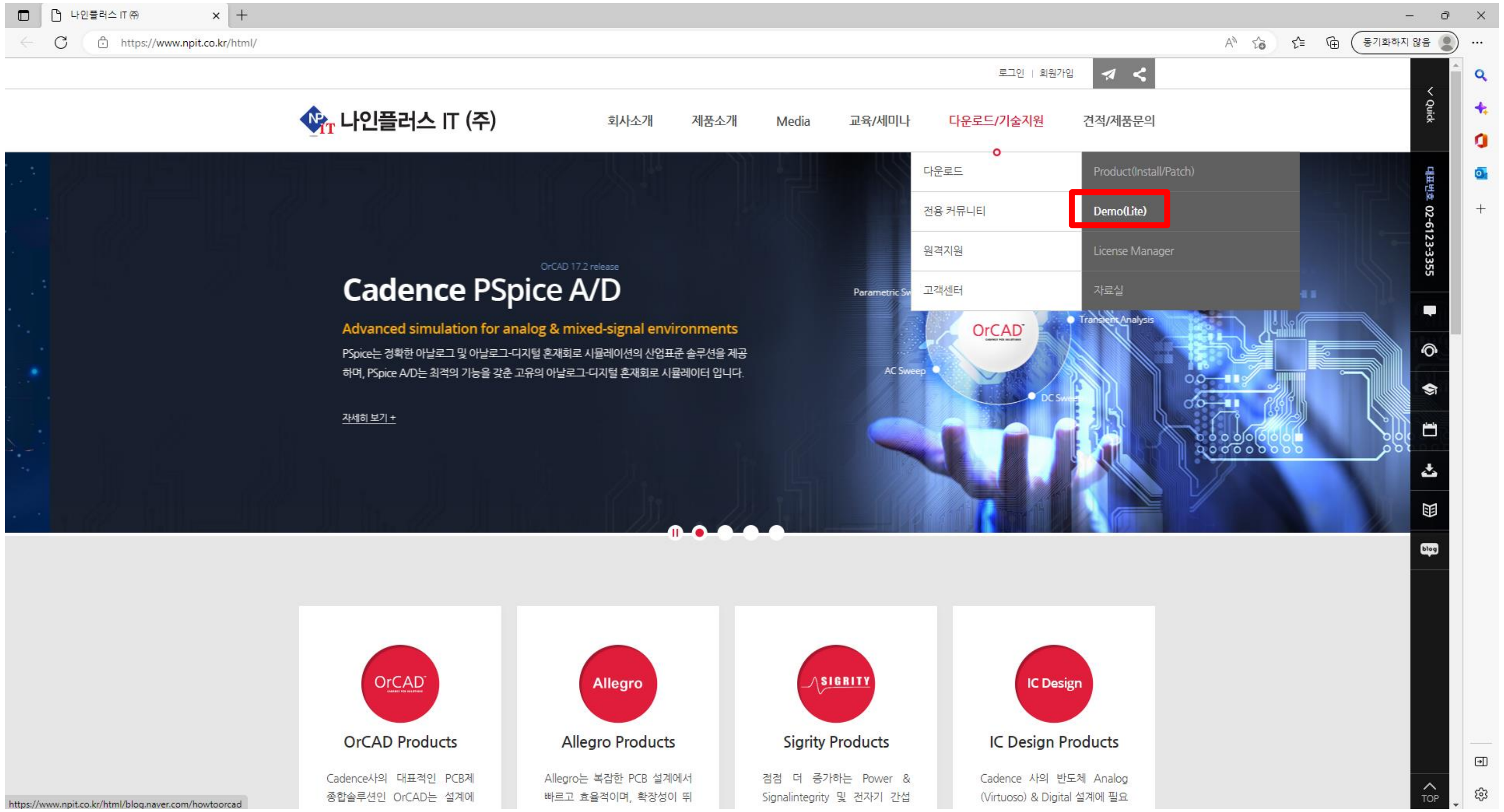
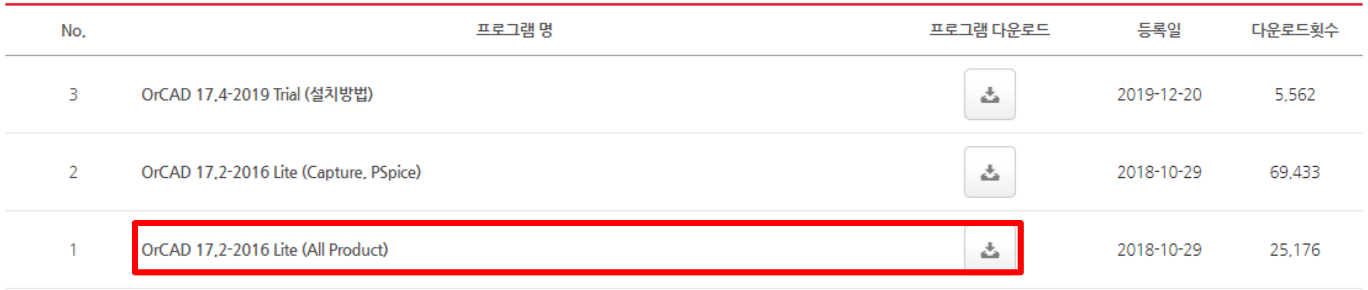
OrCAD 사용법

Install

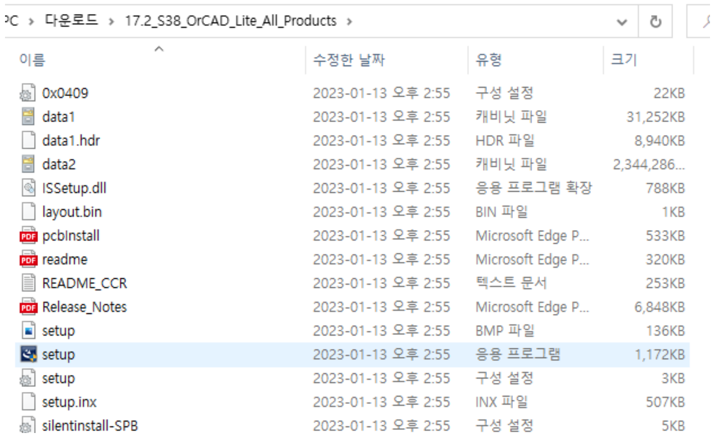




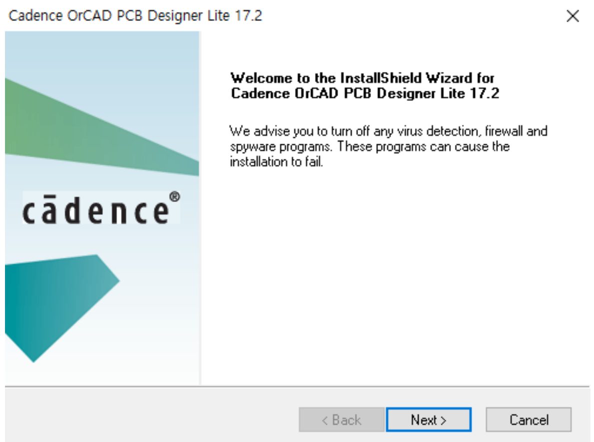
No.	프로그램 명	프로그램 다운로드	등록일	다운로드횟수
3	OrCAD 17.4-2019 Trial (설치방법)		2019-12-20	5,562
2	OrCAD 17.2-2016 Lite (Capture, PSpice)		2018-10-29	69,433
1	OrCAD 17.2-2016 Lite (All Product)		2018-10-29	25,176

http://www.npit.co.kr/ 접속 -> 회원가입 -> 다운로드/기술지원 -> Demo(Lite) -> OrCAD Lite (All Product)

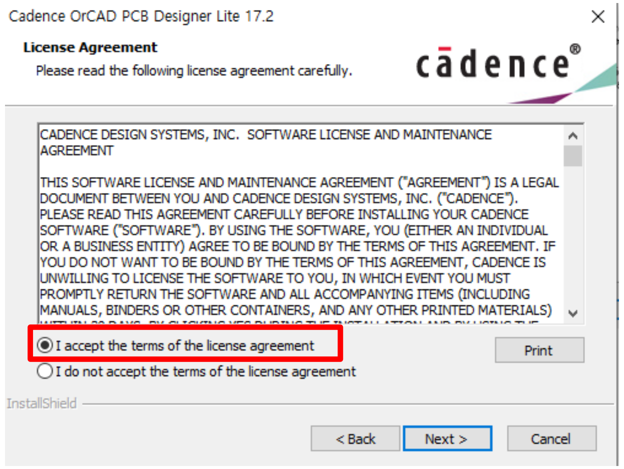
Install



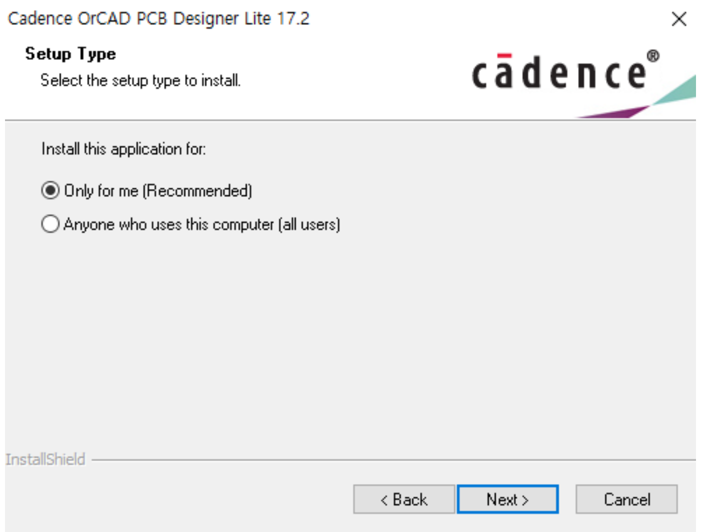
1. 설치 파일 압축 풀어 setup 응용프로그램 실행



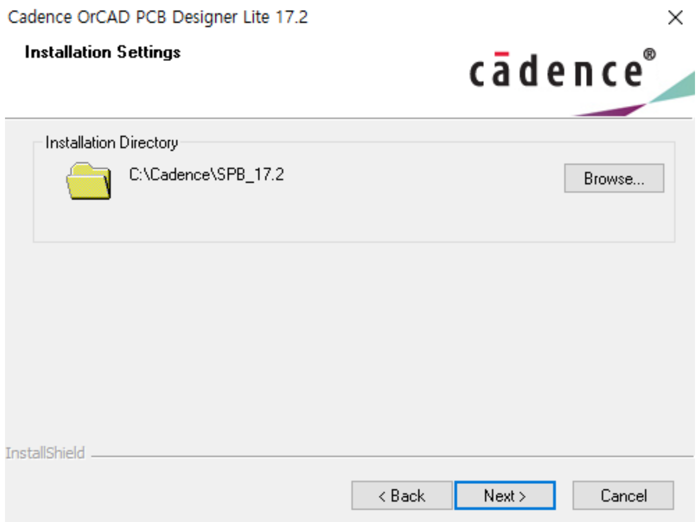
2. Next 누르기



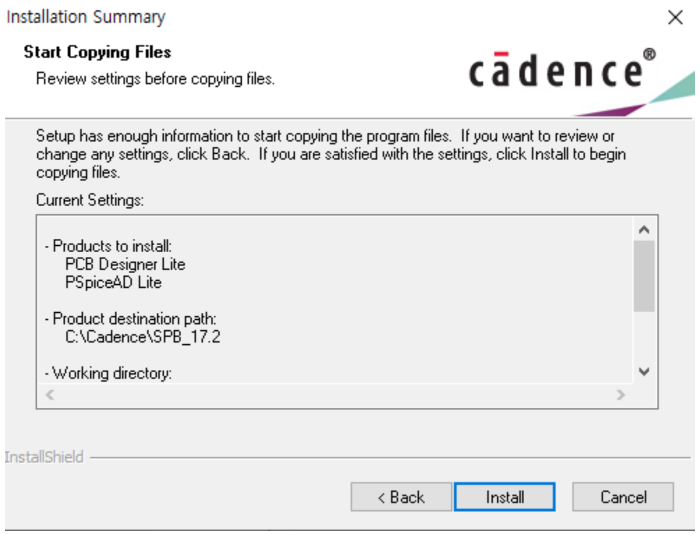
3. 동의로 변경하고 Next



4. 원하는 타입선택

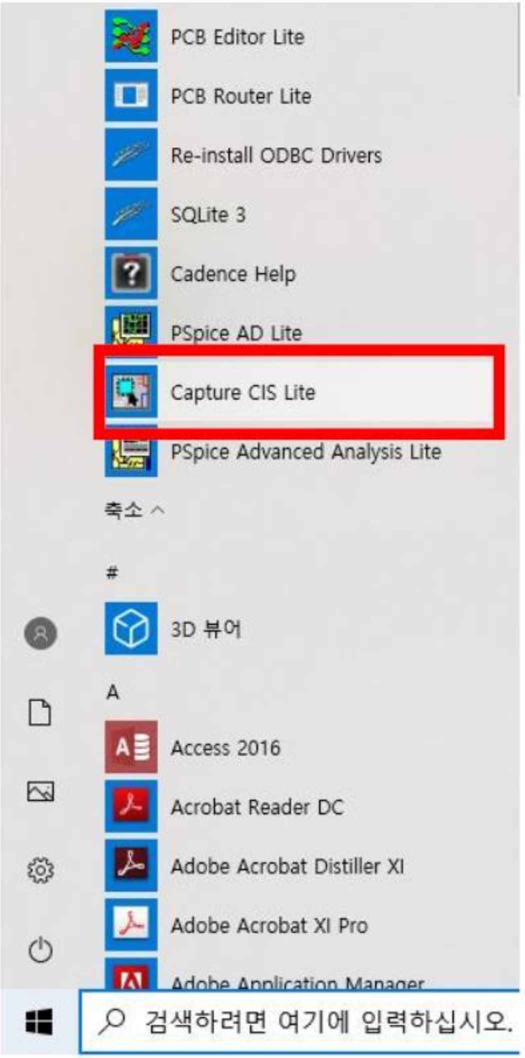


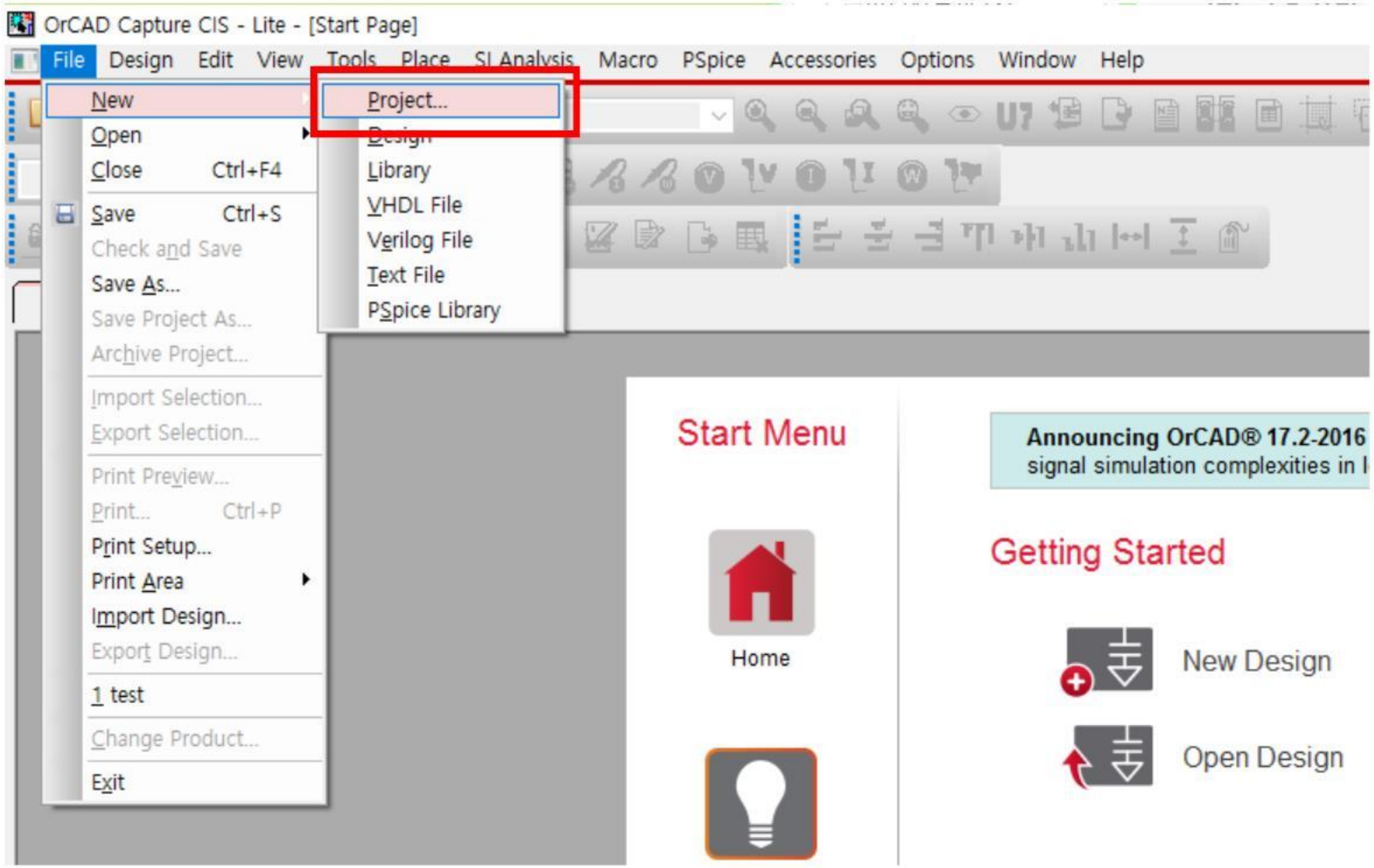
5. 원하는 path 선택 후, Next 선택



6. Install 누르기

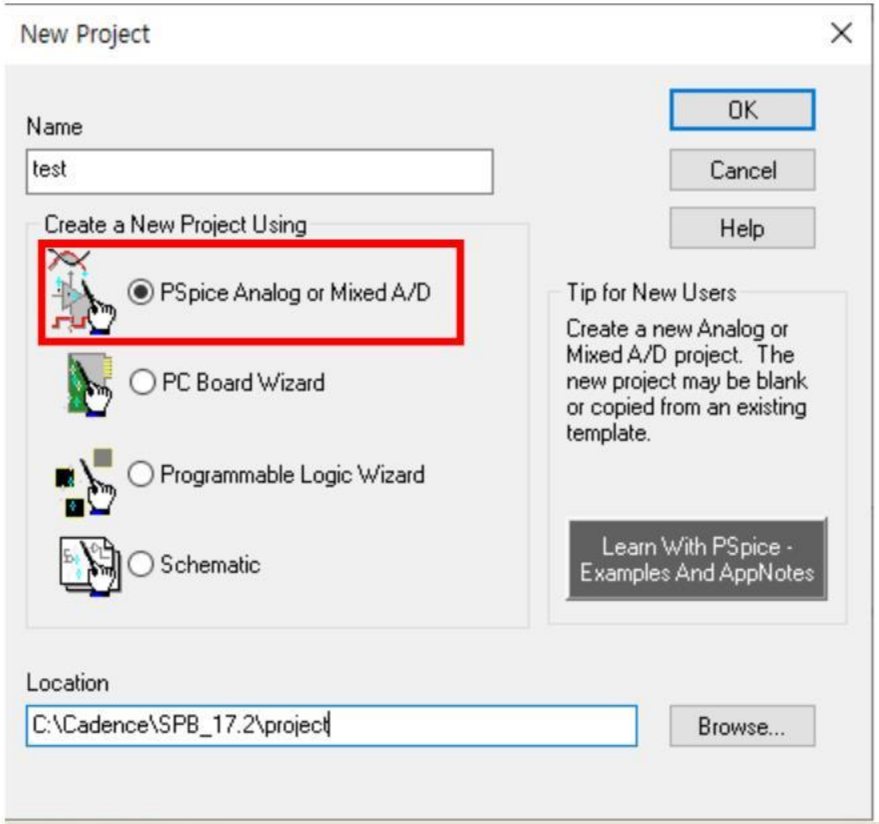
simulation

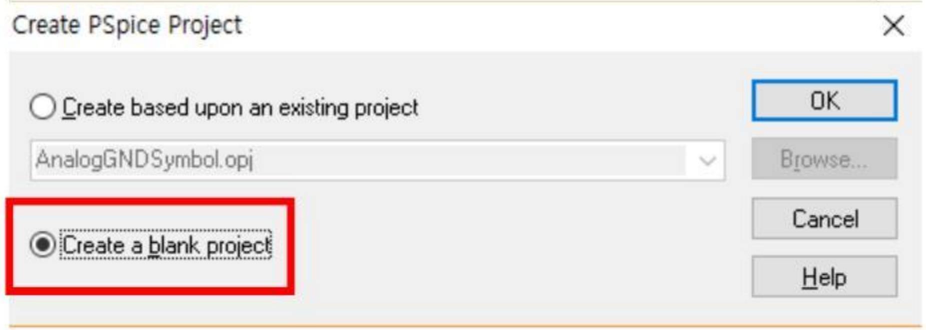


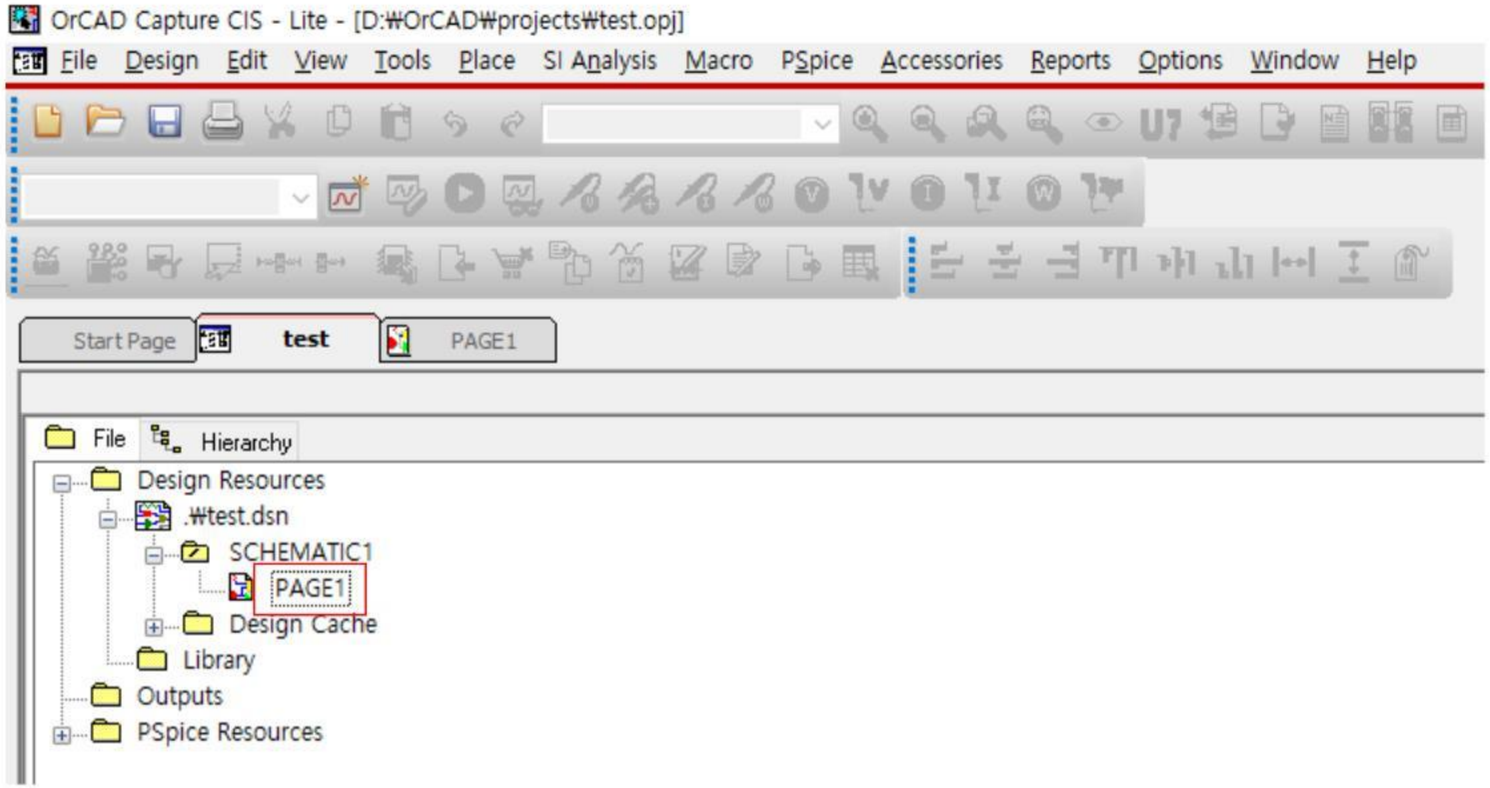


OrCAD Capture CIS Lite 실행 -> File -> New -> Project

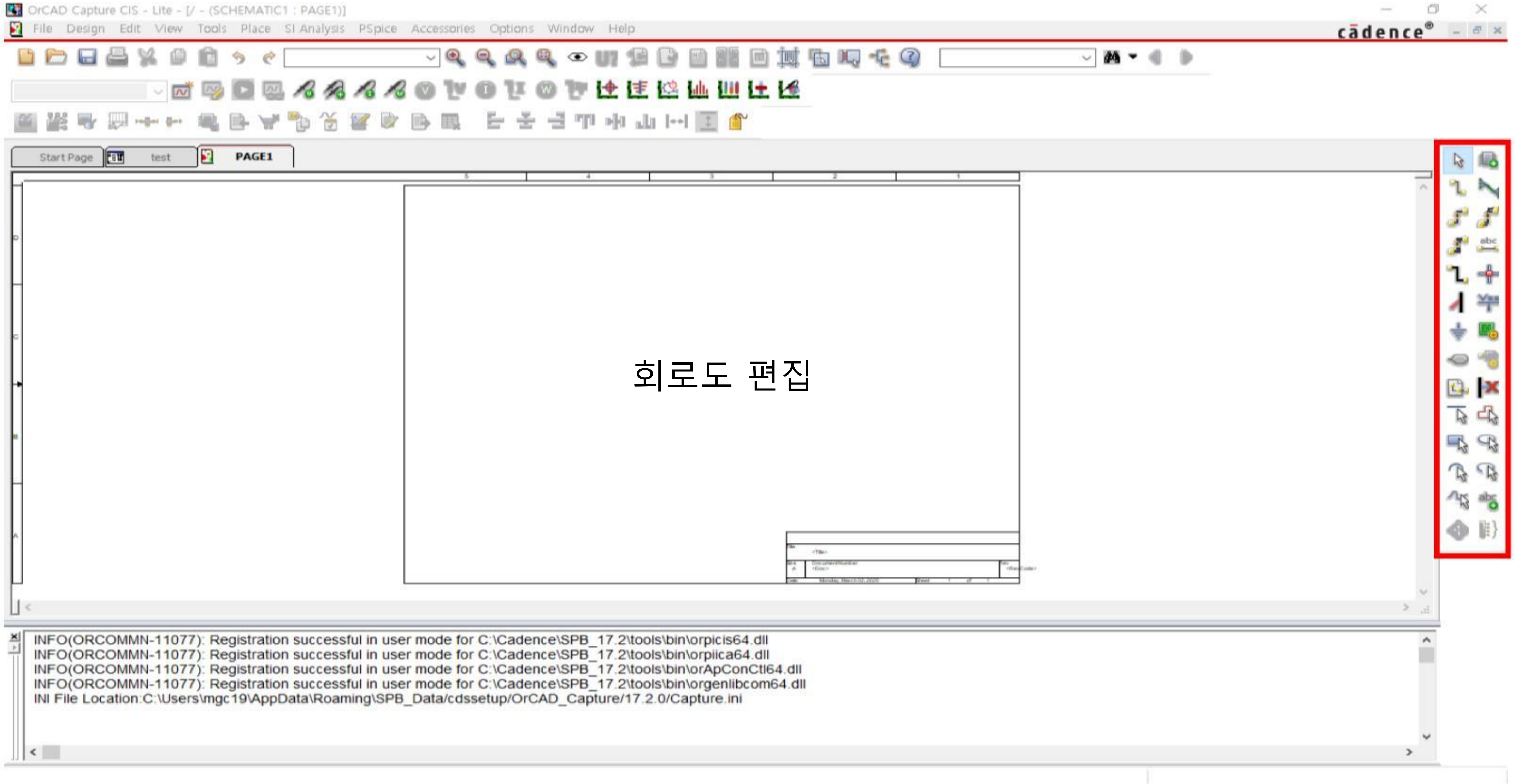
simulation


New Project
Name
test
OK
Cancel
Help
Create a New Project Using
PSpice Analog or Mixed A/D
PC Board Wizard
Programmable Logic Wizard
Schematic
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.
Learn With PSpice - Examples And AppNotes
Location
C:\Cadence\SPB_17.2\project
Browse...


Create PSpice Project
Create based upon an existing project
AnalogGNDSymbol.opj
Create a blank project
OK
Browse...
Cancel
Help


OrCAD Capture CIS - Lite - [D:₩OrCAD₩projects₩test.opj]
File Design Edit View Tools Place SI Analysis Macro PSpice Accessories Reports Options Window Help
Start Page
test
PAGE1
File
Hierarchy
Design Resources
.₩test.dsn
SCHEMATIC1
PAGE1
Design Cache
Library
Outputs
PSpice Resources

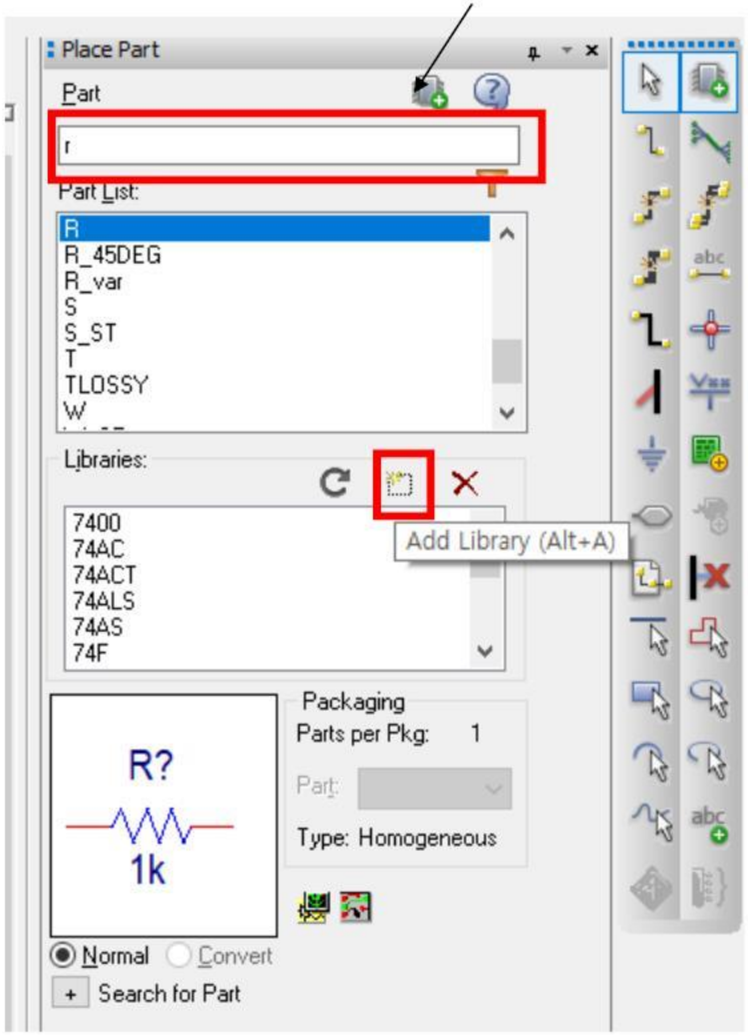
simulation


OrCAD Capture CIS - Lite - [/ - (SCHEMATIC1 : PAGE1)]
File Design Edit View Tools Place SI Analysis PSpice Accessories Options Window Help
cadence
Start Page
test
PAGE1
회로도 편집
INFO(ORCOMMN-11077): Registration successful in user mode for C:\Cadence\SPB_17.2\tools\bin\orpicis64.dll
INFO(ORCOMMN-11077): Registration successful in user mode for C:\Cadence\SPB_17.2\tools\bin\orpiica64.dll
INFO(ORCOMMN-11077): Registration successful in user mode for C:\Cadence\SPB_17.2\tools\bin\orApConCtl64.dll
INFO(ORCOMMN-11077): Registration successful in user mode for C:\Cadence\SPB_17.2\tools\bin\orgenlibcom64.dll
INI File Location:C:\Users\mgc19\AppData\Roaming\SPB_Data/cdssetup/OrCAD_Capture/17.2.0/Capture.ini

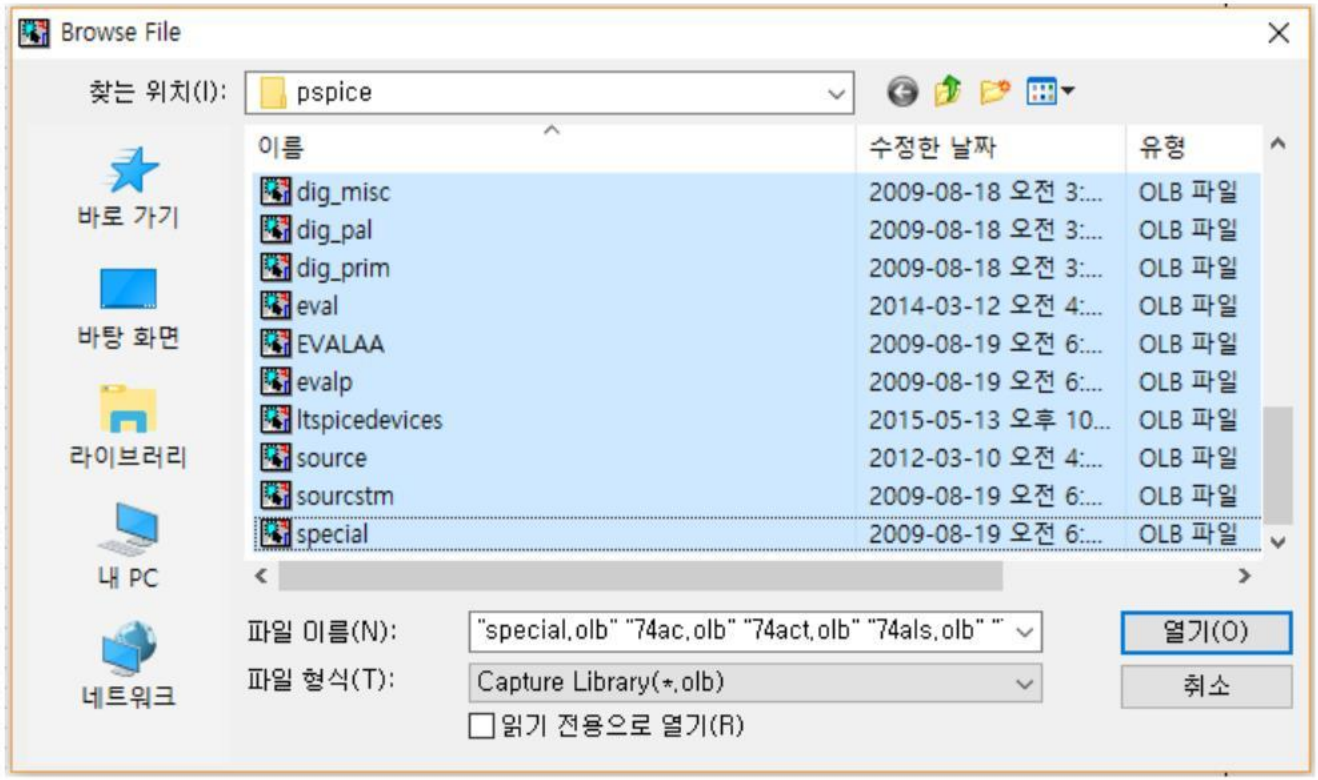
simulation

원하는 부품 이름 입력

Place part
선택

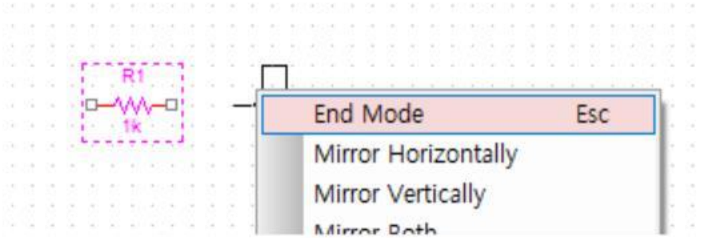


Add Library 선택

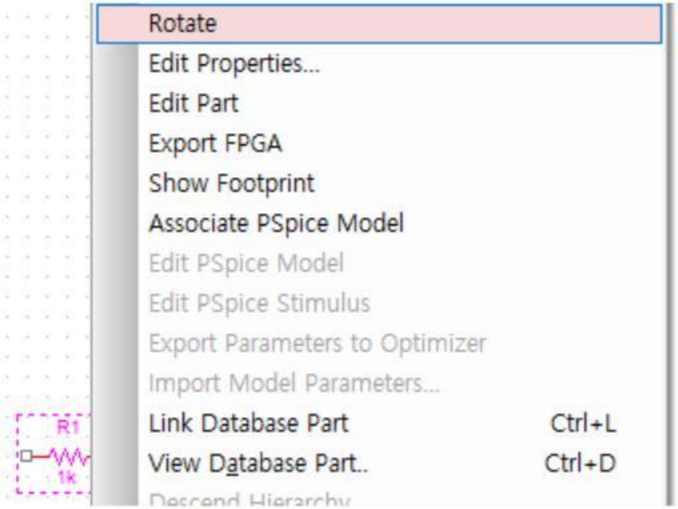


모든 라이브러리 불러오기

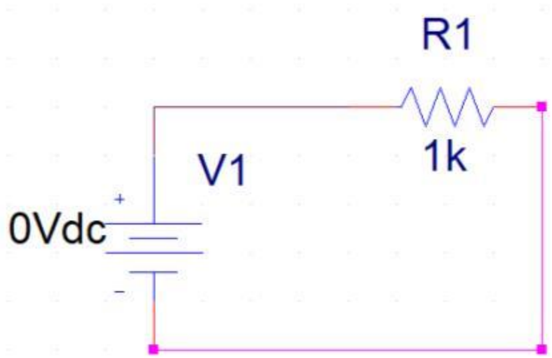
simulation



원하는 위치에 부품을 놓은 뒤, 마우스 오른쪽 버튼 클릭 후, End Mode 클릭하여 현재 작업 종료

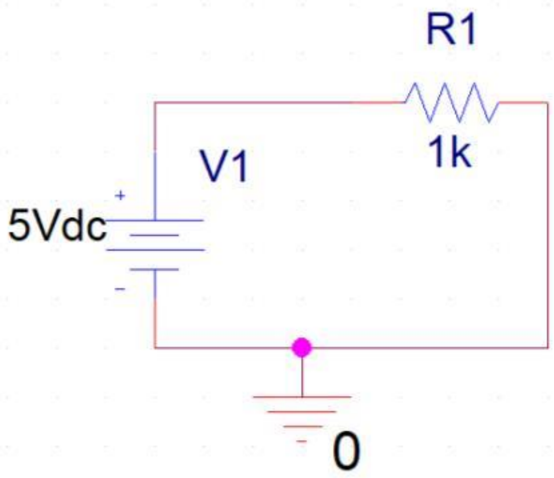


부품을 클릭한 다음 마우스 오른쪽 버튼 클릭 후, Rotate 선택 시 부품 회전 가능



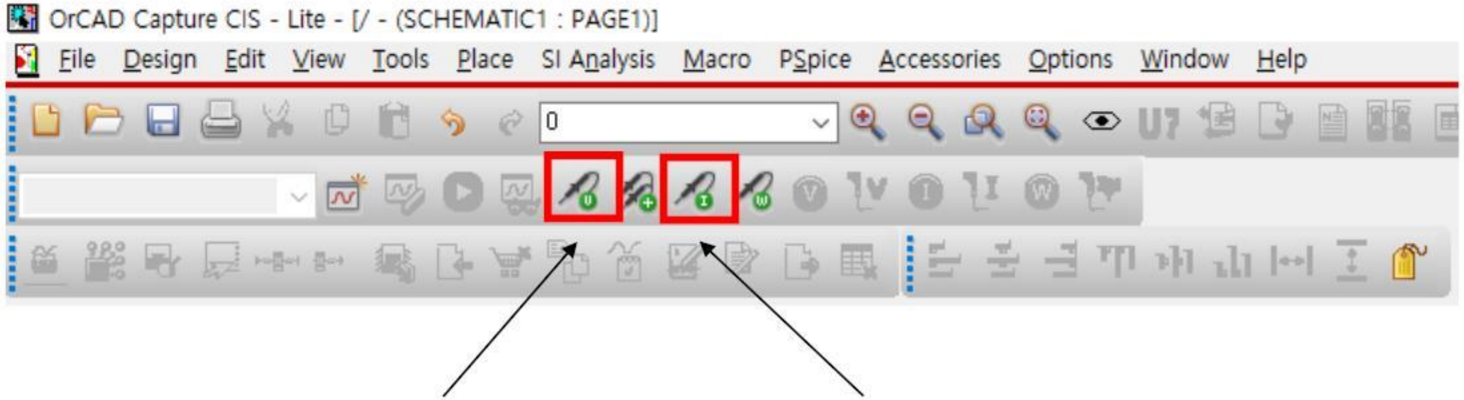
Place wire 선택

Place wire를 통해 선 연결

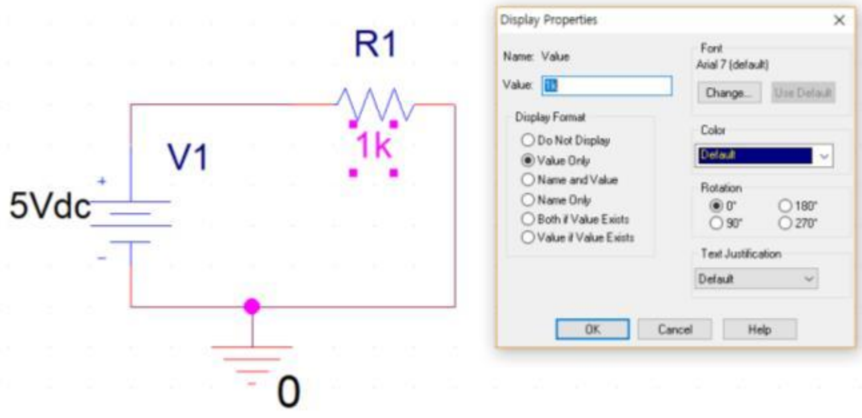


Place ground를 통해 접지

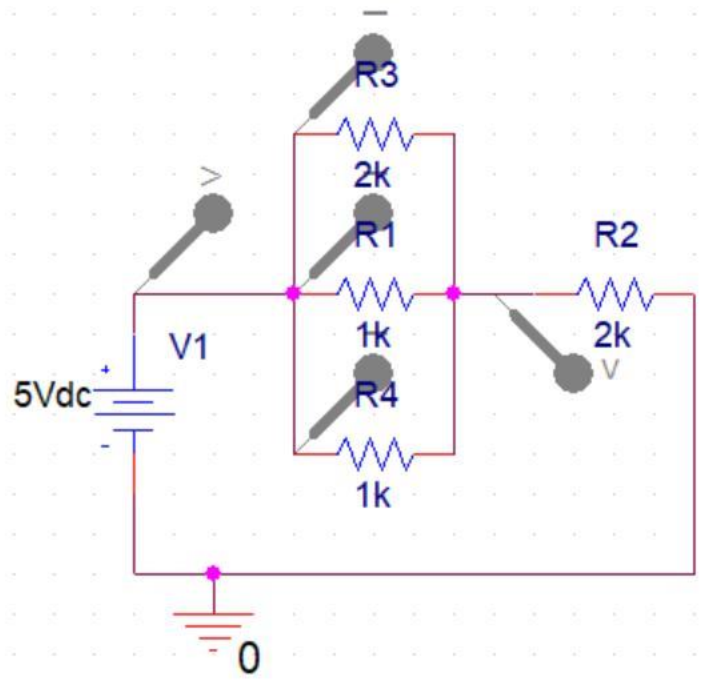
simulation



전압 측정 전류 측정

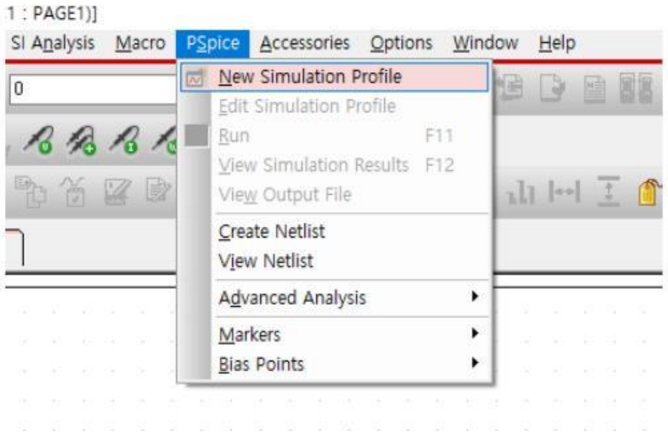


더블 클릭을 통해 값 수정

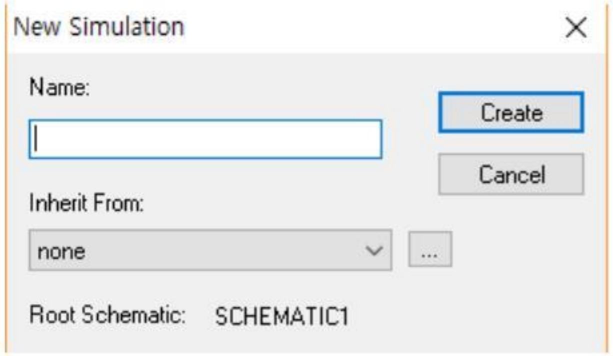


원하는 위치에 프로브

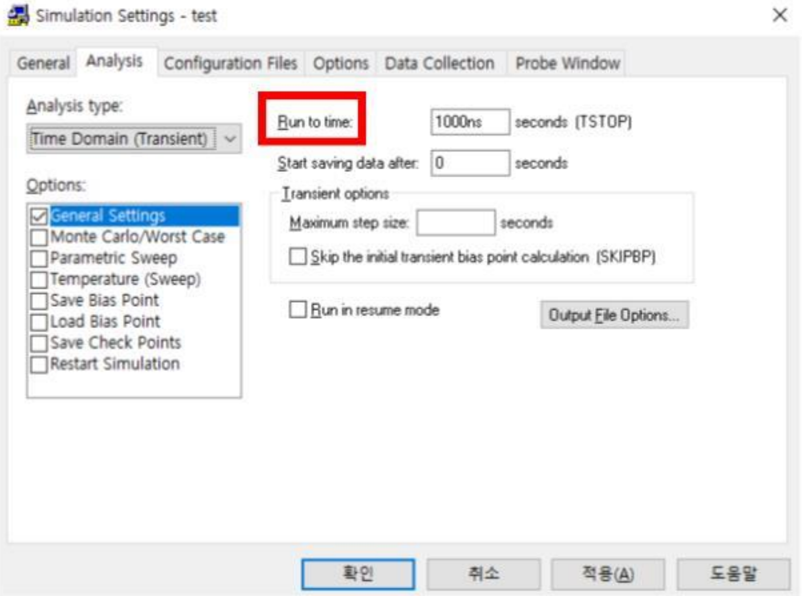
simulation



Pspice->New Simulation Profile 선택

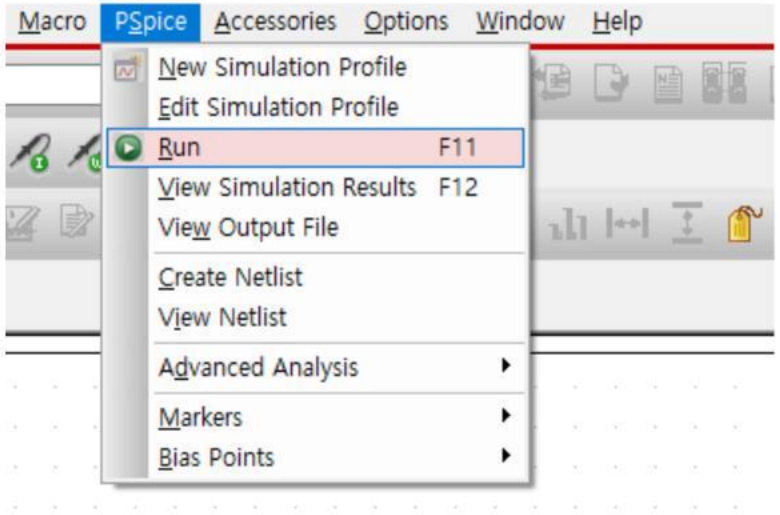


이름 입력후 Create 선택

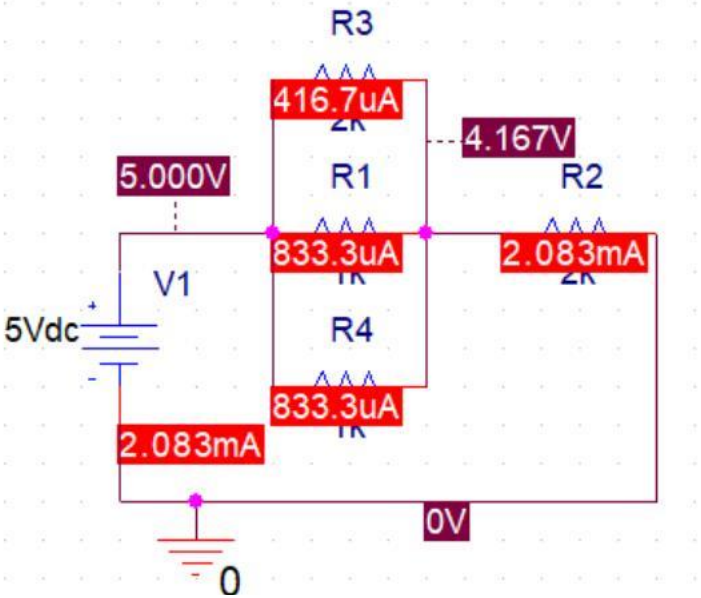


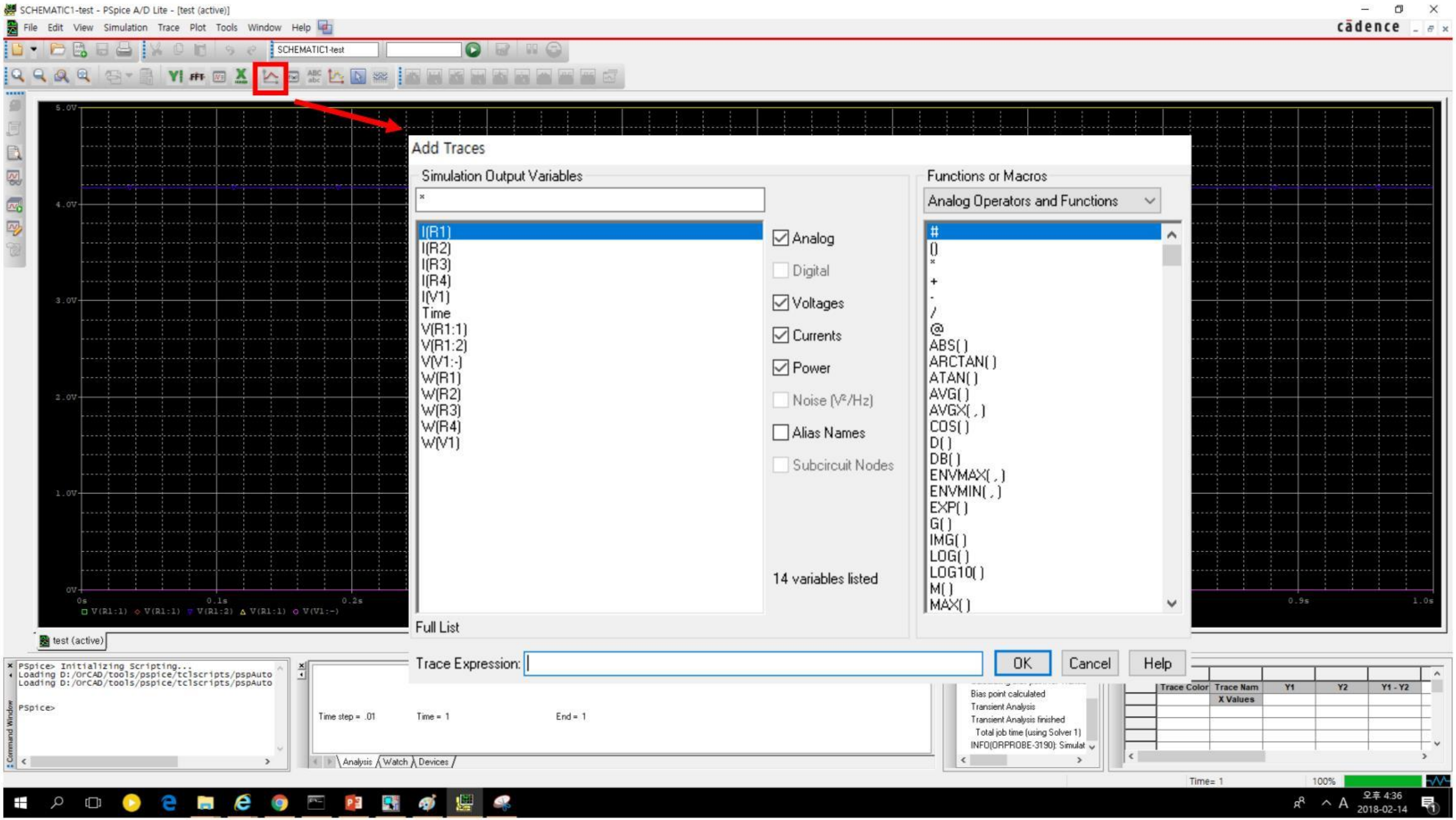
Run to time을 통해 시뮬레이션 시간 설정

simulation



Run을 클릭해 시뮬레이션 실행




SCHEMATIC1-test - PSpice A/D Lite - [test (active)]
File Edit View Simulation Trace Plot Tools Window Help
cādence
SCHEMATIC1-test
Add Traces
Simulation Output Variables
Functions or Macros
Analog Operators and Functions
I(R1)
I(R2)
I(R3)
I(R4)
I(V1)
Time
V(R1:1)
V(R1:2)
V(V1:-)
W(R1)
W(R2)
W(R3)
W(R4)
W(V1)
Analog
Digital
Voltages
Currents
Power
Noise (V²/Hz)
Alias Names
Subcircuit Nodes
14 variables listed
#
()
*
+
-
/
@
ABS( )
ARCTAN( )
ATAN( )
AVG( )
AVGX(,)
COS( )
D( )
DB( )
ENVMAX(,)
ENVMIN(,)
EXP( )
G( )
IMG( )
LOG( )
LOG10( )
M( )
MAX( )
Full List
Trace Expression:
OK
Cancel
Help
test (active)
PSpice> Initializing Scripting...
Loading D:/OrCAD/tools/pspice/tclscripts/pspAuto
Loading D:/OrCAD/tools/pspice/tclscripts/pspAuto
PSpice>
Time step = .01
Time = 1
End = 1
Analysis
Watch
Devices
Bias point calculated
Transient Analysis
Transient Analysis finished
Total job time (using Solver 1)
INFO(ORPROBE-3190): Simulat
Trace Color
Trace Nam
Y1
Y2
Y1 - Y2
X Values
Time= 1
100%
오후 4:36
2018-02-14