

2013  
AUG 13  
TAIPEI

Graser  
User  
Conference

# PSpice SLPS With New Apps

Mark Wu

13/Aug/2013

# Topic

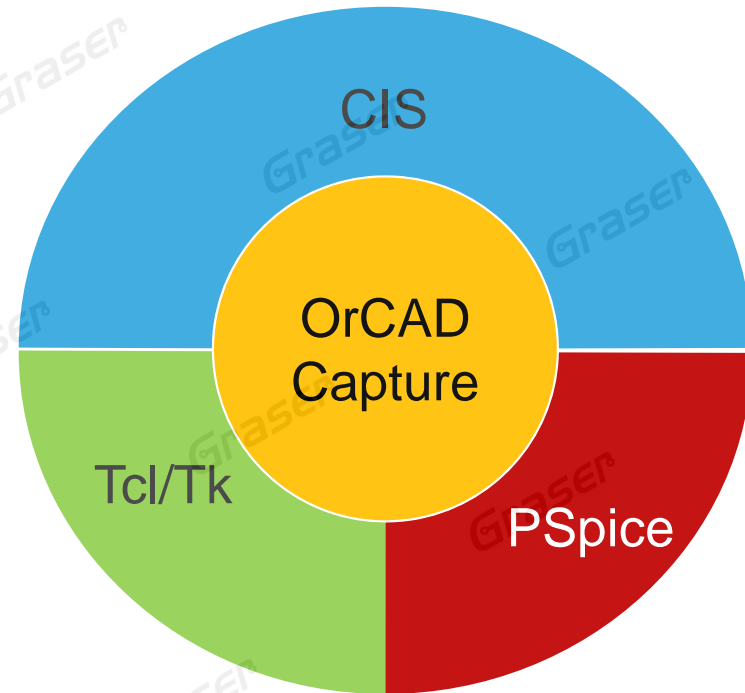
- New Apps Release for Model Creator & Application
  - Source Model
  - Model Application
- SLPS Option
  - Co-Simulation for Multi-Field

# Tcl/Tk Support for PSpice Simulation

Capture support since v16.5

PSpice support since v16.6

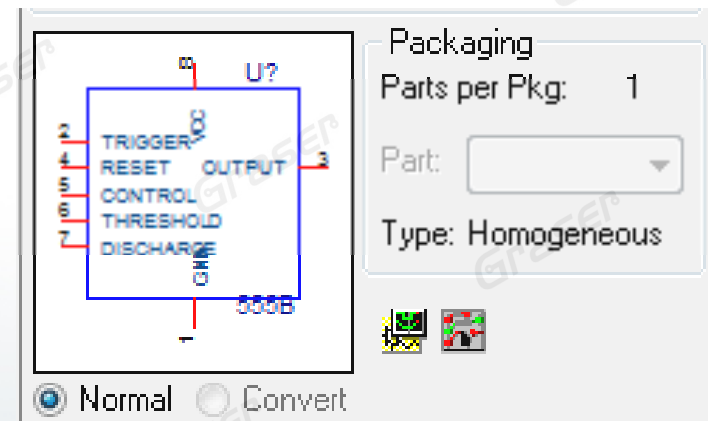
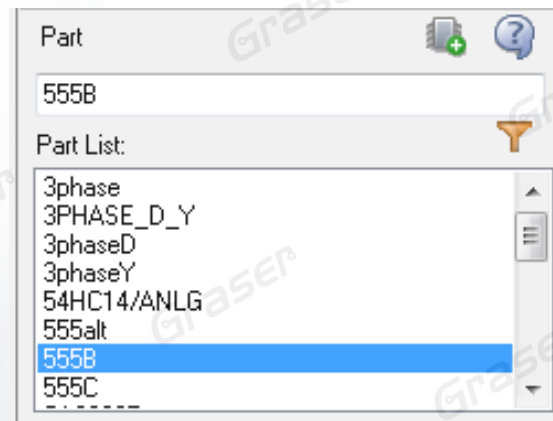
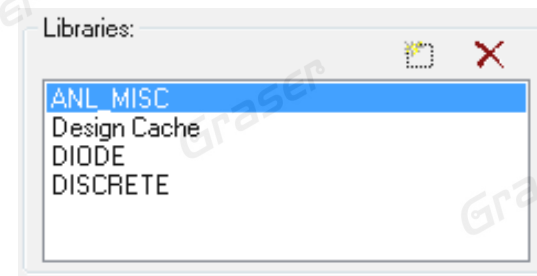
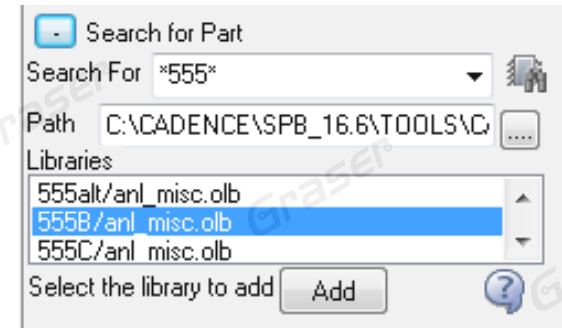
- Easy Model Creation by TCL
  - Source Model
  - Modeling Application
- Direct PSpice simulation driven process by your TCL



# How to Get Symbol with Model in Capture

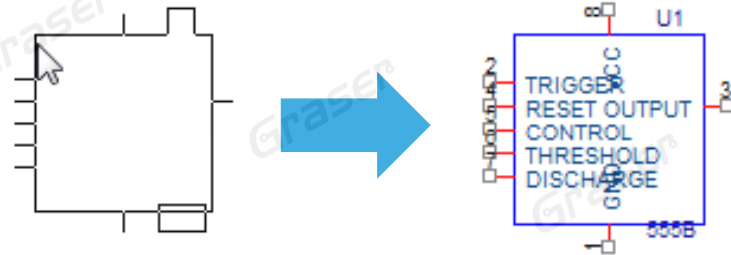
## If we can't make sure the part name

- 1. Type key words in Part Search window
- 2. Add symbol library
- 3. Choose correct symbol library
- 4. Select correct Symbol & place Part



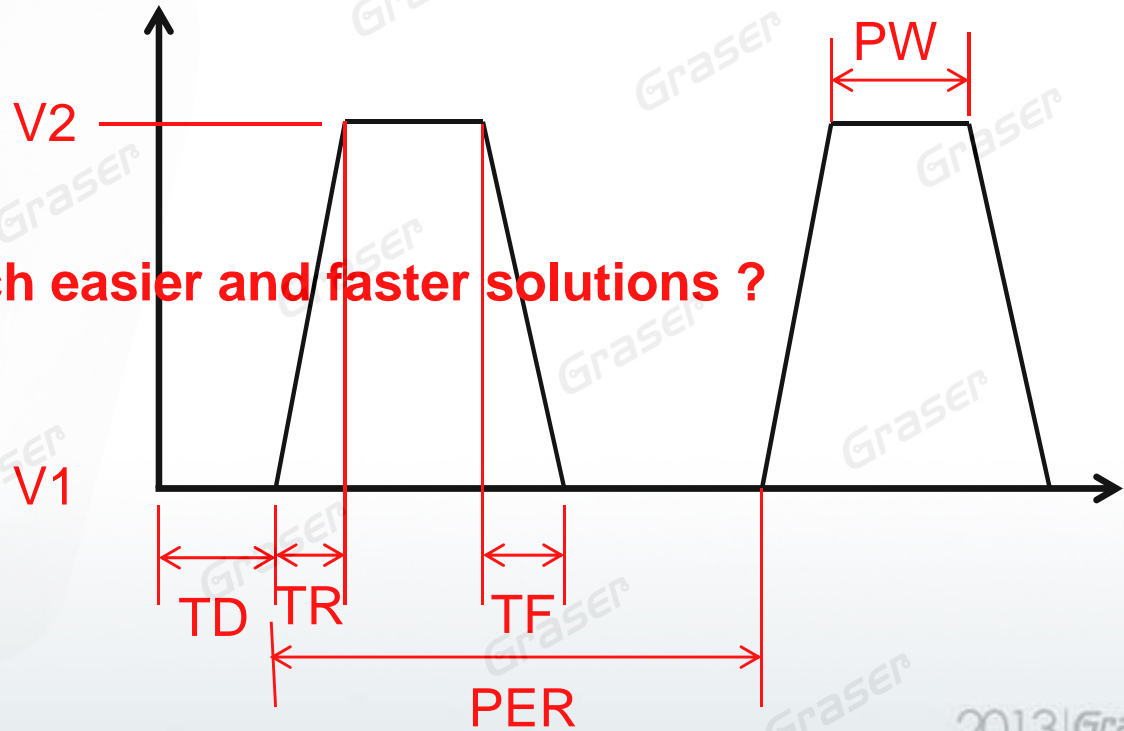
# After Capture Symbols are placed in the Schematic

1. Only place parts on the workspace
2. Need more parameter setups



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

**Do we have much easier and faster solutions ?**



# Download and Install Apps for PSpice Simulation ( For Free )

- Visit OrCAD Marketplace website <http://www.orcadmarketplace.com/>
- Download **PSpice Modeling Apps – Version 1.1**

The screenshot displays the OrCAD Apps marketplace interface. At the top, the text "OrCAD Apps" is shown in red. Below it, a banner reads "Browse for custom or enhanced features, functions, and design capabilities". The navigation bar includes "Store Home", "Products" (highlighted in red), "My Apps", and "My Account". Filter options are set to "OrCAD Capture", "Free", and "All", with a "Reset Filters" button. The "Products" section is sorted by "Name". The featured product is "PSpice Modeling Apps", which is described as significantly reducing development time and effort. It has a "Free Download" button and 0 ratings. A "Latest Additions" sidebar lists "OrCAD Apps adds FREE TRIALS" and "Central Semiconductor Adds 500 PSpice models for Discrete devices".

OrCAD Apps


Browse for custom or enhanced features, functions, and design capabilities

Store Home Products My Apps My Account

Plug-In Type Free/Paid Added Reset Filters

OrCAD Capture Free All Clear All

Products Sort by: Name

 **PSpice Modeling Apps**  
PSpice Modeling Apps significantly reduce the time and effort required to develop components and models needed for simulation during design entry.  
0 Ratings  
**Free Download**

Boosts Productivity and Predictability

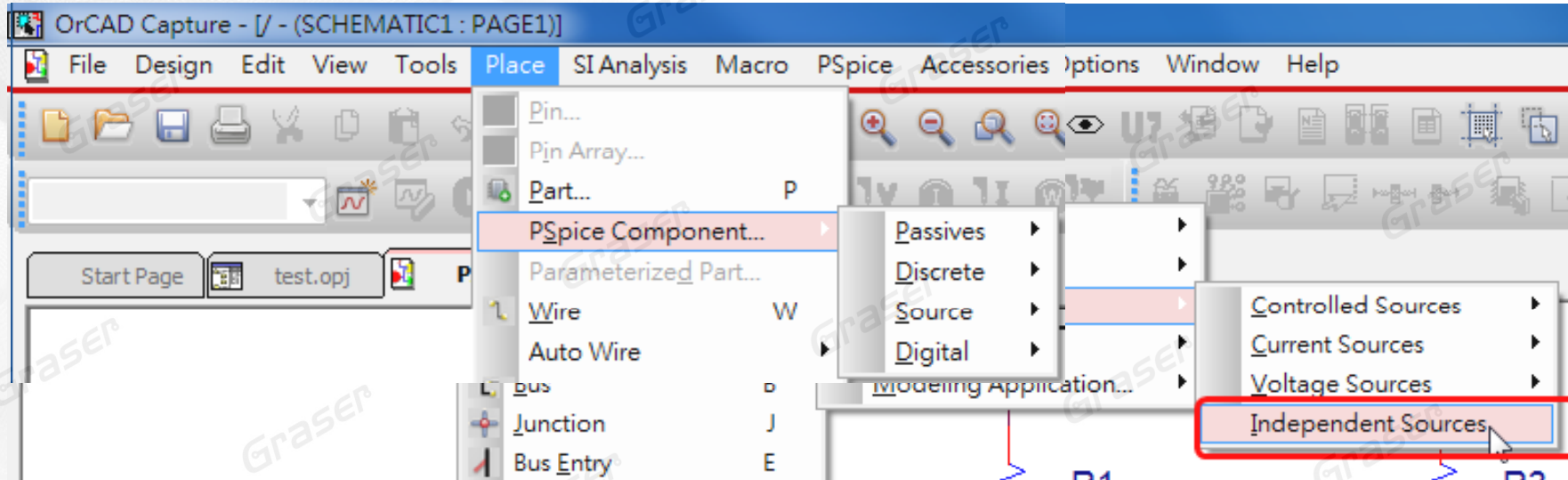
**Latest Additions**

- OrCAD Apps adds FREE TRIALS
- Central Semiconductor Adds 500 PSpice models for Discrete devices

2013 Graser User Conference  
AUG 13  
TAIPEI

# New Source Model Command Menu

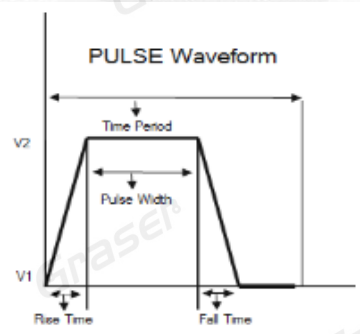
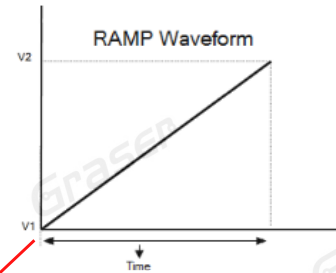
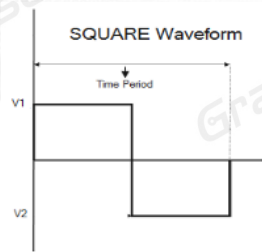
From V16.6



- 5 Type Source Model

- 1. Pulse
- 2. Sin Wave
- 3. DC Source
- 4. Exponential Waive
- 5. FM

# Pulse Source Model Generator



**Pulse** Sine DC Exponential FM

Voltage  Current

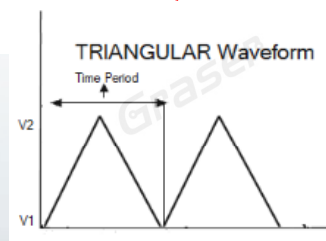
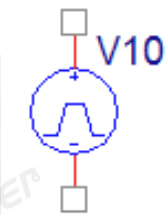
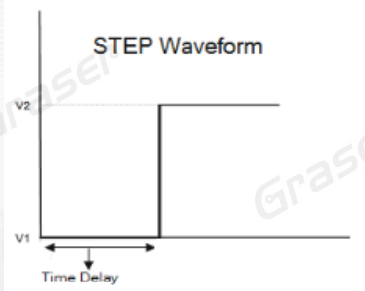
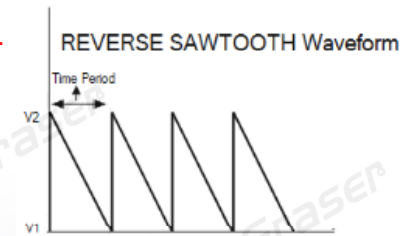
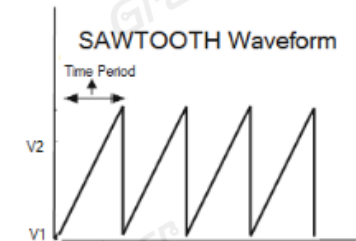
Step  Pulse  Square  Ramp  Sawtooth  Reverse Sawtooth  Triangular

Parameter Name	Parameter Value
V1	0
V2	1
Delay	0
Rise Time	10n
AC	0
DC	0

STEP Waveform

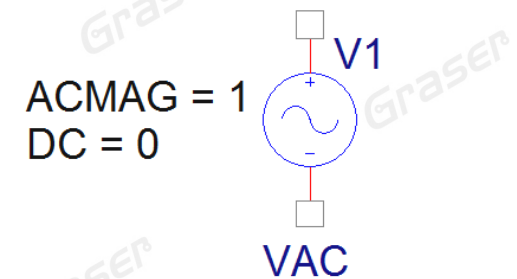
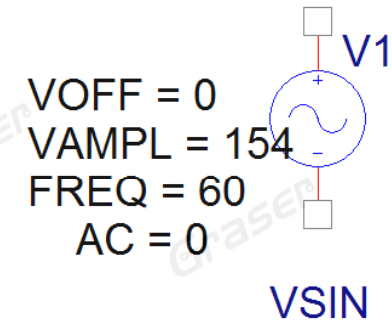
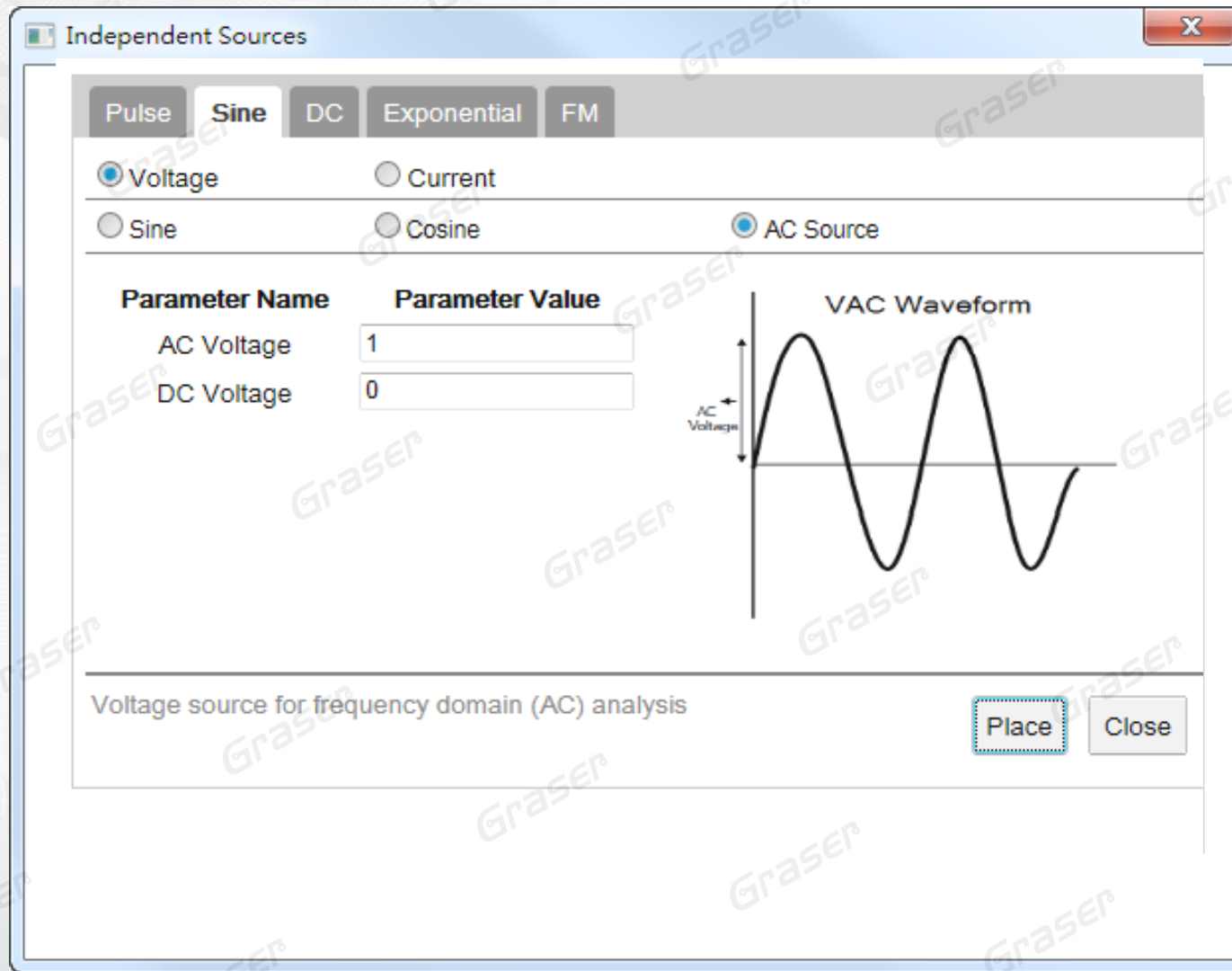
Step voltage source for time domain analysis

Place Close





# Sine Wave Model Generator



# DC Source Generator

The image displays two overlapping windows from a software application. The top window, titled "Independent Sources", has tabs for "Pulse", "Sine", "DC", "Exponential", and "FM". The "DC" tab is selected, and the "Voltage" radio button is chosen. Below, the "Ideal DC" radio button is selected. A table shows a parameter named "DC Voltage" with a value of "5".

Parameter Name	Parameter Value
DC Voltage	5

The bottom window, titled "DC Source with internal source resistance", also has the "DC" tab selected. The "Voltage" radio button is chosen, and the "DC" radio button is selected. A table shows parameters: "DC Voltage" with a value of "5" and "Source Resistance" with a value of "1m". To the right is a "DC Waveform" plot showing a constant horizontal line at a level labeled "V<sub>Dc</sub>". Below the plot is a circuit diagram showing a resistor labeled "R<sub>Dc</sub>" in series with a DC voltage source labeled "V<sub>Dc</sub>".

Parameter Name	Parameter Value
DC Voltage	5
Source Resistance	1m

DC Source with internal source resistance

Place Close

# FM Waveform Generator by Stimulus Editor vs. Apps

The image displays two software windows. The primary window is the 'Independent Sources' dialog box, which is used to configure a stimulus. It features tabs for 'Pulse', 'Sine', 'DC', 'Exponential', and 'FM'. The 'FM' tab is selected, and the 'Voltage' radio button is active. A table of parameters is shown below:

Parameter Name	Parameter Value
Offset	2
Amplitude	1
Carrier Frequency	8
Modulation Index	4
Modulation Frequency	1
AC	0
DC	0

To the right of the table is a preview of the 'FM Waveform', showing a high-frequency sine wave whose frequency varies sinusoidally. Below the table, the text 'Single frequency, frequency modulated (FM) waveform voltage source' is displayed, along with 'Place' and 'Close' buttons.

The second window, titled 'cadence', shows a waveform viewer with a green trace of the FM waveform on a black background. A tooltip for the waveform source 'V1' is overlaid on the viewer, displaying the following parameters:

- VOFF = 2
- VAMPL = 1
- FC = 8
- MOD = 4
- FM = 1

The tooltip also includes a schematic symbol for a voltage source and the label 'VSFFM'.

# Exponential Waive Generator by Stimulus Editor vs. Apps

The image displays two software windows from a circuit simulation environment. The left window, titled "Stimulus Editor - [(untitled) \*]", shows a plot area with a graph and a dialog box titled "EXP Attributes". The dialog box contains the following fields:

- Name: exp
- Initial value: 1
- Peak value: 5
- Rise (fall) delay (sec): 1
- Rise (fall) time constant (sec): 2
- Fall (rise) delay (sec): 2
- Fall (rise) time constant (sec): 0.5

Below the dialog box, a circuit diagram shows a voltage source symbol with parameters:  $V1 = 1$ ,  $V2 = 5$ ,  $TD1 = 1$ ,  $TC1 = 0.2$ ,  $TD2 = 2$ , and  $TC2 = 0.5$ .

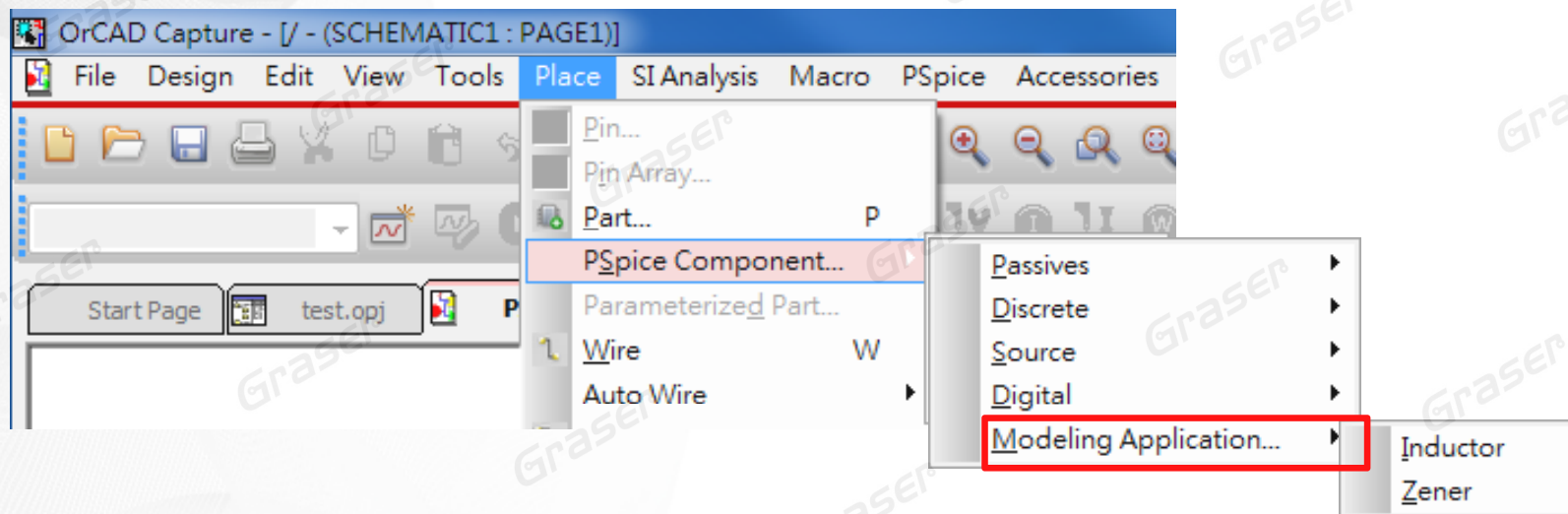
The right window, titled "Independent Sources", shows the configuration for an exponential waveform. The "Exponential" tab is selected, and "Voltage" is chosen. The "Exponential" radio button is selected. A table lists the parameters and their values:

Parameter Name	Parameter Value
V1	1
V2	5
Rise Delay	1
Rise Time Constant	0.2
Fall Delay	2
Fall Time Constant	0.5
AC	0
DC	0

To the right of the table is a graph titled "EXPONENTIAL Waveform" showing a voltage waveform with labels for "Rise Time Constant", "Fall Time Constant", "Rise Delay", and "Fall Delay".

At the bottom of the "Independent Sources" window, the text "Exponentially rising and falling voltage source" is displayed, along with "Place" and "Close" buttons. The "Place" button is highlighted with a red box.

# New Modeling Application from Apps




- **Non Ideal RF Inductor Modeling**
- **Zener Diode Modeling**

PSpice Modeling Application: Zener Diode

This application models Zener diodes. To model a diode using this application, define Vz<sub>t</sub> and TCBV (Temperature Coefficient of Breakdown Voltage). TCBV is also known as temperature Coefficient of zener voltage and given as  $\alpha_{VZ}$ . You can obtain the parameters directly from the zener diode datasheet. Value of TCBV used in this application is in mV/°C. If the value of TCBV is given in %/°C, select the radio button for "TCBV unit in %/°C", the Application will do the conversion to mV/°C.

Parameter Name	Parameter Value
Model Name	MyZenerModel
Zener Voltage (Vz <sub>t</sub> )	5.6
Temp. Coefficient of Vz <sub>t</sub> (TCBV)	1.7

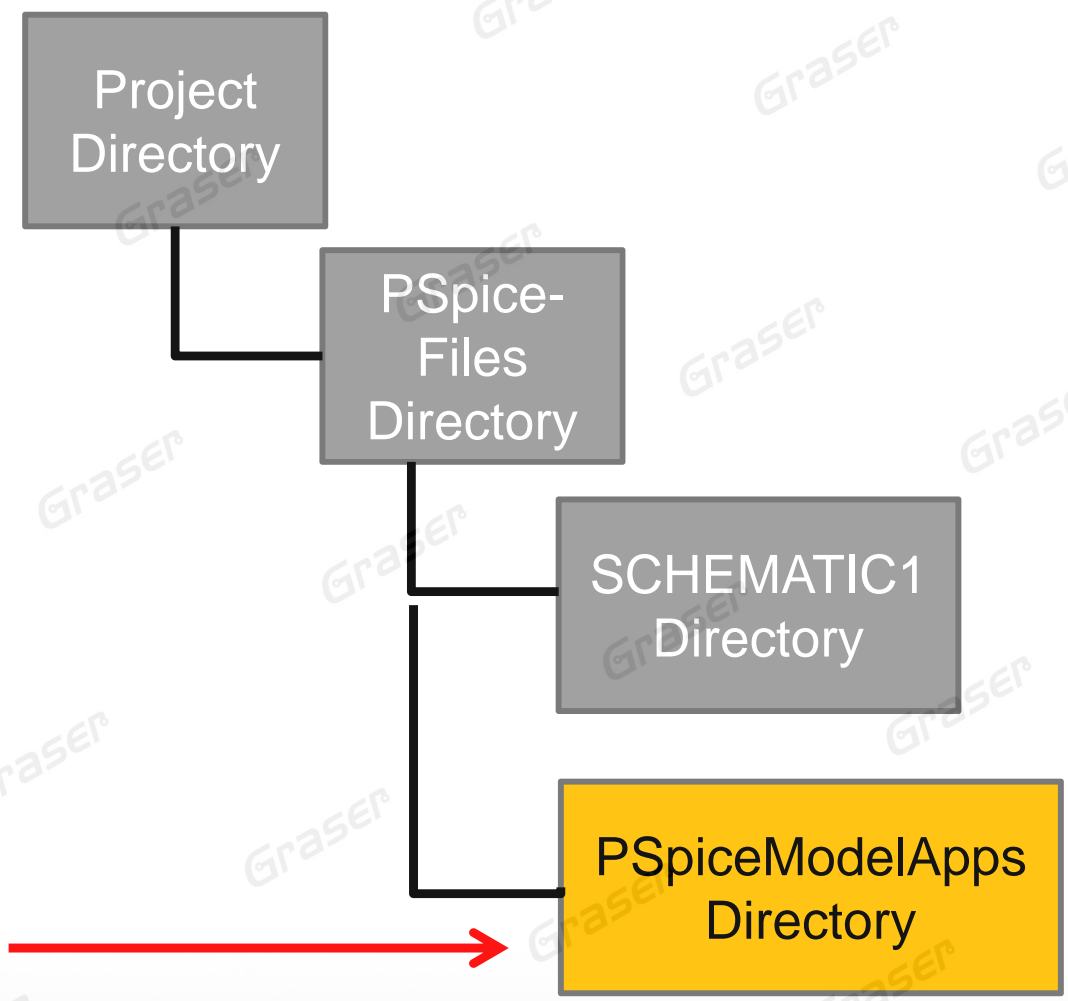
TCBV Unit in mV/°C       TCBV Unit in %/°C



Place      Close      Help

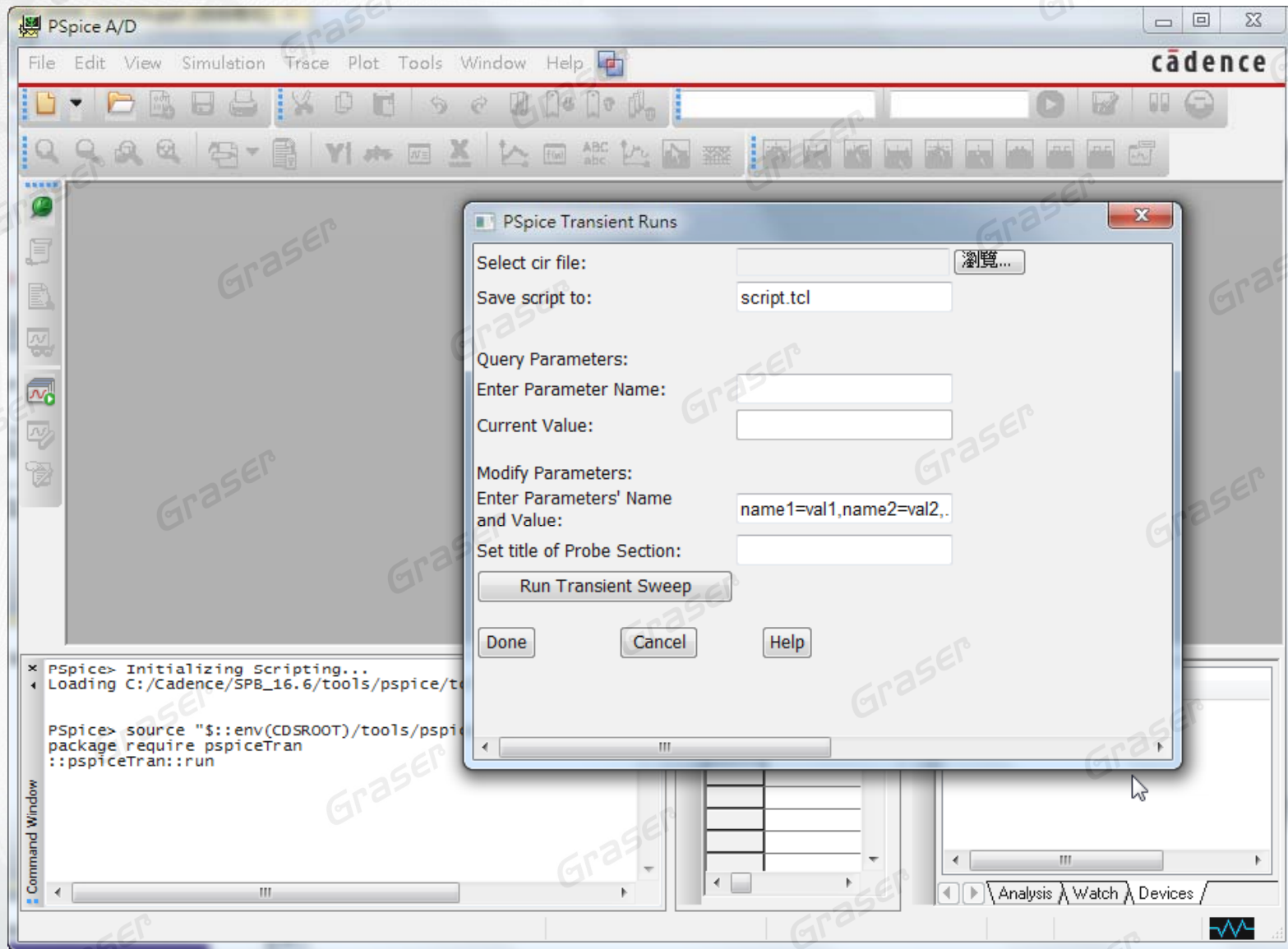
# Combine Model Files

```
PSpiceModelApps_Include.lib x
0 10 20 30 40
1 *PSpice Modeling Apps Index library File
2 *$
3 .SUBCKT MyZenerModel AN CAT
4 * Model generated on 2013 Apr 11
5 * MODEL FORMAT: PSpice
6 * MODEL Type: ZEN
7 + params:
8 + VZT=3.7
9 + IZT= 20m
10 + ZZT= 11
11 + TCBV=1.7
12 + IR= 1u
13 + VR= 1
14 XinstMyZenerModel AN CAT model434
15 + params:
16 + vzt={VZT}
17 + izt={IZT}
18 + zzt={ZZT}
19 + tcbv={TCBV}
20 + ir={IR}
21 + vr={VR}
22 .ends MyZenerModel
23 * - - - - -
24 *$
25 .SUBCKT MyInductorModel IND P1 P2
26 * Model generated on 2013 Apr 11
27 * MODEL FORMAT: PSpice
28 * MODEL Type: IND_R
29 RDC P1 PI 10m
30 L_IND PI P2 100u
```



PSpiceModelApps\_Include.lib

# Direct PSpice Simulation Driven by Apps





# SLPS Option With Matlab

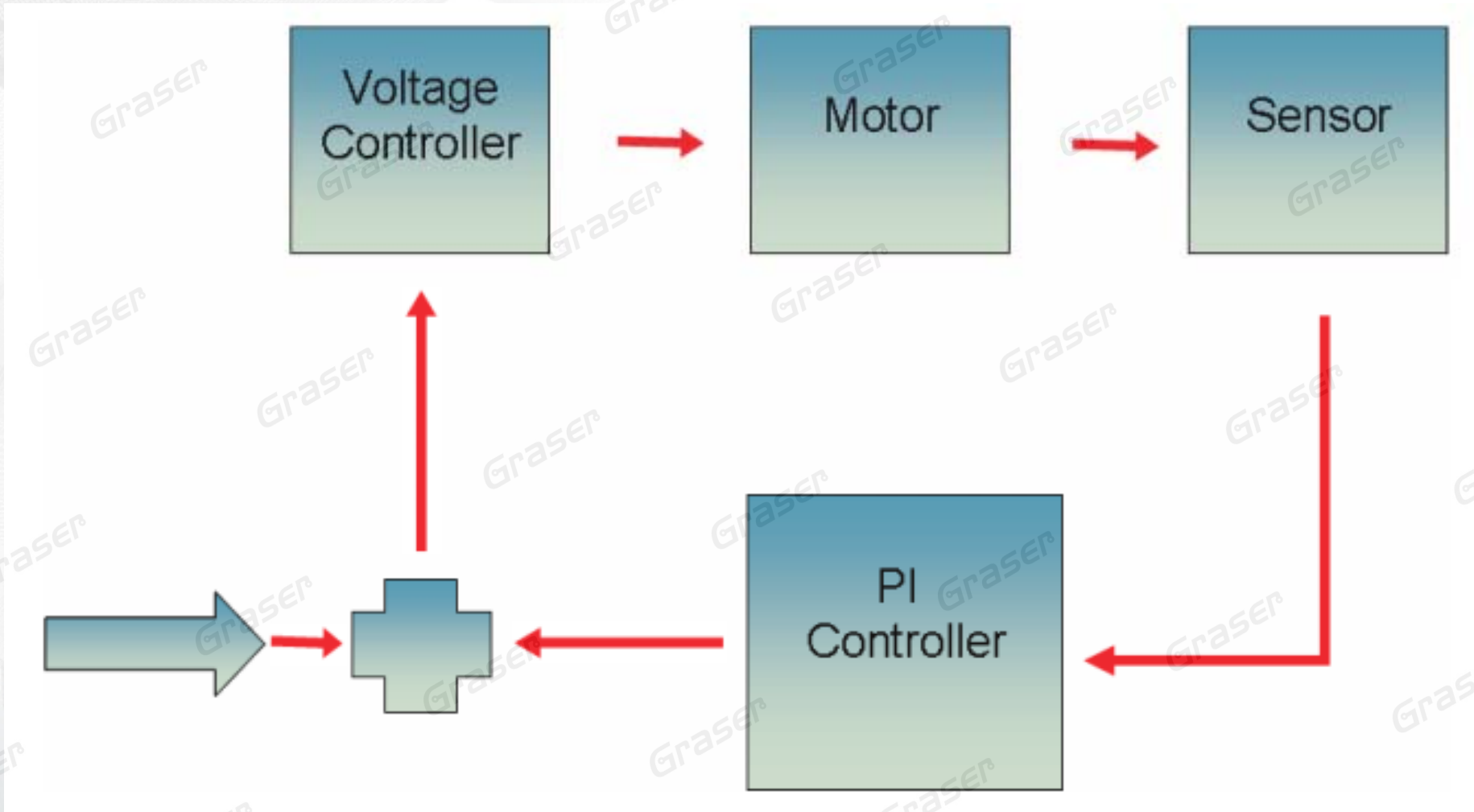
# Multi-System Co-Simulation

- Base on Mathworks Matlab

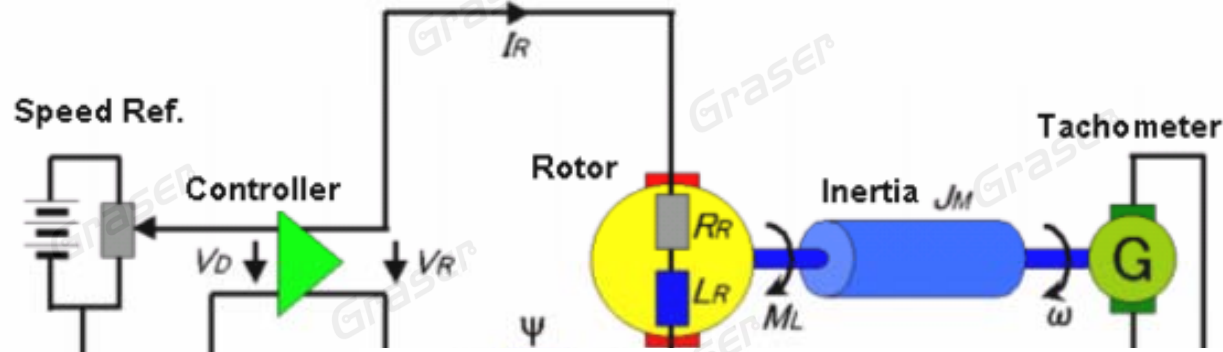
# E-Car Control System Design



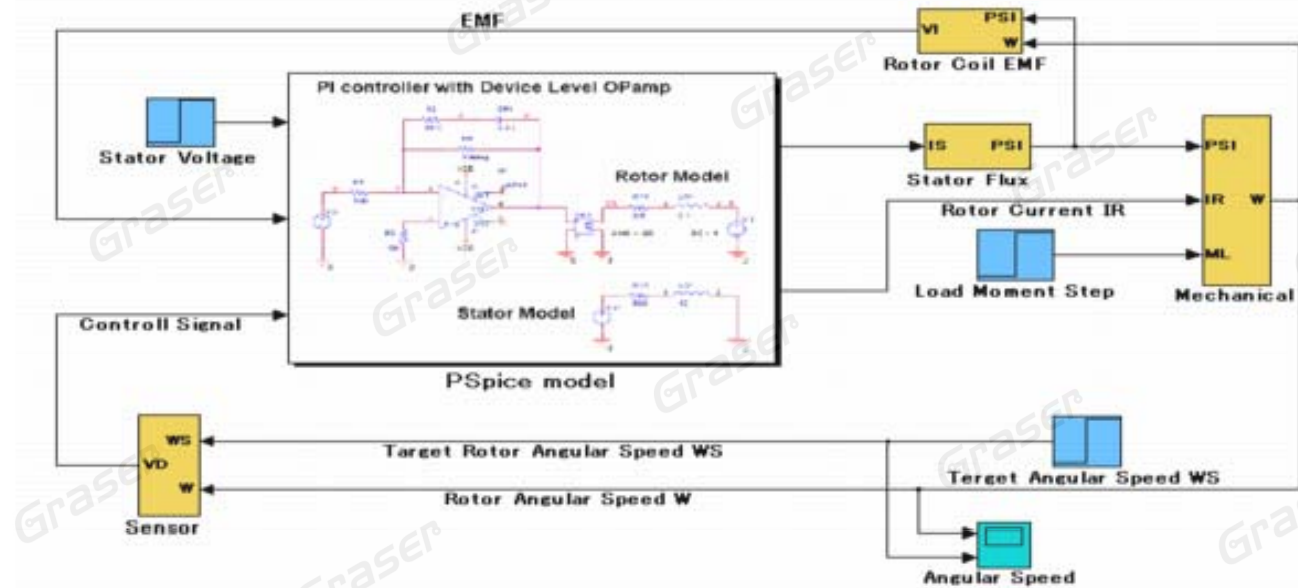
# System Block



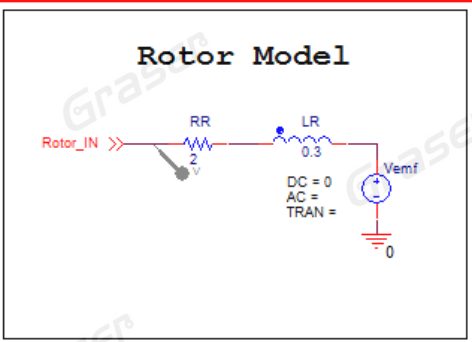
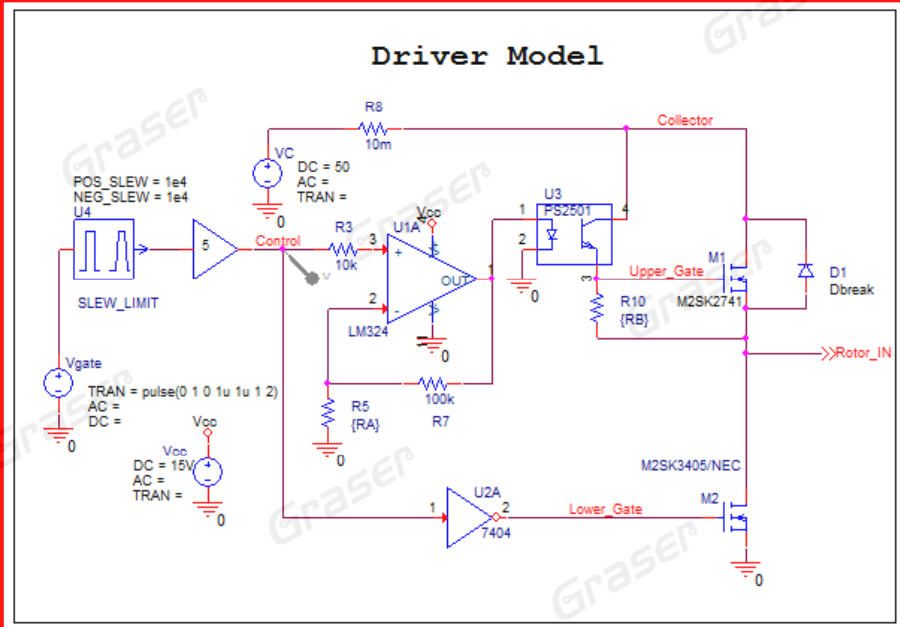
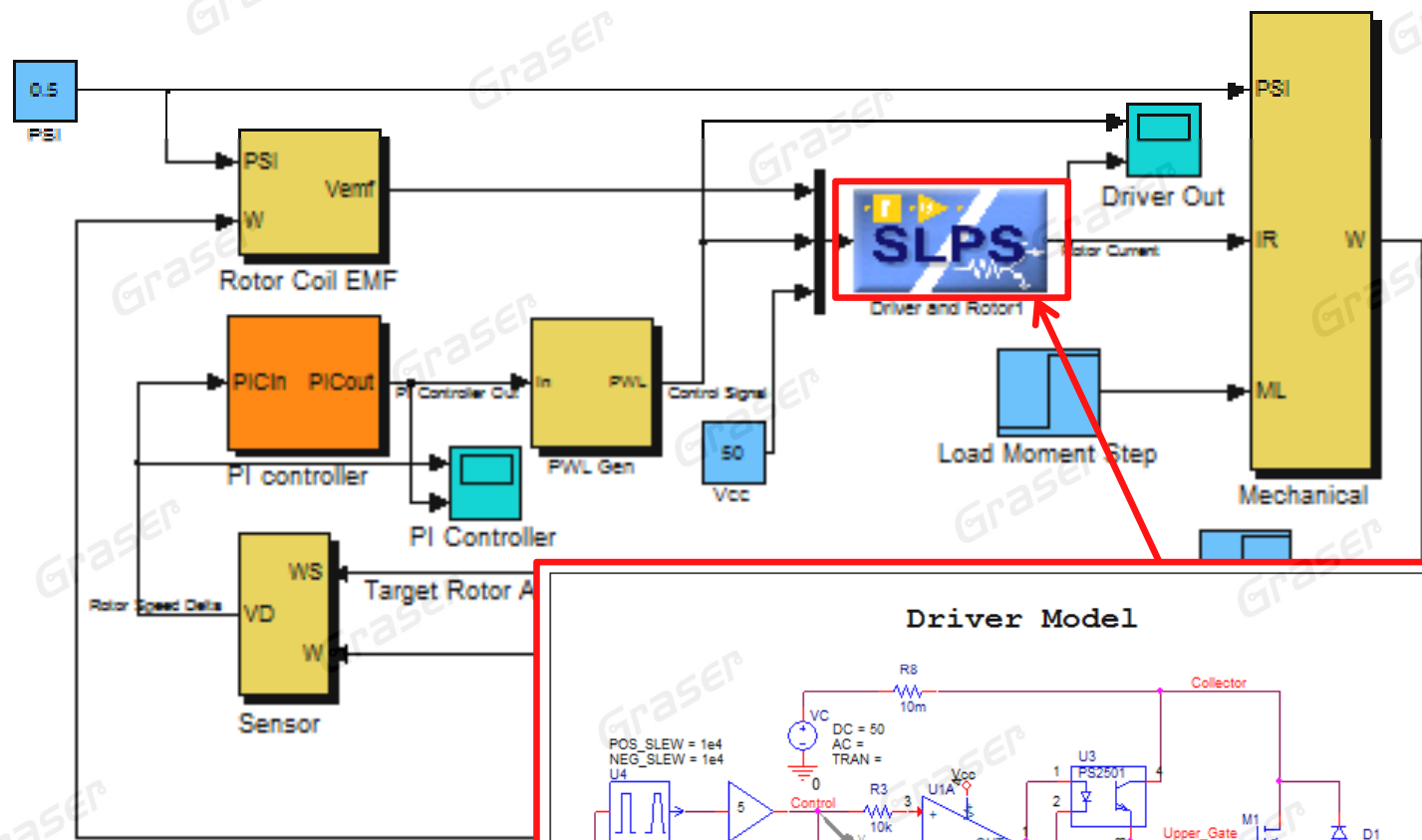
# DC Motor Control Module



*DC Motor speed control model*



# DC Motor Speed Control - Device(Driver)



PARAMETERS:  
 RA = 20k  
 RB = 100k

