Guide to CRT Video Design

INTRODUCTION

This application note provides a design guide for successfully designing high frequency CRT video boards. For better illustration, an example of a complete video board design using the LM1203N RGB preamplifier and the LM2419 RGB CRT driver is provided. The design includes: DC restoration, contrast control, brightness control, cutoff adjustment, delta gain adjustment (for white balance) and blanking at grid G1. The complete circuitry for the video board is shown in *Figure 13*. *Figure 1* shows the pulse response at the cathode for a 45 V_{PP} output signal. Rise and Fall times at the cathode were measured at 6 ns and 7.5 ns respectively and settling time (to within ±5% of final value) was measured at 20 ns. The overshoot and undershoot were measured at 5V. An NEC-5D multi-sync monitor was used to evaluate the video board, the performance of the video board was very good at 1024 x 1024 display resolution. Various sections of *Figure 13's* circuit are described below in detail.

1.0 RGB PREAMPLIFIER (LM1203)

The LM1203 is a wideband video amplifier system specifically designed for RGB CRT monitor applications (Reference 1—LM1203 Data Sheet). The device includes three matched video amplifiers, three matched attenuator circuits for contrast control, and three gated differential input clamp comparators for black level clamping of the video signal. In addition, each video amplifier includes a gain adjustment or "drive" pin for individual gain adjustment of each video channel to allow white balance adjustment.

1.1 RGB Video Signals

The RGB video signals are AC coupled to the inputs of the LM1203 preamplifier. As shown for the Green channel (see *Figure 13*), C1 AC couples the video signal and R8 references the signal to 2.4 V_{DC} reference voltage from pin 11. The 75Ω resistor, R1, is a termination resistor whose value matches the characteristic impedance of the 75Ω coaxial cable. Note that if a 50Ω video generator is used with 50Ω coaxial cables to test the video board then 50Ω termination resistors should be used.

In the absence of R2, the stray input capacitance of the LM1203 would effectively short R1 at high frequencies causing reflections. The 33Ω resistor, R2 maintains reasonable termination at high frequencies thereby minimizing reflections. Note that the value of R2 should not be much larger than 33Ω otherwise the rise and fall times of the output signal would be degraded.

1.2 Gain Adjustment and Black Level Clamping

Potentiometers R16 and R25 allow the user to adjust the gain of the Blue and Green channels respectively for achieving correct white balance. The gain of the Red channel is fixed by resistor R20. Once white balance is achieved, the contrast control potentiometer R11A allows the user to adjust the gain of all three channels simultaneously.

The black level control potentiometer, R27, allows the user to clamp the black level of the video signals to the desired level. Potentiometer R27 should be adjusted such that the video signals at the output of LM2419 (CRT driver) are biased within LM2419's linear operating region. To accomplish black level clamping, however, a back porch clamp signal must be applied to the clamp gate input (pin 14) of LM1203. The MM74HC86 quad exclusive-or gate generates the required back porch clamp signal (see section 4.0).

1.3 Preamplifier Gain Peaking for Improved Rise and Fall Times

Connecting a small capacitor from LM1203's drive pins (pins 18, 22 and 27) to ground peaks the amplifier's high frequency gain and increases the −3 dB frequency. Using 18 pF peaking capacitors (C100, C101 and C102 in *Figure 13*), LM1203's bandwidth is 70 MHz and rise/fall times under 7 ns. If the LM1203 is used to directly drive the LM2419 without the buffer transistors Q1, Q2 and Q3 then 33 pF peaking capacitors should be used. Refer to the LM1203 data sheet for information on frequency response using various peaking capacitor values. To minimize overshoot, the peaking capacitor should be less than 60 pF.

1.4 Buffered Output

Some CRT drivers have large input capacitance which makes it difficult for the preamplifier to directly drive the CRT driver and yet maintain the required bandwidth. In such applications, a buffer transistor is used between the preamplifier and the CRT driver (for example, transistor Q1 for the Green channel in *Figure 13*). The buffer transistor used should have a high f_T at high currents. The LM1203 can directly drive the LM2419 without sacrifice in bandwidth. However, buffer transistors have been used in our design to illustrate a complete design. The overall response of *Figure 13's* circuit is similar with or without the buffer transistors. A 2N5770 transistor was selected which has a minimum f_T of 900 MHz. Note that for fast fall time, the emitter resistor of the buffer transistor should not be much larger than 330Ω.
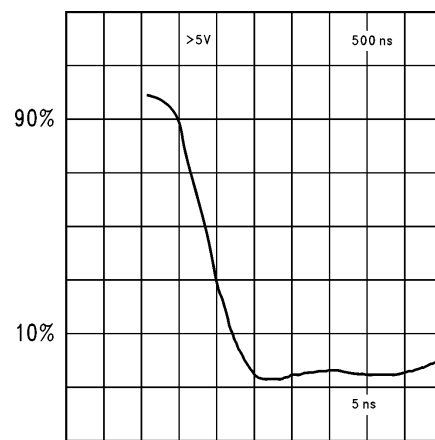
(a)

TL/H/11733–1

(b)

TL/H/11733–2

(c)
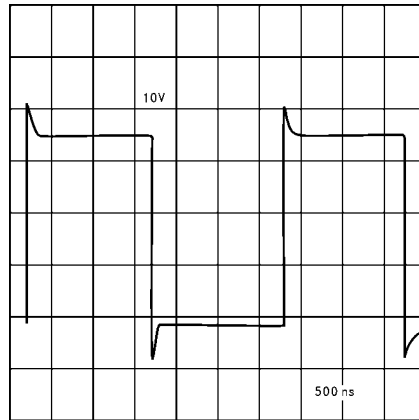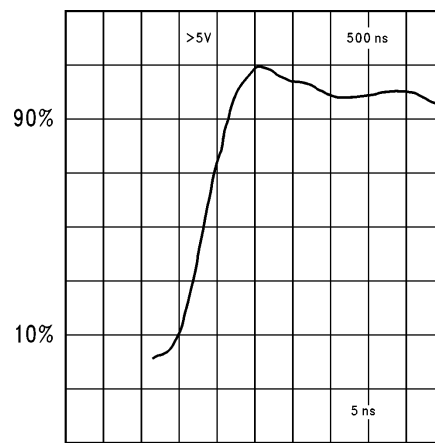
TL/H/11733–3

FIGURE 1. Pulse Response at the Cathode for *Figure 13* 's Circuit

2.0 RGB CRT DRIVER (LM2419)

The LM2419 is a high-voltage, wide-bandwidth amplifier that drives the CRT's cathodes (Reference 2—LM2419 Data Sheet). The outputs of the LM2419 are AC coupled to the cathodes (see *Figure 13*) so that cutoff voltages greater than 80V can be accommodated at the cathodes. Furthermore, large variation in cutoff voltages can be accommodated because of the 120V supply. With the video signal AC coupled to the cathode, the signal's DC information is lost. To restore the DC information of the video signal, the video signal's black level is clamped at the cathode.

2.1 Black Level Clamping (DC Restoration)

Figure 2 shows the black level clamping circuitry for the Green channel. Capacitor C25 AC couples the video signal to the cathode. Transistor Q4 and diode D8 clamp the signal's black level to a voltage two diode drops greater than the voltage at the base of Q4. Adjusting the voltage at the base of Q4 using potentiometer R44 adjusts the clamp voltage thereby providing cutoff adjustment. Note that if the video signal has a sync tip then the clamp circuit will clamp the sync tip to the clamp voltage. The transistor selected for Q4 should have a BVCEO rating greater than 120V and low junction capacitance. For our design, an MPSA92 PNP transistor was selected. Diode D7 is used to protect transistor Q4 from an arc-over. D7 clamps Q4's emitter to a diode drop above 120V and provides a low impedance path for the arc-over current to flow through D8 and D7 to the 120V supply. The diodes used should have a high current rating, low series impedance and low shunt capacitance. An FDH400 diode is recommended.

2.2 Arc Protection

The CRT driver must be protected from arcing within the CRT. To limit the arc-over voltage, a 200V spark gap should be used at each cathode. Diodes D1 and D2 (see *Figure 3*) clamp the voltage at the output of LM2419 to a safe level. The clamp diodes used should have a high current rating, low series impedance and low shunt capacitance. FDH400 or equivalent diodes are recommended. Resistor R54 in *Figure 3* limits the arc-over current while R33 limits the current into the CRT driver. Limiting the current into the CRT driver limits the power dissipation of the output transistors when the output is stressed beyond the supply voltage. The resistor values for R33 and R54 should be large enough to provide optimum arc protection but not too large that the amplifier's bandwidth is adversely affected.
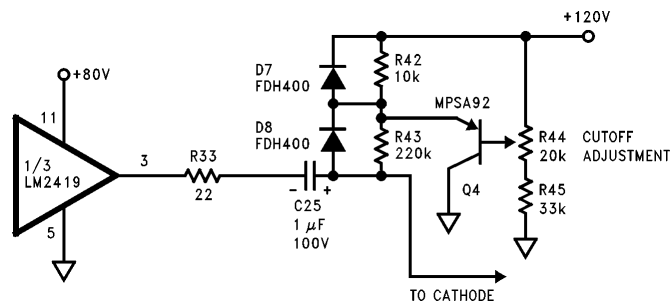
Grids G1 and G2 should also have spark gaps. A 300V and a 1 kV spark gap are recommended for G1 and G2 respectively. The PC board should have separate circuit ground and CRT ground. The board's CRT ground is connected to the CRT's ground pin and also directly connected to the chassis ground. The spark gap's ground return should be to the CRT ground so that high arc-over ground does not directly flow through the circuit ground and damage sensitive circuitry. At some point on the PC board, the circuit ground and the CRT ground should be connected. Often a small resistor is connected between the two grounds to isolate them (see *Figure 3*).

2.3 Overshoot Compensation

LM2419's overshoot is a function of both the input signal rise and fall times and the capacitive loading. The overshoot is increased by either more capacitive loading or faster rise and fall times of the input signal. The circuitry to reduce overshoot is shown in *Figure 4*. Without the compensation circuit (i.e. without R2 and C1) the overshoot and undershoot for the PC board of *Figure 10* were measured at 10% and 15% respectively ($V_{OUT} = 40 V_{P-P}$). With the compensation circuit in place (i.e. with R1 = 24Ω, R2 = 3.9 kΩ and C1 = 3 pF), overshoot and undershoot were reduced to 0% and 3.8% respectively. Inclusion of the compensation circuitry caused the rise and fall times to increase by 1 ns.

The values for the compensation circuit will depend on PC-board layout and LM2419 loading. Here's how to select the correct component values for the compensation circuit shown in *Figure 4*:

(a) R1 and R2 reduce the gain of the CRT driver at high frequencies thereby reducing overshoot.

(b) C1 determines the frequency at which gain is reduced and introduces a time constant in the pulse response.

(c) The time constant, $\tau = R2 \times C1$ should be less than 20 ns. Capacitor C1 should be selected to be 3 pF or slightly larger so as to eliminate the effect of stray capacitance. If C1 is too large such that $\tau \gg 20$ ns then the pulse response will be damped, causing long rise and fall times and therefore picture smearing.

(d) Making R2 too large will cause a damped pulse response, giving rise to picture smearing. If there is a need to change the high frequency gain to adjust the level of overshoot then the value of R1 should be changed since this will not affect the time constant.

FIGURE 2. Black Level Clamp Circuit for DC Restoration

TL/H/11733–4

(e) With the value of R2 fixed, increasing the value of R1 will decrease the high frequency gain and therefore reduce the amplitude of the overshoot. Conversely, decreasing the value of R1 will increase the high frequency gain and therefore increase the amplitude of the overshoot.

(f) Suggested initial starting values for *Figure 4*'s circuit are: R1 = 24Ω to 100Ω, C1 = 3 pF to 6 pF, R2 × C1 < 20 ns.

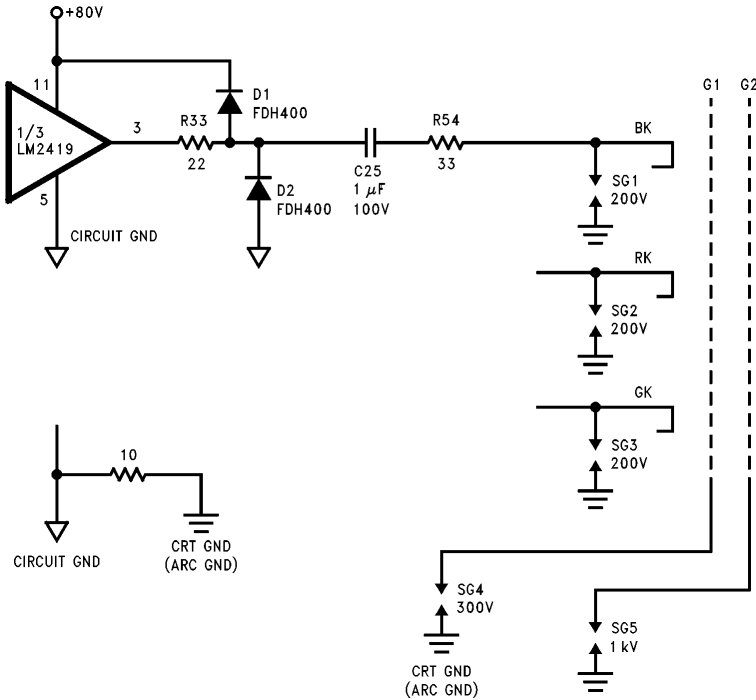*Figure 5* shows the pulse response at the output of LM2419, with and without compensation.
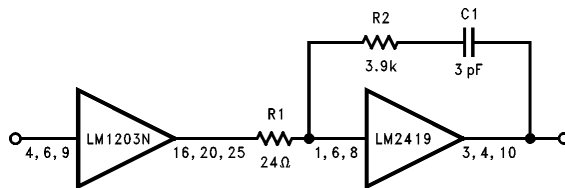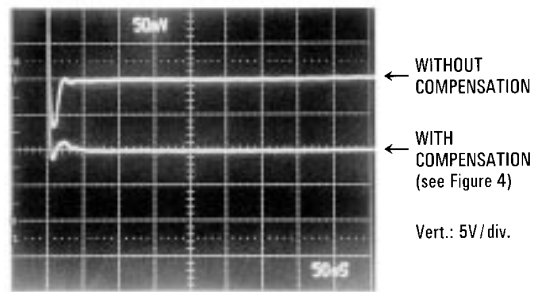

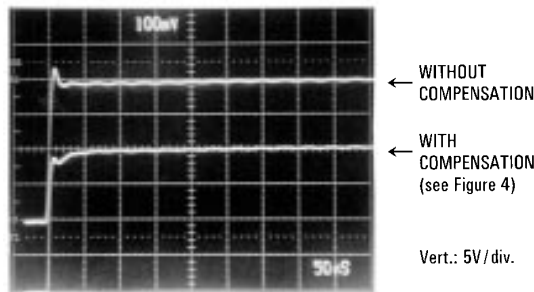FIGURE 3. Arc Protection Circuit

TL/H/11733–5

TL/H/11733–15

FIGURE 4. Overshoot Compensation Circuit for LM2419

4

WITHOUT
COMPENSATION

← WITH
COMPENSATION
(see Figure 4)

Vert.: 5V/div.

TL/H/11733–16

(a)

← WITHOUT
COMPENSATION

← WITH
COMPENSATION
(see Figure 4)

Vert.: 5V/div.

TL/H/11733–17

(b)

FIGURE 5. Pulse Response with and without Overshoot Compensation

2.4 Suppressing Oscillation

As is the case with all wideband amplifiers, PC-board layout precautions must be taken to prevent oscillation. Experimentation had shown that when the LM2419's output was probed with a passive probe (100:1, 5 kΩ probe) connected to a 50Ω oscilloscope, the LM2419's output burst into oscillation. However, when probed with a high input impedance (1 MΩ) FET probe the oscillation was not present. Further investigation showed that the oscillation was caused by: the inductance of the trace connected between the LM1203's output and the LM2419's input; type of loading—in this case, the complex impedance of the 5 kΩ passive probe; and high frequency channel-to-channel cross-talk internal to LM2419. The oscillation can be eliminated by applying the following guidelines:

(a) Minimize trace length between the preamplifier output and LM2419 input. If long trace length is unavoidable then use low value (22Ω for example) series resistors with short lead length in the signal path to damp the inductance of the trace. Use carbon resistors and avoid using wire wound resistors because they are inductive.

5

(b) Avoid long wires to connect the LM2419 output to CRT socket. Inductance of the wire can cause ringing and possible oscillation depending on the characteristics of the complex load. Use a low value resistor (50Ω for example) in series with the output of LM2419 to damp the wire's inductance. Since the input impedance of the LM2419 is high, the device is susceptible to high frequency cross-talk. If oscillation still persists then lowering the input impedance of LM2419 by connecting a 1 kΩ resistor from LM2419's input to ground solves the problem (R500, R501 and R502 in *Figure 13*). The 1 kΩ value is empirically determined, a lower value may be required depending on the PC-board layout and load characteristics.

Two 50Ω series resistors between LM1203 output and LM2419 input were required to damp the inductance of the signal trace. Note that in this application, the LM1203 was used to directly drive the LM2419 without the use of buffer transistors. When the series resistors were changed from carbon to wirewound, the oscillation reappeared because of the inductive characteristics of the wirewound resistors.

Oscillations can often occur due to ground loop currents. Having separate power and low voltage ground planes will help. Refer to Section 7.2 for further details.

2.5 Improving Rise and Fall Times

Because of an emitter follower output stage, the rise and fall times of the LM2419 are relatively unaffected by capacitive loading. However, the series resistors (R33 and R54 for the Green channel, see *Figure 13*) will increase the rise and fall times when driving the CRT's cathode which appears as a capacitive load. The capacitance at the cathode typically ranges from 8 pF to 12 pF and every effort is made to minimize this capacitance.

To improve the rise and fall times at the cathode, a small inductor is often placed in series with the output of the amplifier. The series peaking inductor peaks the amplifier's frequency response at the cathode thus improving the rise and fall times. The value of the inductor is empirically determined. An inductor value of 50 nH is a good initial starting value. Note that peaking the amplifier's frequency response will increase the overshoot. Therefore the value of the inductor selected is a compromise between optimum rise and fall times and acceptable overshoot.

At low output voltage swing (for ex: $V_{OUT} < 5$ $V_{P\text{-}P}$), the rise and fall times may degrade by as much as 50% or more. This is caused by the fact that LM2419 has a class "B" output stage with a 600 mV dead band, thus giving rise to cross-over distortion. Increased rise and fall times may give rise to picture smearing at low contrast settings. Connecting a 20 kΩ (½W) resistor from LM2419's output to ground biases the output stage in class "A" mode thus maintaining similar rise/fall times at both small and large output voltage swing.

2.6 Short Circuit Protection

The output of the LM2419 is not short circuit protected. Shorting the output to either ground or to V_+ will destroy the device. The minimum DC load resistance the LM2419 can drive without damage is 1.6 kΩ to ground or to V_+. However, driving a 1.6 kΩ load for an extended period of time is not recommended because of power dissipation considerations. If the LM2419 is used to drive a resistive load then the load should be 10 kΩ or greater.

3.0 BLANKING AT GRID G1

The circuit used to accomplish blanking is shown in *Figure 6.* A negative voltage is applied to grid G1 using the resistor divider comprised of R58 and R60. Brightness control is achieved by varying the bias at G1 using potentiometer R60. Blanking at the grid is accomplished by R59, R62, D13 and C34.
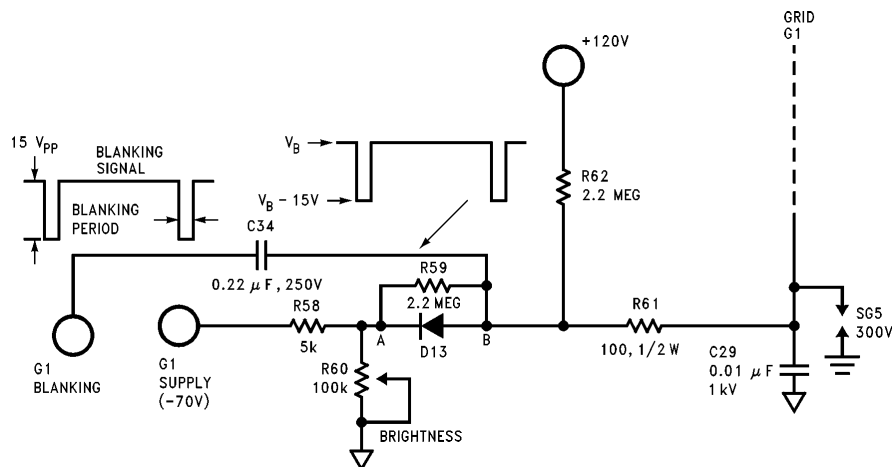

TL/H/11733–6

FIGURE 6. Circuitry for Blanking at the Grid

Resistor R62 biases the clamp diode D13. A 15 V_{PP} blanking signal is AC coupled through C34. Since the voltage at node "B" can not go more than one diode drop above the voltage at node "A", the blanking signal at G1 is clamped at the G1 bias voltage.

During the blanking portion of the video signal, the blanking signal goes low thus reverse biasing D13 and pulling G1 15V negative with respect to its normal bias voltage. This action cuts off the CRT's beam current during the blanking interval and accomplishes blanking.

4.0 BACK PORCH CLAMP GENERATOR

A versatile backporch clamp generator circuit is shown in *Figure 7.* A quad exclusive-or gate (MM74HC86) is used to generate the back porch clamp signal from the composite H-Sync input signal. The composite H-Sync input signal may have either positive or negative polarity. The logic level at pin 11 (Flag Out) indicates the polarity of the H-Sync signal applied to the clamp generator. The Flag output is a logic low (less than 0.8V) if the H-Sync input signal has a negative polarity and is a logic high (greater than 2.4V) if the H-Sync input signal has a positive polarity.

Regardless of the H-Sync input signal's polarity, a negative polarity H-Sync signal is output at pin 8. Furthermore, a negative polarity back porch clamp pulse is output at pin 3. The width of the back porch clamp pulse is determined by the time constant due to R28 and C12. For fast horizontal scan rates, the back porch clamp pulse width can be made narrower by decreasing the value of R28 or C12 or both. Note that an MM74C86 exclusive-or gate may also be used, however, the pin out is different than that of the MM74HC86.

5.0 THERMAL CONSIDERATIONS

The LM1203 preamplifier does not require a heat sink. However, the LM2419 requires a heat sink under all operating conditions. For the LM2419, the worst case power dissipation occurs when a white screen is displayed on the CRT. Considering a 20% black retrace time in a 1024 x 768 display resolution application, the average power dissipation for continuous white screen is less than 4W per channel with 50 V_{PP} output signal (black level at 75V and white level at 25V). Although the total power dissipation is typically 12W for a continous white screen, the heat sink should be selected for 13W power dissipation because of the variation in power dissipation from part to part.

For thermal and gain linearity considerations, the output low voltage (white level) should be maintained above 20V. If the device is operated at an output low voltage below 20V, the power dissipation might exceed 4.7W per channel (i.e., 14W power dissipation for the device). Note that the device can be operated at lower power by reducing the peak-to-peak video output voltage to less than 50V and keeping the clamped video black level close to the supply voltage.

Maximum ratings require that the device case temperature be limited to 90°C maximum. Thus for 50°C maximum ambient temperature and 13W maximum power dissipation, the thermal resistance of the heat sink should be:

$$\theta_{SA} \le (90 - 50)°C/13W = 3°C/W$$

5.1 Designing the Proper Heat Sink

Once the required thermal resistance of the heat sink has been determined, the process of designing the heat sink can begin. *Figure 8* shows the thermal resistance versus the required volume for an anodized or painted aluminum heat sink. Note that the curve in *Figure 8* is based on lab measurement of 1/8" and 1/16" thick sheet aluminum and is only intended as a design guide. The actual thermal resistance of the heat sink is affected by many factors such as the shape of the heat sink, the orientation of the heat sink, etc. Once a heat sink is fabricated, its thermal resistance should be measured under actual operating conditions. The following calculations show how to design a heat sink for the LM2419.
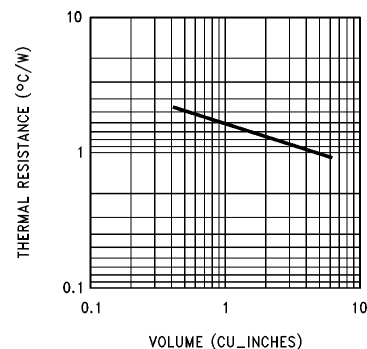
TL/H/11733–8

FIGURE 8. Heat Sink Volume vs Thermal Resistance for Anodized Sheet Aluminum

MM74HC86
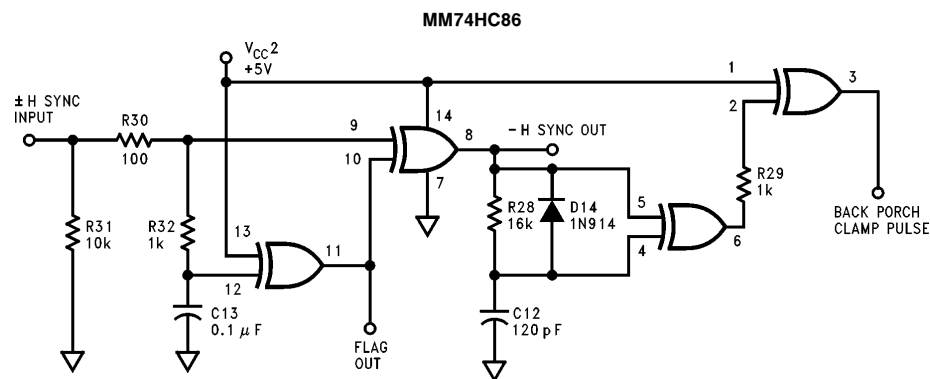
TL/H/11733–7

FIGURE 7. Backporch Clamp Pulse Generator Circuit

Worst case power dissipation for LM2419 (White screen) = 13W

Required heat sink thermal resistance, θ_{SA} = 3°C/W

From thermal resistance curve in *Figure 6,* required volume of Aluminum (Al) = 2.2 cu″

For a sheet of Al with thickness, T = ³⁄₃₂″,
required area, A = 2.2 cu″ /(³⁄₃₂)″ = 24 sq″ = 8″ x 3″

Therefore select an anodized or painted black aluminum heat sink of dimension 8″ x 3″ x ³⁄₃₂″. Note that if the heat sink surface is bright or shiny then the thermal resistance is going to be higher than an anodized or black painted surface because of the heat sink's lower emissivity.

5.2 Measuring the Thermal Resistance of the Heat Sink

Whether a heat sink is designed or a commercially designed heat sink is used, the thermal resistance of the heat sink should be measured under actual operating condition to ensure that the measured thermal resistance meets the specification. If the heat sink's thermal resistance is higher than the required thermal resistance then the CRT driver's case may operate at a temperature higher than the recommended operating temperature thereby adversely affecting the long term reliability of the device.

To measure the heat sink's thermal resistance, a thermocouple may be used. The LM2419's metal tab is bolted on to the heat sink with a screw, a washer and a lock nut. The thermocouple's wire should be securely tightened between the washer and the tab. Next, with the video board mounted in the monitor, the LM2419's power dissipation should be measured. The LM2419's case temperature is then measured under the operating condition using the thermocouple device. The measured thermal resistance is then calculated as follows :

$$\theta_{SA} = (T_C - T_A)/PD$$

where, T_C = Case temperature

T_A = Ambient temperature

PD = Power Dissipation

For our application, a ³⁄₃₂″ x 6″ x 4″ sheet of aluminum was used under actual worst case operating condition. The measured thermal resistance of the shiny (unpainted) heat sink was 5°C/W and decreased to 4°C/W after the heat sink was painted black with an enamel spray paint. Optimizing the shape of the heat sink could have improved the thermal resistance to less than 4°C/W. This excercise illustrates that the guidelines of section 5.1 can be used to design a heat sink and proper characterization under actual worst case operating conditions are needed to finalize the design.

5.3 Getting the Best Performance from the Heat Sink

For best results, the following guidelines should be followed:

(a) A thermal joint compound (such as Thermacote from Thermalloy or the 340 silicone heat sink compound from Dow Corning) should be used between the LM2419's metal tab and the heat sink. The thermal joint compound is a grease that establishes a low thermal resistance between the package and the heat sink by displacing the air gaps.

(b) Proper torquing (i.e., mechanical stress) should be applied so that good thermal contact is established. A torque of 6 lb-inch is commonly applied.

(c) The heat sink should be mounted vertically. This causes the heat sink to lose heat most effectively because cold air is drawn to the bottom of the heat sink, heated and moved to the top of the heat sink by convection. Furthermore, mounting the heat sink vertically is especially useful for heat sinks with fins.

(d) Either an anodized heat sink should be used or black oil paint or a dark varnish should be applied to the heatsink. This further reduces the heat sink's thermal resistance due to improved radiation heat transfer.

6.0 CONTROLLING ELECTROMAGNETIC INTERFERENCE (EMI)

There are stringent requirements on the manufacturers of electronic products to control the emission of electromagnetic waves. Electromagnetic waves not only interfere with radio and TV reception but may also affect other electronic devices in the vicinity of the source of radiation.

Voltage spikes caused by fast switching currents and the impedance of the supply line and ground connection can give rise to EMI radiation. An effective way to combat such a cause of EMI is by making use of power supply filtering and generous use of ground plane on the printed circuit board. The ground plane provides a low impedance return for the fast switching current, thereby suppressing EMI radiation.

Sometimes an undetected very high frequency (several hundred MHz) oscillation in the circuitry can give rise to significant EMI radiation. Such high frequency oscillation may not be noticeable when viewing images on the screen or may go undetected if the oscilloscope used is bandwidth limited when compared with the frequency of oscillation. By looking at the amplitude and frequency spectrum of the EMI radiation, one can discern the presence of high frequency oscillation within the circuitry.

Often long wires carrying high frequency signals can be a big contributor of EMI radiation. If that is the case then shielded cables should be used and the shield should be grounded at both ends. Grounding the shield at both ends allows the signal's return current to flow through the shield at high frequencies. The return current on the shield generates a field that tends to cancel the conductor's field thereby minimizing EMI radiation.

Some very high frequency designs require the use of conductive shield enclosures to minimize EMI radiation. In response to the offending electromagnetic field, the shield produces currents which in turn produce magnetic fields that oppose and cancel the inducing field. A steel enclosure provides excellent attenuation of EMI radiation through reflection and absorption loss (caused by exponential decay of the electromagnetic wave's amplitude as it travels through the medium).

7.0 PC BOARD LAYOUT GUIDELINES

Optimum performance at high frequencies requires careful attention to PC board layout. Before starting the PC board layout the circuit schematic should be carefully studied, high frequency signal paths and sensitive nodes should be marked. Once a well thought out PC board layout plan has been established, the actual board layout can commence. The following guidelines are essential for PC boards designed for 100 MHz or greater bandwidth.

7.1 Adequate Ground Plane

For high frequency layouts, a solid ground plane is a must. The ground plane provides a low inductance path for the circuit's return current. Moreover, a ground strip between two high frequency signal traces can reduce cross talk by referring the stray capacitance between the traces to ground.

Because of the many restrictions placed on the layout, a two sided board is recommended for bandwidths greater than 100 MHz. On a two sided PC board, a full ground plane is placed on the component side, for example the top side of the board. Signal traces are then routed on the bottom side of the board, this minimizes stray capacitance and improves the isolation between high frequency signal paths because the traces can be widely separated without affecting PC board real estate. Furthermore, a two sided board also greatly reduces the number of jumpers required when compared with a single sided board thus allowing optimum layout. A double sided board, however, adds to the cost of the board. A double sided board may cost 30% more than a single sided board. Increasing competition for high volume and low cost consumer products often necessitate the choice of a single sided board. If designed right, a single sided PC board can provide acceptable performance at 100 MHz.

When laying out a single sided PC board, every attempt should be made to layout a solid unbroken ground plane. *Figure 9(a)* shows ground voids along each column of pins of the IC and is not recommended for high frequency layout because it breaks up the flow of ground plane from the left to the right. Such a layout may also compromise the stability of the video amplifiers. The layout is improved by placing a ground void around each pin (see *Figure 9b*) thus ensuring a continuous flow of the ground plane from left to right. Some designers use pin sockets to avoid soldering the IC to the PC board. Pin sockets should be avoided if they reduce the clearance between the pins and make it difficult to achieve the layout of *Figure 9(b)*.
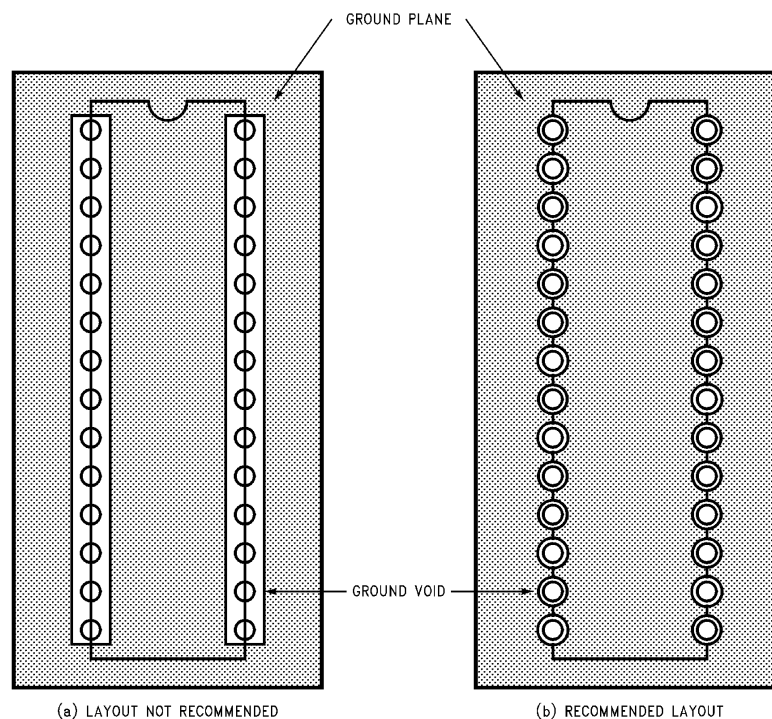

TL/H/11733–9

FIGURE 9. Optimize High Frequency Layout by Having Ground Void around Each Pin (b) instead of Ground Voids along the Entire Column of Pins (a).

Figure 10(a) shows another common error made in many PC board layouts. Trace "a" as shown in *Figure 10(a)* runs both in the horizontal and the vertical direction thus breaking up the flow of the ground plane. Too many zig-zag traces in the horizontal and vertical direction can render the ground plane rather ineffective. *Figure 10(b)* shows that using jumpers to connect traces in the vertical direction maintains the flow of the ground plane from left to right. This technique is especially necessary for single sided PC boards. Note that the effect of jumpers in the signal path must be considered. Normally jumpers are used in low frequency signal paths. Furthermore, with careful planning, passive components can often be used as crossunders such that signal traces can pass under the components thereby minimizing the number of jumpers required.

7.2 Avoiding Ground Loop

Often oscillations can occur due to ground loop currents. When the CRT driver's output slews at high frequencies, large transient current is injected to the ground plane. If both the preamplifier and the CRT driver share a common ground plane then the transient current may couple to the sensitive inputs of the preamplifier and may cause the preamplifier to oscillate thereby causing both amplifiers to oscillate. The problem is severe if a wide bandwidth preamplifier such as the LM1203B (100 MHz bandwidth) is used on a single sided PC-board.

The LM1203 + LM2419 CRT neck board (see *Figure 12*) is a single sided board with a single ground plane. Because of LM1203N's limited bandwidth of 70 MHz, no oscillations were observed. However, when the LM1203N was replaced with the LM1203B, the LM1203B burst into oscillation. Separating the power ground (pin 5) of LM2419 from the

low voltage ground plane and making a single point ground connection with the low voltage ground eliminated the oscillation. So, the recommendations for laying out the PC-board ground plane are as follows:

(a) The PC-board should be laid out with a separate power ground plane for the CRT driver.

(b) The CRT driver's power supply bypass capacitors should be connected to the power ground plane.

(c) The power ground plane should be connected to the low voltage ground plane at some point on the PC-board. The best place to connect the two ground planes should be empirically determined during the prototype design phase.

Use of above guidelines may also reduce ringing at the pre-amplifier's output and therefore further improve the overall system performance.

7.3 Power Supply Bypassing

Proper power supply bypassing is very critical for high frequency PC board layout. The power supply should be a low impedance point. However, the parasitic inductance of the supply lead can cause the power supply to be high impedance at high frequencies. Improper power supply bypassing can not only produce excessive overshoot and ringing on the amplifier's pulse response but can also cause oscillation.

Both the LM1203 preamplifier and the LM2419 CRT driver have very low power supply rejection, especially the LM2419 which has 0 dB power supply rejection. **Thus any noise or ripple or transients, on LM2419's power supply pin will appear directly at the device's output.**
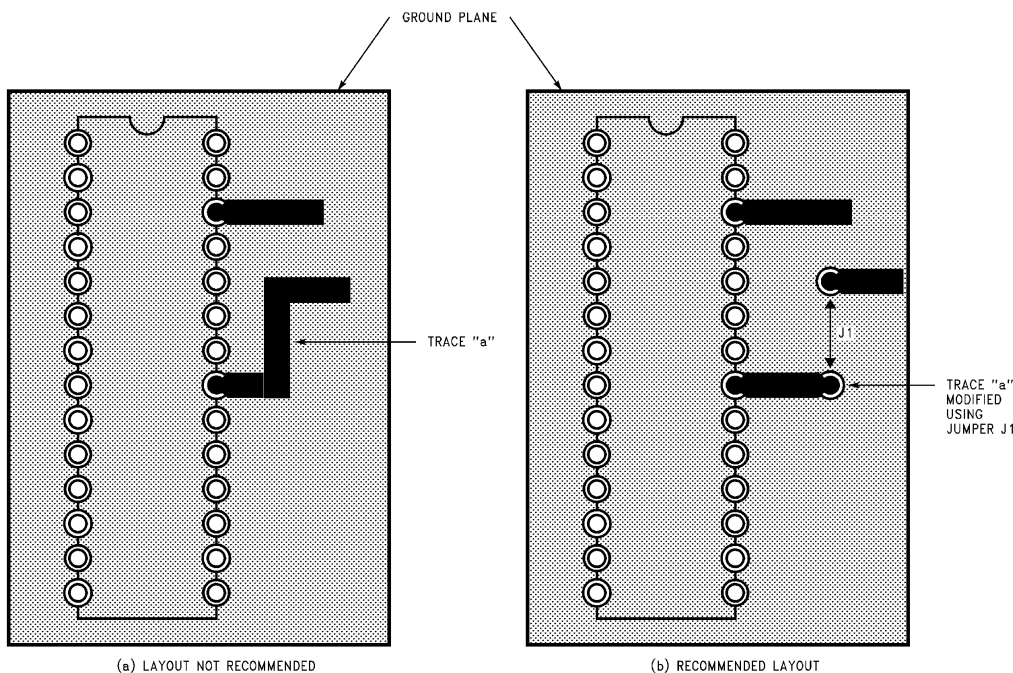

TL/H/11733–10

FIGURE 10. Maintain a Continuous Flow of Ground Plane by Avoiding Zig-Zag Traces (a) and by Making Use of Jumpers (b).

10

A high-frequency ceramic capacitor of value 0.01 μF should be connected less than $\frac{1}{4}$ inch from the power supply pin of the LM1203 and the LM2419. Note that for some wide-bandwidth video amplifiers, the series inductance of the 0.01 μF capacitor is large enough to cause the amplifier to oscillate and a 0.001 μF capacitor may be needed to suppress the oscillations. Since power supply bypassing is so critical, one of the first things to do when starting the PC board layout is to place the bypass capacitors first. Having done so, the designer can then position the rest of the components on the board.

In addition to a high frequency bypass capacitor, a large bypass capacitor of value 10 μF or higher is also required. The large bypass capacitor can be placed a reasonable distance from the supply pin. For LM1203 and the LM2419, 10 μF electrolytic or tantalum capacitor should be connected as close to each supply pin as is practical (see *Figure 13*). The large bypass capacitor acts as an energy storage element and is the source of large transient currents when driving capacitive loads. For example, 5 ns rise and fall times when driving a 12 pF capacitive load at the output of LM2419 would require 120 mA (i = C dV/dt = 12 pF $\times$ 50V/5 ns) current per channel. The 10 μF bypass capacitor provides the 360 mA charge and discharge current for the load while the voltage change on the power supply is only 180 μV

$$\Delta V = 3 \times i \times \Delta t/C = 3 \times 120 \text{ mA} \times 5 \text{ ns}/10 \text{ } \mu\text{F}$$

Also note that the supply connector of LM1203 and LM2419 are bypassed with a 100 μF and a 47 μF capacitor respectively (see *Figure 13*).

Once the video board has been designed and is ready for evaluation, the power supply pin of each IC should be probed and the waveform observed on an oscilloscope. The waveform on the power supply should be observed with the video board operating at the maximum frequency. In a well-designed and well-bypassed board, power supply ripple, noise and transients would be minimum.

7.4 Optimizing the Layout of the High-Frequency Signal Path

Figure 11(a) shows that the components connected to the video input and output pins are placed above the ground plane. In wide bandwidth applications, the stray capacitance from the component to the ground plane can reduce the amplifier's bandwidth. *Figure 11(b)* shows an improved layout where a ground void (absense of ground plane) is placed along the high frequency signal path. Note the large ground void under the components and along the periphery of the signal trace (see *Figure 11(b)*). Furthermore, a large ground void is also placed around all components connected to the CRT socket as well as the CRT driver's outputs.

Every effort should be made to minimize the length of the video signal path. A long trace for instance will behave as a transmission line if the trace length is greater than 1/15th the wavelength of the highest frequency signal. Long trace lengths along the high-frequency signal path can introduce unwanted ringing and overshoot. If a long trace length is unavoidable, then the trace length may be broken up by resistors in series. By minimizing stray capacitance in the layout and using small value series resistors, the reduction in the amplifier's bandwidth can be made negligible.

7.5 Minimizing Crosstalk

Capacitive coupling between two adjacent traces will give rise to crosstalk. The greater the slew rate of the signal propagating through the trace the larger the crosstalk. Crosstalk can be minimized by increasing the separation between traces. Also, a ground strip between high-frequency signal traces can reduce crosstalk by breaking up the inter-trace stray capacitances and referring them to ground. Lowering the impedance of the trace can also reduce crosstalk but this may not always be practical. The following guidelines are recommended:

a. Place a ground void along the high-frequency signal trace (see *Figure 11b*).

b. Keep as much separation between high-frequency signal traces as is possible.

c. Include ground plane between high-frequency signal traces.

d. Keep output signals away from the inputs of the amplifier and from other sensitive nodes in the circuit.

8.0 COMPLETE PC BOARD LAYOUT

A complete PC board layout of *Figure 13's* circuit is shown in *Figure 12*. The ground plane and signal traces are on the bottom side of the board while the silk screen covers the top side. The board makes generous use of ground plane in and around the preamplifier and CRT driver sections. Moreover, jumpers are used to maintain a solid unbroken ground plane.

The high frequency bypass capacitors are placed less than $\frac{1}{4}$ inch from the power supply pins. Also, there is ground void along the high frequency signal path and around the CRT socket.

ACKNOWLEDGEMENT:

The author would like to acknowledge the contribution of Ron Page for the backporch clamp generator circuit (see *Figure 7*) and Tom Mills for the thermal resistance curve (see *Figure 8*).

REFERENCE:

1. LM1203 data sheet, National Semiconductor Corporation.

2. LM2419 data sheet, National Semiconductor Corporation.

TL/H/11733-11

(a) LAYOUT NOT RECOMMENDED

(b) RECOMMENDED LAYOUT
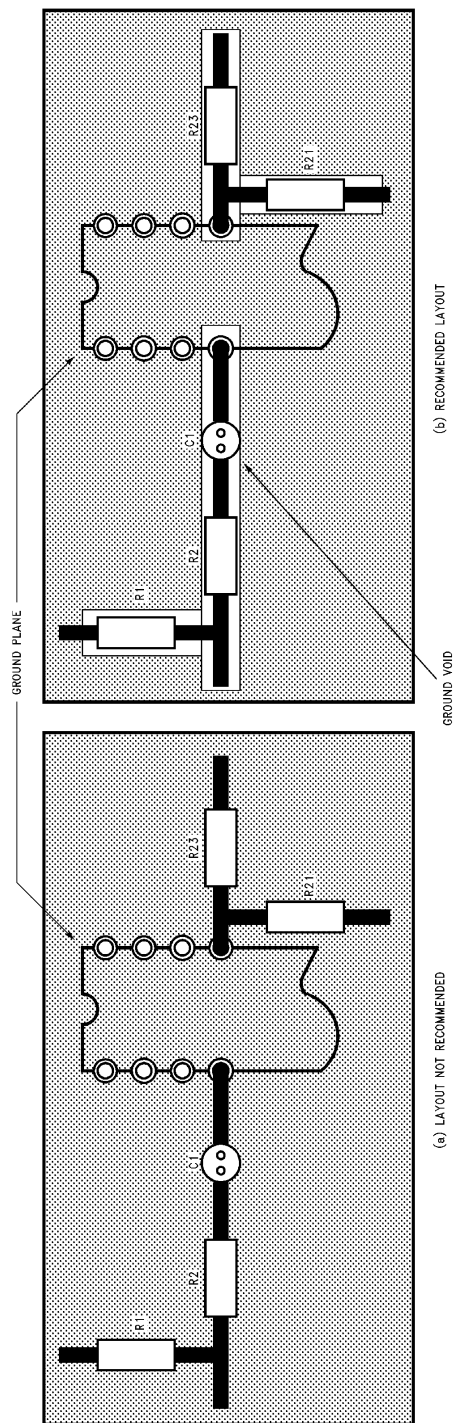
GROUND PLANE

GROUND VOID

R1 R2 C1 R23 R21

FIGURE 11. Using Ground Voids along the High-Frequency Signal Path Minimizes the Effect of Stray Capacitance
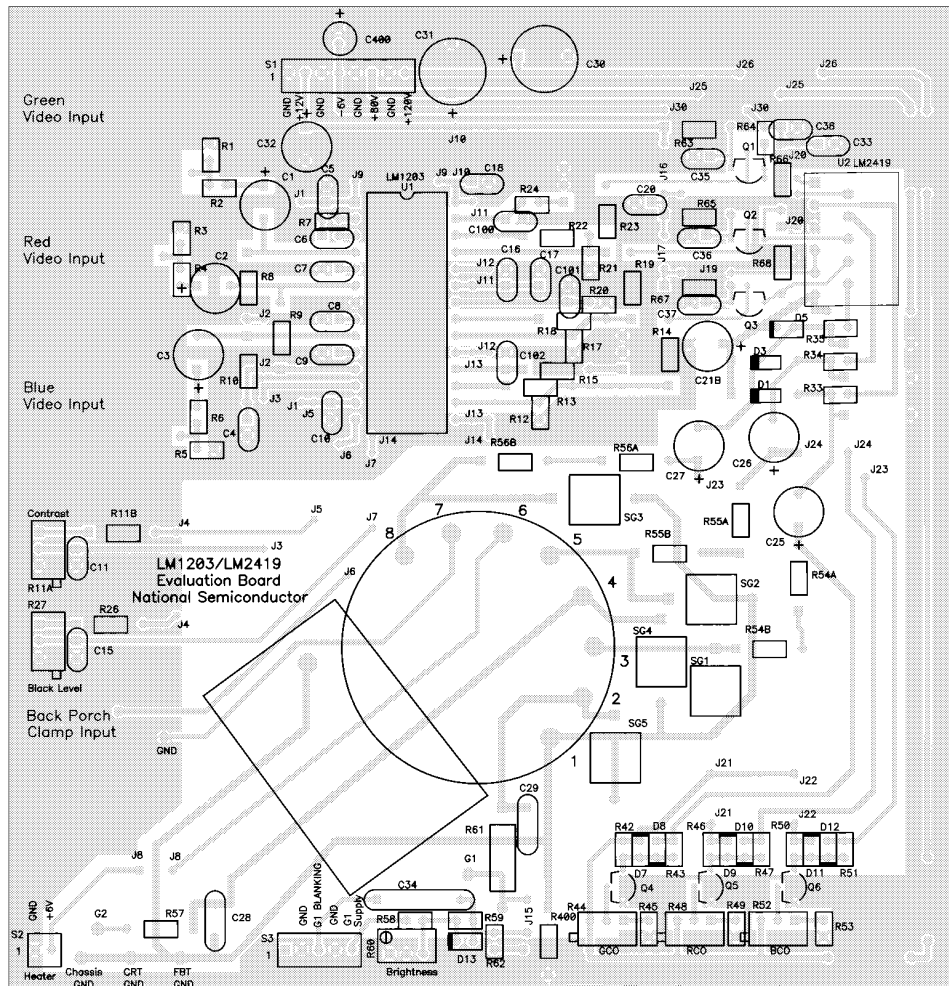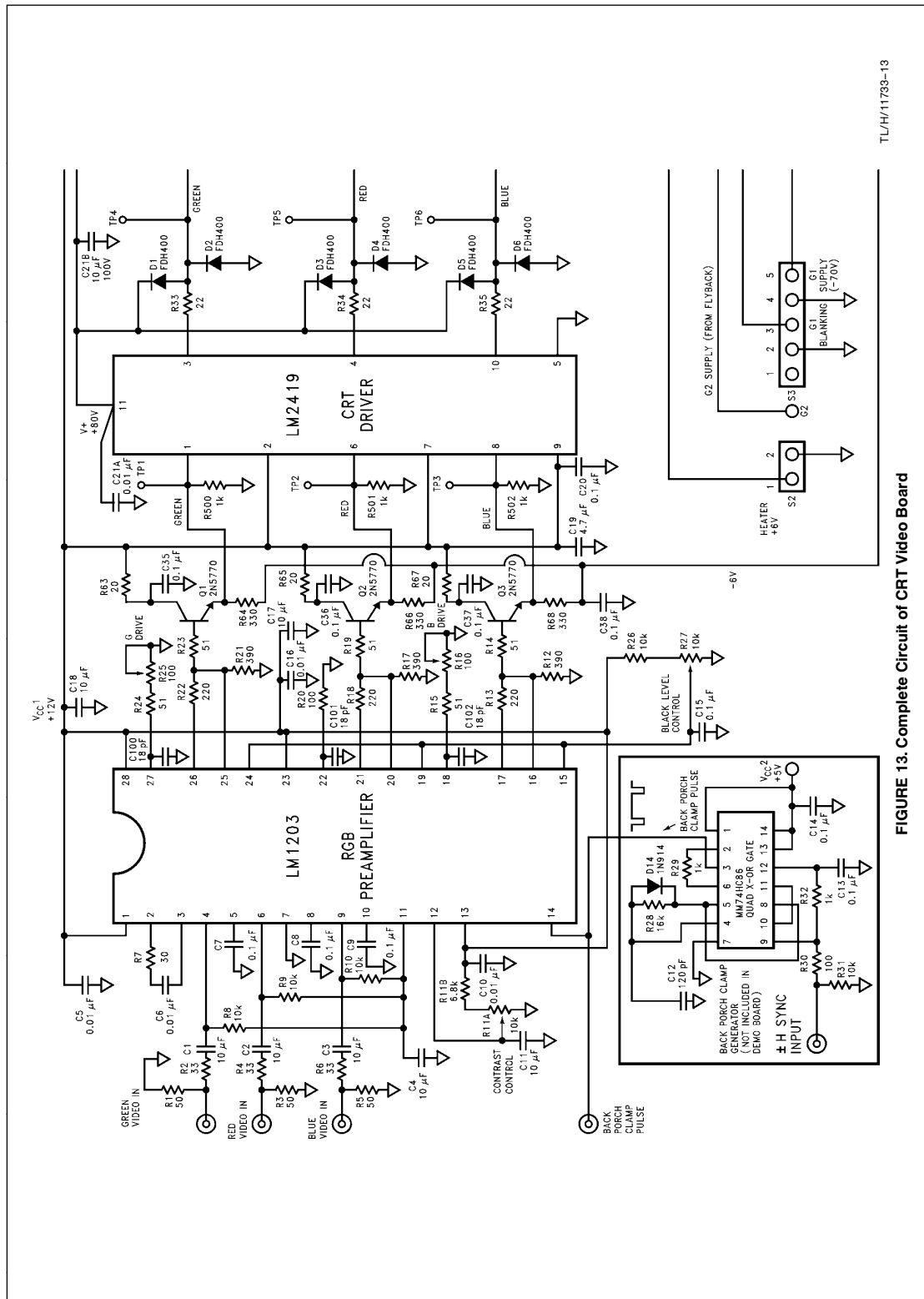
TL/H/11733–12

FIGURE 12. Single Sided PC Board Layout of *Figure 13* 's Circuit

FIGURE 13. Complete Circuit of CRT Video Board

TL/H/11733–13

FIGURE 13. Complete Circuit of CRT Video Board (Continued)

TL/H/11733–14

15