Symbolic and Semisymbolic Analysis of Electronic Circuit in Maple

JIRI HOSPODKA  
Czech Technical University in Prague  
Department of Circuit Theory  
Technicka 2, 166 27 Prague  
CZECH REPUBLIC  
hospodka@fel.cvut.cz

JAN BICAK  
ASICentrum s.r.o.,  
a Company of the SWATCH GROUP  
Novodvorska 994, 142 21 Prague  
CZECH REPUBLIC  
Jan.Bicak@asicentrum.cz

Abstract: This paper presents a usage of Maple program for electric and electronic circuit analysis. The special package PraCAn has been used for this purpose. This contribution introduces possibilities of the package for circuit linearization and following ac symbolic or semisymbolic analyses. Simplification of the active part models is done by their instance parameters to obtain useful results for dominant circuit behavior. This is demonstrated on elementary examples of electronic circuit analysis. The package was developed at the Department of Circuit Theory, CTU for research and teaching support.

Key–Words: Analysis of electronic circuits, Semisymbolic analysis, Symbolic analysis, Maple, Spice

1 Introduction

Nowadays, many systems for circuit analysis are available. Conventional programs like PSpice®, Micro-Cap, WinSpice [1, 2, 3], etc. are single-purpose programs. Evaluation versions of these programs are frequently used for teaching support. Most of them solve the task only numerically. The symbolic analysis of electronic circuits has grown since the late 1980’s, with computer evolution [4, 5]. This is illustrated by number of symbolic analyzers for analog integrated circuits such as ASAP [6], SYNAP [7], SAPEC [8], SSPICE [9], SCAPP [10], GASCAP [11], SNAP [12], and other. These programs usually make analysis of linear or linearized circuits in the frequency domain; hardly any program makes symbolic or semisymbolic analysis of nonlinear circuit possible. In contrast to these single-purpose programs a special package for universal mathematical program Maple™ [13] is designed. Big advantage of this solution is that the analysis is powered by Maple and all results of analyses can be simply treated by native functions of Maple.

Our motivation was to enable electric circuit analysis directly in Maple program. This goal is realized by a special package (library of function) named PraCAn for Maple with wide range of facilities. The structure of the package is modular, additional functionalities and properties can be easily incorporated.

The paper is divided into the two main parts. PraCAn package is introduced in the first part with respect to nonlinear circuit analysis. Examples of circuit analyses are shown to demonstrate usability of the package and its facilities in the second part of the paper.

2 PraCAn Package

PraCAn package is a library of functions for Maple, which facilitates the symbolic and semisymbolic analysis of continuous and discrete-time linearized circuits. PraCAn is acronym for Prague Circuits Analyzer. The properties of the library were presented in [14], [15].

The circuit is processed by modified nodal voltage method [16]. All types of analysis are supported of the package also for continuous-time linear circuit. Results can be obtained in both forms – symbolic or semisymbolic form, i.e. as a function of a variable (for example \( f \) or \( t \) for frequency or transient analysis).

The package contains functions for the analysis of continuous-time nonlinear circuit. These functions enable numeric as well as symbolic analysis of operating point and ac analysis of linearized circuit. Standard Spice semiconductor models were used in the package [17]. It currently supports diode, BJT and MOSFET Level 1 models. Last time the computation of linearized parameters in operating point has been implemented. This contribution is focused on newly added features of the package.

3 Examples of Circuit Analyses

Possibilities of PraCAn package are demonstrated on the two elementary examples of electronic circuits.
The analyses are shown by the list of Maple commands.

### 3.1 Example 1

In this example the output impedance and resistance of the Wilson current mirror (see Fig. 1) are calculated. Symbolic and semisymbolic analyses are done with respect to different setting of transistor model parameters.

![Figure 1: Wilson current mirror with MOSFETs.](image)

#### PraCAn package activation (functions of the package are available at the interactive level).

```maple
> with(PraCAn):
> with(plots):
```

#### Circuit netlist definition and its assignment to a variable.

```maple
> WilsM:="Wils. current mirror
> Iin 0 in dc 20u
> Vo out 0 dc 3 ac 1
> M1 1 2 0 0 mfet-n
> M2 2 2 0 0 mfet-n
> M3 in in 1 0 mfet-n
> M4 out in 2 0 mfet-n
> .model mfet-n nmos lambda=0.01
> .end":
```

#### Symbolic ac analysis.

```maple
> solW:=PraCAn(WilsM,tf,symbolic):
```

#### Output resistance calculation (transistor is modeled without capacitances).

```maple
> rout:=subs(solW,-v("out")/i("Vo"));
```

#### DC numeric analysis – operatin point calculation.

```maple
> PraCAn(WilsM,dc):
```

#### Display of numeric values of M2 transistor linearized model.

```maple
> GetLinPar("M2");
{ gm : M2 = 0.0000285, gds : M2 = 0.197 \times 10^{-6} }
```

#### Numeric value of output resistance.

```maple
> routn:=subs(solW,-v("out")/i("Vo"));
```

#### AC analysis of the same circuit, but with respect to the transistor capacitances. Transistor model is defined for this case as:

NMOS lambda=0.01 CGSO=28E-12 CGDO=3E-12

```maple
> solWm:=PraCAn(WilsMm,ac):
```

#### Frequency response calculation.

```maple
> routm:=subs(solWm,-v("out")/i("Vo"));
```

#### Small frequency output impedance – output resistance.

```maple
> limit(routm,f=0);
```

#### Plot of magnitude of the output impedance vs. frequency.

```maple
> semilogplot(abs(routm)/1e6,
> f=1e4..1e7);
```

### 3.2 Example 2

Transfer characteristics (frequency responses) of the common collector–common base amplifier (Fig. 2) are calculated in this example. The operating point and linearized parameters are calculated at first. Then,
the frequency response in the semisymbolic form is obtained. The response is calculated symbolically as the frequency and a capacitance value function. The magnitude response for the chosen frequency and upper limit frequency of the amplifier are calculated as a function of the capacitance value.

\[ V_{in} \]
\[ V_{out} \]
\[ Q_1 \]
\[ Q_2 \]

Figure 2: The CC-CB configuration of an amplifier.

Circuit netlist definition and its assignment to a variable.

\[
> \text{cccb} := \text{"CCCB multistage amplifier"
> V1 1 0 dc 3
> V2 2 0 dc -3
> Vin in 0 dc 1.5 ac 1
> Rc out 2 3k
> Q1 1 in e bjt-npn
> Q2 out 0 e bjt-pnp
> .model bjt-npn NPN
> .model bjt-pnp PNP CJ=CJC
> + TF = 278.55E-12
> .end
\]

Operating point calculation.

\[
> \text{sol_dc} := \text{PraCAn(cccb,dc)};
\]
Display of transistor Q2 collector current and dc output voltage.

\[
> \text{subs(sol_dc, [i("Q2:C"),v("out")])} ;
> [-3.922011887e-3, -1.823396434]
\]

AC numeric analysis.

\[
> \text{sol_ac} := \text{PraCAn(cccb,ac)} ;
\]
Frequency response – output voltage phasor as a function of frequency.

\[
> \text{fr} := \text{subs(sol_ac, v("out")}) ;
> fr := 0.8258958534 \times 10^{10} / (0.3630980741 \times 10^{9}
+ 0.4556465385 \times 10^{10}) I CJC f
+ 0.003145977463 I f - 39.47841762 CJC f^2)
\]
Plot of the amplifier magnitude frequency response.

\[
> \text{semilogplot}(20*\text{log10(abs(subs(}
> CJC=1.306E-11,fr))), f=1e5..1e8);
\]

Plot of the output voltage magnitude for \( f = 10 \text{ MHz} \) vs. CJC capacitance (Zero bias B-C depletion capacitance) of the transistor Q2.

\[
> \text{semilogplot}(20*\text{log10(abs(subs(}
> f=1e7,fr))), \text{CJC}=1e-12..5E-11);
\]

Calculation of the small frequency gain.

\[
> g := \text{subs(f=0,fr)} ;
\]
Solving the upper frequency (-3dB) as a function of CJC capacitance and its plot.

\[
> \text{fu} := \text{solve(evalc(abs(fr))=g/sqrt(2), f)} ;
> \text{semilogplot(fu[1], \text{CJC}=1e-12..5e-11)} ;
\]

AC analysis of the same circuit, but with default transistor models (NPN BR=0 or PNP BR=0).

\[
> \text{sol_acm} := \text{PraCAn(cccbm,tf,symbolic)} ;
\]
Symbolic calculation of the small frequency gain and input resistance. Simplification of the results using equal symbolic values for the transistor model elements.
4 Conclusion

PraCAn package enables symbolic and semisymbolic analysis of an electric circuit in Maple program. The package contains functions for analysis of continuous-time linear and nonlinear circuits as well as periodically switched linear circuits. Special functions were added to the package to enable analysis of continuous-time nonlinear circuit. Together with this extension models of basic electronic circuit parts were defined. These models can be simplified by special options of the package to enable analysis of dominant circuit behavior without parasitical effects. It is especially necessary for symbolic analysis. The package enables to compute nonlinear characteristic of a circuit by DC analysis and AC analysis of corresponding linearized circuit. The results can be obtained in symbolic or semisymbolic form. They can be easily treated directly in Maple program, which is useful especially for symbolic analysis. It gives very good tool for engineers and students dealing with electronic.

The package was developed at the Department of Circuit Theory, Faculty of Electrical Engineering, Czech Technical University in Prague, for research and teaching support.

Acknowledgements: The work has been supported by the research program No. MSM6840770014 of the CTU in Prague.

Maple is trademark of Waterloo Maple Inc.; Maplesoft is a division of Waterloo Maple Inc. All other trademarks are property of their respective owners.

PSpice is registered trademark of Cadence Design Systems, Inc.

References: