

## OrCAD PSpice 智慧建立模擬模型 實現完整的電路模擬驗證

OrCAD® PSpice® 不僅包含大量豐富的 PSpice 模擬模型，其中有逾 33,000 顆模擬模型可直接由內建零件庫取得，亦提供便利的建模介面 PSpice Modeling Application...等多種不同獲得 PSpice 模擬模型的方式，讓您的電路設計可方便的透過 PSpice 模擬驗證，提升電路設計品質加速整體設計流程。



豐富的PSpice模擬模型零件庫



便利的建立模擬模型介面

• **Date :** 2016 / 08 / 31

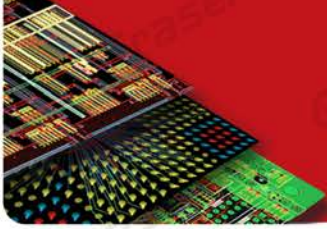
• **Author :** Stacy Chen

• **Revision :**

• **Version :** 16.6 and later

• **備註:**

**Graser®** <http://www.graser.com.tw>

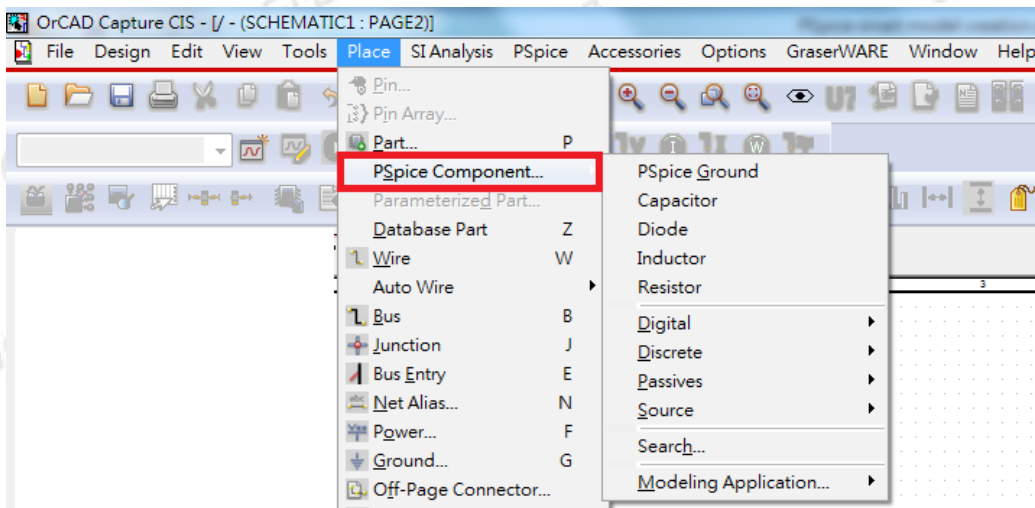


## OrCAD® PSpice® 智慧建立模擬模型 實現完整的電路模擬驗證

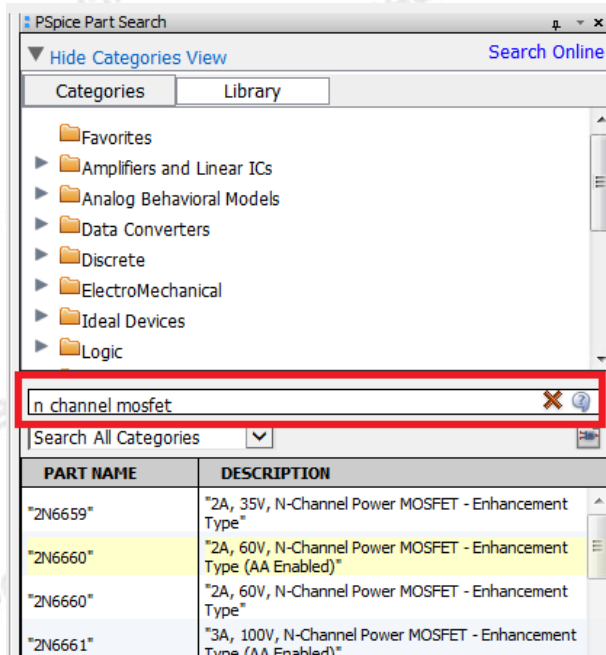
OrCAD PSpice 不僅包含大量豐富的 PSpice 模擬模型，其中有逾 33,000 顆模擬模型可直接由內建零件庫取得，亦提供便利的建模介面 PSpice Modeling Application...等多種不同獲得 PSpice 模擬模型的方式，讓您的電路設計可方便的透過 PSpice 模擬驗證，提升電路設計品質加速整體設計流程。

**直接取得 OrCAD PSpice 模擬模型，兼具零件外觀與模擬模型，電路模擬驗證更便利。**

專司取得具 OrCAD PSpice 模擬模型零件〔含電壓源、電流源、接地〕的選單，取用的零件不僅可進行電路設計更可同時為後續的電路模擬驗證做準備。

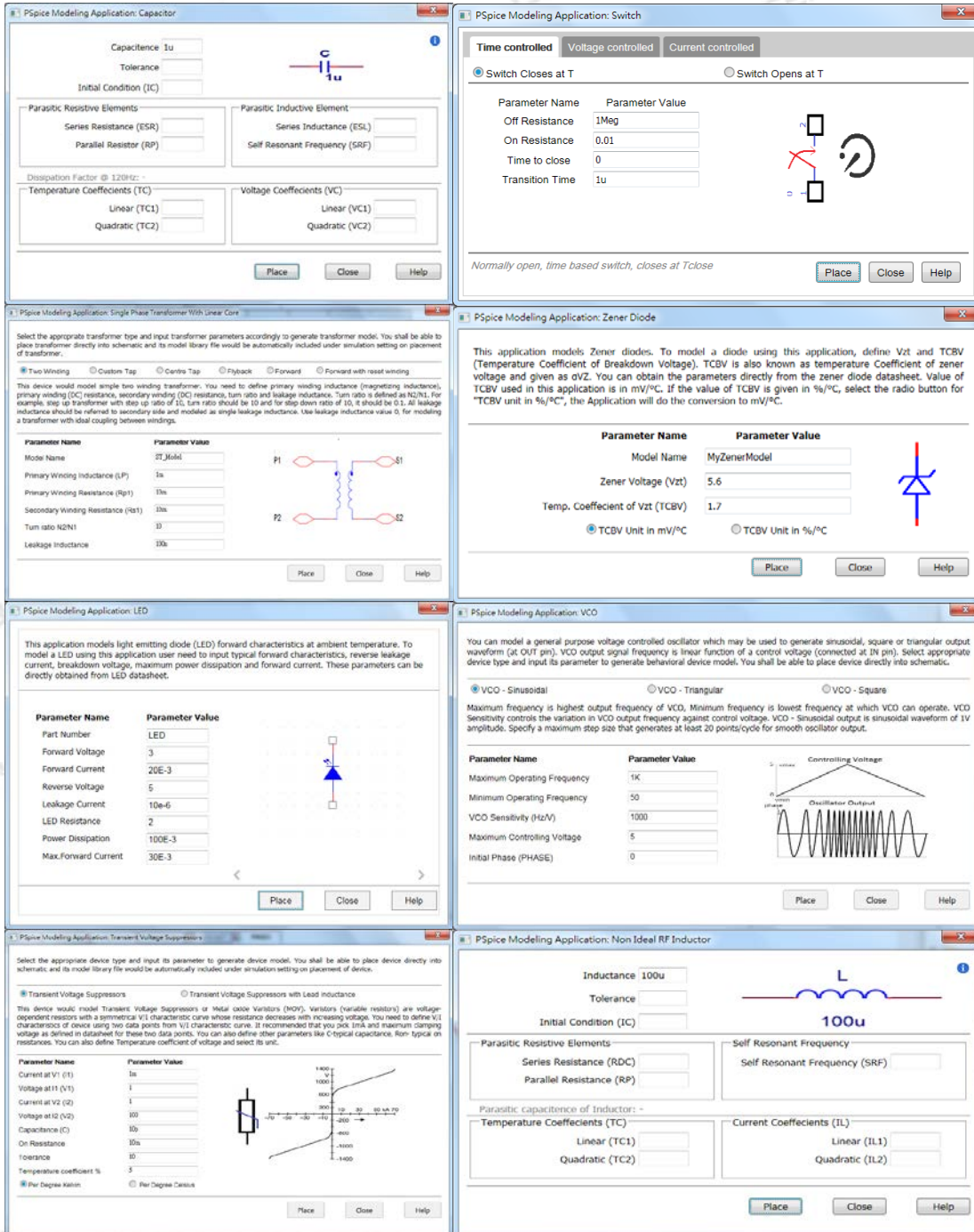


OrCAD PSpice 模擬模型零件也可透過 PSpice Part Search 功能，藉由輸入關鍵字，如：零件名稱、零件描述或是零件庫名稱，快速的搜尋到欲使用零件。



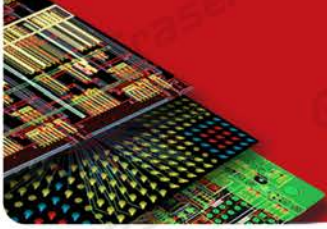
使用 PSpice Modeling Application，大幅減少 PSpice 模擬模型所需的冗長開發時間。

於 PSpice Modeling Application 中可建立 Capacitor、Inductor、Transformer、Zener、Switch、TVS、VCO 及 LED 模擬模型。透過表單填入零件的規格，即可完成此零件的 PSpice 模擬模型。



The image displays six screenshots of the PSpice Modeling Application interface, arranged in a 3x2 grid. Each screenshot shows a different component's configuration window:

- PSpice Modeling Application: Capacitor:** Shows fields for Capacitance (1u), Tolerance, Initial Condition (IC), and various parasitic elements like Series Resistance (ESR), Parallel Resistor (RP), and Self Resonant Frequency (SRF).
- PSpice Modeling Application: Switch:** Features a 'Time controlled' section with options for 'Switch Closes at T' and 'Switch Opens at T'. It includes a parameter table with fields for Off Resistance (1Meg), On Resistance (0.01), Time to close (0), and Transition Time (1u).
- PSpice Modeling Application: Single Phase Transformer With Linear Core:** Includes a section for selecting transformer type and inputting parameters like Mode Name, Primary Winding Inductance (Lp), and Secondary Winding Resistance (Rsr).
- PSpice Modeling Application: Zener Diode:** Contains a parameter table for Model Name (MyZenerModel), Zener Voltage (Vzt) (5.6), and Temp. Coefficient of Vzt (TCBV) (1.7). It also has radio buttons for TCBV Unit in mV/°C or %/°C.
- PSpice Modeling Application: LED:** Shows a parameter table for Part Number (LED), Forward Voltage (3), Forward Current (20E-3), and other characteristics.
- PSpice Modeling Application: VCO:** Features a section for selecting VCO type (Sinusoidal, Triangular, Square) and a parameter table for Maximum Operating Frequency (1K), Minimum Operating Frequency (50), and VCO Sensitivity (1K/V).
- PSpice Modeling Application: Non Ideal RF Inductor:** Includes fields for Inductance (100u), Tolerance, and Initial Condition (IC), along with parasitic elements like Series Resistance (RDC) and Parallel Resistance (RP).



建立完成的 PSpice 模擬模型，即可運用於電路設計中，執行電路模擬驗證。

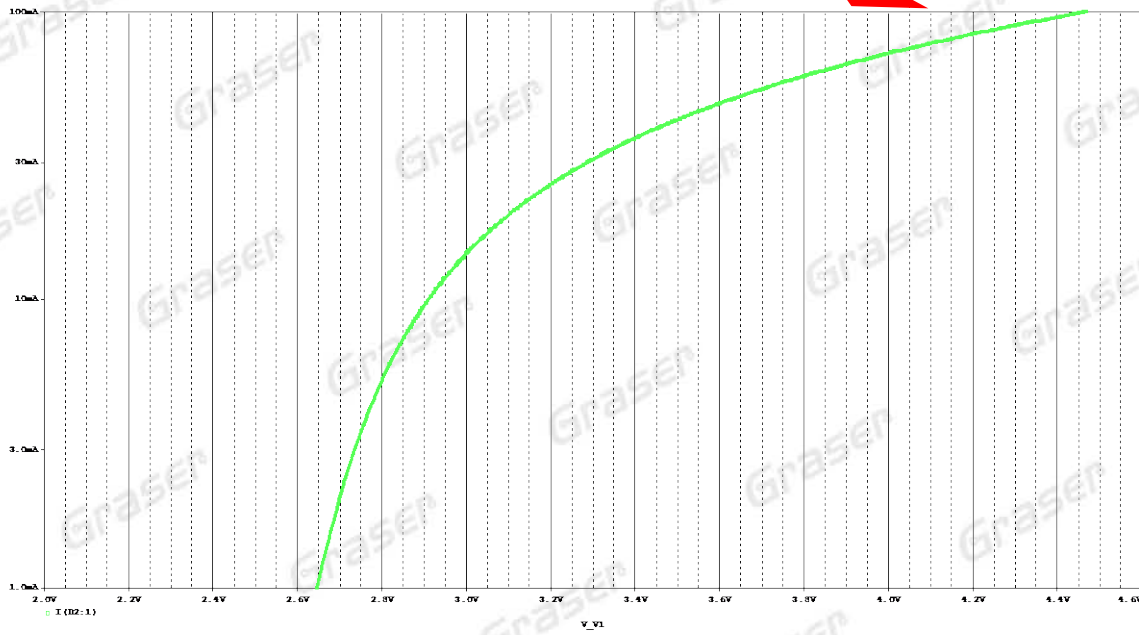
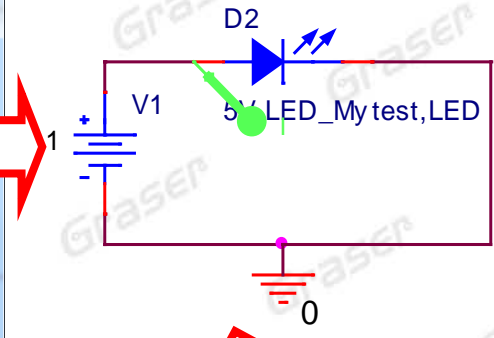
PSpice Modeling Application: LED

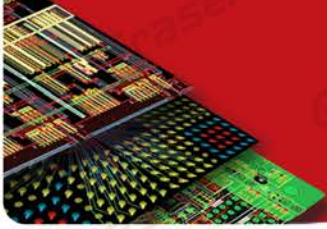
This application models light emitting diode (LED) forward characteristics at ambient temperature. To model a LED using this application user need to input typical forward characteristics, reverse leakage current, breakdown voltage, maximum power dissipation and forward current. These parameters can be directly obtained from LED datasheet.

Parameter Name	Parameter Value
Part Number	LED_Mytest1
Forward Voltage	2.8
Forward Current	20e-3
Reverse Voltage	5
Leakage Current	50e-6
LED Resistance	16
Power Dissipation	100e-3
Max.Forward Current	30e-3

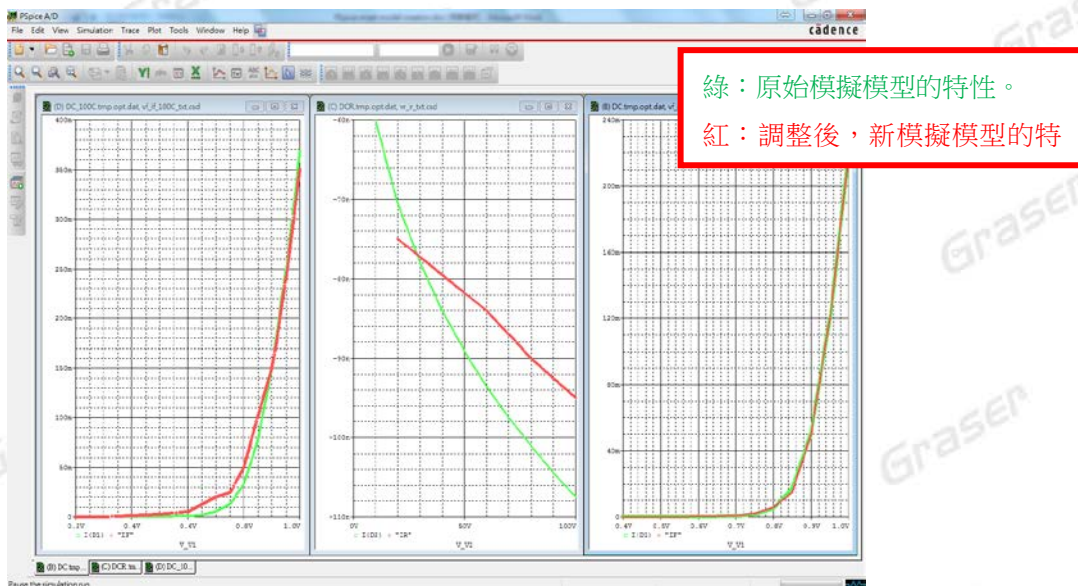
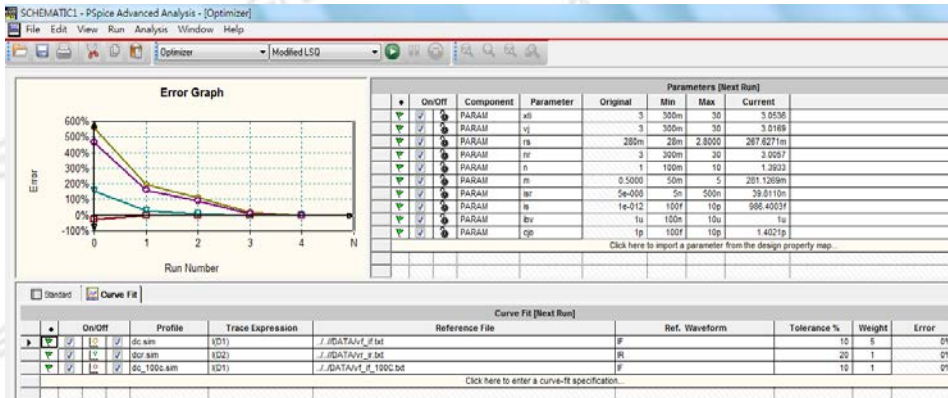
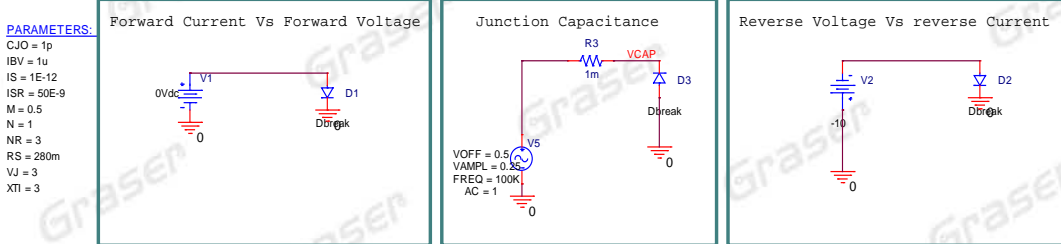
PSpice model name

Place Close Help





使用 PSpice Optimizer Analysis，快速批次的調整模擬模型參數，取得精準 PSpice 模擬模型參數。 PSpice Optimizer Analysis 可批次調整參數變數，便捷地同時滿足多個電路規範，大大降低 trial and error 的驗證時間。因此，我們可應用 PSpice Optimizer Analysis，將模擬模型參數作為變數，批次調整這些模擬模型參數，達成 PSpice 模擬模型的規格。



想了解更多 OrCAD PSpice 資訊快前往 [http://www.graser.com.tw/product\\_or\\_ospice\\_community.htm](http://www.graser.com.tw/product_or_ospice_community.htm)