SIMULATION OF CIRCUITS USING SPARSE TABLEAU ANALYSIS IN MATLAB

M. Vansáč, I. Tomčíková, M. Ocilka
Technical University in Košice
Park Komenského, 3, 04200, Košice, Slovakia.
E-mail: martin.vansac@tuke.sk, iveta.tomcikova@tuke.sk, matus.ocilka@tuke.sk

The paper deals with circuit simulation of DC electric circuits by Graphical User Interface (GUI) in MATLAB environment, which is used to create circuit simulator. As main method there was used Sparse Tableau Analysis (STA). This simulation program in MATLAB/GUI environment allows numerical and also symbolical solution of electric circuit.
Key words: sparse tableau analysis, electric circuit simulation, Matlab.

МОДЕЛЮВАННЯ ЕЛЕКТРИЧНИХ КІЛ З ВИКОРИСТАННЯМ МЕТОДУ РОЗРІДЖЕНИХ ТАБЛИЦЬ У СЕРЕДОВИЩІ MATLAB

М. Вансач, І. Томчікова, М. Оцилка
Технічний університет Кошице
Парк Коменського, 3, м. Кошице, 04200, Словаччина.
E-mail: martin.vansac@tuke.sk, iveta.tomcikova@tuke.sk, matus.ocilka@tuke.sk

Розглядається моделювання електричних кіл постійного струму за допомогою Graphical User Interface (GUI) середовища MATLAB, яке використовується при створенні моделі для симуляції. Основним використовуваним методом є метод розріджених таблиць. Розроблена у середовищі MATLAB/GUI програма моделювання дозволяє отримувати чисельний та символьний розв'язок електричних кіл.
Ключові слова: метод розріджених таблиць, моделювання електричних кіл, Matlab.

PROBLEM STATEMENT. Modeling and simulation are inseparable part for checking, verifying and for the designing of electronic circuits in electronics department. Nowadays, modeling means a process by which the real physical system is transformed into a mathematical expression in form of set of equations. Simulation is the process when a computer is used to obtain numerical or analytical solution.

There are mostly used three formulation methods for formulating, solving and simulation of electronic circuits: nodal analysis (NA), modified nodal analysis (MNA) and sparse tableau analysis (STA) [1].

The key advantages of MNA are: MNA is more compact than STA, preparation and creation of analysis is faster than STA. One disadvantage of MNA is that MNA doesn't contain information about all currents and voltages in electric circuit.

Handicap of STA is that preparation of STA analysis is slower than MNA, but on other hand STA contains information about all voltages and currents. STA can be applied to any circuit, and equations can be assembled directly from the circuit specification [2]. Therefore as formulation method is used sparse tableau analysis for creation of circuit simulator, realized by GUI in MATLAB.

This circuit simulator was used for linear networks with passive and active elements, i.e., networks which consist from linear resistors, capacitors, inductors, independent sources, and linear controlled sources.

The DC analysis of linear networks was achieved with opened capacitors and shortened inductors, where all time-dependent sources and time-varying parameters of each individual components and their derivatives were set to zero.

A main point in this paper is determined branch and nodal voltages and branch currents for each individual part of the electric circuit with sparse tableau analysis, with option of choice between numerical and symbolical solution.

The behavior of the electric circuit is given by set of equations, which are based on combining of the element equations, Kirchhoff's Voltage Law (KVL) and Current Law (KCL) [1].

Sparse tableau analysis consists from the following steps:

1) write Kirchhoff's Current Law in form $Ai=0$, where matrix A presents a reduced incidence matrix and i is a vector of all branch currents;

2) write Kirchhoff's Voltage Law in form $u=A^T v$, where vector u is a vector of all branch voltages and vector v is a vector of all nodal voltages in consideration to ground;

3) write the element equations as $Zi+Yu=s$, where Z and Y are matrices and vector s is a vector of independent sources (voltage or current sources).

To get all element equations in linear algebraic form was used the following conditions:

a) if edge e_k is a resistor, then $u_k - Ri_k = 0$;

b) if edge e_k is an independent voltage or current source, then $u_k=U$ or $I_k=I$;

c) if edge e_k is a controlled source, then we can write for a voltage controlled voltage source $u_k = \alpha u_x$, for a current controlled voltage source $u_k = \alpha i_x$, for a current controlled current source $i_k = \alpha i_x$ and for a voltage controlled current source $i_k = \alpha u_x$.

Combining of these equations with Kirchhoff's Voltage and Current laws, valid the following system:

$$\begin{bmatrix} A & 0 & 0 \\ 0 & I & -A^T \\ Z & Y & 0 \end{bmatrix} \begin{bmatrix} i \\ u \\ v \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ s \end{bmatrix}, \qquad (1)$$

where Z and Y are (m x m) matrices and vector s is a known (m x 1) vector [2].

EXPERIMENTAL PART AND RESULTS OBTAINED. Based on STA, the following electric circuit simulator was created. For practical utilize of designed simulator, program user should to know basic principles of sparse tableau analysis and also properties of electric components (R, L, C, independent and controlled current and voltage sources). To verify of proposed simulator was used electronic circuit, which is shown in Fig. 1.
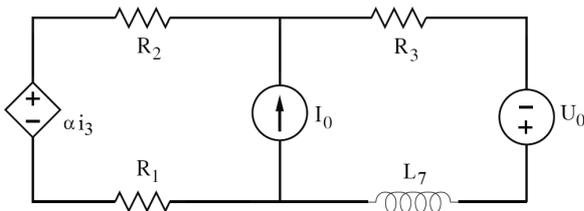
Figure 1 – Tested electric circuit

Before simulation it is suitable to create preparation for solution. This preparation contains design of labeling of each individual nodes, branches voltages and currents. There are many ways, how to do it. One possibility how to create a proposal for labeling is in Fig. 2.
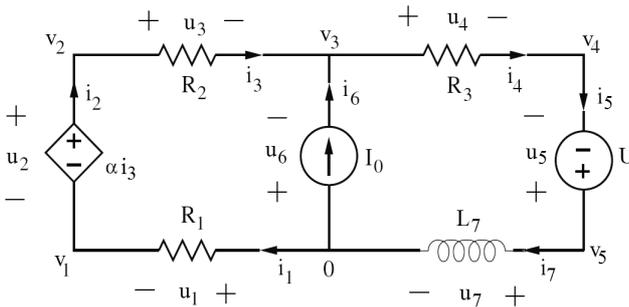
Figure 2 – Preparation of solution

For correct functionality of this circuit simulator is important, that orientation of branch voltage for independent source and also for controlled voltage source has to be the same like voltage orientation of voltage source. Orientation of current in this case is not so important. Analogously for independent current source and also for controlled current source, orientation of branch current has to be the same like current orientation of current source. In this case orientation of voltage is not so important. Also voltage and current orientation for resistors, capacitors and inductors doesn't play important role in this simulator. Result will be either positive or negative. Designed circuit simulator in GUI environment is in Fig. 3.

Field in the left half of the GUI simulator (*set circuit characteristics*) is used to set properties of the circuit. Right side (*calculated currents and voltages*) is used to view obtained results.
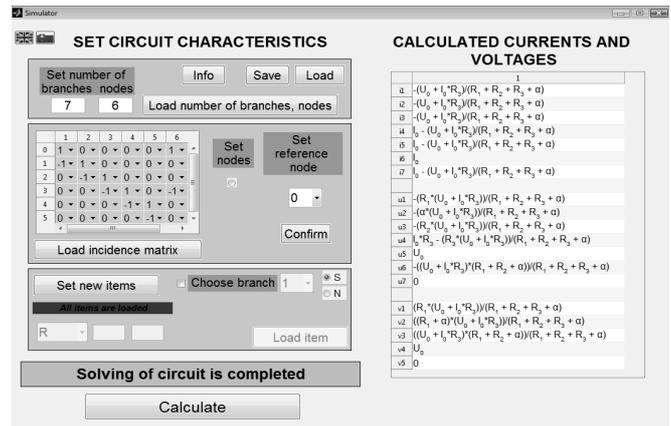
Figure 3 – Proposed electric simulator

After running this simulator, there is the first field with button *Info, Save, Load, Load number of branches, nodes.* After running simulator, there are two options: at the first, it is possible to set number of branches and nodes, set input and output nodes and set all individual items. Or at the second, after running there it is possible to load data from file. Simulator supports English and also Slovak language. Chose of language is providing by buttons in top left corner.

To set all circuit properties, the program user has to set number of branches and nodes and then press the button *Load number of branches, nodes* in the top of GUI simulator. Number of branches or nodes is shown in status field. If those numbers are wrong, then this situation is generated in orange status field.

After this, the second field with table, button *Load incidence matrix* and check box will appear below the previous button. User can set incidence matrix manually, by choosing of input and output node, column by column. If user set nodes incorrectly, then whole corresponding column is red, also is not possible to set nodes in the next column and in the status field is generated error message about this status. If all nodes are inserted, than button *Load incidence matrix* will be enabled to set incidence matrix.

User can set incidence matrix also automatically by choosing checkbox *Set nodes*. In this case button name *Load incidence matrix* will change into name *Enter input node of branch No.1* and user has to enter input node in the first branch. Then in status field is printed information about loaded node and button name *Enter input node of branch No.1* will change into *Enter output node of branch No.1* and user has to enter output node in the first branch. This procedure continues column by column till all nodes are inserted. If user set nodes incorrectly, then in button is generated information about this status. If all nodes are inserted, than button name *Enter output node of branch No.* will change into name *Load incidence matrix* and button will be enabled to set incidence matrix. Example of this procedure is shown in Fig. 4:

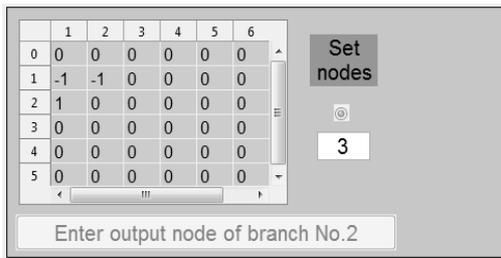Електромеханічні і енергозберігаючі системи. Випуск 4/2012 (20)

74

Figure 4 – Loading of nodes

After previously action, the third field will appear below the previous field. In this field user can set automatically properties of all items: name, value, and for controlled source also number of branch of controlled value. If any property is wrong, then message is also generated in status field. When all items are correctly inserted button *Save* is enabled and in second field will appear button *Confirm* and popup menu, which includes all nodes. In this part, user can choose reference node from popup menu and confirm it by pressing button *Confirm*. Then in the third field will appear check boxes which serve to change of properties of any item and to choose of type of solution. Under this field will appear button *Calculate*. There is possible to choose between analytical and numerical solution. If type of solution is chosen, then button *Calculate* is enabled.

User can solve circuit by pressing this button. Process of calculation is showed in status field. If calculation is done, in the right side will appear table with solution.

By pressing button *Info* user can see all information about circuit in new window. Example of this window is shown in Fig. 5.

By pressing button *Save* (*Load*) is possible to save (load) all information about any circuit. To save (load) is used standard dialog box. There is possible to save (load) data only as type *.mat*. For all others types are generated error message.

If reference node is confirmed, then separately it is possible to change topology of circuit in the table and also separately it is possible set new items by button Set new items, or set individual item by check box. By pressing button Load number of branches, nodes whole process starts all over again, i.e. popup menu and button

Confirm in the second field and the entire third field will be invisible.

Figure 5 – Info

CONCLUSIONS. This work is focused at symbolical and numerical solution of electric circuit by STA in MATLAB/GUI. This simulator allows DC analysis and therefore it can be useful in the study of electric subjects. User can check and verify its theoretical knowledge and he can analyze behavior of circuit before its practical realization. This program is base for future work, for example for AC or transient analysis.

ACKNOWLEDGEMENT.

REFERENCES
1. McCalla W.J. *Fundamentals of Computer-Aided Circuit Simulation* – Dordrecht, Netherlands, 1988. – PP. 1–16.
2. Najm F.N. *Circuit Simulation.* – Hoboken, New Jersey: John Wiley & Sons, Inc. – ISBN 978-0-470-53871-5, 2010. – PP. 3–29.
3. MATLAB. – User's Guide, Math Works 2009.

МОДЕЛИРОВАНИЕ ЭЛЕКТРИЧЕСКИХ ЦЕПЕЙ С ИСПОЛЬЗОВАНИЕМ МЕТОДА РАЗРЕЖЕННЫХ ТАБЛИЦ В СРЕДЕ MATLAB

М. Вансач, И. Томчикова, М. Оцилка
Технический университет Кошице
Парк Коменского, 3, г. Кошице, 04200, Словакия.
E-mail: martin.vansac@tuke.sk, iveta.tomcikova@tuke.sk, matus.ocilka@tuke.sk

Рассматривается моделирование электрических цепей постоянного тока при помощи Graphical User Interface среды MATLAB, которая используется при создании модели для симуляции. Основным используемым методом является метод разреженных таблиц. Разработанная в среде MATLAB/GUI программа моделирования позволяет получать символьное и численное решение электрических цепей.
Ключевые слова: метод разреженных таблиц, моделирование электрических цепей, Matlab.