

How to use dynamic models on OrCAD Capture

Murata Manufacturing Co., Ltd.
Aug. 2021



1. Model files

At first, please see "Comparison_table_for_libraries.pdf" and check model name and file names do you want to use. It is necessary to use both of lib-file and olb-file.

Spice model : *****.lib / OLB file : *****.olb

for MLCC dynamic models

Part Number	Model Name	Lib File	OLB File	Capacitance
GCD188R71H102KA01	GCD188R71H102KA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1000pF
GCD188R71H102MA01	GCD188R71H102MA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1000pF
GCD188R72A102KA01	GCD188R72A102KA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1000pF
GCD188R72A102MA01	GCD188R72A102MA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1000pF
GCD188R71H122KA01	GCD188R71H122KA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1200pF
GCD188R71H122MA01	GCD188R71H122MA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1200pF
GCD188R72A122KA01	GCD188R72A122KA01_NONLIN	GCD18_nonlin.lib	GCD18_nonlin.olb	1200pF

for Power Inductor dynamic models

Part Number	Model Name	Lib File	OLB File	Inductance[uH]
1217AS-H-1R5N	1217ASH1R5N_NONLIN	1217ASH1R5N_nonlin.lib	1217ASH1R5N_nonlin.olb	1.5
1217AS-H-2R2N	1217ASH2R2N_NONLIN	1217ASH2R2N_nonlin.lib	1217ASH2R2N_nonlin.olb	2.2
1217AS-H-3R3N	1217ASH3R3N_NONLIN	1217ASH3R3N_nonlin.lib	1217ASH3R3N_nonlin.olb	3.3
1217AS-H-4R7N	1217ASH4R7N_NONLIN	1217ASH4R7N_nonlin.lib	1217ASH4R7N_nonlin.olb	4.7
1217AS-H-5R6N	1217ASH5R6N_NONLIN	1217ASH5R6N_nonlin.lib	1217ASH5R6N_nonlin.olb	5.6
1217AS-H-6R8N	1217ASH6R8N_NONLIN	1217ASH6R8N_nonlin.lib	1217ASH6R8N_nonlin.olb	6.8
1217AS-H-8R2M	1217ASH8R2M_NONLIN	1217ASH8R2M_nonlin.lib	1217ASH8R2M_nonlin.olb	8.2

Please note that in MLCC and Power Inductor, lib-file and olb- files are named differently.

2. Location of model files

- Unzip zip-file of OrCAD Capture dynamic model
- Place lib-file and olb-file following folder

For example

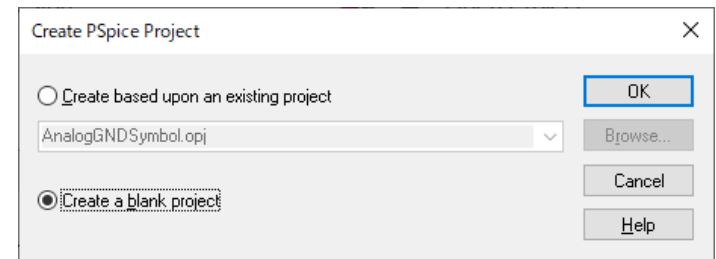
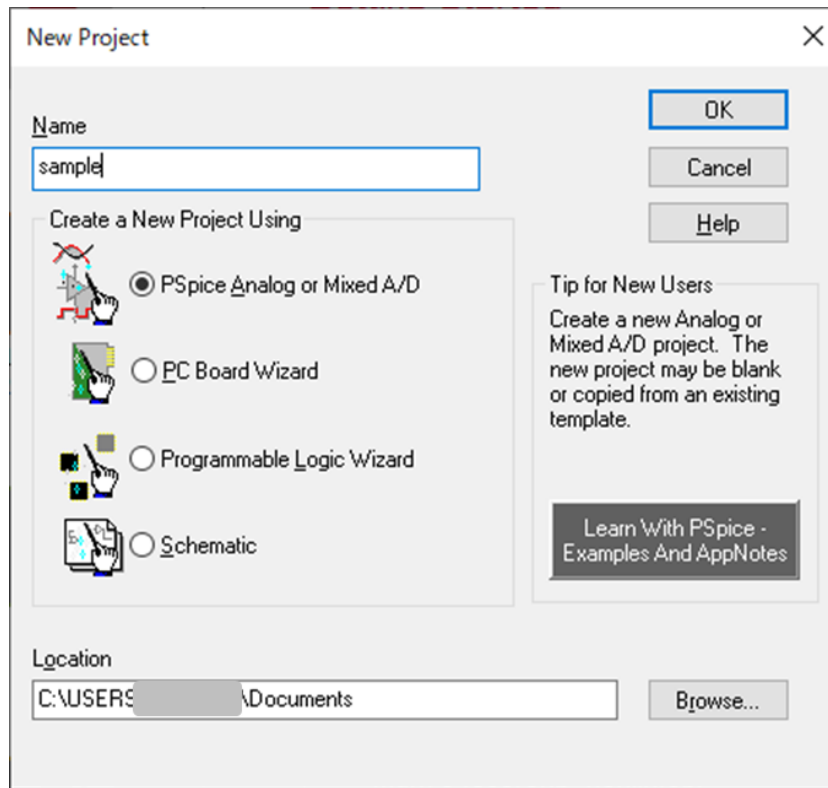
Project file: C:¥Users¥<username>¥Documents
Lib-file: C:¥Users¥<username>¥Documents¥lib
Olb-file: C:¥Users¥<username>¥Documents¥olb

(*) A default location of OrCAD Capture 17.2

Project file: C:¥Users¥<username>¥Documents
Lib-file: C:¥Users¥<username>¥Documents¥<project>¥...
Olb-file: C:¥Cadence¥SPB_17.2¥tools¥capture¥library¥pspice

3. Create New Project

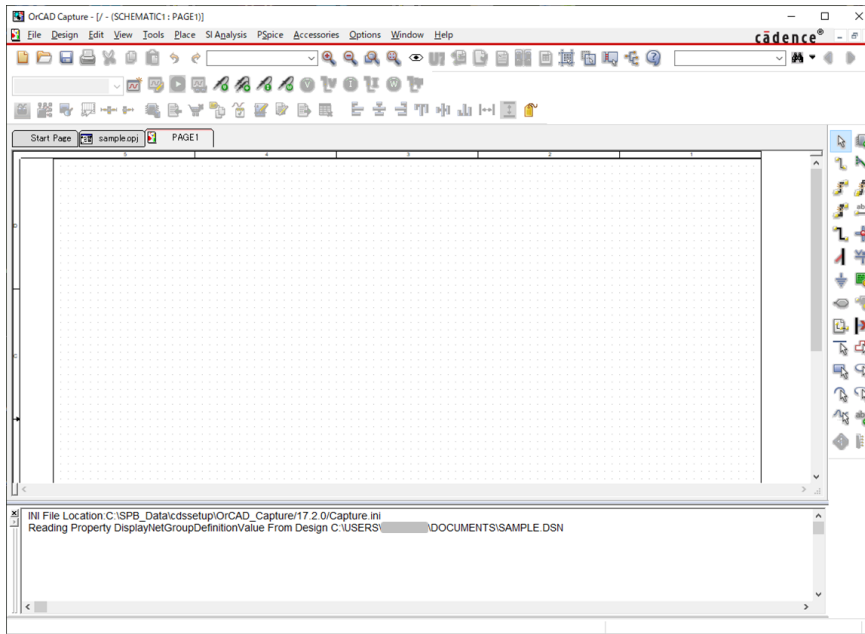
- File menu -> New -> Project
 - Enter a project name "sample" and click "Ok" to create new project



select "Create a blank project"

For example, a default Location path is like this "C:¥Users¥<username>¥Documents".

4. Create simulation profile (AC-simulation)



Create a simulation profile and select lib-file

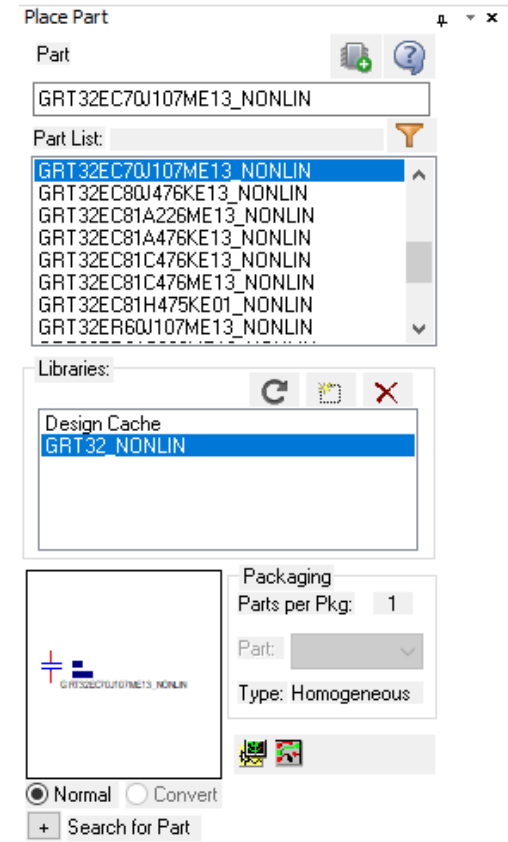
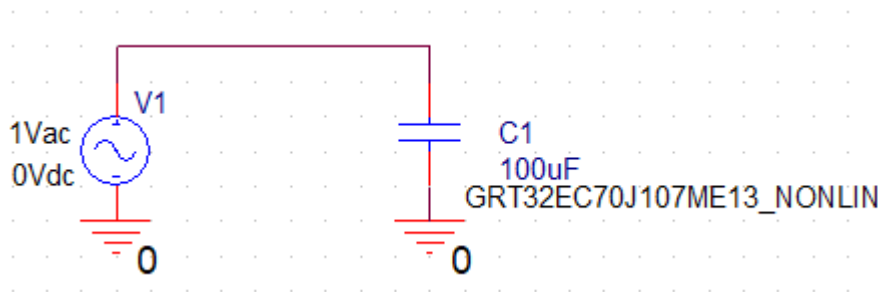
- Create Simulation profile
 - PSpice menu-> New Simulation Profile
- Choose Analysis Tab
 - AC Sweep/Noise
 - Set AC Sweep Type (Start Freq., End Freq., Points etc.)
- Choose Configuration Files
 - Select "Library"
 - Set "Details" (select lib-file and click "Add as ***" button)

5. Create a sample circuit (for MLCC)

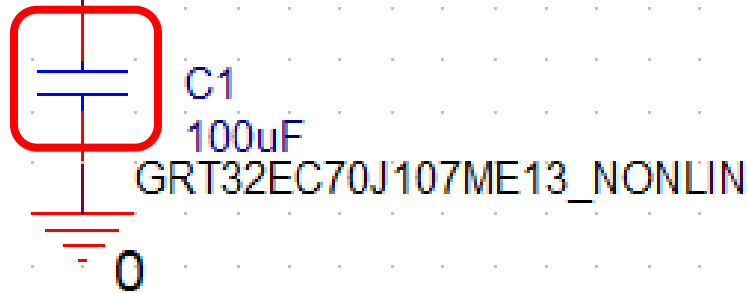
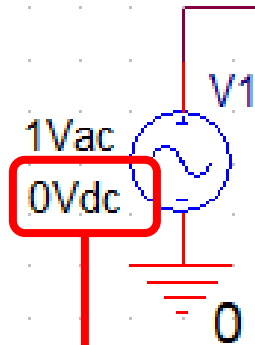


Create a sample circuit included "Murata MLCC dynamic model"

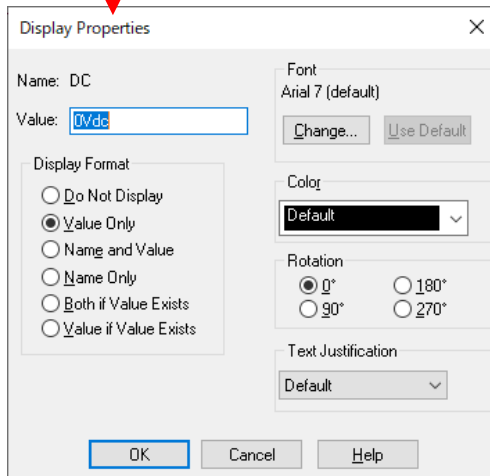
- Place Murata model
 - Place menu -> Parts
 - Set Libnames (Select olb-file)
 - Choose Part from "Part List"
 - Place Capacitor symbol on the schematic
- Place signal component
 - Place menu -> PSpice Component... -> Source
 - > Voltage Sources -> AC



6. Set dc-bias voltage and temperature



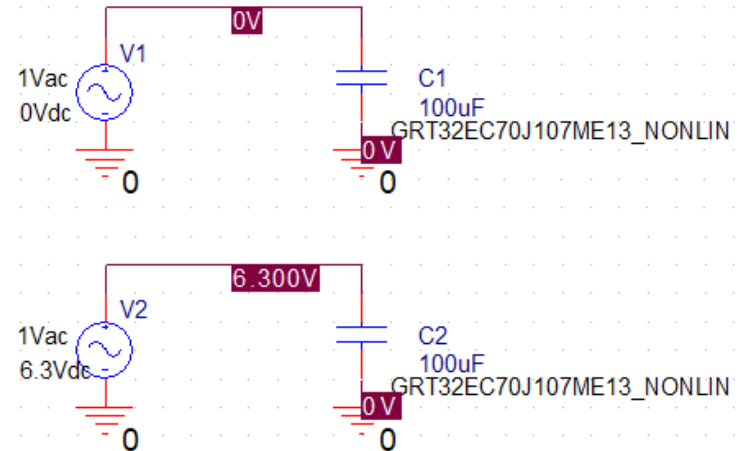
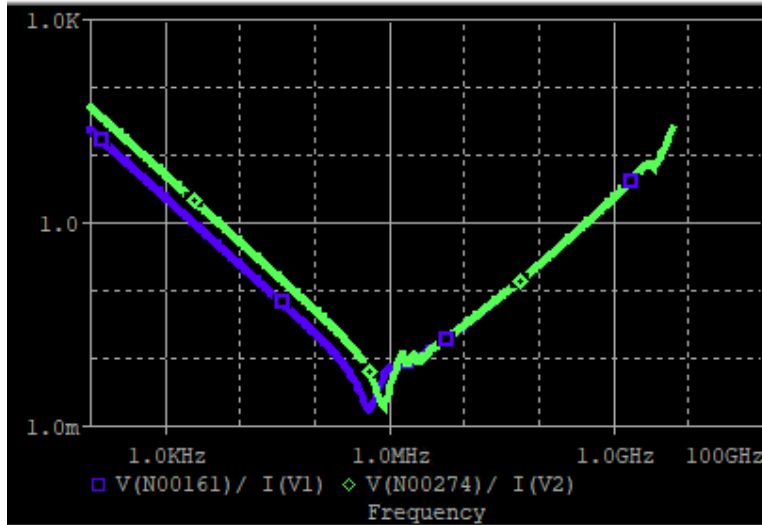
- Select and Right Click a symbol and choose "Edit Properties"
- Click "Pivot" button
- Change Temperature value



Select and Right Click "Vdc" and choose "Edit Properties"

	A
	+ SCHEMATIC1 : PAGE1
BiasValue Power	0W
Color	Default
Designator	
Graphic	GRT32EC70J107ME13_NON
ID	
Implementation	GRT32EC70J107ME13_NON
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	400
Location Y-Coordinate	180
Name	INS453
Part Reference	C1
PCB Footprint	SMDCAP
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceTemplate	X*@REFDES %1 %2 @MOD
Reference	C1
Source Library	C:\USERS\MM02738\D...
Source Package	GRT32EC70J107ME13_N
Source Part	GRT32EC70J107ME13_N
temperature	25
Value	100uF

7. Calculation results



Impedance Characteristics by OrCAD Capture

Part number: GRT32EC70J107ME13

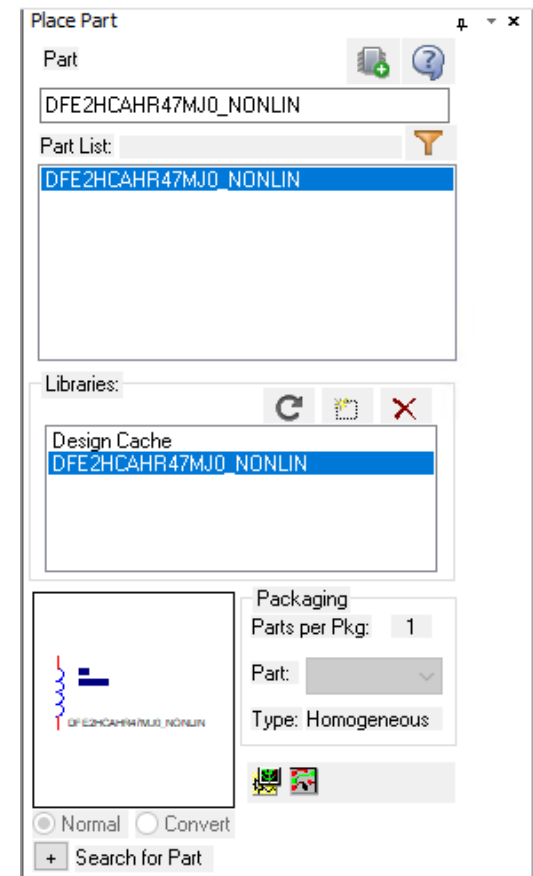
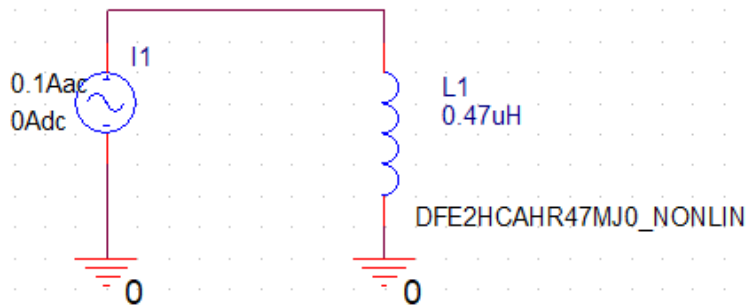
Blue line: 0 Vdc, 25 degC.

Green line: 6.3 Vdc, 125 degC.

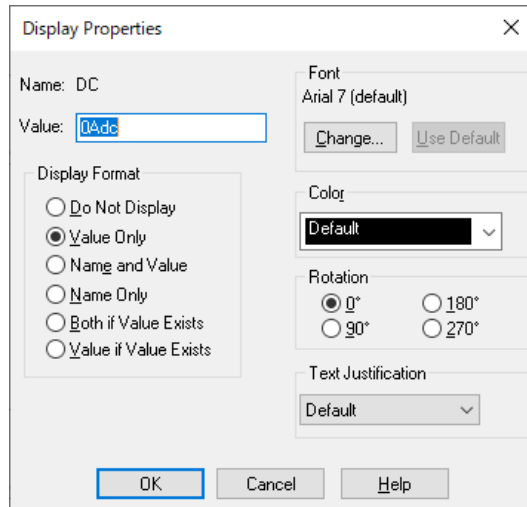
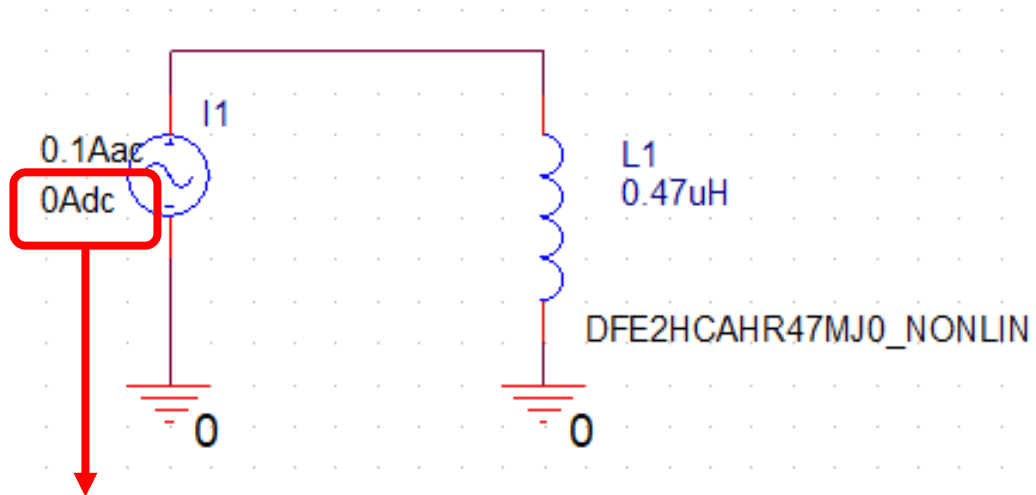
8. Create a sample circuit (for Power Inductor)

Create a sample circuit included "Murata Power Inductor dynamic model"

- Place Murata model
 - Place menu -> Parts
 - Set Libnames (Select olb-file)
 - Choose Part from "Part LIst"
 - Place Inductor symbol on the schematic
- Place signal component
 - Place menu -> PSpice Component... -> Source
 - > Current Sources -> AC

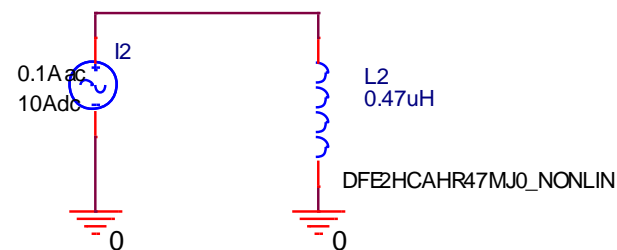
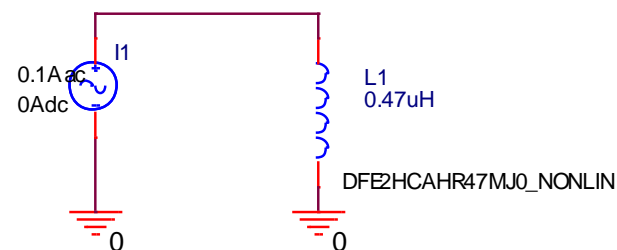
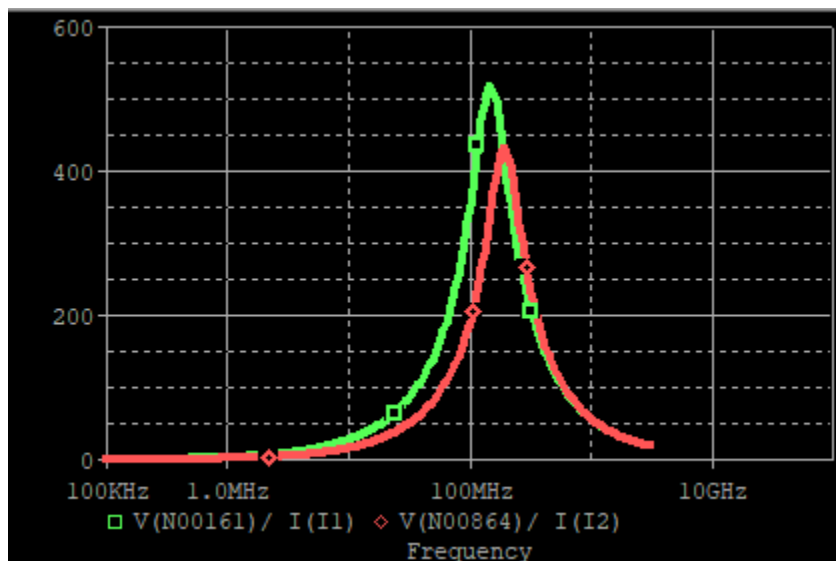


9. Set dc-current



Select and Right Click "Adc" and choose "Edit Properties"

10. Calculation results



Impedance Characteristics by OrCAD Capture

Part number: DFE2HCAHR47MJ0
Green line: 0.1 Aac, 0A dc
Red line: 0.1 Aac, 10A dc



That's all