LECTURE 7

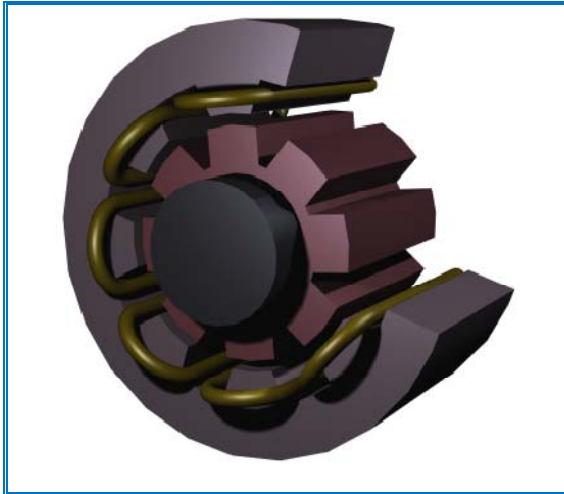**COMSOL MULTIPHISICS**

COMSOL Multiphysics is a finite element analysis, solver and Simulation software / FEA Software package for various physics and engineering applications, especially coupled phenomena, or multiphysics. COMSOL Multiphysics also offers an extensive interface to MATLAB and its toolboxes for a large variety of programming, preprocessing and postprocessing possibilities. The packages are cross-platform (Windows, Mac, Linux). In addition to conventional physics-based user interfaces, COMSOL Multiphysics also allows for entering coupled systems of partial differential equations (PDEs). The PDEs can be entered directly or using the so-called weak form (see finite element method for a description of weak formulation). An early version (before 2005) of COMSOL Multiphysics was called FEMLAB.

COMSOL was started by graduate students to Germund Dahlquist based on code developed for a graduate course at the Royal Institute of Technology (KTH) in Stockholm, Sweden.

In the chapter we will consider how to simulate Generator in 2D. This example shows how the circular motion of a rotor with permanent magnets generates an induced EMF in a stator winding. The generated voltage is calculated as a function of time during the rotation. The model also shows the influence on the voltage from material parameters, rotation velocity, and number of turns in the winding.

The center of the rotor consists of annealed medium carbon steel, which is a material with a high relative permeability. The center is surrounded with several blocks of a permanent magnet made of samarium and cobalt, creating a strong magnetic field. The stator is made of the same permeable material as the center of the rotor, confining the field in closed loops through the winding. The winding is wound around the stator poles. **Figure** shows the generator with part of the stator sliced in order to show the winding and the rotor.

Figure: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.

Modeling in COMSOL Multiphysics

The COMSOL Multiphysics model of the generator is a time-dependent 2D problem on a cross section through the generator. This is a true time-dependent model where the motion of the magnetic sources in the rotor is accounted for in the boundary condition between the stator and rotor geometries. Thus, there is no Lorentz term in the equation, resulting in the PDE

$$\sigma \frac{\partial \mathbf{A}}{\partial t} + \nabla \times \left(\frac{1}{\mu} \nabla \times \mathbf{A} \right) = 0$$

where the magnetic vector potential only has a z component.

Rotation is modeled using a deformed mesh application mode (ALE), in which the center part of the geometry, containing the rotor and part of the air-gap, rotates with a rotation transformation relative to the coordinate system of the stator. The rotation of the deformed mesh is defined by the transformation

$$\begin{bmatrix} x_{rot} \\ y_{rot} \end{bmatrix} = \begin{bmatrix} \cos(\omega t) & -\sin(\omega t) \\ \sin(\omega t) & \cos(\omega t) \end{bmatrix} \cdot \begin{bmatrix} x_{stat} \\ y_{stat} \end{bmatrix}$$

The rotor and the stator are drawn as two separate geometry objects, so it is possible to use an assembly (see "Using Assemblies" on page 405 of the *COMSOL Multiphysics Modeling Guide* for further details). This has several advantages: the coupling between the rotor and the stator is done automatically, the parts are meshed independently, and it allows for a discontinuity in the vector potential at the interface between the two geometry objects (called slits). The rotor problem is solved in a rotating coordinate system where the rotor is fixed (the rotor frame), whereas the stator problem is solved in a coordinate system that is fixed with respect to the stator (the stator frame). An identity pair connecting the rotating rotor frame with the fixed stator frame is created between the rotor and the stator. The identity pair enforces continuity for the vector potential in the global fixed coordinate system (the stator frame).

The material in the stator and the center part of the rotor has a nonlinear relation between the magnetic flux, **B** and the magnetic field, **H**, the so called B-H curve. In COMSOL Multiphysics, the B-H curve is introduced as an interpolation function; see Figure 3-3. The function can be used in the subdomain settings. Usually B-H curves are specified as |**B**| versus |**H**|, but the perpendicular waves application mode must have |**H**| versus |**B**|. It is therefore important that the H-data is entered as $f(x)$-data of the interpolation function and the B-data entered as x-data. This relationship for |**H**| is predefined for the material **Soft Iron (without losses)** in the materials library that is shipped with the AC/DC Module, acdc_lib.txt.
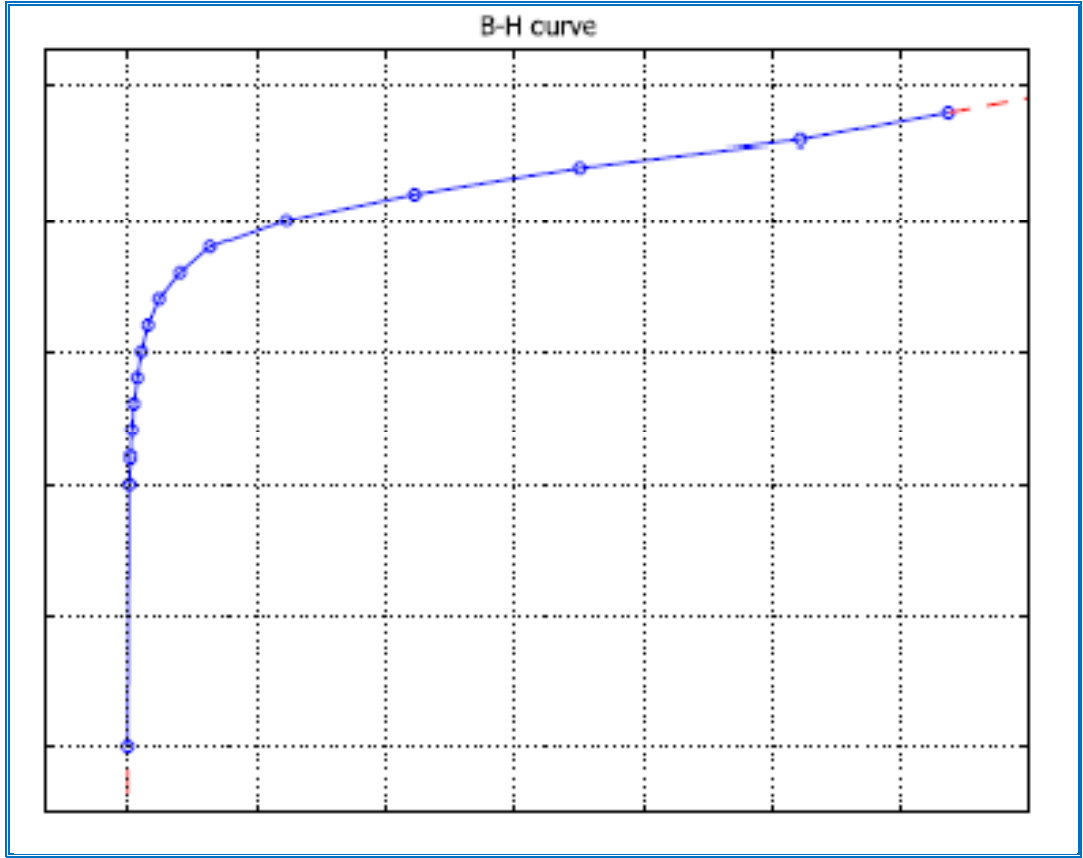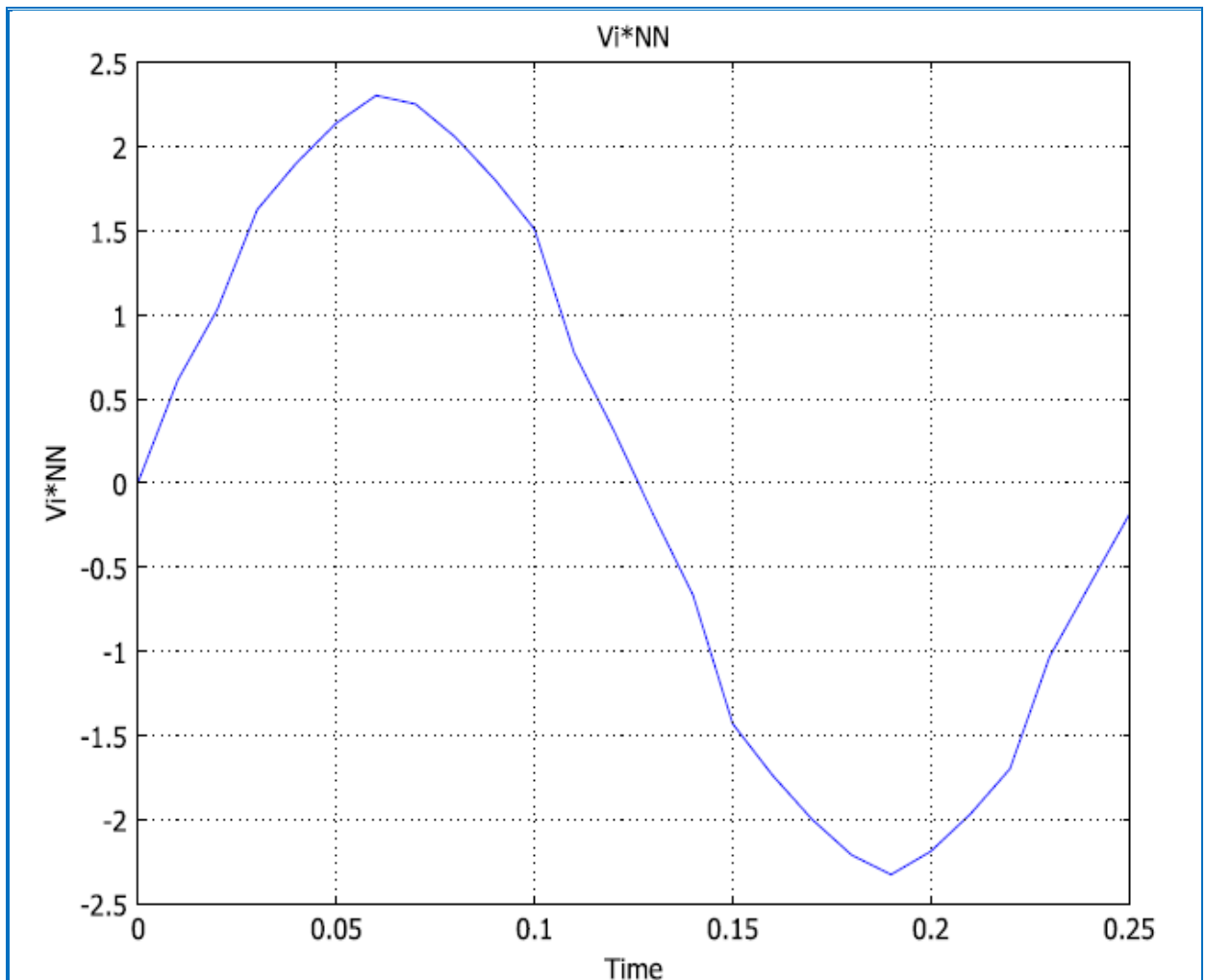
Figure : *The norm of the magnetic flux, |**B**|, versus the norm of the magnetic field, |**H**|, for the rotor and stator materials. The inverse of this curve is used in the calculation.*

The generated voltage is computed as the line integral of the electric field, **E**, along the winding. Because the winding sections are not connected in the 2D geometry, a proper line integral cannot be carried out. A simple approximation is to neglect the voltage contributions from the ends of the rotor, where the winding sections connect. The voltage is then obtained by taking the average z component of the **E** field for each winding cross-section, multiplying it by the axial length of the rotor, and taking the sum over all winding cross sections.

$$V_i = NN \sum_{windings} \frac{L}{A} \int E_Z \, dA$$

where L is the length of the generator in the third dimension, NN is the number of turns in the winding, and A is the total area of the winding cross-section.

The generated voltage in the rotor winding is a sinusoidal signal. At a rotation speed of 60 rpm the voltage will have an amplitude around 2.3 V for a single turn winding; see **Figure**.

Figure: The generated voltage over one quarter of a revolution. This simulation used a single-turn winding.

The norm of the magnetic flux, |**B**|, and the field lines of the **B** field are shown below in **Figure** at time 0.20 s.
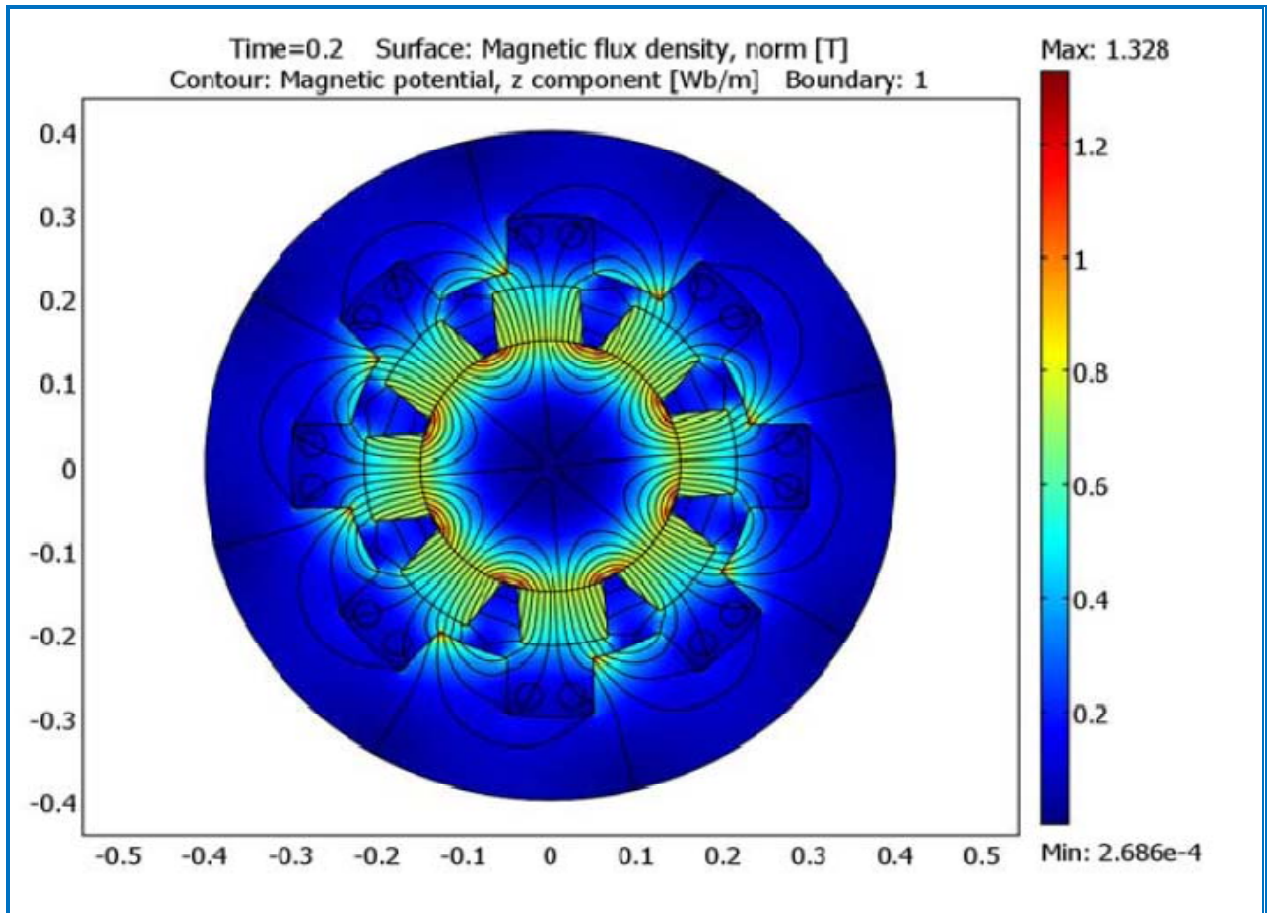
Figure : The norm and the field lines of the magnetic flux after 0.2 s of rotation. Note the brighter regions, which indicate the position of the permanent magnets in the rotor.

Model Library path: ACDC_Module/Motors_and_Drives/generator_2d

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, make sure that **2D** is selected in the **Space dimension** list.

2. In the **AC/DC Module** folder, select **Rotating Machinery>Rotating Perpendicular Currents**.

3. Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

1. From the **Options** menu, choose **Constants**.

2. In the **Constants** dialog box, define the following constants with names and expressions.The description field is optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
t	0[s]	Time for stationary solution
rpm	60[1/min]	Revolutions per minute
A	pi*(1.5[cm])^2	Stator wire cross-section area
L	0.4[m]	Length of the generator
NN	1	Number of turns in stator winding

When you enter the constant t in the table, the text turns red. The reason is that t is the reserved variable name for time in COMSOL Multiphysics. However, because of the solution procedure in this model—you first compute a stationary solution which you then use as the initial condition for a subsequent time-dependent simulation—t needs to be given a value (zero) also for the stationary case. It will subsequently be overridden by the t values from the transient solver.

GEOMETRY MODELING

The geometry modeling of this model is rather extensive, so you can choose to import the generator geometry from a binary file. Select the section that you prefer.

Importing the Geometry from a Binary File

1. From the **File** menu, select **Import>CAD Data From File**.
2. In the **Import CAD Data From File** dialog box, make sure that either **All 2D CAD files** or **COMSOL Multiphysics file** is selected in the **Files of type** list.
3. From the COMSOL Multiphysics installation directory, go to the model library path given on page 46. Select the generator_2d.mphbin file, and click **Import**.
4. Skip the section "Creating the Geometry from Scratch" and continue at "Assemble the Geometry" on page 50.

Creating the Geometry from Scratch

Please follow the steps in the geometry modeling carefully. The instructions in the following sections assume that the boundary numbering is that resulting from these

steps. After all geometry operations and the assembly step, there should be a total of 152 boundaries

1. Choose **Draw>Specify Objects>Circle** to create circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
CI	0.3	Center	0	0	22.5
C2	0.235	Center	0	0	0
C3	0.225	Center	0	0	0
C4	0.4	Center	0	0	0

2. Choose **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
RI	0.1	1	Center	0	0	0
R2	0.1	1	Center	0	0	45
R3	0.1	1	Center	0	0	90
R4	0.1	1	Center	0	0	135

3. Open the **Create Composite Object** dialog box from the **Draw** menu or click on the toolbar button with the same name.

4. In the dialog box clear the **Keep interior boundaries** check box.

5. Type C2+C1*(R1+R2+R3+R4) in the **Set formula** edit field.

6. Click **OK**.

7. Open the **Create Composite Object** dialog box again and select the **Keep interior boundaries** check box.

8. Type C4+CO1-C3 in the **Set formula** edit field.

9. Click **OK**.

10. Create two new circles according to the table below.

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
CI	0.015	Center	0.025	0.275	0
C2	0.015	Center	-0.025	0.275	0

11. Click the **Zoom Extents** button to get a better view of the geometry.

12. Select the circles C1 and C2.

13. Press Ctrl+C to copy the circles.

14. Press Ctrl+V to paste the circles. Make sure that the displacements are zero and click **OK**.

15. Go to **Draw>Modify>Rotate** and type 45 in the **Rotation angle** edit field before clicking **OK**.

16. Repeat the last two steps but with rotation angles 90, 135, 180, −45, −90, and−135.

17. Select all objects by pressing Ctrl+A, then click the **Union** toolbar button.

18. Create four circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
CI	0.215	Center	0	0	0
C2	0.15	Center	0	0	22.5
C3	0.15	Center	0	0	22.5
C4	0.225	Center	0	0	0

19. Create four rectangles with the following properties:

| NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
RI	0.1	1	Center	0	0	22.5
R2	0.1	1	Center	0	0	-22.5
R3	0.1	1	Center	0	0	67.5
R4	0.1	1	Center	0	0	-67.5

20. Open the **Create Composite Object** dialog box and clear the **Keep interior boundaries** check box.

21. Type C2+C1*(R1+R2+R3+R4) in the **Set formula** edit field.

22. Click **OK**.

23. Open the **Create Composite Object** dialog box again and select the **Keep interior boundaries** check box.

24. Type CO2+C3+C4 in the **Set formula** edit field.

25. Click **OK**.

The geometry in the drawing area should now look like that in **Figure.**
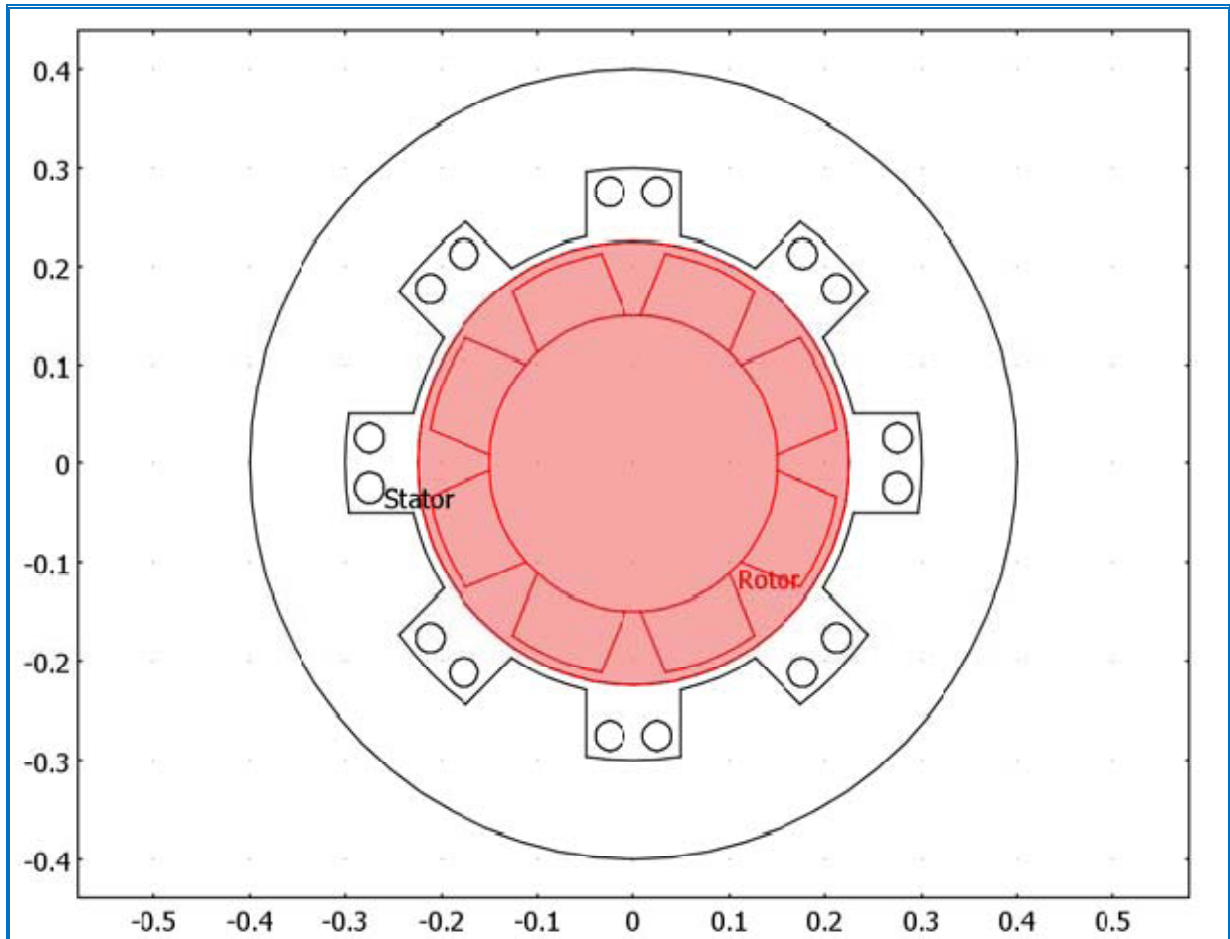
Figure: *Generator geometry.*

Assemble the Geometry

1. Open the **Create Pairs** dialog box from the **Draw** menu.

2. Select both objects (CO1 and CO3), and click **OK**.

You have now created two separate geometry objects in an assembly. You have also created an identity pair connecting the physics in the rotor and the stator, taking the rotation into account. An interface between two assemblies is actually two groups of overlapping boundaries that are, without the pair connection, completely separated in terms of mesh and solution variables. This is necessary to

allow a sliding mesh at the interface between the stator and rotor where the solution variable becomes discontinuous across that interface.

PHYSICS SETTINGS

1. From the **Options** menu point to **Integration Coupling Variables** and click **Subdomain Variables**.

2. In the **Subdomain Integration Variables** dialog box define a variable with integration order 4 and global destination according to the following table:

NAME	EXPRESSION IN SUBDOMAINS 5–8, 13–16	EXPRESSION IN SUBDOMAINS 3, 4, 9–12, 17, 18
Vi	-L*Ez_emqa/A	L*Ez_emqa/A

3. Click **OK**.

Steps 4–6 below are only necessary to check that the library you need is loaded and to view the interpolation function of one of the materials in this library. These steps can be skipped if you know that you have the library loaded. If you later notice that you are missing this library, just go back to these steps and add the library.

4. Open the **Options** menu and select **Materials/Coefficients Library**.

5. Make sure that the library acdc_lib.txt is loaded (click the minus in front of other libraries to remove their content from the view). If you cannot find the library, load the library by clicking the **Add Library** button. The file should be located under the data folder in the COMSOL installation directory.

6. Click **OK** to close the **Materials/Coefficients Library** dialog box.
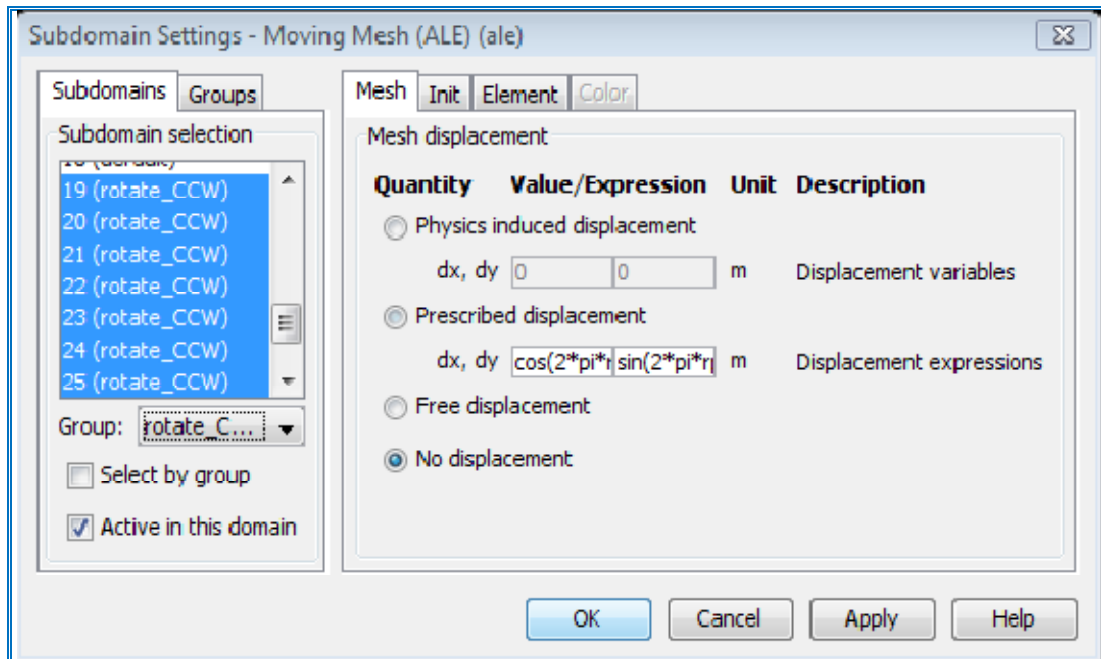
Subdomain Settings

1. From the **Multiphysics** menu, select **Perpendicular Induction Currents, Vector Potential (emqa)** if it is not already selected.

2. Choose **Physics>Properties**.

3. In the **Application Mode Properties** dialog box, choose **On** from the **Weak constraints** list. The ideal weak constraints help conserving the magnetic flux across the assembly boundaries between the rotor and the stator. This allows

you to specify an absolute tolerance for this entity when solving the time-dependent model.

4. Click **OK**.

5. From the **Physics** menu open the **Subdomain Settings**, then select any entry in the **Subdomain selection** list.

6. Click the **Load** button to open the **Materials/Coefficients Library** dialog box.

7. In the dialog box select the **Soft Iron (without losses)** material from the**Electric (AC/DC) Material Properties** library.

8. Click **Apply**. The material **Soft Iron (without losses)** has now been added to the model. You see which materials you have added to the model under the **Model** item in the **Materials** tree view.

9. Repeat the last two steps for the materials **Samarium Cobalt (Radial, inward)** and **Samarium Cobalt (Radial, outward)**. Before you close the dialog box, make sure that you see all three materials under **Model** in the **Materials** tree view.

10. Click **Cancel** to close the **Materials/Coefficients Library** dialog box.

11. Enter subdomain settings and select materials according to the table below. To change the constitutive relation,

SETTINGS	SUBDOMAINS 20, 23, 24, 27	SUBDOMAINS 21, 22, 25, 26	SUBDOMAINS 2, 28	ALL OTHER SUBDOMAINS
$B \leftrightarrow H$	$B = \mu_0\mu_r H + B_r$	$B = \mu_0\mu_r H + B_r$	$H = f(|B|)e_B$	$B = \mu_0\mu_r H$
Library material	Samarium Cobalt (Radial, inward)	Samarium Cobalt (Radial, outward)	Soft Iron (without losses)	

12. Click **OK**.

13. From the **Multiphysics** menu select **Moving Mesh (ALE) (ale)**.

14. From the **Physics** menu open the **Subdomain Settings**.

15. Select Subdomains 19–28, then select the predefined group **rotate_CCW** from the **Group** list.

This group uses a prescribed displacement with expressions that define counterclockwise rotation around the origin. The default setting for all other subdomains is no displacement.

16. Click **OK** to close the dialog box.

Boundary Conditions

Leave the boundary settings at their defaults. The identity pair creates a coupling between the rotor and the stator in the ALE frame.

MESH GENERATION

1. From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.

2. From the **Predefined mesh sizes** list, select **Finer**.

3. Click the **Custom mesh size** option button and type **2** in the **Resolution of narrow regions** edit field.

4. Click **Remesh** to generate the mesh.

5. Click **OK** to close the dialog box.

COMPUTING THE SOLUTION

Begin by computing the initial solution for the subsequent time-dependent simulation.

1. Open the **Solver Parameters** dialog box by clicking the corresponding button on the Main toolbar or from the **Solve** menu.

2. With the **Auto select solver** check box selected, select **Static** from the **Analysis** list.

3. Click **OK**.

4. Click the **Solve** button.

Next, simulate the generator with the rotor in motion. By default, the solution you just calculated will be used as the initial solution.
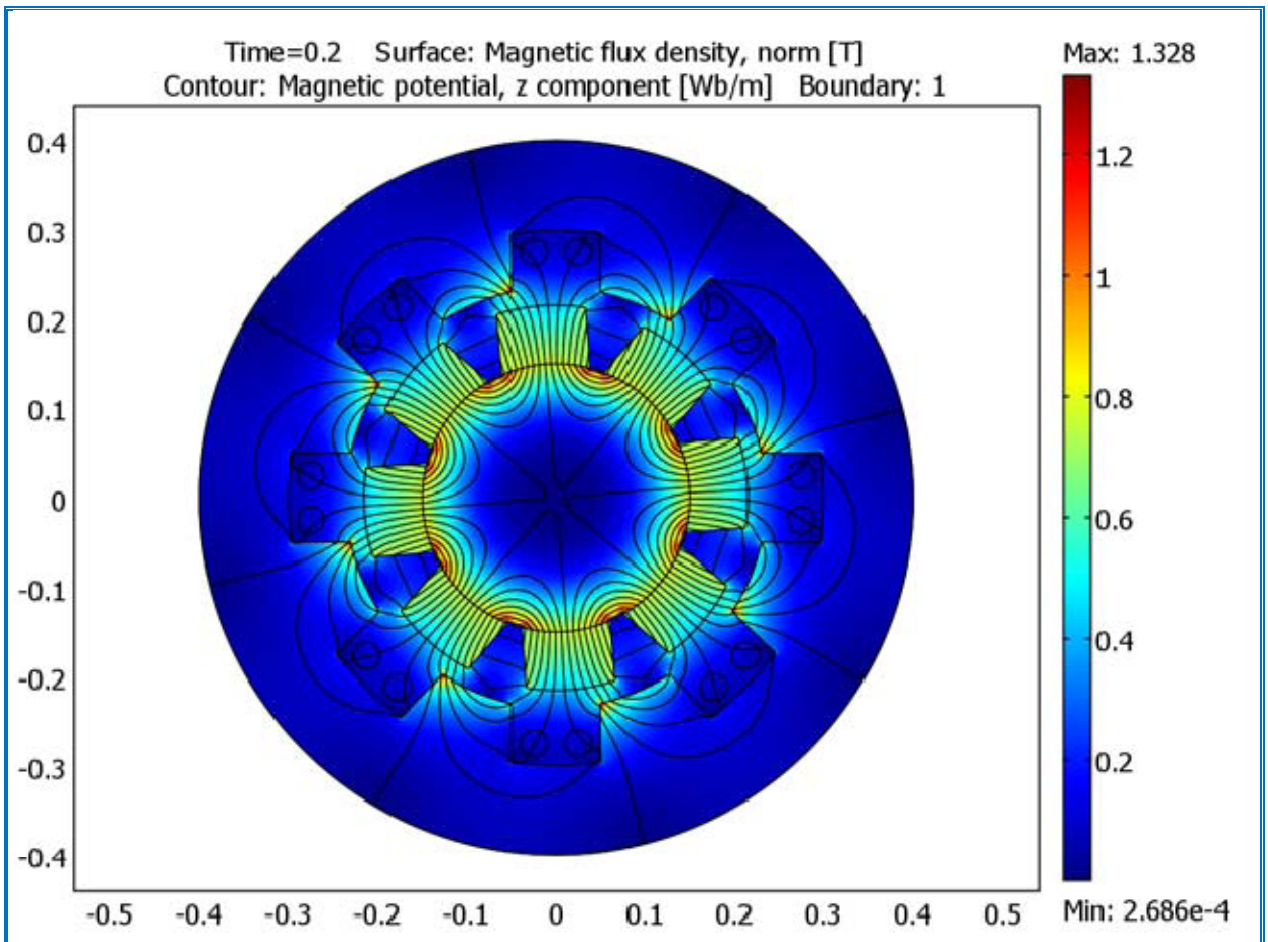
1. Open the **Solver Parameters** dialog box again and select **Transient** from the **Analysis** list.

2. 2In the **Times** edit field type range(0,0.01,0.25), and in the **Absolute tolerance** edit field type Az 1e-3 lm1 5e3.
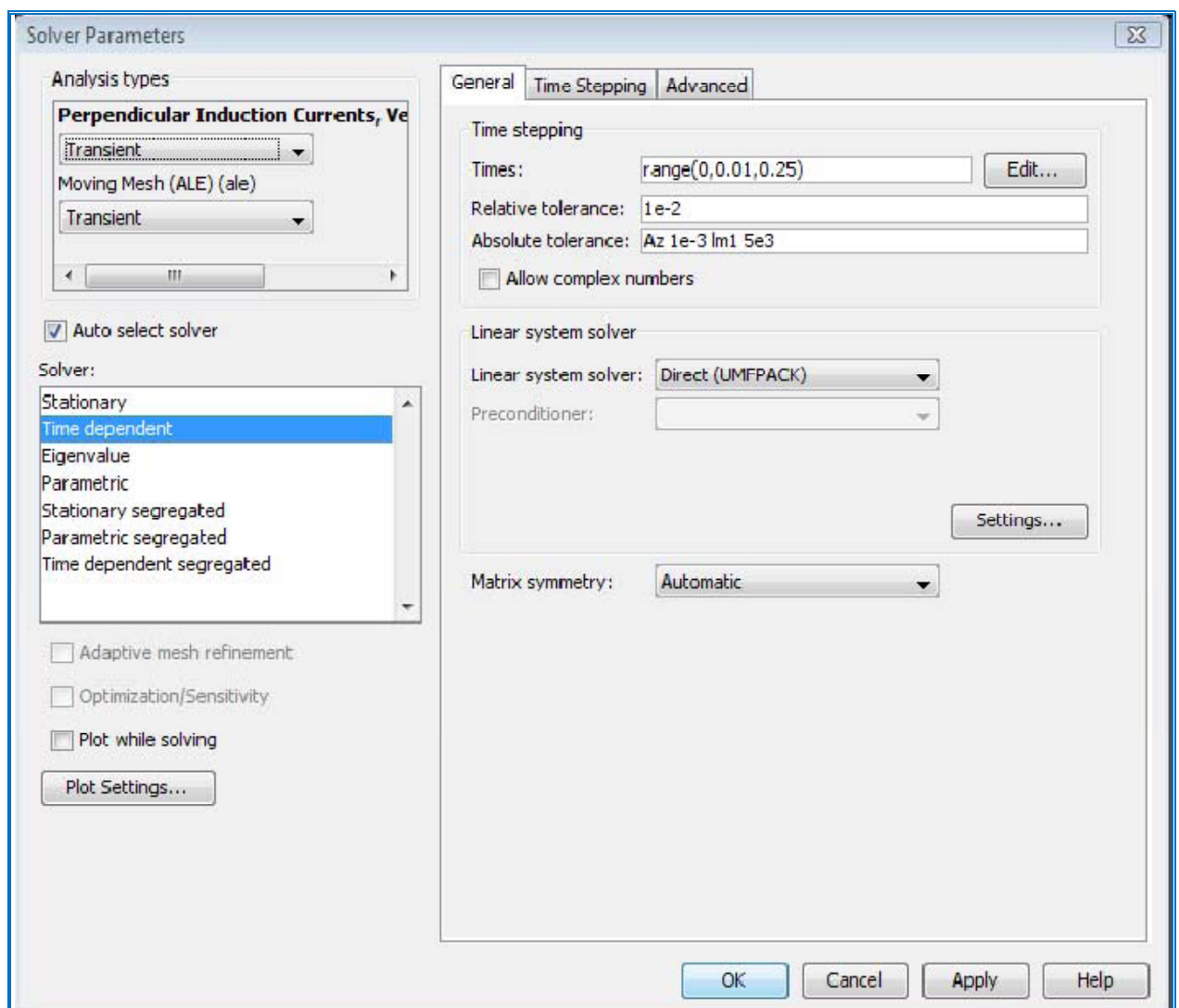
3. Click **OK**.

4. Click the **Restart** button.

POSTPROCESSING AND VISUALIZATION

1. From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2. On the **General** page, select the solution for time 0.2 s from the **Solution at time** list. Clear the **Geometry edges** check box.

3. On the **Surface** page select **Magnetic flux density, norm** from the **Predefined quantities** list or type normB_emqa in the **Expression** edit field.

4. On the **Contour** page type Az in the **Expression** edit field and type 15 in the **Levels** edit field. Click to select the **Contour plot** check box.

5. In the **Contour color** area click the **Uniform color** option button, and clear the **Color legend** check box.

6. Click the **Color** button and select a black color. Click **OK** to close the **Contour Color** dialog box.

7. On the **Boundary** page type 1 in the **Expression** edit field. Click to select the **Boundary plot** check box.

8. In the **Boundary color** area click the **Uniform color** option button.

9. Click the **Color** button and select a black color. Click **OK** to close the **Boundary Color** dialog box.

10. Click **OK** to close the **Plot Parameters** dialog box.

Time=0.2 Surface: Magnetic flux density, norm [T] Max: 1.328
Contour: Magnetic potential, z component [Wb/m] Boundary: 1

Min: 2.686e-4

To view the induced voltage as a function of time, proceed with the steps below.

1. From the **Postprocessing** menu, select **Domain Plot Parameters**.

2. On the **Point** page select an arbitrary point in the **Point selection** list, then type Vi*NN in the **Expression** edit field.

3. Click **OK**.

To create an animation of the rotating rotor, do the following steps.

1. Open the **Plot Parameters** dialog box.

2. Click the **Animate** tab and make sure that all time steps are selected.

3. Click the **Start Animation** button. When the movie has been generated, the *COMSOL Movie Player* opens it.