Inductor Coupling in PSpice

Product Version SPB16.6
July 18, 2015

Contents

Purpose

The purpose of this application note is to know how to use inductor coupling symbol to couple many inductors in PSpice and do the simulation.

Audience

The document is for electronics and telecommunication engineers, power electronics engineers, electronic circuit and magnetic design engineers, as well as for university teachers and students.

Prerequisite

Basic knowledge of inductors, inductor coupling, and electromagnetic theory.
Knowledge of PSpice is must.

Download

References can be found at ‘Attachments’ sections below the PDF. This pdf can be searched with the document title on https://support.cadence.com

Basic Concept on Inductor Coupling

When current flows through a wire and induces electrical current in another wire due to electromagnetic field, it is said that both wires are inductively coupled. The phenomenon of one inductor inducing a voltage in another inductor is known as mutual inductance.

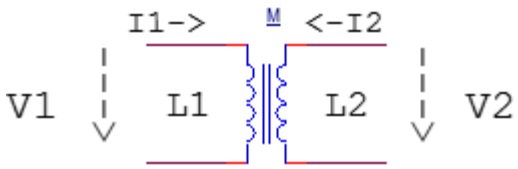


Figure 1 Coupled Inductors

Coupled inductors are characterized by three parameter; self-inductances of two inductors, L1 and L2, and the mutual inductance, M.

For the circuit shown in Figure 1, voltage across each inductor is calculated as:

$$V_1 = L1\frac{dI_1}{dt} + M\frac{dI_2}{dt}$$

$$V_2 = L2\frac{dI_2}{dt} + M\frac{dI_1}{dt}$$

The mutual inductance between two coils is calculated as:

$$M = \sqrt{L_1 \times L_2}$$

However, this formula is based on the assumption that a perfect flux linkage exists between the two inductors. In real life, not all flux lines cut through the coupled inductor, thus affecting the mutual inductance value. Therefore, for non-ideal scenarios, mutual inductance is calculated as:

$$M = k\sqrt{L_1 \times L_2}$$

with $0 \leq k \leq 1$ and k is referred to as *coupling coefficient.*

Inductive coupling is commonly used in transformers, electric motors, radio transmission and reception, non-contact charging devices, and so on.

Inductor Coupling Symbols in PSpice Library

Inductor coupling symbols are available in `analog.olb`, `breakout.olb` and `magnetic.olb`, those are at `<SPB_install>\tools\capture\library\pspice`. These coupling symbols are `K_Linear` for `analog.olb`, and `Kbreak` for `breakout.olb`. In case of `magnetic.olb`, symbols for nonlinear magnetic core models are named as per the core specifications, such as `3_6_6_3C81`, `ER42_3C85`, `I93_28_16_3C85`, `K135T050_3E2A`, `TX63_38_25_3E6`, and so on.

The `K_Linear` symbol is used for specifying linear coupling, between inductors. `Kbreak` is a generic symbol used for specifying arbitrary nonlinear magnetic core models, that has a pre-assigned model attribute.

While `K_Linear` uses inductance values to calculate the turn ratio (where $N \alpha L^2$), `Kbreak` and other coupling symbols use transformer turn ratio directly.

Use of Analog and Breakout Library Coupling Symbol

User can Place coupling symbol, Kbreak or K_Linear, anywhere on the schematic, for each group of coupled inductors. These symbols have no pins; they are represented by the letter K enclosed in a box.

Double-clicking on each coupling symbol (on the K-in-a-box, not the attributes) and enter the reference designators for the coupled inductors as the values for L_i (i=1,2,...,6).
Set the value of the COUPLING attribute to the value of the coupling coefficient, K.

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	310
Location Y-Coordinate	100
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
Value	K_Linear

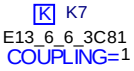
Figure 2 Edit Properties window for K_linear

	A
	SCHEMATIC1 : PAGE1
COUPLING	0.99
Designator	
Graphic	kbreak.Normal
ID	
Implementation	Kbreak
Implementation Path	
Implementation Type	PSpice Model
L1	L7
L2	L8
L3	
L4	
L5	
L6	
Location X-Coordinate	740
Location Y-Coordinate	130
Name	INS43490
Part Reference	K4
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	K^@REFDES L^@L1 ?L2|L
Reference	K4
Source Library	C:\CADENCE\SPB_16.
Source Package	kbreak
Source Part	kbreak.Normal
Value	kbreak

Figure 3 Edit Properties window for Kbreak

Magnetic Library Coupling Symbol Use

This is another inductor coupling symbols that can be used to couple up to six independent inductors on a schematic. The symbol library, “magnetic.olb”, contains one symbol for each nonlinear magnetic core model in the model library file, `magnetic.lib`. For example, for core E13_6_6_3C81 the symbol is:



To use a magnetic coupling symbol, Select it for a desired CORE model and place one coupling symbol, anywhere on the schematic, for each group of coupled

inductors. These symbols have no pins; they are represented by the letter K enclosed in a box.

Double-click on each coupling symbol (on the K-in-a-box) and enter the reference designators for the coupled inductors as the values for L_i (i=1,2,...,6).
Set the value of the COUPLING attribute to the value of the coupling factor, K.

	A
	SCHEMATIC1 : PAGE1
COUPLING	1
Designator	
Graphic	E13_6_6_3C81.Normal
ID	
Implementation	E13_6_6_3C81
Implementation Path	
Implementation Type	PSpice Model
L1	
L2	
L3	
L4	
L5	
L6	
Location X-Coordinate	730
Location Y-Coordinate	570
Name	INS46598
Part Reference	K7
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	Kn^@REFDES L^@L1 ?L2 L
Reference	K7
Source Library	C:\CADENCE\SPB_16.
Source Package	E13_6_6_3C81
Source Part	E13_6_6_3C81.Normal
Value	`M

Figure 4 Edit Properties window for Magnetic Coupling symbols

A Sample Circuit and Simulation Waveforms

Sample circuit shown in Figure 5 is designed in OrCAD Capture uses K_linear symbol.

Using K_Linear:

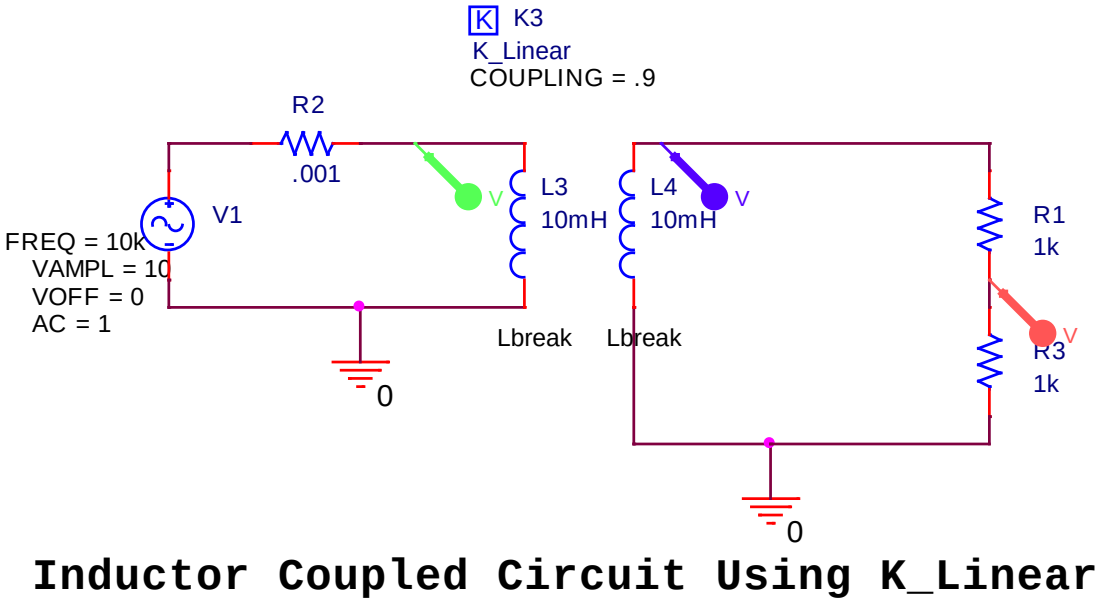


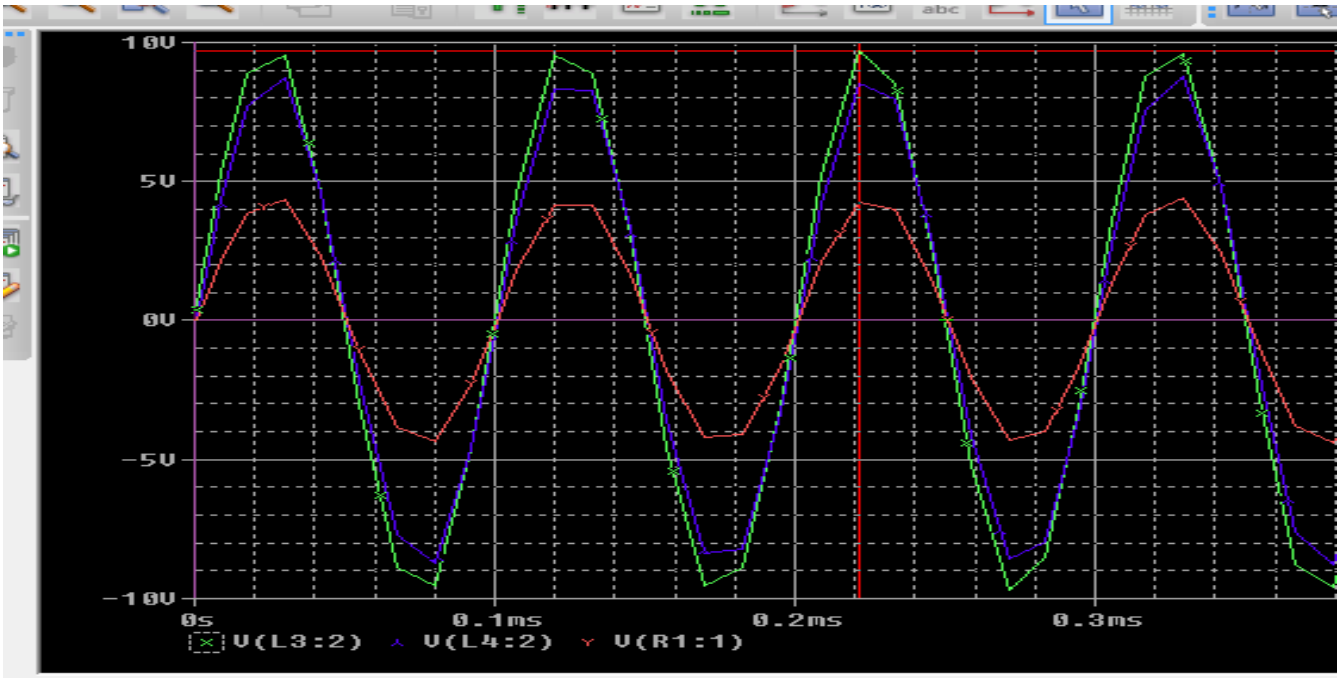
Figure 5

In the above figure, a transformer is designed using two inductors and an inductor coupling symbol, K_Linear. Inductors L3 and L4 are the primary and secondary coil of the transformer. The coupling coefficient is 0.9, which is a practical condition when there is a 90% coupling between two inductors.

The turn ratio is considered as 1:1 for simplicity of the calculation. For an input signal of 10V, there is a negligible voltage drop across R2. The secondary voltage drop at L4 is slightly less than primary voltage drop due to non-ideality and the voltage drop across R3 is half of the secondary voltage drop.

When you simulate the circuit in PSpice, following waveform is visible at the probe window.

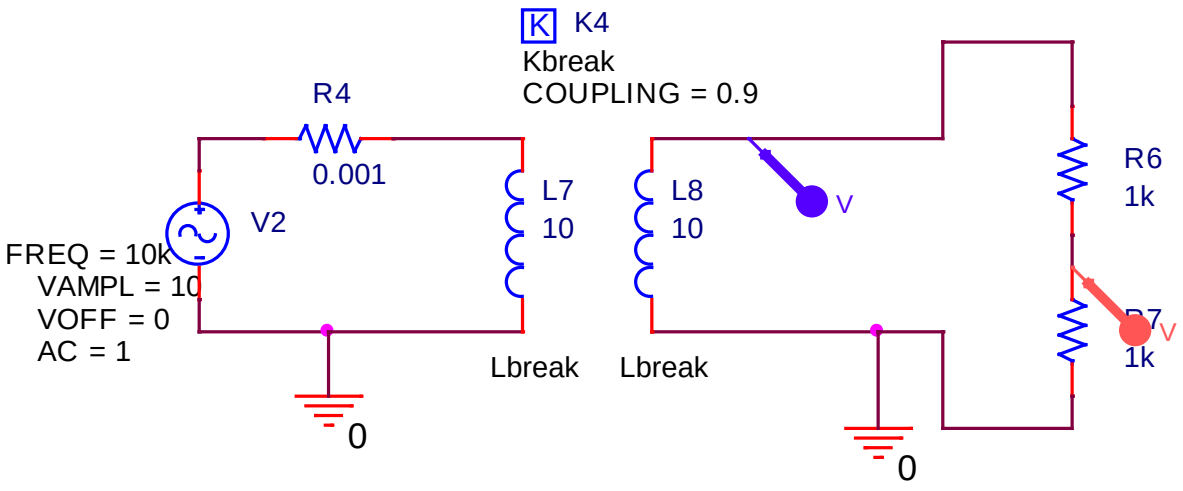
Inductor Coupling in PSpice



	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2
		X Values	221.198u	0.000	221.198u		Y1 - Y1(Cursor1)	Y2 - Y2(Cur
	CURSOR 1,2	V(L3:2)	9.716	0.000	9.716		0.000	0.000
		V(L4:2)	8.5472	8.2192m	8.5390		-1.1688	8.2192m
		V(R1:1)	4.2736	4.1096m	4.2695		-5.4424	4.1096m

It is seen that primary voltage drop is 9.716 V, secondary voltage drop is 8.5472 V, and voltage across R3 is half of the secondary voltage that is 4.2736 V The simulation results and waveforms are as expected.

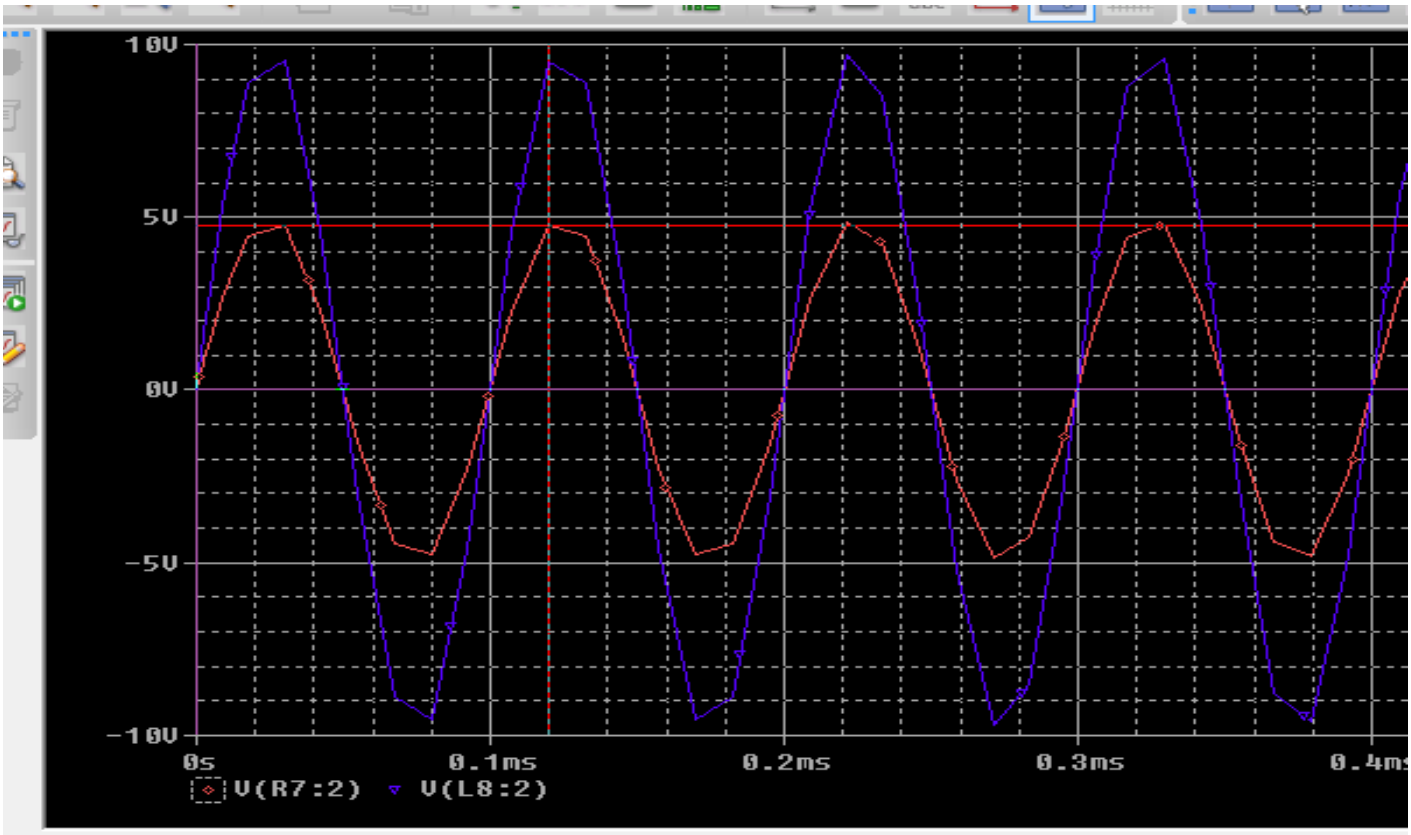
Using KBreak:



Inductor Coupled Circuit Using Kbreak

The above transformer is designed using inductor symbol Kbreak and inductor L7 and L8. The number of primary and secondary turns are 10, which is directly mentioned without any unit, resulting a turn ratio of 1:1

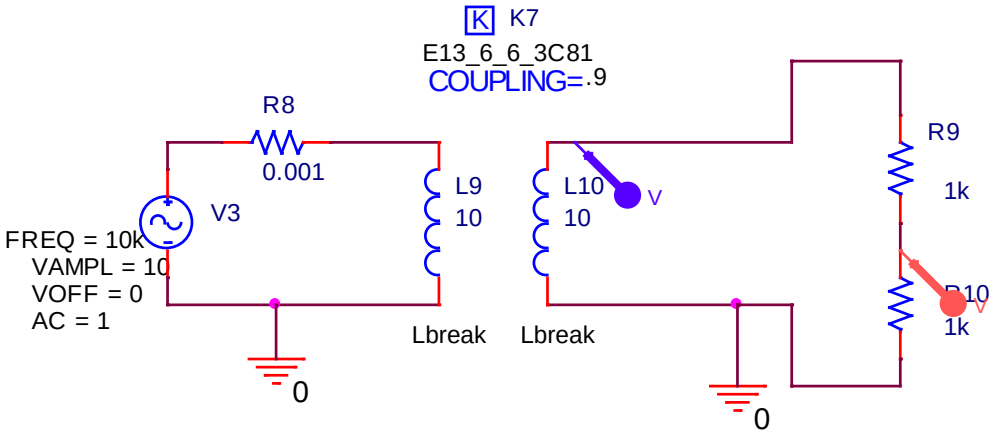
When circuit is simulated, following results are available in the PSpice probe window.



	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)	
		X Values	120.037u	125.000n	119.912u		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)
	CURSOR 1,2	V(R7:2)	4.7560	39.269m	4.7168		0.000	0.000
		V(L8:2)	9.4402	25.445m	9.4148		4.6842	-13.824m

Voltage drop at secondary L8 is 9.44 Volt and across load R7 it is 4.756 Volt. The output voltage and waveforms are as expected.

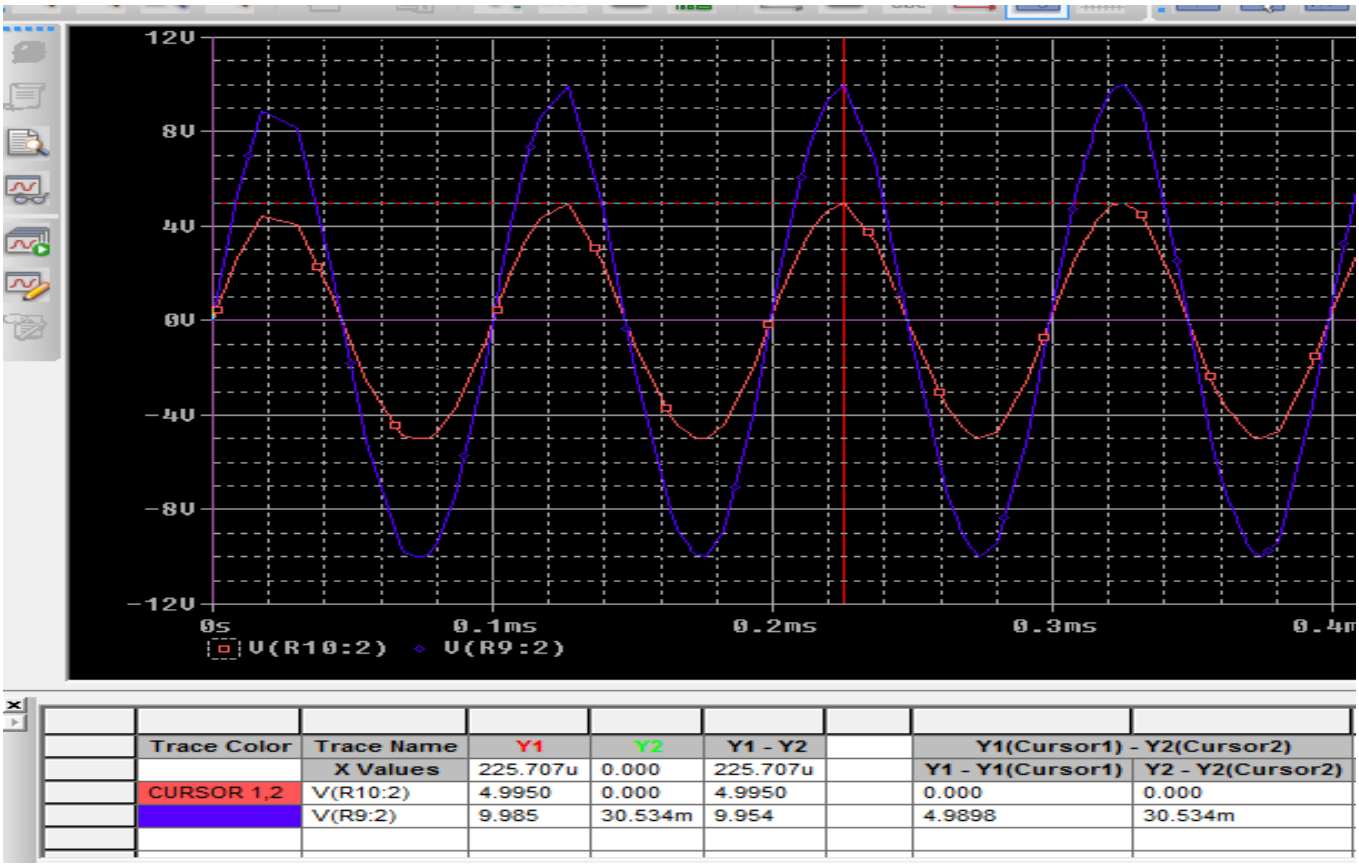
Using Nonlinear Magnetic Core:



Inductor Coupled Circuit Using Symbol for Nonlinear Magnetic Core Model

Similarly, another circuit is designed using nonlinear magnetic core model E13_6_6_3c81, and inductors L9 and L10. The number of primary and secondary turns are 10.

Following results are available in PSpice probe window when simulation is completed.



	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)	
		X Values	225.707u	0.000	225.707u		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)
	CURSOR 1,2	V(R10:2)	4.9950	0.000	4.9950		0.000	0.000
		V(R9:2)	9.985	30.534m	9.954		4.9898	30.534m

Here secondary voltage drop is 9.985 volt and voltage at load R10 is half of the secondary voltage 4.9 V. The waveforms and output voltages are as expected.

Summary

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