Introduction

SPICE simulation is important for students and researchers because it can help them to understand the behavior of the devices or circuit. Unfortunately, SPICE faced the following disadvantages:

• need to run on a workstation
• not multi-platform supported,
• cost much to users on maintenance
• not convenience to use

Features and Objective

Our objective is to provide a more cost-effective solution for educational purposes mainly.

• Circuit Simulation Running
• Model Card from i-MOS Library
• SPICE3f4 netlist format support
• Transistor Models support by NGSPICE
• Working progress saving/retrieving
• Netlist import/export
• Analysis type support
  ➢ DC Transfer
  ➢ AC Transfer (Linear/Decade)
  ➢ Transient Analysis
• Graphs plotting
  ➢ Linear and Log plot rendering
  ➢ AC Decade Plot ready
  ➢ Zooming for plot viewing
  ➢ Plot as image downloading
• RAW result downloading as Spreadsheet

Online Circuit Simulation Platform (CM2-12)

Students: Tsang Wing Hang
Tam Ka Chun Cyrix

Supervisor: Professor Mansun Chan

System Flow Diagram

Results

Input

Output