Generator

Introduction

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 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. In Figure 3-1 shows the generator with part of the stator sliced, in order to show the winding and the rotor.

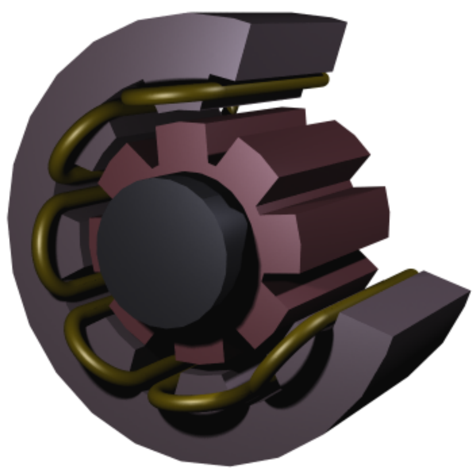


Figure 3-1: A drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.

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

Generator in 3D

Introduction

This model is a 3D version of the 2D generator model on page 39, so most of the details about the model are explained there. The main difference is that this is a static example, calculating the magnetic fields around and inside the generator.

Modeling in COMSOL Multiphysics

The model has some differences compared to the 2D generator model. The PDE is simplified to solve for the magnetic scalar potential, V_m, instead of the vector potential, **A**. It is based on the assumption that currents can be neglected, which holds true when the generator terminals are open. The equation for V_m becomes

$$-\nabla \cdot (\mu \nabla V_m - \mathbf{B}_r) = 0$$

The stator and center of the rotor are made of annealed medium-carbon steel, which is a nonlinear magnetic material. This is implemented in COMSOL Multiphysics as an interpolation function for the relative permeability, μ_r. The difference compared to the 2D version is that the argument to this function needs to be changed to the norm of the magnetic field, $|\mathbf{H}|$, in this model. The reason is that in this problem, **H** is calculated from the dependent variable V_m according to

$$\mathbf{H} = -\nabla V_m$$

B is then calculated from **H** using the relation

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r$$

resulting in an implicit or circular dependence of μ_r if **B** were used as the argument for the interpolation function.

Results and Discussion

Figure 3-9 shows the norm of the magnetic flux for a slice through a centered cross section of the generator. The plot also shows the streamlines of the magnetic flux. The starting points of the streamlines have been carefully selected to show the closed loops between neighboring stator and rotor poles. A few streamlines are also plotted at the edge of the generator, to illustrate the field there.

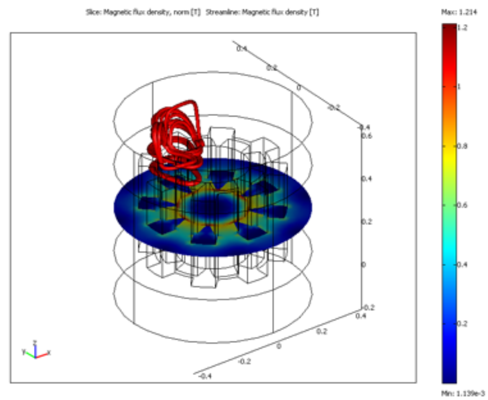


Figure 3-9: A combined slice and streamline plot of the magnetic flux density.

Model Library path: `AC/DC_Module/Motors_and_Drives/generator3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list.

2 In the **AC/DC Module** folder, select **Statics>Magnetostatics, No Currents.**

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

GEOMETRY MODELING

1 Go to **Draw>Work Plane Settings.**

2 In the dialog box that appears, click **OK**.

3 Go to **Draw>Specify Object>Circle** to create circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.3	Center	0	0	22.5
C1	0.235	Center	0	0	0

4 Go to **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	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

5 In the **Create Composite Object** dialog box, clear the **Keep interior boundaries** check box.

6 Enter the formula `C2+C1*(R1+R2+R3+R4)`.

7 Click **OK**.

8 Create a circle with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.215	Center	0	0	0

9 Draw four rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	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

10 In the **Create Composite Object** dialog box, enter the formula `C1*(R1+R2+R3+R4)`.

11 Click **OK**.

12 Create a circle with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.15	Center	0	0	22.5

13 In the **Create Composite Object** dialog box, make sure that the **Keep interior boundaries** check box is not selected.

14 Enter the formula `C1+CO2`.

15 Click **OK**.

16 Draw a circle with parameters according to the table below.

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
`C1`	0.15	`Center`	`0`	`0`	`22.5`

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

18 Go to **Draw>Extrude**.

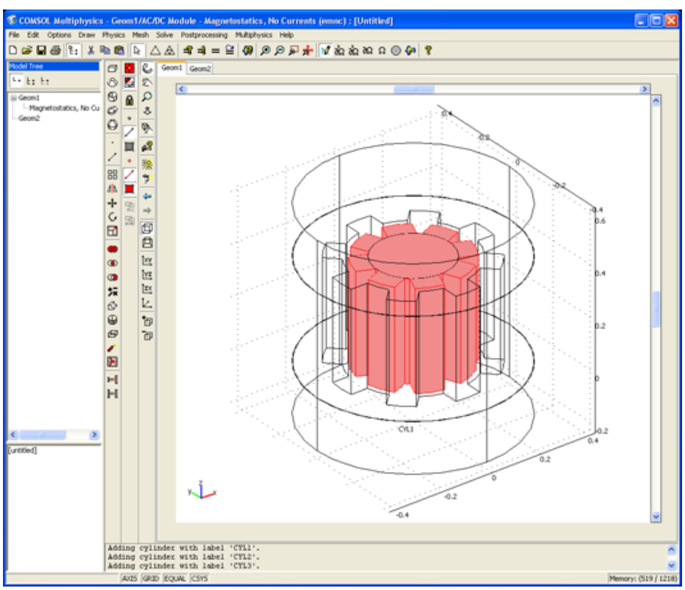
19 In the dialog box that appears, select all objects and type `0.4` in the **Distance** edit field.

20 Click **OK.**

21 Use the **Cylinder** tool to create cylinders with parameters according to the table below.

NAME	AXIS BASE POINT Z	RADIUS	HEIGHT
CYL1	`-0.2`	`0.4`	`0.2`
CYL2	`0`	`0.4`	`0.4`
CYL3	`0.4`	`0.4`	`0.2`

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



PHYSICS SETTINGS

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

2 In the **Constants** dialog box, define the following constants with names and expressions:

NAME	EXPRESSION	DESCRIPTION
BrSmCo	0.84	Remanent flux in permanent magnets (T)
murSmCo	1	Relative permeability in permanent magnets

3 Click **OK**.

4 Go to **Options>Expressions>Scalar Expressions**.

5 In the **Scalar Expressions** dialog box, define:

NAME	EXPRESSION
R	sqrt(x^2+y^2)

6 Click **OK**.

7 Go to **Options>Functions**.

8 In the **Functions** dialog box, click **New**. Type `MUR` as **Function name** and select **Interpolation**, and data from **Table**.

9 Click **OK**.

10 Set the **Interpolation method** to **Linear**, the **Extrapolation method** to **Constant**.

11 Input interpolation points according to the list below.

X	F(X)
4000	281
8000	158
16000	87.0
24000	60.9
32000	47.1
40000	38.5
48000	32.6
64000	24.9
80000	20.3
96000	17.2
112000	14.9

X	F(X)
128000	13.2
144000	11.9
176000	9.93
208000	8.56
240000	7.56
272000	6.78
304000	6.18
336000	5.68
396000	4.96

12 Click **OK**.

Subdomain Settings

1 Enter subdomain settings for the active subdomains according to the table below. In order to set a remanent flux density, you need to click the option button for $\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$ first.

SUBDOMAIN	1, 3, 4	5, 9, 10, 13	6, 8, 11, 12	2, 7
Rel. permeability	1	murSmCo	murSmCo	MUR(normH_emnc)
Rem. flux density x		-BrSmCo*x/R	BrSmCo*x/R	
Rem. flux density y		-BrSmCo*y/R	BrSmCo*y/R	

2 Click the **Init** tab, select all subdomains, and enter R in the **Vm(t$_0$)** edit field.

3 Click **OK**.

Boundary Conditions

1 In the **Boundary Settings** dialog box, apply the following boundary condition:

BOUNDARY	ALL
Boundary condition	Magnetic insulation

2 Click **OK**.

MESH GENERATION

1 In the **Free Mesh Parameters** dialog box, select **Finer** from the **Predefined mesh sizes** list.

2 Click **Remesh** and then **OK.**

COMPUTING THE SOLUTION

Click the **Solve** toolbar button to compute the solution.

POSTPROCESSING AND VISUALIZATION

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

2 Select the check boxes for **Slice** plot and **Streamline** plot on the **General** tab.

3 Click the **Slice** tab. From the **Predefined quantities** list, select **Magnetic flux density, norm**.

4 Type `0` in the edit field for **x levels**, and `1` in the edit field for **z levels**.

5 Click the **Streamline** tab and select **Magnetic flux density** in the **Predefined quantities** list.

6 Click the **Specify start point coordinates** button and enter the values according to the table below.

FIELD	EXPRESSION
`x`	`0 0 0 0 0 0 0`
`y`	`0.35 0.35 0.35 0.35 0.35 0.3 0.2`
`z`	`linspace(0.1,0.59,7)`

7 Select **Tube** in the **Line type** list and click **OK**.