

# Modeling application

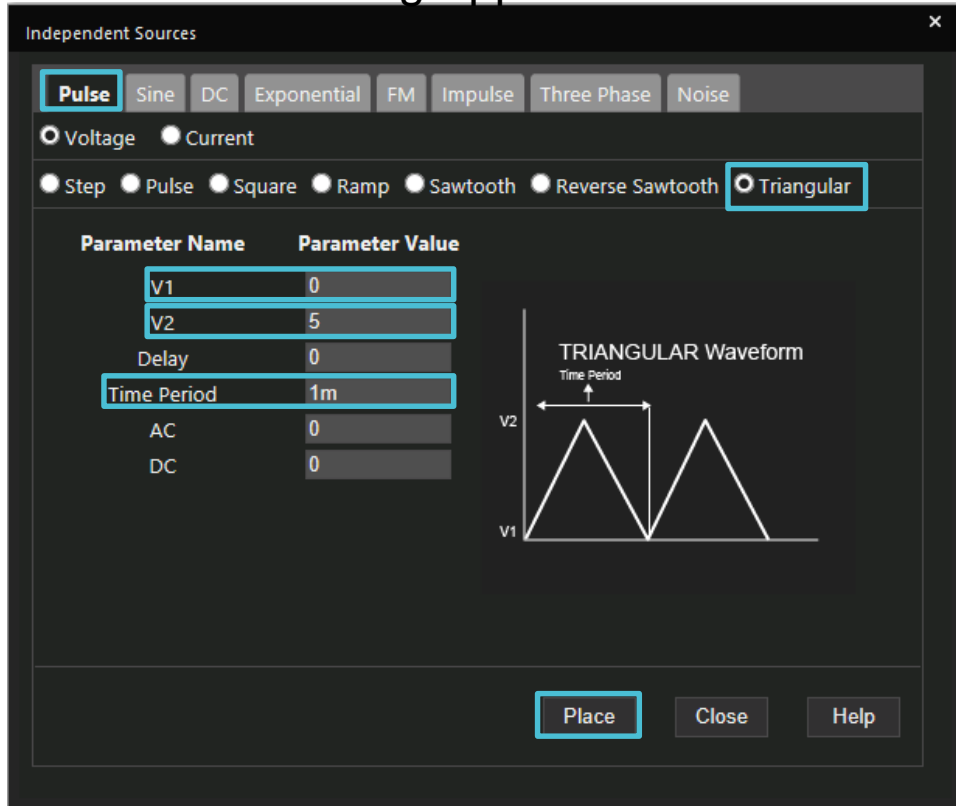
TI Precision Labs – PSpice® for TI



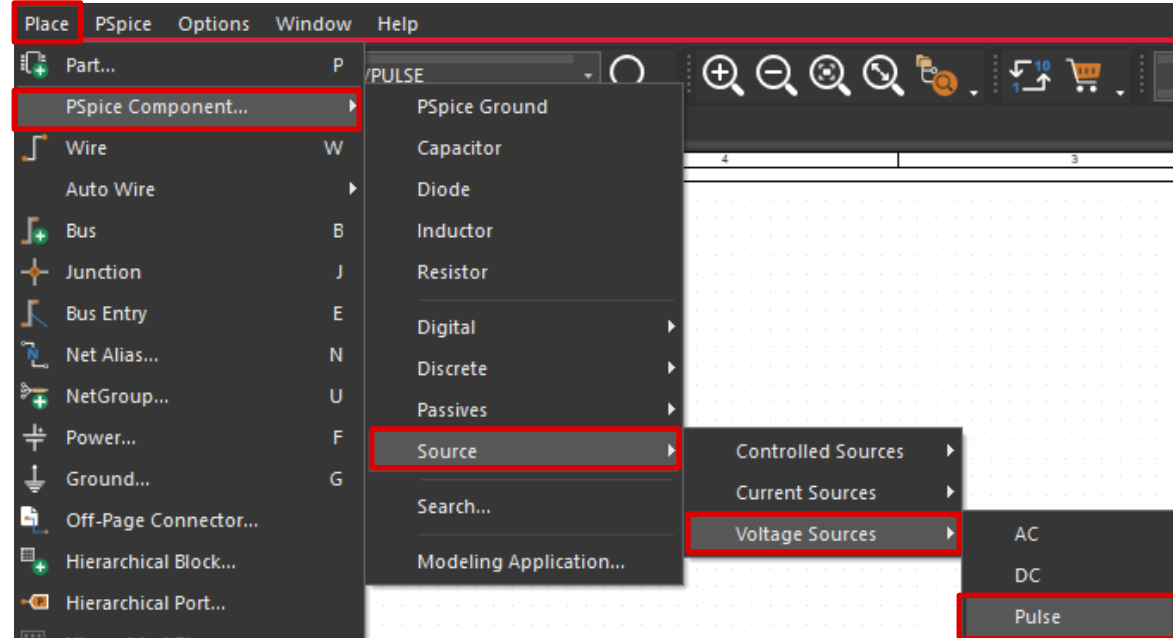
# What is the modeling application?

- Create a 1kHz 5V triangle waveform source


Modeling Application




Place Menu



V1 = 0  
V2 = 5  
TD = 0  
TR = 5.0000e-4  
TF = 5.0000e-4  
PW = 0  
PER = 1m

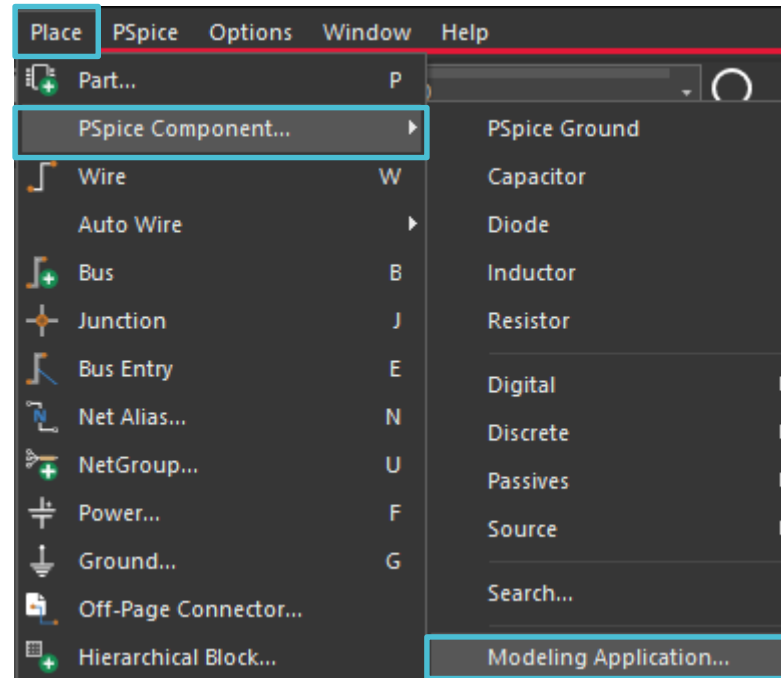


V1 =  
V2 =  
TD =  
TR =  
TF =  
PW =  
PER =



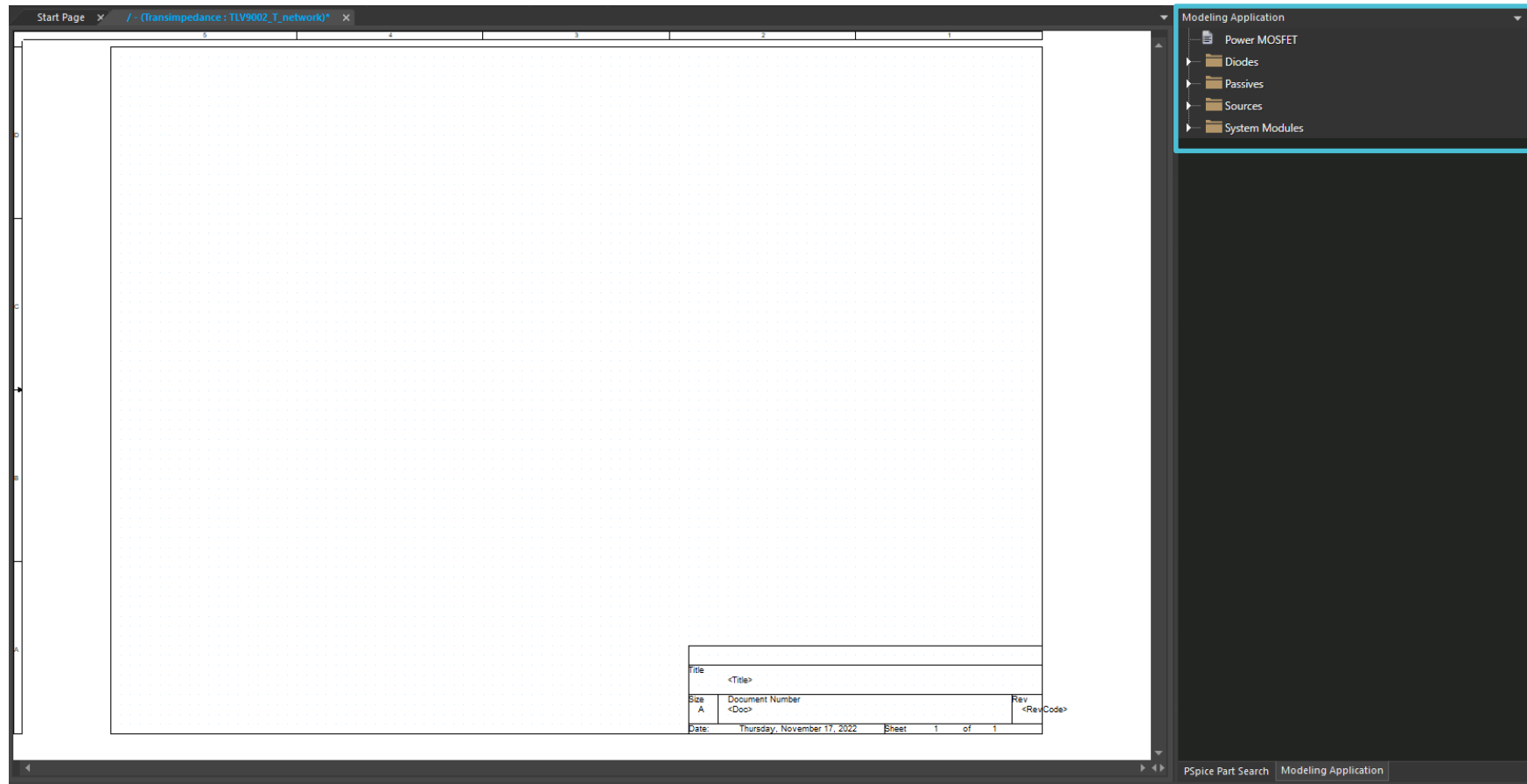
# Taskbar menu

- “Place” > PSpice Component...” > “Modeling Application...”



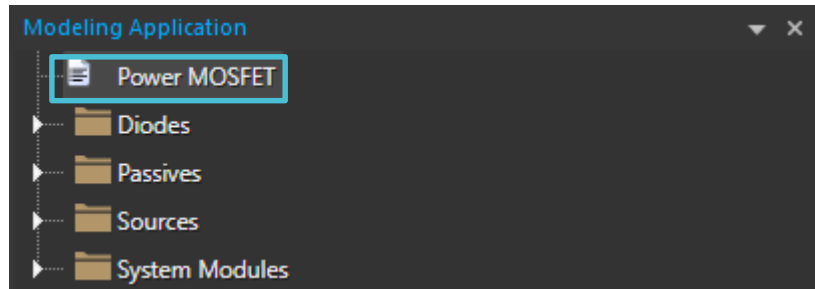
# Modeling application

- By default, the “Modeling Application” docks on the right hand side of the screen



# Power MOSFET

- Select appropriate MOSFET type and input electrical characteristics to generate a model.



A screenshot of the 'PSpice Modeling Application: Power MOSFET' dialog box. The dialog box contains the following information:

Select appropriate Metal Oxide Semiconductor Field Effect Transistor type and input MOSFET electrical characteristics to generate MOSFET PSpice model.

N Channel MOSFET       P Channel MOSFET

**Device Specification**

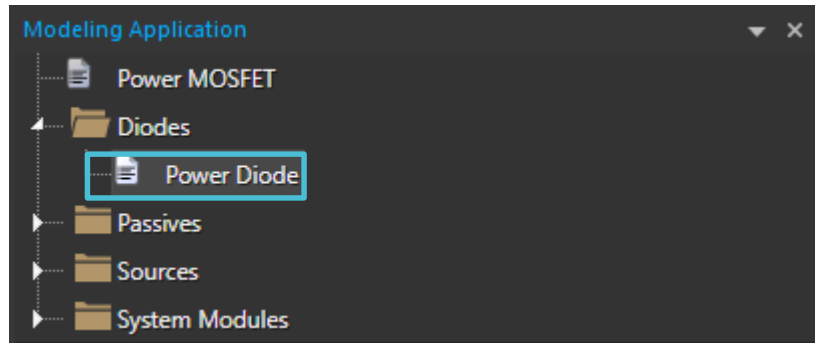
Parameter	Value	Units
Reverse transfer capacitance ( $C_{rss}$ )	58p	F
Input capacitance ( $C_{iss}$ )	1170p	F
Output capacitance ( $C_{oss}$ )	136p	F
Drain-to-source voltage ( $BVD_{ss}$ )	30.0	V
Drain-to-source on-resistance ( $R_{dsON}$ )	0.0059	Ohm
Series gate resistance ( $R_g$ )	1.8	Ohm
Gate-to-source threshold voltage ( $V_{gs\_th}$ )	1.5	V
Diode forward voltage ( $V_{SD}$ )	0.8	V
Drain-to-source leakage current ( $I_{dss}$ )	1u	A
Total gate charge ( $Q_G$ )	17.2n	C
Voltage for specified total gate charge ( $V_{QG}$ )	10	V
Forward trans-conductance ( $G_{FS}$ )	44	S
Body diode reverse recovery time ( $T_{rr}$ )	50n	s

On the right side of the dialog box, there is a schematic diagram of an N-channel MOSFET.

Buttons: Place, Close, Help

# Power diode

- Input diode electrical characteristics to generate a model.

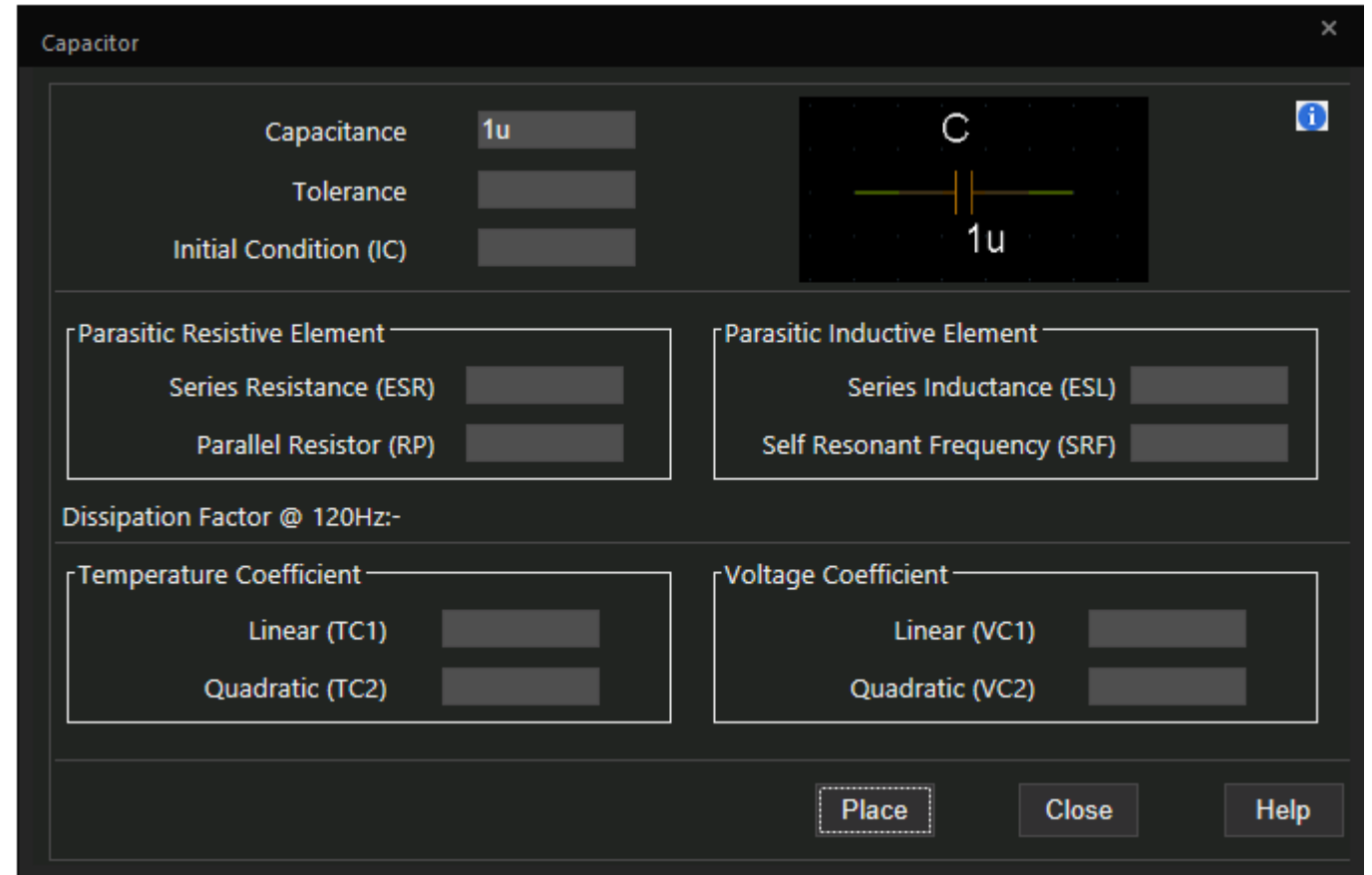
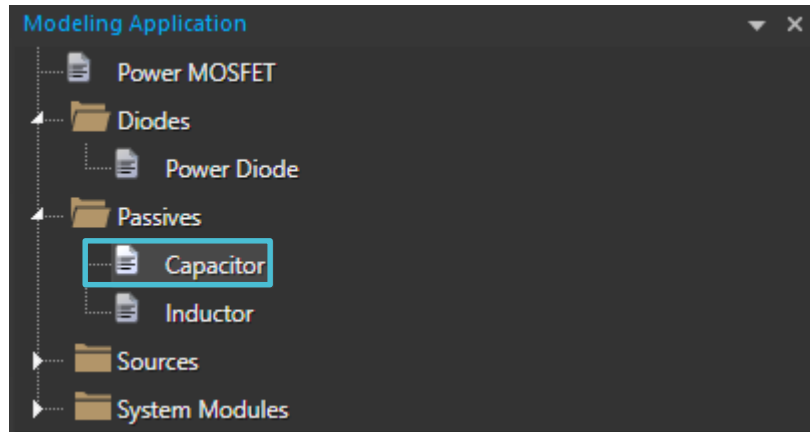


A screenshot of the 'PSpice Modeling Application: Power Diode' dialog box. The title bar reads 'PSpice Modeling Application: Power Diode'. Below the title bar, the text says 'Input diode electrical characteristics to generate power diode PSpice model.' Below this text is a table with columns for 'Parameter', 'Description', 'Value', and 'Units'. The table contains six rows of parameters. To the right of the table is a yellow diode symbol. At the bottom right of the dialog box are three buttons: 'Place', 'Close', and 'Help'.

	Parameter	Description	Value	Units
1	V_BR	Breakdown voltage, reverse peak voltage	600	V
2	I_Rev_Leak	Reverse Leakage current	1e-5	A
3	CT_Zero_Volts	Total junction capacitance at 0V	300p	F
4	VJ_Knee	Threshold voltage for forward current flow	0.9	V
5	VJ_Spec	Diode voltage drop at specified forward current (If_Spec)	1.6	V
6	IF_Spec	Specified forward current	6	A

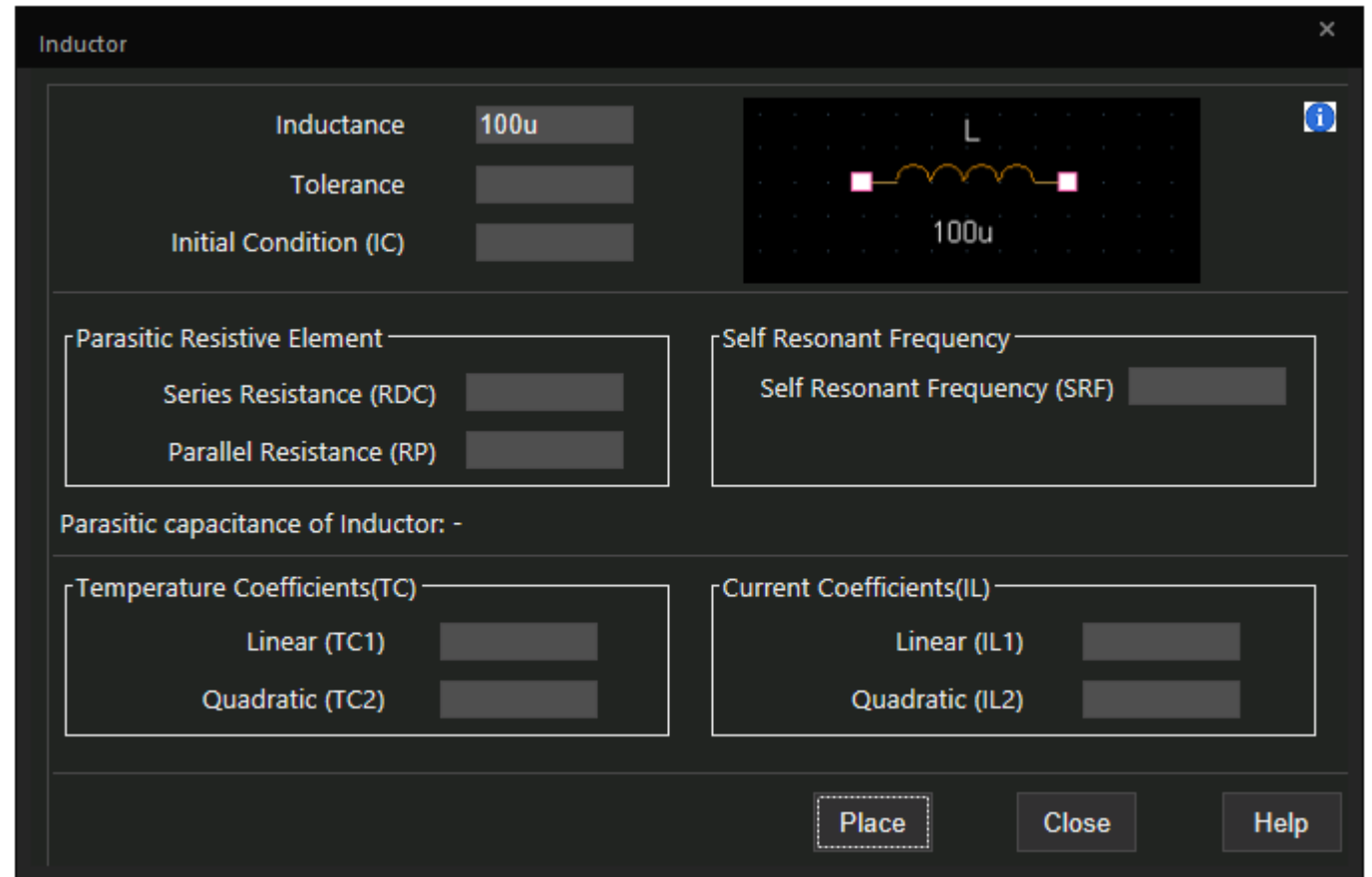
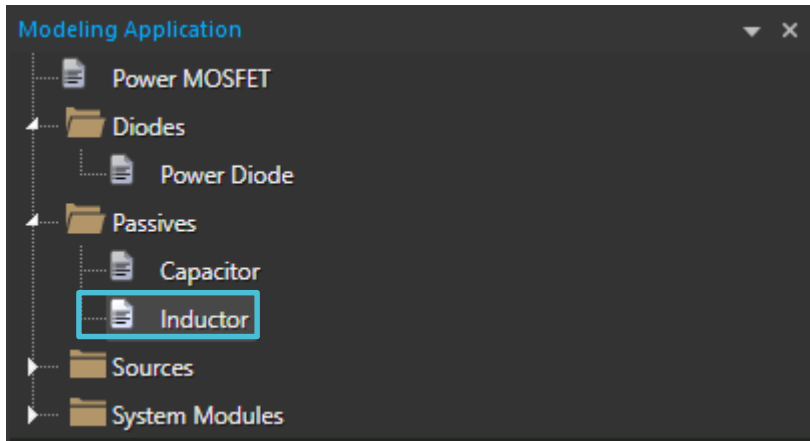
# Capacitor

- Input capacitor electrical characteristics to generate a model.



# Inductor

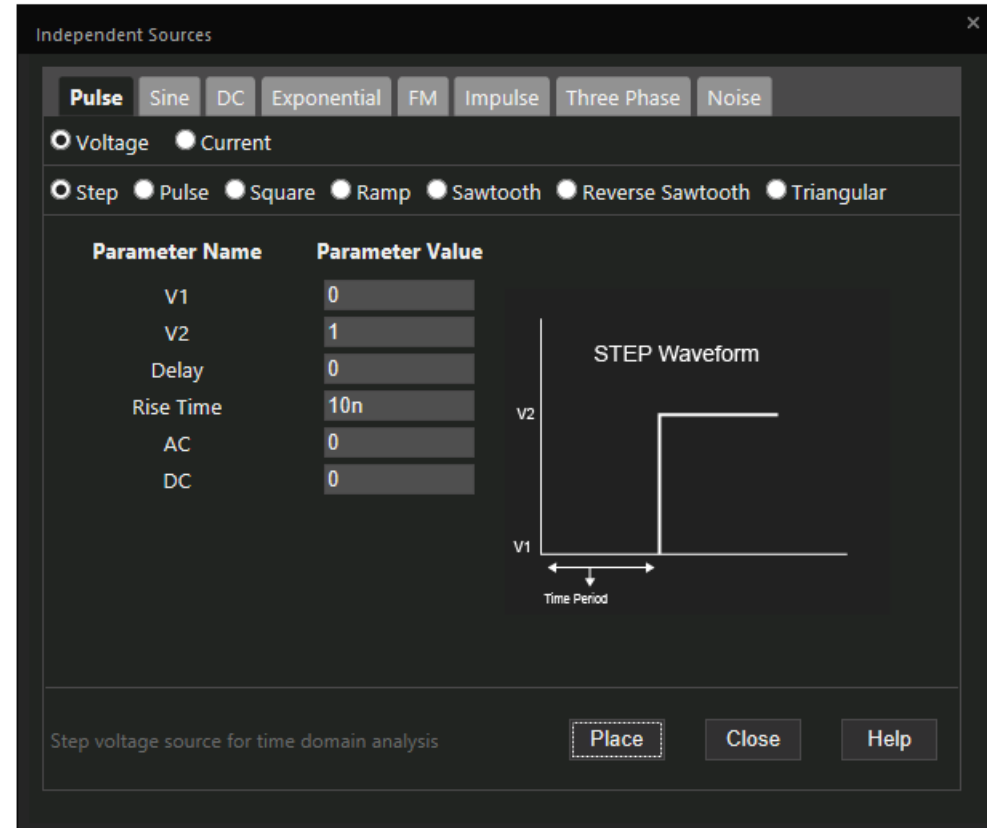
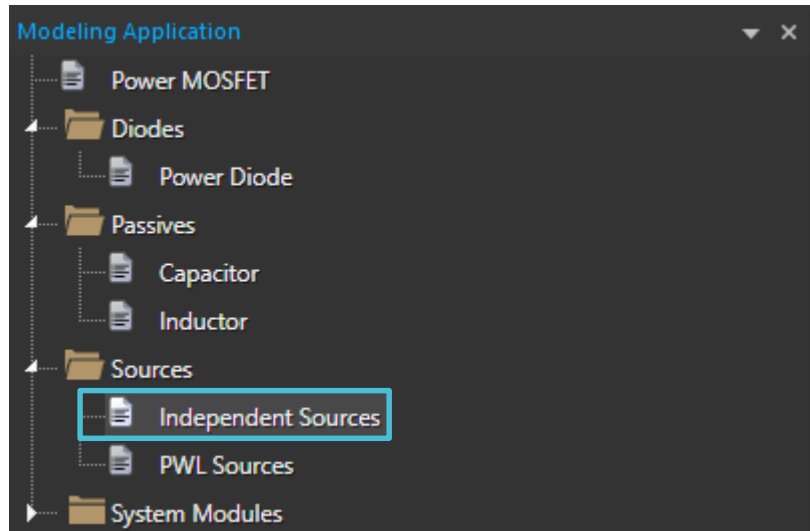
- Input inductor electrical characteristics to generate a model.





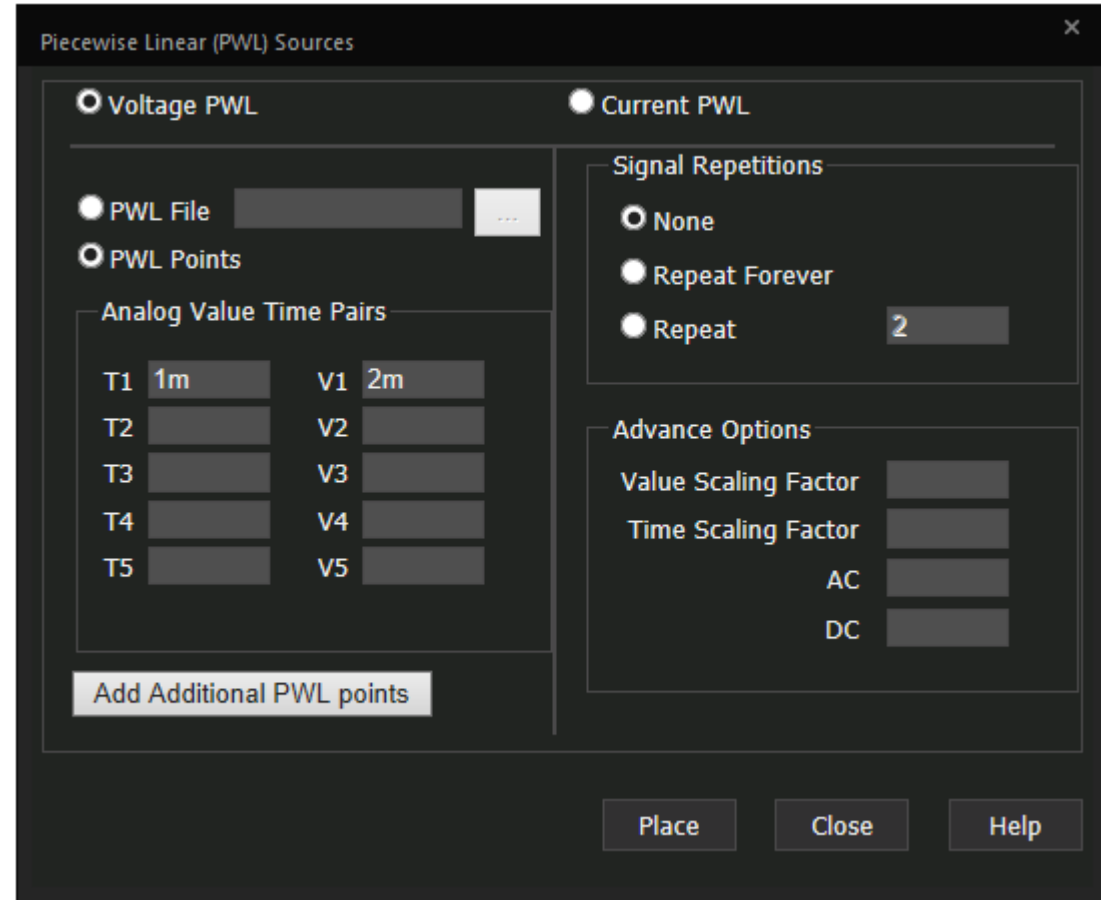
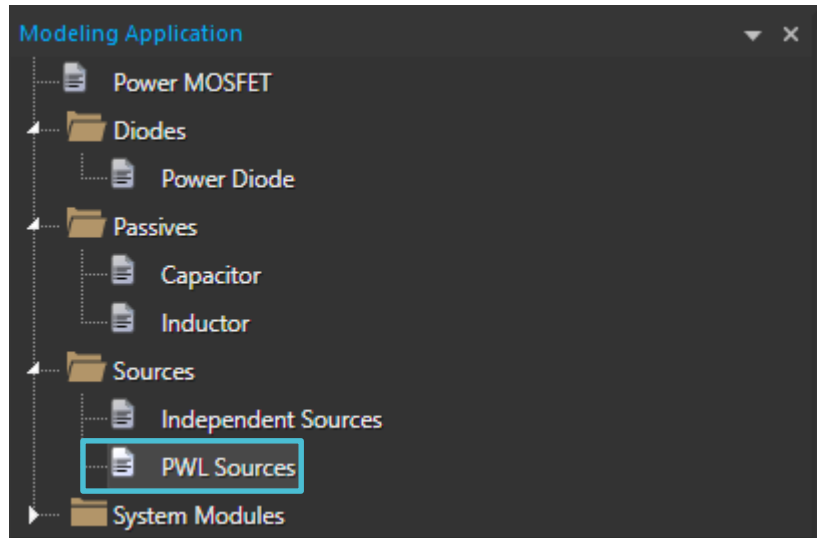
# Independent sources

- Input source parameters to generate a model.



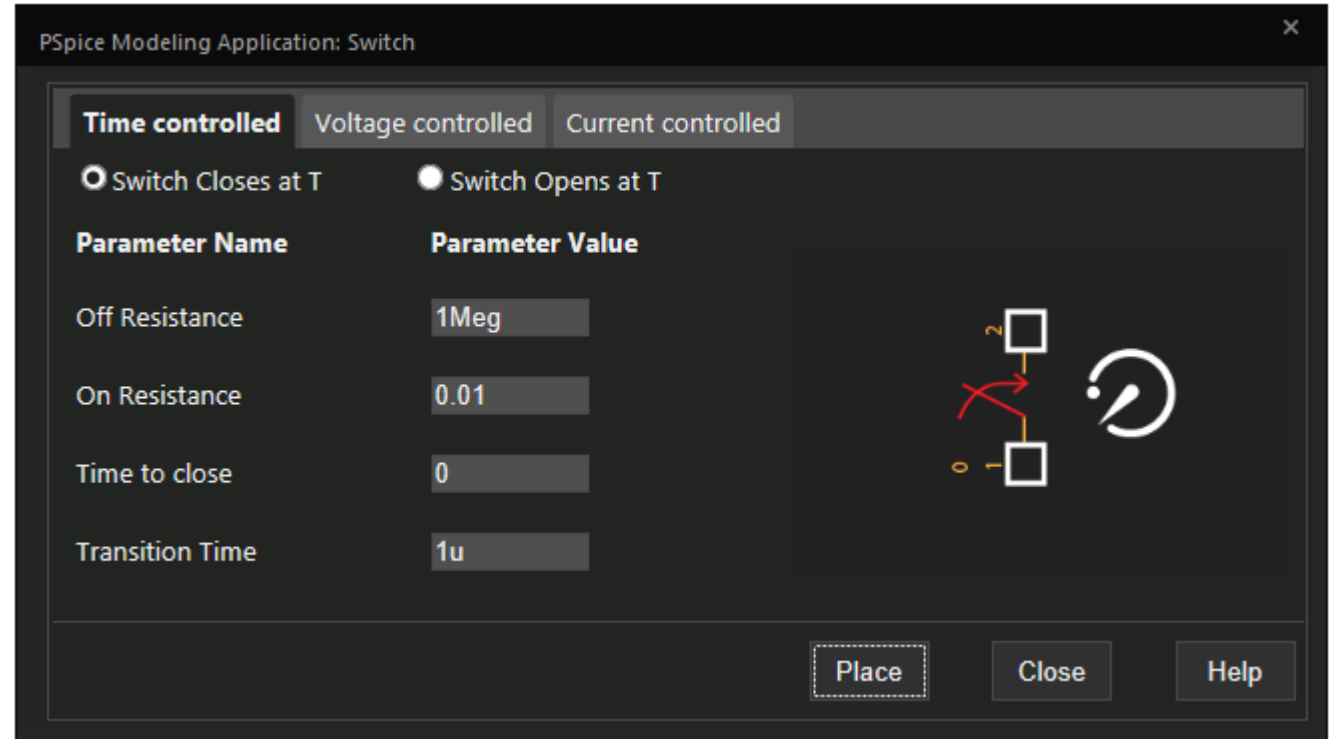
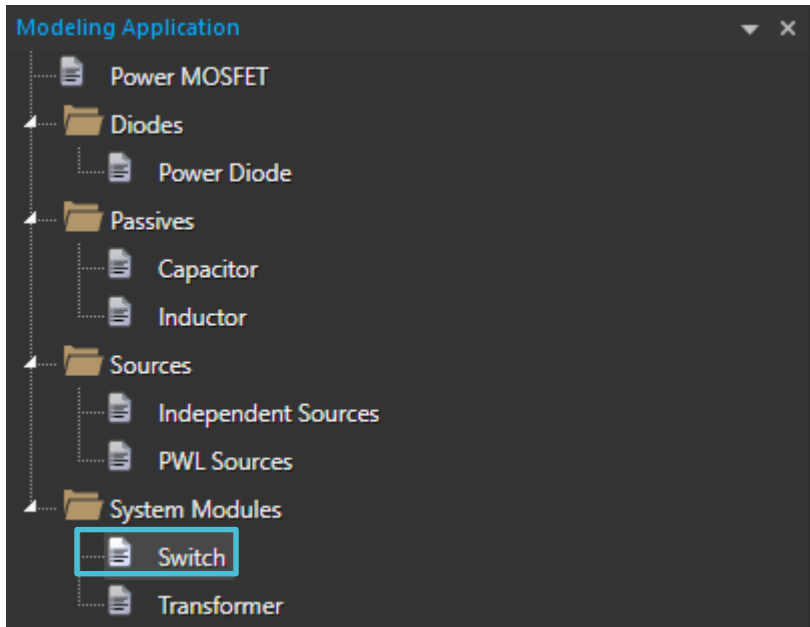
# PWL sources

- Input time pairs and repetitions or upload a source file to generate a model.



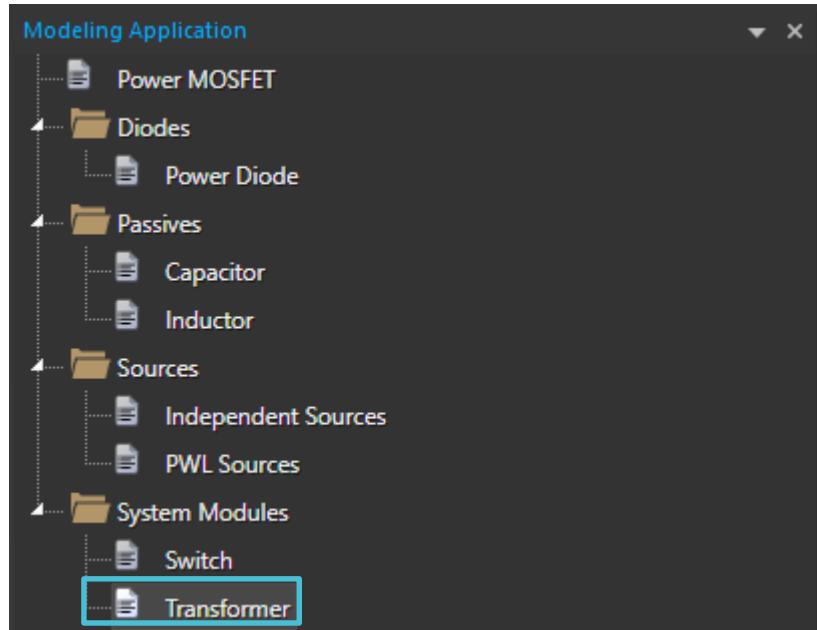
# Switch

- Input switch electrical characteristics to generate a model.



# Single phase transformer with linear core

- Select appropriate transformer type and input electrical characteristics to generate a model.



PSpice Modeling Application: Single Phase Transformer With Linear Core

Select the appropriate transformer type and input transformer parameters accordingly to generate transformer model. You shall be able to place transformer directly into schematic and its model library file would be automatically included under simulation setting on placement of transformer.

Two Winding    Custom Tap    Centre Tap    Flyback    Forward    Forward with reset winding

This device would model simple two winding transformer. You need to define primary winding inductance (magnetizing inductance), primary winding (DC) resistance, secondary winding (DC) resistance, turn ratio and leakage inductance. Turn ratio is defined as  $N2/N1$ . For example, step up transformer with step up ratio of 10, turn ratio should be 10 and for step down ratio of 10, it should be 0.1. All leakage inductance should be referred to secondary side and modeled as single leakage inductance. Use leakage inductance value 0, for modeling a transformer with ideal coupling between windings.

Parameter Name	Parameter Value
Model Name	ST_Model
Primary Winding Inductance (LP)	1m
Primary Winding Resistance (Rp1)	10m
Secondary Winding Resistance (Rs1)	10m
Turn ratio N2/N1	10
Leakage Inductance	100n

The schematic diagram shows a transformer with two primary windings (P1 and P2) and two secondary windings (S1 and S2). The primary windings are connected in series, and the secondary windings are also connected in series. The transformer is represented by two vertical coils with dots at the top, indicating a dot convention.

Place   Close   Help

**Thanks for your time!**

To find more **PSpice® for TI**  
technical resources and search products,  
visit **[ti.com/tool/PSPICE-FOR-TI](https://www.ti.com/tool/PSPICE-FOR-TI)**.