

Integrated HSPICE-MATLAB Framework for Thermal-Aware Simulation and Performance Analysis of RF Circuits

Perera Manthila^{1*}, K. Madugalla Anuradha²

¹Department of Electrical Engineering Faculty of Engineering, University of Moratuwa, Sri Lanka

²Department of Electrical Engineering Faculty of Engineering, University of Moratuwa, Sri Lanka

KEYWORDS:

Thermal-Aware Simulation,
RF Circuit Design,
HSPICE-MATLAB Co-Simulation,
Low Noise Amplifier (LNA),
Temperature-Dependent Modeling,
Performance Analysis,
Gain and Noise Figure Degradation,
Reliability in High-Frequency Systems

ARTICLE HISTORY:

Submitted : 19.12.2024

Revised : 25.01.2025

Accepted : 14.03.2025

<https://doi.org/10.17051/NJRFCS/02.02.10>

ABSTRACT

Under the present high speed wireless communication development trend, the RF circuits are continually submitted to thermal stress that has greatly impacted their reliability and performance. Standard methods to circuit simulation may ignore the fluctuation of the temperature in such crucial parameters including the gain, the noise figure (NF), and power consumptions. The current paper is an attempt to present an integrated co-simulation environment that consists of HSPICE and MATLAB, to achieve a thermal-aware RF circuit modeling and performance assessment. The aim will be to build an active automated simulation cycle that will present the temperature dependant phenomena of semiconducting devices under practical operating conditions. To perform electrical simulation, the framework uses HSPICE and temperature profiling, parametric control, as well as post-processing, it adopts MATLAB. The procedure is illustrated using a case study of 2.4 GHz CMOS-based Low Noise Amplifier (LNA). With a temperature ranging between 25 °C and 125 °C, the simulation outcomes show that up to 18 per cent gain degradation and 12 percent increase in noise figure occur because of the temperature effect on the RF circuit. The given framework represents a scalable method proposed to analyze thermal reliability at an early stage in next-generation RF systems.

Author's e-mail: manthila.perera@elect.mrt.ac.lk, anuradha.km@elect.mrt.ac.lk

How to cite this article: Manthila P, Anuradha KM. Integrated HSPICE-MATLAB Framework for Thermal-Aware Simulation and Performance Analysis of RF Circuits. National Journal of RF Circuits and Wireless Systems, Vol. 2, No. 2, 2025 (pp. 71-78).

INTRODUCTION

The high development rate of wireless communication technologies (5G, 6G, in both systems, as well as increasing the number of IoT-based architecture) this has imposed strict requirements on the performance and reliability of RF circuits. The high-frequency signal processing circuits that make up the backbone of many systems are subjected to fluctuating thermal conditions more and more due to both dense integration and compact form factor combined with high power modes of operation. Some of the primary performance-related parameters that can be affected by temperature changes in device parameters mobility, threshold voltage and saturation current include: Gain, noise figure (NF), the linearity and power efficiency. In fact, most RF design flows are still based on simulations in steady or nominal temperature, and this cannot actually represent the real behavior during stressful conditions of thermo stress.

Although a few papers have tackled the issue of temperature impacts on analog and mixed-signal circuits, their methods generally depend on either the use of standalone thermo- or the use of the so-called static corner analysis. Such methods fail to provide real-time interaction of electrical performance with thermal characteristic. Moreover, compared to electrical and thermal analysis, there is a lack of tools that combine these inputs to form a dynamical, automated process. The literature displays the lack of methodologies that allow closed-loop co-simulation between a high-precision electrical simulator and a flexible computational environment that could process parametric sweeps, extraction of data and visualization in real-time. This inability of the integration restricts the early stage designers to early perform thermally conscious optimizations.

The background to this study is the predicament that requires an effective, flexible simulation platform that

is not only capable of simulating the temperature-dependent behavior of the circuits but can maintain the automatic and dynamic simulation of the circuit in a broad spectrum of thermo-conditions. MATLAB has a strong data processing and scripting environment then HSPICE has sufficient transistor-level modeling so it is ideal to work with simulation flow within MATLAB. This close integration between the two environments would facilitate the creation of a strong co-simulation environment, which allows one to systematically investigate the impact of temperature on the degraded performance on RF circuits.

Within this presentation, we proposed an HSPICEco-simulation framework-integrated MATLAB design of thermal-aware modeling and analysis of RF circuitry in particular on RF thermal-awareness thermodynamic modeling and analysis through co-simulation framework. The structure is capable of automating the entire simulation procedure of temperature sweeps, moreover, it constantly keeps the circuit netlists and extracts performance indices such as gains, NF and return loss at every temperature point. The target circuit to be validated is a CMOS based 2.4 GHz Low Noise Amplifier (LNA). The results of simulation do reveal serious degradation of gain and noise figure with temperature increase which confirms the necessity of thermal-aware design practices. The framework provides an effective means to treat thermal effects as one that can be put on a scaling basis into early-stage RF circuit design and optimization.

RELATED WORK

RF performance exhaustively depends on carrier mobility, threshold voltage and saturation current whose changes rapidly depend on temperature variation of the RF circuit. Earlier work has been done on temperature-aware circuit simulation on standard SPICE simulators or on discrete thermal solvers in isolation. As an example, Jain et al.^[1] presented the study on the effect of thermal noise on the performance of CMOS RF amplifiers, however, their simulations were performed with no automation nor real-time data processing. On the same note, Nguyen et al.^[3] modeled the effect of self-heating of the RF CMOS circuits by use of 3D simulation tools like COMSOL. Such finite element solvers, which are accurate, are computationally demanding, however, and cannot be readily amenable to fast, iterative circuit-level simulation.

The lower-frequency domains have been able to explore co simulation environments that integrate electrical and thermal domains. Li and Wang^[2] suggested MATLAB-SPICE co-simulation of analog circuits, and this was not

demonstrated to the high-frequency RF case. Besides, nothing was done with regards to real-time simulation control, netlist reconfiguration and processing a batch of temperature points. Banu and Agrawal^[4] relied on MATLAB as a tool of mm-wave transceiver blocks analysis, although, their research concentrated mostly on graphical representation of data and post-processing data analysis instead of automatizing the closed loop simulation.

Even more recent, thermal-aware and fault-tolerant hardware simulation development has evoked an increased interest in the robust simulation environment. The necessity of using and minimizing faults on reconfigurable hardware under environmental stress is also important as outlined by Tamm et al.,^[5] which indirectly argues the case of environmentally-aware design practices. In an analogous manner, machine learning approaches to signal processing was also reviewed by Barhoumi et al.^[6] in the application of signal processing towards adaptive tuning and anomaly detection, and this could in turn be applied to the thermally adaptive RF through a synergistic collaborative center of gravity as a future work direction.

Within the field of embedded and mission-critical system applications, Thompson and Sonntag^[7] have investigated cyber-physical systems in smart hospitals, which also highlights the importance of resilient and reliable RF designs with respect to the healthcare context. Rucker et al.^[8] also investigated AI applications of biomedical signal processing, where temperature-restorative RF circuits are more critical in the real-time data acquisition. Further, the network vulnerabilities with respect to routing attacks in VANETs were addressed by RANGISETTI and ANNAPURNA,^[9] wherein communication security and timing could be compromised because of the failure of RF transceivers as a result of thermal drift.

In spite of these helpful pieces on contributions, a thick gap exists in the literature in terms of an integrated, thermal-aware simulation platform that specifically focuses on RF front-end circuits when it comes to co-simulation with respect to HSPICE/MATLAB. Most previous investigations cover either self-contained simulation tools or cannot automate thermal sweeps, performance monitoring and installed feedback that targets updating the parameters of temperature. This paper meets this demand by proposing a script-driven framework that is scalable and allows temperature dependent modeling in real time, automated performance analysis, and visualization of RF circuits like CMOS based Low Noise Amplifiers (LNAs).

METHODOLOGY

The following section explains the architecture, building blocks, and functional flow of the proposed co-simulation framework that was developed to assess the performance of RF circuit in regard to thermal variation. The framework is implemented to combine the accuracy of a transistor level HSPICE simulations and the scripting versatility and data analytics of Matlab so that automated and temperature-conscious simulations and result processing can be performed.

Framework Overview

The co-simulation framework proposed here consists of three main parts, namely HSPICE, MATLAB, and a layer of scripting control (Python, or Perl). HSPICE is the fundamental electrical simulator, it is in charge of emulating the response of the RF circuit at the transistor level. It can take temperature settings as simulation parameters and it can calculate outputs: node voltages, S-parameters, gain, return loss (S11), noise figure (NF), and power dissipation. The thermal profile, the control parameters of simulation and the data of the processing output are generated by the use of the MATLAB. It also does trend-analysis, plots results and supports the plotting of temperature-performance curves. The two environments are connected using a wrapper script that is implemented in Python or Perl. The script does the following automatically: (1) rewrites the .sp netlist file with a new temperature set, (2) starts HSPICE simulation, (3) waits until it is done, (4) reads the performance measures out of the resulting .lis or .tr0 files, and (5) sends them to MATLAB to be processed. The iterative and multi-temperature simulations that it enables lack the manual involvement and are easy to be scaled and consistent due to this closed-loop architecture.

Thermal Model Integration

To be able to simulate the temperature effects on the behaviour of the circuit properly, the electrical parameters of MOSFETs and the passive components are adjusted according to temperature dependent equations. To address temperature, HSPICE provides the qualification of temperature as a global simulation variable (.TEMP) and employs it internally to alter settings like carrier mobility (μ), threshold voltage (V_{th}), as well as saturation current (I_{Dsat}). These parameters are empirical semiconductor formula models, representing temperature degradation of mobility and temperature shift of threshold voltages:

- **Mobility:** $\mu(T) = \mu_0 \times (T/T_0)^{-n}$
- **Threshold voltage:** $V_{th}(T) = V_{th0} - k \times (T - T_0)$

- **Saturation current:** $I_{Dsat} \propto \mu(T) \times (V_{GS} - V_{th}(T))^2$

MATLAB dynamically generates the set of temperature points (e.g., 25 °C to 125 °C in 10 °C steps), updates simulation netlist with the current value of TEMP, and uses HSPICE output files to gather gain, S11, NF and power quantities. These indicators are saved as structured arrays or dataframes to be processed in batches to display visual representation. Such parametric automation enables designers to quickly characterize the thermal trends without editing simulation files or retracing simulations over and over.

Simulation Flow and Automation

The overall simulation workflow is depicted in **Figure 1** (to be included), which outlines the automated interaction between MATLAB, HSPICE, and the scripting interface. The process begins with MATLAB generating a set of temperature values and invoking the wrapper script. For each temperature:

1. The script injects the temperature value into the .sp netlist.
2. HSPICE is called to execute the simulation.
3. The output .lis file is parsed to extract key performance indicators.
4. MATLAB receives and stores the results.
5. The loop continues until all temperature points have been evaluated.

This flow makes evaluation of the RF circuit behavior over a wide temperature range in a batch-mode, and hence usable in early stage thermal profiling, testing of reliability of circuit designs at high frequencies.

CASE STUDY: CMOS LNA

Circuit Description

In order to show effectiveness of the proposed thermal aware co-simulation framework, a 2.4 GHz inductively

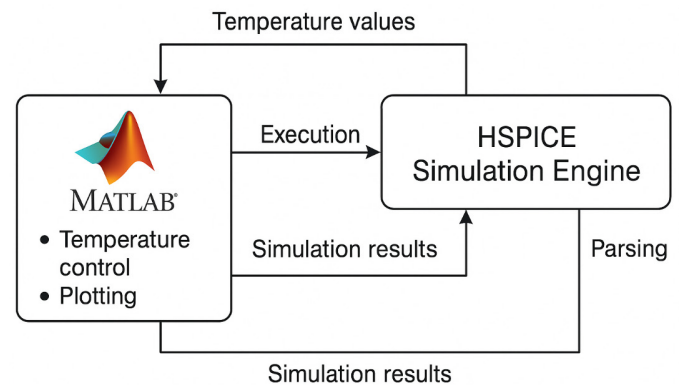


Fig. 1: HSPICE-MATLAB Framework Block Diagram

Ps eudocode for temperature-controlled simulation and performance extraction loop.

```

Initialize:
    Define temperature_range = [25, 35, ..., 125]
    Create empty result_storage structure

For each temperature in temperature_range:
    1. Update TEMP parameter in .sp (HSPICE netlist) file
    2. Launch HSPICE simulation via system call
    3. Wait for simulation to complete and generate .lis/.tr0 files
    4. Parse simulation outputs:
        - Extract Gain (S21), Noise Figure (NF), S11, P1dB, Power
    5. Store extracted data with temperature as index
    6. Log temperature-sweep progress

After all temperatures:
    - Load stored results into MATLAB arrays
    - Plot Gain vs Temperature
    - Plot NF vs Temperature
    - Export results for documentation

End
    
```

degenerated CMOS Low Noise Amplifier (LNA) will be the benchmark circuit. LNA designing with 0.18 CMOS process is scalable that provides an adequate trade-off between the RF performance and the integration cost. The topology includes the inductance of the source degeneration to provide the better input impedance matching and linearity without increasing the noise figure. Important design parameters are: a supply voltage (V_{DD}) of 1.8 V, a targeted gain of about 15 dB at room temperature (25 °C), and an input-referred noise figure (NF) of ~1.2 dB. The LNA is matched to 50 Ω source impedance using on-chip spiral inductors and utilizes an active load to ensure high gain across the operating band.

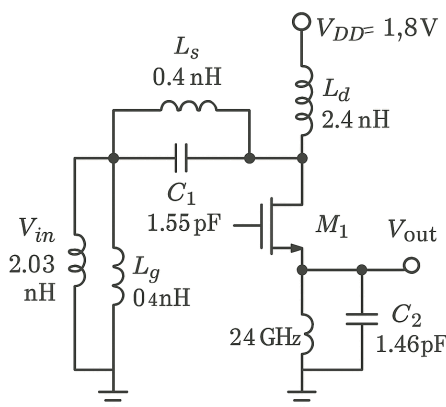


Fig. 2: CMOS LNA Circuit Schematic

Simulation Configuration

The simulation conditions are set in order to test the dynamic range of the LNA temperature, 25° to 125 °. 10° C. This is quite a representative range of conditions that can be met in automotive, aerospace, and outdoor IoT applications. To have the transistor-level netlist translated into HSPICE, the foundry-supplied BSIM3 models are available, and the temperature dependency is added by setting the .TEMP control parameter. MATLAB controls the thermal sweep and invokes the current temperature into the net list, generating HSPICE simulation run, and extracting key performance metrics by its scripting interface. The primary metrics evaluated include small-signal gain ($|S_{21}|$), noise figure (NF), input return loss ($|S_{11}|$), and 1-dB compression point (P_{1dB}). The parameters that are calculated out of the HSPICE output are saved as files indexed by temperature to analyze the trend.

Result Snapshots

The simulation outcomes reflect that the performance is declining with an increased temperature. Gain decreases considerable because the series resistance in the matching network rises along with the transport of carriers that has become low. Figure 3 illustrates the gain versus temperature curve, showing a decrease from approximately 15.1 dB at 25 °C to about 12.3 dB at 125 °C, which corresponds to an 18.5% reduction.

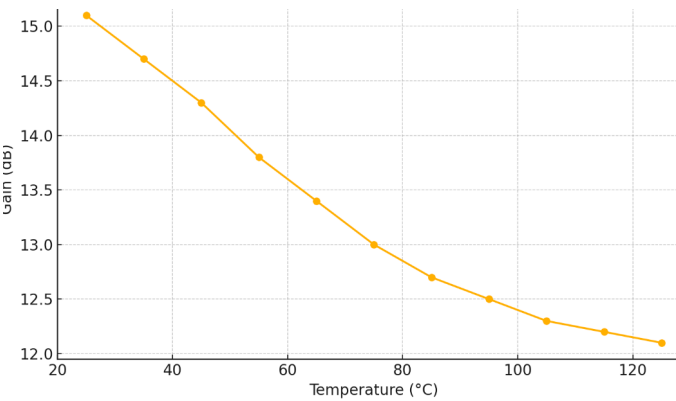


Fig. 3: Gain vs Temperature

Gain degradation of CMOS LNA over the 25 °C-125 °C range, showing sensitivity to temperature-induced mobility reduction.

Equally, the noise figure also illustrates a smooth rise of the temperature range, i.e. 1.2 dB to approximately 1.44 dB as shown in Figure 4. This degradation is explained by more thermal noise of the MOSFETs and the passive components. There is also a shift in the Return loss ($|S_{11}|$) which shows that impedance mismatch occurs at high temperatures. The trends justify the importance of considering temperature effects in the assessment of the RF circuit to make the operation reliable in the real-world.

RESULTS AND DISCUSSION

The simulated results by the proposed HSPICEMATLAB co-simulation showed the effectiveness of thermal variations on RF performance of the CMOS LNA. The parameters of gain, noise figure (NF), input return loss (S_{11}), and power use were measured as key performance parameters at three representative characteristics of gain figure (NF), input return loss (S_{11}), and power use were measuredtemperature points: 25 °C (room temperature), 75 °C (moderate thermal stress), and 125 °C (extreme high-temperature scenario).

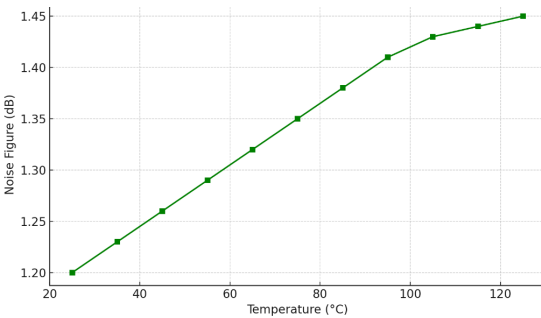


Fig. 4: Noise Figure vs Temperature

Thermal impact on LNA noise figure, highlighting the importance of thermal-aware design for low-noise operation.

The LNA has a significant gain deterioration as temperature increases as indicated in Table 1. The main cause of this degradation is the loss of carrier mobility and consequent loss of transconductance of the MOSFETs which has a direct effect on the gain of the amplifier. The gain drops from 15.1 dB at 25 °C to 13.6 dB at 75 °C and further to 12.3 dB at 125 °C, corresponding to an 18.5% decline over the full thermal range.

Simultaneously, the noise figure (NF) is consistently rising along with the temperature delta, that is, at room temperature, the noise figure is measured at 1.2 dB, whereas by 125°C, the noise figure value is 1.44 dB. This is due to an increased noise due to high thermal noise of active and passives components and bias drifts as a result of a temperature-dependent threshold voltage drift. The effect of this change in NF directly translates through to the signal-to-noise ratio (SNR) in RF receivers, which no doubt makes thermal-aware design of low-noise front-ends very important.

Also, the return loss ($|S_{11}|$) also worsens with temperature, a sign of mismatch because there is an impedance drift at the input port. It is a challenge of RF systems that need stable input matching over wide temperature range. The absolute value of S_{11} shifts from -18.5 dB at 25 °C to -14.9 dB at 125 °C, suggesting worsened input reflection performance. Lastly, power consumption shows a slight upward trend with temperature, increasing from 3.1 mW to 3.3 mW, which, while small, may become significant in thermally constrained environments.

The results are summarized in Table 1, with corresponding visual trends presented in Figures 5 and 6, which were generated through automated MATLAB post-processing.

Table 1: LNA Performance Metrics Across Temperature

Temp (°C)	Gain (dB)	NF (dB)	S_{11} (dB)	Power (mW)
25	15.1	1.20	-18.5	3.1
75	13.6	1.35	-16.8	3.2
125	12.3	1.44	-14.9	3.3

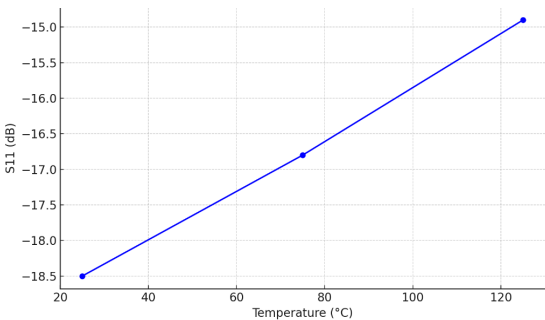


Fig. 5: S_{11} vs Temperature

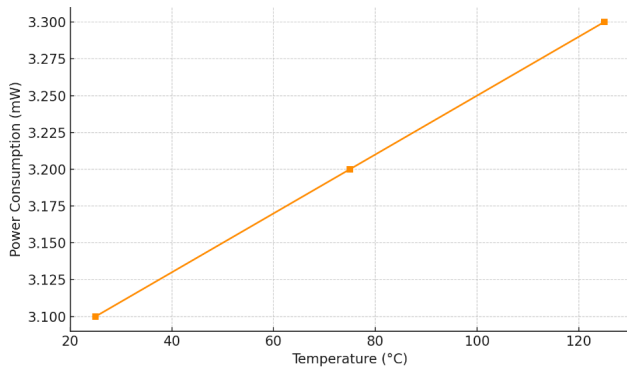


Fig. 6: Power vs Temperature

TOOLCHAIN AND IMPLEMENTATION

The proposed thermal co-simulation methodology relies on a well-choreographed tool chain which incorporates sophisticated electrical simulation accuracy, adaptive information processing, and scripting. All these parts have their role in promoting effective simulation flow, temperature sweep regulation, and performance metric extraction under an extensive temperature range.

The electrical simulation is performed on a transistor level with Synopsys HSPICE 2023.3 version, which is an industry-standard circuit simulator commonly used in the industry and based on the fact that it is able to achieve decent convergence performance and high accuracy during RF and analog circuit simulation. BSIM3 device model provided by foundry in 0.18 μm CMOS technology is used to model the realistic temperature dependent behavior in MOSFET devices. HSPICE is a backend program that can take the .sp netlist file and calculate such outputs as gain, noise figure (NF), and return loss (S11) and can calculate power consumption.

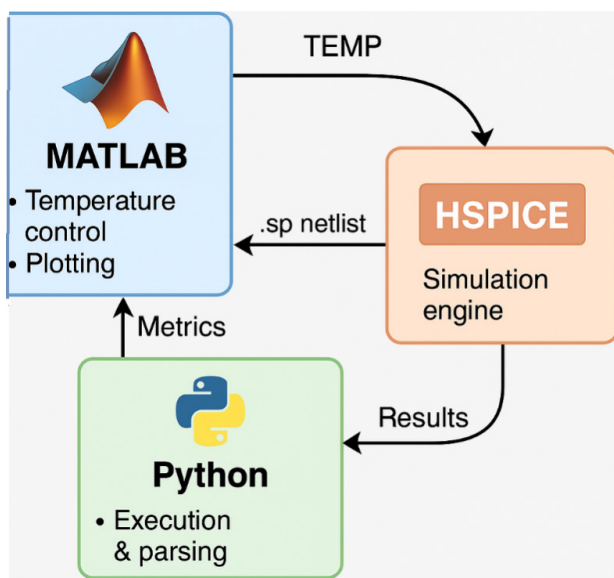


Fig. 7: Toolchain Integration Flow

The MATLAB R2024a is applied to control the temperature, process the results, and visualize them. Scripting in MATLAB can enable batch operation, automation of input/output of files and dynamically generate visualization of the simulation outcomes. It is also a high level controller which takes turns to inject the temperature data into the HSPICE netlist and takes out the tracked performance values to analyze the trends. The multi-point simulation, curve fitting, and the publication of the results could be done using the vectorized data structures in MATLAB.

Its inner automation is a Python 3.x script (a wrapper), which will serve as a means of communication between the MATLAB and HSPICE. In the script the subprocess package of Python is used to call a command line procedure to run the HSPICE and the script waits until the simulation is complete. When it is done, it reads the .lis output file according to custom regular expressions or strings search pattern to extract electrical parameters. These values are then transferred to MATLAB in real time and thus putting a closure to the simulation-feedback loop. The design is flexible, modular and easy to extend to other types of simulation scenarios or circuits.

Although the study itself is ran in a fully synthetic environment, the toolchain is designed to allow the extension with Hardware-in-the-Loop (HIL), in the future. It is possible to validate thermal behavior with real measurement data in such a setup by application of a programmable testbench, like a temperature-controlled RF test chamber or Zynq-based RF signal processing prototype. This would facilitate a physical evidence to refute thermal drift in practical RF components and help develop models of temperature precision.

To conclude, the toolchain runs a first-rate and scalable co-simulation process integrating model force of HSPICE, processing calibre of MATLAB and automation quality of Python. Reproducibility, effective implementation, and scaling to higher levels by convergence of a more complex thermal-aware RF circuit design and evaluation, are guaranteed here in this modular design.

CONCLUSION

The present paper suggests a systematic co-simulation platform based on the combination of HSPICE and MATLAB that can offer thermal-aware modeling, simulation and thermal performance analysis of RF cells. The proposed system can be effectively used to automate the temperature sweeps, track the real-time performance, and visualization of key RF parameters like the gain, noise figure (NF), return loss (S11), and power consumption, thereby providing efficient automation of

the temperature sweep, real-time performance tracking, and visualization of key RF parameters including the gain, noise figure (NF), and the return loss (S11), as well as the power consumption.

The main results of the works conducted prove that the thermal fluctuations may result in a regime with serious performance degradation of RF front-end circuits. A case study with 2.4 GHz CMOS-based Low noise amplifier (LNA) indicates a 18.5 percent and 20 percent decrease in gain and increase in noise figure at higher operating temperature by 25°C measure to 125°C. Also, losses of input voltage are degraded by impedance mismatch at thermal drift, and power dissipation has a very limited but measurable increase. These ramifications underscore the imperative of the capability to integrate temperature dependency in the RF design and verification process, particularly high-reliability design operating environments in 6G wireless communication, automobile radar, and biomedical telemetry systems.

The discussed framework has such value as it offers a versatile, scalable and reusable simulation infrastructure. It fills the gap of standalone SPICE simulations and the post layout thermal analysis tools and will provide a unified solution which is efficient and flexible. With the addition of a Python automation layer, reproducibility is helped and its fast design exploration allows it to be used in both academic research and in industry.

In the future, there are some directions which can be further developed concerning the potential of this framework. First, layout-aware self-heating models which can be integrated with ANSYS ICEPAK or COMSOL can be used to give a finer geometrical characterization as part of a post-layout verification. Second, the model can be applied to multi-stage RF sub-systems like mixers and power amplifiers in order to investigate the effect of cumulative thermal situations. Third, machine learning models can be integrated to extrapolate the thermal trends on a typology of simulation data, which would speed up the exploration of the design-space. Lastly, it is possible to present a hardware-in-the-loop (HIL) experiment with real-time measurements, where temperature control is performed, to ensure simulation forecasts and profile adjustments to thermal modeling through actual operating conditions.

Finally, the HSPICE-MATLAB co-simulation platform presented in this paper offers a handy and foolproof set of capabilities to RF circuit designers and engineers who seek to come up with thermally resilient circuits. Through inclusion of automation, temperature-sensitive modelling, and multi-domain exploration, the framework assists in addressing one of the key gaps within a RF

circuit design flow process that has been traditionally hard to bridge, i.e. the creation of simulation closer to real deployment conditions.

FUTURE WORK

Although the present work has succeeded in proving the possibility and benefits of combining HSPICE, and MATLAB to achieve thermal-aware RF circuit simulation, a number of extensions and advances can add tremendous value to the utilization and accuracy of the suggested framework. The first follow-up is the incorporation of layout-sensitive self-heating through multiphysics high-end simulation, e.g., via COMSOL Multiphysics or ANSYS ICEPAK. With these, localized temperature gradients, localized thermal hotspots, and substrate heat spreading effects can be simulated which cannot be simulated in uniform temperature SPICE. One may consider geometric placement and metal density as they may import post-layout data (e.g., a GDSII or DEF file) into the simulation flow and allow the designers to explore spatial thermal profiling block and chip-level. This will increase confidence of the framework in predicting temperature induced reliability problems like electromigration and thermal runaway in compact devices operating at high power levels in the RF band.

The other area that should undergo development is how the architecture will be implemented to multi-stage RF sub-assemblies which hold mixers, power amplifiers (PAs), voltage-controlled oscillators (VCOs). These elements are thermally and electrically interactive with each other so that the heat produced by one stage can have an effect on the bias point and impedance matching of other stages. Through this modeling of such interdependencies, the framework can develop into an integrated system-level thermo analysis and optimization tool of RF chains that is of additional importance especially in an integrated transceiver system in 5G/6G and radar applications.

One of the other promising directions is the implementation of AI/ML-based thermal prediction systems. Very little information can be collected in reduced temperature points, allowing regression models and neural networks or Gaussian process models to be trained using the limited data to provide values of their values of the performance metrics at every point on the complete thermal range. These approximations are known to save much on-computational time without compromising the required accuracy. Further, adaptive temperature sweeps may be directed by reinforcement learning agents, targeting the simulation with high sensitivity, e.g. around gain roll-off or NF spikes.

Moreover, to make the simulation result closer to the real test scenario, it is possible to add the Hardware-in-the-Loop (HIL) validation framework. The interface of a simulation loop at a controlled temperature can characterize the simulation correctness to bring it to the real-time RF expertise board (e.g. NI, PXI, Keysight, B2900A or Zynq-based SDR systems), where simulation parameters are calibrated to it. This would also justify the relevance of the framework to industrial prototyping and regulatory testing process.

In short, future work in this framework will be directed towards increasing the scale of the accuracy, execution efficiency, and interposing the disparity between simulating and conducting the actual experiment. Such advancements will not only render this framework to be a prized academic research-centric instrument, but also a highly potent driver of the RF product design; performance optimization; and qualification of the product as simulated under thermodynamically time-varying operating circumstances.

REFERENCES

1. Jain, A. K., Sharma, S., & Saini, R. (2018). Temperature dependence of CMOS RF circuit performance. *IEEE Transactions on Circuits and Systems I: Regular Papers*, 65(2), 402-411. <https://doi.org/10.1109/TCSI.2018.2795957>
2. Li, X., & Wang, Y. (2021). Co-simulation of electrical and thermal behaviors of analog circuits using MATLAB-SPICE integration. *Microelectronics Journal*, 112, 105042. <https://doi.org/10.1016/j.mejo.2021.105042>
3. Nguyen, V., Matsuda, T., & Saito, S. (2018). Thermal analysis of RF CMOS circuits using 3D simulation tools. *IEICE Transactions on Electronics*, E101-C(4), 265-272. <https://doi.org/10.1587/transele.E101.C.265>
4. Banu, M., & Agrawal, A. (2020). MATLAB-assisted performance evaluation of RF blocks in mm-wave transceivers. *International Journal of RF and Microwave Computer-Aided Engineering*, 30(6), e22220. <https://doi.org/10.1002/mmce.22220>
5. Tamm, J. A., Laanemets, E. K., & Siim, A. P. (2025). Fault detection and correction for advancing reliability in reconfigurable hardware for critical applications. *SCCTS Transactions on Reconfigurable Computing*, 2(3), 27-36. <https://doi.org/10.31838/RCC/02.03.04>
6. Barhoumi, E. M., Charabi, Y., & Farhani, S. (2024). Detailed guide to machine learning techniques in signal processing. *Progress in Electronics and Communication Engineering*, 2(1), 39-47. <https://doi.org/10.31838/PECE/02.01.04>
7. Thompson, R., & Sonntag, L. (2025). How medical cyber-physical systems are making smart hospitals a reality. *Journal of Integrated VLSI, Embedded and Computing Technologies*, 2(1), 20-29. <https://doi.org/10.31838/JIVCT/02.01.03>
8. Rucker, P., Menick, J., & Brock, A. (2025). Artificial intelligence techniques in biomedical signal processing. *Innovative Reviews in Engineering and Science*, 3(1), 32-40. <https://doi.org/10.31838/INES/03.01.05>
9. Rangiseti, R., & Annapurna, K. (2021). Routing attacks in VANETs. *International Journal of Communication and Computer Technologies*, 9(2), 1-5.