Designing a Transformer in PSpice

Product Version 16.6
April, 2017

Contents

Purpose

This Application Note describes how to design and simulate a transformer using the Cadence PSpice tool.

Audience

The document is for electronics and telecommunication engineers, Power Electronics engineers, and electronic circuit and magnetic design engineers.

Prerequisite

A basic understanding of transformers, coupling of inductors, and the PSpice simulation tool

Download

Click here to download the database used in this Application Note.

Overview

A simple transformer consists of two windings, primary and secondary, and a core.

The primary and secondary windings are magnetically coupled through a transformer core that provides a path for the magnetic lines of force.

An electrical current passes through the primary coil and induces voltage to the secondary due to mutual inductance.

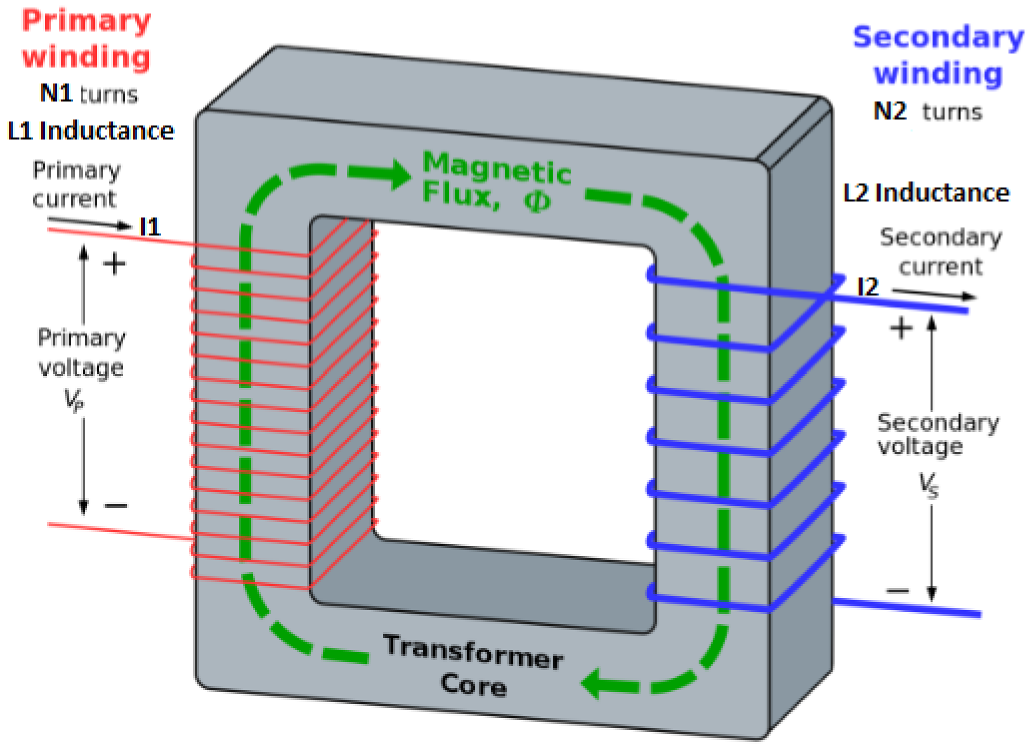


Figure 1: Transformer

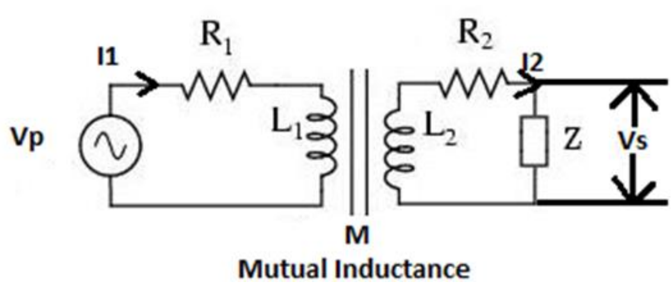


Figure 2: Electrical equivalent circuit of a transformer

The primary and secondary inductances of a transformer can be characterized by three primary parameters:

- L1 inductance (also called a self-inductance)
- L2 inductance (also called a self-inductance)
- Mutual inductance, M=L12

In Figure 2:

Vp=L1 (dI1/dt)+M (dI2/dt)

Vs=L2 (dI2/dt)+M (dI1/dt)

The mutual inductance is:

M=√ (L1*L2)

In an ideal case, mutual inductance is 1, which means a perfect coupling between the primary and secondary coils results in all the magnetic lines of force of the primary passing through the secondary. In a practical case, some magnetic lines of force will not reach the secondary and the mutual inductance will be less than 1.

Considering the coupling coefficient, mutual inductance is:

M=k*√ (L1*L2) where 0<k<1; (k is the "coupling coefficient")

Designing a transformer in PSpice

In PSpice, you can design a transformer using two inductors and couple these by using the coupling symbol.

Inductors are available in the analog.olb library.

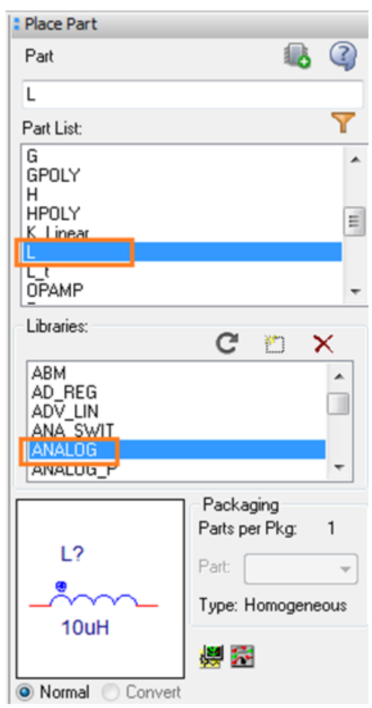


Figure 3: Inductor L in the analog library

Inductors are also available in breakout.olb.

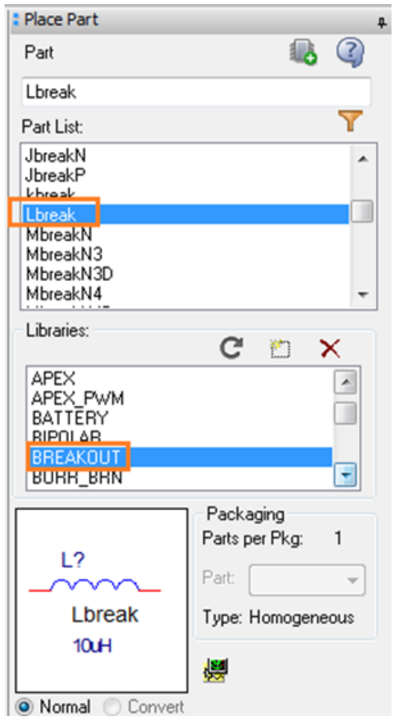


Figure 4: Inductor L in the breakout library

Coupled symbols have no pins; these are represented by the letter K enclosed in a box.

In the coupling symbol, the COUPLING property is the coefficient of mutual coupling, which must be between 0 and 1.0.

For transformers of normal geometry, use 1.0 as the value. Values less than 1.0 occur in air core transformers when the coils do not completely overlap.

There are three types of coupled symbols available in the PSpice library:

- K_Linear in ANALOG.OLB
- KBREAK in BREAKOUT.OLB
- Cores in MAGNETIC.OLB, like X22_4C6

K_Linear part

For linear coupling, PSpice has a K_Linear symbol. The part is present in analog.olb.

Figure 5: K_Linear symbol

The default inductor coupling part, K_Linear, can be used to couple up to six independent inductors on a schematic.

	Klinear : PAGE1 : K3
Color	Default
COUPLING	1
Designator	
Graphic	K_Linear.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
L1	L3
L2	L4
L3	
L4	
L5	
L6	
Location X-Coordinate	525
Location Y-Coordinate	385
Name	INS43188
Part Reference	K3
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	Kn^@REFDES L^@L1 ?L2|L
Reference	K3
Source Library	C:\CADENCE\SPB_16.
Source Package	K_Linear
Source Part	K_Linear.Normal

Figure 6: Six inductors of a K_ Linear part

It provides linear coupling between the primary inductor/coil specified by the value L1 and the secondary coils specified by the values L2 to L6.

The number of secondary coils can be extended to include more inductors by editing the PSpiceTemplate manually.

To do that, go to **New Property** and write `L7` in the **Name** section. Click **Apply** and then click **OK**. `L7` is added as a new property as shown in Figure 6A:

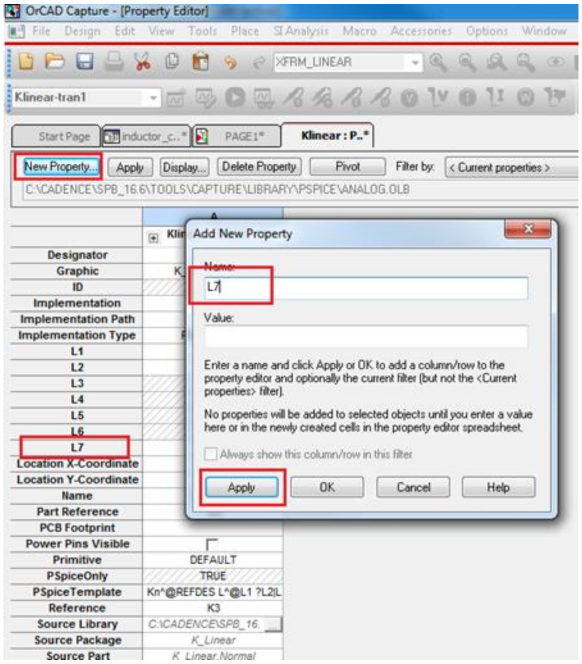


Figure 6A Add New Property

Next, go to PSpiceTemplate and add an entry for the additional inductor, say L7, as `?L7|L^@L7|`. This is shown in Figure 6B:

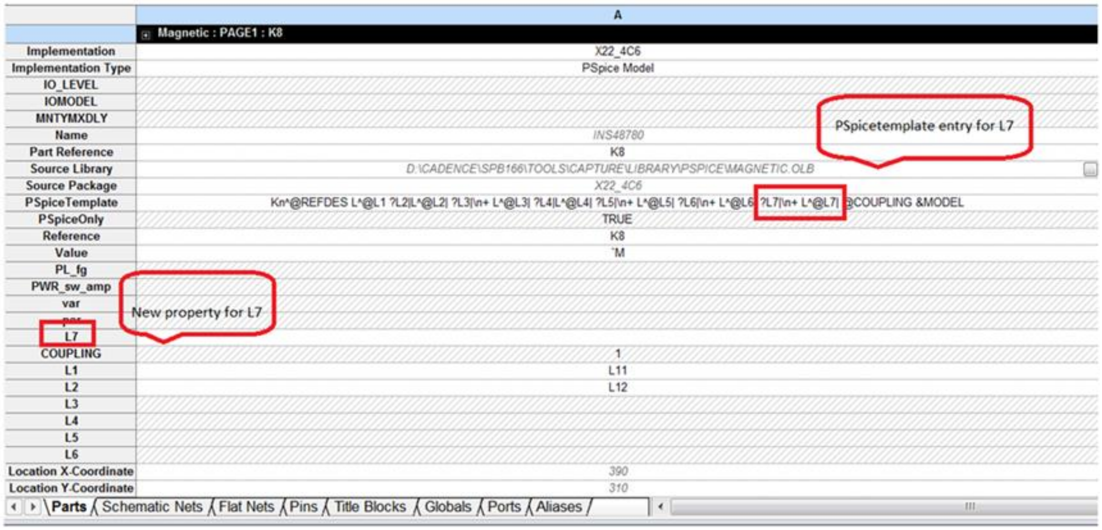


Figure 6B PSpice Template entry

Similarly, L8, L9 ……. Ln can be added. You need to add the new property and update PSpiceTemplate for each new inductor entry.

Note that with the K_Linear cores, the turn ratio of transformer windings is represented by the inductance of the primary and secondary coils.

For example, if the turn ratio of primary and secondary is n1:n2=1:2, it can be represented as Li:Lj=10mH:40mH, because the turn ratio is the square root of inductance, that is $n1/n2=\mathbf{(Li/Lj)^{1/2}}$.

The inductance value is in the mH range to accumulate sufficient magnetic energy that will be transferred to the secondary coil.

$V(OUT)=V(IN)*(n2/n1)=V(IN)*\mathbf{(Lj/Li)^{1/2}}$ (Coupling k is considered as 1)

n1=Primary turn

n2=Secondary turn

Li=Primary inductance

Lj=Secondary inductance

V(OUT)=Voltage induced at secondary coil

V(IN)=Voltage at primary coil

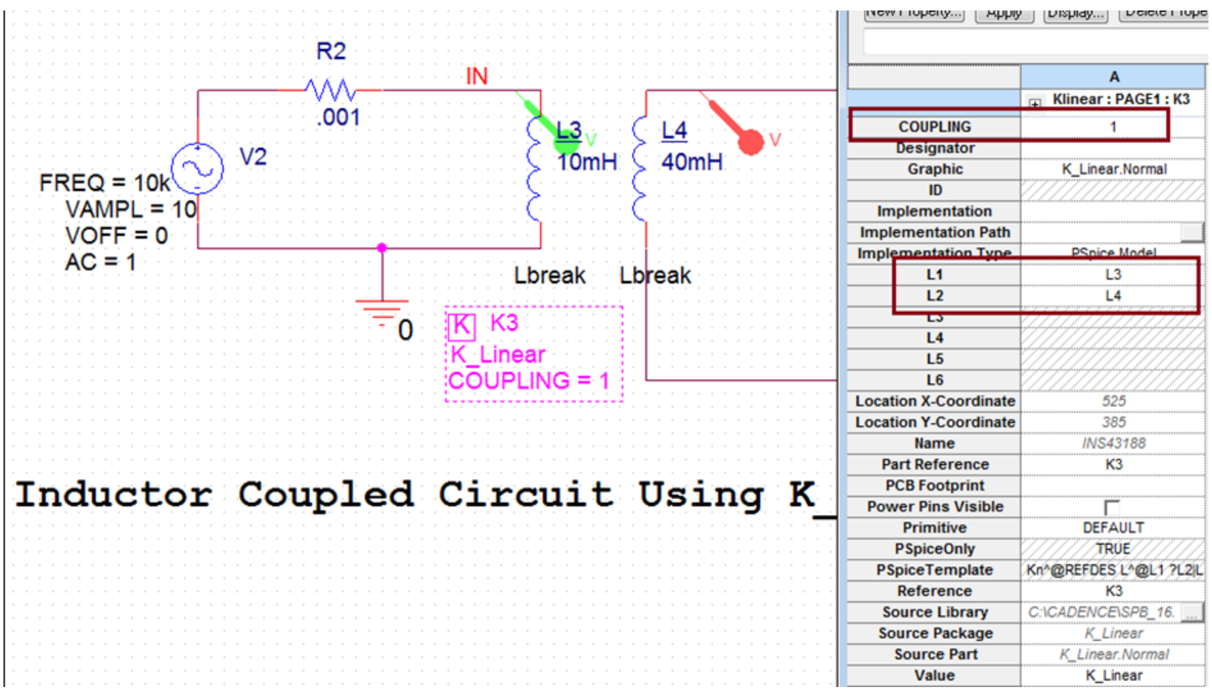


Figure 7: Property editor for K_Linear

Here, the primary coil is L3 and there is one secondary coil, L4.

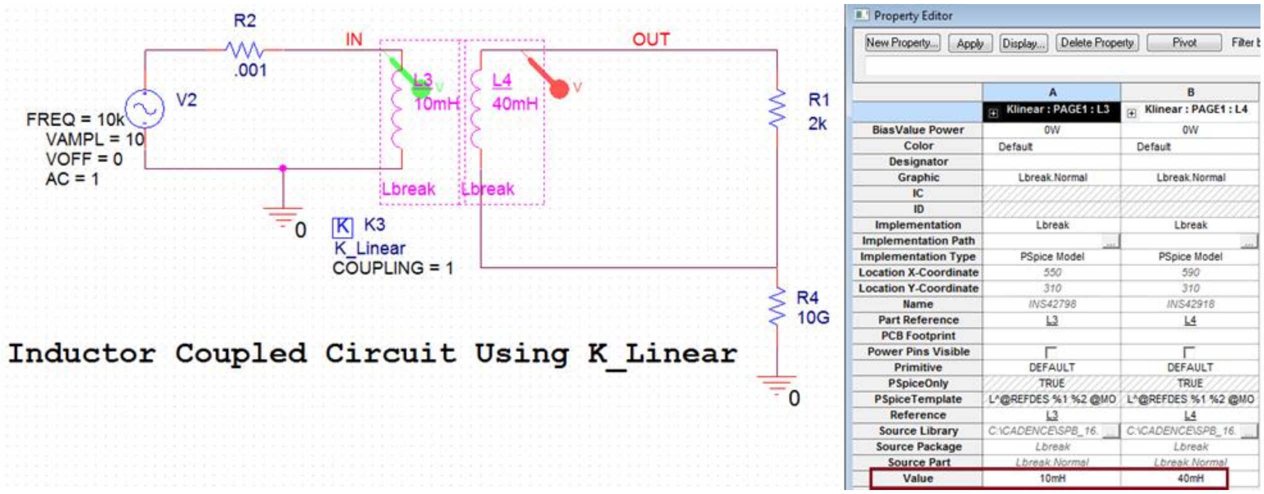


Figure 8: Inductor properties when used with K_Linear

Design and simulation of a transformer using a K_Linear part

Inductor_coupling.dsn has three circuits with the names Klinear, Kbreak, and Magnetic.

For the Klinear transformer, the "K_Linear" symbol is available at <SPB_installation_path>\tools\capture\library\pspice\analog.olb:

1. Open the **PAGE1** design Klinear:

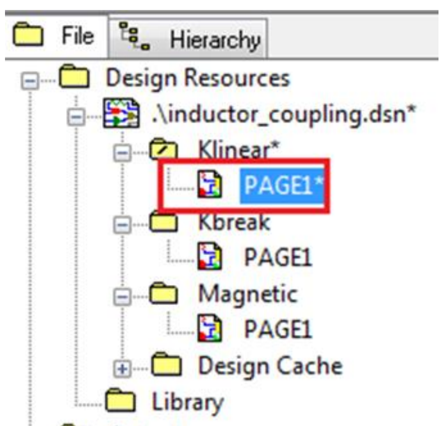


Figure 9: Klinear design at PAGE1

2. Ensure that the active simulation profile is "Klinear-tran1". If not, then make it active by right-clicking the profile name and selecting the **Make Active** option. The active profile is shown in red.

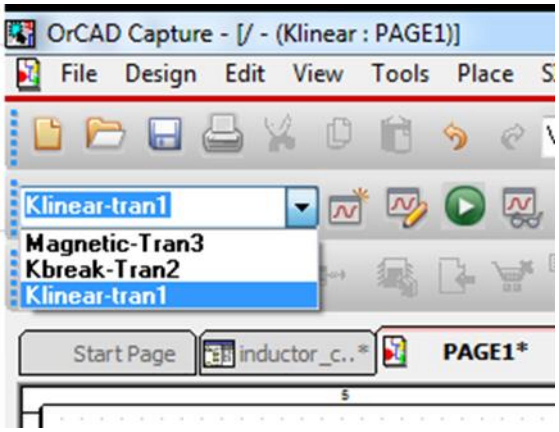


Figure 10: Simulation profile Klinear-tran1

3. For this profile, the schematic is Klinear. When you activate the profile, it must automatically make "Klinear" the root schematic.

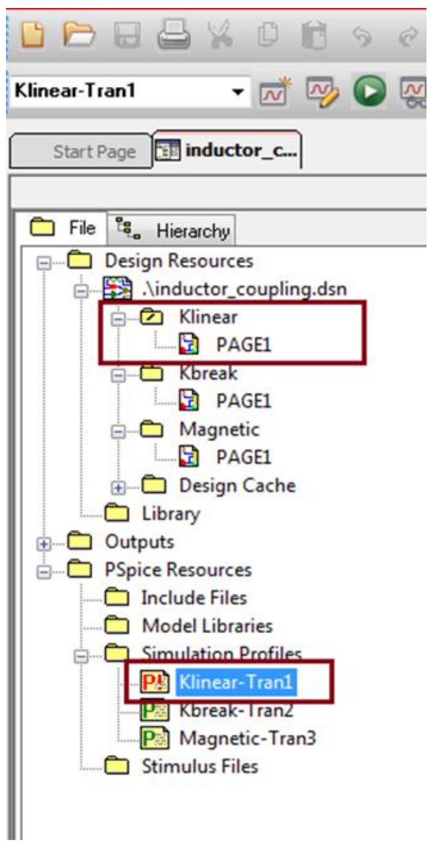


Figure 11: Project Manager view with the K_Linear profile active

4. Opening Page1 of the Klinear schematic shows the following circuit:

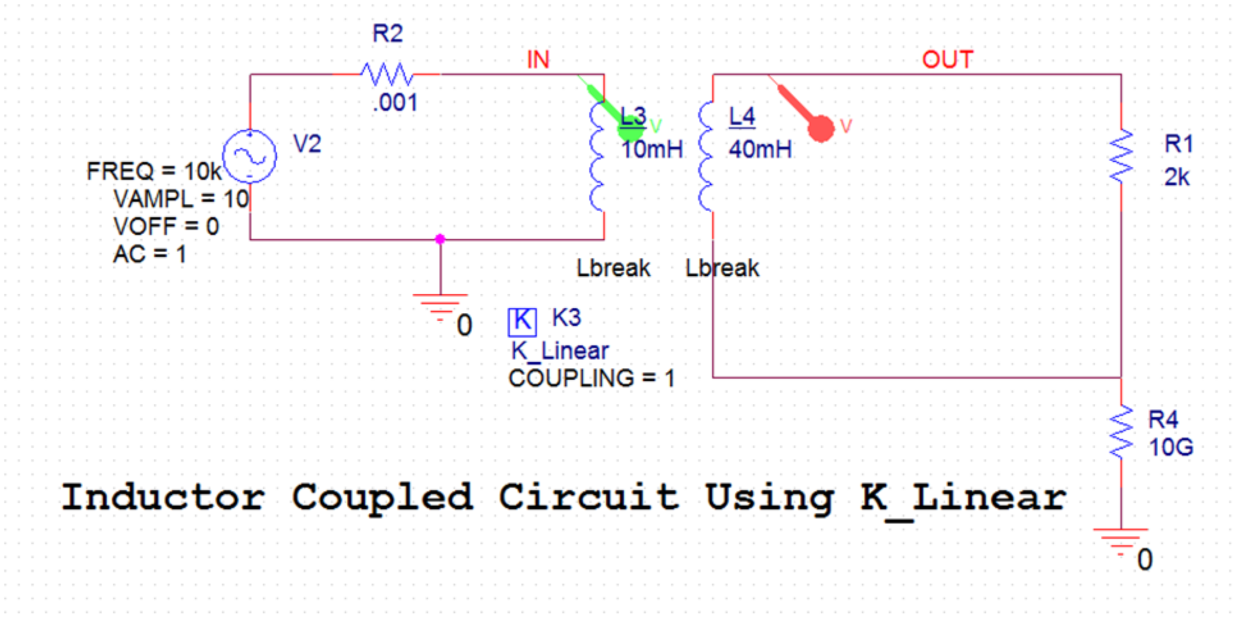


Figure 12: Transformer circuit using a K_Linear part

The above circuit is for a transformer using two inductors and a K_Linear linear coupling symbol. Inductors L1 and L2 behave as the primary and secondary coils of the transformer, respectively.

Turn ratio n2/n1=$\mathbf{(L4/L3)^{1/2}}$ =$(40/10)^{1/2}$ =2

Voltage across the secondary coil will be:

V(OUT)=V(IN)* $\mathbf{(L4/L3)^{1/2}}$
V(OUT)=V(IN)*$(40/10)^{1/2}$
V(OUT)=V(IN)*2=10*2=20Volt

5. Simulation of the design shows V(OUT) as 20V.

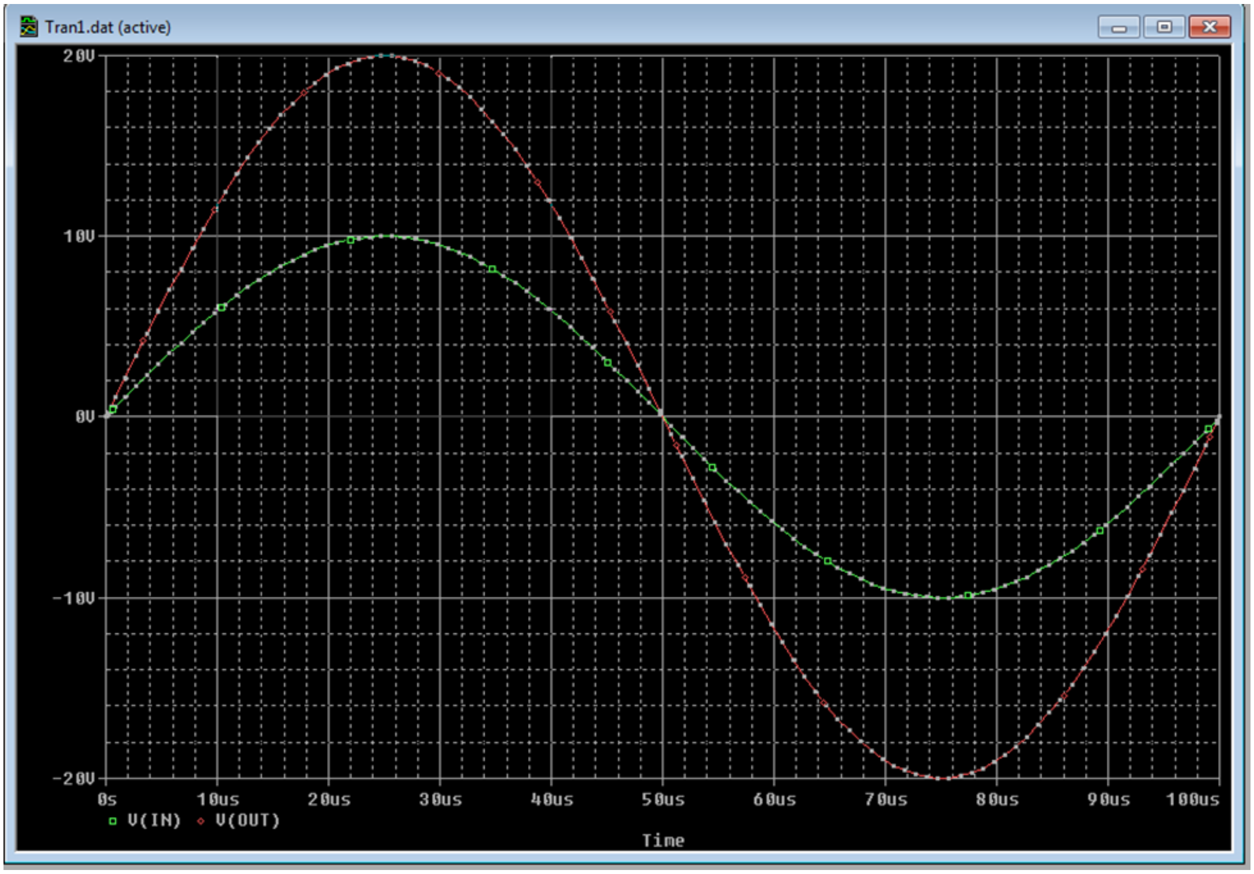


Figure 13: Simulation result

Effect of the turn ratio in a transformer

1. Open inductor_coupling.dsn.
2. Double click L4 to open the property editor.
3. Change the value to 90mH. The turn ratio n2/n1=$\mathbf{(90/10)}^{1/2}$ =3.

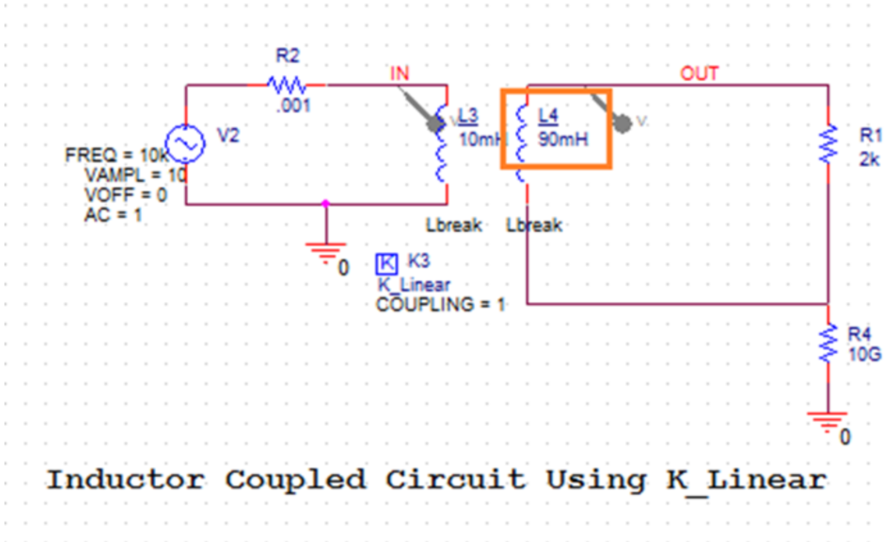


Figure 14: Increased secondary turn

4. Now, V2=10*3=30V.

5. Simulate the circuit to get the following result:

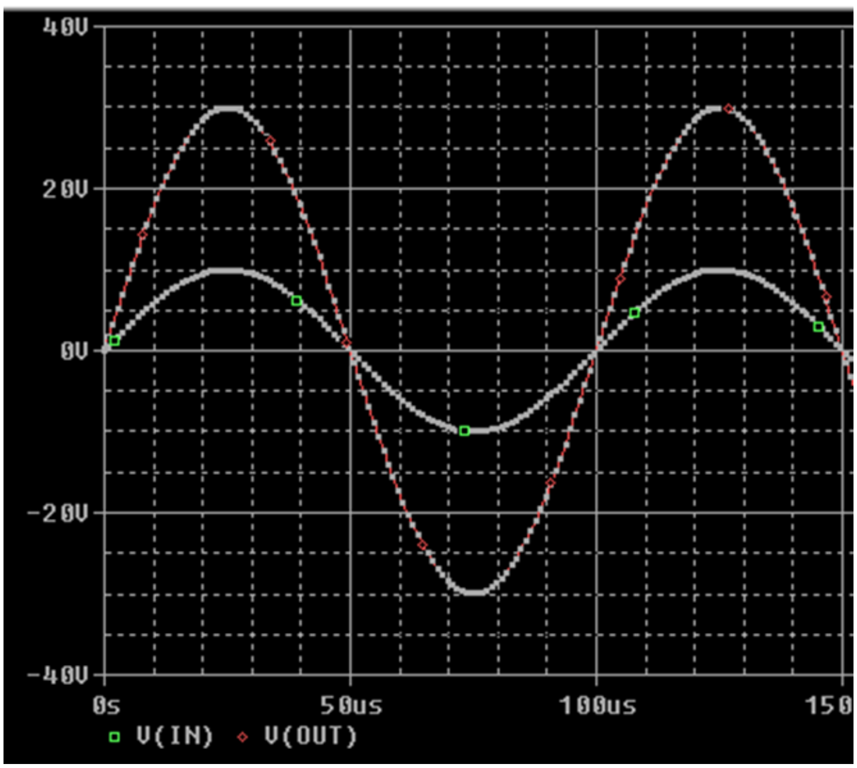


Figure 15: Simulation results

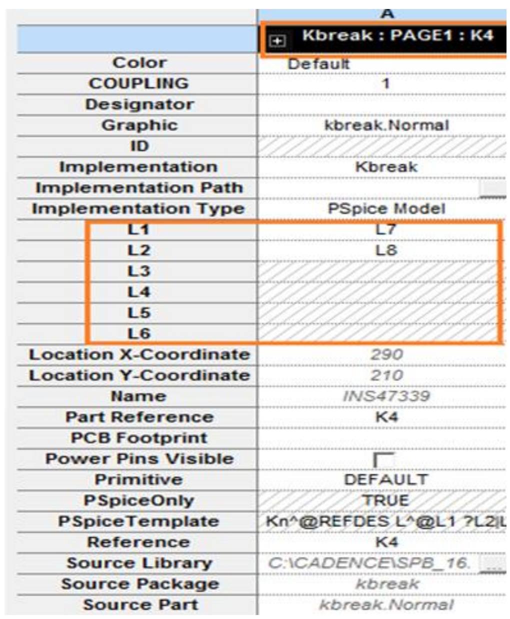
Here, V (OUT) is increased to 30V.

KBREAK part

The KBREAK part is present in breakout.olb:

Figure 16: Kbreak symbol

The KBREAK inductor coupling part, can be used to couple up to six independent inductors on a schematic. It provides linear coupling between the primary inductor/coil specified by the value of L1 and the secondary coils specified by the values of L2 to L6.



	A
	Kbreak : PAGE1 : K4
Color	Default
COUPLING	1
Designator	
Graphic	kbreak.Normal
ID	
Implementation	Kbreak
Implementation Path	
Implementation Type	PSpice Model
L1	L7
L2	L8
L3	
L4	
L5	
L6	
Location X-Coordinate	290
Location Y-Coordinate	210
Name	INS47339
Part Reference	K4
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	Kn^@REFDES L^@L1 ?L2|L
Reference	K4
Source Library	C:\CADENCE\SPB_16.
Source Package	kbreak
Source Part	kbreak.Normal

Figure 17: Primary and secondary inductors of Kbreak

Note that with the KBREAK cores, the turn ratio of the transformer windings can be represented directly by the primary and secondary number of turns.

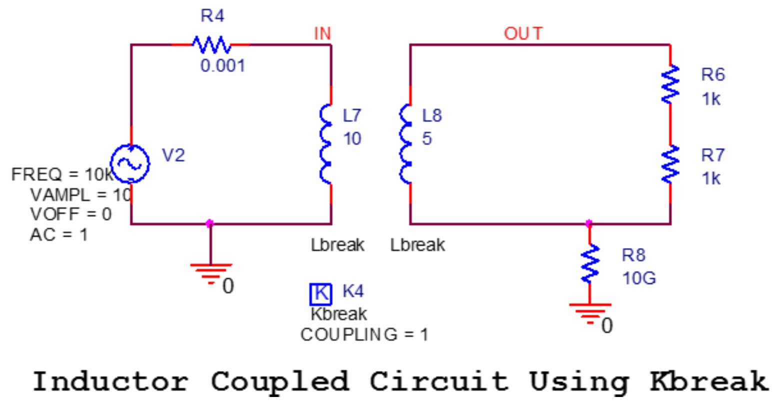
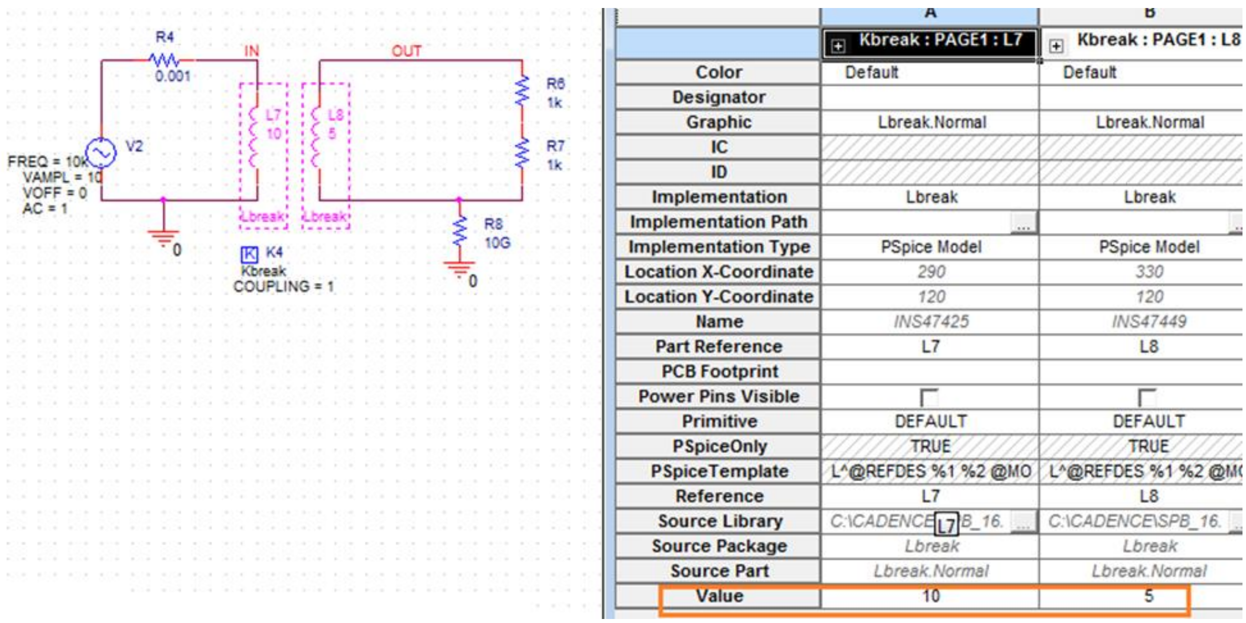


Figure 18: Transformer designed using Kbreak

The above transformer is designed using the Kbreak coupling symbol and the inductors L7 and L8. **The number of the primary turn is 10 and the secondary turn is 5**, which is directly mentioned as a value without any unit. This gives a turn ratio of 2:1.



	Kbreak : PAGE1 : L7	Kbreak : PAGE1 : L8
Color	Default	Default
Designator		
Graphic	Lbreak.Normal	Lbreak.Normal
IC		
ID		
Implementation	Lbreak	Lbreak
Implementation Path		
Implementation Type	PSpice Model	PSpice Model
Location X-Coordinate	290	330
Location Y-Coordinate	120	120
Name	INS47425	INS47449
Part Reference	L7	L8
PCB Footprint		
Power Pins Visible		
Primitive	DEFAULT	DEFAULT
PSpiceOnly	TRUE	TRUE
PSpiceTemplate	L^@REFDES %1 %2 @MO	L^@REFDES %1 %2 @M
Reference	L7	L8
Source Library	C:\CADENCE\SPB_16.	C:\CADENCE\SPB_16.
Source Package	Lbreak	Lbreak
Source Part	Lbreak.Normal	Lbreak.Normal
Value	10	5

Figure 19: Property editor showing the inductor properties when used with a Kbreak part

For example, if the turn ratio of a primary and secondary is n1:n2=2:1, it can be represented as L7:L8=10:5. The number of turns might be more to get sufficient self-inductance that results in enough magnetic energy to get a good waveform.

V(OUT)=V(IN)*(n2/n1)=V(IN)*(L8/L7)=V(IN)*1/2 (Coupling k is considered as 1)

V(OUT)=Voltage induced at secondary coil

V(IN)=Voltage at primary coil

Design and simulation of a transformer using a KBREAK part

A generic symbol, ”Kbreak”, is available at <SPB_installation_path>\tools\capture\library\pspice\breakout.olb.olb.

1. Open Coupling_inductor.dsn.
2. Make the “Kbreak-tran2” simulation profile active.
3. The circuit for Kbreak is as shown here:

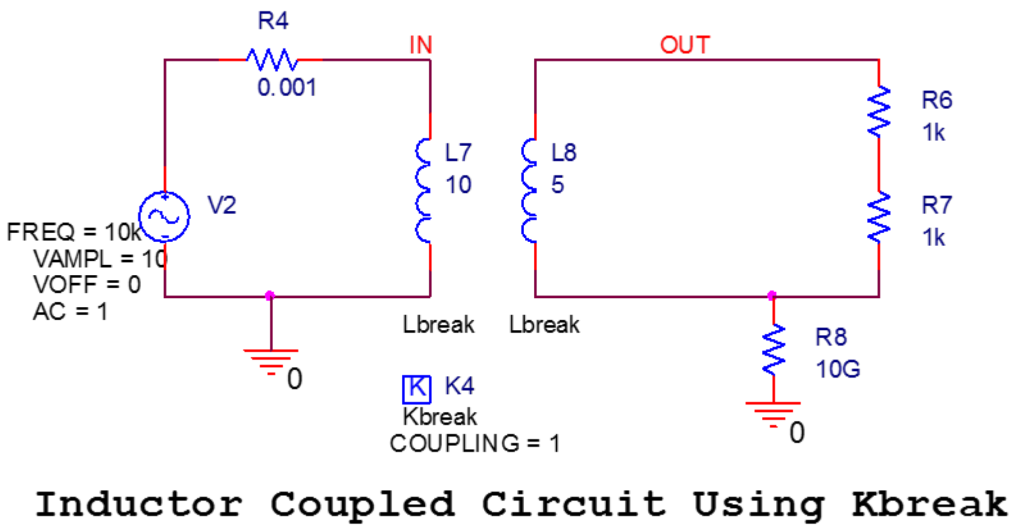


Figure 20: Transformer circuit using a Kbreak part

4. Simulate the design and get the following result:

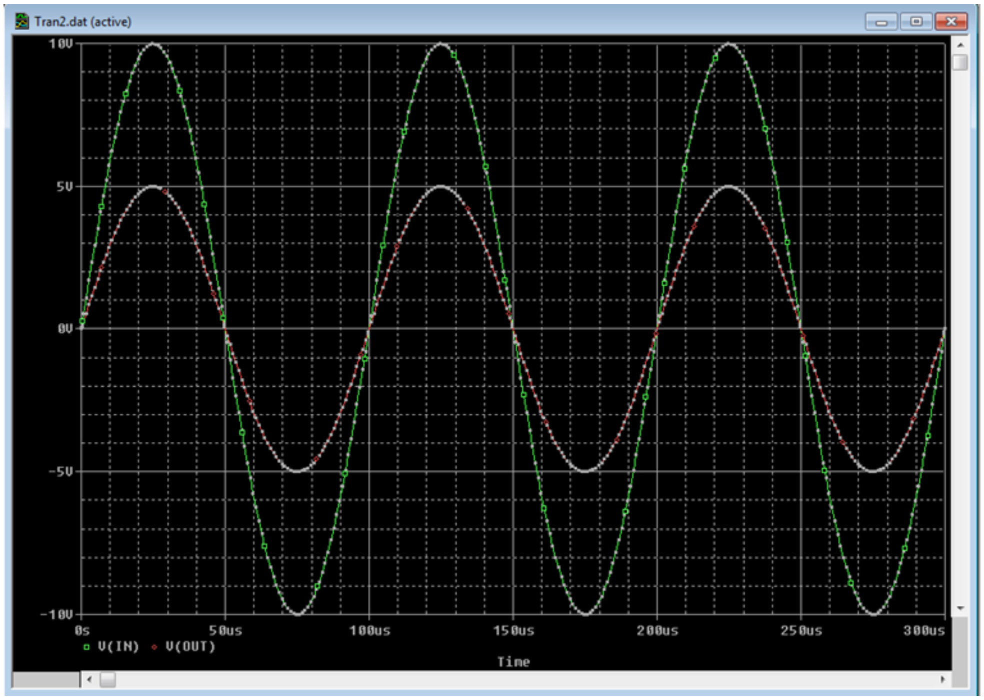


Figure 21: Simulation result

The V(OUT) peak output voltage is 5V. This can be calculated as follows:

The transformer equation is:

n1V2=n2V1; V2=V1*(n2/n1)

In this case:

n1=Primary coil number of turns=10
n2=Secondary coil number of turns=5
So, the turn ratio n2/n1=1/2
V1=Input voltage=10V (peak input voltage (V (IN))
V2=Output voltage

So, V2=10*(1/2)=5Volt

Effect of the turn ratio in a transformer

1. Open inductor_coupling.dsn.
2. Double click L8 to open the property editor.
3. Change the value to 20. The turn ratio n2/n1=2.

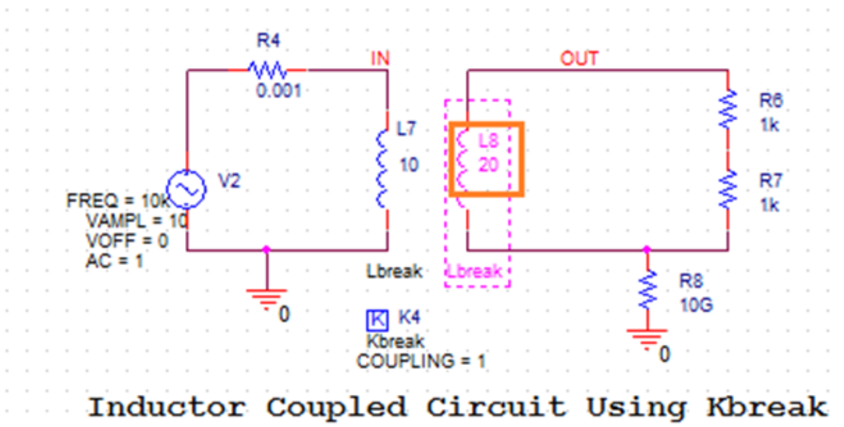


Figure 22: Increased secondary turn

4. Now, V2=10*2=20V.
5. Simulate the circuit to get the following result:

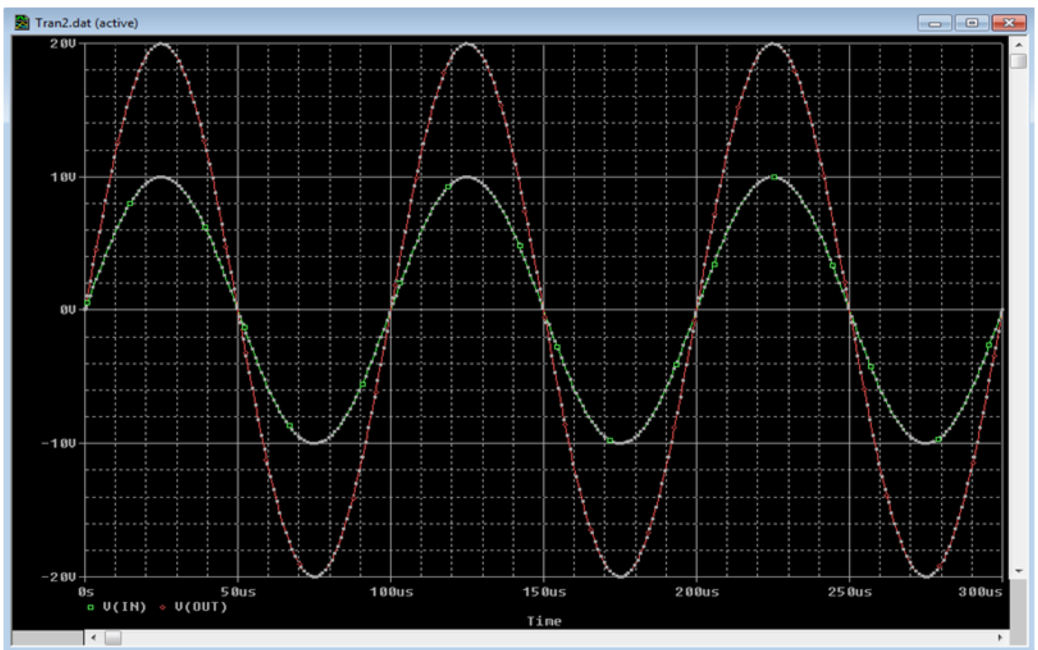


Figure 23: Simulation result

Here, V (OUT)=20Volt.

Magnetic library part

The magnetic.olb Symbol Library file, contains one symbol for each magnetic core model in the magnetic.lib model library file. For example, for the X22_4C6 core, the symbol is shown in the following image:

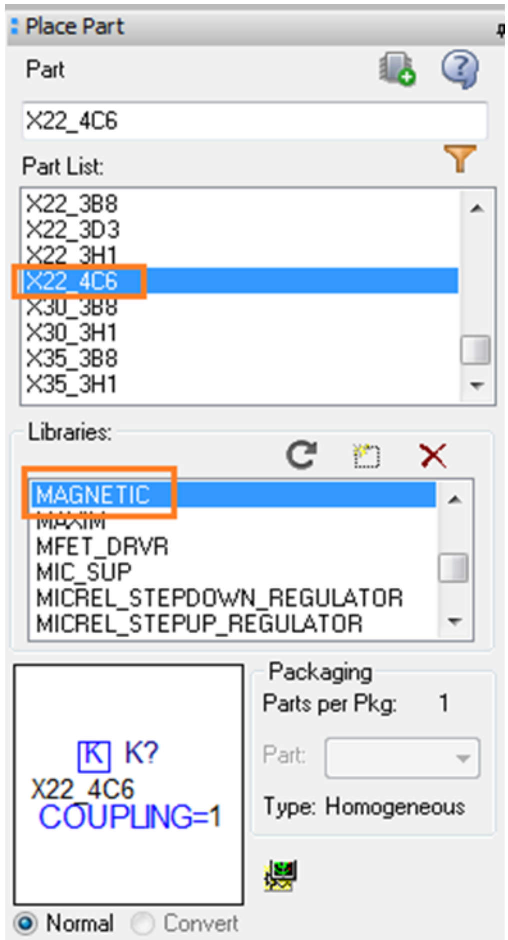


Figure 24: X22_4C6 part in the PSpice library

The reference designators are assigned automatically, but may be changed by double-clicking them.

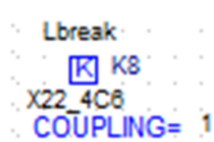


Figure 25: X22_4C6 symbol

The inductor coupling symbols can couple up to six independent inductors on a schematic, L1 to L6; where L1 is the primary coil and L2 to L6 are the secondary coils.

	A
	Magnetic : PAGE1 : K8
Color	Default
COUPLING	1
Designator	
Graphic	X22_4C6.Normal
Implementation	X22_4C6
Implementation Path	
Implementation Type	PSpice Model
L1	L9
L2	L10
L3	
L4	
L5	
L6	
Location X-Coordinate	390
Location Y-Coordinate	310
Name	INS48780
Part Reference	K8
PCB Footprint	
Power Pins Visible	☐
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	Kn^@REFDES L^@L1 ?L2|L
Reference	K8
Source Library	D:\CADENCE\SPB166\
Source Package	X22_4C6
Source Part	X22_4C6.Normal
Value	`M

Figure 26: Property editor for a magnetic coupling symbol

Design and simulation of a transformer using a magnetic part

A generic symbol, ”X22_4C6”, is available at <SPB_installation_path>\tools\capture\library\pspice\magnetic.olb

1. Open Coupling_inductor.dsn.
2. Make the ”Magnetic-Tran3” simulation profile active.
3. The circuit for X22_4C6 is as follows:

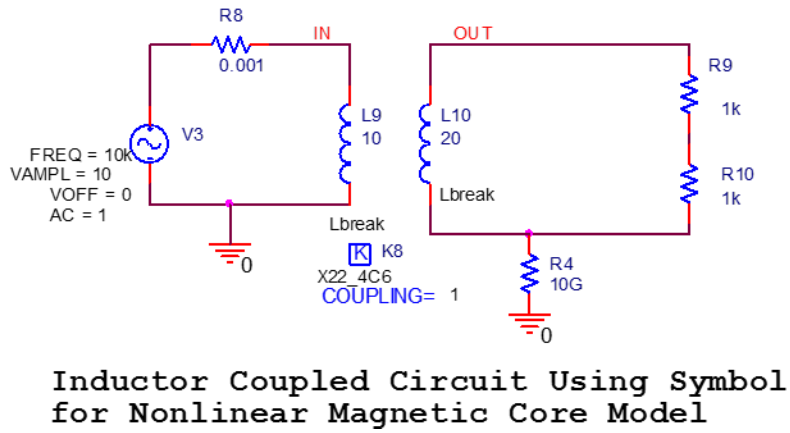


Figure 27: Transformer circuit using a Kbreak part

4. Simulate the design and get the following result:

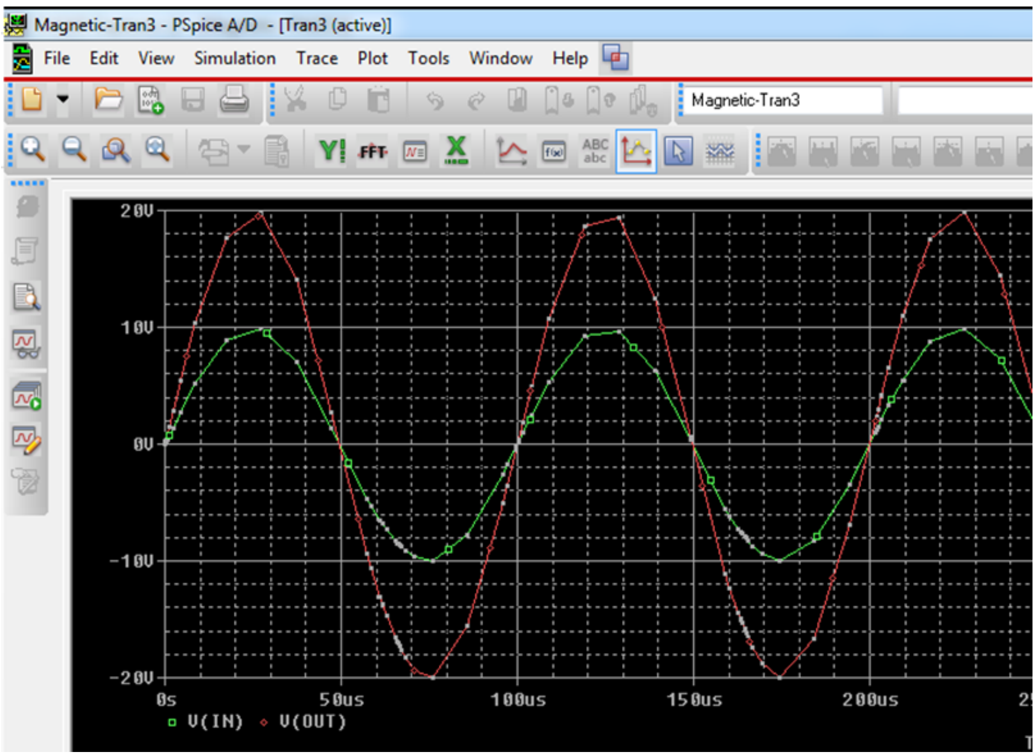


Figure 28: Simulation result

Here, the V(OUT) peak output voltage is 20V. This can be calculated as follows:

The transformer equation is:

n1V2=n2V1; V2=V1*(n2/n1)

In this case:

n1=Primary coil number of turns=10
n2=Secondary coil number of turns=20
So, the turn ratio n2/n1=2
V1=Input voltage=10V (peak input voltage (V (IN))
V2=Output voltage

So, V2=10*2=20Volt

5. If the secondary turns are increased, output voltage will increase.

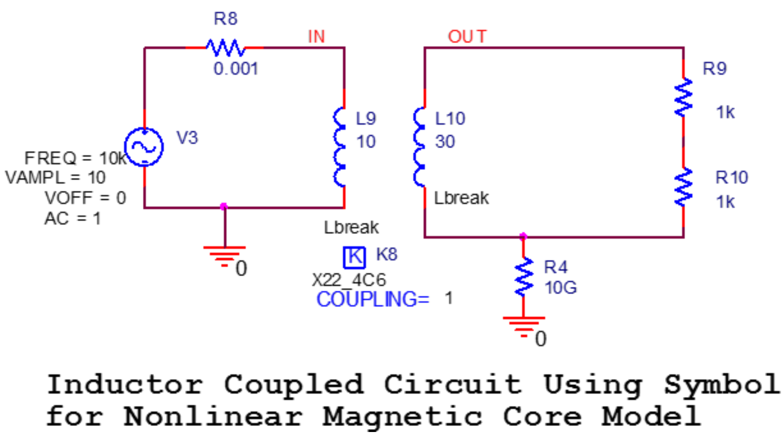


Figure 29: Increased the secondary turn to 30

For n2=30, V (OUT)=3*V (IN)=30V.

The following simulation result shows 30Volt at V (OUT):

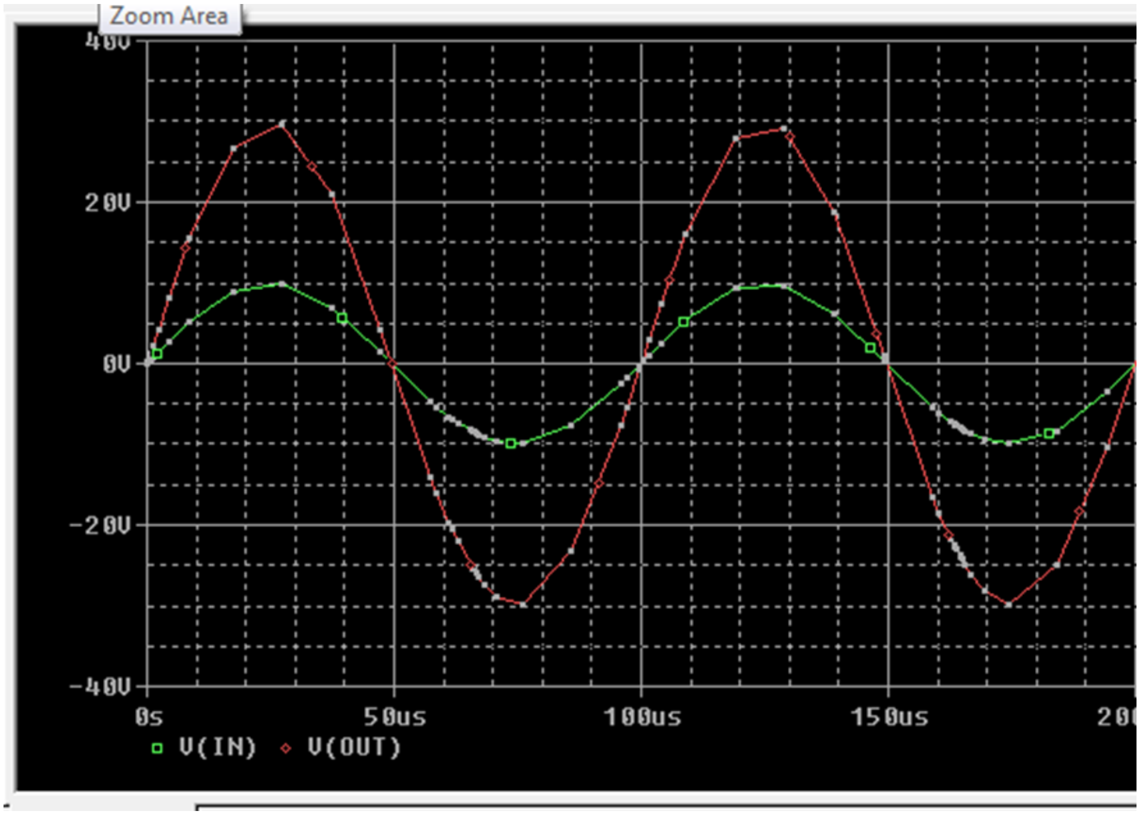


Figure 30: Simulation results

Similarly, the other circuits can be designed using other magnetic core models, like E13_6_6_3c81 and so on, and inductors L9 and L10.

Designing a transformer using the PSpice modelling application

Another easy way to design a transformer in PSpice is to use the PSpice Modelling application. A "how to do it" video, is available in the following Cadence link:

https://support.cadence.com/apex/ArticleAttachmentPortal?id=a1Od0000005xG6DEAU&pageName=ArticleContent&sq=005d0000001T5ZYAA0_20174208444722

Summary

The three inductor coupling parts described earlier can be used to couple up to six independent inductors on a schematic. In the magnetic part, there are more practical coupling symbols like E13_6_6_3C81, ER42_3C85, ETD29_3C30, I93_28_16_3C85, K135T050_3E2A, K1811PL_3D3, TX63_38_25_3E6, U20_16_7_3E25, X30_3B8, and so on available in the library.

The difference between K_Linear, Kbreak, and magnetic part symbols is that K_Linear uses the inductance values to calculate the turn ratio (where L α N2), but Kbreak and the other magnetic part symbols use the turn ratio directly.

Support

Cadence Online Support provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit *https://support.cadence.com.*

Feedback

Email comments, questions, and suggestions to **content_feedback@cadence.com.**