Freescale Semiconductor
Application Note

AN3444
Rev. 0, 10/2007

A Strategy for Routing the MPC8536E in a Six-Layer PCB

by *Michael George*
DSD Applications
Freescale Semiconductor, Inc.
Austin, Texas

This application note is a design guide to assist the customer in creating a low-layer, low-cost PCB design when using the MPC8536E device. Key items of discussion include assumed PCB stackup, power delivery to the device, and proper signal referencing. Additionally, layout plots for the device fan-out are also included.

NOTE

The schematic and Gerber data shown within this application note is intended merely to demonstrate the fan-out and power delivery strategy necessary to achieve a six-layer PCB. The schematic and Gerber data does not include all the decoupling, nor do they show all the necessary pull-ups and AC caps required outside the BGA fan-out area. This level of detail is captured as part of the MPC8536E development system.

Contents

1 Ballmap Organization

The MPC8536E is a 783-pin, 28 × 28 BGA array. The bus organization of the device is shown in Figure 1.

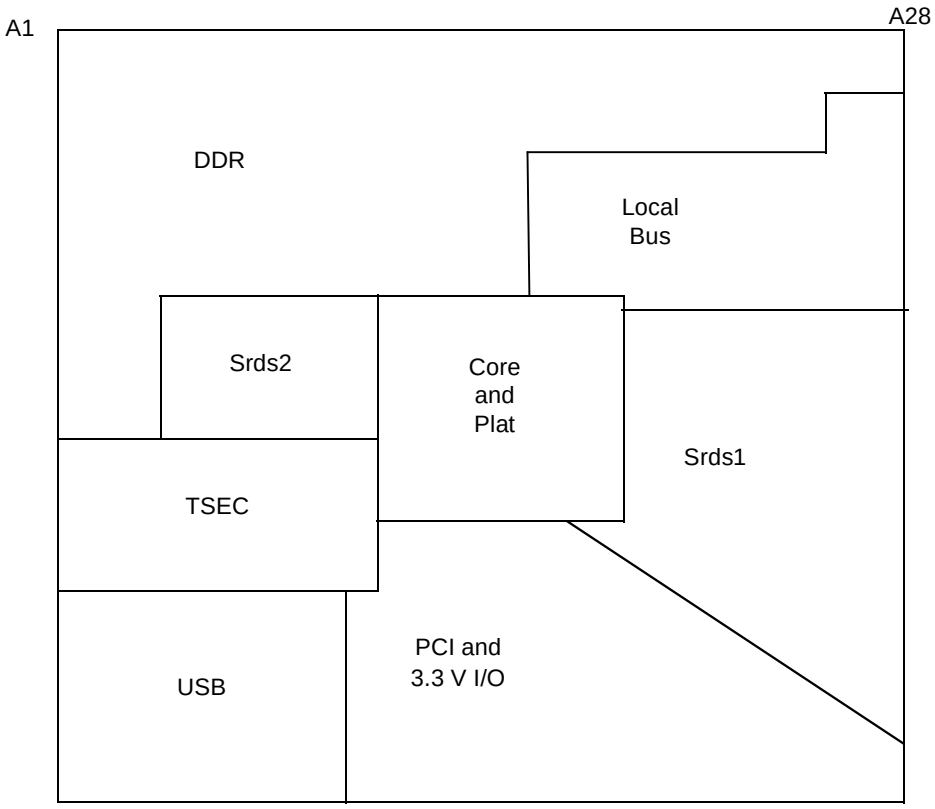


Figure 1. MPC8536E Bus Groupings
(Viewed from the top of the device)

2 Interface Support in Six Layers

Though the MPC8536E device can be fully broken out in six layers, there are a handful of signals that are not necessarily optimal from a signal integrity perspective, mainly due to splits in the reference plane. Such cases are very limited and are, therefore manageable. Table 1 enumerates the MPC8536E interfaces and whether there is proper signal referencing in the six-layer PCB fan-out example.

Table 1. Interface Support in Six Layers

Interface	Configuration	Reference Plane	Comment
DDR2	Full 64-bit + ECC	DDR Data—GND DDR ADDR—1.8 V DDR CLKs—1.8 V	Fully supported.
ETSEC	RGMII	RGMII I/O—2.5 V GTX_CLK125—GND	Both Ethernet ports can be fully routed in RGMII mode with proper signal referencing to the TV_{DD}/LV_{DD} power split, set at 2.5 V. Signals needed for GMII can be broken out, but have breaks in their reference plane.

Table 1. Interface Support in Six Layers (continued)

Interface	Configuration	Reference Plane	Comment
POR Config	All user defined POR pins are accessible.	GND and PWR	Fully supported. Some POR pins cross splits but since they are static, there is no concern.
PCI	Full 32-bit PCI with 3 GNTs and 3 REQs	GND	PCI_REQ[4:5] and PCI_GNT[4:5] not broken out on same layer as rest of the PCI bus.
LB	All signals are accessible	GND and PWR	LB_LAD[23:31] cross split boundaries. These signals are not edge sensitive, therefore imperfections in their transitions can be managed. Use of stitching caps could be used.
SERDES1	All eight lanes accessible	GND	Fully supported
SERDES2	Both lanes accessible	GND	Fully supported
IRQ	All IRQs (0–11) and IRQ_OUT useable	GND and PWR	Fully supported
SPI	All signals are accessible	GND	Fully supported
SDHC	All signals are accessible	GND	Fully supported
JTAG/COP	All signals are accessible	GND and PWR	Fully supported
DUARTS	Both UART ports are fully accessible	GND and PWR	Fully supported
USB	All three ports fully accessible	GND	Fully supported
Power pins and AV_{DD} filters	All unique powers accessible	N/A	Fully supported
I2C	Both I2C port accessible	GND and PWR	Fully supported
Clocking	SYSCLK, RTC, and DDRCLK accessible	GND and PWR	Fully supported
1588	All signals are accessible	GND and PWR	Signals cross split boundaries. Most customers do not use. Use of stitching caps could be used.
GPIO and miscellaneous signals	All signals are accessible	GND and PWR	Fully supported
DEBUG	TRIG_OUT	PWR	Signal crosses split boundaries. Noncritical debug signal. Use of stitching cap could be used.

3 Board Stackup Considerations

This section presents the considerations associated with the PCB and its stackup. Table 2 list the target impedances needed in a six-layer design.

Table 2. Target Impedance for a Typical MPC8536E Design

Interface	Connection	Target Impedance
DDR2	8536E-to-DDR2 memory	Single ended impedance = 55–60 Ω Differential impedance = 100 Ω ± 10%
SERDES1 (PCIe)	8536E-to-PCIe connector or device	MicroStrip Single ended impedance = 60 ± 15% Differential impedance = 100 Ω ± 20% Stripline Single ended impedance = 60 ± 15% Differential impedance = 100 Ω ± 15%
SERDES2 (SGMII/SATA)	8536E-to-connector or device	Differential impedance = 100 Ω ± 15%
All 8536E signals	8536E-to-connector or device	Single ended interface = 55–60 Ω
Other system Items:		
USB differential	USB Phy to connector	Differential impedance = 90 Ω ± 15%
Ethernet differential	Ethernet Phy to connector	Differential impedance = 100 Ω ± 15%

Additional considerations for the stackup include

- Must utilize high volume, low cost PCB technology
- Routing density
- Aspect ratio less than 10:1
- Drill size set at 10 mil
- Common core construction
- Proper signal referencing for all critical signals

3.1 Stackup Proposal

Figure 2 shows a viable stackup for achieving the target impedances noted in Table 2. The target card thickness used is 62 mils (±7 mils). All signal routing is done on the inner two signal layers, or on the top and bottom signal layers. No signal routing is performed on the power and ground layers.

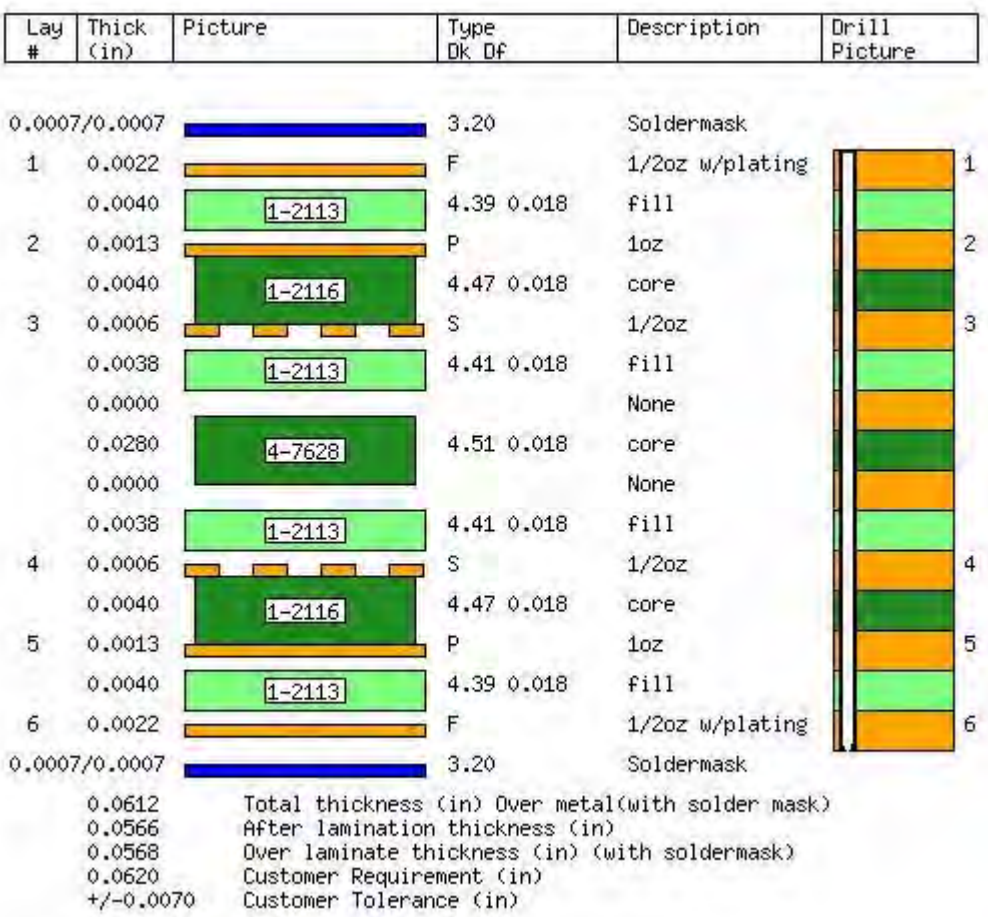


Figure 2. Example Six-Layer PCB Stackup (Provided by Sanmina-SCI; www.sanmina-sci.com)

Table 3. Impedance Information (Provided by Sanmina-SCI; www.sanmina-sci.com)

Layer	Type	Line Width	Center-to-Center	tpd (ps/in.)	Impedance (Ω)
1	Single-ended Microstrip	5 mil	n/a	153	55.15
1	Differential Microstrip	4 mil	8 mil	154	93.28
1	Differential Microstrip	5 mil	12 mil	154	96.54
3	Single-ended Stripline	4 mil	n/a	179	56.0
3	Differential Stripline	4 mil	10 mil	181	96.38
4	Single-ended Stripline	4 mil	n/a	179	56.0
4	Differential Stripline	4 mil	10 mil	181	96.38

Table 3. Impedance Information (Provided by Sanmina-SCI; www.sanmina-sci.com) (continued)

Layer	Type	Line Width	Center-to-Center	tpd (ps/in.)	Impedance (Ω)
6	Single-ended Microstrip	5 mil	n/a	153	55.15
6	Differential Microstrip	4 mil	8 mil	154	93.28
6	Differential Microstrip	5 mil	12 mil	154	96.54

4 Signal Breakout

4.1 Inside the BGA area

The key strategy used in breaking-out the MPC8536E is centered on the use of inexpensive through-hole vias and the use of two track routing for the inner and bottom layers (see Figure 4). For all mainstream PCB fabrication shops, this approach is considered “production-level” technology and is capable of being produced in high volumes. Additionally, this approach produces a cheaper overall PCB design, avoiding the additional cost associated with blind and buried type approaches.

The key items of the strategy are as follows:

- 1 mm (39.3 mil) ball pitch
- Plastic substrate
- 22 mil attach pad on top layer
- 19 mil via pad
- 28 mil anti-pad
- Single track routing on top layer (between attach pads)
- Two track routing on inner layers and on the bottom layer
  - Using 4 mil traces and 4 mil space

Figure 3 and Figure 4 show the track routing strategy used for each of the signal layers.

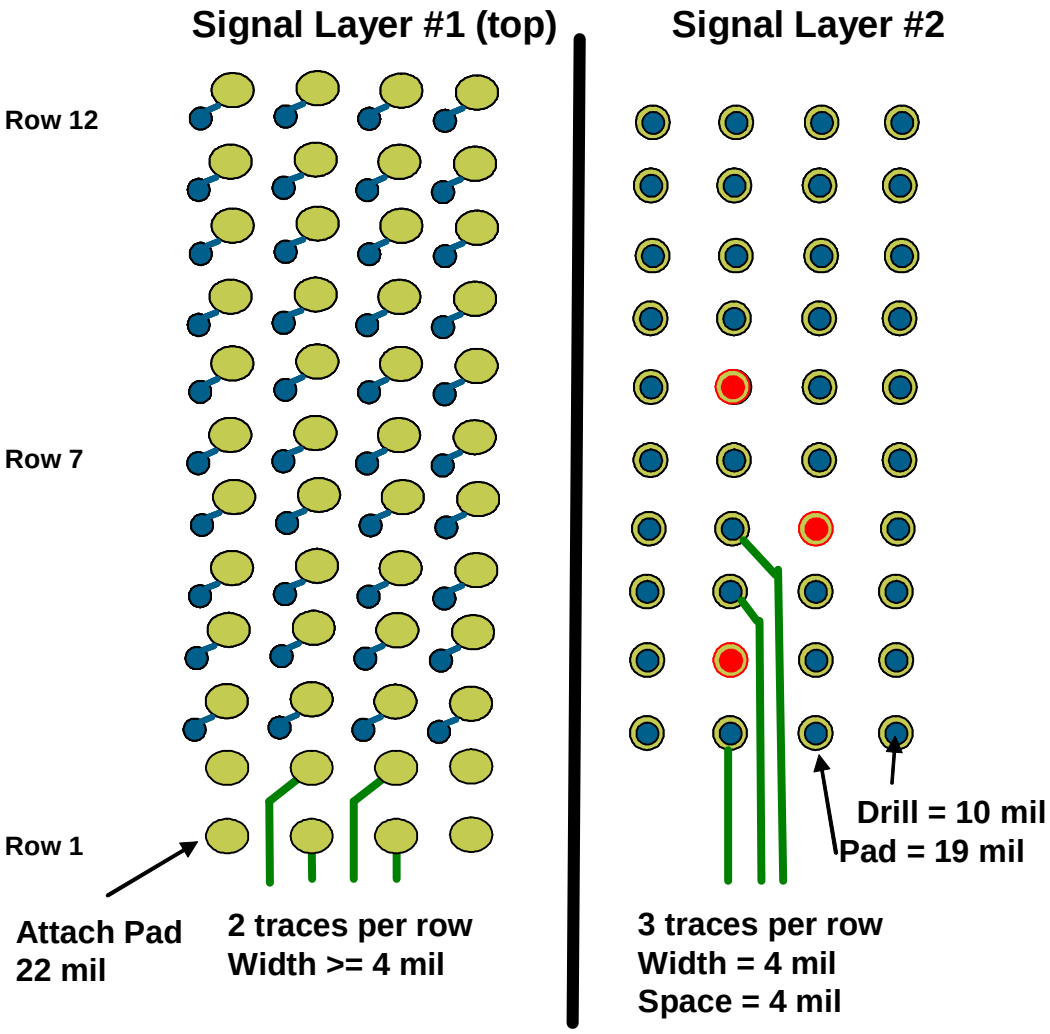


Figure 3. Signal Routing (Top and Layer #2)

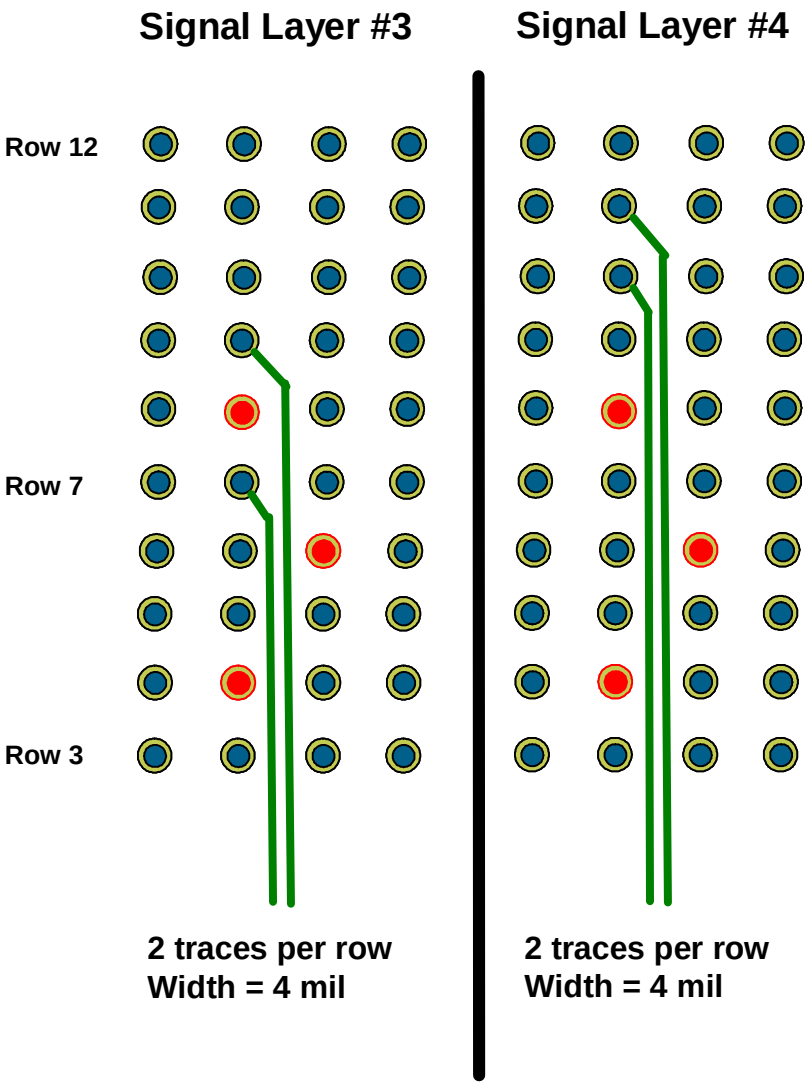


Figure 4. Signal Routing (Signal Layer #3 and Bottom)

4.2 Outside the BGA Area

Inside the BGA area, two track routing limits the minimum trace width to 4 mils and the minimum spacing to 4 mils. Outside the BGA area, this is not the case. Since wider traces have less dielectric loss, the designer may find it useful to use larger trace widths once outside the via array. Similarly, in the cases of differential pairs, the air gaps can be adjusted as needed in order to hit desired impedance targets.

4.3 Trace Spacing Outside the BGA Area

To minimize crosstalk opportunities once outside the BGA area, the spacing between differing signal groups must be observed. This spacing, denoted as ‘S,’ is based on the dielectric thickness ‘H’ (shown in Figure 5). For the stackup shown in Figure 2, the dielectric thickness is 4.0. Using a 3:1 spacing rule, the air gap between differing signal groups should be at least 12 mil.

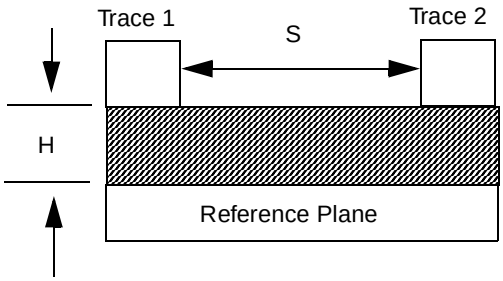


Figure 5. Trace Spacing Outside the BGA Area

5 Via Usage

For the six-layer PCB a 10-mil drill is used. A 10-mil drill has favorably cost advantages since smaller drill sizes incur additional cost. During fabrication, the drill tolerance is specified as +0/–3 mil, giving a finished hole size (FHS) of approximately 7–7.5 mil.

Using a 10-mil drill an aspect ratio of ~ 6:1 is realized. This aspect ratio is based on an 0.062-inch board thickness and is well within the high volume, production capabilities of all mainstream PCB vendors. Figure 6 depicts the via model used for the six-layer PCB.

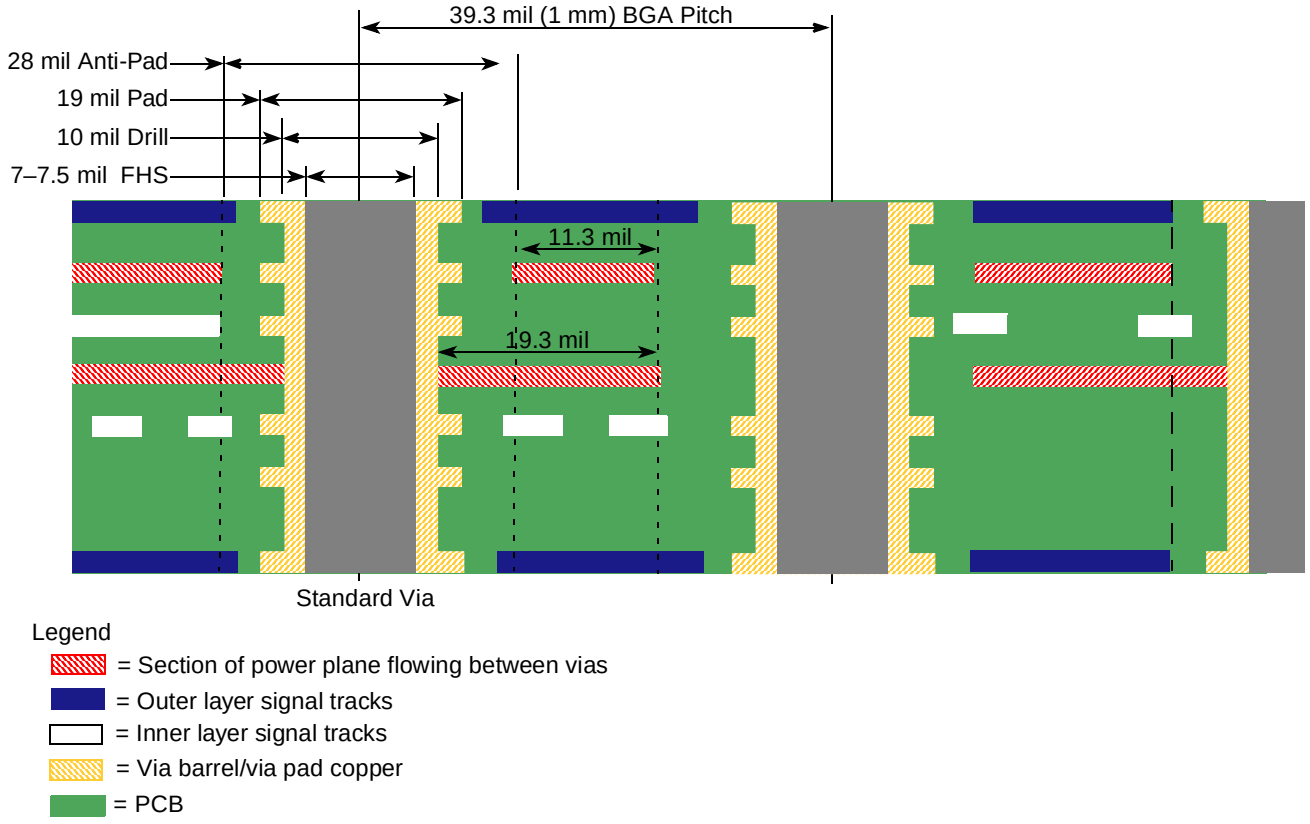


Figure 6. Cross Section of PCB

6 Power and Ground Strategy

The MPC8536E requires several power supplies for each subsystem (DDR, Ethernet, SerDes, and so forth) as well as the interconnect fabric itself (the platform voltage). The total number of supplies a system needs depends on the interfaces, the interface voltages required to connect with external peripherals, and the number of those voltages that are common and shareable.

Consult the *MPC8536E PowerQUICC™ III Integrated Processor Hardware Specifications* for accurate voltage and current requirements. For convenience, Table 4 provides a snapshot of the MPC8536E power requirements at the date of this application note. The table also reflects the unique power splits which where constructed in the six-layer PCB example (refer to the Power Rails column). In short, a total of seven unique power rails are used.

Table 4. MPC8536E Power Requirements [1]

Power Rails	Symbol	Typical Voltage	Tolerance	P_{MAX}
Core power	V_{DD_CORE} AV_{DD_CORE}	1.1 V or 1.0 V	±50 mV	7 W
Platform (internal buses)	V_{DD_PLAT} AV_{DD_PLAT} AV_{DD_PCI} AV_{DD_DDR} AV_{DD_LBIU} AV_{DD_SRDS} AV_{DD_SRDS2}	1.0 V	±50 mV	5 W
SerDes	SV_{DD} XV_{DD}	1.0 V	±50 mV	0.71W
DDR2 bus	GV_{DD}	1.8 V	±90 mV	2.5 W
Ethernet	LV_{DD} TV_{DD}	2.5 V	±125 mV	0.24 W
Local bus	BV_{DD}	3.3 V	±125 mV	0.33 W
Misc/PCI	OV_{DD}	3.3 V	±125 mV	0.33 W

1 These figures are subject to change without notice. Consult the latest version of the *MPC8536E PowerQUICC™ III Integrated Processor Hardware Specifications.*

Note that Table 4 lists the power requirements for the MPC8536E only; it does not include requirements for external devices, memory, and so forth. At the very least, additional power is needed to drive connected I/O pins, and the internal logic of the other devices often shared with the MPC8536E supplies.

6.1 Power Planes

The following pictures (Figure 7 through Figure 10) highlight the power strategy used for the six-layer PCB. The ground reference for the PCB is a solid ground plane at layer 2. Since it is a solid plane with no splits, the ground layer is not shown.

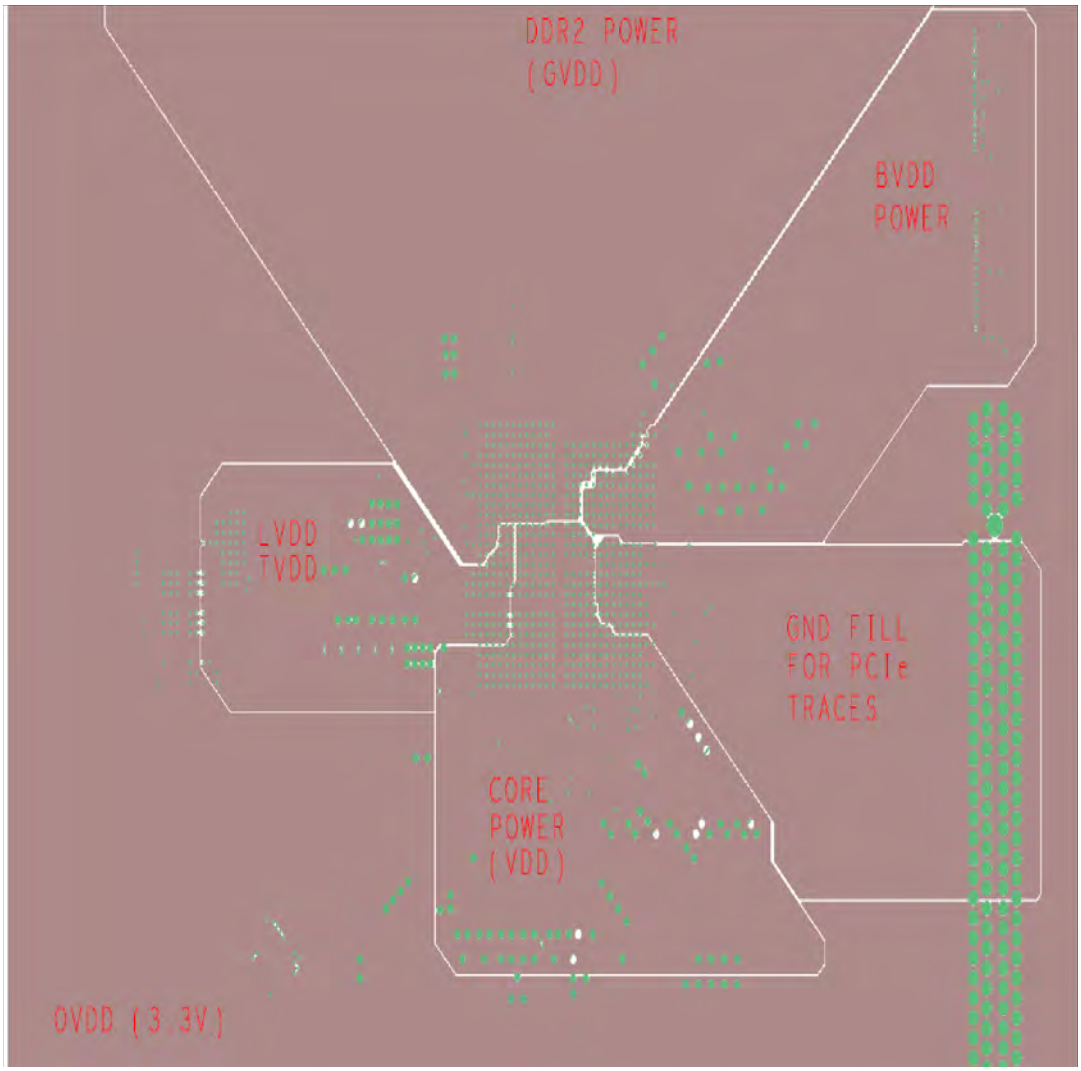


Figure 7. Power Plane (Layer 5 of 6)

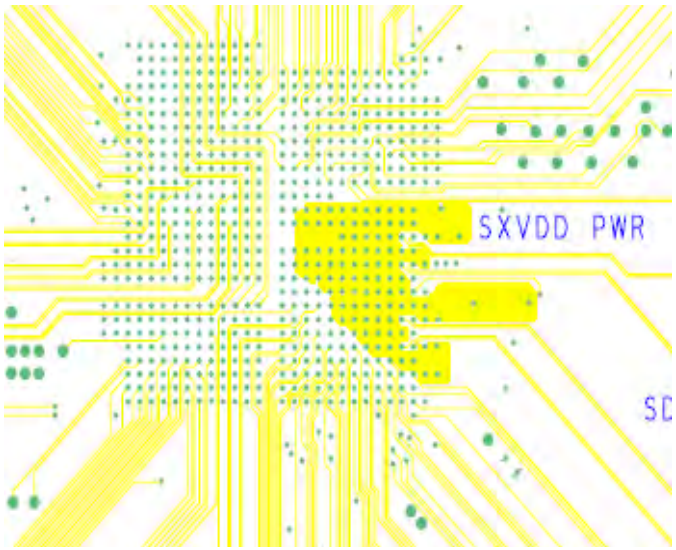


Figure 8. SXV_{DD} Power (Layer 2 of 6)

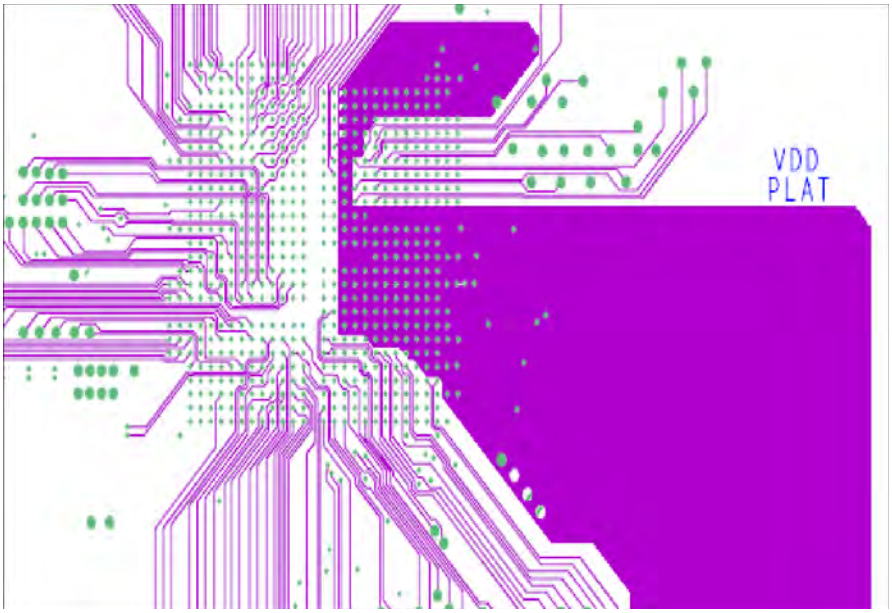


Figure 9. VDD PLAT (Layer 4 of 6)

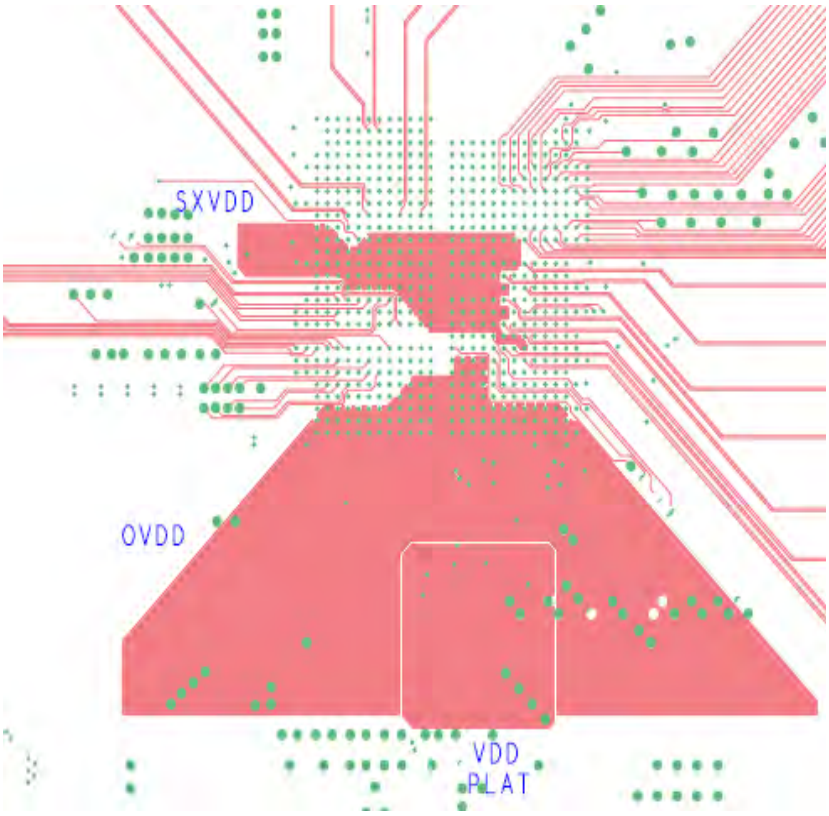


Figure 10. OV_{DD}, V_{DD_PLAT}, and Rest of SXV_{DD} (Layer 6 of 6—Bottom)

For sensitive supplies (such as XV_{DD} and SV_{DD}), a key requirement for power sharing is that the planes be relatively isolated. One effective strategy is to route the power planes separately and tie them together at the power source. See Figure 11.

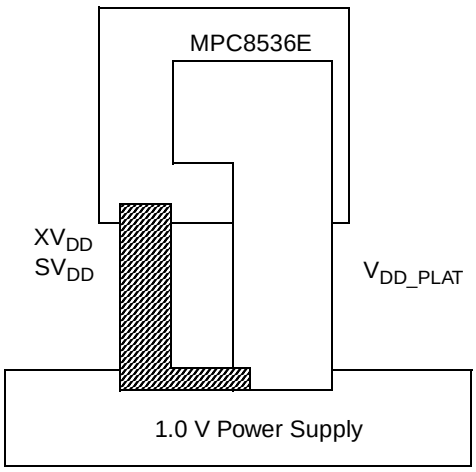


Figure 11. Common Source Split Planes

Another strategy is to use a low-pass filter to eliminate coupled noise, as long as there is sufficient current capacity in the components and connections. In either case, once separated, treat the two as independent power planes and route and bypass them separately.

6.2 Current Delivery

An important consideration in the six-layer PCB is ensuring all power rails have sufficient copper for proper current delivery. The two highest power rails on the MPC8536E are V_{DD_CORE} and V_{DD_PLAT}. The max power requirement for V_{DD_CORE} is 7 W and the max power requirements for V_{DD_PLAT} is 5 W.

Within a congested BGA array, the current travels down multiple 11-mil trace segments as it travels to the power pins, assuming the via stackup referenced in Figure 6 is used. The current-carrying capacity of these 11-mil traces is highly dependent on the current model chosen, either IPC2221A or the newer research performed by Johannes Adam of Flomerics Ltd. For greater detail on these two approaches see Section 9, "Current Delivery in the BGA Field."

In short, Table 5 depicts the current capacity differences between the two approaches assuming a maximum temperature rise of 20°C. Additionally, the last column highlights the current capacity assumed for the MPC8536E six-layer PCB.

Table 5. Current Capacity of an 11-mil Trace Adam's Model vs. IPC

Location	Copper Plating (oz.)	11-mil Trace (Adam's Model) (A)	11-mil Trace (IPC2221A) (A)	11-mil Trace (Capacity Assumed for the MPC8536E Six-Layer PCB) (A)
Outer	0.5	1.210	0.777	1.0
	1.0	1.711	1.285	1.5
	1.5	2.420	1.730	N/A
	2.0	3.422	2.130	N/A
Inner	0.5	1.149	0.390	0.75
	1.0	1.625	0.582	1.1
	1.5	2.299	0.865	N/A
	2.0	3.251	1.065	N/A

Taking the last column from Table 5 and the number of 11-mil channels achieved in the six-layer PCB, Table 6 highlights the current capacity for both V_{DD_CORE} and V_{DD_PLAT}.

Table 6. Current Capacity Achieved in the Six-Layer PCB

Power Rail	Copper Weight Used (oz.)	No. of 11-mil Traces Supplying Power	Current per Trace (A)	Total Current Supplied in Six-Layer PCB (A)	Total Current Required
V_{DD_CORE}	1.0	13	1.1	14.3	6.4 A (7 W/1.1 V)
V_{DD_PLAT}	0.5	11	0.75	8.25	5.0 A (5 W/1.0 V)

6.3 PLL Filters

The MPC8536E has six PLLs which each require a filter circuit. The PLL pins are placed on the package in order to minimize noise and to allow ease during layout. An example layout for the filters are shown in Figure 12. This figure shows the filters for AV_{DD_PCI}, AV_{DD_DDR}, AV_{DD_CORE}, and AV_{DD_PLAT}. The

residual filters would be similar. For the exact filter values refer to the *MPC8536E PowerQUICC™ III Integrated Processor Hardware Specifications.*

To ensure a high quality filter is realized, use the following rules:

- Place the components of the filter as close a possible to their respective pin
- Keep the traces as short as possible
- Use a 10-mil trace width or larger. An area fill is also a possibility.
- Maintain at least 20-mil clearance between the filter and other signals.

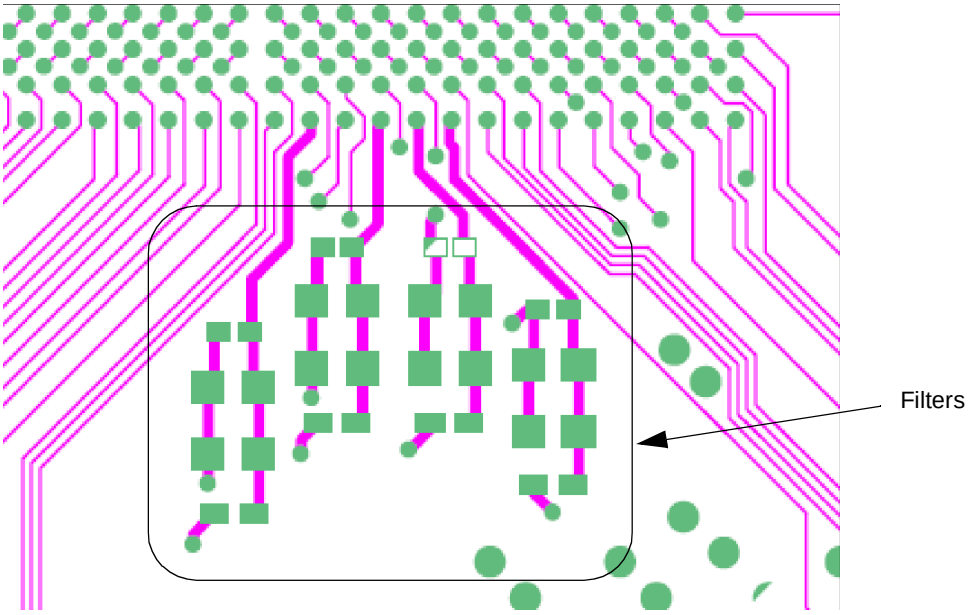


Figure 12. Example Layout for AV_{DD} Filters

6.4 Sharing Power Supplies

The six-layer design referenced in this application note assumes all processor powers referenced in Table 4 are supplied by unique power supplies. If the customer chooses, further sharing of power supplies can be achieved to reduce the number of supplies needed at the board-level. Table 7 shows some acceptable options for sharing MPC8536E power rails.

Table 7. MPC8536E Power Requirements

Power Source	Symbol	Voltage	Comments
Core power + PLL filter	V_{DD_CORE} AV_{DD_CORE}	1.0 V	If core and platform rails are shared, the upper core frequency is restricted. Additionally, there is no option for disabling the core power when the device is in 'deep sleep' state and the platform is alive. PLLs are sourced from RC filter circuit derived from 1.0 V rail.
Platform (internal buses) + other PLL filters	V_{DD_PLAT} AV_{DD_PCI} AV_{DD_DDR} AV_{DD_LBIU} AV_{DD_SRDS} AV_{DD_SRDS2}		
SerDes	SV_{DD} XV_{DD}		Separate filtered plane but derived from the same 1.0 V power regulator.
Memory	GV_{DD}	1.8 V	Unique rail on the MPC8536E device. Could possibly be shared with another 1.8 V device on the board.
Ethernet	LV_{DD} TV_{DD}	2.5 V	2.5 V is readily usable by most PHYs; however, 3.3 V is not suitable for RGMII mode.
Local bus	BV_{DD}		2.5 V requires low-voltage flash memory and so forth
Or alternatively			
Ethernet	LV_{DD} TV_{DD}	3.3 V	3.3 V is not suitable for RGMII mode
Local bus	BV_{DD}		
PCI/Misc	OV_{DD}		

6.5 Bypass Capacitor Placement

Because of the high current transients on the V_{DD_CORE} and V_{DD_PLAT} power pins, take care to bypass these power pins and to provide a good connection between the BGA pads and the power and ground planes. In particular, the SMD capacitors should have pads directly attached to the via ring (or even better, within it using via-in-pad methods). Figure 13 shows the dispersion of V_{DD_CORE} and V_{DD_PLAT} power pins along with the ground pattern.

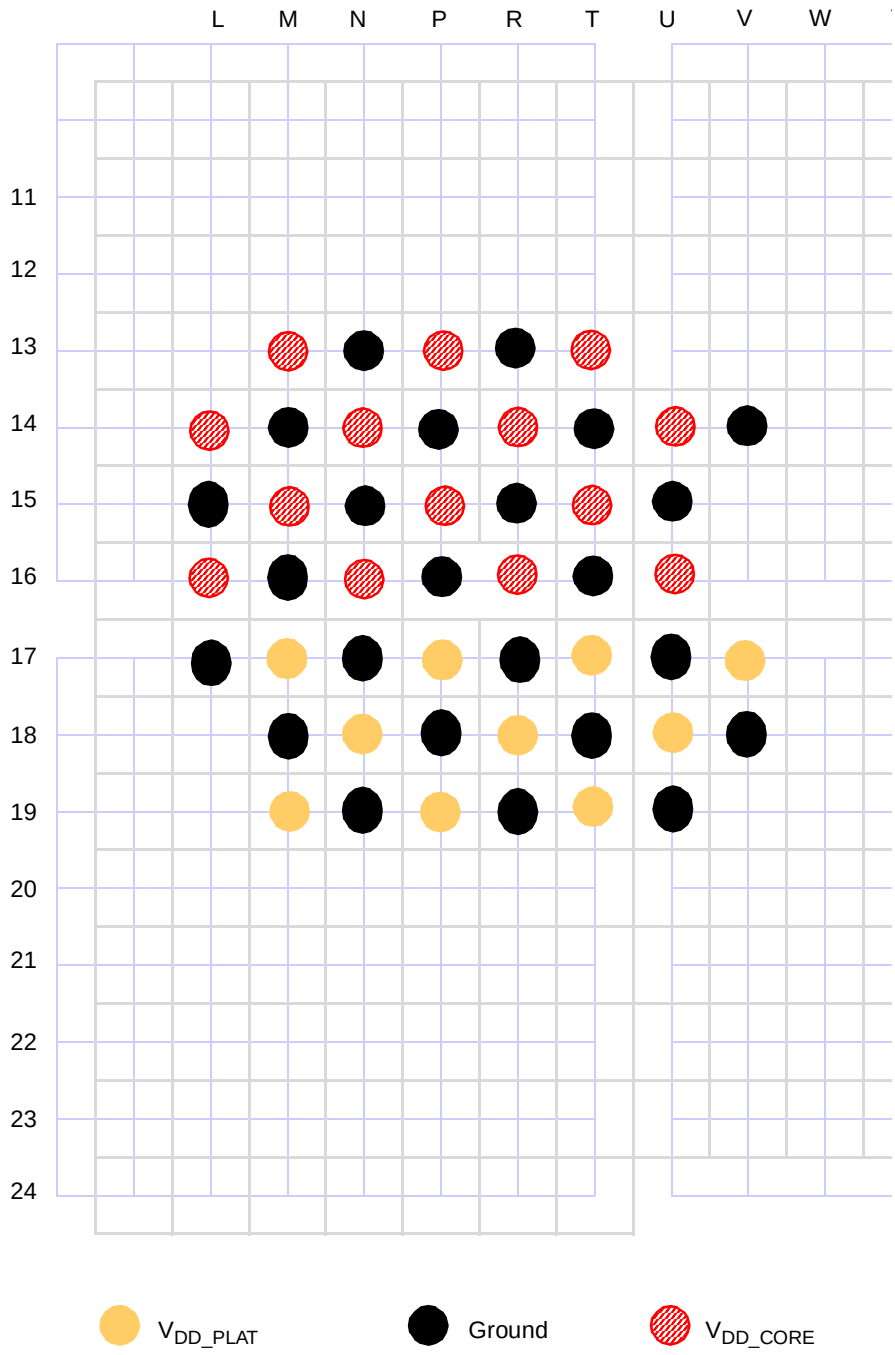


Figure 13. MPC8536E V_{DD_CORE}, V_{DD_PLAT}, and Ground Pattern (Bottom View)

NOTE

This is a bottom view of the center portion of interest within the BGA array.

After the power and grounds are escaped, the SMD 0402 capacitors can be placed so that the capacitor pads attach directly to the power vias on the side. Minimize or eliminate the trace between the via and the capacitor pad. Figure 14 shows three possible attachment methods.

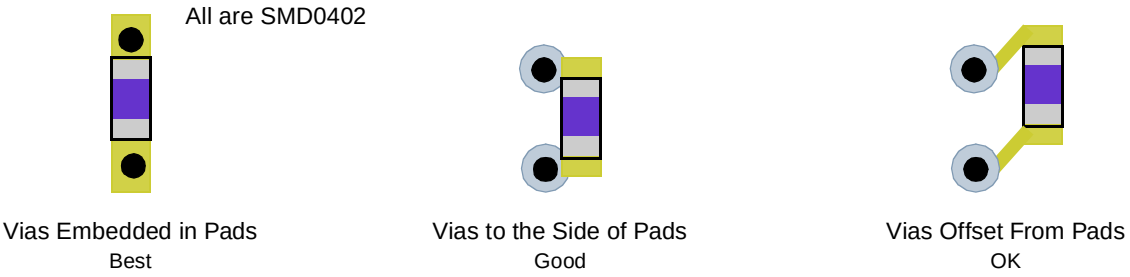


Figure 14. MPC8536E SMD Capacitor Via Placement

Using either of these methods, Figure 15 shows the relative attachment of the SMD 0402 capacitors. Again, this is a view through the top of the board.

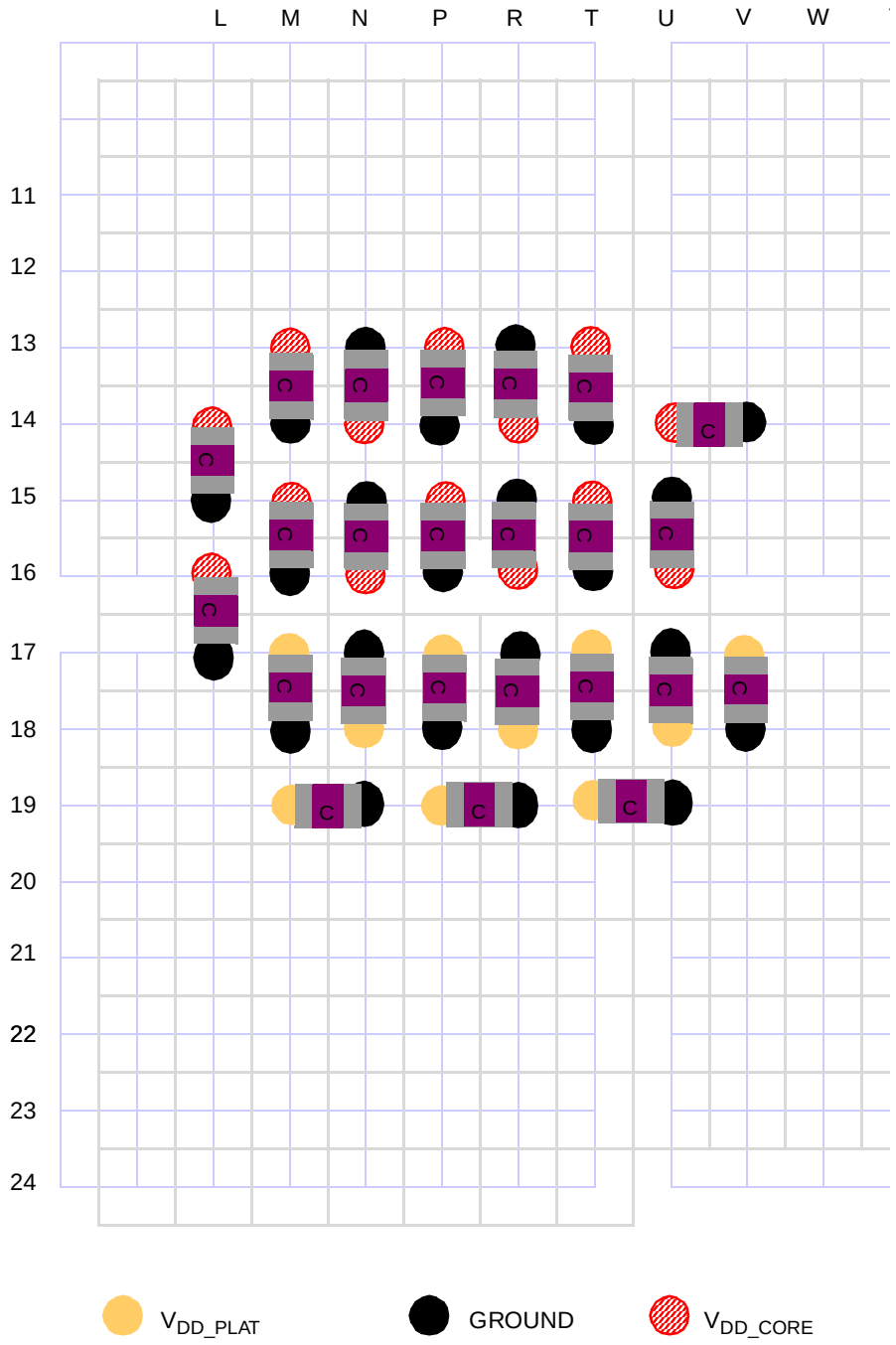


Figure 15. MPC8536E CORE, PLAT, and Ground SMD 0402 Capacitor Placement (Bottom View)

Other capacitors, such as those required for the other power rails, are not shown but can be attached in the same manner. If the PCB is not permitted to use via-in-pad connections, the capacitors can be shifted to attach to a pad in the area between the BGA pads. In that case, use the same relative pattern, but each capacitor shifts a little.

6.6 Bulk Capacitors

Bulk capacitors can be placed outside the periphery of the MPC8536E die, as close as reasonably possible. Systems with heatsinks and/or sockets probably need some additional spacing, but the high-frequency capacitors handle the fastest transients, allowing the bulk capacitors to be spaced a little further away. Keep them within 2 cm. Figure 16 shows the MPC8536E V_{DD_CORE} and V_{DD_PLAT} bulk capacitor placement.

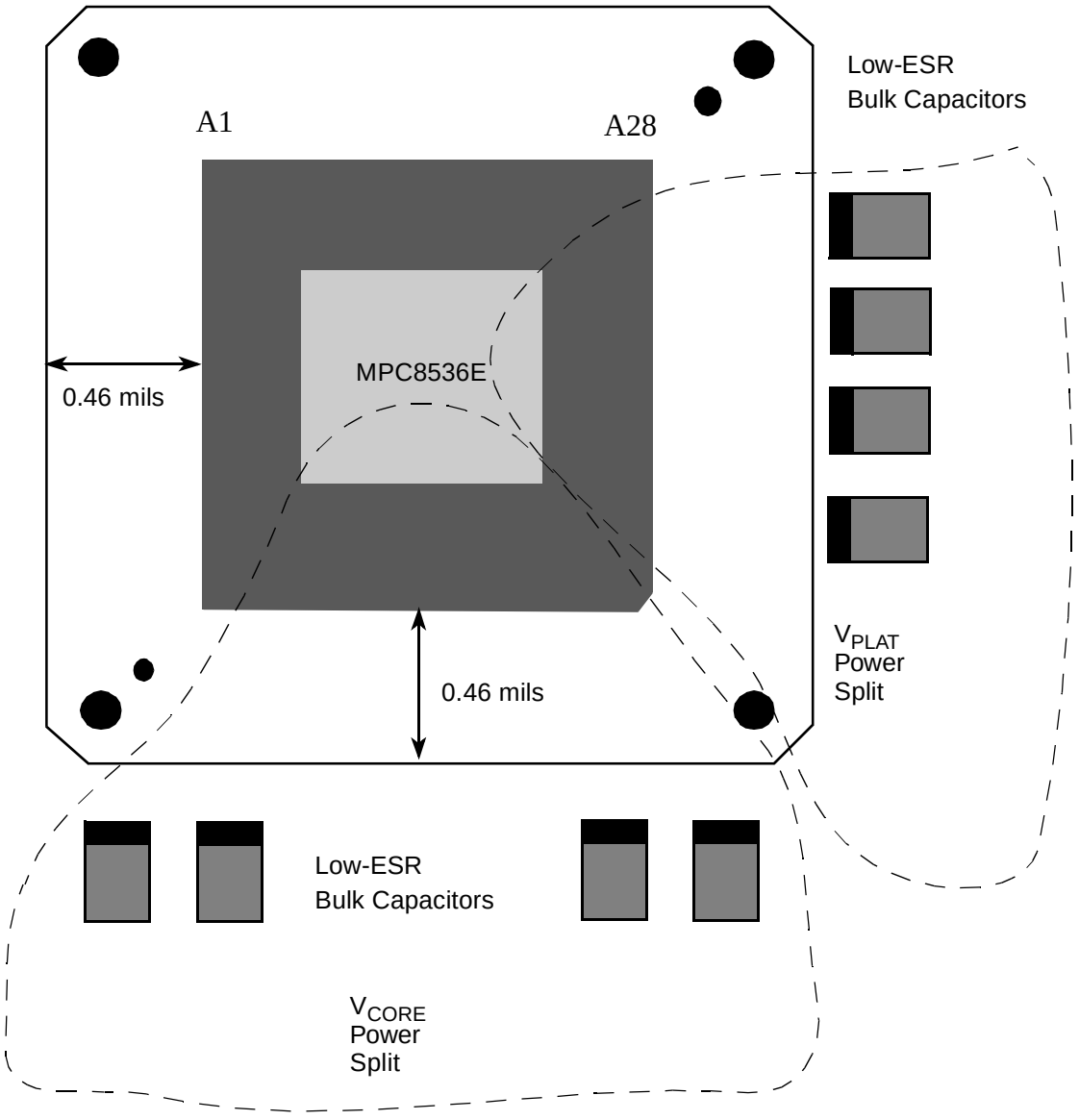


Figure 16. MPC8536E Bulk Capacitor Placement (Top View)

7 Signal Layer Gerber Plot

Figure 17 through Figure 20 show the signal layers for the six-layer PCB example. Again, the intent is to demonstrate that the MPC8536E signals can be broken in a six-layer PCB and does not include all the decoupling, nor does it show all the necessary pull-ups, AC caps, and other circuitry that would be required outside the BGA fan-out area. This level of detail is captured as part of the MPC8536E development system.

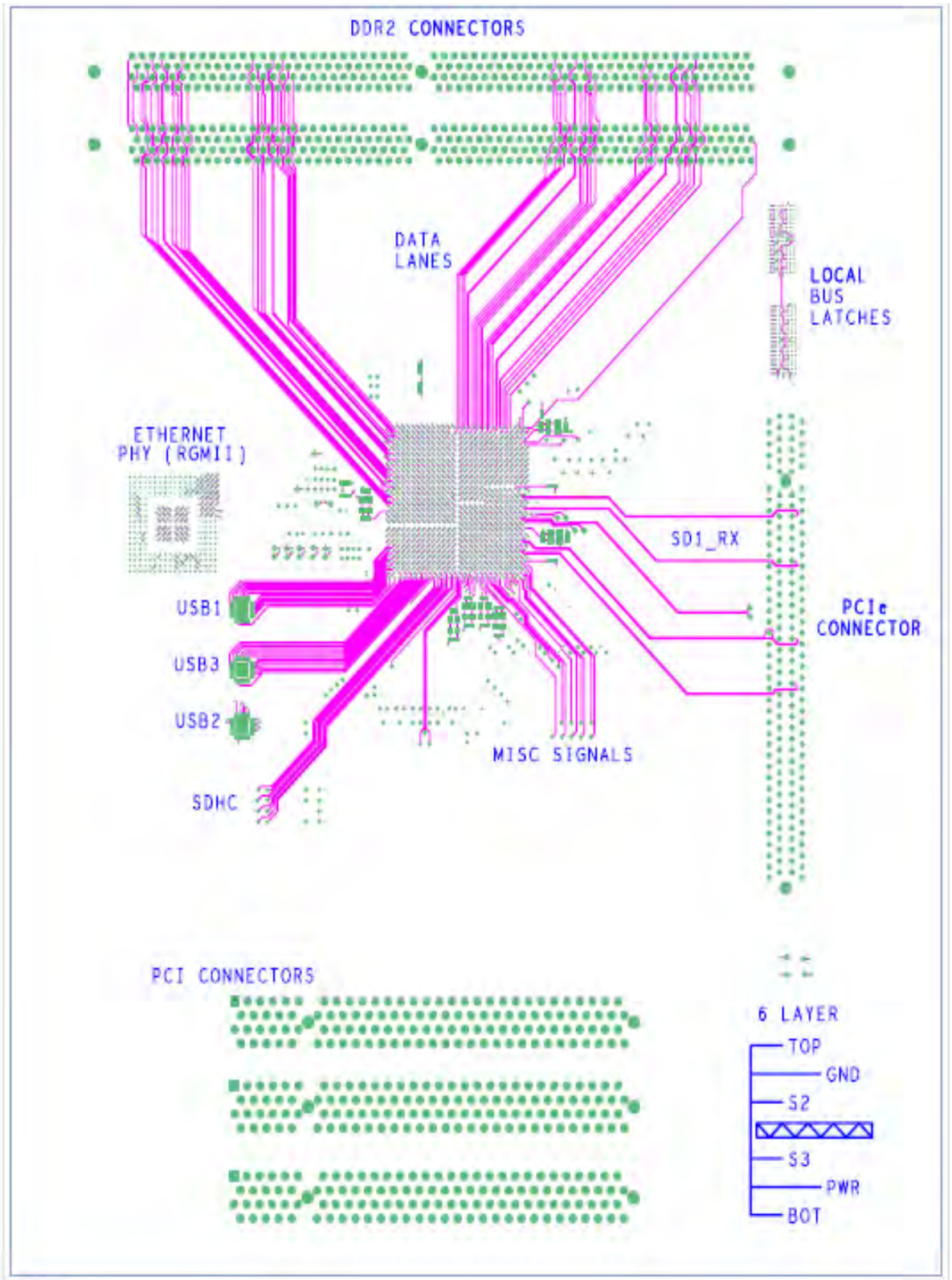


Figure 17. Top Gerber Plot (Layer 1 of 6)

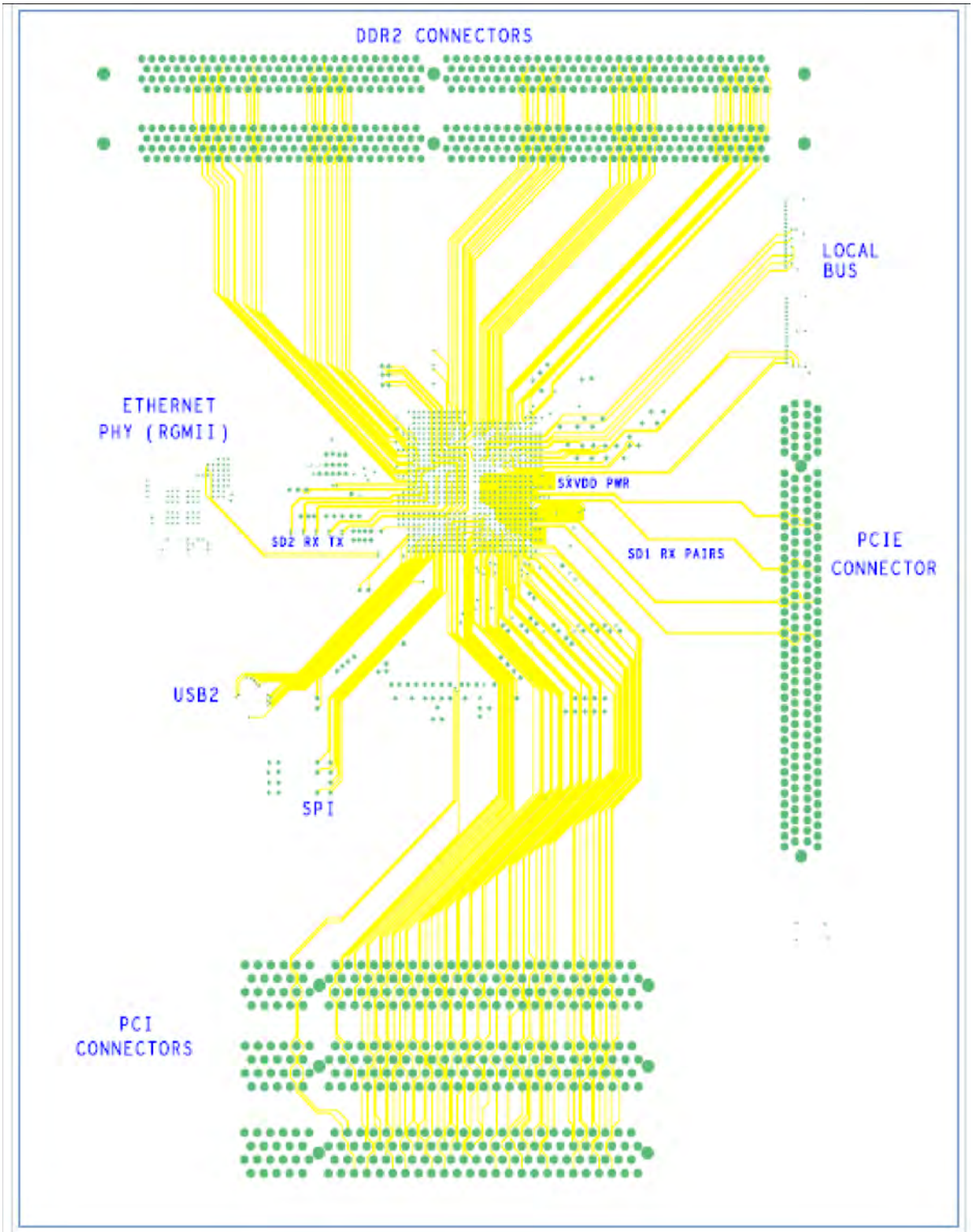


Figure 18. Signal #2 Gerber Plot (Layer 3 of 6)

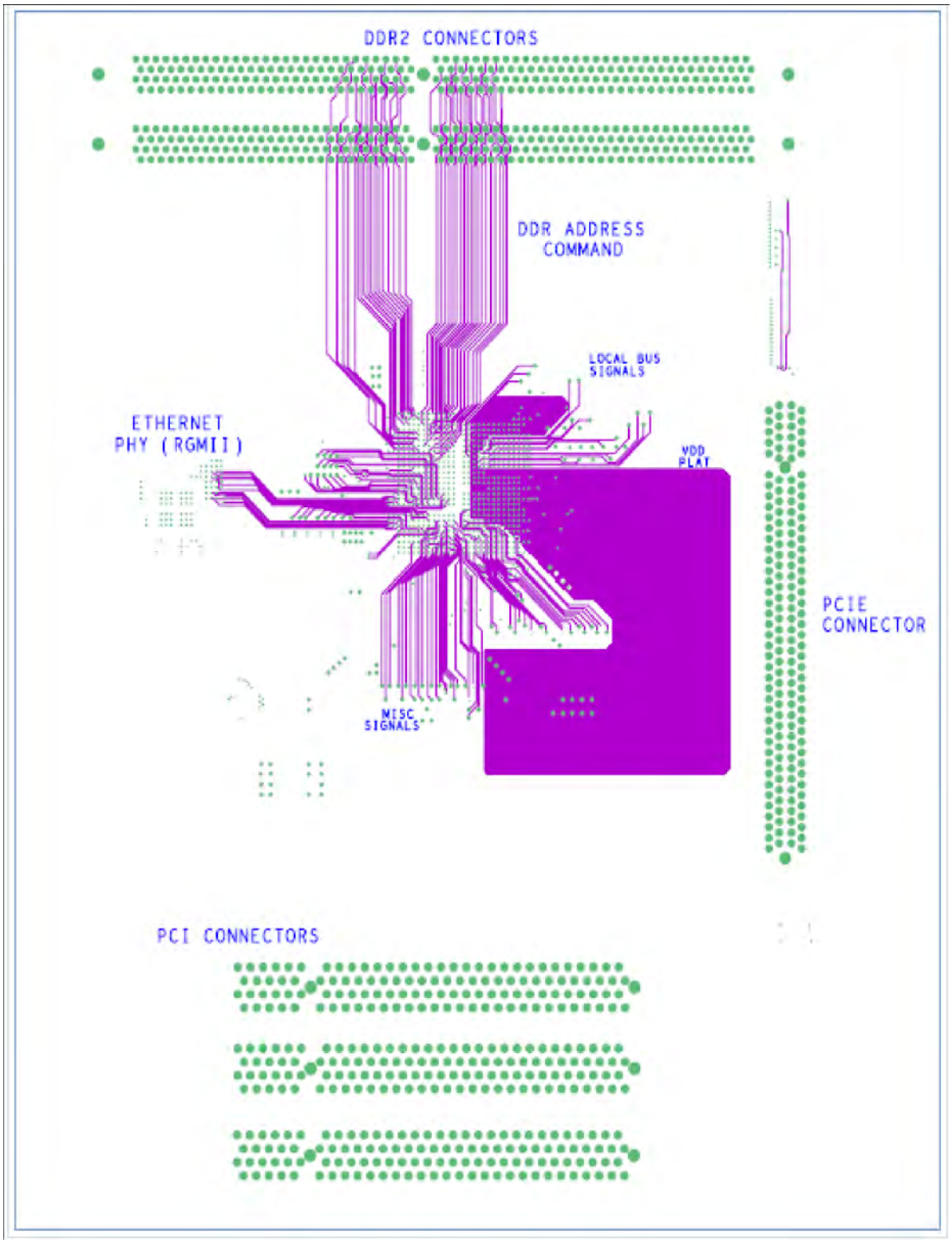


Figure 19. Signal #3 Gerber Plot (Layer 4 of 6)

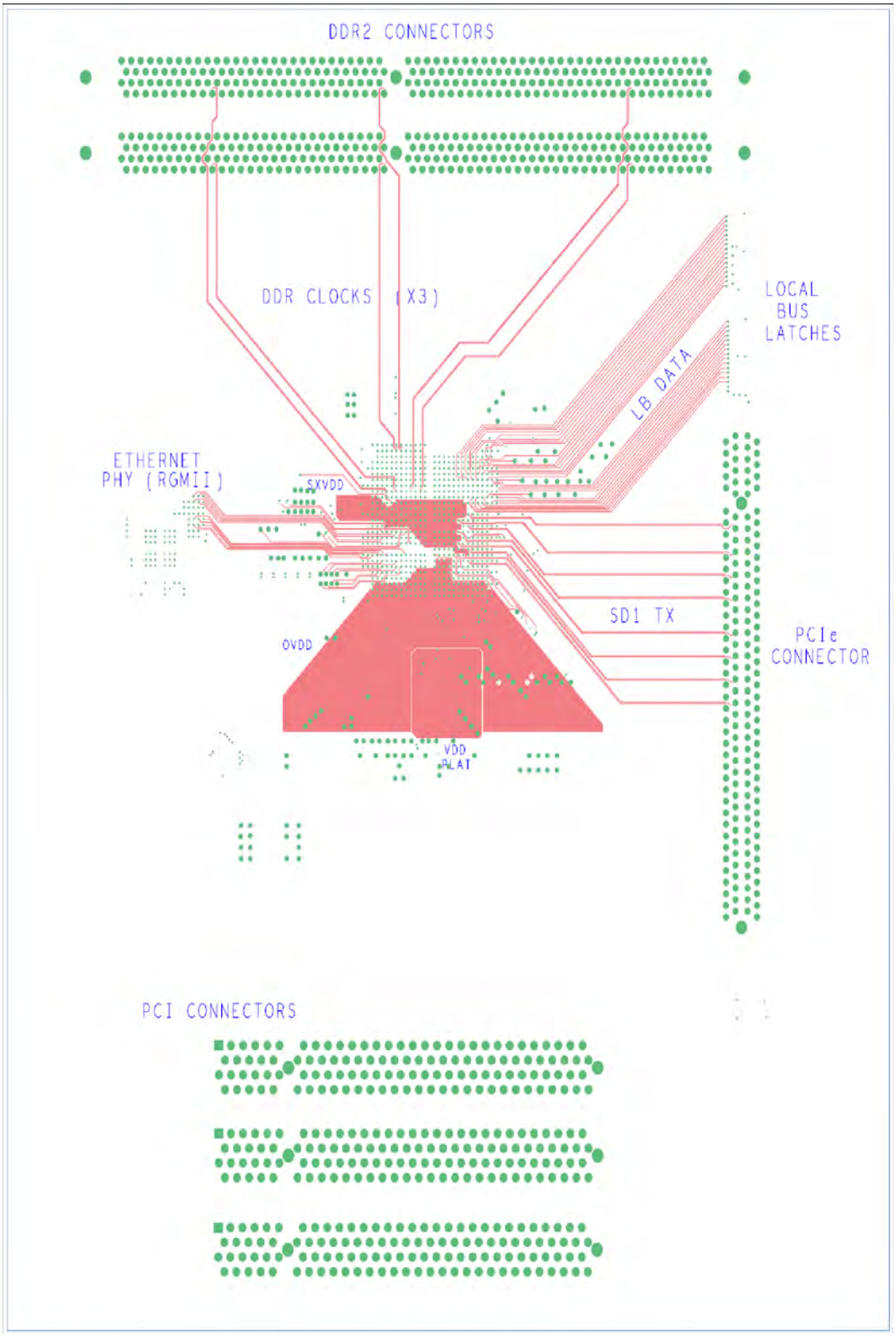
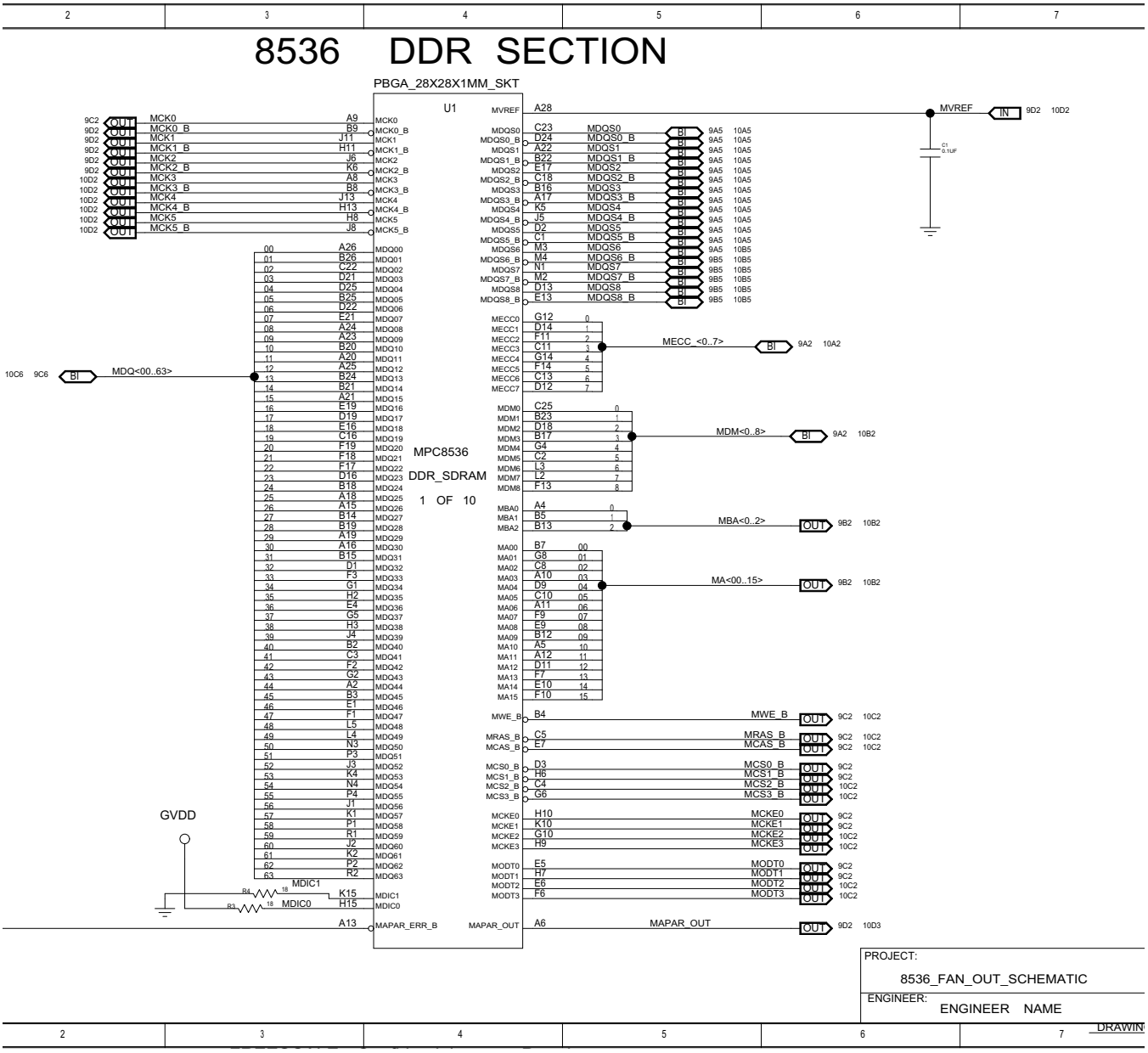
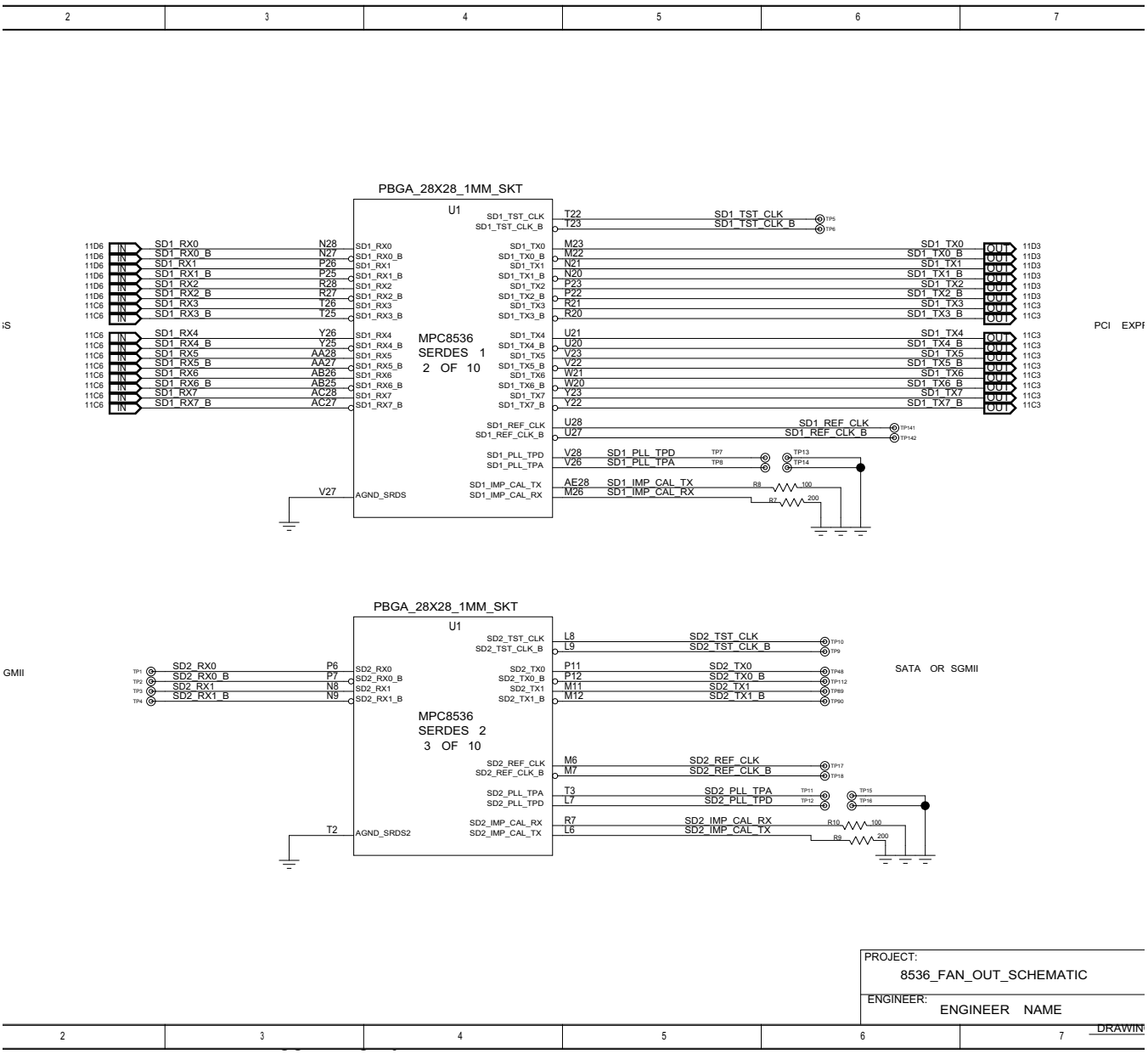


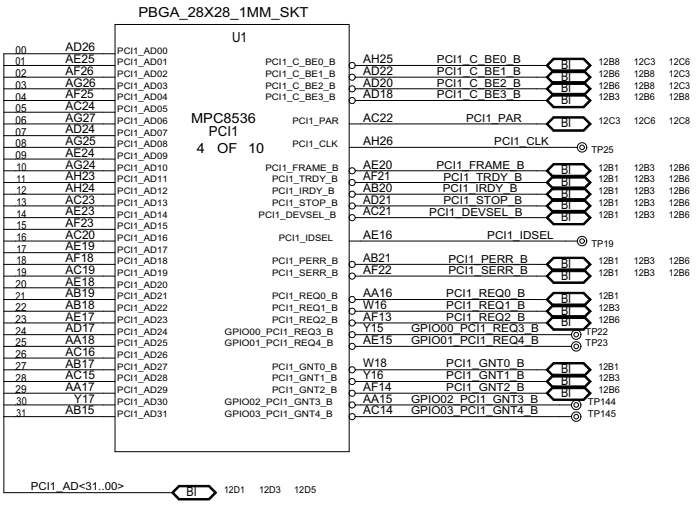
Figure 20. Bottom Gerber Plot (Layer 6 of 6)

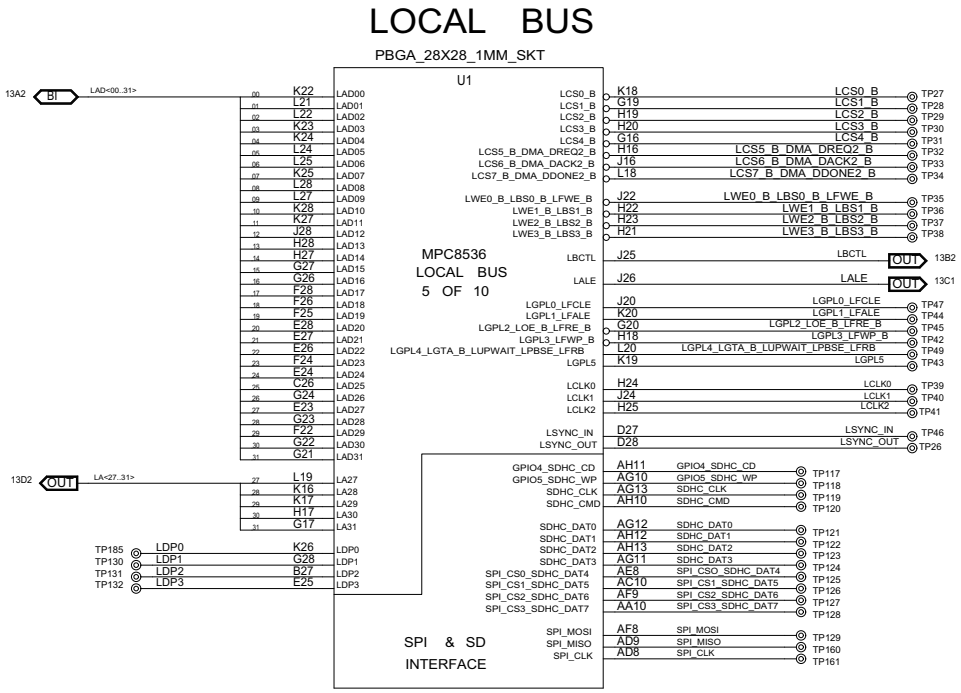
8 Fan-Out Schematics

This section contains the fan-out schematics used to perform the six-layer PCB study. It does not include all the decoupling, nor does it show all the necessary pull-ups, AC caps, and other circuitry that would be required outside the BGA fan-out area. This level of detail is captured as part of the MPC8536E development system.




2
3
4
5
6
7
PBGA_28X28_1MM_SKT
U1
MPC8536
SERDES 1
2 OF 10
SD1_RX0
SD1_RX0_B
SD1_RX1
SD1_RX1_B
SD1_RX2
SD1_RX2_B
SD1_RX3
SD1_RX3_B
SD1_RX4
SD1_RX4_B
SD1_RX5
SD1_RX5_B
SD1_RX6
SD1_RX6_B
SD1_RX7
SD1_RX7_B
SD1_TX0
SD1_TX0_B
SD1_TX1
SD1_TX1_B
SD1_TX2
SD1_TX2_B
SD1_TX3
SD1_TX3_B
SD1_TX4
SD1_TX4_B
SD1_TX5
SD1_TX5_B
SD1_TX6
SD1_TX6_B
SD1_TX7
SD1_TX7_B
SD1_TST_CLK
SD1_TST_CLK_B
SD1_REF_CLK
SD1_REF_CLK_B
SD1_PLL_TPD
SD1_PLL_TPA
SD1_IMP_CAL_TX
SD1_IMP_CAL_RX
AGND_SRDS
PCI EXPR
PBGA_28X28_1MM_SKT
U1
MPC8536
SERDES 2
3 OF 10
SD2_RX0
SD2_RX0_B
SD2_RX1
SD2_RX1_B
SD2_TX0
SD2_TX0_B
SD2_TX1
SD2_TX1_B
SD2_TST_CLK
SD2_TST_CLK_B
SD2_REF_CLK
SD2_REF_CLK_B
SD2_PLL_TPA
SD2_PLL_TPD
SD2_IMP_CAL_RX
SD2_IMP_CAL_TX
AGND_SRDS2
GMII
SATA OR SGMII
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME
DRAWIN

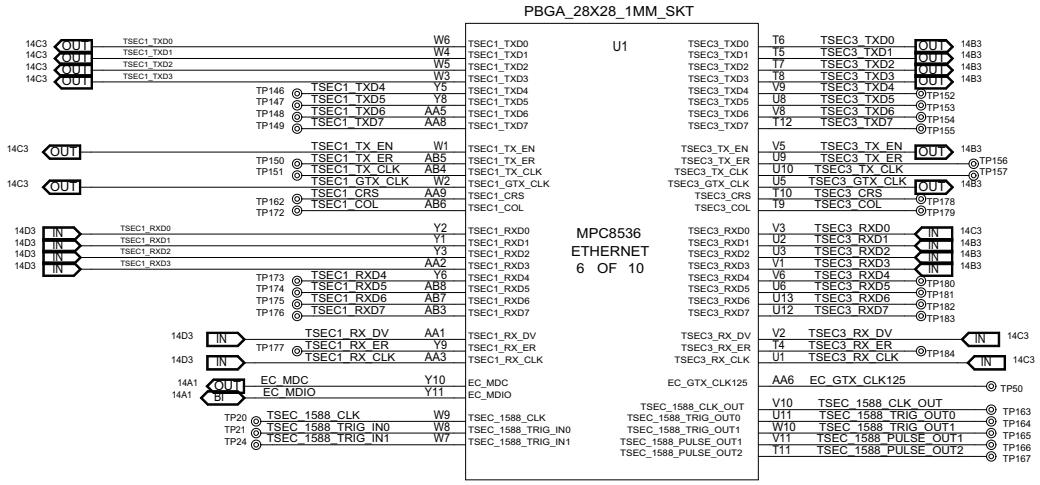

PCI 32 BIT
PBGA_28X28_1MM_SKT
U1
MPC8536
PCI1
4 OF 10
PCI1_AD<31..00>
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME



PROJECT: 8536_FAN_OUT_SCHEMATIC

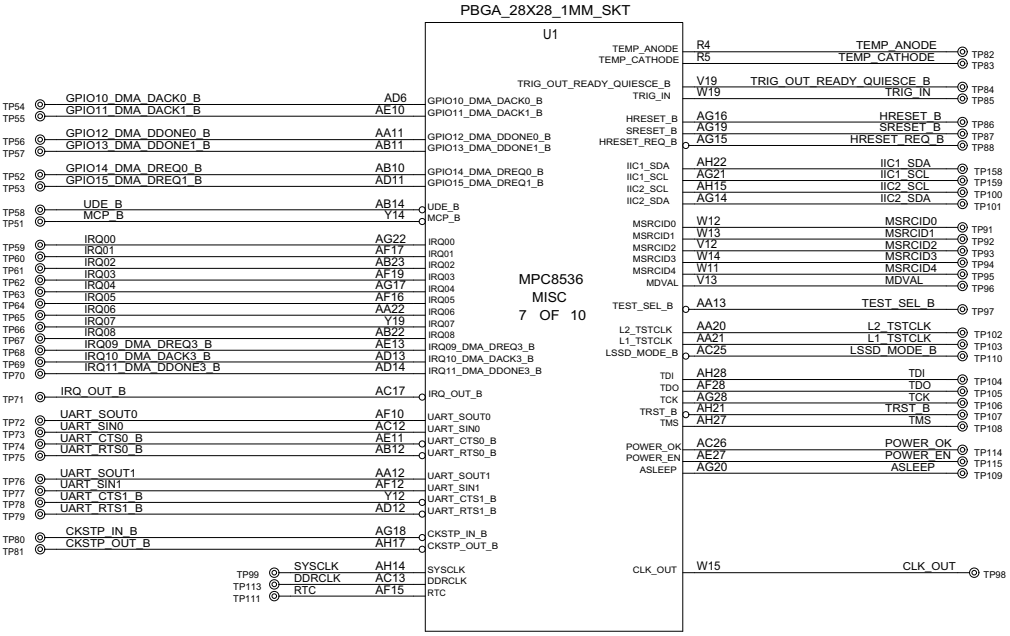
ENGINEER: ENGINEER NAME

FREESCALE Confidential Proprietary

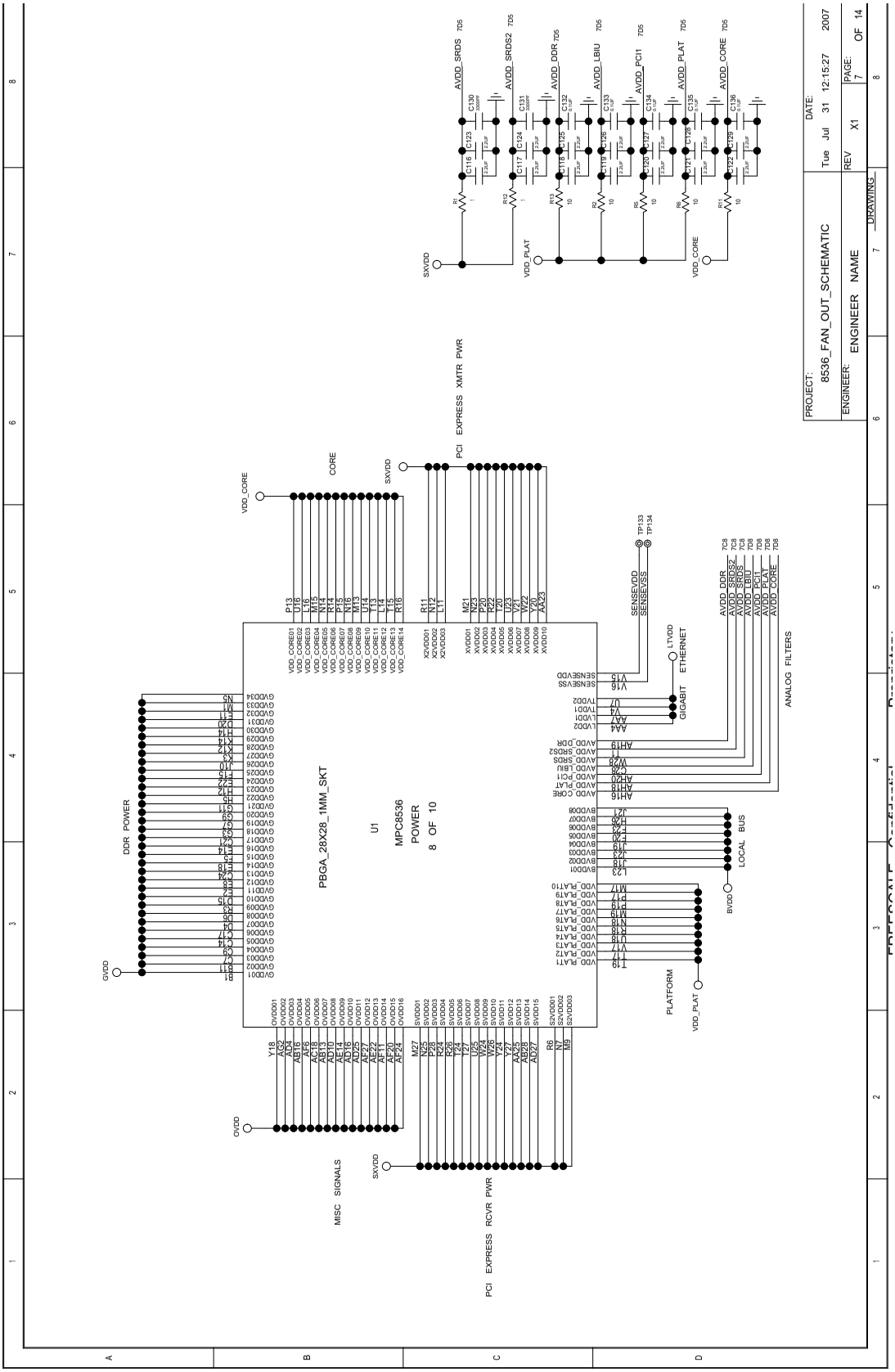


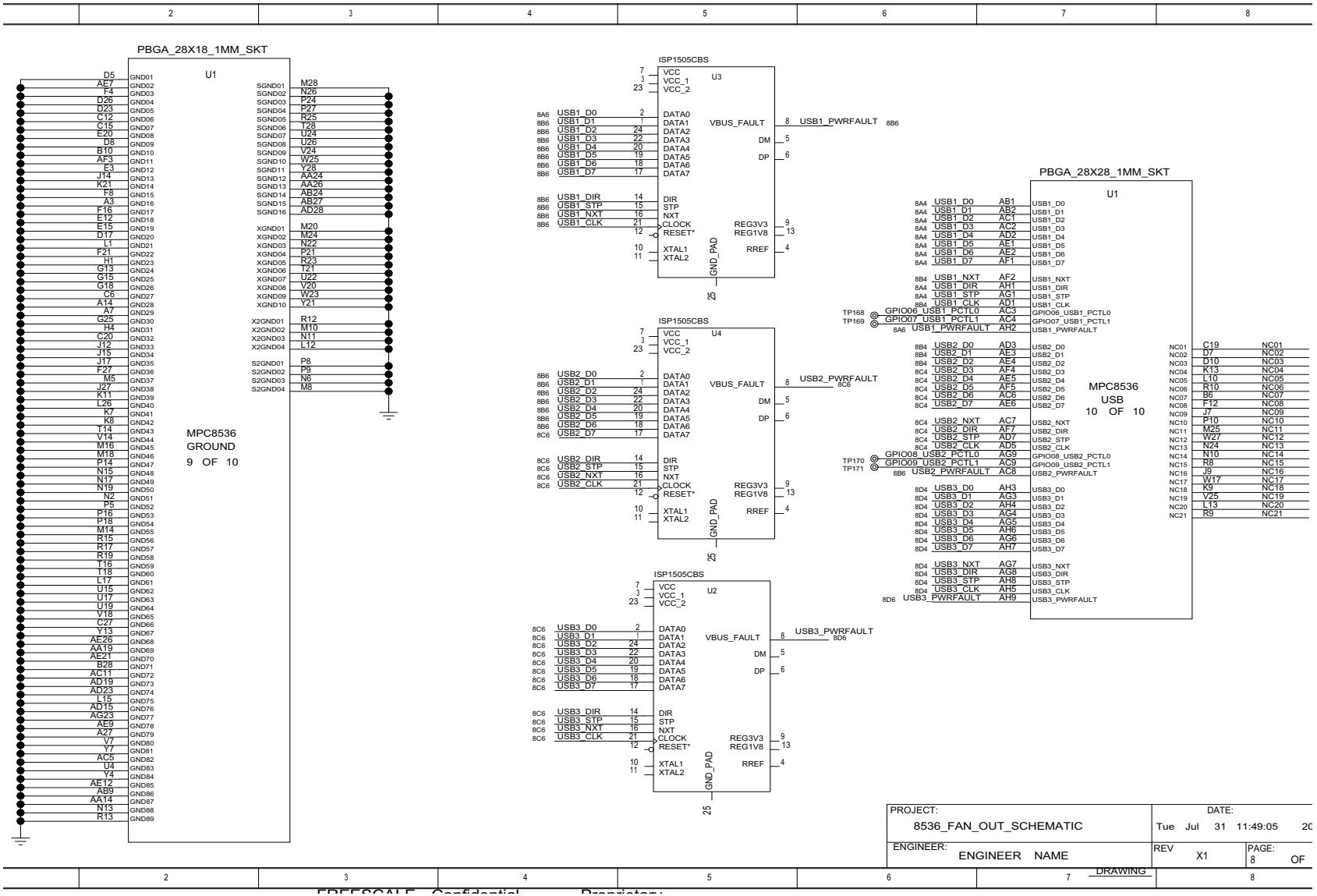
PROJECT:	8536_FAN_OUT_SCHEMATIC
ENGINEER:	ENGINEER NAME

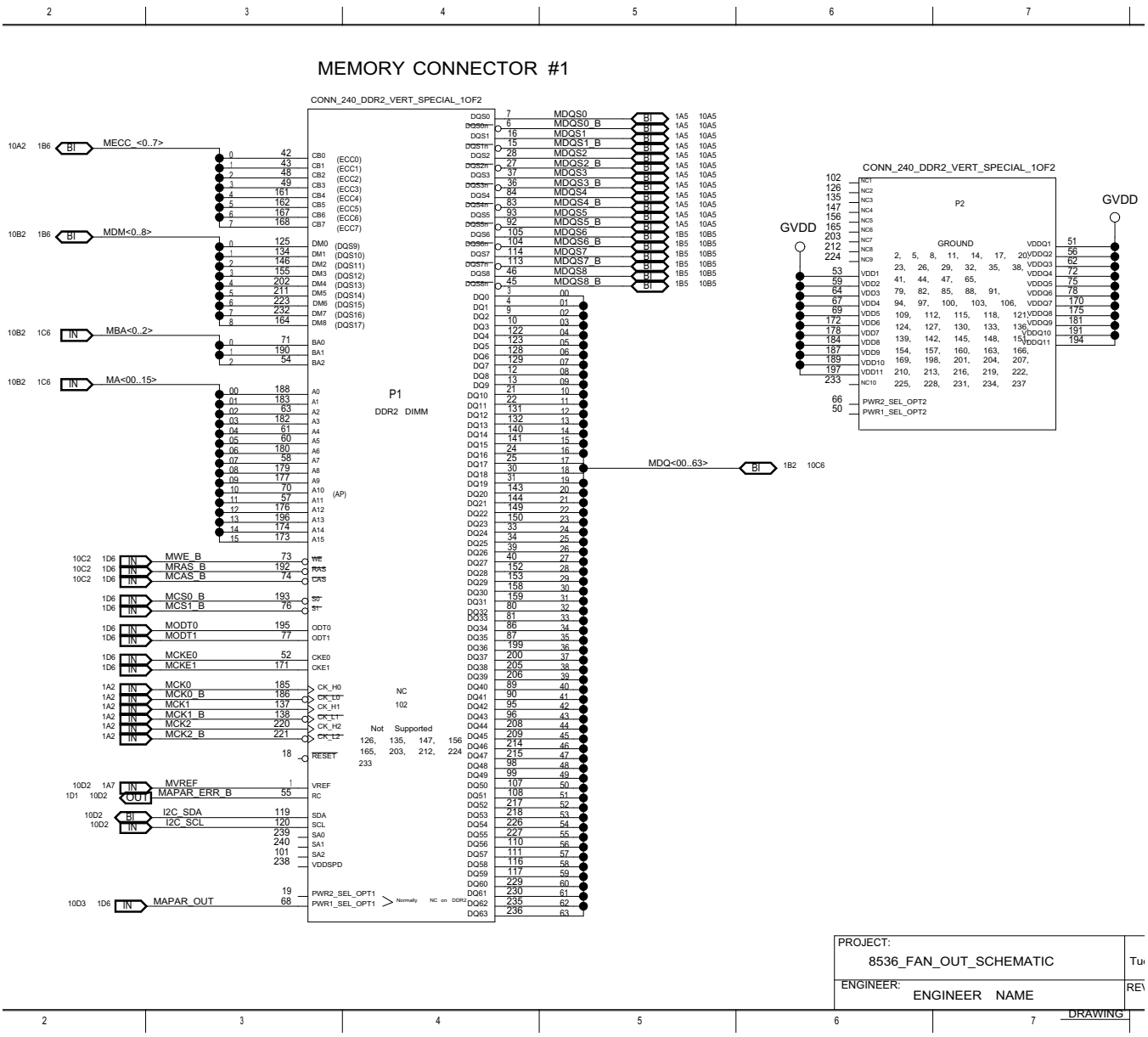
MISC / OTHERS

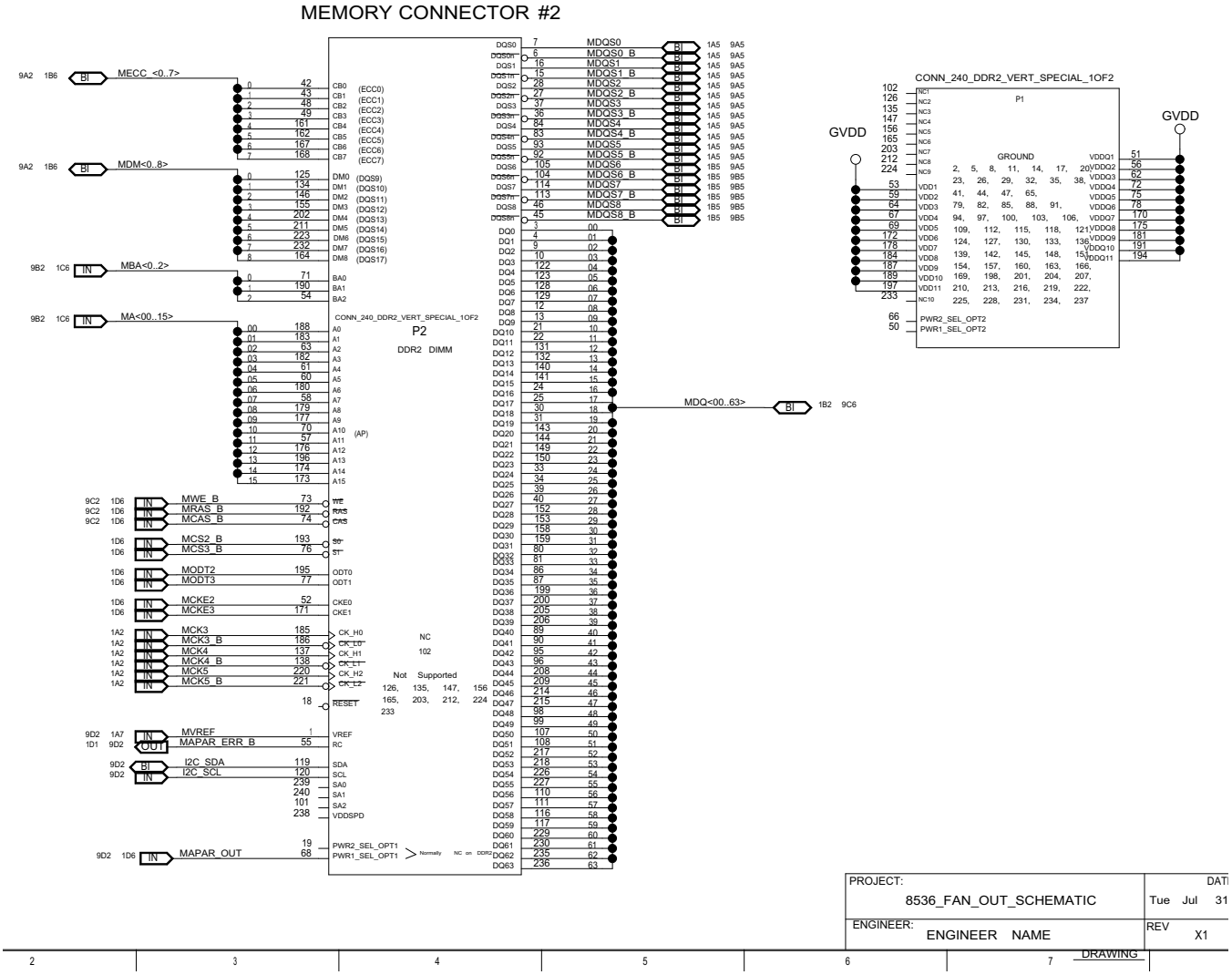


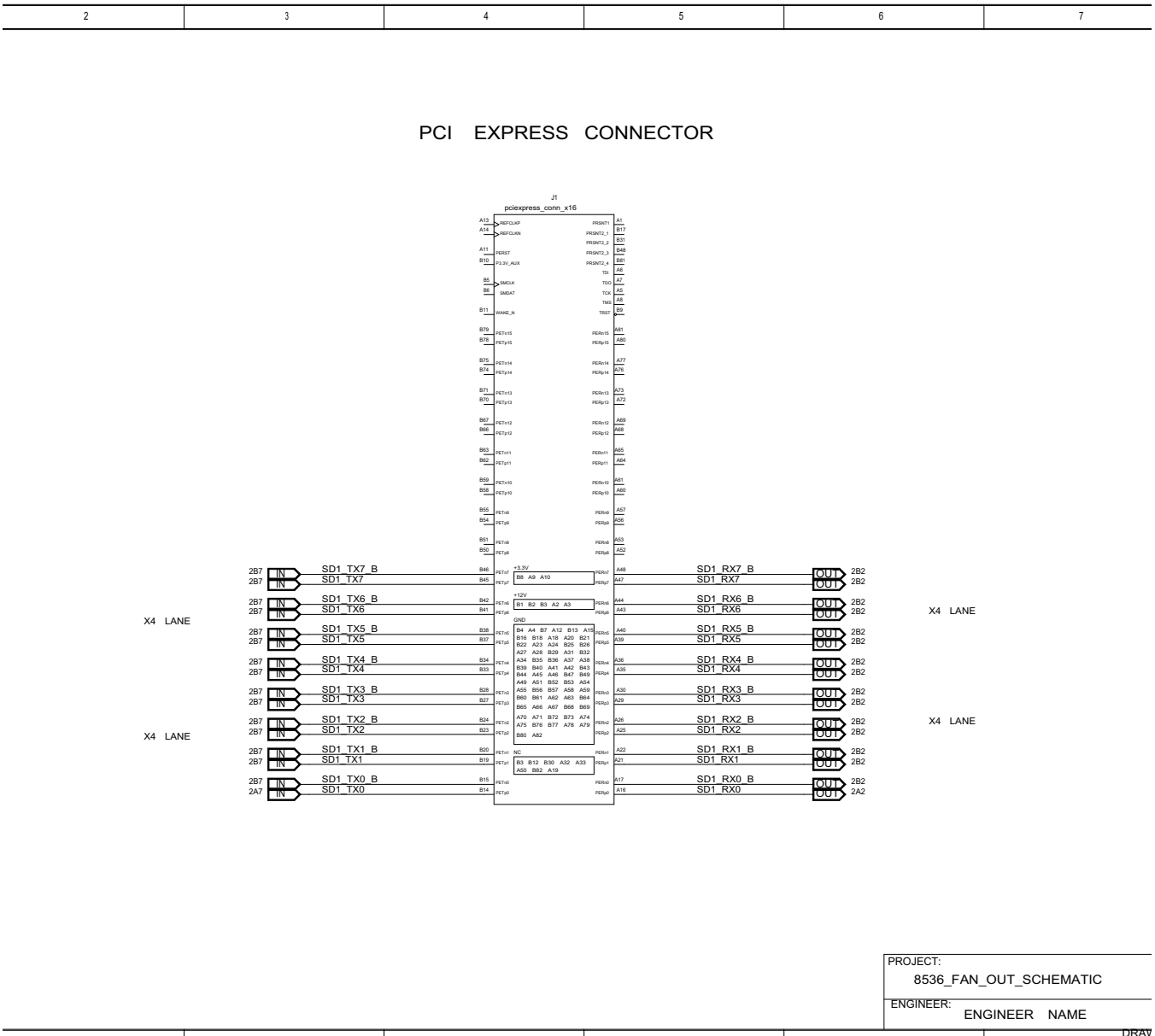
PROJECT:	8536_FAN_OUT_SCHEMATIC
ENGINEER:	ENGINEER NAME

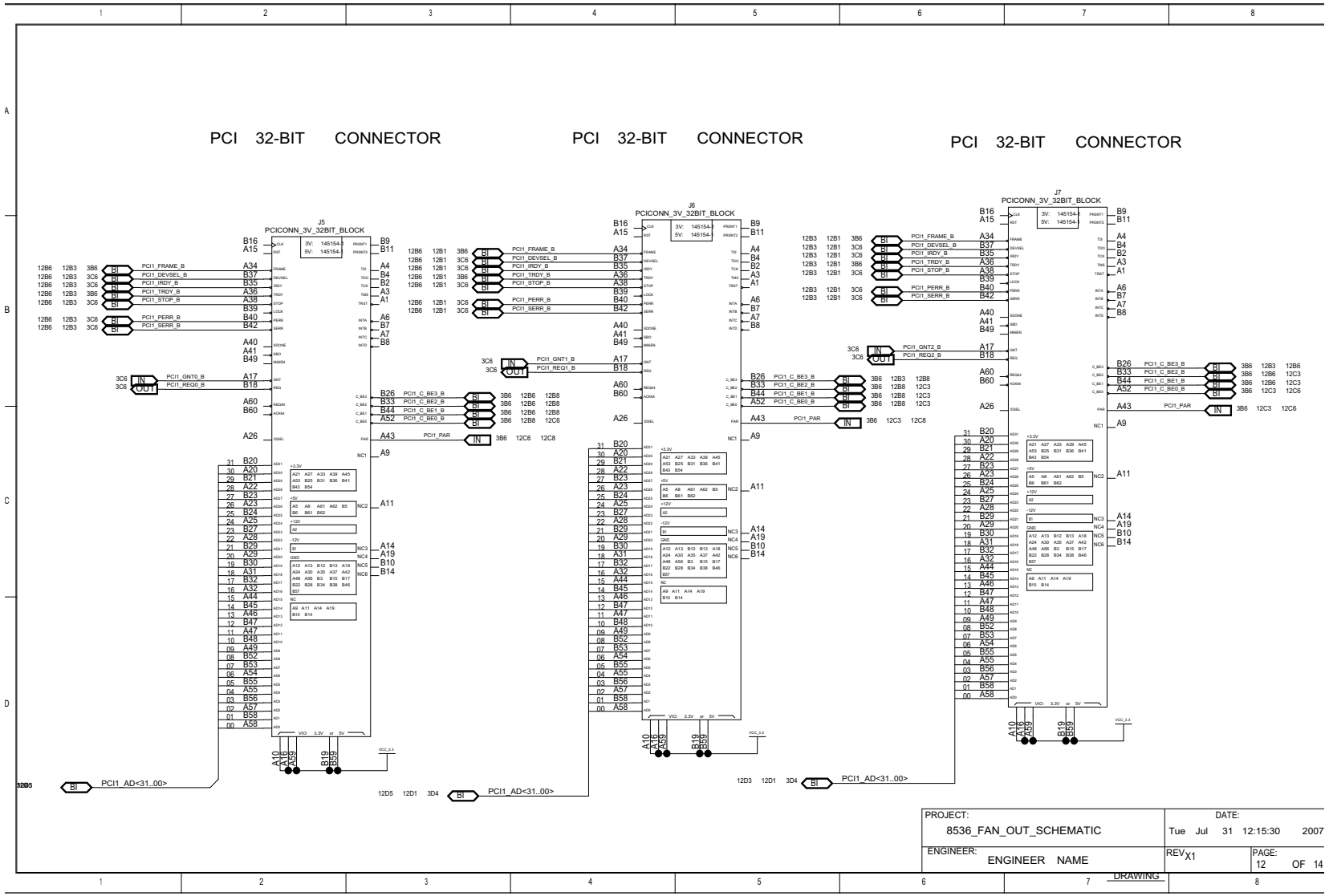

PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME
DATE:
Tue Jul 31 12:15:27 2007
REV X1
PAGE: 7 OF 14
DRAWING
U1
MPC8536
POWER
8 OF 10
PBGA_28X28_1MM_SKT
DDR POWER
PCI EXPRESS XMTR PWR
CORE
SXVDD
GVDD
OVDD
BVDD
LVDD
TVDD
LTVDD
ETHERNET
GIGABIT
LOCAL BUS
PLATFORM
ANALOG FILTERS
MISC SIGNALS
PCI EXPRESS RCVR PWR
AVDD_SRDS
AVDD_SRDS2
AVDD_DDR
AVDD_LBIU
AVDD_PCI1
AVDD_PLAT
AVDD_CORE
VDD_CORE
VDD_PLAT
SENSEVDD
SENSEVSS

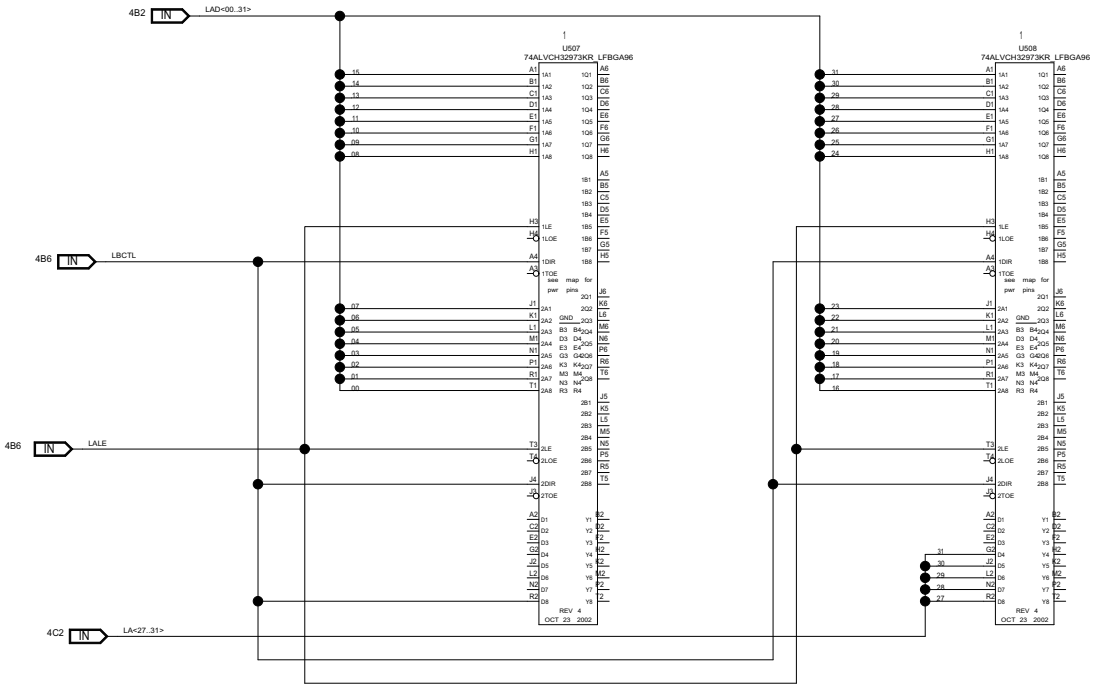

PBGA_28X18_1MM_SKT
U1
MPC8536
GROUND
9 OF 10
ISP1505CBS
U3
U4
U2
VBUS_FAULT
USB1_PWRFAULT
USB2_PWRFAULT
USB3_PWRFAULT
PBGA_28X28_1MM_SKT
MPC8536
USB
10 OF 10
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME
DATE:
Tue Jul 31 11:49:05
REV
X1
PAGE:
8 OF
DRAWING


MEMORY CONNECTOR #1
CONN_240_DDR2_VERT_SPECIAL_1OF2
P1
DDR2 DIMM
P2
GROUND
GVDD
MECC_<0..7>
MDM<0..8>
MBA<0..2>
MA<00..15>
MDQ<00..63>
MDQS0
MDQS0_B
MDQS1
MDQS1_B
MDQS2
MDQS2_B
MDQS3
MDQS3_B
MDQS4
MDQS4_B
MDQS5
MDQS5_B
MDQS6
MDQS6_B
MDQS7
MDQS7_B
MDQS8
MDQS8_B
MWE_B
MRAS_B
MCAS_B
MCS0_B
MCS1_B
MODT0
MODT1
MCKE0
MCKE1
MCK0
MCK0_B
MCK1
MCK1_B
MCK2
MCK2_B
MVREF
MAPAR_ERR_B
I2C_SDA
I2C_SCL
MAPAR_OUT
PWR2_SEL_OPT1
PWR1_SEL_OPT1
Not Supported
126, 135, 147, 156
165, 203, 212, 224
233
NC
102
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME
DRAWING

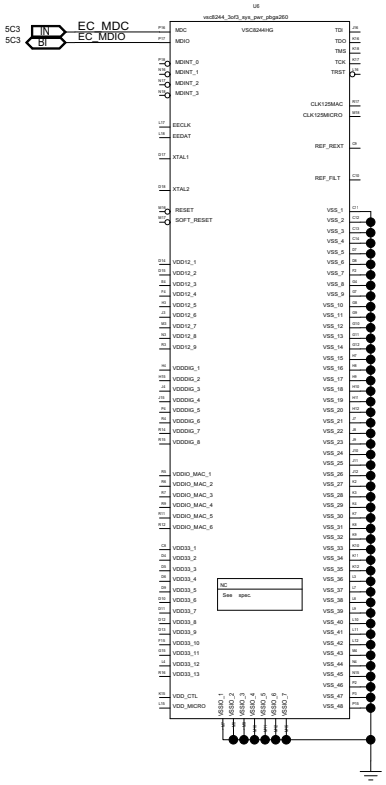

MEMORY CONNECTOR #2
CONN_240_DDR2_VERT_SPECIAL_1OF2
P2
DDR2 DIMM
MECC_<0..7>
MDM<0..8>
MBA<0..2>
MA<00..15>
MDQ<00..63>
MWE_B
MRAS_B
MCAS_B
MCS2_B
MCS3_B
MODT2
MODT3
MCKE2
MCKE3
MCK3
MCK3_B
MCK4
MCK4_B
MCK5
MCK5_B
MVREF
MAPAR_ERR_B
I2C_SDA
I2C_SCL
MAPAR_OUT
MDQS0
MDQS0_B
MDQS1
MDQS1_B
MDQS2
MDQS2_B
MDQS3
MDQS3_B
MDQS4
MDQS4_B
MDQS5
MDQS5_B
MDQS6
MDQS6_B
MDQS7
MDQS7_B
MDQS8
MDQS8_B
CONN_240_DDR2_VERT_SPECIAL_1OF2
P1
GVDD
GROUND
PWR2_SEL_OPT2
PWR1_SEL_OPT2
NC
Not Supported
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME
REV
X1
DRAWING


2
3
4
5
6
7
PCI EXPRESS CONNECTOR
J1
pciexpress_conn_x16
X4 LANE
X4 LANE
X4 LANE
X4 LANE
SD1_TX7_B
SD1_TX7
SD1_TX6_B
SD1_TX6
SD1_TX5_B
SD1_TX5
SD1_TX4_B
SD1_TX4
SD1_TX3_B
SD1_TX3
SD1_TX2_B
SD1_TX2
SD1_TX1_B
SD1_TX1
SD1_TX0_B
SD1_TX0
SD1_RX7_B
SD1_RX7
SD1_RX6_B
SD1_RX6
SD1_RX5_B
SD1_RX5
SD1_RX4_B
SD1_RX4
SD1_RX3_B
SD1_RX3
SD1_RX2_B
SD1_RX2
SD1_RX1_B
SD1_RX1
SD1_RX0_B
SD1_RX0
PROJECT:
8536_FAN_OUT_SCHEMATIC
ENGINEER:
ENGINEER NAME

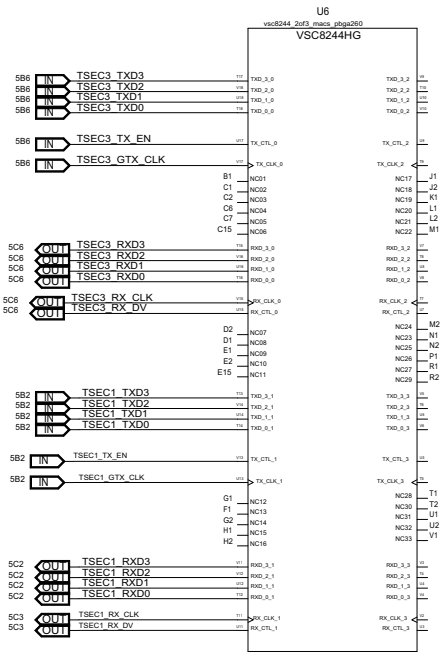

PCI 32-BIT CONNECTOR
PCI 32-BIT CONNECTOR
PCI 32-BIT CONNECTOR
J5
PCICONN_3V_32BIT_BLOCK
J6
PCICONN_3V_32BIT_BLOCK
J7
PCICONN_3V_32BIT_BLOCK
PCI1_AD<31..00>
PROJECT:
8536_FAN_OUT_SCHEMATIC
DATE:
Tue Jul 31 12:15:30 2007
ENGINEER:
ENGINEER NAME
REV X1
PAGE:
12 OF 14
DRAWING


LOCAL BUS LATCHES
4B2
IN
LAD<00..31>
4B6
IN
LBCTL
4B6
IN
LALE
4C2
IN
LA<27..31>
U507
74ALVCH32973KR_LFBGA96
U508
74ALVCH32973KR_LFBGA96
REV 4
OCT 23 2002
PROJECT:
8536_FAN_OUT_SCH
ENGINEER:
ENGINEER NAM

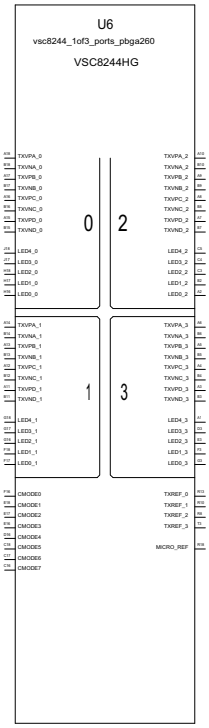
GBE PHY 1 OF 3



GBE PHY 2 OF 3



GBE PHY 3 OF 3



PROJECT:	8536_FAN_OUT_SCHEMAT
ENGINEER:	ENGINEER NAME

9 Current Delivery in the BGA Field

The MPC8536E comes in a 1 mm pitch, 783 BGA package arranged as a 28 × 28 array (with pin A1 missing). The BGA pads and vias usually have the physical parameters listed Table 8.

Table 8. MPC8536E Escape Dimensions

Dimension	Size	Notes
Drill diameter	10 mils	Via drill
Finished hole size (FHS)	7–7.5 mils	After plating
Pad	19 mils	
Anti-pad	28 mils	Clearance from via pads to adjacent plane area-fills
Thermal relief	33 mils[2]	
Via keepout	23 mils	As above, but for traces on signal (inner) planes

Keep the following points in mind while reading this section:

- The remainder of this section is based on the padstack dimensions in Table 8. If a different padstack is used, most or all of the numbers derived probably need to be recalculated.
- It is assumed that each BGA connection carries current equally, no matter its proximity to the power supply and/or processor internal power pads. In reality, the current is nonuniformly distributed due to internal BGA-to-die substrate connections and other factors. There are no guaranteeable distributions among the power pins.

All comments in this section about power planes apply equally to the return ground. Multilayer PCBs often have multiple ground planes, so this restriction is occasionally overlooked; however, the ground return path requirements are exactly equal to the power source path requirements.

In addition to bypass capacitor placement, the designer must also consider the wholesale delivery of power to the V_{DD} pins. On most PCBs, the power plane is not actually solid, but is perforated by numerous vias. This is illustrated in Figure 21, where the wide plane is actually only as wide as its narrowest point.

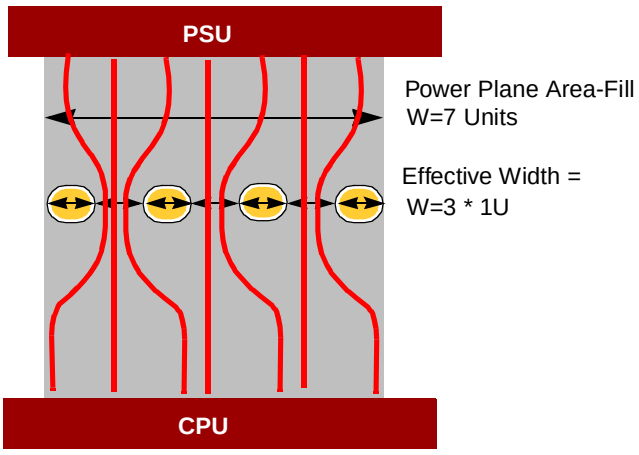


Figure 21. Restricted Current Flow

For many applications, the effect of the vias passing through the power plane on the flow of current is insignificant. However, as the power reaches the outer periphery of the MPC8536E, it must flow through the outer array of signal BGA vias. While some BGA pads may not need vias (outer BGA pads may be able to route out on the top layers); in most cases the number of interfering vias ranges from 8 to 11.

For many layouts, the outer one or two BGA pads do not require vias as these signals are usually “escaped” on the top layer of the PCB. Delivering power properly to the power pins requires planning the flow of current around these vias, or more properly, the antivias or antipads that surround the via and cause a hole on the power plane layer. When coupled with a 1-mm spacing, the cross section of the PCB typically resembles that shown in Figure 6.

The anti-pad is larger than the spacing between the via barrels, which in effect reduces the width and capacity of the trace. In effect, the power plane is not a solid plane but a parallel array of traces with one of the following widths:

- 11.3 mils, when planes must be isolated from both vias
- 19.3 mils, when planes connect to one of the vias

The standard formulae to determine the limits of the copper trace are described in the IPC-2221A standard and are given by:

$$I_{(Trace,\ Outer)} = 0.048 \times (maxTempRise)^{0.44} \times (viaPlating \times viaDiameter \times \pi)^{0.725} \quad \textit{Eqn. 1}$$

$$I_{(Trace,\ Inner)} = 0.024 \times (maxTempRise)^{0.44} \times (viaPlating ¥ viaDiameter \times \pi)^{0.725} \quad \textit{Eqn. 2}$$

Using these formulae, and using an 11-mil trace as a standard (rounded down from the actual 11.3-mil max trace width), for various layout rules the amount of current that can be safely carried, assuming a maximum temperature rise of 20°C, is shown in Table 9.

Table 9. Current-Carrying Capacity per Standard IPC2221A

Location	Copper Plating (oz.)	11-mil Trace (A)
Outer	0.5	0.777
	1.0	1.285
	1.5	1.730
	2.0	2.130
Inner	0.5	0.390
	1.0	0.582
	1.5	0.865
	2.0	1.065

9.1 Via Current Capacity

Unless the power is delivered on the top plane and attaches directly to the BGA pad of the processor (which as noted is fairly difficult to accomplish), power is delivered from one or more PCB planes to the processor through vias. Given the padstack parameters in Table 8 and the number of vias allocated, the next step to

ensure a quality PDS is to evaluate the individual and aggregate via current capacity using the following formula:

$$IVIA = 0.048 \times (maxTempRise)^{0.44} \times (viaPlating \times viaDiameter \times \pi)^{0.725} \quad \textit{Eqn. 3}$$

Table 10 shows a few via current capacities for given PCB process parameters.

Table 10. Via Capacity

maxTempRise (°C)	viaPlating (mil)	viaDiameter (mil)	I_{VIA} (A)	Notes
10	1	10	1.609	Standard
10	1	11	1.724	—
10	1	15	2.159	—
10	1.5	10	2.159	—
20	1	10	2.183	—

At 1.609 A/via, we have a maximum current capacity of 22.5 A for V_{DD_CORE} (14 vias * 1.609), and 16 A for V_{DD_PLAT} (10 * 1.609).

9.2 Alternate Plane Current Capacity

Research performed by Johannes Adam of Flomerics as preparatory work for a revision to one of the rules of the IPC2221 standard (specifically, design rule 2152), found that the formula for outer traces was based on materials and assumptions no longer prevalent. Additionally, he discovered that the 50 percent derating factor used for inner traces is wholly arbitrary and should be approximately 5 percent. In particular, thermal conduction to adjacent traces and power planes was found to be vastly more important than conduction to free space. Refer to Section 10, “References,” for details on this paper and associated historical data.

At the time of publication, the IPC2221 has not been updated based on these findings, so the designer can either follow the published standards and accept overly conservative numbers or incorporate the pending changes. Freescale cannot recommend one course over the other and advises designers to consult the original source documentation and decide for themselves. Examining the IEEE paper, “case 5” shows an outer trace of variable width over a 35 um (1 oz. copper) plane. This correlates nicely with an external power connection residing over an inner ground plane immediately adjacent to it. Such a plane is desirable not only for power purposes but for routing differential pairs such as those on the SerDes or DDR interfaces. Extracting the data for a 0.3 mm trace (~11 mils), and applying a curve fitting process to the data produces the following formula, which is represented in Figure 22:

$$T(trace) = 19.473 + 3.228IMAX + 4.469IMAX^2 + 0.385IMAX^3 \quad \textit{Eqn. 4}$$

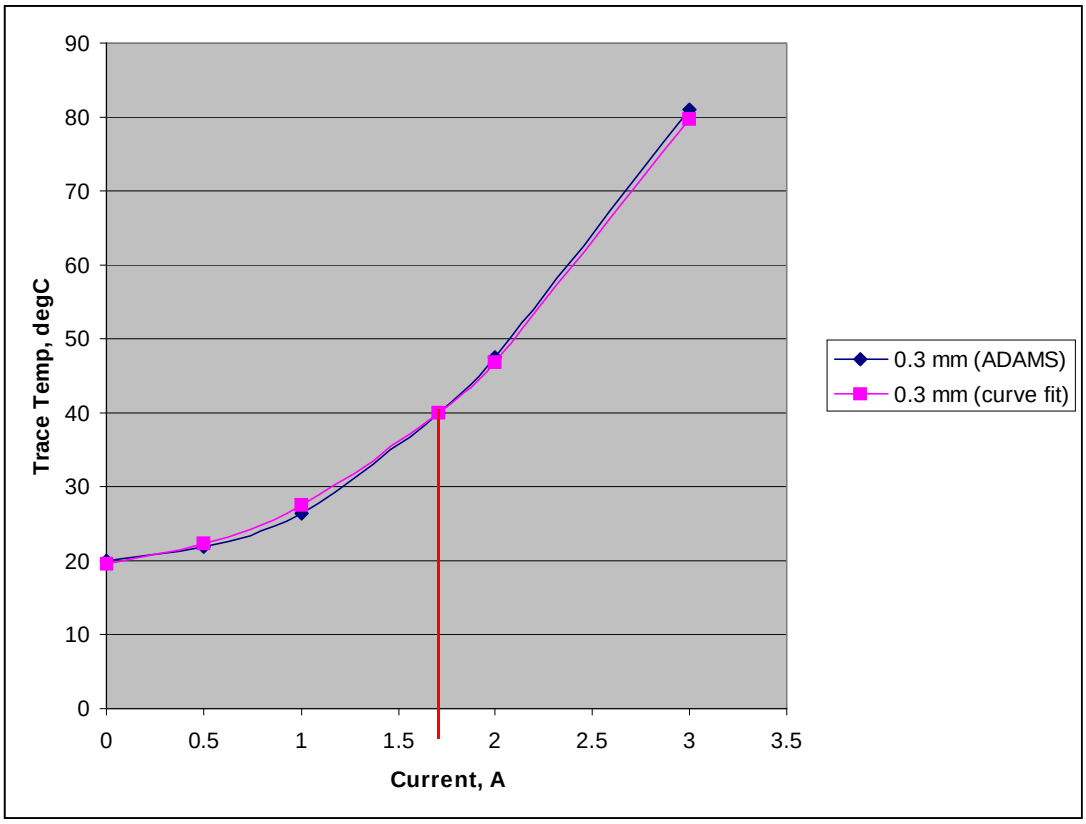


Figure 22. Extrapolated Current Flow

CAUTION

This curve is valid only for the 0.3 mm trace. The IPC standard will likely include a more complicated curve equation that accepts both current and trace width as a parameter.

Note that the Y-axis is the total trace temperature. Because we are concerned with a maximum 20°C rise, the Y-axis starts at 20 as the ambient temperature is also assumed to be 20°C. The crossover point of 40°C occurs when I = 1.711 A. Using this new data, Table 9 can be updated as shown in Table 11:

Table 11. Current-Carrying Capacity Extrapolated from Adams

Location	Copper Plating (oz.)	A/11-mil Trace (A)
Outer	0.5	1.210
	1.0	1.711
	1.5	2.420
	2.0	3.422

Table 11. Current-Carrying Capacity Extrapolated from Adams (continued)

Location	Copper Plating (oz.)	A/11-mil Trace (A)
Inner	0.5	1.149
	1.0	1.625
	1.5	2.299
	2.0	3.251

The IEEE paper included in the references explains why the classic IPC-275/IPC2221A values for inner current layers are extremely misleading. IPC-2152 is not published at the time of this application note.

10 References

Table 12 lists useful resources for further reading.

Table 12. References

Document	Source
IPC Standards: IPC-D-275 IPC-2221A IPC-2152	www.ipc.org
New Correlations Between Electrical Current and Temperature Rise in PCB Traces Johannes Adam, Flomerics Ltd.	20th IEEE SEMI-THERM Symposium 0-7803-8363-X/04/$20.00 ©2004 IEEE www.flomerics.com/flotherm/technical_papers/t341.pdf
Martin Tarr, University of Bolton	www.ami.ac.uk/courses/ami4817_dti/u02/pdf/meah0221.pdf
Decoupling Capacitors, a Designer's Roadmap to Optimal Decoupling Networks for Integrated Circuits (CSC240_MUCCIOLI) Freescale Semiconductor	http://www.freescale.com/files/ftf_2005/doc/reports_presentations/CSC240_MUCCIOLI.pdf
Interactive trace-width calculator	http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator
DC Line Resistance and Current Carrying Capacity Merix Inc.	http://www.merix.com/technology.php?section=processes&page=technology/_processes.html

11 Revision History

Table 13 provides a revision history for this application note.

Table 13. Document Revision History

Rev. Number	Date	Description
0	10/07	Initial release.

How to Reach Us:

Home Page:
www.freescale.com

Web Support:
http://www.freescale.com/support

USA/Europe or Locations Not Listed:
Freescale Semiconductor, Inc.
Technical Information Center, EL516
2100 East Elliot Road
Tempe, Arizona 85284
+1-800-521-6274 or
+1-480-768-2130
www.freescale.com/support

Europe, Middle East, and Africa:
Freescale Halbleiter Deutschland GmbH
Technical Information Center
Schatzbogen 7
81829 Muenchen, Germany
+44 1296 380 456 (English)
+46 8 52200080 (English)
+49 89 92103 559 (German)
+33 1 69 35 48 48 (French)
www.freescale.com/support

Japan:
Freescale Semiconductor Japan Ltd.
Headquarters
ARCO Tower 15F
1-8-1, Shimo-Meguro, Meguro-ku
Tokyo 153-0064
Japan
0120 191014 or
+81 3 5437 9125
support.japan@freescale.com

Asia/Pacific:
Freescale Semiconductor Hong Kong Ltd.
Technical Information Center
2 Dai King Street
Tai Po Industrial Estate
Tai Po, N.T., Hong Kong
+800 2666 8080
support.asia@freescale.com

For Literature Requests Only:
Freescale Semiconductor
Literature Distribution Center
P.O. Box 5405
Denver, Colorado 80217
+1-800 441-2447 or
+1-303-675-2140
Fax: +1-303-675-2150
LDCForFreescaleSemiconductor
@hibbertgroup.com

Document Number: AN3444
Rev. 0
10/2007