2013 AUG 13 TAIPEI | Graser User Conference

PSpice SLPS With New Apps

Mark Wu

13/Aug/2013

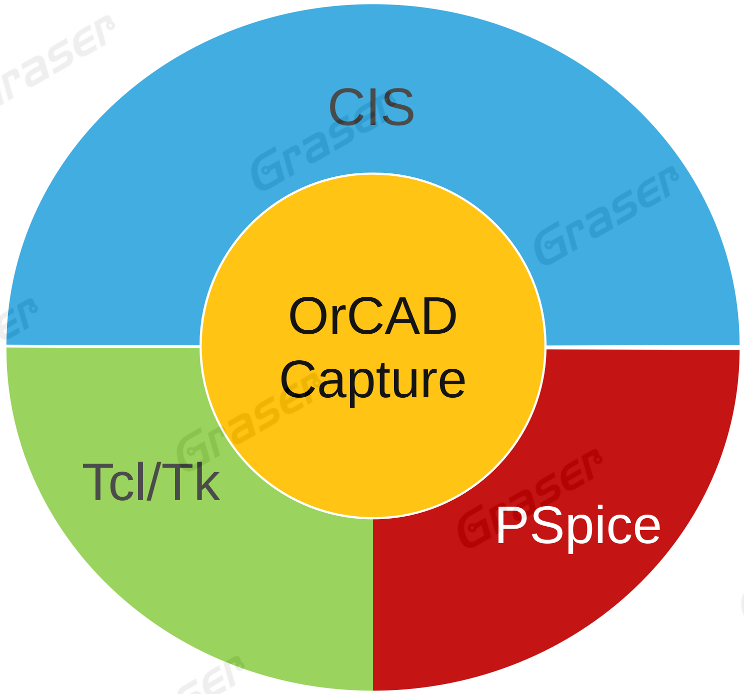
Topic

- 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



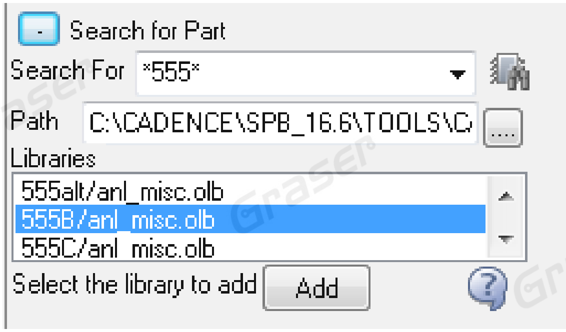
- Easy Model Creation by TCL
  - Source Model
  - Modeling Application

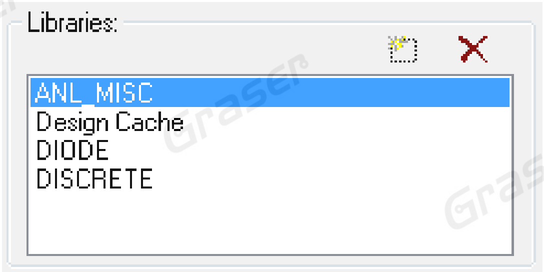
- Direct PSpice simulation driven process by your TCL

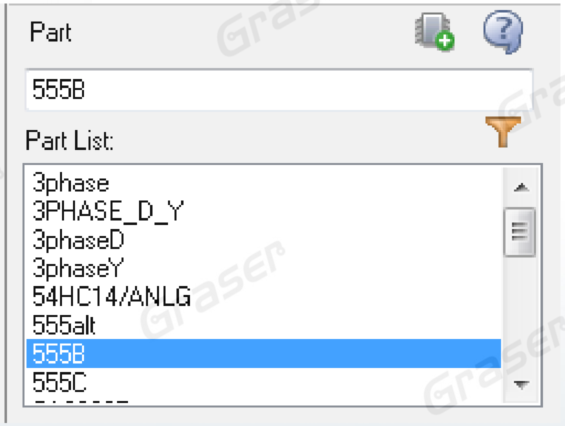
How to Get Symbol with Model in Capture

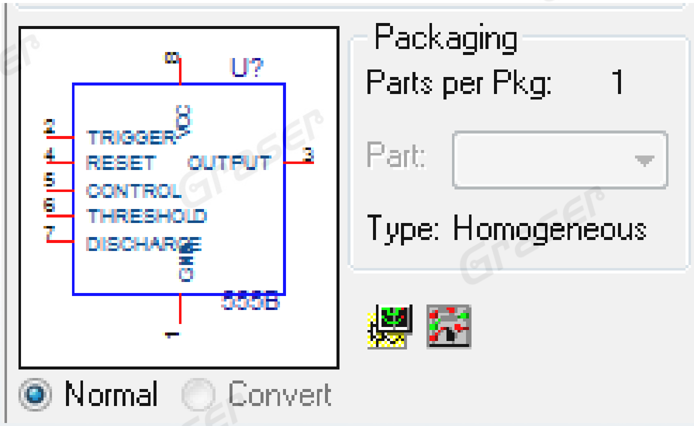
If we can’t make sure the part name

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



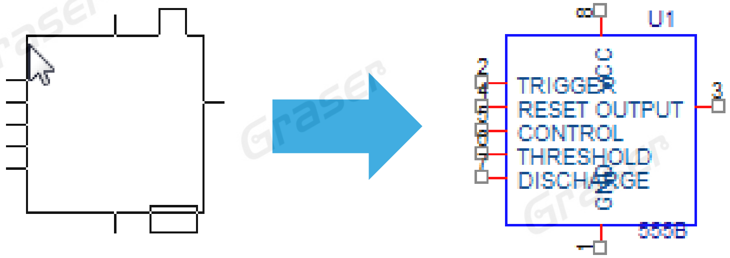




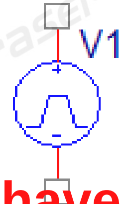


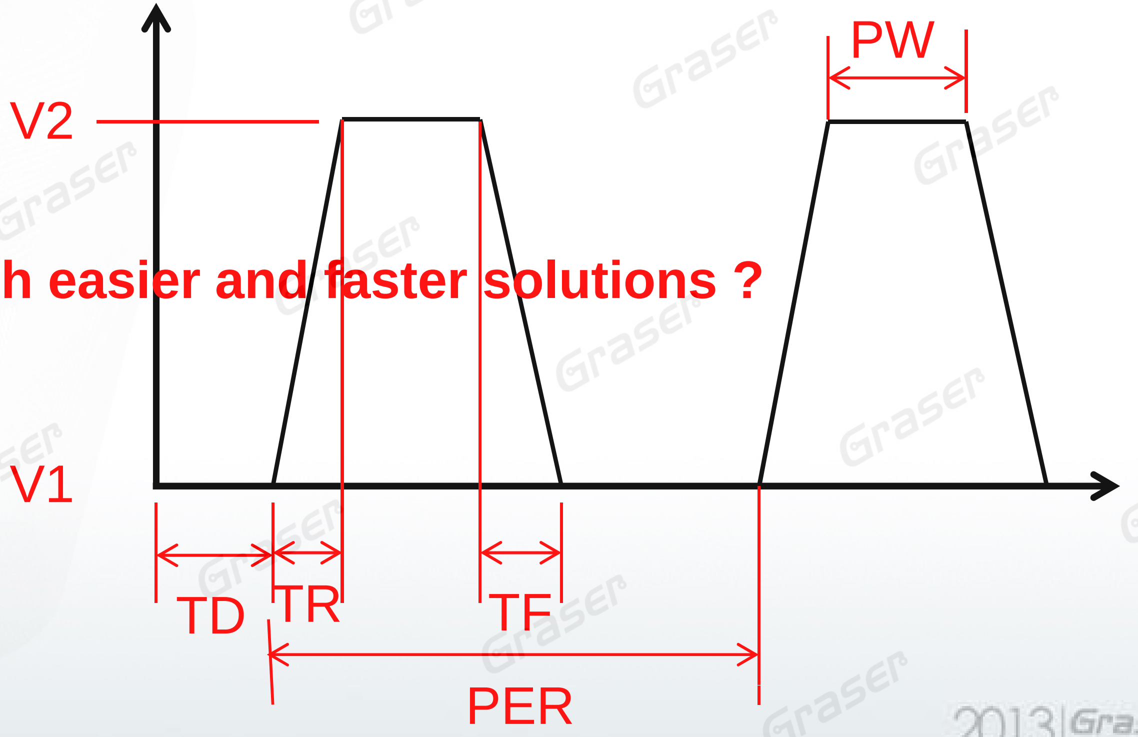
After Capture Symbols are placed in the Schematic

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



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

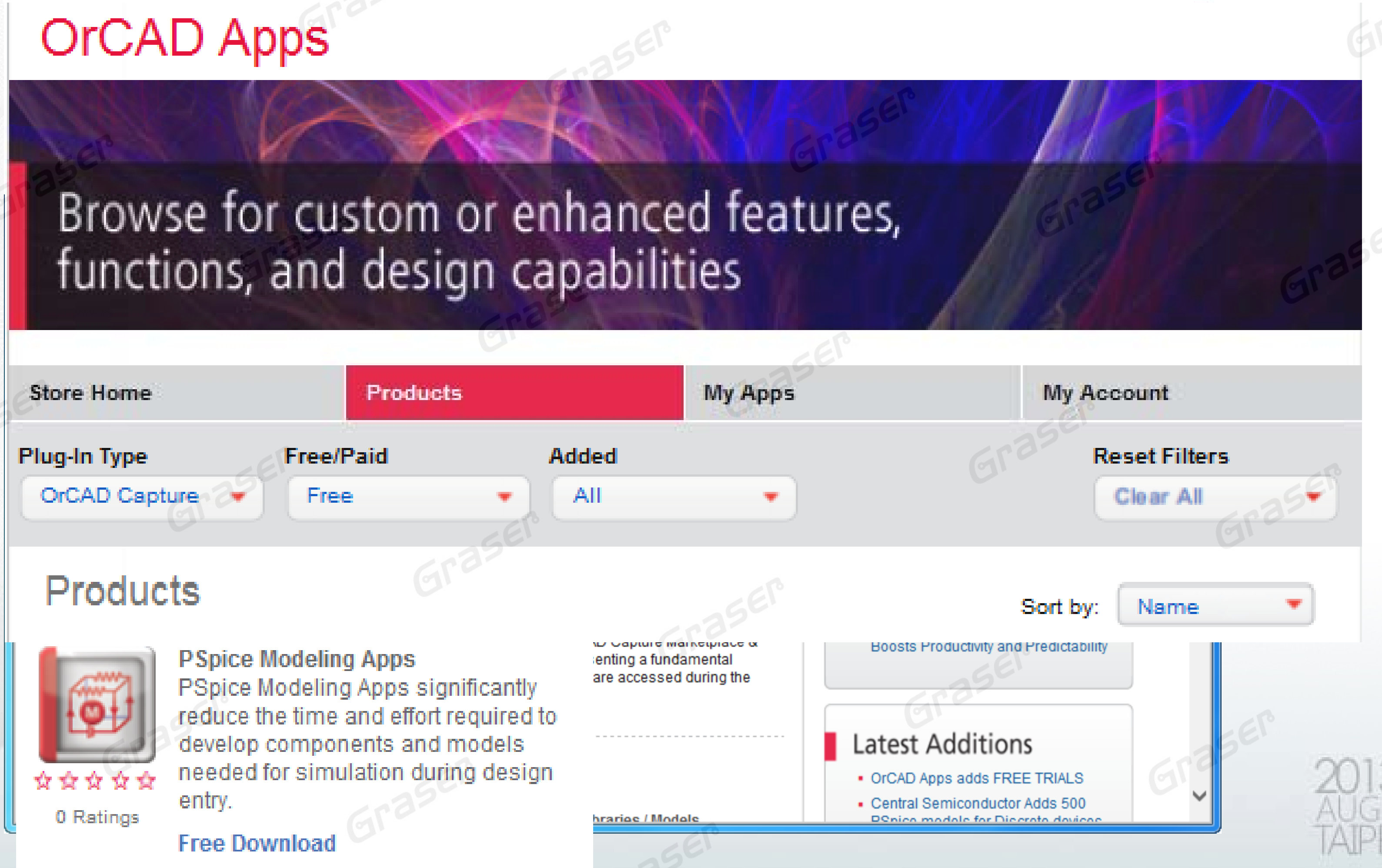




Do we have much easier and faster solutions ?

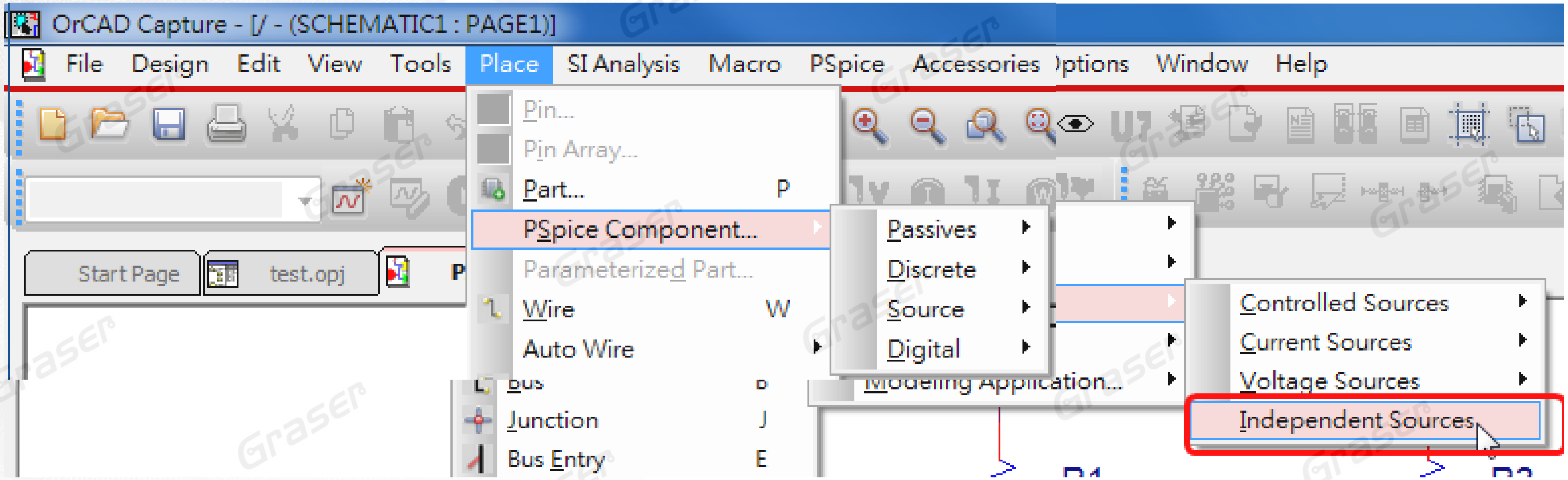
Download and Install Apps for PSpice Simulation (For Free)

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



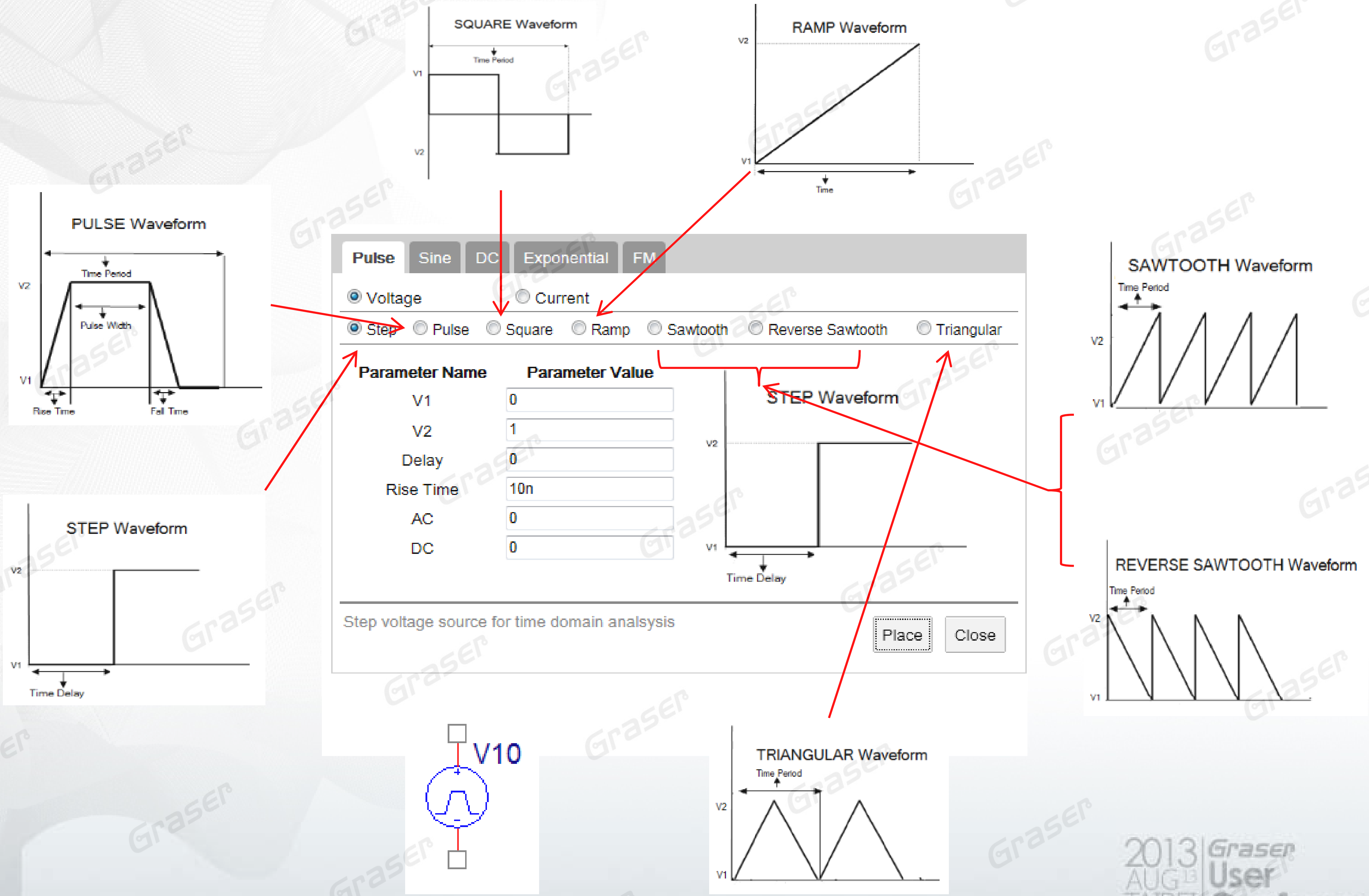
New Source Model Command Menu

From V16.6

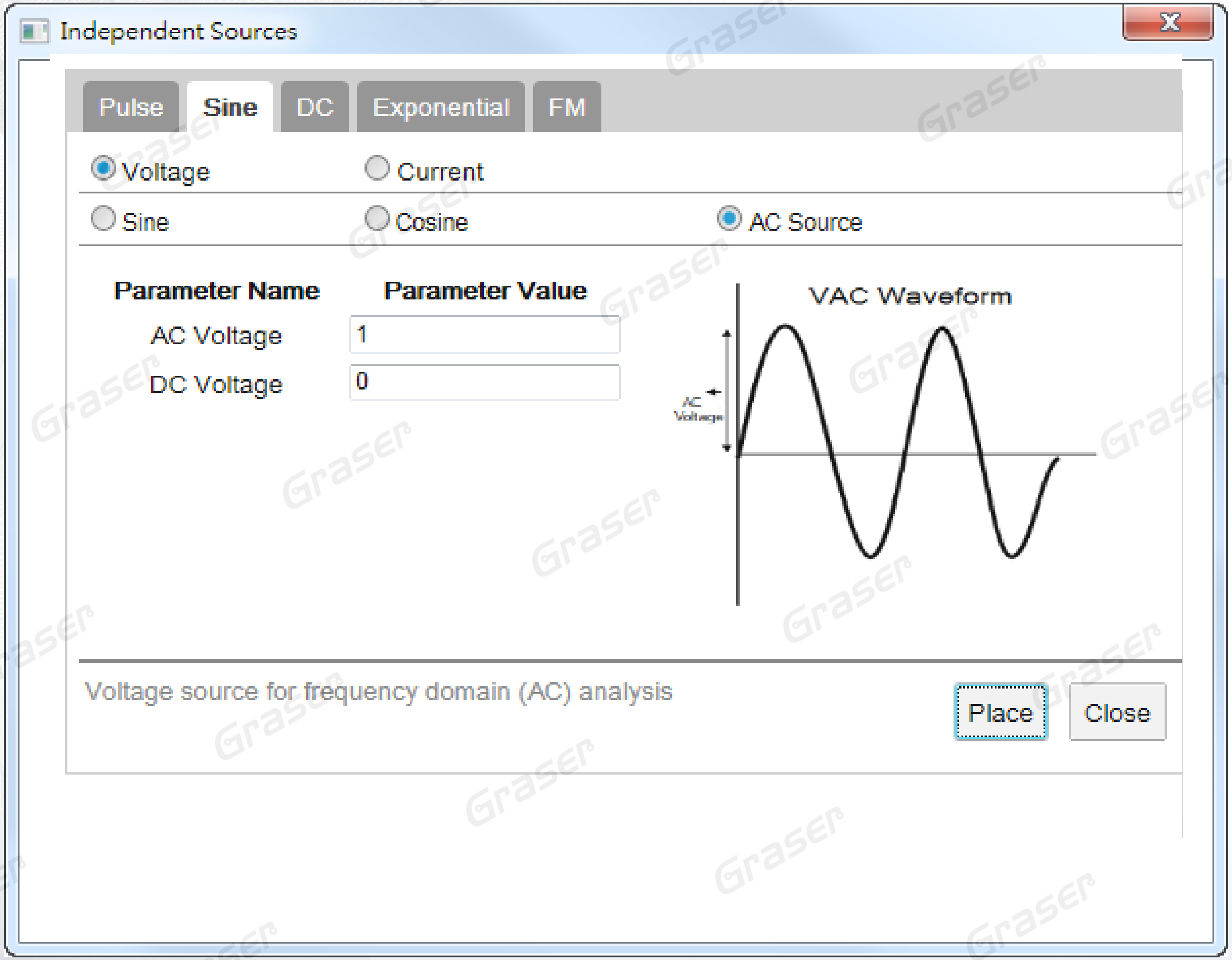


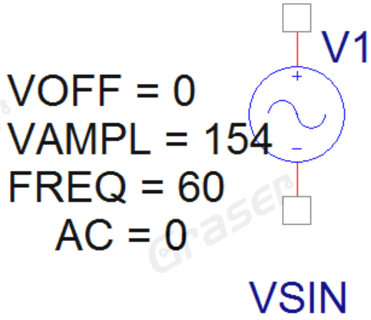
- 5 Type Source Model
  - 1. Pulse
  - 2. Sin Wave
  - 3. DC Source
  - 4. Exponential Waive
  - 5. FM

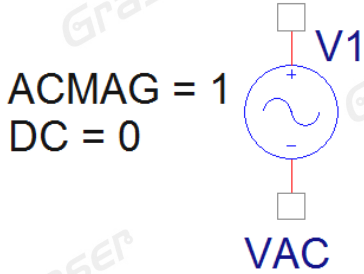
Pulse Source Model Generator



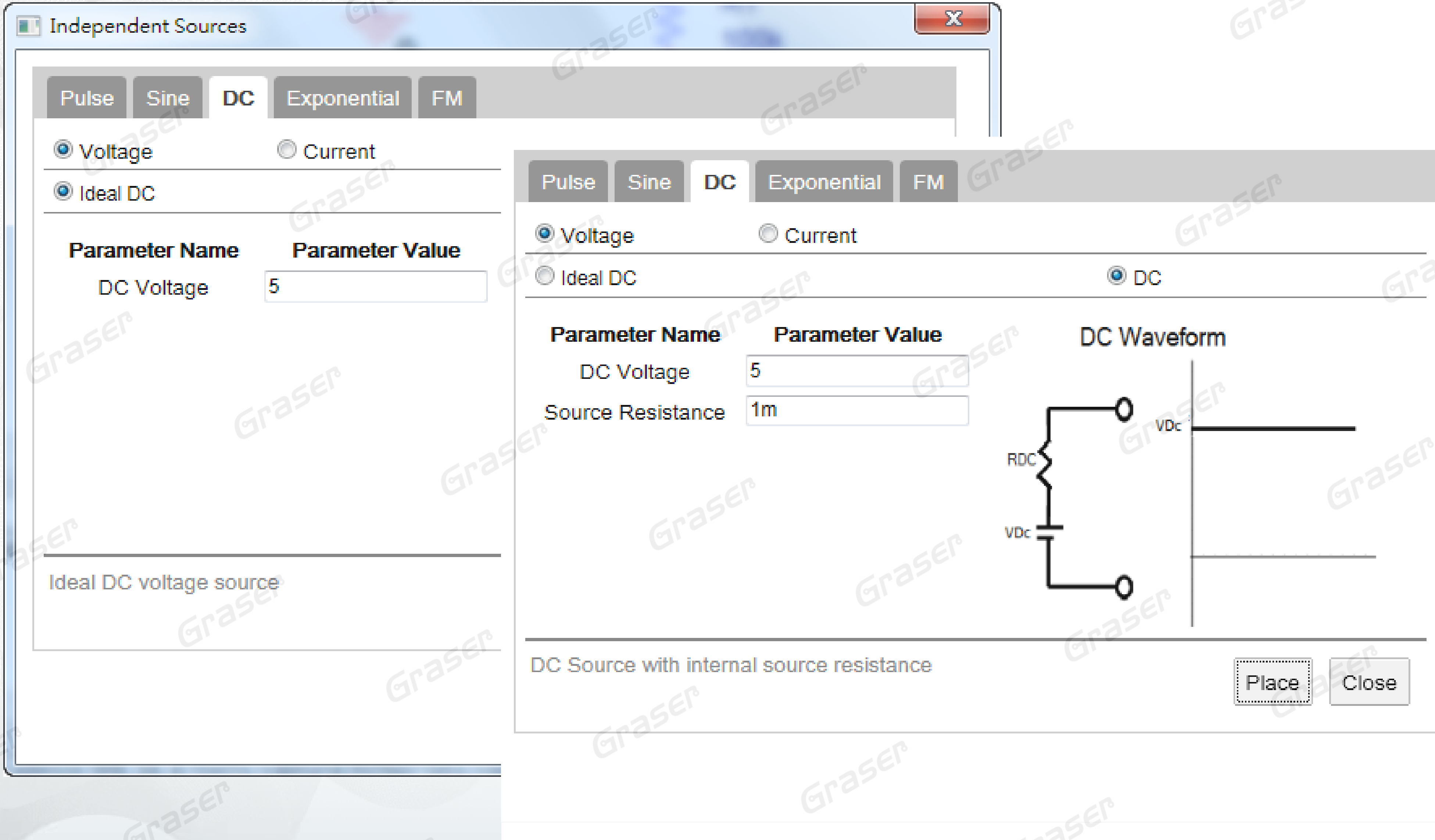
Sine Wave Model Generator



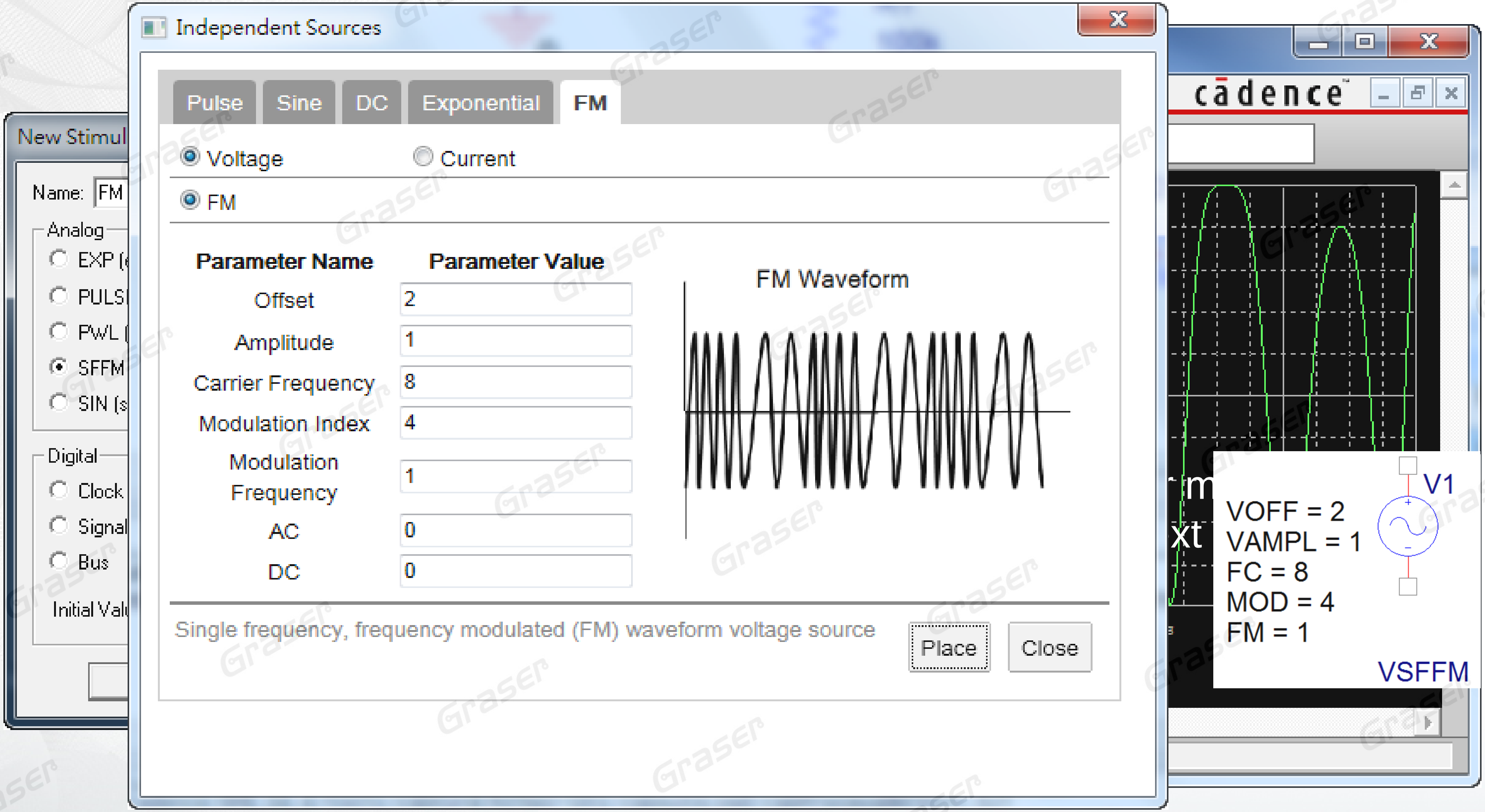




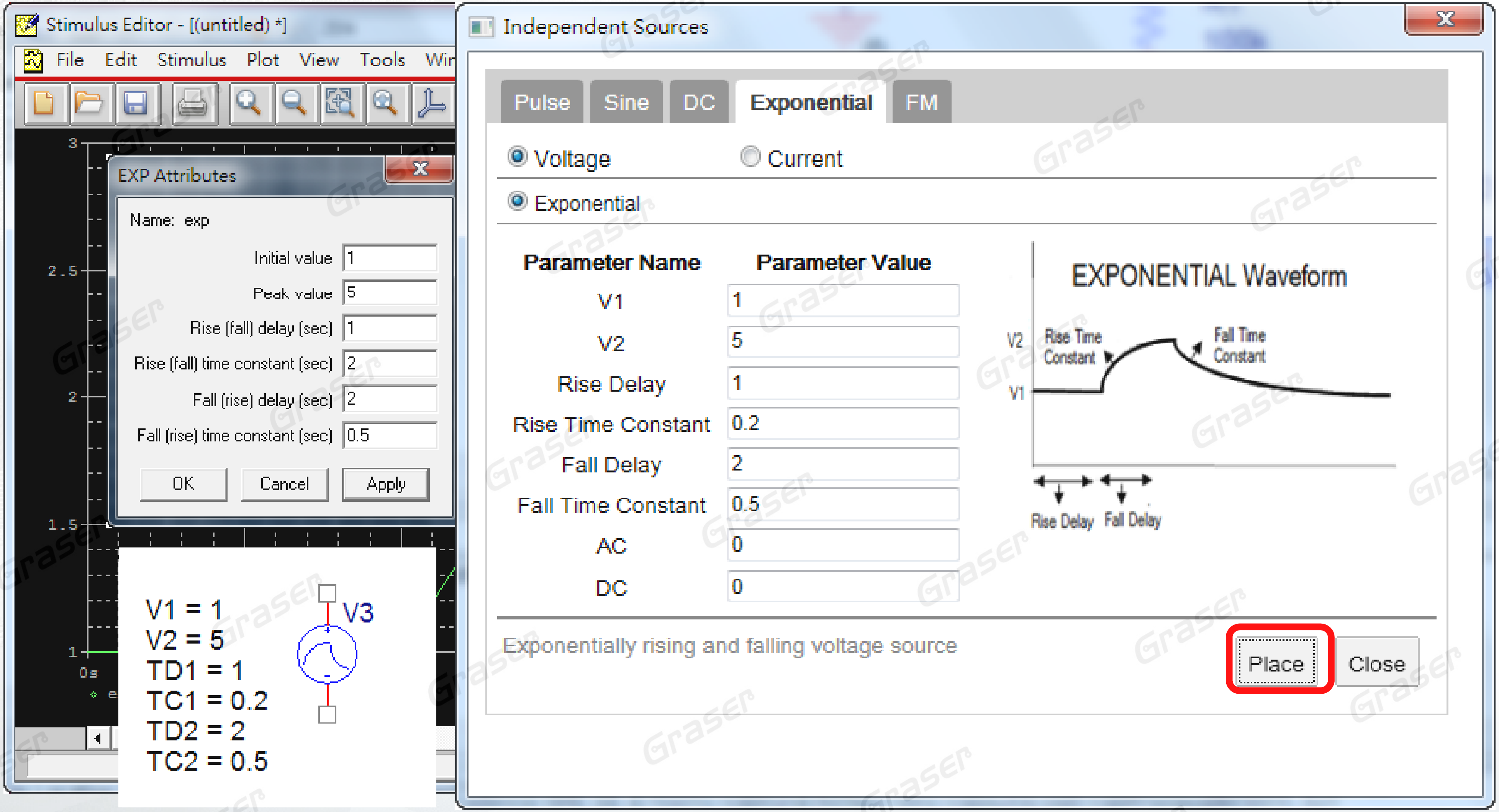
DC Source Generator



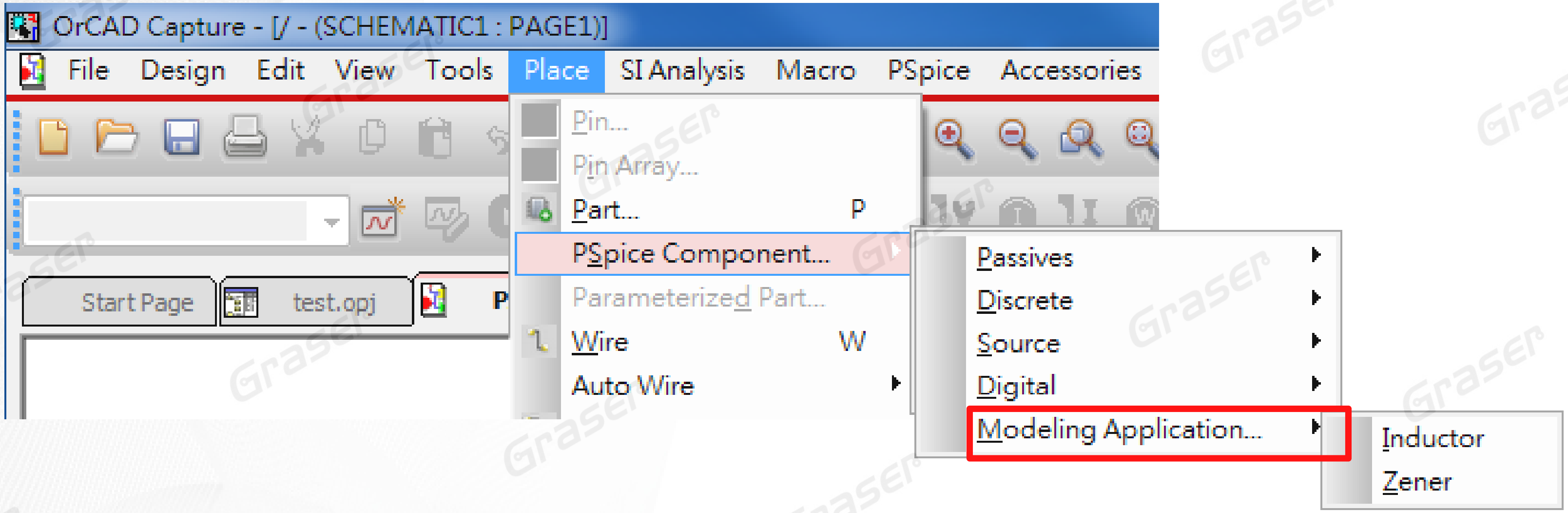
FM Waveform Generator by Stimulus Editor vs. Apps



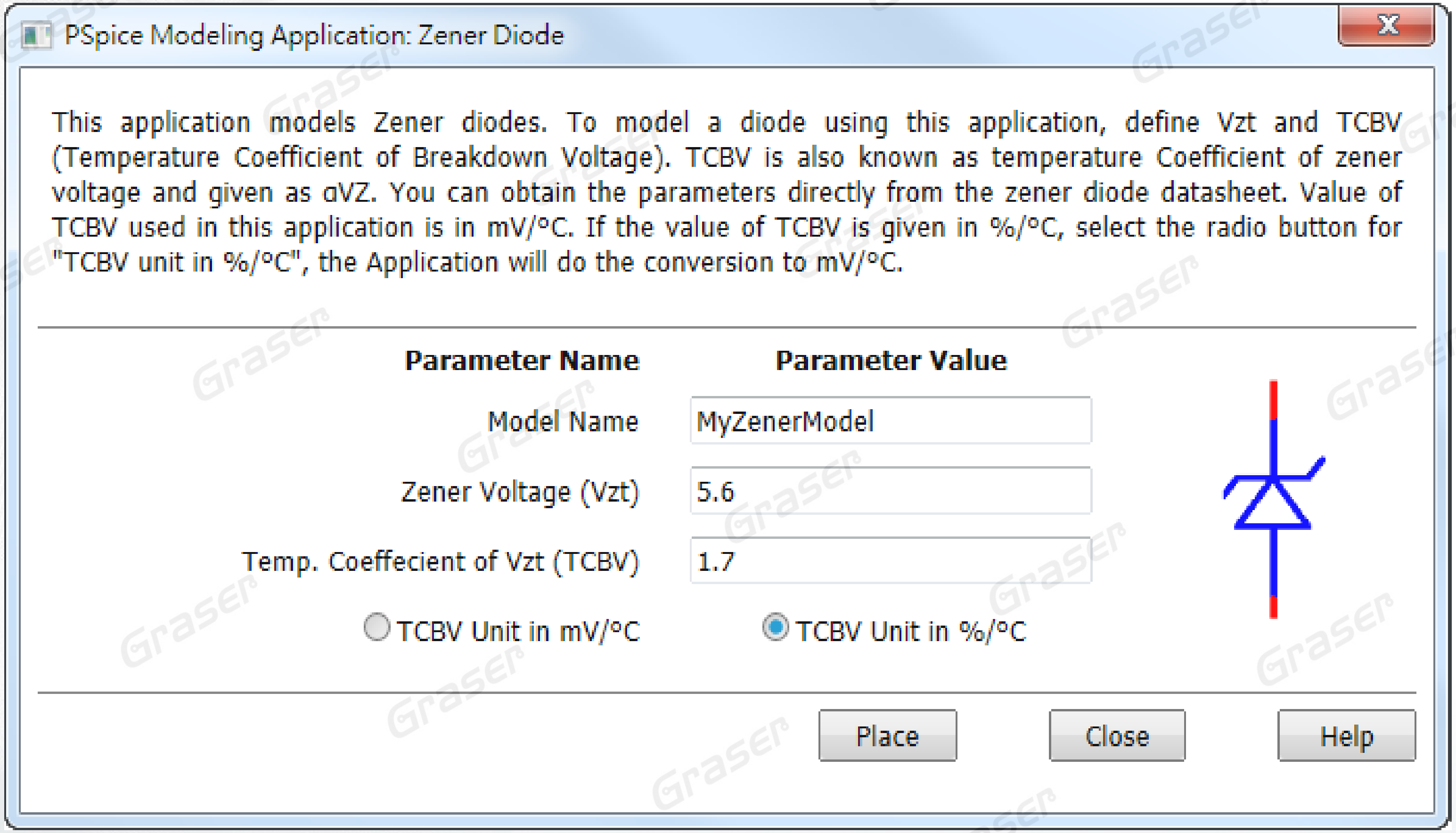
Exponential Waive Generator by Stimulus Editor vs. Apps



New Modeling Application from Apps



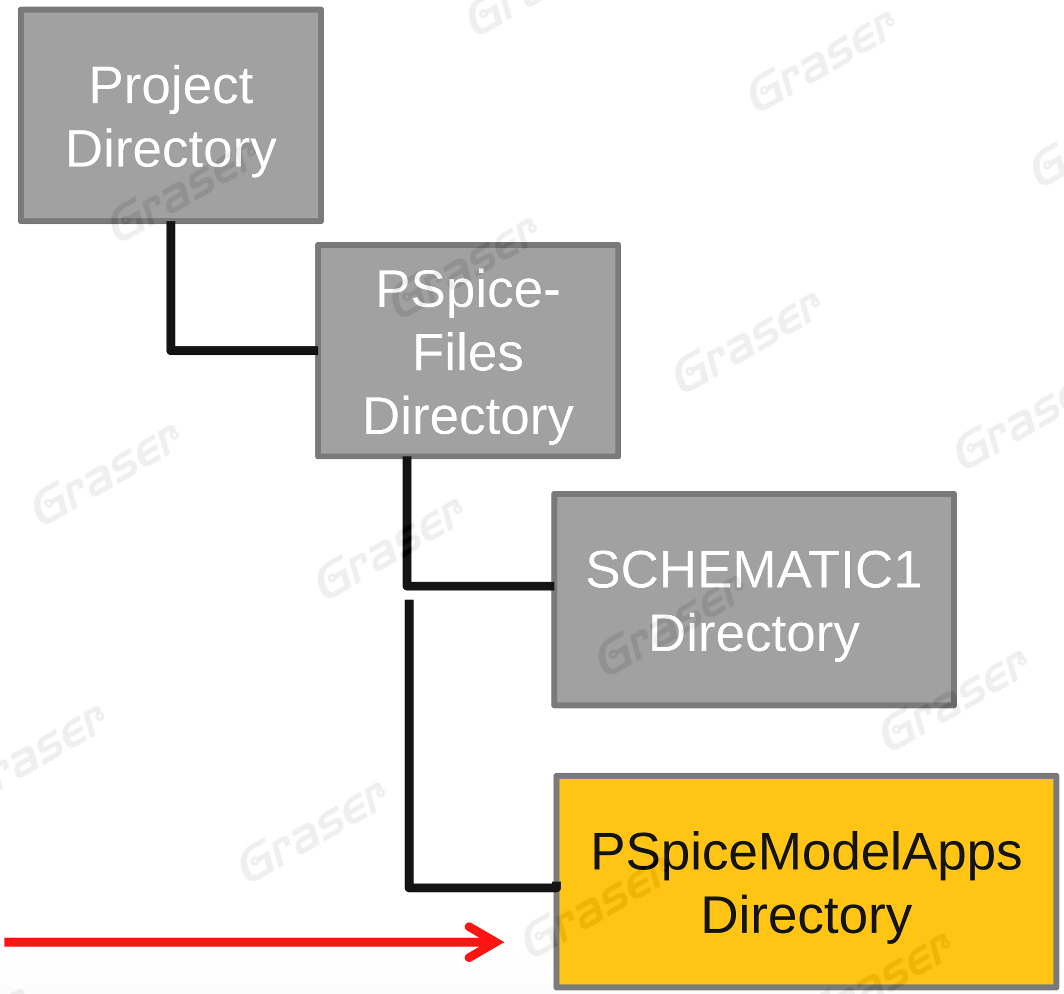
- **Non Ideal RF Inductor Modeling**
- **Zener Diode Modeling**



Combine Model Files

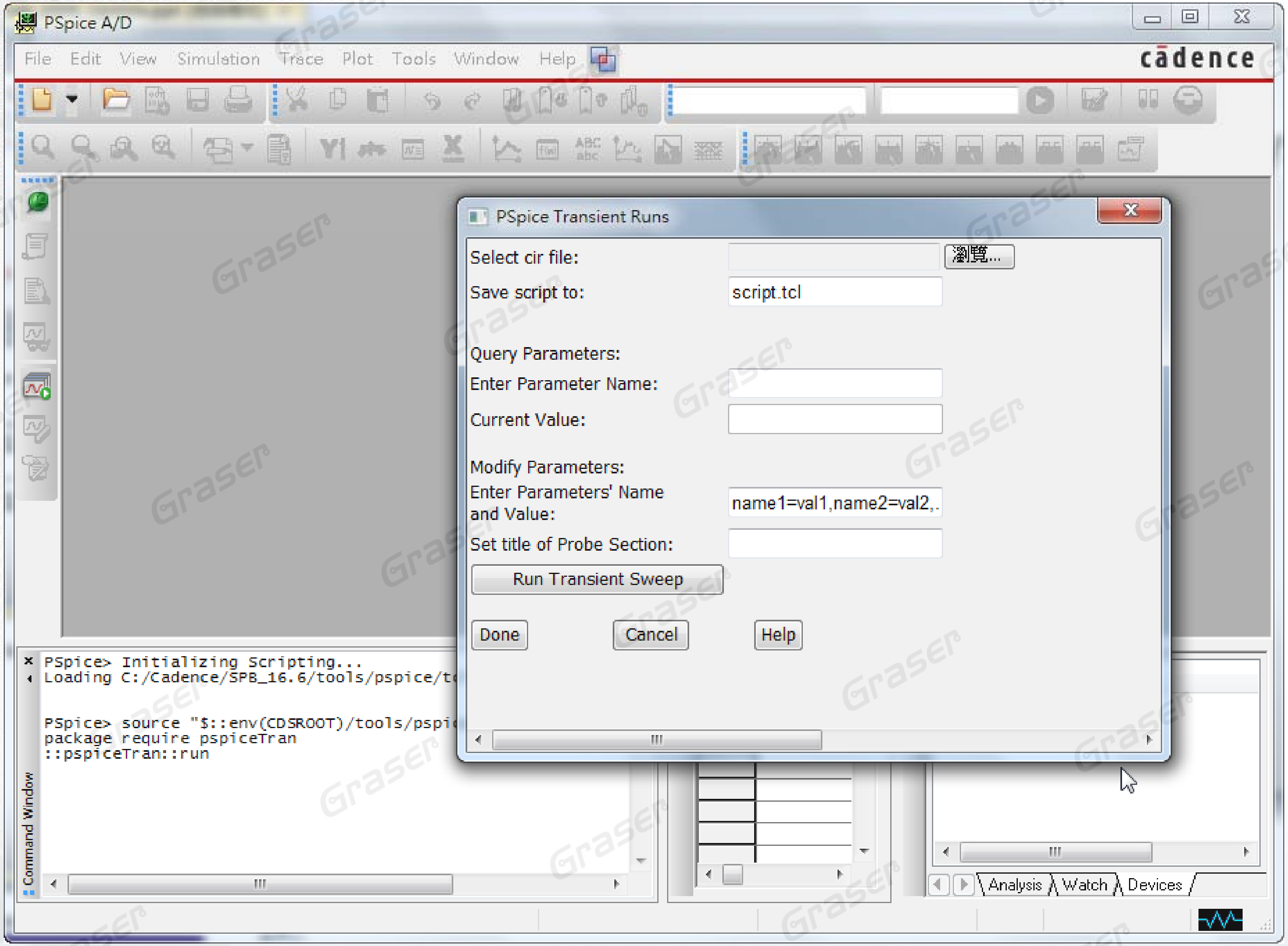
PSpiceModelApps_Include.lib

```
*PSpice Modeling Apps Index library File
*$
.SUBCKT MyZenerModel AN CAT
* Model generated on 2013 Apr 11
* MODEL FORMAT: PSpice
* MODEL Type: ZEN
+ params:
+ VZT=3.7
+ IZT= 20m
+ ZZT= 11
+ TCBV=1.7
+ IR= 1u
+ VR= 1
XinstMyZenerModel AN CAT model434
+ params:
+ vzt={VZT}
+ izt={IZT}
+ zzt={ZZT}
+ tcbv={TCBV}
+ ir={IR}
+ vr={VR}
.ends MyZenerModel
* - - - - - - - - - - - - - - - - - - - - - - - -
*$
.SUBCKT MyInductorModel IND P1 P2
* Model generated on 2013 Apr 11
* MODEL FORMAT: PSpice
* MODEL Type: IND_R
RDC P1 PI 10m
L_IND PI P2 100u
```



PSpiceModelApps_Include.lib

Direct PSpice Simulation Driven by Apps



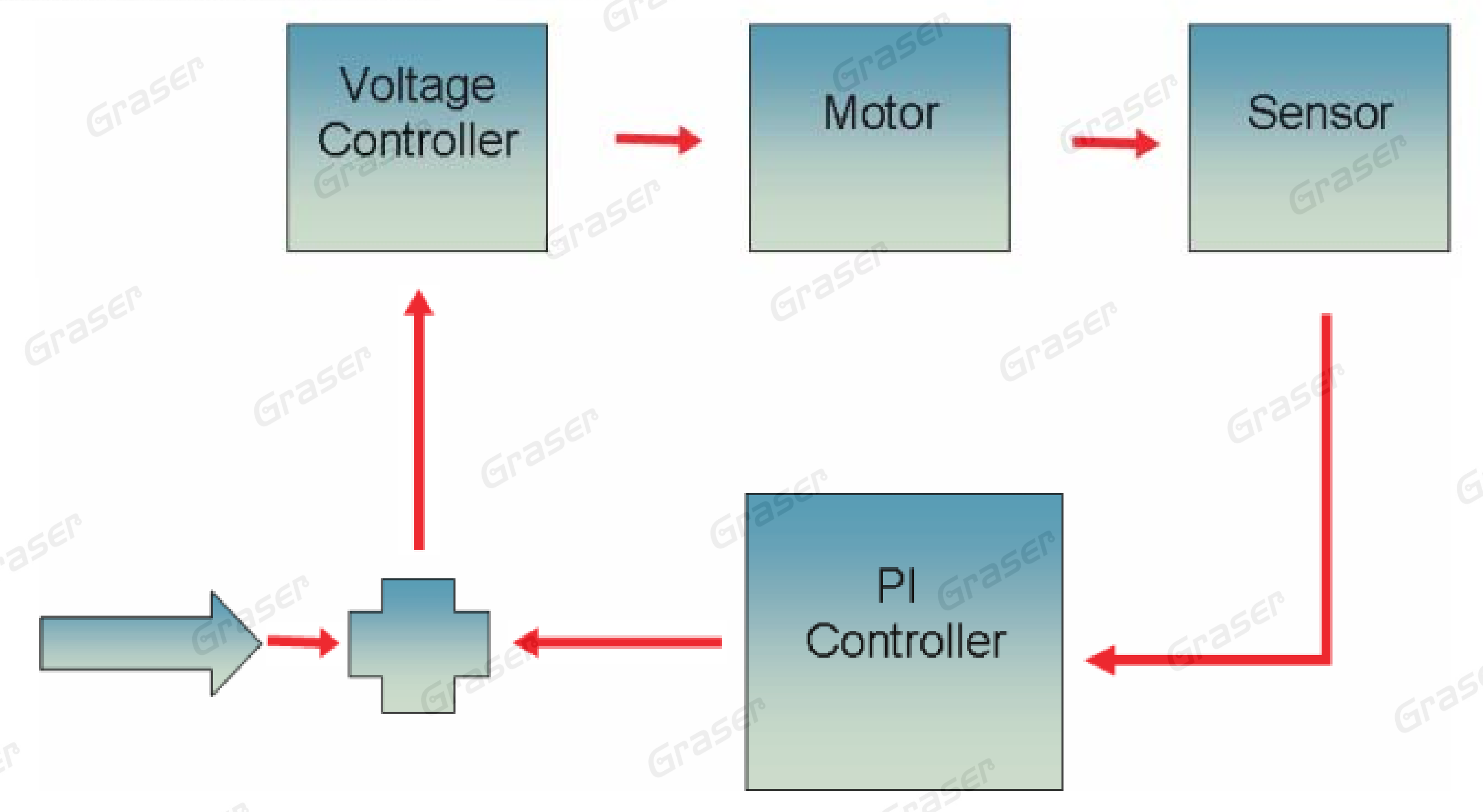
SLPS Option With Matlab

Multi-System Co-Simulation

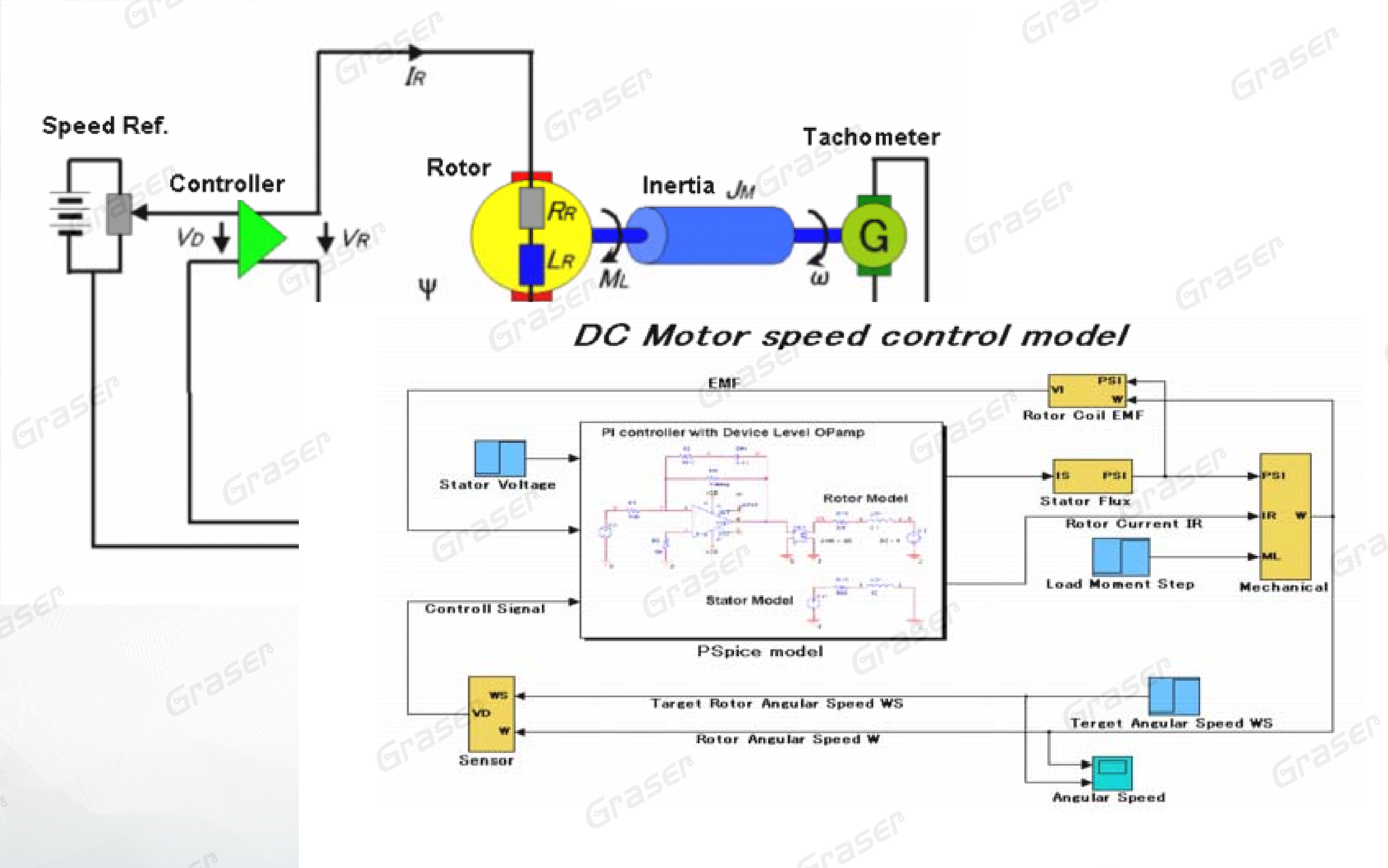
- Base on Mathworks Matlab

E-Car Control System Design

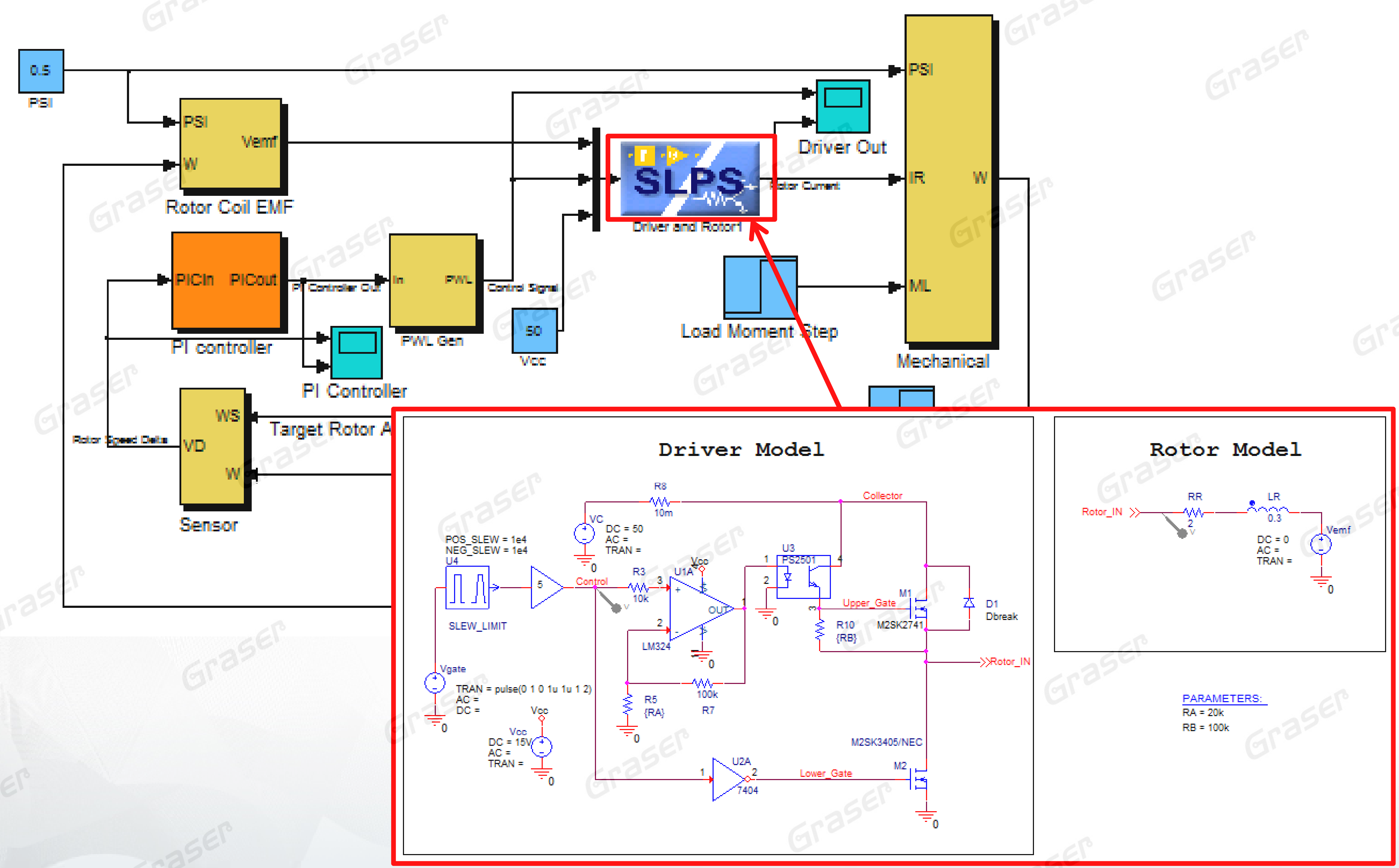
System Block

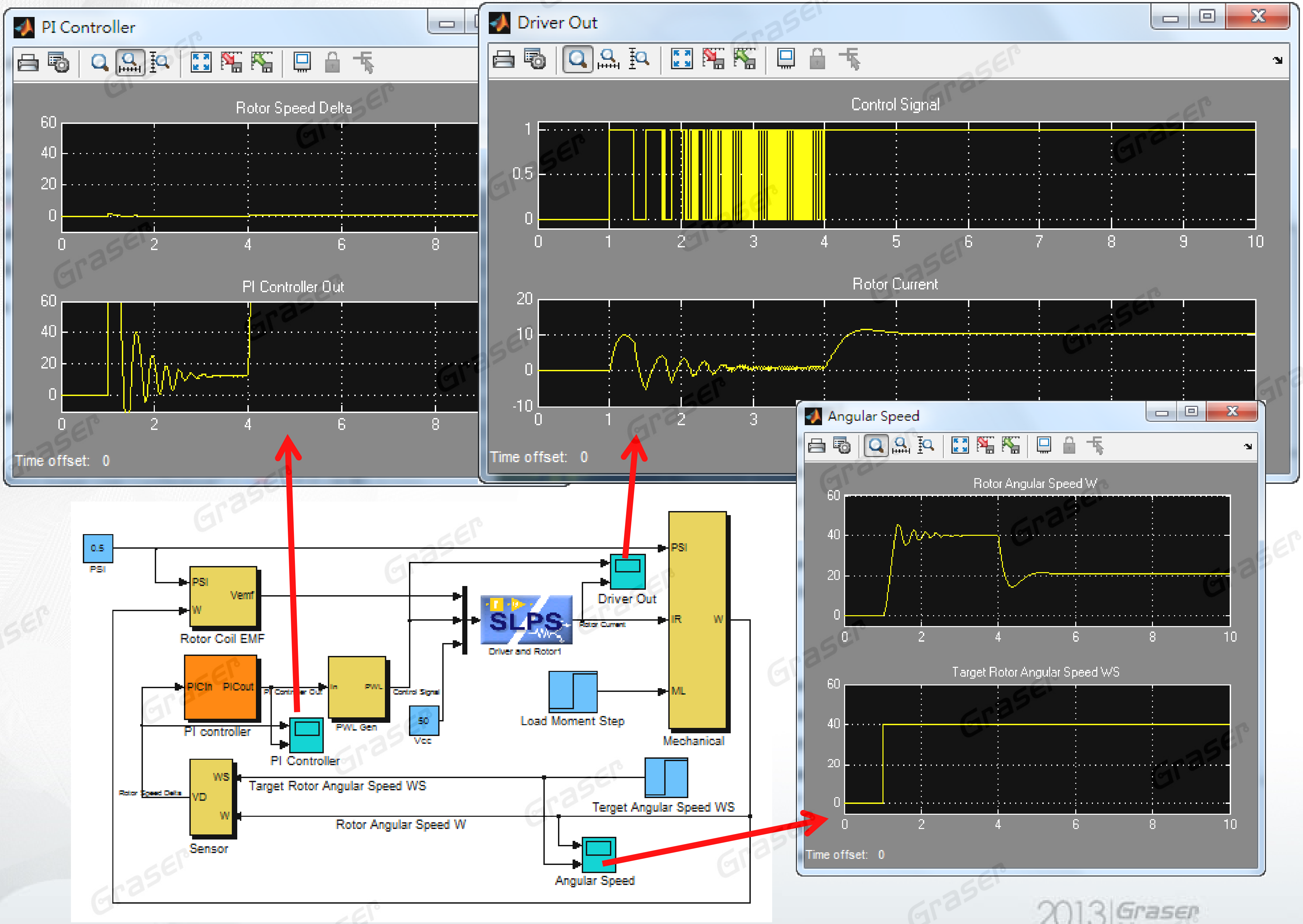


DC Motor Control Module



DC Motor Speed Control - Device(Driver)





2013 AUG 13 TAIPEI Graser User Conference