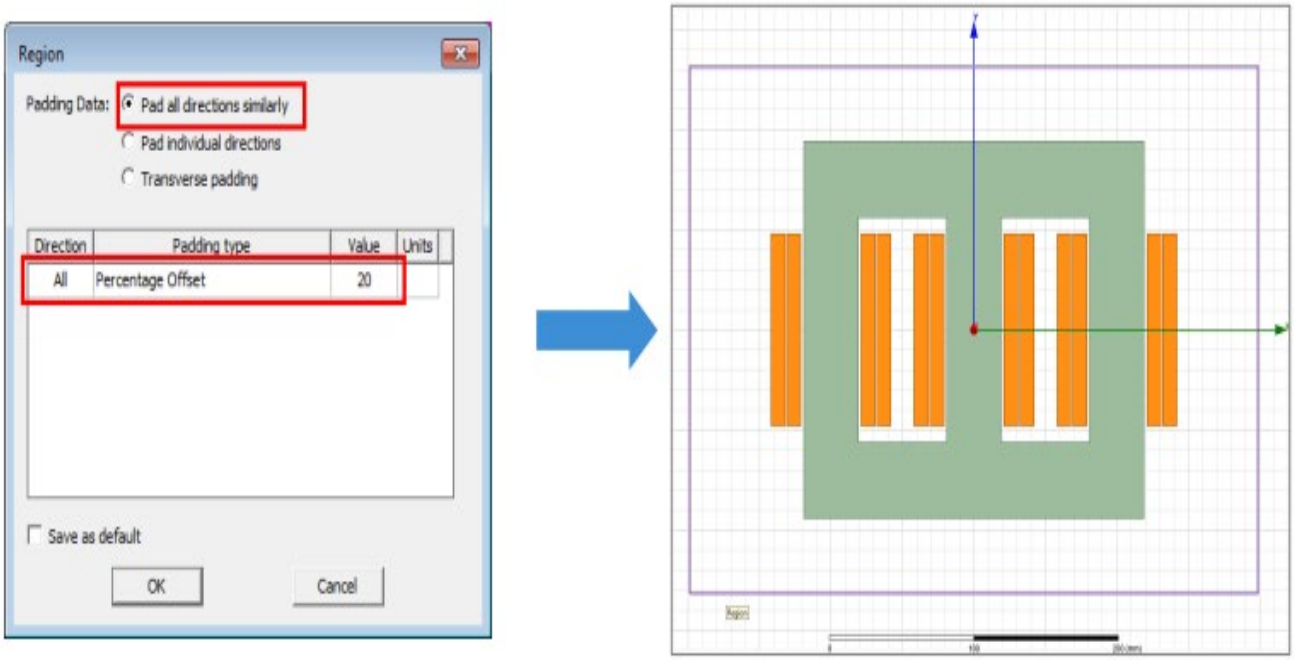
This example will go through all steps to create the 2D and 3D models based on a customer supplied base model. For core losses, only a single magnetizing winding needs to be considered. Core material will be characterized for nonlinear BH and core loss characteristics. An exponentially increasing voltage source will be applied in order to eliminate inrush currents and the need for an unreasonably long simulation time. Finally, the core loss will be averaged over time and the core flux density will be viewed in an animated plot.

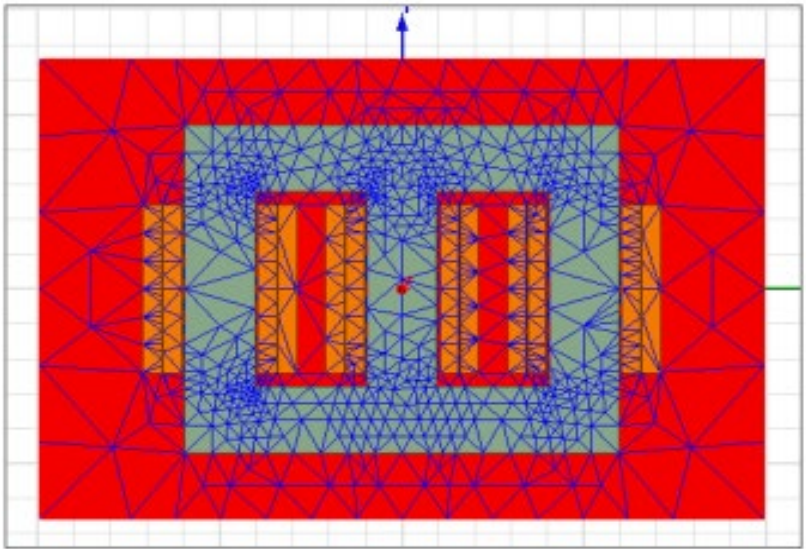
This example will be solved in two parts using the 2D Transient and 3D Transient solvers. The model consists of a magnetic core and low voltage winding on each core leg.





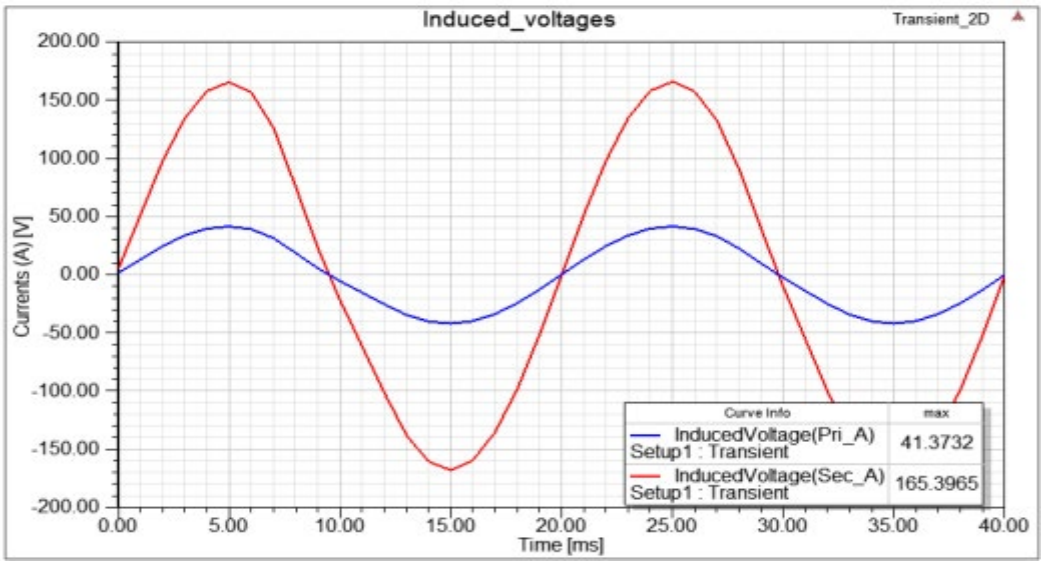



Region
Padding Data:
Pad all directions similarly
Pad individual directions
Transverse padding
Direction
Padding type
Value
Units
All
Percentage Offset
20
Save as default
OK
Cancel

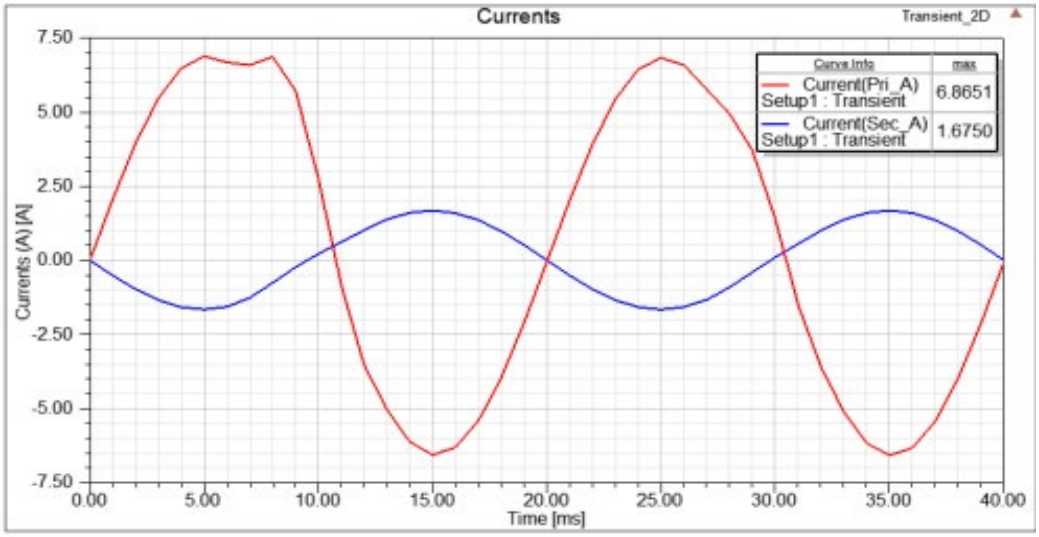


fig, Mesh

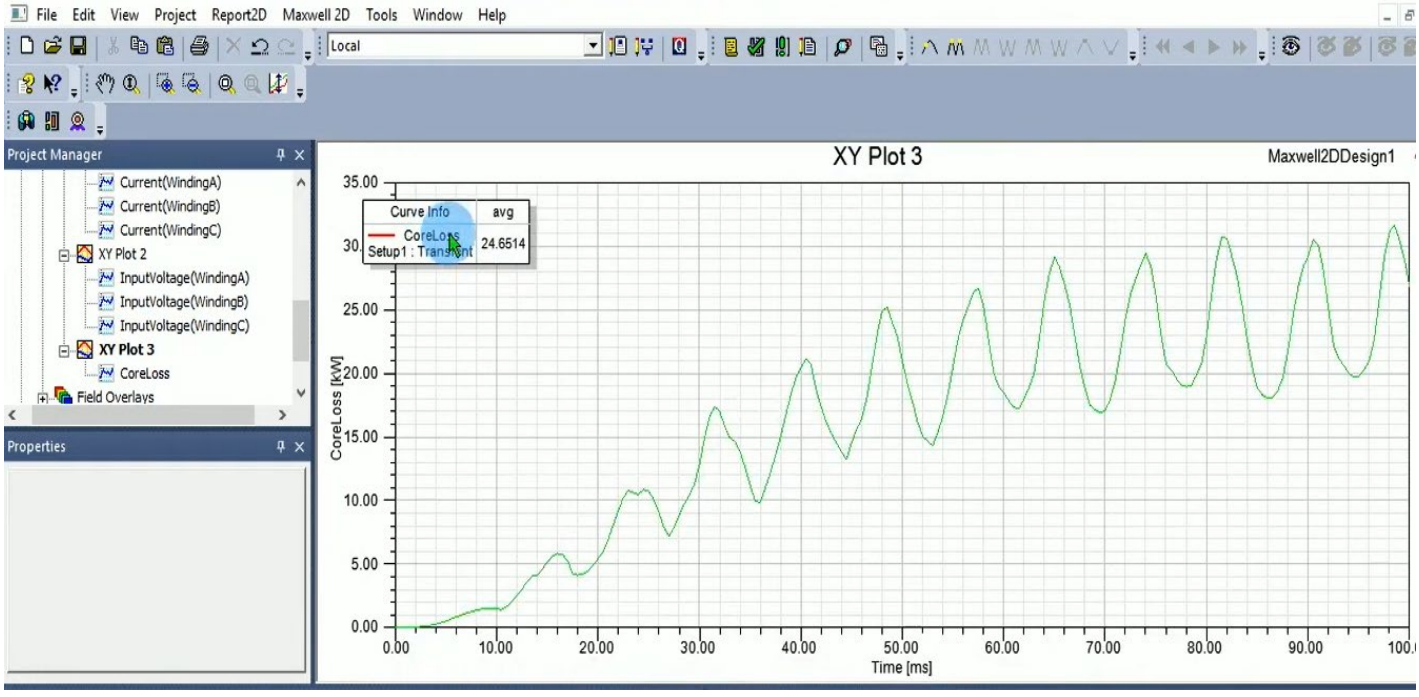
Plot the values of Primary and Secondary Induced Voltages vs Time:



Plot the values of Primary and Secondary Current vs Time:

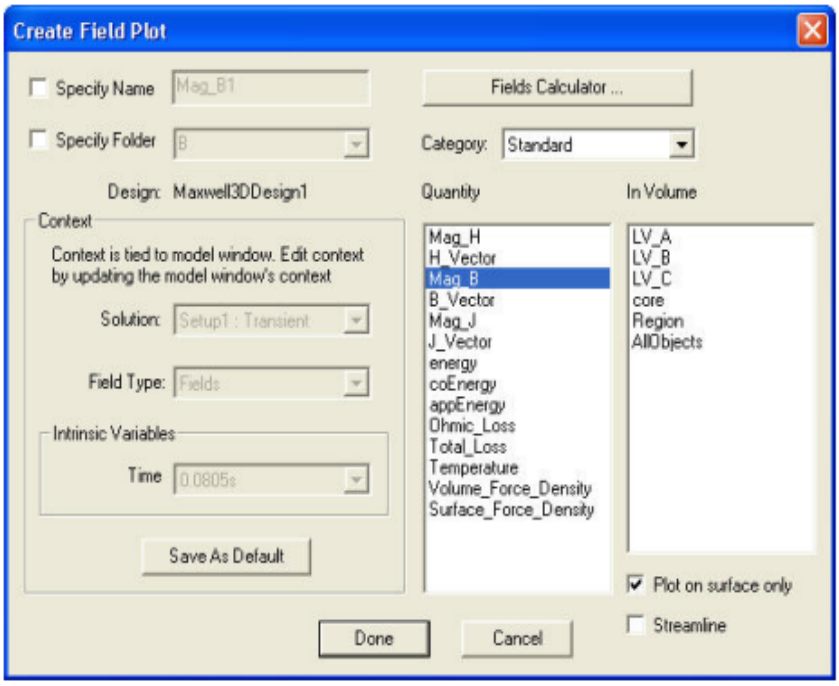


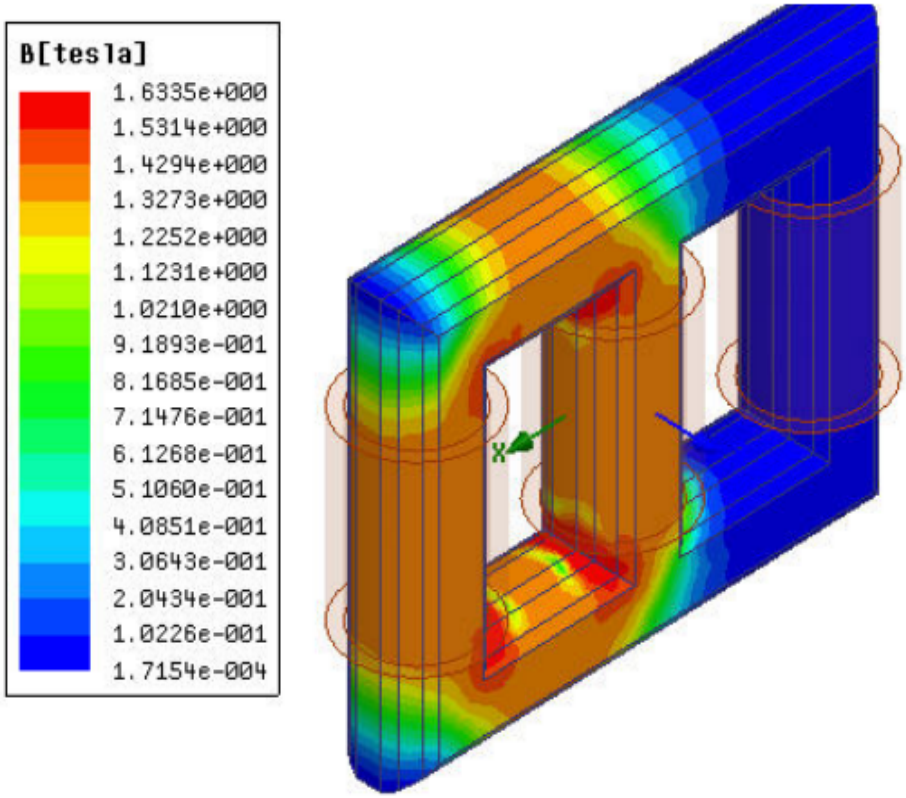
Average Core Loss:

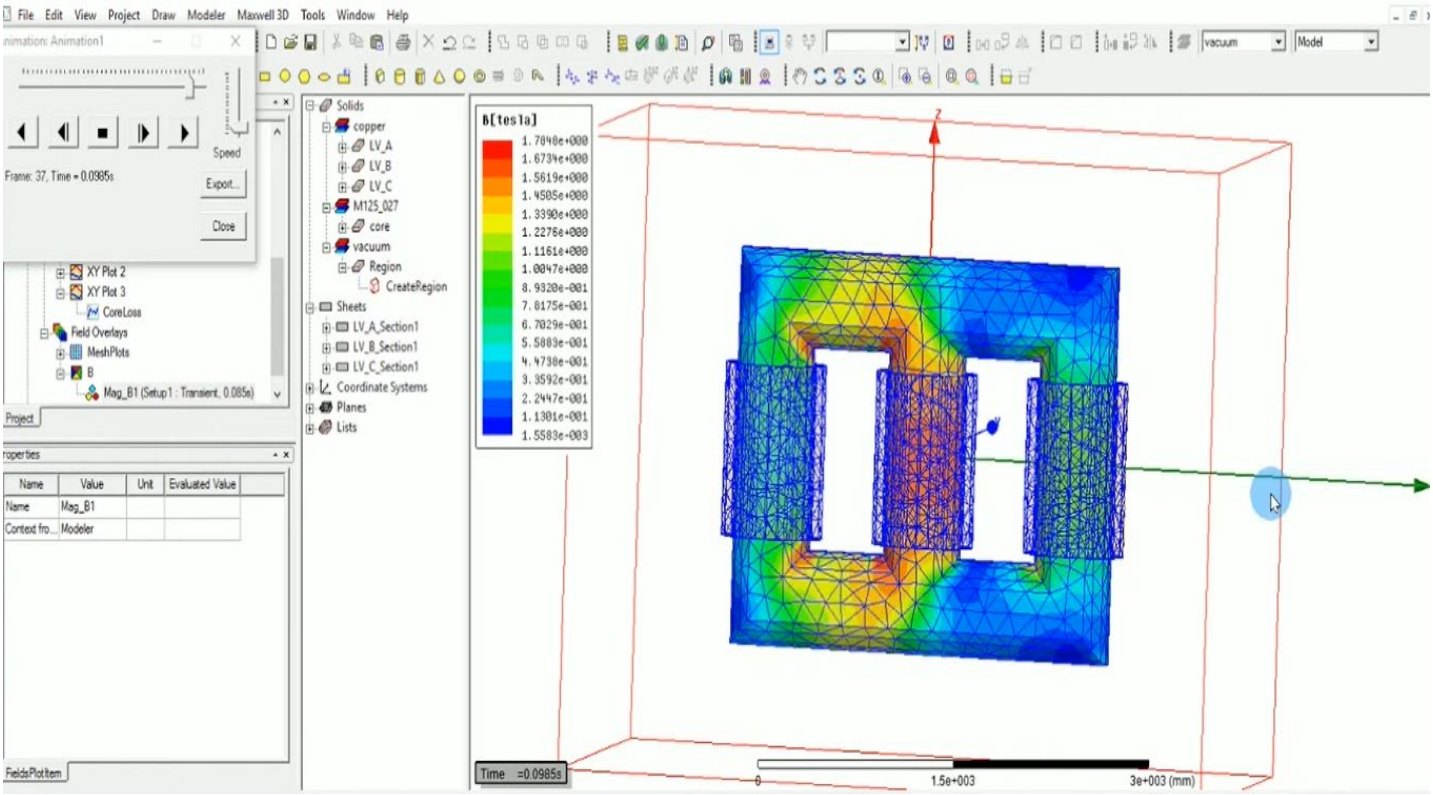


Create Flux Density Plot

- **To Plot Flux Density on Core**
  - Double click on **Maxwell3DDesign1** in Project Manager window to exit Plot view
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    - Plot on surface Only: ☑ **Checked**
    - Press **Done**







fig, Thermal Analysis of 3 phase Transformer

Reference:

- Ansys Learning Library.
- Ansys Learning Hub.
- Ansys Education Resources.

Certificate:

Conclusion:

The report on "Introduction to ANSYS Electronics Desktop and Motor-CAD" offers a thorough examination of utilizing ANSYS Maxwell for designing and analyzing electromagnetic systems. By delving into the creation of parametric rotors and the computation of transformer core losses in both 2D and 3D environments, it demonstrates a comprehensive approach to electrical system analysis.

Through adept utilization of ANSYS Maxwell, the report adeptly navigates through the complexities of rotor design, core loss calculation, and thermal considerations, providing valuable insights into electromagnetic system optimization.

Overall, it highlights the importance of employing advanced simulation tools like ANSYS Maxwell in electrical engineering, showcasing their potential to enhance system efficiency and performance while also setting the stage for future innovations in the field.