Qucs

Report

Verilog-A Modular Macromodel for Operational Amplifiers

Mike Brinson

Introduction

Since the release of Qucs version 0.0.11 the Qucs team has spent a lot of time and effort improving and debugging the Qucs device and circuit modelling facilities. Initially, the primary device and circuit modelling technique available to users was based on a subcircuit approach that allowed new models to be formed from the built in components distributed with each release of the package[1]. Releases 0.0.11 and 0.0.12 added model construction centred around the nonlinear equation defined device (EDD) and ADMS compact device models. For these modelling techniques preliminary documentation and application information has been posted on the Qucs Web site[2], outlining the fundamental basis of these procedures. Unfortunately, both the EDD and ADMS modelling techniques do require a level of specialised knowledge which many Qucs users may be unfamiliar with. These notes have been written in the hope that the information they provide will help add to the body of information published on Qucs modelling and also be useful to other Qucs users who would like to try constructing, and experimenting with, new device and circuit models using the ADMS software. Today Qucs has moved on from simply a GPL circuit simulator with user friendly schematic capture, and has started to evolve into an advanced circuit and device modelling tool which offers features that were not commonly available in previous generations of GPL circuit simulators. This report introduces an ADMS synthesised compact macromodel for the modular operational amplifier previously described in the Qucs tutorial on operational amplifiers.[3]. This macromodel is characterised by a similar symbol to the primitive operational amplifier gain block included with all Qucs releases. However, it provides Qucs with a more realistic general purpose operational amplifier model that can be used with confidence when designing many practical circuits which are dependent on amplifier characteristics for correct operation. The process of compiling and linking ADMS generated C++ models with the Qucs C++ code has been a learning vehicle on my part, allowing some of the complexities of the ADMS Verilog-A compiler to be decoded and understood. The ADMS synthesised operational amplifier macromodel also marks a departure from the previously described Qucs compact device models in that it represents a circuit function rather than a semiconductor device characteristic. The source code for

---

[1]It is also possible to construct models directly in C++ and compile and link them to the main body of the Qucs code. This is the approach taken by the package developers. However, for the average Qucs user who is primarily interested in using the package, rather than taking part in it's development, this route to model development is not really an option.

[2]Mike Brinson, "Qucs Tutorial: Component, compact device and circuit modelling using symbolic equations", `http://qucs.sourceforge.net/docs.html`, and Stefan Jahn, Hélène Parruitte, "Qucs Description: Verilog-AMS interface", `http://qucs.sourceforge.net/docs.html`.

[3]Mike Brinson, "Qucs Tutorial: Modelling Operational Amplifiers", `http://qucs.sourceforge.net/docs.html`.

the modular macromodel is included in this report and the ADMS generated C++ code can be found in the Qucs release source tarballs archive[4]. The procedure used to synthesize and link the model to Qucs followed the route suggested by Stefan and Hélène in their report on the Verilog-AMS/Qucs interface.

A Modular Macromodel for an Operational Amplifier

An introduction to the technical specification and structure of a modular macromodel for a general purpose operational amplifer is given on page 7 of the Qucs operational amplifier tutorial. The characteristics of this operational amplifier macromodel are:

- Input stage: Offset voltage and current, bias current, differential resistance and capacitance.

- Gain stages: Two pole differential response and single zero common-mode response.

- Large signal response: Slew rate limiting.

- Output stage: Resistance, output voltage and current limiting.

Parameters

Name	Symbol	Description	Unit	Default*
GBP	GBP	Gain bandwidth product	Hz	1e6
AOLDC	$AOLDC$	Open-loop differential gain at DC	dB	106.0
FP2	$FP2$	Second pole frequency	Hz	3e6
RO	RO	Output resistance	Ω	75
CD	CD	Differential input capacitance	F	1e-12
RD	RD	Differential input resistance	Ω	2e6
IOFF	$IOFF$	Input offset current	A	20e-9
IB	IB	Input bias current	A	80e-9
VOFF	VOFF	Input offset voltage	V	7e-4
CMRRDC	$CMRRDC$	Common-mode rejection ratio at DC	dB	90.0
FCM	FCM	Common-mode zero corner frequency	Hz	200.0
PSRT	$PSRT$	Positive slew rate	V/s	5e5

---

[4]Available from `http://qucs.sourceforge.net/download.html`

Name	Symbol	Description	Unit	Default*
NSRT	*NSRT*	Negative slew rate	V/s	5e5
VLIMP	*VLIMP*	Positive output voltage limit	V	14
VLIMN	*VLIMPN*	Negative output voltage limit	V	-14
ILMAX	*ILMAX*	Maximum output current at DC	A	35e-3
CSCALE	*CSCALE*	Current limit scale factor		50

* The default parameters are for a typical UA741 Operational Amplifier.

Verilog-A model code

```
//    Qucs modular OP AMP model:
//    Default parameters are for a typical UA741.
//
// This is free software; you can redistribute it and/or modify
// it under the terms of the GNU General Public License as published by
// the Free Software Foundation; either version 2, or (at your option)
// any later version.
//
// Copyright (C), Mike Brinson, mbrin72043@yahoo.co.uk, November 2007.
//
`include "disciplines.vams"
`include "constants.vams"
//
module mod_amp (in_p, in_n, out_p);
inout in_p, in_n, out_p;
electrical in_p, in_n, out_p;
electrical n2, n3, n4, n5, n6, n7, n8, n9, n10, n11, n12;
//
`define attr(txt) (*txt*)
//
parameter real GBP =1e6 from [1 : inf]
  `attr(info="Gain_bandwidth_product_(Hz)");
parameter real AOLDC = 106.0 from [0.01 : inf]
  `attr(info="Open_loop_DC_gain_(dB)");
parameter real FP2 = 3e6 from [0.01 : inf]
  `attr(info="Second_pole_frequency_(Hz)");
parameter real RO = 75 from [0.01 : inf]
  `attr(info="Output_resistance_(Ohm)");
parameter real CD = 1e-12 from [1e-20 : inf]
  `attr(info="Differential_input_capacitance_(F)");
parameter real RD = 2e6 from [0.01 : inf]
  `attr(info="Differential_input_resistance_(Ohm)");
parameter real IOFF = 20e-9 from [1e-20 : inf]
  `attr(info="Input_offset_current_(A)");
parameter real IB = 80e-9 from [1e-20 : inf]
  `attr(info="Input_bias_current_(A)");
parameter real VOFF = 7e-4 from [0 : inf]
  `attr(info="Input_voltage_offset_(A)");
parameter real CMRRDC = 90.0 from [1 : inf]
  `attr(info="Common_mode_rejection_ratio_at_DC_(dB)");
parameter real FCM = 200 from [0.01 : inf]
  `attr(info="Common_mode_zero_corner_frequency_(Hz)");
parameter real PSRT = 5e5 from [1 : inf]
  `attr(info="Positive_slew_rate_(V/s)");
```

3

```verilog
  parameter real NSRT = 5e5 from [1 : inf]
   `attr(info="Negative_slew_rate_(V/s)");
  parameter real VLIMP = 14 from [0.01 : inf]
   `attr(info="Positve_output_voltage_limit_(V)");
  parameter real VLIMN = −14 from [−inf : 0]
   `attr(info="Negative_output_voltage_limit_(V)");
  parameter real ILMAX = 35e−3 from [1e−9 : inf]
   `attr(info="Maximum_DC_output_current_(A)");
  parameter real CSCALE = 50 from [0 : inf]
   `attr(info="Current_limit_scaling_factor");
//
  real RP1, CP1, RP2, CP2;
  real Rdiff, Voffset;
  real CMRR0, CMgain, CCM;
  real Slewratepositive, Slewratenegative;
  real MTWOPI;
//
  analog begin
//
//  Constants
//
    MTWOPI =   6.28318530717958647693;
//
//  Design equations
//
    Voffset = VOFF*5;
    Rdiff = RD/2;
    CMRR0 = pow(10, CMRRDC/20);
    CMgain = 1e6/CMRR0;
    CCM = 1.0/(MTWOPI*1e6*FCM);
    RP1 = pow(10, AOLDC/20);
    CP1 = 1/(MTWOPI*GBP);
    RP2 = 1;
    CP2 = 1/(MTWOPI*FP2);
    Slewratepositive = PSRT/(MTWOPI*GBP);
    Slewratenegative = NSRT/(MTWOPI*GBP);
//
//  Input voltage offset
//
    I(in_p, n7) <+ V(in_p, n7);
    I(in_p, n7) <+ Voffset;
//
    I(in_n, n9) <+ V(in_n, n9);
    I(in_n, n9) <+ −Voffset;
//
//  Input bias currents
//
    I(n7) <+ IB;
    I(n9) <+ IB;
//
//  Input current offset
//
    I(n7,n9) <+ IOFF/2;
//
//  Input differential resistance and capacitance
//
  I(n7, n8) <+ V(n7, n8)/Rdiff;
  I(n9, n8) <+ V(n9, n8)/Rdiff;
  I(n7, n9) <+ ddt(CD*V(n7, n9));
//
//  Common mode stage
//
```

4

```verilog
    I(n6) <+ -CMgain*V(n8);
    I(n6) <+ V(n6);
    I(n6, n10) <+ V(n6, n10)/1e6;
    I(n6, n10) <+ ddt(CCM*V(n6, n10));
    I(n10) <+ V(n10);
//
// Differential mode and common mode signal adder stage
//
    I(n11) <+ -V(n10);
    I(n11) <+ -V(n7, n9);
    I(n11) <+  V(n11);
//
// Slew rate limiting stage
//
    if (V(n11) > Slewratepositive)
                    I(n12) <+ -Slewratepositive;
    else if (V(n11) < -Slewratenegative)
                    I(n12) <+  Slewratenegative;
    else I(n12) <+ -V(n11);

    I(n12) <+ V(n12);
//
// First pole
//
    I(n3) <+ -V(n12);
    I(n3) <+ V(n3)/RP1;
    I(n3) <+ ddt(CP1*V(n3));
//
// Second pole
//
    I(n5) <+ -V(n3);
    I(n5) <+ V(n5)/RP2;
    I(n5) <+ ddt(CP2*V(n5));
//
// Current limiter stage
//
    if (V(n2, out_p) >= ILMAX)
       begin
         I(n4) <+  -V(n5);
         I(n4) <+ CSCALE*V(n5)*(V(n2, out_p) - ILMAX);
         I(n4) <+ V(n4);
       end
    else if (V(n2, out_p) <= -ILMAX)
       begin
         I(n4) <+  -V(n5);
         I(n4) <+ -CSCALE*V(n5)*(V(n2, out_p) + ILMAX);
         I(n4) <+ V(n4);
       end
    else
       begin
         I(n4) <+ -V(n5);
         I(n4) <+  V(n4);
       end
//
// Output resistance
//
    I(n4, n2) <+ V(n4, n2)/(RO-1);
    I(n2, out_p) <+ V(n2, out_p);
//
//
// Voltage limiter stage
//
```

```
   if (V( out_p ) > VLIMP)
      begin
         I( out_p ) <+   −10.0∗VLIMP;
         I( out_p ) <+ 10.0∗V( out_p );
      end
   else if (V( out_p ) < VLIMN)
      begin
         I( out_p ) <+ −10.0∗VLIMN;
         I( out_p ) <+ 10.0∗V( out_p );
      end
   end
   endmodule
```

The ADMS syntax is a subset of Verilog-A. Allowed language structures are outlined in a SYNTAX-SUPPORTED file which can be downloaed from `http://mot-adms.sourceforge.net`.

Model test examples

In the following sections a series of test results are presented. These illustrate how the modular macromodel performs in comparison to the transistor level[5] model for the UA741 operational amplifier. Typical parameters for the UA741 operational amplifier are listed in the introduction to this report. These values have been extracted from UA741 device data sheets provided by manufacturers. In the test simulations the modular UA741 parameters have been adjusted to give similar simulation results obtained from the transistor level model. A number of interesting differences between the simulation results obtained with the modular macromodel, plus default parameters, and the transistor level model are observed, for example the transistor level simulation results yield a value for IOFF of approximately zero amperes. With a real device this is unlikely to occur due to mismatches in the input transistor properties. In the transistor level model the input transistors are identical implying perfect matching. This is, of course, unlikely to be the case with a real device. Once again the results demonstrate one of the most important rules in circuit simulation, namely that the accuracy of the results from a specific simulation does largely depend on how well a model represents a physical device or circuit. Further comments about the Verilog-A code and the accuracy of the simulation results are given with each set of test results.

Input voltage offset

Input voltage offset is represented by the Verilog-A code listed in this section. This models the input voltage offset by two batteries of value VOFF/2. These are formed by current generators in parallel with one Ohm resistors. Notice that the direction of the current generator current flow determines the polarity of the

---

[5]The UA741 transistor level model can be found in the Qucs OpAmps component library.

batteries.

```
// Input voltage offset
//
    I(in_p, n7) <+ V(in_p, n7);
    I(in_p, n7) <+ Voffset;
//
    I(in_n, n9) <+ V(in_n, n9);
    I(in_n, n9) <+ -Voffset;
```
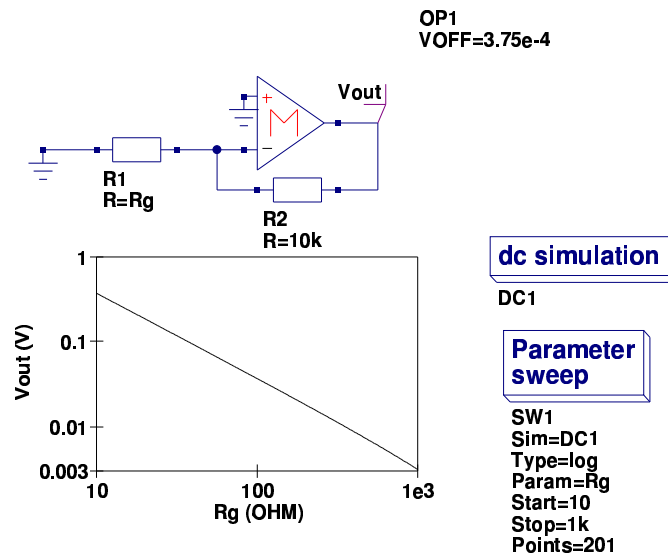

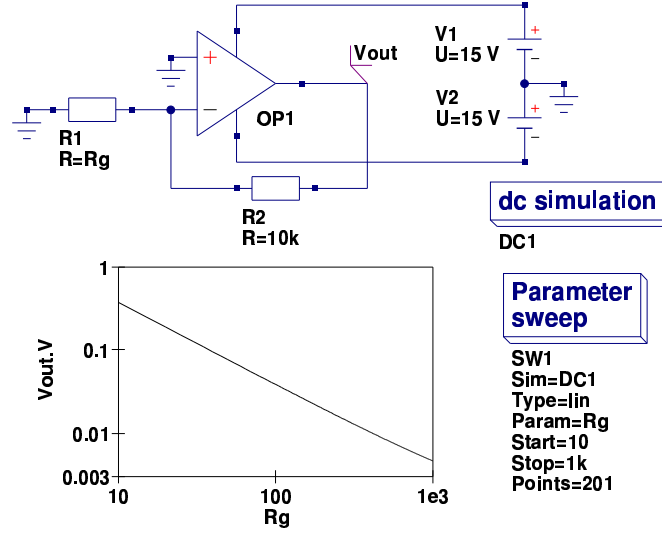Figure 1: UA741 modular macromodel input voltage offset test circuit and results

Figure 2: UA741 transistor level input voltage offset test circuit and simulation results

Input bias and offset currents

Input bias and offset currents are represented by the Verilog-A code listed in this section. Simple current generators are employed to model the input current effects. Values for IB and IOFF can be extracted from the results given in Figs. 3 and 4, using equations (1) and (2).

$$Vp = -\left(IB + \frac{IOFF}{2}\right) \cdot 1e3 \qquad (1)$$

$$Vn - Vs = \left(\frac{IOFF}{2} - IB\right) \cdot 1e3 \qquad (2)$$

The remainder of the input stage Verilog-A code adds differential input resistance and capacitance to the input stage of the operational amplifier macromodel.

```
// Input bias currents
//
    I(n7) <+ IB;
    I(n9) <+ IB;
//
// Input current offset
//
    I(n7,n9) <+ IOFF/2;
//
// Input differential resistance and capacitance
//
  I(n7, n8) <+ V(n7, n8)/Rdiff;
  I(n9, n8) <+ V(n9, n8)/Rdiff;
  I(n7, n9) <+ ddt(CD*V(n7, n9));
```

8

Figure 3: UA741 modular macromodel input bias and offset current test circuit and simulation results
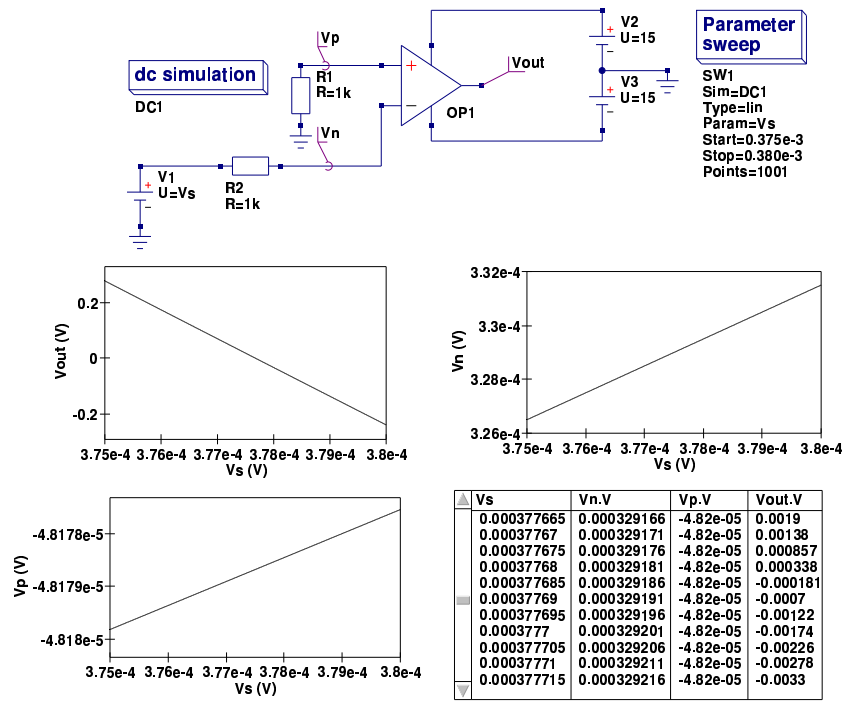
Figure 4: UA741 transistor level input bias and offset current test circuit and simulation results

Open loop differential voltage gain

Differential voltage gain is represented by the Verilog-A code listed in this section.
Differential gain is modelled with three distinct quantities. These are 1. resistor
RP1 which is set equal to the open loop differential gain at DC, 2. the primary
pole in the voltage gain frequency response which is set by capacitor CP1, and 3.
a high frequency pole set by CP2 and resistor RP2. Each pole stage is driven by
a voltage controlled current generator. Note the current generator negative sign.
This is required to maintain the correct signal phase. The derivation, and a more
detailed discussion, of the model open-loop differential voltage gain properties can
be found in pages 13 to 19 of the Qucs operational amplifier tutorial. Simulated
results for the open-loop differential gain response are shown in Figs. 5 and 6
The AOLDC and CD parameters given in Fig. 5 have been adjusted to give the
same response as the Qucs transistor level model. Again due to perfect transistor
matching a value of CD roughly zero is required to produce similar high frequency
responses above the second pole frequency.

```
//
// First pole
//
    I(n3) <+ -V(n12);
    I(n3) <+ V(n3)/RP1;
    I(n3) <+ ddt(CP1*V(n3));
//
// Second pole
//
    I(n5) <+ -V(n3);
    I(n5) <+ V(n5)/RP2;
    I(n5) <+ ddt(CP2*V(n5));
//
```
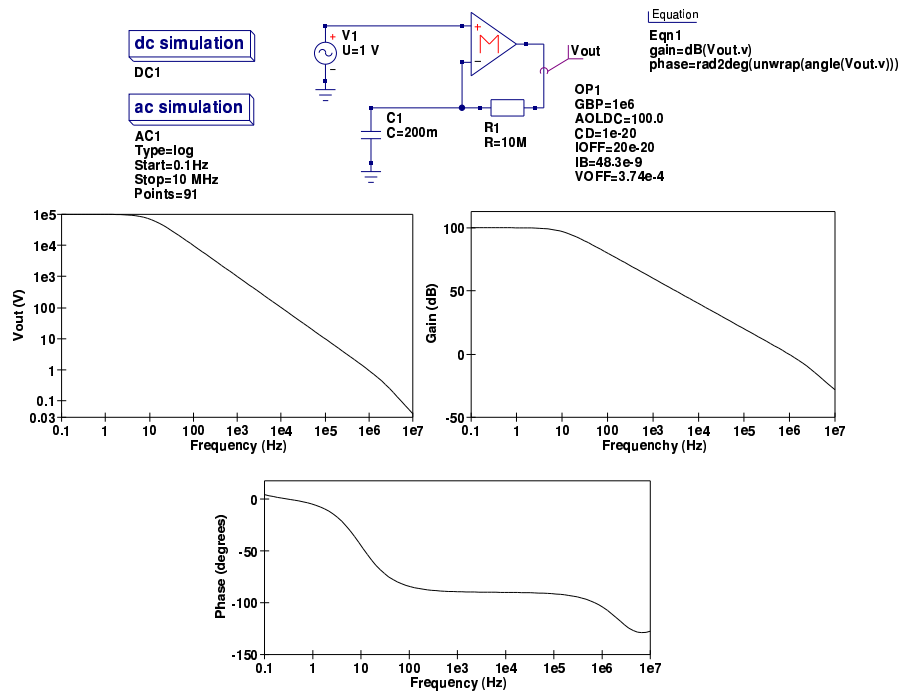
Figure 5: UA741 modular macromodel open-loop differential voltage gain test circuit and simulation results
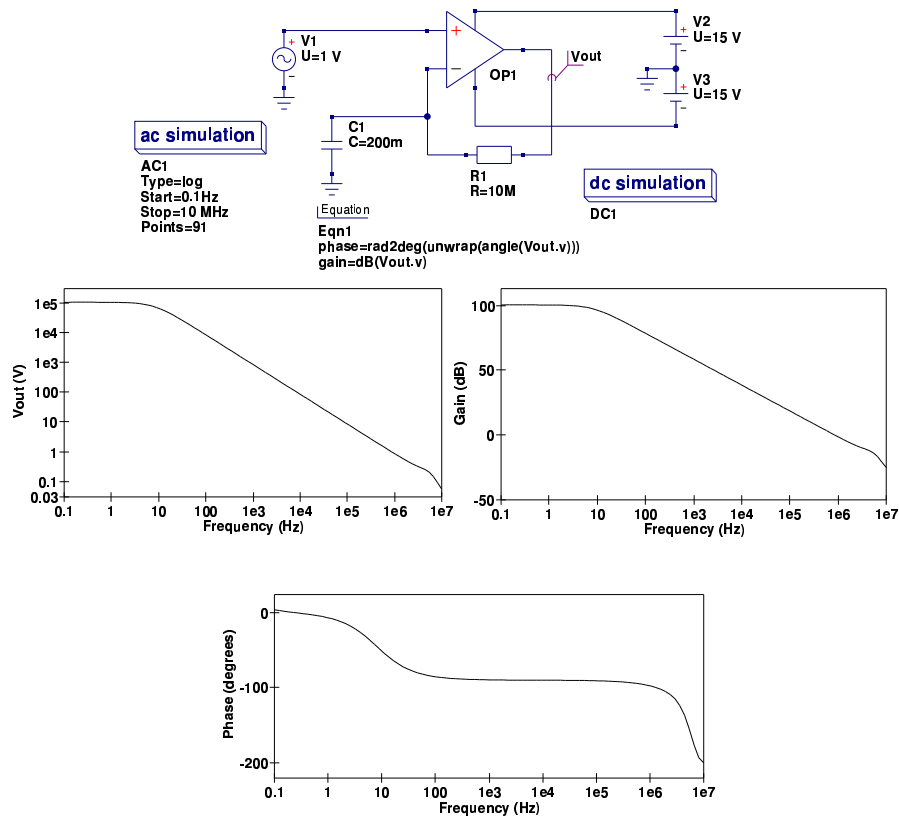
Figure 6: UA741 transistor level open-loop differential voltage gain test circuit and simulation results

Common mode effects

Operational amplifier common-mode effects are represented by the Verilog-A code listed in this section. Common-mode effects are simulated using an identical network to that outlined in page 20 the Qucs operational amplifier tutorial. The simulated results for a unity gain CMMR test circuit are illustrated in Figs. 7 and 8. The macromodel parameters CMRR and FCM have been set at their default values to demonstrate the difference when compared to the transistor level model. Common-mode simulation results observed with the transistor level model are significantly better than those obtained from measurements on real devices, mainly because the simulation model is constructed from perfectly matched transistors. Associated with the differential amplifier stages and the common-mode section of the macromodel is a signal adder which combines the differential and common-mode signals. The Verilog code for this adder stage is also repeated with common-mode code. An adder can be formed by a one Ohm resistor driven by differential and common-mode currents. The resulting volt drop across the one Ohm resistor then becomes the sum of the two signal voltages.

```
//
// Common mode stage
//
   I(n6) <+ -CMgain*V(n8);
   I(n6) <+ V(n6);
   I(n6, n10) <+ V(n6, n10)/1e6;
   I(n6, n10) <+ ddt(CCM*V(n6, n10));
   I(n10) <+ V(n10);
//
// Differential mode and common mode signal adder stage
//
   I(n11) <+ -V(n10);
   I(n11) <+ -V(n7, n9);
   I(n11) <+  V(n11);
//
```
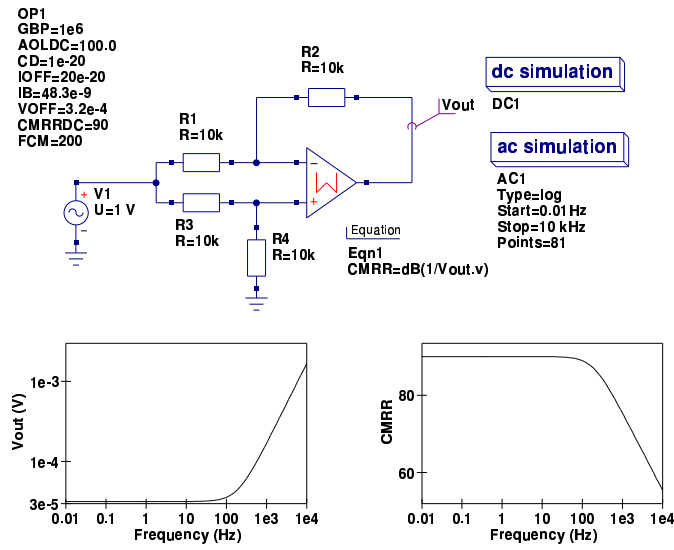
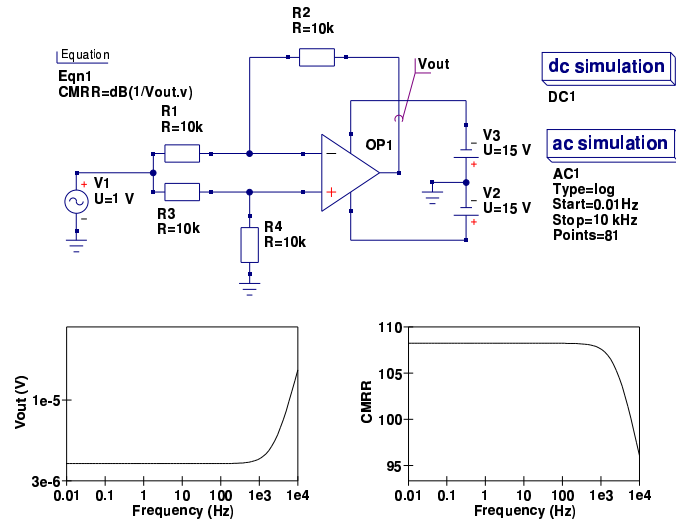Figure 7: UA741 modular macromodel CMRR test circuit and simulation results

Figure 8: UA741 transistor level CMRR test circuit and simulation results

15

Slew rate limiting

Operational amplifier slew rate limiting effects are represented by the Verilog-A code listed in this section. Slew rate limiting can be modelled in Verilog-A using the if-else language statement. This allows the maximum signal current at node n12 to be limited to a value set by variables Slewratepositve or Slewratenegative which in turn are functions of parameters PSRT and NSRT (see page 25 of the Qucs operational amplifier tutorial). Again please note the sign of I(n12). The simulated results for a UA741 slewrate limiting test circuit are illustrated in Figs. 9 and 10. In general the results are very similar for both models. However, there is one point worth commenting on; in the case of the transistor level model there is a marked difference in the level of slewing for negative and positive signals (see waveform Vout3 in Fig. 10). This effect is probably due to the fact that the UA741 operational amplifier circuitry is very different near the power supply rails, resulting in significant differences in signal shape at high signal swings. This effect is not modelled by the modular macromodel.

```
//
// Slew rate limiting stage
//
   if (V(n11) > Slewratepositive)
                     I(n12) <+ -Slewratepositive;
   else if (V(n11) < -Slewratenegative)
                     I(n12) <+  Slewratenegative;
   else I(n12) <+ -V(n11);
   I(n12) <+ V(n12);
```
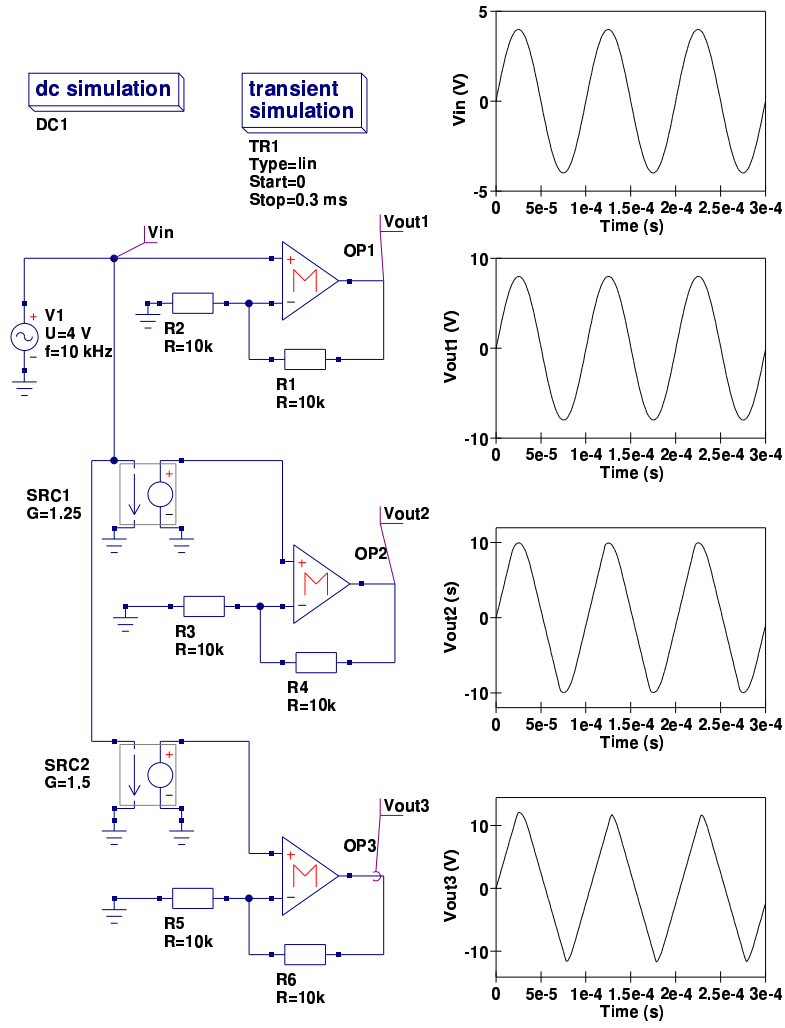
Figure 9: UA741 modular macromodel slewrate limiting test circuit and simulation results

Figure 10: UA741 transistor level slewrate limiting test circuit and simulation results

Output voltage limiting

Operational amplifier output voltage limiting effects are represented by the Verilog-A code listed in this section. A Verilog-A if-else statement is used to model voltage limiting. This language construction also demonstrates the use of the Verilog-A block construction formed with begin-end code words. Effectively the if-else statement swaps circuit elements as the voltage signal polarity changes. The simulated results for a UA741 voltage limiting test circuit are illustrated in Figs. 11 and 12. Again the results are very similar for both models. However, in the case of the macromodel there is no attempt to reduce the differential gain when output voltage limiting occurs and as a result some waveform distortion takes place. In practice if, as in the case of pure AC simulation, output voltage limiting is not required then simply set VLIMP and VLIMN well outside the range of required operating voltage and voltage limiting is never triggered.

```
//
// Voltage limiter stage
//
    if (V(out_p) > VLIMP)
       begin
         I(out_p) <+  -10.0*VLIMP;
         I(out_p) <+ 10.0*V(out_p);
       end
  else if (V(out_p) < VLIMN)
       begin
         I(out_p) <+ -10.0*VLIMN;
         I(out_p) <+ 10.0*V(out_p);
       end
    end
```
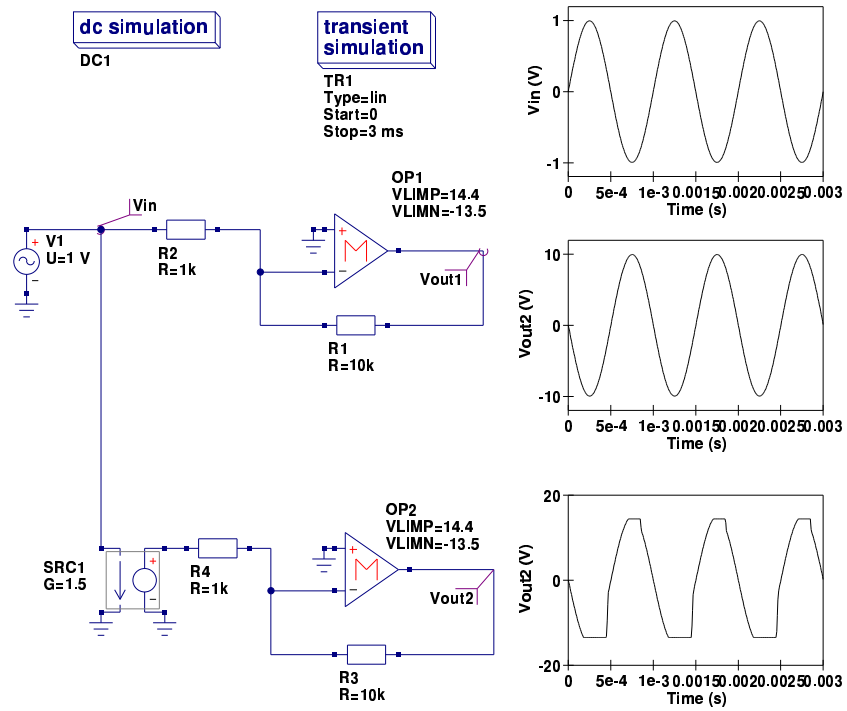
Figure 11: UA741 modular macromodel output voltage limiting test circuit and simulation results
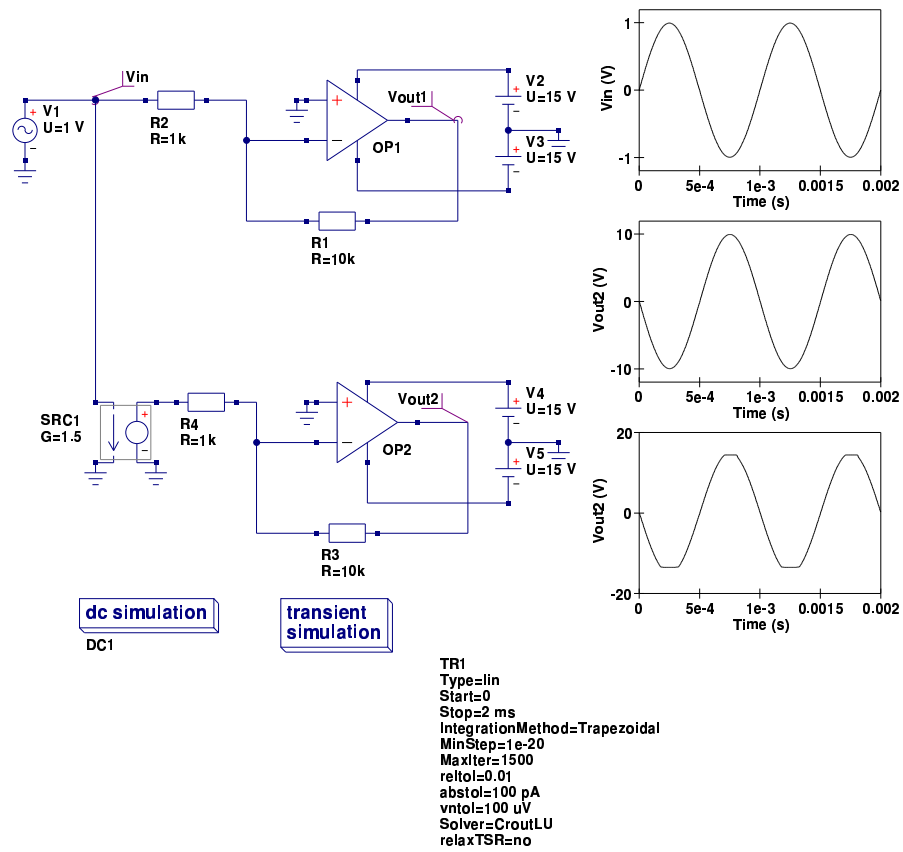
Figure 12: UA741 transistor level output voltage limiting test circuit and simulation results

Output current limiting

Operational amplifier output current limiting effects are represented by the Verilog-A code listed in this section. A Verilog-A if-else statement is used to model current limiting. The effect is modelled by a feedback mechanism which reduces the magnitude of current I(n4) which is proportional to the difference of the current flowing in the output path and ILMAX. A scale factor CSCALE is used to adjust maximum clamped output current. The simulated results for a UA741 current limiting test circuit are illustrated in Figs. 13 and 14. Current clamping induces distortion in the output waveform. Both the macromodel and the transistor level model give roughly the same clamped output currents but the wave shapes, and hence the distortion, are somewhat different. This is not surprising as the macromodel does not include a mechanism to control clamping distortion. Setting CSCALE to zero removes the current limiting process from the operational amplifier macromodel.

```
//
// Current limiter stage
//
    if (V(n2, out_p) >= ILMAX)
      begin
        I(n4) <+  -V(n5);
        I(n4) <+ CSCALE*V(n5)*(V(n2, out_p) - ILMAX);
        I(n4) <+ V(n4);
      end
    else if (V(n2, out_p) <= -ILMAX)
        begin
         I(n4) <+  -V(n5);
         I(n4) <+ -CSCALE*V(n5)*(V(n2, out_p) + ILMAX);
         I(n4) <+ V(n4);
        end
    else
        begin
            I(n4) <+ -V(n5);
            I(n4) <+  V(n4);
        end
//
```
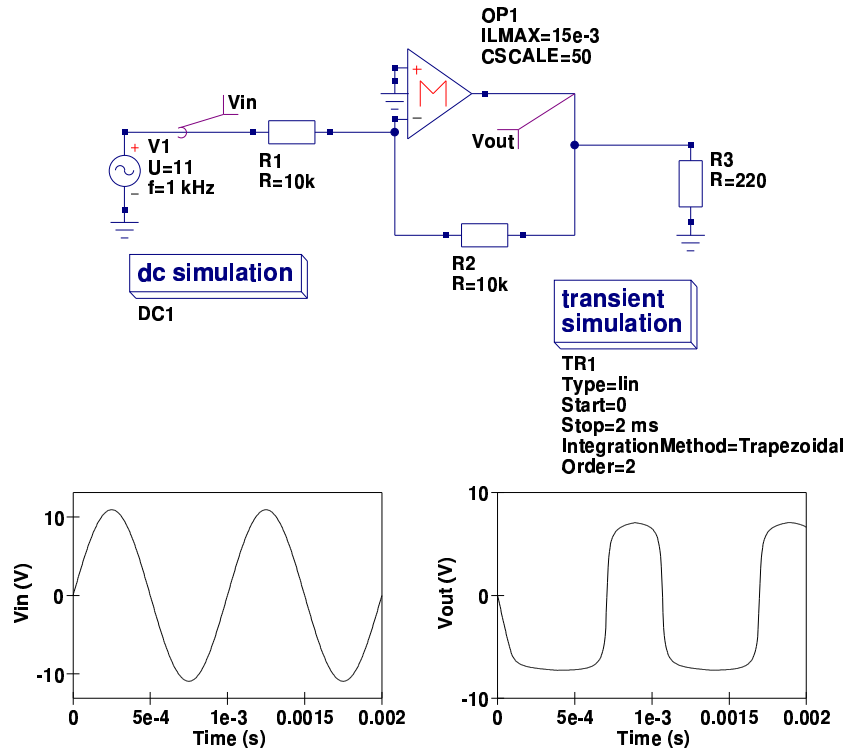
Figure 13: UA741 modular macromodel output current limiting test circuit and simulation results
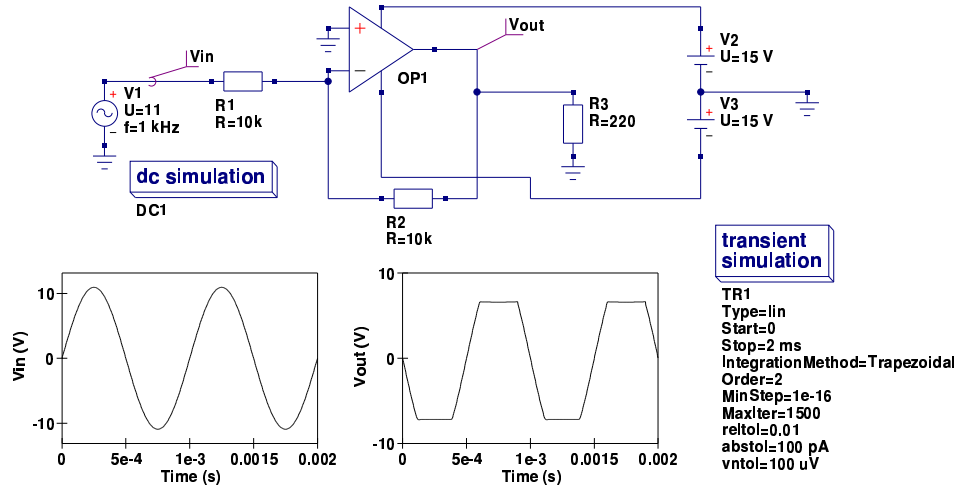

Figure 14: UA741 transistor level output current limiting test circuit and simulation results

23

End note

These notes summarise a number of techniques for modelling operational amplifier functions using Verilog-A. Verilog-A is primarily intended for modelling compact semiconductor devices. However, as this report demonstrates it can be equally employed for macromodelling of integrated circuits and circuits in general. The Qucs C++ code for the modular operational amplifier model, generated by ADMS, can be found in the Qucs release source tarballs at `http://qucs.sourceforge.net`. The procedure followed to convert the Verilog-A code into C++ for compilation and linking with Qucs closely followed that described by Sefan Jahn and Hélène Parruitte. Although it takes more work to construct models using ADMS the effort is worthwile because the finished models are very efficient in terms of memory usage and have significantly reduced run times. One interesting observation which is worth recording here is the fact that Qucs equation defined device (EDD) models and ADMS Verilog-A models have a very similar structure, implying that EDD models can be used as prototypes prior to compilation and linking via the compact device modelling route using ADMS. Once again a special thanks to Stefan Jahn for all his help and encouragement over the period that I have been developing the Verilog-A version of the operational amplifier macromodel, writing this report and testing the examples it includes.