Salvaging Old Designs Through EDIF 2 0 0

EDIF (Electronic Design Interchange Format) is an ANSI/EIA standard file format used to transfer data from one CAD/CAE system to another. In Gateway, the EDIF file format is used to transfer schematic drawings between other vendors and Silvaco. Gateway only supports importing and exporting of EDIF Version 2 0 0 files.

Since EDIF is not a foundry driven standard, the look, feel, and scope of what is transferred by the EDIF process varies greatly between software vendors. This is one way that the EDIF standard is different from the GD-SII standard. For this reason, it is unlikely that a schematic may be able to netlist or simulate after import without some amount of editing. For example, the original schematic may have parameters defined using a specific syntax. The importing tool uses another simulator with a completely different engine and syntax. In this case, there is no definition in the EDIF standard to translate them into meaningful attributes for the imported design. In many cases, the imported drawing is only a drawing of symbols and wires but unable to netlist or run a simulation. This document highlights some of the basics of how to get the raw imported EDIF schematic into a condition that is useful.

The Gateway EDIF importer uses all available information from the imported file to regenerate the same information on cellviews in Gateway. This includes wire connectivity, net and node naming (where possible), symbol reference designators, and symbol attributes with their instance values. In addition to that, schematic hierarchies should be imported and capable of descending and ascending through the hierarchy. The few exceptions are bus rippers and naming because of the variety of bus implementations between vendors, and other cellviews such as netlist views. If netlist views were present on the original drawing, attach the netlist in the control deck (*.ctr file) and manually create the symbol (*.body file).

Gateway successfully imports files that comply to the EDIF 2 0 0 standard. Figure 1 shows the EDIF Import dialog box. The options there are grouped by vendor specific options and general options. All options are designed to maximize productivity wherever possible by making use of data and converting it to reduce the amount of editing after conversion.

General Options

These options can be used on any EDIF file coming from any vendor. They are described in the following:

- **Fix case sensitivity conflicts** – Resolves conflicts for library, cell, and net names duplicate alphanumeric strings in the EDIF file by naming them uniquely.

- **Automatic schematic pin conversion** – Replaces ipin, opin, iopin symbol spicelib/inschpin, spicelib/outschpin, spicelib/bischpin symbol instances. This option is disabled if the current workspace does not include spicelib, the native spice primitive library supplied by Gateway.

- **Discard invalid characters in node names** – Strips out node name characters for SmartSpice.

- **Verbose output** – Produces extra information during import that may be useful import errors. If an import fails for some reason, you may want to turn on Verbose importing again. The verbose messages might help to locate the problem.

- **Filter attributes** – Ignores specified attribute names during the EDIF import and from importing. To specify these attributes, click the Edit button on the dialog. This is useful in reducing clutter as well as attribute types that may be cosmetic or have no impact on netlisting.

Further options are classified as vendor specific. These should be chosen as necessary and are dependent on which vendor generated the EDIF 2 0 0 file. They are described following sections.

Cadence Options

- **Convert iPar/pPar parameters** - Attempts to find iPar/pPar attributes on symbol instances and adds them as properties of a spicelib/parameters symbol instance on the parent schematic. This sets up the feature for adding passable parameters down a hierarchy.

- **Discard Cadence cds annotations** - Throws away symbol drawing annotations starting with the "cds" prefix. For example, "cdsTerm("G")", "cdsName()", and "cdsParam(1)".

Viewdraw Options

- **Add Viewdraw pins** – Adds pins to schematic files where necessary for the purpose of hierarchical connectivity and netlisting. Because the incoming schematics may not have the required pins on some schematics, the parent symbols will be unable to push down unless this option is checked.

- **Retain Viewdraw snap spacing** - Converts Viewdraw schematics/symbols from the imported file to the Gateway schematic grid.

MicroSim PSPICE and ORCAD Options

- **Retain OrCAD snap spacing** - Converts MicroSIM PSPICE and OrCAD schematics/symbols from the imported file to the Gateway schematic grid.

Note that schematics from MicroSIM PSPICE must be first converter to OrCAD Capture first. When that has been done, Capture can generate EDIF 2 0 0 from there.

Because of the extra development above and beyond the EDIF 2 0 0 base standard, Gateway customers can benefit from getting the most out of their old designs and can move faster toward the goal of a usable SPICE netlist. Because the retention of object names, parameters, and parameter values, the editing after import is greatly reduced. Adding to those conversions are the wiring connectivity retained as originally drawn, a hierarchy in tact, and symbol footprints converted to the Gateway grid. With some minor adjustment for primitive symbols so they can be recognized for SmartSpice, the netlist can be achieved without the need for redrawing.
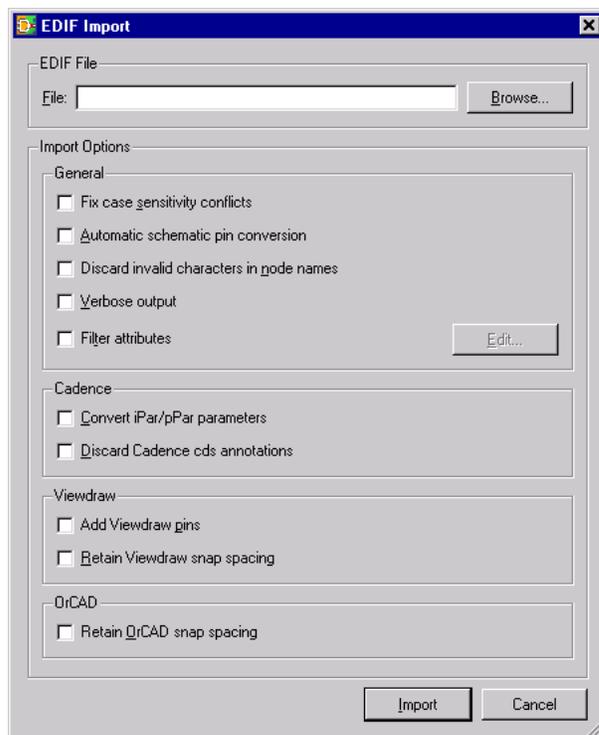

Figure 1. Gateway EDIF Import Dialog.