5.3 Ideal Transformers

The XFRM_LINEAR part shown in the second row of Table 5.2 can also be used to represent coupled coils. The properties of the XFRM_LINEAR include the inductance of each coil, $L_1$ and $L_2$, and the coupling coefficient, $k$. When $k = 1$ and, also, $L_1$ and $L_2$ are large enough that the impedances of the coils are much larger than the other impedances in the circuit, the XFRM_LINEAR will represent an ideal transformer. The turns ratio of the ideal transformer is related to the coil inductances by

$$n = \frac{N_2}{N_1} = \sqrt{\frac{L_2}{L_1}} \quad (5.2)$$

where $N_i$ is the number of turns in the coil having inductance $L_i$ and $N_2$ is the number of turns in the coil having inductance $L_2$.

Example 5.3 Figure 5.12a shows a circuit that includes an ideal transformer. Ideal transformers can be represented using the PSpice part called XFRM_LINEAR. XFRM_LINEAR is shown in the second row of Table 5.2.

Step 1 Formulate a circuit analysis problem.

Analyze the circuit shown in Figure 5.12a to determine the currents $i_i(t)$ and $i_d(t)$ and the voltages $v_i(t)$ and $v_d(t)$.

Step 2 Describe the circuit using Orcad Capture.

Figure 5.13 shows the circuit described in Capture. The device TX1 is a XFRM_LINEAR part representing the ideal transformer. We need to specify the coil inductances and the coupling coefficient of TX1. The value of the coupling coefficient of an ideal transformer is $k = 1$. Figure 5.12a indicates that the turns ratio of the ideal transformer is $n = 2$. Equation 5.2 requires that $L_2 = 4 L_1$ to obtain a turns ratio equal to 2. Figure 5.12b indicates that the largest impedance in the circuit is 5 Ω. Choosing $L_1 = 10$ Ω and $L_2 = 40$ Ω cause the coil impedances to be $j100$ Ω and $j400$ Ω, both much larger than 5 Ω.
In Figure 5.14 the property editor is used to specify the value of the coupling coefficient and the values of the coil inductances.

The circuit shown in Figure 5.12a consists of two separate parts. One part is comprised of the voltage source, capacitor, the 5 $\Omega$ resistor and the primary (left) coil of the transformer. The other part is comprised of the 20 mH inductor, the 2 $\Omega$ resistor and the secondary (right) coil of the transformer. These two parts are connected magnetically, but not electrically. No charge flows from one part to the other. PSpice will only analyze circuits that consist of a single part, electrically. The resistor R3 is added in Figure 5.13 to connect the two parts electrically. The resistance of R3 is very large so that R3 acts like an open circuit and the circuit in Figure 5.13 will behave like the circuit in Figure 5.12a.

Figure 5.13 shows that printers have been added to print the values of $I_1(\omega)$, $I_2(\omega)$, $V_1(\omega)$ and $V_2(\omega)$ into the PSpice output file.

![Circuit Diagram](image)

**Figure 5.12** A circuit represented in the (a) time and (b) frequency domain.
Step 3 Simulate the circuit using PSpice.

Select PSpice / New Simulation Profile from the Orcad Capture menus to pop up the "New Simulation" dialog box. Specify a simulation name, then select "Create" to close "New Simulation" dialog box and pop up the "Simulation Settings" dialog box. In the "Simulation Settings" dialog box, select "AC Sweep/Noise" as the analysis type. We only want to simulate the circuit at a single frequency so we will set the "Start Frequency" and "End Frequency" to the same value 10 rad/s = 1.591549 Hertz. Finally, select "OK" to close the "Simulation Settings" dialog box and return to the Capture screen.

Select PSpice / Run from the Orcad Capture menus to run the simulation.
Step 4  Display the results of the simulation, for example, using Probe.

After a successful AC Sweep/Noise simulation, Probe, the graphical post-processor for PSpice, will open automatically in a Schematics window. Select View / Output File from the Schematics menus. Scroll through the output file to find results of the AC analysis. These results indicate that

\[ I_1(\omega) = 0.6759 \angle 41.98^\circ = 0.5025 - j \, 0.4521, \]
\[ I_2(\omega) = 0.3378 \angle -137.7^\circ = -0.2500 - j \, 0.2272 \]

and

\[ V_1(\omega) = 0.3395 \angle 47.97^\circ = 0.2273 + j \, 0.2522, \]
\[ V_2(\omega) = 0.6789 \angle 47.97^\circ = 0.4545 + j \, 0.5043 \]

Step 5  Verify that the simulation results are correct.

The ideal transformer in Figure 5.12b is represented by the equations

\[ \frac{V_2(\omega)}{V_1(\omega)} = n = 2 \quad \text{and} \quad \frac{I_1(\omega)}{I_2(\omega)} = \frac{1}{n} = -0.5 \]

Substituting the values of the voltages \( V_1(\omega) \) and \( V_2(\omega) \) obtained using PSpice gives

\[ \frac{0.6789 \angle 47.97^\circ}{0.3395 \angle 47.97^\circ} = 1.9997 \angle 0^\circ = 2 \]

and substituting the values of the currents \( I_1(\omega) \) and \( I_2(\omega) \) obtained using PSpice gives

\[ \frac{0.3378 \angle -137.7^\circ}{0.6759 \angle 41.98^\circ} = 0.4998 \angle -179.68^\circ = -0.5 \]

The simulation results are correct.
Step 6  **Report** the answer to the circuit analysis problem.

The currents and voltages of the ideal transformer in the circuit shown in Figure 5.12a are

\[ i(t) = 0.6759 \cos(5t + 41.98°) \, \text{A} \]
\[ i_s(t) = 0.3378 \cos(5t - 137.7°) \, \text{A} \]

and

\[ v_1(t) = 0.3395 \cos(5t + 47.97°) \, \text{V} \]
\[ v_2(t) = 0.6789 \cos(5t + 47.97°) \, \text{V} \]

### 5.4 **SUMMARY**

PSpice can be used to analyze an AC circuit. The phasors corresponding to the currents and voltages of AC circuits are printed into the PSpice output file using the printers shown in Table 5.1.

Coupled coils and transformers can be incorporated into AC circuits using the PSpice part `K_Linear` and `XFRM_LINEAR` shown in Table 5.2.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>PSpice Name</th>
<th>Library</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="symbol1.png" alt="Symbol" /></td>
<td>Mutual Inductance</td>
<td>K_Linear</td>
<td>ANALOG</td>
</tr>
<tr>
<td><img src="symbol2.png" alt="Symbol" /></td>
<td>Transformer</td>
<td>XFRM_LINEAR</td>
<td>ANALOG</td>
</tr>
</tbody>
</table>