

Technical Note

Land Pad Design for NOR Flash Memories

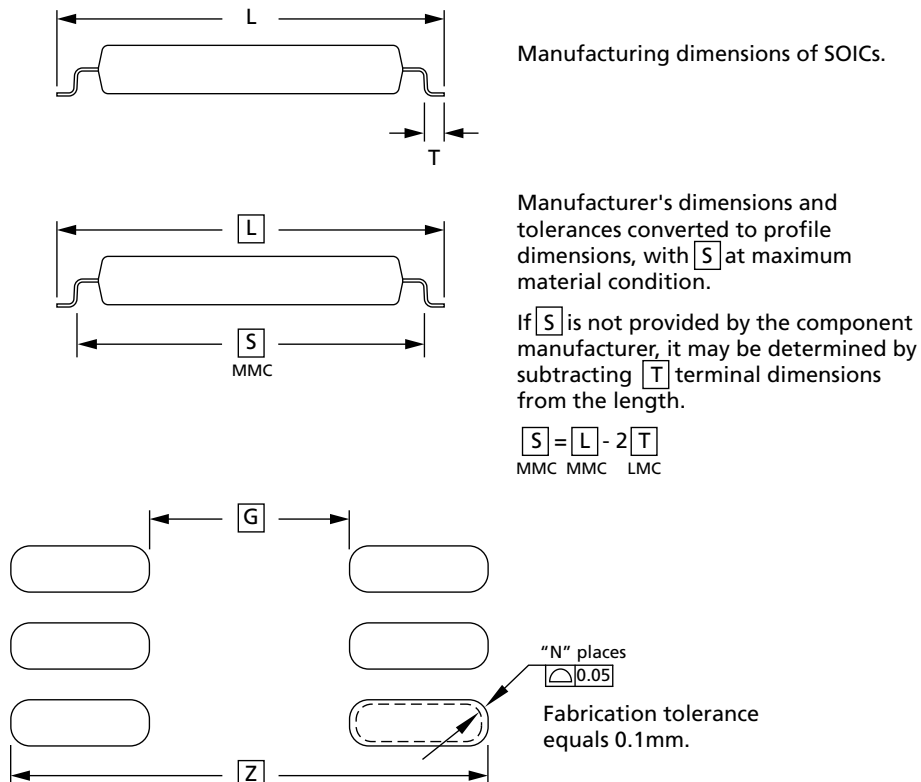
Introduction

This document provides high-level information on Micron's NOR Flash memory packages and their suggested PCB land patterns. For additional information and assistance for any manufacturability issues, contact your Micron representative or log on to www.micron.com.

Package Outlines and Suggested Land Pattern Designs

The following sections contain detailed mechanical descriptions of Micron's NOR Flash packages and the related suggested PCB land pattern designs. The suggested land pattern (mm) designs were developed using Mentor's *PCB Matrix Calculator software*, which is based on the IPC-7351B standard. For additional information refer to the NOR Flash section of Micron's website.

Figure 1: Key Land Pattern Sizes Definitions (from IPC-7351B)



There are many options available to determine the tolerances for each component. The IPC-7351B formulas for calculating the Z_{MAX} distance are shown below (based on the RMS method).

$$Z_{MAX} = L_{MIN} + 2J_T + \sqrt{C_L^2 + F^2 + P^2}$$

$$G_{MIN} = S_{MAX} - 2J_H - \sqrt{C_S^2 + F^2 + P^2}$$

$$X_{MAX} = W_{MIN} + 2J_S + \sqrt{C_W^2 + F^2 + P^2}$$

Where:

Z_{MAX} = overall length of land pattern

G_{MIN} = minimum distance between land of the pattern

X_{MAX} = maximum width of the pattern

S_{MAX} = maximum distance among component termination

W_{MIN} = minimum width of the lead

L_{MIN} = minimum distance of the component length

J_T = toe joint dimension

C_L = tolerance for component length

F = fabrication tolerance

P = positioning tolerance

J_H = solder fillet or land protrusion at heel

C_S = tolerance on distance between component terminations

C_W = tolerance on the lead width

Table 1: Tolerance Definitions (from IPC-7351b)

Tolerance Element	Description
Compound tolerance	The difference of maximum material condition (MMC) and least material condition (LMC) of each component dimension length, width, and distance between terminations or leads. This number is the "C" tolerance in the equations.
Printed board tolerance	The difference of MMC and LMC of each land pattern dimension length. This number is the "F" tolerance in the equations.
Positional accuracy	The diameter of true position (DTP). This is the variation of the part centroid related to the land pattern theoretical center (includes feature location tolerance from Table 2).

All tolerances for lands are intended to provide a projected land pattern with individual lands at maximum size, while unilateral tolerance will result in a smaller area for solder joint formation. Land patterns are within these outer and inner extremities.

The LPW tool uses the distance between pin centers ("C" as shown in the land pattern figures) instead of the distance between the pin ends (Z). This is the way most of tools used in the industry manage that dimension.

Some rounding off of values occurs, which is controlled by the calculator settings tab. These settings should be reviewed and set to meet the user's fabrication and assembly best practices for their industry and their fabrication/assembly technology. The tolerances shown below have been used in all of the land pattern calculations that follow in this document.

Table 2: Applied Tolerances

Tolerance Element	Tolerance (mm)
Land-to-land MIN	0.20

Table 2: Applied Tolerances (Continued)

Tolerance Element	Tolerance (mm)
Silkscreen to Land MIN	0.25
Solder mask web MIN	0.075
Fabrication	0.05
Placement	0.05
Land place round-off (C)	0.10
Land size round-off	0.05

Definitions of the areas and dimensions provided in this document:

- **Land pattern:** The combinations of lands used for the mounting of a particular component.
- **Courtyard:** The smallest rectangular area that provides a minimum electrical and mechanical clearance around the combined component body and land pattern boundaries.
- **Silkscreen:** The printed area of the position where the component body will be placed, in relation to the orientation indicator.
- **Assembly area:** Maximum area allowed for component body mounting on the PCB.

SO Packages

Figure 2: Calculated SO Parameters

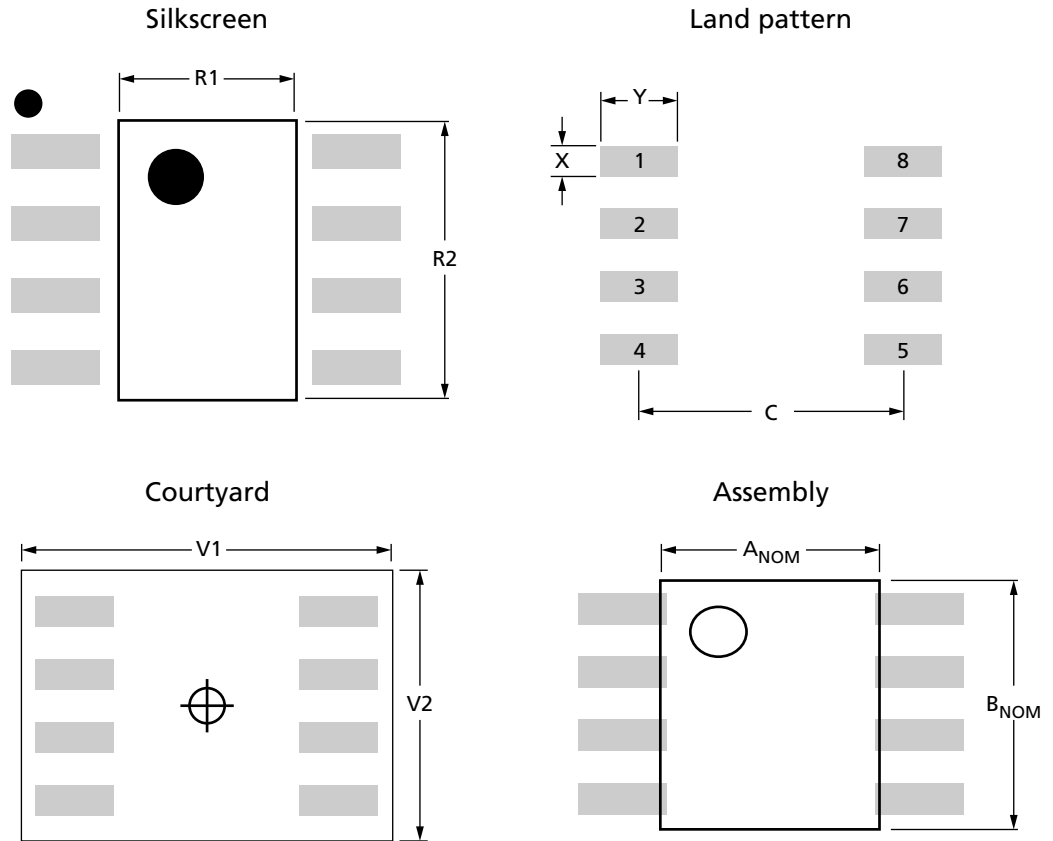
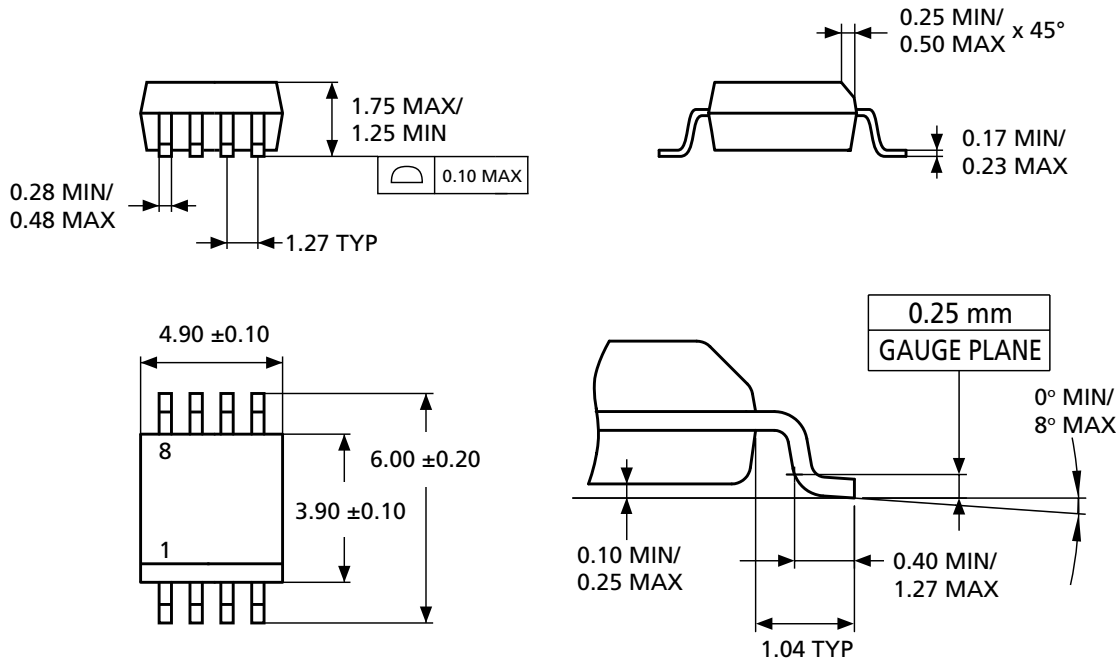


Table 3: SO Packages and Parameter Values

Package Type	Pins	Width (mm)	Length (mm)	Max Thickness (mm)	Pitch (mm)
SOP2-8 150 mils body width (SO8N)	8	4.9	6	1.75	1.27
SOP2-8 208 mils body width (SO8W)	8	5.28	7.9	2.5	1.27
SOP2-16 300 mils body width (SO16W)	16	10.3	10.3	2.65	1.27

SO8N

Figure 3: SO8N Package Outline



Note: 1. All dimensions are in millimeters.

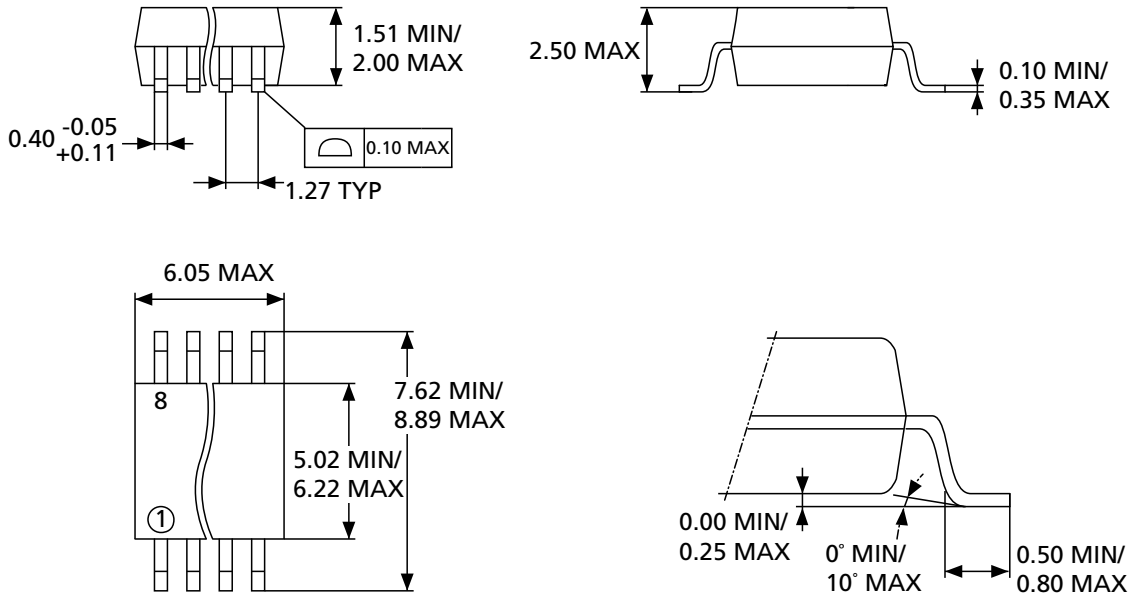
Table 4: SO8N Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)			Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	Y	X	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	5.2	1.35	0.55	4.9	3.9	3.9	4.9	6.8	5.2

Note: 1. See Figure 2 (page 4) for suggested land pattern.

SO8W

Figure 4: SO8W Package Outline



Note: 1. All dimensions are in millimeters.

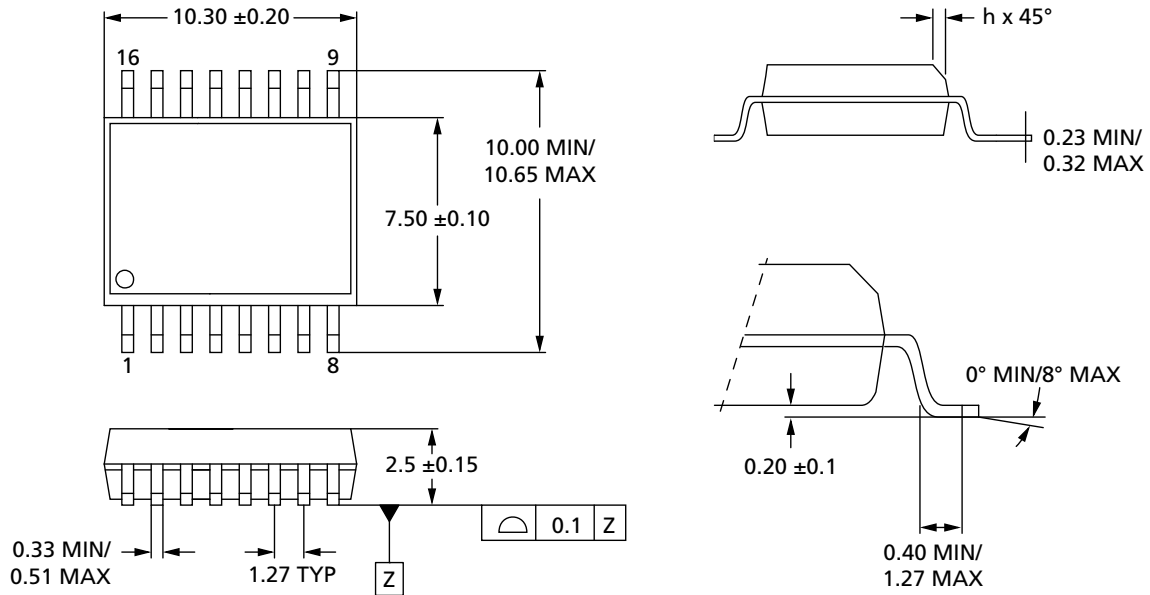
Table 5: SO8W Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)			Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	Y	X	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	7.5	1.7	0.6	5.15	6	5.62	5.95	9.4	6.3

Note: 1. See Figure 2 (page 4) for suggested land pattern.

SO16W

Figure 5: SO16W Package Outline



Note: 1. All dimensions are in millimeters.

Table 6: SO16W Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)			Silkscreen (mm)		Assembly (mm)		Courtyard (mm)
	C	Y	X	R1	R2	ANOM	BNOM	V1
Typical dimension	9.2	1.75	0.6	6.8	10.3	7.5	10.3	10.7

Note: 1. See Figure 2 (page 4) for suggested land pattern.

DFN Packages

Figure 6: Calculated DFN Parameters

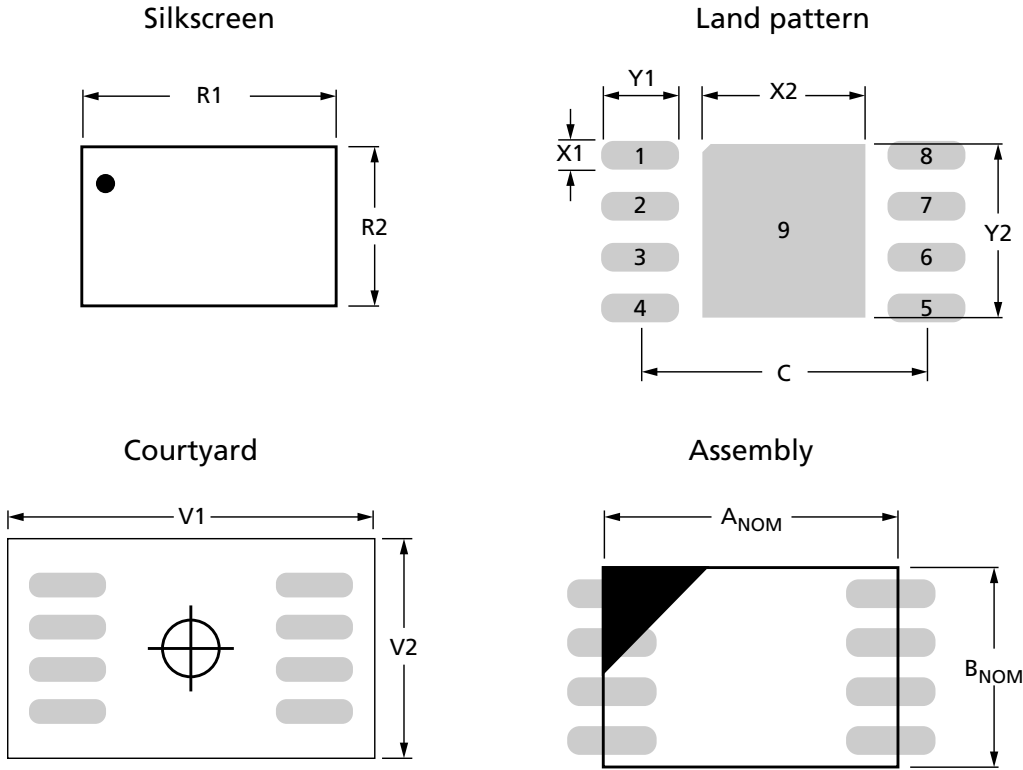
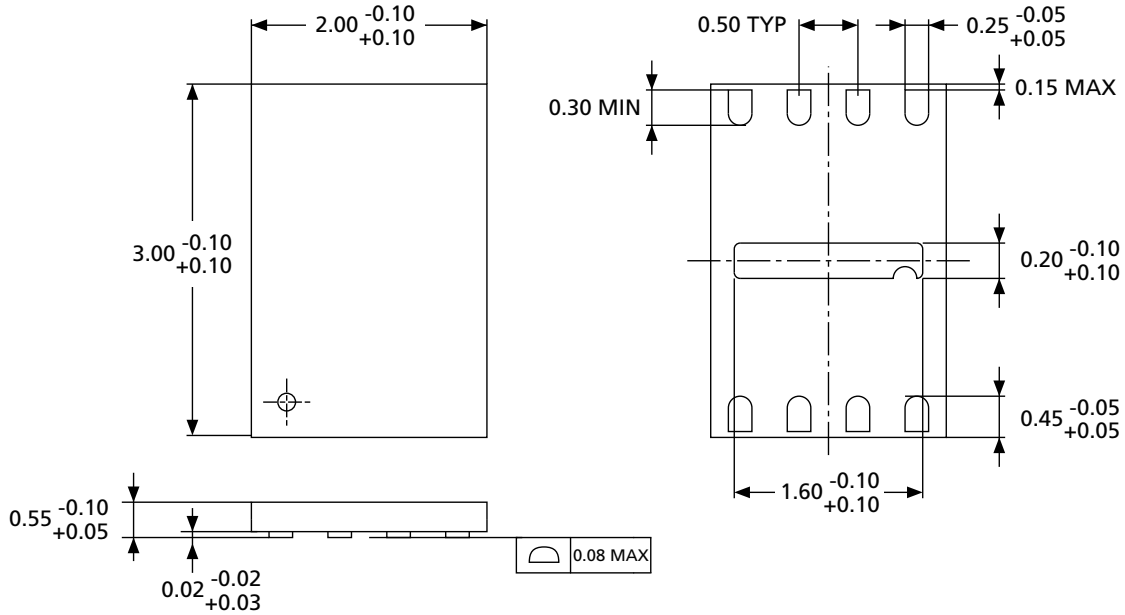


Table 7: DFN Packages and Parameter Values

Package Type	Pins	Width (mm)	Length (mm)	Max Thickness (mm)	Pitch (mm)
U-PDFN-8 2 x 3mm (MLPA8)	8	2	3	0.6	0.5
U-PDFN-8 4 x 3mm (MLPA8)	8	4	3	0.6	0.8
V-PDFN-8 6 x 5mm (MLP8)	8	5	6	1	1.27
V-PDFN-8 6 x 5mm Sawn (MLP8)	8	5	6	0.9	1.27
VPDFN-8 8 x 6mm (MLP8)	8	6	8	1	1.27

DFN-8 2mm x 3mm

Figure 7: DFN-8 2mm x 3mm Package Outline



Note: 1. All dimensions are in millimeters.

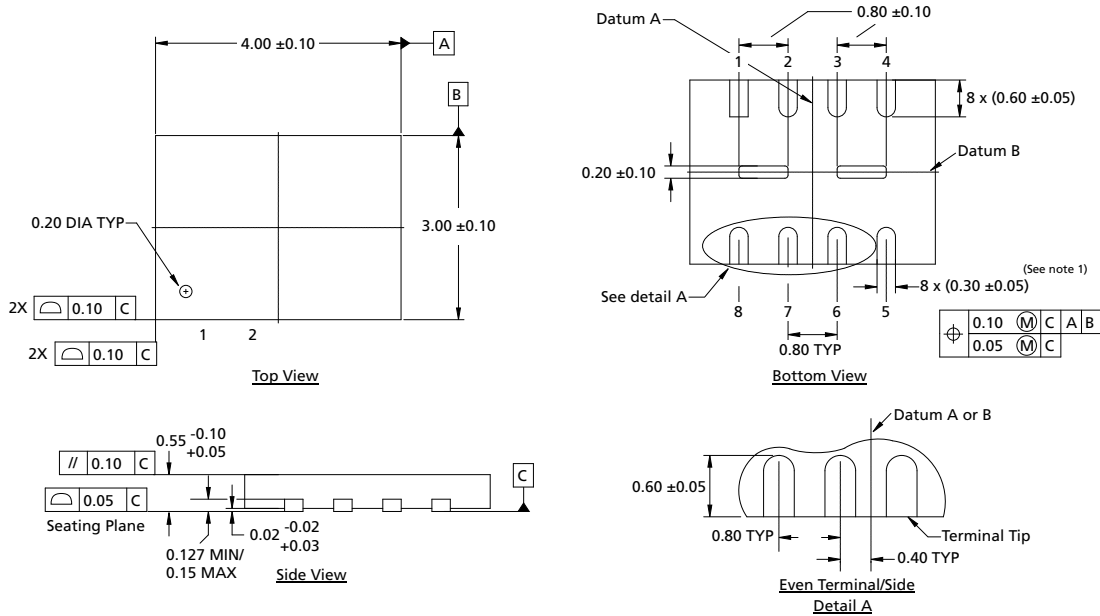
Table 8: DFN-8 2mm x 3mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)					Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X1	Y1	X2	Y2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	2.8	0.3	0.75	0.3	1.7	1.3	2	3	2	3.8	2.3

Note: 1. See Figure 6 (page 8) for suggested land pattern.

DFN-8 4mm x 3mm

Figure 8: DFN-8 4mm x 3mm Package Outline



Note: 1. All dimensions are in millimeters.

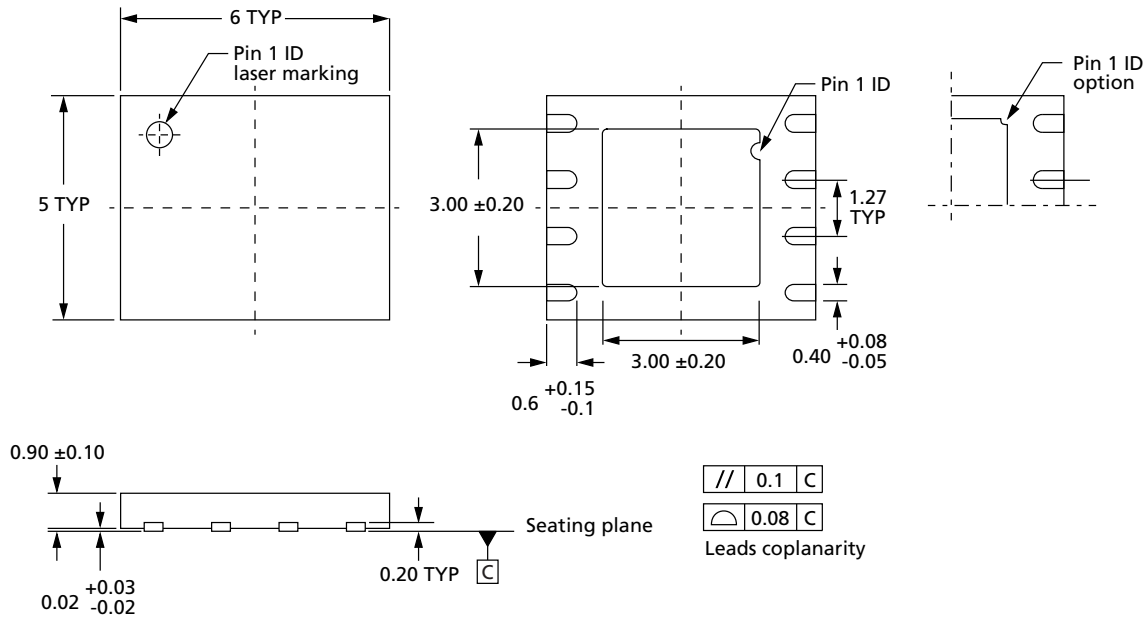
Table 9: DFN-8 4mm x 3mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm) ²					Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X1	Y1	X 2	Y2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	2.6	0.35	0.95	0.3	2.4	3	4	3	4	3.8	4.3

- Notes:
1. See Figure 6 (page 8) for suggested land pattern.
 2. The software used to generate the land pattern dimensions, does not have the capability to split the thermal tab in two as shown in the drawing above. Please consider doing one of the following:
 - 1) Screen print such that the part is not lifted excessively by the solder, or
 - 2) Break the thermal tab in two with .05mm clearance around the pad, similar to dimension X2.

DFN-8 6mm x 5mm

Figure 9: DFN-8 6mm x 5mm Package Outline



Note: 1. All dimensions are in millimeters.

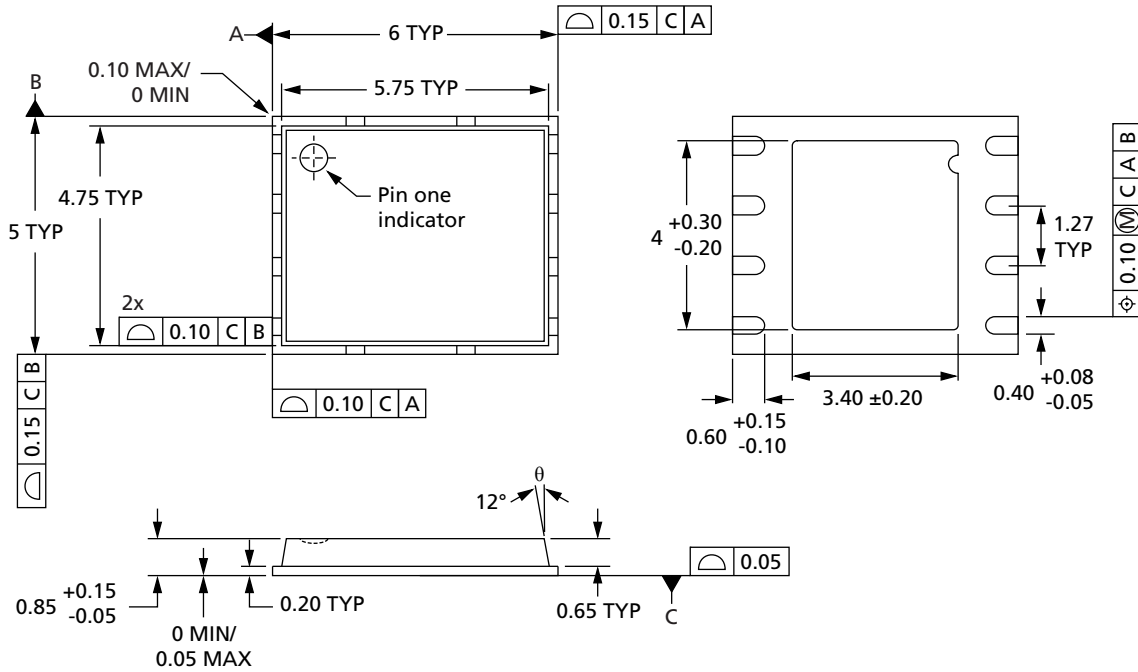
Table 10: DFN-8 6mm x 5mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)					Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X1	Y1	X2	Y2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	5.5	0.45	1.05	3.2	3.2	3.7	5	6	5	6.8	5.4

Note: 1. See Figure 6 (page 8) for suggested land pattern.

DFN 5mm x 6mm (Not Sawn)

Figure 10: DFN 5mm x 6mm (Not Sawn) Package Outline



Note: 1. All dimensions are in millimeters.

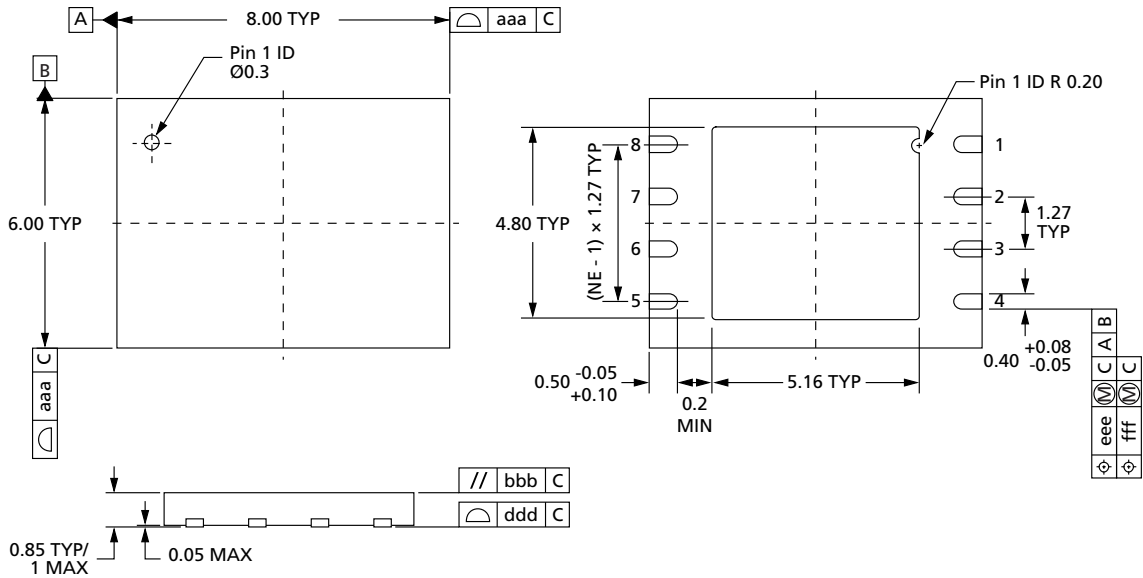
Table 11: DFN 5mm x 6mm (Not Sawn) Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)					Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X1	Y1	X2	Y2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	5.5	0.45	1.05	3.6	4.3	3.8	5	6	5	6.8	5.4

Note: 1. See Figure 6 (page 8) for suggested land pattern.

DFN-8 8mm x 6mm

Figure 11: DFN-8 8mm x 6mm Package Outline



Note: 1. All dimensions are in millimeters.

Table 12: DFN-8 8mm x 6mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)					Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X1	Y1	X2	Y2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	7.7	0.45	0.9	5.15	4.8	8	6	8	6	8.8	6.4

Note: 1. See Figure 6 (page 8) for suggested land pattern.

TSOP Packages

Figure 12: Calculated TSOP Parameters

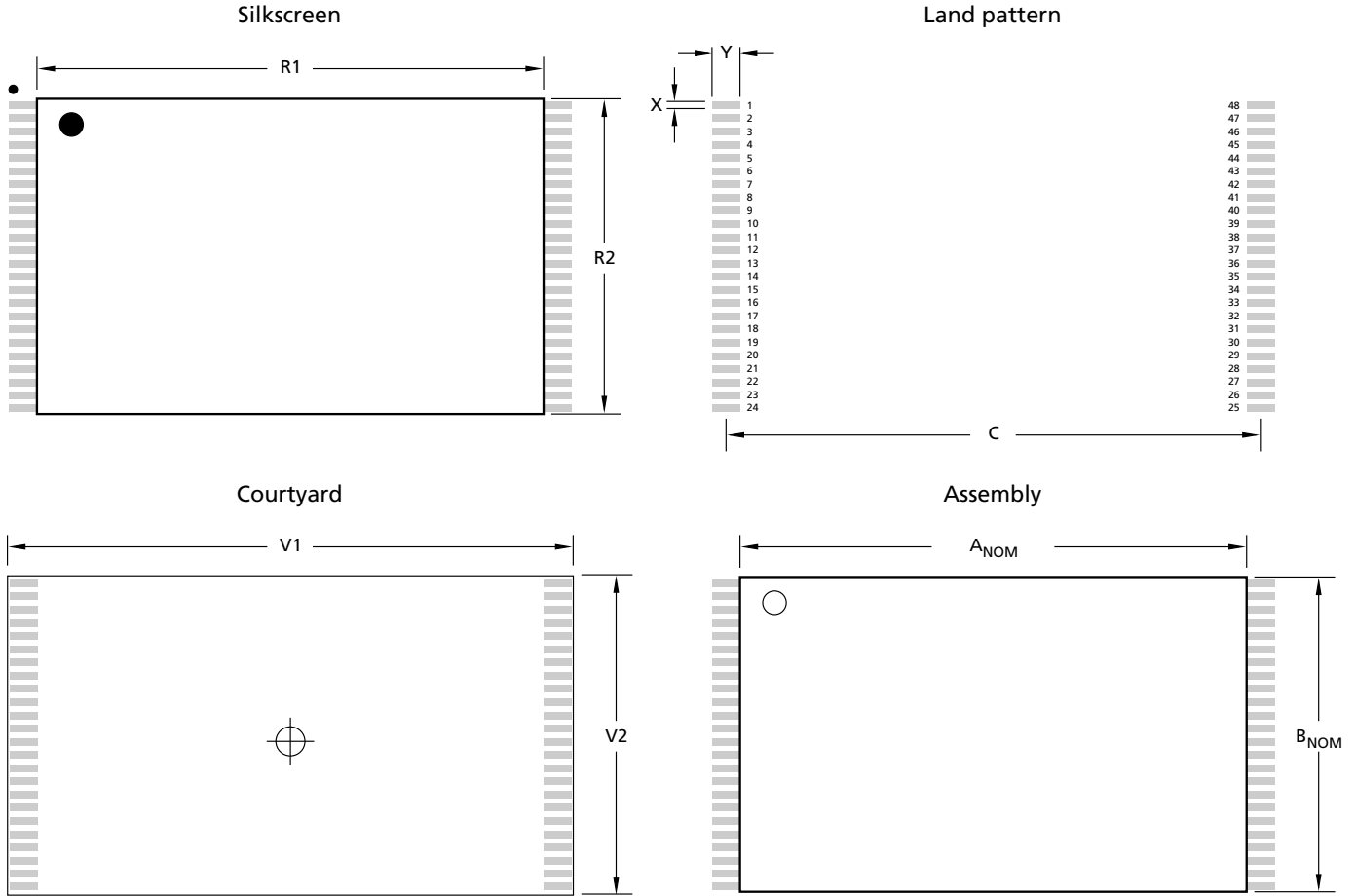
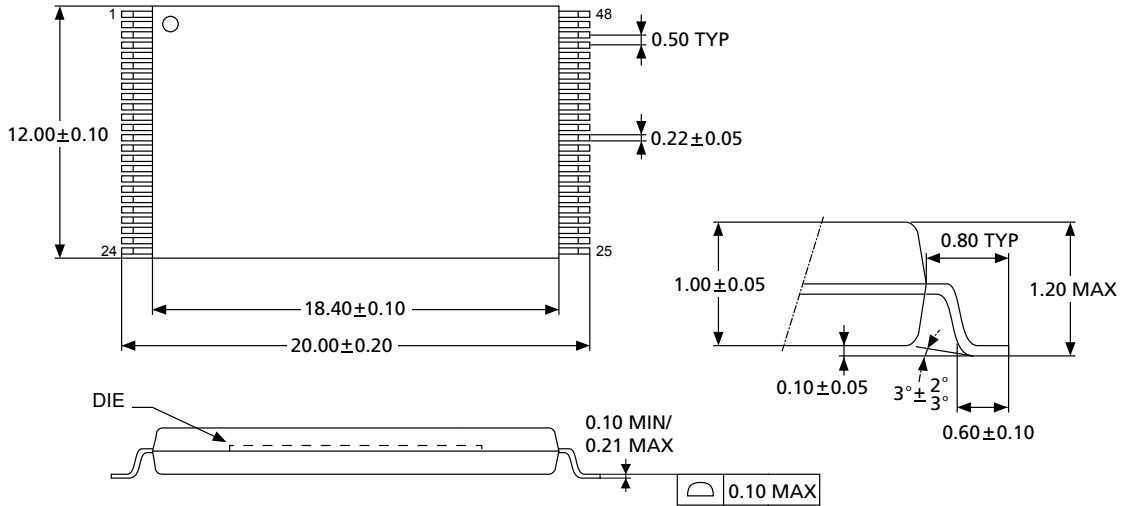


Table 13: TSOP Packages and Parameter Values

Package Type	Pins	Width (mm)	Length (mm)	Max Thickness (mm)	Pitch (mm)
TSOP48: 12mm x 20mm	48	12	20	1.2	0.5
TSOP56: 14mm x 20mm	56	14	20	1.2	0.5

TSOP-I 48 12mm x 20mm

Figure 13: TSOP-I 48 12mm x 20mm Package Outline



Note: 1. All dimensions are in millimeters.

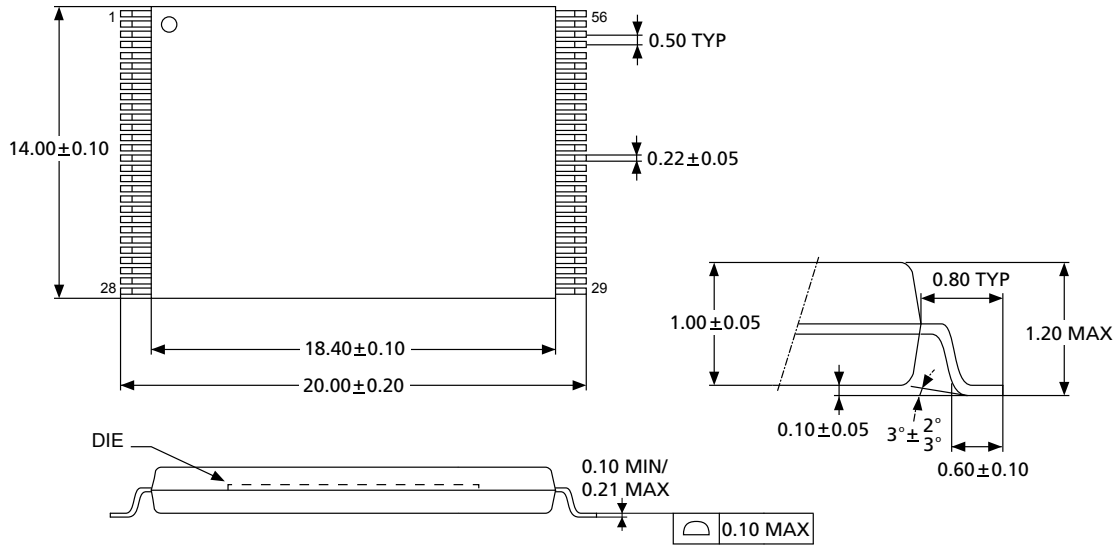
Table 14: TSOP-I 48 12mm x 20mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)			Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X	Y	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	19.5	0.25	1.1	17.75	12	18.4	12	20.7	12.3

Note: 1. See Figure 12 (page 14) for suggested land pattern.

TSOP-I 56 14mm x 20mm

Figure 14: TSOP-I 56 14mm x 20mm Package Outline



Note: 1. All dimensions are in millimeters.

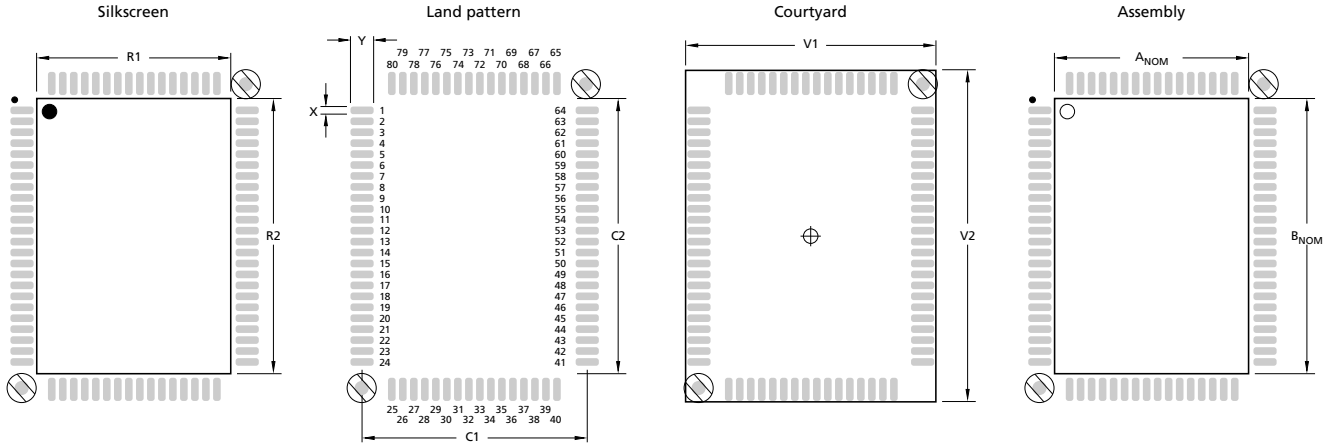
Table 15: TSOP-I 56 14mm x 20mm Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)			Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X	Y	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	19.4	0.25	1.1	17.65	14	18.4	14	20.7	14.3

Note: 1. See Figure 12 (page 14) for suggested land pattern.

QFP Packages

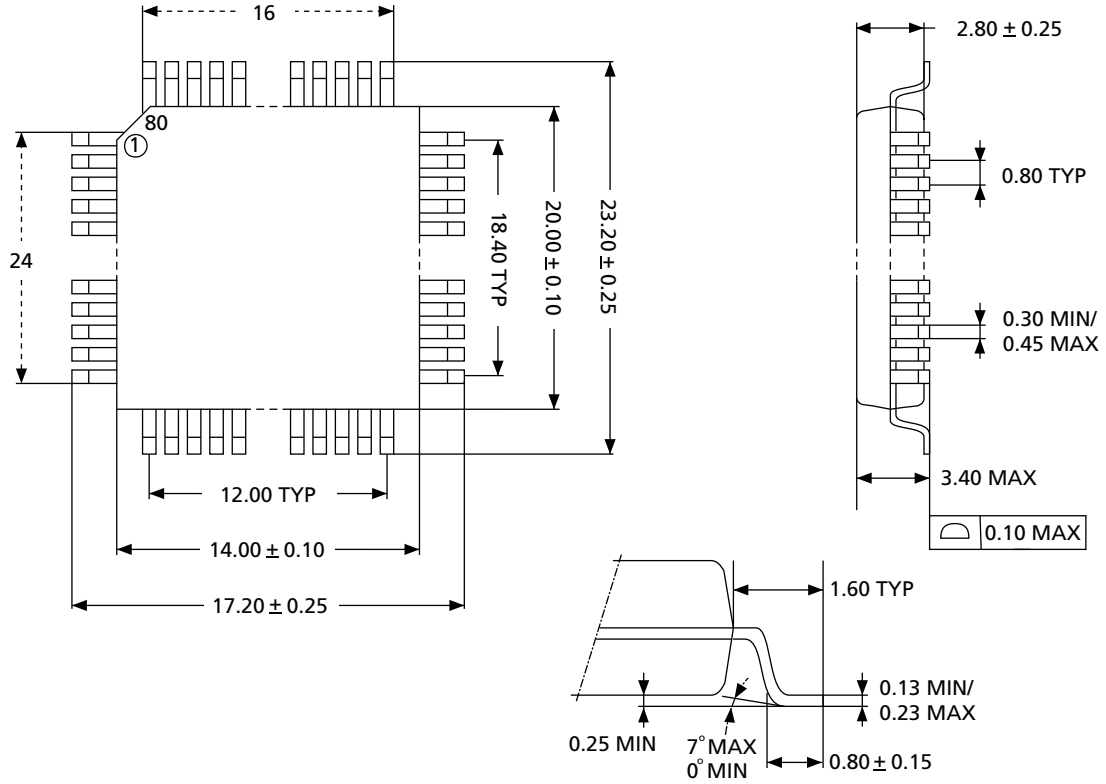
Figure 15: Calculated QFP Parameters



QFP 48 12mm x 20mm

QFP assumptions:

- Toe (outside) goal = 0.15
- Toe MIN = 0.17
- Toe MAX goal = 0.43
- Heal (in side) goal = 0.25
- Heal MIN = 0.23
- Heal MAX = 0.57
- Side goal = 0.25
- Side MIN = 0.23
- Side MAX = 0.57

Figure 16: PQFP80 Package Outline


Note: 1. All dimensions are in millimeters.

Table 16: PQFP80 Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)				Silkscreen (mm)		Assembly (mm)		Courtyard (mm)	
	C	X	Y	C2	R1	R2	ANOM	BNOM	V1	V2
Typical dimension	16.3	0.55	1.5	14.1	14.1	20.1	14	20	18	27

Note: 1. See Figure 15 (page 17) for suggested land pattern.

BGA Packages

BGA land pattern calculations are mainly based on ball size, but other features are also considered. Variations that affect BGA land patterns include pitch, ball diameter, the positional accuracy of the balls vs. the true position of the component on the PCB, and the manufacturing allowance that can be held for the land on the substrate that mounts the ball. Basically, the land pattern of the component (where the ball is attached) and the land pattern of the substrate mounting structure (PCB) should be as close in size as possible. Reducing the PCB land size by some percentage of the ball diameter is acceptable as well. The IPC-7351 table below show the parameters that may cause variations of land pattern needed to describe variations in the system. This data is usually described at the MMC for NSMD lands.

Table 17: BGA Tolerances (Reproduced from the IPC-7351B Standard)

Tolerance values are in millimeters

Land Size		Location Allowance	Ball Variation	PCB Fabrication Allowance	Nominal	Ball Size		% Reduction from Nominal	Variation Allowance
MMC	LMC					MMC	LMC		
0.60	0.50	0.10	0.25	0.10	0.75	0.90	0.65	25%	0.29
0.50	0.40	0.10	0.20	0.10	0.60	0.70	0.50	25%	0.24
0.45	0.35	0.10	0.15	0.10	0.55	0.65	0.45	25%	0.20
0.45	0.35	0.10	0.10	0.10	0.50	0.55	0.45	20%	0.17
0.40	0.30	0.10	0.10	0.10	0.45	0.50	0.40	20%	0.17
0.35	0.25	0.10	0.10	0.10	0.40	0.45	0.35	20%	0.17
0.25	0.20	0.05	0.10	0.05	0.30	0.35	0.25	20%	0.15
0.20	0.17	0.05	0.06	0.03	0.25	0.28	0.22	20%	0.08
0.20	0.14	0.05	0.04	0.03	0.20	0.22	0.18	15%	0.07
0.20	0.14	0.05	0.04	0.02	0.17	0.17	0.13	15%	0.07
0.18	0.12	0.05	0.04	0.02	0.15	0.15	0.10	15%	0.07

BGA Calculated Land Pattern Parameters

Figure 17: BGA Calculated Land Pattern Parameters

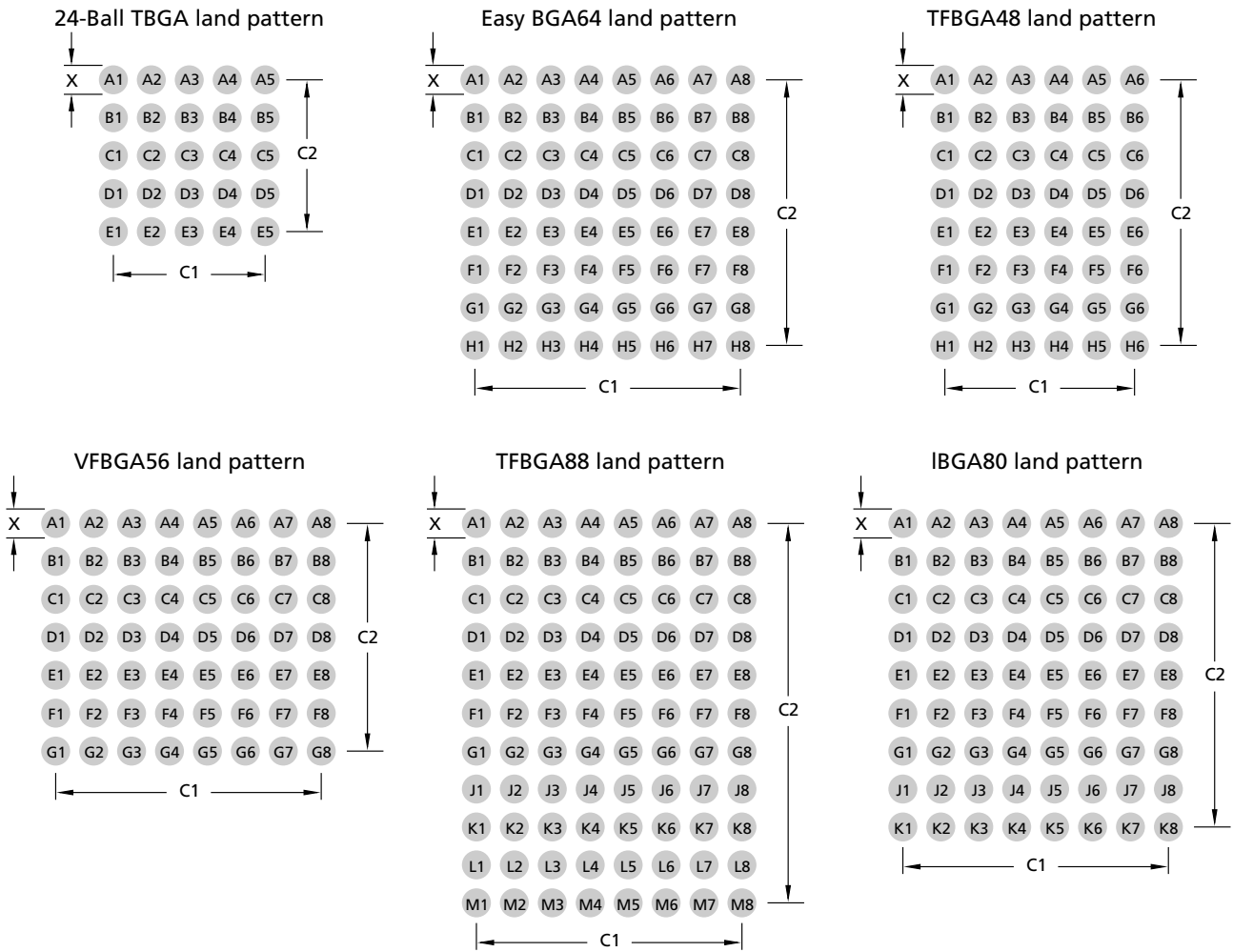
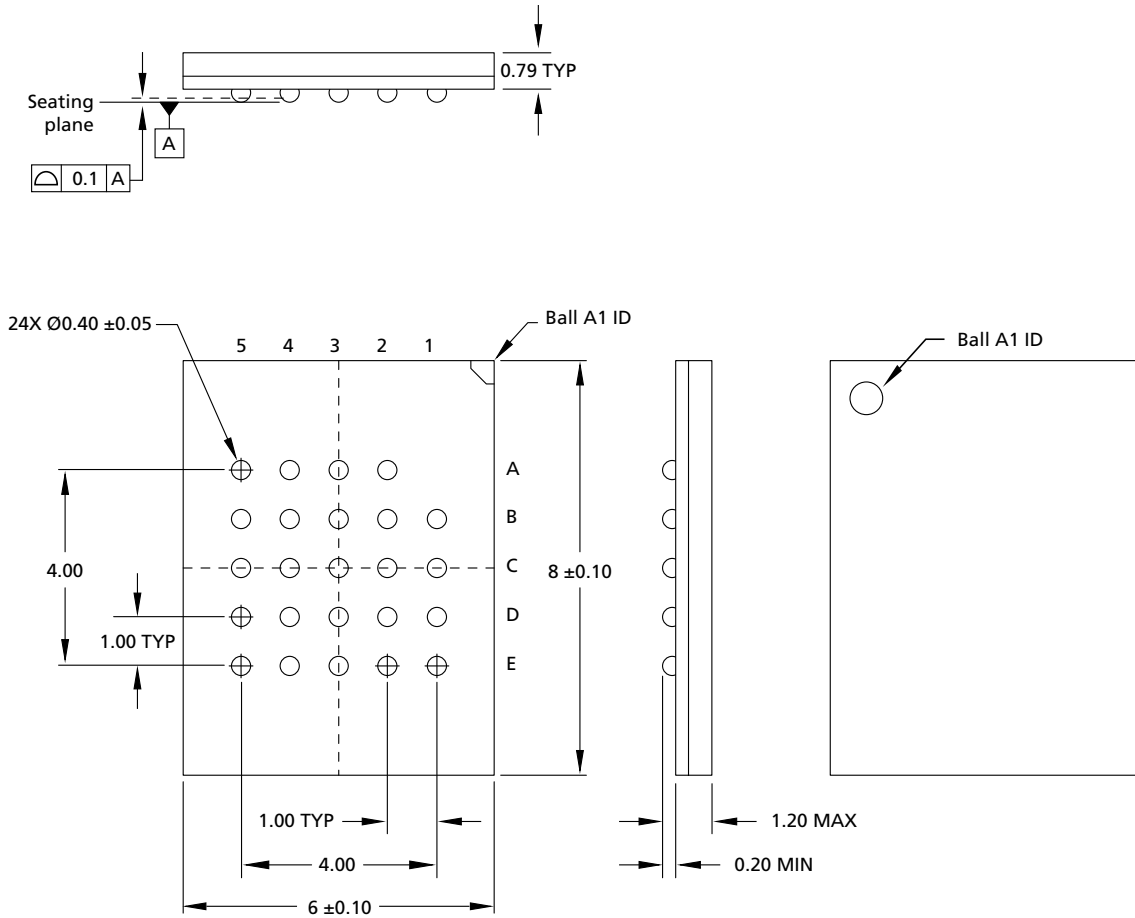


Table 18: BGA Packages and Parameter Values

Package Type	Balls	Width (mm)	Length (mm)	Max Thickness (mm)	Pitch (mm)
Easy BGA64 10 x 13 x 1.2mm	64	10	13	1.2	1
TBGA24 6 x 8 x 1mm	24	6	8	1	1
LBGA80 10 x 12 x 1.4mm	80	10	12	1.4	0.6
TFBGA88 8 x 10 x 1.2mm	88	8	10	1.2	0.8
TFBGA48 6 x 9 x 1.2mm	48	6	9	1.2	0.8
VFPGA56 7.7 x 9 x 1mm	56	7.7	9	1	0.75

24-Ball TBGA

Figure 18: 24-Ball TBGA Package Outline



Note: 1. All dimensions are in millimeters.

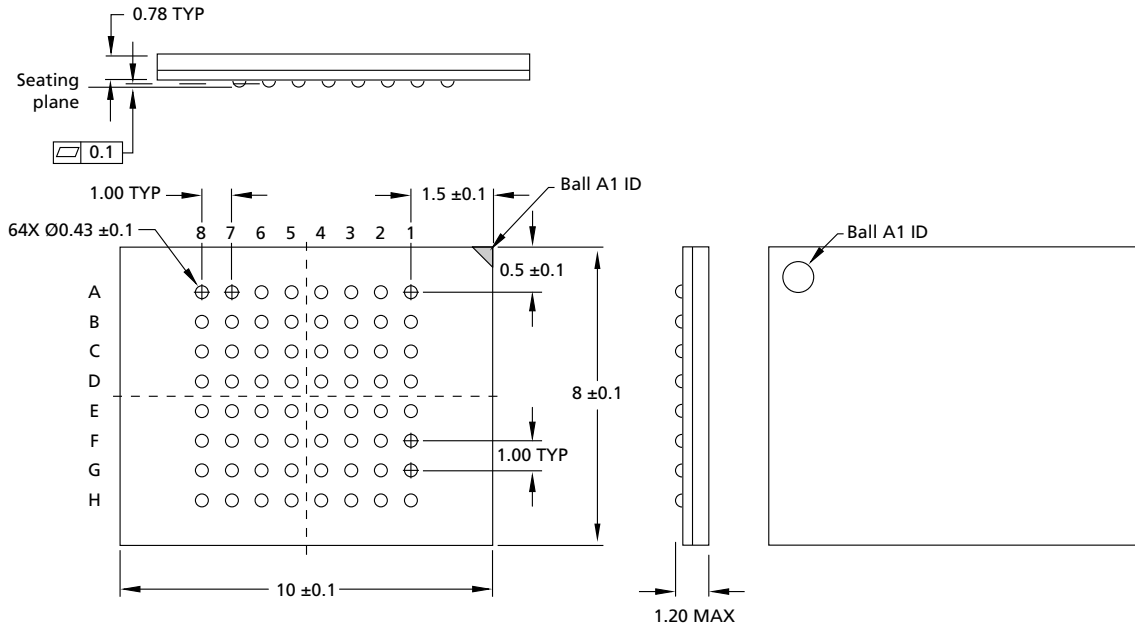
Table 19: 24-Ball TBGA Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	4	4	0.35

Note: 1. See Figure 17 (page 20) for suggested land pattern.

Easy BGA64

Figure 19: Easy BGA64 Package Outline



Note: 1. All dimensions are in millimeters.

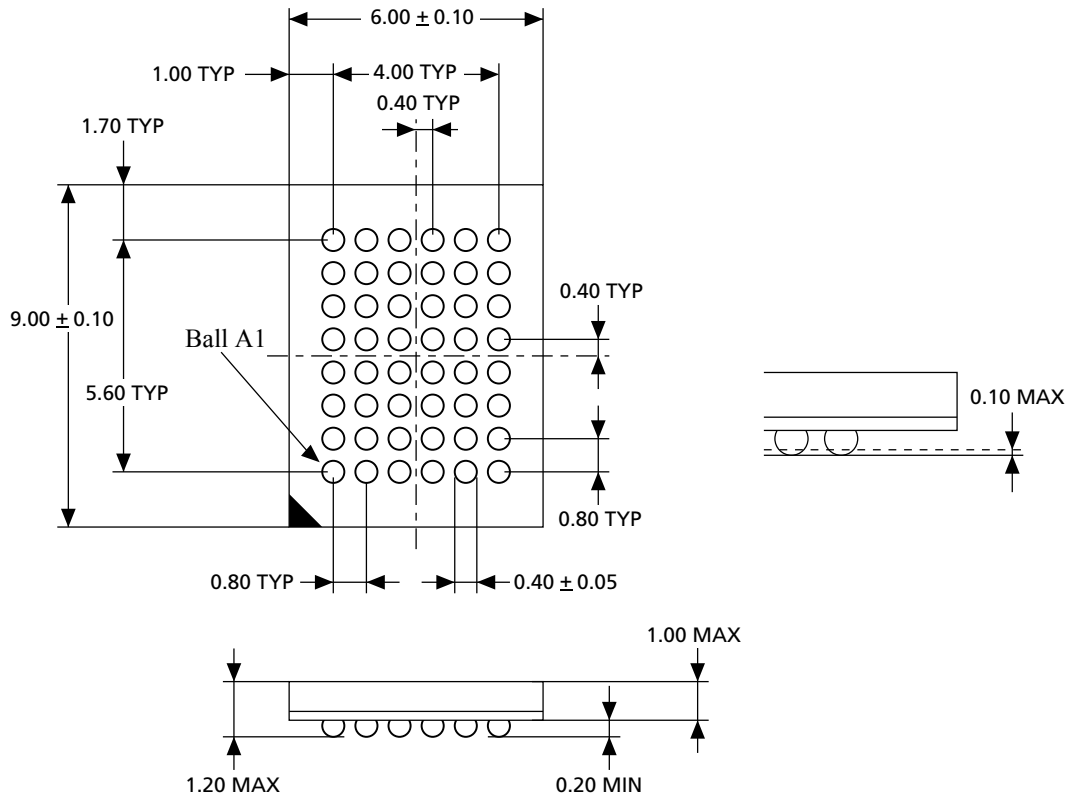
Table 20: Easy BGA64 Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	7	7	0.4

Note: 1. See Figure 17 (page 20) for suggested land pattern.

TFBGA48

Figure 20: TFBGA48 Package Outline



Note: 1. All dimensions are in millimeters.

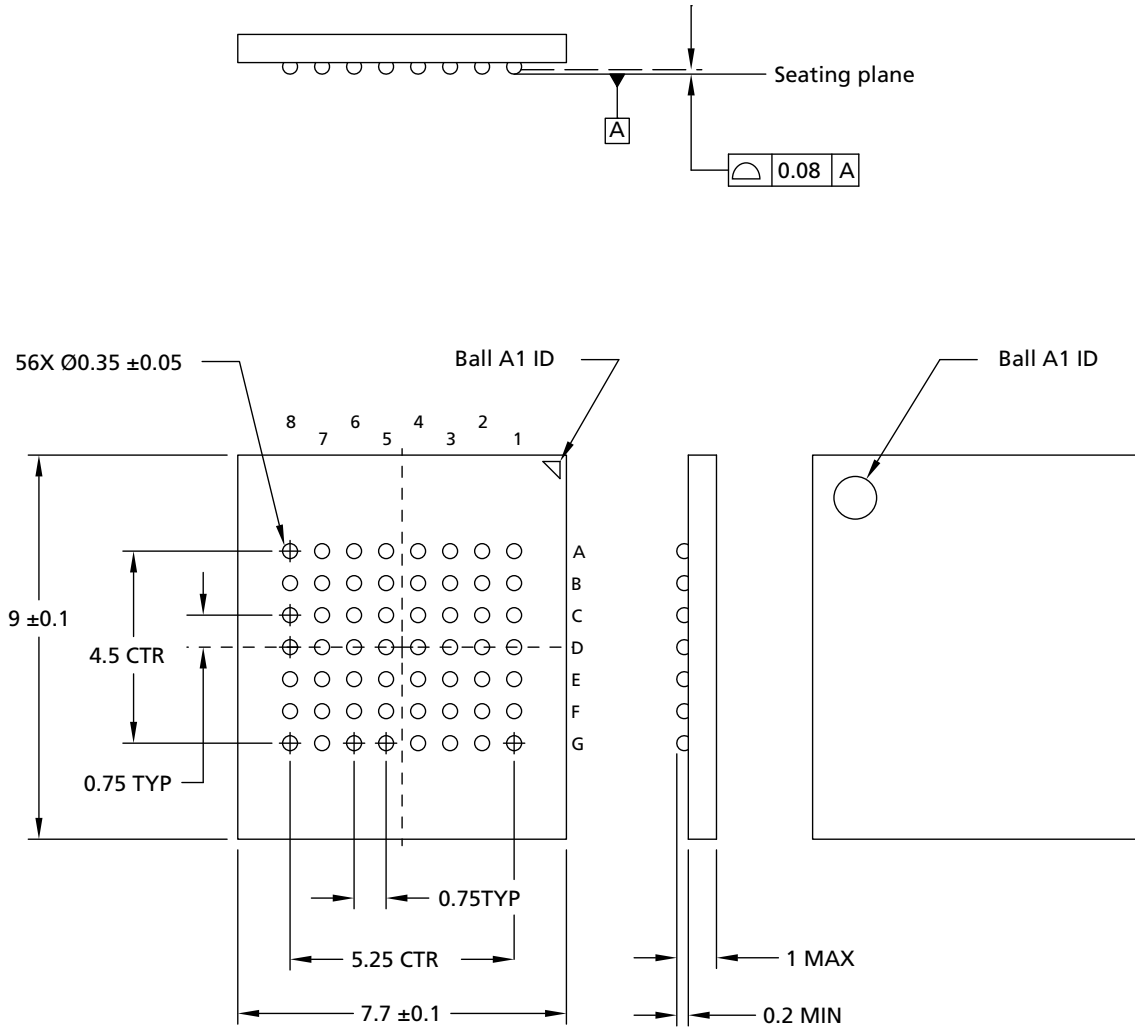
Table 21: TFBGA48 Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	4	5.6	0.35

Note: 1. See Figure 17 (page 20) for suggested land pattern.

VFBGA56

Figure 21: VFBGA56 Package Outline



Note: 1. All dimensions are in millimeters.

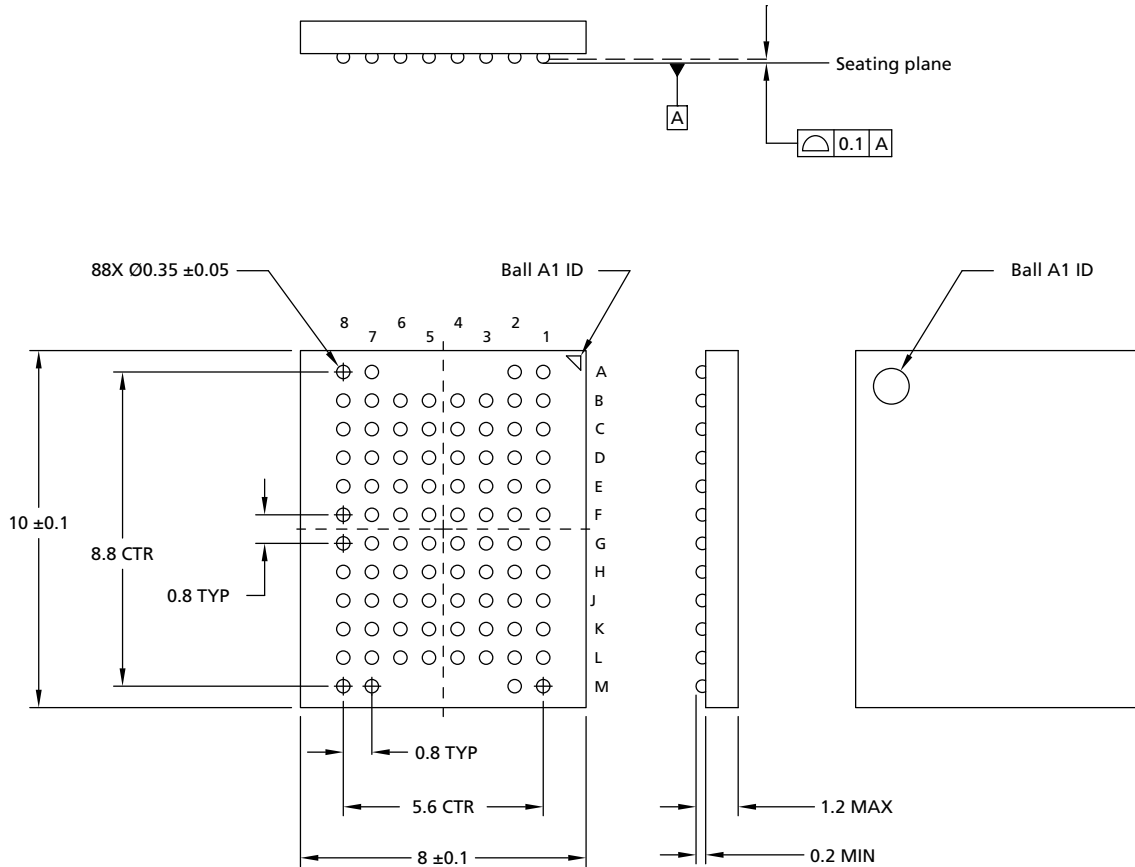
Table 22: VFBGA56 Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	5.3	4.5	0.35

Note: 1. See Figure 17 (page 20) for suggested land pattern.

TFBGA88

Figure 22: TFBGA88 Package Outline



Note: 1. All dimensions are in millimeters.

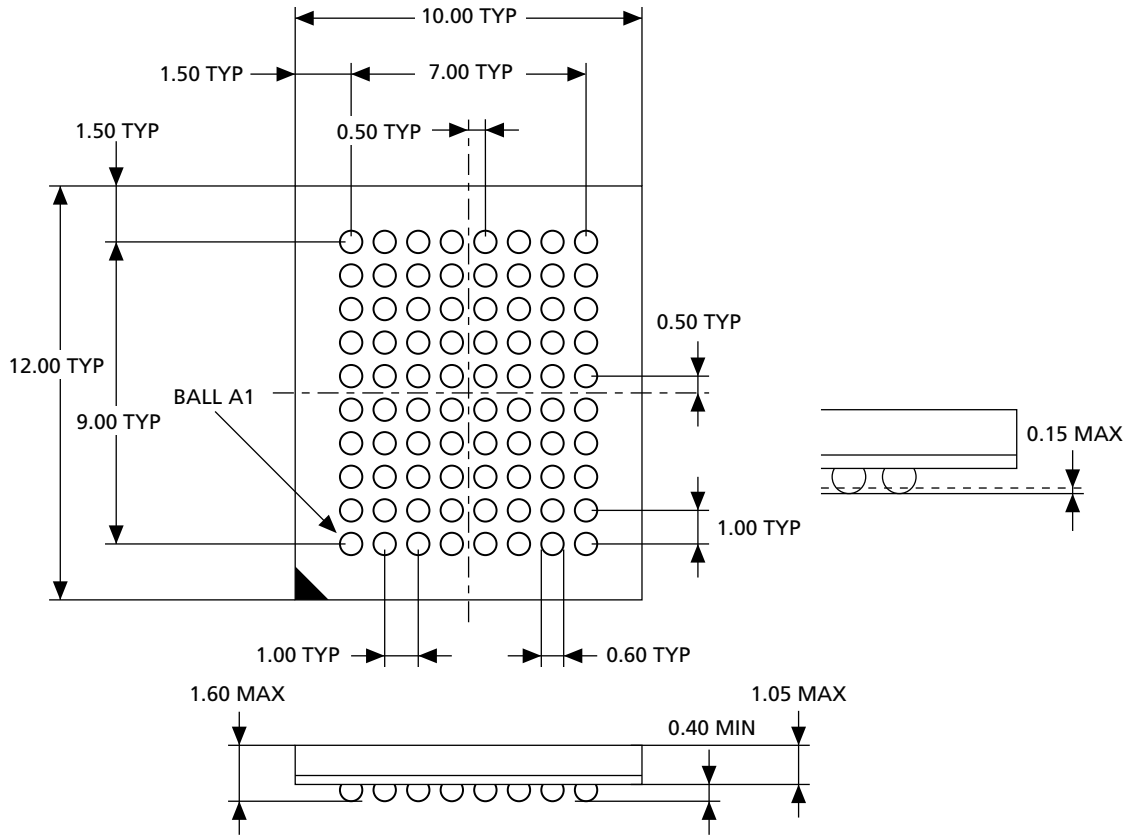
Table 23: TFBGA88 Suggested Land Pattern Dimensions

Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	5.6	8.8	0.35

Note: 1. See Figure 17 (page 20) for suggested land pattern.

LBGA80

Figure 23: LBGA80 Package Outline



Note: 1. All dimensions are in millimeters.

Table 24: LBGA80 Suggested Land Pattern Dimensions

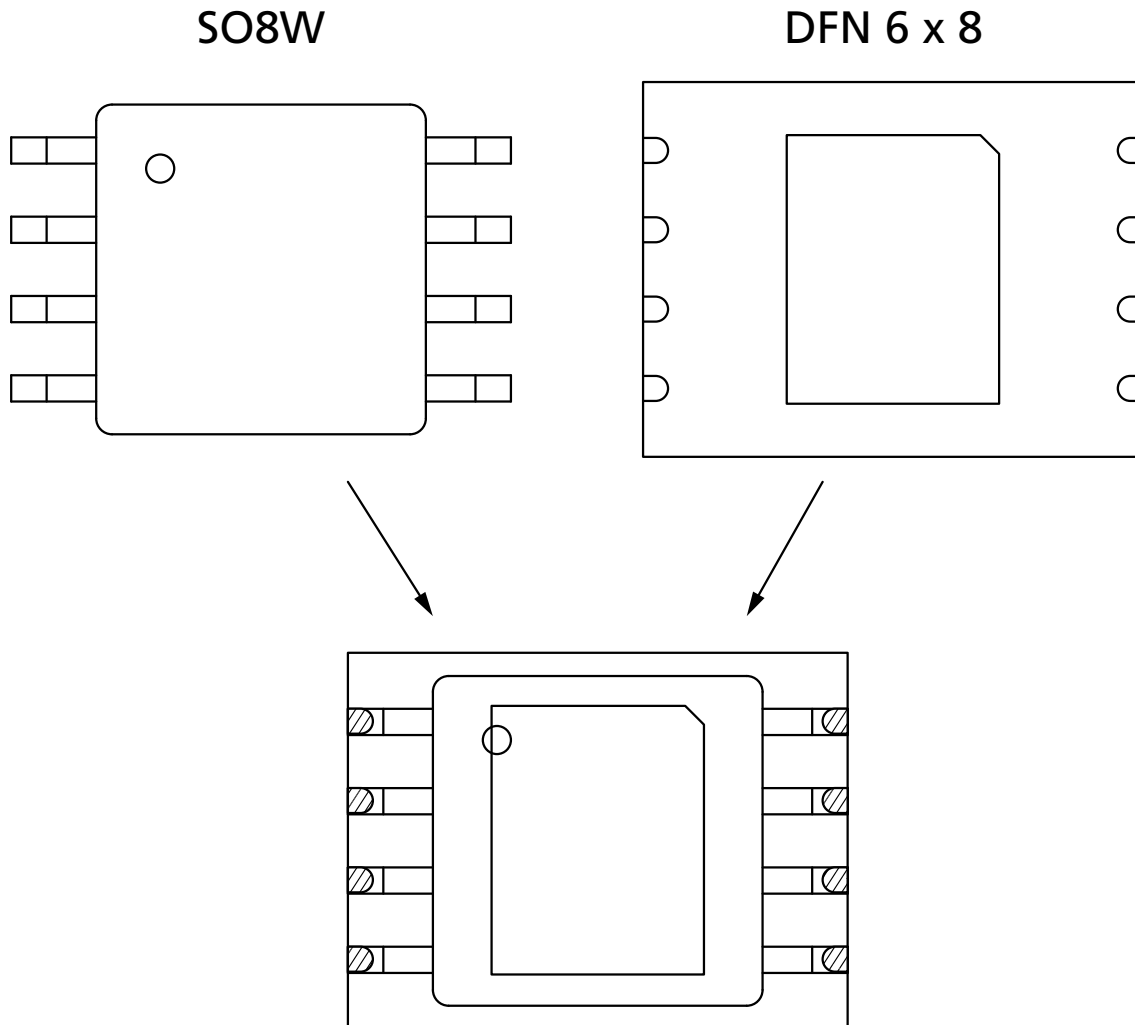
Reference	Land Pattern (mm)		
	C1	C2	X
Typical dimension	7	9	0.5

Note: 1. See Figure 17 (page 20) for suggested land pattern.

Land Pattern Compatibility

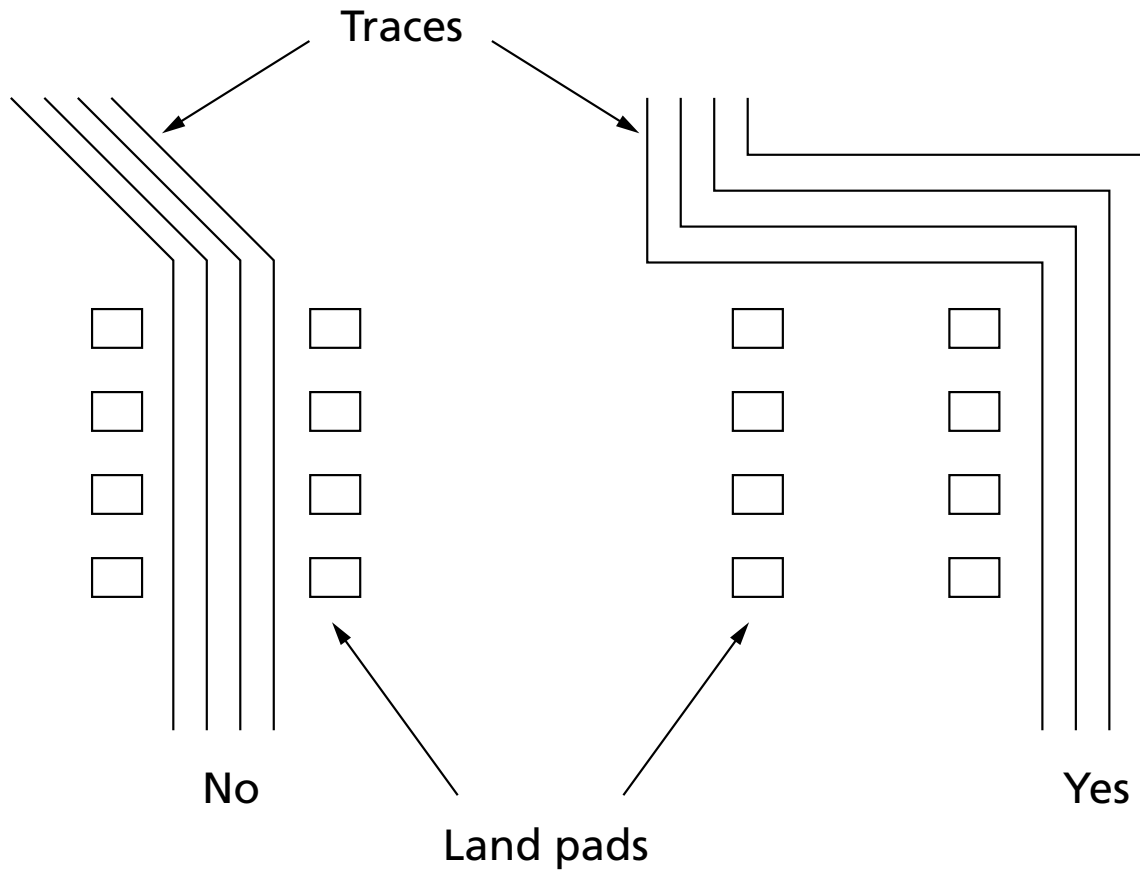
Some small outline (SO) and dual flat no-lead (DFn) packages can use the same land pattern on a PCB. SO8W and DFN8L 6 x 8mm packages have a very similar overall footprint; both packages fit the same area on the PCB. The shaded areas in the picture below indicate the areas that match between the DFN8L 6 x 8mm and the SO8W packages. All of the dimensions for the DFN 6 x 8mm, particularly pads, are smaller than the corresponding maximum dimensions for the SO8W packages.

Figure 24: SO8W and DFN 6 x 8mm Land Pattern Compatibility



Another difference between the packages is the DFN's central conductive pad, which does not require a solder connection. Micron recommends that no PCB traces run below the DFN package, because they could short-circuit to the central pad. A recommended trace routing in the PCB specifically for DFN packages is shown in the figure below.

Figure 25: Recommended Trace Routing for DFN and SO Packages



The same recommendations apply for SO8N and DFN 8L 5 x 6mm package compatibility. A complete analysis and verification of the design and requirements regarding the variations between the packages should be made before attempting to match the PCB land pad.



Revision History

Rev. A – 10/13

- Initial release

8000 S. Federal Way, P.O. Box 6, Boise, ID 83707-0006, Tel: 208-368-3900
www.micron.com/productsupport Customer Comment Line: 800-932-4992
Micron and the Micron logo are trademarks of Micron Technology, Inc.
All other trademarks are the property of their respective owners.