cadence®

PSpice Library Models Data Book

Product Version16.6
October 2012

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

Preface

Before you begin

This manual provides reference information needed to work with models in the libraries of PSpice. The chapters in this manual describe selected models, list model limitations, if any, specify the data sheet referred while developing the models, and provide application circuit examples with simulation results for key functions of the models.

PSpice library contains a large number of models under different categories organized as library files. This manual does not explain each model but has comprehensive reference material for complex models shipped with PSpice that are mostly representative of the different categories, which include:

- PSpice A/D
- Allegro AMS Simulator

Structure of Manual

The chapters of this book are organized according to specific application areas to categorize the models into the following areas:

- Power Management Controller IC Models
- Solid State Relay
- Resolver
- Battery
- Special Components

This manual assumes that you are familiar with basic PSpice simulation terminologies, including how to use PSpice, setting up analysis, and creating schematics.

Note: Refer to the *PSpice User Guide* for information about using PSpice to set up analysis, and the user guides for the design entry tool you use for information on creating schematics.

1

Power Management Controller IC Models

This chapter starts with an introduction to Power Management controller IC models present in the PSpice library available with the Cadence installed hierarchy. These models can be categorized into the following broad categories

- DC-DC Converters with Internal MOSFET/Switch
  - MAX8566
  - IR3840
  - MAX8855
  - MAX5090
- Off Line PWM Controller
  - UC3841
- Charge-Pump-Based DC-DC Controller
  - TPS6050x
- MOSFET Driver
  - MIC4123
- Off Line Switches
  - LNK304

DC-DC Converters with Internal MOSFET/Switch

This section describes the *DC-DC converters with internal MOSFET/Switch* models from various semiconductor manufacturers.

MAX8566

This is a high-efficiency, 10A, PWM Internal-Switch Step-Down Regulator from Maxim Semiconductor. This model is developed based on the datasheet for MAX8566 available at http://datasheets.maxim-ic.com/en/ds/MAX8566.pdf.

This device is used for low voltage, on-board, or point-of-load application.

Following are the key features of this device:

- 10A Output PWM Step-Down Regulator with Internal MOS
- Input supply variation from 2.3V to 3.6V
- Adjustable Output from 0.6V to (0.87 x VIN)
- Programmable Soft-Start Time
- Power-Good Output
- 250kHz to 2.4MHz Adjustable Frequency.

Using MAX8566 in Circuit

Figure 1-1 shows the typical application circuit from vendor data sheet.

Figure 1-1 MAX8566 Application in Datasheet

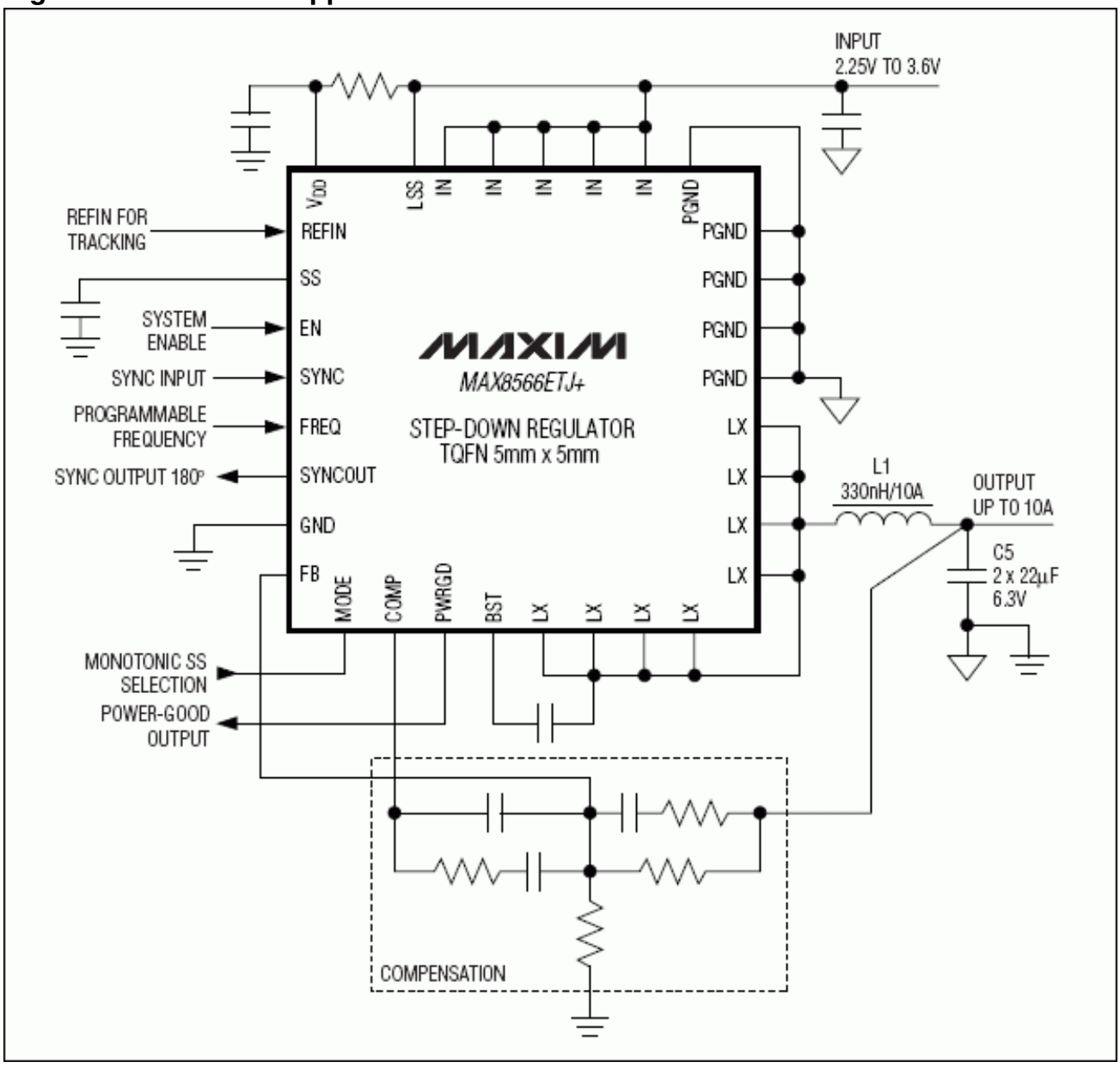


Figure 1-2 shows an application circuit for 5Amp, 1.5Voutput from 2.5-3.2V input DC bus, step down buck controller using MAX8566 model.

Figure 1-2 Simulation Circuit for MAX8566

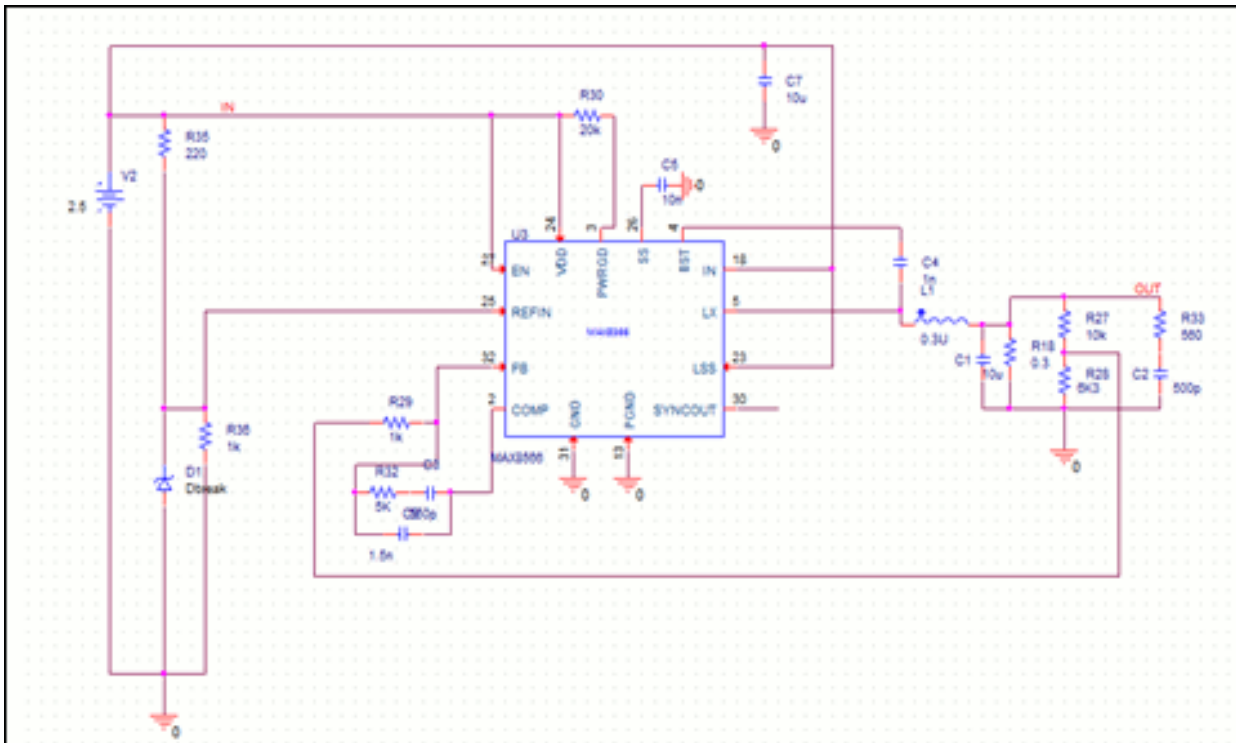


Figure 1-3 shows the voltage waveform and current across output filter inductor from PSpice Simulation of application circuit described in Figure 1-2 above. 2 MHz switching frequency is used for this simulation.

Figure 1-3 Voltage Waveform and Current across Output Filter

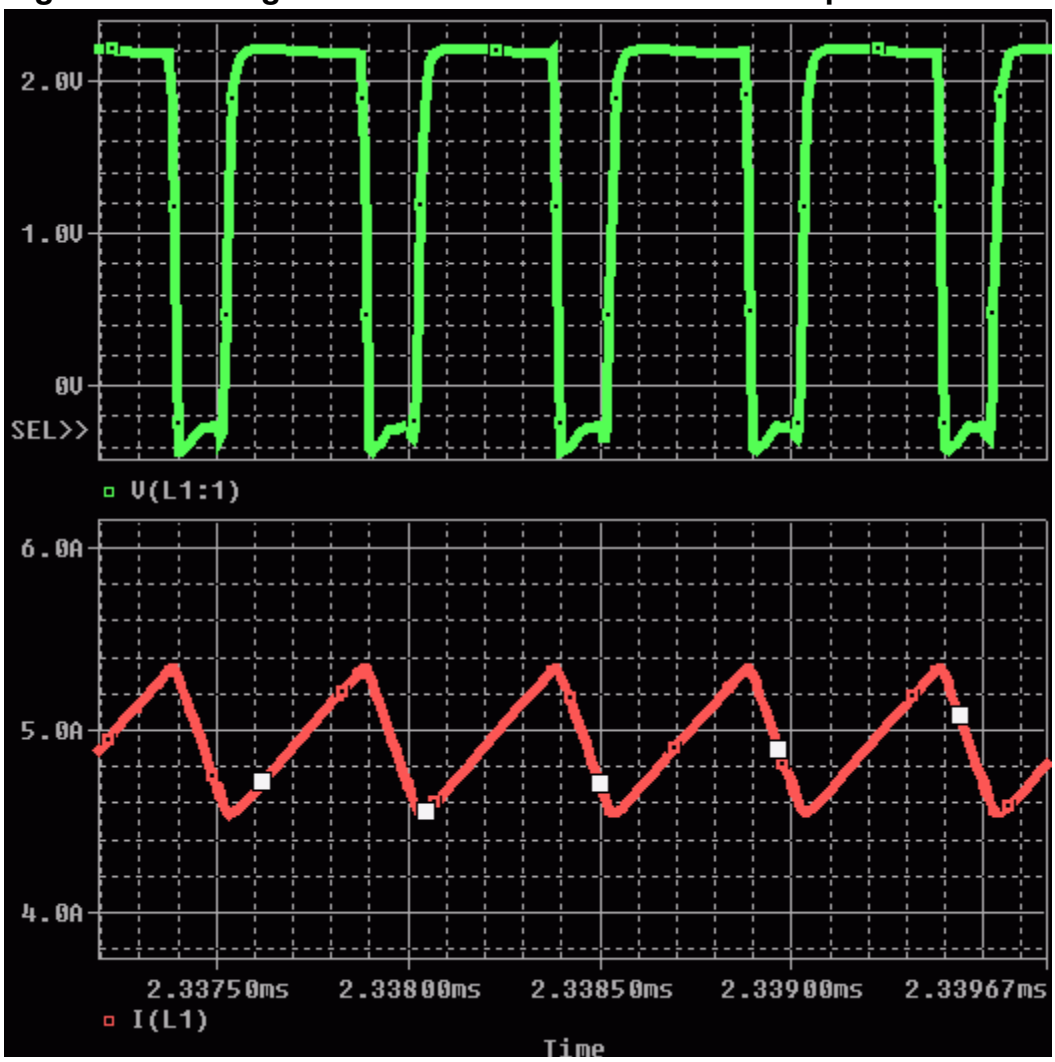


Figure 1-4 shows simulation of the programmable slow-start behavior of this model. Slow start time can be programmed by appropriately choosing a capacitor value connected with the *SS* pin. Higher the capacitance value, slower is the output voltage buildup. The two output waveforms are generated with two different slow start capacitors [C5], leading to different rates of rise of output voltage. The green waveform has lower slow start capacitance value.

Figure 1-4 Rate of Rise of Output Voltage

Limitations

The internal oscillator section of MAX8566 is modeled so that power supply, external resistor, and capacitor variations have no affect on the output voltage. To adjust oscillator time period (PWM output pulse time period), modify the `PER` property on the symbol instance. For example, to set `500KHz` output frequency, set `PER` to `2u` or `2E-06`.

Other limitations are:

- Temperature variations are not modeled
- Sync pin is not modeled

Locating the Model

The MAX8566 simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/pspice/swit_reg.olb`
- Allegro Design Entry HDL: `<INSTALLDIR>/share/library/swit_reg`

Following are some models similar to MAX8566 in the `swit_reg` library:

- MAX5090A
- MAX5090B
- MAX5090C
- MAX8566
- MAX8643
- MAX8643A
- MAX15038
- MAX15039

IR3840

IR3840 is a fully integrated and highly efficient DC/DC synchronous Buck regulator. The MOSFETs co-packaged with the on-chip PWM controller make IR3840 a space-efficient solution, providing accurate power delivery for low output voltage applications.

IR3840 is a versatile regulator which can be programmed for slow start, switching frequency, and current limit while operating in a wide input and output voltage range. The switching frequency is programmable from 250kHz to 1.5MHz.

It also features important protection functions such as hiccup current limit. This model is developed based on datasheet for IR3840M available at http://www.irf.com/product-info/datasheets/data/ir3840m.pdf.

Following are the key features of this model:

- Programmable Switching Frequency up to 1.5MHz
- Programmable Over Current Protection (Hiccup)
- PGood output
- Precision Reference Voltage (0.7V, +/-1%)
- Programmable Soft-Start
- Enable Input with Voltage Monitoring Capability

Using IR3840 in circuit

Figure 1-5 shows the typical regulator application circuit diagram for IRF3840.

Figure 1-5 : IRF3840 Regulator Application Circuit

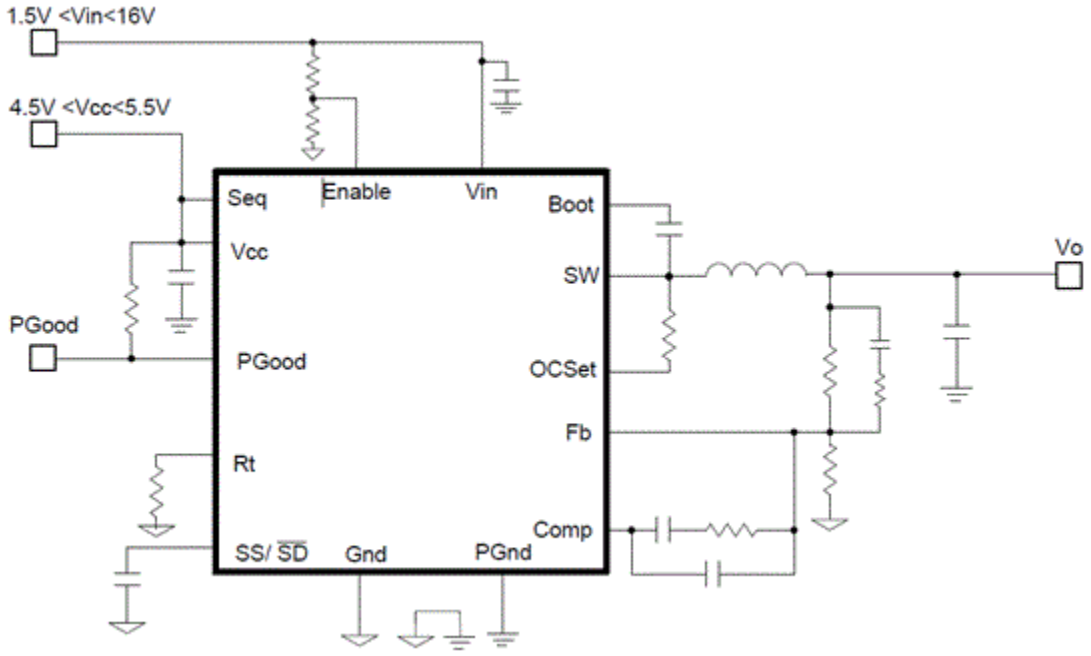


Figure 1-6 shows the typical regulator application circuit diagram based on this model for 3.3V/10A output.

Figure 1-6 Regulator Application of IRF3840 for 3.3v/10A Output

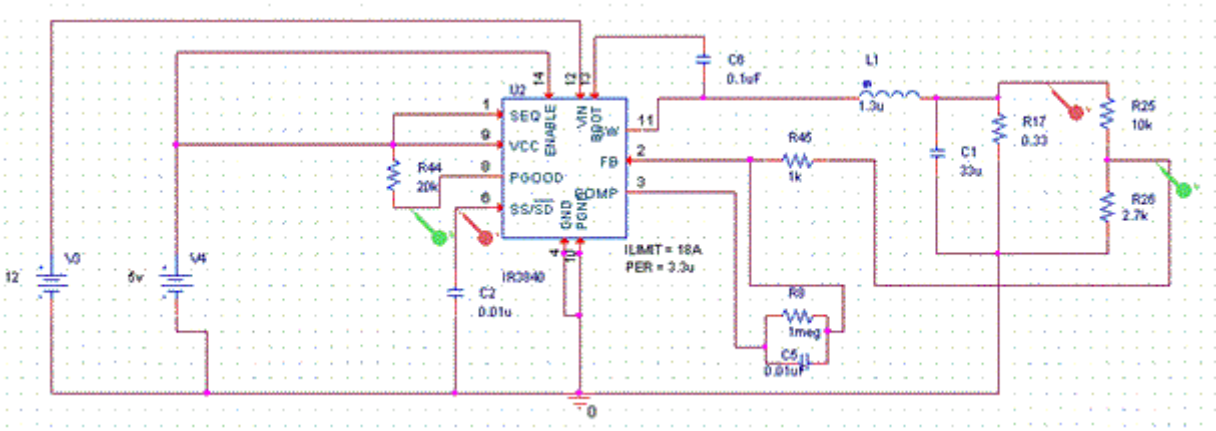
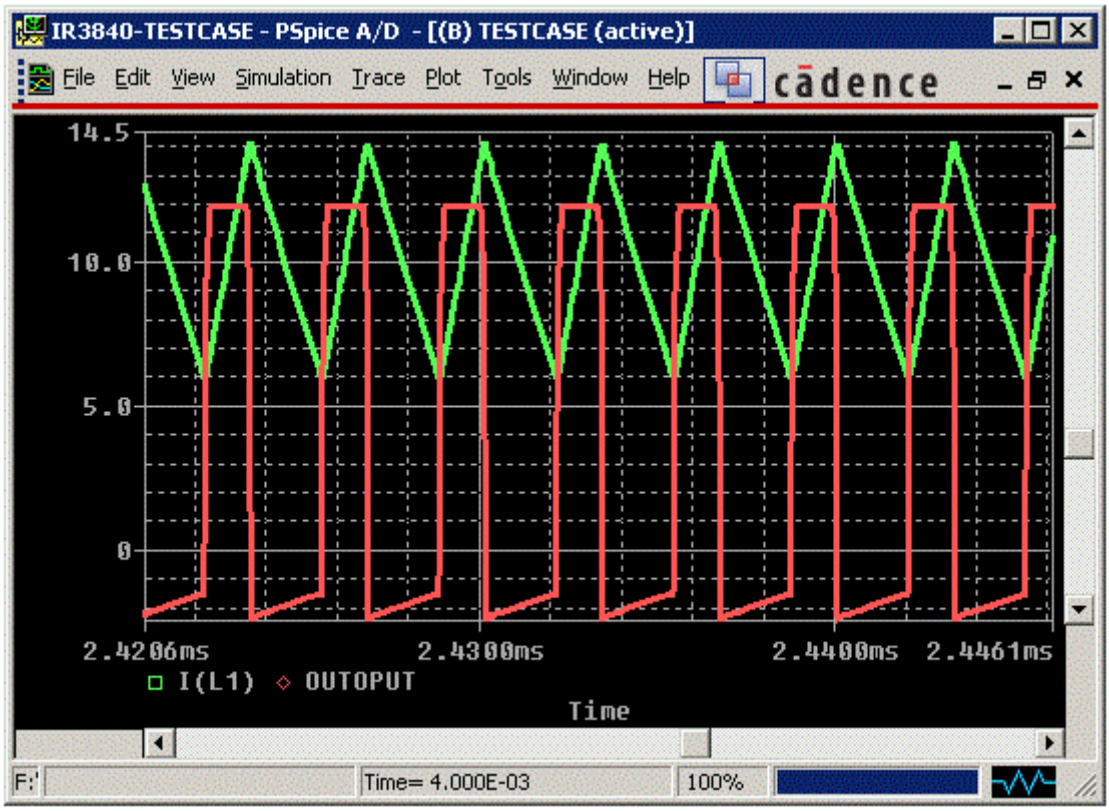


Figure 1-7 shows voltage waveform at switch terminal and current into output filter inductor.

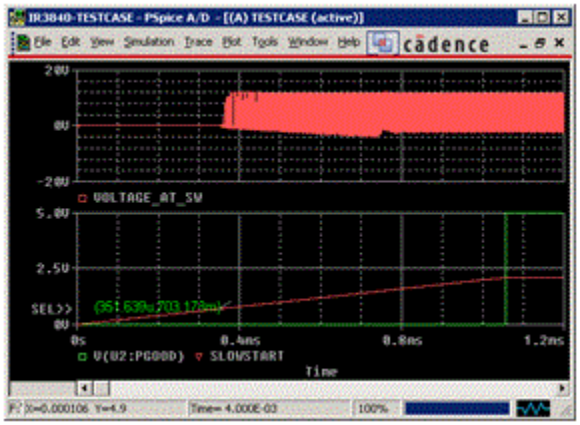
Figure 1-7 Voltage Waveform at Switch Terminal

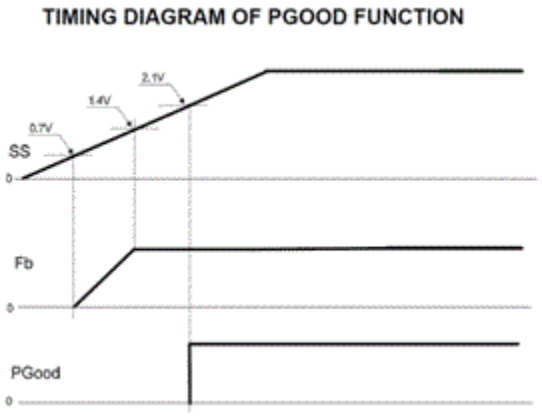


Special Pins and Functions

- **POWERGOOD PIN**: Figure 1-8 shows the status of power good pin, slow start pin, and startup sequence from the device datasheet and simulation results side by side.

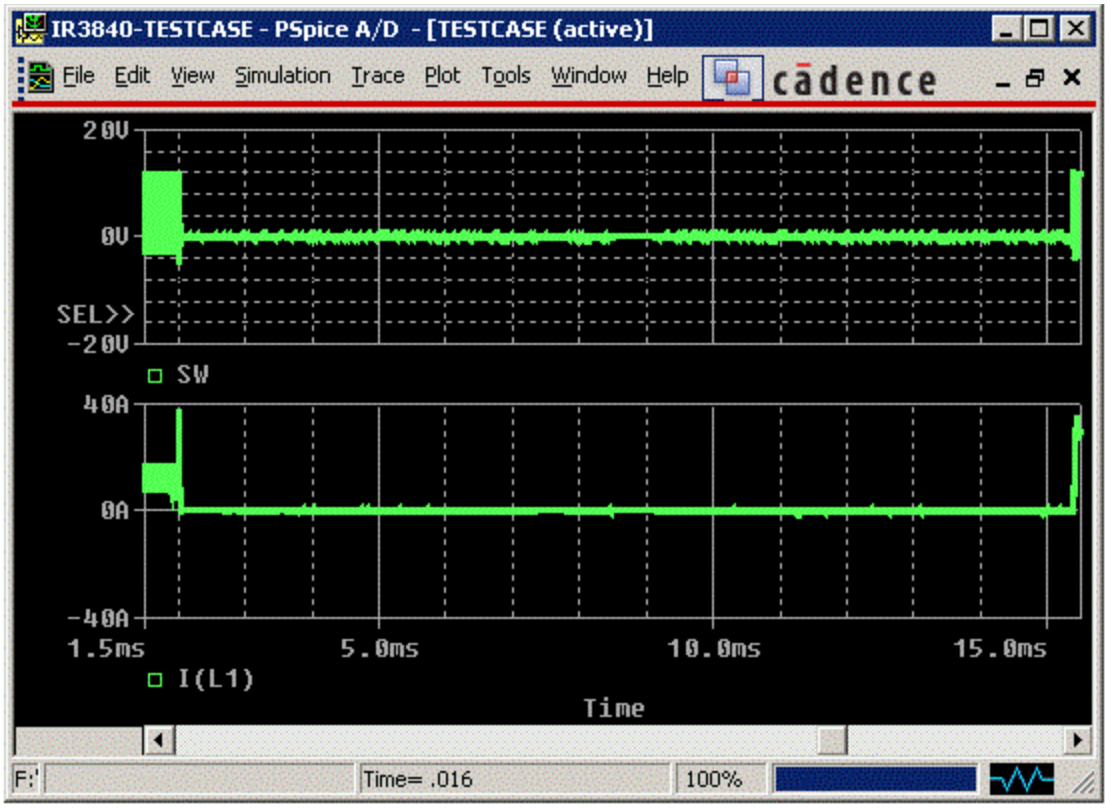
Figure 1-8 POWERGOOD Pin Status





- **ENABLE PIN**: This model also includes the functionality of ENABLE pin. Pulling this pin to any voltage below 1.2V disables the output.
- **Over-current Protection**: This model also includes functionality of over-current protection in hiccup mode. Over-current threshold is set by setting the value of *ILIMIT* parameter on the IR3840 symbol instance in schematic editor. For example to set `ILIMIT = 15A` set the *ILIMIT* parameter to `15`. Once the overcurrent is detected, controller goes into shutdown mode and waits for 4096 cycles before restarting. For example, if over-current is detected at 1mSec simulation time point, controller will shutdown and it will wait for 4098*3.3u sec (where 3.3u sec is time period of oscillator) that is 13.52mSec before restarting. Thus it will restart at ~15.52mSec.In Figure 1-9 the bottom waveform indicates over-current condition, while top waveform shows voltage at the SW pin. It goes into shutdown at 2m and restarts at 15.5m.

Figure 1-9 Over Current Condition



- **Setting up oscillator frequency**: You can set the oscillator frequency by modifying the value of `PER` property on IR3840 symbol instance in schematic. Value of this property should be `1/FREQUENCY`; for example to set 1Meg HZ as switching frequency, value of `PER` property should be set to `1u`.

Limitations

Thermal shutdown and Voltage sequencing functionality are not modeled.

Locating the Model

The IR3840 simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/PSpice/switch_reg.olb`
- Allegro Design Entry HDL `<INSTALLDIR>/share/library/switch_reg`

MAX8855

The MAX8855 dual step-down regulator is capable of delivering up to 5A at each output. The device operates from a 2.35V to 3.6V supply, and provides output voltages from 0.6V to 0.9 x VIN, making it ideal for on-board point-of-load applications like ASIC/CPU/DSP/DDR power supplies.

This model is developed based on the datasheet for MAX8855 available at http://datasheets.maxim-ic.com/en/ds/MAX8855.pdf.

Following are the key features of this device:

- Dual, 5A, PWM Step-Down Regulators
- 27$m\Omega$ On-Resistance Internal MOSFETs
- Adjustable Output from 0.6V to 0.9 x VIN
- Soft-Start Reduces Inrush Supply Current
- 0.5MHz to 2MHz Adjustable Switching
- Individual Enable Inputs and PWRGD Outputs

Using MAX8855 in Circuit

Figure 1-10 shows a typical application circuit from vendor datasheet. Here MAX8855 is used to deliver two voltage output 1.2V and 1.5V from a voltage bus of ~3V.

Figure 1-10 MAX8855 Application in Datasheet

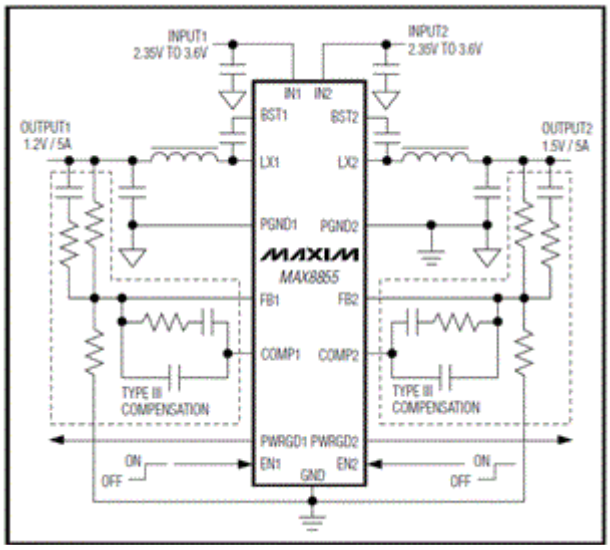


Figure 1-11 shows the simulation circuit using MAX8855 regulator designed to give 1.2V and 1.8V.

Figure 1-11 Simulation Circuit for MAX8855

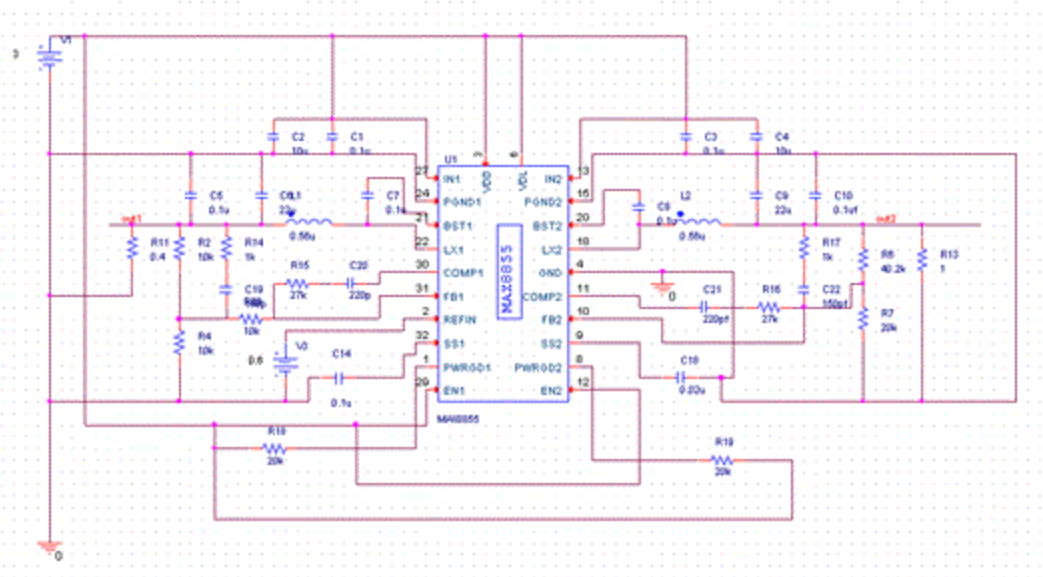
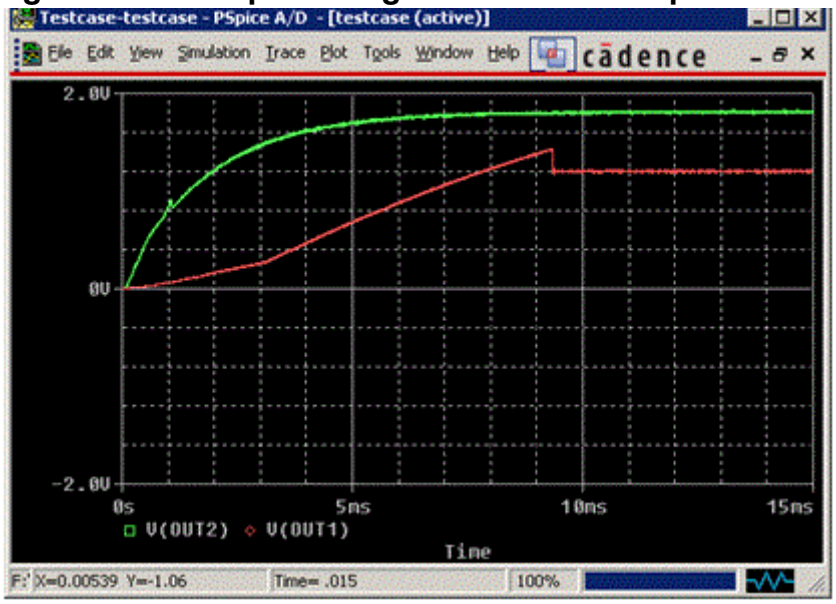


Figure 1-12 shows the two output voltage waveforms from simulation result of circuit in Figure 1-11 . Different rates of rise of output voltage from these waveforms are due to the different slow start capacitors. You can use this model to program slow start timing of each output independently.

Figure 1-12 Output Voltage Waveform in PSpice



Limitations

This model does not offer external sync capability.

Locating the Model

The MAX8588 simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/swit_reg.olb`
- Allegro Design Entry HDL: `<INSTALLDIR>/share/library/swit_reg`

MAX5090

The MAX5090A/B/C easy-to-use, high-efficiency, high voltage step-down DC-DC converters from Maxim. The MAX5090A/MAX5090B versions have fixed output voltages of 3.3V and 5V, respectively, while the MAX5090C features an adjustable 1.265V to 11V output voltage. This pulse-width-modulated (PWM) converter operates at a fixed 127 kHz switching frequency and includes internal frequency compensation simplifying circuit implementation.

This model is developed based on datasheet for MAX5090 available at http://datasheets.maxim-ic.com/en/ds/MAX5090-MAX5090C.pdf.

Following are the key features of this model:

- Wide Input Voltage Range: 6.5V to 76V
- Fixed (3.3V, 5V) and Adjustable (1.265V to 11V)
- Output-Voltage Versions
- 2A Output Current
- Efficiency Up to 92%
- Internal 0.26Ω High-Side DMOS FET
- Internal Frequency Compensation
- Fixed 127kHz Switching Frequency

Using MAX5090 in a circuit

Figure 1-13 shows the typical regulator application circuit diagram for MAX5090:

Figure 1-13 Regulator Application Circuit for MAX5090

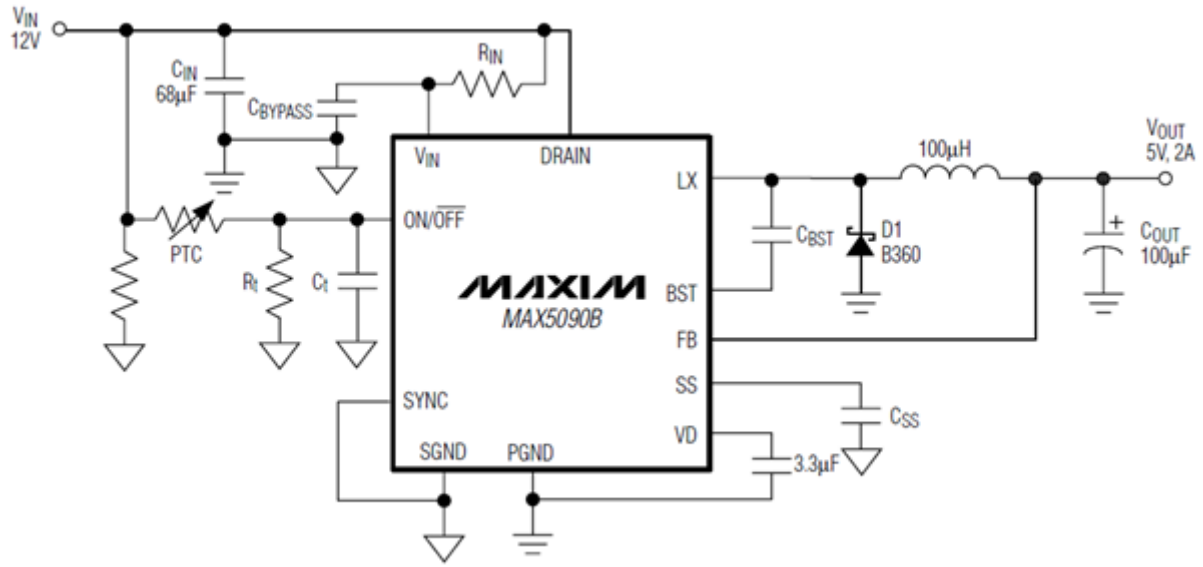


Figure 1-14 shows the typical regulator application circuit diagram based on this model for 3.3V output.

Figure 1-14 Application Circuit for MAX5090 for 3.3V

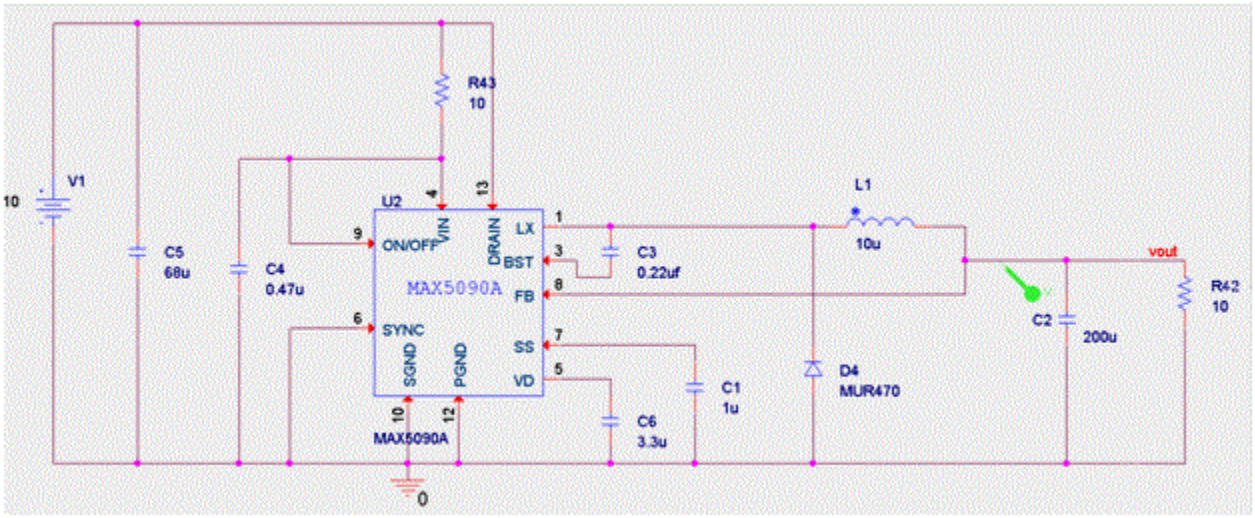
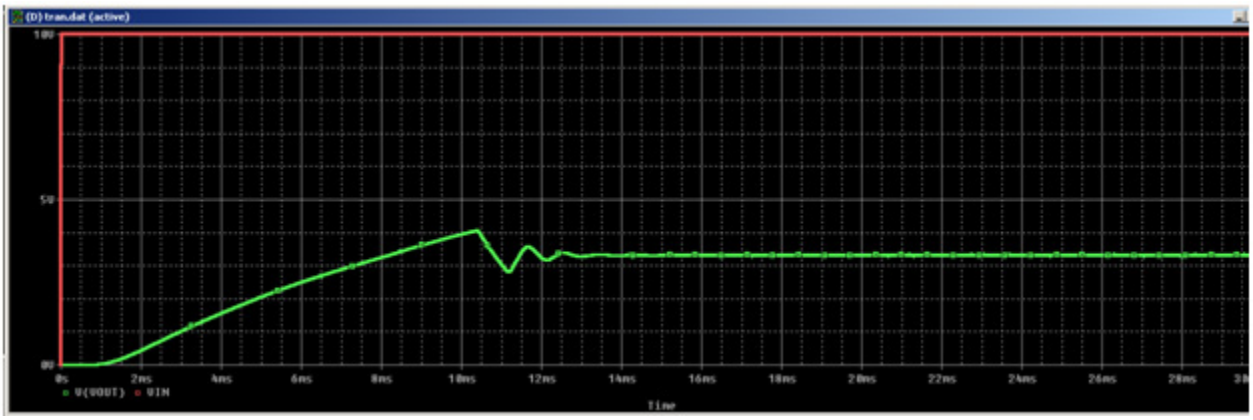


Figure 1-15 shows voltage waveform at input terminal VIN and regulated output voltage at load.

Figure 1-15 Input and Output Voltage Waveforms

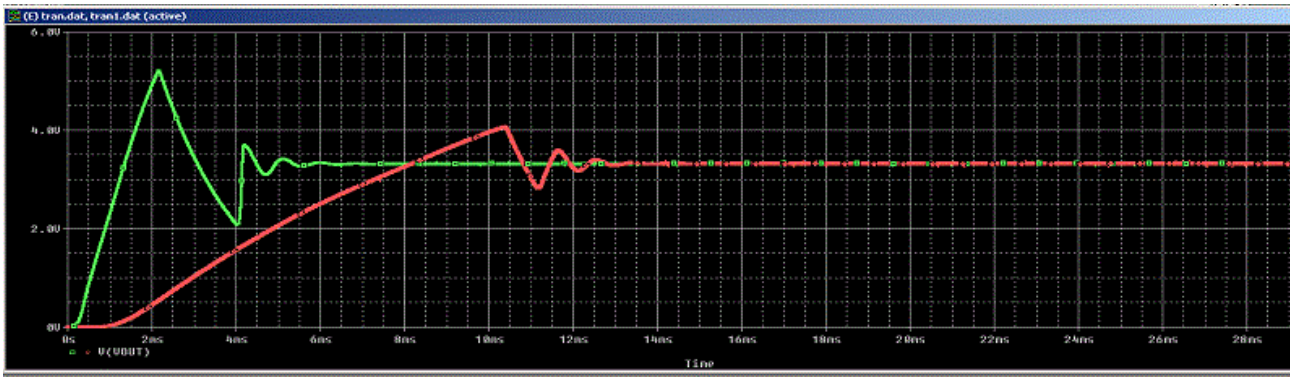


Special Pins and Functions

- **SS Pin**: The MAX5090 provides the flexibility to externally program a suitable soft-start time for a given application. Connect an external capacitor from *SS* to *SGND* to use the external soft-start. Soft-start gradually ramps up the reference voltage seen by the error amplifier to control the output's rate of rise and reduce the input surge current during startup.

  Figure 1-16 shows the effect of soft start on output voltage. The effect can be seen by changing capacitor *C1* in application circuit shown in Figure 1-14. In Figure 1-16 the green waveform is plotted using `C1=0.1u` and the red waveform is plotted using `1u`.

Figure 1-16 Affect of Soft Start on Output Voltage



- **SYNC Pin**: *SYNC* controls the oscillator frequency. Connect *SYNC* to *SGND* to select 127 KHz operation. Use the *SYNC* input to synchronize to an external clock. *SYNC* has

a guaranteed frequency range of 119 kHz to 200 kHz when using an external clock. When *SYNC* is connected to *SGND*, the internal clock is used to generate a ramp.

Figure 1-17 shows application circuit for MAX5090A with use of SYNC Pin.

Figure 1-17 Application Circuit of MAX5090 with SYNC Pin

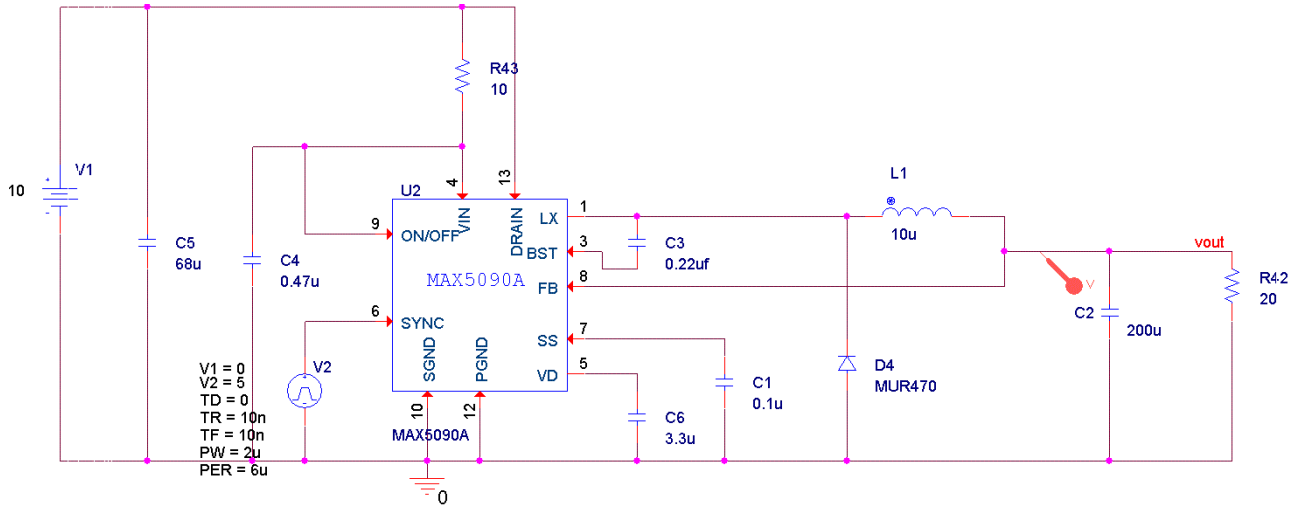
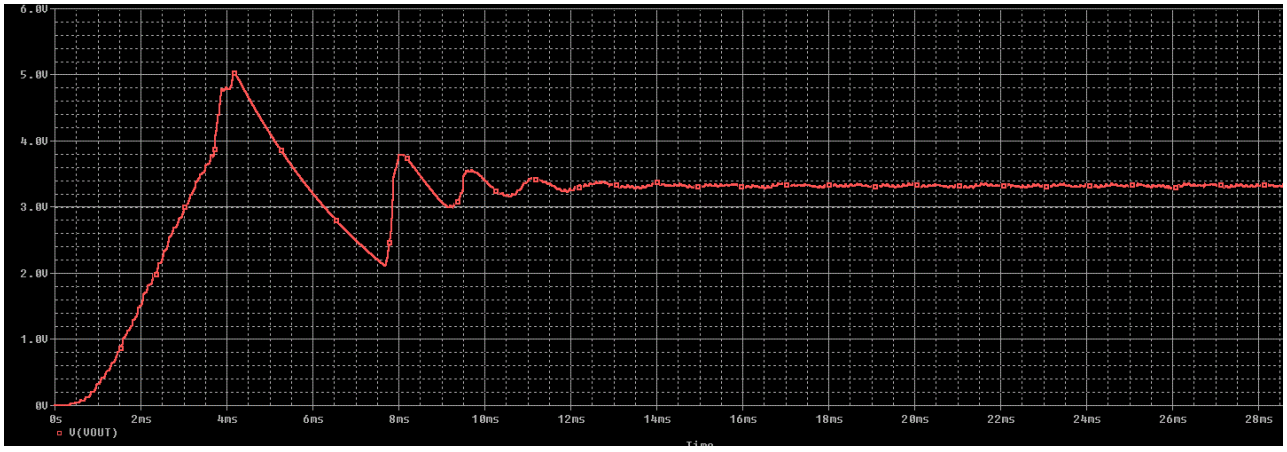


Figure 1-18 shows the output voltage plotted with application circuit shown in Figure 1-17.

Figure 1-18 Output Voltage for Circuit with SYNC Pin



Limitations

Thermal Shutdown and BST Pin are not modeled.

Locating the Model

The MAX5090A simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/PSpice/switch_reg.olb`
- Allegro Design Entry HDL:`<INSTALLDIR>/share/library/switch_reg`

Following are some models similar to MAX5090 in the `swit_reg` library:

- MAX5090A
- MAX5090B
- MAX5090C
- MAX5033A
- MAX5033B
- MAX5033C
- MAX5033D

Off Line PWM Controller

This section describes the *off line PWM controller* models from various semiconductor manufacturers.

UC3841

UC3841 is a versatile off line current mode AC-DC & DC-DC controller. This regulator model offers programmability of switching frequency and current limit while operating in a wide input and output voltage range. It also features important protection functions, such as pulse by pulse current limit with shutdown for over current fault and external stop pin. This model is developed based on datasheet for UC3841 available at http://focus.ti.com/lit/ds/symlink/uc3841.pdf.

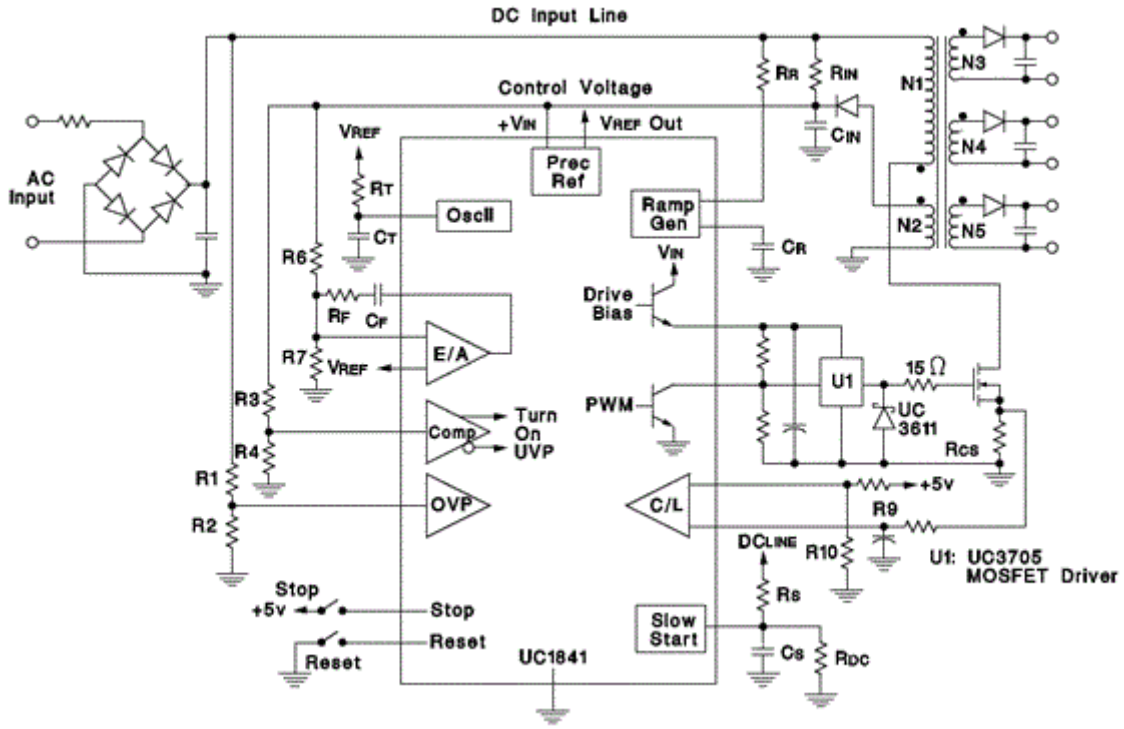
Following are the key features of this model:

- Programmable Switching Frequency up to 0.5MHz
- Programmable Over Current Protection
- External Stop
- Over Voltage Protection
- Under Voltage lockout

Using UC3841 in a circuit

Figure 1-19 shows the typical regulator application circuit diagram for IRF3840:

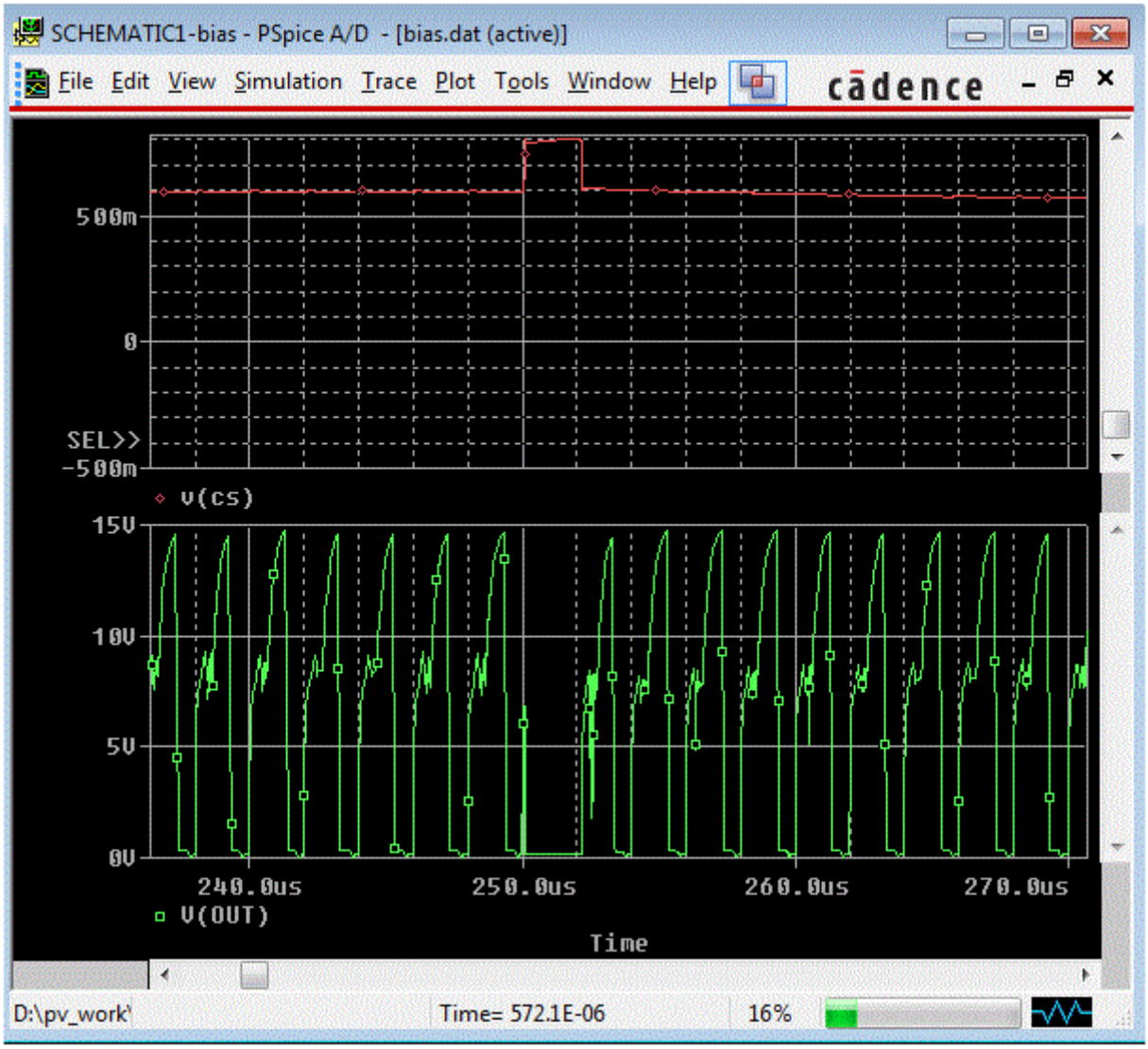
Figure 1-19 Regulator Application Circuit for IRF3840



Special Pins and Functions

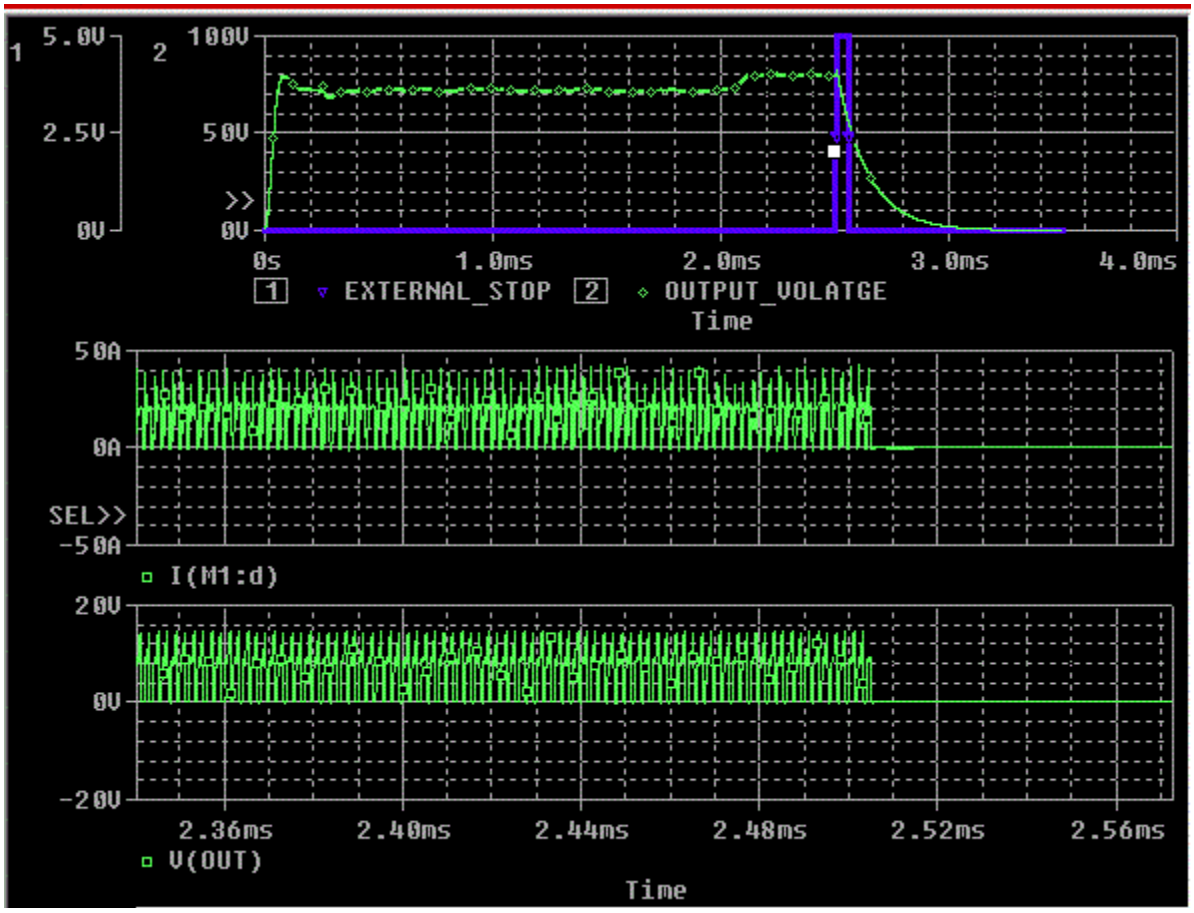
- **Pulse By Pulse Current Limit**: Figure 1-20 shows the operation of pulse-by-pulse current limit. Top waveform shows status of *CS* (current sense pin). On this pin a sudden jump in voltage is sensed and output pulses are terminated at same instance. Since this overshoot is less than the over-current threshold, output pulses are release for next cycle; effectively stopping the operation for just one cycle.

Figure 1-20 Pulse by Pulse Current Limit



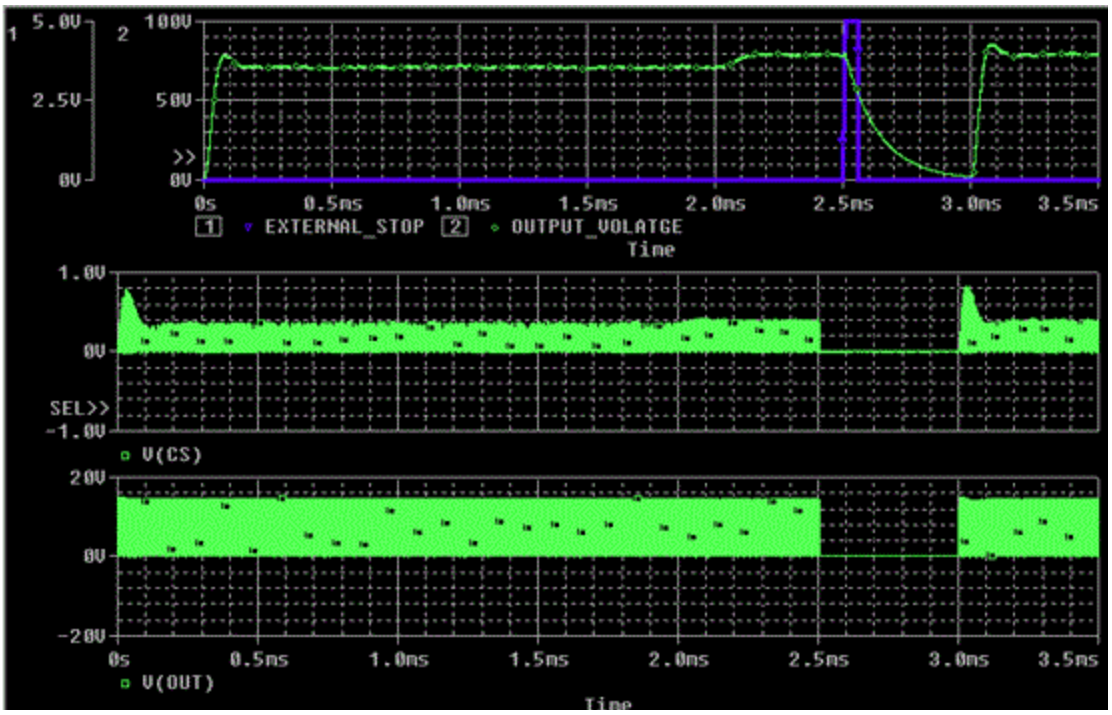
- **External Stop PIN**: This model also includes the functionality of external stop pin. Pulling this pin to voltage above 1.2V disables the output pulse and hold the output in off state. In Figure 1-21 an external stop signal is applied at 2.5mSec and immediately output pulses are stopped. They remain in off state ever after external stop signal is removed.

Figure 1-21 Output Pulses in PSpice



- **RESET Pin**: This model also includes functionality of *RESET* Pin. This pin works in conjunction with external stop or over-current protection functionality. A voltage greater than 3.2v at *RESET* pin with clear the fault condition and turn on the output pulse again. In Figure 1-22 the *RESET* signal is applied at 3mSec and the output pulses are started again.

Figure 1-22 Output Pulses with RESET



- **Setting up oscillator frequency**: You can set the oscillator frequency by modifying the value of *PER* property on UC3843 symbol instance in schematic. Value of this property should be `1/FREQUENCY`; for example, to set 500KHz as switching frequency, value of *PER* property should be set to 2u.

Locating the Model

The UC3841 simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/PSpice/switch_reg.olb`
- Allegro Design Entry HDL :`<INSTALLDIR>/share/library/switch_reg`

Charge-Pump-Based DC-DC Controller

This section describes the *charge-pump-based DC-DC controller* models from various semiconductor manufacturers.

TPS6050x

The TPS6050x devices are a family of step-down charge pumps that generate a regulated, fixed 3.3-V, 1.8-V, 1.5-V, or adjustable output voltage. Only four small ceramic capacitors are required to build a complete high efficiency dc/dc charge pump converter. To achieve the high efficiency over a wide input voltage range, the charge pump automatically selects between three different conversion modes. The output can deliver a maximum of 250-mA output current. The power good function supervises the output voltage and goes high when the output voltage rises to 97% of its nominal value.

This model is developed based on the datasheet for TPS60500 available at http://www.datasheetcatalog.org/datasheet/texasinstruments/tps60500.pdf.

Following are the key features of this device:

- Regulated 3.3-V, 1.8-V, 1.5-V, or Adjustable Output Voltage
- Up to 250-mA Output Current
- 1.8-V to 6.5-V Input Voltage
- Up to 90% Efficiency
- Output Voltage Supervisor Included (Power Good)
- Internal Soft Start
- Load Isolated From Battery During Shutdown
- Over current Protected

Using TPS6050x in Circuit

Figure 1-23 shows the typical application circuit from vendor datasheet. TPS60502 shown in figure is a fixed output voltage charge pump with 1.8Volts.

Figure 1-23 Application Circuit for TPS6050

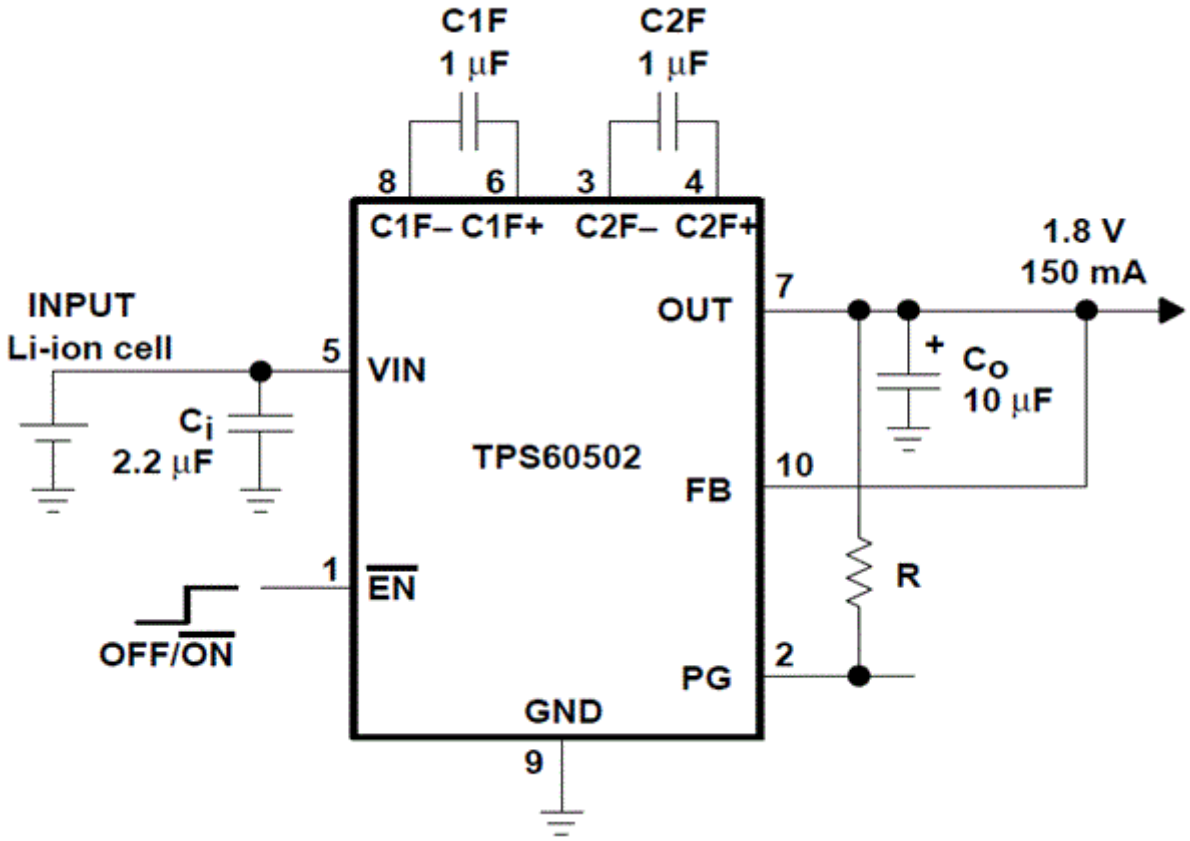


Figure 1-24 shows the simulation circuit using TPS60500. TPS60500 is an adjustable charge pump.

Figure 1-24 Simulation Circuit for TPS60500

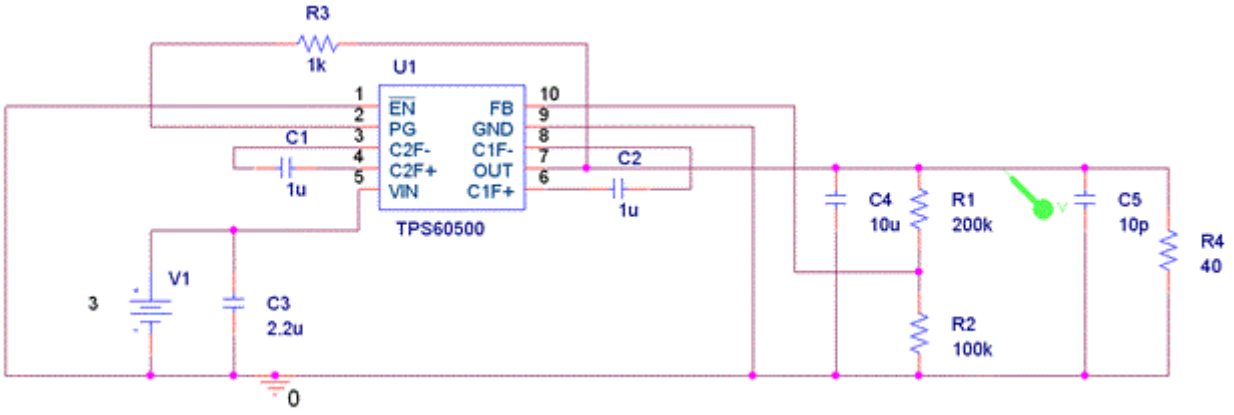
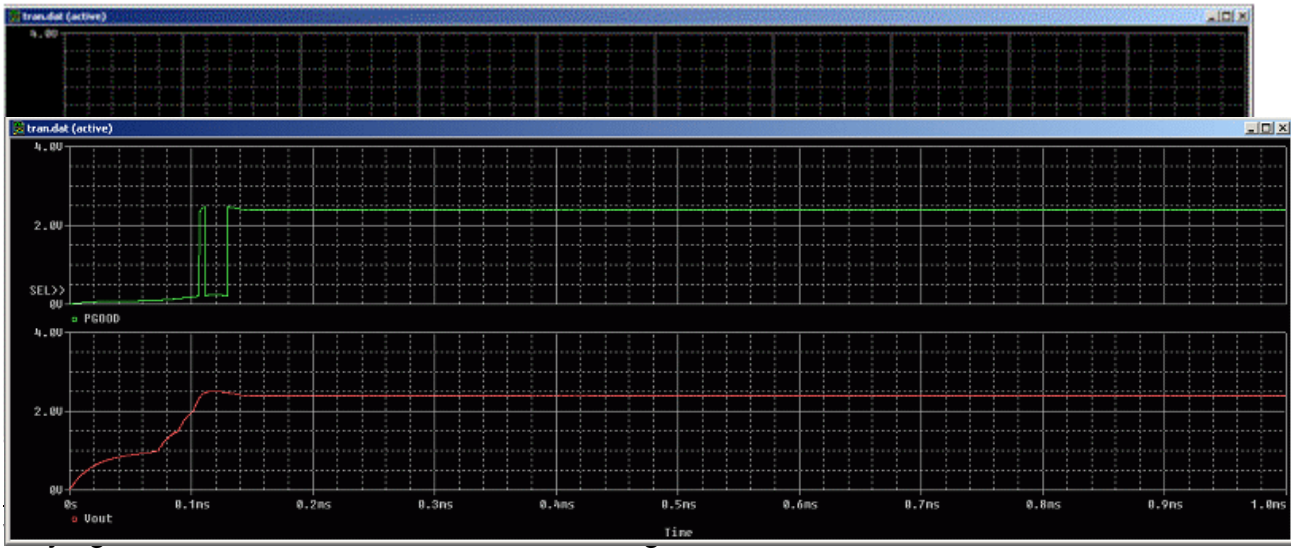


Figure 1-25 shows the input and output voltage waveforms. The blue waveform shows input voltage (3Volts) and the pink waveform shows output voltage (2.4Volts).

Figure 1-25 Input and Output voltage Waveforms

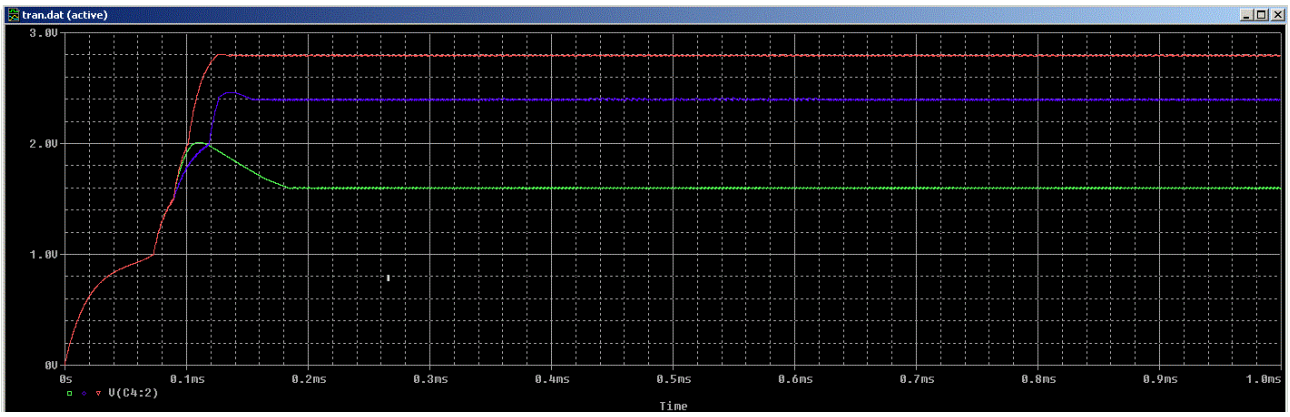


$$Vout = ((R1 + R2) \bullet Vfb)/(R2)$$

Where `Vfb = 0.8V`

Figure 1-26 shows output voltages `1.6V`, `2.4V`, and `2.8V` corresponding to `R1=100k`, `200k`, `250k` and `R2=100k`.

Figure 1-26 Output Voltages Varying with Feedback Voltage



In Figure 1-27, the green waveform is the PGOOD signal that represents the state of the output voltage. PGOOD pulls high when output voltage is in good state. It can help enabling or disabling the peripheral devices communicating to TPS60500.

Figure 1-27 Output Voltage Supervisor PGOOD

Limitations

This model does not have any effect on temperature variation.

Locating the Model

The TPS60500 simulation model is available in the `CP_cnvtr` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/CP_cnvtr.olb`
- Allegro Design Entry HDL: `<INSTALLDIR>/share/library/CP_cnvtr`

Following are some models similar to TPS60500 in the `CP_cnvtr` library:

- TPS60500
- TPS60501
- TPS60502
- TPS60503
- LTC3250-1.2

MOSFET Driver

This section describes the *MOSFET Driver* models from various semiconductor manufacturers.

MIC4123

MIC4123 is dual 3A peak low side MOSFET driver. This can drive the FET gate circuit from logic input. This model is developed based on the datasheet for MIC4123 available at http://www.micrel.com/_PDF/MIC4123v3.pdf.

Following are the key features of this device:

- High 3A-peak output current
- Wide 4.5V to 20V operating range
- Drives 1800pF capacitance in 25ns
- Short less than 50ns typical delay time
- Matched rise and fall times
- TTL logic input independent of supply voltage
- Low 2.3Ω typical output impedance

Figure 1-28 shows the typical application circuit from vendor datasheet.

Figure 1-28 Application Circuit for MIC4123

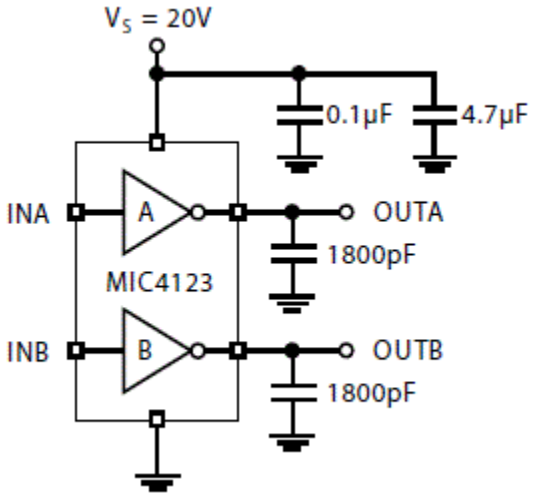


Figure 1-29 shows the application circuit driving two low side MOSFET from logic level input using MIC4123 PSpice Model.

Figure 1-29 Simulation Circuit for MIC4123

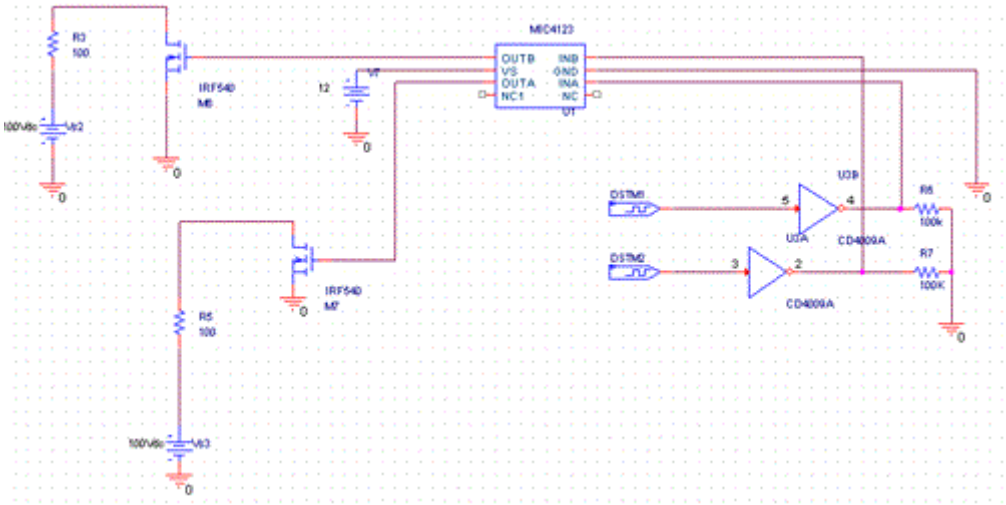
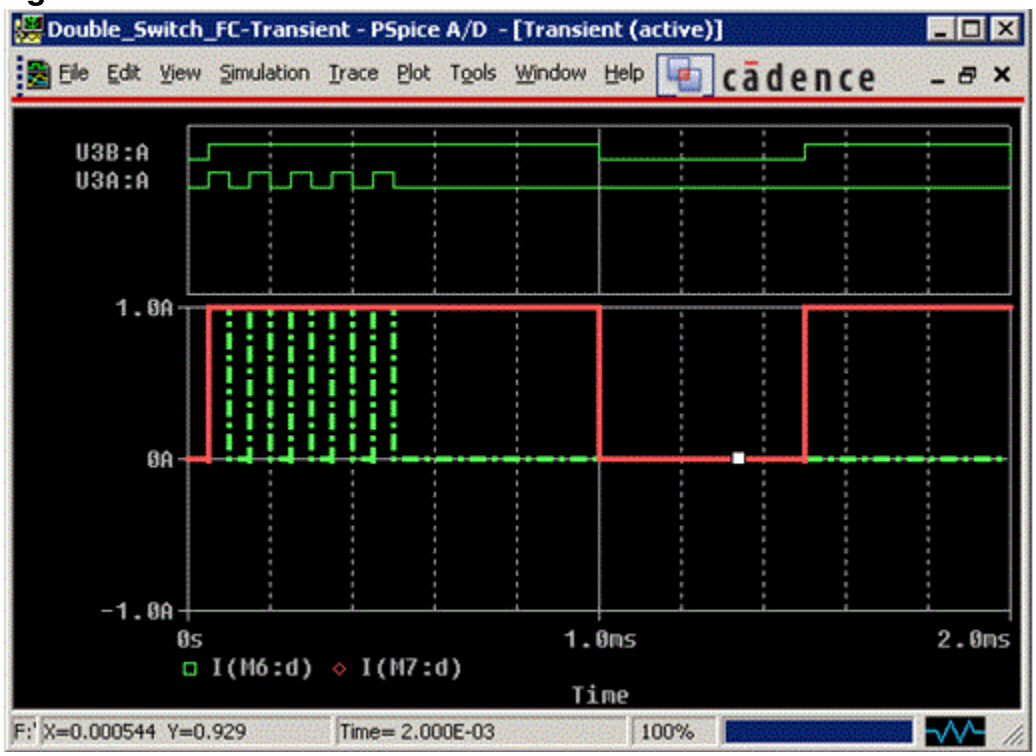


Figure 1-30 shows the MOSFET current vis-à-vis logic level digital input waveform driving these MOSFETs. Signal *U3A:A* is driving MOSFET M6, and Signal *U3B:A* is driving MOSFET M7.

Figure 1-30 MOSFET Current



Locating the Model

The MIC4123 simulation model is available in the `MFET_DRVR` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/mfet_drvr.olb`

Allegro Design Entry HDL: `<INSTALLDIR>/share/library/mfet_drvr`

Following are some models similar to MIC4123 in the `MFET_DRVR` library:

- MIC4120/ MIC4129
- MIC4123/ MIC 4125
- MIC4126/ MIC 4127/ MIC 4128
- MIC4223/ MIC 4224/ MIC 4225
- MIC4416/ MIC 4417
- MIC4451/ MIC 4452
- TSC428
- MAX626/MAX627/MAX628

Off Line Switches

This section describes the *off line switch* models from various semiconductor manufacturers.

LNK304

LNK304 is a Lowest Component Count, Energy Efficient Off-Line Switcher IC. These are typically targeted replacement of linear regulators. This model is developed based on datasheet for IR3840M available at http://www.es.co.th/Schemetic/PDF/LNK304-306.PDF.

Following are the key features of this model:

- Programmable Switching Frequency up to 66KHz
- Over Current Protection and pulse by pulse current limiting
- Supports buck, buck-boost and flyback topologies

Using LNK304 in a circuit

Figure 1-31 shows the typical regulator application circuit diagram for LNK304:

Figure 1-31 Application Circuit for LNK304

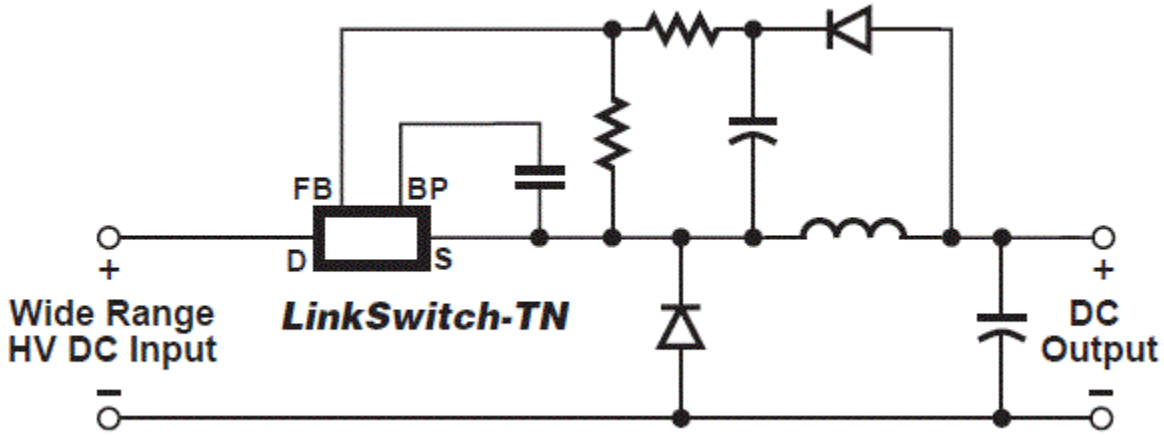


Figure 1-32 shows the typical regulator application circuit diagram based on this model for 12V/120mAmp output.

Figure 1-32 Regulator Application Circuit for 12v/120mA output

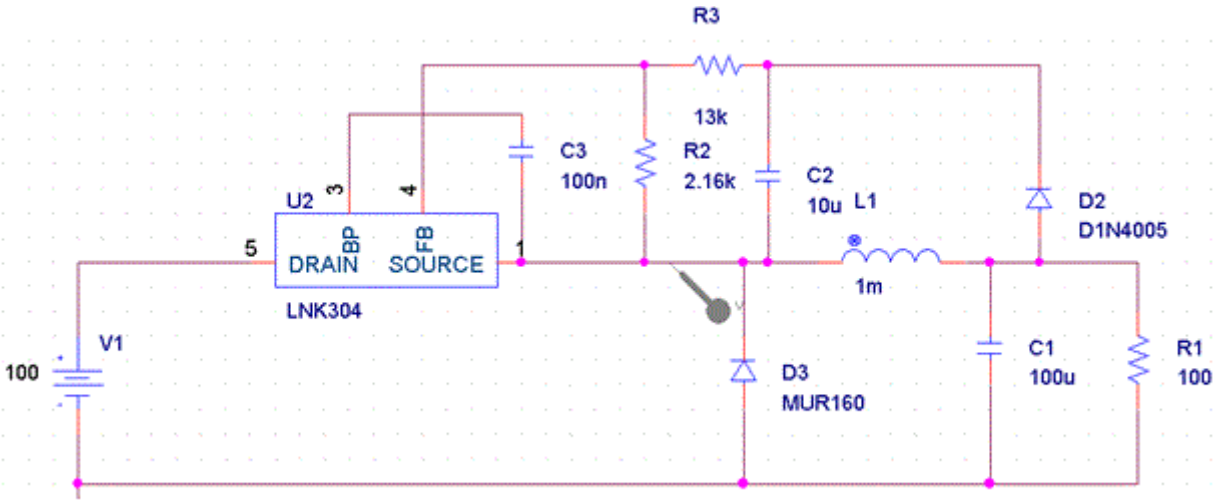


Figure 1-33 shows voltage waveform at switch terminal and current into output filter inductor.

Figure 1-33 Voltage Waveform and Output Current

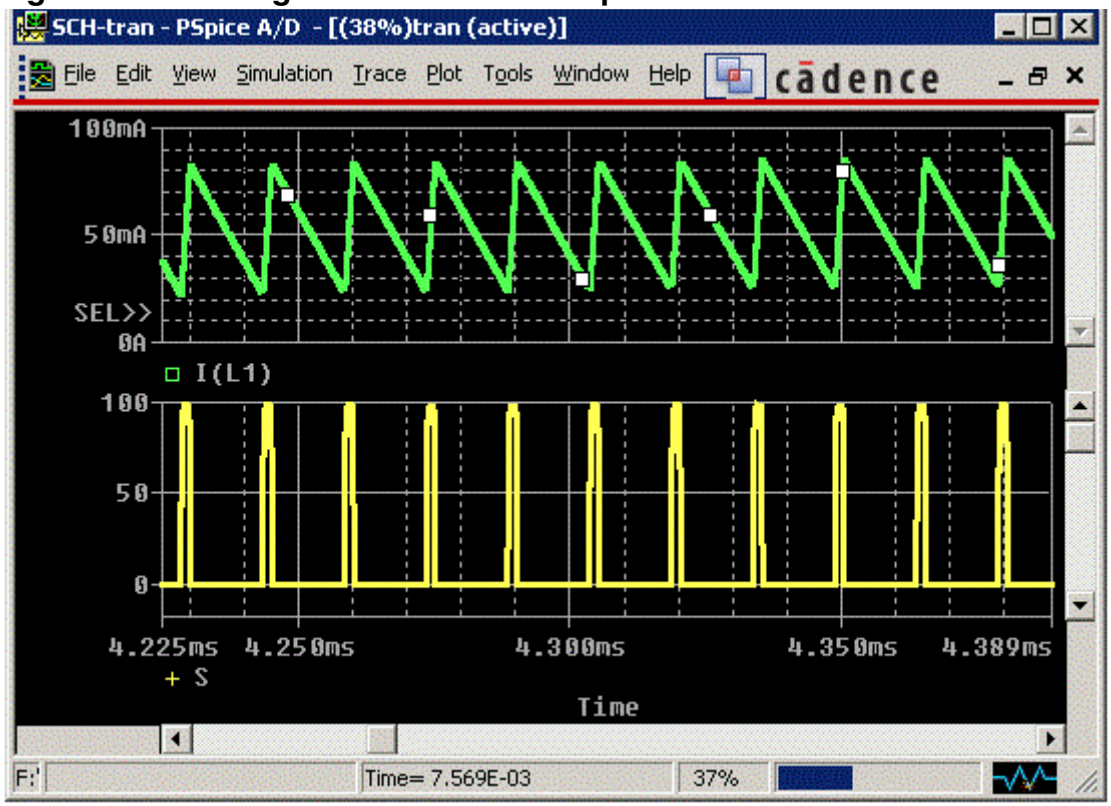
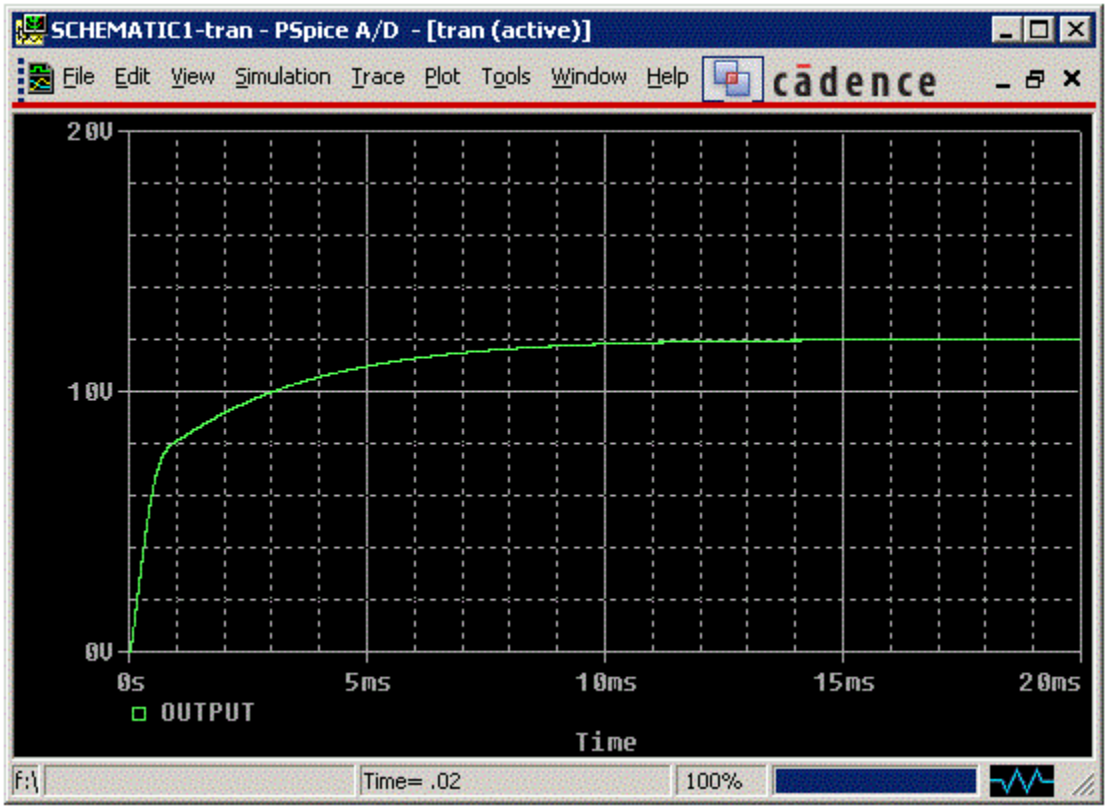


Figure 1-34 shows the output voltage waveform.

Figure 1-34 Output Voltage Waveform



Special Functions of LNK304

- **Over-current Protection**: This model also includes functionality of over-current protection in pulse by pulse current limiting mode. Switch current is sensed and pulses are blocked if current through switch exceed 345mAmpOver-current threshold. This protection system is intelligent to ignore initial current spikes caused by diode reverse recovery current. This leading edge blanking time of 215nSec introduced at start of every cycle.
- **Setting up oscillator frequency**: This device operates at fix frequency of 66KHz.

Limitations

Thermal shutdown functionality and auto-restart functionality has not been modeled.

Locating the Model

The LNK302 simulation model is available in the `swit_reg` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/PSpice/switch_reg.olb`
- Allegro Design Entry HDL:`<INSTALLDIR>/share/library/switch_reg`

2

Solid State Relay

This chapter describes the following *solid state relay* models:

- CPC1117N
- HSR412

CPC1117N

CPC1117N is a Single-Pole, Normally Closed 4-Lead SOP OptoMOS Relay from Clare. It employs optically coupled MOSFET technology to provide 1500Vrms of input/output isolation. It employs an active current limit circuitry enabling the device to withstand current surge transients.

This model is developed based on the datasheet for CPC1117N available at http://www.clare.com/home/pdfs.nsf/www/CPC1117N.pdf/$file/CPC1117N.pdf.

Following are the key features of this device:

- Only 1mA of LED current required to operate
- 1500Vrms Input/output Isolation
- Small 4-Lead SOP Package
- TTL/CMOS Compatible input

Using CPC1117N in Circuit

Figure 2-1 shows typical switching characteristics of normally closed devices.

Figure 2-1 Switching Characteristics of Normally Closed Devices

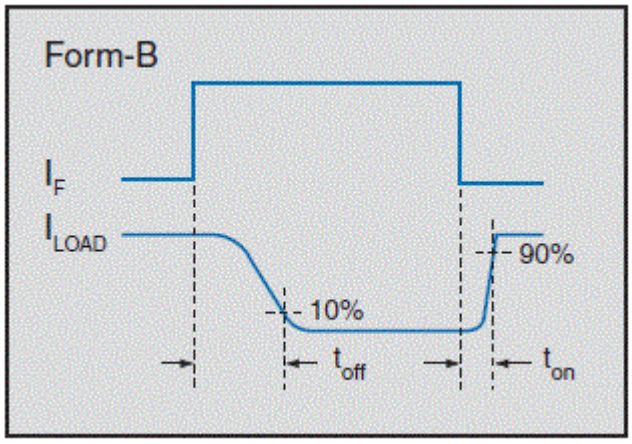


Figure 2-2 shows a simulation circuit for CPC1117N.

Figure 2-2 Simulation Circuit for CPC1117N

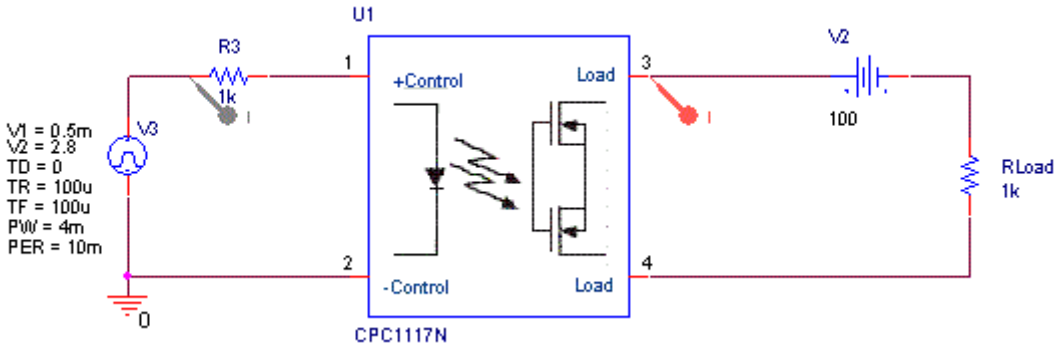


Figure 2-3 shows the switching characteristics obtained by simulated circuit shown in Figure 2-2.

Figure 2-3 Switching Characteristics Simulated in PSpice

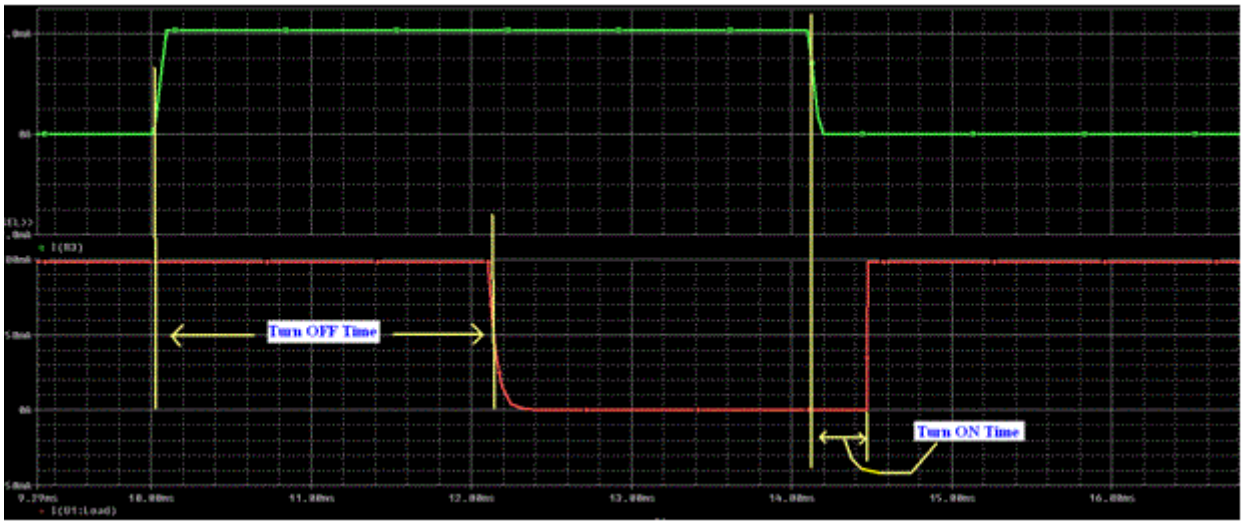
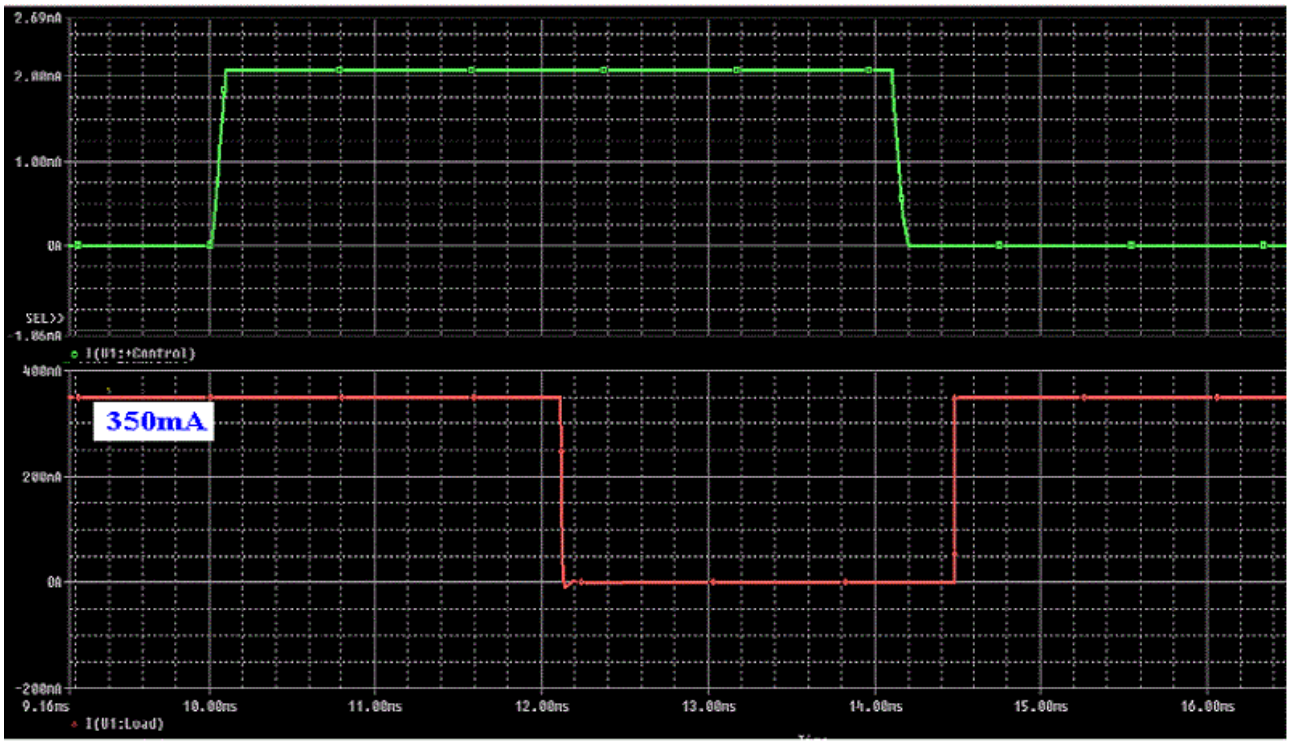


Figure 2-4 shows the CPC117N internal current limiting capability. In the circuit shown in Figure 2-2, change `Rload` to `0.1k`.

As ON Resistance for MOSFET mentioned in datasheet is 16 ohm (max). So, by applying KVL, output current should be more than 800m. But its internal current limiting circuitry limits current to 350mA.

Figure 2-4 Switching Characteristics with Current Limiting



Locating the Model

The CPC1117N simulation model is available in the `SSR` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/SSR.olb`
- Allegro Design Entry HDL: `<INSTALLDIR>/share/library / SSR`

Following are some models similar to CPC1117N in the SSR library:

- CPC1117N
- CPC1150N
- HSR312
- HSR312L
- HSR412L

- LH1503AAC

HSR412

HSR412 is a Photovoltaic Solid-State Relay Optocoupler. It consists of AlGaAs infrared emitting diode optically coupled to a power MOSFET detector, which is driven by a photovoltaic generator.

Solid state relays are available in two types, Normal Open (Form A) and Normal closed (Form B). HSR412 falls in the category of “Form A” SSR’s.

This model is developed based on the datasheet for HSR412 available at http://www.fairchildsemi.com/ds/HS/HSR312.pdf.

Following are the key features of this device:

- 4,000 VRMS Isolation
- Wide operating voltage range
- Solid-State Reliability
- Bounce-Free Operation

Using HSR412 in Circuit

Figure 2-5 shows the typical application circuit from vendor datasheet.

Figure 2-5 Application Circuit for HSR412

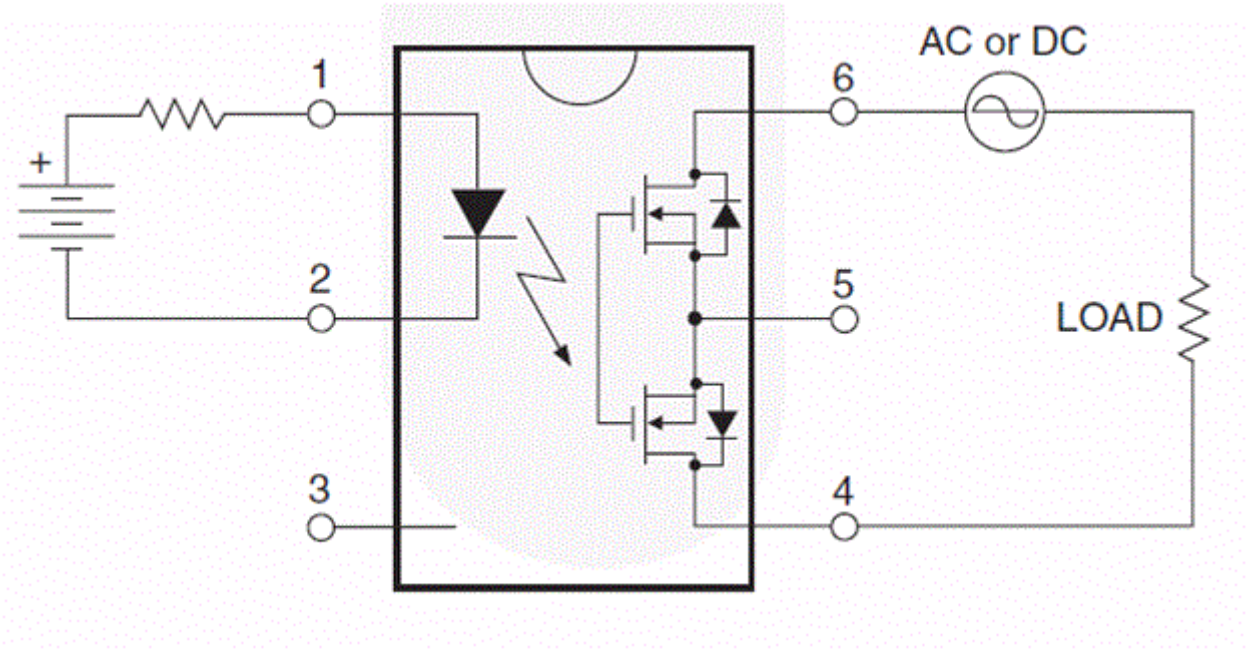


Figure 2-6 shows the PSpice simulation circuit for HSR412 solid state relay. Solid state relay circuits can be designed to switch either AC or DC to the load. In addition to these, it can be used in series as well as parallel by which you can have different RDSon of output MOSFETS.

Figure 2-6 HSR412 SSR Connected in Series

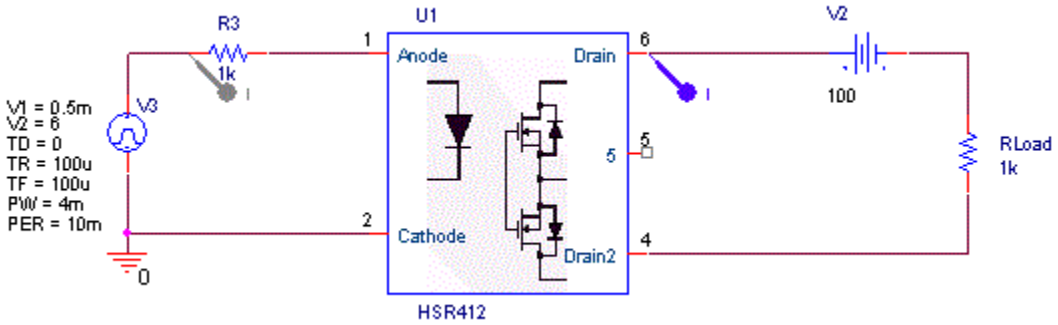


Figure 2-7 shows the PSpice simulation circuit for HSR412 connected in parallel.

Figure 2-7 HSR412 SSR Connected in Parallel

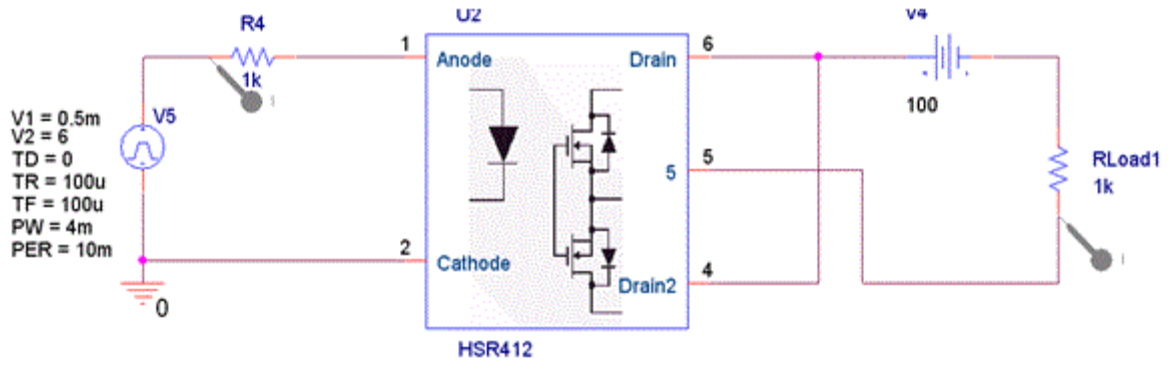
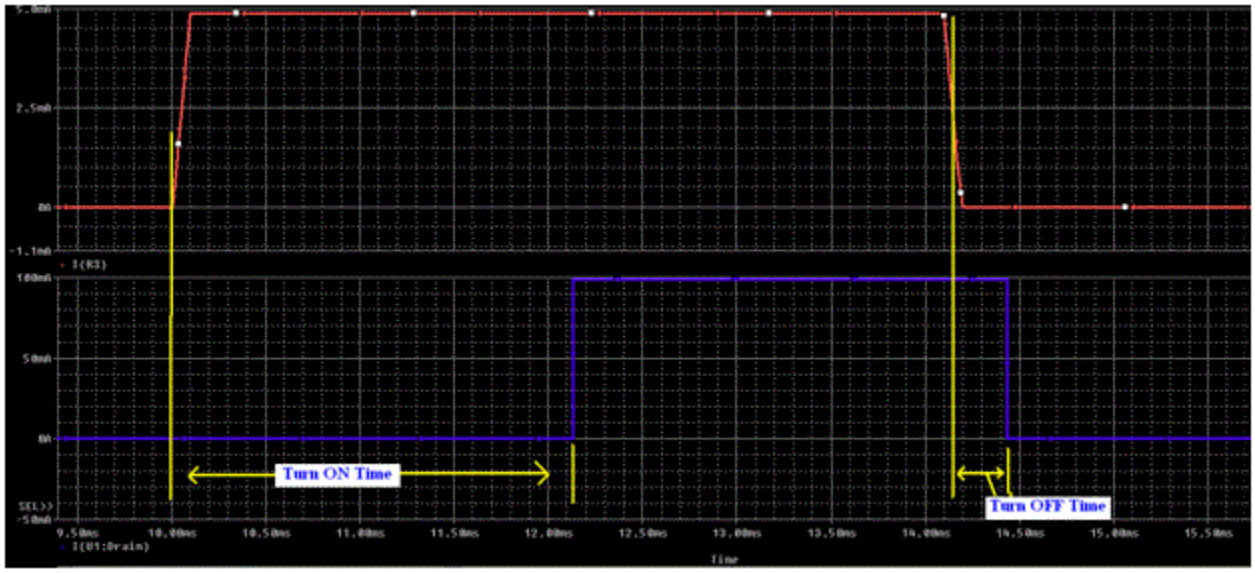


Figure 2-8 shows the timing waveforms for input and output voltages. In SSR, the turn ON and turn OFF time depends on the current through the input light emitting diode. Larger the input current, smaller the turn ON time and vice versa.

Figure 2-8 Timing Waveform for Input-Output Voltages



Locating the Model

The HSR412 simulation model is available in the `SSR` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/SSR.olb`

- Allegro Design Entry HDL: `<INSTALLDIR>/share/library/SSR`

Following are some models similar to HSR412 in the SSR library:

- CPC1117N
- CPC1150N
- HSR312
- HSR312L
- HSR412L
- LH1503AAC

3

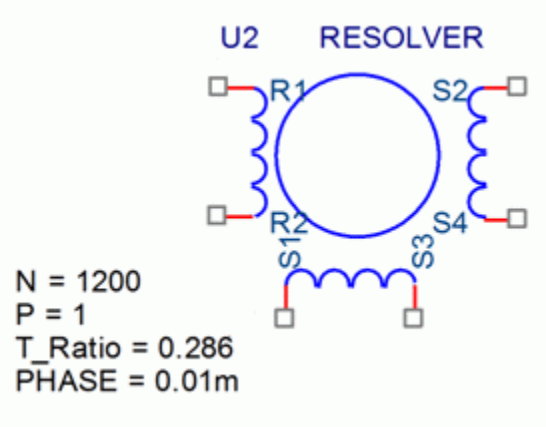
Resolver

This chapter describes the following *resolver* model:

- RESOLVER

RESOLVER

A resolver is rotatory transformer which provides output or information on the basis of its rotor position angle θ. The output is used in control circuits to provide velocity and position information of rotating objects. The symbol of the resolver is given below.



The resolver is energized with AC voltage applied to input pins R1 and R2. The AC voltage is transferred to the rotor winding (not shown) with transfer ratio given by the parameter T_Ratio. Rotor winding then induces voltage on the two output stator winding with pins S1-S3 and S2-S4; these windings are configured 90degrees from each other.

The phase and the frequency of the voltage induced on the two output stator windings are same as the input signal. However, the amplitude of the output voltages is a function of sine and cosine of the rotor position angle.

The equation of input output voltage and their relationship is given below:

- Input

  $E_{R1-R2} = A \cdot \sin(\theta t)$

- Output voltage

  E_{S1-S3} = T_Ratio. A .sin(θ).sin(θt +θ)

  E_{S2-S4} = T_Ratio. A .cos(θ).cos(θt +θ)

  Where,

  A = Amplitude of input signal

  E_{R1-R2} = Input voltage

E_{S1-S3} = Output voltage of winding S1-S2

E_{S2-S4} = Output voltage of winding S3-S4

T_Ratio = Transfer ratio from input to output

θ? = Rotor angle

ψ? = Phase shift

The value of the rotor angle is given as:

$$\theta = 2\Pi \times \frac{N}{60} \times \text{Time}$$

Where,

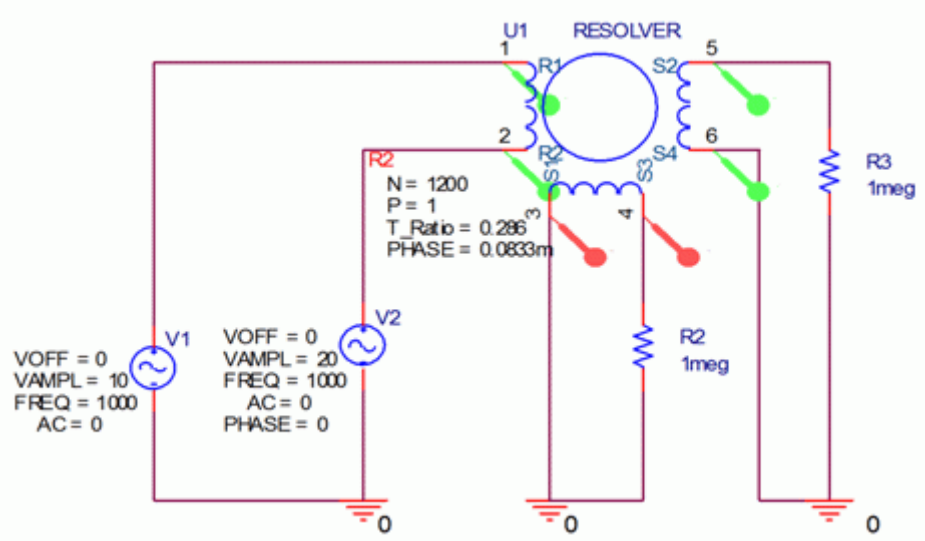
N = Speed of the rotor

P = Magnetic pole pairs in rotor

T_Ratio, N, P and ψ (PHASE) are directly configured in the symbol.

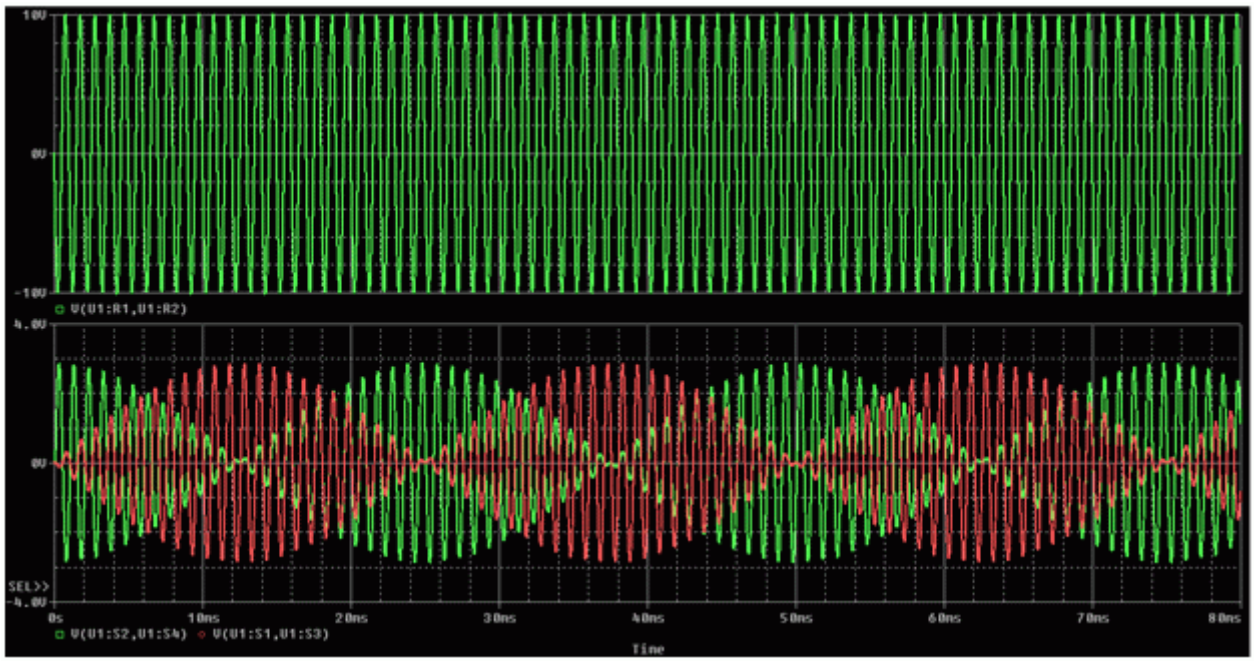
Note: This model expects the parameter PHASE or (ψ) in absolute time instead of angle. Thus user needs to convert phase shift in terms of time. For example, if the input signal competes its half cycle at 0.5ms then in order to get phase shift of 30^{o}, divide 0.5ms by 6 which gives PHASE equal to 0.083ms.

Using RESOLVER in Circuit



The application circuit shown has input pin `R1` and `R2` connected to two sine sources. The input voltage is thus differential voltage across the two pins, E_{R1-R2}.

On simulating the above circuit in transient analysis the output waveform is:



Locating the Model

The RESOLVER simulation model is available in the `ANL_MISC` library.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/Library/PSpice/ANL_MISC.olb`
- Allegro Design Entry HDL: `<INSTALLDIR>/share/library/ANL_MISC`

4

Battery

This chapter describes the following *battery* model:

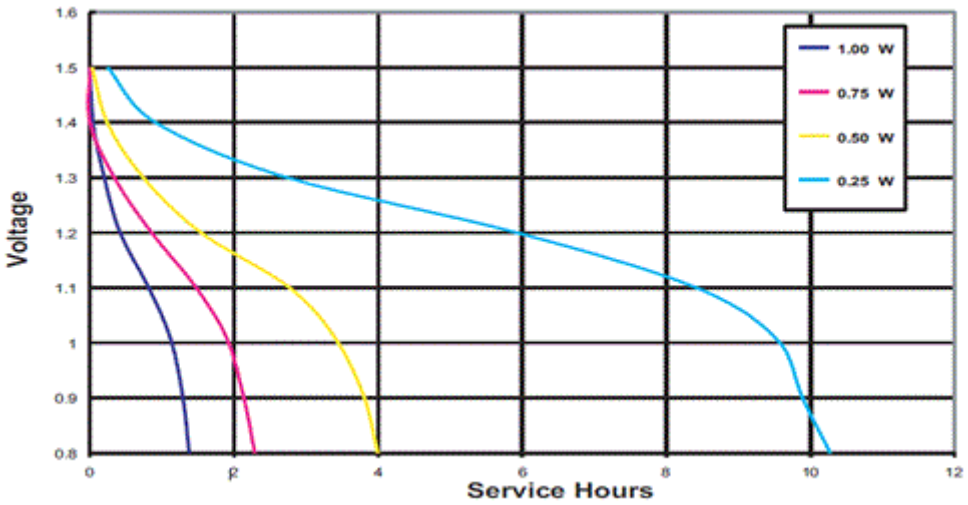
- MX1500

MX1500

MX1500 is 1.5V alkaline-manganese dioxide battery from Duracell Batteries. Having a model with accurate discharge characteristics would help designer to estimate appliance run time for a given battery. This model is developed based on the datasheet for MX1500 available at http://www1.duracell.com/oem/Pdf/MX1500.pdf.

Following are the key features of this device:

- Alkaline-Manganese Dioxide Battery
- Nominal Voltage : 1.5V
- Typical discharge characteristics as shown:



Using MX1500 in a circuit:

Figure 4-1 shows the battery MX1500 connected to constant power load:

Figure 4-1 MX1500 Connected to Constant Power Load

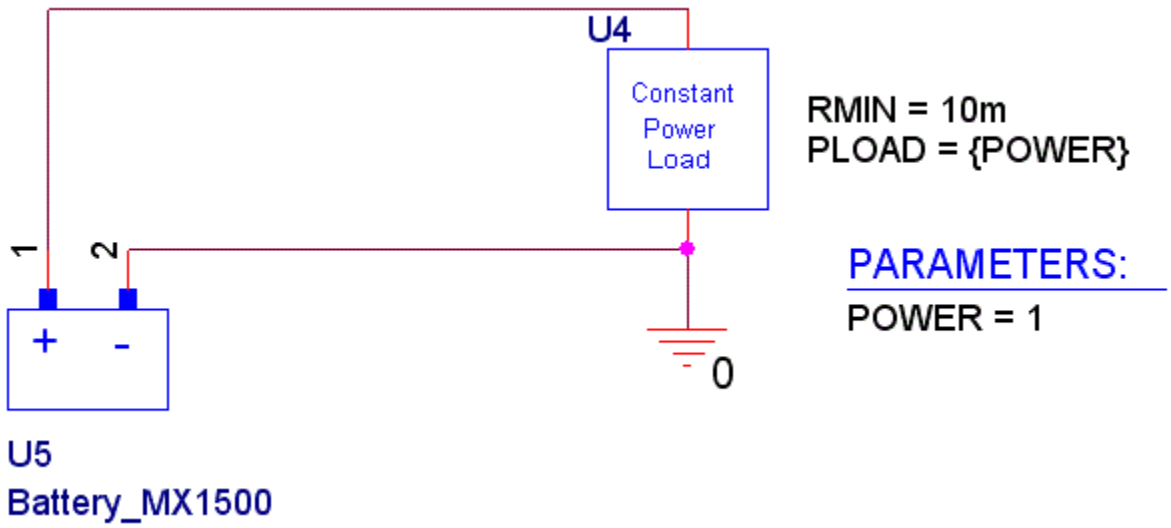


Figure 4-2 shows the typical discharge characteristics of battery MX1500 simulated in PSpice at different power loads.

Figure 4-2 Battery Discharge Characteristics Simulated in PSpice

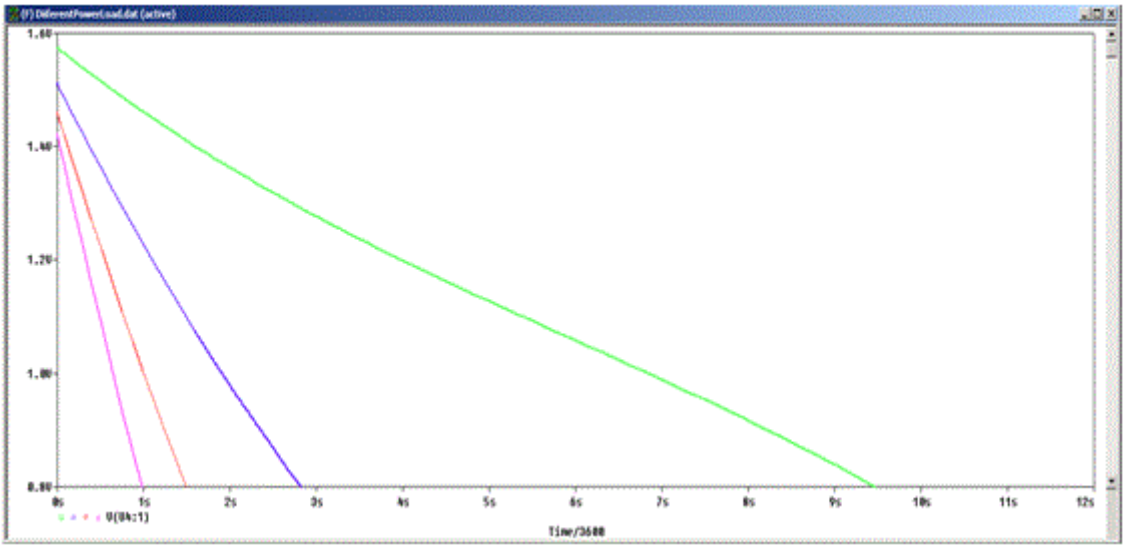


Figure 4-3 shows two batteries connected in series and simulated at constant power of 500mWatt.

Figure 4-3 MX1500 Connected in Series

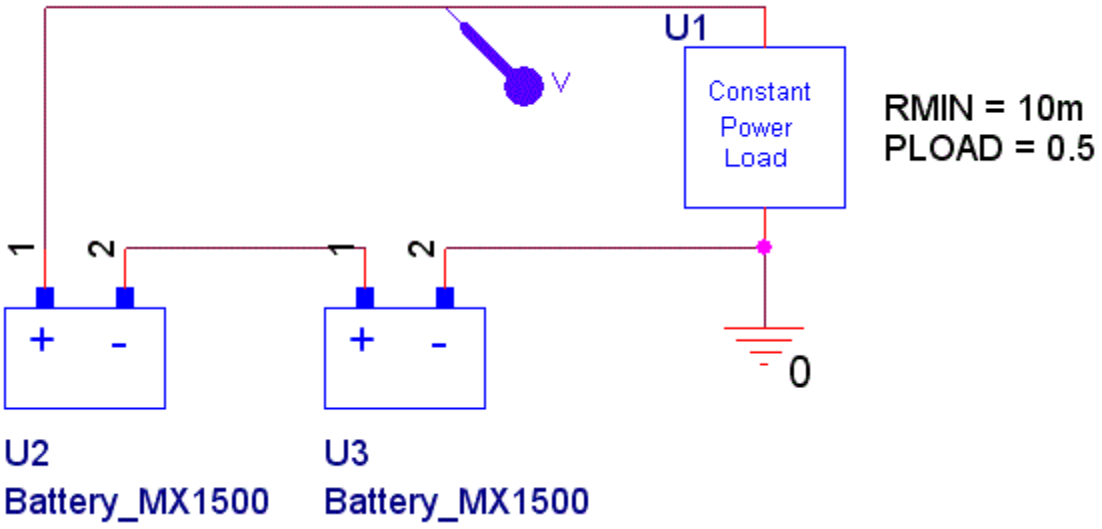


Figure 4-4 shows the output waveform for circuit shown in Figure 4-3. Connecting two batteries in series, doubles the output voltage while maintaining the same capacity rating (amp hours).

Figure 4-4 Output Voltage of Two Batteries in Series

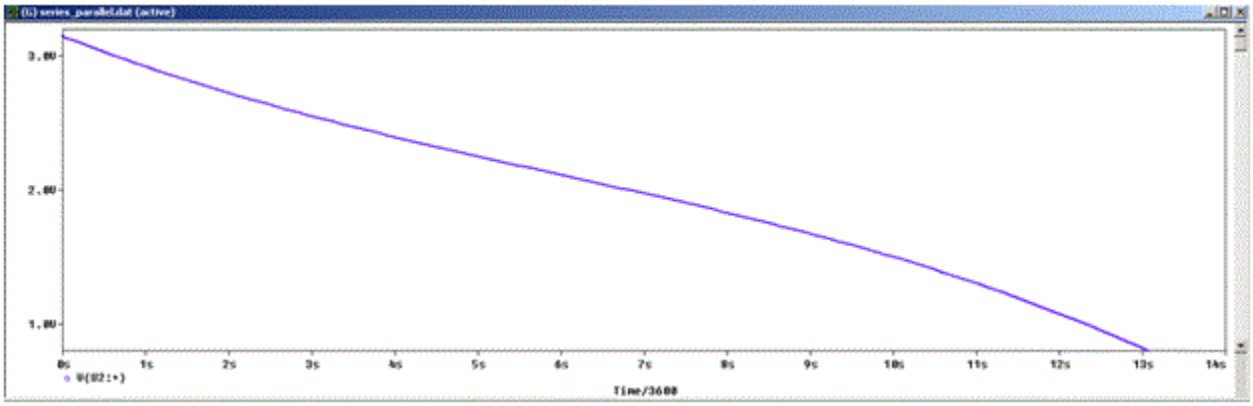


Figure 4-5 shows a boost converter using MX1500.

Figure 4-5 Boost Converter Using MX1500

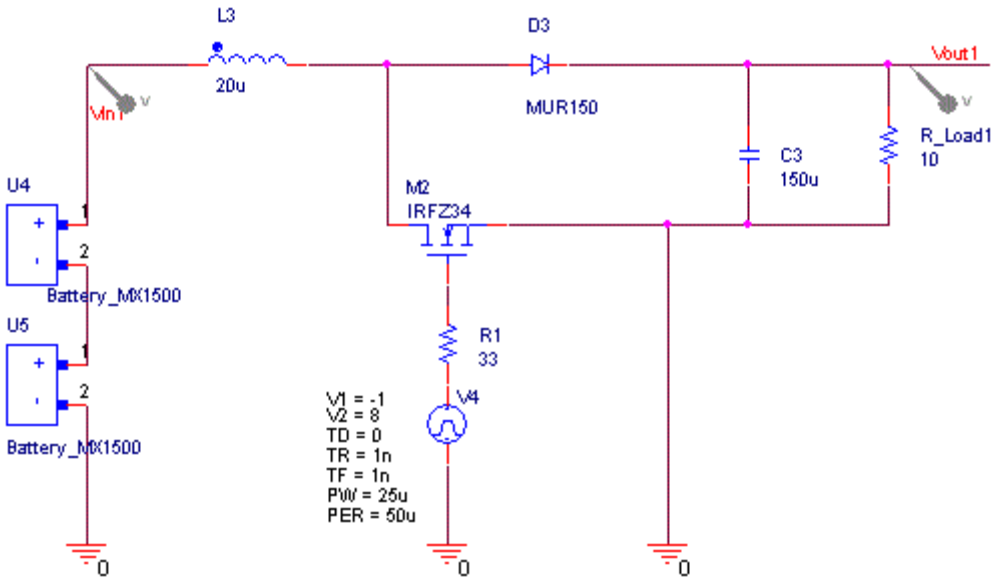
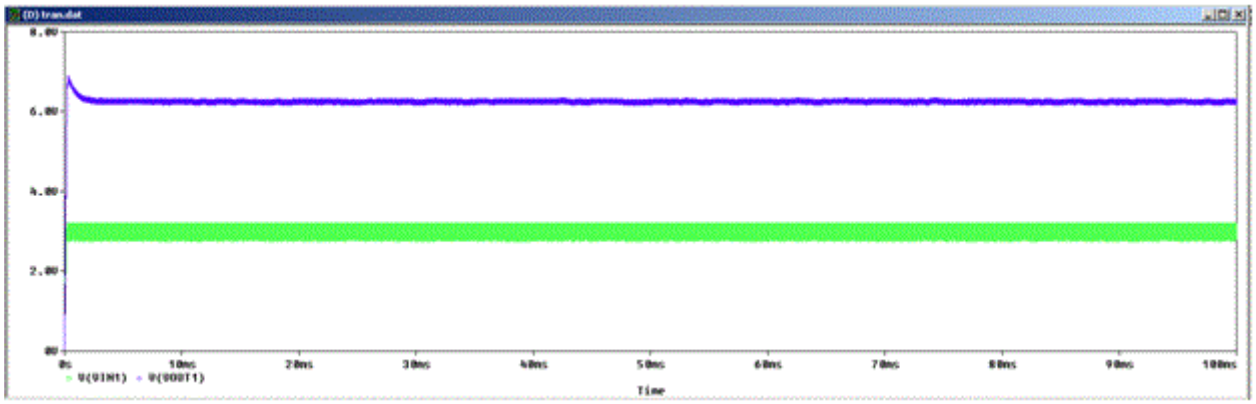


Figure 4-6 shows the input and output voltage waveform for the circuit shown in Figure 4-5.

Figure 4-6 Boost Converter Waveforms Simulated in PSpice



Limitations

Temperature variations have no effect on the output voltage.

Locating the Model

The MX1500 simulation model is available in the `battery` library. This model is in encrypted form; therefore, you will not be able to modify or view its internal details.

The schematic symbols for the model can be found in following location.

- OrCAD Capture/CIS: `<INSTALLDIR>/tools/capture/library/PSpice/battery.olb`
- Allegro Design Entry HDL :`<INSTALLDIR>/share/library /battery`

5

Special Components

This chapter describes parts used in simulation flow for special purposes. These are not electrical parts in themselves. These parts are used in schematic to perform special functions.

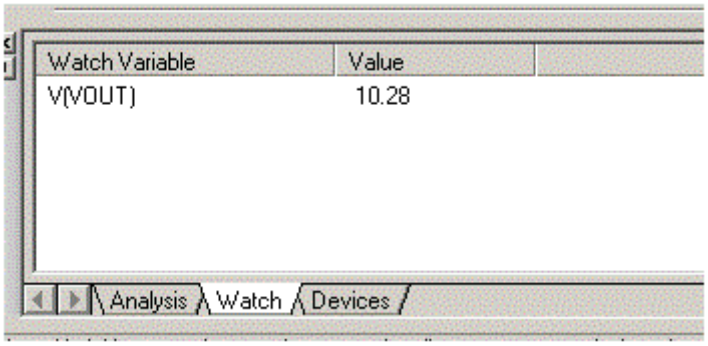
Watch1

Use this component to insert `.WATCH` command in your simulation. This component enables you to observe voltage at node in simulation status windows while simulation is in progress as shown in Figure 5-1. Attach this part at node or net which you will like to observe during the simulation. After this, you need to define following three parameters on symbol instance:

- Analysis: Assign the one of the following three values depending upon the type of analysis being performed:
  - *DC*
  - *AC*
  - *TRAN*
- *HI*: Assign a numeric value to define upper limit of voltage
- *LO*: Assign a numeric value to define lower limit of voltage

Simulator pauses if voltage at node is outside of range defined by *HI* and *LO*.

Figure 5-1 Voltage in Simulation Status Window



Print1

Use this component to insert .PRINT command in your simulation. This component enables you to print voltage at node in simulation output (.out) file.

You need to configure one or more parameters on this symbol instances from following list to use in simulation flow.

- Analysis Type: Assign any value to one of the variables AC/DC/TRAN, depending upon the analysis you are performing. For example if your simulation profile is configured for Transient Analysis you should set TRAN property value as 1.
- Functions: You can use following special function on output nodes
  - DB
  - PHASE
  - REAL
  - MAG
  - IMAG

  You can use one or more of the parameters at a time. To select these you need to assign any value to these parameter, say, 1. For example, if you have set DB and PHASE values to 1 on symbol instance and this component is connected to node VOUT, it will print following in output file:

```
DB of Voltage at VOUT node

PHASE of Voltage at VOUT node
```

Print2

PRINT2 performs the same function as PRINT1, only difference being this component should be used for differential voltage between two nodes.

PRINTDGTLCHG

Use this component for digital nodes. It lists every change of state for a given node in the output file. You need not specify any parameter on this symbol instance.

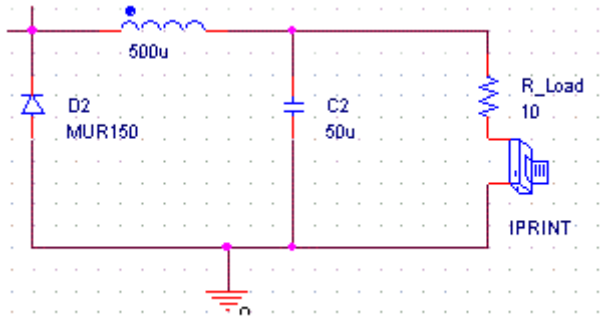
VPLOT1 and VPLOT2

VPLOT1 and VPLOT2 perform same function as the PRINT component, only difference being that these generate PLOTs instead of tabular data.

IPRINT

Use this component to print current through a specific device in your simulation output file. This component works like an ammeter. To print current through a specific device you need to insert this component in series. This component enables you to print current through the device in simulation output (`.out`) file.In Figure 5-2 ,`IPRINT` lists current in tabular format through the device `R_LOAD`.

Figure 5-2 Circuit using IPRINT



You need to configure one or more parameters on this symbol instance from following list to use this in simulation flow.

- Analysis Type: Assign any value to one of the variables `AC`/`DC`/`TRAN`, depending upon the analysis you are performing. For example if your simulation profile is configured for Transient Analysis you should set `TRAN` property value as 1.
- Functions: You can use following special function on output nodes
  - `DB`
  - `PHASE`
  - `REAL`
  - `MAG`
  - `IMAG`

You can use one or more of the parameters at a time. To select these you need to assign any value to these parameter, say, `1`. For example if you have set `DB` and `PHASE` values to 1 on symbol instance and this component is connected to node `VOUT`, it will print following in output file:

```
DB of Voltage at VOUT node

PHASE of Voltage at VOUT node
```

IPLOT

Use this component to plot current through a specific device in your simulation output file. This works identical to IPRINT. Refer IPRINT on page 76 for additional details on this component.

INCLUDE

Use this component to include any file in your simulation. This is an alternate to including the files in simulation profile. To include a file in your simulation, place this instance in your schematic and modify the `FILENAME` property. This property value should be set to the name of the file with complete path. If path is not defined then the tool looks for the file in the profile folder.

LIB

Use this component to add a model library file in your simulation. This is an alternative to adding library file in simulation profile. To include a file in your simulation place this instance in your schematic and modify the `FILENAME` property. This property value should be set to the name of the library file to be include with complete path. If path is not defined then tool looks for the file in the profile folder.

IC1

Use this component to define *Initial Condition* on a given node/net in your simulation. To define *Initial Condition* in your simulation, place this instance in your schematic and attach it to the node/net and modify the *Value* property. This property should be set to the desired initial condition voltage on that node/net.

IC2

Use this component to define differential voltage as initial condition. IC1 component defines voltage with respect to ground whereas this component, IC2, defines initial condition between a node pair.