

Application Note

Product Name: 82C916 Serial Codec/Mixer
Title: Board Level Design Considerations and Interfacing with the 82C750
Date: January 28, 1997

Scope

The 82C916 Serial Codec/Mixer is intended to be an analog companion chip to the 82C750 (Vendetta) Integrated Core Logic/Audio Controller. This application note addresses circuit and layout issues to deploy the 82C916 as an all-in-one PC motherboard, notebook motherboard, a separate daughter board, or an ISA add-on card. Also attached are schematics showing a sample layout for the 82C916.

Discussion

Review of 82C916 Pin Functions

As an audio codec/mixer chip, the analog portion of the 82C916 is like a stripped-down version of OPTi's 82C931 and packaged in a 28-pin Small Outline Package (SOP, 300mm). Pins 5 through 13 are used to carry digital related signals; all the other pins are analog in nature and analog layout rules should be followed. This means that:

1. AVDD (Pin 25) should be supplied with a regulated source, such as 78L05 type. **This regulator should not be used to supply any digital circuits.**
2. Among the nine pins for digital connection, five are mandatory for interfacing the 82C916 and 82C750: SCLK (Pin 9), FS (Pin 10), SDI (Pin 11), SDO (Pin 12), and LR2 (Pin 13). In particular, **SDI (Pin 11) must connect to the 82C750's SDO (Pin A16) and SDO (Pin 12) must connect to the 82C750's SDI (Pin B16).** For connectivity information, refer to Figure 1 and Figure 2.
3. Pin 5 (OSCO) and Pin 6 (OSCI) are for connecting to external oscillating capacitors (22pF to 27pF). Pin 6 can also be connected to an external clock source if the source has enough fan-outs.
4. CINL (Pin 16) and CINR (Pin 14) are internal codec I/Os to be connected to corresponding mixer I/Os as MIX-OUTL (Pin 17) and MIXOUTR (Pin 15). The connection should be AC coupled through a capacitor of 0.22 μ F, minimum. (Do not use 0.1 μ F or less, otherwise frequency response will be narrowed at the low frequency edge.)

5. VDD (Pin 7) can be supplied from near +5.0V sources on-board, preferably not taken from those that connect to high-speed digital circuits (e.g. a high-speed graphic processor). VSS (Pin 8) should be connected to digital ground bypassing the ground plane of high-speed digital circuits also, even if this means tying Pin 8 to the far side.
6. If Game Port/Ext. MIDI are supported, the transmit and receive signals must be supplied from the 82C750. Use the following signals from the 82C750: TxD (Pin AE13), RxD (Pin AF13), GP0-GP3 (Pins A7, B7, C7, J23, respectively).
7. If MIC recording is supported, be sure to supply 2.5V for MIC biasing which is necessary for most Condenser MICs. This voltage can be taken from either an LM385-2.5 precision silicon reference diode, a 2.7V Zener Diode (e.g. ITT ZPD2.7), or stepped up from the reference voltage supplied by the 82C916. Be sure to check the voltage at MIC Jack's bias terminal to determine if it is properly biased. There is no standard value in the impedance of a Condenser MIC.

If MIC bias is taken from the 82C916, be sure to buffer this voltage with a unity-gain buffer. **VREF (Pin 22) is not recommended to be used as a direct current source.**

8. LOUT and ROUT from the 82C916 - If a headset or power speaker is the only output device to be used, devise a LINE_OUT buffer using an OP-Amp with high current driving capability. The 82C916 is not recommended to drive a 32 ohm headset directly.

If a (pair of) passive speaker is to be used as the output device, a power amplifier (such as Philips TDA1517P or SGS Thomson TEA2025B) is recommended. TEA2025B has two or three pin-to-pin compatible parts and is less expensive, but TDA1517P has better quality and has mute control pin available for Microsoft APM (Advanced Power Management) control purpose. For applications where power consumption is important, such as a note-

Application Note

book environment, a LINE_OUT buffer device is recommended.

PCB Layout General Precautions

1. An analog/digital ground boundary should be set across the 82C916's layout footprint so that Pins 5 through 13 are sitting on top of the digital ground plane (assuming a 4-layer PCB) and the remaining pins are on top of the analog ground plane.
2. As analog output pins, OUTL (Pin 28) and OTR (Pin 27) should maintain a clearance with analog input traces to avoid possible input/output crosstalk. A minimum 100mil clearance is recommended.
3. All analog input/output routes should remain a reasonable clearance with all digital routes (i.e., these traces should not run directly in parallel with digital routes but maintain at least a 100mil clearance). In general, analog traces are usually surrounded by the analog ground plane (in a 2-layer PCB) or running above analog ground (in a 4-layer PCB).
4. All analog input/output traces between left and right channels must maintain reasonable clearance to avoid left/right inter-channel crosstalk. A standard practice is to deploy one trace on the component side while the other channel on the copper side. If this method can not be sustained on every analog I/O, the next alternative is to deploy both on the same side but maintain reasonable clearance, 100mil apart typically.
5. Routing of the five interface signals mentioned above (in "Review of 82C916 Pin Functions", Item 2) should be deployed as a group along digital ground (if 2-layer) or above digital ground (if 4-layer). Preferably not running along side-by-side with other digital traces.
6. If both a joystick connector and wave table (e.g., Wave-Blaster) are supported, the Tx/D trace length between the joystick connector and wave table connector (or on-board wave table circuit) must be taken into consideration. Do not exceed a trace length of more than 6 inches or else the transmission line may produce unwanted noise resulting in having to make the trace width thicker, or terminate the trace altogether. *In general, trace width for audio related digital signals should be at least 12mil.*
7. As previously stated, noise can be induced by improper analog/digital ground separation. Analyzing this noise source is a very tedious process and is not limited to add-on cards, daughter cards, or all-in-one motherboards. The chances for this type of noise to occur is equal in each case.

The most obvious case of this type of noise is when the humming of the noise becomes (much) louder when no

audio is playing. A minor case is when it only sounds noisier than other boards of carefully deployed grounding. The worst case is that it is no longer generating quiet humming (known as white noise), but very loud humming instead, sometimes interleaved with high-pitch noise caused by power supply noise coupling. Such occurrence can only be avoided by careful grounding, it is not 82C916 specific.

Layout Precautions of All-In-One Motherboard

1. In an all-in-one PC/Notebook motherboard, available room for running circuit routes is much more restricted as opposed to an add-on card. Layout principles described in the "PCB Layout General Precautions" section may not be fully feasible and compromise may be necessary. If that's the case, it is more important to follow Items 1 through 4 in the "PCB Layout General Precautions" section rather than 5 and 6.
2. Do not arbitrarily take digital power (+5.0V) and digital ground just because it is available nearby. Examine the nature of the supply and ground plane to assure no high-speed circuit is residing closely to the 82C916's layout area. Analog signals can easily be disturbed by digital noise either through power/ground or crosstalk/stray coupling.

For power, either an L shape or π shape inductor/ferrite filter can reduce digital noise. For ground, make sure that digital ground does not join analog ground at a junction where large ground-return current may happen or that digital ground does not join with another digital ground plane where high-speed pulses are generated/running nearby. This is because high-speed pulses create many lower speed harmonics which may eventually couple into digital ground and propagate to analog ground through the analog/digital ground joint. Again, a ferrite may be used to decouple analog ground with digital ground.
3. A switching power supply is usually noisy, regardless of the brand chosen. The supply to AVDD can be reduced by using a voltage regulator to produce a clean, low-ripple voltage output from a noisy +12.0V input from the switching power supply. The supply to VDD is generally taken from +5.0V elsewhere on-board. It is an important consideration to devise a method to reduce 33KHz switching power supply noise that is commonly encountered. Placing an L/ π shape filter at the entry point where the VDD supply trace is to be injected to the 82C916 is good practice. It is also important to note that:
 - a. The VDD source is already heavily filtered at the location where switching power supply connectors are connected to motherboard.



- b. The VDD pin has no direct feed (along its traveling path) to any high-speed circuits. The feeding of this power trace to other digital circuits should be minimized.

Note: It is vague to say "high-speed" digital circuits. Generally speaking, this means those circuits that generate/carry multiple data/clock streams at 20MHz or higher.

82C916-based Daughter Board Or Add-on Card Layout Precautions

When the 82C916 circuit is built on a daughter board or as an ISA add-on card, there is a potential noise problem caused by the five digital interface signals carried on the ribbon cable from 82C750 to 82C916:

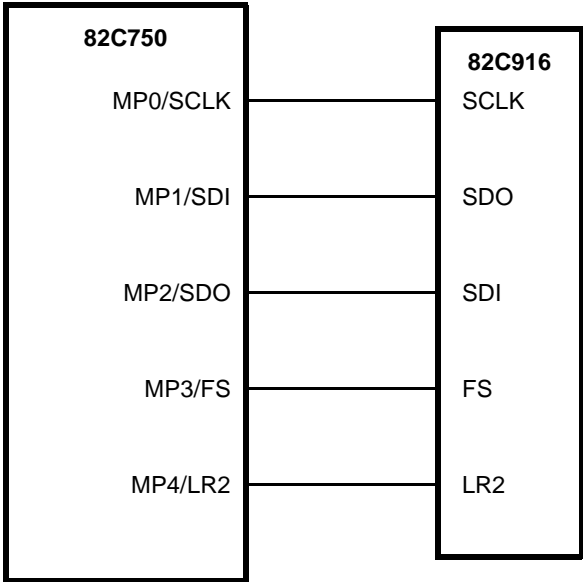
1. Noise induced by cable - Shield the cable with digital ground (as in the OPTi 82C750/82C916 reference design). The shorter the cable length, the better. If the cable length is more than 6 inches, consider using a non-

inverting buffer. If noise still exists despite this effort, change the clock source from an outside source to an internal source (refer to "Review of 82C916 Pin Functions" section, Item 3), or try to terminate the five signals with an RC filter.

Summary

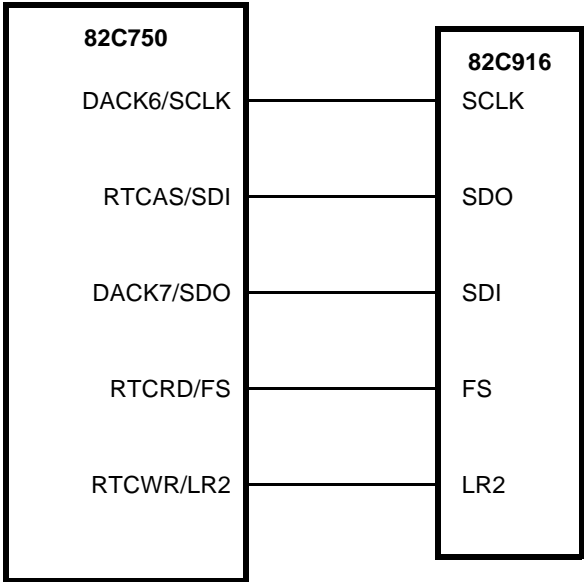
Notebook/motherboard design often (inevitably) involves a 4-layer PCB process for saving routing real estate, and the add-on card/daughter board approach is often a 2-layer effort for cost reasons. Grounding practice for a 2-layer PCB takes more effort than 4-layer PCB to the layout designer. However, regardless of whether a 2-layer or 4-layer approach is used, the circuit designer needs to pay great attention to 82C916's grounding as much as to a conventional sound card design because of the nature of the chip as a Serial Audio Codec, other than that, the circuit itself for interfacing 82C916 to 82C750 is quite straightforward (refer to Figure 1 and Figure 2).

Figure 1 Scenario 1: Memory Parity and Codec Muxed



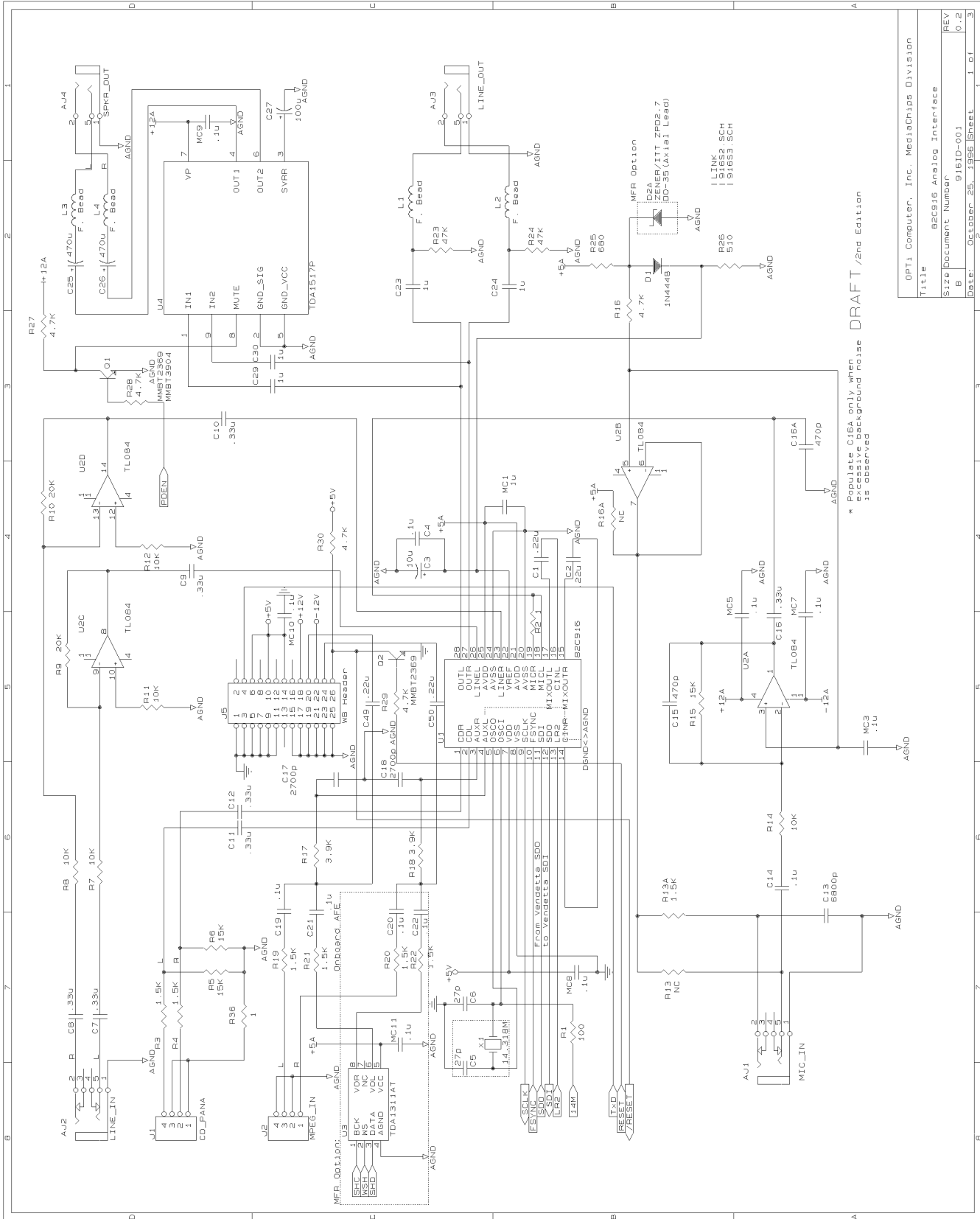
Note: The MP or ECC functionality is not available if this scheme is used.

Figure 2 Scenario 2: DACK and Codec Muxed



Note: If the encoded DACK scheme is used, an external F138 is required. ECC can be used with this scheme. No discrete RTC chip can be used in this mode.

Application Note

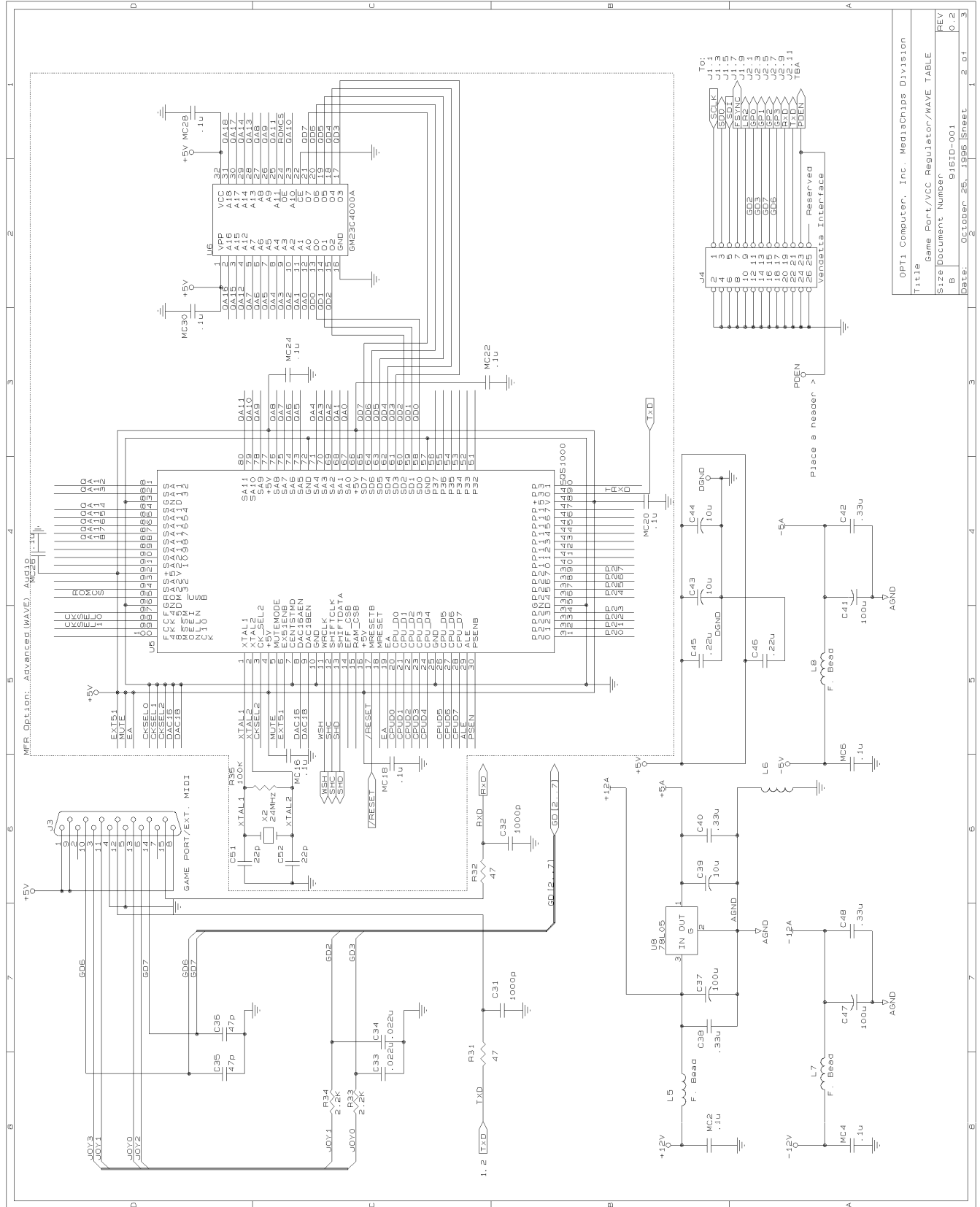


* Populate C16A only when background noise is observed

DRAFT / 2nd Edition

OPTi Computer, Inc. MediaChips Division	
Title	BEC916 Analog Interface
Size	Document Number
REV	916ID-001
B	0.2
Date:	October 25, 1996 Sheet
	1 of 3

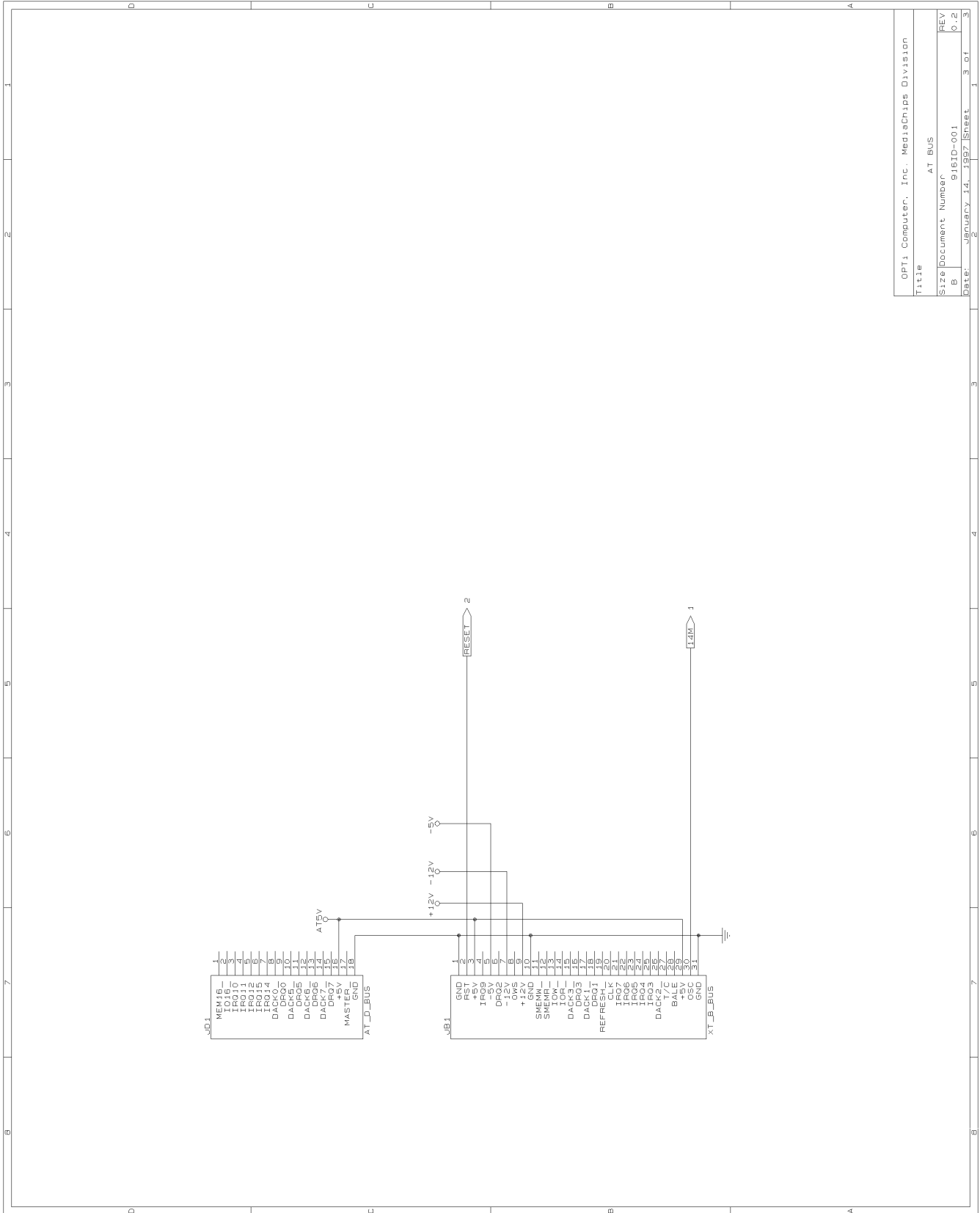




OPTI Computer, Inc. MediaChips Division		
Title	Game Port/VCC Regulator/WAVE TABLE	
Size	Document Number	9161D-001
REV	0.2	
Date:	October 25, 1996	Sheet 2 of 3



Application Note



OPT1 Computer, Inc. MediaChips Division	
Title	AT BUS
Size	Document Number
B	916ID-001
REV	0.2
Date:	January 14, 1997
Sheet	3 of 3

