

ELEC-E8407

Electromechanics

Simulation exercise:

Transformer

Simulation exercise: E core type Transformer

The simulation exercise aims to present typical setups for the estimation of electrical quantities in power transformers to create a better understanding of the basic operating principles of a power transformer. The transformer model studied in this simulation exercise is implemented using the commercial finite element software **COMSOL Multiphysics**. The primary winding of the transformer is connected to a primary resistor, R_p and the AC voltage source, V_{ac} while the secondary winding is connected to the secondary load resistor, R_s as shown in Figure 1 below. The model is solved in frequency domain for a line frequency of 50 Hz.

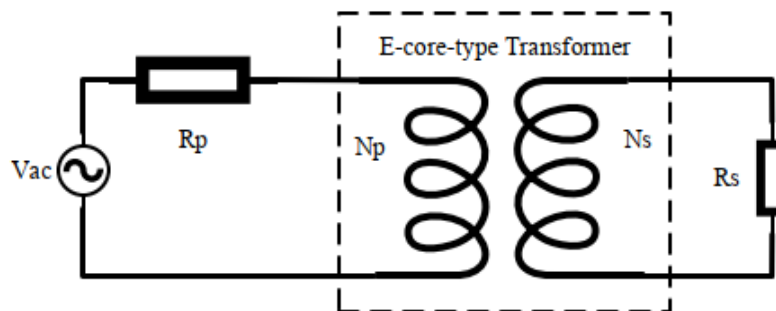


Figure 1: Transformer supply setup

An equivalent circuit model based on physical reasoning is often used to compute important quantities such as voltage, current, reactance etc. of the practical transformer. In the equivalent circuit model, the quantities of primary or secondary side can be transferred to another side by referring all the quantities to the primary or secondary (most convenient one). Figure (2) shows a single-phase equivalent circuit model where all the secondary side quantities are transferred to the primary side. The primed mark indicates all the referred quantities. The following symbols are used for the quantities on the single-phase equivalent circuit:

- U_{v1} Primary phase to neutral voltage
- I_1 Primary current
- R_1 Primary resistance
- $X_{\sigma 1}$ Primary stray reactance
- U'_{v2} Secondary phase to neutral voltage
- I'_2 Secondary current
- R'_2 Secondary resistance
- $X'_{\sigma 2}$ Secondary stray reactance
- R_{Fe} Iron loss resistance
- X_m Magnetizing (principle) reactance

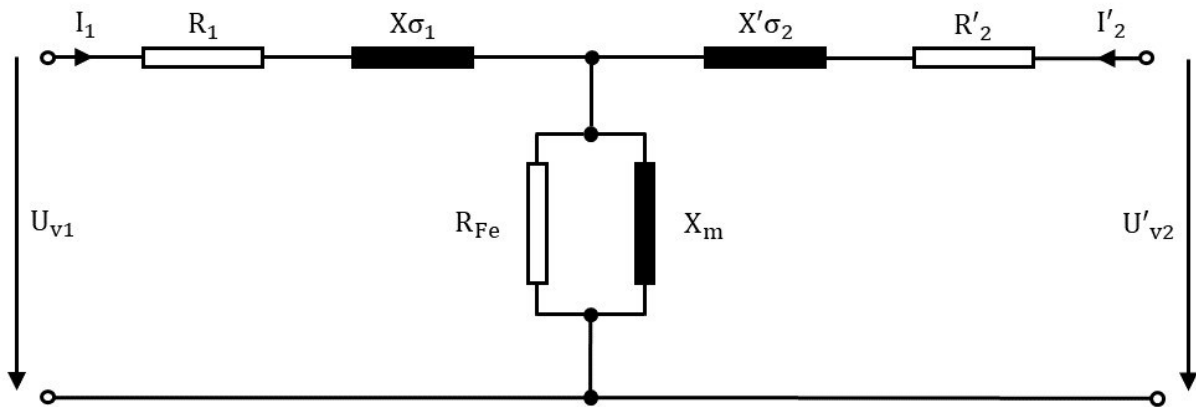


Figure 2: A single phase equivalent circuit model referred to the primary side

The input parameters shown in table 1 are parameterized and can therefore easily be changed.

Table 1: Input parameters

Parameter	Values	Description
Vac	24V	Primary (Supply) voltage
nu	50Hz	Frequency of supply voltage
Np	240	Number of turns in primary winding
Ns	150	Number of turns in secondary winding
R_p	0.05Ω	Primary side resistance
R_s	5Ω	Load resistance
Cs	2.5mF	Capacitive load
Ls	5mH	Inductive load

The following simulation will be carried out:

- Standard tests (i.e. short-circuit and open circuit simulation)
- Load tests (i.e. resistive, inductive, and capacitive loads)

Simulation setup instructions

Download and install **COMSOL Multiphysics 5.5** software on your personal computer from www.download.aalto.fi and Open the file trans_elec_course_sim. Follow the steps to run the simulations.

a) Input parameters

To define the input parameters: In the **model builder** window locate **Global Definitions – Parameters 1**. **Note Always change all input parameters here**

b) Open Circuit Test: (question 1)

1. Define the primary coil input: In the **model builder** window locate on **Component** (double click) - **magnetic fields** (double click) - **coil 1** (single click), From the **setting window** locate **Coil excitation** list, choose **Circuit (current)**. (i.e. $V_{ac} = 24 \cdot \sqrt{2}$)

2. Define the Secondary coil input: In the **model builder** window locate **coil 2**(single click), From the **setting window** locate **Coil excitation** list, choose **Current** input $I_{coil} = 0$.
3. In the **model builder** window locate **Electrical circuit** (double click) - **External I vs U1**, From the **setting window** locate **Electric potential** list, choose **Coil voltage (mf/coil1)**.
4. In the **model builder** window locate on **Study-1** (single click), next go to **setting window** and click on =**Compute**.
5. Results: In the **model builder** window locate **Results** (double click) - **derived values** (double click) - **Global Evaluation 1**, From the **setting window** click on = **Evaluate**.

c) **Short Circuit Test: (question 1)**

1. Define the primary coil input: The **model builder** window locates **Electric circuit - Voltage source V1**, From the **setting window** locate **voltage V_{src}** and use 10% of the rated supply voltage V_{ac} . (i.e. $0.1 \cdot V_{ac} \cdot \sqrt{2}$)
2. Define the Secondary coil input: In the **model builder** window locate **coil 2**, From the **setting window** locate **Coil excitation** list, choose **Voltage** input $V_{coil} = 0$
- 3 In the **model builder** window locate on **Study-2** (single click), next go to **setting window** and click on =**Compute**.
- 4 Results: In the **model builder** window locate **Results - derived values - Global Evaluation 2**, From the **setting window** click on = **Evaluate**.

d) **Load simulation: (question 2 - 5)**

- 1 Define the primary coil input: The **model builder** window locates **Electric circuit- Voltage source V1**, From the **setting window** locate **voltage V_{src}** and use 100% of rated supply voltage V_{ac} . (i.e. $V_{ac} = 24 \cdot \sqrt{2}$)
- 2 Define the Secondary coil input: In the **model builder** window locate **coil 2**, From the **setting window** locate **Coil excitation** list, choose **Circuit (current)**.
- 3 In the **model builder** window locate **Electrical circuit - External I vs U2**, right click on it and select **Enable**, From the **setting window** locate **Electric potential** list, choose **Coil voltage (mf/coil2)**.
- 4 In the **model builder** window locate **Electrical circuit - Resistor R2**. Right click to enable **Resistor R2**, keep (**Inductor L1** and **Capacitor C1**) disabled.
- 5 In the **model builder** window locate on **Study-3– load_sim** (single click), next go to **setting window** and click on =**Compute**.
- 6 Results: In the **model builder** window locate **Results - derived values - Global Evaluation 3**, From the **setting window** click on = **Evaluate**.
- 7 In the **model builder** window locate **Electrical circuit - Inductor L1**. Right click to enable **Inductor L1**, keep (**Resistor R2** and **Capacitor C1**) disabled.
- 8 In the **model builder** window locate **Electrical circuit - Capacitor C1**. Right click to enable **Capacitor C1** keep (**Resistor R2** and **Inductor L1**) disabled.
- 9 Repeat steps 5 and 6 in each case to obtain the results.

Questions

1. Calculate the equivalent circuit parameters shown in figure 2 from the results of open-circuit and short circuit simulations.
2. Carry out a load simulation by increasing the resistive load from 10Ω - 100Ω in step of 10Ω and explain the effects of increase resistive load on the transformer.
3. Simulate the transformer at resistive (5Ω), capacitive (2.5mF) and inductive (5mH) loads and calculate efficiency and power factor for each load condition.
4. Explain the difference in results obtained in 3
5. Compare the Volumetric flux density distribution in all simulated cases and give comments. Add one figure showing the magnetic flux density distribution of the transformer in any of the case simulated.
6. Give overall comments about the simulation exercise.