AN3017

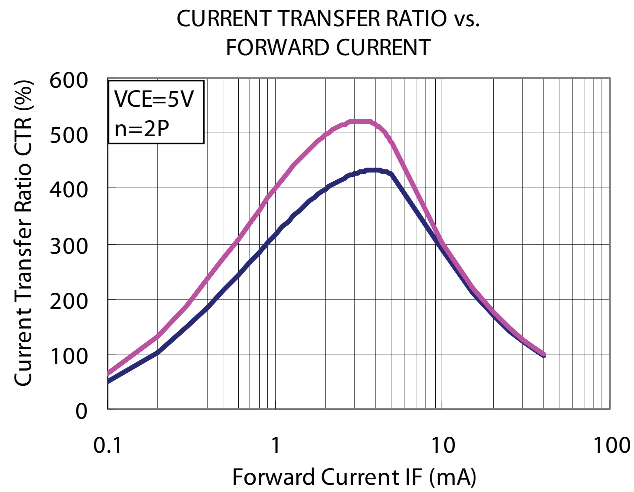
PSPICE Model for a Phototransistor

Authors: Van N. Tran — CEL Staff Application Engineer, CEL Opto Semiconductors
Larry Sisken — CEL Product Marketing Manager, CEL Opto Semiconductors
Wei Z. Jiang — Graduate Intern (MSEE), SJSU

Introduction

PSPICE is a circuit simulation program, used to provide a reasonably detailed analysis of circuits containing active components such as bipolar transistors, field effect transistors, diodes, opamps, and lumped components such as resistors, capacitors and inductors. Its measurement parameters include voltage and current. However, when it comes to model opto electronics components, the SPICE program does not possess the capability to evaluate or simulate opto electronics components with outputs measured in radiometric or photometric units like watt (w) or lumen (lm), or other variables like optical intensity, radiant power, irradiance with unit measurements in mW/sr, mW/m2, lumens. In many instances, the desired output will still end up with measurement units in voltage or current and this application note provides an example of a model of an optocoupler using the SPICE model.

CTR Characteristics



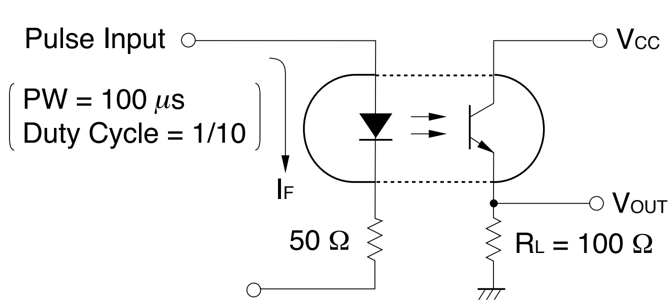
Note: TA=25°C, unless otherwise specified
This data is reference only, not guaranteed.

An Optocoupler Device

Typically, an optocoupler is an optically coupler isolator using a GaAs LED as a light source and a bipolar NPN phototransistor as a receiver. The forward current, If, through the LED emitter produces the photons that impinge on the active area of a phototransistor and its output, collector current, will be a product of If and Current Transfer Ratio (CTR). The PS2561F-1 will be used as an example for this application note.

From the PS2561F-1 data sheet, the CTR can vary from 300% to 600% at If = 5 mA and VCE = 5.0 V from device to device. Below is a summary of the PS2561F-1 electrical characteristics that can be found in the PS2561F-1 data sheet at www.cel.com for reference.

Test circuit for switching time

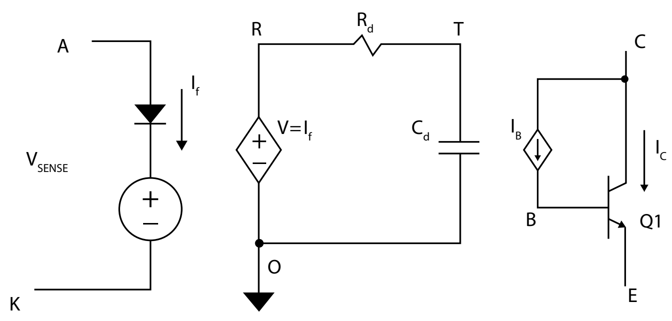


Electrical Characteristics (TA = 25˚C)

Parameter		Symbol	Conditions	MIN	TYP.	MAX.	Unit
Diode	Forward Voltage	V_F	I_F = 10 mA		1.2	1.4	V
	Reverse Current	I_R	V_R = 5 V			5	μA
	Terminal Capacitance	C_t	V = 0 V, f = 1.0 MHz		10		pF
Transistor	Collector to Emitter Dark Current	I_{CEO}	I_F = 0 mA, V_{CE} = 80 V			100	nA
Coupled	Current Transfer Ratio (IC/IF)*1	CTR	I_F = 5 mA, V_{CE} = 5 V	300	450	600	%
			I_F = 1 mA, V_{CE} = 5 V	60			
	Collector Saturation Voltage	$V_{CE\ (sat)}$	I_F = 10 mA, I_C = 2 mA			0.3	V
	Isolation Resistance	$R_{I\text{-}O}$	$V_{I\text{-}O}$ = 1.0 kV_{DC}	10^{11}			Ω
	Isolation Capacitance	$C_{I\text{-}O}$	V = 0 V, f = 1.0 MHz		0.5		pF
	Rise Time*2	t_r	V_{CC} = 10 V, I_C = 2 mA		5		μs
	Fall Time*2	t_f	R_L = 100 W		7		

An Optocoupler Circuit Model

Below is the schematic of the optocoupler with photo-transistor output used in this application note for PSPICE modeling.



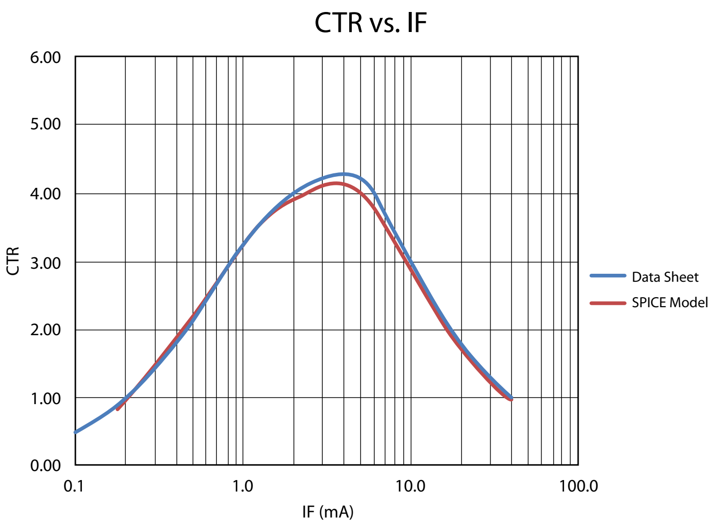
Below is the written text file of the PSPICE model for operation at TA = 250°C

```
Model: PS2561F-1
* A = PIN 1: diode anode
* K = PIN 2: diode cathode
* E = PIN 3: BJT emitter
* C = PIN 4: BJT collector
.SUBCKT PS2561F  A K E C
D1 A D LED
*OptoLED
Vsense D K 0
*Forward If current sense
*using Vsense
Hd   R 0 Vsense 1
*I-V (current dependent voltage source)
Rd   R T 100K
Cd   T 0 20p
Gctr C B TABLE
* Photodetector {(IC vs IF) / * Q1 BF}
* where the data of the collector current
* versus forward current
* are derived from the CTR
* graph using polynomial fit
+ {If(V(T)<=3m,
+ ((-80000000000*PWR(V(T),5)+800000000*
+ PWR(V(T),4)-3000000*PWR(V(T),3)
++5177.2*PWR(V(T),2)+0.2453*V
+ (T)-0.00005)*1.04/700),
+ ((9000000*PWR(V(T),5)-
+998113*PWR(V(T),4)+42174*PWR
+ (V(T),3)-
+ 861.32*PWR(V(T),2)+
+ 9.0836*V(T)-0.0078)*0.945/700))}
+ (0,0) (0.1m, 0.1m)
Q1 C B E detector
.model LED D IS=1p N=1.999644
+ RS=0 BV=6 IBV=10u
+ CJO=10p EG=1.424 TT=500n
.model detector NPN IS=2.857P BF=700
+ NF=1.1786653 BR=20 TF=80.64p
+ TR=4.56886n
+ CJE=16.315P CJC=21.1189P VAF=250 ISS=0
+ CJS=44.5657p ISC=120p
.ends
*$
```

The Results of the PSPICE Model

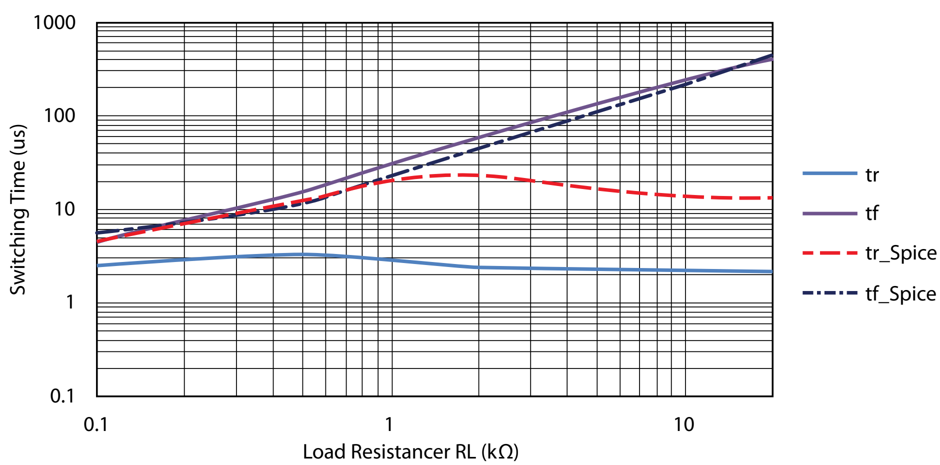
A. CTR

The PSPICE model for the CTR curve comes quite close to the data in the data sheet as shown below.



B. Switching Time

When it comes to match the same performance of the rise and fall time of the device, it becomes difficult to select the parameters of the NPN transistor model that will duplicate the same performance, as a result, a compromise is made, such that, the model could be considered as the typical or worst case performance as shown below.



Comments:

This SPICE model is intended to simulate the device behavior at Ta=25°C. Please note that the model cannot duplicate exact device performance in the real world under all conditions and they can't replace the breadboarding for final design verification.

Conclusion

PSPICE model can be a helpful tool for simulation of an optocoupler in a circuitry that incorporates optocouplers in its design. However, care has to be taken and need to test and verify with the real devices in the test circuit to ensure that the result. Please visit our website for more PSPICE models of Renesas/CEL optocouplers at http://www.cel.com/spice.do?command=showByType&group=2

CEL California Eastern Laboratories
4590 Patrick Henry Drive, Santa Clara, CA 95054-1817
Tel. 408-919-2500 FAX 408-988-0279 www.cel.com