University of Technology, Sydney

48551 ADVANCED ELECTRONICS

Faculty of Engineering

OrCAD 10.0 Demo: Computer-Aided Circuit Analysis

3. Time Domain Analysis

Introduction

Time Domain Analysis (also called Transient Analysis or Large-Signal Analysis) allows us to examine the behaviour of a circuit when excitations (input signals) are either non-periodic, or periodic non-sinusoidal, or their amplitudes drive the devices into the nonlinear regions of operation.

It is important to note that the Frequency Domain Analysis (AC Sweep), discussed in Section 2 of this tutorial, uses the small-signal models of nonlinear devices, linearised at the DC operating point. Such small-signal equivalent circuit is linear and its response is always proportional to the input signal. It means that if the circuit's output has the amplitude of 100 mV with the input of 1 mV, then when the input amplitude is increased to 1 V, the output signal, as calculated by PSpice, will have the amplitude of 100V. Even if the real circuit is not capable of producing such an output! In other words, frequency domain analysis results are only valid in the small-signal regime. If you wish to explore the large-signal behaviour of a circuit (a typical example would be the analysis of nonlinear distortions), then the Time Domain Analysis must be used.

Analysis of a class-B output stage with a V_{BE} multiplier

Consider a class-B power amplifier in Fig. 1 (see Comer&Comer, pp. 116-118).

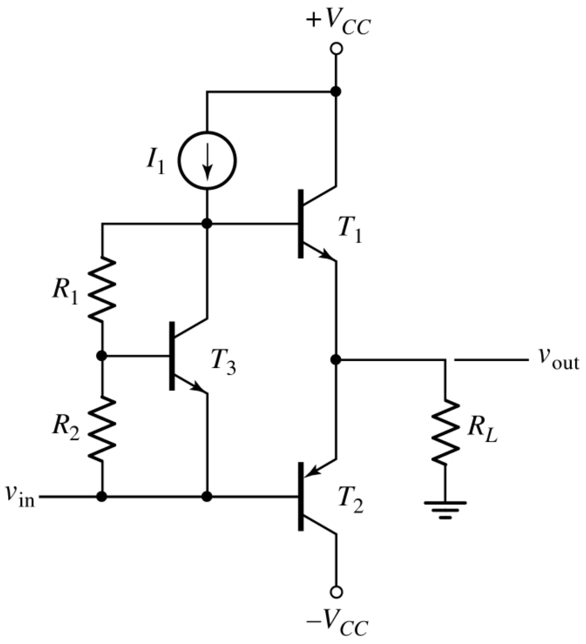


Fig. 1. Class-B output stage with a V_{BE} multiplier.

Using discrete transistors 2N3904, 2N3906 and 2N2222 for T_1, T_2 and T_3, respectively, the other components of the circuit were chosen to be: R_1 = 20 kΩ, R_2 = 10 kΩ, R_L = 8 Ω, I_1 = 39.4 µA and V_{CC} = 15 V.

We want to investigate the operation of this amplifier when a signal of 5V amplitude and 1 kHz frequency is applied to its input.

Transistors T_1 and T_2 should be in a near off state with no signal, so the signal source must have a carefully chosen DC offset (this can be done either by a trial-and-error method or, better, by running a DC sweep on v_{in}).

Create a new project and draw the circuit diagram in the schematics window. First, place the symbols for the n-p-n and the p-n-p BJTs from the BREAKOUT symbol library. Then, change the model labels from `QbreakP` to `Q2N3906` and from `QbreakN` to `Q2N3904` and `2N2222`, as appropriate. After placing all devices, entering their values, wiring and grounding, your diagram should look like the one shown in Fig. 2. The signal source is a sinusoidal voltage source, `VSIN`.

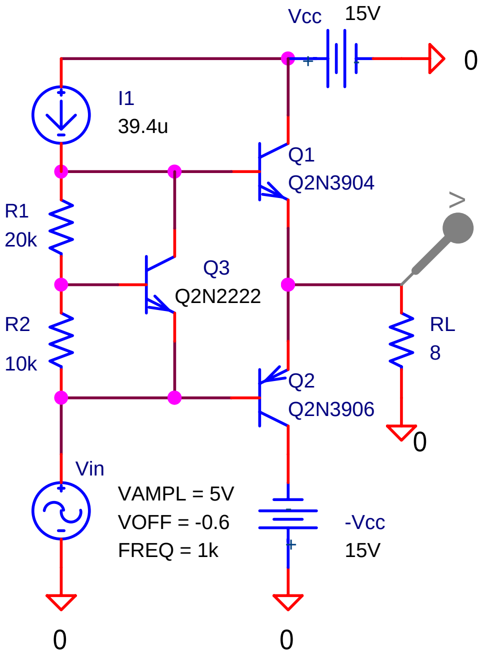
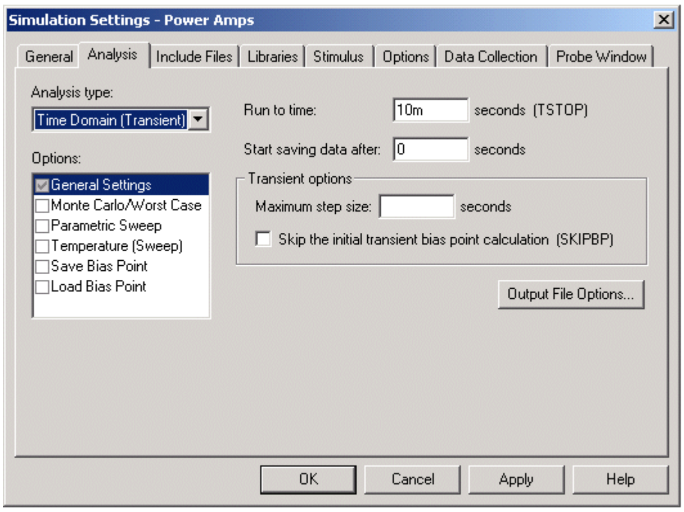


Fig. 2. Class-B output stage with V_{BE} multiplier.

It is always a good idea to run a **Bias Point** simulation first (not shown here) to make sure that the quiescent operating points of all devices are correct.

To run a time domain simulation, use the **PSpice -> Edit Simulation Profile** command to bring up the *Simulation Settings* window. Then, click on the *Analysis* tab and select the `Time Domain (Transient)` analysis type; specify the *Run to time* at 10 ms, as shown in Fig. 3a. If you wish to calculate the nonlinear distortion coefficient, click on the **Output File Options...** button and specify the Fourier analysis parameters in the *Transient Output File Options* window, shown in Fig. 3b.



Analysis type: Time Domain (Transient)

Options:
- ☑ General Settings
- ☐ Monte Carlo/Worst Case
- ☐ Parametric Sweep
- ☐ Temperature (Sweep)
- ☐ Save Bias Point
- ☐ Load Bias Point

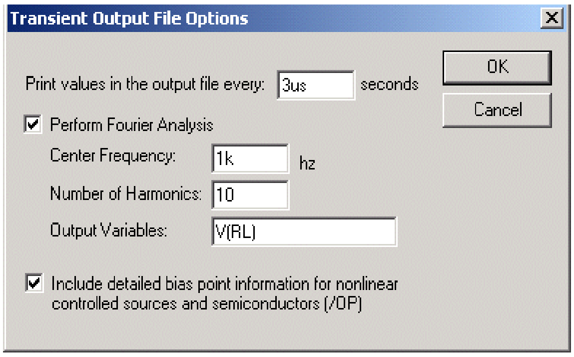
Run to time: 10m seconds (TSTOP)

Start saving data after: 0 seconds

Transient options

Maximum step size: seconds

☐ Skip the initial transient bias point calculation (SKIPBP)



Print values in the output file every: 3us seconds

☑ Perform Fourier Analysis

Center Frequency: 1k hz

Number of Harmonics: 10

Output Variables: V(RL)

☑ Include detailed bias point information for nonlinear controlled sources and semiconductors (/OP)

Fig. 3a. Specifying Time Domain Analysis.

Fig. 3b. Requesting Fourier Analysis.

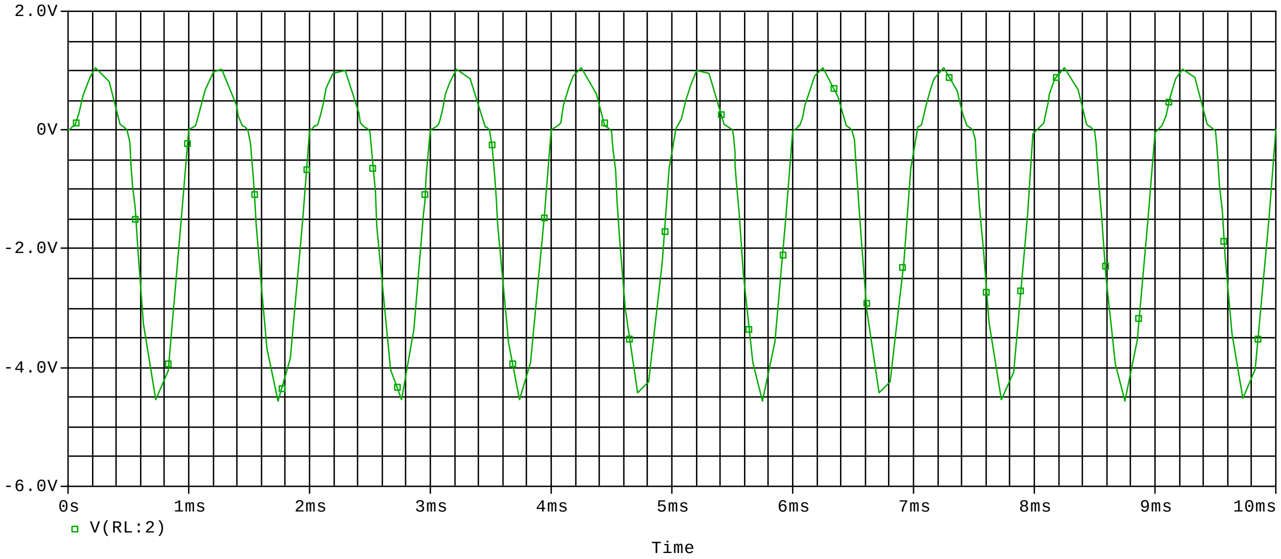
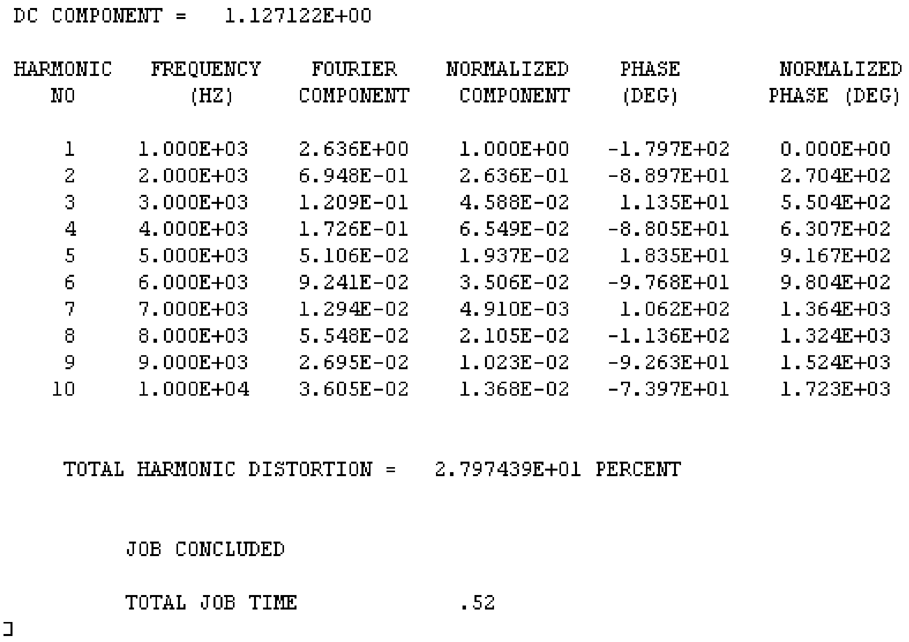


Fig. 4. Output voltage waveform for a circuit in Fig. 2.

Running the transient analysis produces a time plot of the output voltage, as shown in Fig. 4. A very significant distortion of the positive-going output waveform is clearly evident. Since we requested Fourier analysis, we can find the amplitudes of the fundamental frequency component (harmonic #1) and all other requested harmonics (in the *Transient Output File Options*, Fig. 3b, we've specified the number of harmonics as 10). The total harmonic distortion is also calculated [cf. equation (3.4) on p. 109 of Comer&Comer]. To display the output file, use **View -> Output File** command in the Probe window. The relevant fragment of the output file for this simulation is shown in Fig. 5.

It is obvious that the stage needs to be re-designed in order to reduce the distortion to acceptable levels (a few % would be the maximum acceptable). It is beyond the scope of this tutorial, but you are encouraged to try improving this circuit's performance as an exercise.



```
DC COMPONENT =   1.127122E+00

HARMONIC   FREQUENCY    FOURIER    NORMALIZED    PHASE        NORMALIZED
   NO         (HZ)      COMPONENT   COMPONENT    (DEG)        PHASE (DEG)

    1      1.000E+03    2.636E+00   1.000E+00   -1.797E+02    0.000E+00
    2      2.000E+03    6.948E-01   2.636E-01   -8.897E+01    2.704E+02
    3      3.000E+03    1.209E-01   4.588E-02    1.135E+01    5.504E+02
    4      4.000E+03    1.726E-01   6.549E-02   -8.805E+01    6.307E+02
    5      5.000E+03    5.106E-02   1.937E-02    1.835E+01    9.167E+02
    6      6.000E+03    9.241E-02   3.506E-02   -9.768E+01    9.804E+02
    7      7.000E+03    1.294E-02   4.910E-03    1.062E+02    1.364E+03
    8      8.000E+03    5.548E-02   2.105E-02   -1.136E+02    1.324E+03
    9      9.000E+03    2.695E-02   1.023E-02   -9.263E+01    1.524E+03
   10      1.000E+04    3.605E-02   1.368E-02   -7.397E+01    1.723E+03


    TOTAL HARMONIC DISTORTION =   2.797439E+01 PERCENT


         JOB CONCLUDED

         TOTAL JOB TIME            .52
]
```

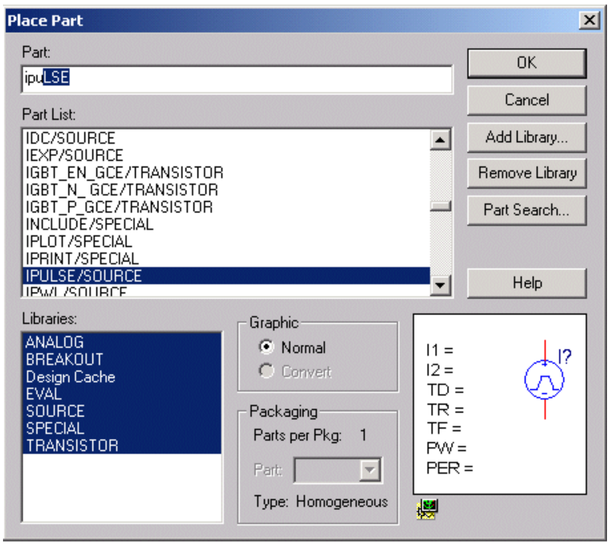
Fig. 5. Fragment of the PSpice output file, showing Fourier analysis of the amplifier's output waveform.

Step response of a parallel resonant circuit

Another application of the Time Domain Analysis is obtaining circuit response to arbitrary waveforms. Suppose we want to analyse the time domain response to a step current of an inductor current in a parallel RLC circuit. We may start placing parts on the diagram by selecting the pulse current source, IPULSE, as shown in Fig. 6. After placing the pulse current source, we need to specify its parameters. Various pulse parameters are defined in Table 1.

Table 1: Pulse current source parameters.

Parameter	Description	Units	Default
<i1>	Initial current	A	none
<i2>	Pulsed current	A	none
<td>	Delay	s	TSTEP
<tr>	Rise time	s	TSTEP
<tf>	Fall time	s	TSTEP
<pw>	Pulse width	s	TSTOP
<per>	Period	s	TSTOP



Part: ipuLSE

● Normal ○ Convert

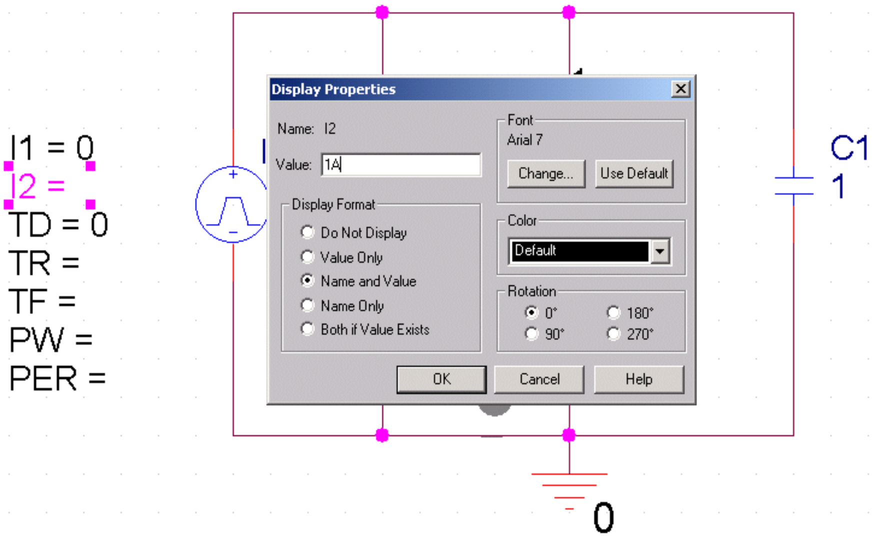
Parts per Pkg: 1

Type: Homogeneous

Fig. 6. Selecting a pulse current source.

To enter/change a source parameter, double click on the required parameter and enter its value in the *Display Properties* window, as shown in Fig. 7. From the Table 1 we see that in order to specify a step pulse of 1 A, beginning at $t = 0$, we need only to specify three parameters: I1 = 0, I2 = 1A and TD = 0. The other parameters will be given their default values.

If we chose the values of L = 0.4 H and C = 0.4 F, the undamped resonance frequency will be $\omega_0 = 1/\sqrt{LC} = 2.5\,\mathrm{s}^{-1}$. The quality factor, given by $Q = R/(\omega_0 L)$, will be numerically equal to



Name: I2

Value: 1A

Display Format: ● Name and Value

Color: Default

Rotation: ● 0°

Fig. 7. Changing a source parameter.

R. We can now easily examine the behaviour of the circuit under overdamped ($Q < ½$), critically damped ($Q = ½$) and underdamped ($Q > ½$) conditions. We could simply set the resistance to a required value and run the time domain simulation to obtain a single plot of $i_L(t)$. For example, Fig. 8 shows the inductor current when $R = 10\ \Omega$.

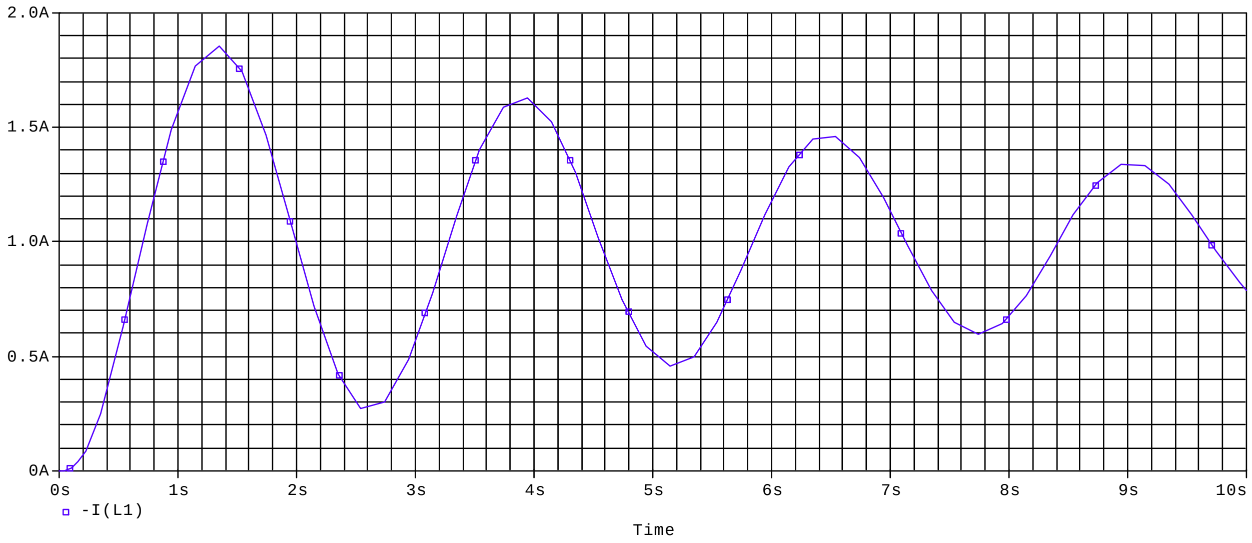
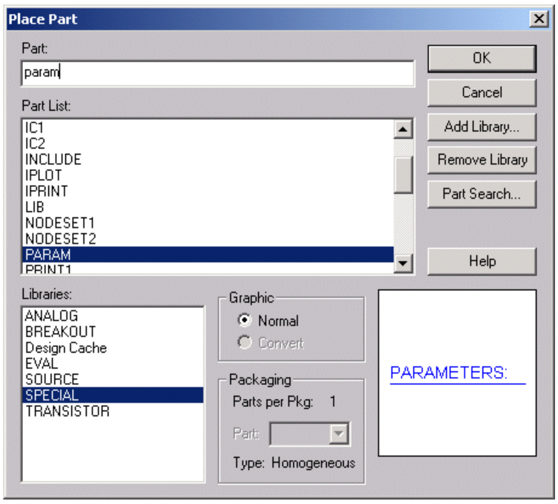


Fig. 8. Inductor current in underdamped conditions ($R = 10\ \Omega$).

It may be beneficial, however, to be able to plot several responses on a single graph. This can be accomplished in PSpice by the use of the **Parametric Sweep** function. First, double-click on the resistor value to bring up the *Display Properties* dialogue box. In the *Value* text box, replace `10` with `{Rd}`. Then, select a PARAM part from the SPECIAL.OLB library, as shown in Fig. 9. Place this 'part' in any open area on the schematic page. Double click on the PARAM part to display the *Property Editor* spreadsheet, then click the **New Column** button. In the *Add New Column* dialogue box enter Rd (no curly braces!) into the *Name:* text box and `0.1` (initial value) in the *Value:* text box. Close the *Add New Column* dialogue box by clicking on the **Apply** button. This creates a new property for the PARAM part, as shown by the new column label Rd in the spreadsheet. Select the cell below the Rd label. Click **Display**. In the *Display Properties* dialogue box, select *Display Format: Name and Value*. Click **Apply** and close the Property Editor. Your circuit diagram should look like the one shown in Fig. 10.


Place Part

Part: param

Part List: PARAM (selected)

Libraries: SPECIAL (selected)

Graphic: ● Normal ○ Convert

Packaging: Parts per Pkg: 1

Type: Homogeneous

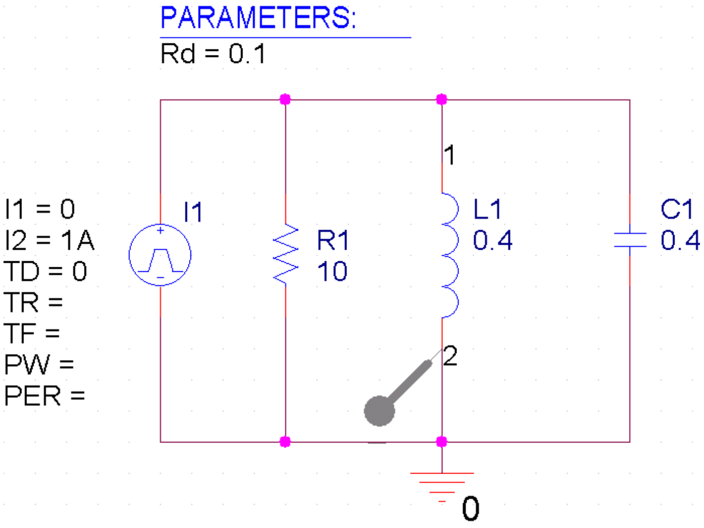
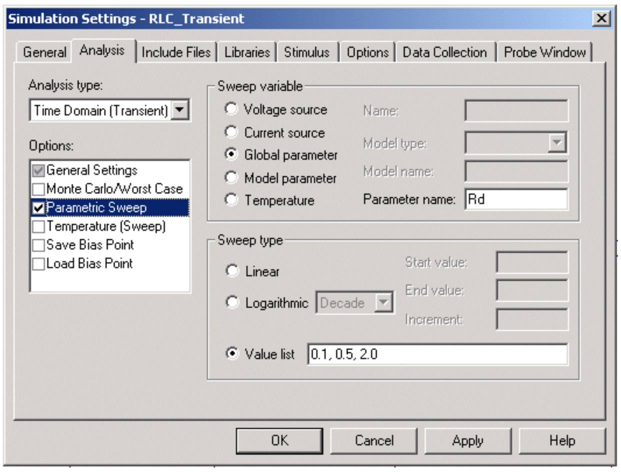


Fig. 9. Adding PARAM part.

Fig. 10. Circuit ready for a parametric sweep on R1.

To set up and run a parametric sweep to step the value of R1 using Rd, select **PSpice -> Edit Simulation Profile** from Capture menu. Under the *Analysis* tab, select *Analysis type:* Time Domain (Transient), check the Parametric Sweep option and enter the sweep variable and type as shown in Fig. 11. Under General Settings, set the *Run to time* value to `10` seconds. Close the simulation Settings dialogue box and run PSpice.



Simulation Settings - RLC_Transient

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

Sweep variable
- ○ Voltage source
- ○ Current source
- ● Global parameter
- ○ Model parameter
- ○ Temperature

Name:
Model type:
Model name:
Parameter name: Rd

Sweep type
- ○ Linear
- ○ Logarithmic Decade
- ● Value list 0.1, 0.5, 2.0

Start value:
End value:
Increment:

OK | Cancel | Apply | Help

Fig. 11. Setting up a parametric sweep.

You should get the result as shown in Fig. 12 (the trace labels have been entered in Probe, using the *Text Label* tool).

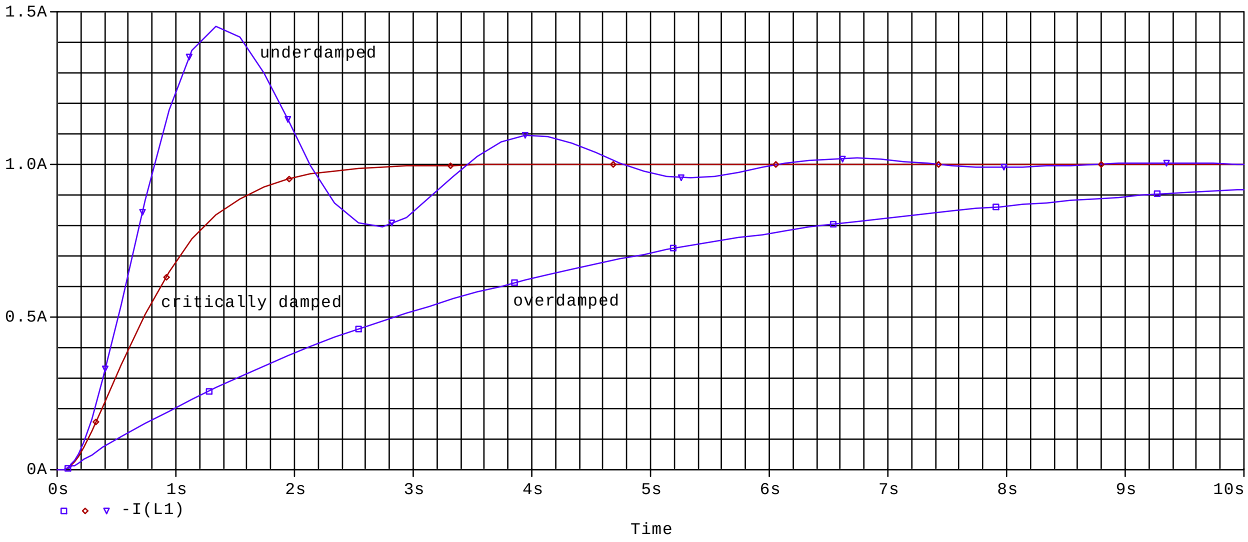


Fig. 12. Results of a parameter sweep for the parallel RLC circuit in Fig. 10.

Note: To obtain a 'black on white' plot, like the one shown above, in the Probe click on `Window -> Copy to Clipboard` and select the `Foreground: change white to black` option. Then paste the plot from Clipboard to your document.