

OEM Board Integration Guide



Start Developing Your Application On Day 1!

Embedded Artists AB

Södra Promenaden 51

SE-211 38 Malmö

Sweden

info@EmbeddedArtists.com

<http://www.EmbeddedArtists.com>

Copyright 2010 © Embedded Artists AB. All rights reserved.

No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, electronic, mechanical, magnetic, optical, chemical, manual or otherwise, without the prior written permission of Embedded Artists AB.

Disclaimer

Embedded Artists AB makes no representation or warranties with respect to the contents hereof and specifically disclaim any implied warranties or merchantability or fitness for any particular purpose. Information in this publication is subject to change without notice and does not represent a commitment on the part of Embedded Artists AB.

Feedback

We appreciate any feedback you may have for improvements on this document. Please send your comments to support@EmbeddedArtists.com.

Trademarks

All brand and product names mentioned herein are trademarks, services marks, registered trademarks, or registered service marks of their respective owners and should be treated as such.

Table of Contents

1	Document Revision History	4
2	Introduction	5
2.1	OEM Board Concept	5
2.2	Typical Development Process	5
3	Integration	7
3.1	SO-DIMM Connector	7
3.1.1	Handling and Mounting SO-DIMM Boards	8
3.2	Power Supply	9
3.3	Reset	9
3.4	Ethernet	10
3.4.1	Magnetic	10
3.4.2	PCB Trace Routing and Controlled Impedance	10
3.4.3	ESD Protection	11
3.5	USB	11
3.5.1	External Circuits	11
3.5.2	PCB Trace Routing and Controlled Impedance	11
3.5.3	ESD Protection	11
3.6	External Memory Bus	12
3.6.1	LPC3141 and LPC3152	12
3.6.2	LPC3250 and LPC1788-16/32	12
3.6.3	LPC2478-16/32	12
3.7	Operating Conditions	12
3.8	ESD/EMI Protection and EMC Directive	12
3.8.1	ESD/EMI Protection	13
3.8.2	EMC Directive	13
3.9	Memories	13
3.9.1	NAND FLASH Bad Block	13
3.9.2	Brand of Memory Chips	13

1 Document Revision History

<i>Revision</i>	<i>Date</i>	<i>Description</i>
A	2010-11-04	First version

2 Introduction

This manual describes the essential steps of integrating an Embedded Artists' *OEM Board* into a design. The advantages of using an *OEM Board* in a design are many:

- **Shorter design time – resulting in shorter time-to-market**
The core part of the hardware is pre-designed and verified, thereby eliminating substantial design issues and development risks.
- **Reduced development and production costs**
Embedded Artists' *OEM boards* have lower total cost for volumes up to 1-5K, when compared to custom development.
- **Allow focus on real application**
All resources can be allocated to the value adding application, as opposed to having to allocate resources to also develop common hardware.

Embedded Artists' *OEM Boards* can be deployed in a variety of embedded applications. They are design to be general control platforms. The small SO-DIMM form factor makes the *OEM Board* easy to integrate in a target system.

The *OEM Boards* come in different flavors;

- **Standard *OEM Boards*** for lower volume applications, typically up to 2000 pieces
- **Cost optimized versions** when 2000 pieces, or more, are needed.
- **Unique boards** for specific application needs based on re-use of Embedded Artists' proven designs.

2.1 OEM Board Concept

An *OEM Board* integrates all core components around a specific NXP processor. *OEM Boards* have a standardized form factor of 68 x 48 mm, based on the SO-DIMM standard. This standard is well-known and a proven high-speed signal interface connector that is commonly used in the industry (used for example in virtually all laptops for memory cards).

Pinning is not 100% compatible between different *OEM Boards*, but are quite similar in order to ease switch between boards (for example to upgrade to a more powerful processor).

When integrating an *OEM Board* into a design an application specific base board must be developed. The *Developer's Kit* contains a flexible base board that can serve as reference implementation. Another name for an application specific base board is a *carrier board* (for the *OEM Board*), but the term *base board* is used throughout this document.

A design utilizes only exactly what is needed on an *OEM Board*. Only interfaces and signals needed for a design need to be connected on the base board that is created. This flexibility makes it possible to create a dense and (cost) optimized base board. There are multiple interfaces given by the core processor that can be used, for example: Ethernet, USB, memory card, I2C, SPI, UART, memory bus and more.

2.2 Typical Development Process

Embedded Artists have created *Developer's Kits* for all OEM Boards. The kits make it easy to get started immediately with evaluation and prototyping work. A *Developer's Kit* contains a base board for simple *OEM Board* integration and has typically multiple display options. The picture below shows an example of the *LPC3141 Developer's Kit*. This specific kit also includes headphones since the *OEM Board* in the kit supports audio. Exact kit content varies from between kits.



Figure 1 – Typical Developer's Kit Content

The typical development flow to integrate an *OEM Board* into a target system is as follows:

1. Start with a *Developer's Kit* that contains both an *OEM Board* and a suitable base board. The kit makes it possible to start working with the system very quickly, typically from day 1.
2. Prototype and develop the core application. Utilize the software sample applications that exist for the *Developer's Kit*. Embedded Artists offers support packages to further speedup development.
3. When the specification for the new product is set, develop a base board for the specific end-application. This document contains information about what to consider when designing the specific base board.

3 Integration

This chapter addresses different interface and integration issues that must be addressed for an application specific base board design. The list of subjects presented can be viewed as a checklist that should be checked during design work.

3.1 SO-DIMM Connector

The *OEM Board* is based on the SO-DIMM form factor standard. A SO-DIMM connector is needed on the base board and there are many different models and manufacturers to choose from.

More specifically it is the DDR2 SO-DIMM standard that is used. This is also called to have a 1.8V keyed SO-DIMM board. There are also 2.5V keyed SO-DIMM boards and these are called DDR1 SO-DIMM boards.

A typical DDR2 SO-DIMM socket specifications looks like below (with minor variations between different models):

- Durability : 25 Cycles
- Voltage Rating: 25VAC
- Current Rating: 0.5A
- Contact Resistance: 50mΩ max.
- Dielectric Withstanding Voltage: 250V AC/1 min.
- Insulation Resistance: 50MΩ
- Operating Temperature: -40°C to +85°C

The table below lists some DDR2 SO-DIMM connectors from two manufacturers. The list is by no means complete and is only an example of what can be found. For detailed information about specific sockets, for example exact measures and recommended PCB layout, see the respective datasheet.

Part number	Socket height	Distance between base board and SO-DIMM board	Manufacturer
1565691-4	4.0 mm	1.7 mm	Tyco Electronics
AS0A426-N4SN-7F		Note that this is typically too small margin!	Foxconn
1473005-4	5.2 mm	2.9 mm	Tyco Electronics
AS0A426-N2SN-7F			Foxconn
1717468-4	6,5mm	4.2 mm	Tyco Electronics
AS0A426-N6SN-7F			Foxconn
1827341-4	8.0 mm	5.6 mm	Tyco Electronics
AS0A426-N8SN-7F			Foxconn
1612618-4	9.2 mm	6.8 mm	Tyco Electronics
AS0A426-NASN-7F			Foxconn

Always check height of components placed under the SO-DIMM socket on the base board and make sure that there is enough clearance. At least 1 mm clearance is recommended.

Total clearance under the SO-DIMM connector will be (see tables above and below for exact numbers):

'Distance between base board and SO-DIMM board' - 'Component height on bottom side'

OEM Board	Component height on top side	Component height on bottom side
LPC1788-16/32	TBD	TDB
LPC2478-16/32	2.0 mm	2.0 mm
LPC3141	1.4 mm	2.0 mm
LPC3152	2.1 mm	2.1 mm
LPC3250	2.6 mm	1.2 mm

Total height of SO-DIMM connector and *OEM Board* will be:

'Distance between base board and SO-DIMM board' + 1.0 mm + 'Component height on top side'

3.1.1 Handling and Mounting SO-DIMM Boards

See picture below for instructions about how to mount/remove an *OEM Board*.

To install an *OEM Board*, align it to the socket (1). Push the board gently, and with even force between the board edges, fully into the socket (2). Then push the board down in a rotating move (3) until it snaps into place (4). The *OEM Board* shall lie flat and parallel to the base board.

To remove an *OEM Board*, spread the two arms of the SO-DIMM socket apart slightly. The board will pop up (5). Gently rise the board in a rotating move (6) and then extract the board from the socket (7). Apply even force between board edges when removing so that the board is removed parallel to the locking arms.

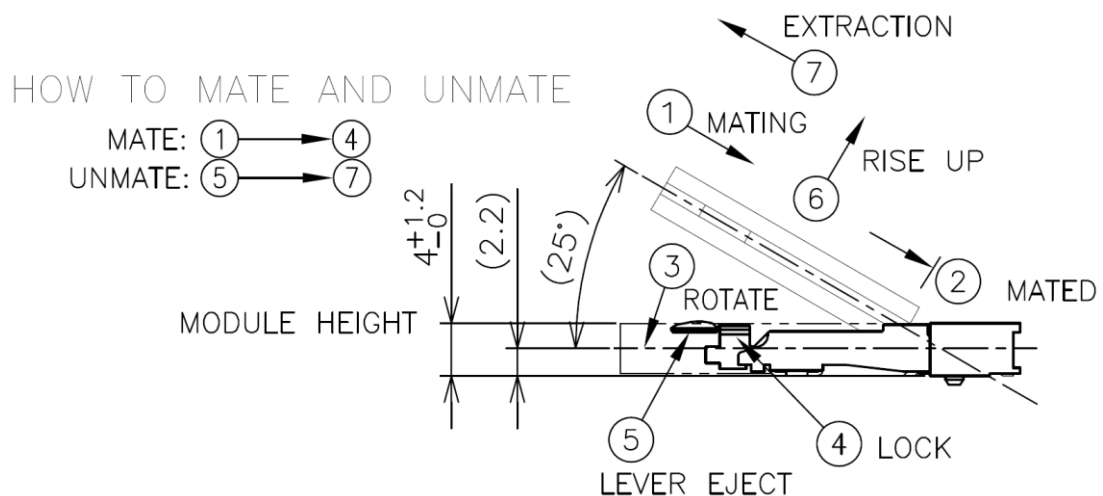


Figure 2 – Instructions how to Mount/Remove an OEM Board

Do not forget to follow standard ESD precaution routines when mounting/unmounting an *OEM Board*. Most signals exposed on the 200 edge contact fingers on the SO-DIMM board are unprotected. Maintain the same electrical potential of the *OEM Board* (to be mounted) and the base board. Do not

touch the *OEM Board* edge connectors. Handle the *OEM Board* only by the three other edges. Also, do not touch the components on the board.

3.2 Power Supply

The *OEM Board* needs a stable power supply. See the respective *OEM Board* datasheet for details about power supply levels. Normally it is 3.3V nominal, but there are also *OEM Boards* that operate on +5V nominal input voltage.

A linear power supply is recommended because of the lower noise ripple levels, but not strictly needed. A switched supply is also possible but it must be well filtered. Noise ripple on the supply voltage must be less than 50mV for frequency content up to 100 kHz. Above this frequency, ripple should be below 10mV. Voltage dips below the minimum level (or above the maximum level) specified in the *OEM Board* datasheet are not allowed and this includes noise ripple.

One 22µF low-ESR decoupling capacitor (tantal or ceramic) is recommended on the base board close to the SO-DIMM connector pins (37/39) related to power supply.

Two 22µF low-ESR decoupling capacitors (tantal or ceramic) is also recommended on the base board close to the SO-DIMM connector pins (165/199) related to power supply for buffers for memory bus expansion.

The power supply shall be designed for the maximum current consumption of the *OEM Board*, and should be able to deliver this at the maximum temperature for the system. The current consumption very much depends on how the processor and memories on the *OEM Board* are used. Current consumption is for example much higher when program executes from external SDRAM than from internal SRAM. Usage of Ethernet, USB and NAND FLASH also affect current consumption. For simplicity, the power supply can be designed for the maximum current consumption listed in the *OEM Board* datasheet. For designs requiring a more optimized design, it is recommended to measure the current consumption with the final application running. Then design the power supply with some reasonable margin.

The VBAT supply can supply the *OEM Board* processor during low-power/idle mode. It is normally for keeping a real-time clock running and keeping a small SRAM intact. Refer to the processor's datasheet for details about current consumption on VBAT and the valid voltage levels. Normally a small backup battery (lithium button cell, or similar), a rechargeable battery cell or a super-capacitor is connected to this supply. The duration requirements for the application determine what type of power supply to use.

3.3 Reset

The *OEM Boards* contains on-board reset generation on the supply voltage. There is no need from the *OEM Board* perspective to generate a reset signal. If there is an additional external reset source, the RESET_IN signal can be driven with an open-drain driver.

3.4 Ethernet

For *OEM Boards* with Ethernet interface, care must be observed when integrating this interface. Follow the guidelines in this section for smoothest integration. The design supports both 10 and 100Mbps Ethernet, i.e., IEEE 10BASE-T and 100BASE-TX standards.

3.4.1 Magnetic

The magnetic is an important component for the overall IEEE and emission conformance.

Suggested magnetic modules (integrated in RJ45) that are used on Embedded Artists base boards in *Developer's Kits* are Bel Stewart SI-50170F and HanRun HY931168C.

These magnetics has not been tested to verify proper operation under all circumstances. They have been evaluated by the content of the vendor supplied datasheet and legacy performance only. It can be assumed, with some degree of confidence, that with proper PCB layout the suggested magnetic will perform to high standards. For qualified magnetic (tested by the PHY vendor in order to verify proper operation) please refer to the respective Ethernet-PHY vendor of the *OEM Board* used (check schematics for details).

The picture below illustrated how to connect the SI-50170F connector (with integrated magnetic) from Bel Stewart.

RJ45 Ethernet Connector

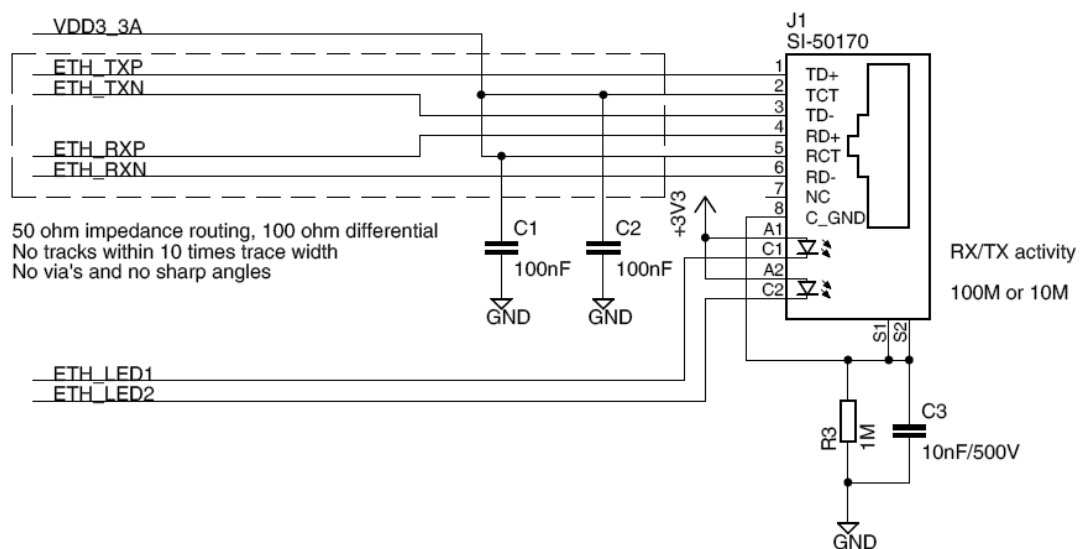


Figure 3 – Typical Ethernet Interface Design

3.4.2 PCB Trace Routing and Controlled Impedance

The transmit and receive data pairs shall be routed with 50 ohm impedance to ground and 100 ohm differential (per data pair). The differential pairs shall maintain symmetry in length, shape and number of vias. Vias shall be avoided as much as possible.

Controlled impedance for PCB traces are created by having a certain trace width and spacing between data pairs. Affecting factors (for width and spacing) are dielectric relative permittivity, trace thickness and distance to ground plane (i.e., dielectric height). Use an impedance calculator or consult a PCB manufacturer for details how to create PCB traces with controlled impedance.

The transmit and receive data pairs shall also be isolated from nearby circuitry and signals. As a general rule, no other tracks shall be within 10 times the trace width (or alternatively the distance between the data pair tracks).

Consult the following application note from SMSC for detailed guidelines:

AN 18-6 SMSC Ethernet Physical Layer Layout Guidelines

http://www.smsc.com/media/Downloads/Application_Notes/an186.pdf

3.4.3 ESD Protection

Depending on application, special ESD protection of the Ethernet interface can be needed. There are specific ESD protection diodes for Ethernet interfaces on the market, see for example the many different ESD protection diodes from NXP.

3.5 USB

For *OEM Boards* with USB interface, care must be observed when integrating this high-speed interface. Signal integrity is important to maintain.

3.5.1 External Circuits

Consult the *Developer's Kit* base board schematic for reference designs around the USB interfaces.

3.5.2 PCB Trace Routing and Controlled Impedance

Below is a list of general routing guidelines that will help minimize signal quality problems and maximize possibility for a successful design. The USB data pair (DP/DM) shall be routed with about 45-50 ohm impedance to ground and 90 ohm differential. This data pair traces are called the *USB differential traces* below.

- Route the USB differential traces with minimum trace lengths. Maintain maximum possible distance between high-speed clocks/periodic signals to USB differential traces (at least 50 mil distance). Low speed/non-periodic signal traces can be allowed somewhat closer to the data pair, but a minimum of 20 mil distance should always be observed.
- Route the USB differential traces using a minimum of vias and corners/turns, since this reduce signal reflections and impedance discontinuities.
- Avoid stubs on the USB differential traces since these will cause signal reflections and negatively affect signal quality.
- The USB differential traces shall maintain symmetry in length, shape and number of vias.
- Route all USB differential traces over continuous GND plane with no interruptions.
- Use 45° turns or an arc in order to reduce signal reflections and minimize impedance discontinuities. A 90° turn is for example created by using two 45° turns.
- The USB differential traces shall never be routed under crystals, oscillators, clock synthesizers, magnetic devices or other components that use and/or duplicate clocks.

Controlled impedance for PCB traces are created by having a certain trace width and spacing between data pairs. Affecting factors (for width and spacing) are dielectric relative permittivity, trace thickness and distance to ground plane (i.e., dielectric height). Use an impedance calculator or consult a PCB manufacturer for details how to create PCB traces with controlled impedance.

3.5.3 ESD Protection

Always add ESD protection diodes to an USB interface. See the base board design in *Developer's Kits* for recommended ESD protection design of the USB interfaces.

3.6 External Memory Bus

There are three different *OEM Board* principles how the buffers for external memory bus expansion are controlled.

3.6.1 LPC3141 and LPC3152

Signal N_ABUF_EN shall be grounded to enable the address buffers. It is not recommended to gate this signal with any logic. It should always be grounded when utilizing the external memory bus.

Signal N_DBUF_EN shall be driven by a logical XNOR (can be created with the 74LVC1G57 logic chip) of signals B_N_STCS0 and B_N_STCS1. This way, the address space covered by signals N_STCS0 and N_STCS1 is available for external expansion.

3.6.2 LPC3250 and LPC1788-16/32

The address buffers are always enabled; signal ABUF_EN is pulled low on the *OEM Board* to always enable the address buffers.

The databus buffers are automatically controlled by logic on the *OEM Board* for the static memory address ranges. For both LPC3250 and LPC1788 this covers CS0, CS1, CS2 and CS3.

Note that CS1 is used on the *LPC1788 OEM Board* for NAND flash chip select, so the address space covered by CS1 cannot be used for external expansion.

3.6.3 LPC2478-16/32

Signal N_ABUF_EN shall be grounded to enable the address buffers. It is not recommended to gate this signal with any logic. It should always be grounded when utilizing the external memory bus.

Signal DBUS_EN shall be driven by a logical AND of all external chip select (CS) signals that are used.

Note that both CS0 and CS1 are used on the *LPC2478 OEM Board* for NOR and NAND flash chip selects, so the address space covered by CS0 and CS1 cannot be used for external expansion. CS2 and CS3 can however be used.

Note that there is no failsafe control of the data DBUS_EN buffer enable signal. If it is pulled low wrong (i.e., enable the buffers) it will cause contention on the internal databus. In worst case it can damage the internal circuits on the *OEM Board*.

3.7 Operating Conditions

The temperature range is specified in the *OEM Board* datasheet. Normally it is 0-70 degrees Celsius. Extended temperature range is typically available on special request.

Note that self heating in an application can sometimes be significant (depending on ventilation and cooling). Always measure the temperature the *OEM Board* operates in when the complete system operates at its maximum temperature and make sure the maximum temperature limit is not exceeded.

Also make sure the relative humidity (RH) limits are met. The non-condensing requirement is important to meet. This can be a problem if temperature in the system is varying rapidly.

3.8 ESD/EMI Protection and EMC Directive

Many nations and market areas require commercial electronic products to meet certain electromagnetic compatibility (EMC) standards before they may be marketed and sold there. The European Union (EU), and to a growing extension other regions as well, also require commercial electronic products to meet certain electromagnetic interference (EMI) standards and electrostatic discharge (ESD) standards.

3.8.1 ESD/EMI Protection

In general it is very important to protect a design for the effects of ESD and EMI. External signals entering the base board, and eventually the *OEM Board*, must be properly protected from both ESD and EMI. Exactly what type of protection that is needed is very application dependent. The different standards must be consulted for details about needed protection.

3.8.2 EMC Directive

The goal of electromagnetic compatibility (EMC) is correct operation of a system (immunity of EMI) and the avoidance of generating unwanted effects to other systems (emission of EMI).

Note that the EU EMC Directive concerns primarily the finished products. In applying the Directive to components, the Guidelines [1] should be consulted to determine whether the component in question has a “direct function”. Electric motors, power supply units or temperature controls represent examples of such components with “direct function”. These types of components must be provided with a CE marking.

Components which are integrated into a device (not by the end-user), such as an *OEM Board*, do not have an independent function of their own. A given *OEM Board* may perform differing functions in different devices. Consequently, *OEM Boards* must be considered components without “direct function” which are not subject to the EMC Directive.

OEM Board shall not be labeled with a CE marking nor shall a declaration of conformity be issued within the scope of the EMC Directive. The end product that an *OEM Board* is integration into is however very likely to need CE marking and a declaration of conformity should be issued within the scope of the EMC Directive.

The *OEM Boards* have been designed according to best practice for reducing electro-magnetic emission with multilayer PCB, appropriate decoupling, component placement and trace routing. Measurements must however be performed on the final products and the result very much depends on the environment into which it is integrated to. Shielding around the product might be needed for compliance.

[1] Guidelines (version dated February 8, 2010) for the Application of the Council Directive 2004/108/EC, see http://ec.europa.eu/enterprise/sectors/electrical/emc/index_en.htm and http://ec.europa.eu/enterprise/sectors/electrical/documents/emc/guidance/index_en.htm

3.9 Memories

3.9.1 NAND FLASH Bad Block

Some *OEM Boards* have a NAND Flash memory. These chips may include invalid blocks when shipped from factory. Additional invalid blocks may develop while being used. Invalid blocks are defined as blocks that contain one or more bad bits. Do not erase or program factory-marked bad blocks. Refer to technical notes from chip manufacturer for appropriate management of invalid blocks.

3.9.2 Brand of Memory Chips

There is no guarantee for a certain brand or version of memory chips mounted on *OEM Boards*; serial/parallel NOR flash, parallel NAND flash and SDRAM. The lifetime of memory chips is limited and availability can also be limited from time to time. Embedded Artists make every effort to mount the original design chip on the board. In case that is impossible a compatible chip will instead be mounted without any prior notice. There can be small programming differences between mounted brands. The application program shall always read the chip id of flash devices to make certain which chip is actually mounted on the board.

The support page contains datasheets to the different memory devices and information about mounted devices on different board versions.