GS2989

Design Guide

Version	ECR	Date	Changes and / or Modifications
2	157313	December 2011	Corrections to the sink or source value for input and output pins in Section 4. Adjustments to Figure 2-1, Figure 2-3, Figure 2-4, Figure 2-5.
1	154995	September 2010	Removed “Proprietary & Confidential” from document footer.
0	151405	September 2009	New document.

Contents

Overview

Together with the GS2989 Evaluation Board (EB-GS2989), the EB-GS2989 User Guide and the GS2989 Data Sheet, this document serves as a reference for designing with the GS2989, a Multi-Rate Dual Output Cable Driver. This document contains two main areas of focus:

1. Considerations in the schematic design.
2. Recommended PCB layout practices when designing with the GS2989.

Figure A below shows a typical GS2989 application block diagram.

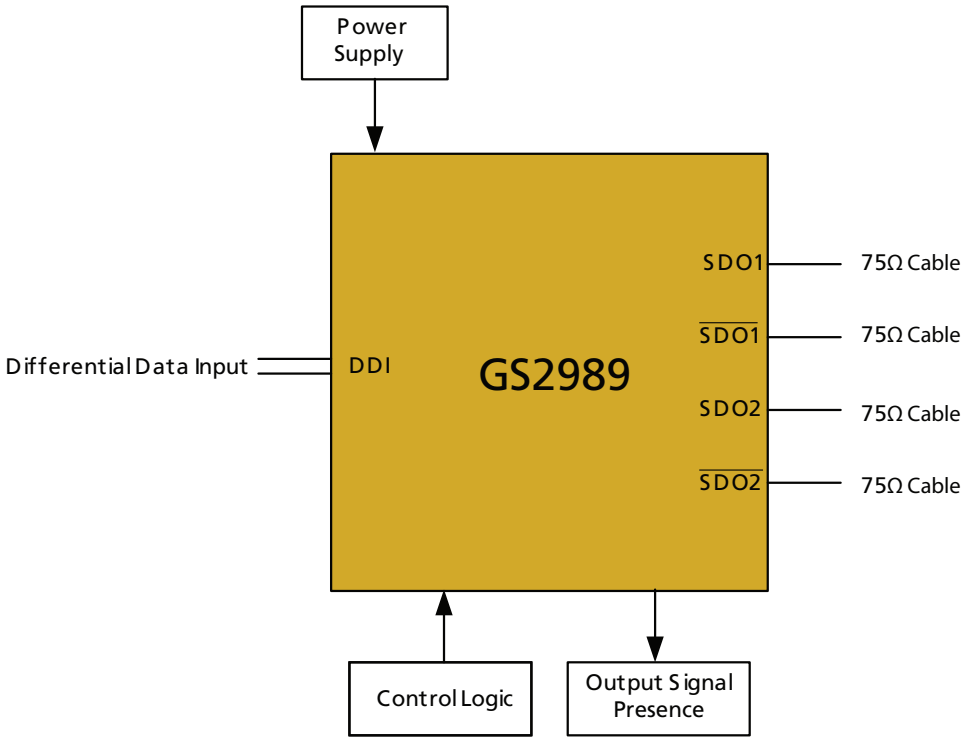

Figure A: Typical Application of the GS2989

1. Power

The GS2989 requires a single power supply at either 3.3V or 2.5V, provided a high output swing is not required. If a high output swing is required, the termination voltage should be capable of being connected to 5V, while the supply of the device remains at 3.3V or 2.5V.

The following guidelines are recommended when designing power for the GS2989:

- Use coupled power and ground planes i.e. use minimum spacing between power and ground planes. Power and ground planes form a natural capacitor, which will increase capacitance
- Do not overlap power planes. If it is unavoidable, different power planes should be isolated from each other with a ground plane between them
- Power and ground planes should be placed near the component side, which will reduce inductance of vias
- Bulk supply decoupling capacitors of 10µF and 100nF are recommended to filter-out high frequency noise
- It is recommended to decouple each power supply pin individually with a 10nF capacitor. All decoupling capacitors should be placed as close as possible to the power and ground pin pairs of the GS2989. Decoupling capacitors for the termination supply should be placed on the same side of the board as the GS2989 and as close as possible to the termination resistors. Wherever possible, use two vias for each VCC and GND pin and flood the copper area
- Central paddle should be connected to ground with multiple vias for proper thermal dissipation. At least 5 vias are recommended

2. SDI Input and Output

The GS2989 takes in 100Ω differential input, and is capable of driving up to four 75Ω single ended co-axial cables. The GS2989 can drive data rates up to 2.97Gb/s, and provides two selectable slew rates in order to achieve compliance to SMPTE 424M, SMPTE 292M and SMPTE 259M.

The following general guidelines are recommended when designing the transmission lines for the GS2989:

- Input and output trace lengths should be minimized and the input differential traces should be closely matched, and curved rather than sharp-angled
- Any discontinuity (ie. impedance transitions) should be avoided on the transmission lines
- No silk-screen should be allowed on these traces to ensure an accurate impedance
- 4.7µF ceramic AC-coupling capacitors are recommended for the input and output transmission lines as shown in the schematic

- Input transmission lines should have 100Ω differential controlled impedances, while output transmission lines should have 75Ω single-ended controlled impedances
- Non-transmission line components should be at a minimum distance of 10x the transmission line width (example: if transmission line is 1mm wide, the closest non transmission line component needs to be at least 10mm away)
- Antipads are recommended to deal with transmission line component parasitics. The antipad sizes used on the evaluation board are given in Figure 2-1, Figure 2-2, Figure 2-3, Figure 2-4 and Figure 2-5. These antipad sizes are specific to this design, which uses ISOLA FR-410 dielectric with a 10mil thickness between the top routing layer and the associated GND layer. NOTE: Antipads are defined as the absence of copper in the underlying GND and PWR plane layers. As well, there should be no routing on any routing layer underneath the antipads

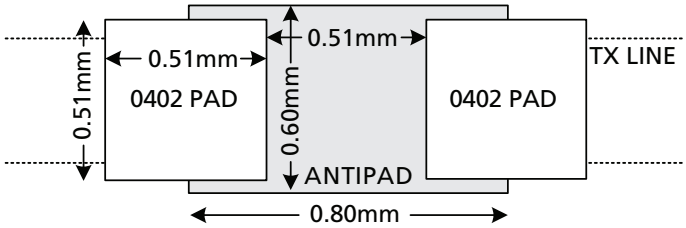

Figure 2-1: Series 0402 Component Antipad optimized for 50Ω traces

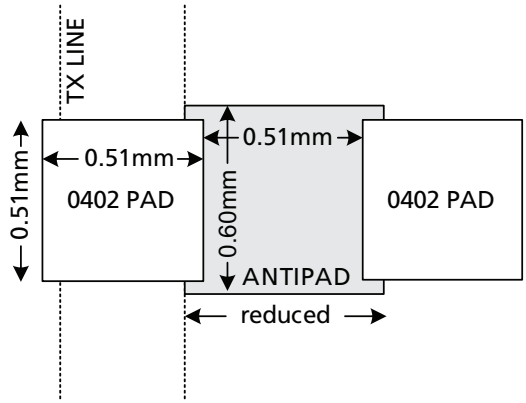

Figure 2-2: Shunt 0402 Component Antipad optimized for 50Ω and 75Ω traces

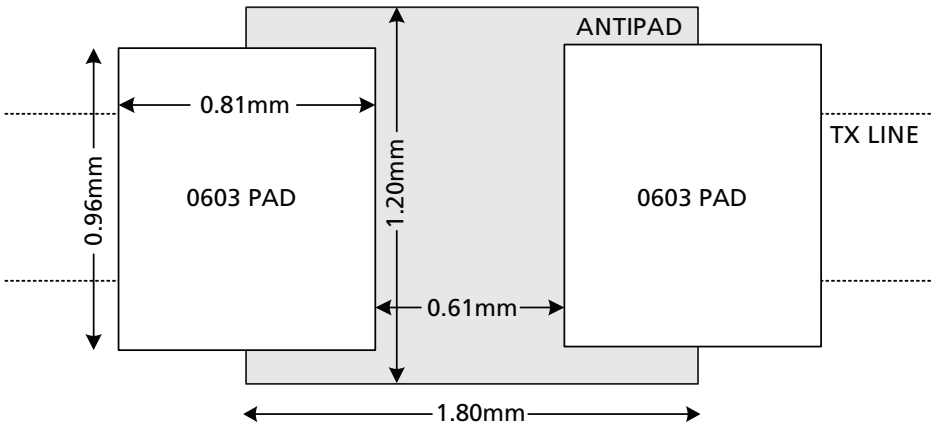

Figure 2-3: Series 0603 Component Antipad optimized for 50Ω traces

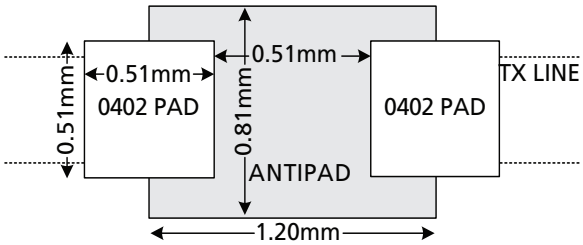

Figure 2-4: Series 0402 Component Antipad optimized for 75Ω traces

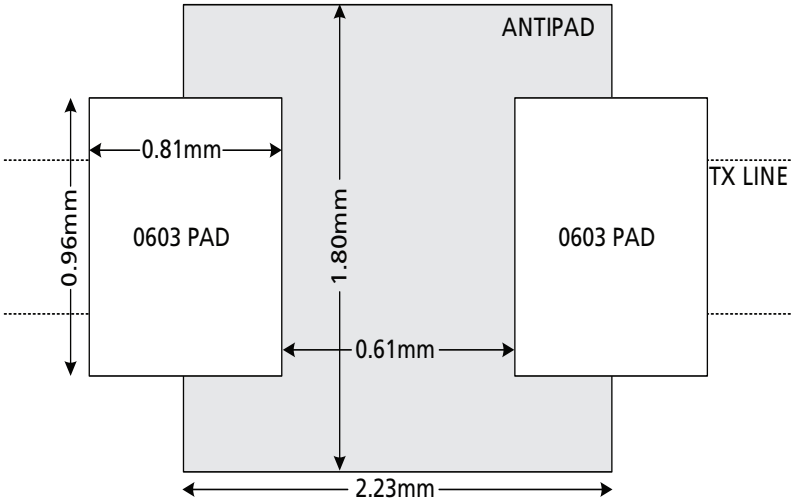

Figure 2-5: Series 0603 Component Antipad optimized for 75Ω traces

3. ORL Design Recommendations

This section contains recommendations that should be followed to obtain good Output Return Loss (ORL) when designing with the GS2989.

3.1 Termination Resistor Placement

It is recommended to place the termination resistors 4-5mm away from the device pins. The distance to the termination resistor will change the way the ORL compensation network is designed.

Generally, placing the termination resistor closer to the device improves the ORL and has the same effect as increasing the ORL network inductor value.

Placing the termination resistor farther away from the device improves the eye shape and has the same effect as decreasing the ORL network inductor value as shown in Figure 3-1.

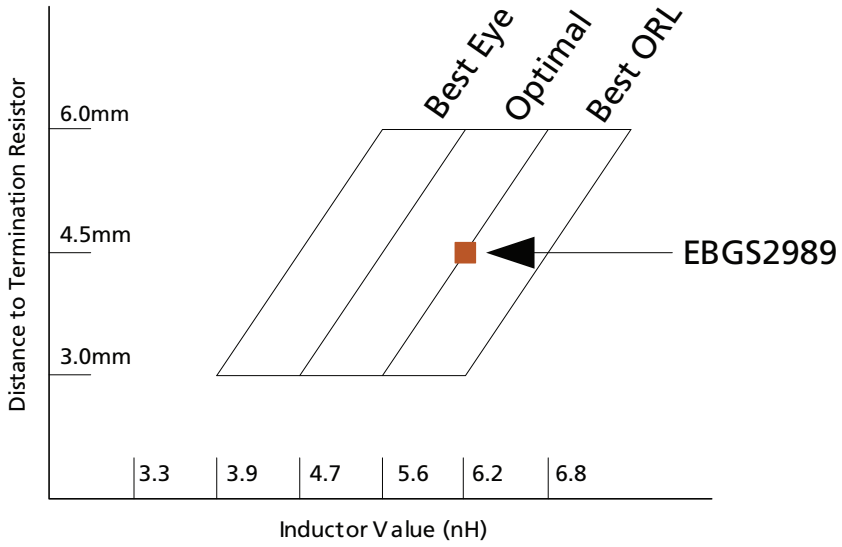

Figure 3-1: Termination Resistor Placement

To obtain a good trade-off between ORL and eye quality, the termination resistors should be placed no closer than 2mm away and no farther than 6mm away from the device. The evaluation board design uses a distance of 4.5mm.

3.2 ORL Compensation Network

The ORL compensation network should consist of a 75Ω resistor in parallel with an inductor. The optimum inductor value will depend on how far the termination resistor is placed from the device. See Figure 3-1.

The type of inductor used in the ORL compensation network is important for good performance. For best performance, select a chip inductor with good tolerance and high self resonant frequency, such as the LQG15HS series Murata inductors.

3.3 AC-coupling Capacitor

It is recommended to use a 4.7µF ceramic capacitor to AC-couple the outputs with a voltage rating of 10V or better. The pad size should closely match the transmission line width to avoid impedance discontinuities. It is also recommended to use antipads (absence of copper in the underlying ground and power planes) to compensate for the parasitic effects of the capacitor package.

3.4 Termination Voltage Decoupling

The termination voltage should be decoupled with 10nF ceramic capacitors. Decoupling capacitors should be placed on the same side and as close as possible to the termination resistors.

3.5 BNC Edge Connector Footprint

It is important to have a good BNC edge launch. The BNC footprint central pin should be as close as possible to the transmission line width and antipads should be used. Please refer to Figure 3-2 for details on the footprint used in the evaluation board.

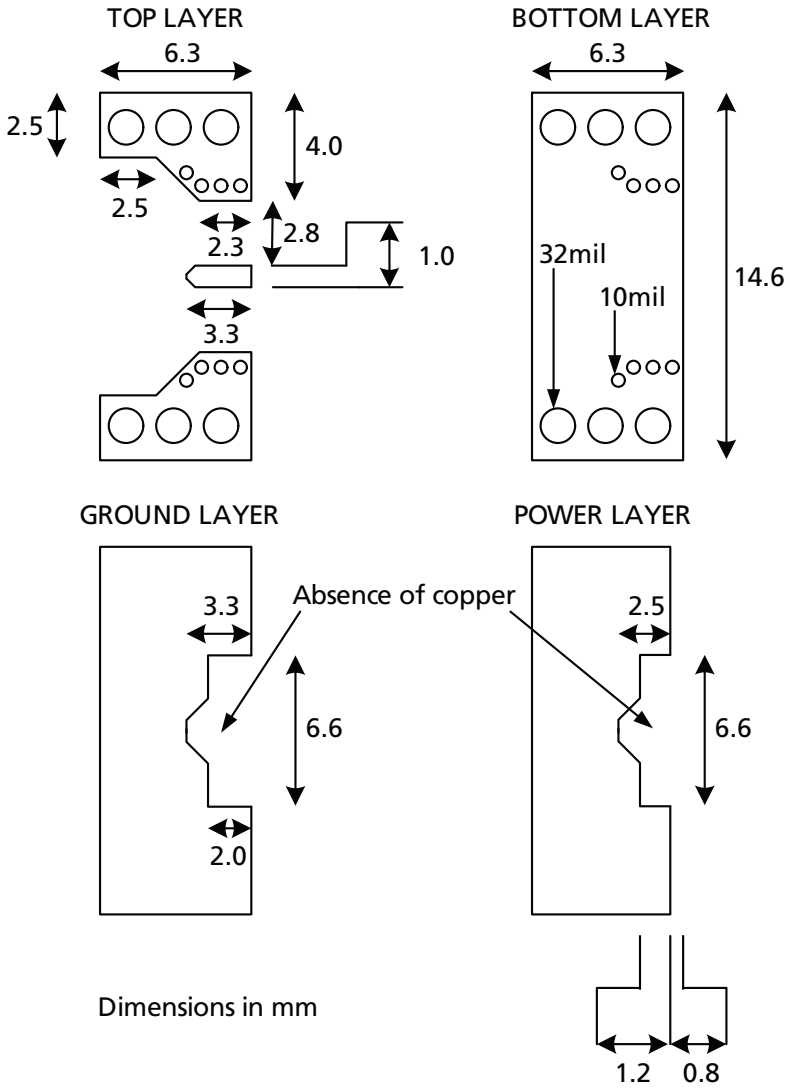

Figure 3-2: BNC Footprint

4. Other Considerations

- Input and output pins can only sink or source 2mA. Therefore, the $\overline{\text{OSP}}$ output pin should not be used to drive LEDs directly, LED drivers should be used instead.
- For good performance, the RSET resistor should be placed as close as possible to the RSET pin.

DOCUMENT IDENTIFICATION
DESIGN GUIDE

Information relating to this product and the application or design described herein is believed to be reliable, however such information is provided as a guide only and Gennum assumes no liability for any errors in this document, or for the application or design described herein. Gennum reserves the right to make changes to the product or this document at any time without notice.

CAUTION

ELECTROSTATIC SENSITIVE DEVICES

DO NOT OPEN PACKAGES OR HANDLE EXCEPT AT A STATIC-FREE WORKSTATION

GENNUM CORPORATE HEADQUARTERS
4281 Harvester Road, Burlington, Ontario L7L 5M4 Canada
Phone: +1 (905) 632-2996
E-mail: corporate@gennum.com
Fax: +1 (905) 632-2055
www.gennum.com

CANADA
Suite 320, 3553 31st St. N.W.
Calgary, Alberta T2L 2K7
Canada
Phone: +1 (403) 284-2672
Fax: +1 (905) 632-2055

415 Legget Drive, Suite 200
Kanata, Ontario K2K 2B2
Canada
Phone: +1 (613) 270-0458
Fax: +1 (613) 270-0429

GERMANY
Gennum Canada Limited
Niederlassung Deutschland
München, Germany
Phone: +49 89 309040 290
Fax: +49 89 309040 293
E-mail: gennum-germany@gennum.com

INDIA
#208(A), Nirmala Plaza,
Airport Road, Forest Park Square
Bhubaneswar 751009
India
Phone: +91 (674) 65304815
Fax: +91 (674) 259-5733

JAPAN KK
Shinjuku Green Tower Building 27F
6-14-1, Nishi Shinjuku
Shinjuku-ku, Tokyo, 160-0023
Japan
Phone: +81 (03) 3349-5501
Fax: +81 (03) 3349-5505
E-mail: gennum-japan@gennum.com
Web Site: http://www.gennum.co.jp

MEXICO
Venustiano Carranza 122 Int. 1
Centro, Aguascalientes
Mexico CP 20000
Phone: +1 (416) 848-0328

NORTH AMERICA WESTERN REGION
691 South Milpitas Blvd., Suite #200
Milpitas, CA 95035
United States
Phone: +1 (408) 934-1301
Fax: +1 (408) 934-1029
E-mail: naw_sales@gennum.com

NORTH AMERICA EASTERN REGION
4281 Harvester Road
Burlington, Ontario L7L 5M4
Canada
Phone: +1 (905) 632-2996
Fax: +1 (905) 632-2055
E-mail: nae_sales@gennum.com

TAIWAN
6F-4, No.51, Sec.2, Keelung Rd.
Sinyi District, Taipei City 11502
Taiwan R.O.C.
Phone: (886) 2-8732-8879
Fax: (886) 2-8732-8870
E-mail: gennum-taiwan@gennum.com

UNITED KINGDOM
South Building, Walden Court
Parsonage Lane,
Bishop's Stortford Hertfordshire, CM23 5DB
United Kingdom
Phone: +44 1279 714170
Fax: +44 1279 714171

2, West Point Court, Great Park Road
Bradley Stoke, Bristol BS32 4PY
Great Britain
Phone: +44 1454 462200
Fax: +44 1454 462201

SNOWBUSH IP - A DIVISION OF GENNUM
439 University Ave. Suite 1700
Toronto, Ontario M5G 1Y8
Canada
Phone: +1 (416) 925-5643
Fax: +1 (416) 925-0581
E-mail: sales@snowbush.com
Web Site: http://www.snowbush.com