Circuit models and SPICE macro-models for quantum Hall effect devices

Massimo Ortolano\textsuperscript{1,2} and Luca Callegaro\textsuperscript{2}

\textsuperscript{1} Dipartimento di Elettronica e Telecomunicazioni, Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129 Torino, Italy
\textsuperscript{2} INRIM - Istituto Nazionale di Ricerca Metrologica, Strada delle Cacce, 91 - 10135 Torino, Italy

E-mail: massimo.ortolano@polito.it

Received 9 January 2015, revised 8 May 2015
Accepted for publication 26 May 2015
Published 16 July 2015

Abstract

Precise electrical measurement technology based on the quantum Hall effect is one of the pillars of modern quantum electrical metrology. Electrical networks including one or more QHE elements can be used as quantum resistance and impedance standards. The analysis of these networks allows metrologists to evaluate the effect of the inevitable parasitic parameters on their performance as standards. This paper presents a concise review of the various circuit models for QHE elements proposed in the literature, and the development of a new model. This last model is particularly suited to be employed with the analogue electronic circuit simulator SPICE. The SPICE macro-model and examples of SPICE simulations, validated by comparison with the corresponding analytical solution and/or experimental data, are provided.

Keywords: quantum Hall effect, quantum Hall resistance standards, SPICE

(Some figures may appear in colour only in the online journal)

1. Introduction

The quantum Hall effect (QHE) has been the basis of resistance metrology for 25 years in dc [1–3] and more recently in the ac regime [4–7]. In dc, networks of several QHE elements have been developed to realize quantum Hall array resistance standards (QHARS) [8–16], intrinsically-referenced voltage dividers [17], and Wheatstone bridges [18]. In ac, Schurr et al [19] included two QHE elements in a quadrature bridge to realize the unit of capacitance.

In 1988, Ricketts and Kemeny (RK) [20] proposed a circuit model suitable for the symbolic analysis of arbitrary electrical networks containing QHE elements, in dc or ac. The predictions of the RK model were also verified experimentally [20, 21]. Other circuit models were developed in later years [21–28] and a general method of analysis based on the indefinite admittance matrix has been recently proposed [29].

The circuit models proposed in the literature are better suited for the symbolic analysis of networks of QHE elements; the matrix method can be employed both for symbolic and numerical analyses, typically with the aid of a computer algebra system or a numerical computing environment. Nowadays, however, the numerical analysis of electrical circuits is commonly carried out by employing analogue electronic circuit simulation tools, like SPICE [30] and its derivatives. SPICE can perform simulations of linear and non-linear networks in time and frequency domains, and noise analysis. Hence, it could be a very useful tool for the simulation of electrical networks containing QHE elements. Unfortunately, even though the above cited circuit models can be directly coded in SPICE, several issues arise when trying to run a simulation. The aim of this work is to address these issues and to present a working SPICE macro-model for the simulation of networks of QHE elements.

In section 2, we review the existing circuit models for QHE elements and present a new one. In section 3, we discuss the issues of SPICE modelling and provide a SPICE macro-model for an 8-terminal QHE element, which is based on the new circuit model. In section 4, three simulation examples are worked out in detail: (i) a dc analysis taking into account parasitic resistances; (ii) an ac analysis of a network containing a QHE element and a capacitor; (iii) a noise analysis. The simulation of example (i) is compared with an analytical result; those of (ii) and (iii) are compared with analytical and experimental results. Appendix A reports the full derivation of the new circuit model.
with the convention $e_{n+1} \equiv e_1$. Ideal QHE elements are memoryless, passive (actually, they dissipate power) and nonreciprocal\(^4\).

The sets of equations (1) and (2) are, indeed, a consequence of the QHE phenomenology, but can also be derived theoretically on the basis of the Landauer-Büttiker formalism (see, e.g. [39, section 16.3] and references therein).

A real QHE device operating at appropriate values of temperature and magnetic flux density can be modelled, with a certain approximation, either as an ideal cw or an ideal ccw element depending on the orientation of the magnetic field applied to the device and on the type of the majority charge carriers. The Hall resistance is quantized, $R_H = R_K/i$, where $R_K = h/e^2$ is the von Klitzing constant and $i$ is a positive integer called plateau index. In QHE devices realized with GaAs technology, the plateau index of interest for resistance metrology is $i = 2$, so $R_H = R_K/2$. The recommended value of the von Klitzing constant is $R_K = 25.812.807 \, 4434(84) \, \Omega$, which yields $R_H = R_K/2 = 12.906.403 \, 7217(42) \, \Omega$ [40]. The conventional value adopted internationally for resistance metrology is $R_{K-90} = 25.812.807 \, \Omega$, which yields $R_{H-90} = R_{K-90}/2 = 12.906.4035 \, \Omega$. In section 4, we will use this last value for the Hall resistance of the QHE elements in the example circuits.

Table 1 reports known circuit models listed in order of appearance in the literature: The first row reports the well-known circuit of Ricketts and Kemeny [20]; the last one, a new circuit model whose derivation is given in appendix A. For each referenced work, the second and third columns show the circuit models corresponding to the ideal cw and ccw 4-terminal QHE elements (the cited works might not provide both); in order to see how these models can be extended to elements with more than 4 terminals, the fourth column shows the outlines of the circuit models corresponding to a cw 6-terminal element. All circuit models contain only resistors (because of memorylessness and passivity) and controlled sources (because of nonreciprocity); none of them are related to the actual device physics.

We remark that the sets of equations (1) and (2) are all that is needed to analyze circuits containing ideal QHE elements [29], and that the external behavior of all the models listed in table 1 is exactly described by these equations. Nonetheless, a circuit model can be a useful basis for taking into account nonidealities (e.g. nonzero longitudinal resistances [24, 27], parasitic elements in ac regime [41]) or to develop a SPICE macro-model, as we will see in the next section.

3. SPICE modelling

In SPICE a circuit is represented by a list of statements (netlist), written according to a certain specific syntax [30, 42], which describes how the elements that compose the circuit are interconnected. A SPICE macro-model (or subcircuit netlist) is a netlist with a designated name that can be treated as any other

\(^4\)They are memoryless because the sets of equations (1) and (2) relate terminal voltages and currents at the same time instant. For definitions of passivity and reciprocity see e.g. [38, chapter 2].
Table 1. List of known circuit models for QHE elements.

| Reference       | 4-terminal cw model | 4-terminal ccw model | 6-terminal model (cw) |
|-----------------|---------------------|----------------------|-----------------------|
| Ricketts–Kemeny [20] | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |
| Hartland et al [22, 23] | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |
| Jeffery et al [21, 24] | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |
| Sosso–Capra [25, 26] | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |
| Schurr et al [27, 28] | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |
| This work (appendix A) | ![Circuit Diagram] | ![Circuit Diagram] | ![Circuit Diagram] |

Note: Column 1 reports references to the literature. Columns 2 and 3 report the corresponding circuit models for ideal cw and ccw 4-terminal QHE elements. Terminal voltages and currents are chosen according to figure 1: $e_{ij} = e_i - e_j; r = \frac{R_0}{2}$. The symbol → represents a controlled voltage source; the symbol ←, a controlled current source. In the last column, outlines of the circuit models for a cw 6-terminal element are sketched (see the given references for details).
Table 2. Properties satisfied by the circuit models listed in table 1.

| Model                          | Property (i) | Property (ii) | Property (iii) |
|--------------------------------|--------------|---------------|----------------|
| 1. Rickets–Kemeny              | •            | •             | •              |
| 2. Hartland et al              | •            | •             | p = 2r(j1 + j2)(j1 + j2) |
| 3. Jeffery et al               | •            | •             | •              |
| 4. Sosso–Capra                 | •            | •             | p = c1j2 + c2j1 + c3j4 + c4j3 |
| 5. Schurr et al*               | •            | •             | p = 2r(j2j3 + j3j4) |
| 6. This work                   | •            | •             | •              |

* Property (i) can be violated by model 5 if a voltage source is connected between terminals 1 and 2, or 3 and 4. Note: A bullet indicates that a certain property holds. For property (iii), the table reports the power delivered by the controlled sources (in the cw case) when this is not identically zero.

SPICE element to compose larger circuits in a hierarchical way. Circuits composed exclusively of resistors and controlled sources can be directly translated into SPICE macro-models. Given that all the models reported in table 1 are equivalent from the point of view of circuit theory, and that many more possessing this property can be conceived, how can we select one with the purpose of writing a SPICE macro-model?

In reality, SPICE modelling requires at least two additional properties, (i) and (ii) below, but one more, (iii), would be useful:

(i) **No loops of voltage sources.** SPICE requires a circuit that does not contain loops composed only of ideal voltage sources (independent or controlled). In fact, violating this requirement would lead to two scenarios: Either Kirchhoff’s voltage law would be violated or the current crossing the loop would be indeterminate and the circuit would not have a unique solution. In both cases, SPICE would not be able to solve the circuit.

(ii) **No cut sets of current sources.** This property is the dual of (i). If, in a circuit, there is a node where only current sources join, either Kirchhoff’s current law would be violated or the node potential would be indeterminate (floating node).

(iii) **Non-dissipative/generative sources.** As stated in section 2, QHE elements are passive elements that dissipate power. One property that could then be required by the circuit model is that all the power is dissipated in the resistors and that the overall power absorbed or delivered by the controlled sources is identically zero. This property would also allow SPICE to predict the thermal noise generated by the QHE elements correctly (see appendix A for more details on this). Even though this property is not strictly required, and even though SPICE has a few limitations in noise analysis (e.g. it is not able to analyse correlations directly and a workaround is needed [43]), it might be useful to have a circuit model which is as complete as possible.

Table 2 summarizes which properties hold for each circuit model. Properties (i) and (ii) can be verified by inspecting the circuit diagrams given in table 1; for models which do not satisfy property (iii), table 2 reports the power delivered by the controlled sources in the cw case.

Models 2, 4, and 5 cannot be easily modified to satisfy property (iii) and will not be considered further. Models 1 and 3 can be modified to satisfy property (i) either by adding a small series resistance to the loop of voltage sources [44] or by removing one of the controlled sources in the loop (this does not alter the model’s behavior). However, neither of the above two remedies is completely satisfactory: The former because the effect of the additional resistance on the simulation accuracy cannot be easily predicted in the case of many interconnected QHE elements; the latter because it destroys the symmetry of the model, making it more difficult to develop possible refinements to take account of nonidealities. For model 6, it is worth noting that the potential ε of the floating node can be fixed arbitrarily without altering the model’s behavior; this allows the satisfaction of property (ii) without the need for additional elements and without destroying the circuit symmetry. Therefore, model 6 was selected to implement a SPICE macro-model.

Listings 1 and 2 report, respectively, the SPICE macro-models for ideal cw and ccw 8-terminal QHE elements5. These macro-models are a direct translation of the general circuit models derived in appendix A. The actual number of terminals is nine: The ninth terminal, labelled C, corresponds to the node joining the controlled current sources and has to be connected to an arbitrary potential (e.g. ground). The default value of $R_{H}$ is 1 Ω, but other values can be assigned when calling the macro-model (see next section for an example).

Listing 1. SPICE macro-model for an ideal cw 8-terminal element.

```
.subckt qhe8cw 1 2 3 4 5 6 7 8 C params: RH=1
R1 1 2 [2*RH] R2 2 3 [2*RH] R3 3 4 [2*RH] R4 4 5 [2*RH] R5 5 6 [2*RH] R6 6 7 [2*RH] R7 7 8 [2*RH] R8 8 1 [2*RH] G1 1 C 2 8 [1/(2*RH)] G2 2 C 3 1 [1/(2*RH)] G3 3 C 4 2 [1/(2*RH)] G4 4 C 5 3 [1/(2*RH)] G5 5 C 6 4 [1/(2*RH)]
```

5 These macro-models should work out of the box, or with just slight modifications, with any modern SPICE simulator that accepts subcircuits with parameters. In this work, we have chiefly used LTspice IV from Linear Technology Corporation [45], but tests have been carried out also with the open source simulator Ngspice [46], with PSpice A/D from Cadence Design Systems and with TINA-TI from Texas Instruments. The authors can provide advice on adapting the circuit models and the simulations here described to other SPICE-based analogue circuit simulators.
Listing 2. SPICE macro-model for an ideal ccw 8-terminal element.

```
.subckt qhe8ccw 1 2 3 4 5 6 7 8 C params:
    RH = 1
R1 1 2 {2*RH}
R2 2 3 {2*RH}
R3 3 4 {2*RH}
R4 4 5 {2*RH}
R5 5 6 {2*RH}
R6 6 7 {2*RH}
R7 7 8 {2*RH}
R8 8 1 {2*RH}
G1 C 1 2 8 {1/(2*RH)}
G2 C 2 3 1 {1/(2*RH)}
G3 C 3 4 2 {1/(2*RH)}
G4 C 4 5 3 {1/(2*RH)}
G5 C 5 6 4 {1/(2*RH)}
G6 C 6 7 5 {1/(2*RH)}
G7 C 7 8 6 {1/(2*RH)}
G8 C 8 1 7 {1/(2*RH)}
.ends
```

4. Examples

4.1. Double-series interconnection of two QHE elements

Several QHE elements can be interconnected to obtain multiples, submultiples, or fractions of the Hall resistance. Multiple-series, parallel [47] and bridge connections [48] can be employed to reject the effect of the inevitable contact and wiring resistances. In this section, SPICE is used to analyze, in the dc regime, the effect of the parasitic resistances in a double-series connection; the result of the SPICE analysis is then compared with an analytical solution obtained with the technique described in [29].

Figure 2 shows the complete circuit diagram for the analysis of the double-series connection. For ease of analysis, only two parasitic resistances, \( r_1 \) and \( r_2 \), represent the wiring and contact resistances between the two devices; other contact resistances are neglected for simplicity.

```
Listing 3. SPICE netlist corresponding to the circuit of figure 2.

The included file qhe8cw.sub should contain the SPICE macro-model of listing 1. In this example we have set \( \epsilon_1 = 0.15r \) and \( \epsilon_2 = 0.35r \), where \( r \) is a dimensionless scaling parameter. The dot command .op declares that SPICE should perform a dc operating point analysis. The dot command .meas (LTspice IV specific) performs the calculation of the parameter \( \delta \) as defined by (3).

QHE double-series circuit simulation
* Includes the macro-model
  .inc qhe8cw.sub
* Definition of circuit parameters
  .param RH=12906.4035 {r=RH/2}
  .param t=0.01
  .param r1=0.15*t*r
  .param r2={0.35*t*r}
* Circuit netlist
  XU1 1A 2A 1A 4A 5A 6A 7A 8A 0 qhe8cw
  XU2 1B 2B 3B 4B 0 6B 0 8B 0 qhe8cw
  .param: RH={RH}
  .param: t={t}
  .param: r1={r1}
  .param: r2={r2}
  I0 0 1A 1
  r1 5A 1B {r1}
  r2 7A 3B {r2}
* Analysis directives
  .op
  .meas op delta param (V(1A)-2*RH)/(2*RH)
.ends
```

Figure 2. Circuit diagram for the dc analysis of a double-series connection. \( r_1 \) and \( r_2 \) represent the wiring and contact resistances between the two devices; other contact resistances are neglected for simplicity.

\[
\delta = \frac{R_S^{(2)} - 2R_{H1}}{2R_{H1}}, \quad (3)
\]

where \( 2R_{H1} \) is the resistance of the double-series when \( r_1 = r_2 = 0 \Omega \). At the third order in \( \epsilon_1 \) and \( \epsilon_2 \), the analytical solution yields\(^6\)

\[
\delta_{theo} = \frac{\epsilon_1\epsilon_2}{16} - \frac{\epsilon_1^2\epsilon_2 + \epsilon_1\epsilon_2^2}{64}. \quad (4)
\]

\(^6\)The authors can provide the Mathematica® notebook used to obtain the analytical solution.
floating point arithmetic and numerical algorithms. From (3), \( \delta \) is expected to scale as \( t^2 \).

Table 3 reports a comparison, for different values of \( t \), between the values of \( \delta^{\text{theo}} \) obtained from (3) and those of \( \delta^{\text{sim}} \) obtained from the SPICE simulation of listing 3. The third column \( \delta^{\text{sim}} \) (alt) reports the results obtained with the LTspice IV’s alternate solver, instead, the results obtained with the LTspice IV’s default solver; similar results are obtained with other SPICE-based simulators. The column \( \delta^{\text{sim}} \) (alt) reports, instead, the results obtained with the LTspice IV’s alternate solver with reduced round-off errors. In any case, whatever the simulator employed in this kind of analysis, we suggest writing all parasitic resistances as functions of a scaling parameter to keep round-off errors under control.

4.2. QHE gyrator

QHE elements, like their classical counterparts, can be employed as gyrators [49, 50]. At the angular frequency \( \omega \), the circuit of figure 3, which includes the load capacitor \( C_L \) with negative reactance \( X_L = -(\omega C_L)^{-1} \), realizes between terminal 1 and ground an impedance \( Z(\omega) = V/I_0 = R(\omega) + jX(\omega) \) having positive reactance, that is, \( X(\omega) > 0 \) for any \( \omega \). Therefore, at any given frequency, the circuit of figure 3 behaves like an \( RL \) two-terminal element (the values of \( R \) and \( L \) are frequency-dependent).

Network analysis yields

\[
Z(\omega) = \frac{R_0}{1 + \omega^2 C_L^2 R_0^2} + j\omega C_L R_0, \tag{5}
\]

for which \( X(\omega) = \text{Im } Z(\omega) > 0 \).

The behavior of such an unconventional QHE circuit was checked experimentally with an 8-terminal GaAs-AlGaAs Hall bar working at the temperature \( T = 1.6 \) K and plateau index \( i = 2 \). Terminals 3 and 7 were connected to a variable capacitance box; terminal pairs (1, 2) and (5, 6) were connected in a double-series configuration to an LCR meter (Agilent mod. 4284A) which performed the impedance measurements. The measurement frequency \( f = \omega/2\pi \) was chosen at 1233 Hz, so that the reactance of a 10 nF capacitor is \( X_L \approx -R_H \).

However, the above described experimental set-up is not accurately modelled by the circuit of figure 3 or by the corresponding equation (5), because in figure 3 the cable capacitances and the real operation of the LCR meter are not taken into account (see [7] for an example about the effect of the cable capacitances in ac QHE measurements). The experimental set-up is instead better modelled by the circuit of figure 4, where \( C_{p1} \) and \( C_{p2} \) represent the cable capacitances, and where the transimpedance \( Z_{\text{meas}} = V/I_0 \) represents the impedance actually measured by the LCR meter, that is, the ratio of the meter’s high-side voltage to the meter’s low-side current.

Even though the circuit in figure 4 can be solved analytically, this kind of analysis is not straightforward, and SPICE simulation can provide a quicker response. Listing 4 reports the SPICE netlist for the ac analysis of the circuit in figure 4, for different values of \( C_L \) at the fixed frequency of 1233 Hz. The given values for \( C_L \) are those used in the experiment: 0 nF, 1 nF, 3 nF, 7 nF, 10 nF, 15 nF, 20 nF, and 30 nF. The values of the parasitic capacitances reported in the listing, \( C_{p1} = C_{p2} = 300 \) pF, are just rough estimates and not the result of a measurement.
Listing 4. SPICE netlist corresponding to the circuit of figure 4. The .step directive is used to automatically run simulations with different values of the load capacitance $C_L$ at the fixed frequency of 1233 Hz; the .ac directive declares that SPICE should perform an ac sweep analysis, here of just one frequency point; the .probe directive is used to save node voltages and branch currents for further analysis, plots etc.

QHE gyrator simulation
* Includes the macro-model
  .inc qhe8cw.sub
* Definition of circuit parameters
  .param RH=12906.4035
  .step param CL list 0 1n 3n 7n 10n 15n 20n 30n
* Circuit netlist
  XU1 1 3 4 5 5 7 8 0 qhe8cw+params: RH={RH}
  I0 0 1 ac 1
  CL 3 7 {CL}
  Cp1 7 0 300p
  Cp2 3 0 300p
  VG 5 0 0
* Analysis directives
  .ac lin 1 1233 1233
  .probe V(1) I(VG)
.end

The results are reported in figure 5, where the measured values of $R(C_L)$ and $X(C_L)$ at the frequency of 1233 Hz are compared with the values obtained from the SPICE simulation of listing 4 and with those obtained from (5). The agreement between the measured values and those obtained from the SPICE simulation is within 1%. The gyrator circuit was implemented and simulated for didactic purposes. The improved agreement between the SPICE simulation of the circuit of figure 4 and the experimental data does not mean that this circuit gives a complete description of the experiment. Further model refinements can be easily implemented in the SPICE simulation.

4.3. Noise analysis

Noise is a fundamental limit to the resolution of a measuring system. Noise analysis is therefore a useful tool to predict the resolution achievable in a measurement. SPICE has built-in noise models for resistors and semiconductor devices which allow a complete noise analysis of circuits, taking into account all the main noise sources. In the case of circuits containing QHE elements, the noise analysis can include the thermal noise generated by these elements and the noise generated by possible auxiliary elements, such as voltage or current sources, detectors etc.

In this section we present the noise analysis of the circuit of figure 6, for which the thermal noise properties have been investigated both theoretically and experimentally in [51]. The one-sided cross-spectral density function $S_{ab}(f)$ between the voltages $v_a$ and $v_b$ due to thermal noise is $S_{ab}(f) = 2k_B T R_{th}$ [51], where $k_B$ is the Boltzmann’s constant and $T$ is the thermodynamic temperature of the QHE element.

Figure 7 shows the circuit needed to evaluate $S_{ab}(f)$ with SPICE. Since SPICE cannot evaluate directly cross-spectral density functions, but only (auto-) spectral density functions, a trick should be employed [43]. Four voltage controlled current sources, with transconductances of 1 S, generate two currents, one proportional to the sum of $v_a$ and $v_b$, the other proportional to their difference; it can be shown that
$$S_{ab}(f) = \frac{1}{4}[S_a(f) - S_b(f)],$$

where $S_a(f)$ is the spectral density function of $v_a + v_b$ and $S_b(f)$ is the spectral density function of $v_a - v_b$. Both $S_a(f)$ and $S_b(f)$ can be directly evaluated by SPICE. The additional, noiseless, voltage source $V_0$ is actually ineffective in this analysis, but it is required by SPICE to have a fictitious input.

Listing 5 reports the SPICE netlist for the noise analysis described above. The subcircuit `probe_correlation` is used to calculate the sum and the difference of $v_a$ and $v_b$ in two successive steps: When the parameter $gb$ is set to 1 the output node (OUT) of `probe_correlation` yields the sum of the two voltages; when $gb$ is set to -1, the difference is obtained instead. The simulation temperature is set to 2 K. The calculation of $S_{ab}(f)$ or of the ratio $S_{ab}(f)/(4k_B R_T T)$ can be done automatically by the SPICE graphical post-processor, but we will not dwell on the details here. The result of the simulation is $S_{ab}(f)/(4k_B R_T T) = 0.500 000 46$ (independent of frequency because there are no reactive elements), with a relative discrepancy of about $10^{-6}$ from the theoretical value of 0.5.

In this paper we reviewed the modelling of the ideal quantum Hall effect element as a $n$-terminal electrical network. The unique set of network equations can be represented with different equivalent circuit models published in the literature. An analysis of these models shows that they are unsuitable to be directly coded in circuit simulation software such as SPICE, because they generate singular representations or because of errors in the noise simulation. A new QHE circuit model, specifically designed to be implemented in SPICE, is proposed. This paper provides several examples of electrical circuits including QHE elements, of their coding in SPICE, and hints for proper simulation runs. For each case, the simulation outcome is compared with the results from analytical modelling and also, in two cases, with experimental data.

### Acknowledgments

The authors are grateful to F J Ahlers and J Schurr from the Physikalisch-Technische Bundesanstalt (PTB), Braunschweig (Germany), for their collaboration in the experiment described in section 4.2, which was carried out at the Electrical Quantum Metrology Department of the PTB.

The authors would also like to thank H Sennewald and the LTspice IV users’ group [52] for their useful suggestions on the usage of LTspice IV.

### Appendix A. Yet another model

#### A.1 Model derivation

In this section we derive model 6 of table 1 for an ideal $n$-terminal QHE element on the basis of the analysis method described in [29] and briefly reviewed below. In the derivation, a sinusoidal regime is assumed: Voltages and currents are to be understood as voltage and current phasors and are denoted with capital letters. Labelling of terminals and reference directions are again that of figure 1.

The sets of equations (1) and (2) can be rewritten in matrix form as [29]

$$J = \tilde{Y}E,$$  \hspace{1cm} (A.1)

where $J = (J_1, ..., J_n)^T$ (the superscript T denotes transposition) and $E = (E_1, ..., E_n)^T$ are column vectors, and where the $n \times n$ indefinite admittance matrix (see [53] for a definition) $\tilde{Y}$ is either

$$\tilde{Y}_{icw} = \frac{1}{R_H} \begin{pmatrix} 1 & 0 & \cdots & 0 & -1 \\ -1 & 1 & 0 & \cdots & 0 \\ \vdots & \vdots & \ddots & \ddots & \vdots \\ 0 & 0 & \cdots & -1 & 1 \end{pmatrix},$$  \hspace{1cm} (A.2)

in the case of an ideal cw $n$-terminal element, or

$$\tilde{Y}_{icw} = \frac{1}{R_H} \begin{pmatrix} 1 & -1 & 0 & \cdots & 0 \\ 0 & 1 & -1 & \cdots & 0 \\ \vdots & \vdots & \ddots & \ddots & \vdots \\ 0 & 0 & \cdots & 0 & 1 -1 \\ -1 & 0 & \cdots & 0 & 1 \end{pmatrix},$$  \hspace{1cm} (A.3)

in the case of an ideal ccw $n$-terminal element.

The two above matrices are real matrices. A basic theorem of algebra [54, chapter 1] states that every matrix can be decomposed into the sum of a symmetric part and an antisymmetric one:

$$\tilde{Y} = \tilde{Y}^S + \tilde{Y}^A,$$  \hspace{1cm} (A.4)
with
\[
\tilde{Y}_i^S = \frac{1}{2}(\tilde{Y}_i + \tilde{Y}_i^T) \quad \text{and} \quad \tilde{Y}_i^A = \frac{1}{2}(\tilde{Y}_i - \tilde{Y}_i^T). \quad (A.5)
\]

Since, by definition, \((\tilde{Y}_i^S)_{ij} = (\tilde{Y}_i^A)_{ij}\), \(\tilde{Y}_i^S\) is associated with a reciprocal network [55, chapter 16]; in addition, this associated network can be realized with just resistors because \(\tilde{Y}_i^S\) is also real. The antisymmetric part \(\tilde{Y}_i^A\) is instead associated with a nonreciprocal network that can be realized with voltage controlled current sources. For the matrices in (A.2) and (A.3) the symmetric and antisymmetric parts are
\[
\tilde{Y}_{\text{Lcw}}^S = \tilde{Y}_{\text{Lccw}}^S = \frac{1}{R_{\text{H}}} \begin{pmatrix}
1 -\frac{1}{2} & 0 & \cdots & -\frac{1}{2} \\
-\frac{1}{2} & 1 -\frac{1}{2} & 0 & \cdots \\
\vdots & \vdots & \ddots & \vdots \\
0 & \cdots & -\frac{1}{2} & 1 -\frac{1}{2} \\
-\frac{1}{2} & 0 & \cdots & -\frac{1}{2} & 1
\end{pmatrix},
\]
\[
(A.6)
\]
\[
\tilde{Y}_{\text{Lcw}}^A = \frac{1}{R_{\text{H}}} \begin{pmatrix}
0 & 1 & 0 & \cdots & -\frac{1}{2} \\
-\frac{1}{2} & 0 & 1 & \cdots & 0 \\
\vdots & \vdots & \ddots & \vdots & \vdots \\
0 & \cdots & -\frac{1}{2} & 0 & 1 \\
1 & 0 & \cdots & -\frac{1}{2} & 0
\end{pmatrix},
\]
\[
(A.7)
\]
\[
\tilde{Y}_{\text{Lccw}}^A = -\tilde{Y}_{\text{Lccw}}^A = \frac{1}{R_{\text{H}}} \begin{pmatrix}
0 & -\frac{1}{2} & 0 & \cdots & \frac{1}{2} \\
\frac{1}{2} & 0 & -\frac{1}{2} & \cdots & 0 \\
\vdots & \vdots & \ddots & \vdots & \vdots \\
0 & \cdots & 1 & 0 & -\frac{1}{2} \\
-\frac{1}{2} & 0 & \cdots & 1 & 0
\end{pmatrix}.
\]
\[
(A.8)
\]

From (A.1), taking into account (A.4), we obtain that the terminal currents can be decomposed into the sum of two contributions, one associated with \(\tilde{Y}_i^S\) and one with \(\tilde{Y}_i^A\):
\[
J = J^S + J^A,
\]
\[
(A.9)
\]
\[
\text{with}
J^S = \tilde{Y}_i^S E \quad \text{and} \quad J^A = \tilde{Y}_i^A E.
\]
\[
(A.10)
\]

This means that \(\tilde{Y}_i^S\) and \(\tilde{Y}_i^A\) can be considered as associated with two electrical networks connected in parallel.

Let \(J^S = (J_{1}^S, \ldots, J_{n}^S)^T\): from (A.10) and (A.6) the current at the \(m\)th terminal is \((m = 1, \ldots, n; E_0 \equiv E_n \text{ and } E_{n+1} \equiv E_1)\)
\[
J_m^S = \frac{E_m - E_{m-1}/2 + E_{m+1}/2}{2R_{\text{H}}}, \quad R_{\text{H}},
\]
\[
(A.11)
\]
\[
= \frac{E_m - E_{m-1}}{2R_{\text{H}}} + \frac{E_{m} - E_{m+1}}{2R_{\text{H}}}. \quad (A.12)
\]

The last equation above corresponds to a network where each terminal is connected to its adjacent terminals through a resistor with resistance \(2R_{\text{H}}\), as shown in figure A1. This can also be obtained directly from the properties of the indefinite admittance matrices [56, chapter 2, section 2.2].

Now, let \(J^A = (J_1^A, \ldots, J_n^A)^T\): from (A.10), (A.7) and (A.8), the current at the \(m\)th terminal is
\[
J_m^A = \pm \frac{E_{m+1} - E_{m-1}}{2R_{\text{H}}}, \quad (A.13)
\]

where the plus sign holds for a cw element and the minus for a ccw one. This equation can be obtained with a voltage controlled current source, driven by the voltage difference \(E_{m+1} - E_{m-1}\), which draws from the \(m\)th terminal the current \(\pm(E_{m+1} - E_{m-1})/(2R_{\text{H}})\) injecting it into an additional internal node \(E\) (figure A2).

Finally, combining in parallel the two networks, one obtains the circuit model of figure A3. This circuit model obviously satisfies property (i); property (ii) can be satisfied by fixing the potential \(E\) of the central node of figure A3 to an arbitrary value.

A.2. Power

To prove that the circuit model of figure A3 satisfies property (iii), let us recall that, in sinusoidal regime, the average power entering an \(n\)-terminal element is given by [53, 55]
\[
P_{av} = \frac{1}{2} \text{Re} \left( \sum_{m=1}^{n} J_m^* E_m \right) = \frac{1}{2} \text{Re}(J^*E), \quad (A.14)
\]
\[
= \frac{1}{4}(J^*E + E^*J), \quad (A.15)
\]
where the operator Re denotes the real part and the asterisk denotes the conjugate transpose. Substituting (A.1) in (A.15), and taking into account that \( \bar{Y} \) is real, yields

\[
P_{av} = \frac{1}{4} E^* (\bar{Y}^S_{pq} + \bar{Y}_{pq}) E ,
\]

(A.16)

\[
= \frac{1}{4} E^* (\bar{Y}^T_{pq} + \bar{Y}_{pq}) E ,
\]

(A.17)

\[
= \frac{1}{2} E^* \bar{Y}^S_{pq} E .
\]

(A.18)

Equation (A.18) says that the power dissipated in the ideal QHE element is related only to the symmetric part of the corresponding indefinite admittance matrix. Thus, no power is delivered or absorbed by the circuit associated with the antisymmetric part, and property (iii) is satisfied.

**A.3. Noise modelling**

When SPICE performs the noise analysis of a circuit, it automatically assigns noise sources to certain circuit elements in accordance with known noise models. The insertion of arbitrary noise sources is not permitted. In the case of the macro-models of listings 1 and 2, SPICE will assign noise sources to the resistors only: The controlled sources are considered noiseless. To each resistor, SPICE assign a noise source which generates the corresponding thermal (equilibrium) noise at the simulation temperature (specified by the SPICE parameter TNOM, 27 °C by default). Therefore, to prove that the satisfaction of property (iii) allows SPICE to predict the equilibrium noise generated by the QHE elements correctly, we need only to prove that the equilibrium noise of a QHE element coincides with that of the resistive network associated with \( \bar{Y}^S_{pq} \).

The equilibrium noise properties of a general passive multiterminal network were derived by Twiss [57] from thermodynamic considerations. Later, Büttiker derived an equivalent result for multiterminal conductors from a quantum scattering theory [58, 59]\(^8\). These results were verified experimentally by the authors in [51].

For the present purposes, the result given by Twiss is in a more convenient form. This result can be stated as follows [57]: Consider a passive \( n \)-terminal element and assume that all the terminals are grounded and one, say terminal \( n \) without loss of generality, is considered as reference; then, the one-sided cross-spectral density function \([62]\) \( S_{j_j}(f) \) between the noise currents \( j_p \) and \( j_q \) at the terminals \( p = 1, \ldots, n - 1 \) and \( q = 1, \ldots, n - 1 \) is given by

\[
S_{j_j}(f) = 2 k_B T (Y_{pq} + Y^*_{qp}) ,
\]

(A.19)

where \( k_B \) is the Boltzmann’s constant, \( T \) is the thermodynamic temperature, and \( Y = (Y_{pq}) \) is the short-circuit \((n - 1) \times (n - 1)\) admittance matrix obtained by considering the \( n \)-terminal element as an \((n - 1)\)-port with ports defined between the terminals \( 1, \ldots, n - 1 \) and the reference node \( n \).

The short-circuit admittance matrix can be obtained by the indefinite admittance matrix associated with the \( n \)-terminal element by deleting the \( n \)th row and column [53]; then, for and ideal QHE element, \( Y_{pq} = (\bar{Y})_{pq} \) and

\[
S_{j_j}(f) = 2 k_B T (\bar{Y}^S_{pq} + \bar{Y}^S_{qp}) ,
\]

(A.20)

\[
= 2 k_B T [(\bar{Y})_{pq} + (\bar{Y})_{qp}^*] ,
\]

(A.21)

\[
= 2 k_B T [(\bar{Y})_{pq} + (\bar{Y})_{qp}] ,
\]

(A.22)

\[
= 4 k_B T (\bar{Y}^S_{pq}) ,
\]

(A.23)

\[
= 2 k_B T [(\bar{Y}^S_{pq} + (\bar{Y}^S_{qp})^*] .
\]

(A.24)

Hence, and taking into account that the reference element is arbitrary, the equilibrium noise of a QHE element coincides with that of the resistive network associated with \( \bar{Y}^S \).

\(^8\)The two approaches are complementary: The derivation of Twiss can be considered analogous to that of Nyquist [60] for the Johnson–Nyquist noise generated by a resistor; the derivation of Büttiker can be considered analogous to that of Landauer [61].
References

[1] Delahaye F and Jeckelmann B 2003 Revised technical guidelines for reliable dc measurements of the quantized Hall resistance Metrologia 40 217

[2] Jeckelmann B and Jeanneret B 2003 The quantum Hall effect as an electrical resistance standard Meas. Sci. Technol. 14 1229

[3] Poirier W, Schopfer F, Guignard J, Thévenot O and Gournay P 2011 Application of the quantum Hall effect to resistance metrology C. R. Phys. 12 347–68

[4] Overney F, Jeanneret B, Jeckelmann B, Wood B M and Schurr J 2006 The quantized Hall resistance: towards a primary standard of impedance Metrologia 43 409

[5] Schurr J, Ahlers F J, Hein G and Pierz K 2007 The ac quantum Hall effect as a primary standard of impedance Metrologia 44 15–23

[6] Ahlers F J, Jeanneret B, Overney F, Schurr J and Wood B M 2009 Compendium for precise ac measurements of the quantum Hall resistance Metrologia 46 R1–R12

[7] Hernandez C, Consejo C, Degiovanni P and Chauvet C 2014 Admittance of multiterminal quantum Hall hall conductors at kilohertz frequencies J. Appl. Phys. 115 123710

[8] Piquemal F P M, Blanchet J, Géneves G and André J-P 1999 A first attempt to realize (multiple-QHE devices)-series array resistance standards IEEE Trans. Instrum. Meas. 48 296–300

[9] Poirier W, Bounouh A, Hayashi H, Fhima H, Piquemal F, Géneves G and André J-P 2002 Rg/100 and Rg/200 quantum Hall array resistance standards J. Appl. Phys. 92 2844–54

[10] Bounouh A, Poirier W, Piquemal F, Géneves G and André J-P 2003 Quantum resistance standards with double 2DEG IEEE Trans. Instrum. Meas. 52 555–8

[11] Poirier W, Bounouh A, Piquemal F and André J P 2004 A new generation of QHARS: discussion about the technical criteria for quantization Metrologia 41 285

[12] Hein G, Schumacher B and Ahlers F-J 2004 Preparation of quantum Hall effect device arrays 2004 Conf. on Precision Electromagnetic Measurements Digest (London, UK) pp 273–4

[13] Oe T, Kaneko N, Urano C, Itatani T, Ishii H and Kiryu S 2008 Development of quantum Hall array resistance standards at NMJ CPEM 2008. Conf. on Precision Electromagnetic Measurements Digest (Broomfield, CO, USA, 2008) pp 20–1

[14] Oe T, Matsuhiro K, Itatani T, Gorwałdak S, Kiryu S and Kaneko N 2011 Development of quantum Hall array resistance standards at NMJII IEEE Trans. Instrum. Meas. 60 2590–5

[15] Woszczyńska M, Friedemann M, Dziomba T, Weimann Th and Ahlers F J 2011 Graphene p–n junction arrays as quantum-Hall resistance standards Appl. Phys. Lett. 99 022112

[16] Oe T, Matsuhiro K, Itatani T, Gorwałdak S, Kiryu S and Kaneko N 2013 New design of quantized Hall resistance array device IEEE Trans. Instrum. Meas. 62 1755–9

[17] Domae A, Oe T, Matsuhiro K, Kiryu S and Kaneko N 1–2012 Development of a one-chip quantized hall resistance voltage divider Meas. Sci. Technol. 23 124008

[18] Schopfer F and Poirier W 2007 Testing universality of the quantum Hall effect by means of the wheatstone bridge J. Appl. Phys. 102 054903

[19] Schurr J, Bürkel V and Kibble B P 2009 Realizing the farad from two ac quantum Hall resistances Metrologia 46 619

[20] Ricketts B W and Kemeny P C 1988 Quantum Hall effect devices as circuit elements J. Phys. D: Appl. Phys. 21 483

[21] Jeffery A, Elmiquisite R E and Cage M E 1995 Precision tests of a quantum Hall effect device dc equivalent circuit using double-series and triple-series connections J. Res. Natl Inst. Stand. 100 677–85

[22] Hartland A, Kibble B P, Rodgers P J and Bohacek J 1995 Ac measurements of the quantized hall resistance IEEE Trans. Instrum. Meas. 44 245–8

[23] Chua S W, Hartland A and Kibble B P 1999 Measurement of the ac quantized hall resistance IEEE Trans. Instrum. Meas. 48 309–13

[24] Cage M E, Jeffery A, Elmiquisite R E and Lee K C 1998 Calculating the effects of longitudinal resistance in multi-series-connected quantum Hall effect devices J. Res. Natl Inst. Stand. 103 561–92

[25] Sossio A and Capra P P 1999 Electronic simulation of a multiterminal quantum hall effect device Rev. Sci. Instrum. 70 2082–6

[26] Sossio A 2001 Derivation of an electronic equivalent of qhe devices IEEE Trans. Instrum. Meas. 50 223–6

[27] Schurr J, Ahlers F-J, Hein G, Melcher J, Pierz K, Overney F and Wood B M 2006 Ac longitudinal and contact resistance measurements of quantum hall devices Metrologia 43 163

[28] Schurr J, Ahlers F and Pierz K 2014 Magnetocapacitance and loss factor of GaAs quantum Hall effect devices Metrologia 51 235–42

[29] Ortolano M and Callegaro L 2012 Matrix method analysis of quantum Hall effect device connections Metrologia 49 88

[30] The SPICE page Online http://bwrcs.eecs.berkeley.edu/Classes/EB002/SPICE/

[31] Garg J M and Carlh J 1965 Theory network of semiconductor hall-plate circuits IEEE Trans. Circuit Theory pp 59–73

[32] Arnold E 1982 Computer simulation of conductivity and Hall effect in inhomogeneous invasion layers Surf. Sci. 113 239–43

[33] Popovic R S 1985 Numerical analysis of MOS magnetic field sensors Solid-State Electron. 28 711–6

[34] Salim A, Manku T, Nathan A and Kung W 1992 Modeling of magnetic-field sensitive devices using circuit simulation tools Technical Digest of the 5th IEEE Solid-State Sensor and Actuator Workshop pp 94–7

[35] Salim A, Manku T and Nathan A 1995 Modeling of magnetic field sensitivity of bipolar magentotransistors using HSPICE IEEE Trans. Comput.-Aided Des. Integr. Circuits Syst. 14 464–9

[36] Chua L O 1980 Device modeling via basic nonlinear circuit elements IEEE Trans. Circuits Syst. 27 1014–44

[37] Willems J C 2010 Terminals and ports IEEE Circuits Syst. Mag. 10 8–16

[38] Belevitch V 1968 Classical Network Theory (San Francisco, CA: Holden-Day)

[39] Ihn T 2010 Semiconductor Nanostructures: Quantum States and Electronic Transport (Oxford: Oxford University Press)

[40] Mohr P J, Taylor B N and Newell D B 2012 CODATA recommended values of the fundamental physical constants: 2010 Rev. Mod. Phys. 84 1527–605

[41] Cage M E, Jeffery A and Matthews J 1999 Equivalent electrical circuit representations of ac quantized hall resistance standards J. Res. Natl. Inst. Stand. 104 529–56

[42] Steer M B 2007 SPICE: User’s guide and reference Online www.freedo.org/doc/SPICE/spice.pdf

[43] McAndrew C, Coram G, Blumsoy P and Pil loud O 2005 Correlated noise modeling and simulation Technical Proc. of the 2005 Workshop on Compact Modeling (Anahiem, CA) pp 40–5

[44] Rashid M H and Rashid H M 2006 SPICE for Power Electronics and Electric Power 2nd edn (Boca Raton, FL: CRC Press)

[45] Linear Technology LTspice IV www.linear.com/designtools/software/LTspice

[46] Ngspice home page http://ngspice.sourceforge.net/
[47] Delahaye F 1993 Series and parallel connection of multiterminal quantum hall-effect devices J. Appl. Phys. 73 7914–20
[48] Ortolano M, Abrate M and Callegaro L 2015 On the synthesis of quantum Hall array resistance standards Metrologia 52 31
[49] Tellegen B D H 1948 The gyrator, a new electric network element Philips Res. Rep. 3 81–101
[50] Viola G and DiVincenzo D P 2014 Hall effect gyrators and circulators Phys. Rev. 4 021019
[51] Callegaro L, Ortolano M and Schurr J 2013 Equilibrium noise correlations in quantum Hall effect devices as a test of a formula by büttiker Europhys. Lett. 101 50003
[52] LTspice IV users’ group (http://groups.yahoo.com/neo/groups/LTspice/info)
[53] Shekel J 1954 Voltage reference node—its transformations in nodal analysis Wirel. Eng. 31 6–10
[54] Roman S 2010 Advanced Linear Algebra 3rd edn (Berlin: Springer)
[55] Desoer C A and Kuh E S 1969 Basic Circuit Theory (Singapore: McGraw–Hill)
[56] Chen W K 1991 Active Network Analysis (Singapore: World Scientific)
[57] Twiss R Q 1955 Nyquist’s and thevenin’s theorems generalized for nonreciprocal linear networks J. Appl. Phys. 26 599–602
[58] Büttiker M 1990 Scattering theory of thermal and excess noise in open conductors Phys. Rev. Lett. 65 2901–4
[59] Büttiker M 1992 Scattering theory of current and intensity noise correlations in conductors and wave guides Phys. Rev. B 46 12485–507
[60] Nyquist H 1928 Thermal agitation of electric charge in conductors Phys. Rev. 32 110–3
[61] Landauer R 1989 Johnson–Nyquist noise derived from quantum mechanical transmission Physica D 38 226–9
[62] Priestley M B 1981 Spectral Analysis and Time Series vol 2 (New York: Academic)