PSPICE Quickstart

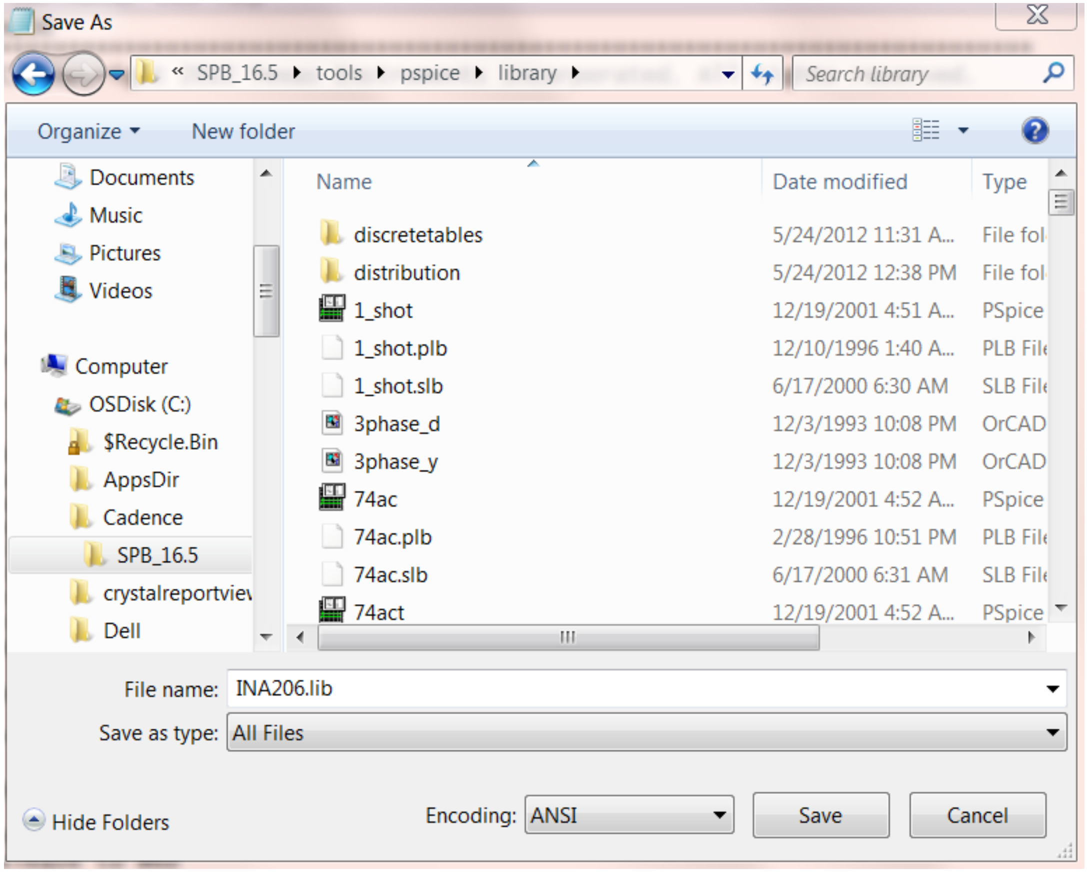
Revision 0

Pete Semig

Analog Applications Engineer, Precision Linear

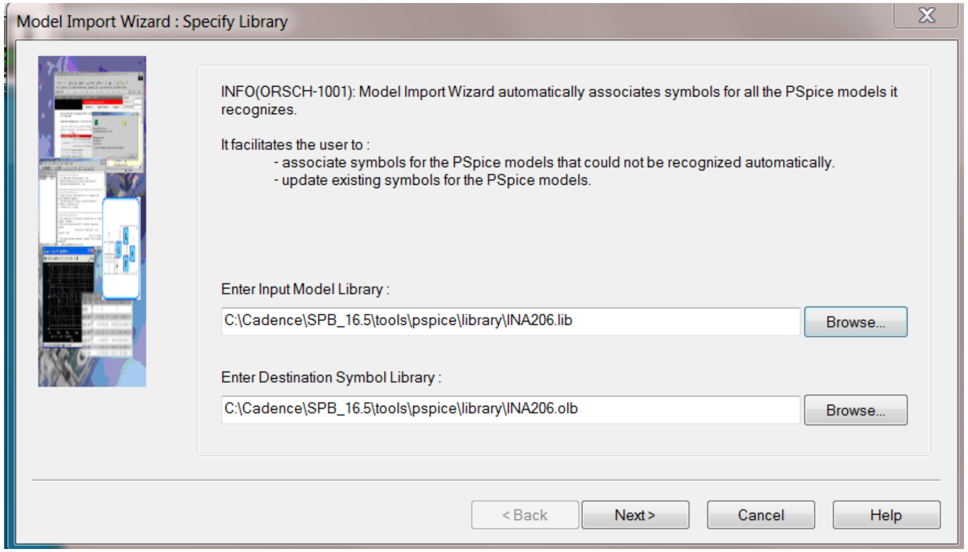
Create Model

- Copy text model as *.lib file to
  - C:\Cadence\SPB_16.x\tools\pspice\library



Create Model

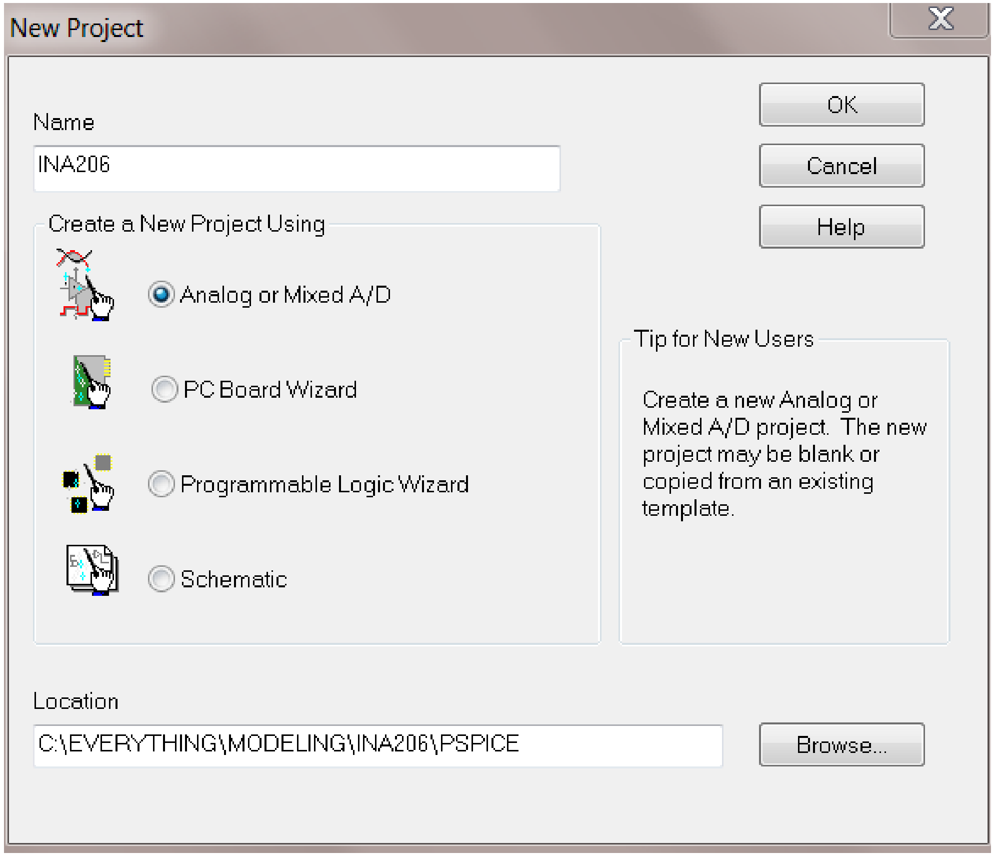
- Open Model Editor
  – Start->All Programs->Cadence->Release 16.5->PSpice Accessories->Model Editor
- File->Model Import Wizard
- Browse to *.lib file
- Click Next, then Finish



- Allow the program to create a symbol for the model
- Click OK
- Close Model Editor

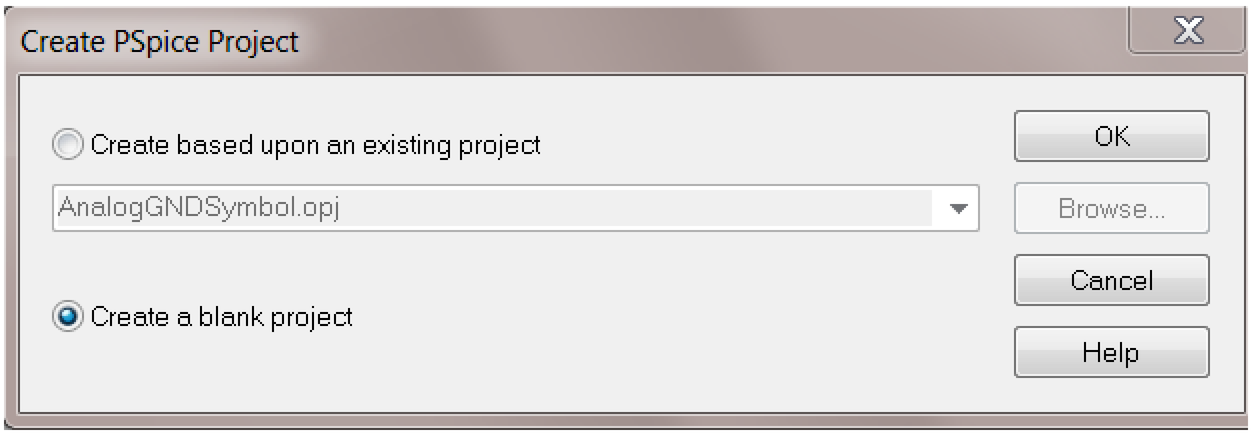
Create New Project

- Start->All Programs->Cadence->Release 16.5->OrCAD Capture
- File->New Project



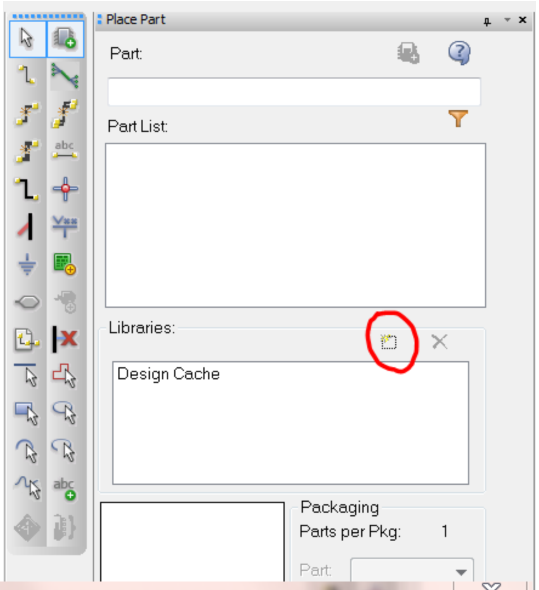
Create New Project

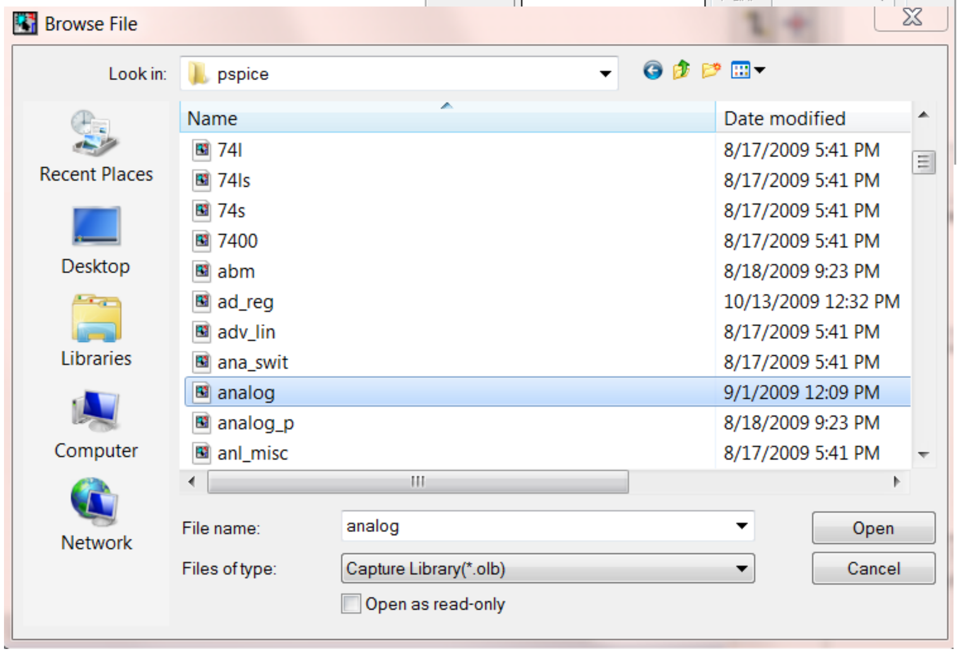
- Use a blank project



Add Libraries

- Type “p” to place part
- Click button shown to add library
- Add the following (in capture\library\pspice)
  - ‘**analog.olb**’
  - Use for basic components
    - Resistors
    - Capacitors
    - Inductors
    - Ideal op amp
    - Switches
    - Transmission line
    - Transformer
    - Controlled sources





Add Libraries

- Also add ‘**source.olb**’

If the analog portion of your circuit requires DC power, then you need to include a DC source in your design. To specify a DC source, use one of the following parts.

Table 12

For this source type...	Use this part...
voltage	VDC or VSRC
current	IDC or ISRC

- So, for DC and AC analysis use VSRC and ISRC
- For transient, you can use VSRC and ISRC but Cadence recommends using VSTIM and ISTIM from ‘**sourcestm.olb**’ library

Analog stimuli include both voltage and current sources. The following table shows the part names for voltage sources.

Table 13

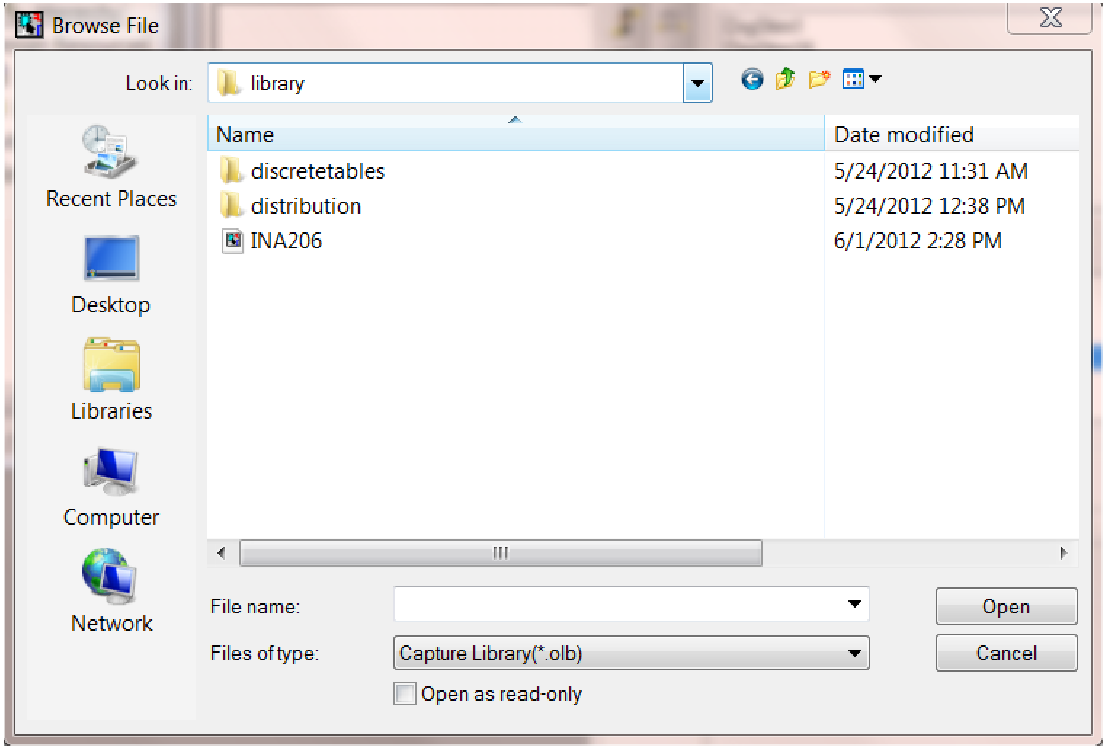
If you want this kind of input...	Use this part for voltage...
For DC analyses	
DC bias	VDC or VSRC
For AC analyses	
AC magnitude and phase	VAC or VSRC
For transient analyses	
exponential	VEXP or VSTIM*
periodic pulse	VPULSE or VSTIM*
piecewise-linear	VPWL or VSTIM*
piecewise-linear that repeats forever	VPWL_RE_FOREVER or VPWL_F_RE_FOREVER**
piecewise-linear that repeats n times	VPWL_N_TIMES or VPWL_F_N_TIMES**
frequency-modulated sine wave	VSFFM or VSTIM*
sine wave	VSIN or VSTIM*

* VSTIM and ISTIM parts require the Stimulus Editor to define the input signal.

** VPWL_F_RE_FOREVER and VPWL_F_N_TIMES are file-based parts; the stimulus specification is saved in a file and adheres to PSpice netlist syntax.

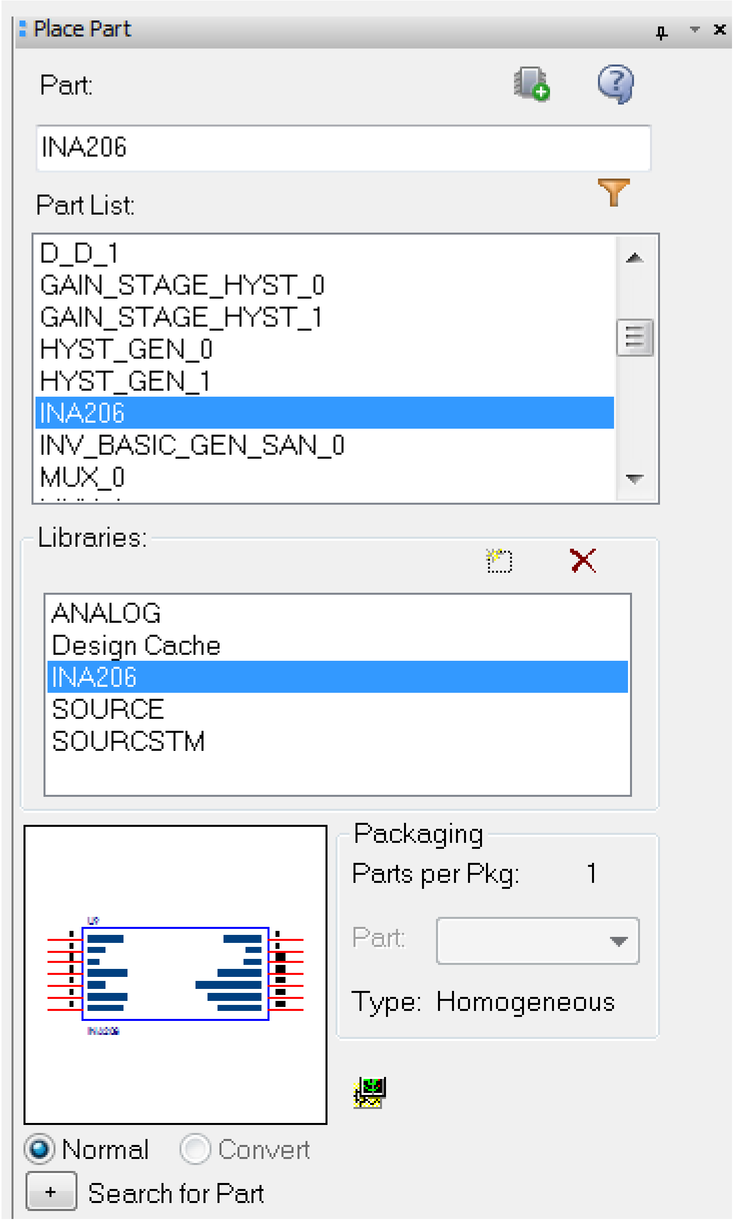
Add Libraries

- We also need to add the library for the model we just created.
- It was saved to C:\Cadence\SPB_16.5\tools\pspice\library
  - As opposed to C:\Cadence\SPB_16.5\tools\capture\library\pspice for the other libraries



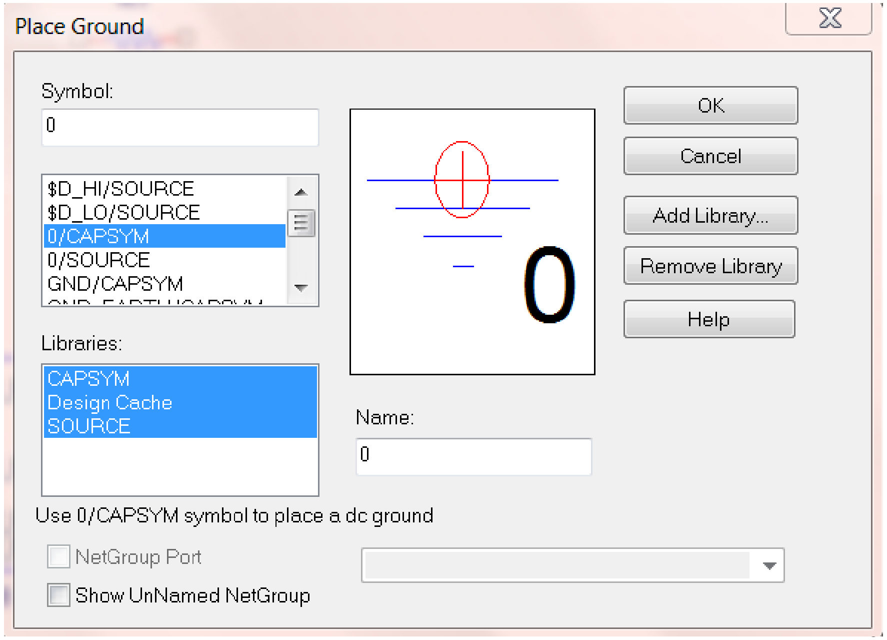
Build Circuit

- Now that we have all of the libraries we can build the circuit.
- Note that for this example, the INA206 subcircuit is made of a plethora of other subcircuits. Therefore we will instantiate the top-level subcircuit (INA206)

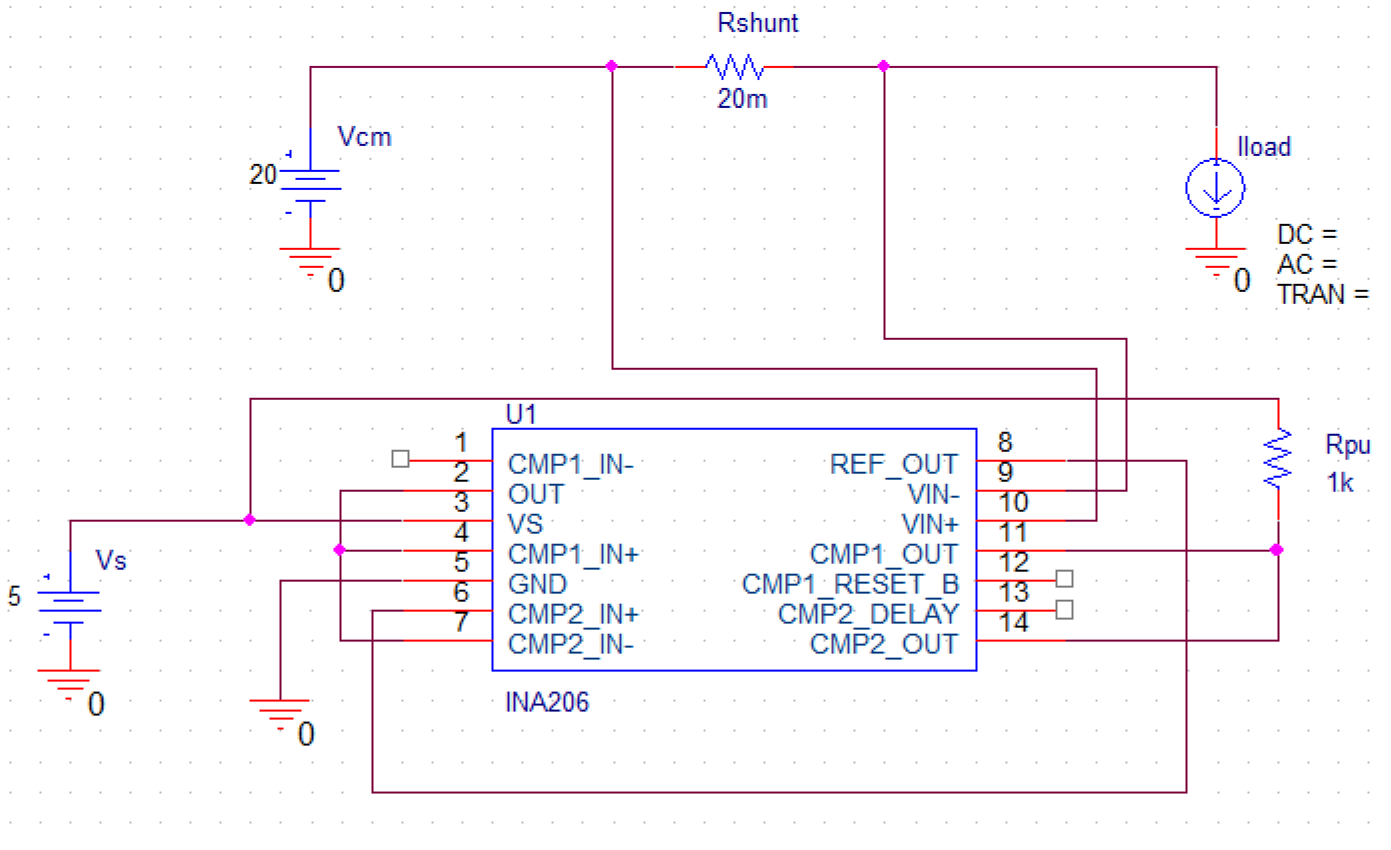


Build Circuit

- Use VDC for power supply and common-mode voltage
  - To change name and value, simply double-click the text
- Use ISRC for load current
- For ground, press ‘g’ and use 0/CAPSYM
- Press ‘w’ to place a wire
- Use ‘R’ from Analog library

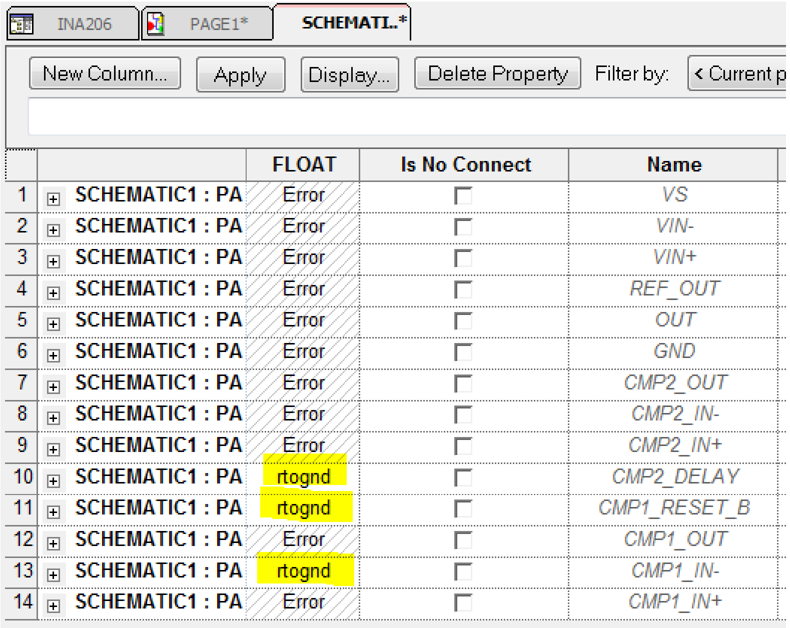


Build Circuit



Build Circuit

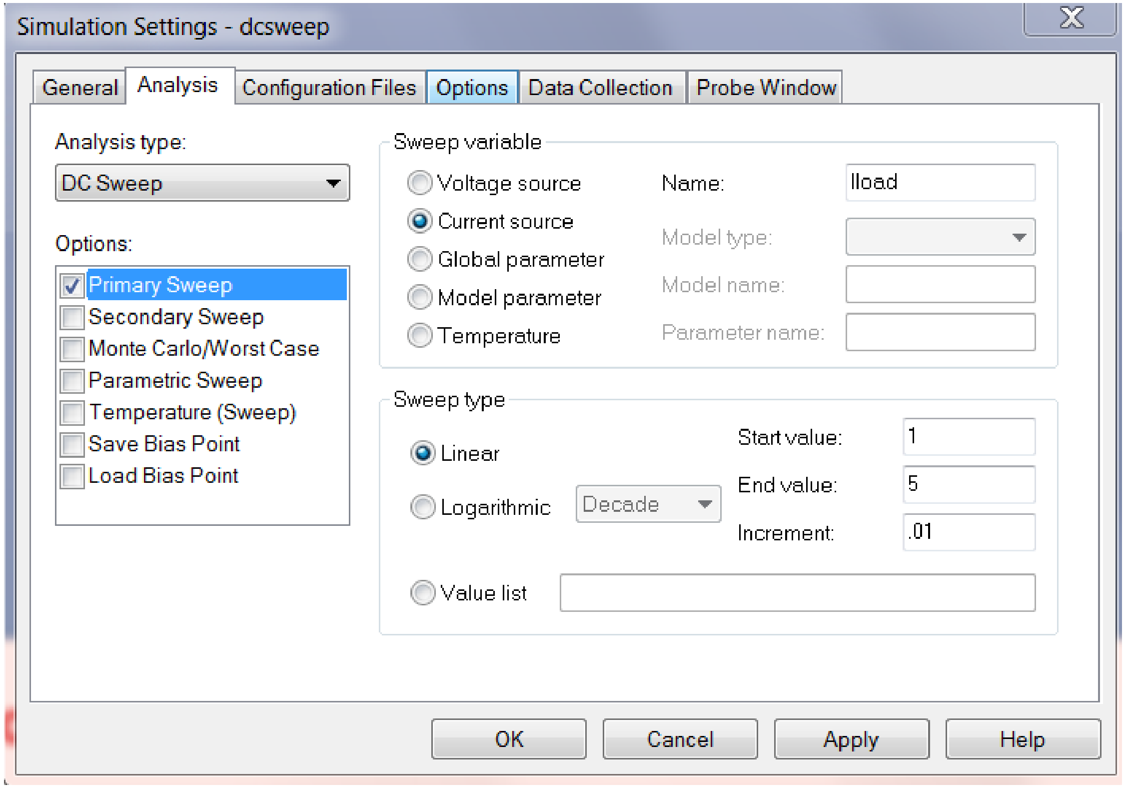
- PSpice may give errors because of the unconnected pins. To fix:
  - Select the symbol (single left-click)
  - Right-click
  - Edit Properties
  - Select “Pins” tab at the bottom
  - In FLOAT column right-click and select Edit
  - Change to “rtognd” meaning ‘resistor to ground’
  - Do this for all unconnected pins



		FLOAT	Is No Connect	Name
1	SCHEMATIC1 : PA	Error		VS
2	SCHEMATIC1 : PA	Error		VIN-
3	SCHEMATIC1 : PA	Error		VIN+
4	SCHEMATIC1 : PA	Error		REF_OUT
5	SCHEMATIC1 : PA	Error		OUT
6	SCHEMATIC1 : PA	Error		GND
7	SCHEMATIC1 : PA	Error		CMP2_OUT
8	SCHEMATIC1 : PA	Error		CMP2_IN-
9	SCHEMATIC1 : PA	Error		CMP2_IN+
10	SCHEMATIC1 : PA	rtognd		CMP2_DELAY
11	SCHEMATIC1 : PA	rtognd		CMP1_RESET_B
12	SCHEMATIC1 : PA	Error		CMP1_OUT
13	SCHEMATIC1 : PA	rtognd		CMP1_IN-
14	SCHEMATIC1 : PA	Error		CMP1_IN+

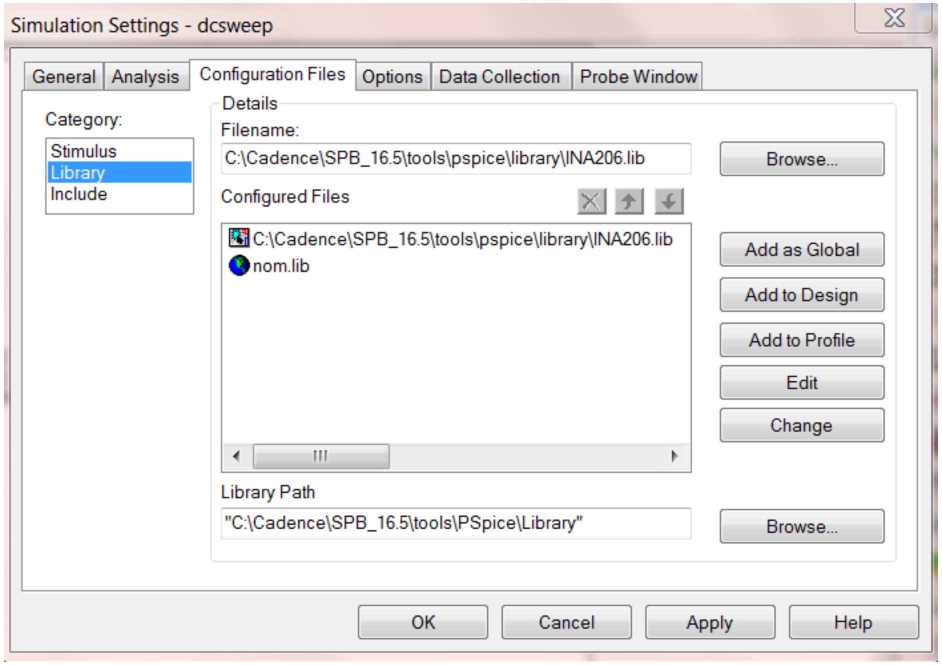
Simulate-DC Sweep

- PSpice->New Simulation Profile
- Give it a name (e.g. dcsweep)

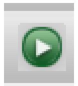


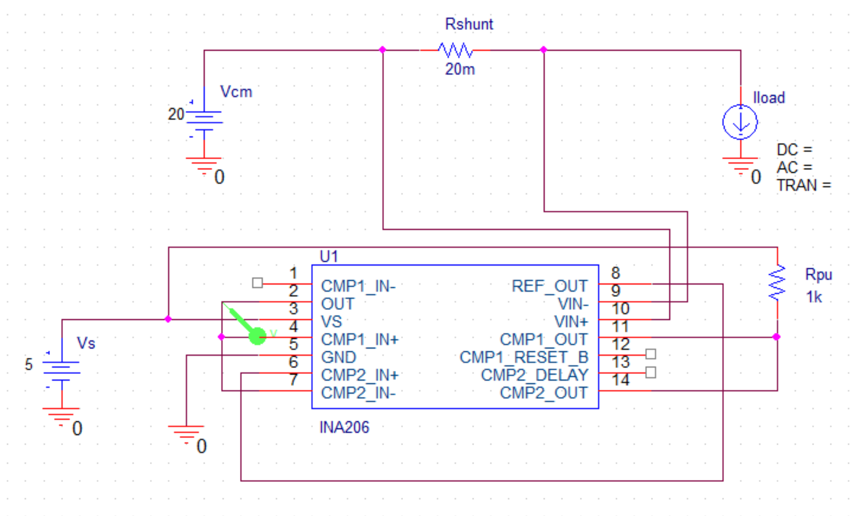
Simulate-DC Sweep

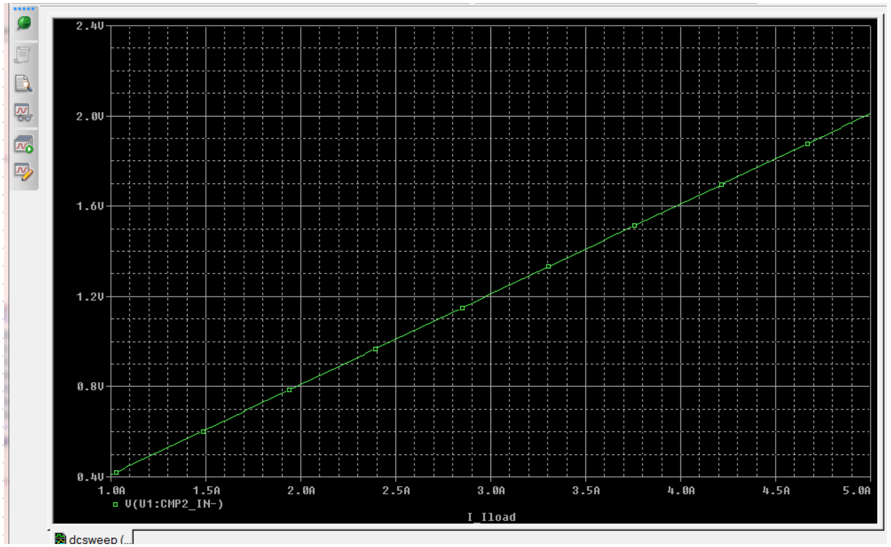
- This is very important: We need to add the library file to the simulation so it can find the subcircuits of the INA206 subcircuit
  - Select Configuration Files tab
  - Change Category to Library and Browse to INA206.lib file
    - C:\Cadence\SPB_16.5\tools\pspice\library\
  - Add to Design
  - Apply
  - OK



Simulate-DC Sweep

- Add probes to schematic as needed 
- Run PSpice 



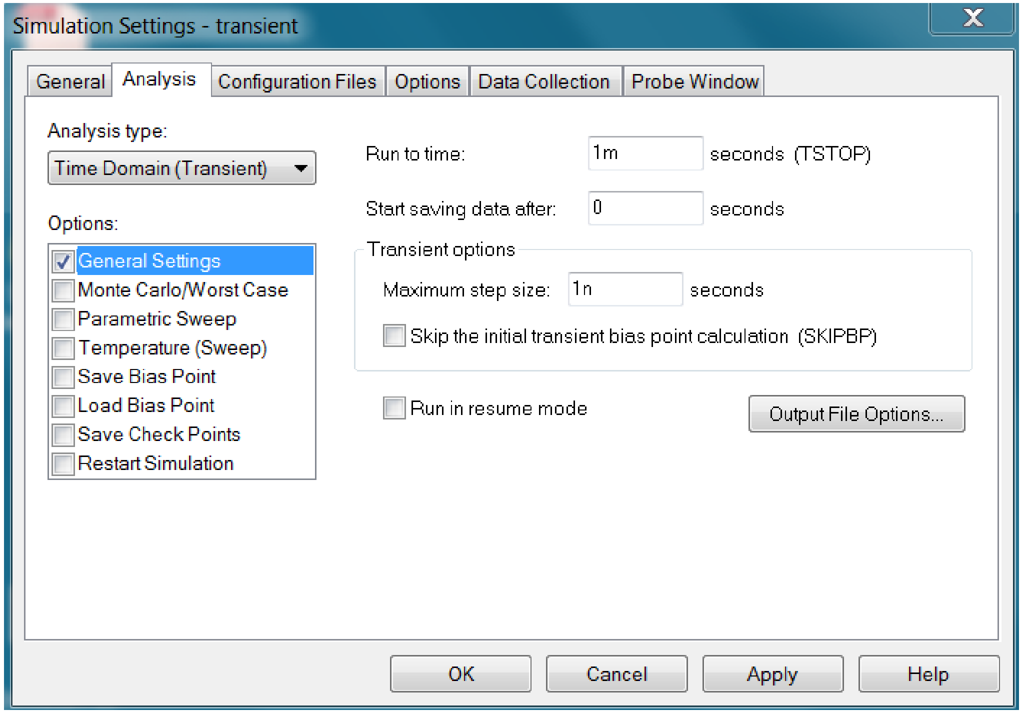


- Note: you can interactively add probes in schematic window and the simulation window will update accordingly

Texas Instruments

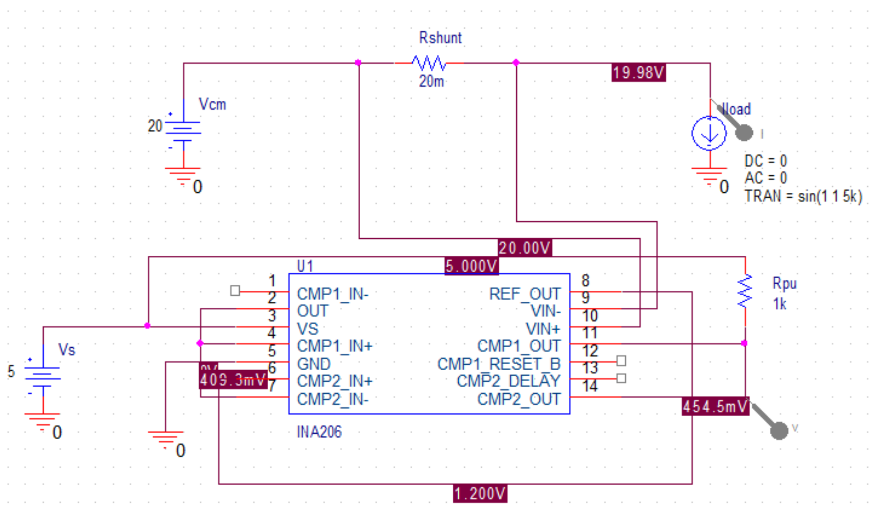
Simulate-Transient

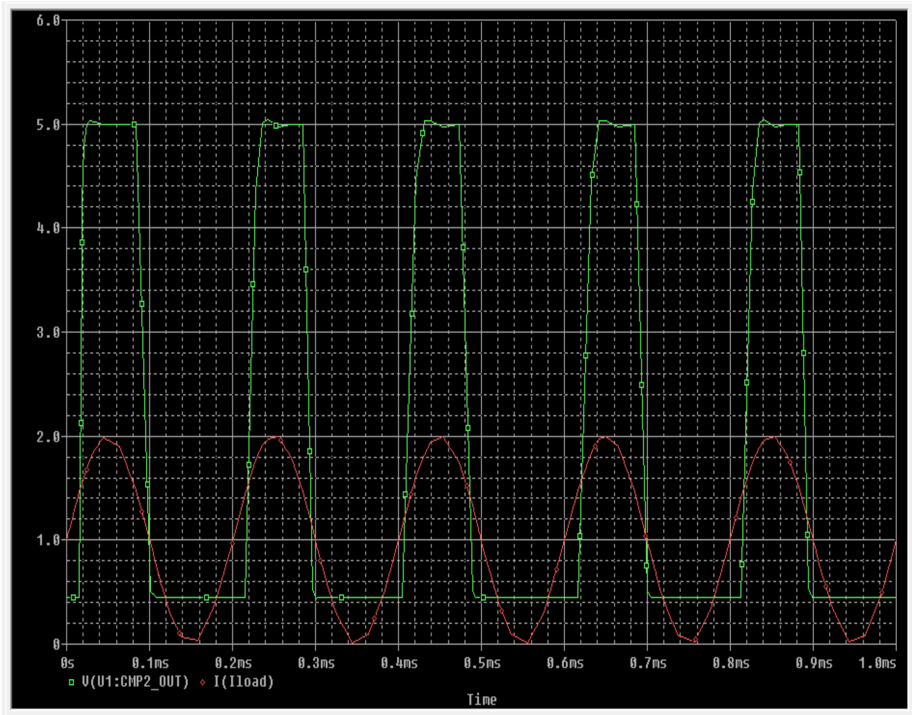
- PSpice->New Simulation Profile
- Give it a name (e.g. transient)
- Fill in parameters accordingly
- Apply
- OK



Simulate-Transient

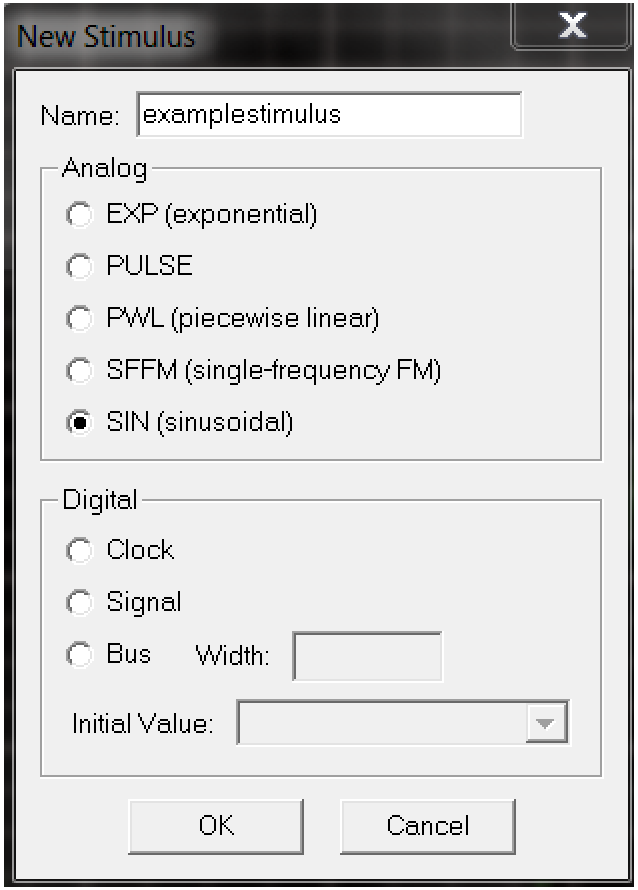
- Need to edit Iload for transient analysis
- Double-click TRAN and enter “sin(1 1 5k)”
- Run simulation





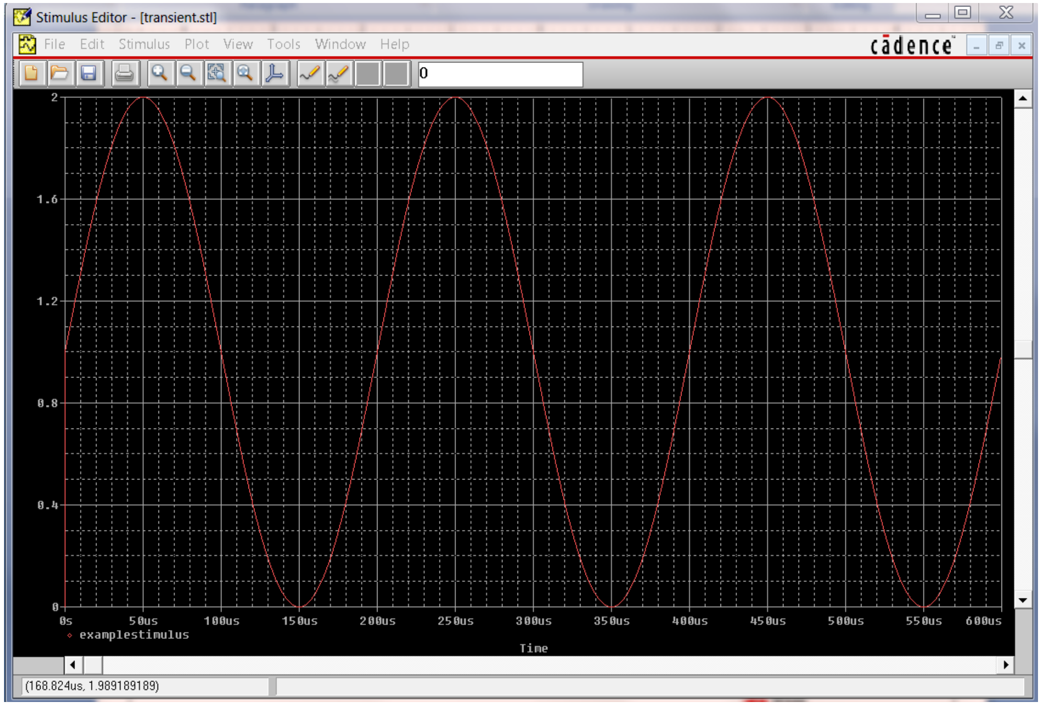
Simulate-Transient

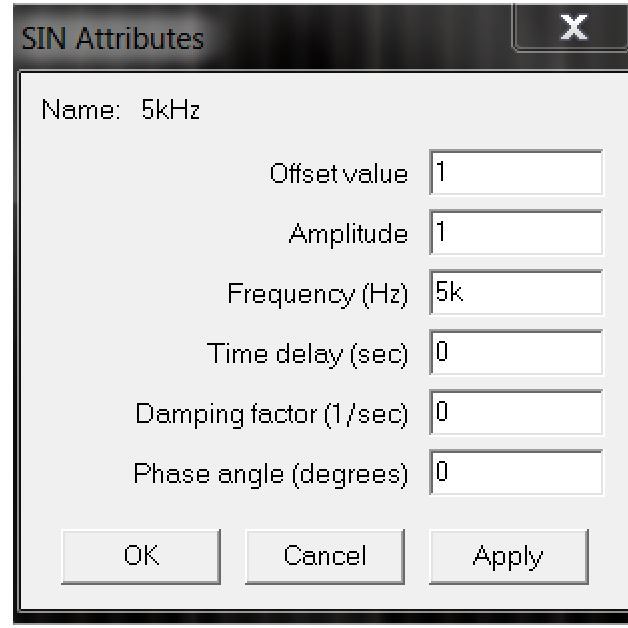
- Or you could change ISRC to ISTIM (from SOURCESTM library) and use Stimulus Editor
- Right-click, select Edit PSpice Stimulus
- Give the implementation a name and choose stimulus type
- OK



Simulate-Transient

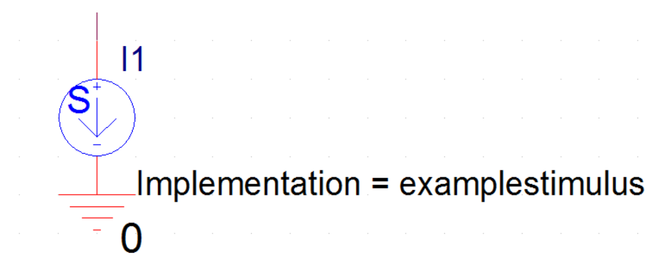
- Fill in attributes accordingly and click OK





- Close Stimulus Editor (let it update schematic)

- Run Simulation



Backup Slides

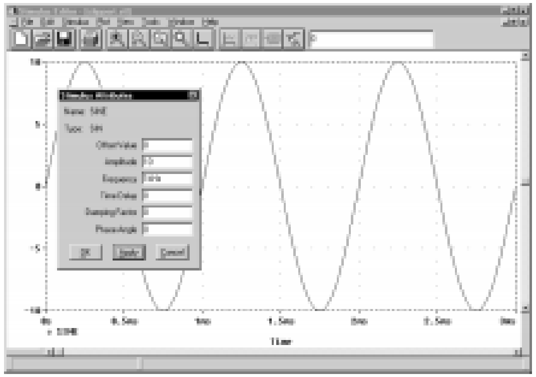
Stimulus Editor

What is the Stimulus Editor?

The Stimulus Editor is a graphical input waveform editor that lets you define the shape of time-based signals used to test your circuit's response during simulation. Using the Stimulus Editor, you can define:

- analog stimuli with sine wave, pulse, piecewise linear, exponential pulse, single-frequency FM shapes

The Stimulus Editor lets you draw analog piecewise linear stimuli by clicking at the points along the timeline that correspond to the input values that you want at transitions.

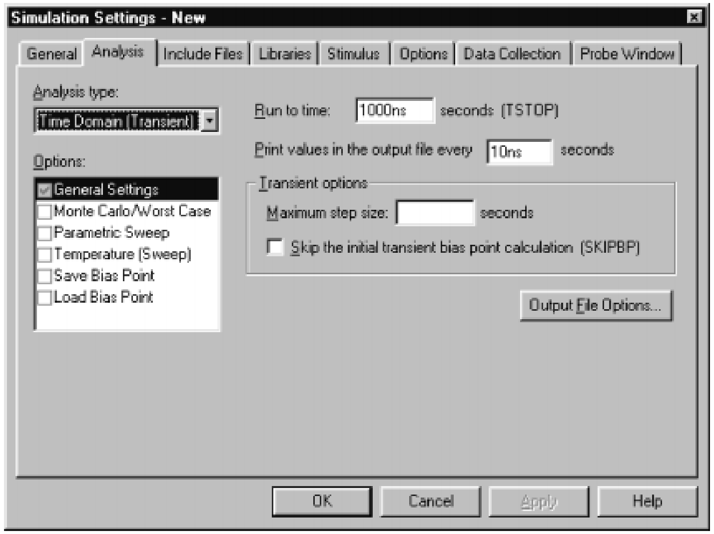


Add Simulation

Setting up analyses

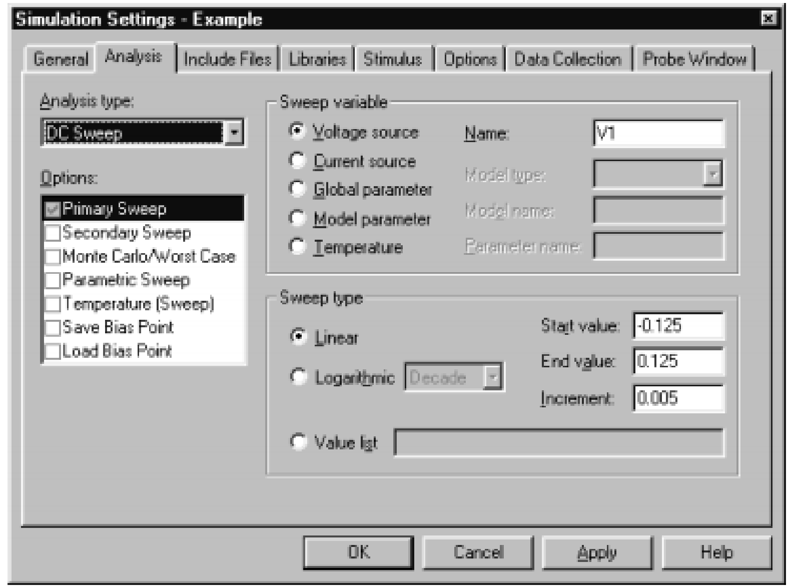
To set up one or more analyses

1. From the PSpice menu, choose New Simulation Profile.
2. Enter the name of the profile and click OK.
3. Click the Analysis tab if it is not already the active tab in the dialog box.



4. Enter the necessary parameter values and select the appropriate check boxes to complete the analysis specifications.
5. Set up any other analyses you want to perform for the circuit by selecting any of the remaining analysis types and options, then complete their setup dialog boxes.

DC Sweep



1 In Capture, select New Simulation Profile or Edit Simulation Settings from the PSpice menu. (If this is a new simulation, enter the name of the profile and click OK.)

The Simulation Settings dialog box appears.

2 Under Analysis type, select DC Sweep.

3 For the Primary Sweep option, enter the necessary parameter values and select the appropriate check boxes to complete the analysis specifications.

4 Click OK to save the simulation profile.

5 Select Run under the PSpice menu to start the simulation.

Note *Do not specify a DC sweep and a parametric analysis for the same variable.*

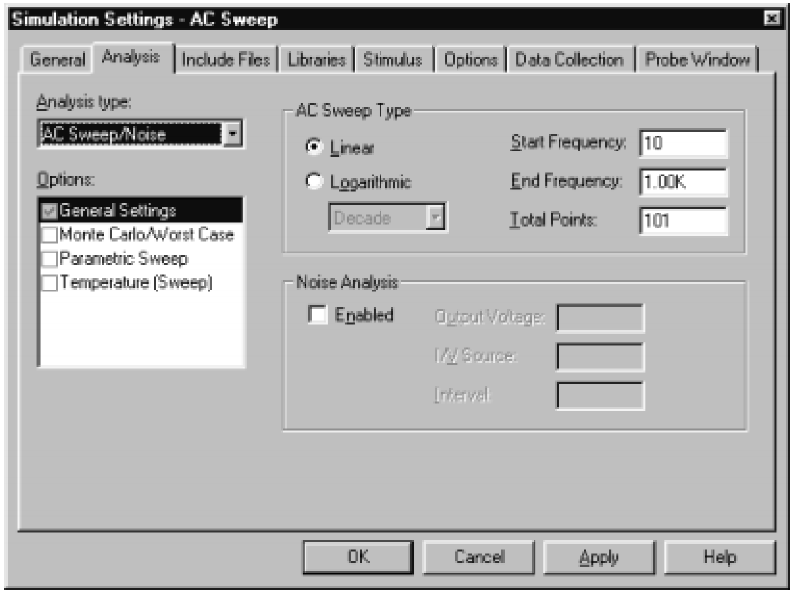
AC Sweep

Setting up an AC analysis

To set up the AC analysis

1 From the PSpice menu, choose New Simulation Profile or Edit Simulation Settings. (If this is a new simulation, enter the name of the profile and click OK.)

The Simulation Settings dialog box appears.

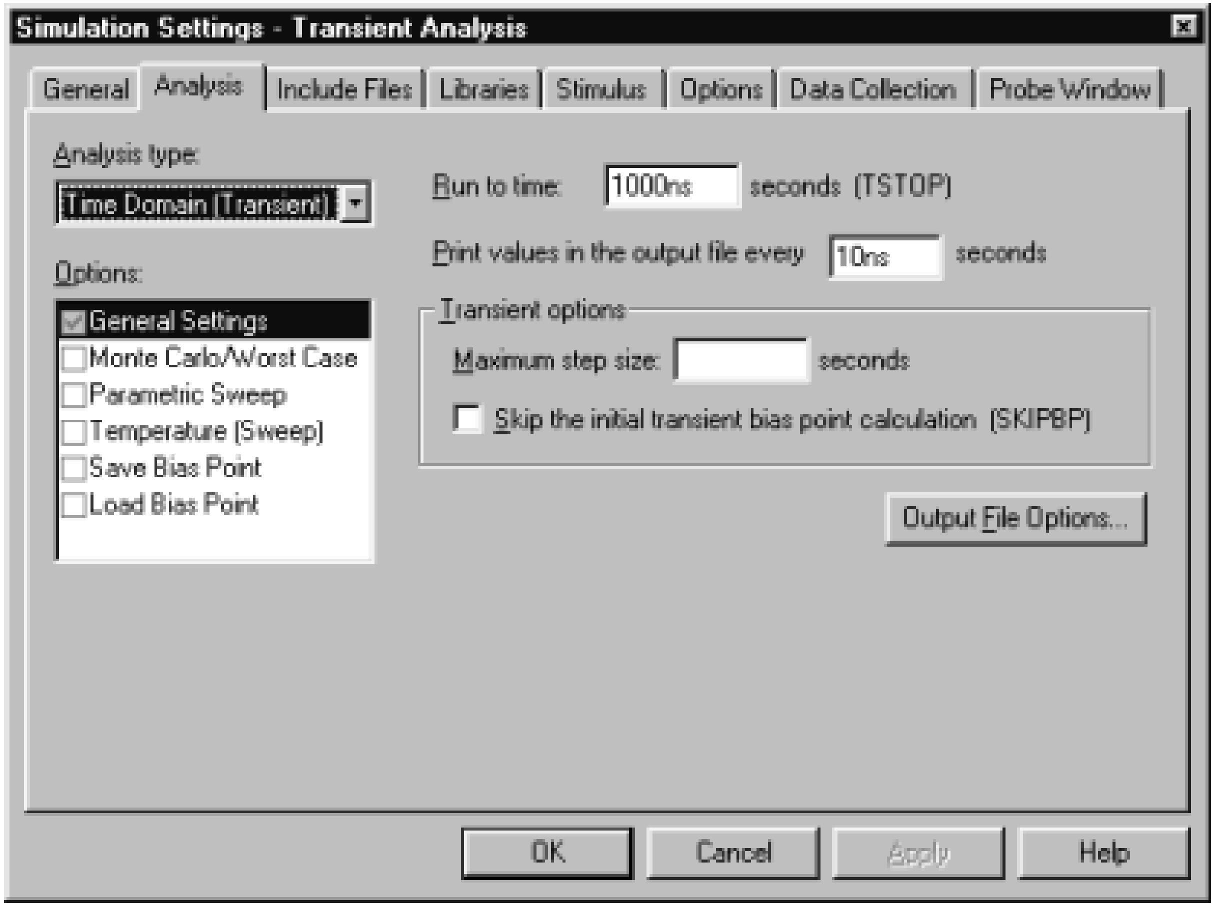


2 Choose AC Sweep/Noise in the Analysis type list box.

3 Under Options, select General Settings if it is not already enabled.

4 Set the number of sweep points as follows:

Transient



Simulation Settings - Transient Analysis
General
Analysis
Include Files
Libraries
Stimulus
Options
Data Collection
Probe Window
Analysis type:
Time Domain (Transient)
Options:
General Settings
Monte Carlo/Worst Case
Parametric Sweep
Temperature (Sweep)
Save Bias Point
Load Bias Point
Run to time:
1000ns
seconds (TSTOP)
Print values in the output file every
10ns
seconds
Transient options
Maximum step size:
seconds
Skip the initial transient bias point calculation (SKIPBP)
Output File Options...
OK
Cancel
Apply
Help