



Get Started with DesignKit

Class D Audio Amplifier Using IRS2092

Contents

1.	DesignKit Simulations folders	3
2.	How the initial condition are set?	4-5
3.	Example of Using Design Kit	6
4.	How to Estimate Design %Efficiency?	7-8
5.	How to Estimate Output THD?	9-11
6.	How to Estimate Frequency Response?	12-13
7.	How to Create Reference Waveforms?	14-15
8.	Change R_{IN} (R2) and simulate to see change in G_{V}	16-17
9.	Use Design Kit to select proper VR value	18-19
10.	Use Design Kit to Predict Spike Voltage vs. Dead-time setting	20-21
11.	Use Design Kit to Develop the Design (Change the FETs)	22-23
12.	MOSFET Professional Model	24

Slide #

1. DesignKit Simulations folders

rCAD Capture C	IS					
Edit View Place M	acro PSpice	Accessories Op	tions Window Help			
28 a X h f	1 2 2 1	RFIZ24N		& 	7 0 R I I I	E 🗈 🛍 🤽 🦉
tUp-StartUp	*	🕨 🦻 🦻 🕈	$\mathfrak{A} \gg \mathbf{V} \neq \mathbf{I}$	± y, W w∤		
D:\2009\Design}	(it\DAUDA ■ / - (Sta Open	MP\Simulat	ions 💶 🗖 (tUp(Page1))		?×	
Startup-psp Startup-psp Startup-psp Startup-psp Simulation Pr Simulation Pr Startup-ssp Simulation Pr Startup-ssp Simulation Pr Startup-ssp Simulation Pr Startup-ssp Startup-ssp Startup-ssp Simulation Pr Startup-ssp Startup-ssp	Look in My Recent Documents Desktop My Documents My Computer	Simulations DT Efficiency FET FET(STD) FFT(STD) FFT(STD) FreqResp Gv Loss Meas OSC Short ShrtRsp StartUp Stress THD	₩F ₩VF00			R3 47k 601m 0.01m 0.01m C 02 FR1 4.7k 4
		File name:		~	Open	8.2k = 10 = Tkm
	My Network	Files of type:	Orcad Project (*.opj)	R6 8.2k OCS	Cancel	
			0 items	selected		Scale=100%
			O Nerris	selected		Scale=100%
	<	. p	Concession (1984)	N 13 80		

- Ready to use simulation projects
 - Test conditions are set and easily changeable.
 - Appropriate simulation settings and Initial Condition (.IC).
 - Option setting is done without convergence problem.
 - Libraries are included and added.
 - Simulation results (ex. Power and %Efficiency) are calculated and displayed.

2. How the initial condition are set?

- 1. Open Project: ...¥Simulations¥StartUp¥StartUp.opj .
- 2. Set initial value of charged-up capacitors (C2, C3, C7, C8, C9, and C10) to be zero (IC=0).
- 3. Run the simulation (0-1sec. or until circuit is startup).

ile Edit View	Place Macro PSpice Accessories	Options Window Help		
) 6 2 2	🛞 🖻 🖻 😫 😂 🛛 IRFIZ24N	V 8 8 9 8	い ロ ジ 的 図 田 三致 [1 1. 8
StartUp-StartUp	💌 🏝 🗖 🕨 🖗 🖗 🖗	PP 🧟 🖉 V 🕮 I 🦮 W 🛸		
D:\2009\	DesignKit\DAUDAMP\Simul	ations 🔳 🗖 🗙		
— N / -	(Startlin : Startlin(Page 1))		
File	s	•	3	- 1
		Display Properties	X	
	VOFF = 0 VAMPL = 0 FRE0 = 1k	R8 Value: Display Format Do Not Display Value Only Value Only Value Only Value Only Name and Value Name Only Both if Value Exists C1 IC In C7 C20 VSS VREF R8 2.2k OCSET R7 I 22k R7 I 22k	Font Arial 7 Use Default Color ✓ Default ✓ Rotation ○ 0~ 180~ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 180~ ✓ ○ 0~ ○ 270~ ✓ ○ 0~ ○ 270~ ✓ ○ 0~ ○ 270~ ✓ ○ 0~ ○ 270~ ✓ ○ 0~ ○ 270~ ✓ ○ 0~ ○ 3.3k ○ ○ 0 ○ 3.3k ○ ○ 10 ○ 3.3k ○	R9 4.7k MUR120RL6 D1 R13 R14 4.7 R15 10 1 20 20 20 20 20 20 20 20 20 20 20 20 20

2. How the initial condition are set?

- 4. Initial conditions are the startup voltage at each capacitor.
- 5. Change the IC values ,then run the simulation (0-100usec. with maximum time step 10nsec.).





3. Example of Using Design Kit

Estimate design specification.

- ✓ %Efficiency.
- ✓ %THD.
- Frequency response.
- Create reference waveforms.
- Change the design parameters and simulate to see results.
- Component stress test.
- Simulate switching losses.
- Simulate Short-circuit scenarios.

4. How to Estimate Design %Efficiency?

- 1. Open Project: ...¥Simulations¥Efficiency¥Efficiency.opj .
- 2. Enter test condition parameters: P_0 =25W, G_V =15.85(24dB), R_L =4ohm, and f_{in} =1kHz.
- 3. Run the simulation from 1 to 3 ms. (about 3×1kHz output cycles)



4. How to Estimate Design %Efficiency?

4. Add traces: "AVG(W(LOAD))" for PO[W],

"-(AVG(W(+B))+AVG(W(-B)))" for Supply power [W], and

"-100*AVG(W(LOAD))/(AVG(W(+B))+AVG(W(−B)))" for %Efficiency



5. How to Estimate Output THD?

- 1. Open Project: ...¥Simulations¥THD¥THD.opj .
- 2. Enter test condition parameters: P_0 =10W, G_V =15.85(24dB), R_L =4ohm, and f_{in} =1kHz.



5. How to Estimate Output THD?

3. THD is calculated by checking the box "Perform Fourier Analysis" in the Output File Options setting. Center Frequency is 1kHz same as f_{in} and "V(OUT)" is the Output Variable(s).

	V I W	· · · · · · · · · · · · · · · · · · ·		
Simulation Settings General Analysis Configuratio Analysis type: Time Domain (Transient • Options: © General Settings Onorle Carlo-Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Goad Bias Point Save Restart Simulation	For THD n Files Options Data Collection Piles Bun to time: 3ms Start saving data after: 0 Iransient options Maximum step size: 100n Maximum step size: 100n s If guin in resume mode Image: Start saving data after: 1000	seconds (TSTOP) seconds econds int calculation (SKIPBP) Output Ele Options Transient Print values in t I♥ Perform Fox	2 20nH R17 LST CTI MM FFTT PERTIN 1 CTI MM PERTIN 1 CTI MM 20nH LS2 20nH LS3 t Output File Opt the output file every:	
	OK Cancel	Apply Number of H	Harmonics:	23
		Output Vari Include det controlled s	tables: V(DUT) tailed bias point information for n sources and semiconductors (/D	onlinear P)

5. How to Estimate Output THD?

- 4. Run the simulation 0 to 3ms. (maximum time step 100ns.).
- 5. View Output File to see the simulated result THD(%)



* Please note that the simulated result is only an estimate of %THD and the value is influenced by maximum step size.

6. How to Estimate Frequency Response?

- 1. Open Project: ...¥Simulations¥FrqRsp¥FreqResp.opj.
- 2. Enter test condition parameters: $V_{OUT}=2V$, $G_V=15.85(24dB)$, $R_L=4/8$ ohm, and $f_{in}=20kHz$.
- 3. Run the simulation from 0 to 2 ms. (about 40×20kHz output cycles). Use Parametric Sweep (Global parameter: RL with value = 4 and 8)

8 % D	6 22	- 🔍 🔍 🔍 🧐 印 紅学 創 阿 🏾 🔅 🖸 🛍 🤽 🌹	
IC1-FqRsp	💽 🎽 🗖 🕨 🦻 🖉 🖓 🖗 🖗	\mathfrak{P} V \cong I \hookrightarrow W \cong	
5	•	3 2	
			-
		R3 VS	清
	RB	471× +B	÷
	820	Simulation Settings - FqRsp	
		General Analysis Configuration Files Options Data Collection Probe Window	
		Analuit han	
		Time Domain (Transient C Voltage source Name	_
	75 R4 VR1	VAAT C Quirent source	_
		<u>Options:</u> Global parameter	_
INO	C2 C1 100 K2 = 100 11 000	CS Model parameter Model name:	_
₹ R1 10		InF COMP Parametric Sweep Carameter name: RL	
1	10u (C7	Vss Save Bias Point Sweep type	
=0	IC = 14 10u	VREF Load Bias Point C Linear Start value	
= {tin}	Real Provide P	6 8.2k OCSET Restart Simulation C Loganthmic Decade * End value:	
CONDITION:	*	R7 [ncrement:	
= 20k		r Value ligt ↓ 8	_
= 8 = 15.85	BK		
JT = 2			
	820	OK Cancel Apply	Help

6. How to Estimate Frequency Response?

4. Add traces: "DB(RMS(V(OUT))/2)" for the frequency response of $2V_{RMS}$ output in dB.



7. How to Create Reference Waveforms?

- 1. Open Project: ...¥Simulations¥Waveforms¥Waveform.opj .
- 2. Enter test condition parameters: V_{OUT} =2V, G_V =15.85(24dB), R_L =4ohm, and f_{in} =1kHz.
- 3. Run the simulation from 100n to 3 ms. (about 3×1kHz output cycles)
- 4. Put the Voltage/Level Marker (or Current Marker) to see the waveform(s).



7. How to Create Reference Waveforms?

- 4. Set X and Y Data Range in Axis Setting according to oscilloscope scale.
- 5. Use simulated waveforms as reference to compare with real circuit waveforms.



8. Change R_{IN} (R2) and simulate to see change in G_V

- 1. Open Project: ...¥Simulations¥Gv¥Gv.opj.
- Enter test condition parameters: V_{IN}=100mV_{RMS}(0.14142V_{PEAK}), R_{IN}=2.4 / 3 kohm, R_L=8 ohm (speaker), and f_{in}=1kHz.
- 3. Run the simulation from 0 to 1 ms. (about 1×1kHz output cycles). Use Parametric Sweep (Global parameter: RIN with value = 2.4k and 3k)



8. Change R_{IN} (R2) and simulate to see change in G_V

- 4. Simulated result shows Vo with different gain G_V .
- 5. Change parameter: RIN until you get a satisfied result.



9. Use Design Kit to select proper VR value

- 1. Open Project: ...¥Simulations¥OSC¥Waveform.opj.
- 2. Input voltage: V_{IN} =0, R_L =8 ohm (speaker).
- f_{OSC}=400kHz is chosen for this design, C4=C5=1nF, R4=220, a variable resistor: VR1 value will be varied.



9. Use Design Kit to select proper VR value

- 4. Change VR1 value until simulation result with f_{OSC} =400kHz (VR1=750hm).
- 5. Choose VR1 that value more than 100 ohm for the design (this time VR1:1k is chosen).



Simulated waveform with VR1: Value=75

Measured waveform from real circuit using VR1: 1k

10. Use Design Kit to Predict Spike Voltage vs. Dead-time setting

- 1. Open Project: ...¥Simulations¥DT¥Waveform.opj.
- 2. Input voltage: V_{IN} =0, R_L =8 ohm (speaker).
- 3. Select dead-time setting DT1: R_{20} =3.3k / R_{21} =8.2k, Simulate and compare result with dead-time setting DT3: R_{20} =8.2k / R_{21} =3.3k



10. Use Design Kit to Predict Spike Voltage vs. Dead-time setting

4. Compare the results to see that spike voltages are acceptable or not for each dead-time setting.



11. Use Design Kit to Develop the Design (Change the FETs)

- Use the simulation files for the performance evaluation (ex. Efficiency, THD, and Waveform). 1.
- 2. Replace MOSFET model IRFIZ24N with IRFI4024H-117P.
- 3. Run simulation file to check the design performance.



V

А

mΩ

nC

ns

ns

nC

11. Use Design Kit to Develop the Design (Change the FETs)

3. Compare the performance of the circuit with difference FET.



	IRFIZ24N	IRFI4024H-117P
Efficiency (@ 25W, 4 Ω)	93.505%	94.578%
Distortion (@ 1kHz, 4Ω , 10W)	0.0144 %THD	0.0201 %THD

12. MOSFET Professional Model

- Library and symbol files are in folder ...¥Parts¥IRFIZ24N¥IRFIZ24N(PRO)
- IRFIZ24N Professional Model consists of MOSFET Professional (MIRFIZ24N_P), body diode DIRFIZ24N, and body diode Professional DIRFIZ24N_P.
- Use MOSFET Professional model to improve an accuracy Gate Charge characteristics Q_g of FET model.
- Use body diode Professional model to improve an accuracy Reverse Recovery Time characteristic T_{RR} of FET's body diode model.



* Using Professional model will slow down the simulation time and might cause some convergence error.