ANSYS Maxwell Magnetic Field Formulation

There are various variational electromagnetic field formulations using FEA to numerically solve Maxwell's equations. When choosing the right formulation to be implemented in FEA special mathematical handling is required in order to avoid unphysical solutions and to provide numerical stability and computational efficiency. This application brief describes the basis for formulation employed in ANSYS Maxwell.

ANSYS Maxwell magnetic field formulation is founded on Maxwell's equations starting with the basic field equations:

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \qquad \text{(Faraday's law)} \qquad (1)$$

$$\nabla \cdot \mathbf{B} = 0 \qquad \text{(Gauss's law)} \qquad (2)$$

$$\nabla \times \mathbf{H} = \mathbf{J} \qquad \text{(Ampere's law)} \qquad (3)$$

in which **E** is the electric field strength, **B** is the magnetic flux density, **H** is the magnetic field strength, and **J** is the electric current density. Obviously, these equations are considered together with the constitutive material equations for both electric fields as **E**=f (**J**) and magnetic field as **B**=f (**H**).

Numerical solution of such equations is based on **T**-Ω formulation in which Ω is nodal-based magnetic scalar potential, defined in the entire solution domain, and **T** is edge-based electrical vector potential, defined only in the conducting eddy-current region (Figure 1).

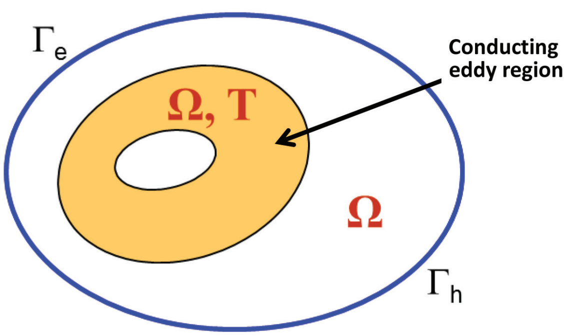

Figure 1. Domain representation for T-Ω formulation

There are several advantages of this formulation:

- Avoid unphysical solution due to utilization of edge elements to model a source component and induced eddy current
- Computationally efficient because in the nonconducting region, only scalar potential is employed
- Numerical stability because no gauging is required to obtain unique solutions

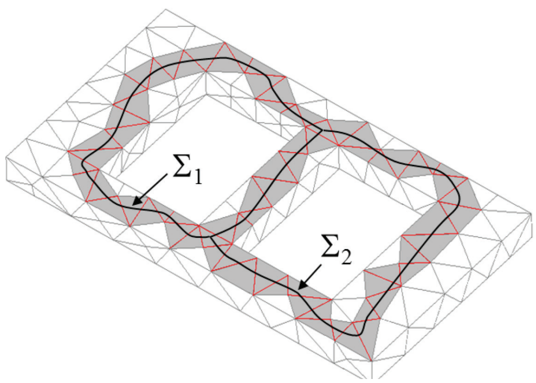

Figure 2. Identification of multiple connected region and creation of cutting domain are all done automatically based on tree/cotree technology.

In **T**-Ω formulation, the key to allow the use of scalar potential is that the solution domain has to be single connected. In the eddy-conducting region, field **H** is described by both electrical vector potential **T** and magnetic scalar potential Ω, and **curl** of the electrical vector potential **T** is the induced eddy-current density. In the source conductor region, field **H** is described by both applied source field **Hp** and magnetic scalar potential Ω, in which **curl** of **Hp** is the source current density **J**. In the non-source, non-eddy-conductor region, since $\nabla \times \mathbf{H} = 0$ and **curl** of any gradient is always zero, then **H** can be represented by the gradient of the magnetic scalar potential as long as the domain is single connected.

To make the domain single connected, you need to introduce a cut so that Ampere's law can hold with respect to **T** in the cut region. This means in the nonconducting cut region, even though there is no current, field **H** is also described by both Ω and **T**, not just by Ω alone.

Therefore, for the **T**-Ω formulation with multiple connected domains, you identify the nonconducting cut domain. In ANSYS Maxwell, the process of cut domain generation is automatically done based on the automatic identification of tree and cotree algorithm (Figure 2).

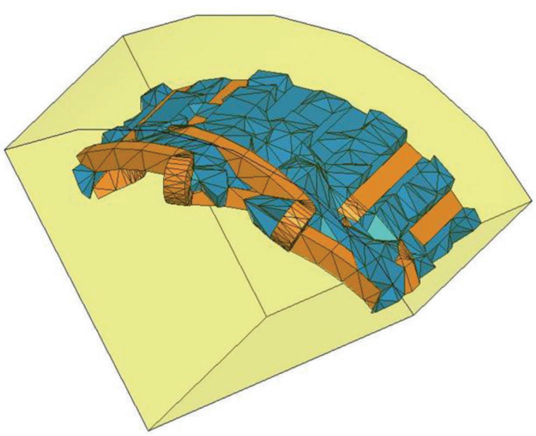

Figure 3. Quarter model of three-phase synchronous generator with damper

Case Study

This case study illustrates the automatic creation of a cut domain for a one-phase winding in a three-phase synchronous generator with damper. The one-phase winding is colored brown (Figure 3), and the automatically identified cut represented by one layer of elements is blue. Taking advantage of the periodic boundary condition, only one quarter is modeled. For the damper with induced eddy current, a total of 16 cuts are automatically identified, which precisely matches the number of 16 holes; even one hole is cut into two halves by the master/slave boundary (Figure 4).

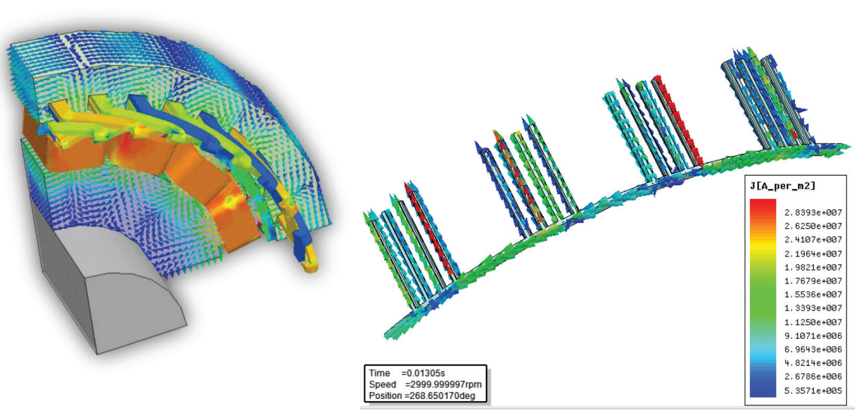

Figure 4. Magnetic flux density (left) and eddy-current distribution on damper (right), 16 cuts

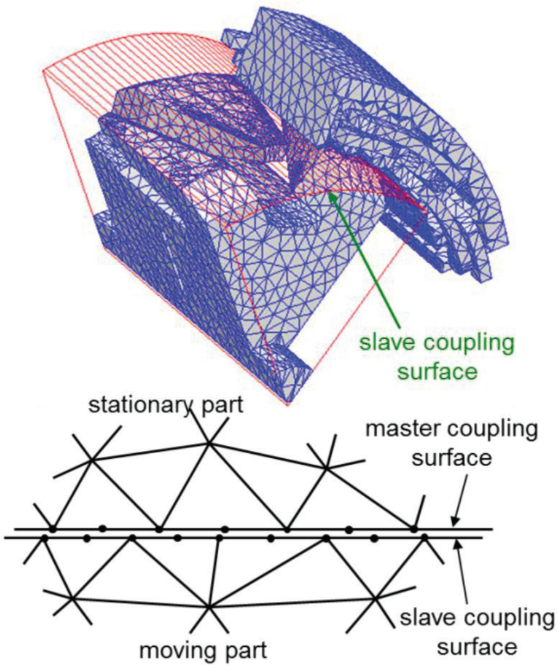

Figure 5. Moving mesh coupling technology

When rigid motion is involved in magnetic transient analysis, two independent meshes must be coupled together after an arbitrary displacement of the moving part. To achieve maximum flexibility, non-conforming meshes are used for the coupling (Figure 5). This means that the scalar potential at each node, the vector field at each edge on the slave coupling surface, has to be mapped onto the master coupling surface to eliminate all unknowns on the slave surface.

For mapping a vector field, the process of splitting slave edge variables with respect to the trace of the master mesh while preserving valid cutting domains is very complicated. To overcome this difficulty, a separation technique is introduced to confine each cut generated to either the stationary or moving part without crossing the coupling interface. As a result, the process of mapping the vector field is completely avoided, and only the node-based scalar potential is involved in the coupling.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

© 2013 ANSYS, Inc. All Rights Reserved.

ANSYS, Inc. is one of the world's leading engineering simulation software providers. Its technology has enabled customers to predict with accuracy that their product designs will thrive in the real world. The companyoffers a common platform of fully integrated multiphysics software tools designed to optimize product development processes for a wide range of industries, including aerospace, automotive, civil engineering, consumer products, chemical process, electronics, environmental, healthcare, marine, power, sports and others. Applied to design concept, final-stage testing, validation and trouble-shooting existing designs, software from ANSYS can significantly speed design and development times, reduce costs, and provide insight and understanding into product and process performance.Visit www.ansys.com for more information.