



Technical Reference for Altium's Xilinx® Virtex®-4 Daughter Board DB36

Summary

Technical Reference
TR0160 (v1.0) April 28, 2008

This reference document provides detailed information on Altium's Xilinx Virtex-4 daughter board DB36, including the physical FPGA device it offers and any additional resources available to an FPGA design targeting that device.

The target FPGA devices to which a design can be downloaded reside on separate satellite boards, referred to as daughter boards. These boards plug-in to Altium's Desktop NanoBoard NB2DSK01. By keeping each programmable device on its own daughter board, engineers can easily change the target project architecture, while keeping the NB2DSK01 a truly FPGA vendor-independent, reconfigurable hardware platform.

Altium's Xilinx® Virtex®-4 daughter board DB36 provides an XC4VLX25-10FF668C device, as well as a range of on-board memories available for use by a design running within that device.

Note: All daughter boards available from Altium may be plugged into either the Desktop NanoBoard NB2DSK01 or the NanoBoard-NB1.

For information on the Desktop NanoBoard NB2DSK01, refer to the document [TR0143 Technical Reference Manual for Altium's Desktop NanoBoard NB2DSK01](#).

For information on the complete range of daughter boards currently available, and additional documentation specific to each, go to www.altium.com/nanoboard/resources.

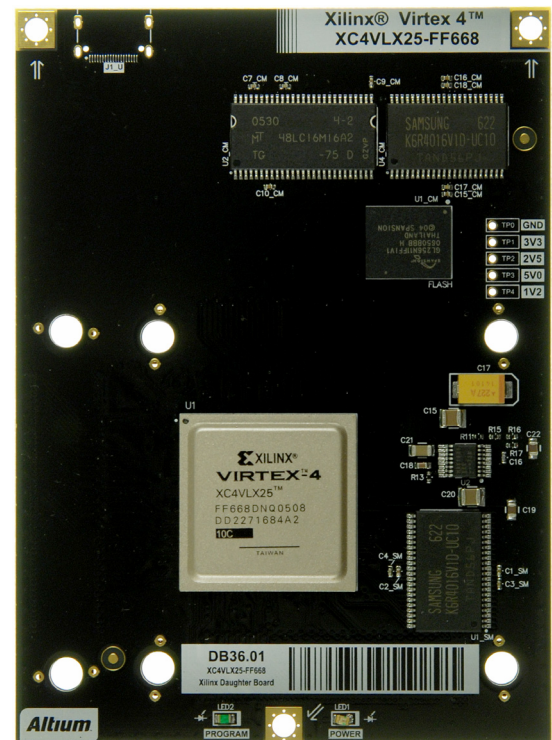


Figure 1. Altium's Xilinx Virtex-4 daughter board DB36.

Key features

- Xilinx Virtex-4 FPGA (XC4VLX25-10FF668C)
- On-board memories available for use by FPGA design:
 - 256K x 32-bit common-bus SRAM (1MByte)
 - 16M x 32-bit common-bus SDRAM (64MByte)
 - 16M x 16-bit common-bus Flash memory (32MByte)
 - Dual 256K x 16-bit independent SRAM (512KByte each)
- 1-Wire® memory device used to store board ID and related information
- Three 100-way connectors for attachment to NB2DSK01 motherboard. These connectors provide:
 - Interface to resources on the NB2DSK01 motherboard and plugged-in peripheral boards
 - SPI bus interface
 - I2C bus interface
 - 1-Wire bus interface
 - JTAG, power and additional control lines from the motherboard.

Functional overview

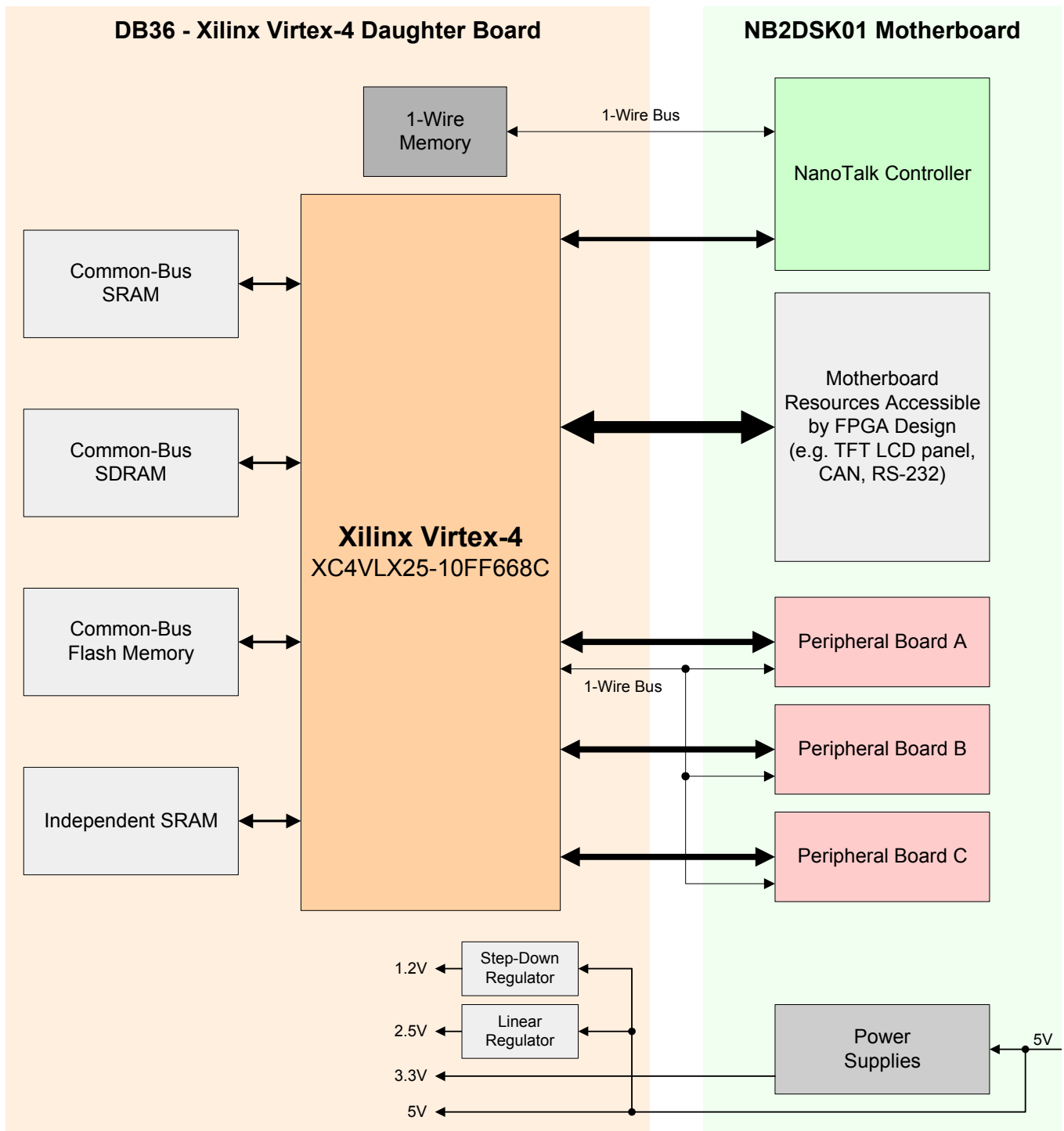


Figure 2. Xilinx Virtex-4 daughter board (DB36) block diagram.

3-connector daughter board

Daughter boards available with the Desktop NanoBoard NB2DSK01 and beyond each have three connectors (top, bottom and left). These are referred to as '3-connector' daughter boards. Those available previously for the NanoBoard-NB1 have two such connectors (top and bottom) and are referred to as '2-connector' daughter boards. The Xilinx Virtex-4 daughter board (DB36) is a 3-connector daughter board.


3-connector daughter boards also have three holes that align with threaded standoffs on the NB2DSK01. These can be used to affix the board securely to the motherboard.

Attachment to the NB2DSK01 motherboard

The daughter board is mounted onto the NB2DSK01 motherboard by plugging its three 100-way Male docking connectors into the motherboard's corresponding three 100-way Female docking connectors – designated HDR_T1, HDR_L1 and HDR_B1. These are referred to as 'NANOCONNECT' interfaces. The board can be securely fixed in place using the available threaded standoffs.

Accessing motherboard resources

The daughter board connectors on the motherboard map I/O resources on the NB2DSK01 directly to the pins of the daughter board FPGA device, as if that device were mounted directly on the motherboard.

 For more information on the NB2DSK01 motherboard resources made available to a daughter board FPGA device, see the section *Resources accessible from an FPGA design*, in the document [TR0143 Technical Reference Manual for Altium's Desktop NanoBoard NB2DSK01](#).

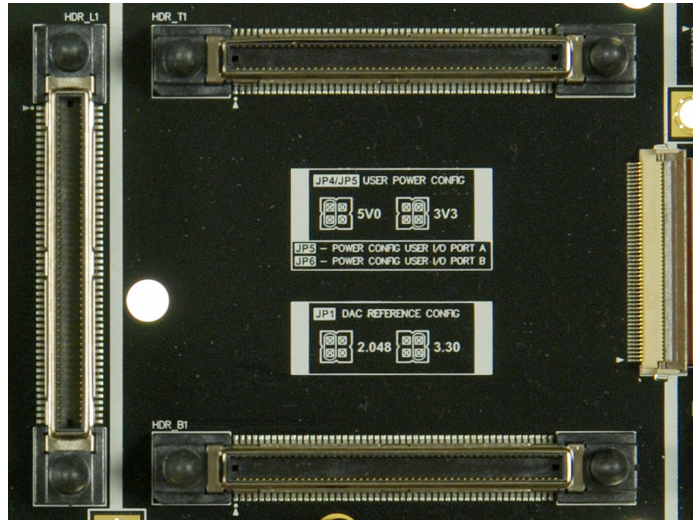


Figure 3. Daughter board connectors on the NB2DSK01 motherboard – the docking point for a satellite daughter board.


Accessing peripheral board resources

Resources on the plug-in peripheral boards are also made available to the daughter board FPGA device. The corresponding I/O pins from each peripheral board connector – 50 each – are wired directly to pins of the daughter board connectors on the motherboard:

- Peripheral Board A and B connector signals are wired to daughter board connector HDR_L1
- Peripheral Board C connector signals are wired to daughter board connector HDR_T1

These signals are subsequently wired to I/O pins of the daughter board's FPGA device. By keeping the signals generic – based on the peripheral board connector position and not on the resources – the peripheral boards can be attached to the NB2DSK01 in any of the three peripheral board positions.

Note: Some peripheral boards (e.g. PB01 – the Audio / Video Peripheral Board) are twice the width of other available peripheral boards and, as such, can only be connected to the NB2DSK01 using the 'PERIPHERAL BOARD A' or 'PERIPHERAL BOARD C' connectors.

 For more information on peripheral boards, see the section *Peripheral Boards*, in the document [TR0143 Technical Reference Manual for Altium's Desktop NanoBoard NB2DSK01](#). For information on the range of peripheral boards currently available and additional documentation specific to each, go to www.altium.com/nanoboard/resources.

Additional services

In addition to the user-available I/O on the motherboard and peripheral boards, the daughter board connectors provide pins for a series of other functions, including implementation of the NanoTalk communications protocol, power, and programming of the FPGA device. The following sections detail these additional signals, in relation to the daughter board connectors on the daughter board itself.

Hard JTAG signals

All daughter board devices that are JTAG-equipped are connected to signals FPGA_TMS, FPGA_TCK, FPGA_TDI and FPGA_TDO. This allows the NB2DSK01 and Altium Designer to address the daughter board hardware using the JTAG protocol.

Soft JTAG signals

Four FPGA I/O pins are reserved for JTAG signals that are utilized by the FPGA design. Altium Designer uses JTAG IP to communicate directly with the FPGA fabric, allowing applications to be debugged live. These signals (NEXUS_TMS, NEXUS_TCK, NEXUS_TDI and NEXUS_TDO) are derived in the NB2DSK01's NanoTalk Controller, which is implemented in a Xilinx Spartan-3 on the motherboard.

 For more information on the JTAG communications, refer to the article [AR0130 PC to NanoBoard Communications](#).

Daughter board identification signals

FPGA devices from different manufacturers and families require differing auto-configuration processes, so it is necessary for the NanoTalk Controller to be able to identify the device family.

Board identification is achieved through use of a 1-Wire slave memory device located on the daughter board. This device carries more than enough storage capacity to hold information such as Board ID, physical Device ID, board revision, and so on. For more information, see the section [Board ID memory](#) later in this document.

To satisfy NanoBoard-NB1 compatibility requirements, four signals (`FPGA_ID0` . . `FPGA_ID3`) are hardwired on the daughter board to provide the required identification to the NanoTalk Controller. For the Xilinx Virtex-4 daughter board (DB36), these four signals are tied to `GND`, giving an `FPGA_ID` value of `0h`.

SPI bus interface


The NB2DSK01 SPI system involves a variety of SPI-compatible slave resources, located across the hardware system – on the NB2DSK01 motherboard itself and also on certain peripheral boards that plug in to the motherboard. These SPI resources are accessible by three distinct SPI masters:

- Altium Designer (via the parallel or USB connection)
- The firmware – more specifically a TSK3000A processor therein – loaded onto the motherboard's Spartan-3 FPGA device (the NanoTalk Controller)
- The design loaded onto the currently plugged-in daughter board FPGA device.

Providing the required SPI bus arbitration between the masters, and access to the SPI devices, is the NB2DSK01's SPI Controller. The Controller, which is part of the NanoBoard firmware, plays the role of multiplexer/router – determining which master has access to the SPI bus and which SPI slave device is selected for communications.

From an FPGA design perspective, the NB2DSK01's SPI Controller provides an SPI path from the daughter board to each of the SPI slave resources resident in the system.

Daughter board connector signals `SPI_DIN`, `SPI_DOUT`, `SPI_CLK`, `SPI_SEL` and `SPI_MODE` provide the required connectivity from the daughter board. During operation, the daughter board FPGA design communicates with the NB2DSK01's SPI Controller to establish a path between the design and a specific motherboard/peripheral board SPI device.

 For detailed information on the Desktop NanoBoard's SPI communications system, refer to the document [AP0163 SPI Communications on the Desktop NanoBoard NB2DSK01](#).

1-Wire[®] bus interface

A 1-Wire serial bus interface signal is provided (`ONE_WIRE_DB_PB`) which is connected through to each of the NB2DSK01's peripheral board connectors. This provides the ability to communicate directly from a processor in an FPGA design, with one or more slave 1-Wire compatible devices located across plugged-in peripheral boards (where such devices exist). As the 1-Wire bus is made available to all three peripheral board sites it allows development of a dedicated network of 1-Wire devices – a micro-LAN if you will.

Daughter board power signals

Daughter board connectors `HDR_T` and `HDR_B` provide two power supplies to the daughter board, as well as ground signals. The power supply voltages – sourced from the NB2DSK01 motherboard – are 5V and 3.3V.

Note: The maximum current available to the daughter board will depend upon the number and type of peripheral boards attached to the motherboard, as well as the power requirements of each individual rail. In any case, the total connector contact rating of 2.0A for both the 3.3V and 5V rails is not to be exceeded. The maximum return ground current for all rails should not exceed 3A.

Daughter board FPGA control signals

Daughter board connector `HDR_T` handles various signals between the NanoTalk Controller and the daughter board FPGA device, that are used to control that device. Such signals include detection of an FPGA device when a daughter board is plugged-in to the motherboard, as well as signals used to actually program the device with an FPGA design.

A word about changing daughter boards

When changing FPGA daughter boards, please take care not to damage the connectors that attach the daughter board to the NB2DSK01. The following procedure is recommended:

1. Ensure that the NB2DSK01's power is switched off.
2. If using a 3-connector daughter board, ensure that any screws affixing it to the motherboard are removed.
3. Grip the two sides of the daughter board between your thumb and fingers and gently pull the daughter board upwards. Gently rocking the daughter board from side to side can help loosen the connectors.
4. As the daughter board disengages, ensure that you keep the board parallel to the motherboard and pull it straight up until all connectors are fully disengaged.
5. Install a different daughter board by gently positioning it so that its connectors are aligned with the corresponding connectors on the motherboard – the posts of which slot through the holes in the daughter board.

If using a 2-connector daughter board, place its locator 'legs' into the corresponding holes in the motherboard.

Once located, press the daughter board firmly onto the NanoBoard. If using a 3-connector daughter board, such as the Xilinx Virtex-4 daughter board (DB36), you can secure it to the motherboard using the available threaded standoffs.

6. Switch the NB2DSK01's power on.

The system software interrogates the NB2DSK01 at regular intervals to determine the FPGA device installed. If you change daughter boards, the system will automatically detect the change and show the correct device in the **Devices** view. When the **Devices** view is active you can force the system to poll the NB2DSK01 by pressing the **F5** key.

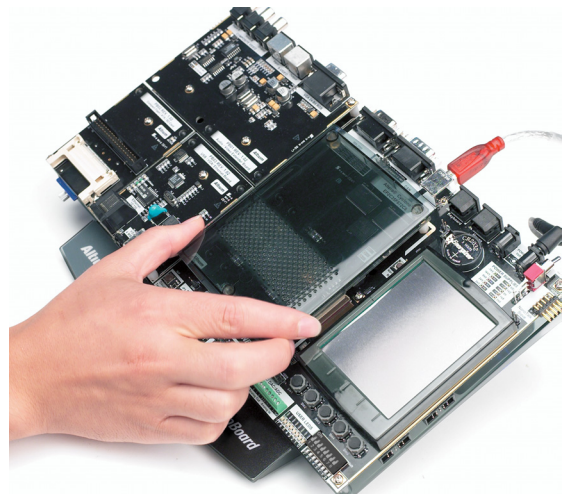


Figure 4. Switch daughter boards to change the target architecture.

Daughter board resources

The resources on the Xilinx Virtex-4 daughter board (DB36) can be divided into the following key areas:

- General board resources – including connectors, power supplies and LEDs
- Target FPGA device – the primary element of the board, into which an FPGA design is programmed
- User resources – resources made available for use by the FPGA design itself, such as memory storage devices.

The following sections take a closer look at these three areas.

Note: For each of the following resources, reference is made to the corresponding sheet(s) of the daughter board schematics on which the circuitry of interest can be found. A pdf document of these schematics – [DB36 Xilinx Virtex-4 \(XC4VLX25-FF668\) Daughter Board Schematics.pdf](#) – can be found at www.altium.com/nanoboard/resources.

General board resources

The following is a list of general resources found on the DB36.

Docking connectors

The DB36 is a 3-connector daughter board and, as such, has three 100-way Male connectors. These are used to connect the daughter board to the NB2DSK01 motherboard, which has corresponding 100-way Female connectors ('NANOCONNECT' interfaces).

The connectors on the daughter board are designated HDR_T, HDR_L and HDR_B

Note: HDR_T and HDR_B connectors are wired such that the daughter board may be attached to either the Desktop NanoBoard NB2DSK01 or the NanoBoard-NB1.

Location on board

The docking connectors (HDR_T, HDR_L and HDR_B) are located on the solder side of the board.

Schematic reference

The docking connectors can be found on sheet `DB_MotherBoardConnectors.SchDoc` (entitled *NB2 to Daughter Board Connectors*) of the daughter board schematics.

Board ID memory

Board identification is handled courtesy of a DS2406 device (from Dallas Semiconductor). Although the device is actually a dual-addressable switch, it is used for the additional 1kbit of memory that it possesses.

The DS2406 is a 1-Wire[®] compatible device, primarily used to contain a code with which to identify the daughter board when plugged in. The NanoTalk Controller interrogates this device over a single wire, the associated signal of which arrives at the Controller as `ONE_WIRE_DBID`.

The DS2406 supports half-duplex communication at a rate of up to 16.3kbits/s. Although power for the device is sourced over the same single wire as the data itself – courtesy of a parasitic capacitor which charges when the 1-Wire bus signal line is High – the device is also fed from the daughter board's 3.3V supply, to ensure continuous power.

Similar devices on all peripheral boards and 3-connector daughter boards allow the NanoTalk Controller to detect which specific boards are available to the system.

Location on board

The DS2406 device (designated `U1_ID`) is located on the solder side of the board, to the top-left of the 100-way docking connector `HDR_T`.

Schematic reference

The 1-Wire memory device can be found on sheet `1WB_DS2406_EPROM.SchDoc` (entitled *1-Wire Bus ID*) of the daughter board schematics.

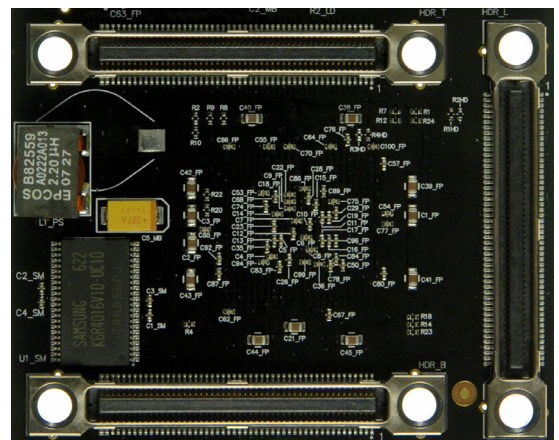


Figure 5. Docking connectors on the daughter board, used to attach the board to the NB2DSK01 motherboard.

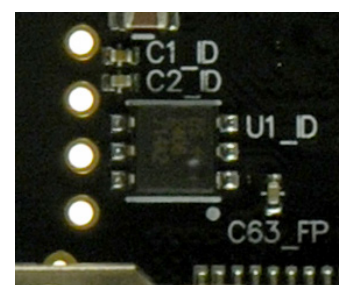


Figure 6. 1-Wire memory used to contain the ident code for the board.

Further device information

For more information on the DS2406 device, refer to the datasheet (DS2406.pdf) available at www.maxim-ic.com.

Power

In addition to the 3.3V and 5V supply voltages and ground signals fed to the daughter board from the NB2DSK01 motherboard, the daughter board also has its own 1.2V and 2.5V supplies.

The 1.2V supply is generated by passing the regulated 5V supply through a low-voltage step-down regulator – a MAX1831 device, from Maxim.

The 2.5V supply is sourced from the output of an adjustable linear voltage regulator – a MAX8860 device, from Maxim. Powered by the DB36's 5V supply, the output level is determined by the resistance between the device's OUT and SET pins, and is calculated using the following formula:

$$V_{OUT} = 1.25 * (1 + RA/100K)$$

Where,

V_{OUT} is the voltage output of the device

RA is the resistance between the device's OUT and SET pins.

On the DB36, a resistance of 100K is used, giving a nominal analog power supply of 2.5V.

The daughter board provides an array of test points for checking the integrity of each of the board's power supplies, as well as GND.

Location on board

The MAX1831 device (designated U2) is located on the component side of the board, to the right of the Xilinx Virtex-4 device. The MAX8860 device (designated U11) is located on the solder side of the board.

The power supply test points (designated TP0 – TP4) are located on the component side of the board, to the right of the Flash memory device (U1_CM).

Schematic reference

The power supply-related circuitry can be found on the following sheets of the daughter board schematics.

- PSU_SchDoc (entitled *Power Supply Top Level*)
- PSU_MAX1831_1V2_ALT.SchDoc (entitled *Power Supply MAX1831 (1V2)*)
- PSU_MAX8860_ADJ.SchDoc (entitled *Power Supply MAX8860 (2V5)*)

Further device information

For more information on the MAX1831 and MAX8860 devices, refer to the datasheets (MAX1830-MAX1831.pdf and MAX8860.pdf) available at www.maxim-ic.com.

Power and Program LEDs

The daughter board provides the following two LEDs:

- **Power LED** – this will light (RED) when the daughter board is correctly plugged into the motherboard and the NB2DSK01's power is switched on. It signifies presence of the 3.3V supply to the board.
- **Program LED** – this will light (GREEN) when the target device on the daughter board has been successfully programmed with an FPGA design.

Location on board

The two LEDs, labeled 'POWER' (LED1) and 'PROGRAM' (LED2) respectively, are located at the bottom of the board, on the component side.

Schematic reference

The LEDs can be found on sheet DB_LEDS.SchDoc (entitled *Daughter Board LEDs*) of the daughter board schematics.

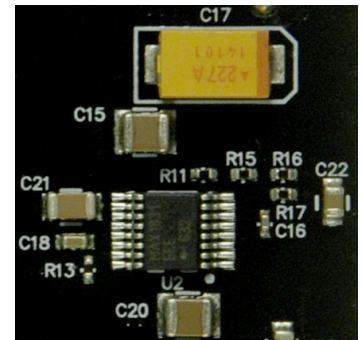


Figure 7. MAX1831 (U2) device.

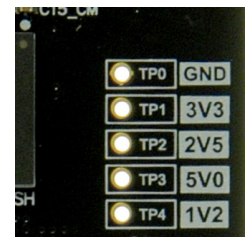


Figure 8. Power supply test points.

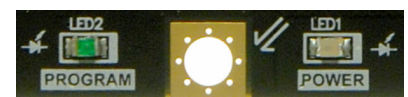


Figure 9. Power and Program LEDs.

Target FPGA device

The XC4VLX25-10FF668C device on the daughter board is a member of the Virtex-4 family of FPGAs. The Virtex-4 family is comprised of 17 devices in total, over three platform families – LX, SX and FX. The LX series, of which the daughter board device is a member, provide a high performance solution for logic applications.

The Virtex-4 family provides devices offering densities ranging from 12,312 to 200,448 Logic Cells. The XC4VLX25-10FF668C offers 24,192 Logic Cells. Table 1 provides an information summary for this device.

Note: In order to use the Virtex-4 device on this daughter board, you will need to install the relevant Vendor tools – Xilinx[®] ISE™ or Xilinx ISE WebPACK[®] from www.xilinx.com.



Figure 10. Xilinx Virtex-4 FPGA (XC4VLX25-10FF668C).

Table 1. Feature summary for the Virtex-4 FPGA device.

Feature	Description
Device Name	XC4VLX25-10FF668C
Vendor	Xilinx
Family	Virtex-4
Package	668-Ball Flip Chip Fine Pitch Ball Grid Array (FF668)
Speed Grade	10 (Lowest)
Temperature Grade	Commercial
Pin Count	668
Total I/O Banks	11
Maximum User I/O Pins	448
Max. Differential I/O Pairs	224
Max. High-Speed Differential I/O Pairs	0
Xilinx Logic Cells	24,192
CLB Array	2,688 CLBs (96 rows by 28 columns). (1 CLB = 4 Slices, giving 10,752 Slices).
Embedded (Block) RAM	1296K bits (72 dual-port RAM blocks, each 18K bits)
Distributed RAM	168K bits
Embedded Multipliers (18x18)	48 (each 18x18 multiplier is part of an XtremeDSP™ slice, which also includes an accumulator and an adder/subtractor).
Digital Clock Managers (DCM)	8
Phase Matched Clock Dividers (PMCD)	4
Global Clock Resources	32
Configuration Memory Required	8,037,312 bits
On-Chip Termination Support	Yes

Supported single-ended I/O standards

Table 2 lists the single-ended I/O standards supported by the XC4VLX25-10FF668C device.

Table 2. Supported single-ended I/O standards.

I/O Standard	Description
GTL	Gunning Transceiver Logic
GTLP	Gunning Transceiver Logic Plus
HSTL_I	High-Speed Transceiver Logic (1.5V) Class I
HSTL_II	High-Speed Transceiver Logic (1.5V) Class II
HSTL_III	High-Speed Transceiver Logic (1.5V) Class III
HSTL_IV	High-Speed Transceiver Logic (1.5V) Class IV
HSTL_I_18	High-Speed Transceiver Logic (1.8V) Class I
HSTL_II_18	High-Speed Transceiver Logic (1.8V) Class II
HSTL_III_18	High-Speed Transceiver Logic (1.8V) Class III
HSTL_IV_18	High-Speed Transceiver Logic (1.8V) Class IV
LVC MOS15	Low-Voltage Complementary Metal-Oxide Semiconductor (1.5V)
LVC MOS18	Low-Voltage Complementary Metal-Oxide Semiconductor (1.8V)
LVC MOS25	Low-Voltage Complementary Metal-Oxide Semiconductor (2.5V)
LVC MOS33	Low-Voltage Complementary Metal-Oxide Semiconductor (3.3V)
LV TTL	Low-Voltage Transistor-Transistor Logic (3.3V)
PCI33_3	Peripheral Component Interconnect (33MHz, 3.3V)
PCI66_3	Peripheral Component Interconnect (66MHz, 3.3V)
PCIX	Peripheral Component Interconnect Extended (133MHz, 3.3V)
SSTL18_I	Stub Series Terminated Logic (1.8V) Class I
SSTL18_II	Stub Series Terminated Logic (1.8V) Class II
SSTL2_I	Stub Series Terminated Logic (2.5V) Class I
SSTL2_II	Stub Series Terminated Logic (2.5V) Class II

Supported differential I/O standards

Table 3 lists the differential I/O standards supported by the XC4VLX25-10FF668C device.

Table 3. Supported differential I/O standards.

I/O Standard	Description
BLVDS_25	Bus Low-Voltage Differential Signaling (2.5V)
DIFF_HSTL_II	Differential High-Speed Transceiver Logic (1.5V) Class II
DIFF_HSTL_II_18	Differential High-Speed Transceiver Logic (1.8V) Class II
DIFF_SSTL18_II	Differential Stub Series Terminated Logic (1.8V) Class II
DIFF_SSTL2_II	Differential Stub Series Terminated Logic (2.5V) Class II
LDT_25	Lightning Data Transport (2.5V) – also referred to as Hypertransport™
LVDS_25	Low-Voltage Differential Signaling (2.5V)

LVDSEXT_25	Extended Low-Voltage Differential Signaling (2.5V)
LVPECL_25	Low-Voltage Positive Emitter-Coupled Logic (2.5V)
RSDS_25	Reduced Swing Differential Signaling (2.5V)
ULVDS_25	Ultra Low-Voltage Differential Signaling with Differential Termination (2.5V)

Supported single-ended DCI I/O standards

Table 4 lists the single-ended DCI (Digitally Controlled Impedance) I/O standards supported by the XC4VLX25-10FF668C device.

Table 4. Supported single-ended DCI I/O standards.

I/O Standard	Description
GTL_DCI	Gunning Transceiver Logic with Digitally Controlled Impedance technology
GTL_P_DCI	Gunning Transceiver Logic Plus with Digitally Controlled Impedance technology
HSLVDCI_15	High-Speed Low-Voltage Complementary Metal-Oxide Semiconductor (1.5V) with Digitally Controlled Impedance technology
HSLVDCI_18	High-Speed Low-Voltage Complementary Metal-Oxide Semiconductor (1.8V) with Digitally Controlled Impedance technology
HSLVDCI_25	High-Speed Low-Voltage Complementary Metal-Oxide Semiconductor (2.5V) with Digitally Controlled Impedance technology
HSLVDCI_33	High-Speed Low-Voltage Complementary Metal-Oxide Semiconductor (3.3V) with Digitally Controlled Impedance technology
HSTL_I_DCI	High-Speed Transceiver Logic (1.5V) Class I with Digitally Controlled Impedance technology
HSTL_II_DCI	High-Speed Transceiver Logic (1.5V) Class II with Digitally Controlled Impedance technology
HSTL_III_DCI	High-Speed Transceiver Logic (1.5V) Class III with Digitally Controlled Impedance technology
HSTL_IV_DCI	High-Speed Transceiver Logic (1.5V) Class IV with Digitally Controlled Impedance technology
HSTL_I_18_DCI	High-Speed Transceiver Logic (1.8V) Class I with Digitally Controlled Impedance technology
HSTL_II_18_DCI	High-Speed Transceiver Logic (1.8V) Class II with Digitally Controlled Impedance technology
HSTL_III_18_DCI	High-Speed Transceiver Logic (1.8V) Class III with Digitally Controlled Impedance technology
HSTL_IV_18_DCI	High-Speed Transceiver Logic (1.8V) Class IV with Digitally Controlled Impedance technology
LVDCI_15	Low-Voltage Complementary Metal-Oxide Semiconductor (1.5V) with Digitally Controlled Impedance technology
LVDCI_18	Low-Voltage Complementary Metal-Oxide Semiconductor (1.8V) with Digitally Controlled Impedance technology
LVDCI_25	Low-Voltage Complementary Metal-Oxide Semiconductor (2.5V) with Digitally Controlled Impedance technology
LVDCI_33	Low-Voltage Complementary Metal-Oxide Semiconductor (3.3V) with Digitally Controlled Impedance technology
LVDCI_DV2_15	Low-Voltage Complementary Metal-Oxide Semiconductor (1.5V) with Digitally Controlled Impedance technology supporting controlled impedance drivers with half impedance
LVDCI_DV2_18	Low-Voltage Complementary Metal-Oxide Semiconductor (1.8V) with Digitally Controlled Impedance technology supporting controlled impedance drivers with half impedance
LVDCI_DV2_25	Low-Voltage Complementary Metal-Oxide Semiconductor (2.5V) with Digitally Controlled Impedance technology supporting controlled impedance drivers with half impedance

SSTL18_I_DCI	Stub Series Terminated Logic (1.8V) Class I with Digitally Controlled Impedance Technology
SSTL18_II_DCI	Stub Series Terminated Logic (1.8V) Class II with Digitally Controlled Impedance Technology
SSTL2_I_DCI	Stub Series Terminated Logic (2.5V) Class I with Digitally Controlled Impedance Technology
SSTL2_II_DCI	Stub Series Terminated Logic (2.5V) Class II with Digitally Controlled Impedance Technology

Supported differential DCI I/O standards

Table 5 lists the differential DCI (Digitally Controlled Impedance) I/O standards supported by the XC4VLX25-10FF668C device.

Table 5. Supported differential DCI I/O standards.

I/O Standard	Description
LVDS_25_DCI	Low-Voltage Differential Signaling (2.5V) with Digitally Controlled Impedance Technology
LVDS_25_DCI	Extended Low-Voltage Differential Signaling (2.5V) with Digitally Controlled Impedance Technology
DIFF_HSTL_II_DCI	Differential High-Speed Transceiver Logic (1.5V) Class II with Digitally Controlled Impedance technology
DIFF_HSTL_II_18_DCI	Differential High-Speed Transceiver Logic (1.8V) Class II with Digitally Controlled Impedance technology
DIFF_SSTL18_II_DCI	Differential Stub Series Terminated Logic (1.8V) Class II with Digitally Controlled Impedance technology
DIFF_SSTL2_II_DCI	Differential Stub Series Terminated Logic (2.5V) Class II with Digitally Controlled Impedance technology

Location on board

The Virtex-4 device (designated U1) is located on the component side and in the lower half of the board.

Schematic reference

The Virtex-4 device and related circuitry can be found on the following sheets of the daughter board schematics:

- DEVICES.SchDoc (entitled *FPGA, LEDs and SRAM Memory*)
- FPGA.SchDoc (entitled *FPGA Connections*)
- FPGA_NonIO.SchDoc (entitled *FPGA Power and Programming*)
- Bypass_FPGA_1V2.SchDoc (entitled *FPGA Bypass Capacitors for 1V2*)
- Bypass_FPGA_2V5.SchDoc (entitled *FPGA Bypass Capacitors for 2V5*)
- Bypass_FPGA_3V3.SchDoc (entitled *FPGA Bypass Capacitors for 3V3*).

Further device information

For more information on the XC4VLX25-10FF668C device, refer to the datasheet (ds112.pdf) available at www.xilinx.com.

User resources

The following sections detail the resources on the DB36 which are accessible by, and available for use within, a design running on the board's target FPGA.

Accessing resources from an FPGA design

Before taking a closer look at the FPGA-accessible resources on the daughter board, it is worth taking time to elaborate just exactly how access to these resources is made possible.

Normally you would use ports to connect from the nets in a design to the pins on the FPGA. However, since the connectivity from the FPGA to the memory components on the daughter board is fixed by the routing, there is no need to place ports and then define the net-to-pin mapping. Instead, Altium Designer provides special components that can be placed – allowing the daughter board's memory resources to be easily incorporated in designs. These components, which can be thought of as design interface components and are commonly referred to as port components, are available in the FPGA DB Common Port-Plugin integrated library (`\Library\Fpga\FPGA DB Common Port-Plugin.IntLib`).

The port components automatically establish connectivity between the resource and FPGA I/O pins, allowing the same design to be built for different FPGA devices from different manufacturers. They are placed on the top sheet of the FPGA design project (*.PrjFpg), instead of ports. They are recognized as being external to the FPGA design by the presence of the `PortComponent = True` parameter. They are automatically converted to ports during synthesis.

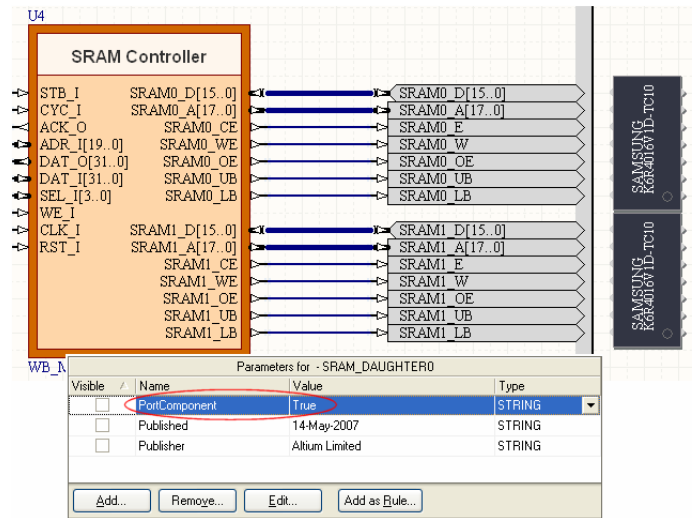


Figure 11. Example port components.

Converting a port component to standard ports

It can often be clearer to represent the signal lines to the physical pins of the FPGA device using standard ports, rather than the port components. Altium Designer provides a conversion command to quickly change a port component to standard ports. From the schematic document, this command can be accessed by:

- Choosing **Tools » Convert » Convert Part To Ports** from the main menus and clicking on the required port component you wish to convert.
- Right-clicking over the required port component and choosing **Part Actions » Convert Part To Ports** from the menu that appears.

After launching the command (and choosing the port component if applicable), the component will be converted into electrically equivalent ports.

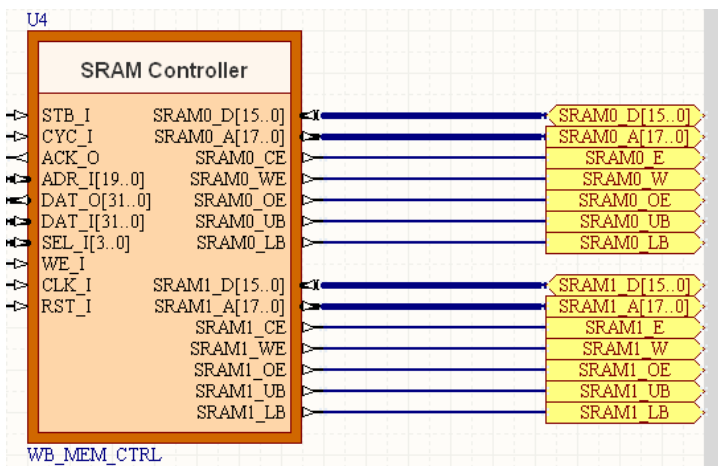


Figure 12. Converted port components.

Common-Bus SRAM

The DB36 includes Static RAM as part of the common-bus block of memory resources available to the FPGA.

The SRAM is provided in the form of two 4Mbit, high-speed CMOS SRAM devices. Each device is organized as 256K x 16 bits – combined together to give 256K x 32-bit storage (1MByte). Both devices are powered by the daughter board's 3.3V supply.

Although the devices require 18 address lines (SRAM_A[19..2]), the common bus interface makes provision for a nineteenth address line (SRAM_A20), giving the daughter board the flexibility to accommodate 512 x 16-bit devices, should they require to be fitted.

Location on board

The common-bus SRAM device U4_CM is located on the component side of the board, to the right of the common-bus SDRAM device U2_CM.

The common-bus SRAM device U5_CM is located in a corresponding position on the solder side of the board, to the left of the common-bus SDRAM device U3_CM.

Schematic reference

The common-bus SRAM devices can be found on sheet SRAM_256Kx32_TSOP44_1.SchDoc (entitled *256K x 32 SRAM – TSOP44 x 2*) of the daughter board schematics.

The common-bus memory block and interface wiring can be found on sheet NB2_CommonMemory.SchDoc (entitled *Common-Bus Memory Block*).

Common-Bus SDRAM

The DB36 includes Synchronous Dynamic RAM as part of the common-bus block of memory resources available to the FPGA.

The SDRAM is provided in the form of two MT48LC16M16A2TG devices (from Micron Technology). Each device is a 256Mbit, high-speed CMOS SDRAM, organized as 16M x 16 bits (4M x 16 bits x 4 banks) – combined together to give 16M x 32-bit storage (64MByte). Both devices are powered by the daughter board's 3.3V supply.

Location on board

The common-bus SDRAM device U2_CM is located on the component side of the board, to the left of the common-bus SRAM device U4_CM.

The common-bus SDRAM device U3_CM is located in a corresponding position on the solder side of the board, to the right of the common-bus SRAM device U5_CM.

Schematic reference

The common-bus SDRAM devices can be found on sheet SDRAM_MT48LC16M16A2TG_16Mx32.SchDoc (entitled *16M x 32 SDRAM TSOP54 x 2*) of the daughter board schematics.

The common-bus memory block and interface wiring can be found on sheet NB2_CommonMemory.SchDoc (entitled *Common-Bus Memory Block*).

Further device information

For more information on the MT48LC16M16A2TG device, refer to the datasheet (*256MSDRAM.pdf*) available at www.micron.com.



Figure 13. Common-bus SRAM (U4_CM).



Figure 14. Common-bus SRAM (U5_CM).



Figure 15. Common-bus SDRAM (U2_CM).



Figure 16. Common-bus SDRAM (U3_CM).

Common-Bus Flash memory

The DB36 includes Flash memory as part of the common-bus block of memory resources available to the FPGA.

The Flash memory is provided in the form of an S29GL256N11FFIV10 device (from Spansion). The Page Mode 256Mbit device is manufactured using 110nm MirrorBit™ technology and is organized as 16M x 16 bits (32MByte). It is powered by the daughter board's 3.3V supply and offers a 110ns access time.

Although this device can operate with 8-bit or 16-bit data bus widths, it has been fixed at 16-bit for the daughter board.

Location on board

The common-bus Flash memory device (designated U1_CM) is located on the component side of the board, directly below the common-bus SRAM and to the left of the power supply test points.

Schematic reference

The common-bus Flash memory device can be found on sheet FLASH_S29GL256N11FFIV10_16Mx16.SchDoc (entitled 16M x 16 Flash Memory (BGA)) of the daughter board schematics.


The common-bus memory block and interface wiring can be found on sheet NB2_CommonMemory.SchDoc (entitled Common-Bus Memory Block).

Further device information

For more information on the S29GL256N11FFIV10 device, refer to the datasheet (s29gl-n_00_b3_e.pdf) available at www.spansion.com.

Accessing common-bus memory devices

Table 6 summarizes the available design interface components that can be placed from the FPGA DB Common Port-Plugin.IntLib for access to, and communications with, any or all of the common-bus memory resources on the daughter board.

 Only ONE common-bus related port component can be placed in an FPGA design. The design interface component used will depend on which particular memory resource(s) you wish to access, and how you have configured the Shared Memory Controller – the intermediate design peripheral that sits between a processor in the design and the common-bus memory on the daughter board.

The Shared Memory Controller has an option to control the visibility of unused pins – in terms of its physical memory interface. By default, this option is enabled and all pins of the interface will be shown. In terms of wiring, it is more convenient to leave all interface pins visible and wire all pins to the corresponding ports of the SHARED_MEM_DAUGHTER port component. Those signals not required for the design, in accordance with the Controller's configuration, are internally handled. For example, active Low outputs relating to unused memory types are tied to '1', unused inputs are ignored and, if not using the SDRAM, the associated clock output signal will be tied to '0'.

However, you may not be using all three memory types in your design, or may prefer to hide the 'clutter' of pins associated with memory types that are not being used. You would then place the port component that truly corresponds to the memory you are wishing to access, only wiring from the Shared Memory Controller to those pins of relevance. For the remaining pins of the port component chosen, use the orange guidance text on each unused port to terminate it correctly:

- VCC – connect the unused port to VCC
- GND – connect the unused port to GND
- X – place a No ERC directive on the unused port.

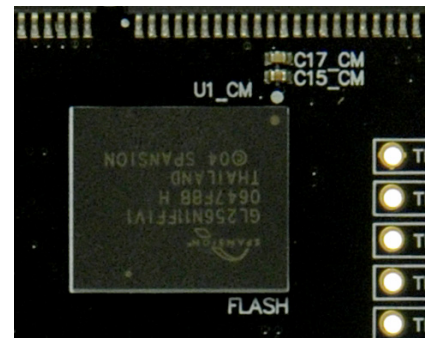

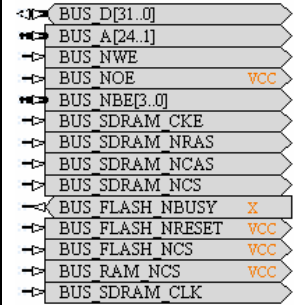
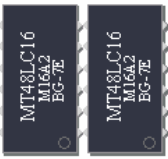
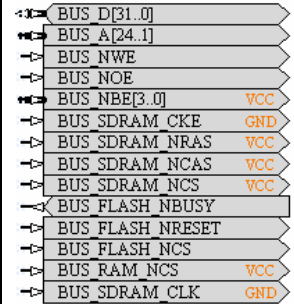
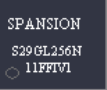

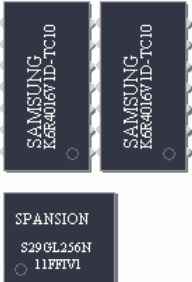


Figure 17. Common-bus Flash memory.

Table 6. Common-bus memory port-plugin components.

Component Symbol	Component Name	Description
	 <p>SHARED_SRAM_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus SRAM.</p>
	 <p>SHARED_SDRAM_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus SDRAM.</p>
	 <p>SHARED_FLASH_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus Flash memory.</p>
	 <p>SHARED_SRAM_SDRAM_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus SRAM and SDRAM.</p>
	 <p>SHARED_SRAM_FLASH_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus SRAM and Flash memory.</p>

<ul style="list-style-type: none"> ← BUS_D[31..0] ← BUS_A[24..1] ← BUS_NWE ← BUS_NOE ← BUS_NBE[3..0] ← BUS_SDRAM_CKE ← BUS_SDRAM_NRAS ← BUS_SDRAM_NCAS ← BUS_SDRAM_NCS ← BUS_FLASH_NBUSY ← BUS_FLASH_NRESET ← BUS_FLASH_NCS ← BUS_RAM_NCS VCC ← BUS_SDRAM_CLK 		<p>SHARED_SDRAM_FLASH_DAUGHTER</p>	<p>Place this component to interface to the daughter board's common-bus SDRAM and Flash memory.</p>
<ul style="list-style-type: none"> ← BUS_D[31..0] ← BUS_A[24..1] ← BUS_NWE ← BUS_NOE ← BUS_NBE[3..0] ← BUS_SDRAM_CKE ← BUS_SDRAM_NRAS ← BUS_SDRAM_NCAS ← BUS_SDRAM_NCS ← BUS_FLASH_NBUSY ← BUS_FLASH_NRESET ← BUS_FLASH_NCS ← BUS_RAM_NCS ← BUS_SDRAM_CLK 		<p>SHARED_MEM_DAUGHTER</p>	<p>Place this component to interface to all of the daughter board's common-bus memory resources.</p>

For information on available peripheral devices – used to provide the control interface between a processor in the design and the daughter board memory resource – go to www.altium.com/nanoboard/resources.

Independent SRAM

The DB36 also includes independent Static RAM as part of the memory resources available to the FPGA device, or more specifically the design running within. The term 'independent' is used in this case to distinguish this SRAM – which is interfaced using dedicated address and data lines – from the SRAM that is accessed over a common bus (also used to access on-board SDRAM and Flash memory).

The SRAM is provided in the form of two 4Mbit, high-speed CMOS SRAM devices. Each device is organized as 256K x 16 bits and powered by the daughter board's 3.3V supply.

The devices are accessed separately, giving two distinct 256K x 16-bit storage areas (512KByte each, 1MByte in total).

Although the devices require 18 address lines (SRAM_A[17..0]), the interface makes provision for a nineteenth address line (SRAM_A18), giving the daughter board the flexibility to accommodate 512 x 16-bit devices, should they require to be fitted.

Location on board

The independent SRAM device U1_SMA is located on the component side of the board, towards the bottom-right corner.

The independent SRAM device U1_SMB is located in a corresponding position on the solder side of the board, towards the bottom-left corner.

Schematic reference

The independent SRAM device can be found on sheet SRAM_256Kx16_TSOP44.SchDoc (entitled 256K x 16-Bit SRAM) of the daughter board schematics.

Design interface component

Table 7 summarizes the available design interface components that can be placed from the FPGA DB Common Port-Plugin.IntLib for access to, and communications with, the independent SRAM on the daughter board.

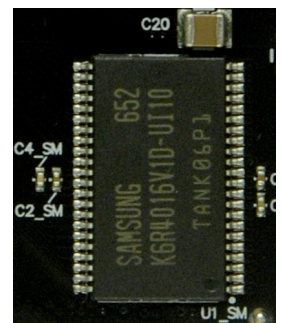


Figure 18. Independent SRAM (U1_SMA).

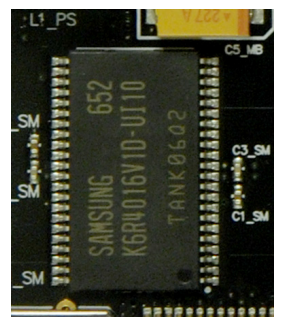
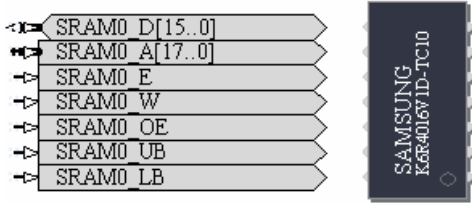
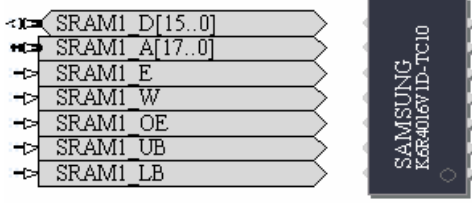


Figure 19. Independent SRAM (U1_SMB).

Table 7. Independent SRAM port-plugin components.

Component Symbol	Component Name	Description
	SRAM_DAUGHTER0	Place this component to interface to the independent SRAM device U1_SMA on the daughter board.
	SRAM_DAUGHTER1	Place this component to interface to the independent SRAM device U1_SMB on the daughter board.


For information on available peripheral devices – used to provide the control interface between a processor in the design and the daughter board memory resource – go to www.altium.com/nanoboard/resources.


Desktop NanoBoard NB2DSK01 Constraint system

The process of mapping or constraining a design to its physical implementation is done by creating constraint files – files that specify implementation detail such as the target device, the port-to-pin mapping, pin IO standards, and so on. The minimum information required to synthesize a design is the device specification.

Sets of constraint files are targeted to a design by creating a configuration, which is simply a named list of constraint files.

Setting up to implement a design on the NB2DSK01 is simplified through use of an auto-configuration feature, whereby a target configuration for the FPGA design project is automatically created. The required constraint files are automatically determined and added to this configuration, based on the IDs of the hardware (motherboard, daughter board and peripheral boards) in the system.

 For more information on the concept of configurations and constraints, and their role in design portability, refer to the article [AR0124 Design Portability, Configurations and Constraints](#).

 For more detailed information on the Desktop NanoBoard NB2DSK01 constraint system, including auto-configuration, refer to the application note [AP0154 Understanding the Desktop NanoBoard NB2DSK01 Constraint System](#).

Revision History

Date	Version No.	Revision
28-Apr-2008	1.0	Initial release

Software, hardware, documentation and related materials:

Copyright © 2008 Altium Limited. All Rights Reserved.

The material provided with this notice is subject to various forms of national and international intellectual property protection, including but not limited to copyright protection. You have been granted a non-exclusive license to use such material for the purposes stated in the end-user license agreement governing its use. In no event shall you reverse engineer, decompile, duplicate, distribute, create derivative works from or in any way exploit the material licensed to you except as expressly permitted by the governing agreement. Failure to abide by such restrictions may result in severe civil and criminal penalties, including but not limited to fines and imprisonment. Provided, however, that you are permitted to make one archival copy of said materials for back up purposes only, which archival copy may be accessed and used only in the event that the original copy of the materials is inoperable. Altium, Altium Designer, Board Insight, DXP, Innovation Station, LiveDesign, NanoBoard, NanoTalk, OpenBus, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed. v8.0 31/3/08.