

OrCAD 사용법

Install

The screenshot shows the OrCAD website with the 'RESOURCES' menu expanded. The 'Download OrCAD Lite' option is highlighted in a white box. The website header includes a search bar and a language dropdown set to 'English'. The main navigation bar contains links for PRODUCTS, HOW TO BUY, RESOURCES, ABOUT OrCAD, and FREE TRIAL. The 'RESOURCES' dropdown menu lists OrCAD Downloads, Service Bureaus, Resource Library, Download OrCAD Lite, OrCAD User Forum, Tutorials, PSpice Community, and OrCAD Blog. A banner on the left promotes OrCAD Cert ISO 26262, mentioning Cadence PCB design flow and automotive functional safety, with a 'Learn More' button. Below the banner are four small images: a red sports car, a circuit board, a person working on a device, and a PSpice Simulink logo.



Future-proof Scalability

Unlike other PCB solutions, OrCAD has the ability to grow with evolving design challenges providing the security of scalability for future challenges



Outstanding Technology

OrCAD technologies provide maximum robustness and functionality depth to meet the most demanding customer requirements for all types of designs

<http://www.orcad.com/> 접속 -> RESOURCES -> Download OrCAD Lite

Install

Download Lite Request

First Name *

Last Name *

Email *

Company/University *

Job Title *

Address

Country *

Address 1 *

Address 2

District

City *

Province *

Postal code *

임시주번



Gn



받은

별표

보낸

임시주번

더보기

Kim



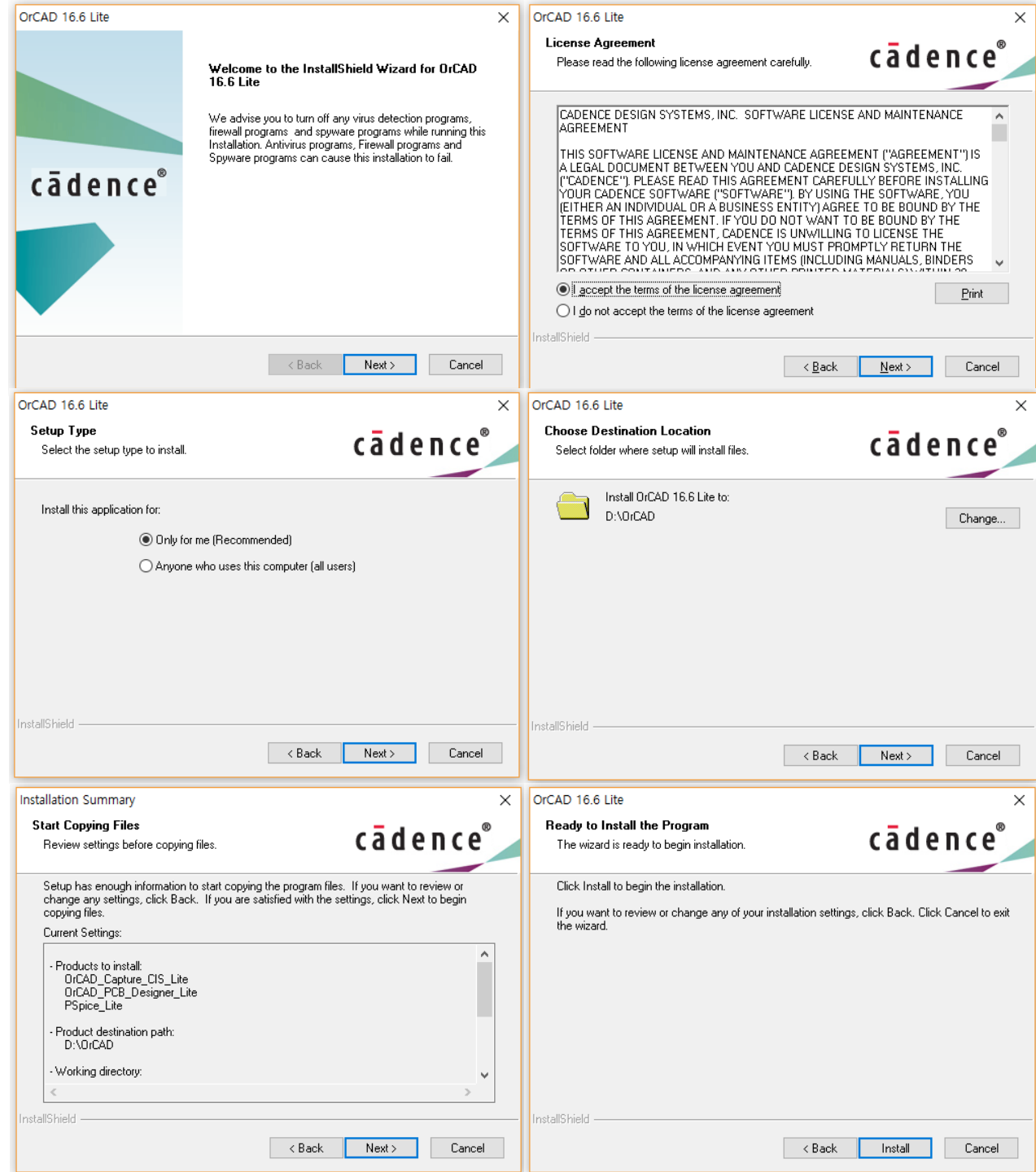
Thank you for your interest in OrCAD!

OrCAD 16.6 Lite Software (All products): [\(Download\)](#)

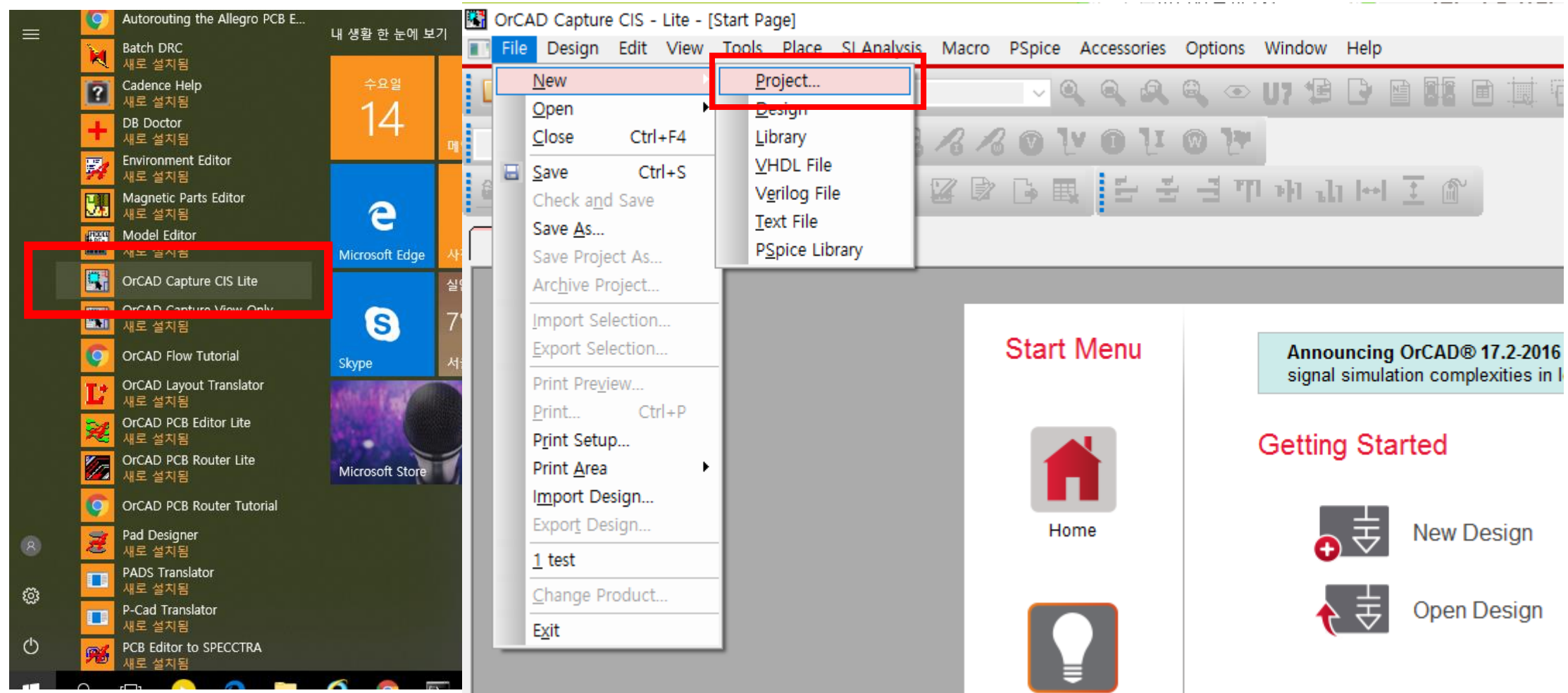
If you have any questions you can [contact us](#) directly or you can find your local Cadence Channel Partner [here](#)

Sincerely,
The OrCAD Team

* 표시된 부분 작성 후
이메일로 설치 파일
확인 가능

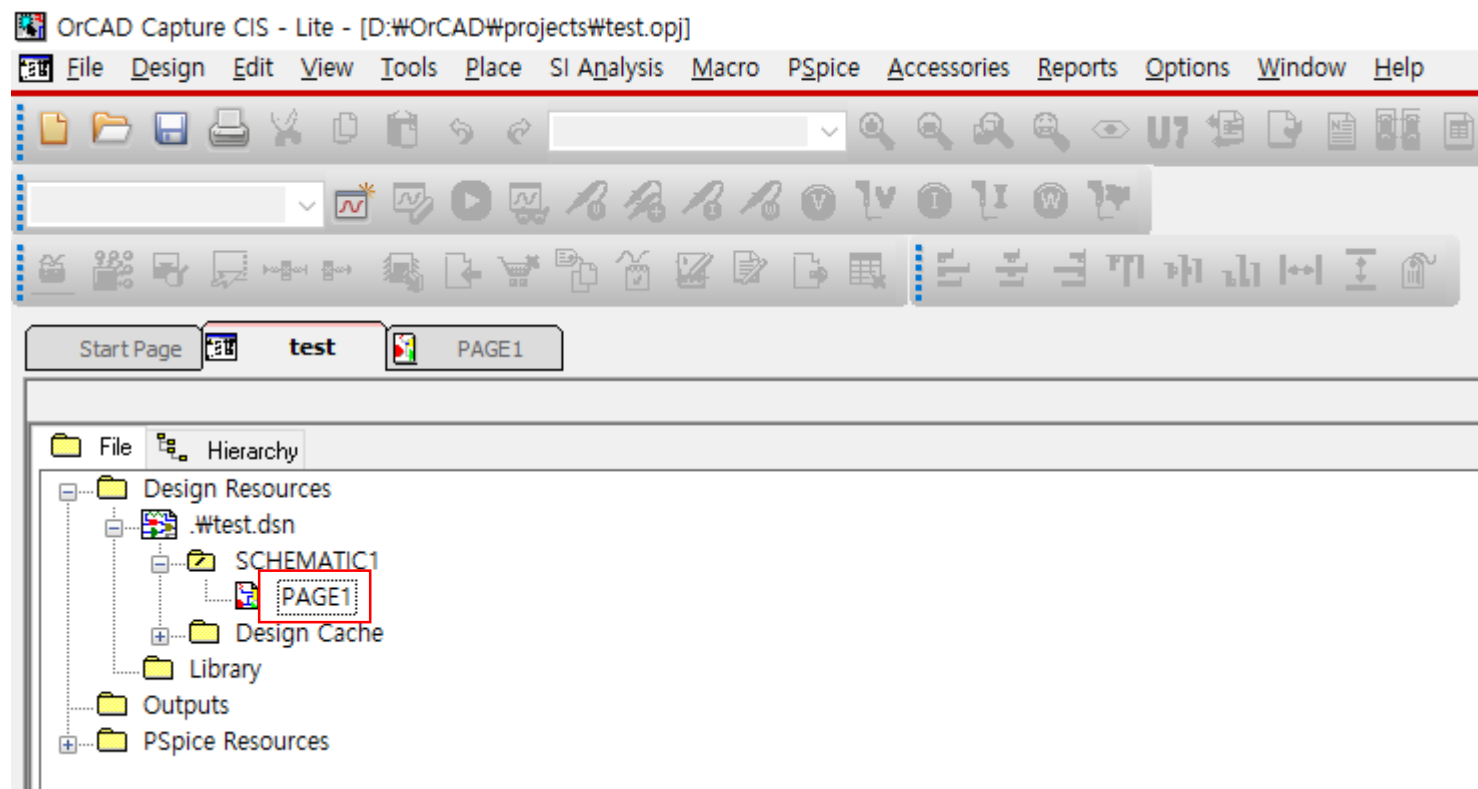
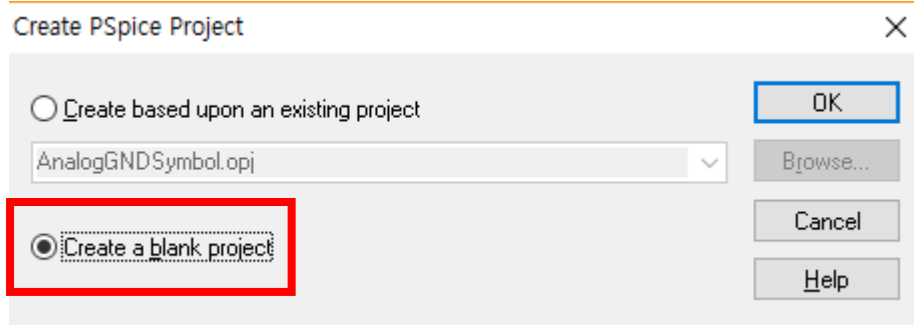
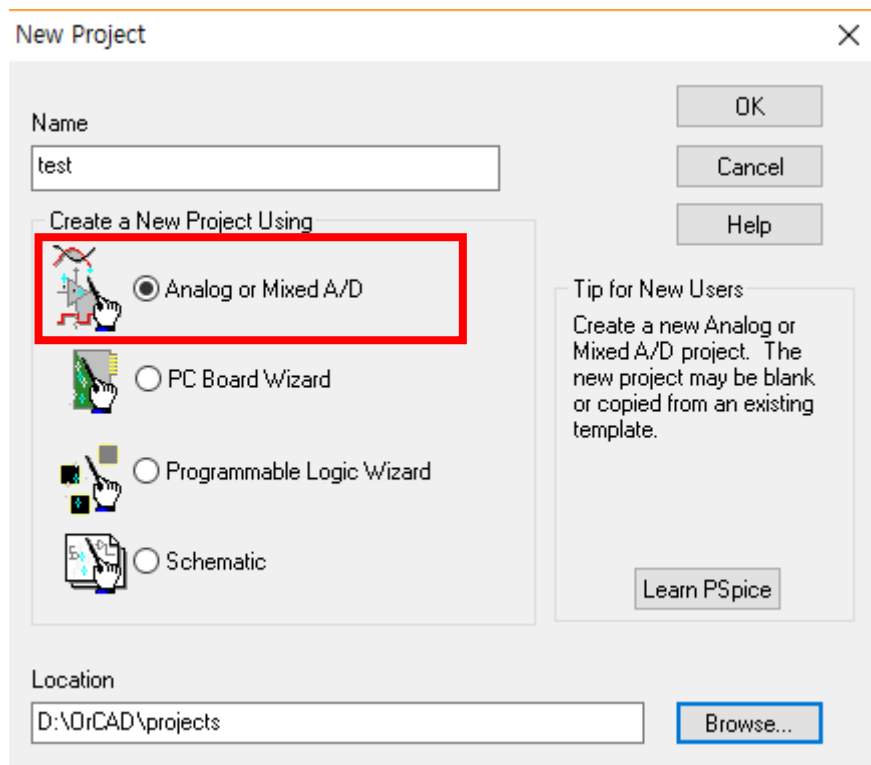


Simulation

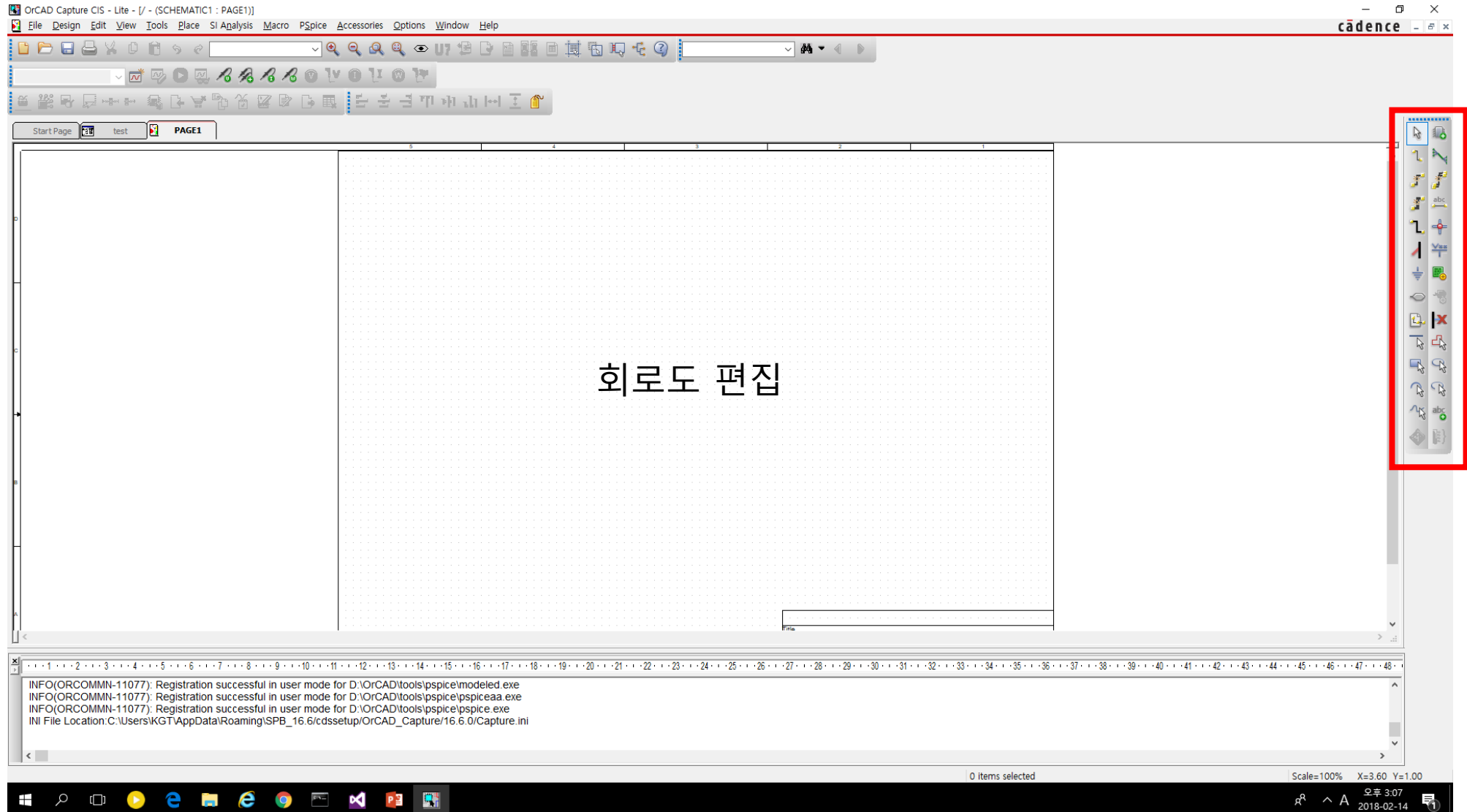


OrCAD Capture CIS Lite 실행 -> File -> New -> Project

Simulation



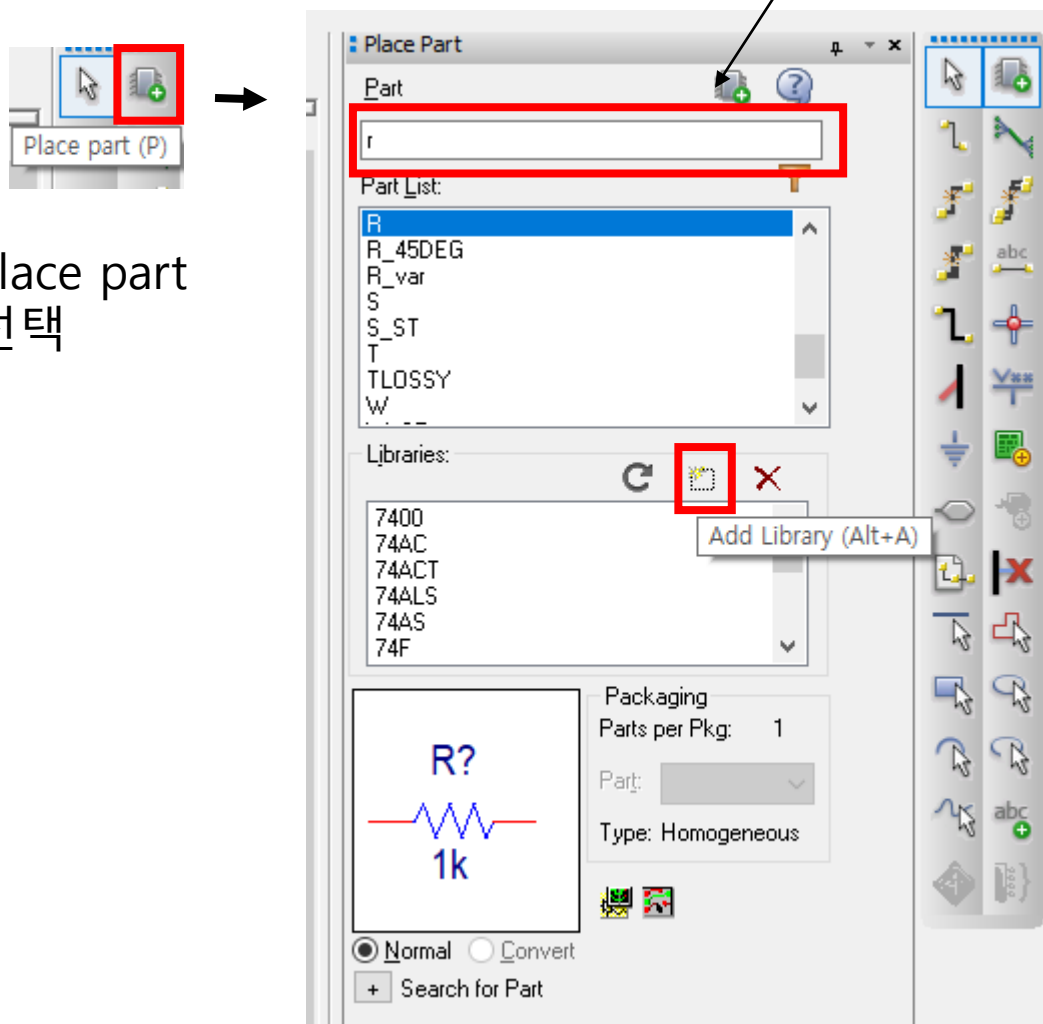
Simulation



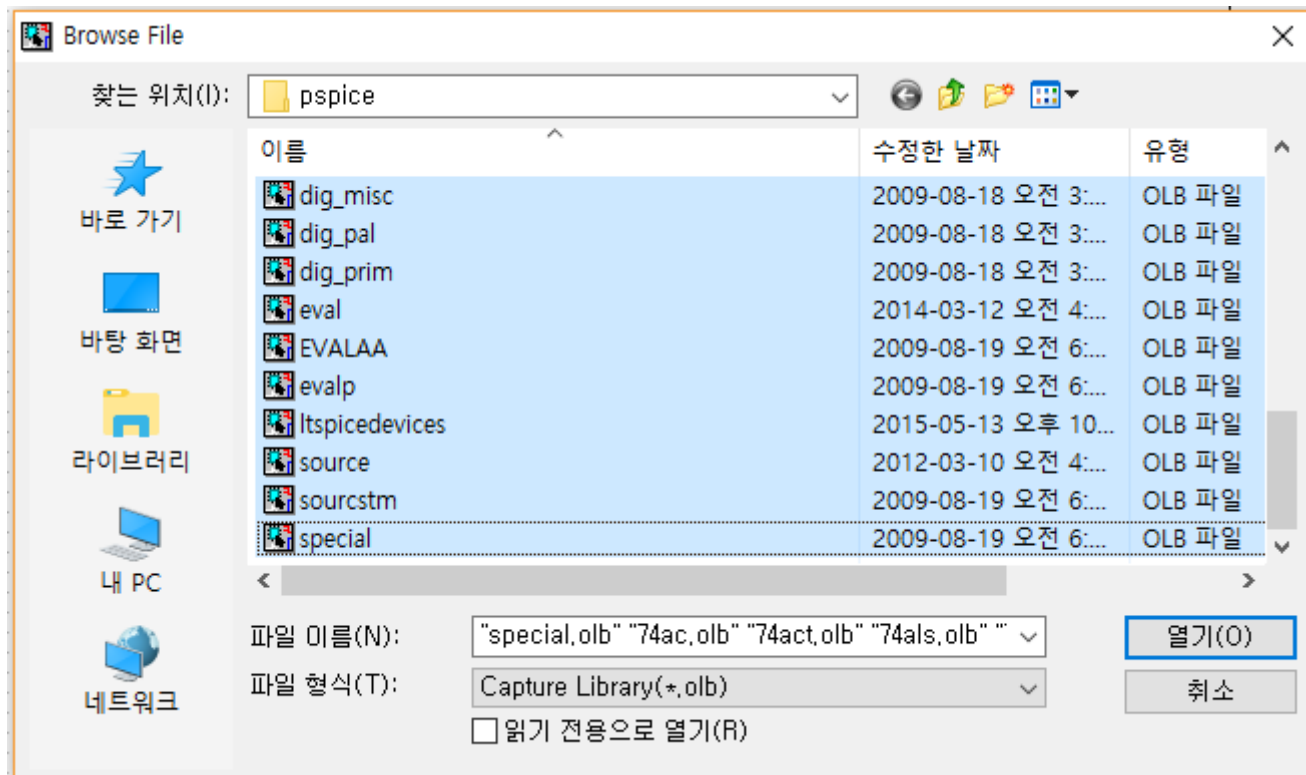
Simulation

원하는 부품 이름 입력

Place part
선택

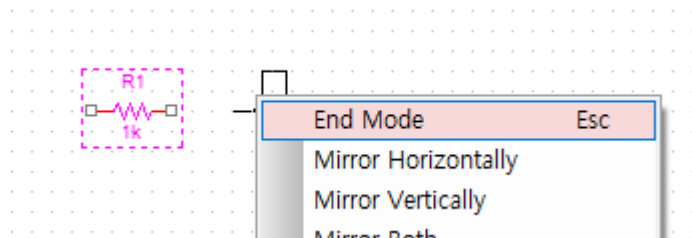


Add Library 선택

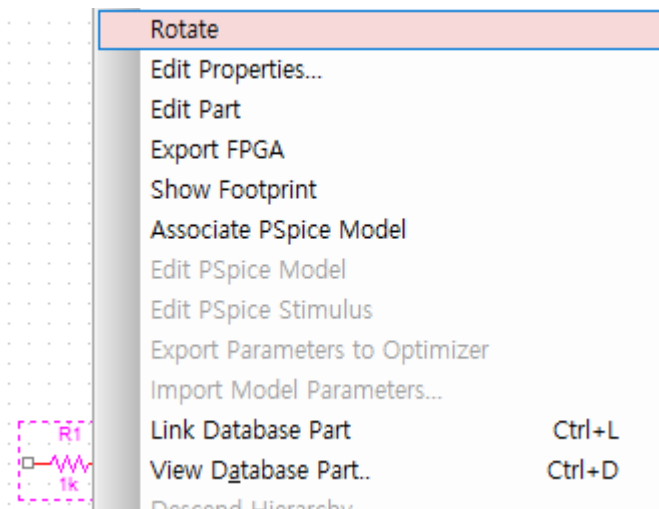


모든 라이브러리 불러오기

Simulation



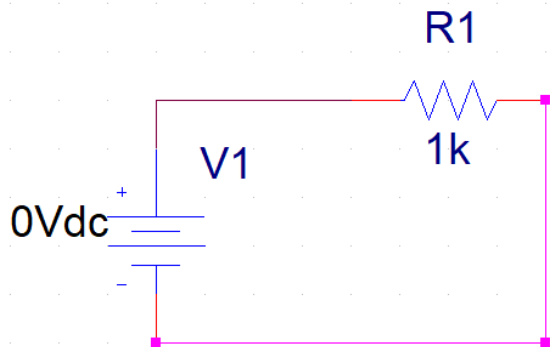
원하는 위치에 부품을 놓은 뒤, 마우스 오른쪽 버튼 클릭 후, End Mode 클릭하여 현재 작업 종료



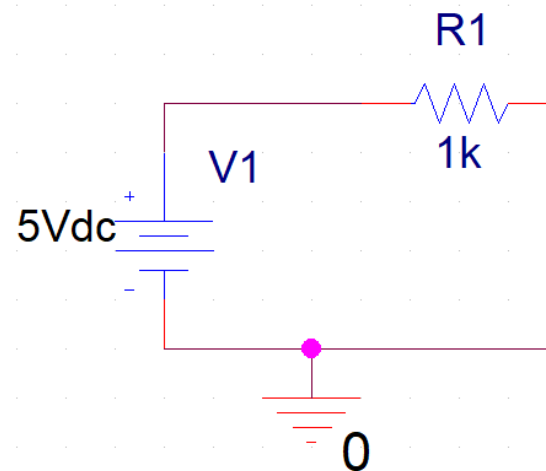
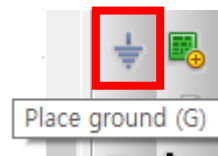
부품을 클릭한 다음 마우스 오른쪽 버튼 클릭 후, Rotate 선택 시 부품 회전 가능



Place wire 선택

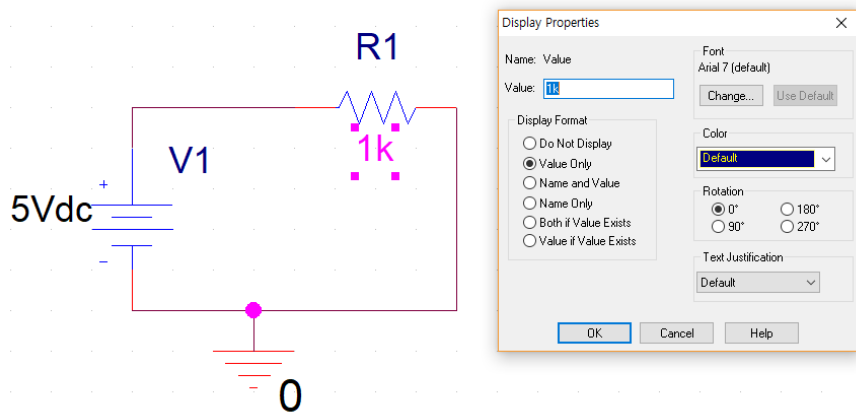


Place wire를 통해 선 연결

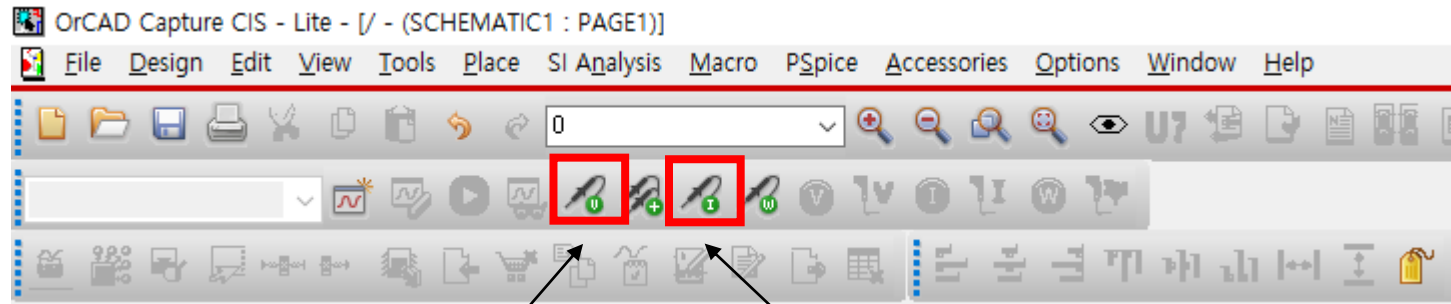


Place ground를 통해 접지

Simulation

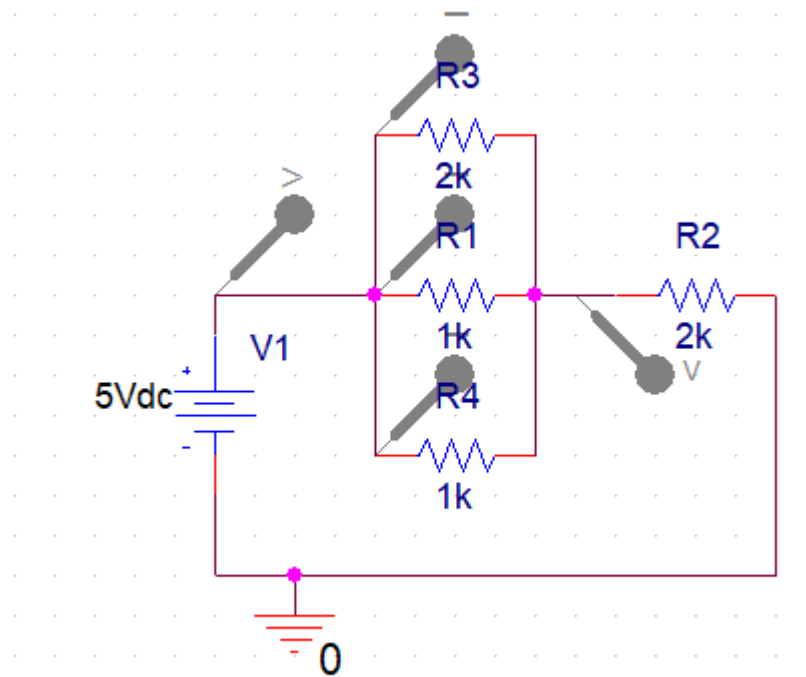


더블 클릭을 통해 값 수정



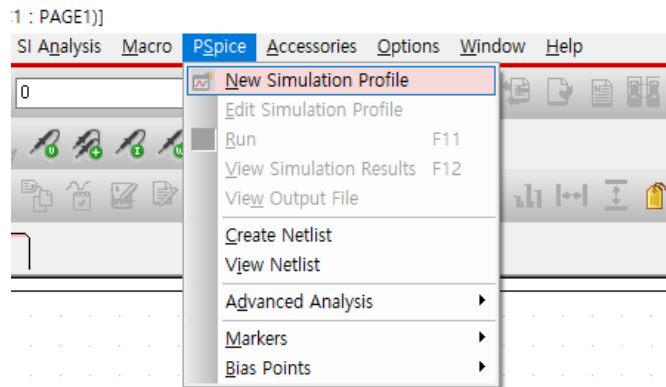
전압 파형 측정

전류 파형 측정

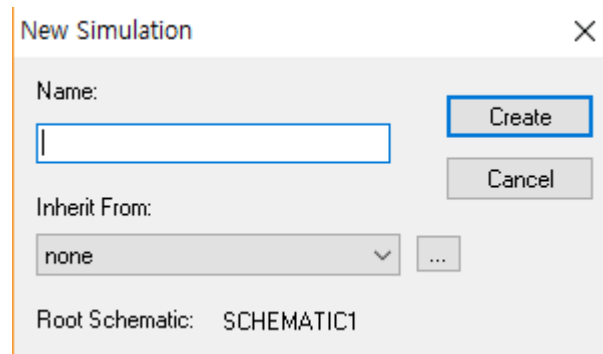


원하는 위치에 프로브

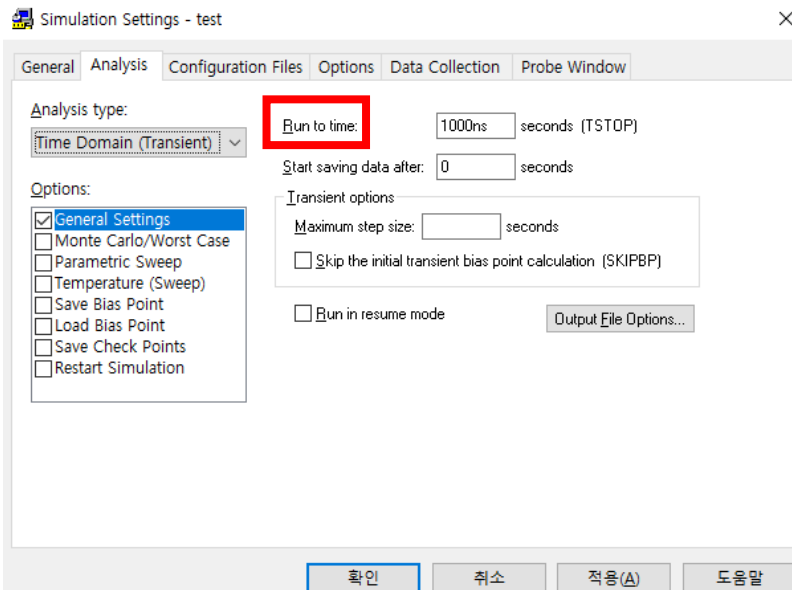
Simulation



Pspice->New Simulation Profile 선택

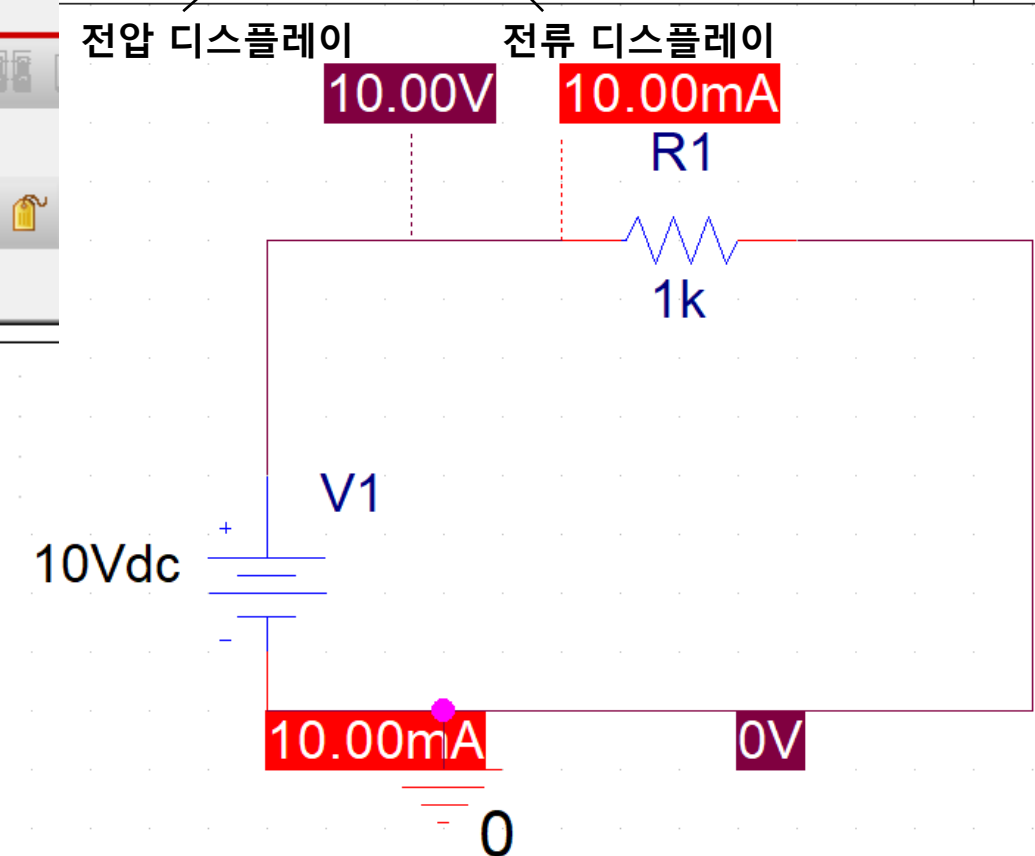
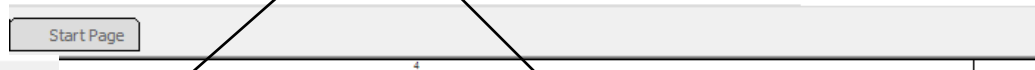


이름 입력후 Create 선택

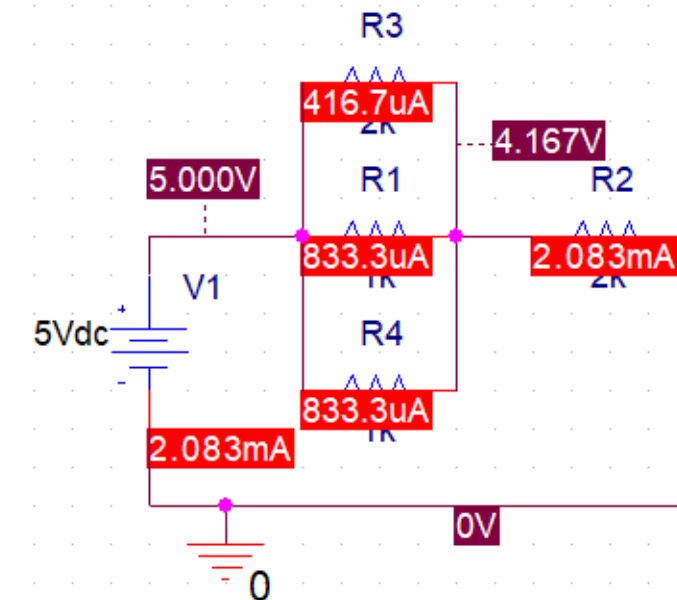


Run to time을 통해 시뮬레이션 시간 설정

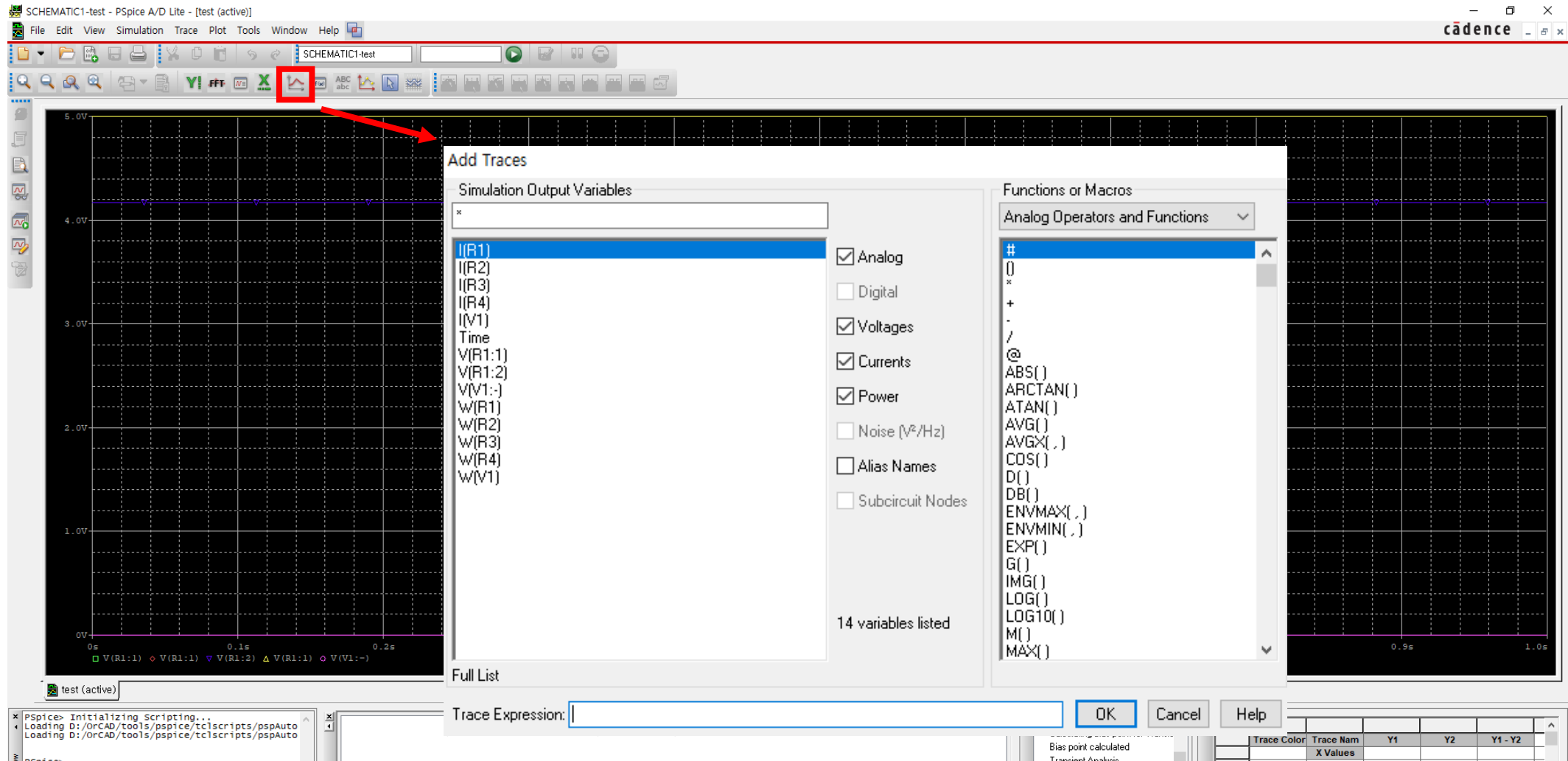
Simulation



Run을 클릭해 시뮬레이션 실행



Simulation



Trace하고자 하는 대상을 선택하여 파형을 관측함.