

Faculty of Engineering
ASPIRE Summer Research Program 2026-2027

Project Title: Bringing SPICE Circuit Simulation into Mathematica: a Wolfram Language package for symbolic and numerical analysis of electronic circuits

Supervisor(s): Dr. Uthpala Ekanayake, Prof. Malin Premaratne

Department: Electrical and Computer Systems Engineering

Email: uthpala.ekanayake1@gmail.com

Website profile of project supervisor: <https://research.monash.edu/en/persons/uthpala-ekanayake/>

Objective:

Developing an open-source Wolfram Language (Mathematica) package that imports industry standard SPICE netlists and performs core circuit combining symbolic computation, numerical methods, and advanced visualisation in a unified environment.

Project Details:

SPICE (Simulation Program with Integrated Circuit Emphasis) is the standard software used for electronic circuit simulation. This project will develop an open-source Mathematica package that reads standard SPICE netlists and performs DC, AC, and transient circuit analysis, allowing existing SPICE circuit files to be simulated directly within Mathematica. The package will include a netlist parser, Modified Nodal Analysis, and numerical integration for transient simulation. If time permits, support for basic nonlinear devices (e.g. diodes and transistors) will also be included. The implementation will be validated against ngspice and released as open source for research and teaching. By integrating SPICE simulation into Mathematica, the project will provide a unified environment for circuit simulation, symbolic computation, optimization, and use of machine learning, that are not possible with conventional compiled circuit simulators.

Project Timeline

TASK	1	2	3	4	5	6	7	8	9	10
Onboarding; SPICE & MNA background; Wolfram setup										
Netlist parser (R, L, C, sources)										
Linear DC solver (MNA); validate vs ngspice										
AC / frequency response; symbolic transfer functions										
Transient analysis (numerical integration)										
Nonlinear devices (diode, level-1 transistor)										
Validation suite, benchmarking & documentation										
Final Report, presentation and open source code release										

Prerequisites

Basic programming experience (such as Python, C, MATLAB, or Mathematica), introductory circuit theory, and an interest in numerical methods and scientific computing are recommended. Experience with Mathematica is beneficial but not required. The project requires persistence and a willingness to code throughout. Some use of AI coding assistants is permitted to support development.

Student Placement Agreement

A student placement agreement must be signed by the project supervisor and the student prior to commencement of the ASPIRE program: [ASPIRE Program Co-Curricular-Student-Placement-Agreement-Short-term.docx](#)

Additional Information

Applicants may be invited to a short interview. This project is well suited to students interested in scientific computing, electronic circuit simulation, or computational electronics, and provides an opportunity to develop an open source software package with potential for publication