An Academic EDA Suite for the Full-Custom Design of Mixed-Mode Integrated Circuits

J. Pallarès¹, K. Sabine², L. Terés¹,³ and F. Serra-Graells¹,³

¹Institut de Microelecètronica de Barcelona, IMB-CNM(CSIC) ²Peardrop Design Systems Ltd. ³Universitat Autònoma de Barcelona

Introduction

- Expensive commercial EDA tools of complex installation, maintenance and learning.
- Free framework for full-custom IC design:
  - Schematic and layout edition
  - Mixed A/D, event/electrical simulation
  - Automatic circuit optimization
  - All physical verification steps
- Physical design kit (PDK) for target CMOS technology
- Available for:
  - Windows
  - Linux

Schematic Edition

- All SPICE3 and XSpice devices are supported:

XSpice HDL Simulation

- Mixing C-language models with electrical SPICE devices:

Circuit Optimization

- Usage of SpiceOpus optimize command:

Schematic-Driven Layout

- Python-based layout PCells
- Design-rule driven (DRD)
- Multi-path (MPP)
- Matching groups
- Layout generation:

Design Rule Checker

- Python-based rule scripting:
  - geometrical/boolean operations
  - verification functions:
    - off-grid
    - width
    - spacing
    - match
    - extension
    - overlap
    - enclosure
    - area...
  - User-friendly error debugging:

Layout versus Schematic

- Full support of extracted views:

3D Parasitics Extraction

- Python-based scripting:
  - pin/net identification
  - extraction functions:
    - resistor
    - capacitor
    - diodes
    - BJTs
    - MOSFET
    - TFT
    - arbitrary device...
  - Integrated Fastcap finite-element engine
  - Automatic mesh generation

Filtering, merging and backannotation

Array of transcapacitances