ANALOG | edge℠

Dealing with EMI in Class D Audio Applications

Application Note AN-1737

Bill McCulley, Applications Engineer

EMI – The Basics

Electromagnetic Interference (EMI) is an unwanted disturbance caused in an electrical circuit by electromagnetic radiation emitted from an external source. The disturbance may interrupt, obstruct, or otherwise degrade the effective performance of the circuit.

In today's portable and consumer applications, space has become a premium, and engineers are often required to eliminate enclosures and shielding, and suppress EMI and noise through other means such as better segregation at the circuit level. Smaller space and higher functionality require high-density PCBs, and the use of wafer-scale packaging with tiny PCB design rules makes EMI more of a concern.

EMI encompasses two aspects. Emissions refer to the scope to which equipment generates radiated noise. Susceptibility is the scope to which equipment is affected by emissions generated from other electromagnetic waves. The degree to which a designer controls unintended emissions may make the task of susceptibility easier. Emissions are generally classified as radiated and conducted emissions. Radiated emissions leave a circuit board, trace, or wire, and propagate through the air in the form of electromagnetic waves to interfere with a nearby receiver. It is important to note that a "receiver" refers to any circuit whose operation can be affected adversely by the reception of electromagnetic energy – such as a PCB trace or even the lead of an IC. Conducted emissions refer to energy which escapes, or is conducted, out of a circuit through wires or cables. Conducted emissions may cause problems directly or manifest themselves as radiated emissions.

In order to understand emissions, it is important to understand antennas. In *Figure 1*, the well-known physical relationship between wavelength and frequency is shown:

$$\lambda = \frac{Velocity\ \ of\ \ Light}{Frequency\ x\sqrt{\in_r}} = \frac{300}{f\,(MHz)\ x\sqrt{\in_r}}$$

$\in_r$ is the relative permittivity

Figure 1. Wavelength as a Function of Frequency

The shortest length required to be an efficient antenna is λ/4. In the case of air, permittivity is 1, but in the case of FR4 or glass-epoxy PCBs, permittivity is reduced to approximately 4.8. The effect causes a signal traveling a trace to slow once it reaches the dielectric gradient created by the FR4 material, causing essentially, a "wavelength-shortening" effect. For example:

A 200 MHz signal has a quarter wavelength in air of 16.7 cm.

In an inner-layer PCB trace, it is: $16.7/4.8^{(1/2)} = 7.6$ cm.

A PCB trace can act as an unintentional antenna even at lengths shorter than λ/4, increasing both emissions and susceptibility. Surface traces also exhibit this wavelength-shortening effect, as one side of the dielectric serves to change the overall permittivity of the transmission.

Unintended antennas, such as PCB traces, are the key culprit behind radiated noise in digital systems. As we will see, the Class D audio amplifier is, in essence, a digital system from the perspective of radiated emissions. One key principle underlying electromagnetism is that of reciprocity. A flow of current can create an electric field, and a change in electric flux can induce a current to flow. Likewise, an antenna that is good for receiving is also good for transmitting. If any of the dimensions of an unintentional antenna approach quarter wavelength at the frequency at which that antenna is excited by noise current, radiated emissions can be expected.

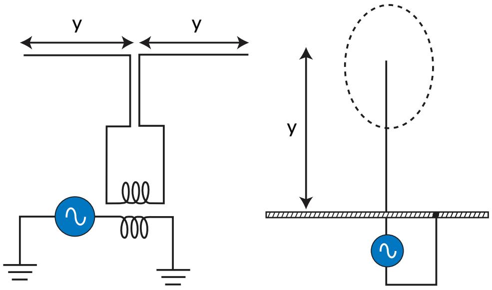


Figure 2. Common Antennas

National Semiconductor
The Sight & Sound of Information

These are two common antenna designs, the simple dipole and whip, shown in ***Figure 2***. An interesting observation is that the whip is essentially one half of a dipole with the horizontal ground essentially being induced to act as the other half of a dipole.

Antennas are intended to transmit and receive signals through the radiation of electric energy. But, as illustrated in ***Figure 3***, *unintended* antennas in circuit boards can include:

- Long traces
- Vias
- Component leads and pins
- Unpopulated PCB board connectors and headers

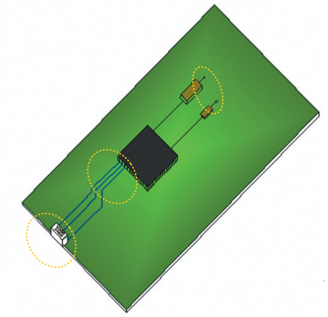


Figure 3. Unintended Antennas in PCBs

An unterminated surface trace or an unterminated buried trace can become an unintended whip antenna. Segments of traces at different RF potential due to poor layout can become unintentional dipole antennas. Also, the conductive layers of the PCB can act as the other leg of a dipole antenna with the plane itself being coupled into the electric field.

The Class D Audio Amplifier

The Class D audio amplifier has emerged as a popular topology for the consumer market due to its high efficiency. It modulates a high-frequency square wave by the incoming analog signal. The square wave itself could be fixed frequency, variable frequency, or simply random pulses with no fixed frequency. A low-pass filter, typically a 2-pole Butterworth, is used to filter the high-frequency content and recover the original audio signal. In “filterless” topologies, the inductance of the speaker itself is incorporated as part of the filter. One common Class D topology, Pulse Width Modulation (PWM), uses a fixed-frequency waveform and changes the duty cycle to create a moving average of the signal after a low-pass filter as seen in ***Figure 4***.

The benefits of a switching topology are apparent – high efficiency, low-power consumption, and small thermal designs. But increased efficiency is not without cost. In order to drive efficiencies up, a sharp, rapidly switching square wave is required. This can lead to the same undesirable artifacts that are present in digital systems as the spectral energy is highly concentrated on the edges of the square wave. Also, there will be some overshoot, causing the waveform to go beyond the maximum and minimum voltages for a short time. Overshoot creates additional high-frequency content in the output spectrum and is undesirable for EMI and audio performance.

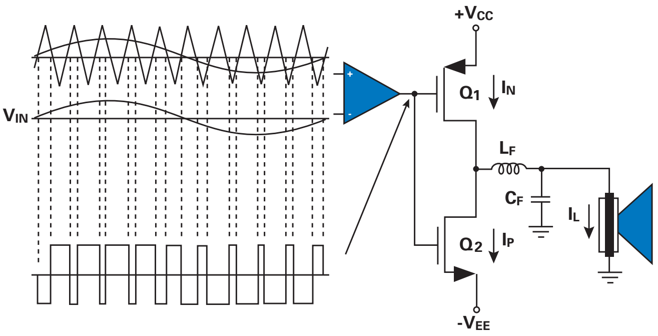


Figure 4. The Class D Audio Amplifier PWM

Countering EMI

To counter EMI, it is essential to include PCB floor planning as part of the circuit design with the electrical engineer, PCB layout engineer, and manufacturing engineer. General PCB guidelines for dealing with EMI include:

- Placement of decoupling capacitors between power and ground where voltage fluctuations are determined to exist. Haphazard placement of caps can exacerbate the EMI problem. References for the use of decoupling capacitors are included at the end of this article.
- Power planes should be backed off from the edges of the PCB
- Avoidance of traces cut within ground or power planes – which can create an unintended aperture
- Adequate termination of all high-frequency clock lines
- Proper filtering of PCB connectors
- Good PCB design will avoid loop antennas. Loop antennas encompass any route in which both forward and return currents are on a well-defined conducting path:

Optimally, you will stop radiation by suppressing the source of current that is feeding the antenna.

For the audio designer, it is important to consider the following:

- Keep traces from the audio amplifier to the speaker as short as possible. PCB traces and/or wires act as antennas with significant radiation occurring once the trace length reaches λ/4.
- For filterless Class D systems, the trace and cable length connecting the amplifier’s output to the speakers will likely be the largest source of RF emissions.

The practice of placing ferrite beads in series with the loudspeakers close to the amplifier can be effective. In order to better understand the suppression mechanism of ferrite beads, it is useful to break the ferrite bead down into frequency-dependent resistive and inductive elements – R(f) and L(f) - as shown in ***Figure 5***.

Regarding EMI suppression purposes, ferrite beads act as resistors, but because $R_{DC} = 0$, there is no DC voltage drop. This makes them useful in cases where frequencies of interest are significantly below 1 MHz. Also, as shown in the figure, it is important to understand that the ferrite bead is effective when considered part of a two-element voltage divider. Both Z_1 and Z_2 are frequency dependent. To achieve the desired function of a low-pass filter, the following relationships should be present: $Z_2 > Z_1$ at desired frequencies and $Z_1 > Z_2$ at noise frequencies.

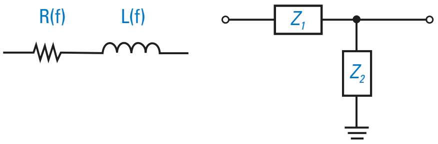
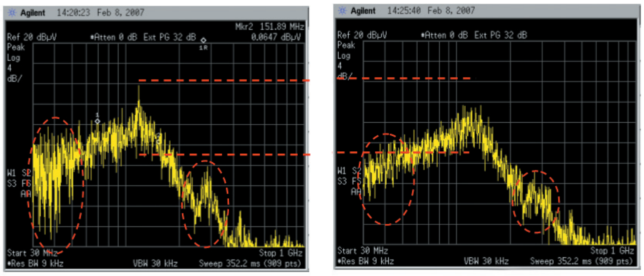


Figure 5. Resistance and Inductance as a Function of Frequency

The ferrite will usually serve as the series element, and the shunt element is a capacitance – either a physical capacitance or a lumped capacitance. The transfer function indicates that Z_1 will increase and Z_2 will decrease with frequency ($1/j\omega C$) and that the system will be damped to the extent that resonance effects are significantly diminished.

The primary difficulty with periodic square waves inherent in Class D is the concentration of energy at the harmonic intervals. In the push to create a "quiet" low-EMI class D amplifier, one approach is to dither that frequency back and forth – or spread the spectrum of the switching so the energy at any one point in the spectrum is reduced. Compared to traditional Class D amplifiers, the spread-spectrum-modulation scheme has some key advantages. Efficiency and low THD+N are maintained, but radiated noise and EMI can be significantly reduced, as seen in *Figure 6*.



Fixed-Frequency **Spread-Spectrum**

Figure 6. Comparison of Fixed-Frequency and Spread-Spectrum Modulation

The LM48511 is a spread-spectrum-modulated class D audio amplifier. It also includes a built-in boost regulator which drives the supply voltage to 7V, increasing amplifier output power and the audio-sound pressure level compared to an unboosted amplifier. The boost regulator allows the amplifier to maintain a constant output level, even when powered from a decaying voltage source such as a battery.

The LM48511 amplifier features a logic-selectable, spread-spectrum modulator that reduces EMI, eliminating the need for output filters or chokes. As shown in ***Figure 7***, the spread-spectrum modulator feeds a standard H-Bridge which drives the bridge-tied-load speaker. In spread-spectrum mode, the switching frequency varies randomly by 10% around a 330 kHz center frequency, decreasing EMI emissions radiated by the speaker and associated cables and traces. Where a fixed-frequency Class D exhibits large amounts of spectral energy at multiples of the switching frequency, the spread-spectrum architecture of the LM48511 amplifier spreads that energy over a larger bandwidth, reducing peak noise power in the circuit.

Electromagnetic interference is a system-level concern, and it is essential for today's audio engineer to design with EMI in mind, including the best possible design practices and judicious choice of components and materials.

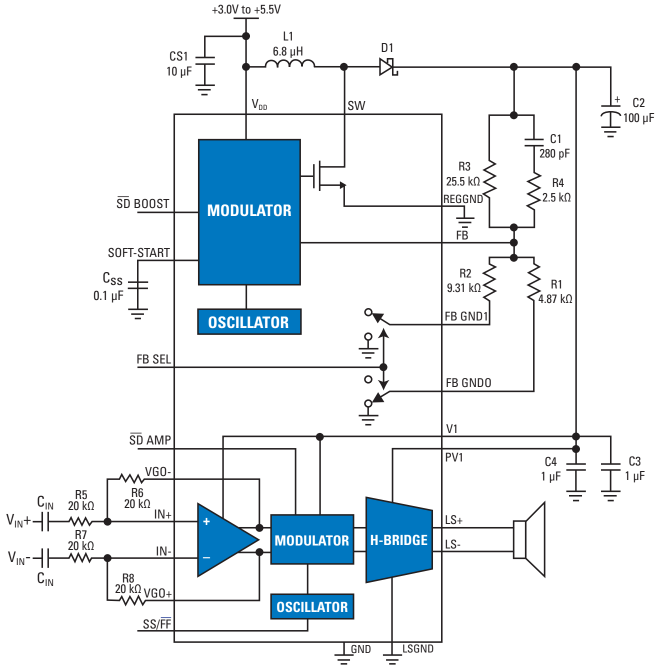


Figure 7. LM48511 – A Spread-Spectrum Modulated Class D Amplifier

References:

- Bruce Archambeault and James Drewniak, "PCB Design for Real-World EMI Control", 2002
- David Terrell and R. Kenneth Keenen, "Digital Design for Interference Specifications", 1997
- Howard Johnson, "High-Speed Digital Design: A Handbook of Black Magic", 1993

National Semiconductor
2900 Semiconductor Drive
Santa Clara, CA 95051
1 800 272 9959

Mailing Address:
PO Box 58090
Santa Clara, CA 95052

National
Semiconductor
The Sight & Sound of Information

570259-013