

Answers to Frequently Asked Questions

About

The LM4651/52 Class D Solution

1. Can the LM4651 be used at a separate supply voltage than the output FETs (LM4652)?

Yes and No. The bootstrap capacitors (labeled C_{BT}) are charged to 6 Volts. When the high side of the output FETs is turned on the gate voltage is 6 Volts above V_{CC} of the LM4651. For example if the V_{CC} of the LM4651 is 20 Volt, the gate voltage on the topside FETs will be 26 Volts giving a V_{gs} of 6 Volts. This value of 6 Volts was chosen because the LM4652 does not need a high gate voltage to turn fully on minimizing $R_{DS(on)}$. If the LM4651 is used at a lower supply voltage than the LM4652 the FETs may not turn fully on causing a higher $R_{DS(on)}$ resulting in lower efficiency. But the LM4651 can be used at a supply voltage that is HIGHER than the output FETs or the LM4652. This would result in a higher V_{gs} than 6 Volts. The limit for V_{gs} of the LM4652 is 10 Volts before damage can result. The low side FETs do not use bootstrap capacitors since the gate voltage is 6 volts above V_{EE} (-20 Volts, $V_{gate} = -14$ Volts).

2. Can the LM4651 be used to drive discrete output FETs for higher output current and lower impedance loads for higher output power?

As question number 1 helps explain there can be some difficulty with driving discrete FETs with the LM4651. The major problem is that the bootstrap capacitors (C_{BT}) charge to 6 Volts. Some discrete FETs specify a minimum of 10 Volts for V_{gs} to have the minimal $R_{DS(on)}$. Since the V_{gs} will be only 6 Volts the FETs may not turn fully on and $R_{DS(on)}$ will not be minimize. This will result in lower efficiency. If the discrete FETs use a slightly lower supply voltage than the LM4651 then V_{gs} can be increased. For example, if the LM4651 has a +/-22 Volt supply and the discrete output FETs have a +/-18V supply then V_{gs} will be 10 Volts. There are two other possible problems with using discrete FETs instead of the LM4652. One is that Thermal Shut Down (TSD) will have to be implemented separately. The LM4652 has a TSD output flag that goes back to the LM4651. This flag transitions to 6V when the die temperature exceeds approximately 150° C. The LM4651 needs a 5V signal to shut down the output stage. The second problem is that the current limit control of the LM4651 will not be accurate. The LM4652 and LM4651 have very closely matched output FETs. If discrete FETs are used they will not be matched as closely to the LM4651 output FETs. Current limit may still work but would need to be recalibrated with each different discrete FET.

3. Can two or more LM4652 ICs be put in parallel to drive lower impedance loads for higher power.

Yes. The PCB layout is a little more difficult but two different layouts have been tested on the bench. One layout has the two LM4652's side by side with a space of 0.5 inches between for easy mounting to a common heat sink. The other layout has the two LM4652's back to back with just enough space between to allow for a heat sink to be mounted. Both layouts were designed with minimal gate trace lengths as possible. Similar performance in Noise, THD, and power were observed for each board. Changes in the application circuit that were required for high power operation was the addition of $10\mu\text{F}$ capacitors on the supply pins (Vcc and Vee) of the LM4651 and $47\mu\text{F}$ capacitors on the supply pins (Vcc and Vee) of each LM4652. Larger capacitors are recommended for unregulated supplies.

The dual LM4652 configuration can be used to drive two loads with the same input signal (i.e. dual voice coil woofer) with each LM4652 having it's own output filter. The dual configuration can also be used to drive lower impedance loads connecting the outputs of the LM4652's in parallel and using a single, high power filter before the load. When using the dual configuration the current limit function is now current per LM4652. For example, if the current limit is set to by using a $100\text{k}\Omega$ resistor for R_{SCKT} then each LM4652 will be allowed to produce 11 Amps of peak current before current limit is activated. Dual configuration allows for total output power levels up to 300W for the LM4651/52 class D solution. A demo board for dual configuration is not available at this time.

4. What is the purpose of the Schottky diodes?

The Schottky diodes protect the LM4652 from fly back voltages. The output filter contains inductors (L_1) and driving a real work load speaker also has inductance. These inductors cause fly back voltages. If the fly back voltage is too high the LM4652 will be permanently damaged. The diodes also help clamp the output square waveform to have less overshoot. The recommended value for the Schottky diodes is 50 Volts and 3 Amps or 50 Volts and 1Amp with a high surge current ($>20\text{A}$) rating.

5. Can the Schottky diodes be integrated into the LM4652 die?

No. The fabrication process of the LM4652 does not have this ability. A larger diode can be put in parallel with the body diode but this is costly in die size.

6. If the supply voltage is lowered can lower current Schottky diodes be used or eliminated completely?

Possibly. THD may increase due to the switching waveform have more overshoot and

less ideal. EMI may also increase since the overshoot contains high frequency components. The topside diodes (from output to Vcc) may not be necessary for protection in most cases. The bottom side diodes (from output to Vee) are needed. The fly back voltages are really a function of the output stage and inductance of the load. The factors all contribute so the amount of diodes and current rating are all application dependent. Lower current diodes like 1Amp may be ok in most cases.

7. What is the maximum supply voltage with out any input signal or output power?

The maximum supply voltage of the LM4651/52 is limited by the fabrication process. The +/-22 Volt guarantee is for full output power. The LM4651/52 chips can normally go to +/-25 Volts with out damage but not with full output power in all cases. Because there is often confusion about operating and maximum ratings one specification is given. In most cases an unregulated supply is used in the application with the voltage rails increasing as the output power decreases. With no input signal or output power the supply rails must not go above +/-22 Volts or there is the possibility that a small number of parts will fail.

8. The demo board is a double-sided board. Can a single-sided board be used?

Yes. For high frequency design a double-sided board is much better. Planes can be used for power and ground. If these planes are laid out over each other a small amount of parasitic capacitance (pF) is created. This small parasitic capacitance filters very high frequency off the supply planes to the ground plane giving a cleaner voltage supply for the LM4651/52. A double-sided board also allows for a smaller layout size. A single-sided board would need larger supply bypass capacitors close to the supply pins of the LM4651/52 than a dual sided board.

9. What is the damping factor of the LM4651/52 solution?

Damping factor is the ratio of the output impedance to the load impedance. The higher the factor the better the amplifier can control the movement of the speaker. If the damping factor is too low the bass will not sound "tight". The output impedance of a class D amplifier also depends on the output filter. Typical values for damping factor for the LM4651/52 chipset are 40 to 100 at 100Hz. The value depends on the DC resistance of the inductors used in the output filter. To improve damping factor the lowest DC resistance inductor possible should be used. For a damping factor of 50 the DC resistance of the inductor should be less than 0.04Ω .

10. Noise is not acceptable. How can I decrease the audible noise?

The audible noise can be decreased several ways. Noise is mostly affected by the ratio of

R_f/R_2 . For good performance it is recommended that this ratio be higher than 100. For example, R_f would be $1M\Omega$ and R_2 would be less than $10k\Omega$. Then the value of R_1 would be adjusted for the desired gain. As the ratio increases the noise will be reduced. Another way to lower noise is to lower the gain of the class D amplifier. Changing the value of R_2 and R_1 plus using a pre-amplifier will help lower audible noise. National's LM833 dual audio operation amplifier is a good solution. Often a pre-amplifier is necessary already so adjusting the gain on the pre-amplifier to account for the decreased gain of the class D amplifier will maintain a constant system gain but will lower the audible noise. A third alternative is to decrease the output filter frequency response. If the application is a subwoofer or bass amplifier the output filter can be set to a lower 3dB frequency. This approach will often cost more due to larger values for the inductors and capacitor in the output filter. A fourth alternative is to lower the supply voltage. This may not be a possibility based on the target output power specification.

11. Is there a recommended range for gain for the LM4651/52 class D amplifier?

Yes. Unlike Class AB amplifiers the class D amplifier can be run with unity gain to over 100 V/V. High gain is not recommended. The recommended range for gain of the LM4651/52 class D amplifier is 1 – 20 V/V.

12. How can the DC offset seen at the load be reduced to eliminate clicks and pops at turn on/off?

The DC offset seen at the load is due to the duty cycle error. When there is no input signal present the class D amplifier should show a 50% duty cycle square wave. Any error from exactly 50% creates some DC offset at the load. To minimize the DC offset the ratio of R_f/R_2 should be above 100. A higher ratio will give a lower DC offset. If the DC offset is still unacceptable then adjusting the value of the resistor labeled R_{OFFSET} shown in the Application Circuit in the Data Sheet (Figure 1, page 4) will reduce the DC offset to 0V. This resistor puts a small DC voltage at the input so there is no duty cycle error. The value of this resistor will often be very large, several mega - ohms.

13. Why does the Application Circuit in the Data Sheet (Figure 1, page 4) show two different capacitors on the feedback filter (C_{f1} and C_{f2})? Why not just one larger capacitor?

The purpose of using two capacitors in the feedback filter is to lower EMI. The first capacitor (C_{f1}) is placed very near the output of the LM4652. The feedback traces to the LM4651 can be long and the longer a trace is the better it radiates electrical noise (EMI). Having the switching square wave output on a long trace will only increase EMI. Having a capacitor right near the output filters the audio signal from the switching square wave so an analog low frequency waveform is present on the longer feedback traces. The second capacitor (C_{f2}) is used right near the input to the LM4651. This capacitor is placed near the LM4651 to filter off any additional high frequency noise picked up by the

long feedback traces. The combination of these two capacitors lowers EMI and provides a cleaner feedback signal to the LM4651.

14. What is the purpose of R_{GATE} , the series gate resistors?

The gate resistors increase the rise and fall time of the gate voltage. This helps remove some of the highest frequencies in the square waveform. EMI is lowered due to lower frequency content in the square waveform.

15. How do I choose a value for R_{SCKT} setting the current limit?

The current limit is a function of the output current and the ripple current in the output inductors (L_1). Unless a load impedance lower than 4Ω is used a value of $100k\Omega$ for R_{SCKT} should be fine. Lower impedance loads require more output current so a lower value for R_{SCKT} like $39k\Omega$ can be used setting the current limit around 13 Amps. The ripple current will increase as the value of the inductor is lowered.

16. Why is Over-Modulation Protection necessary?

The over-modulation protection's main purpose is to create an artificial clipping point a little below where the actual clipping would occur without the protection. This allows the LM4651/52 class D amplifier to control the output when clipping so no extra distortion is introduced from oscillations or other imperfections. Also, if there was no protection signal then the output FETs would stay on for an extended period of time increasing the heating of the FET which also increases the $R_{DS(on)}$.

17. How does the output LC filter for a bridge application versus a single-ended application?

A 2-pole output filter consists of an inductor in series with the output and a capacitor going to ground after the inductor. This is the correct design for a single-ended configuration using a 2-pole output filter. When the design is a bridge tied load (BTL) often the same output filter is used; namely an inductor in series with each output and a capacitor to ground after the inductor to ground or four components. For a BTL configuration this is correct but the pole location is not the same. What the filter actually looks like to the output is a capacitor to ground from one output then from ground through a capacitor to the other output. In effect, two capacitors in series or $1/2$ the capacitance value than originally calculated. This is easily solved by connecting one capacitor across the load at $1/2$ the calculated value. Each output sees a capacitor so the effect is double the value used is seen in the output filter. The BTL configuration therefore requires a smaller capacitor than the single-ended configuration for a filter response that is the same. Often times in the single-ended configuration a small capacitor is put in parallel with the output filter capacitor. This small capacitor is a high quality

capacitor with the purpose filter the high frequency to ground. These capacitors are needed in the BTL design. A capacitor value around 10 – 20% of the large output filter capacitor should be used on each output to ground. A correctly designed output for a BTL configuration is two inductors, one large capacitor across the load and two capacitors that are 10 – 20% in size of the large capacitor across the load connected from each output to ground, five total components.

18. At high power the LM4651/52 seems to be going into some protection mode?

If driving a load that is smaller than 4Ω the current limit protection may be turning on at higher output power levels. The current limit set by R_{SCKT} is the sum of the peak output current in the load and the ripple current in the inductors. For a 4Ω load and 125W output the peak current in the load is 7.9 amps. The ripple current is affected by the inductor value, supply voltage and switching frequency. Some head room in the current limit should be designed in when setting the value of R_{SCKT} . Normally R_{SCKT} is set to $100k\Omega$ giving 11Amps of output current. Lowering this resistor will increase the peak output current allowed from the FETs.

This symptom can also be due to noisy or low supply lines. Under voltage detection is activated when either supply is within 10.7 Volts of GND. If the supply is unregulated and momentarily droops near this level UVD may be activated unintentionally. Increasing the capacitors on the supply lines will often solve this problem.

19. What is the gate capacitance of the LM4652 FET H-bridge?

The gate capacitance is typically 1000pF on each FET contained in the LM4652.

20. What is the V_T of the FETs in the LM4652 H-bridge?

The V_T of the FETs in the LM4652 FET H-bridge is right around 1.0 Volts.

21. What is the ripple current in the inductors on the demo board?

The ripple current is approximately 0.8 Amps in each inductor giving 1.6 Amps total ripple current.

22. How to I calculate the ripple current for different inductor values?

Ripple current depends on the value of the inductor, the frequency of modulation (switching frequency) and the supply voltage. The general equation for calculating ripple current is: $\frac{1}{4} (V_{cc}/L)(1/2*SWF)$.

23. What is the output impedance of the LM4652 FETs?

It would be the $R_{DS(on)}$ of one of the transistors since at any time one of them is fully on. The value of $R_{DS(on)}$ depends on the temperature of the die. At room temperature (25° C) the $R_{DS(on)}$ of the FETs in the LM4652 is typically 200m Ω .

24. What is the output impedance of the LM4651 small signal FETs?

It would be the $R_{DS(on)}$ of one of the transistors since at any time one of them is fully on. At room temperature (25° C) the $R_{DS(on)}$ of the FETs in the LM4651 is typically 1.0 Ω .

23. What is the forward voltage on the body diode in the LM4652 FETs?

It had not be measured exactly but from the die design the forward voltage is around 0.55 V to 0.65 V.

25. Does the Thermal Shut Down output from the LM4652 an analog signal or a digital?

The output of the TSD pin on the LM4652 is at zero volts until the die temperature reaches about 140° C. The voltage then starts to increase quickly as the die temperature continues to increase to over 150° C. When the voltage reaches 5V the LM4651 goes into STBY mode. The TSD output will go as high as 6V. When the die temperature cools below 150° C the TSD voltage transitions down to less than 2V.

26. Can the Lm4651/52 be used for frequency ranges other than subwoofer?

Yes. The audio frequency range is controlled by the external components and is not limited by the LM4651/52 chip set. Several external values need to be changed to increase or decrease the audio frequency range of operation. To increase the audio frequency range of operation the following values must change: L_1 and C_{BYP} (the output filter) must decrease, C_{f11} , C_{f12} and C_f may need to decrease but normally this is not the case, R_1 , R_2 , and R_f may need to change to lower noise and gain to meet target specifications, R_{OSC} may need to decrease, R_{DLY} will need to be reduced or set to 0 Ω . A pre-amplifier with a gain of 10V/V and a 2 pole active input filter is recommended. The input filter will control the audio frequency range while the output LC filter will be set to high levels to allow the frequency range of interest to have a flat, linear response through the output filter. This pre-amplifier/filter can be implemented easily with a low cost dual op. amp. like the LM833.

27. The lower -3dB point is not low enough for my application. How can I decrease the lower -3dB point for more bass response?

The lower 3dB point is controlled by the relationship of C_{in} , R_1 and R_p on the demo boards. Increasing the value of C_{in} is the easiest way to fix this problem. Alternately, R_1 or R_p can be increased but changing these values will effect the gain and the upper 3dB point of operation.

28. Can the LM4651/52 be used with a single supply instead of a dual supply?

Yes. The performance with single supply is the same as with dual supply. The single supply voltage minimum is double the dual supply minimum. The dual supply minimum is $\pm 11\text{V}$ so the single supply minimum is 22V . A reference voltage of 12V up to V_{cc} must be provided as the GND connections to the LM4651/52 chip set. Vee connections then become 0V (single supply GND) and Vcc connections are the single supply voltage. This voltage reference can be created with a resistor divider, a resistor and a Zener diode or a voltage regulator. Bypass capacitors will need to change connections with capacitors now added on the reference voltage to the single supply GND. The voltage rating of these capacitors will need to be higher than a dual supply configuration.

29. When using two or more LM4651/52 chipsets in a system an output signal is heard when there is no input signal. What is the cause and how can it be eliminated?

When using two LM4651/52 class D chipsets the modulation frequency or switching frequency (SWF) of each chipset is independently set by the R_{OSC} resistor connected between the OSC pin (pin 16) and GND. Due to resistor variation, fab process variation and the operating temperature the SWF of each chipset will vary slightly. This difference between the SWF of each chip results in a "beat frequency" that occurs in the audio band. For example, if the SWF is set to 100kHz for each chip and there is just a 0.1% difference then it is possible a 100Hz beat frequency will be heard when there is no input signal. The most likely cause of the beat frequency is due to the output filter inductors coupling between channels. Coupling between PCB traces can also be a cause or very poor supply bypassing on the power lines. Possible solutions are listed below:

A. The simplest and lowest cost solution is to set the SWF far enough apart to eliminate any possible beat frequency from developing in the audio band. It is recommended that the SWF be set 25kHz apart. For example, 75kHz and 100kHz or 90kHz and 115kHz , etc. If more than two channels are used then each channel should have it's SWF set 25kHz higher or lower than the other channels. If the application is for subwoofer only then a lower difference can be used due to the output filter helping to reduce the signal level of the beat frequency and the frequency response of the speaker itself acting as a natural low-pass filter (10kHz spacing, for example). The minimum amount of difference between the SWF of each channel would have to be determined by the application. The only negative impacts of using different SWF for subwoofer applications are a small decrease in clipping output power (1% THD power point) and a slight decrease in efficiency for the channel set to the higher SWF. For full band applications the THD

performance at higher audio frequencies might be negatively impacted for the channel with lower SWF.

B. Another solution is to reduce any coupling between channels. Solutions could be as simple as using toroid inductors in the output filter stages for lower coupling. Designing the PCB so there is more physical distance between the two channels will also aid in coupling. Shielding around the output stage filters and around the LM4652 IC may also reduce or eliminate the coupling.

C. The SWF is set by a voltage that is developed across the R_{OSC} resistor. The difference in SWF from IC to IC when using the same value of R_{OSC} is in the range of 1 – 3%. The corresponding voltage for a given frequency will vary from LM4651 IC to LM4651 IC so forcing a fixed voltage (1.5V for example) across R_{OSC} is not a possible solution as there will still be a 1 – 3% variation in SWF. However, since a voltage can be forced on to the OSC pin (pin 16) as long as R_{OSC} is not removed, an error signal in the form of a voltage can be used to synchronize the switching frequencies of all LM4651/52 chipsets in a given system. The next two solutions are theoretical only. They have not been tested on the bench.

1) Using a Phase Lock Loop (PLL) the SWF of one LM4651/52 chipset can serve as the master and the error signal from the PLL can be used to control the other LM4651/52 chipsets as slaves. Possible problems occur when an audio input signal is present. Perhaps a high-pass filter would be needed at the input of the PLL to remove the audio pulse width modulation portion of the output waveform.

2) Another possible solution is to create an error signal using a differential op. amp. The low gate drive signals coming from the LM4651 (pins 25 & 26) are 6V signals in amplitude (from $-20V$ to $-14V$ in a $\pm 20V$ system). LG_1 and LG_2 are exactly 180 degrees out of phase. LG_1 from one LM4651 IC and LG_2 from the other LM4651 IC (in a two channel system) can be added and filtered to produce an error signal. When the two switching frequencies are in phase a zero error signal will be produced. The error signal would need to be adjusted to be in the correct range for the LM4651 OSC pin. A low-pass filter, a resistor (R_{OSC}) and capacitor would probably be sufficient to create a smooth and slow changing error signal. When the OSC pin is tied directly to GND the SWF is 210kHz – 220kHz. At 1.0V the SWF is approximately 160kHz, at 1.4V the SWF is approximately 115kHz, and at 2.0V the SWF is approximately 60kHz. The same high-pass filter as describe above may be needed to remove the audio portion of the signal.