ICIC Express Letters
Part B: Applications
Volume **10**, Number **10**, October **2019**

ICIC International ©2019 ISSN 2185-2766
pp. **911–918**

STUDY ON THREE-PHASE POWER FLOW APPROACH FOR UNBALANCED DISTRIBUTION NETWORKS BASED ON CIRCUIT MODELS

Hsin-Ching Chih[1], Wei-Tzer Huang[1,*], Wei-Chen Lin[1], Kai-Chao Yao[1]
Yih-Der Lee[2], Jheng-Lun Jiang[2] and Hui-Min Lai[2]

[1]Department of Industrial Education and Technology
National Changhua University of Education
No. 2, Shida Road, Changhua City 50007, Taiwan
{ d0631008; m0731011 }@mail.ncue.edu.tw; kcyao@cc.ncue.edu.tw
*Corresponding author: vichuang@cc.ncue.edu.tw

[2]Institute of Nuclear Energy Research
Atomic Energy Council, Executive Yuan
No. 1000, Wenhua Road, Jiaan Village, Longtan Township, Taoyuan County 32546, Taiwan
{ ydlee; Jhenglun; hml0301 }@iner.gov.tw

Received May 2019; accepted August 2019

Abstract. *This work demonstrates the new simulation approach for unbalanced distribution networks. An open source software Ngspice and Python language associate the new distribution network calculation method. Using the proposed circuit models, the simulation results of the proposed method are with high accuracy compared with the test results of the IEEE 4-bus system and OpenDSS. Moreover, the voltage waveform in the neutral line voltage and its harmonic analysis are accomplished, which is not possible in the OpenDSS software. The outcomes of this work provide a friendly and practical approach for polyphaser power system simulation.*
Keywords: Open source software, Ngspice, Python, OpenDSS, Neutral line voltage

1. **Introduction.** Distribution networks inherit with many estimate challenges, such as the polyphase transformers, the unbalance load, and the required transient analysis. There are many power system analysis tools available for today practice. Huang et al. involved the Leblanc connection polyphase transformer by EMTP [1] commercial tool. Chen and Liao utilized the Matlab/Simulink [2] commercial tool in railway system transient analysis. A popular OpenDSS simulation tool was widely used in the smart grid system simulation. It required to joint venture with Matlab commercial tool via GridPV toolbox in Mortazavi et al. [3] study. All these commercial tools not only require installation cost but also suffer the national politics risk [4]. It becomes vital to build up commercial tool-free distribution analysis tool in infrastructure engineering.

The Simulation Program with Integrated Circuit Emphasis engine (SPICE) serves the Integrated Chip (IC) design industry in its high reliability and accuracy [5,6]. Python computer language (Python) manages the proposed method in the SPICE initiation, unbalance load parameters calculation, steady state, transient state, and harmonic analysis. Both SPICE and Python are the open source tools beyond authority control.

The following Section 2 explains the implementation concept. In the elements Section 3, it presents the power system elements in mathematical equations and SPICE models. Section 4 introduces the simulation procedure. In Section 5, a simulation case is presented

DOI: 10.24507/icicelb.10.10.911

with the corresponding discussion. The voltage and current result in the Steady-State-Analysis, time domain waveforms in Transient-State-Analysis, and harmonic component in Numerical-Analysis are in comparison. The conclusions are in Section 6.

2. **Concepts.** OpenDSS software is widely used in scenarios. It comes from the Electric Power Research Institute (EPRI) in United State. Figure 1 shows the software application concept illustration. It is a generic Steady-State-Analysis tool in text style programming environment. Hence, it requires trained engineers and maintenance overhead.

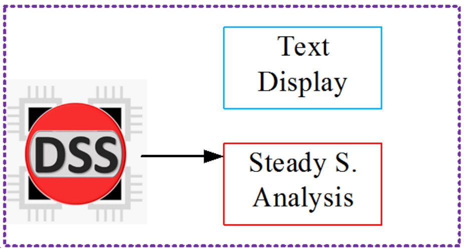

FIGURE 1. OpenDSS software application

The proposed method starts by creating the circuit topology file for SPICE in Figure 2 illustration. This paper contributes to the dedicated power system element models to constructing the whole system model. With the current and voltage control sources philosophy, this method behaves in user-friendly by the visual drawing work. It becomes intuitive operation and supports many analysis requirements.

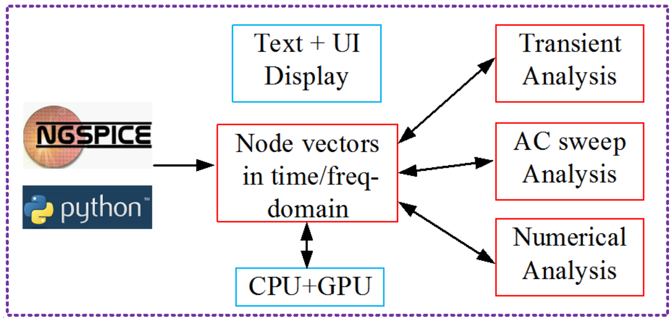

FIGURE 2. Proposed Ngspice and Python simulation method

The open source Ngspice program is the open source SPICE engine in this study. It supports the steady state sweep, transient, and run time parameters adjustment for iteration. Python asserts the Ngspice simulation command via corresponding package for duplex communication during the run time [7]. In this study, the Python routine calculates the unbalance load parameters from Steady-State-Analysis iteration function. Later, Python starts the Transient-State-Analysis task in the fine time interval, e.g., 400 points per power cycle time (16.67msec). It results in the vast data vector matrix in every node. With this data, both plotting and the Numerical-Analysis processes take over the rest services.

3. **Element Models.** The models of CCCS (linear Current-Control Current Source) and VCVS (linear Voltage-Control Voltage Source) associate the transmission line and transformer elements. Most of the SPICE simulators contain these two basic models in the library. As shown in Figure 3, the output current i_o and voltage v_o sources denote in Equations (1) and (2):

$$i_o = \alpha \cdot i_c, \tag{1}$$

$$v_o = \beta \cdot v_c, \tag{2}$$

where α and β denote the gain values of the user input number in the real number format.

A transmission line element, which also utilizes the CCCS and VCVS models to satisfy the traditional Z bus matrix format, is presented in Figure 4. Assume that the transmission line has two endings in mark-x and mark-y. The voltage drop over the line impedance and other phase-coupled voltages denote in Equation (3). Hence, the graphical schematic expression is derived for power system schematic drawing.

$$\begin{bmatrix} V_{ax} \\ V_{bx} \\ V_{cx} \end{bmatrix} - \begin{bmatrix} V_{ay} \\ V_{by} \\ V_{cy} \end{bmatrix} = \begin{bmatrix} R_{aa} + j\omega L_{aa} & R_{ab} + j\omega L_{ab} & R_{ac} + j\omega L_{ac} \\ R_{ba} + j\omega L_{ba} & R_{bb} + j\omega L_{bb} & R_{bc} + j\omega L_{bc} \\ R_{ca} + j\omega L_{ca} & R_{cb} + j\omega L_{cb} & R_{cc} + j\omega L_{cc} \end{bmatrix} \begin{bmatrix} I_a \\ I_b \\ I_c \end{bmatrix} \tag{3}$$

An ideal transformer and its SPICE element model are in Figure 5. Despite the possible core and copper loss factors, the CCCS and VCVS models configure the ideal transformer characteristic in Equations (4) and (5). Be careful in the current flow direction spot symbols while connecting the SPCIE network. It inevitably causes the wrong phase angle result if improperly connected. In the ideal transformer, the parameters of α and β values are equal to one.

$$\alpha = N_2/N_1 \tag{4}$$

$$\beta = N_2/N_1 \tag{5}$$

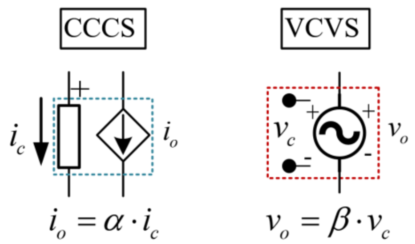

FIGURE 3. CCCS and VCVS models

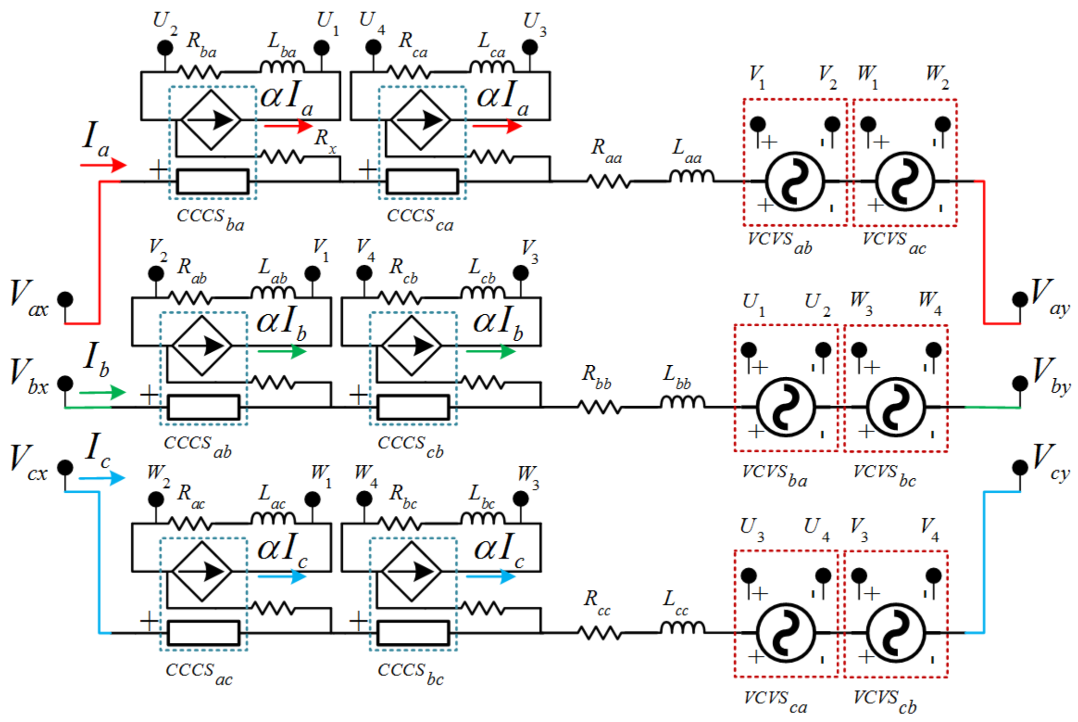

FIGURE 4. Three-phase transmission line model

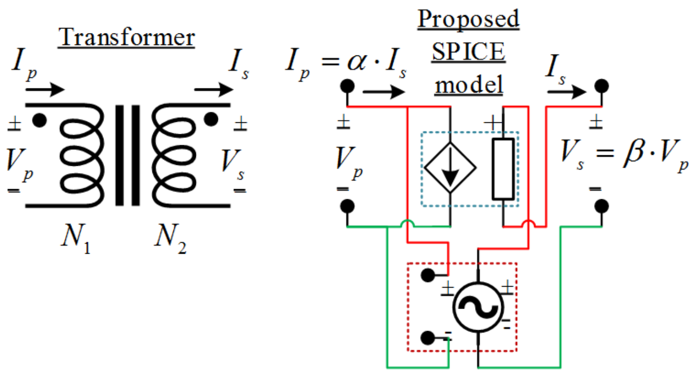

FIGURE 5. Single-phase transformer (left) and the proposed SPICE model (right)

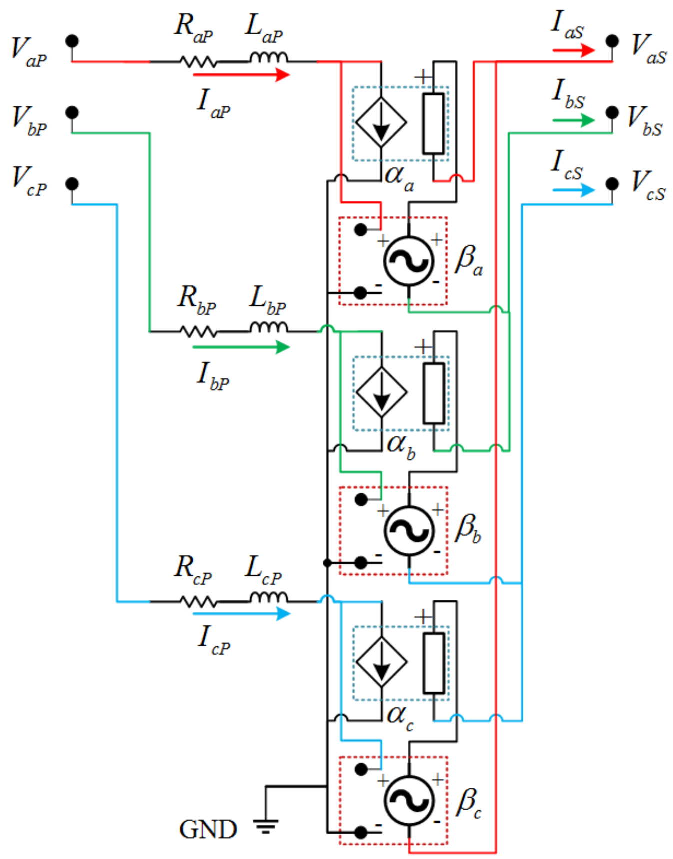

FIGURE 6. A wye-Δ connection transformer model

A traditional three-phase wye-Δ transformer element is associated with the three individual single-phase SPICE elements shown in Figure 6. Assume that the voltage conversion ratio is N_2/N_1. Thus, the primary and secondary side voltages denote in Equation (6).

$$\begin{bmatrix} \alpha_a \\ \alpha_b \\ \alpha_c \end{bmatrix} = \begin{bmatrix} \beta_a \\ \beta_b \\ \beta_c \end{bmatrix} = \begin{bmatrix} \frac{N_2}{N_1}(V_{aP} - V_{bP})\Big/V_{aP} \\ \frac{N_2}{N_1}(V_{bP} - V_{cP})\Big/V_{bP} \\ \frac{N_2}{N_1}(V_{cP} - V_{aP})\Big/V_{cP} \end{bmatrix} = \begin{bmatrix} \frac{N_2}{N_1} \cdot \sqrt{3} \\ \frac{N_2}{N_1} \cdot \sqrt{3} \\ \frac{N_2}{N_1} \cdot \sqrt{3} \end{bmatrix} \quad (6)$$

Copper loss part is in R_{aP}, L_{aP}, R_{bP}, L_{bP}, R_{cP} and L_{cP} components with the true measurement value. These values are from manufacturer or converted from per unit

Table 1. Common three-phase transformer element gain setting

Gain \ Conn.	wye-wye	wye-Δ	Δ-wye	Δ-Δ
α, β	$\frac{N_2}{N_1}$	$\frac{N_2}{N_1}\cdot\sqrt{3}$	$\frac{N_2}{N_1}\cdot\frac{1}{\sqrt{3}}$	$\frac{N_2}{N_1}$

format data. The α and β gain vector settings in the common three phase transformer connections are summarized in Table 1.

4. **Simulation Procedure.** Simulation procedure, shown in Figure 7, starts from designing the schematic with proposed elements. After converting to SPICE file with the .cir extension file, Python initiates the tuning parameters and initiates the Ngspice simulation engine. Loading estimation block extends the flexibility, such as calculating the loading network resistance, inductance, capacitance values for the fixed P/Q loading case. It is also capable of performing the complex step simulation for searching the optimal values.

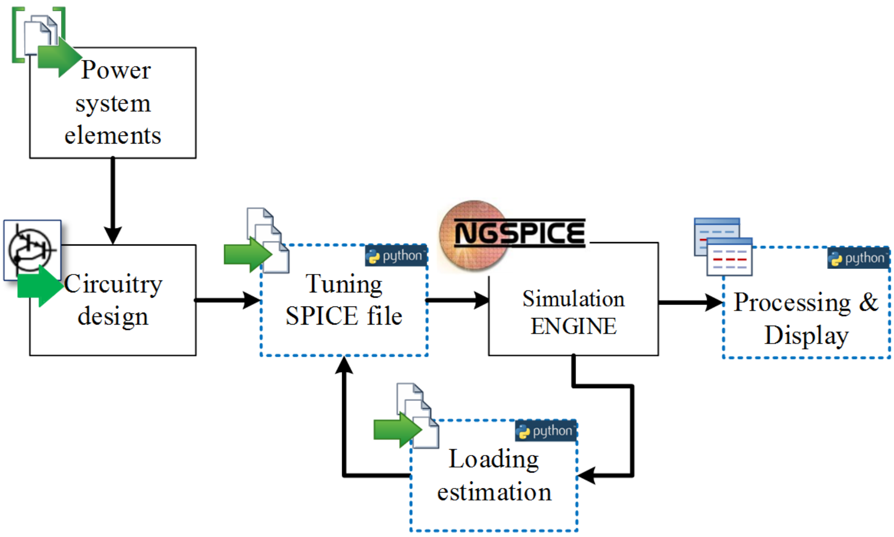

Figure 7. Proposed simulation flow chart

Processing and Display block aims to communicate the result to a text file or to show in the graphical waveforms. By utilizing the various Python packages, this block can directly send out the simulation result to a cloud server for application expansion. GPU acceleration package, e.g., CuPy, is optionally called for speeding up the vast voltage and current vector computation process routine.

5. **Simulation Results.** A complicated condition of the IEEE 4-bus test model [8] is chosen for simulation. It is the stepdown, unbalanced load with an open wye-delta connection transformer in between. The detailed schematic with proposed models is in Figure 8. AC power source configures in the three-phase power supply with neutral grounding. There are two transmission line segments in 2,000 and 2,500 feet length, respectively. An unbalanced load is attached to the system. For sensing the neutral line current, an additional R_{res} (0.001Ω) is installed between power source and transformer.

The Ngspice program's version number is 29 and Python runs in 3.7.2 version with PyCharm® development environment in the Windows 10. The schematic drawing tool is Microcap 11® demo version from Spectrum Software™. The initial hardware setup is the Intel™ i7-5820K CPU with Nvidia™ P2000 GPU with 1024 CUDA® cores. The OpenDSS software runs in the 8.5.9.1 version. Each analysis outcome is in below.

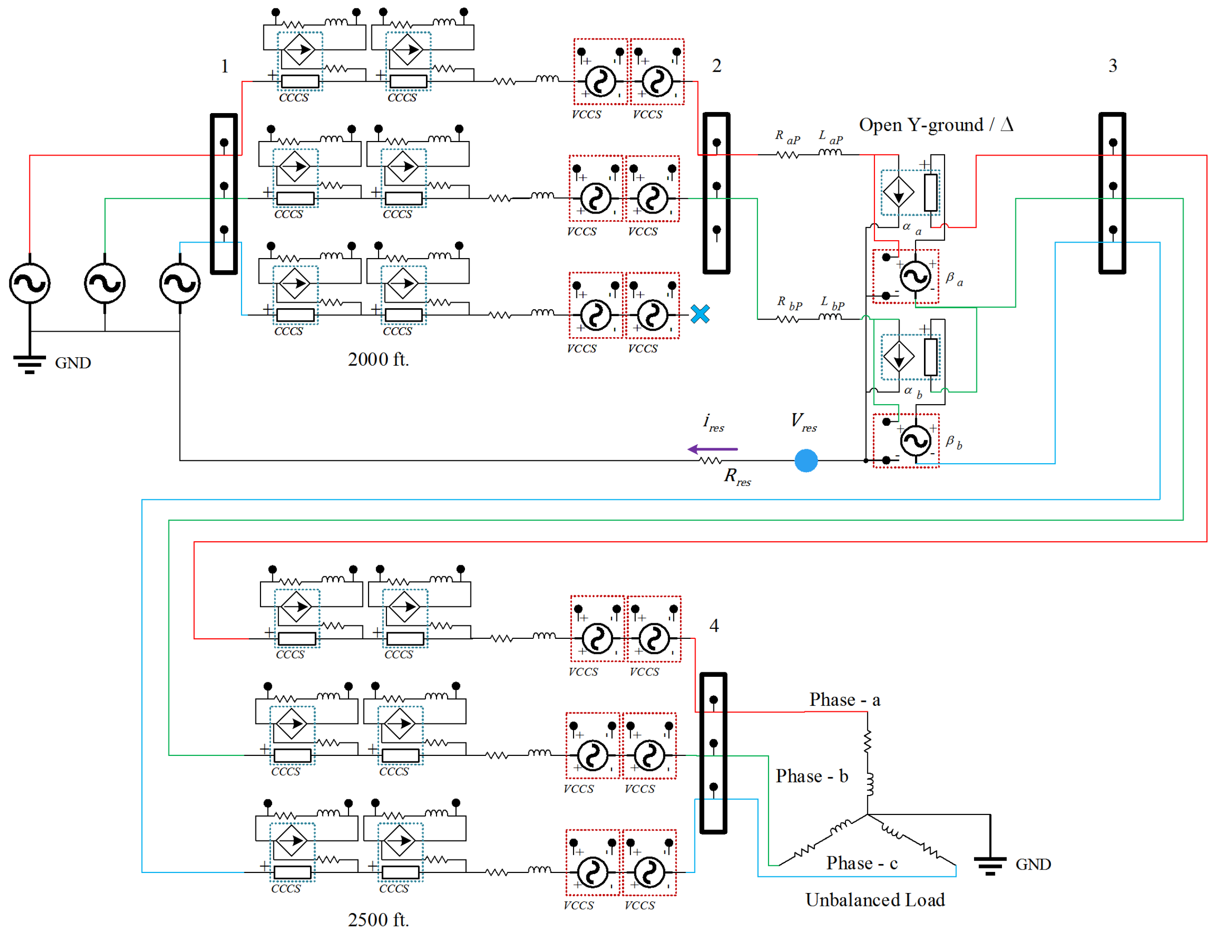

FIGURE 8. A step-down, unbalanced mode with an open wye-delta connection transformer

TABLE 2. Comparison in the IEEE document (IEEE), proposed method (SPICE) and OpenDSS software (OpenDSS) in the Steady-State Analysis

		Node2	Node3	Node4	Current I_{12}	Current I_{34}	*Load* model
V_a	IEEE	**6952/0.7**	**3632/0.1**	**3307/ − 1.5**	**424.8/ − 73.8**	**735.2/ − 73.8**	–
	SPICE	**6950.2/0.6**	**3632.4/ − 0.1**	**3300.8/ − 1.5**	**423.7/ − 74**	**733.2/ − 74.0**	**2.0333/2.3306**
	OpenDSS	**6954.8/0.7**	**3641.7/0.0**	**3322.8/ − 1.6**	**417.11/ − 73.3**	**721.95/ − 73.3**	–
V_b	IEEE	**7172/ − 122**	**4121/127.6**	**3907/ − 131.9**	**440.3/ − 118.5**	**569.9/176.3**	–
	SPICE	**7188.0/ − 121.9**	**4128.5/ − 127.5**	**3921.9/ − 131.9**	**439.7/ − 118.7**	**569.2/176.2**	**3.8445/6.1333**
	OpenDSS	**7173.4/ − 122.0**	**4120.9/ − 127.4**	**3908.9/ − 131.7**	**431.97/ − 118.6**	**567.4/176.4**	–
V_c	IEEE	**7313/120.5**	**3450/108.9**	**3073/103.1**		**762.0/61.5**	–
	SPICE	**7303.1/120.8**	**3466.6/108.9**	**3080.7/102.8**		**761.0/61.3**	**2.2371/1.8638**
	OpenDSS	**7310.7/120.5**	**3465.9/109.2**	**3099/103.5**	**0**	**747.64/61.4**	–

Unit: Voltage/DEG, Ampere/DEG, Ω/mH

Steady-State-Analysis: Table 2 shows the result between the IEEE document, proposed SPICE method, and OpenDSS. The SPICE iterates the loading model parameters in AC sweep for satisfying with the rated power consumption. It is obvious that the proposed SPICE method result closes to an IEEE document and OpenDSS result. Moreover, the proposed SPICE method presents the unbalanced load model parameters as a plus.

Transient-State-Analysis: The unbalanced loading voltage waveform is presented in Figure 9(a) and V_{res} voltage waveform in Figure 9(c). It exhibits the startup waveforms scenario till 90ms. V_{res} voltages are in $0.7834\angle-96.4°$ volt, and $0.7998\angle-100.33°$ volt, respectively in OpenDSS and the proposed method. Neither IEEE document nor OpenDSS are capable in demonstrating the transient waveform feature.

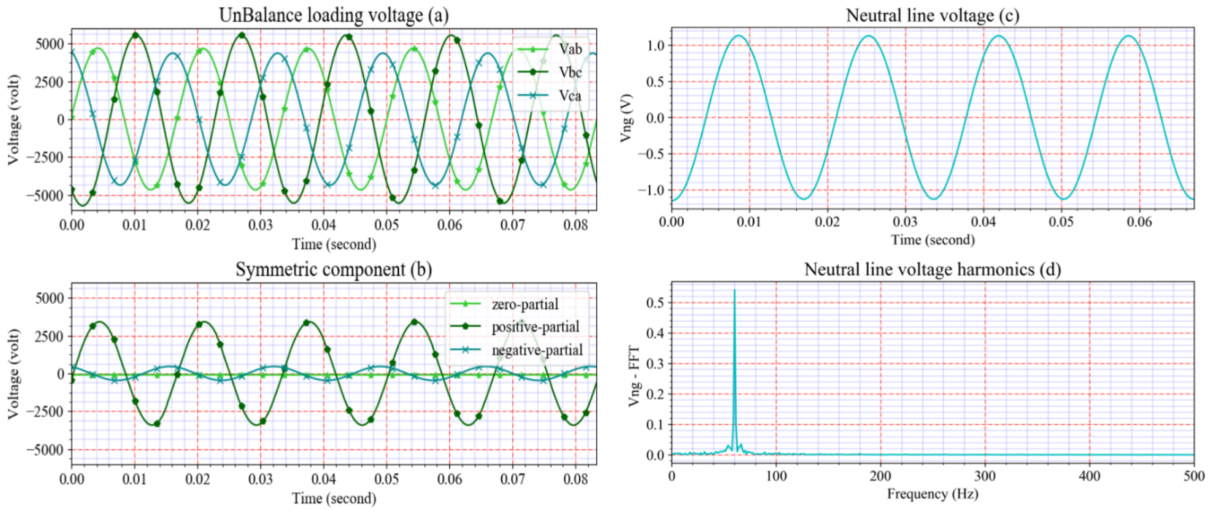

FIGURE 9. Waveforms of the unbalanced loading voltage (a), symmetric component (b), neutral line voltage V_{res} (c) and spectrum of V_{res} (d)

Numerical-Analysis: The harmonic and symmetric components analysis are in Figure 9(d), and Figure 9(b), respectively. Python routine converts the time-domain vector data and generates the spectrum and component waveform charts.

6. **Conclusions.** This study proposed that the new distribution system simulation method utilized the SPICE and Python concept. New power system elements were introduced and simulated in the Steady-State-Analysis. These results were closer to the IEEE and OpenDSS outcome. The vast voltage and current vector data were from the Transient-State-Analysis for plotting the loading waveforms, neutral line voltage V_{res} waveform, harmonic analysis, and symmetric component analysis. All these achievements demonstrate the proposed method sufficiently replaces these commercial tools.

Future research focuses on to implement the proposed SPICE models into the commercial Microcap® library for a simple application. The conditional SPICE model for line short/open fault case is mandatory for system protection topic. For PV solar source, the non-linear power source SPICE model is necessary for Green power grid application. With the above improvement, Python extends its role into the load forecast in machine learning algorithms with edge computing structure. In summary, this proposed SPICE method provides all purposes, cost-free, and flexible features in a will.

Acknowledgment. The authors would like to acknowledge the financial support of the Institute of Nuclear Energy Research of Taiwan through its Grant No. 108A001.

REFERENCES

[1] S.-R. Huang, Y.-L. Kuo, B.-N. Chen, K.-C. Lu and M.-C. Huang, A short circuit current study for the power supply system of Taiwan railway, *IEEE Trans. Power Delivery*, vol.16, no.4, pp.492-497, 2001.

[2] T. Chen and R. Liao, Modelling, simulation, and verification for detailed short-circuit analysis of a 1 × 25 kV railway traction system, *IET Generation, Transmission & Distribution*, vol.10, no.5, pp.1124-1135, 2016.

[3] H. Mortazavi, H. Mehrjerdi, M. Saad, S. Lefebvre, D. Asber and L. Lenoir, An impedance-based method for distribution system monitoring, *IEEE Trans. Smart Grid*, vol.9, no.1, pp.220-229, 2018.

[4] T.-F. Cheng and L. Li, Huawei loses access to vital chip design updates from Synopsys, *Nikkei Asian Review*, 2019.

[5] A. Cross and D. Strickland, Using SPICE for power system simulations, *Proc. of the 2011 14th European Conference on Power Electronics and Applications*, 2011.

[6] Y. Cai, C. Deng, Q. Zhou, H. Yao, F. Niu and C. N. Sze, Obstacle-avoiding and slew-constrained clock tree synthesis with efficient buffer insertion, *IEEE Trans. Very Large Scale Integration (VLSI) Systems*, vol.23, no.1, pp.142-155, 2015.

[7] Ignamv, *Python Bindings for the Ngspice Simulation Engine*, https://github.com/ignamv/ngspyce, 2018.

[8] D. S. A. Subcommittee, *IEEE 4 Node Test Feeder*, http://sites.ieee.org/pes-testfeeders/resources/, 2006.