

SN65LVDTxx multi-channel LVDS transceivers

SN65LVDT14—One LVDS driver plus four LVDS receivers

SN65LVDT41—Four LVDS drivers plus one LVDS receiver

1 Features

- Integrated 110-Ω nominal receiver line termination resistor
- Single 3.3-V power supply (3-V to 3.6-V range)
- Supports signaling rates of at least 250 Mbps
- Flow-through pinout simplifies PCB layout
- LVTTTL-compatible logic I/Os
- ESD protection on bus pins exceeds 16 kV
- Meets or exceeds the requirements of ANSI/TIA/EIA-644A standard for LVDS
- 20-pin PW thin shrink small-outline package with 26-mil terminal pitch

2 Applications

- Serial Peripheral Interface™ (SPI) over LVDS allows long interconnects between master and slave
- Board-to-board communication
- Test and measurement
- Motor drives
- LED video walls
- Wireless infrastructure
- Telecom infrastructure
- Rack servers

3 Description

The SN65LVDTxx devices are multi-channel LVDS transceivers that operate using LVDS line drivers and receivers. The SN65LVDTxx devices support signaling rates of at least 250 Mbps, and the devices operate from a single supply (typically at 3.3 V) in a 20-pin TSSOP package designed for easy PCB layout.

The SN65LVDT14 and SN65LVDT41 provide general-purpose, asymmetric, bidirectional communication with the added benefit of high noise immunity, low electromagnetic interference (EMI), and increased cable length through the use of LVDS lines. The SN65LVDT14 and SN65LVDT41 are primarily used for SPI over LVDS applications.

The SN65LVDT14 combines one LVDS line driver with four terminated LVDS line receivers in one package. The SN65LVDT14 can be used to extend asymmetric, bidirectional interfaces such as SPI over long distances, and should be located at the SPI slave.

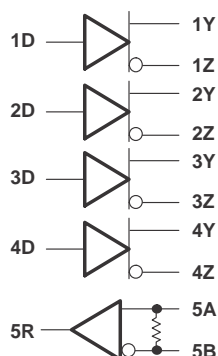
The SN65LVDT41 combines four LVDS line drivers with a single terminated LVDS line receiver in one package. The SN65LVDT41 can be used to extend asymmetric, bidirectional interfaces such as SPI over long distances, and should be located at the SPI master.

Device Information⁽¹⁾

PART NUMBER	PACKAGE	BODY SIZE (NOM)
SN65LVDT14	TSSOP (20)	6.50 mm × 4.40 mm
SN65LVDT41		

(1) For all available packages, see the orderable addendum at the end of the data sheet.

SN65LVDT41 Functional Diagram



SN65LVDT14 Functional Diagram

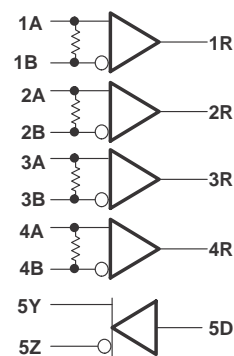


Table of Contents

1 Features 1 2 Applications 1 3 Description 1 4 Revision History 2 5 Pin Configuration and Functions 3 6 Specifications 5 6.1 Absolute Maximum Ratings 5 6.2 ESD Ratings 5 6.3 Recommended Operating Conditions 5 6.4 Thermal Information 6 6.5 Receiver Electrical Characteristics 6 6.6 Driver Electrical Characteristics 7 6.7 Device Electrical Characteristics 7 6.8 Receiver Switching Characteristics 7 6.9 Driver Switching Characteristics 7 6.10 Typical Characteristics 8 7 Parameter Measurement Information 9 8 Detailed Description 12 8.1 Overview 12	8.2 Functional Block Diagram 12 8.3 Feature Description 12 8.4 Device Functional Modes 14 9 Application and Implementation 15 9.1 Application Information 15 9.2 Typical Application 16 10 Power Supply Recommendations 20 11 Layout 20 11.1 Layout Guidelines 20 11.2 Layout Examples 24 12 Device and Documentation Support 26 12.1 Related Documentation 26 12.2 Receiving Notification of Documentation Updates 26 12.3 Related Links 26 12.4 Community Resources 26 12.5 Trademarks 26 12.6 Electrostatic Discharge Caution 26 12.7 Glossary 27 13 Mechanical, Packaging, and Orderable Information 27
--	--

4 Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

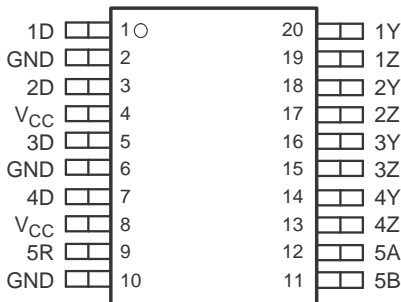
Changes from Revision B (February 2006) to Revision C

Page

<ul style="list-style-type: none"> • Added <i>Device Information</i> table, <i>ESD Ratings</i> table, <i>Feature Description</i> section, <i>Device Functional Modes</i>, <i>Application and Implementation</i> section, <i>Power Supply Recommendations</i> section, <i>Layout</i> section, <i>Device and Documentation Support</i> section, and <i>Mechanical, Packaging, and Orderable Information</i> section..... 1 • Added navigation links and removed the NRND banner from the top of the datasheet page 1 • Moved power dissipation parameter to the <i>Absolute Maximum Ratings</i> table..... 5 	1 1 5
---	-------------

5 Pin Configuration and Functions

**SN65LVDT41 PW Package
20-Pin TSSOP
Top View**



*marked as LVDT41

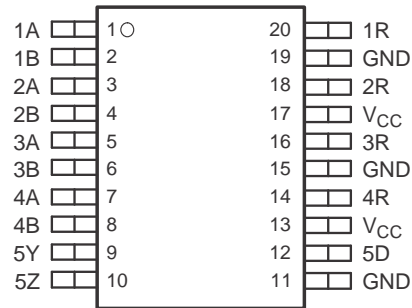
SN65LVDT41 Pin Functions

PIN		I/O	DESCRIPTION
NAME	NO.		
1D ⁽¹⁾	1	I	LVTTTL Driver Input Pin
2D ⁽¹⁾	3		
3D ⁽¹⁾	5		
4D ⁽¹⁾	7		
1Y ⁽¹⁾	20	O	Noninverting LVDS Driver Output Pin
2Y ⁽¹⁾	18		
3Y ⁽¹⁾	16		
4Y ⁽¹⁾	14		
1Z ⁽¹⁾	19	O	Inverting LVDS Driver Output Pin
2Z ⁽¹⁾	17		
3Z ⁽¹⁾	15		
4Z ⁽¹⁾	13		
5R	9	O	LVTTTL Receiver Output Pin
5A	12	I	Noninverting LVDS Receiver Input Pin
5B	11	I	Inverting LVDS Receiver Input Pin
V _{CC}	4, 8	I	Power Supply Pin, +3.3 V ± 0.3 V
GND	2, 6, 10	I	Ground Pin

(1) x = 1, 2, 3, 4 indicating channel number of SN65LVDT41

SN65LVDT14, SN65LVDT41

SLLS530C – APRIL 2002 – REVISED FEBRUARY 2019

www.ti.com
**SN65LVDT14 PW Package
20-Pin TSSOP
Top View**


*marked as LVDT14

SN65LVDT14 Pin Functions

PIN		I/O	DESCRIPTION
NAME	NO.		
1A ⁽¹⁾	1	I	Noninverting LVDS Receiver Input Pin
2A ⁽¹⁾	3		
3A ⁽¹⁾	5		
4A ⁽¹⁾	7		
1B ⁽¹⁾	2	I	Inverting LVDS Receiver Input Pin
2B ⁽¹⁾	4		
3B ⁽¹⁾	6		
4B ⁽¹⁾	8		
1R ⁽¹⁾	20	O	LVTTTL Receiver Output Pin
2R ⁽¹⁾	18		
3R ⁽¹⁾	16		
4R ⁽¹⁾	14		
5Y	9	I	Noninverting LVDS Driver Output Pin
5Z	10	I	Inverting LVDS Driver Output Pin
5D	12	O	LVTTTL Driver Input Pin
GND	11, 15, 19	I	Ground Pin
V _{CC}	13, 17	I	Power Supply Pin, +3.3 V ± 0.3 V

(1) x = 1, 2, 3, 4 indicating channel number of SN65LVDT41

6 Specifications

6.1 Absolute Maximum Ratings

over operating free-air temperature range unless otherwise noted⁽¹⁾

		MIN	MAX	UNIT
Supply voltage ⁽²⁾	V _{CC}	-0.5	4	V
Input voltage	D or R	-0.5	6	V
	A, B, Y, or Z	-0.5	4	V
Lead temperature 1,6 mm (1/16 inch) from case for 10 seconds			260	°C
Continuous total power dissipation	T _A < 25°C power rating		774	mW
	T _A = 85°C power rating		402	
	Operating factor above T _A = 25°C		6.2	mW/°C
Storage temperature, T _{stg}		-65	150	°C

- Stresses beyond those listed under absolute maximum ratings may cause permanent damage to the device. These are stress ratings only, and functional operation of the device at these or any other conditions beyond those indicated under recommended operating conditions is not implied. Exposure to absolute-maximum-rated conditions for extended periods may affect device reliability.
- All voltage values, except differential I/O bus voltages are with respect to network ground terminal.

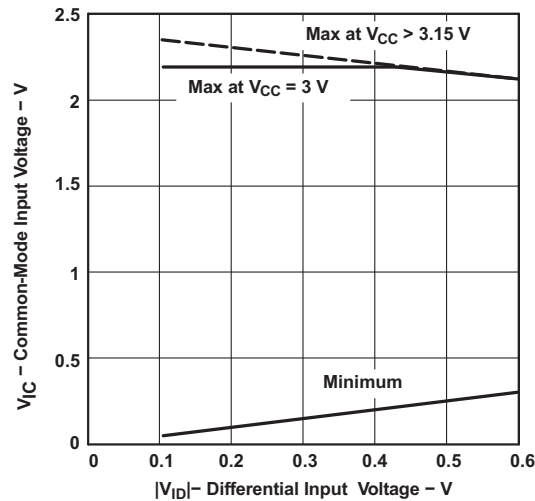
6.2 ESD Ratings

			VALUE	UNIT
V _(ESD) Electrostatic discharge	Human-body model (HBM), per ANSI/ESDA/JEDEC JS-001 ⁽¹⁾	All pins except A, B, Y, Z, and GND ⁽²⁾	±8000	V
		Pins A, B, Y, Z, and GND ⁽³⁾	±16000	
	Charged-device model (CDM), per JEDEC specification JESD22-C101 ⁽⁴⁾⁽⁵⁾			

- Tested in accordance with JEDEC Standard 22, Test Method A114-A.
- JEDEC document JEP155 states that 500-V HBM allows safe manufacturing with a standard ESD control process. Pins listed as ±8000 V may actually have higher performance.
- JEDEC document JEP155 states that 500-V HBM allows safe manufacturing with a standard ESD control process. Pins listed as ±16000 V may actually have higher performance.
- JEDEC document JEP157 states that 250-V CDM allows safe manufacturing with a standard ESD control process. Pins listed as ±500 V may actually have higher performance.
- Tested in accordance with JEDEC Standard 22, Test Method C101.

6.3 Recommended Operating Conditions

		MIN	NOM	MAX	UNIT
V _{CC}	Supply voltage	3	3.3	3.6	V
V _{IH}	High-level input voltage	2			V
V _{IL}	Low-level input voltage			0.8	V
V _{ID}	Magnitude of differential input voltage	0.1		0.6	V
V _{IC}	Common-mode input voltage, See Figure 1	$\frac{ V_{ID} }{2}$	2.4	$-\frac{ V_{ID} }{2}$	V
				V _{CC} - 0.8	V
T _A	Operating free-air temperature	-40		85	°C


Figure 1. V_{IC} vs V_{ID} and V_{CC}

6.4 Thermal Information

THERMAL METRIC ⁽¹⁾		SN65LVDTxx	UNIT
		PW (TSSOP)	
		20 PINS	
$R_{\theta JA}$	Junction-to-ambient thermal resistance	86.9	°C/W
$R_{\theta JC(top)}$	Junction-to-case (top) thermal resistance	28.4	°C/W
$R_{\theta JB}$	Junction-to-board thermal resistance	38.2	°C/W
Ψ_{JT}	Junction-to-top characterization parameter	1.4	°C/W
Ψ_{JB}	Junction-to-board characterization parameter	37.8	°C/W

(1) For more information about traditional and new thermal metrics, see the [Semiconductor and IC Package Thermal Metrics](#) application report (SPRA953).

6.5 Receiver Electrical Characteristics

over operating free-air temperature range unless otherwise noted

PARAMETER	TEST CONDITIONS	MIN	TYP ⁽¹⁾	MAX	UNIT
V_{ITH+}	Positive-going differential input voltage threshold			100	mV
V_{ITH-}	Negative-going differential input voltage threshold	-100			
V_{OH}	High-level output voltage	$I_{OH} = -8$ mA	2.4		V
V_{OL}	Low-level output voltage	$I_{OL} = 8$ mA		0.4	V
I_I	Input current (A or B inputs)	$V_I = 0$ V and $V_I = 2.4$ V, other input open		±40	µA
$I_{I(OFF)}$	Power-off input current (A or B inputs)	$V_{CC} = 0$ V, $V_I = 2.4$ V		±40	µA
C_i	Input capacitance, A or B input to GND	$V_I = A \sin 2\pi ft + CV$	5		pF
Z_t	Termination impedance	$V_{ID} = 0.4 \sin 2.5E09 t$ V	88	132	Ω

(1) All typical values are at 25°C and with a 3.3-V supply.

6.6 Driver Electrical Characteristics

over operating free-air temperature range unless otherwise noted

PARAMETER		TEST CONDITIONS	MIN	TYP ⁽¹⁾	MAX	UNIT
$ V_{OD} $	Differential output voltage magnitude	$R_L = 100 \Omega$, See Figure 9 and Figure 12	247	340	454	mV
$\Delta V_{OD} $	Change in differential output voltage magnitude between logic states		-50		50	
$V_{OC(SS)}$	Steady-state common-mode output voltage	See Figure 13	1.125		1.375	V
$\Delta V_{OC(SS)}$	Change in steady-state common-mode output voltage between logic states		-50		50	mV
$V_{OC(PP)}$	Peak-to-peak common-mode output voltage			50	150	mV
I_{IH}	High-level input current	$V_{IH} = 2 V$			20	μA
I_{IL}	Low-level input current	$V_{IL} = 0.8 V$			10	μA
I_{OS}	Short-circuit output current	V_{OY} or $V_{OZ} = 0 V$			± 24	mA
		$V_{OD} = 0 V$			± 12	
$I_{O(OFF)}$	Power-off output current	$V_{CC} = 1.5 V, V_O = 2.4 V$			± 1	μA

(1) All typical values are at 25°C and with a 3.3-V supply.

6.7 Device Electrical Characteristics

over operating free-air temperature range unless otherwise noted

PARAMETER		TEST CONDITIONS	MIN	TYP ⁽¹⁾	MAX	UNIT	
I_{CC}	Supply current	SN65LVDT14	Driver $R_L = 100 \Omega$, Driver $V_I = 0.8 V$ or $2 V$, Receiver $V_I = \pm 0.4 V$			25	mA
		SN65LVDT41				35	

(1) All typical values are at 25°C and with a 3.3-V supply.

6.8 Receiver Switching Characteristics

over operating free-air temperature range unless otherwise noted

PARAMETER		TEST CONDITIONS	MIN	NOM	MAX	UNIT
t_{PLH}	Propagation delay time, low-to-high-level output	$C_L = 10 pF$, See Figure 11	1	2.6	3.8	ns
t_{PHL}	Propagation delay time, high-to-low-level output		1	2.6	3.8	ns
t_r	Output signal rise time		0.15		1.2	ns
t_f	Output signal fall time		0.15		1.2	ns
$t_{sk(p)}$	Pulse skew ($ t_{PHL} - t_{PLH} $)			150	600	ps
$t_{sk(o)}$	Output skew ⁽¹⁾			100	400	ps
$t_{sk(pp)}$	Part-to-part skew ⁽²⁾				1	ns

(1) $t_{sk(o)}$ is the magnitude of the time difference between the t_{pLH} or t_{pHL} of all the receivers of a single device with all of their inputs connected together.

(2) $t_{sk(pp)}$ is the magnitude of the difference in propagation delay times between any specified terminals of two devices when both devices operate with the same supply voltages, at the same temperature, and have identical packages and test circuits.

6.9 Driver Switching Characteristics

over operating free-air temperature range unless otherwise noted

PARAMETER		TEST CONDITIONS	MIN	NOM	MAX	UNIT
t_{PLH}	Propagation delay time, low-to-high-level output	$R_L = 100 \Omega, C_L = 10 pF$, See Figure 14	0.9	1.7	2.9	ns
t_{PHL}	Propagation delay time, high-to-low-level output		0.9	1.6	2.9	
t_r	Differential output signal rise time		0.26		1	
t_f	Differential output signal fall time		0.26		1	

Driver Switching Characteristics (continued)

over operating free-air temperature range unless otherwise noted

PARAMETER		TEST CONDITIONS	MIN	NOM	MAX	UNIT
$t_{sk(p)}$	Pulse skew ($ t_{PHL} - t_{PLH} $)	$R_L = 100 \Omega$, $C_L = 10 \text{ pF}$, See Figure 14		150	500	ps
$t_{sk(o)}$	Output skew ⁽¹⁾			80	150	ps
$t_{sk(pp)}$	Part-to-part skew ⁽²⁾				1.5	ns

(1) $t_{sk(p)}$ is the magnitude of the time difference between the high-to-low and low-to-high propagation delay times at an output.

(2) $t_{sk(pp)}$ is the magnitude of the difference in propagation delay times between any specified terminals of two devices when both devices operate with the same supply voltages, at the same temperature, and have identical packages and test circuits.

6.10 Typical Characteristics

6.10.1 Receiver

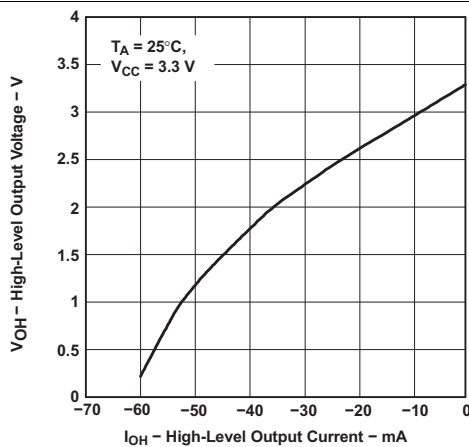


Figure 2. High-Level Output Voltage vs High-Level Output Current

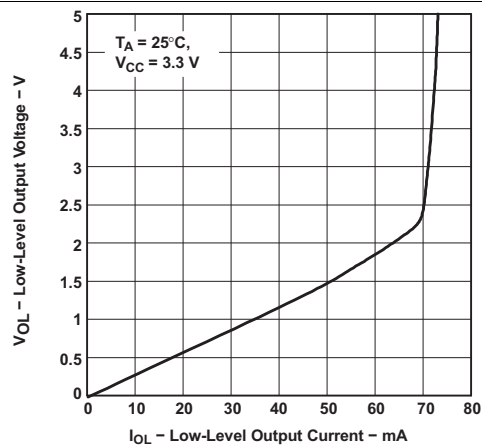


Figure 3. Low-Level Output Voltage vs Low-Level Output Current

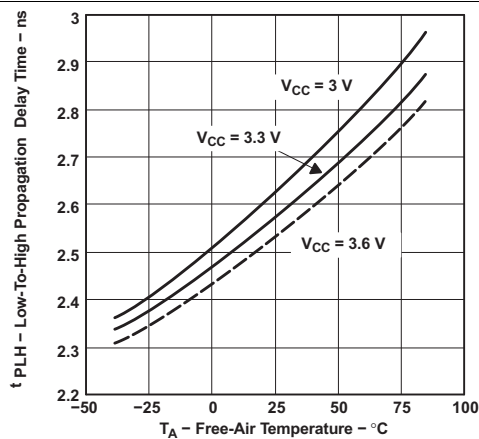


Figure 4. Low-to-High Propagation Delay Time Vs Free-Air Temperature

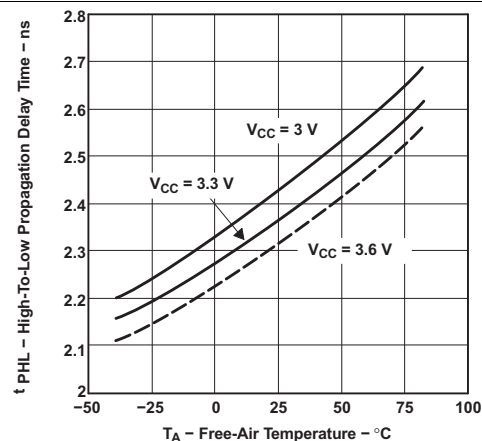
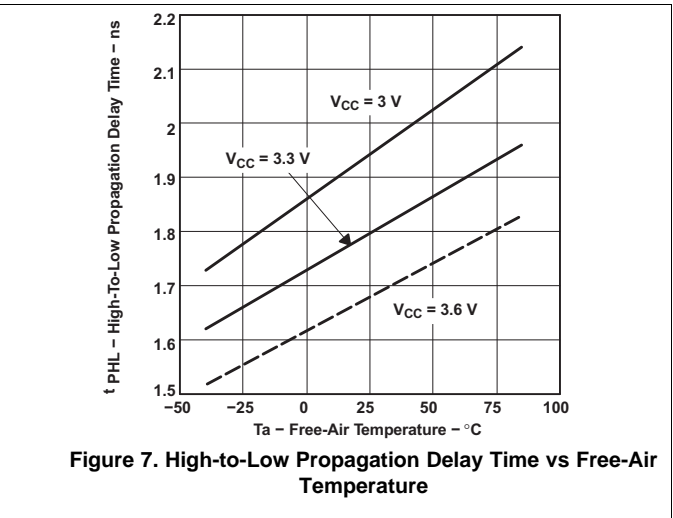
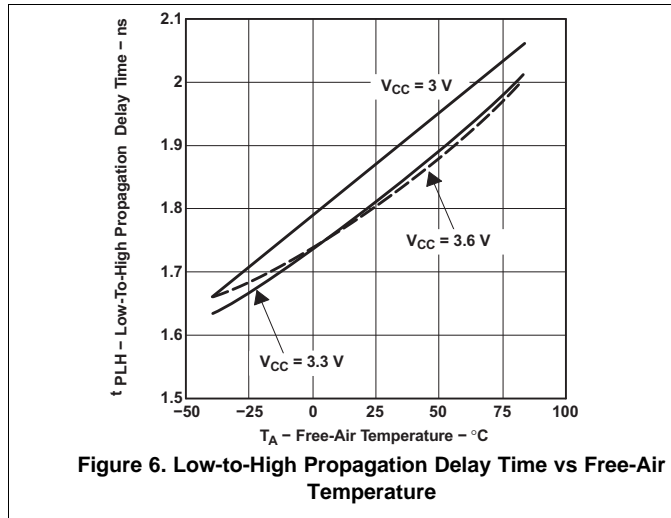


Figure 5. High-to-Low Propagation Delay Time vs Free-Air Temperature

6.10.2 Driver



7 Parameter Measurement Information

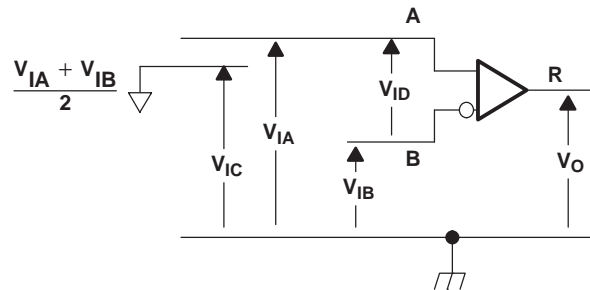
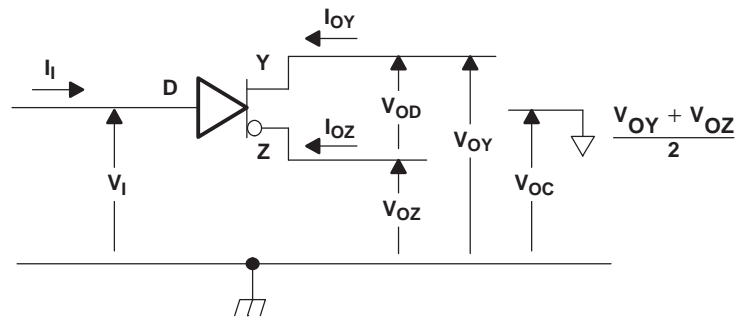
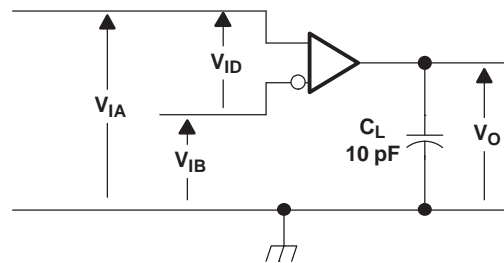
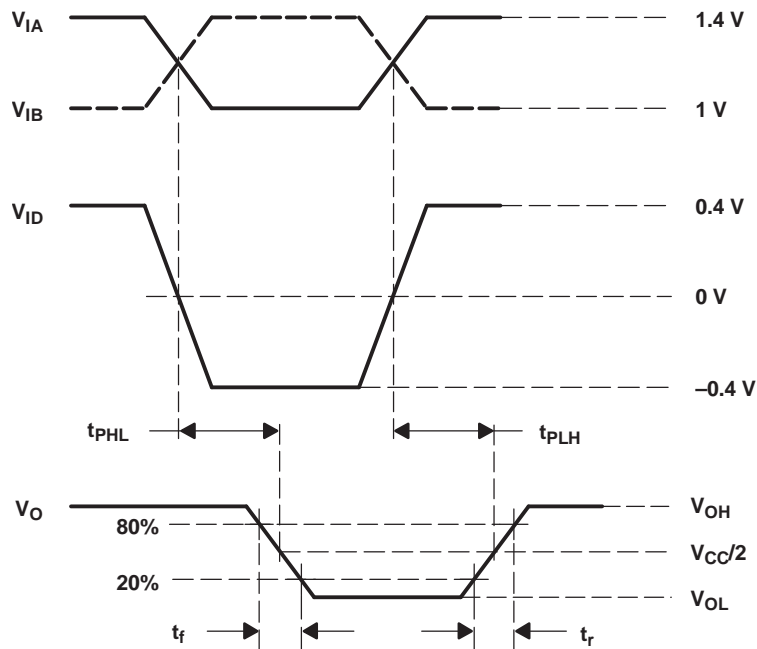


Figure 8. Receiver Voltage Definitions

Table 1. Receiver Minimum and Maximum Input Threshold Test Voltages

APPLIED VOLTAGES		RESULTING DIFFERENTIAL INPUT VOLTAGE	RESULTING COMMON-MODE INPUT VOLTAGE
V_{IA}	V_{IB}	V_{ID}	V_{IC}
1.25 V	1.15 V	100 mV	1.2 V
1.15 V	1.25 V	-100 mV	1.2 V
2.4 V	2.3 V	100 mV	2.35 V
2.3 V	2.4 V	-100 mV	2.35 V
0.1 V	0.0 V	100 mV	0.05 V
0.0 V	0.1 V	-100 mV	0.05 V
1.5 V	0.9 V	600 mV	1.2 V
0.9 V	1.5 V	-600 mV	1.2 V
2.4 V	1.8 V	600 mV	2.1 V
1.8 V	2.4 V	-600 mV	2.1 V
0.6 V	0.0 V	600 mV	0.3 V
0.0 V	0.6 V	-600 mV	0.3 V


Figure 9. Driver Voltage and Current Definitions

Figure 10. Receiver Timing Test Circuit


- A. All input pulses are supplied by a generator having the following characteristics: t_r or $t_f \leq 1$ ns, pulse repetition rate (PRR) = 1 Mpps, pulse width = $0.5 \pm 0.05 \mu\text{s}$. C_L includes instrumentation and fixture capacitance within 0,06 m of the D.U.T.

Figure 11. Receiver Timing Test Circuit Waveforms

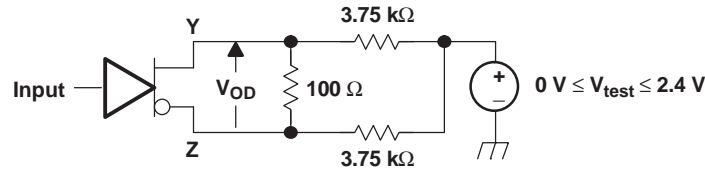
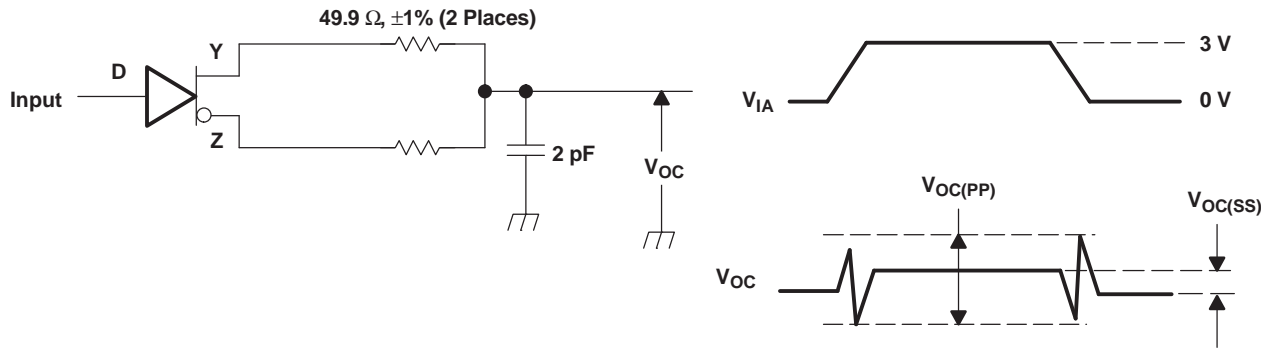
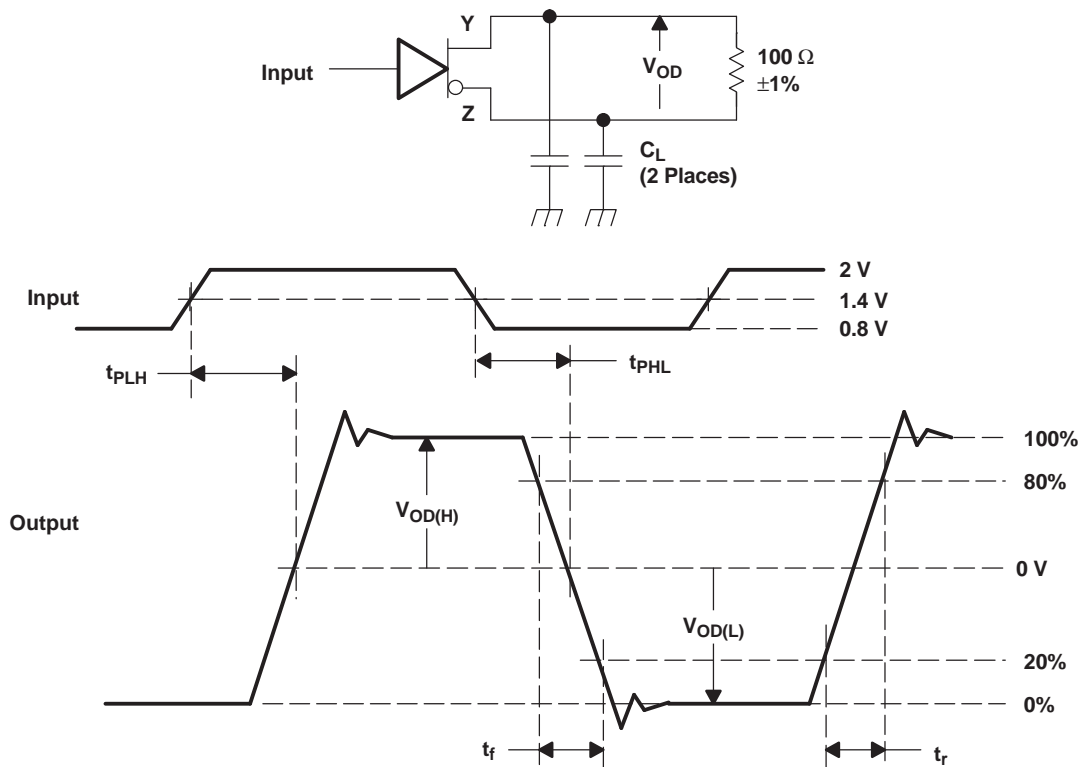


Figure 12. Driver VDO Test Circuit



- A. All input pulses are supplied by a generator having the following characteristics: t_r or $t_f \leq 1$ ns, pulse repetition rate (PRR) = 0.5 Mpps, pulse width = 500 ± 10 ns. C_L includes instrumentation and fixture capacitance within 0.06 mm of the D.U.T. The measurement of $V_{OC(PP)}$ is made on test equipment with a -3 -dB bandwidth of at least 1 GHz.

Figure 13. Test Circuit and Definitions for the Driver Common-Mode Output Voltage



- A. All input pulses are supplied by a generator having the following characteristics: t_r or $t_f \leq 1$ ns, pulse repetition rate (PRR) = 1 Mpps, pulse width = 0.5 ± 0.05 μ s. C_L includes instrumentation and fixture capacitance within 0.06 mm of the D.U.T.

Figure 14. Test Circuit, Timing, and Voltage Definitions for the Differential Output Signal

8 Detailed Description

8.1 Overview

The SN65LVDTxx integrates both low-voltage differential signaling (LVDS) line drivers, with a balanced current source design, and LVDS line receivers into a single package. This device operates from a single supply that is nominally 3.3 V, but the supply can be as low as 3 V and as high as 3.6 V. The input to the SN65LVDTxx LVDS drivers is a LVCMOS/LVTTL signal, and the output is a differential signal complying with the LVDS standard (TIA/EIA-644). The input to the SN65LVDTxx LVDS receivers is a differential signal complying with the LVDS Standard (TIA/EIA-644), and the output is a 3.3-V LVCMOS/LVTTL signal. The differential output signal of the SN65LVDTxx LVDS line drivers operates with a signal level of 350 mV, nominally, at a common-mode voltage of 1.2 V. This low differential output voltage results in low electromagnetic interference (EMI). The differential input signal of the SN65LVDTxx LVDS line receivers operates with a signal level of 350 mV, nominally, at a common-mode voltage of 1.2 V. The differential nature of the LVDS outputs and inputs can provide immunity to common-mode coupled signals (noise) that the driven/received signal may experience, along with a low EMI solution.

The SN65LVDTxx can be used to extend asymmetric bidirectional interface buses. The SN65LVDT41 combines four LVDS line drivers with a single terminated LVDS line receiver in one package, and the SN65LVDT14 combines one LVDS line driver with four terminated LVDS line receivers in one package. The SN65LVDTxx can be used to extend asymmetric bidirectional interface buses, such as Serial peripheral interface (SPI) over LVDS, to achieve long-distance and low-cost SPI communication.

The SN65LVDTxx is primarily used in point-to-point configurations, as seen in [Figure 19](#). This configuration provides a clean signaling environment for the fast edge rates of the SN65LVDTxx and other LVDS components. The SN65LVDTxx should be connected through a balanced media, which could be a standard twisted pair cable, a parallel pair cable, or simply PCB traces to a LVDS receiver. Typically, the characteristic differential impedance of the media is in the range of 100 Ω. The SN65LVDTxx device is intended to drive a 100-Ω transmission line. The 100-Ω termination resistor is selected to match the media and is located as close to the LVDS receiver input pins as possible.

8.2 Functional Block Diagram

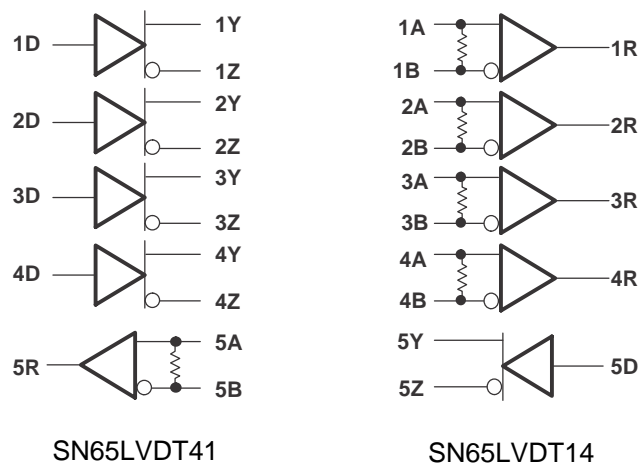


Figure 15. SN65LVDT41 (left) and SN65LVDT14 (Right) Functional Diagram

8.3 Feature Description

8.3.1 SN65LVDTxx Driver and Receiver Functionality

[Table 2](#) shows how the LVDS receiver differential input to single-ended output relationship is defined for SN65LVDTxx. The SN65LVDTxx receiver is capable of detecting signals as low as 100 mV over a ± 1 -V common-mode range centered around 1.2 V.

Feature Description (continued)

Table 2. SN65LVDTxx Receiver Functionality

INPUTS	OUTPUT
$V_{ID} = V_A - V_B$	R
$V_{ID} \geq 100 \text{ mV}$	H
$-100 \text{ mV} < V_{ID} < 100 \text{ mV}$?
$V_{ID} \leq -100 \text{ mV}$	L
Open	H

Table 3 shows how the LVDS driver single-ended input to differential output relationship is defined for SN65LVDTxx.

Table 3. SN65LVDTxx Receiver Functionality

INPUT	OUTPUTS	
D	Y	Z
H	H	L
L	L	H
Open	L	H

8.3.2 Integrated Termination

SN65LVDTxx integrates termination resistors for the LVDS receiver internal to the device. The resistor value will be between 88 Ω and 132 Ω. Additional termination resistors are not required on the receiver inputs of SN65LVDTxx.

8.3.3 SN65LVDTxx Equivalent Circuits

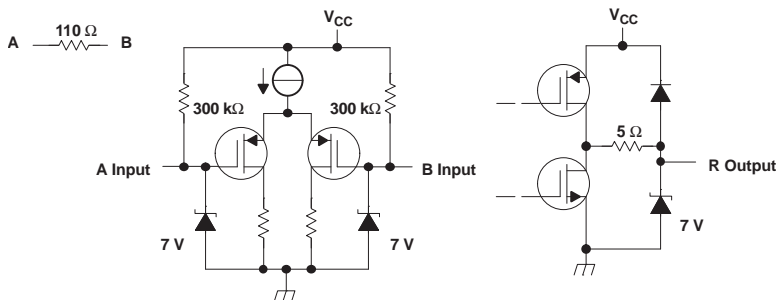


Figure 16. Receiver Equivalent Input and Output Schematic Diagrams

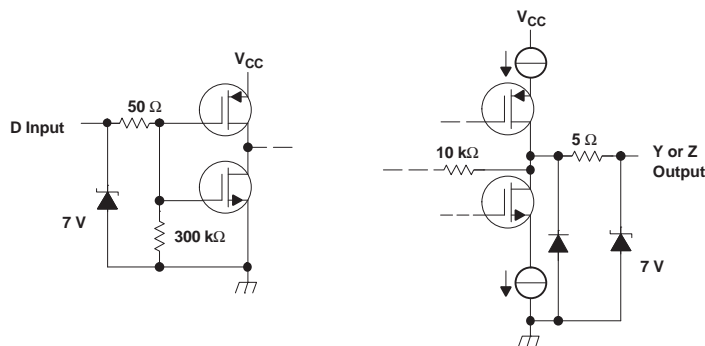


Figure 17. Driver Equivalent Input and Output Schematic Diagrams

8.4 Device Functional Modes

The device has one mode of operation that applies when operated within the [Recommended Operating Conditions](#).

9 Application and Implementation

NOTE

Information in the following applications sections is not part of the TI component specification, and TI does not warrant its accuracy or completeness. TI's customers are responsible for determining suitability of components for their purposes. Customers should validate and test their design implementation to confirm system functionality.

9.1 Application Information

The SN65LVDTxx devices are multi-channel LVDS driver and receiver pairs. The functionality of this device is simple yet extremely flexible, leading to its use in designs ranging from test and measurement to LED video wall. The varied class of potential applications share features and applications discussed in the paragraphs below.

9.1.1 Extending a Serial Peripheral Interface Using LVDS Signaling Over Differential Transmission Cables

Serial Peripheral Interface (SPI) is found in numerous applications as the communication method between processor and peripheral devices using single-ended signals over short distances. However, there is increasing demand for longer range SPI communication on the same PCB or from board to board. As distance increases, external noise, and electromagnetic interference (EMI) with single-ended SPI signals becomes an issue. Furthermore, increased distance limits the data rate due to propagation delay, and affects the signal quality due to potential ground shift between boards. A long distance SPI communication is achievable with the help of LVDS. LVDS, as specified by the TIA/EIA-644-A standard, provides several benefits when compared to alternative long-distance signaling technologies: low EMI, high noise immunity, low power consumption, and inexpensive interconnect cables.

SPI operates in a master-slave architecture, with four unidirectional signal lines. The master supplies data named Master-Out-Slave-In (MOSI), a clock (SCK), and a optional Chip Select (CS) signal to control the operation of the system with multiple slave devices. The MOSI, SCLK, and \overline{CS} signals are unidirectional from the master device to slave devices. The serial data from slave to master device is a unidirectional signal named Master-In-Slave-Out (MISO). The flow of data can be seen in [Figure 18](#).

SN65LVDT14 and SN65LVDT41 provide the necessary LVDS drivers and receivers specifically targeted at implementing a long distance SPI application. It uses the unidirectional links for the MOSI, MISO, SCK, and CS signals and converts the single-ended data into a unidirectional LVDS links. SN65LVDT41 combines four LVDS line drivers with a single terminated LVDS line receiver in one package should be located at the SPI master device. SN65LVDT14 combines one LVDS line driver with four terminated LVDS line receivers in one package and should be located at the SPI device.

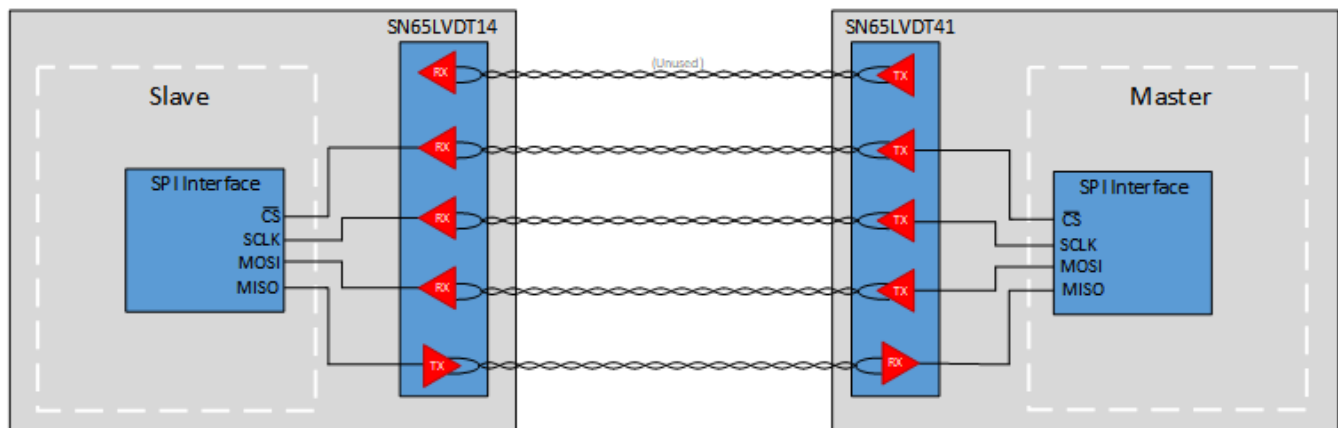


Figure 18. Typical SPI Application With LVDS

9.2 Typical Application

Point-to-Point applications provide a clean signaling environment for the fast edge rates of the SN65LVDTxx and other LVDS components. The SN65LVDTxx is connected through a balanced media which may be a standard twisted-pair cable, a parallel pair cable, or simply PCB traces to a LVDS receiver. Typically, the characteristic differential impedance of the media is in the range of 100 Ω. The SN65LVDTxx device is intended to drive a 100-Ω transmission line. The 100-Ω termination resistor is selected to match the media and is located as close to the LVDS receiver input pins as possible.

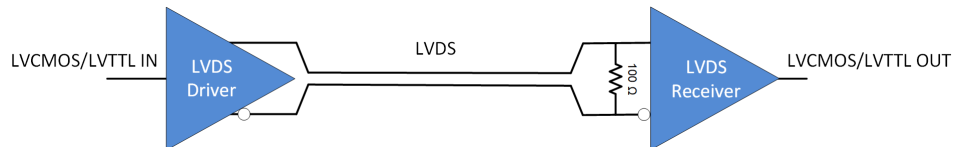


Figure 19. Typical LVDS Point-to-Point Application

9.2.1 Design Requirements

Table 4 lists the design parameters for typical point-to-point applications.

Table 4. Typical Design Parameters

Design Parameter	Example Value
Supply Voltage (V_{DD})	3 to 3.6 V
Single-ended Input Voltage	0 to V_{DD}
Data Rate	0 to 400 Mbps
Interconnect Characteristic Impedance	100 Ω
Number of LVDS Channel	5
Number of Receiver/Transmitter Nodes	5
Ground shift between driver and receiver	±1 V

Table 5. SPI Design Parameters

Design Parameter	Example Value
Supply Voltage (V_{DD})	3 to 3.6 V
Single-ended Input Voltage	0 to V_{DD}
SPI Data Rate	0 to 10 Mbps
Interconnect Characteristic Impedance	100 Ω
Number of LVDS Channel	4
Number of Transmitter Nodes	3
Number of Receiver Nodes	1
Ground shift between driver and receiver	±1 V

9.2.2 Detailed Design Procedure

9.2.2.1 SPI Propagation Delay Limitations

In typical SPI communication, the SPI master decides the sampling rate and data transfer rate, sends data at the rising edge of one clock cycle, and receives data on the falling edge within the same clock cycle. In a low latency system, the data in peripheral device should be made available to the host system with minimum delay. However in systems with high latency, the total round trip propagation delay of the SPI system must be less than half the SCLK period to avoid missing bits. There are three major delay contributors in a typical system—the SPI peripheral, data link device, and transmission media. Both the SPI peripheral and the data link device have fixed delay. The delay in transmission media, however, increases as communication distance increases. The relationship between cable length and SPI clock frequency can be seen in Figure 22. Figure 22 refers to a system where both MISO and MOSI are used, accounting for the case of slave-to-master data transmission, including roundtrip delay. The specific setup is described in [Extending SPI and McBSP with differential interface products](#) (SLLA142)

9.2.2.2 Interconnecting Media

The physical communication channel between the LVDS driver and the LVDS receiver may be any balanced and paired metal conductors meeting the requirements of the LVDS standard, the key points of which are included here. This media may be shielded twisted-pair cables, twinax cables, flat ribbon cables, or PCB traces. The nominal characteristic impedance of the interconnect media should be between 100 Ω and 120 Ω with a variation of no more than 10% (90 Ω to 132 Ω). Balanced cables (for example, twisted-pair) are usually better than unbalanced cables (like ribbon and simple coax cables) for noise reduction and signal quality. Balanced cables tend to generate less EMI due to field-canceling effects and also tend to pick up electromagnetic radiation a common-mode (not differential mode) noise which is rejected by the receiver. There should not introduce major impedance discontinuities in the system.

9.2.2.3 Input Fail-Safe Biasing

External pullup and pulldown resistors may be used to provide enough of an offset to enable an input fail-safe under open-circuit conditions. This configuration ties the positive LVDS input pin to VDD through a pullup resistor and the negative LVDS input pin is tied to GND by a pulldown resistor. The pullup and pulldown resistors should be in the 5-kΩ to 15-kΩ range to minimize loading and waveform distortion to the driver. The common-mode bias point should be set to approximately 1.2 V (less than 1.75 V) to be compatible with the internal circuitry. Refer to application note [AN-1194 Fail-safe biasing of LVDS interfaces](#) (SNLA051) for more information.

9.2.2.4 Power Decoupling Recommendations

Bypass capacitors must be used on power pins. Use high-frequency, ceramic (surface mount is recommended), 0.1-μF and 0.001-μF capacitors in parallel at the power supply pin with the smallest value capacitor closest to the device supply pin.

Bypass capacitors play a key role in power distribution circuitry. Specifically, they create low-impedance paths between power and ground. At low frequencies, a good digital power supply offers very low-impedance paths between its terminals. However, as higher frequency currents propagate through power traces, the source is quite often incapable of maintaining a low-impedance path to ground. Bypass capacitors are used to address this shortcoming. Usually, large bypass capacitors (10 μF to 1000 μF) at the board-level do a good job up into the kHz range. Due to their size and length of their leads, they tend to have large inductance values at the switching frequencies of modern digital circuitry. To solve this problem, one must resort to the use of smaller capacitors (nF to μF range) installed locally next to the integrated circuit.

Multilayer ceramic chip or surface-mount capacitors (size 0603 or 0805) minimize lead inductances of bypass capacitors in high-speed environments, because their lead inductance is about 1 nH. For comparison purposes, a typical capacitor with leads has a lead inductance around 5 nH.

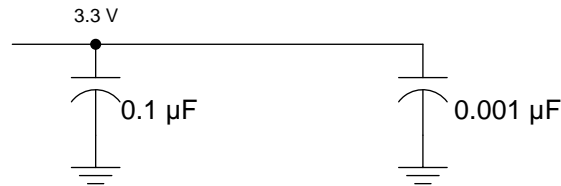
The value of the bypass capacitors used locally with LVDS chips can be determined by [Equation 1](#) and [Equation 2](#) according to Johnson⁽¹⁾ equations 8.18 to 8.21. A conservative rise time of 200 ps and a worst-case change in supply current of 1 A covers the whole range of LVDS devices offered by Texas Instruments. In this example, the maximum power supply noise tolerated is 200 mV. However, this figure varies depending on the noise budget available in the design. ⁽¹⁾

$$C_{\text{chip}} = \left(\frac{\Delta I_{\text{Maximum Step Change Supply Current}}}{\Delta V_{\text{Maximum Power Supply Noise}}} \right) \times T_{\text{Rise Time}} \quad (1)$$

$$C_{\text{LVDS}} = \left(\frac{1\text{A}}{0.2\text{V}} \right) \times 200 \text{ ps} = 0.001 \mu\text{F} \quad (2)$$

[Figure 20](#) lowers lead inductance and covers intermediate frequencies between the board-level capacitor (>10 μF) and the value of capacitance found above (0.001 μF). TI recommends that the user place the smallest value of capacitance as close to the chip as possible.

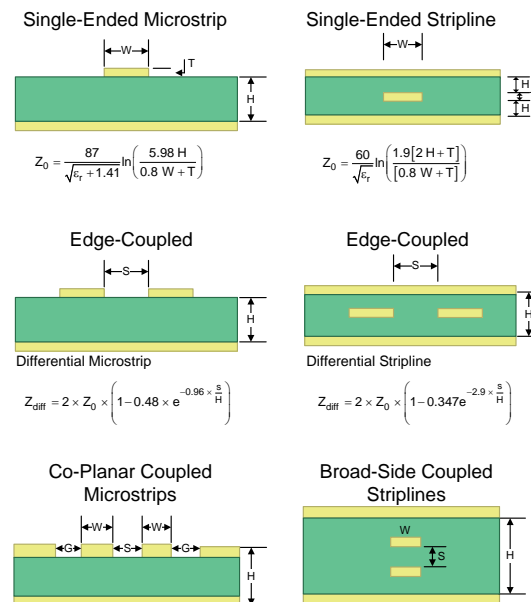
(1) Howard Johnson & Martin Graham. 1993. High Speed Digital Design – A Handbook of Black Magic. Prentice Hall PRT. ISBN number 013395724.


Figure 20. Recommended LVDS Bypass Capacitor Layout

9.2.2.5 PCB Transmission Lines

As per the *LVDS owner's manual design guide, 4th edition* (SNLA187), [Figure 21](#) depicts several transmission line structures commonly used in printed-circuit boards (PCBs). Each structure consists of a signal line and return path with a uniform cross section along its length. A microstrip is a signal trace on the top (or bottom) layer, separated by a dielectric layer from its return path in a ground or power plane. A stripline is a signal trace in the inner layer, with a dielectric layer in between a ground plane above and below the signal trace. The dimensions of the structure along with the dielectric material properties determine the characteristic impedance of the transmission line (also called controlled-impedance transmission line).

When two signal lines are placed close by, they form a pair of coupled transmission lines. [Figure 21](#) shows examples of edge-coupled microstrip lines, and edge-coupled or broad-side-coupled striplines. When excited by differential signals, the coupled transmission line is referred to as a differential pair. The characteristic impedance of each line is called odd-mode impedance. The sum of the odd-mode impedances of each line is the differential impedance of the differential pair. In addition to the trace dimensions and dielectric material properties, the spacing between the two traces determines the mutual coupling and impacts the differential impedance. When the two lines are immediately adjacent (like if S is less than $2W$, for example), the differential pair is called a tightly-coupled differential pair. To maintain constant differential impedance along the length, it is important to keep the trace width and spacing uniform along the length, as well as maintain good symmetry between the two lines.


Figure 21. Controlled-Impedance Transmission Lines

9.2.2.6 Probing LVDS Transmission Lines on PCB

Always use high impedance ($> 100 \text{ k}\Omega$), low capacitance ($< 2 \text{ pF}$) scope probes with a wide bandwidth (1 GHz) scope. Improper probing will skew results.

9.2.3 Application Curve

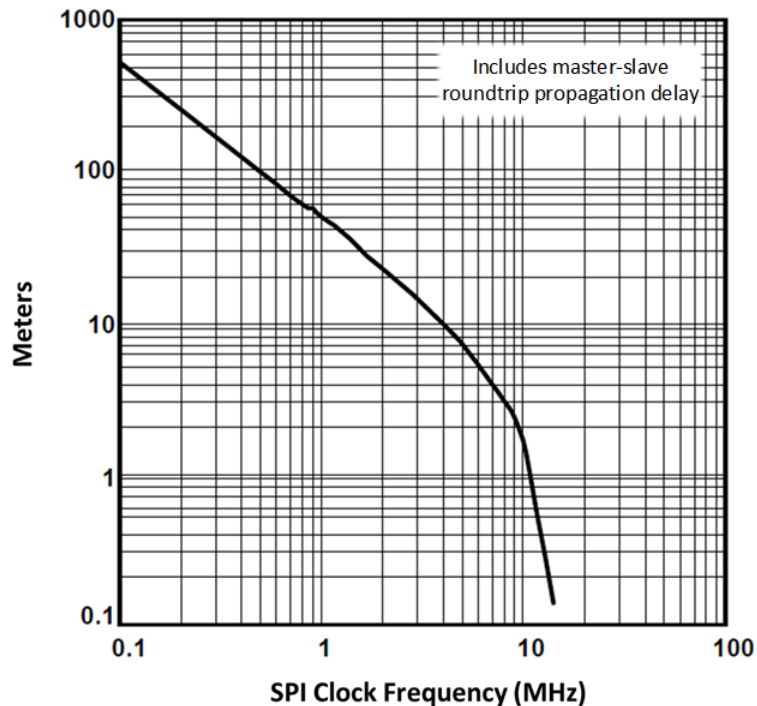


Figure 22. SN65LVDTxx SPI Performance: Cable Length vs SPI Clock

10 Power Supply Recommendations

The SN65LVDTxx devices are designed to operate from a single power supply with a supply voltage range of 3 V to 3.6 V. In a typical point-to-point application, a driver and a receiver may be on separate boards, or even separate equipment. In these cases, separate supplies would be used at each location. The expected ground potential difference between the driver power supply and the driver power supply would be less than $|\pm 1\text{ V}|$. Board level and local device level bypass capacitance should be used.

11 Layout

11.1 Layout Guidelines

11.1.1 Microstrip vs. Stripline Topologies

As per the *LVDS application and data handbook* (SLLD009), printed-circuit boards usually offer designers two transmission line options: microstrip and stripline. Microstrips are traces on the outer layer of a PCB, as shown in Figure 23.

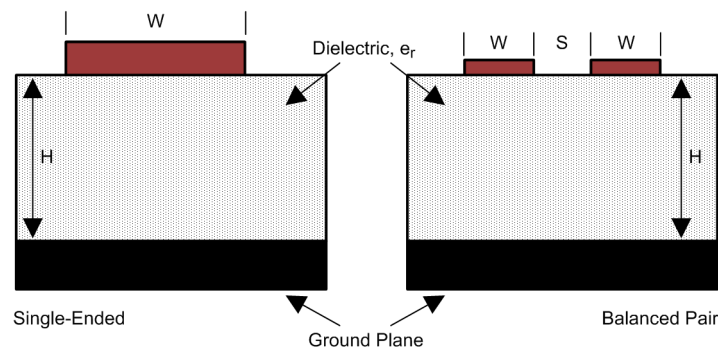


Figure 23. Microstrip Topology

On the other hand, striplines are traces between two ground planes. Striplines are less prone to emissions and susceptibility problems because the reference planes effectively shield the embedded traces. However, from the standpoint of high-speed transmission, juxtaposing two planes creates additional capacitance. TI recommends routing LVDS signals on microstrip transmission lines when possible. The PCB traces allow designers to specify the necessary tolerances for Z_0 based on the overall noise budget and reflection allowances. Footnotes 1⁽²⁾, 2⁽³⁾, and 3⁽⁴⁾ provide formulas for Z_0 and t_{PD} for differential and single-ended traces. ⁽²⁾ ⁽³⁾ ⁽⁴⁾

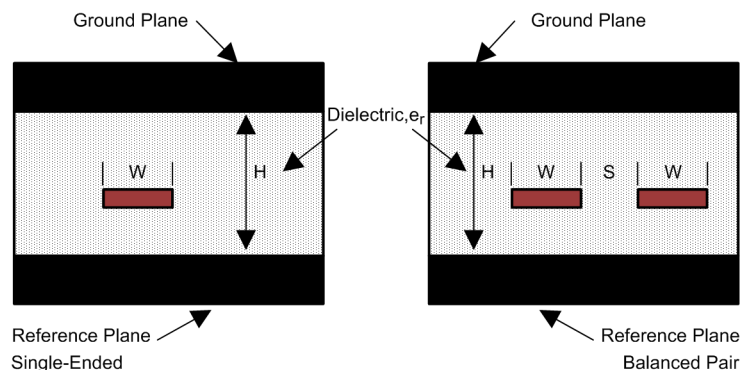


Figure 24. Stripline Topology

(2) Howard Johnson & Martin Graham. 1993. High Speed Digital Design – A Handbook of Black Magic. Prentice Hall PRT. ISBN number 013395724.

(3) Mark I. Montrose. 1996. Printed Circuit Board Design Techniques for EMC Compliance. IEEE Press. ISBN number 0780311310.

(4) Clyde F. Coombs, Jr. Ed, Printed Circuits Handbook, McGraw Hill, ISBN number 0070127549.

Layout Guidelines (continued)

11.1.2 Dielectric Type and Board Construction

The speeds at which signals travel across the board dictates the choice of dielectric. FR-4, or an equivalent, usually provides adequate performance for use with LVDS signals. If rise or fall times of LVCMOS/LVTTL signals are less than 500 ps, empirical results indicate that a material with a dielectric constant near 3.4, such as Rogers™ 4350 or Nelco N4000-13, may be desired. Once the designer chooses the dielectric, there are several parameters pertaining to the board construction that can affect performance. The following set of guidelines were developed experimentally through several designs involving LVDS devices:

- Copper weight: 15 g or 1/2 oz start, plated to 30 g or 1 oz
- All exposed circuitry should be solder-plated (60/40) to 7.62 μm or 0.0003 in (minimum).
- Copper plating should be 25.4 μm or 0.001 in (minimum) in plated-through-holes.
- Solder mask over bare copper with solder hot-air leveling

11.1.3 Recommended Stack Layout

Following the choice of dielectrics and design specifications, the designer must decide how many levels to use in the stack. To reduce the LVCMOS/LVTTL to LVDS crosstalk, it is good practice to have at least two separate signal planes as shown in [Figure 25](#).

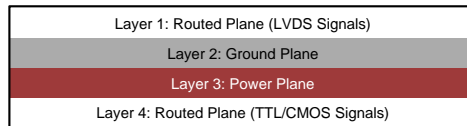


Figure 25. Four-Layer PCB Board

NOTE

The separation between layers 2 and 3 should be 127 μm (0.005 in). By keeping the power and ground planes tightly coupled, the increased capacitance acts as a bypass for transients.

One of the most common stack configurations is the six-layer board, as shown in [Figure 26](#).

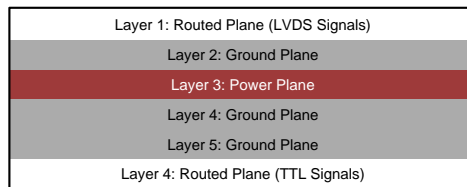


Figure 26. Six-Layer PCB Board

In this particular configuration, it is possible to isolate each signal layer from the power plane by at least one ground plane. The result is improved signal integrity, but fabrication is more expensive. Using the 6-layer board is preferable, because it offers the layout designer more flexibility in varying the distance between signal layers and referenced planes in addition to ensuring reference to a ground plane for signal layers 1 and 6.

11.1.4 Separation Between Traces

The separation between traces depends on several factors, but the amount of coupling that can be tolerated usually dictates the actual separation. Low-noise coupling requires close coupling between the differential pair of an LVDS link to benefit from the electromagnetic field cancellation. The traces should be 100- Ω differential and thus coupled in the manner that best fits this requirement. In addition, differential pairs should have the same electrical length to ensure that they are balanced, thus minimizing problems with skew and signal reflection.

Layout Guidelines (continued)

In the case of two adjacent single-ended traces, one should use the 3-W rule, which stipulates that the distance between two traces must be greater than two times the width of a single trace, or three times its width measured from trace center to trace center. This increased separation effectively reduces the potential for crosstalk. The same rule should be applied to the separation between adjacent LVDS differential pairs, whether the traces are edge-coupled or broad-side-coupled.

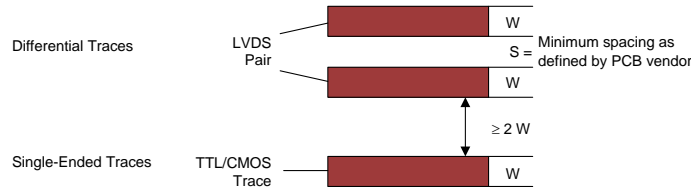


Figure 27. 3-W Rule for Single-Ended and Differential Traces (Top View)

Exercise caution when using autorouters, because they do not always account for all factors affecting crosstalk and signal reflection. For instance, it is best to avoid sharp 90° turns to prevent discontinuities in the signal path. Using successive 45° turns tends to minimize reflections.

11.1.5 Crosstalk and Ground Bounce Minimization

To reduce crosstalk, it is important to provide a return path to high-frequency currents that is as close to its originating trace as possible. A ground plane usually achieves this. Because the returning currents always choose the path of lowest inductance, they are most likely to return directly under the original trace, thus minimizing crosstalk. Lowering the area of the current loop lowers the potential for crosstalk. Traces kept as short as possible with an uninterrupted ground plane running beneath them emit the minimum amount of electromagnetic field strength. Discontinuities in the ground plane increase the return path inductance and should be avoided.

11.1.6 Decoupling

Each power or ground lead of a high-speed device should be connected to the PCB through a low inductance path. For best results, one or more vias are used to connect a power or ground pin to the nearby plane. TI recommends that the user place a via immediately adjacent to the pin to avoid adding trace inductance. Placing a power plane closer to the top of the board reduces the effective via length and its associated inductance.

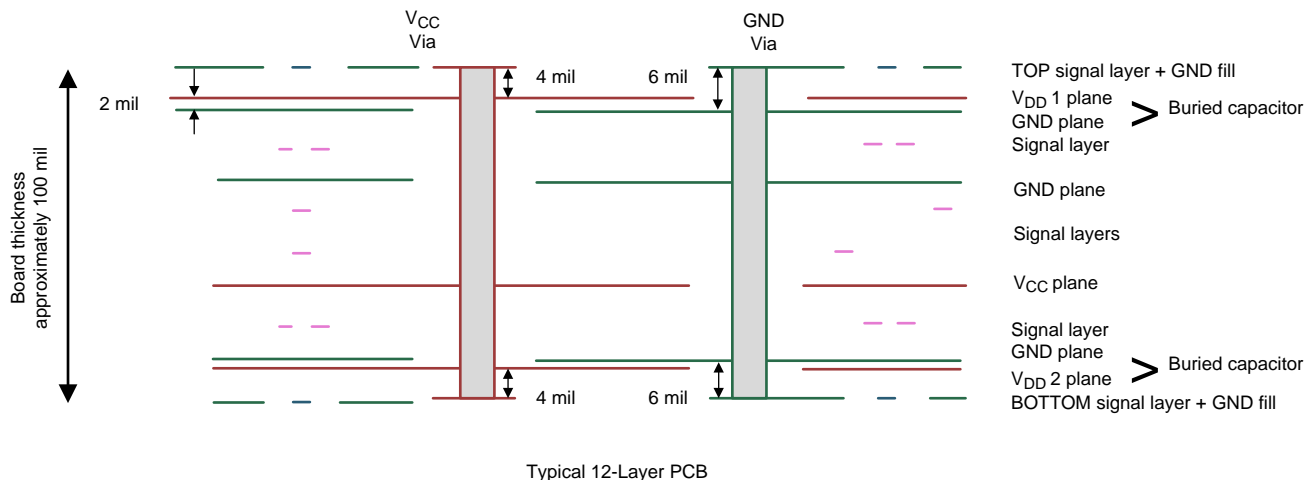


Figure 28. Low Inductance, High-Capacitance Power Connection

Layout Guidelines (continued)

Bypass capacitors should be placed close to V_{DD} pins. They can be placed conveniently near the corners or underneath the package to minimize the loop area. This extends the useful frequency range of the added capacitance. Small-physical-size capacitors, such as 0402 or even 0201, or X7R surface-mount capacitors should be used to minimize body inductance of capacitors. Each bypass capacitor is connected to the power and ground plane through vias tangent to the pads of the capacitor as shown in Figure 29(a).

An X7R surface-mount capacitor of size 0402 has about 0.5 nH of body inductance. At frequencies above 30 MHz or so, X7R capacitors behave as low-impedance inductors. To extend the operating frequency range to a few hundred MHz, an array of different capacitor values like 100 pF, 1 nF, 0.03 μ F, and 0.1 μ F are commonly used in parallel. The most effective bypass capacitor can be built using sandwiched layers of power and ground at a separation of 2 to 3 mils. With a 2-mil FR4 dielectric, there is approximately 500 pF per square inch of PCB. Refer back to Figure 21 for some examples. Many high-speed devices provide a low-inductance GND connection on the backside of the package. This center dap must be connected to a ground plane through an array of vias. The via array reduces the effective inductance to ground and enhances the thermal performance of the small Surface Mount Technology (SMT) package. Placing vias around the perimeter of the dap connection ensures proper heat spreading and the lowest possible die temperature. Placing high-performance devices on opposing sides of the PCB using two GND planes (as shown in Figure 21) creates multiple paths for heat transfer. Often thermal PCB issues are the result of one device adding heat to another, resulting in a very high local temperature. Multiple paths for heat transfer minimize this possibility. In many cases the GND dap that is so important for heat dissipation makes the optimal decoupling layout impossible to achieve due to insufficient pad-to-dap spacing as shown in Figure 29(b). When this occurs, placing the decoupling capacitor on the backside of the board keeps the extra inductance to a minimum. It is important to place the V_{DD} via as close to the device pin as possible while still allowing for sufficient solder mask coverage. If the via is left open, solder may flow from the pad and into the via barrel. This will result in a poor solder connection.



Figure 29. Typical Decoupling Capacitor Layouts

At least two or three times the width of an individual trace should separate single-ended traces and differential pairs to minimize the potential for crosstalk. Single-ended traces that run in parallel for less than the wavelength of the rise or fall times usually have negligible crosstalk. Increase the spacing between signal paths for long parallel runs to reduce crosstalk. Boards with limited real estate can benefit from the staggered trace layout, as shown in Figure 30.

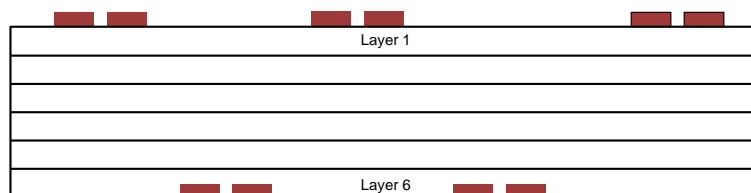


Figure 30. Staggered Trace Layout

This configuration lays out alternating signal traces on different layers. Thus, the horizontal separation between traces can be less than 2 or 3 times the width of individual traces. To ensure continuity in the ground signal path, TI recommends having an adjacent ground via for every signal via, as shown in Figure 31. Note that vias create additional capacitance. For example, a typical via has a lumped capacitance effect of 1/2 pF to 1 pF in FR4.

Layout Guidelines (continued)

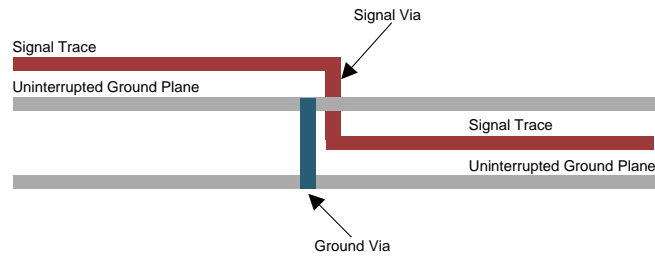


Figure 31. Ground Via Location (Side View)

Short and low-impedance connection of the device ground pins to the PCB ground plane reduces ground bounce. Holes and cutouts in the ground planes can adversely affect current return paths if they create discontinuities that increase returning current loop areas.

To minimize EMI problems, TI recommends avoiding discontinuities below a trace (for example, holes, slits, and so on) and keeping traces as short as possible. Zoning the board wisely by placing all similar functions in the same area, as opposed to mixing them together, helps reduce susceptibility issues.

11.2 Layout Examples

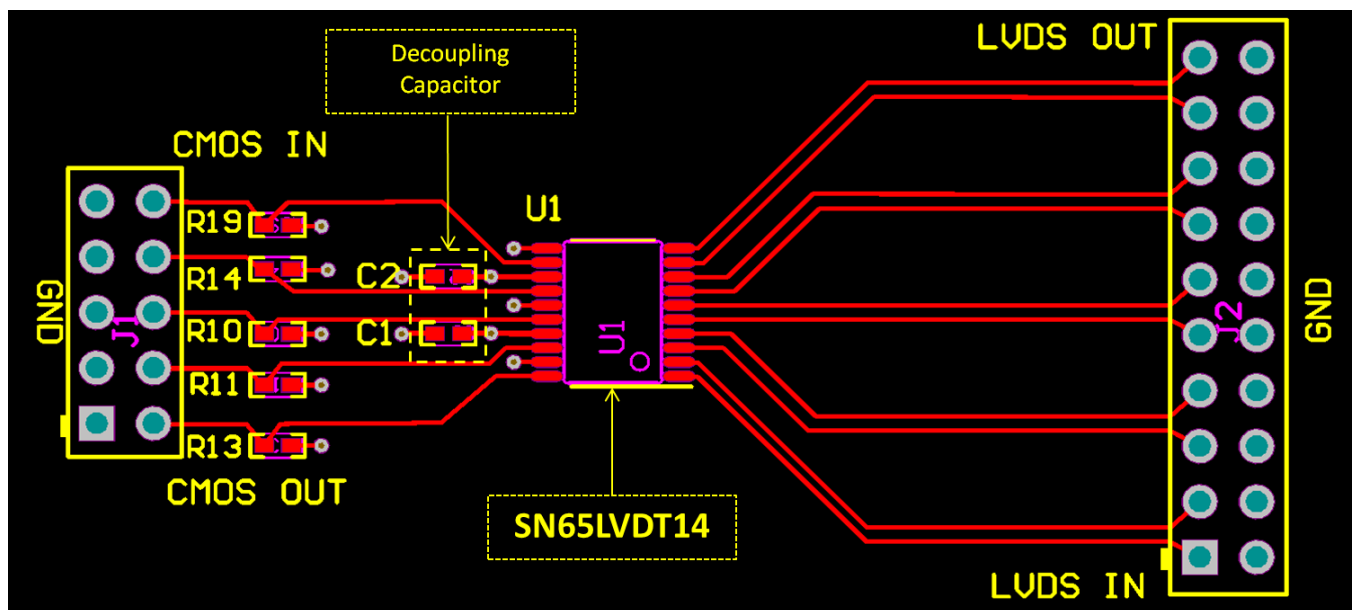


Figure 32. Example SN65LVDT14 Layout

Layout Examples (continued)

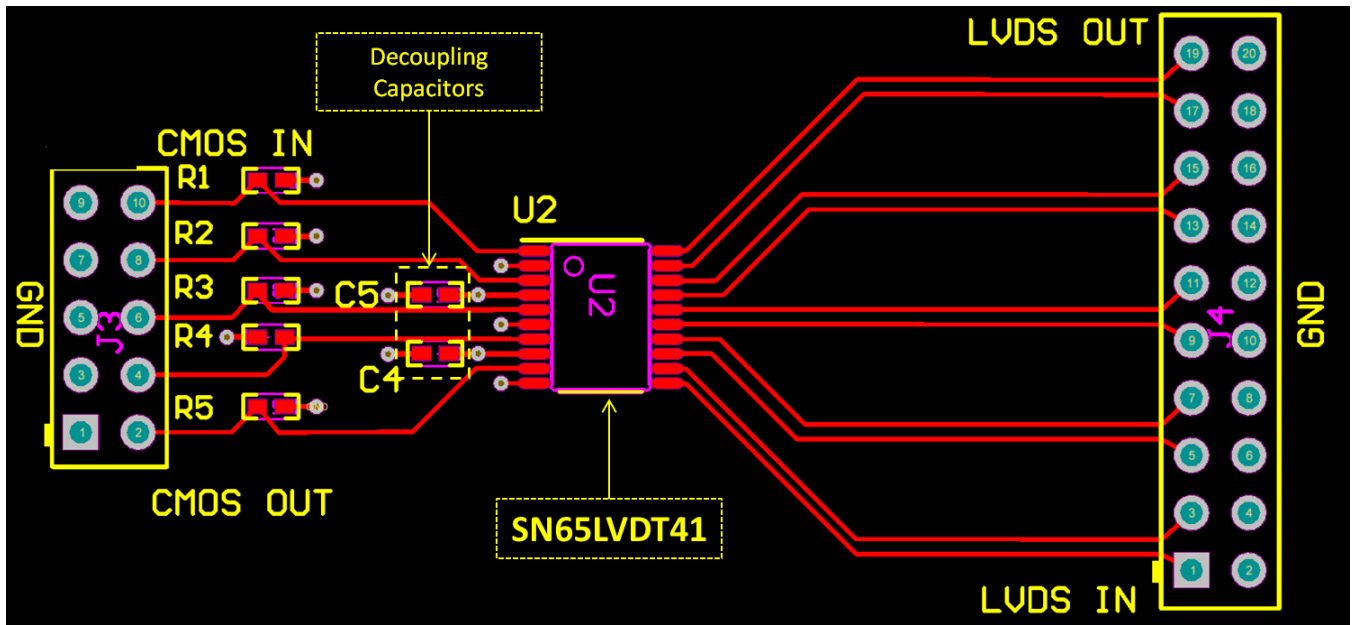


Figure 33. Example SN65LVDT41

12 Device and Documentation Support

12.1 Related Documentation

For related documentation, see the following:

- [LVDS owner's manual](#) (SNLA187)
- [AN-808 Long transmission lines and data signal quality](#) (SNLA028)
- [AN-977 LVDS signal quality: jitter measurements using eye patterns test report #1](#) (SNLA166)
- [AN-971 Overview of LVDS technology](#) (SNLA165)
- [AN-916 Practical guide to cable selection](#) (SNLA219)
- [AN-805 Calculating power dissipation for differential line drivers](#) (SNOA233)
- [AN-903 Comparison of differential termination techniques](#) (SNLA034)
- [AN-1194 Fail-safe biasing of LVDS interfaces](#) (SNLA051)

12.2 Receiving Notification of Documentation Updates

To receive notification of documentation updates, navigate to the device product folder on ti.com. In the upper right corner, click on *Alert me* to register and receive a weekly digest of any product information that has changed. For change details, review the revision history included in any revised document.

12.3 Related Links

The table below lists quick access links. Categories include technical documents, support and community resources, tools and software, and quick access to order now.

Table 6. Related Links

PARTS	PRODUCT FOLDER	ORDER NOW	TECHNICAL DOCUMENTS	TOOLS & SOFTWARE	SUPPORT & COMMUNITY
SN65LVDT14	Click here	Click here	Click here	Click here	Click here
SN65LVDT41	Click here	Click here	Click here	Click here	Click here

12.4 Community Resources

The following links connect to TI community resources. Linked contents are provided "AS IS" by the respective contributors. They do not constitute TI specifications and do not necessarily reflect TI's views; see TI's [Terms of Use](#).

TI E2E™ Online Community *TI's Engineer-to-Engineer (E2E) Community*. Created to foster collaboration among engineers. At e2e.ti.com, you can ask questions, share knowledge, explore ideas and help solve problems with fellow engineers.

Design Support *TI's Design Support* Quickly find helpful E2E forums along with design support tools and contact information for technical support.

12.5 Trademarks

E2E is a trademark of Texas Instruments.

Serial Peripheral Interface is a trademark of Motorola.

Rogers is a trademark of Rogers Corporation.

All other trademarks are the property of their respective owners.

12.6 Electrostatic Discharge Caution



This integrated circuit can be damaged by ESD. Texas Instruments recommends that all integrated circuits be handled with appropriate precautions. Failure to observe proper handling and installation procedures can cause damage.

ESD damage can range from subtle performance degradation to complete device failure. Precision integrated circuits may be more susceptible to damage because very small parametric changes could cause the device not to meet its published specifications.

12.7 Glossary

[SLYZ022](#) — *TI Glossary*.

This glossary lists and explains terms, acronyms, and definitions.

13 Mechanical, Packaging, and Orderable Information

The following pages include mechanical, packaging, and orderable information. This information is the most current data available for the designated devices. This data is subject to change without notice and revision of this document. For browser-based versions of this data sheet, refer to the left-hand navigation.

PACKAGING INFORMATION

Orderable Device	Status (1)	Package Type	Package Drawing	Pins	Package Qty	Eco Plan (2)	Lead finish/ Ball material (6)	MSL Peak Temp (3)	Op Temp (°C)	Device Marking (4/5)	Samples
SN65LVDT14PW	ACTIVE	TSSOP	PW	20	70	RoHS & Green	NIPDAU	Level-1-260C-UNLIM	-40 to 85	LVDT14	Samples
SN65LVDT14PWG4	ACTIVE	TSSOP	PW	20	70	RoHS & Green	NIPDAU	Level-1-260C-UNLIM	-40 to 85	LVDT14	Samples
SN65LVDT14PWR	ACTIVE	TSSOP	PW	20	2000	RoHS & Green	NIPDAU	Level-1-260C-UNLIM	-40 to 85	LVDT14	Samples
SN65LVDT41PW	ACTIVE	TSSOP	PW	20	70	RoHS & Green	NIPDAU	Level-1-260C-UNLIM	-40 to 85	LVDT41	Samples
SN65LVDT41PWR	ACTIVE	TSSOP	PW	20	2000	RoHS & Green	NIPDAU	Level-1-260C-UNLIM	-40 to 85	LVDT41	Samples

(1) The marketing status values are defined as follows:

ACTIVE: Product device recommended for new designs.

LIFEBUY: TI has announced that the device will be discontinued, and a lifetime-buy period is in effect.

NRND: Not recommended for new designs. Device is in production to support existing customers, but TI does not recommend using this part in a new design.

PREVIEW: Device has been announced but is not in production. Samples may or may not be available.

OBSELETE: TI has discontinued the production of the device.

(2) **RoHS:** TI defines "RoHS" to mean semiconductor products that are compliant with the current EU RoHS requirements for all 10 RoHS substances, including the requirement that RoHS substance do not exceed 0.1% by weight in homogeneous materials. Where designed to be soldered at high temperatures, "RoHS" products are suitable for use in specified lead-free processes. TI may reference these types of products as "Pb-Free".

RoHS Exempt: TI defines "RoHS Exempt" to mean products that contain lead but are compliant with EU RoHS pursuant to a specific EU RoHS exemption.

Green: TI defines "Green" to mean the content of Chlorine (Cl) and Bromine (Br) based flame retardants meet JS709B low halogen requirements of <=1000ppm threshold. Antimony trioxide based flame retardants must also meet the <=1000ppm threshold requirement.

(3) MSL, Peak Temp. - The Moisture Sensitivity Level rating according to the JEDEC industry standard classifications, and peak solder temperature.

(4) There may be additional marking, which relates to the logo, the lot trace code information, or the environmental category on the device.

(5) Multiple Device Markings will be inside parentheses. Only one Device Marking contained in parentheses and separated by a "~" will appear on a device. If a line is indented then it is a continuation of the previous line and the two combined represent the entire Device Marking for that device.

(6) Lead finish/Ball material - Orderable Devices may have multiple material finish options. Finish options are separated by a vertical ruled line. Lead finish/Ball material values may wrap to two lines if the finish value exceeds the maximum column width.

Important Information and Disclaimer:The information provided on this page represents TI's knowledge and belief as of the date that it is provided. TI bases its knowledge and belief on information provided by third parties, and makes no representation or warranty as to the accuracy of such information. Efforts are underway to better integrate information from third parties. TI has taken and continues to take reasonable steps to provide representative and accurate information but may not have conducted destructive testing or chemical analysis on incoming materials and chemicals. TI and TI suppliers consider certain information to be proprietary, and thus CAS numbers and other limited information may not be available for release.

In no event shall TI's liability arising out of such information exceed the total purchase price of the TI part(s) at issue in this document sold by TI to Customer on an annual basis.

OTHER QUALIFIED VERSIONS OF SN65LVDT14, SN65LVDT41 :

- Enhanced Product : [SN65LVDT14-EP](#), [SN65LVDT41-EP](#)

NOTE: Qualified Version Definitions:

- Enhanced Product - Supports Defense, Aerospace and Medical Applications

TAPE AND REEL INFORMATION

QUADRANT ASSIGNMENTS FOR PIN 1 ORIENTATION IN TAPE


*All dimensions are nominal

Device	Package Type	Package Drawing	Pins	SPQ	Reel Diameter (mm)	Reel Width W1 (mm)	A0 (mm)	B0 (mm)	K0 (mm)	P1 (mm)	W (mm)	Pin1 Quadrant
SN65LVDT14PWR	TSSOP	PW	20	2000	330.0	16.4	6.95	7.1	1.6	8.0	16.0	Q1
SN65LVDT41PWR	TSSOP	PW	20	2000	330.0	16.4	6.95	7.1	1.6	8.0	16.0	Q1

TAPE AND REEL BOX DIMENSIONS


*All dimensions are nominal

Device	Package Type	Package Drawing	Pins	SPQ	Length (mm)	Width (mm)	Height (mm)
SN65LVDT14PWR	TSSOP	PW	20	2000	367.0	367.0	38.0
SN65LVDT41PWR	TSSOP	PW	20	2000	356.0	356.0	35.0

TUBE


*All dimensions are nominal

Device	Package Name	Package Type	Pins	SPQ	L (mm)	W (mm)	T (μm)	B (mm)
SN65LVDT14PW	PW	TSSOP	20	70	530	10.2	3600	3.5
SN65LVDT14PW	PW	TSSOP	20	70	530	10.2	3600	3.5
SN65LVDT14PWG4	PW	TSSOP	20	70	530	10.2	3600	3.5
SN65LVDT14PWG4	PW	TSSOP	20	70	530	10.2	3600	3.5
SN65LVDT41PW	PW	TSSOP	20	70	530	10.2	3600	3.5
SN65LVDT41PW	PW	TSSOP	20	70	530	10.2	3600	3.5

EXAMPLE BOARD LAYOUT

PW0020A

TSSOP - 1.2 mm max height

SMALL OUTLINE PACKAGE



LAND PATTERN EXAMPLE
EXPOSED METAL SHOWN
SCALE: 10X



SOLDER MASK DETAILS

4220206/A 02/2017

NOTES: (continued)

- 6. Publication IPC-7351 may have alternate designs.
- 7. Solder mask tolerances between and around signal pads can vary based on board fabrication site.

EXAMPLE STENCIL DESIGN

PW0020A

TSSOP - 1.2 mm max height

SMALL OUTLINE PACKAGE



SOLDER PASTE EXAMPLE
BASED ON 0.125 mm THICK STENCIL
SCALE: 10X

4220206/A 02/2017

NOTES: (continued)

8. Laser cutting apertures with trapezoidal walls and rounded corners may offer better paste release. IPC-7525 may have alternate design recommendations.
9. Board assembly site may have different recommendations for stencil design.

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), 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 will 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](#) or other applicable terms available either on [ti.com](https://www.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.

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

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2024, Texas Instruments Incorporated