

APPLICATION NOTE

AN1613 Rev.0.00

Dec 8, 2010

From SPICE Netlist to Allegro Design Sub-Circuit

Introduction

Intersil provides a SPICE model for all our new precision Opamps. The SPICE model netlist is included in the data sheet, along with simulation vs characterization curves. Refer to <u>AN1556</u> for more details about making SPICE models.

This application note will walk the user through the process of taking the netlist from the data sheet and creating a sub-circuit to drop into a Cadence Allegro Design simulator.

Copying the SPICE Netlist

Download the Intersil data sheet from the web. The data sheet will be in .pdf format. Open the .pdf document and right click to enable the select tool, if it is not already selected (Figure 1). This will enable you to then copy and paste the entire list into Notepad.

ISL28114, ISL28214, ISL28414

* source ISL28114_SPIC * Revision C, LaFontaine * Model for Noise, supply	Emodel October 7th 2009 currents, CMRR 72dB f=80kHz ,	AVOL 90dB	R_R11 R_R12	16 V++ 1 TC=0,0 V- 16 1 TC=0,0
* SR = 2.5V/us, GBWP {	🗸 📐 Select Tool		¹ 2nd Gair G G3	1 Stage V++ VG 16 VMID 24.893e-3
short ckt I limit *Copyright 2009 by Inte *Refer to data sheet "LI	⁽¹⁾ <u>H</u> and Tool ⁽¹⁾ Marquee <u>Z</u> oom		G_G4 V_V5 V_V6	V- VG 18 VMID 24.893e-3 19 VG .604 VG 20 .604
*this model indicates yo *terms and provisions in	Previous View	Alt+Left Arrow	D_D5 D_D6	19 V++ DX V 20 DX
* Connections:	<u>R</u> otate Clockwise	Shift+Ctrl+Plus	R_R13 R_R14	VG V++ 636.658e3 TC=0,0 V- VG 636.658e3 TC=0,0
:	🖶 Print	Ctrl+P	C_C2 C_C3	VG V++ 2E-09 TC=0.0 V VG 2E-09 TC=0.0



Scroll towards the end of the datasheet and find the SPICE netlist (Figure 3) and copy it into notepad. Name the file with the extension **.MOD** (not case sensitive) as shown in Figure 2. This file needs to be saved in a common directory with all the other SPICE files for this design.



FIGURE 2. SAVING NOTE PAD FILE AS . MOD

ISL28114, ISL282	214, ISL28414
Lawren (D) 05114 0D(OEmotio)	B BH (811) (70-00
* Revision C. J. a Footblan Ortobar 7th 2000	R R12 V- 16 1 TC=0.0
* Model for Noise supply currents CMRR 72dB f=80kHz AVOI 90dB	10110-00
f=125Hz	"2nd Gain Stage
* SR = 2.5V/us, GBWP 5MHz, 2nd pole 6MHz Output voltage clamp and shart att Limit.	G_G3 V++ VG 16 VMID 24.893e-3
Short dat Filmt	G_G4 V-VG 16 VMID 24.893e-3
"Befor to data choot "LICENCE STATEMENT" Lico of	V_V5 19 VG .604
"this model indicates your accentance with the	V_V6 VG 20.604
farms and provisions in the Linense Statement	D_D5 19 V++ DX
* Connections: +input	B B13 M2 VAA 636 65943 TC=0.0
-input	R R14 V=VG 636 658e3 TC=0.0
+Vsupply	C C2 VG V++ 2E-09 TC=0.0
- Vsupply	C_C3 V-VG 2E-09 TC-0,0
• output	· -
	*Mid supply Ref
Source ICI 28114 DC mud	E_E4 VMID V- V++ V- 0.5
source is the intervention of the intervention	E_E2 V++0V+01
"Voltage Noise	E_E3 V=0V=01
E_En VIN+ EN 28 0 1	100 00000
D_D13 29 28 DN	"Common Mode Gain Stage with Zero
V_V9 290.00035	G_G5 V++ VC VCM VMID 2.5118E-10
R_R21 28 0 500E3 TC=0,0	G_G6 V-VC VCM VMID 2.5118E-10
Name diama	E_EOS 1 EN VC VMID 1
Input stage	R_R15 VC 21 1e6 TC=0,0
M_M14 3133 NCHANNELMOSFET	R_R16 22 VC 1e6 TC=0,0
M M16 11 VIN-99 PMOSISIL	R_R22 EN VCM 5611 TC=0,0
M_M17 12 1 10 10 PMOSISIL	1 1 21 Vaa 1 9895
L 11 7 V- DC 5e-3	L L2 22 V- 1.9895
1_12 V++ 8 DC 5e-3	-
LOS VIN-1 DC 25e-12	"Pole Satge
G_G1A V++ 14 4 3 1404	G_G7 V++ 23 VG VMID 376.98e-6
G_G2A V=1411121404	G_G8 V-23 VG VMID 376.98e-6
V V2 13V-1e6	R_R17 23 V++ 2652.66 1C=0,0
R R1 32 1.0004 TC=0.0	C C4 23 Vet 10e 12 TC=0.0
R_R2 4 2 1.0004 TC=0,0	C C5 V-23 10-12 TC-0.0
R_R3 57 10 TC=0,0	
R_R4 7610TC=0,0	*Output Stage with Correction Current Sources
R_R5 98 10 TC=0,0	G_G9 25 V- VOUT 23 0.02
R R5 810 10 10 -0,0	G_G10 27 V-23 VOUT 0.02
R 88 1312 1TC=0.0	G_G11 VOUT V++ V++ 23 0.02
R RA1 14V++ 1TC-0.0	G_G12 V=V00123V=0.02
R_RA2 V-14 1 TC-0.0	V V8 VOUT 25.08
C_CIND# VIN-EN 1.02E-12 TC=0,0	D D7 2324 DX
C_CIn1 V-EN 1.26e-12 TC=0,0	D D8 25 23 DX
C_Cin2 V-VIN- 1.26e-12 TC=0,0	D_D9 V++ 26 DX
Net Only Orac	D_D10 V++ 27 DX
15L Gam Gage	D_D11 V-26 DY
G G2 V-16 15 VMID 334 753e-3	U_U12 V= 27 DY
V V3 17 16.61	
V_V4 16 18.61	model propisitiones (kp=15e-3 vtp=-0.5)
D_D1 15 VMID DX	model NCHANNELMOSFET mos (kp=3e-3 vto=0.6)
D_D2 VMID 15 DX	.model DN D(KF=6.69e-9 AF=1)
D_D3 17 V++ DX	MODEL DX D(IS=1E-12 Rs=0.1)
D_D4 V-18 DX	MODEL DY D(IS=1E-15 BV=50 Rs=1)
R_R9 1514 100 10-0,0	.ends ISL28114subckt
S S IV IS YOUL IN IVELU	

12 intersil

FIGURE 3. NETLIST FROM DATA SHEET

FN6800.4 October 8, 2010

Model Editor

Open the Cadence model editor via the path shown in Figure 4 (Cadence SPB 16.2\AMS Simulator\Simulation Accessories \setminus Model Editor)

FIGURE 21. SPICE NET LIST

	Pliciosofic Office			AMS Simulator			Simulation Accessories		240	Magnetic Parts Editor
	Oracle 8i	۲	(in)	PCB Editor Utilities	Þ	5	AMS Advance Analysis		e	Model Editor
	Oracle - ORANT9I	٠		Tutorials	۲	<u>الله</u>	AMS Simulator		28	Simulation Manager
	Oracle Installation Products	•		What's New in Release 16.2	×	Г		-	1	Stimulus Editor
•	Reflection Web	•		Cadence Help						
	Report2Web PageMart Viewer	۲		Design Entry CIS						
	Startup	×	R	Design Entry HDL						
	WinZip	•	*	Design Entry HDL Rules Checker						
入	Adobe Reader 9		2	Layout Plus						
6	Internet Explorer			Layout Plus SmartRoute Calibrate						
1	Jinitiator Clear Cache		۵Į	Library Explorer						
٩,	JInitiator Control Panel 1.1.8.19		遇	License Client Configuration Utility						
62	Microsoft Office Communicator 2007 R2		1	Model Integrity						
3	Outlook Express			Package Designer						
Ð	Remote Assistance		×	PCB Editor						
Θ	Windows Media Player		6	PCB Router						
18	Windows Messenger		×	PCB SI						
۹.	Windows Movie Maker		×	Physical Viewer						
•••	Cisco Systems VPN Client	•	₽.	Project Manager						
	Dell ControlPoint	•		README CCR						
	Mathcad	۲	ing Nelson	SigXplorer						
	Roxio Creator DE	۲		SIP		L				
0	Using Intersil WIFi		Z	SIP Digital Architect						
	Adobe FrameMaker 9	•	Z	System Architect						
Control	Cadence SPB 16.2	•	Ð	Uninstall Cadence SPB 16.2						

FIGURE 4. PATH TO CADENCE MODEL EDITOR

Note: This document is written using the SPB16.2 software. The look and feel may change with different revisions of the Cadence software, but the procedure will be the same.



After selecting the Model Editor, the screen in Figure 5 will open up. Select Capture and click DONE.



FIGURE 5. SELECT DESIGN ENTRY TOOL

Click on File in the tool bar and select New. Figure 6 will appear.

🛗 Untitled1.lib - AMS Model E	litor	- 🗆 ×
File Edit View Model Plot Tools	Window Help	cādence
🖻 🗁 🖬 🖴 🖪	XOB QQQQ 🖬 XM 🖻	
Models List 🗶		
Model Name Type Modifier		

FIGURE 6. BEGINNING OF NEW MODEL

Click on Model in the tool bar and select Import. Then browse to the folder where you put the .MOD file. Figure 7 will appear. Select the desired .MOD file and click Open.



FIGURE 7. SELECT .MOD NETLIST

This will load the netlist into the Model editor tool as shown in Figure 8.

178		
Untitled1.lib:ISL28114subckt -	AMS Model Editor - [Model Text]	- 🗆 ×
File Edit View Model Plot Tools	Window Help Cadence	_ 8 ×
Models Lat X Model Name Type Mos 152.0114sb SLBCXT	<pre>* source ISL20114_SPICEmodel * Revision C, LaFontaine October 7th 2009 * Model Ico Noise, supply currents, CHNR 72dD f=00kHz , AVoL 90dB f=125Hs * 25.5Vus, GUWP SHHz, 2nd pole GHHZ Output voltage classp and whort oth I limit "copyright 2009 by Intersil Corporation "Refer to data abset "LICENSE STATEMENT" Use of "this model indicates your acceptance with the terms and provisions in the License Statement. * Connections: +input * -iAuput * -iAuput * -iAuput * -Vaupply * -vaupply * -vaupply * -vaupply * output * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * 0 utput * </pre>	*
	*Input Stage M Mi4 3 1 5 S NCHIMNELMOSFET M M15 4 VIN- 6 6 NCHANNELMOSFET	-

FIGURE 8. NETLIST LOADED INTO MODEL EDITOR

Click on File in the tool bar and select Save As. Then type the part name as the file name in Figure 9 and click Save. The file with the complete netlist is now saved as a .lib library file.

Save As				? ×
Save in	C SPICE Mode	Folder	💽 🕝 🤌 🕑 🗖	
My Recent Documents Desktop My Documents	ISL28114.lb			
My Computer: NBK8QJBPH1				
My Network	File name:	ISL28114.lib	•	Save
	Save as type:	Model Library Files (*.lib)	•	Cancel

FIGURE 9. FILE SAVED AS .lib

Click on File in the tool bar and select Export to Capture Part Library. The Input Model Library path and the Output Part Library path will automatically be loaded as shown in Figure 10.

Create Parts for Library	×
Enter Input Model Library:	
ppnotes\AN1613\SPICE Model Folder\ISL28114.lib Browse	1
	-
Enter Output Part Library:	
pnotes\AN1613\SPICE Model Folder\ISL28114.olb Browse	1
OK Cancel Help	

FIGURE 10. LIBRARY CREATION

Verify that the files paths are the same with the only difference being the .lib and .olb extensions.



Click OK and verify no Error messages or Warning messages as shown in Figure 11. Click OK.



FIGURE 11. SCHEMATIC TO CAPTURE TRANSLATOR CHECK

Click on File in the tool bar and select Import Wizard [Capture]. Like before, both path names will load automatically and should have the same file paths with the only difference being the .lib and .olb extensions as shown in Figure 12.



FIGURE 12. ASSOCIATE SYMBOL WITH SPICE MODEL

Click Next and the screen shown in Figure 13 will appear. This is the screen in which we will associate the pins of our SPICE model to the pins of the sub-circuit model. The symbol shown is a generic 5 pin device. We want our Opamp symbol to look like an Opamp. To do this click on the Replace Symbol button and select from the list of symbols provided with the Cadence program. This list is located at the following location on your C drive. C:\Cadence\SPB.16.2\tools\capture\libary\OPAmp.olb



FIGURE 13. REPLACE GENERIC SYMBOL

If the location of your Cadence software was loaded in a different location, then search for Cadence \SPB.

When selecting your symbol, all that matters is the pin count. The numbers assigned to the symbol pins can be changed later. Just scroll through the list to find a symbol that matches a desired pinout and pin count of your device. In this example, we selected the TLC2201. Click Next.

Model Import Wizard : Select Matching					
	Select library to pick matching symbols : [C:\Cadence\SPB_16.2\tools\capture\library\OF	'Amp.olb	.		
	Model : ISL28114SUBCKT		Symbol : TLC2201		
	Matching Symbols TLC2201 LMC6492 LM25492 L2150C-W EL2150C-W L2725C LM127D LM754 LM127D LM1750L MAX430 MAX430		3 2 4 6 4		
View Model Text	< Back	Ne	xt> Cancel Help		

FIGURE 14. ASSOCIATE OP AMP SYMBOL WITH MODEL

Then click on the row under the Symbol Pin column to activate pull down menu box under the symbol column. Now pick the associated pin to match the Model Terminal function in the model terminal column. As shown in Figure 15.



FIGURE 15. DEFINE PINS OF SYMBOL TO PINS OF MODEL

Repeat for all Model Terminal pins as shown in Figure 16.



FIGURE 16. ALL PINS ASSOCIATED TO SYMBOL

Click Save Symbol, and Figure 17 will appear. Verify no Error messages or Warning messages appear. Click OK and then close the Model Editor.

Model Import Wizard : H:\AMS Files\1 Precision and MicroPower Amplifiers\ 🗙					
Log File Error File					
STATUS: PSpice Model Import Wizard for "Capture" (16.2.0.s003) STATUS: INFD: LBB driven flow INFD: Input Flie: H:VAMS Files\1 Precision and MicroPower Amplifiers\Appnotes\AN1613\SPICE INFD: Output Flie: H:VAMS Files\1 Precision and MicroPower Amplifiers\Appnotes\AN1613\SPICE STATUS: STATUS: STATUS: INFD: Symbol "ISL28114SUBCKT" already exists for model "ISL28114SUBCKT". STATUS: STATUS: STATUS: Completed identifying matching symbols automaticallyat Tuesday, November 09, 2010 STATUS: INFD: Symbol "ISL28114SUBCKT" updated for model "ISL28114SUBCKT". STATUS: 0 Error messages, 0 Warning messages					
OK					

FIGURE 17. MODEL IMPORT WIZARD CHECK

Using the New Sub-Circuit to Run Simulations

Open the Cadence Software. Figure 18 shows the path to select the Design Entry CIS.



FIGURE 18. PATH TO DESIGN ENTRY CIS

Figure 19 shows the Cadence Product Choices. Select Allegro Design Entry CIS and Click OK.

Cadence Product Choices	
Please select the suite from which to check out the OrCAD Capture	feature:
Allegro Design Entry CIS Allegro PCB Design CIS L Allegro PCB Librarian XL Allegro PCB Design CIS XL	OK Cancel
🔲 Use as default	

FIGURE 19. CADENCE PRODUCT CHOICES

Click on File in the tool bar and select New, and then Project. The screen shown in Figure 20 will appear. Type in the name of the project and select Analog of Mixed A/D button. Browse to where you saved the Netlist in the common directory (you must have all the files located in the same directory) and click OK.

×
OK Cancel Help Tip for New Users Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
s\PSPICE Browse

FIGURE 20. SCREEN TO SETUP NEW PROJECT

The screen shown in Figure 21 will appear. The user can select to base their new project on an existing project or start a new one. Selecting to base upon an existing project will carry over the existing project with all the simulation profiles and schematics. This can be a real time saver if the new project is very similar to an old project. In this example, we will chose to create a new project.

Create PSpice Project	×
C Create based upon an existing project	OK
H:\AMS Files\1 Precision and MicroPower Amplifiers\PSPICE model	Browse
Create a blank project	Cancel
	Help

FIGURE 21. CREATING A NEW PROJECT OPTIONS



Click OK, and the screen in Figure 22 will appear. Click on the SCHEMATIC1 to open the Page tab and then Click on the page tab to open the Schematic1 page. This is where the new sub-circuit will be placed to run the simulations.

Allegro Design Entry CIS		
File Edit View Tools Place Macro PSp	ice Accessories Options Window Help	
15120114 PAGE1		
🖬 H: WMS Files M Pre 🗕 🗆 🗙	S /- (SCHEMATIC1 : PAGE1)	- 🗆 🗙
Analog or Mixed A/D		
File 4. Hierarchy		111 -
E- Design Resources		
- 13228114_an1613.dsn		
B-CO SCHEMATICI		
PWGE1		
Theory		
Contracts		
F D PSpice Resources		
	f	
	•	111
		111
		111
		C
	•	111
		333
		100 -
	[•	► //.

FIGURE 22. SIMULATION SCHEMATIC PAGE

Before we can place the new sub-circuit model and run a simulation, we need to set-up the simulation profile and add the library. Click on PSpice in the tool bar and select New Simulation Profile. Figure 23 will appear. Then type in any name that will help you keep track of the different simulations. Then click Create and Figure 24 will appear.

New Simulation	×
Name:	Create
AC Analysis	Groate
· · · · -	Cancel
Inherit From:	
none 💌	
Root Schematic: SCHEMATIC1	

FIGURE 23. NAMING SIMULATION PROFILE

Click the Configuration Files tab. Then click on Library in the Category field (highlighted in blue). Browse to where to saved the Library file. Then click the Add to Design button. The Simulation Settings screen should look like that shown in Figure 24 with the file path name being the location of the common directory. Click the Apply button.

Simulation Setting	gs - AC Analysis	×
General Analysis Category:	Configuration Files Options Data Collection Probe Window Details Filename:	1
Stimulus	\Appnotes\AN1613\SPICE Model Folder\ISL28114.lib Browse	
Include	Configured Files	
	H:\AMS Files\1 Precision and MicroPower Amplifie Add as Global	
	Add to Design	
	Add to Profile	
	Edit	
	Change	
	Library Path	
	"C:\Cadence\SPB_16.2\tools\pspice\library" Browse	
	OK Cancel Apply Help	

FIGURE 24. CONFIGURATION FILE TO ADD LIBRARY

Now click the analysis tab and configure the simulation as shown in Figure 25. The analysis selected for this example is an AC Sweep/Noise. Other types of analysis are: Time Domain (Transient), DC Sweep and Bias Point. Just click the down arrow in the analysis type section to access the different Analysis options. When done, click OK.

Simulation Settings - AC Ana	lysis	د				
General Analysis Configuration	n Files Options Data Col	lection Probe Window				
Analysis type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	AC Sweep Type C Linear C Logarithmic Decade Noise Analysis Enabled Out I/V Inte	Start Frequency: 0.1 End Frequency: 100Meg Points/Decade: 100 put Voltage: Source: rval:				
Output File Options Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP) OK Cancel Apply Help						

FIGURE 25. SCREEN TO SET-UP THE ANALYSIS PROFILE CONFIGURED

Add the Library .olb to the simulator. Click Place in the tool bar and select Part. This will bring up the part placement tool at the far right of the simulator as shown in Figure 26.



To add the library, click on the tab where the arrow is pointing to in Figure 26.



FIGURE 26. PART PLACEMENT TOOL

This will bring up the screen shown in Figure 27. Browse to where you saved the Netlist in the common directory and click Open.

Browse File		? X
Look in	x 🗀 SPICE Model Folder 💽 🕓 🈥 📰 🗸	
My Recent Docktop Desktop My Documents	ESL28114_ANI613-PSpiceFiles ■ ISL28114.OLB	
My Network Places	File name: ISL28114.0LB Image: Op Files of type: Capture Library(*.olb) Image: Can Image: Open as read-only Open as read-only	en icel

FIGURE 27. CONNECTING THE SYMBOL LIBRARY TO SIMULATOR

Now you are ready to add the sub-circuit to your simulation schematic and start your simulations.

Adding the Sub-Circuit to Your Simulation Schematic

With the .lib file added to the simulation profile and the .olb file added to the Part placement tool, your are now ready to place the Opamp sub-circuit into your simulation schematic. Figure 28 shows the part placement tool after the .olb has been added to it. Under the Libraries section, find the new .olb symbol you added in the previous step (highlighted in blue). Double click the file to add the sub-circuit to the Part list section (also highlighted in blue). Double click the Part in the part list section to add the sub-circuit to the simulation schematic.



FIGURE 28. ADDING A SUB-CIRCUIT TO A SIMULATION SCHEMATIC

Figure 29 shows sub-circuit in a basic non-inverting application circuit. The simulation result showing AVOL (green trace) and Phase (pink) are shown in Figure 30.



FIGURE 29. SIMULATION SCHEMATIC





AN1613 Rev.0.00 Dec 8, 2010



Notice

- 1. Descriptions of circuits, software and other related information in this document are provided only to illustrate the operation of semiconductor products and application examples. You are fully responsible for the incorporation or any other use of the circuits, software, and information in the design of your product or system. Renesas Electronics disclaims any and all liability for any losses and damages incurred by you or third parties arising from the use of these circuits, software, or information
- 2. Renesas Electronics hereby expressly disclaims any warranties against and liability for infringement or any other claims involving patents, copyrights, or other intellectual property rights of third parties, by or arising from the use of Renesas Electronics products or technical information described in this document, including but not limited to, the product data, drawings, charts, programs, algorithms, and application examples
- 3. No license, express, implied or otherwise, is granted hereby under any patents, copyrights or other intellectual property rights of Renesas Electronics or others.
- 4. You shall not alter, modify, copy, or reverse engineer any Renesas Electronics product, whether in whole or in part. Renesas Electronics disclaims any and all liability for any losses or damages incurred by you or third parties arising from such alteration, modification, copying or reverse engineering.
- Renesas Electronics products are classified according to the following two quality grades: "Standard" and "High Quality". The intended applications for each Renesas Electronics product depends on the product's quality grade, as indicated below.
 - "Standard" Computers: office equipment; communications equipment; test and measurement equipment; audio and visual equipment; home electronic appliances; machine tools; personal electronic equipment: industrial robots: etc.

"High Quality": Transportation equipment (automobiles, trains, ships, etc.); traffic control (traffic lights); large-scale communication equipment; key financial terminal systems; safety control equipment; etc. Unless expressly designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not intended or authorized for use in products or systems that may pose a direct threat to human life or bodily injury (artificial life support devices or systems; surgical implantations; etc.), or may cause serious property damage (space system; undersea repeaters; nuclear power control systems; aircraft control systems; key plant systems; military equipment; etc.). Renesas Electronics disclaims any and all liability for any damages or losses incurred by you or any third parties arising from the use of any Renesas Electronics product that is inconsistent with any Renesas Electronics data sheet, user's manual or other Renesas Electronics document.

- 6. When using Renesas Electronics products, refer to the latest product information (data sheets, user's manuals, application notes, "General Notes for Handling and Using Semiconductor Devices" in the reliability handbook, etc.), and ensure that usage conditions are within the ranges specified by Renesas Electronics with respect to maximum ratings, operating power supply voltage range, heat dissipation characteristics, installation, etc. Renesas Electronics disclaims any and all liability for any malfunctions, failure or accident arising out of the use of Renesas Electronics oroducts outside of such specified ranges
- 7. Although Renesas Electronics endeavors to improve the quality and reliability of Renesas Electronics products, semiconductor products have specific characteristics, such as the occurrence of failure at a certain rate and malfunctions under certain use conditions. Unless designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not subject to radiation resistance design. You are responsible for implementing safety measures to guard against the possibility of bodily injury, injury or damage caused by fire, and/or danger to the public in the event of a failure or malfunction of Renesas Electronics products, such as safety design for hardware and software, including but not limited to redundancy, fire control and malfunction prevention, appropriate treatment for aging degradation or any other appropriate measures. Because the evaluation of microcomputer software alone is very difficult and impractical, you are responsible for evaluating the safety of the final products or systems manufactured by you.
- 8. Plea e contact a Renesas Electronics sales office for details as to environmental matters such as the environmental compatibility of each Renesas Electronics product. You are responsible for carefully and sufficiently investigating applicable laws and regulations that regulate the inclusion or use of controlled substances, including without limitation, the EU RoHS Directive, and using Renesas Electronics products in compliance with all these applicable laws and regulations. Renesas Electronics disclaims any and all liability for damages or losses occurring as a result of your noncompliance with applicable laws and regulations.
- 9. Renesas Electronics products and technologies shall not be used for or incorporated into any products or systems whose manufacture, use, or sale is prohibited under any applicable domestic or foreign laws or regulations. You shall comply with any applicable export control laws and regulations promulgated and administered by the governments of any countries asserting jurisdiction over the parties or transactions
- 10. It is the responsibility of the buyer or distributor of Renesas Electronics products, or any other party who distributes, disposes of, or otherwise sells or transfers the product to a third party, to notify such third party in advance of the contents and conditions set forth in this document.
- 11. This document shall not be reprinted, reproduced or duplicated in any form, in whole or in part, without prior written consent of Renesas Electronics
- 12. Please contact a Renesas Electronics sales office if you have any questions regarding the information contained in this document or Renesas Electronics products
- (Note 1) "Renesas Electronics" as used in this document means Renesas Electronics Corporation and also includes its directly or indirectly controlled subsidiaries
- (Note 2) "Renesas Electronics product(s)" means any product developed or manufactured by or for Renesas Electronics.

(Rev.4.0-1 November 2017)

RENESAS

SALES OFFICES

Renesas Electronics Corporation

http://www.renesas.com

Refer to "http://www.renesas.com/" for the latest and detailed information

Renesas Electronics America Inc. 1001 Murphy Ranch Road, Milpitas, CA 95035, U.S.A. Tel: +1-408-432-8888, Fax: +1-408-434-5351 Renesas Electronics Canada Limited 9251 Yonge Street, Suite 8309 Richmond Hill, Ontario Canada L4C 9T3 Tel: +1-905-237-2004 Renesas Electronics Europe Limited Dukes Meadow, Miliboard Road, Bourne End, Buckinghamshire, SL8 5FH, U.K Tei: +44-1628-651-700, Fax: +44-1628-651-804 Renesas Electronics Europe GmbH Arcadiastrasse 10, 40472 Düsseldorf, Germar Tel: +49-211-6503-0, Fax: +49-211-6503-1327 Renesas Electronics (China) Co., Ltd. Room 1709 Quantum Plaza, No.27 ZhichunLu, Haidian District, Beijing, 100191 P. R. China Tel: +86-10-8235-1155, Fax: +86-10-8235-7679 Renesas Electronics (Shanghai) Co., Ltd. Unit 301, Tower A, Central Towers, 555 Langao Road, Putuo District, Shanghai, 200333 P. R. China Tel: +86-21-2226-0888, Fax: +86-21-2226-0999 Renesas Electronics Hong Kong Limited Unit 1601-1611, 16/F., Tower 2, Grand Century Place, 193 Prince Edward Road West, Mongkok, Kowloon, Hong Kong Tel: +852-2265-6688, Fax: +852 2886-9022 Renesas Electronics Taiwan Co., Ltd. 13F, No. 363, Fu Shing North Road, Taipei 10543, Taiwan Tel: +886-2-8175-9600, Fax: +886 2-8175-9670 Renesas Electronics Singapore Pte. Ltd. 80 Bendemeer Road, Unit #06-02 Hyflux Innovation Centre, Singapore 339949 Tel: +65-6213-0200, Fax: +65-6213-0300 Renesas Electronics Malaysia Sdn.Bhd. Unit 1207, Block B, Menara Amcorp, Amco Amcorp Trade Centre, No. 18, Jln Persiaran Barat, 46050 Petaling Jaya, Selangor Darul Ehsan, Malaysia Unit 1207, Block B, Menara Amcorp, Amcorp Tel: +60-3-7955-9390, Fax: +60-3-7955-9510 Renesas Electronics India Pvt. Ltd. No.777C, 100 Feet Road, HAL 2nd Stage, Indiranagar, Bangalore 560 038, India Tel: +91-80-67208700, Fax: +91-80-67208777 Renesas Electronics Korea Co., Ltd. 17F, KAMCO Yangjae Tower, 262, Gangnam-daero, Gangnam-gu, Seoul, 06265 Korea Tei: +822-558-3737, Fax: +822-558-5338