

Sitara™ Processor Power Distribution Networks: Implementation and Analysis



Sitara MPU

ABSTRACT

The purpose of a power distribution network (PDN) is primarily to provide clean and reliable power to the active devices on the system. The printed circuit board (PCB) is a critical component of the system-level PDN delivery network. As such, optimal design of the PCB power distribution network is of utmost importance for high performance microprocessors. This application report provides implementation guidelines and recommendations for designing printed circuit board (PCB) power delivery networks (PDN) for the Texas Instruments Sitara™ family of microprocessors.

Table of Contents

1 Introduction	2
1.1 Acronyms Used in This Document.....	2
2 Guidelines for PCB Stack-Up	3
3 Physical Layout Optimization of the PDN	5
4 Static PDN Analysis (IR Drop Optimization)	8
5 Dynamic Analysis of PCB PDN	10
5.1 Selecting Decoupling Capacitors to Meet Z_{TARGET}	11
6 Checklist for PDN	13
7 Implementation Examples and PDN Targets	14
7.1 AM570x.....	14
7.2 AM571x.....	16
7.3 AM572x.....	19
7.4 AM574x.....	22
7.5 AM65xx/DRA80xM.....	25
7.6 AM62xx.....	28
7.7 AM64xx.....	29
7.8 AM62Ax.....	32
7.9 AM62Px.....	35
7.10 AM62Lx.....	38
8 Revision History	40

List of Figures

Figure 2-1. Minimize Loop Inductance by Optimizing Layer Assignments in the PCB.....	3
Figure 2-2. Example Stack-Up Utilizing High-Density Interconnect Vias.....	4
Figure 2-3. Example Stack-Up Utilizing Plated-Through-Hole (PTH) Vias.....	4
Figure 3-1. Characteristics of a “Real” Capacitor With ESL and ESR.....	5
Figure 3-2. Typical Impedance Profile of a Capacitor.....	6
Figure 3-3. Example of “Co-Planar” Shielding of Power Net Using Ground Guard-Band.....	6
Figure 3-4. Capacitor Mounting Geometries.....	7
Figure 4-1. Depiction of Sheet Resistivity and Resistance.....	8
Figure 4-2. PCB IR Drop Budget.....	8
Figure 4-3. Sense Line Placement.....	9
Figure 4-4. Allowable Power Input Voltage Difference.....	9
Figure 5-1. Components of a Typical Power Distribution Network (PDN).....	10
Figure 5-2. Decoupling Frequency Range of PCB Components.....	10
Figure 5-3. Example Capacitor Placement.....	12

Figure 7-1. AM62P PCB Stack up.....	35
Figure 7-2. AM62P PCB Impedance.....	36
Figure 7-3. AM62L PCB Stack up.....	38
Figure 7-4. AM62L PCB Impedance.....	38

List of Tables

Table 1-1. Acronyms.....	2
Table 7-1. AM570x PDN Targets and Decoupling Example.....	15
Table 7-2. AM571x PDN Targets and Decoupling Example.....	18
Table 7-3. AM572x PDN Targets and Decoupling Example.....	21
Table 7-4. AM574x PDN Targets and Decoupling Example.....	24
Table 7-5. AM65xx/DRA80xM PDN Targets and Decoupling Example	27
Table 7-6. AM62xx PDN Targets and Decoupling Example.....	29
Table 7-7. AM64xx PDN Targets and Decoupling Example.....	30
Table 7-8. AM62Ax PDN Targets and Decoupling Example.....	33
Table 7-9. AM62Px PDN Targets and Decoupling Example.....	37
Table 7-10. AM62Lx PDN Targets and Decoupling Example.....	39

Trademarks

Sitara™ is a trademark of Texas Instruments.

All trademarks are the property of their respective owners.

1 Introduction

This application report provides implementation guidelines and recommendations for designing printed circuit board (PCB) power delivery networks (PDN) for the Texas Instruments Sitara family of microprocessors. This application report will:

- Discuss PCB PDN design methodology
- Provide general PCB PDN design recommendations and requirements
- Discuss the rationale behind these design requirements
- Provide suggestions and methods that PCB designers can implement to ensure that the PDN requirements for a specific processor are met.
- Provide PDN targets for Sitara-class microprocessors

1.1 Acronyms Used in This Document

Table 1-1. Acronyms

Acronym	Description
AC	Alternating current
BGA	Ball grid array
DC	Direct current (static)
Df	Loss tangent
Dk	Dielectric constant
EDA	Electronic design automation
EM	Electromigration
ESL	Effective series inductance
ESR	Effective series resistance
FDTIM	Frequency domain target impedance method
HDI	High density interconnect (for example, buried/blind via)
IR	Product of current (I) x resistance (R)
PCB	Printed circuit board
PDN	Power distribution network
PM-IC/PMIC	Power management integrated circuit
PTH	Plated through hole
RLC	Resistance, inductance, and capacitance
SMPS	Switch mode power supply

Table 1-1. Acronyms (continued)

Acronym	Description
SMT	Surface mount technology
SRF	Self resonant frequency
VIP	Via in pad
VRM	Voltage regulator module

Note

Notwithstanding any provision to the contrary, TI makes no warranty expressed, implied, or statutory, including any implied warranty of merchantability of fitness for a specific purpose, for customer boards. The data described in this document are intended as guidelines only.

2 Guidelines for PCB Stack-Up

The PCB stack-up (or layer assignment) is an important factor in ensuring optimal performance of a power distribution scheme. An optimized PCB stack-up for improved power integrity performance can be achieved by following these recommendations:

- Power and ground plane pairs/“islands” should be closely coupled together. The capacitance formed between the planes can be used to decouple the power supply. Whenever possible the power and ground planes should be solid to provide a continuous return path for return current.
- Use a thin dielectric between the power and ground plane pair. Capacitance is inversely proportional to the separation of the plane pair so minimizing the separation distance (the dielectric thickness) will help to maximize the capacitance.
- Keep the power and ground plane pair as close to the PCB top and bottom surfaces as possible (see [Figure 2-1](#)). This will help to minimize the associated loop inductance of the decoupling capacitors, vias, and the power/ground plane pair spreading loop inductance.

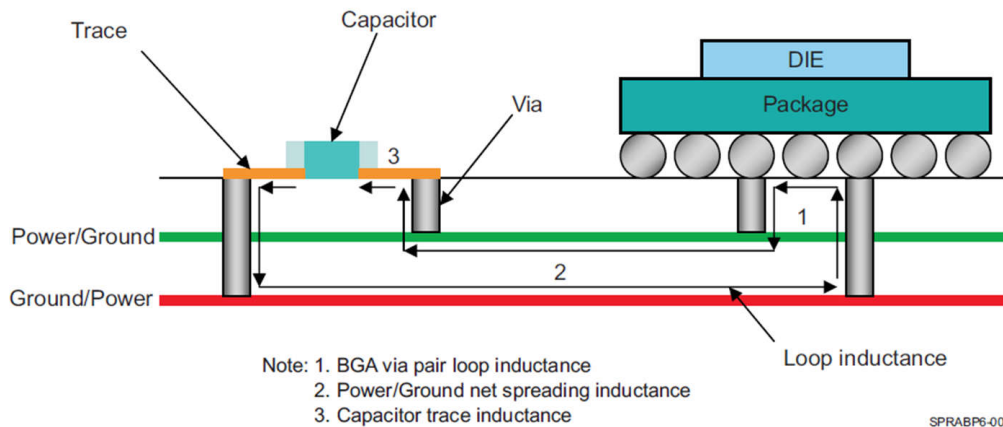


Figure 2-1. Minimize Loop Inductance by Optimizing Layer Assignments in the PCB

The placement of power and ground planes in the PCB stack-up (determined by layer assignment) has a significant impact on the parasitic inductance of the power current path as shown above. For this reason, it is recommended to consider layer order in the early stages of the PCB PDN design cycle, putting high priority supplies in the top half of the stack-up and low-priority supplies in the bottom half of the stack-up as shown in the examples below. [Figure 2-2](#) and [Figure 2-3](#) show examples of typical PCB stack-ups designed with power distribution performance in mind. Device-specific stack-up examples can be found in Section 8.

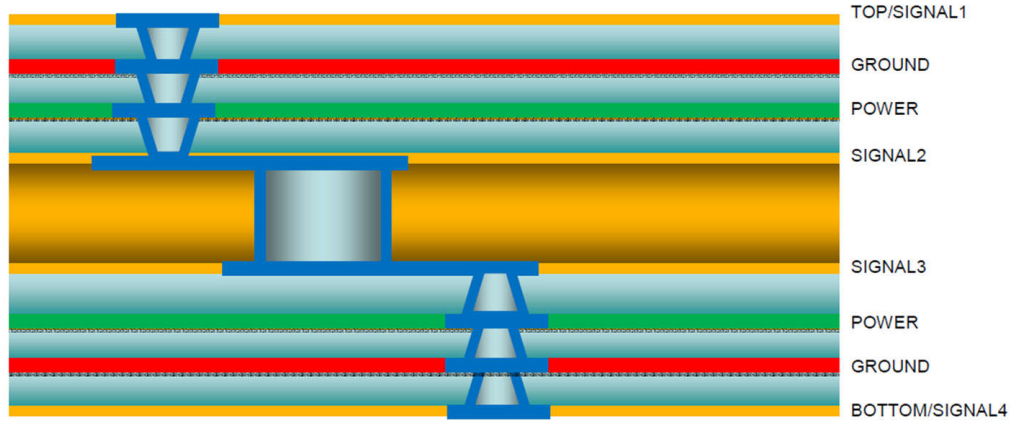


Figure 2-2. Example Stack-Up Utilizing High-Density Interconnect Vias

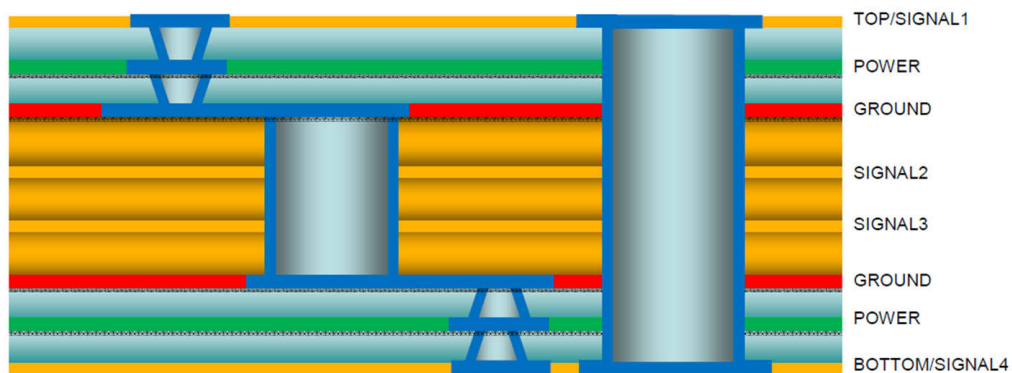


Figure 2-3. Example Stack-Up Utilizing Plated-Through-Hole (PTH) Vias

3 Physical Layout Optimization of the PDN

The following are important requirements that need to be implemented in the PCB PDN design:

- External power trace routing between components should be as wide as possible as wider traces result in reduced DC resistance and consequently a lower static IR drop. They also provide for lower loop inductance and higher capacitance.
- Whenever possible, attempt to achieve a ratio of 1:1 (or better) for component pins and associated vias. Do not share vias among multiple capacitors.
- Placement of decoupling capacitors and associated vias should be as close to the processor ball as possible.
- The maximum current-carrying capacity of each transitional via should be evaluated through simulation to determine the appropriate number of vias required to connect components. This ensures that the current-carrying ability of a via interconnect network is sufficient for the needs of each particular component. When a via interconnect, or a network of same, is unable to supply sufficient current, this is referred to as “via starvation”.
- TI highly recommends that both static and dynamic IR drop analysis (discussed in [Section 4](#) and [Section 5](#)) be performed on any new PCB design prior to fabrication. These analyses should be used to assess the appropriate number of vias and geometrical trace width dimensions required to meet the IR drop requirements of system components.
- Whenever possible for the internal layers (both signal routing and power plane), wide traces and copper area fills are recommended for PDN layout. As discussed in previous sections of this document, routing power nets in planes provides for more inter-plane capacitance and improves high frequency performance of the PDN.
- Decoupling capacitors should be mounted with minimum impact to inductance. A capacitor has characteristics not only of capacitance but also inductance and resistance. [Figure 3-1](#) shows the parasitic model of a real capacitor. A real capacitor should be treated as an RLC circuit with effective series resistance (ESR) and effective series inductance (ESL).



Figure 3-1. Characteristics of a “Real” Capacitor With ESL and ESR

The magnitude of the impedance of this series model is given in [Equation 1](#).

$$|Z| = \sqrt{ESR^2 + \left(wESL - \frac{1}{wC} \right)^2}$$

where

$$w = 2\pi f$$

(1)

Figure 3-2 shows the resonant frequency response of a typical capacitor with self-resonant frequency of 55 MHz. The impedance of the capacitor is a combination of its series resistance and reactive capacitance and inductance as shown per Equation 1.

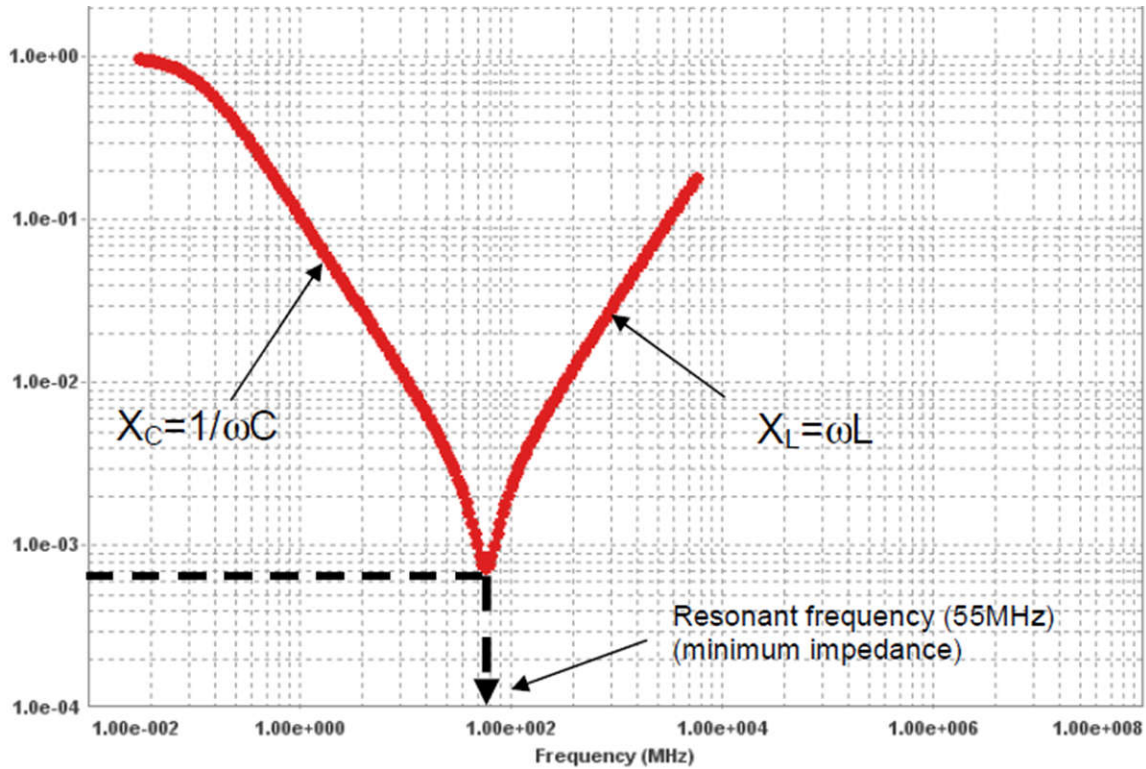


Figure 3-2. Typical Impedance Profile of a Capacitor

- Try to avoid different power nets coupling on the PCB by using “co-planar” shielding whenever appropriate. Figure 3-3 depicts an example of co-planar shielding for two different power nets (VDD_MPU_IVA and VDD_CORE).

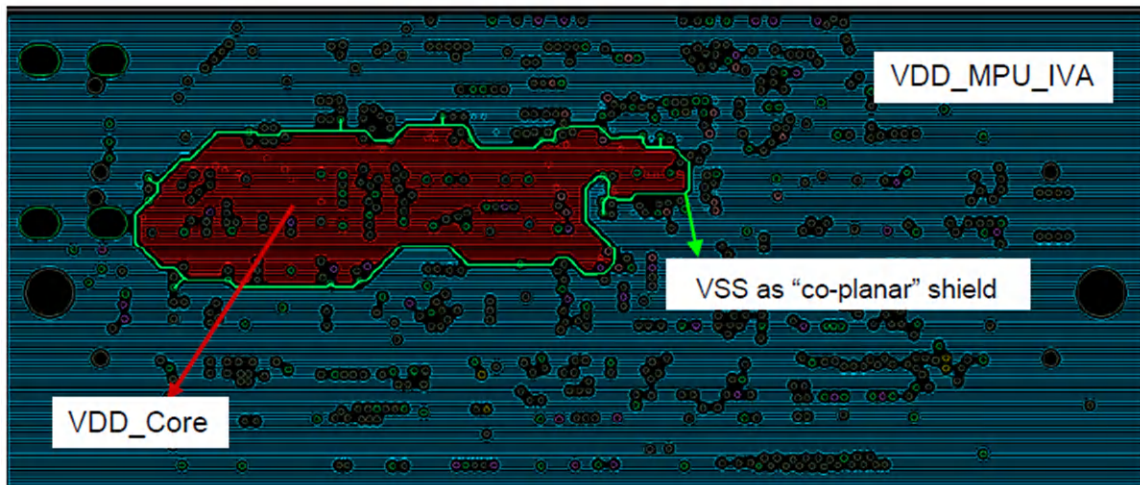


Figure 3-3. Example of “Co-Planar” Shielding of Power Net Using Ground Guard-Band

As the capacitors have both series inductance and resistance that will impact their effectiveness, it is critical that the following recommendations are adopted in placing them on the power distribution network. Whenever possible make sure to mount the capacitor with the geometry that will minimize the mounting inductance and resistance. The capacitor mounting inductance and resistance includes the inductance and resistance of the pads, the trace, and the associated vias.

The length of a trace used to connect a capacitor has a significant impact on the parasitic inductance and resistance of the mounting. This trace should be as short and wide as possible. Wherever possible, minimize the trace by locating vias near the solder pad landing. Further improvements can be made to the mounting by placing vias to the side of capacitor lands or by doubling the number of vias. If the PCB manufacturing process allows, and if cost-effective, via-in-pad (VIP) geometries are strongly recommended.

The most common via placement geometries are proved below, in order of preference for reducing parasitic impact:

1. Via-In-Pad (VIP)
2. Dual Offset Via
3. Single Offset Via
4. Pad to Via Trace (short)
5. Pad to Via Trace (long)

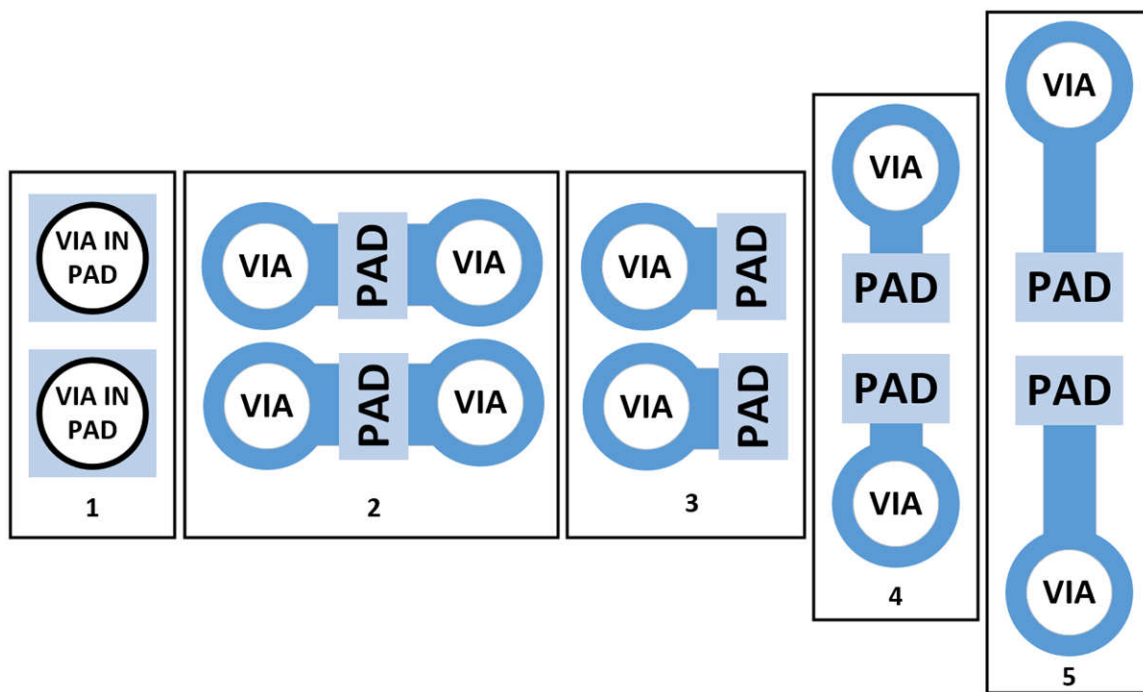


Figure 3-4. Capacitor Mounting Geometries

In addition to mounting inductance and resistance associated with placing a capacitor on the PCB, the effectiveness of a decoupling capacitor also depends on the spreading inductance and resistance that the capacitor sees with respect to the load. The spreading inductance and resistance is strongly dependent on the layer assignment in the PCB stack-up (see [Figure 2-1](#)).

4 Static PDN Analysis (IR Drop Optimization)

Delivering reliable power to circuits is always of critical importance as IR drops can occur at every level in a chip, package, and board system. Components that are distant from their associated power source are particularly susceptible to IR drop, and designs that rely on battery power must further minimize voltage drop to avoid unacceptable power loss. Early DC assessments made through simulation help to determine power distribution basics such as the best available entry point for power, layer stack-up choices, and estimates for the amount of copper needed to carry the required current.

The resistance R_s of a plane conductor for a unit length and unit width is called the **surface resistivity** (ohms per square).

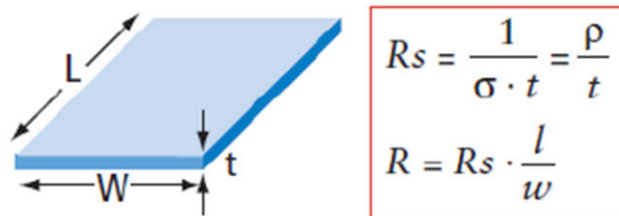


Figure 4-1. Depiction of Sheet Resistivity and Resistance

Ohm's Law ($V=IR$) relates conduction current to voltage drop, and at DC, the relation coefficient is a constant representing the resistance of the conductor. Conductors also dissipate power due to their resistance. Both voltage drop and power dissipation are proportional to the resistance of the conductor. Static IR or DC analysis/design methodology consists of designing the power distribution network such that the voltage drop (under DC operating conditions) across the power and ground pads of the application processor device is within a specified value of the nominal voltage to ensure proper functionality of the device. The PCB-level static IR drop budget is defined between the *pins/pads* of the power management device (PMIC/VRM/SMPS) and the *BGA balls* on the application processor device to which the power management device is supplying power (see [Figure 4-2](#)).

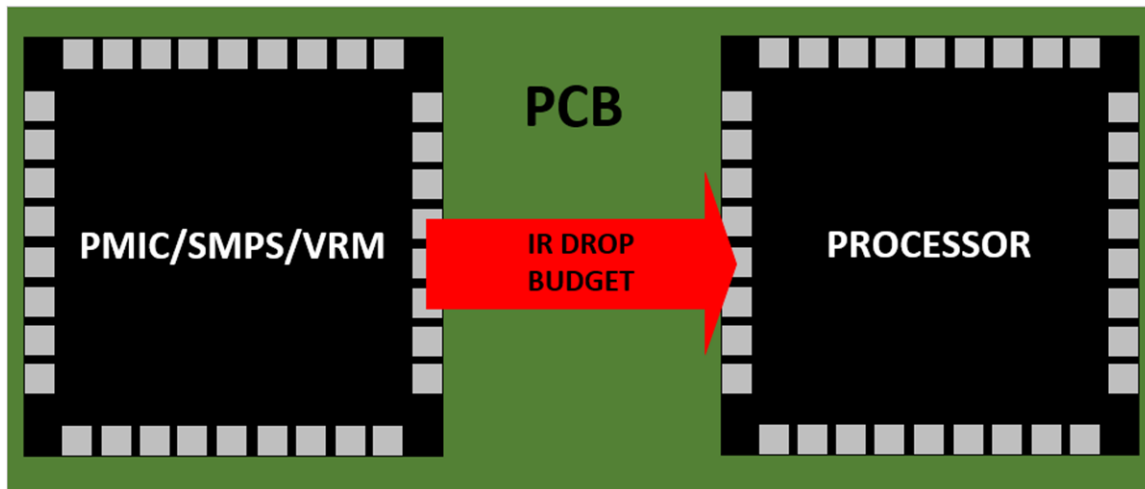


Figure 4-2. PCB IR Drop Budget

Given the total system-level margin allowed for proper device functionality, allowable voltage variation at the BGA of the device is typically specified at 2.5% of the nominal voltage.¹ For devices implementing remote-sense functionality, it is a requirement that the power management device feedback/sense line(s) be placed as close to the relevant processor power balls as physically possible (see Figure 4-3) and that a supply input voltage difference of ≤ 5 mV under maximum current loading is maintained across all balls connected to a common power rail. This 5 mV maximum represents any voltage difference that may exist between a remote sense point and any associated power input (see Figure 4-4).

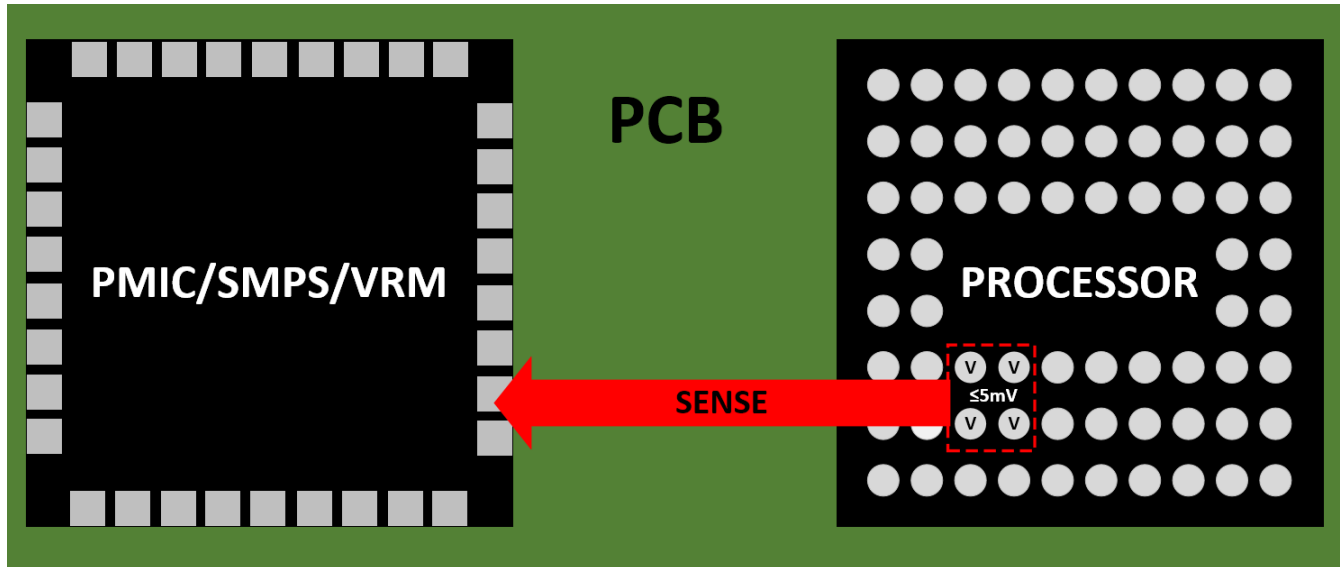


Figure 4-3. Sense Line Placement

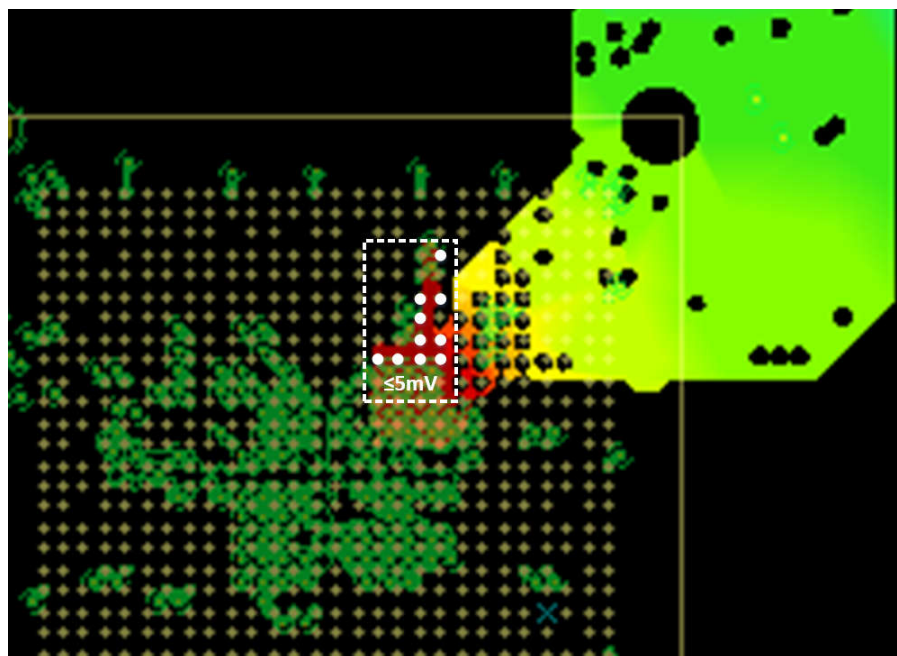


Figure 4-4. Allowable Power Input Voltage Difference

¹ This is a generic recommendation and may not apply to your specific processor. For processor requirements, see the device-specific data sheet.

5 Dynamic Analysis of PCB PDN

The typical elements of the PDN are shown in Figure 5-1; which includes the chip-level power distribution with thin-oxide decoupling capacitors. The package-level power distribution with planes and mid-frequency decoupling capacitors; and the board-level (for example, PCB) power distribution with planes, low-frequency ceramic and bulk decoupling capacitors, and the voltage regulator module (VRM).

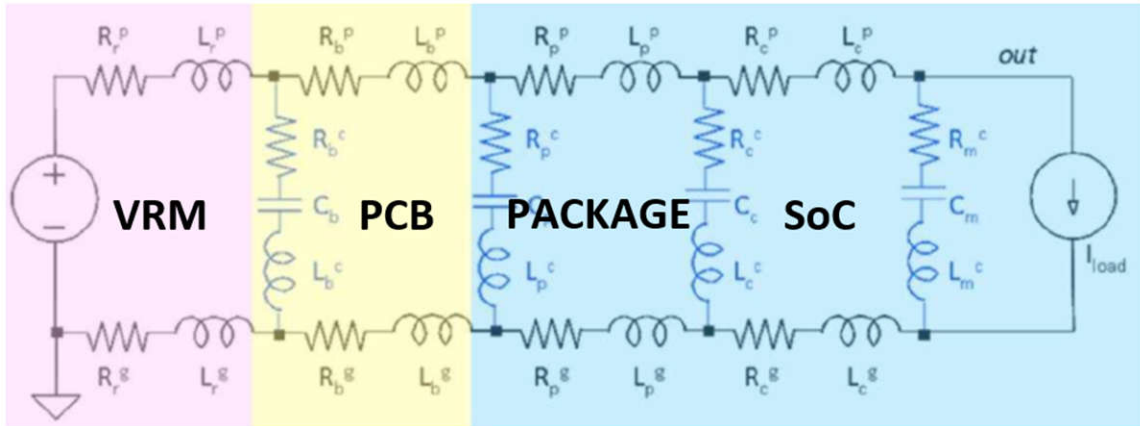


Figure 5-1. Components of a Typical Power Distribution Network (PDN)

The frequency ranges covered by these elements are shown in Figure 5-2. As the primary focus is on optimizing the PCB PDN for high performance, the methodology is developed around the areas over which the PCB designer has control over and can influence early in the design phase.

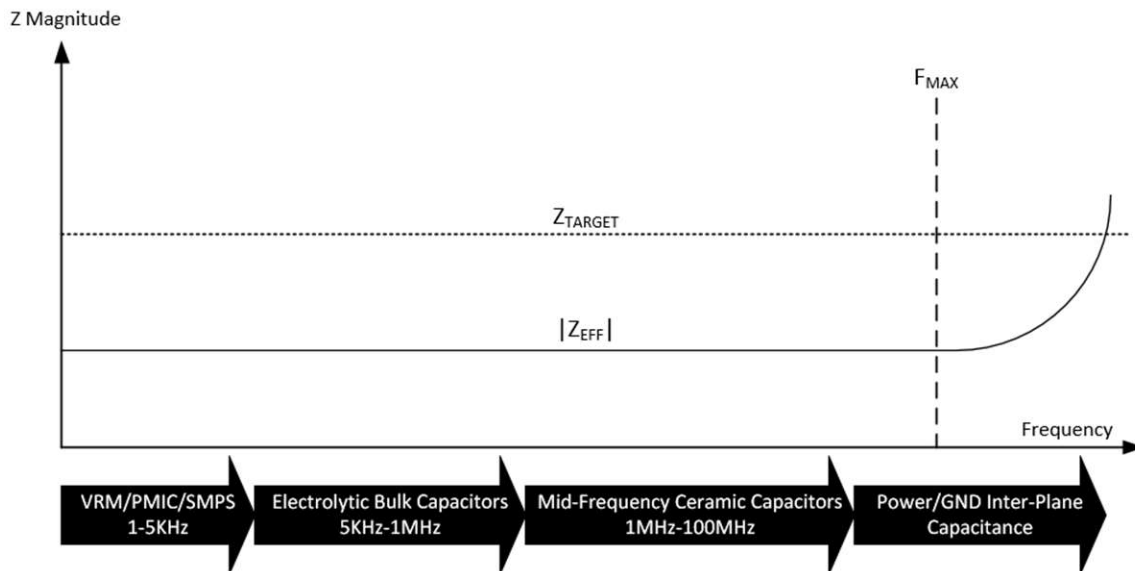


Figure 5-2. Decoupling Frequency Range of PCB Components

The VRM/PMIC/SMPS (or simply voltage regulator device) is the first major component of the PDN. It observes its output voltage and adjusts the amount of current being supplied to keep the voltage constant. Most common voltage regulators make this adjustment on the order of milliseconds to microseconds. They are effective at maintaining output voltage for events at all frequencies from DC to a few kilohertz (depending on the regulator dynamic response time). For all transient events that occur at frequencies above this range, there is a time delay before the voltage regulator can respond to the new level of demand. During this time delay the rail suffers from voltage droop. A power delivery network has an impedance (Z_{PDN}) associated with the path from the voltage regulator module to the processor. The magnitude of noise (voltage ripple) seen on a given power rail is proportional to the impedance (Z_{PDN}) and the transient current ($I_{TRANSIENT}$) draw associated with that rail.

Based on Ohm's Law,

$$V_{\text{RIPPLE}} = I_{\text{TRANSIENT}} \times Z_{\text{PDN}} \quad (2)$$

Typically the transient current is application-specific and is determined by a particular switching scenario. As a board designer, you have the ability to minimize the voltage ripple by reducing Z_{PDN} either by reducing the inductance, or by maximizing the capacitance. To ensure that the voltage ripple noise is within the processor's specification, the Z_{PDN} must be designed to meet a certain impedance, referred to as the target impedance (Z_{TARGET}). Using frequency domain target impedance method (FDTIM) to describe the behavior of a power delivery system has been widely accepted.

The key concept of the FDTIM is the determination of the target impedance Z_{TARGET} (see [Equation 3](#)) for the power rail under consideration. For reliable operation of a power delivery system, its impedance spectrum needs to be maintained below the target value at the frequencies from DC to F_{MAX} (see [Figure 5-2](#)). F_{MAX} is the point in frequency after which adding a reasonable number of decoupling capacitors does not bring the power rail impedance $|Z_{\text{EFF}}|$ below the target impedance (Z_{TARGET}) due to the dominance of the parasitic planar spreading inductance and package inductances.

$$Z_{\text{TARGET}} = \frac{\text{Voltage Rail} \times \% \text{Ripple}}{0.5 \times I_{\text{Max}}} \quad (3)$$

5.1 Selecting Decoupling Capacitors to Meet Z_{TARGET}

To maintain power integrity throughout the entire frequency range of interest, the power distribution network relies on the voltage regulator module (VRM/SMPS), the on-board discrete bulk electrolytic and ceramic decoupling capacitors, and the inter-plane capacitances (capacitance from the power-ground sandwich in the board stack-up). For a first-order analysis, the VRM of the power management integrated circuit (PMIC) can be modeled as a series-connected resistor and inductor. A PMIC, at low frequencies (up to 500 KHz-1 MHz), typically has low impedance and is capable of responding to the instantaneous requirements of the processor. The ESR and ESL values for the VRM are therefore very low. Beyond lower frequencies, the VRM impedance is primarily inductive, making it incapable of meeting the transient current requirement of the device. The bulk and ceramic discrete decoupling capacitors must provide the required low impedance from the point at which the VRM becomes inductive. The effectiveness of the bulk and mid-frequency decoupling capacitors (1 MHz-70 MHz, depending on the capacitor's ESL and ESR) is limited by its placement (due to loop inductance), value, and type. Refer to [Figure 5-3](#) for a capacitor placement example. Note that in order to minimize the loop inductance, the mid-frequency capacitors have been placed directly underneath the processor (on the bottom side of the PCB).

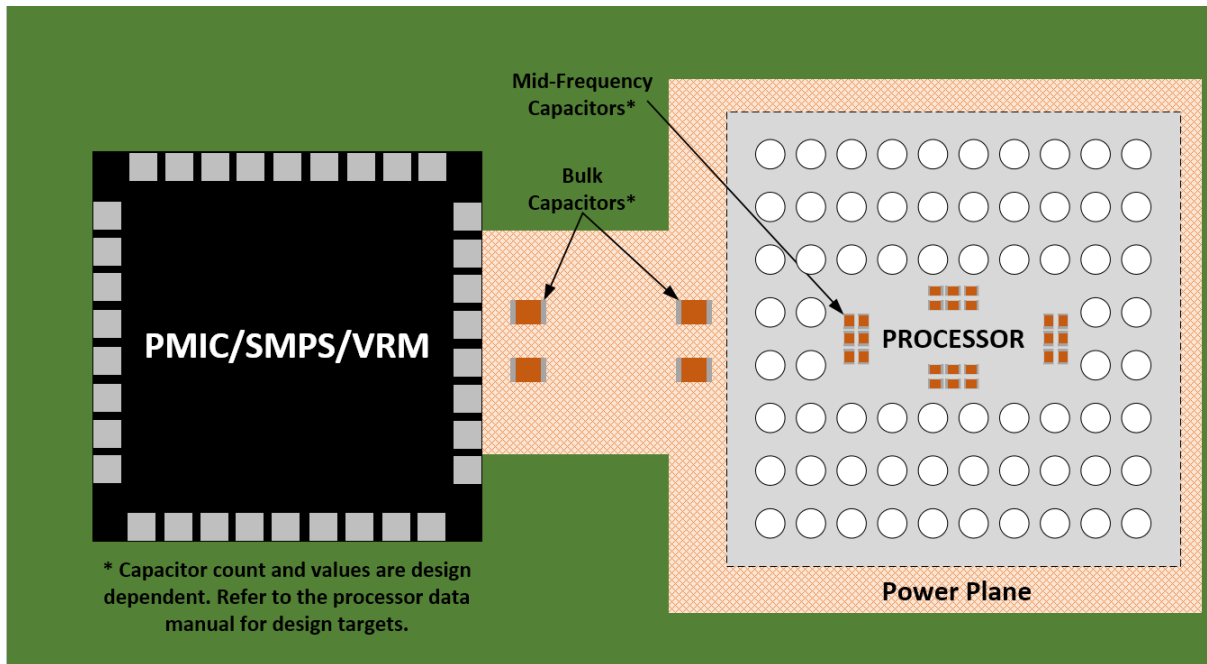


Figure 5-3. Example Capacitor Placement

The bulk capacitors should be located near the point of entry of the supply entrance to the board. The decoupling capacitors maintain the PDN impedance at the required value beyond the VRM frequency and until the frequency at which mid-frequency capacitors become useful. The mid-frequency SMT capacitors are useful in the 10 to 150 MHz range and higher. These capacitors are primarily ceramic capacitors that come in several dielectric types (NPO, X7R, X5R, and Y5V) and several sizes (1206, 0805, 0603, 0402, and so forth). The mid-frequency capacitors are much smaller than the bulk capacitors and can be placed closer to the transistor circuit. Since the ceramic capacitors are smaller, they have lower ESR and ESL and lower capacitance than bulk capacitors, leading to a higher resonance frequency with smaller impedance at resonance. Therefore, ceramic capacitors can be used at higher frequencies. Typical mid-frequency capacitors have capacitance in the range of 1 nF – 100 nF, ESR in the range of 10-100 mΩ, and ESL in the range of 0.5 nH – 1 nH.

The concept of “loop inductance” is a useful metric for quantifying the effectiveness of the decoupling capacitors of a power distribution network. To calculate the “loop inductance” associated with decoupling capacitor placement, [Equation 4](#) can be used:

$$L_{\text{eff}} = \frac{\text{Imaginary}(Z(\text{Power, GND pads of decap}))}{2 \times \pi \times \text{Frequency}} \quad (4)$$

Where L_{eff} is the effective loop inductance, $Z(\text{power, GND pads of decap})$ represents the Z-parameters of the port defined across the power and ground pads of the corresponding decap. The frequency should be chosen in the “flat” region of the Z-parameter response, typically in the 50 MHz – 70 MHz range.

6 Checklist for PDN

Although this list is not inclusive of every parameter and variable that must be considered when designing a PCB, a PDN-optimized PCB design will implement these guidelines:

- Power and ground plane pairs (or “islands”) should be closely coupled together. The capacitance formed between the planes can be used to decouple the power supply at high frequencies.
- Whenever possible, the power and ground planes should be solid as this provides a continuous return path for return current.
- Use a thin dielectric thickness between the power and ground plane pair. Capacitance is inversely proportional to the separation of the plane pair so minimizing the separation distance (such as, the dielectric thickness) will maximize the resulting capacitance.
- The placement of power and ground planes in the PCB stack-up (determined by layer assignment) has a significant impact on the parasitic inductances of power current path. For this reason, it is recommended to consider layer order in the early stages of the PCB PDN design cycle, placing high priority supplies in the upper-half of the stack-up and low priority supplies in the lower half of the stack-up. This helps to minimize the loop inductance caused by decoupling capacitors and their associated vias.
- External power trace routing between components should be as wide as possible as wider traces result in reduced DC resistance and consequently a lower static IR drop.
- Whenever possible, attempt to achieve a ratio of 1:1 (or better) for component pins and associated vias. Do not share vias among multiple capacitors.
- Placement of decoupling capacitors and their associated vias should be as close to the processor ball as possible. Reserve the space directly underneath the processor for this purpose.
- Use of short and wide surface traces to connect capacitor pads to the vias connected to the planes below is preferred.
- Use of large diameter vias is preferred for reduced inductance/resistance.
- 1 oz – 2 oz Cu weight for power / ground plane is preferred to enable better PCB heat spreading, which helps to reduce processor junction temperatures. In addition, it is preferable to have the power / ground planes be located adjacent to the PCB surface on which the processor is mounted.
- Place the VRM as close as possible to the processor and on the same side of the PCB. In cases where a power management IC (PMIC) is implemented as the VRM, it should be aligned to minimize distance for the highest current rail.

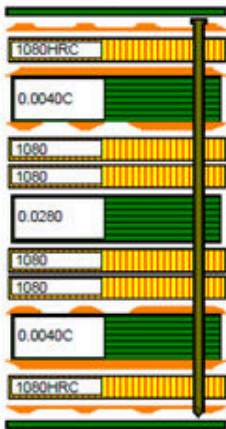
7 Implementation Examples and PDN Targets

This section provides device-specific PDN targets and implementation examples that meet these targets.

Note







The decoupling capacitor counts and/or capacitor values presented in this document are PCB design-specific and should not be used in lieu of PCB simulations. It is the responsibility of the PCB designer to ensure that any design meets the provided PDN targets.

7.1 AM570x

Layer	Calc Thickness	Primary Stack	Description	Dk / Df	
Layer - 1	0.0005		Taiyo 4000-MP 1/4oz Sig (Std Plt)	4.70 / 0.0330	
	0.0020		370H	4.03 / 0.0210	
Layer - 2	0.0033		0.0040C	1oz P/G	4.54 / 0.0190
	0.0012		1080	1/2oz Mix	
Layer - 3	0.0041		1080	370H	4.17 / 0.0210
	0.0006		0.0280	370H	4.30 / 0.0180
Layer - 4	0.0054		1080	370H	4.16 / 0.0210
	0.0280		1080	1/2oz Mix	
Layer - 5	0.0055		0.0040C	370H	4.54 / 0.0190
	0.0006		1080HRC	1oz P/G	
Layer - 6	0.0041		1080HRC	370H	4.05 / 0.0210
	0.0012			1/4oz Sig (Std Plt) Taiyo 4000-MP	4.70 / 0.0330
Layer - 6	0.0032				
Layer - 6	0.0020				
Layer - 6	0.0005				

Materials: Isola 370H High-Tg FR4

Requirement	Req. Thickness	Tol +	Tol -	Calc Thick
Incl. Plating & Mask	0.0620	0.0062	0.0062	0.0622
Incl. Mask over Laminate	0.0580	0.0058	0.0058	0.0582
Incl. Plating	0.0610	0.0061	0.0061	0.0612
After Lamination	0.0576	0.0029	0.0029	0.0578
Over Laminate	0.0570	0.0057	0.0057	0.0572

Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
1  Surface MS	L1	0.0050	0.0050	-	-	50	5	49.89
	-	-	-	-	L2			
2  EC Microstrip	L1	0.00480	0.0046	0.0110	-	90	9	90.71
	-	0.00480	0.0046	-	L2			
3  EC Microstrip	L1	0.00440	0.0042	0.0140	-	100	10	100.04
	-	0.00440	0.0042	-	L2			
4  Stripline	L3	0.00525	0.0054	-	L2	50	5	50.01
	-	-	-	-	L5			
5  EC Stripline	L3	0.00450	0.0045	0.0140	L2	100	10	100.18
	-	0.00450	0.0045	-	L5			
6  EC Stripline	L3	0.0050	0.0050	0.0110	L2	90	9.0	89.67
	-	0.0050	0.0050	-	L5			







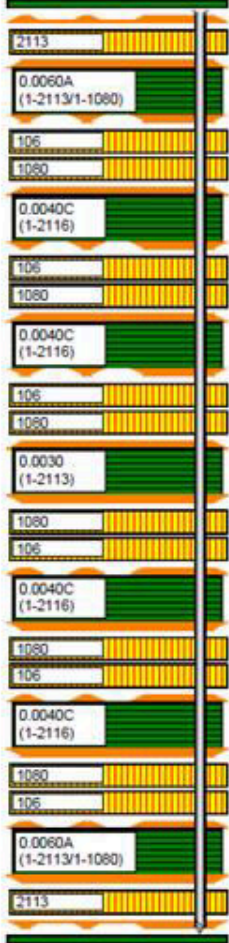
Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
 Stripline	L4	0.00525	0.0054	-	L2	50	5	50.03
	-	-	-	-	L5			
 EC Stripline	L4	0.00450	0.0045	0.0140	L2	100	10	100.22
	-	0.00450	0.0045	-	L5			
 EC Stripline	L4	0.0050	0.0050	0.0110	L2	90	9	89.71
	-	0.0050	0.0050	-	L5			
 EC Microstrip	L6	0.00440	0.0041	0.0140	L5	100	10	99.6
	-	0.00440	0.0041	-	-			
 EC Microstrip	L6	0.00480	0.0046	0.0110	L5	90	9	89.44
	-	0.00480	0.0046	-	-			
 Surface MS	L6	0.0050	0.0048	-	L5	50	5	49.88
	-	-	-	-	-			

Table 7-1. AM570x PDN Targets and Decoupling Example

Supply Name ⁽¹⁰⁾	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply ^{(1) (2) (3) (4) (5) (6) (9)}							
	Max R _{eff} (mΩ) ⁽⁷⁾	Dec Cap Max LL (nH) ^{(6) (8)}	Max Impedance (mΩ)	Frequency of Interest (MHz)	100 nF	220 nF	470 nF	1 μF	2.2 μF	4.7 μF	10 μF	22 μF
VDD_CORE	18	2	57	≤20			5	4			1	
VDD_DSP	22	2.5	54	≤20			6	5			2	
VDDS_DDR1	10	2.5	200	≤100			12	1			1	
CAP_VBBLDO_DSP	N/A	6	N/A	N/A				1				
CAP_VBBLDO_GPU	N/A	6	N/A	N/A				1				
CAP_VBBDLO_IVA	N/A	6	N/A	N/A				1				
CAP_VBBLD0_MPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE2	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE3	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE4	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSP	N/A	6	N/A	N/A				1				
CAP_VDDRAM_GPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_IVA	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU	N/A	6	N/A	N/A				1				

- (1) For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- (2) ESL must be as low as possible and not exceed 0.5 nH.
- (3) The power delivery network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table in the *Specifications* chapter of the device-specific data manual.
- (4) The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- (5) Assuming that the external SMPS (power IC) feedback sense is taken close to processor power balls.
- (6) High-frequency (30 MHz – 70 MHz) PCB decoupling capacitors.
- (7) Maximum R_{eff} from VRM/SMPS/PMIC to processor.
- (8) Maximum loop inductance for decoupling capacitor.
- (9) Decoupling capacitor counts and values are provided as a baseline recommendation only. TI recommends that all PCB designs be simulated prior fabrication to ensure that the processor PDN requirements are met.
- (10) Ganged rails must meet all requirements of each member rail.

7.2 AM571x
















Layer	Calc Thickness	Primary Stack	Description	Dk / Df	
Layer - 1	0.0005 0.0017		Taiyo 4000-BN 1/4oz Mix (Std Plt)	4.71 / 0.0330	
Layer - 2	0.0037		2113	370H	4.34 / 0.0210
Layer - 3	0.0012		0.0060A (1-2113/1-1080)	1oz P/G 370H	4.33 / 0.0210
Layer - 4	0.0060		106	1/2oz Mix 370H	4.06 / 0.0210
Layer - 5	0.0006		1080	1/2oz P/G 370H	4.54 / 0.0190
Layer - 6	0.0048		0.0040C (1-2116)	1/2oz Mix 370H	4.06 / 0.0210
Layer - 7	0.0006		106	1/2oz P/G 370H	4.54 / 0.0190
Layer - 8	0.0048		1080	1/2oz Mix 370H	4.06 / 0.0210
Layer - 9	0.0006		0.0040C (1-2116)	1/2oz P/G 370H	4.57 / 0.0190
Layer - 10	0.0048		106	1/2oz P/G 370H	4.06 / 0.0210
Layer - 11	0.0006		1080	1/2oz Mix 370H	4.54 / 0.0190
Layer - 12	0.0048		106	1/2oz P/G 370H	4.06 / 0.0210
Layer - 13	0.0006		0.0040C (1-2116)	1/2oz Mix 370H	4.54 / 0.0190
Layer - 14	0.0048		1080	1/2oz P/G 370H	4.06 / 0.0210
Layer - 15	0.0006		106	1/2oz Mix 370H	4.33 / 0.0210
Layer - 16	0.0060		0.0060A (1-2113/1-1080)	1oz P/G 370H	4.34 / 0.0210
	0.0012	2113	1/4oz Mix (Std Plt) Taiyo 4000-BN	4.71 / 0.0330	
	0.0037				
	0.0017				
	0.0005				

Materials: Isola 370H High-Tg FR4

Requirement	Req. Thickness	Tol +	Tol -	Calc Thick
Incl. Plating & Mask	0.0800	0.0080	0.0080	0.0816
Incl. Mask over Laminate	0.0766	0.0077	0.0077	0.0782
Incl. Plating	0.0790	0.0079	0.0079	0.0806

Requirement	Req. Thickness	Tol +	Tol -	Calc Thick
After Lamination	0.0762	0.0038	0.0038	0.0778
Over Laminate	0.0756	0.0076	0.0076	0.0772

Job Comment
Sim to pn 516581 Rev F and 517391A

Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
 Surface MS	L1	0.00550	0.0055	-	-	50	5	49.83
	-	-	-	-	L2			
 EC Microstrip	L1	0.00470	0.0047	0.0100	-	90	9	89.73
	-	0.00470	0.0047	-	L2			
 EC Microstrip	L1	0.0040	0.0040	0.0105	-	100	10	99.29
	-	0.0040	0.0040	-	L2			
 Stripline	L3	0.00450	0.0045	-	L2	50	5	50.71
	-	-	-	-	L4			
 EC Stripline	L3	0.0050	0.0050	0.0120	L2	90	9.0	90.96
	-	0.0050	0.0050	-	L4			
 EC Stripline	L3	0.00370	0.0037	0.0090	L2	100	10	100.20
	-	0.00370	0.0037	-	L4			
 Stripline	L5	0.00370	0.0037	-	L4	50	5	49.84
	-	-	-	-	L6			
 EC Stripline	L5	0.00360	0.0036	0.0120	L4	100	10.0	98.59
	-	0.00360	0.0036	-	L6			
 Stripline	L7	0.00370	0.0037	-	L6	50	5	49.84
	-	-	-	-	L8			
 EC Stripline	L7	0.00360	0.0036	0.0120	L6	100	10.0	98.59
	-	0.00360	0.0036	-	L8			
 Stripline	L10	0.00370	0.0037	-	L9	50	5	49.84
	-	-	-	-	L11			
 EC Stripline	L10	0.00360	0.0036	0.0120	L9	100	10.0	98.59
	-	0.00360	0.0036	-	L11			
 Stripline	L12	0.00370	0.0037	-	L11	50	5	49.84
	-	-	-	-	L13			
 EC Stripline	L12	0.00360	0.0036	0.0120	L11	100	10.0	98.59
	-	0.00360	0.0036	-	L13			
 Stripline	L14	0.00450	0.0045	-	L13	50	5	50.71
	-	-	-	-	L15			








Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
 EC Stripline	L14	0.00370	0.0037	0.0090	L13	100	10	100.20
	-	0.00370	0.0037	-	L15			
 Surface MS	L16	0.00550	0.0055	-	L15	50	5	49.84
	-	-	-	-	-			
 EC Microstrip	L16	0.00470	0.0047	0.0100	L15	90	9	89.73
	-	0.00470	0.0047	-	-			
 EC Microstrip	L16	0.0040	0.0040	0.0105	L15	100	10	99.29
	-	0.0040	0.0040	-	-			
 EC Microstrip	L1	0.00350	0.0035	0.0120	-	110	11.0	109.02
	-	0.00350	0.0035	-	L2			
 EC Microstrip	L1	0.00560	0.0056	0.0120	-	87	8.7	86.46
	-	0.00560	0.0056	-	L2			
 EC Microstrip	L16	0.00560	0.0056	0.0120	L15	87	8.7	86.46
	-	0.00560	0.0056	-	-			

Table 7-2. AM571x PDN Targets and Decoupling Example

Supply Name ⁽¹⁰⁾	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply ^{(1) (2) (3) (4) (5) (6) (9)}							
	Max R_{eff} (m Ω) ⁽⁷⁾	Dec Cap Max LL (nH) ^{(6) (8)}	Max Impedance (m Ω)	Frequency of Interest (MHz)	100 nF	220 nF	470 nF	1 μ F	2.2 μ F	4.7 μ F	10 μ F	22 μ F
VDD_CORE	27	2	87	≤ 50	6	1	1	1	1	1		
VDD_MPU	10	2	57	≤ 20	8	1	1	1	1	1		1
VDD_DSP VDD_GPU VDD_IVA	13	2.5	54	≤ 20	8	1	1	1	1	1	1	1
VDDS_DDR1	10	2.5	200	≤ 100	8	4		2		2		1
CAP_VBBLDO_DSP	N/A	6	N/A	N/A				1				
CAP_VBBLDO_GPU	N/A	6	N/A	N/A				1				
CAP_VBBDLO_IVA	N/A	6	N/A	N/A				1				
CAP_VBBLDO_MPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE3	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE4	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSP	N/A	6	N/A	N/A				1				
CAP_VDDRAM_GPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_IVA	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU	N/A	6	N/A	N/A				1				

- (1) For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- (2) ESL must be as low as possible and not exceed 0.5 nH.
- (3) The power delivery network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table in the *Specifications* chapter of the device-specific data manual.
- (4) The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- (5) Assuming that the external SMPS (power IC) feedback sense is taken close to processor power balls.
- (6) High-frequency (30 MHz – 70 MHz) PCB decoupling capacitors.
- (7) Maximum R_{eff} from VRM/SMPS/PMIC to processor.
- (8) Maximum loop inductance for decoupling capacitor.
- (9) Decoupling capacitor counts and values are provided as a baseline recommendation only. TI recommends that all PCB designs be simulated prior fabrication to ensure that the processor PDN requirements are met.
- (10) Ganged rails must meet all requirements of each member rail.

7.3 AM572x

Layer	Calc Thickness	Primary Stack	Description	Dk / Df
Layer - 1	0.0005 0.0017		Taiyo 4000-BN 1/4oz Mix (Std Plt)	4.71 / 0.0330
Layer - 2	0.0039		370H	3.95 / 0.0210
Layer - 3	0.0025 0.0060		2oz P/G 370H	4.33 / 0.0210
Layer - 4	0.0006 0.0047		1/2oz Mix 370H	4.07 / 0.0210
Layer - 5	0.0006 0.0035		1/2oz P/G 370H	4.40 / 0.0210
Layer - 6	0.0006 0.0039		1/2oz Mix 370H	3.95 / 0.0210
Layer - 7	0.0006 0.0035		1/2oz P/G 370H	4.40 / 0.0210
Layer - 8	0.0006 0.0040		1/2oz Mix 370H	3.94 / 0.0210
Layer - 9	0.0006 0.0030		1/2oz P/G 370H	4.57 / 0.0190
Layer - 10	0.0006 0.0040		1/2oz Mix 370H	3.94 / 0.0210
Layer - 11	0.0006 0.0035		1/2oz P/G 370H	4.40 / 0.0210
Layer - 12	0.0006 0.0040		1/2oz Mix 370H	3.94 / 0.0210
Layer - 13	0.0006 0.0035		1/2oz P/G 370H	4.40 / 0.0210
Layer - 14	0.0006 0.0048		1/2oz Mix 370H	4.06 / 0.0210
Layer - 15	0.0006 0.0060		1/2oz Mix 370H	4.33 / 0.0210
Layer - 16	0.0025 0.0039 0.0005		2oz P/G 370H 1/4oz Mix (Std Plt) Taiyo 4000-BN	3.95 / 0.0210 4.71 / 0.0330

Materials: Isola 370H High-Tg FR4

Requirement	Req. Thickness	Tol +	Tol -	Calc Thick
Incl. Plating & Mask	0.0800	0.0080	0.0080	0.0788
Incl. Mask over Laminate	0.0766	0.0077	0.0077	0.0754
Incl. Plating	0.0790	0.0079	0.0079	0.0778
After Lamination	0.0762	0.0038	0.0038	0.0750
Over Laminate	0.0756	0.0076	0.0076	0.0744
















Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
 Surface MS	L1	0.0050	0.0060	-	-	50	5	51.11
	-	-	-	-	L2			
 EC Microstrip	L1	0.00440	0.0052	0.0105	-	90	9	90.68
	-	0.00440	0.0052	-	L2			
 EC Microstrip	L1	0.00380	0.0048	0.0120	-	100	10	99.26
	-	0.00380	0.0048	-	L2			
 Stripline	L3	0.00450	0.0045	-	L2	50	5	50.37
	-	-	-	-	L4			
 EC Stripline	L3	0.0050	0.0050	0.0120	L2	90	9	90.45
	-	0.0050	0.0050	-	L4			
 Stripline	L5	0.0040	0.0033	-	L4	50	5	48.65
	-	-	-	-	L6			
 Stripline	L7	0.00330	0.0033	-	L6	50	5	48.96
	-	-	-	-	L8			
 Stripline	L10	0.00330	0.0033	-	L9	50	5	48.96
	-	-	-	-	L11			
 Stripline	L12	0.0040	0.0033	-	L11	50	5	48.96
	-	-	-	-	L13			
 Stripline	L14	0.00450	0.0045	-	L13	50	5	50.71
	-	-	-	-	L15			
 Surface MS	L16	0.0050	0.0060	-	L15	50	5	51.11
	-	-	-	-	-			
 EC Microstrip	L16	0.00350	0.0044	0.0105	L15	100	10	99.49
	-	0.00350	0.0044	-	-			
 EC Microstrip	L1	0.00380	0.0038	0.0120	-			110.77
	-	0.00380	0.0038	-	L2			
 EC Microstrip	L1	0.0060	0.0060	0.0120	-			87.59
	-	0.0060	0.0060	-	L2			
 EC Microstrip	L16	0.0060	0.0060	0.0120	L15			87.60
	-	0.0060	0.0060	-	-			

Table 7-3. AM572x PDN Targets and Decoupling Example

Supply Name ⁽¹⁰⁾	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply ^{(1) (2) (3) (4) (5) (6) (9)}							
	Max R _{eff} (mΩ) ⁽⁷⁾	Dec Cap Max LL (nH) ^{(6) (8)}	Max Impedance (mΩ)	Frequency of Interest (MHz)	100 nF	220 nF	470 nF	1 μF	2.2 μF	4.7 μF	10 μF	22 μF
VDD_CORE	27	2	87	≤50	6	1	1	1	1	1		
VDD_MPU	10	2	57	≤20	12	2	2	3	1	1		1
VDD_DSPEVE	13	2.5	54	≤20	8	1	1	2	1	1	1	
VDD_IVA	48	2	800	≤100	5		1			1		
VDD_GPU	18	2.5	207	≤50	6	1	1	1	1	1		
VDDS_DDR1	10	2.5	200	≤100	8	4		2		2		1
VDDS_DDR2	10	2.5	200	≤100	8	4		2		2		1
CAP_VBBLDO_DSPEVE	N/A	6	N/A	N/A				1				
CAP_VBBLDO_GPU	N/A	6	N/A	N/A				1				
CAP_VBBDLO_IVA	N/A	6	N/A	N/A				1				
CAP_VBBLDO_MPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE2	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE3	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE4	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE5	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSPEVE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSPEVE2	N/A	6	N/A	N/A				1				
CAP_VDDRAM_GPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_IVA	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU2	N/A	6	N/A	N/A				1				

- (1) For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- (2) ESL must be as low as possible and not exceed 0.5 nH.
- (3) The power delivery network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table in the *Specifications* chapter of the device-specific data manual.
- (4) The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- (5) Assuming that the external SMPS (power IC) feedback sense is taken close to processor power balls.
- (6) High-frequency (30 MHz – 70 MHz) PCB decoupling capacitors.
- (7) Maximum R_{eff} from VRM/SMPS/PMIC to processor.
- (8) Maximum loop inductance for decoupling capacitor.
- (9) Decoupling capacitor counts and values are provided as a baseline recommendation only. TI recommends that all PCB designs be simulated prior fabrication to ensure that the processor PDN requirements are met.
- (10) Ganged rails must meet all requirements of each member rail.

7.4 AM574x

Layer	Cust Thickness	Calc Thickness	Primary Stack	Description	Dk / Df
Layer - 1		0.0005 0.0017		Taiyo 4000 LD1 1/4oz Sig (Std Plt)	3.50 / 0.0019
Layer - 2		0.0033	1080HRC	370H	4.03 / 0.0210
Layer - 3		0.0012		1oz P/G	
Layer - 4		0.0060 (1-1652)		370H	4.43 / 0.0200
Layer - 5		0.0006	106	1/2oz Sig	
Layer - 6		0.0044	1080	370H	4.12 / 0.0210
Layer - 7		0.0006		1/2oz P/G	
Layer - 8		0.0041	0.0040C (1-2116)	370H	4.54 / 0.0190
Layer - 9		0.0006	106	1/2oz Sig	
Layer - 10		0.0044	1080	370H	4.12 / 0.0210
Layer - 11		0.0006		1/2oz P/G	
Layer - 12		0.0041	0.0040C (1-2116)	370H	4.54 / 0.0190
Layer - 13		0.0006	106	1/2oz Sig	
Layer - 14		0.0044	1080	370H	4.12 / 0.0210
Layer - 15		0.0030 (1-2113)		1/2oz P/G	
Layer - 16		0.0006	1080	370H	4.12 / 0.0210
Layer - 17		0.0044	106	370H	4.12 / 0.0210
Layer - 18		0.0006		1/2oz Sig	
Layer - 19		0.0041	0.0040C (1-2116)	370H	4.54 / 0.0190
Layer - 20		0.0006	1080	1/2oz P/G	
Layer - 21		0.0044	106	370H	4.12 / 0.0210
Layer - 22		0.0006		1/2oz Sig	
Layer - 23		0.0041	0.0040C (1-2116)	370H	4.54 / 0.0190
Layer - 24		0.0006	1080	1/2oz P/G	
Layer - 25		0.0044	106	370H	4.12 / 0.0210
Layer - 26		0.0006		1/2oz Sig	
Layer - 27		0.0060 (1-1652)		370H	4.43 / 0.0200
Layer - 28		0.0012		1oz P/G	
Layer - 29		0.0033	1080HRC	370H	4.03 / 0.0210
Layer - 30		0.0017 0.0005		1/4oz Sig (Std Plt) Taiyo 4000 LD1	3.50 / 0.0019

Requirement	Req. Thickness	Tol +	Tol -	Calc Thick
Incl. Plating & Mask	0.0800	0.0080	0.0080	0.0784
Incl. Mask over Laminate	0.0766	0.0077	0.0077	0.0750
Incl. Plating	0.0790	0.0079	0.0079	0.0774
After Lamination	0.0762	0.0038	0.0038	0.0748


















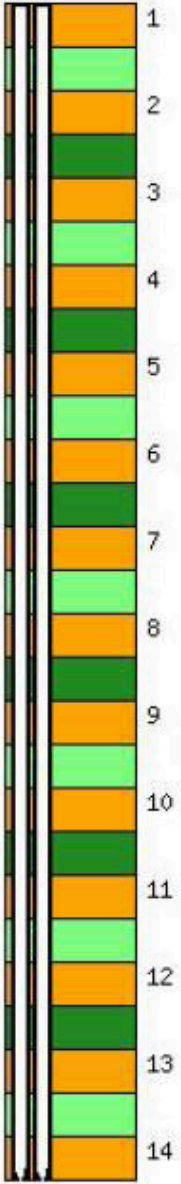
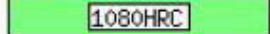

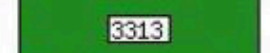

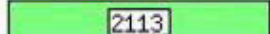

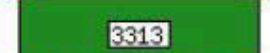

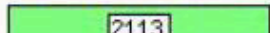

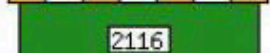

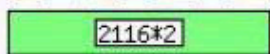





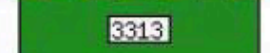

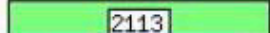
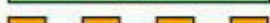
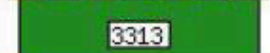

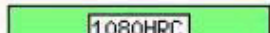


Impedance Type	Layer	Design	Actual	Plotted	Pitch	Plane	Target	Tol (ohms)	Predict
 Surface MS	L1	0.00520	0.0052	0.00558	-	-	50	5	49.67
	-	-	-	-	-	L2			
 EC Microstrip	L1	0.00390	0.0039	0.00428	0.0105	-	100	10	99.86
	-	0.00390	0.0039	0.00428	-	L2			
 EC Microstrip	L1	0.00540	0.0054	0.00578	0.0120	-	87	8.7	87.08
	-	0.00540	0.0054	0.00578	-	L2			
 Stripline	L3	0.00430	0.0043	0.0048	-	L2	50	5	50.09
	-	-	-	-	-	L4			
 EC Stripline	L3	0.00350	0.0035	0.0040	0.0090	L2	100	10	100.69
	-	0.00350	0.0035	0.0040	-	L4			
 Stripline	L5	0.00350	0.0035	0.0040	-	L4	50	5	50.05
	-	-	-	-	-	L6			
 Stripline	L7	0.00350	0.0035	0.0040	-	L6	50	5	50.05
	-	-	-	-	-	L8			
 Stripline	L10	0.00350	0.0035	0.0040	-	L9	50	5	50.05
	-	-	-	-	-	L11			
 Stripline	L12	0.00350	0.0035	0.0040	-	L11	50	5	50.05
	-	-	-	-	-	L13			
 EC Stripline	L12	0.00340	0.0034	0.0039	0.0120	L11	100	10	99.46
	-	0.00340	0.0034	0.0039	-	L13			
 Stripline	L14	0.00430	0.0043	0.0048	-	L13	50	5	50.09
	-	-	-	-	-	L15			
 EC Stripline	L14	0.00350	0.0035	0.0040	0.0090	L13	100	10	100.69
	-	0.00350	0.0035	0.0040	-	L15			
 Surface MS	L16	0.00520	0.0052	0.00558	-	L15	50	5	49.67
	-	-	-	-	-	-			
 EC Microstrip	L16	0.00390	0.0039	0.00428	0.0105	L15	100	10	99.86
	-	0.00390	0.0039	0.00428	-	-			
 EC Microstrip	L16	0.00540	0.0054	0.00578	0.0120	L15	87	8.7	87.08
	-	0.00540	0.0054	0.00578	-	-			

Table 7-4. AM574x PDN Targets and Decoupling Example

Supply Name ⁽¹⁰⁾	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply ^{(1) (2) (3) (4) (5) (9)}							
	Max R_{eff} (m Ω) ⁽⁷⁾	Dec Cap Max LL (nH) ^{(6) (8)}	Max Impedance (m Ω)	Frequency of Interest (MHz)	100 nF	220 nF	470 nF	1 μ F	2.2 μ F	4.7 μ F	10 μ F	22 μ F
VDD_MPU	18	2	57	≤ 20	2		4	5			2	
VDD_DSPEVE	22	1.6	40	≤ 30	2		4	5			2	
VDD_CORE	32	1.6	43	≤ 30			5	4			1	
VDD_GPU	22	2.1	48	≤ 30	2		4	3			1	
VDD_IVA	48	2.1	179	≤ 30	2		2	2			1	
VDDS_DDR1	18	1.5	130	≤ 100			8	1			1	
VDDS_DDR2	18	1.5	130	≤ 100			8	1			1	
CAP_VBBLDO_DSPEVE	N/A	6	N/A	N/A				1				
CAP_VBBLDO_GPU	N/A	6	N/A	N/A				1				
CAP_VBBDLO_IVA	N/A	6	N/A	N/A				1				
CAP_VBBLDO_MPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE2	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE3	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE4	N/A	6	N/A	N/A				1				
CAP_VDDRAM_CORE5	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSPEVE1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_DSPEVE2	N/A	6	N/A	N/A				1				
CAP_VDDRAM_GPU	N/A	6	N/A	N/A				1				
CAP_VDDRAM_IVA	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU1	N/A	6	N/A	N/A				1				
CAP_VDDRAM_MPU2	N/A	6	N/A	N/A				1				


























- (1) For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- (2) ESL must be as low as possible and not exceed 0.5 nH.
- (3) The power delivery network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table in the *Specifications* chapter of the device-specific data manual.
- (4) The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- (5) Assuming that the external SMPS (power IC) feedback sense is taken close to processor power balls.
- (6) High-frequency (30 MHz – 70 MHz) PCB decoupling capacitors.
- (7) Maximum R_{eff} from SMPS/PMIC to processor.
- (8) Maximum loop inductance for decoupling capacitor.
- (9) Decoupling capacitor counts and values are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior fabrication to ensure that the processor PDN requirements are met.
- (10) Ganged rails must meet all requirements of each member rail.

7.5 AM65xx/DRA80xM

Lay #	Thick (in)	Picture	Type Dk Df	Description	Drill Picture
0.0006/0.0013			4.5 0.019	Soldermask	
1	0.0020		F / S	0.5oz w/plating	
	0.0032		3.10 0.0059	fill	
2	0.0013		P	1oz	
	0.0040		3.34 0.0060	core	
3	0.0013		S	1oz	
	0.0035		3.42 0.0060	fill	
4	0.0013		P	1oz	
	0.0040		3.34 0.0060	core	
5	0.0013		S	1oz	
	0.0035		3.43 0.0060	fill	
6	0.0013		S	1oz	
	0.0050		3.36 0.0060	core	
7	0.0013		S	1oz	
	0.0095		3.41 0.0060	fill	
8	0.0013		S	1oz	
	0.0050		3.36 0.0060	core	
9	0.0013		P	1oz	
	0.0035		3.42 0.0060	fill	
10	0.0013		S	1oz	
	0.0040		3.34 0.0060	core	
11	0.0013		P	1oz	
	0.0035		3.42 0.0060	fill	
12	0.0013		S	1oz	
	0.0040		3.34 0.0060	core	
13	0.0013		P	1oz	
	0.0032		3.10 0.0059	fill	
14	0.0020		F / S	0.5oz w/plating	
0.0006/0.0013			4.5 0.019	Soldermask	
	0.0760	Total thickness (in) Over mask on plated copper			
	0.0720	After lamination thickness (in)			
	0.0734	Over laminate thickness (in) (with soldermask)			
	0.0754	Customer Requirement (in)			
	+/-0.0075	Customer Tolerance (in)			

Notes and Recommendations:
 Trace widths measured at base of trace
 All dimensions in inches (unless otherwise noted)

Impedance Constraint Information (I)

Imp #	Impedance Type		Affect Lyr		Cust L/W	Line Width		CenterToCenter		Ref Plane		Targ ohms	Tol ohms	Predicted ohms@2GHz
			(1)	(2)		(1)	(2)	(1)	(2)	Top	Bot			
1	EC CPW MS		1	None	0.0105	0.0112	0.1	0.0155	0.06925	None	4	85	8.5	85.73
2	EC MS		1	None	0.0062	0.0062	0.0062	0.0115		None	2	80	8	79.89
3	EC MS		1	None	0.0041	0.0041	0.0041	0.0085		None	2	90	9	89.50
4	EC MS		1	None	0.0042	0.0042	0.0042	0.0116		None	2	100	10	99.65
5	Surf MS		1	None	0.0082	0.0082				None	2	40	4	41.01
6	Surf MS		1	None	0.0052	0.0052				None	2	50	5	51.29
7	EC SL		3	None	0.0043	0.0043	0.0043	0.0087		4	2	80	8	79.81
8	Stripline		3	None	0.005	0.005				4	2	40	4	40.60
9	Stripline		3	None	0.0033	0.0033				4	2	50	5	49.63
10	EC SL		5	None	0.0035	0.0035	0.0035	0.0085		6	4	90	9	88.81
11	Stripline		5	None	0.005	0.005				6	4	40	4	40.39
12	Stripline		5	None	0.0033	0.0033				6	4	50	5	49.41
13	EC SL		10	None	0.0043	0.0043	0.0043	0.0087		9	11	80	8	79.75
14	Stripline		10	None	0.005	0.005				9	11	40	4	40.56
15	Stripline		10	None	0.0033	0.0033				9	11	50	5	49.59
16	EC SL		12	None	0.0043	0.0043	0.0043	0.0087		11	13	80	8	79.88
17	EC SL		12	None	0.003	0.003	0.003	0.0109		11	13	100	10	100.52
18	Stripline		12	None	0.005	0.005				11	13	40	4	40.64
19	Stripline		12	None	0.0033	0.0033				11	13	50	5	49.68
20	EC MS		14	None	0.0062	0.0062	0.0062	0.0115		None	13	80	8	80.12
21	EC MS		14	None	0.0041	0.0041	0.0041	0.0085		None	13	90	9	89.70
22	EC MS		14	None	0.0042	0.0042	0.0042	0.0116		None	13	100	10	99.92
23	Surf MS		14	None	0.0082	0.0082				None	13	40	4	41.19
24	Surf MS		14	None	0.0052	0.0052				None	13	50	5	51.49

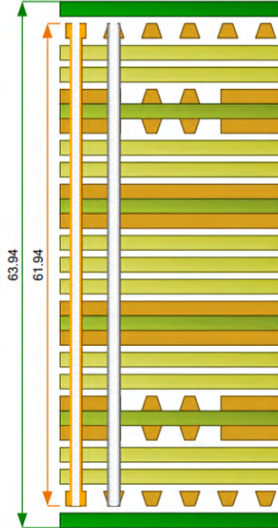
Trace widths measured at base of trace
All dimensions in inches (unless otherwise noted)

Table 7-5. AM65xx/DRA80xM PDN Targets and Decoupling Example

Supply Name ⁽¹⁾	Frequency of Interest (MHz)	Number of Decoupling Capacitors Per Supply ^{(2) (3) (4)}				
		0.01 μ F	0.1 μ F	1 μ F	2.2 μ F	10 μ F
VDD_CORE	≤ 25		23	12		4
VDD_MPU0 VDD_MPU1	≤ 25		11	4		4
VDD_MCU	≤ 25		3	1		2
VDD_DLL_MMC0 VDD_DLL_MMC1	≤ 25		2			
VDD_WKUP0 VDD_WKUP1	≤ 25		4			
CAP_VDDAR_WKUP	N/A			1		
CAP_VDDAR_MPU1_1	N/A			1		
CAP_VDDAR_MPU1_0	N/A			1		
CAP_VDDAR_MPU0_1	N/A			1		
CAP_VDDAR_MPU0_0	N/A			1		
CAP_VDDAR_MCU	N/A			1		
CAP_VDDAR_CORE4	N/A			1		
CAP_VDDAR_CORE3	N/A			1		
CAP_VDDAR_CORE2	N/A			1		
CAP_VDDAR_CORE1	N/A			1		
CAP_VDDAR_CORE0	N/A			1		
CAP_VDDSHV_SDIO CAP_VDDA_1P8_SDIO CAP_VDDA_1P8_IOLDO1 CAP_VDDA_1P8_IOLDO0	N/A	Use-case dependent. Refer to the device-specific data sheet.				
CAP_VDDA_1P8_IOLDO_WKUP	N/A				1	
CAP_VDD_WKUP	N/A			1		

- (1) Ganged rails must meet all requirements of each member rail.
- (2) The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
- (3) For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data sheet.
- (4) ESL must be as low as possible and not exceed 0.5 nH.

7.6 AM62xx

Layer	Stack up	Description	Processed Thickness	Isolation Distance (Summed)	Copper Coverage	ϵ_r	Impedance ID	Supplier Description	Tg
1		Taiyo PSR 4000 HFX DI-GREEN	1.000			3.500		PSR 4000	
		Copper Foil 12 microns	1.850		100.000		1, 2, 3, 4, 5, 6, 7	HI-Q Foil	
2		Iteq IT180A Prepreg 106 RC71-NEW	1.779	3.558		3.790		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A Prepreg 106 RC71-NEW	1.779	-		3.790		IPC-4101E / 99/ 101/ 126	170.000
3		Iteq IT180A 4 mil core 1/1	1.260		60.000	4.400		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A 4 mil core 1/1	4.000	4.000		4.400	8, 9, 10, 11, 12	IPC-4101E / 99/ 101/ 126	170.000
4		Iteq IT180A Prepreg 2113 RC58-NEW	3.298	6.597		4.130		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A Prepreg 2113 RC58-NEW	3.298	-		4.130		IPC-4101E / 99/ 101/ 126	170.000
5		Iteq IT180A 5 mil core 1/1	1.260		60.000	4.210		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A 5 mil core 1/1	5.000	5.000		4.210		IPC-4101E / 99/ 101/ 126	170.000
6		Iteq IT180A Prepreg 106 RC71-NEW	1.817	9.823		3.790		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A Prepreg 7628 RC43-NEW	6.190	-		4.450		IPC-4101E / 99/ 101/ 126	170.000
7		Iteq IT180A Prepreg 106 RC71-NEW	1.817	-		3.790		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A 5 mil core 1/1	1.260		51.000	4.210		IPC-4101E / 99/ 101/ 126	170.000
8		Iteq IT180A 5 mil core 1/1	5.000	5.000	61.000	4.210		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A Prepreg 2113 RC58-NEW	1.260	6.622		4.130		IPC-4101E / 99/ 101/ 126	170.000
9		Iteq IT180A Prepreg 2113 RC58-NEW	3.311	-		4.130		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A 4 mil core 1/1	1.260		47.000	4.400	13, 14, 15, 16, 17	IPC-4101E / 99/ 101/ 126	170.000
10		Iteq IT180A 4 mil core 1/1	4.000	4.000		4.400		IPC-4101E / 99/ 101/ 126	170.000
		Iteq IT180A Prepreg 106 RC71-NEW	1.260		60.000			IPC-4101E / 99/ 101/ 126	170.000
10		Iteq IT180A Prepreg 106 RC71-NEW	1.779	3.558		3.790		IPC-4101E / 99/ 101/ 126	170.000
		Copper Foil 12 microns	1.850		100.000		18, 19, 20, 21, 22, 23	HI-Q Foil	
		Taiyo PSR 4000 HFX DI-GREEN	1.000			3.500		PSR 4000	

Impedance ID	Impedance Signal Layer	Structure Name	Ref. Plane 1 in Layer	Ref. Plane 2 in Layer	Lower Trace Width (W1)	Trace Separation (S1)	Ground Strip Separation (D1)	Target Impedance	Calculated Impedance	Tol (+/- %)
1	1	Coated Microstrip 1B	2	0	9.050	0.000	0.000	40.000	40.030	10.000
2	1	Coated Microstrip 1B	2	0	5.800	0.000	0.000	50.000	50.540	10.000
3	1	Edge Coupled Coated Microstrip 1B	2	0	4.250	7.150	0.000	100.000	101.270	10.000
4	1	Edge Coupled Coated Microstrip 1B	2	0	4.200	4.400	0.000	90.000	91.560	10.000
5	1	Edge Coupled Coated Microstrip 1B	2	0	6.020	4.800	0.000	80.000	81.230	10.000
6	1	Edge Coupled Coated Microstrip 2B	3	0	4.100	5.750	0.000	120.000	118.350	10.000
7	1	Coated Coplanar Strips With Lower Ground 2B	3	0	16.000	0.000	14.000	50.000	49.090	10.000
8	3	Offset Stripline 1B1A	2	4	5.800	0.000	0.000	40.000	39.930	10.000
9	3	Edge Coupled Offset Stripline 1B1A	2	4	3.200	7.500	0.000	100.000	98.350	10.000
10	3	Edge Coupled Offset Stripline 1B1A	2	4	3.700	6.000	0.000	90.000	89.870	10.000
11	3	Edge Coupled Offset Stripline 1B1A	2	4	4.200	4.300	0.000	80.000	80.240	10.000
12	3	Offset Stripline 1B1A	2	4	3.600	0.000	0.000	50.000	49.920	10.000
13	8	Offset Stripline 1B1A	7	9	5.800	0.000	0.000	40.000	39.930	10.000
14	8	Edge Coupled Offset Stripline 1B1A	7	9	3.200	7.500	0.000	100.000	98.350	10.000
15	8	Edge Coupled Offset Stripline 1B1A	7	9	3.700	6.000	0.000	90.000	89.870	10.000
16	8	Edge Coupled Offset Stripline 1B1A	7	9	4.200	4.300	0.000	80.000	80.240	10.000
17	8	Offset Stripline 1B1A	7	9	3.600	0.000	0.000	50.000	49.920	10.000
18	10	Coated Microstrip 1B	9	0	9.050	0.000	0.000	40.000	40.030	10.000
19	10	Coated Microstrip 1B	9	0	5.800	0.000	0.000	50.000	50.540	10.000
20	10	Edge Coupled Coated Microstrip 1B	9	0	4.250	7.150	0.000	100.000	101.270	10.000
21	10	Edge Coupled Coated Microstrip 1B	9	0	4.200	4.400	0.000	90.000	91.560	10.000
22	10	Edge Coupled Coated Microstrip 1B	9	0	6.020	4.800	0.000	80.000	81.230	10.000
23	10	Edge Coupled Coated Microstrip 2B	8	0	4.100	5.750	0.000	120.000	118.350	10.000

Table 7-6. AM62xx PDN Targets and Decoupling Example

Supply Name (10) (11)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2)(3) (4) (5) (6) (9)			
	Max R _{eff} (mΩ) (7)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (6) (8)	Z _{TARGET} (mΩ)	0.1 μF	1 μF	4.7 μF	10 μF
VDD_CORE	23	≤1	1.5	23	17	1	1	1
		1-20		31				
		20-50		35				
VDDS_DDR	For more information, see AM62x DDR Board Design and Layout Guidelines .							

- For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- Loop ESL (excluding the intrinsic decap ESL) from the capacitor pads to the SoC BGA must be as low as possible and not exceed 1.5 nH.
- The Power Delivery Network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table of the *Specifications* chapter of the device-specific processor data manual.
- The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- This assumes that the external SMPS (power IC) feedback sense is located very close to processor power balls.
- High-frequency (30 MHz to 70 MHz) PCB decoupling capacitors.
- Maximum R_{eff} from VRM/SMPS/PMIC to Processor.
- Maximum Loop Inductance for decoupling capacitor when placed underneath processor BGA.
- The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
- Ganged rails must meet all requirements of each member rail.
- Rails not listed in this table are not simulated by TI due to low load transients. For more information, see the device-specific EVM layout for example implementation of these rails.

7.7 AM64xx

Layer	Stack up	Description	Type	Processed Thickness	Isolation Distance (Summed)	Copper Coverage	ε _r	Impedance ID
1	↑	Taiyo PSR 4000BN GREEN	SolderMask	2.000			3.900	
		Copper Foil 12 microns	Copper	1.850		100.000		1, 2, 3, 4, 5, 6
2	↑	Iteq IT180A Prepreg 106 RC71.5	Dielectric	1.848	3.696		3.790	
		Iteq IT180A Prepreg 106 RC71.5	Dielectric	1.848	-		3.790	
3	↑	Iteq IT180A 4 mil core 1/1	FR4	1.260	4.000	60.000	4.400	
		Iteq IT180A Prepreg 2113 RC58	Dielectric	3.322	6.644		4.130	
4	↑	Iteq IT180A Prepreg 2113 RC58	Dielectric	3.322	-	30.000	4.130	
		Iteq IT180A 5 mil core 1/1	FR4	1.260	5.000	60.000	4.210	7, 8, 9, 10, 11
5	↑	Iteq IT180A Prepreg 2113 RC58	Dielectric	3.586	9.047		4.130	
		Iteq IT180A Prepreg 106 RC71.5	Dielectric	1.875	-		3.790	
6	↑	Iteq IT180A Prepreg 2113 RC58	Dielectric	3.586	-		4.130	
		Iteq IT180A 5 mil core 1/1	FR4	1.260	5.000	54.000	4.210	
7	↑	Iteq IT180A Prepreg 2113 RC58	Dielectric	3.297	6.594	56.000	4.130	
		Iteq IT180A Prepreg 2113 RC58	Dielectric	3.297	-		4.130	
8	↑	Iteq IT180A 4 mil core 1/1	FR4	1.260	4.000	30.000	4.400	12, 13, 14, 15, 16
		Iteq IT180A Prepreg 106 RC71.5	Dielectric	1.848	3.696		3.790	
9	↑	Iteq IT180A Prepreg 106 RC71.5	Dielectric	1.848	-		3.790	
		Copper Foil 12 microns	Copper	1.850		100.000		17, 18, 19, 20, 21, 22
10	↓	Taiyo PSR 4000BN GREEN	SolderMask	2.000			3.900	

Copper Thickness = 13.779 | Dielectric Thickness = 47.677 | Solder Mask Thickness = 4.000 | Stack Up Thickness = 61.456 | Stack Up Thickness with Soldermask = 65.456

Impedance ID	Impedance Signal Layer	Structure Name	Ref. Plane 1 in Layer	Ref. Plane 2 in Layer	Lower Trace Width (W1)	Trace Separation (S1)	Ground Strip Separation (D1)	Calculated Impedance	Target Impedance	Tol (+/- %)
1	1	Coated Microstrip 1B	2	0	5.800	0.000	0.000	50.040	50.000	10.000
2	1	Coated Microstrip 1B	2	0	9.050	0.000	0.000	40.010	40.000	10.000
3	1	Edge Coupled Coated Microstrip 1B	2	0	4.200	4.400	0.000	89.900	90.000	10.000
4	1	Edge Coupled Coated Microstrip 1B	2	0	4.250	7.150	0.000	100.000	100.000	10.000
5	1	Edge Coupled Coated Microstrip 1B	2	0	6.020	4.800	0.000	80.140	80.000	10.000
6	1	Edge Coupled Coated Microstrip 1B	4	0	4.100	5.750	0.000	119.080	120.000	10.000
7	3	Edge Coupled Offset Stripline 1B1A	2	4	3.700	6.000	0.000	89.920	90.000	10.000
8	3	Offset Stripline 1B1A	2	4	3.600	0.000	0.000	49.980	50.000	10.000
9	3	Edge Coupled Offset Stripline 1B1A	2	4	4.200	4.300	0.000	80.280	80.000	10.000
10	3	Edge Coupled Offset Stripline 1B1A	2	4	3.200	7.500	0.000	98.410	100.000	10.000
11	3	Offset Stripline 1B1A	2	4	5.800	0.000	0.000	39.980	40.000	10.000
12	8	Edge Coupled Offset Stripline 1B1A	7	9	3.700	6.000	0.000	89.920	90.000	10.000
13	8	Offset Stripline 1B1A	7	9	3.600	0.000	0.000	49.980	50.000	10.000
14	8	Edge Coupled Offset Stripline 1B1A	7	9	4.200	4.300	0.000	80.280	80.000	10.000
15	8	Edge Coupled Offset Stripline 1B1A	7	9	3.200	7.500	0.000	98.410	100.000	10.000
16	8	Offset Stripline 1B1A	7	9	5.800	0.000	0.000	39.980	40.000	10.000
17	10	Edge Coupled Coated Microstrip 1B	9	0	4.200	4.400	0.000	89.900	90.000	10.000
18	10	Coated Microstrip 1B	9	0	5.800	0.000	0.000	50.040	50.000	10.000
19	10	Coated Microstrip 1B	9	0	9.050	0.000	0.000	40.010	40.000	10.000
20	10	Edge Coupled Coated Microstrip 1B	7	0	4.100	5.750	0.000	119.080	120.000	10.000
21	10	Edge Coupled Coated Microstrip 1B	9	0	4.250	7.150	0.000	100.000	100.000	10.000
22	10	Edge Coupled Coated Microstrip 1B	9	0	6.020	4.800	0.000	80.140	80.000	10.000



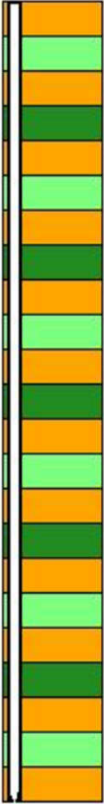
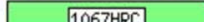

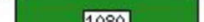

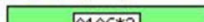






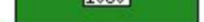
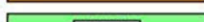
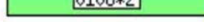

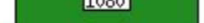

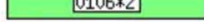

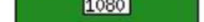

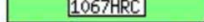

Table 7-7. AM64xx PDN Targets and Decoupling Example

Supply Name (10) (11)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2)(3) (4) (5) (6) (9)			
	Max R_{eff} (m Ω) (7)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (6) (8)	Z_{TARGET} (m Ω)	0.1 μ F	1 μ F	4.7 μ F	10 μ F
VDD_CORE	23	≤ 1	1.5	10	11	1	1	1
		1-20		34				
		20-50		35				
VDDS_DDR	For more information, see AM64x/AM243x DDR Board Design and Layout Guidelines .							

- For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- Loop ESL (excluding the intrinsic decap ESL) from the capacitor pads to the SoC BGA must be as low as possible and not exceed 1.5 nH.
- The Power Delivery Network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table of the *Specifications* chapter of the device-specific processor data manual.
- The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- This assumes that the external SMPS (power IC) feedback sense is located very close to processor power balls.
- High-frequency (30 MHz to 70 MHz) PCB decoupling capacitors.
- Maximum R_{eff} from VRM/SMPS/PMIC to Processor.
- Maximum Loop Inductance for decoupling capacitor when placed underneath processor BGA.
- The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
- Ganged rails must meet all requirements of each member rail.

11. Rails not listed in this table are not simulated by TI due to low load transients. For more information, see the device-specific EVM layout for example implementation of these rails.

7.8 AM62Ax

Lay #	Thick (in)	Picture	Type Dk Df	Description	Drill Picture
	0.0006/0.0013		4.5 0.019	Soldermask	
1	0.0020		F/S/LPHTe	0.5oz w/plating	
	0.0030		2.91 0.0058	fill	
2	0.0006		P/RTF5P	0.5oz	
	0.0030		3.14 0.0059	core	
3	0.0006		M/RTF5P	0.5oz	
	0.0044		2.93 0.0058	fill	
4	0.0006		P/RTF5P	0.5oz	
	0.0030		3.14 0.0059	core	
5	0.0006		M/RTF5P	0.5oz	
	0.0044		2.93 0.0058	fill	
6	0.0006		P/RTF5P	0.5oz	
	0.0030		3.14 0.0059	core	
7	0.0006		P/RTF5P	0.5oz	
	0.0045		2.92 0.0058	fill	
8	0.0006		P/RTF5P	0.5oz	
	0.0030		3.14 0.0059	core	
9	0.0006		P/RTF5P	0.5oz	
	0.0044		2.93 0.0058	fill	
10	0.0006		M/RTF5P	0.5oz	
	0.0030		3.14 0.0059	core	
11	0.0006		P/RTF5P	0.5oz	
	0.0030		2.91 0.0058	fill	
12	0.0020		F/S/LPHTe	0.5oz w/plating	
	0.0006/0.0013		4.5 0.019	Soldermask	
	0.0486	Total thickness (in) Over plated copper			
	0.0458	After lamination thickness (in)			
	0.0472	Over laminate thickness (in) (with soldermask)			
	0.0490	Customer Requirement (in)			
	+/-0.0049	Customer Tolerance (in)			

Notes and Recommendations:

1. Assume copper usage: 75% for GND layer, 60% for Mix layer.
2. In order to meet impedance, suggest to use 3mil core instead of 4mil core, L1-2&L12-11 Pre-preg is about 2.95mil(1x 1067 PP)
The final board thickness is 49mil exclude solder mask.
3. Suggest to relax tolerance to +/-5ohm for 400HM impedance.
4. Suggest to relax tolerance to +/-5ohm for 330HM impedance.
5. For "L10 33E single ended(6.3mils for 33E SE)"
--> We need adjust 6.3mil to 8mil to meet 33ohm+/-5ohm.
6. For "L10 66E differential(5.5mil/6.3mil for 66E differential)"
--> We need adjust 5.5mil/6.3mil to 7.4mil/4.4mil to meet 66ohm+/-10%.
7. For "L3 66E single ended(3 mils for 66E SE)"
--> 3mil line can not be adjusted to more thin.
the impedance value only meet 57ohm.
8. For "L3 133E differential(3mils/6.5mil for 133E differential)"
--> 3mil line can not be adjusted to more thin.
the impedance value only meet 110ohm.

Impedance Constraint Information (I)

Imp #	Impedance Type	Affect Lyr (1) (2)	Cust L/W	Line Width		Spacing		Ref Plane		Targ ohms	Tol ohms	Predicted ohms@2GHz
				(1)	(2)	(1)	(2)	Top	Bot			
1	EC MS	1	None	0.004	0.004	0.004	0.0043	None	2	90	9	89.38
2	EC MS	1	None	0.0035	0.0035	0.0035	0.0055	None	2	100	10	99.52
3	Surf MS	1	None	0.0055	0.0055			None	2	50	5	49.28
4	EC SL	3	None	0.005	0.005	0.005	0.004	4	2	80	8	81.17
5	EC SL	3	None	0.004	0.004	0.004	0.004	4	2	90	9	90.86
6	EC SL	3	None	0.0035	0.0035	0.0035	0.005	4	2	100	10	99.70
7	EC SL	3	None	0.003	0.003	0.003	0.0065	4	2	133	13.3	109.88
8	Stripline	3	None	0.0055	0.0058			4	2	40	5	40.04
9	Stripline	3	None	0.004	0.004			4	2	50	5	49.33
10	Stripline	3	None	0.003	0.003			4	2	66	6.6	56.75
11	EC SL	5	None	0.005	0.005	0.005	0.004	6	4	80	8	81.17
12	EC SL	5	None	0.004	0.004	0.004	0.004	6	4	90	9	90.86
13	EC SL	5	None	0.0035	0.0035	0.0035	0.005	6	4	100	10	99.70
14	Stripline	5	None	0.0055	0.0058			6	4	40	5	40.04
15	Stripline	5	None	0.004	0.004			6	4	50	5	49.33
16	EC SL	10	None	0.0055	0.0074	0.0074	0.0044	9	11	66	6.6	65.37
17	EC SL	10	None	0.005	0.005	0.005	0.004	9	11	80	8	81.17
18	EC SL	10	None	0.004	0.004	0.004	0.004	9	11	90	9	90.86
19	EC SL	10	None	0.0035	0.0035	0.0035	0.005	9	11	100	10	99.70
20	Stripline	10	None	0.0063	0.008			9	11	33	5	32.59
21	Stripline	10	None	0.0055	0.0058			9	11	40	5	40.04
22	Stripline	10	None	0.004	0.004			9	11	50	5	49.33
23	EC MS	12	None	0.004	0.004	0.004	0.0043	None	11	90	9	89.38
24	EC MS	12	None	0.0035	0.0035	0.0035	0.0055	None	11	100	10	99.52
25	Surf MS	12	None	0.0055	0.0055			None	11	50	5	49.28

Trace widths measured at base of trace
All dimensions in inches (unless otherwise noted)

Table 7-8. AM62Ax PDN Targets and Decoupling Example

Supply Name (10) (11)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2)(3) (4) (5) (6) (9)		
	Max R _{eff} (mΩ) (7)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (6) (8)	Z _{TARGET} (mΩ)	0.1 μF	1 μF	2.2 μF
VDD_CORE	6.8	≤1	1.5	6	4	9	1
		1-20		11			
		20-50		22			

Table 7-8. AM62Ax PDN Targets and Decoupling Example (continued)

Supply Name (10) (11)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2)(3) (4) (5) (6) (9)		
	Max R_{eff} (m Ω) (7)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (6) (8)	Z_{TARGET} (m Ω)	0.1 μ F	1 μ F	2.2 μ F
VDDS_DDR	For more information, see AM62Ax DDR Board Design and Layout Guidelines .						

- For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- Loop ESL (excluding the intrinsic decap ESL) from the capacitor pads to the SoC BGA must be as low as possible and not exceed 1.5 nH.
- The Power Delivery Network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table of the *Specifications* chapter of the device-specific processor data manual.
- The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- This assumes that the external SMPS (power IC) feedback sense is located very close to processor power balls.
- High-frequency (30 MHz to 70 MHz) PCB decoupling capacitors.
- Maximum R_{eff} from VRM/SMPS/PMIC to Processor.
- Maximum Loop Inductance for decoupling capacitor when placed underneath processor BGA.
- The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
- Ganged rails must meet all requirements of each member rail.
- Rails not listed in this table are not simulated by TI due to low load transients. For more information, see the device-specific EVM layout for example implementation of these rails.

7.9 AM62Px

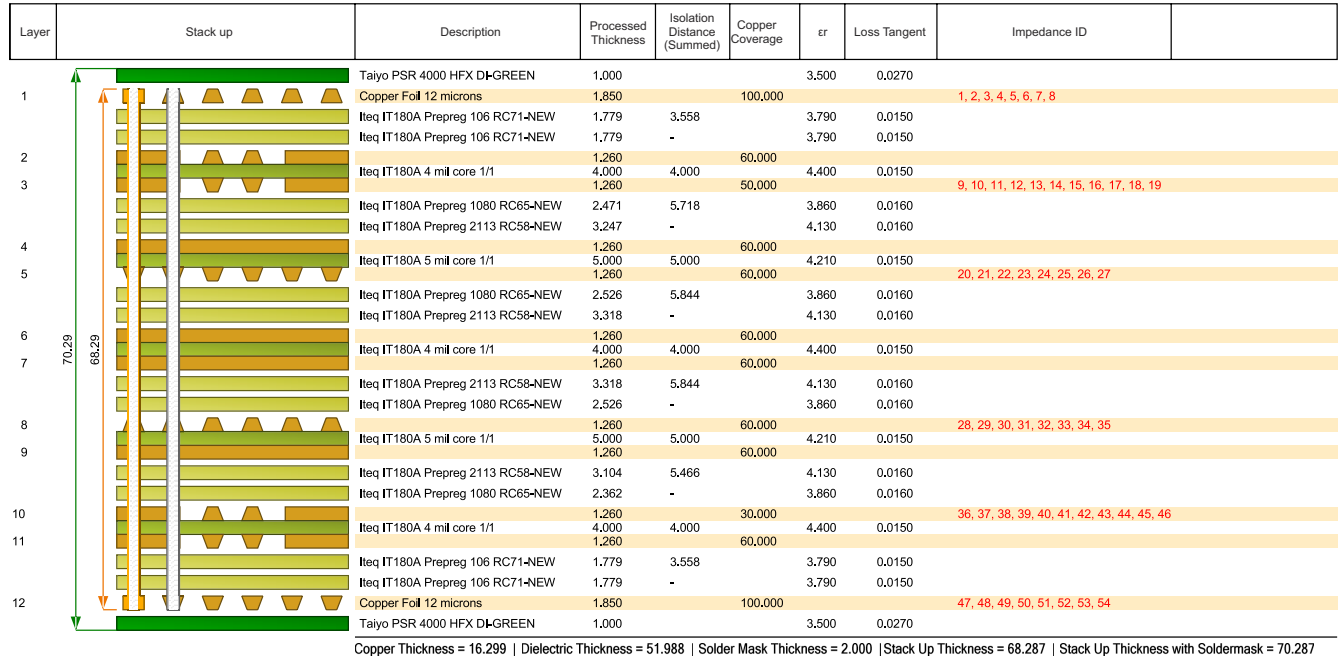


Figure 7-1. AM62P PCB Stack up

Impedance ID	Impedance Signal Layer	Structure Name	Ref. Plane 1 Layer	Ref. Plane 2 Layer	Lower Trace Width (W1)	Trace Separation (S1)	Ground Strip Separation (D1)	Calculated Impedance	Calculated Target Impedance	Calculated Target (%)
1	1	Coated Microstrip 1B	2	0	5,900	0,000	0,000	50,130	50,000	10,000
2	1	Edge Coupled Coated Microstrip 1B	2	0	4,100	6,200	0,000	99,940	100,000	10,000
3	1	Edge Coupled Coated Microstrip 2B	3	0	4,000	5,500	0,000	119,710	120,000	10,000
4	1	Coated Microstrip 1B	2	0	12,500	0,000	0,000	32,930	33,000	10,000
5	1	Coated Microstrip 1B	2	0	9,000	0,000	0,000	40,160	40,000	10,000
6	1	Edge Coupled Coated Microstrip 1B	2	0	9,100	4,500	0,000	66,170	66,000	10,000
7	1	Edge Coupled Coated Microstrip 2B	3	0	4,000	7,800	0,000	133,220	133,000	10,000
8	1	Edge Coupled Coated Microstrip 1B	2	0	6,000	4,500	0,000	80,200	80,000	10,000
9	3	Offset Stripline 1B2A	2	4	8,000	0,000	0,000	32,610	33,000	10,000
10	3	Offset Stripline 1B2A	2	4	3,500	0,000	0,000	49,870	50,000	10,000
11	3	Edge Coupled Offset Stripline 1B2A	2	4	4,300	4,500	0,000	80,400	80,000	10,000
12	3	Edge Coupled Offset Stripline 1B2A	2	4	3,200	8,000	0,000	99,070	100,000	10,000
13	3	Edge Coupled Offset Stripline 1B2A	2	4	6,600	4,500	0,000	65,900	66,000	10,000
14	3	Edge Coupled Offset Stripline 1B2A	2	4	3,700	4,500	0,000	85,450	85,000	10,000
15	3	Edge Coupled Offset Stripline 1B2A	2	4	3,600	5,800	0,000	90,450	90,000	10,000
16	3	Offset Stripline 1B2A	2	4	5,700	0,000	0,000	39,530	40,000	10,000
17	3	Offset Stripline 1B2A	2	4	3,000	0,000	0,000	53,120	66,000	10,000
18	3	Edge Coupled Offset Stripline 1B2A	2	4	3,000	16,000	0,000	105,990	133,000	10,000
19	3	Edge Coupled Offset Stripline 1B2A	2	4	6,600	4,500	0,000	65,900	66,000	10,000
20	5	Offset Stripline 1B2A	4	6	4,250	0,000	0,000	49,850	50,000	10,000
21	5	Offset Stripline 1B2A	4	6	9,150	0,000	0,000	33,350	33,000	10,000
22	5	Edge Coupled Offset Stripline 1B2A	4	6	3,500	6,300	0,000	98,360	100,000	10,000
23	5	Edge Coupled Offset Stripline 1B2A	4	6	8,300	6,166	0,000	66,030	66,000	10,000
24	5	Offset Stripline 1B2A	4	6	6,600	0,000	0,000	40,230	40,000	10,000
25	5	Edge Coupled Offset Stripline 1B2A	4	6	4,500	4,500	0,000	83,590	85,000	10,000
26	5	Edge Coupled Offset Stripline 1B2A	4	6	5,200	4,500	0,000	78,770	80,000	10,000
27	5	Edge Coupled Offset Stripline 1B2A	4	6	4,000	4,700	0,000	88,320	90,000	10,000
28	8	Edge Coupled Offset Stripline 1B2A	7	9	3,500	6,300	0,000	98,360	100,000	10,000
29	8	Offset Stripline 1B2A	7	9	4,250	0,000	0,000	49,850	50,000	10,000
30	8	Offset Stripline 1B2A	7	9	9,150	0,000	0,000	33,350	33,000	10,000
31	8	Edge Coupled Offset Stripline 1B2A	7	9	4,500	4,500	0,000	83,590	85,000	10,000
32	8	Edge Coupled Offset Stripline 1B2A	7	9	8,300	6,166	0,000	66,030	66,000	10,000
33	8	Offset Stripline 1B2A	7	9	6,600	0,000	0,000	40,230	40,000	10,000
34	8	Edge Coupled Offset Stripline 1B2A	7	9	4,000	4,700	0,000	88,320	90,000	10,000
35	8	Edge Coupled Offset Stripline 1B2A	7	9	5,200	4,500	0,000	78,770	80,000	10,000
36	10	Offset Stripline 1B2A	9	11	3,500	0,000	0,000	49,870	50,000	10,000
37	10	Edge Coupled Offset Stripline 1B2A	9	11	6,600	4,500	0,000	65,900	66,000	10,000
38	10	Edge Coupled Offset Stripline 1B2A	9	11	4,300	4,500	0,000	80,400	80,000	10,000
39	10	Offset Stripline 1B2A	9	11	8,000	0,000	0,000	32,610	33,000	10,000
40	10	Offset Stripline 1B2A	9	11	5,700	0,000	0,000	39,530	40,000	10,000
41	10	Edge Coupled Offset Stripline 1B2A	9	11	3,700	4,500	0,000	85,450	85,000	10,000
42	10	Edge Coupled Offset Stripline 1B2A	9	11	3,200	8,000	0,000	99,070	100,000	10,000
43	10	Edge Coupled Offset Stripline 1B2A	9	11	6,600	4,500	0,000	65,900	66,000	10,000
44	10	Offset Stripline 1B2A	9	11	3,000	0,000	0,000	53,120	66,000	10,000
45	10	Edge Coupled Offset Stripline 1B2A	9	11	3,600	5,800	0,000	90,450	90,000	10,000
46	10	Edge Coupled Offset Stripline 1B2A	9	11	3,000	16,000	0,000	105,990	133,000	10,000
47	12	Edge Coupled Coated Microstrip 1B	11	0	4,100	6,200	0,000	99,940	100,000	10,000
48	12	Coated Microstrip 1B	11	0	5,900	0,000	0,000	50,130	50,000	10,000
49	12	Coated Microstrip 1B	11	0	12,500	0,000	0,000	32,930	33,000	10,000
50	12	Coated Microstrip 1B	11	0	9,000	0,000	0,000	40,160	40,000	10,000
51	12	Edge Coupled Coated Microstrip 2B	10	0	4,000	5,500	0,000	119,710	120,000	10,000
52	12	Edge Coupled Coated Microstrip 2B	10	0	4,000	7,800	0,000	133,220	133,000	10,000
53	12	Edge Coupled Coated Microstrip 1B	11	0	6,000	4,500	0,000	80,200	80,000	10,000
54	12	Edge Coupled Coated Microstrip 1B	11	0	9,100	4,500	0,000	66,170	66,000	10,000

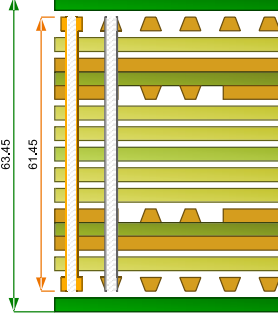
Figure 7-2. AM62P PCB Impedance

Table 7-9. AM62Px PDN Targets and Decoupling Example

Supply Name (10) (11)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2) (3) (4) (5) (6) (9)			
	Max R _{eff} (mΩ) (7)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (6) (8)	Z _{TARGET} (mΩ)	0.1 μF	1 μF	2.2 μF	10 μF
VDD_CORE	3	≤1	1.5	12	5	16	1	2
		1-20		20				
		20-50		40				
VDDS_DDR	For more information, see AM62Px DDR Board Design and Layout Guidelines .							

- For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
- Loop ESL (excluding the intrinsic decap ESL) from the capacitor pads to the SoC BGA must be as low as possible and not exceed 1.5 nH.
- The Power Delivery Network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table of the *Specifications* chapter of the device-specific processor data manual.
- The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
- This assumes that the external SMPS (power IC) feedback sense is located very close to processor power balls.
- High-frequency (30 MHz to 70 MHz) PCB decoupling capacitors.
- Maximum R_{eff} from VRM/SMPS/PMIC to Processor.
- Maximum Loop Inductance for decoupling capacitor when placed underneath processor BGA.
- The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
- Ganged rails must meet all requirements of each member rail.
- Rails not listed in this table are not simulated by TI due to low load transients. For more information, see the device-specific EVM layout for example implementation of these rails.

7.10 AM62Lx

Layer	Stack up	Description	Processed Thickness	Isolation Distance (Summed)	Copper Coverage	εr	Loss Tangent	Impedance ID
1		Taiyo PSR 4000 HFX DI-GREEN	1,000			3,500	0,0270	
		Copper Foil 12 microns	1,850		100,000			1, 2, 3, 4, 5, 6, 7
2		Iteq IT180A Prepreg 2113 RC58-NEW	3,387	3,387		4,130	0,0150	
		Iteq IT180A 3 mil core 1/1	1,260	3,000	60,000	3,930	0,0150	8, 9, 10, 11, 12
3		Iteq IT180A Prepreg 1080 RC65-NEW	2,580	39,940		3,860	0,0150	
		Iteq IT180A Prepreg 2113 RC58-NEW	3,390	-		4,130	0,0150	
4		Iteq IT180A 28 mil core H/H	28,000	-		4,530	0,0150	
		Iteq IT180A Prepreg 2113 RC58-NEW	3,390	-		4,130	0,0150	
5		Iteq IT180A Prepreg 1080 RC65-NEW	2,580	-		3,860	0,0150	
		Iteq IT180A 3 mil core 1/1	1,260	3,000	30,000	3,930	0,0150	
6		Iteq IT180A Prepreg 2113 RC58-NEW	3,387	3,387		4,130	0,0150	
		Copper Foil 12 microns	1,850		100,000			13, 14, 15, 16, 17, 18, 19
		Taiyo PSR 4000 HFX DI-GREEN	1,000			3,500	0,0270	

Copper Thickness = 8.740 | Dielectric Thickness = 52.713 | Solder Mask Thickness = 2.000 | Stack Up Thickness = 61.453 | Stack Up Thickness with Soldermask = 63.453

Figure 7-3. AM62L PCB Stack up

Impedance ID	Impedance Signal Layer	Structure Name	Ref. Plane 1 Layer	Ref. Plane 2 Layer	Lower Trace Width (W1)	Lower Trace Width (W1)	Trace Separation (S1)	Ground Strip Separation (D1)	Calculated Impedance	Calculated Target Impedance	Tol (+/- %)
1	1	Coated Microstrip 1B	2	0	8.000	7.000	0.000	0.000	40.160	40.000	10.000
2	1	Coated Microstrip 1B	2	0	5.200	4.200	0.000	0.000	50.070	50.000	10.000
3	1	Edge Coupled Coated Microstrip 1B	2	0	7.800	6.800	4.000	0.000	66.220	66.000	10.000
4	1	Coated Microstrip 1B	2	0	11.100	10.100	0.000	0.000	33.090	33.000	10.000
5	1	Edge Coupled Coated Microstrip 1B	2	0	3.500	2.500	5.800	0.000	99.890	100.000	10.000
6	1	Edge Coupled Coated Microstrip 1B	2	0	5.400	4.400	4.500	0.000	80.030	80.000	10.000
7	1	Edge Coupled Coated Microstrip 1B	2	0	4.000	3.000	4.500	0.000	89.900	90.000	10.000
8	3	Edge Coupled Offset Stripline 1B1A	2	4	4.000	3.000	4.500	0.000	80.260	80.000	10.000
9	3	Offset Stripline 1B1A	2	4	3.700	2.700	0.000	0.000	50.130	50.000	10.000
10	3	Offset Stripline 1B1A	2	4	6.000	5.000	0.000	0.000	40.140	40.000	10.000
11	3	Edge Coupled Offset Stripline 1B1A	2	4	3.200	2.200	10.000	0.000	99.160	100.000	10.000
12	3	Edge Coupled Offset Stripline 1B1A	2	4	3.500	2.500	6.250	0.000	89.860	90.000	10.000
13	6	Coated Microstrip 1B	5	0	8.000	7.000	0.000	0.000	40.160	40.000	10.000
14	6	Coated Microstrip 1B	5	0	11.100	10.100	0.000	0.000	33.090	33.000	10.000
15	6	Coated Microstrip 1B	5	0	5.200	4.200	0.000	0.000	50.070	50.000	10.000
16	6	Edge Coupled Coated Microstrip 1B	5	0	7.800	6.800	4.000	0.000	66.220	66.000	10.000
17	6	Edge Coupled Coated Microstrip 1B	5	0	4.000	3.000	4.500	0.000	89.900	90.000	10.000
18	6	Edge Coupled Coated Microstrip 1B	5	0	3.500	2.500	5.800	0.000	99.890	100.000	10.000
19	6	Edge Coupled Coated Microstrip 1B	5	0	5.400	4.400	4.500	0.000	80.030	80.000	10.000

Figure 7-4. AM62L PCB Impedance

Table 7-10. AM62Lx PDN Targets and Decoupling Example

Supply Name (9) (10)	Static PDN Target	Dynamic PDN Targets			Number of Decoupling Capacitors Per Supply (1) (2) (3) (4) (5) (8)				
	Max R_{eff} (m Ω) (6)	Frequency of Interest (MHz)	Dec. Cap. Max LL (nH) (5) (7)	Z_{TARGET} (m Ω)	0.1 μ F	1 μ F	2.2 μ F	4.7 μ F	10 μ F
VDD_CORE	5	≤ 1	1.5	19	4	12	1	1	2
		1-20		32					
		20-50		65					
VDDS_DDR	For more information, see AM62Lx DDR Board Design and Layout Guidelines .								

1. For more information on peak-to-peak noise values, see the *Recommended Operating Conditions* table in the device-specific data manual.
2. Loop ESL (excluding the intrinsic decap ESL) from the capacitor pads to the SoC BGA must be as low as possible and not exceed 1.5 nH.
3. The Power Delivery Network (PDN) impedance characteristics are defined versus the device activity (that runs at different frequency) based on the *Recommended Operating Conditions* table of the *Specifications* chapter of the device-specific processor data manual.
4. The static drop requirement drives the maximum acceptable PCB resistance between the PMIC or the external SMPS and the processor power balls.
5. High-frequency (30 MHz to 70 MHz) PCB decoupling capacitors.
6. Maximum R_{eff} from VRM/SMPS/PMIC to Processor.
7. Maximum Loop Inductance for decoupling capacitor when placed underneath processor BGA.
8. The decoupling capacitor counts and values presented here are provided as a baseline recommendation only and are based on a specific PCB design. TI recommends that all PCB designs be simulated prior to fabrication to ensure that all processor PDN requirements are met.
9. Ganged rails must meet all requirements of each member rail.
10. Rails not listed in this table are not simulated by TI due to low load transients. For more information, see the device-specific EVM layout for example implementation of these rails.

8 Revision History

Changes from Revision G (August 2023) to Revision H (October 2025)		Page
• Added Section 7.9 and AM62L PDN targets		35
• Added Section 7.10 and AM62L PDN targets		38

Changes from November 1, 2022 to February 29, 2024 (from Revision F (November 2022) to Revision G (February 2024))		Page
• Updated the numbering format for tables, figures and cross-references throughout the document.....		2
• Updated AM62xx section.....		28
• Updated Section 7.7		29
• Added Section 7.8		32

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you fully indemnify TI and its representatives against any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to [TI's Terms of Sale](#), [TI's General Quality Guidelines](#), or other applicable terms available either on [ti.com](#) or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products. Unless TI explicitly designates a product as custom or customer-specified, TI products are standard, catalog, general purpose devices.

TI objects to and rejects any additional or different terms you may propose.

Copyright © 2025, Texas Instruments Incorporated

Last updated 10/2025