

EDIF Conversion Flow with Gateway

Analog/Mixed-Signal Simulation

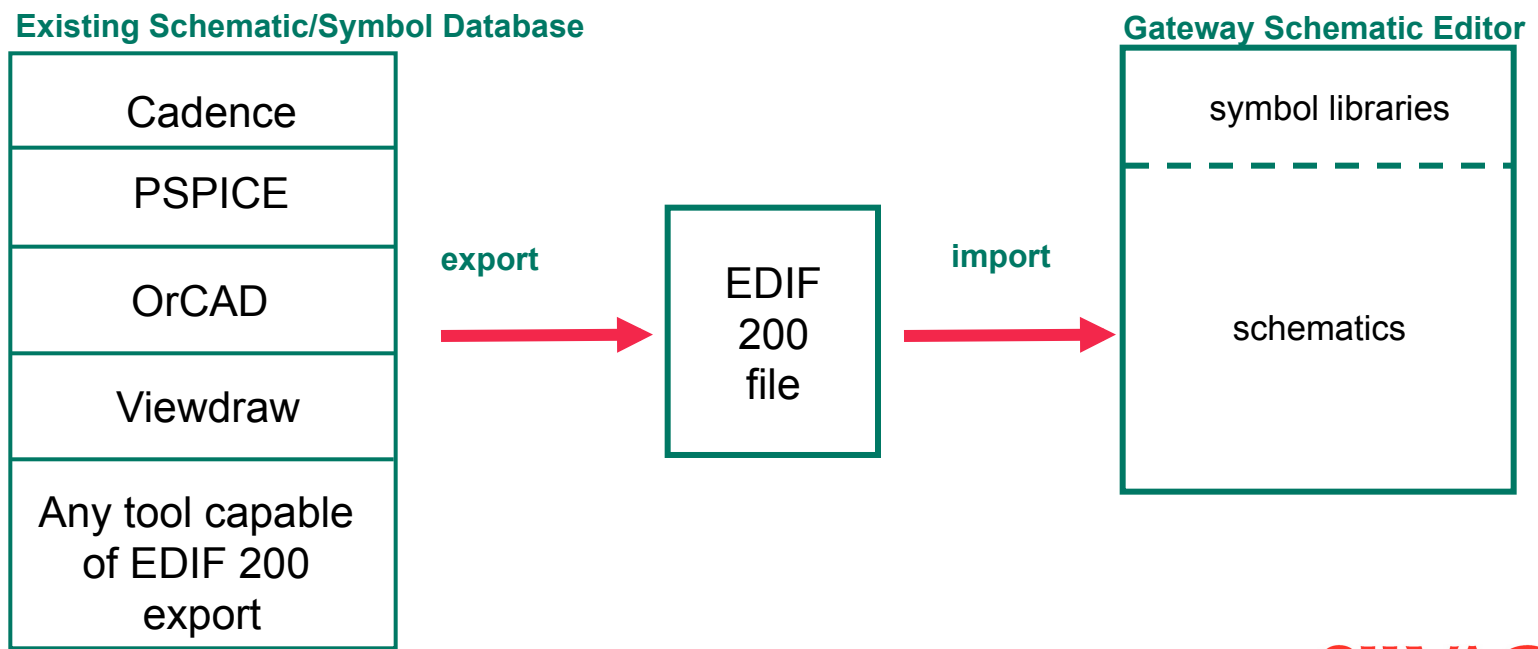


SILVACO

Analog/Mixed-Signal Simulation

What is EDIF 200?

- EDIF is the acronym for Electronic Document Interchange Format
- EDIF is a standard for transferring schematic data between two environments
- EDIF 200 files from Cadence, Viewdraw, PSPICE, or OrCAD can all be imported



Analog / Mixed-Signal Simulation

Limitations of the EDIF Standard

▪ EDIF 200 General Limitations

- EDIF is a university-driven standard, not foundry-driven
- Schematics, unlike GDSII files, cannot be streamed-in and out seamlessly
- Information may be lost through both exporting and importing of the same file
- Successful parsing by University of Manchester Parser is the only guarantee that a file is EDIF 200 compliant
- Some EDA vendors use vendor-specific fields where information cannot be seen by other vendors
- Arrayed instances are not converted easily
- Buses and ripping are generally handled differently between vendors
- Many symbol attribute names are not similar between vendors
- Schematic grids and snap-grids are vastly different between vendors
- Schematic hierarchies may not ascend and descend after export/import

Analog / Mixed-Signal Simulation

Vendor Specific Limitations

■ PSPICE-OrCAD

- Uses floating-point grid instead of integer grid
- Symbol sizes can be exceedingly large in comparison to standard Gateway symbols
- Some PSPICE device attributes differ from the corresponding devices in SmartSpice
- Some pins and wires are off-grid and must be attached using auto-snapping
- OrCAD and PSPICE exports differ and must be scaled accordingly
- Differences between PSPICE and SmartSpice in terms of general syntax and subcircuit instantiation
- MicroSIM PSPICE users need to first convert designs to OrCAD CAPTURE and then EDIF out

■ Viewdraw

- Viewdraw schematics do not require pins when pushing down from parent symbols
- Device parameter name fields are different than for SmartSpice

Analog / Mixed-Signal Simulation

Gateway Exceeds the EDIF Standard

■ Conversion Capabilities of Gateway

- Most nodes are retained by name and probe-able through hierarchy
- Device reference designators retained wherever possible
- Imported symbol libraries automatically written to gateway workspace
- Symbol browser automatically generated for imported symbols
- Attribute visibilities retained wherever possible
- Attribute values retained for all OrCAD “VALUE” fields
- Hierarchy recognizable in Gateway
- Hierarchy navigation up and down through cells
- Device wiring retained for connectivity and rubberbanding

■ Additional Importing Options

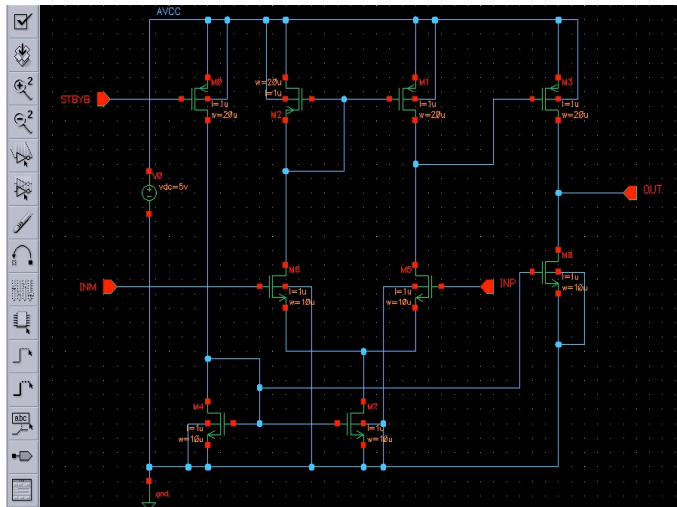
- Ability to filter out unwanted, cosmetic, or irrelevant device parameters during import
- Resolves differences in case when importing
- Filters out node and net name characters which Smartspice does not understand
- Option to convert symbol footprints and designs to the Gateway grid

Analog/Mixed-Signal Simulation

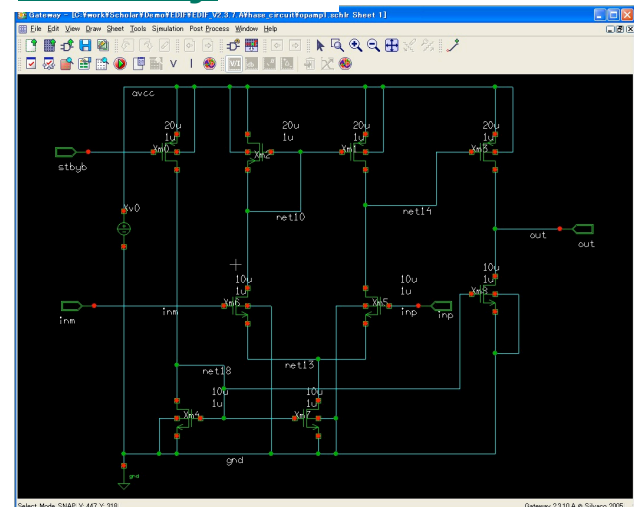
'Before and After' example

- Positions, connectivity, parameter values are kept
- Native libraries are faithfully imported
- Naming, wiring, and connectivity is intact

Composer



Gateway



Analog/Mixed-Signal Simulation

Schematic Imported Into Gateway

Libraries automatically imported and written to the workspace →

Nodes and names are retained and highlighted thru hierarchy →

Imported symbols scaled and previews generated →

Searchable schematic database →

Hierarchy Browser →

Verbose session log containing All import messages and errors →

Preview pane to show zoom Location relative to entire schematic →

The screenshot shows the Gateway 2.6.3.R interface. The main window displays a schematic diagram with components like 74LS04, 74LS08, and 74LS138. A search dialog is open, showing search results for 'Wires'. The session log at the bottom left provides detailed information about the import process, including warnings and errors. The preview pane at the bottom right shows a zoomed-in view of the schematic.

Analog / Mixed-Signal Simulation

Guidelines for Productivity

- Do NOT EDIF export IP that does not need rework
- Review all Export fields accordingly before actually exporting to EDIF
- Export designs from top-level schematic
- Export symbol libraries by instantiating all instances in the library to a schematic and then export the schematic
- Primitive devices (M, Q, R, C, etc..) must be converted first and given SmartSpice attributes for netlisting
- Use both “Instance Replacement Method” and “Edit-in-Place Method”
 - Edit-in-Place – converting symbol master file from instances on imported drawings
 - Instance Replacement – use of the Change Symbol dialog to globally convert when changes are made
- Use the Gateway User Manual Chapter 9 for specific detailed instruction