AN ANALOG CIRCUIT FAULT WAVEFORM DATA EXTRACTION METHOD

[1]**BAORU HAN**, [2]**JINGBING LI** [1]**YIIIANG XING**

[1]Department of Electronic Engineering，Hainan Software Profession Institute，Qionghai 571400, Hainan, China
[2]College of Information Science and Technology, Hainan University，Haikou570228，Hainan, China
(Corresponding Author) **JINGBING LI**,

ABSTRACT

The fault waveform data extraction is the key of the analog circuit fault diagnosis. The paper presents a tolerance analog circuit fault waveform data extraction method. First, tolerance analog circuit simulation uses OrCAD / Pspice circuit simulation software in order to generate fault waveform data. Then, program in MATLAB automatically and accurately extract fault waveform data. Finally, the fault waveform data in binary form is stored in the matlab working space. And the bridge rectifier capacitor filter circuit fault data extraction set as an example is explained in detail, through which the effectiveness of the method is verified. This method solves the problem of data interface between the two, laying the foundation for the realization of analog circuit fault diagnosis fault.

Keywords: *Analog Circuit, Fault Waveform, Data Extraction, Sensitivity Analysis*

1. INTRODUCTION

With the development of electronic industry, the electronic equipment is becoming more and more complex. The analog circuit is indispensable. Theoretical analysis and practical application show that in these devices analog circuit fault is more likely to occur than the digital circuit fault [1]. In addition, along with the development of the large scale analog circuit and the enhancement of the complicacy of electronic device, the analog circuit reliability requirement is stricter. Therefore analog circuit fault diagnosis technology is increasingly important [2].The fault waveform data extraction is the key of the analog circuit fault diagnosis. Meanwhile, fault waveform data extraction is also configured the basis of the sample set. How to extract the analog circuit fault waveform data effectively is the difficulty of circuit fault diagnosis and test. When designing analog circuit fault diagnosis system, the key to fault diagnosis is extracting the characteristics of circuit fault quickly and efficiently.

First of all, with the combination of Pspice and Matlab,Y.S.LEE of Hong Kong Polytech University developed the CAD procedures MatSpice that is used in optimization power electronic circuit. The key of combination is how to obtain the simulation data of Pspice with the matlab; the [3] has not made detailed narrative literature. Currently, there is no good method to get Pspice simulation data with the matlab. The two softwares used in combination makes it easily access to the Pspice simulation data [4-6].

This paper presents a tolerance analog circuit fault waveform data extraction method. Firstly, tolerance analog circuit is simulated by OrCAD / Pspice circuit simulation software. Secondly, program in matlab automatically and accurately extracts fault waveform data. Lastly, the fault waveform data in binary form is stored in the matlab working space. The method can be used to solve communication problems in data between the two softwares. It realizes the fault waveform data extraction.

2. EXTRACTION METHOD

OrCAD / Pspice software, which has powerful functions in schematic drawing, circuit simulation, graphics processing and components production symbols, is the best general-purpose circuit simulation software [7]. It has the advantages of intuitive interface, easy to operate and analyze more powerfully. So it has broad application prospects. It can be very intuitive and easy to draw a circuit by using a graphical input mode in circuit design window, and do circuit various simulation analyses. The software system has a variety of electrical components factory model parameters, which are used to circuit analysis and calculation. At the same time, it also can be based on the

particularity of the actual components from external to modify the device parameters. This will be able to fully meet the requirements of calculation accuracy and very convenient for the setting of the various fault model.

As mathematical software, matlab is a high-level language for science and engineering computing, and is widely used in numerical computing, graphics processing, symbolic computation, mathematical modeling, signal analysis, system identification, real-time control, and dynamic simulation research [8]. Because it has a powerful waveform signal analysis and processing functions, it can be employed together with OrCAD/Pspice software effectively, through which can maximize their respective strengths.

OrCAD / Pspice software simulation of fault data is imported into matlab [9]. This is the basis for performing signal processing, also a first problem to be solved [10]. In most cases, what we need to import is a waveform of the response signal of the circuit. If it is the result based on Monto Cano analysis, there will be dozens or even hundreds of waveform needed to import. Manual operation is a laborious process, which will affect the progress of other research work. In this article, the Pspice simulation waveform data are accurately extracted by programming in MATLAB. This programming method successfully resolved a large number of signal waveform data communication problems between Pspice and MATLAB, parameter setting is simple, slight adjustment can be used for data transmission under other conditions ask.

3. CIRCUIT SIMULATION

3.1 Simulation Software

OrCAD / Pspice is a powerful circuit simulation software. It can not only carry out the basic circuit characteristics of DC, AC, transient analysis, but also can undertake Monte Carlo analysis, the worst-case analysis, optimization of design, and advanced analysis. OrCAD /Pspice software has a circuit schematic drawing, printed circuit board design and digital/analog circuit simulation. Its components library is also the most abundant of all the EDA software. Its probe module can display the signal waveform after the end of the simulation and perform some basic arithmetic processing, including Fourier analysis, bandwidth calculation and waveform extreme computing. The OrCAD / Pspice simulation types are as follows:

DC Sweep：The DC Sweep analysis varies a voltage source over a range of voltages in an assigned number of increments in a linear or logarithmic fashion.

AC Sweep：The AC Sweep analysis varies the operating frequency in a linear or logarithmic manner. Its role is calculating circuit AC small signal frequency response characteristics.

Time Domain (transient)：The Time Domain (transient) analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time. The starting and ending times for the various plots can be input. The accuracy (smoothness) of the output plots can also be controlled by regulating the maximum (time) step size.

Bias Point: the Bias Point analysis is the starting point for all analysis. In this mode, the simulator calculates the DC bias circuit.

Monte Carlo analysis: in order to simulate the circuit dispersion characteristics, Pspice provides Monto Carlo analysis functions. The Monto Carlo analysis prerequisite is that the components must have deviation properties.

Parameter sweep analysis: to analyze the corresponding circuit characteristics in the specified parameter value changes.

Temperature analysis: to analyze the circuit characteristics in a specific temperature.

Sensitivity analysis: to calculate the influence on the output variables with component parameter changes in circuit. It contains the DC sensitivity and exchanges sensitivity .

3.2 Circuit Fault Simulation

3.2.1 Circuit fault simulation process
Circuit fault simulation process as is shown in figure 1，imulation generated fault waveform data are saved in text form for matlab extraction.
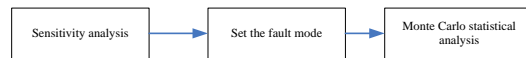

Figure 1. Circuit Fault Simulation Process

3.2.2　Sensitivity analysis
Under normal circumstances, the slight change of the circuit component values will change the characteristics of certain aspects of the circuit. The sensitivity is used to measure the change of physical quantities. Electrical network function T with respect to the rate of change of a parameter X is its specific definition. Sensitivity is represented by the symbol S_X^T

$$S_X^T = \frac{\partial T}{\partial X} \qquad (1)$$

Where T is a circuit network function (output variables), such as input impedance, output impedance, the transfer function, and the output voltage (or current). X are the component values or influences certain physical parameters of the component values, such as temperature. Tolerance analog circuit sensitivity analysis process is shown in figure 2.
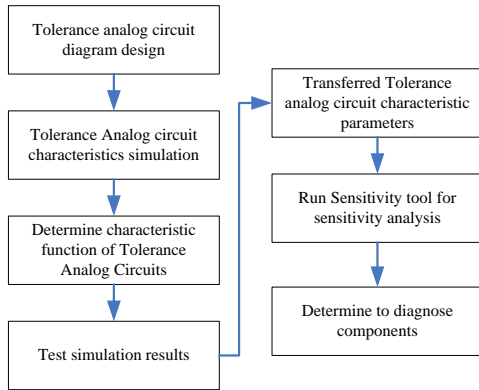

Figure 2. Sensitivity Analysis Process

3.2.3   Set fault mode

There are two types of analog circuit fault.

(1) Soft fault

It is caused by deviation from the normal tolerance range of the component, also called the offset fault. It is a continuous quantity. And so far there has been no model that can be used to reference. This would be very difficult to implement fault diagnosis. When the range of a component in nominal value tolerance is 50%, we believe that the component has a soft fault.

(2) Hard fault

It is caused by open or shorted; including open, shorted and line-to-line short-circuit. In Orcad10.5, we use fixed nominal value of large resistance to simulate the open-circuit fault mode, and use fixed nominal value of small resistance to simulate short-circuit fault mode.

According to the diagnostic components, we divide the fault mode in order to obtain various mode data. The data is used to construct and train classifiers in order to realize fault diagnosis.

3.2.4   Monte Carlo analysis

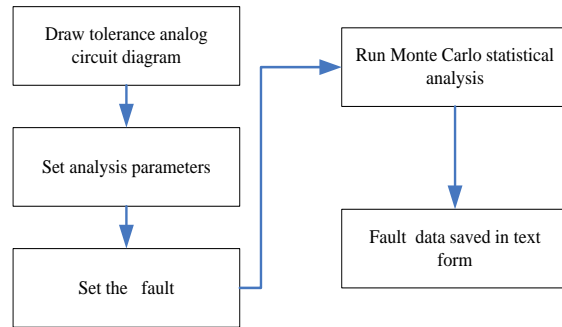Tolerance analog circuits Monte Carlo analysis process is shown in figure 3.

Figure 3. Monte Carlo Analysis Process

4.   EXTRACT FAULT WAVEFORM DATA

4.1 Commonly Used Functions

Matlab software can be programmed directly in the command window and can also create M-files to be programmed. It can make the program seem simple and easy to modify and maintain. Matlab software provides a series of low-level I/O functions in order to be able to read and write arbitrary binary or formatted ASCII file. [5-6]

(1) FID =FOPEN (FILENAME) opens the file FILENAME for read access. FILENAME is a string containing the name of the file to be opened. FILENAME can be a MATLABPATH relative partial pathname. If the file is not found in the current working directory, FOPEN searches for it on the MATLAB search path.

(2) ST =FCLOSE (FID) closes the file associated with file identifier FID, which is an integer value obtained from an earlier call to FOPEN. FCLOSE returns 0 if successful or -1 if not.

(3)TEXTREAD Read formatted data from text file. TEXTREAD is useful for reading text files with a known format. Both fixed and free format files can be handled.

(4)NUM2STR convert numbers to a string. If the input array is integer-valued, num2str returns the exact string representation of that integer. The term integer-valued includes large floating-point numbers that lose precision due to limitations of the hardware.

(5)STR2DOUBLE convert string to double precision value. X = STR2DOUBLE(S) converts the string S, which should be an ASCII character representation of a real or complex scalar value, to MATLAB's double representation

4.2 Fault Waveform Data Extraction Program Flow Chart

Combined with the matlab syntax of sequential structure, loop structure, branch structure and control statements, we can use the above functions to do some of the more complex operations in order

to achieve text file extraction. Fault waveform data extraction program flow chart is shown in figure 4.
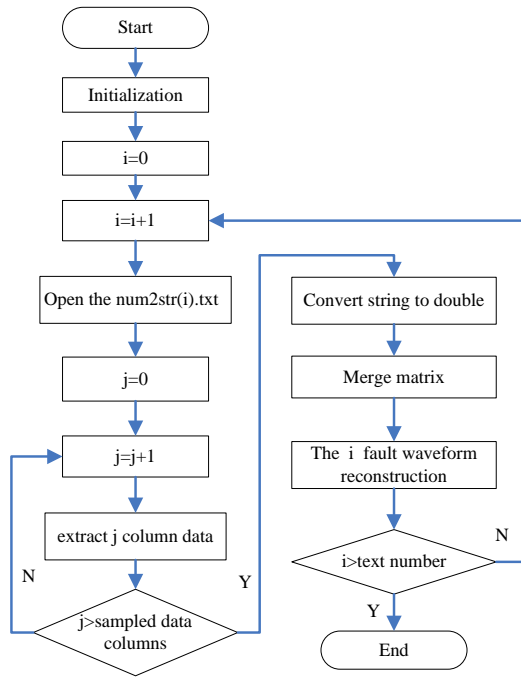

Figure 4. Program Flow Chart

5. SIMULATION

Circuit shown in figure 5, it's called the bridge rectifier capacitor filter circuit, composed of four diodes, a capacitor and a resistor. The parameter values are shown in figure 5. Diode, resistance and capacitance tolerance range are 10%.
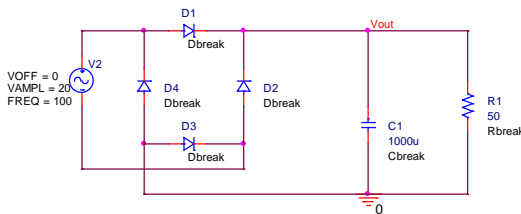

Figure 5. Bridge Rectifier Capacitor Filter Circuit

Because the influence of some components parameters on the output signal is not heavy, not all the fault components of analog circuit can be diagnosed. Therefore, it is necessary to calculate the sensitivity of the circuit before diagnosis. Therefore, the circuit is carried to Sensitivity analysis using Orcad10.5 sensitivity tool. First, select the sinusoidal voltage source as the excitation source. Sinusoidal voltage source initial value is 0 V. And the peak value is 20V. Set the Run to time to 20 ms. second, select circuit characteristic function and run sensitivity tools. I found that the

capacitance C, the diode D_3 and the diode D_4 of the changes in parameter values have a great impact on the circuit output. Therefore the capacitor C, the diode D_3 and the diode D_4 is selected as the components to be diagnosed. Consequently, fault modes are the capacitor C open, the diode D3 open, the diode D4 open and normal state. The circuit is a total of four modes. Then for each fault mode to do Monte Carlo analysis, the output waveform is shown in figure 6, figure 7, figure 8 and figure 9.
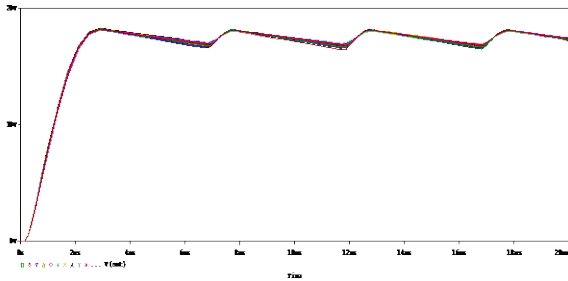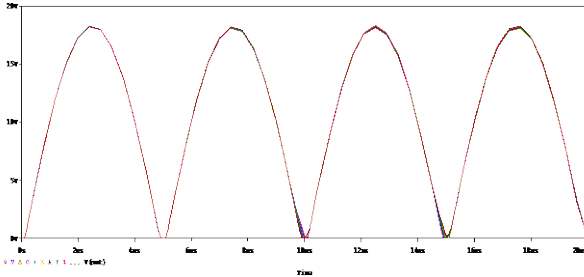

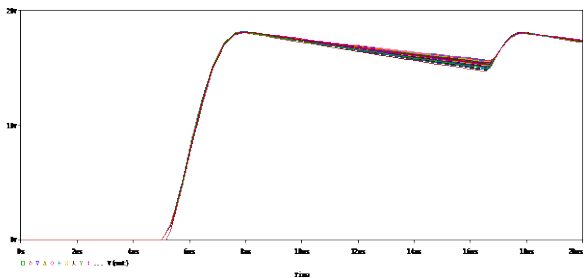*Figure 6. Normal Waveform*

Figure 7. C Open Waveform

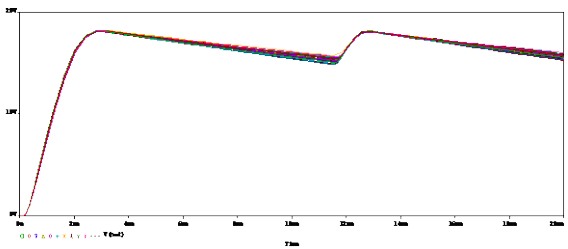*Figure 8. D_3 Open Waveform*

Figure 9. D_4 Open Waveform

Every time, Monte Carlo analysis completes, and the output waveform will be stored in the form of data in the text. Finally, automatically extract fault

waveform data using matlab programming approach. Reconstruction fault waveform using the extraction data. Reconstruction of the waveform is shown in figure 10, figure 11, figure 12 and figure 13. Compared the extraction data with the fault data stored in the text, two sets of data are exactly the same. So the method can correctly extract the fault waveform data.
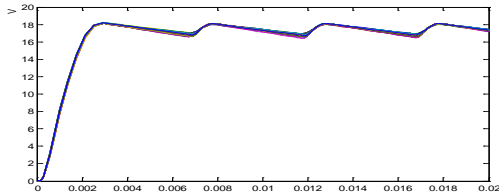

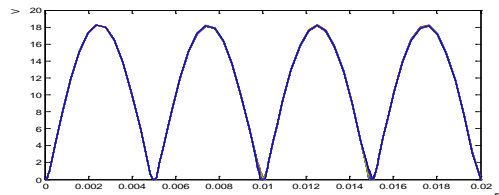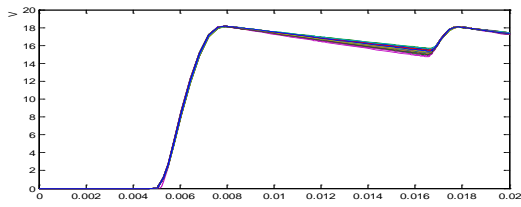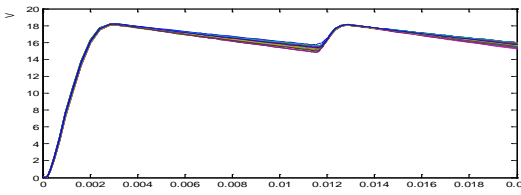*Figure 10. Normal Waveform Of Reconstruction*

Figure 11. C Open Waveform Of Reconstruction

Figure 12. D_3 Open Waveform Of Reconstruction

Figure13. D_4 Open Waveform Of Reconstruction

6. CONCLUSION

OrCAD / Pspice software is the most popular electronic systems simulation software, which can effectively simulate analog, digital, and analog-to-digital hybrid system. Matlab software has powerful data processing and graphics capabilities. The paper proposes an effective method for the fault data waveform extraction, which is a good solution to the waveform data communication problems by the combination of two softwares. The method is to lay the foundation for tolerance analog circuit fault diagnosis with a practical value.

REFERENCES:

[1] Bandler J W, Salama A E, "Fault Diagnosis of Analog Circuits", *Proceedings of the IEEE*, Vol. 73, No. 8, 1985, pp. 1279-1327

[2] ARTUR R, ROMUALD Z, "Fault diagnosis of analog piecewise linear circuits based on homogony", *IEEE Transactions on Instruments and Measurement,* Vol.51, No.4, 2002, pp. 876−881.

[3] Y.S.LEE, "Use of spice in design of power electronic circuits and systems", *IPEMS,* 1997, pp. 7l7-721

[4] Aminian M, Aminian F, "Neural network based analog -circuit fault diagnosis using wavelet transform as processor", *IEEE Trans. on CAS - II: Analog and Digital Signal Processing,* Vol. 47, No. 2, 2000, pp. 151-156

[5] Aminian M, Aminian F, "A modular fault - diagnostic system for analog electronic circuits using neural networks with wavelet transform as a preprocessor", *IEEE Trans.Instrum. Meas.* Vol. 56, No. 5, 2007, pp.1546-1554.

[6] Li F, Woo P Y, "Fault detection for linear analog IC the method of short-circuit admittance parameters", *IEEE Trans. on CAS I: Fundamental Theory and Applications,* Vol. 49, No. 1, 2002, pp. 105-108.

[7] L. T. Pillage, "An early introduction to circuit simulation techniques", *IEEE Transactions on Education,* vol. 36, 1993, pp. 16-19

[8] Abou-Fakher, Y.H.l, Robinson, F.V.P, "Modeling transformers in MATLAB", *Proceedings of the Universities Power Engineering Conference,* September 10-12, 1997, pp.137-141

[9] Liangyu, Peng, Bao, Lei, "Application of Matlab/Simulink and Orcad/PSpice software in theory of circuits", *Advances in Intelligent and Soft Computing,* vol. 114, 2012, pp. 1055-1064,

[10] Le-Huy, Hoang, Sybille, Gilbert, "MATLAB/ simulink and Pspice as modeling tools for power systems and power electronics", *Proceedings of the IEEE Power Engineering Society Transmission and Distribution Conference,* vol.2, 2000, pp. 766-767