

SPICE-Based Lumped Circuit Model of Multiconductor Lines Excited by an Incident Plane Wave

Moustafa Raya

*Chair for Electromagnetic Compatibility
Otto von Guericke University Magdeburg
Magdeburg, Germany
moustafa.raya@ovgu.de*

Mathias Magdowski

*Chair for Electromagnetic Compatibility
Otto von Guericke University Magdeburg
Magdeburg, Germany
mathias.magdowski@ovgu.de*

Sergey V. Tkachenko

*Chair for Electromagnetic Compatibility
Otto von Guericke University Magdeburg
Magdeburg, Germany
sergey.tkachenko@ovgu.de*

Ralf Vick

*Chair for Electromagnetic Compatibility
Otto von Guericke University Magdeburg
Magdeburg, Germany
ralf.vick@ovgu.de*

Abstract—This paper presents a SPICE-based circuit model for multiconductor lines excited by an incident plane wave. The introduced method allows engineers to calculate field coupling and crosstalk for multiconductor lines directly in circuit simulators without the need for field simulation programs. The presented model is suitable for EMC studies where the voltage response at the loads of the lines can be calculated due to plane wave coupling or due to lumped interference sources. The conductors are divided into cells and represented in terms of lumped elements and controlled sources. The presented model of multiconductor lines can be used in the frequency domain or in the time domain with nonlinear elements. The model validation shows excellent agreement of the results with those obtained with conventional field simulations.

Index Terms—Multiconductor lines, SPICE, field coupling

I. INTRODUCTION

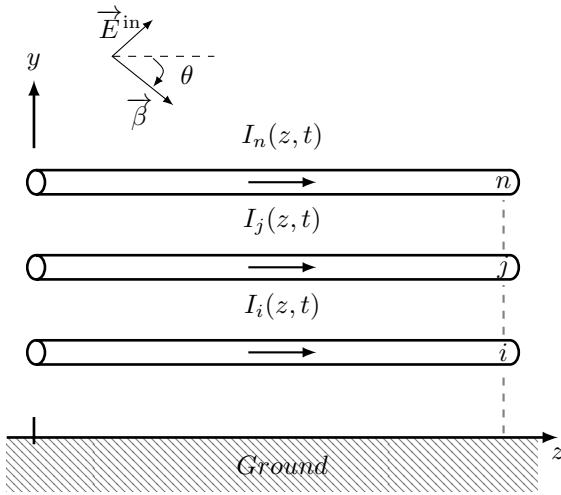
Since transmission lines represent a significant source of electromagnetic field coupling in electrical systems, the study of field coupling on such lines continues to be of great interest in the EMC world. Conductor lines could be illuminated by extraneous electromagnetic fields, such as emissions from high-power radars, radio towers, and nuclear electromagnetic pulses (EMP). Analytical solutions for single and multiconductor cables have been presented in various forms [1]–[4]. The transfer of these solutions into SPICE environments has been done in several works, such as [2], [5]–[9]. However, the integration of such models into SPICE programs has not yet been achieved. In previous work, circuit models were developed for a single conductor and a shielded cable over a ground plane [10]–[12], but models for multiconductor lines have not yet been developed.

With the current state of the art for calculating field coupling on transmission lines, engineers only have the option of using full-wave field simulations or co-simulations. Full-wave simulations require a higher computational effort and reach their limits for long cable lengths. Co-simulation of field and circuit solvers based on defined cable libraries has led to a significant reduction in the computational effort. However, the field coupling must first be calculated in the field simulator, which is then transferred into the circuit simulator, as in CST Cable Studio [13].

This solution is not ideal as engineers typically need a license for field simulation programs. In addition, they perform analysis of complex circuits directly in a circuit simulation program and may encounter field simulation limitations when considering complex conductor structures. A better conception of EMC studies is when the field coupling on multiconductor lines can be analyzed directly in a circuit simulator like SPICE, taking crosstalk into account. Unfortunately, SPICE simulation programs are still far away from this idea, since they usually only have simple transmission line models in their libraries.

In this work, this problem is addressed by presenting a lumped circuit model for multiconductor lines. The field coupling of a plane wave is considered in the model without the need for field simulations. The user only needs to define the properties of the conductors and the incident plane wave to run the model directly in SPICE. Excellent results were obtained when compared with field simulations in CST [13].

The rest of this paper is organized as follows. In Section II, the circuit model for the multiconductor lines is developed. The model of three conductors over a ground plane is validated in Section III. The conclusion follows in Section IV.


 Fig. 1. n -uniform cylindrical wires over a ground plane.

II. CIRCUIT MODEL

Consider n uniform conductors above a ground plane in Fig. 1, which are excited by an incident plane wave. The incident electric field of the plane wave \vec{E}^{in} with an amplitude E_0 and the reflected electric field \vec{E}^{re} are defined as follows:

$$\vec{E}^{\text{in}} = E_0 [e_y \vec{a}_y + e_z \vec{a}_z] e^{-j\beta_y y} e^{-j\beta_z z} \quad (1a)$$

$$\vec{E}^{\text{re}} = E_0 [e_y \vec{a}_y - e_z \vec{a}_z] e^{j\beta_y y} e^{-j\beta_z z}. \quad (1b)$$

The elements e_y and e_z are given by $e_y = \cos(\theta)$ and $e_z = \sin(\theta)$ with the unit vectors \vec{a}_y and \vec{a}_z . The phase constants along the axes are defined by $\beta_y = -\beta \sin(\theta)$ and $\beta_z = \beta \cos(\theta)$. The parameter $\beta = \omega/v_0$ is the phase constant in free space with the phase velocity $v_0 = 1/\sqrt{\mu_0 \epsilon_0}$, where μ_0 and ϵ_0 represent the permeability and permittivity of free space.

The differential equation for field coupling on n -conductor lines was presented by Agrawal et al. [1] and can be written as follows:

$$\frac{d\mathbf{V}^s(z)}{dz} + j\omega \mathbf{L}' \cdot \mathbf{I}(z) = \mathbf{V}'_{\text{tan}}(z) \quad (2)$$

$$\frac{d\mathbf{I}(z)}{dz} + j\omega \mathbf{C}' \cdot \mathbf{V}^s(z) = 0. \quad (3)$$

The $n \times n$ matrix \mathbf{L}' represents the per-unit-length (p.u.l.) inductance of the lines, where L'_{ii} stands for the p.u.l. self-inductance of the i -th conductor and L'_{ij} for the mutual inductance between the i -th and j -th conductor. The same applies to the p.u.l. capacitance matrix \mathbf{C}' . The $n \times 1$ vectors $\mathbf{V}^s(z)$ and $\mathbf{I}(z)$ represent the scattered voltage and total current with respect to the ground plane. The $n \times 1$ vector $\mathbf{V}'_{\text{tan}}(z)$ represents the p.u.l. voltage induced by the superposition of the tangential electric field on the conductors. For simplicity,

$V'_{\text{tan},i}(z)$ is called the p.u.l. tangential voltage of the i -th conductor and is defined as follows:

$$\begin{aligned} V'_{\text{tan},i}(z) &= E_z^{\text{in}}(h_i, z) + E_z^{\text{re}}(h_i, z) \\ &= -j2E_0 \sin(\theta) \sin(\beta_y h_i) e^{-j\beta_z z}. \end{aligned} \quad (4)$$

Note that h_i represents the height of the i -th conductor above the ground.

Since the total induced voltage is of interest and not the scattered one, the total voltage of the i -th conductor can be represented as follows [14]:

$$V_i(z) = V_i^s(z) - \underbrace{\int_0^{h_i} (E_y^{\text{in}}(z) + E_y^{\text{re}}(z)) dy}_{V_i^t(z)}. \quad (5)$$

The elements $E_y^{\text{in}}(z)$ and $E_y^{\text{re}}(z)$ represent the transverse incident and reflected electric field. The so-called transverse voltage $V_i^t(z)$ of the i -th conductor, was added to represent the total one.

A. Circuit Model Approach

To develop the lumped circuit model for multiconductor lines, the wire bundle is divided into short segments of length $l_{\text{step}} < (\lambda/10)$, where λ is the wavelength of the maximum frequency of interest. The maximum frequency of interest can be calculated by the reciprocal of the rise or fall time for the case of a symmetrical trapezoidal function in the time domain. Each wire segment is represented in SPICE by the parameters p.u.l., where the coupling between wires is given by L'_{ij} and C'_{ij} for the i -th and j -th wires, respectively (see Fig. 4). Based on (2) and (3), the field coupling is considered by including (4) in each segment. This creates a model that calculates only the scattered voltages. To represent the total voltages on the loads of the i -th conductor, the transverse voltages from (5) must be added as follows [14]:

$$V_i(0) = V_i^s(0) - \underbrace{\int_0^{h_i} (E_y^{\text{in}}(0) + E_y^{\text{re}}(0)) dy}_{V_{t,i}^1} \quad (6)$$

$$V_i(\mathcal{L}) = V_i^s(\mathcal{L}) - \underbrace{\int_0^{h_i} (E_y^{\text{in}}(\mathcal{L}) + E_y^{\text{re}}(\mathcal{L})) dy}_{V_{t,i}^2}, \quad (7)$$

where \mathcal{L} is the length of the wire. The elements $V_{t,i}^1$ and $V_{t,i}^2$ are added in the form of lumped voltage sources, as shown in Fig. 4.

The antenna type current induced in the conductors is neglected. This means that the sum of the induced currents in the conductors is equal and opposite in sign to the corresponding z -component of the current in the reference conductor.

Since (4) as well as $V_{t,i}^1$ and $V_{t,i}^2$ from (6) and (7) are in a mathematical form containing integrals and frequency-dependent exponential functions, they cannot be directly included in SPICE programs. For this reason, circuit models for these functions will be developed in the next steps.

1) *Tangential Voltages*: The calculation of the tangential voltage for a single cable over a ground plane was performed in [11] and is briefly described here for the i -th conductor $V'_{\text{tan},i}(z)$.

A multiplication of $\beta_y h$ on the numerator and denominator of (4) leads to

$$\begin{aligned} V'_{\text{tan},i}(z) &= -j2E_0 \sin(\theta) \frac{\sin(\beta_y h_i)}{\beta_y h_i} \cdot \beta_y h_i \cdot e^{-j\beta_z z} \\ &= -j2E_0 \sin(\theta) \text{sinc}(-\omega \sin(\theta) T_y) \cdot \beta_y h_i \cdot e^{-j\beta_z z} \\ &\approx -j2E_0 \sin(\theta) \cdot \beta_y h_i \cdot e^{-j\beta_z z}. \end{aligned} \quad (8)$$

The transit time T_y between the ground plane and the i -th conductor was neglected by assuming a short distance compared to the length of the conductor. Therefore, $\text{sinc}(-\omega \sin(\theta) T_y) = 1$.

The representation of (8) in Laplace form in preparation for the transformation to the time domain leads to

$$V'_{\text{tan},i}(s) \approx 2h_i \cdot \frac{\sin^2(\theta)}{v_0} \cdot sE_0(s) \cdot e^{-s \frac{\cos(\theta)}{v_0} z}. \quad (9)$$

The transformation into the time domain results in

$$V'_{\text{tan},i}(t) = 2h_i \frac{\sin^2(\theta)}{v_0} \frac{d \left(E_0 \left(t - \frac{\cos(\theta)}{v_0} z \right) \right)}{dt}. \quad (10)$$

In order to eliminate the physical position dependency, a multiplication with $n \cdot l_{\text{step}}$ was carried out instead of z , where

$$\begin{aligned} V'_{\text{tan},i}(t) &= \\ &\underbrace{\frac{\Gamma}{2l_{\text{step}} h_i}}_{\Gamma} \frac{\sin^2(\theta)}{v_0} \frac{d \left(E_0 \left(t - \frac{\cos(\theta)}{v_0} \cdot \overbrace{n \cdot l_{\text{step}}}^{T_{d,i}} \right) \right)}{dt}. \end{aligned} \quad (11)$$

The segment length l_{step} was also multiplied externally to remove the dependence on the length per unit. The element n in (11) stands for the order of the section. This means that the first section of the conductor has the order $n = 1$, while the last section has the order $n = N_s$, where N_s is the total number of subdivided sections. The parameter l_{step} is the length of the subdivided sections with

$$l_{\text{step}} = \frac{\mathcal{L}}{N_s}. \quad (12)$$

Function (11) involves a time-shifted electric field $E_0(t)$, which can be modeled in a SPICE environment by a voltage source feeding a matched transmission line with a time delay $T_{d,i}$, as shown in Fig. 2. The derivation is obtained by calculating the current I_{C1} for a voltage source v_i feeding a 1 F capacitor. A controlled voltage source multiplied by the factor Γ finally gives the solution of $V'_{\text{tan},i}(t)$. This solution can be easily integrated into the sources $V_{\text{tan},i}^{1,2,3\dots}$ in Fig. 4.

2) *Transverse Voltages*: The functions $V_{\text{t},i}^1$ and $V_{\text{t},i}^2$ from (6) and (7) can also be simplified by neglecting the sinc function [11], where

$$\begin{aligned} V_{\text{t},i}^1 &= 2h_i E_0 \cos(\theta) \text{sinc}(\beta_y h_i) \\ &\approx 2h_i E_0 \cos(\theta). \end{aligned} \quad (13)$$

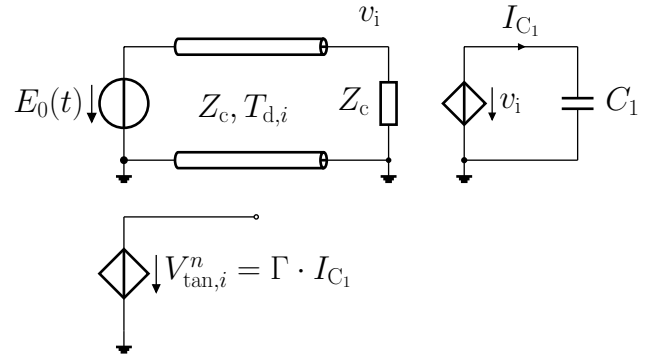


Fig. 2. Equivalent circuit for $V'_{\text{tan},i}(t)$.

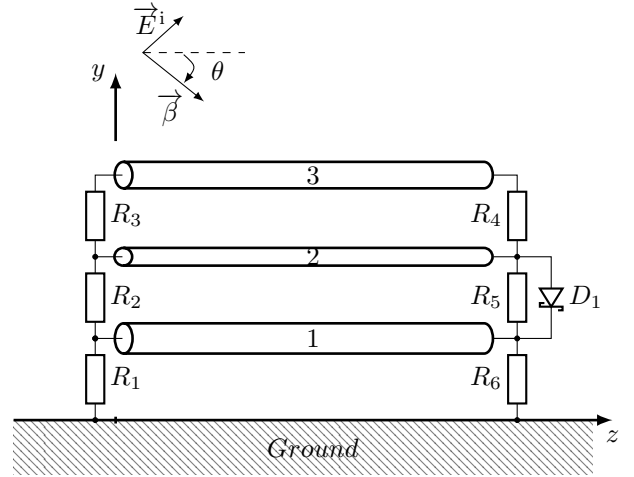


Fig. 3. Three wires over a ground plane.

The solution of $V_{\text{t},i}^1$ can be directly integrated into SPICE. The solution at $z = \mathcal{L}$ is given by $V_{\text{t},i}^2 = V_{\text{t},i}^1 e^{-j\beta_z \mathcal{L}}$ [11]. The transfer into the time domain leads to

$$V_{\text{t},i}^2 = V_{\text{t},i}^1 \left(t - \frac{\cos(\theta) \mathcal{L}}{v_0} \right). \quad (14)$$

A realization of this solution in SPICE can be done in a similar way as before, where a voltage source with amplitude $V_{\text{t},i}^1$ has to be connected to a matched transmission line with time delay $\cos(\theta) \mathcal{L} / v_0$. The result from (13) and (14) can be directly integrated in Fig. 4.

B. Three-conductor Lines over a Ground Plane

For a better understanding, three conductors with different diameters are considered in Fig. 3. All conductors have a length of $\mathcal{L} = 2$ m with heights of $h_1 = 10$ mm, $h_2 = 20$ mm and $h_3 = 40$ mm above the ground plane. The diameters are $d_1 = 1$ mm, $d_2 = 0.5$ mm and $d_3 = 2$ mm. The parameters p.u.l. were calculated with a static solver and are given by

$$\mathbf{L}' = \begin{bmatrix} 742.7 & 221.1 & 104.6 \\ 221.1 & 1019.1 & 224.2 \\ 104.6 & 224.2 & 886.4 \end{bmatrix} \text{ nH/m}$$

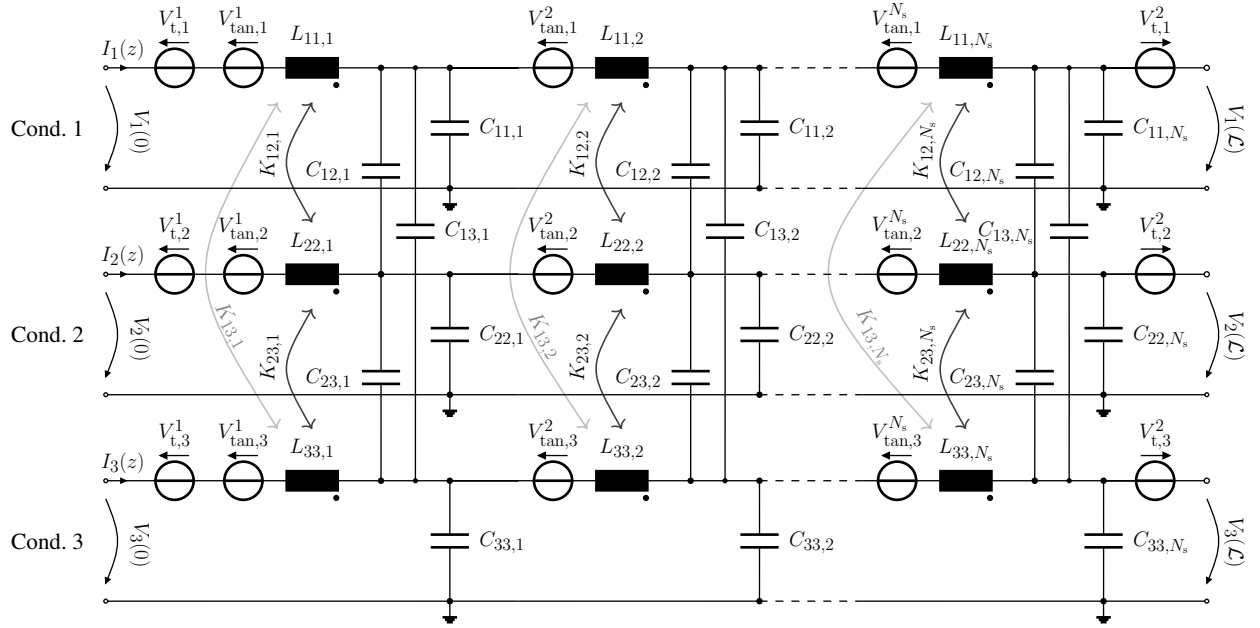


Fig. 4. Equivalent circuit consisting of N_s sections of three conductor lines above a ground plane, taking into account the field coupling.

$$\mathbf{C}' = \begin{bmatrix} 16.10 & -3.26 & -1.08 \\ -3.26 & 12.22 & -2.71 \\ -1.08 & -2.71 & 13.36 \end{bmatrix} \text{ pF/m.}$$

The loads in Fig. 3 are defined as follows: $R_1 = R_6 = 10 \Omega$, $R_2 = R_5 = 50 \Omega$, and $R_3 = R_4 = 100 \Omega$. A diode of type MBR0520L is connected in parallel with R_5 .

A circuit model of the three conductors is shown in Fig. 4. The inductive and capacitances of the i -th and j -th conductors at the n -th section are given by

$$L_{ii,n} = L'_{ii} \cdot l_{\text{step}} \quad (15a)$$

$$C_{ii,n} = C'_{ii} \cdot l_{\text{step}} \quad (15b)$$

$$C_{ij,n} = C'_{ij} \cdot l_{\text{step}}, \quad (15c)$$

where i and j vary between 1 and 3, and n between 1 and N_s . The segment length l_{step} was defined in (12).

The inductive coupling between the i -th and j -th conductors at the n -th cell is realized in SPICE by the inductance coupling coefficient $K_{ij,n}$, where

$$K_{ij,n} = \frac{L'_{ij}}{\sqrt{L'_{ii}L'_{jj}}}. \quad (16)$$

The voltage sources $V_{\text{tan},i}^n$ are included in each cell in Fig. 4 to represent the tangential voltage from Fig. 2 due to field coupling. To represent the total voltage, $V_{t,i}^1$ and $V_{t,i}^2$ from (13) and (14) are added at the ends. Since all components of the model in Fig. 4 were developed directly in SPICE, the model is now ready for circuit simulation.

III. MODEL VALIDATION

The setup with three wires shown in Fig. 3 is used for validation. The wires were divided into N_s sections and

a netlist based on Fig. 4 was created using MATLAB. It would also be quite easy and possible to use an open source program like GNU Octave to create the netlist instead of MATLAB. The netlist was imported into LTspice to perform the simulation [15]. The results of the presented SPICE model are compared with simulations performed in CST Studio Suite [13]. In CST's cable solver, a minimum sample length of 5 mm was specified for the mesh and losses were not considered. A PC with an Intel i7-6700 CPU 3.4 GHz and 16 GB of installed RAM was used for the simulation.

A. Time Domain

In Fig. 3, a plane wave was excited from a distance of $z = -50 \text{ cm}$ at $t = 0 \text{ s}$. The wave has an angle of incidence of $\theta = 45^\circ$ and an electric field strength of 100 V m^{-1} with a trapezoidal time function. The rise, fall and hold times are 2 ns, 3 ns and 10 ns, respectively.

The induced voltage at D_1 is presented in Fig. 5 for a different number of cells N_s . The rise time of 2 ns corresponds to $\lambda/10 = 5.9 \text{ cm}$, so the number of cells N_s should be at least 34 to obtain correct results. It can be seen in Fig. 5 that the deviation is significant with $N_s = 8$, but with $N_s = 34$ the results agree. Further increasing the number of elements with $N_s = 68$ slightly improves the results, but also increases the simulation time. The simulation time was 2.1 s for $N_s = 34$ and 7.3 s for $N_s = 68$. In comparison, the simulation time with CST was up to 3.2 min. The result for the voltage response at R_1 is shown in Fig. 6.

B. Frequency Domain

For the investigations in the frequency domain, the diode D_1 in Fig. 3 was removed. The induced voltage across R_6 is shown in Fig. 7 when a plane wave with $\theta = 45^\circ$ and

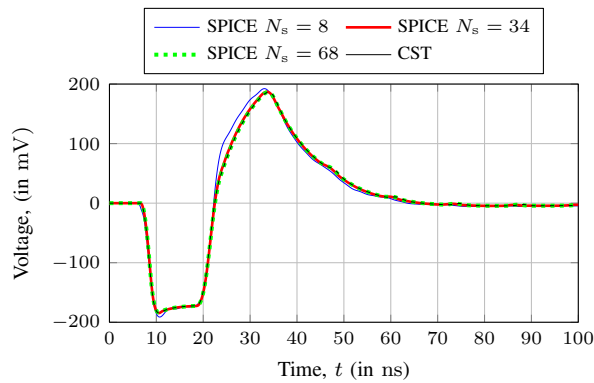


Fig. 5. Voltage response across D_1 .

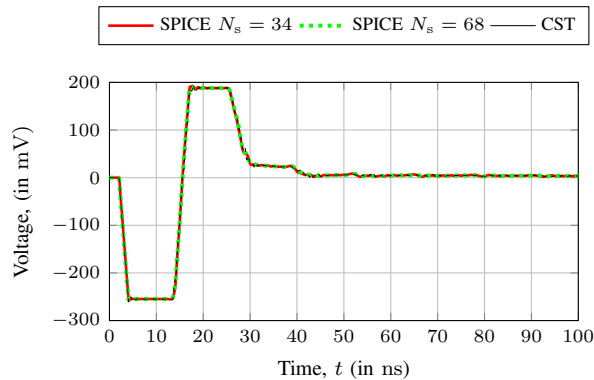


Fig. 6. Voltage response across R_1 .

$E_0 = 10 \text{ V m}^{-1}$ is excited. From the results, it can be seen that when the cable is divided into $\lambda/10$ sections with $N_s = 34$, a slight deviation is observed where a frequency shift occurs mainly at higher frequencies. As the number of sections increases, this shift is compensated. The simulation time of the developed lumped model in LTspice was 3.6 s for $N_s = 102$, compared to 4.6 min with CST.

IV. CONCLUSION

This paper presents a model of multiconductor transmission lines for SPICE simulations. An advanced simulation method is presented that allows users to simulate field coupling on transmission lines directly in SPICE programs without the need for field simulations. The transmission lines are represented by coupled, cascaded cells. Each cell consists of LC elements with controlled sources and coupling coefficients K . The field coupling of a plane wave is accounted for in the model by developing equivalent circuits directly in SPICE. The model can be used for simulation in the frequency domain or together with nonlinear elements in the time domain. The results show excellent agreement compared to field simulations as long as the assumptions and approximation of classical transmission line theory are valid.

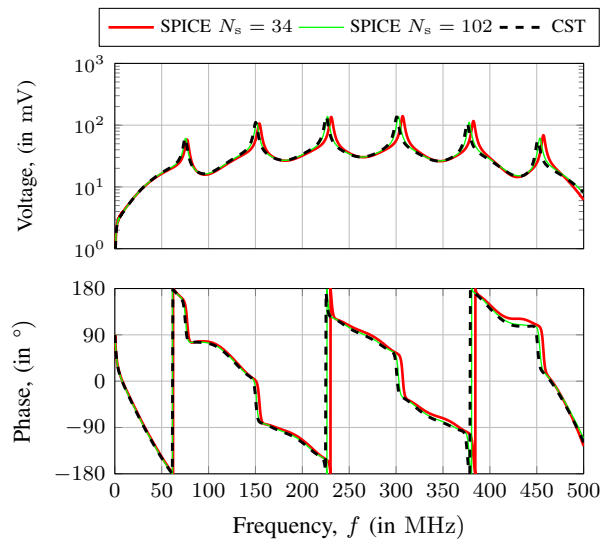


Fig. 7. Voltage response across R_6 .

REFERENCES

- [1] A. Agrawal, H. Price, and S. Gurbaxani, "Transient response of multi-conductor transmission lines excited by a nonuniform electromagnetic field," in *1980 Antennas and Propagation Society*, June 1980, pp. 432–435.
- [2] C. R. Paul, *Analysis of multiconductor transmission lines*, 2nd ed. Hoboken, N.J. and Chichester: Wiley, 2008.
- [3] C. Taylor, R. Satterwhite, and C. Harrison, "The response of a terminated two-wire transmission line excited by a nonuniform electromagnetic field," *IEEE Trans. Antennas Propag.*, vol. 13, no. 6, pp. 987–989, 1965.
- [4] R. Perez, *Handbook of electromagnetic compatibility*. San Diego and London: Academic Press, 1995.
- [5] I. Maio, F. G. Canavero, and B. Dilecce, "Analysis of crosstalk and field coupling to lossy MTLs in a SPICE environment," *IEEE Trans. Electromagn. Compat.*, vol. 38, no. 3, pp. 221–229, 1996.
- [6] I. Wuyts and D. de Zutter, "Circuit model for plane-wave incidence on multiconductor transmission lines," *IEEE Trans. Electromagn. Compat.*, vol. 36, no. 3, pp. 206–212, 1994.
- [7] Y. Kami and R. Sato, "Circuit-concept approach to externally excited transmission lines," *IEEE Trans. Electromagn. Compat.*, vol. EMC-27, no. 4, pp. 177–183, Nov 1985.
- [8] M. Raya and R. Vick, "SPICE models for one-conductor and three-conductor lines excited by a uniform plane wave," in *2019 IEEE International Symposium on Electromagnetic Compatibility, Signal Power Integrity (EMC+SIPI)*, July 2019, pp. 360–365.
- [9] M. Raya and R. Vick, "SPICE models of shielded single and multiconductor cables for EMC analyses," *IEEE Trans. Electromagn. Compat.*, vol. 62, no. 4, pp. 1563–1571, 2020.
- [10] M. Raya, S. V. Tkachenko, and R. Vick, "A SPICE model for a field-coupled conductor based on the scattered voltage formulation," in *2021 Asia-Pacific International Symposium on Electromagnetic Compatibility (APEMC)*, 2021, pp. 1–4.
- [11] M. Raya, M. Magdowski, and R. Vick, "SPICE-based lumped circuit model of shielded cables for EMC analyses," in *2020 International Symposium on Electromagnetic Compatibility - EMC EUROPE*, 2020, pp. 1–5.
- [12] —, "SPICE-Based lumped circuit model of shielded multiconductor cables," in *2021 Asia-Pacific International Symposium on Electromagnetic Compatibility (APEMC)*, 2021, pp. 1–4.
- [13] CST, (accessed February 1, 2022). [Online]. Available: <https://www.3ds.com>
- [14] F. M. Tesche, M. Ianoz, and T. Karlsson, *EMC analysis methods and computational models*. New York and Chichester: Wiley, 1997.
- [15] LTspice, (accessed February 1, 2022). [Online]. Available: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html/>