PHYSICS AND CHEMISTRY OF SOLID STATE
V. 21, No.3 (2020) pp. 519-524

DOI: 10.15330/pcss.21.3.519-524

Vasyl Stefanyk Precarpathian National University

ФІЗИКА І ХІМІЯ ТВЕРДОГО ТІЛА
Т. 21, № 3 (2020) С. 519-524

UDC 621.382 ISSN 1729-4428

G.I. Barylo[1], R.L. Holyaka[1], I.I. Helzhynskyy[1], Z.Yu. Hotra[1], M.S. Ivakh[1], R.L. Politanskyi[2]

Modeling of Organic Light Emitting Structures

[1]*Lviv Polytechnic National University, Lviv, Ukraine.* hryhorii.i.barylo@lpnu.ua
[2]*Yuriy Fedkovych Chernivtsi National University, Chernivtsi, Ukraine*

The paper has been shown the results of the study of the parameters of organic light-emitting structures based on the SPICE (Simulation Program with Integrated Circuit Emphasis) model studies. A SPICE model of diode structure has been developed, which is implemented in the form of a substitution scheme based on the basic components of the simulator. This model can be extended by introducing additional components of the substitution scheme, which provides higher accuracy in representing the structure specifics. Graphical results of researches of the model of OLED structure at the change of internal parameters have been presented. The obtained data well represent the parameters of real structures and are characterized by a fairly effective adaptation to the experimental data of specific samples.

Keywords: light emitting structures, model, SPICE modeling.

Received 21 July 2020; Accepted 15 September 2020.

Introduction and problem statement

The synthesis of new organic semiconductor materials creates the preconditions for the development of organic electronics. Creating effective of the light-emitting structures have much attention in this direction. The development of such devices is largely determined by the efficiency of software used in the study of electro-optical parameters of organic semiconductors. Currently, various software tools for computer modeling of OLED structures are used [1, 2]. However, among the significant number of software products should be noted the electronic circuits simulator with open source SPICE (Simulation Program with Integrated Circuit Emphasis). This software product is used in circuit modeling applications, in particular SPICE (Personal SPICE), Proteus, LTSPICE (Linear Technology SPICE), MicroCap. Much of the element base of inorganic electronics is represented by ready-made mathematical models created by the manufacturers of these elements. However, in the direction of organic electronic components, such models are used very rarely. This is due to certain difficulties in manufacturing technology, which is constantly looking for optimal solutions.

This paper presents a variant of constructing a SPICE model of a light-emitting organic structure. The mathematical model takes into account the internal structure of the semiconductor displayed equivalent circuit. Open source code allows you to change the parameters of the elements of the equivalent circuit according to the internal structure of OLED. The use of such an approach of construction allows us to carry out model researches of parameters of the developed structures and to define their change in the course of operation. Model studies make it possible to obtain volt-ampere characteristics of the studied structures and determine the influence of external factors on them.

The obtained simulation results can be used to build control systems that control the drift characteristics and color temperature of OLED radiation directly during their operation (such solutions use the term - "in-situ", which literally means "inside" the body or system.

I. SPICE model of the OLED structure

The developed model is based on one of the traditional SPICE substitution schemes (equivalent schemes) of diode structures, in particular, Level 1

Standard SPICE or Level 2 PSpice. If necessary, provide higher accuracy represent the specific structure of the OLED, the model is supplemented by other components, which, in particular, provide higher flexibility in shaping the CVC OLED structure and its thermal characteristics. The SPICE model substitution diagram is shown in Fig. 1 in addition to diode D contains a series RS and parallel RL resistance, as well as two dependent on the bias mode and capacitance current - CJ and CD (hereinafter according to SPICE syntax model parameters are written without indexes). Serial resistance RS describes the resistance of the electrodes and layers of the passive structure of OLED, and parallel RL - parasitic current seepage through the structure of ohmic (linear) nature. The CJ component describes the barrier and the CD diffusion capacitance of the diode structure. The first of them CJ determines the dynamic characteristic at the reverse bias of the diode and depends on the voltage on the diode, and the second CD - the transient process at direct bias and is determined by the diode current due to the accumulation effect in the structure of non-basic charge carriers.

During SPICE model specification diode structures using a set of parameters that describe their CVC, performance, and temperature dependence of these characteristics. The main parameters of the model of diode structures [4] are: IS (Saturation current) - saturation current; N (Emission coefficient) - emission coefficient; ISR (Recombination current) - recombination current; IKF (High-injection "knee" current) - current of transition to a high level of injection; BV (Reverse breakdown "knee" voltage) - reverse breakdown voltage; RS (Series resistance) - series ohmic resistance of the structure; RL (Junction Leakage Resistance) - parallel ohmic impregnation resistance; TT (Transit time) - time of flight of carriers; CJ0 (Zero-bias junction capacitance) - capacity at zero offset; VJ (Junction potential) - barrier potential; M (Junction grading) - barrier potential gradient; EG (Energy gap eV) - bandwidth; XTI (Temperature exponent for IS) - exponential saturation current coefficient; KF (Flicker noise coefficient) - linear flicker noise coefficient; AF (Flicker noise exponent) - exponential flicker noise coefficient; TIKF (IKF linear temperature coefficient) - linear temperature coefficient IKF; TBV1 (BV linear temperature coefficient) - linear temperature coefficient BV; TBV2 (BV quadratic temperature coefficient) - quadratic temperature coefficient BV; TRS1 (RS linear temperature coefficient) linear temperature coefficient RS; TRS2 (RS quadratic temperature coefficient) RS quadratic temperature coefficient; T_MEASURED (Measured temperature Celsius), T_ABS (Absolute temperature), T_REL_GLOBAL (Relative to current temperature), T_REL_LOCAL (Relative to AKO model temperature) - temperatures in degrees Celsius.

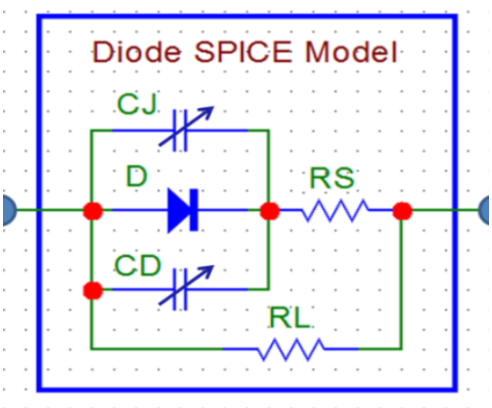


Fig. 1. Scheme of substitution of SPICE model of OLED structure.

CVC diode structure that describes the dependence of the current ID on the voltage VD, a SPICE model is defined as [%1]:

$$I_D = I_S\left[exp\left(\frac{V_D}{N\cdot\varphi_T}\right)\right],$$

$$\varphi_T = \frac{k\cdot T}{q},$$

where: k - Boltzmann constant, q - electron charge, T - thermodynamic temperature.

The above barrier and diffusion capacitances of the diode are determined by the equations:

$$C_J = \frac{C_{0J}}{\left(1-\frac{V_D}{V_J}\right)^M};$$

$$C_D = \frac{I_D\cdot TT}{\varphi_T}.$$

The model is supplemented with resistive components if necessary. This makes it possible to more accurately represent the temperature dependences of the resistance of the layers of the passive structure of OLED. For this purpose, the extended model of the resistor which includes the following parameters is used: R (Resistance multiplier) - the multiplier of resistance; CP (Parallel capacitance) - parallel capacity; LS (Series inductance) - series inductance; TC1 (Linear temperature coefficient) linear temperature coefficient; TC2 (Quadratic temperature coefficient) - quadratic coefficient; TCE (Exponential temperature coefficient) - exponential coefficient; NM (Noise multiplier) - noise multiplier; T_MEASURED (Measured temperature, Celsius) - measuring temperature in C; T_ABS (Absolute temperature, Celsius) - the absolute temperature in °C.

According to the SPICE syntax, the temperature is denoted by TEMP. In addition, it is important to note that all temperature values, including T_ABS (absolute temperature), are not absolute thermodynamic temperatures, but Celsius temperature for the whole circuit. The resistance function of the resistor to temperature is determined by the quadratic and exponential dependences:

$$R(TEMP) = R(T_{NOM})\cdot K_{TQ}\cdot K_{TE};$$

$$K_{TQ} = 1 + TC1\cdot(T - T_{NOM}) + TC2\cdot(T - T_{NOM})^2;$$

$$K_{TE} = 1.01^{TCE(T-T_{NOM})}.$$

The TNOM value is the nominal temperature value, which in SPICE models corresponds to 27°C, that is, approximately 300 K of absolute thermodynamic temperature. When specifying the model of a certain component, it is possible to change the temperature of this component relative to the temperature of its base model. Use the concept of AKO model clone (an acronym for A Kind Of) with the corresponding temperature values: T_REL_GLOBAL (Relative to current temperature, Celsius) - relative temperature in °C and T_REL_LOCAL (Relative to AKO model temperature, Celsius) - relative temperature of the cloned model in °C. In this specification version, the actual temperature of the component (Device operating temperature) is determined by one of three values: T_ABS, T_REL_LOCAL + T_ABS (of AKO parent), T_REL_GLOBAL + TEMP (global temperature, °C). The SPICE specification window of the resistor model and an example of temperature dependences of resistance R (RDT) for a certain set of temperature coefficients are presented in Fig. 2, and examples of temperature dependences of absolute values of current ABS (I (DOLE)) of the diode on temperature - in Fig.2, b.

II. Study of the OLED structure model

Research CVC OLED semiconductor structure by changing certain parameters is shown in Fig. 3 (RL =

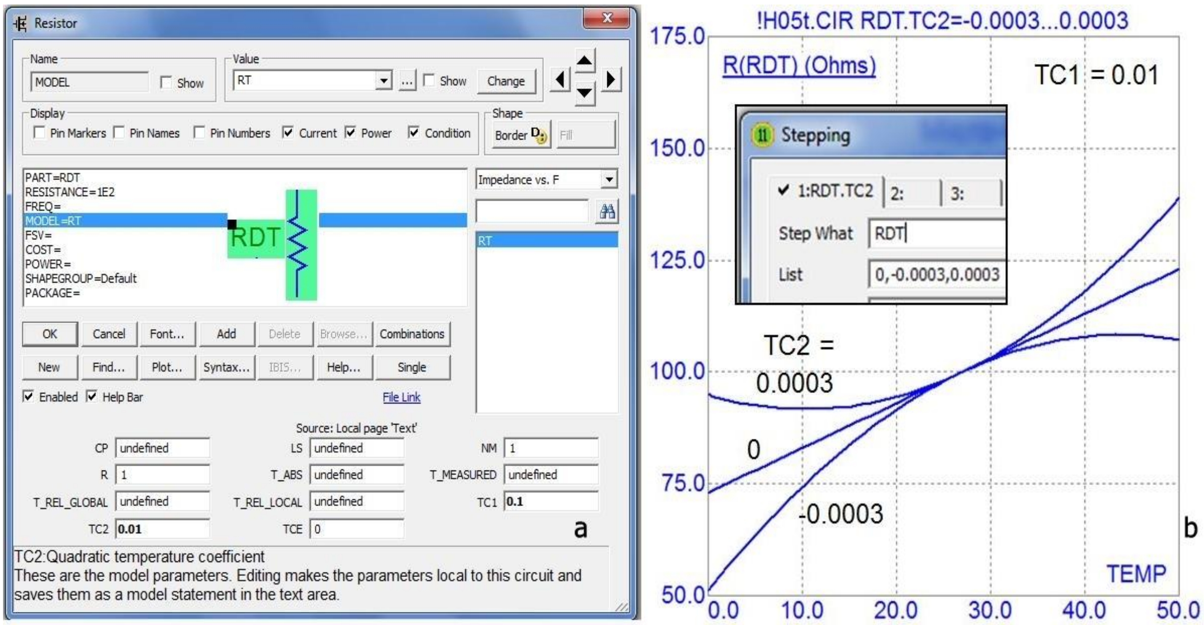


Fig. 2. SPICE specification window of the resistor model (a) and an example of temperature dependences of resistance R (RDT) at: TC1 = 0,001; TC2 = -0.0003, 0, 0.0003 (b).

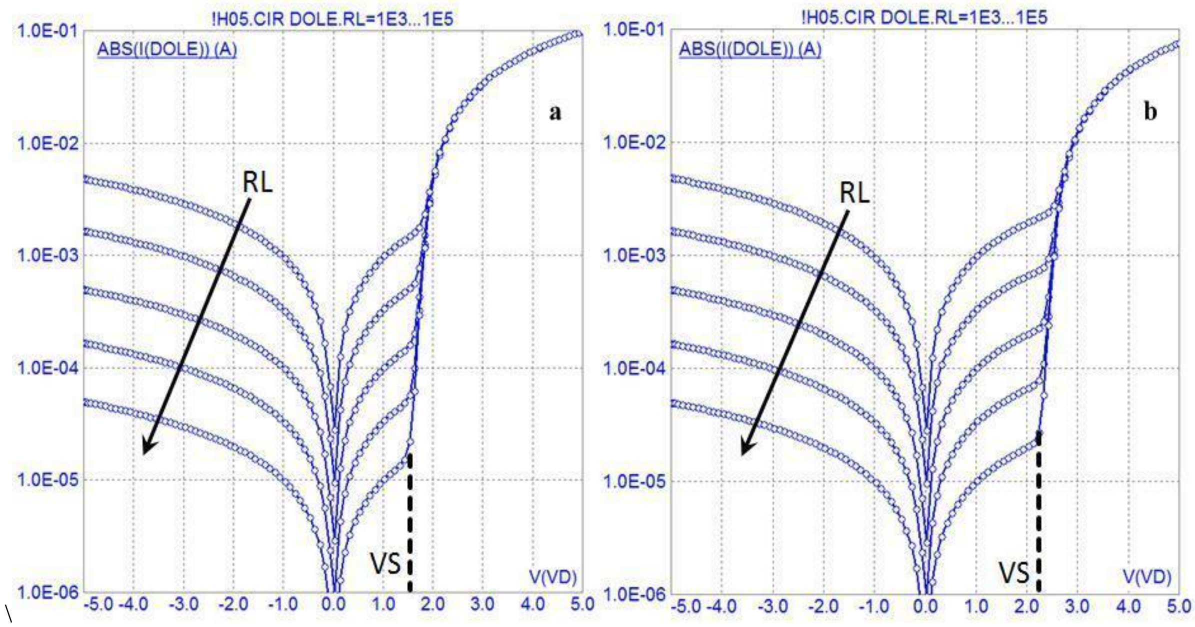
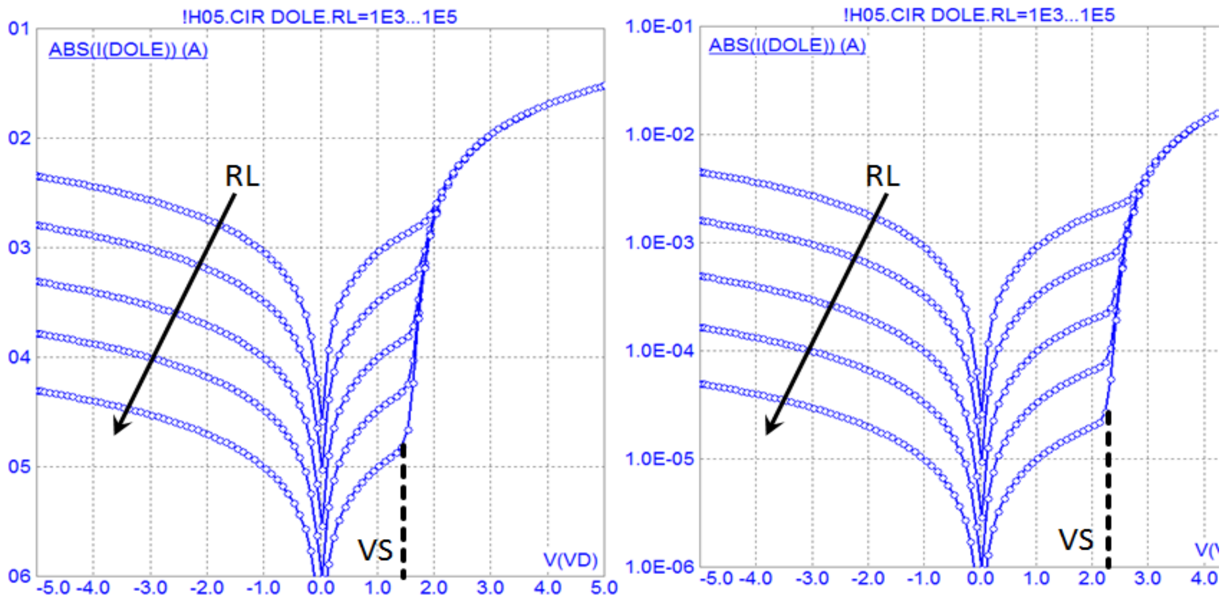


Fig. 3. VAC models OLED structure with IS=1E-18 (a) end IS=1E-24 (b)
[RL = 1E3, 3E3, 1E4, 3E4, 1E5 @ RS = 30].



a) b)

Fig.4. VAC models OLED structure with IS=1E-18 (a) end IS=1E-24 (b)
[RL = 1E3, 3E3, 1E4, 3E4, 1E5 @ RS = 100]

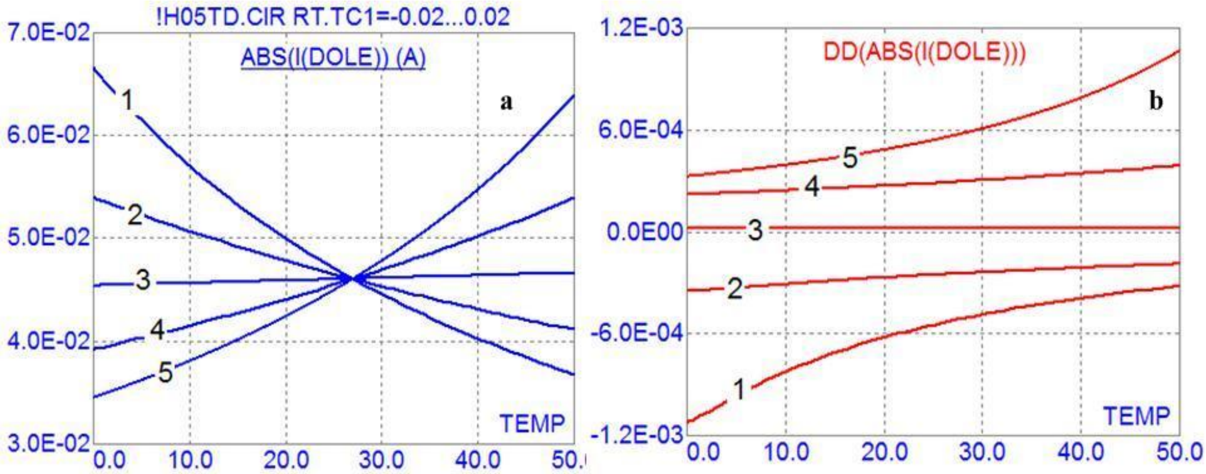


Fig. 5. Examples of temperature dependences of absolute values of current ABS (I (DOLE)) (a) and their steepness DD (ABS (I (DOLE))) (b) at: TC1 = -0,02 (1), -0,01 (2), 0 (3), 0,01 (4), 0,02 (5).

1E3, 3E3, 1E4, 3E4, 1E5; RS = 30) and Fig. 4 (RL = 1E3, 3E3, 1E4, 3E4, 1E5; RS = 100). The notation V (D) corresponds to the value of the voltage on the diode, and ABS (I (DOLE)) - the absolute values of the current, where DOLE - the model name of the OLED structure (according to SPICE syntax, the name of the diode model must begin with the letter D - DOWN). The designation VS corresponds to the transition voltages from the barrier to the predominantly ohmic current-carrying mechanism.

You can see that the CVC received generally well represent actual specifics OLED structures and are characterized by the ability to adapt sufficiently effective under these experimental data specific models.

The temperature dependence of the studied structure is reflected by the dependence of the absolute values of the current ABS (I (DOLE)) and its steepness (Fig. 5, a). For greater clarity, the temperature dependence can be represented as a derivative (slope) of the temperature, denoted by DD (A), where A is the studied value. This is shown in Fig. 5, b, where DD (ABS (I (DOLE))) - the slope of the current function ABS (I (DOLE)).

The study of the dynamic characteristics of the SPICE model of OLED structure is to establish the time of flight of TT carriers (Transit time) taking into account the values of barrier CJ and diffusion CD capacity. 2E-6 (sec) are presented in Fig. 6 and Fig. 7.

The simulation results reflect the transients in the OLED structure and can be used to determine the mobility of charge carriers by extraction in the case of linear voltage rise (CELIV) [4-6]. The use of this method allows the calculation of technological parameters, in particular, to determine the film thickness of

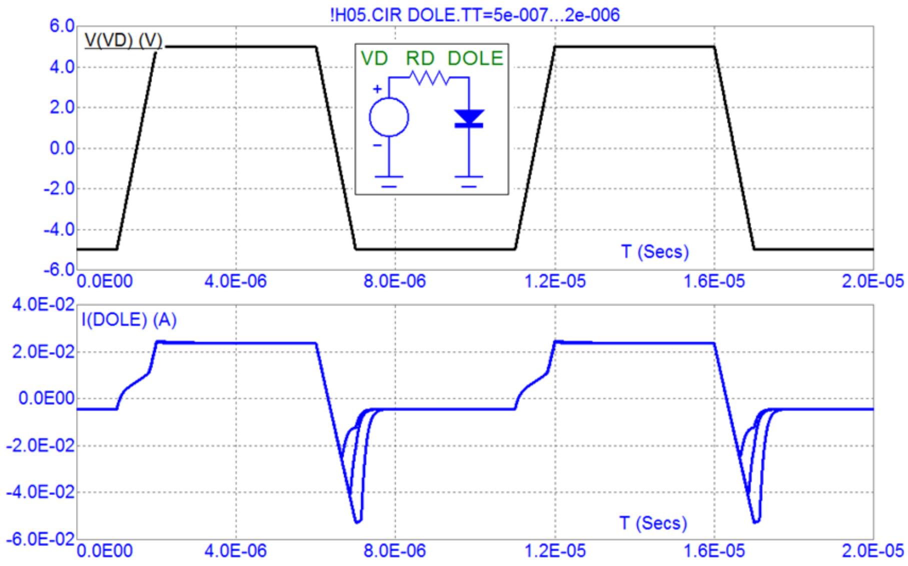


Fig. 6. An example of the study of transients at CJ0 = 1E-9F.

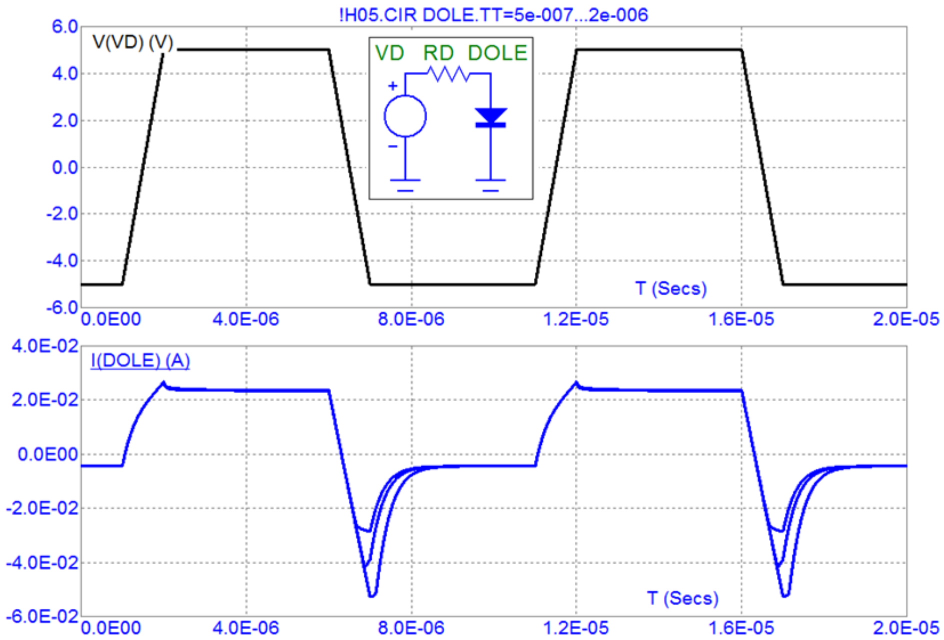


Fig. 7. An example of the study of transients at CJ0 = 3E-9F.

semiconductor structures.

Conclusions

The problem of building a SPICE model for studying the parameters of OLED structures is substantiated and solved. The model is based on the basic elements of the simulator in accordance with the generalized equivalent scheme that reflects the internal structure. For modeling results that match the actual parameters of the structures set up data structures diode SPICE model describing the CVC, performance, and temperature dependence of these characteristics.

The mathematical study of the measurement process CVCs and shows the value and impact of the barrier Cj Cd capacity diffuse transition. Calculations of the magnitude of thermal influence on the parameters of the elements are presented, which in the software specification of the model are included.

The CVC characteristics of OLED structures are obtained and the magnitude of the drift caused by the change in the value of RL under the influence of temperature is established. The possibility and example of research of temperature dependence of absolute values of current and its steepness are shown.

The possibilities of using the developed model for research of dynamic characteristics of the organic structure are shown. The results of such modeling can be used to determine the mobility of the charge carriers of the semiconductor junction and the film thickness of the test sample.

The proposed approach to the implementation of the model provides for the possibility of expanding its functionality by expanding the detail of the structure with a corresponding expansion of the equivalent scheme, which will provide higher accuracy of the results.

The model can be successfully used for the construction of power management OLED which uses dynamic control of the CVC and potential adjustments.

Barylo G.I. - DScTech., associated professor, associated professor of electronic devices department;
Holiaka R.L. - DScTech., professor, professor of electronics and information technology department
Helzhynskyi I.I. - PhDTech., associated professor, associated professor of electronic devices department;
Hotra Z.Yu. - DScTech., professor, head of electronic devices department;
Ivakh M.S. - PhDTech., senior lecturer of electronic devices department;
Politanskyi R.L. - DScTech., professor, professor of radio engineering and information security department.

[1] R.L. Politanskyi, M.V. Vistak, G.I. Barylo , A.S. Andrushchak, Optical Materials Volume 102 (2020) (doi: 10.1016/j.optmat.2020.109782).

[2] V. Cherpak, A. Gassmann, P. Stakhira, D. Volyniuk, J. Grazulevicius, A. Michaleviciute, A. Tomkeviciene, G. Barylo, H. Von Seggern, Organic Electronics 15, 7 (2014) (doi: 10.1016/j.orgel.2014.04.002).

[3] H.V. Baryshnikov, D.Iu. Volyniuk, I.I. Helzhynskyi, Z.Yu. Hotra, B.P. Minaiev, P.I. Stakhira, V.V. Cherpak, Orhanichna elektronika (Lvivska politekhnika, Lviv 2019).

[4] G. Barylo, V. Cherpak, G. Pakhomov, Z. Hotra, I. Helzhynskyy, M. Hladun, G.Wiosna-Salyga, B Luszczynska, P. Stakhira, Molecular Crystals and Liquid Crystals 639, 1 (2016) (doi: 10.1080/15421406.2016.1255075).

[5] K. Ivaniuk, M. Chapran, V. Cherpak, G. Barylo, P. Stakhira, Z. Hotra, M. Hladun, T. Dudok, Ukrainian Journal of Physical Optics 16 (2015) (doi: 10.3116/16091833/16/2/95/2015).

[6] T. Lawicki, G. Barylo, M. Vistak, R. Holyaka, V. Virt, F. Vezyr, Nursanat Askarova, Proc. SPIE 10808, Photonics Applications in Astronomy, Communications, Industry, and High-Energy Physics Experiments 2018, 108086E (doi: 10.1117/12.2501630).

Г.І. Барило[1], Р.Л. Голяка[1], І.І. Гельжинський[1], З.Ю. Готра[1], М.С. Івах[1], Р.Л. Політанський[2]

Моделювання органічних світловипромінювальних структур

[1]Національний університет «Львівська політехніка», Львів, Україна, hryhorii.i.barylo@lpnu.ua
[2]Чернівецький національний університет ім. Юрія Федьковича, Чернівці, Україна

В роботі приведені результати дослідження параметрів органічних світловипромінюючих структур на основі SPICE (Simulation Program with Integrated Circuit Emphasis) модельних досліджень. Розроблено SPICE модель діодної структури, яка реалізована у вигляді схеми заміщення на основі базових компонентів симулятора. Дана модель може бути розширена шляхом введення додаткових компонентів схему заміщення, що забезпечує вищу точність представлення специфіки структури. Представлено графічні результати досліджень моделі OLED структури при зміні внутрішніх параметрів. Отримані дані добре представляють параметри реальних структур та характеризуються достатньо ефективною адаптацією до експериментальних даних конкретних взірців.

Ключові слова: світловипромінювальні структури, модель, SPICE моделювання.