
Designer Series Development System

R1-1999 Release Notes Addendum

This is a supplement to the R1-1999 Release Notes. Please read through this document thoroughly for important information on new features and known limitations.

New Features and Enhancements

This section lists the new features and enhancements of the R1-1999 release.

Designer

Chip Edit has been enhanced to display input and output nets of any specified macro or set of macros. To display input and output nets of any specified macro or set of macros, select the "Nets" pull down menu. The Nets menu contains two options, Display and Display Algorithm.

Display allows users to select which type of nets they wish to view. The options under the Display category are: none, input only, output only, and both. The default is both.

The Display Algorithm option allows users to select between two different modes of viewing nets. The default mode, Fanout, displays all nets that are a direct input or output to the selected module(s). The second mode, Minimum Spanning Trees, displays the total fanout of the input or output nets, and does so by connecting pins with a set of segments such that the total length of its segments are minimized.

For example, a design has a pin Q that drives signals A, B, and C. If Fanout mode is selected, 3 nets are displayed: one from Q to A, one from Q to B, and one from Q to C. If Minimum Spanning Trees mode is selected, 1 net is displayed with 3 segments. If A is physically closest to Q, and B is closer to A than C, then the first segment will span from Q to A, the second segment will span from A to B, and the third segment will span from B to C.

A new command, Static Objects has been added to the View menu in Chip Edit. Users can now insert a static grid in Chip Edit to help keep track of, and count rows and columns. This is especially useful for larger devices. The available grid sizes are: 2 by 2 lines, 4 by 4 lines, 4 by 4 rectangles, and 8 by 8 rectangles.

Known Limitations

This section describes known limitations for the R1-1999 release. For the latest information on known limitations, check the GURU on-line technical support system on the Actel Web site (<http://www.actel.com>). Also check the “software updates” section of the User Area frequently for software fixes.

Designer

Designer may unexpectedly exit when you select layout on a PC if a design name is 20 characters or longer, and the design was compiled on a PC version of Designer. A patch will be available soon in the software updates section of the User’s Area on Actel’s Web site.

For the A42MX family, it is currently not possible to select “MIL” temperature range for plastic packages. In order to design a military device using a plastic package with the correct timing, users must select Custom temperature range, and manually enter the correct Military operating conditions. The correct Military operating conditions are as follows:

Best: -55 C, 5.50V

Typical: 25 C, 5.00V

Worst: 125 C 4.5V

Designer Series Development System’s netlist translator function has difficulty translating a signal that passes completely through a design and is carried through different levels of hierarchy. In rare cases, the direction of ports will be switched. The workaround for this is to either not pass the signal through the hierarchy of the design (keep it on the top level), or insert a buffer on the signal to preserve the direction.

ACTmap

Do not select SXA in ACTmap. Instead, select SX if you wish to create a design targeted for SXA. Afterward, follow the instructions listed in the release notes.

If SXA is selected in ACTmap VHDL, you will get an error as follows:

```
ERROR : (VHDL_0047). Not allowed -tech value : 54SXA.
```

If you select SXA in Netlist Optimizer, you will get the following error:

```
Error during netlist optimization mapping.
```

When you invoke ACTmap on the HP platform, the xterm window closes automatically.

APSW

Do not select Action Probe from APSW. If you do, the APSW software session will end. In order to use the Silicon Explorer, run the Silicon Explorer 3.0 Software. To install this software, go to the Silicon Explorer 3.0 directory on your R1-1999 Designer series Development System PC CD-ROM and run setup.exe (Windows Only).