펌웨어 개발 및 회로 설계 기초

-5-

2019-02-28

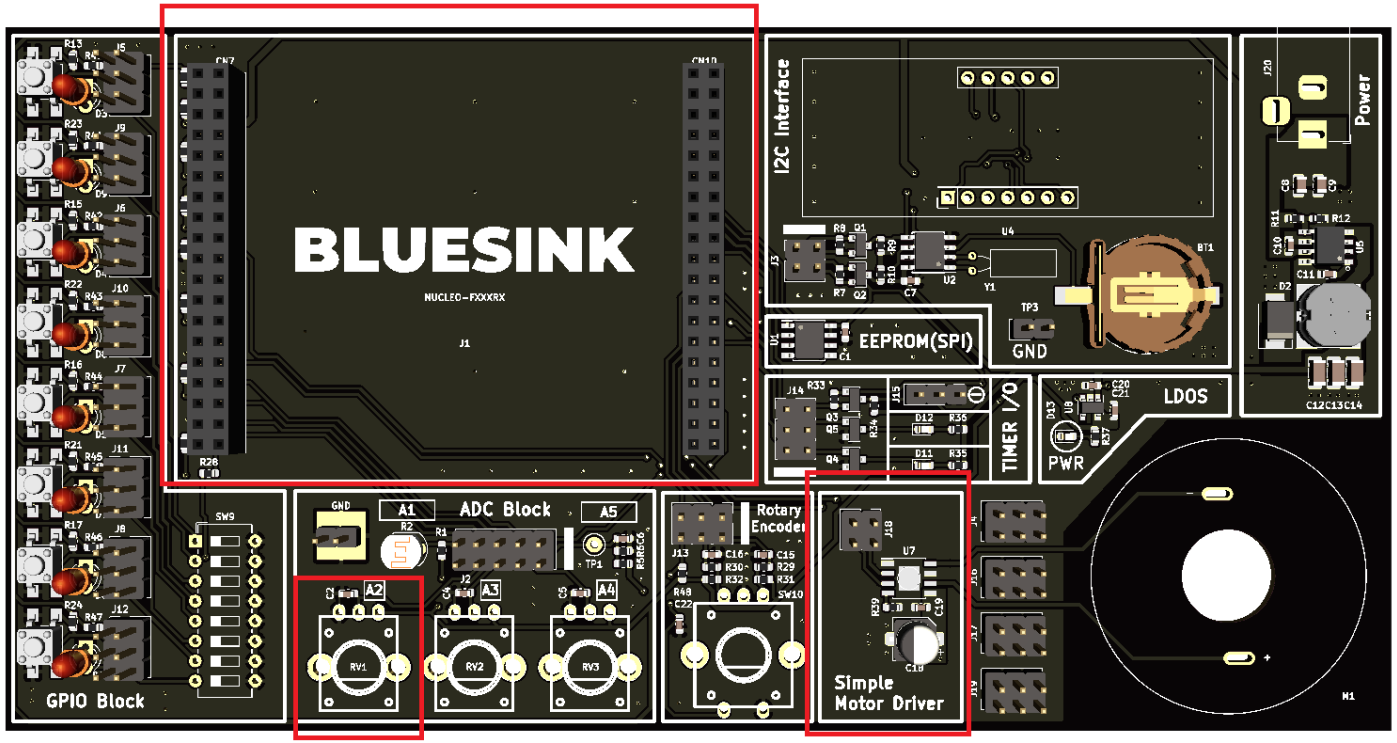
BLUESINK

0. 회로설계

0. 회로설계

• 회로설계 목표

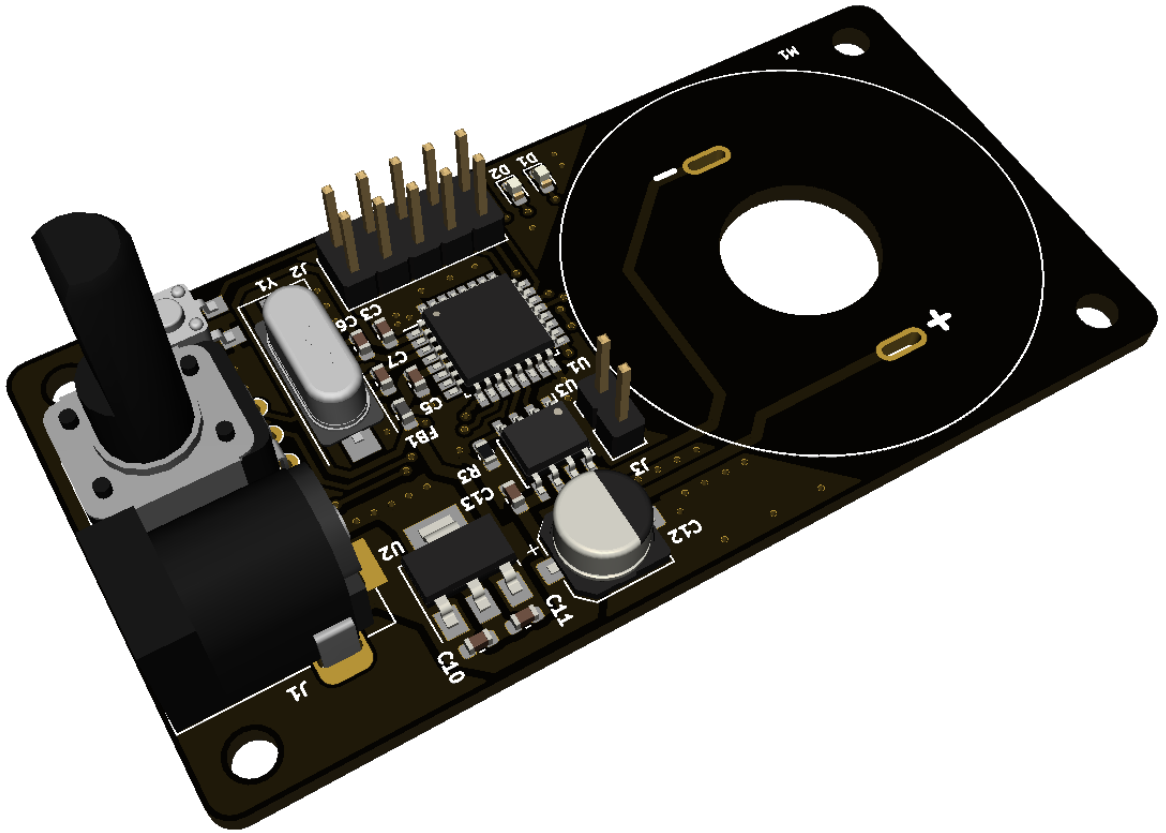
- 현재의 개발보드에서 **MCU, 가변저항, 모터 제어** 블록만 추려서 하나의 소형 보드로 제작한다.
- 가변저항을 통해 모터의 속도를 제어하는 회로 기판을 만들어본다.



0. 회로설계

• 회로설계 목표

- 현재의 개발보드에서 **MCU, 가변저항, 모터 제어** 블록만 추려서 하나의 소형 보드로 제작한다.
- 가변저항을 통해 모터의 속도를 제어하는 회로 기판을 만들어본다.

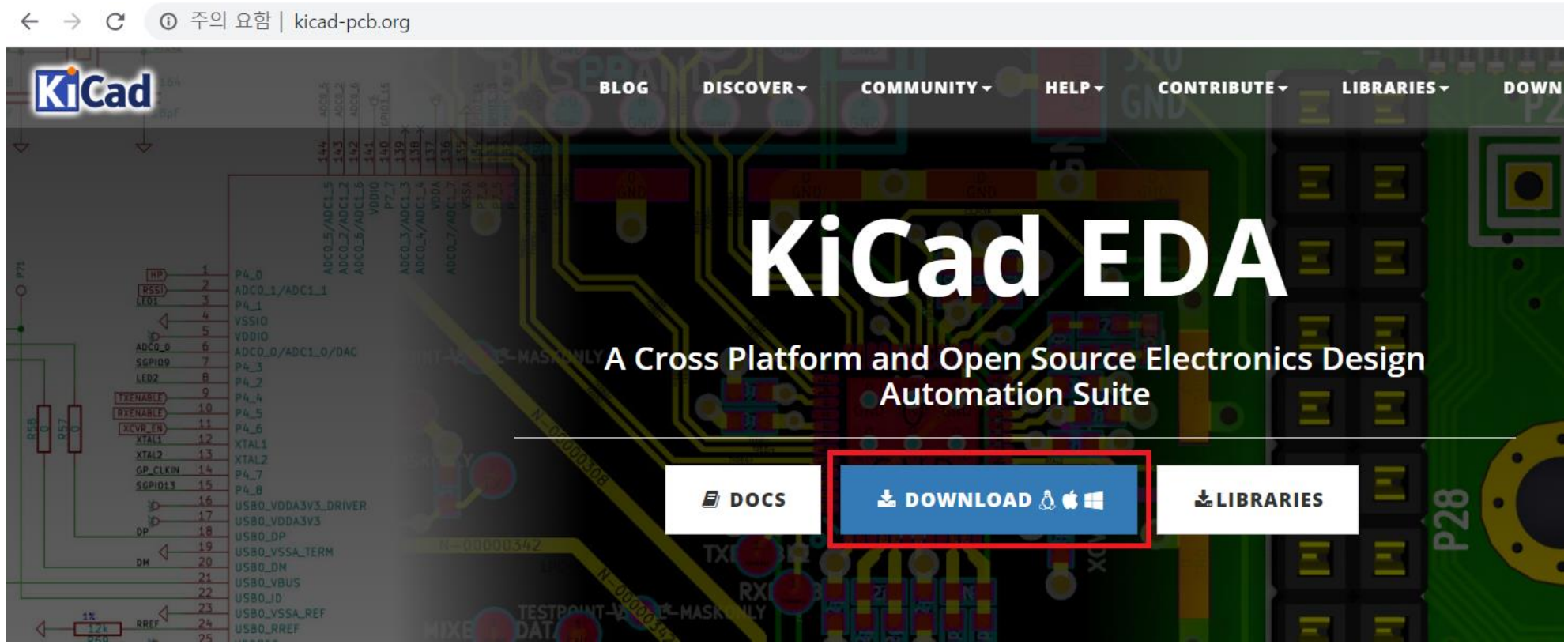


1. KiCad

1. KiCad

• KiCad 설치하기

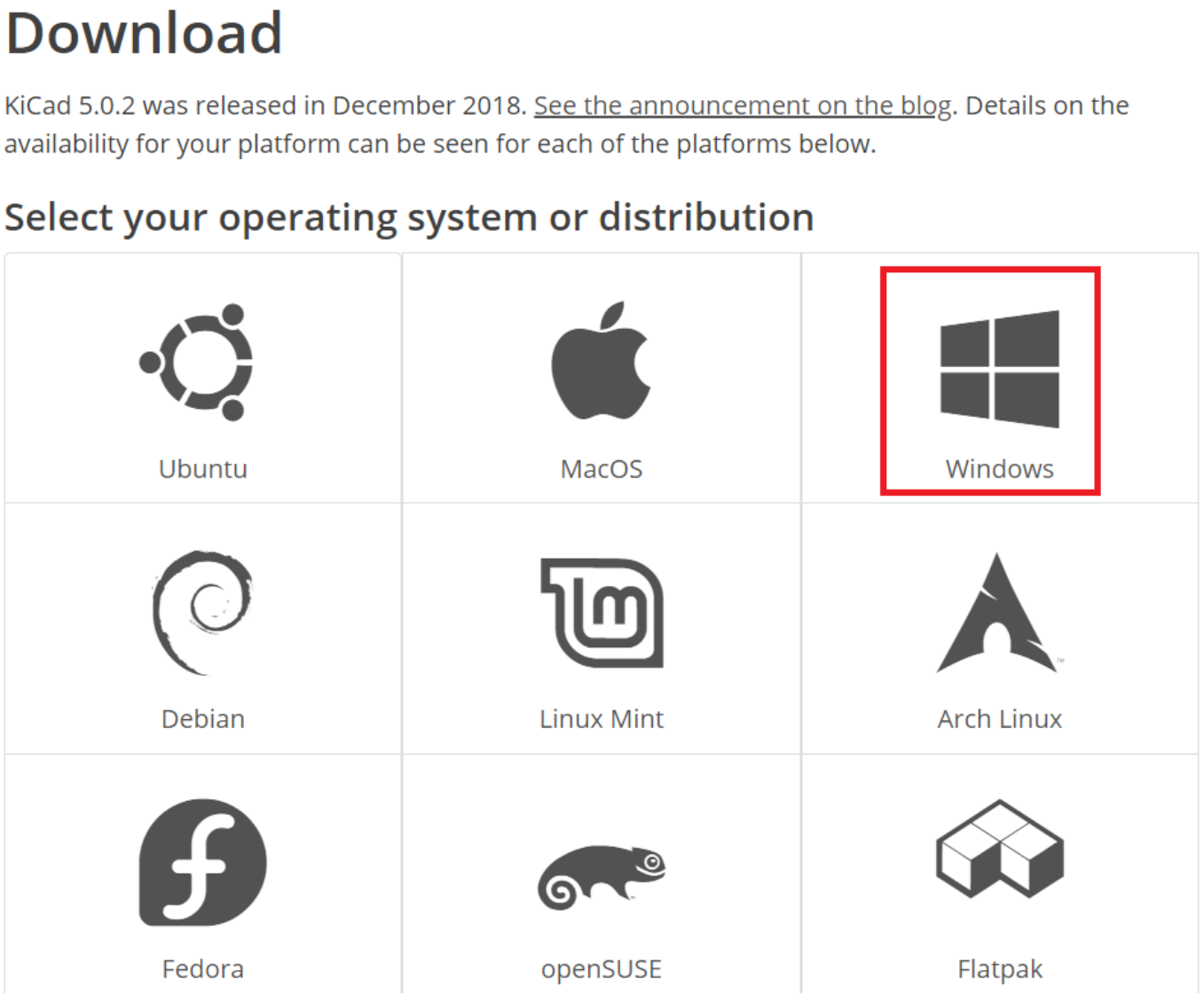
- http://kicad-pcb.org/
- KiCad 공식 홈페이지로 들어가 DOWNLOAD 클릭



1. KiCad

• KiCad 설치하기

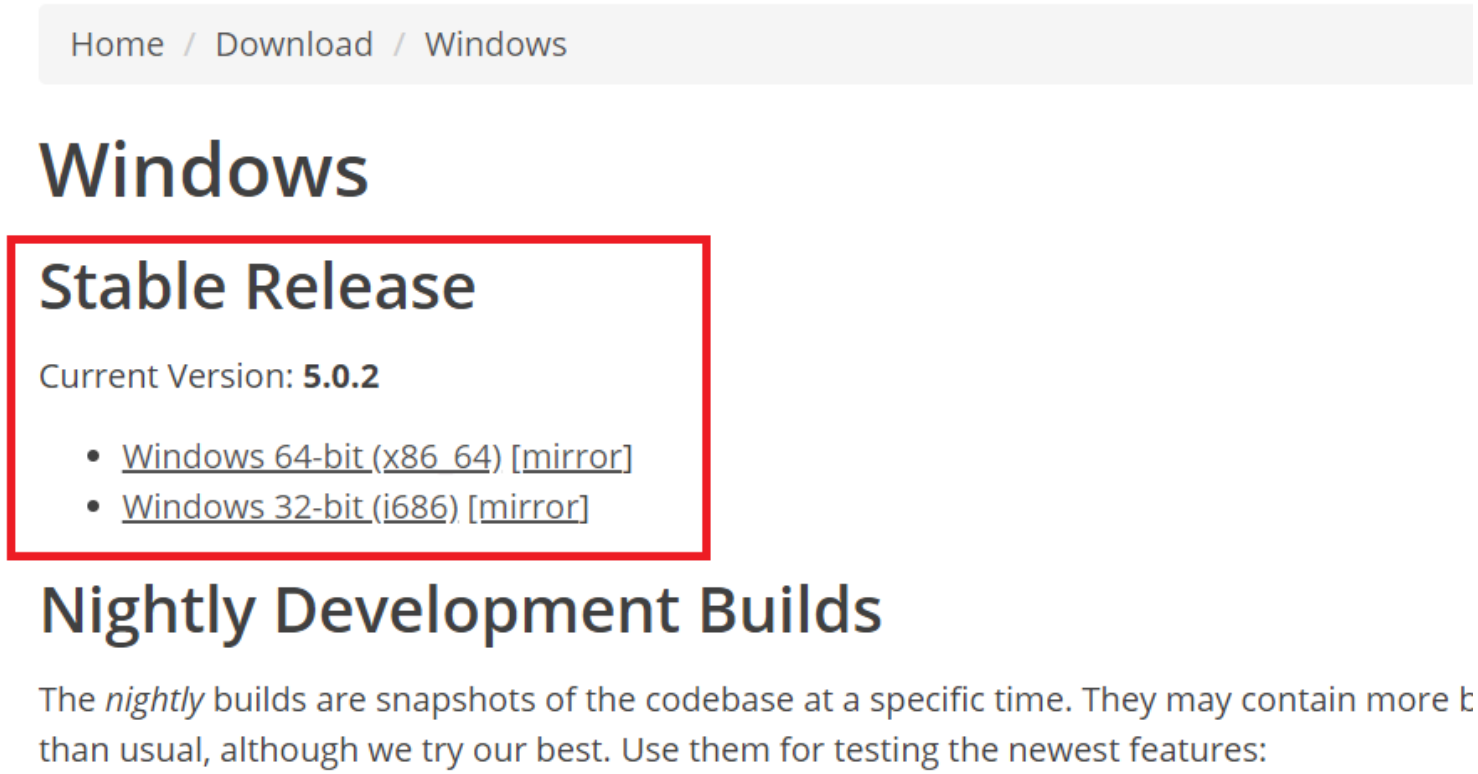
- 자신에게 맞는 OS 클릭



1. KiCad

• KiCad 설치하기

- Stable Release 버전 다운로드 및 설치

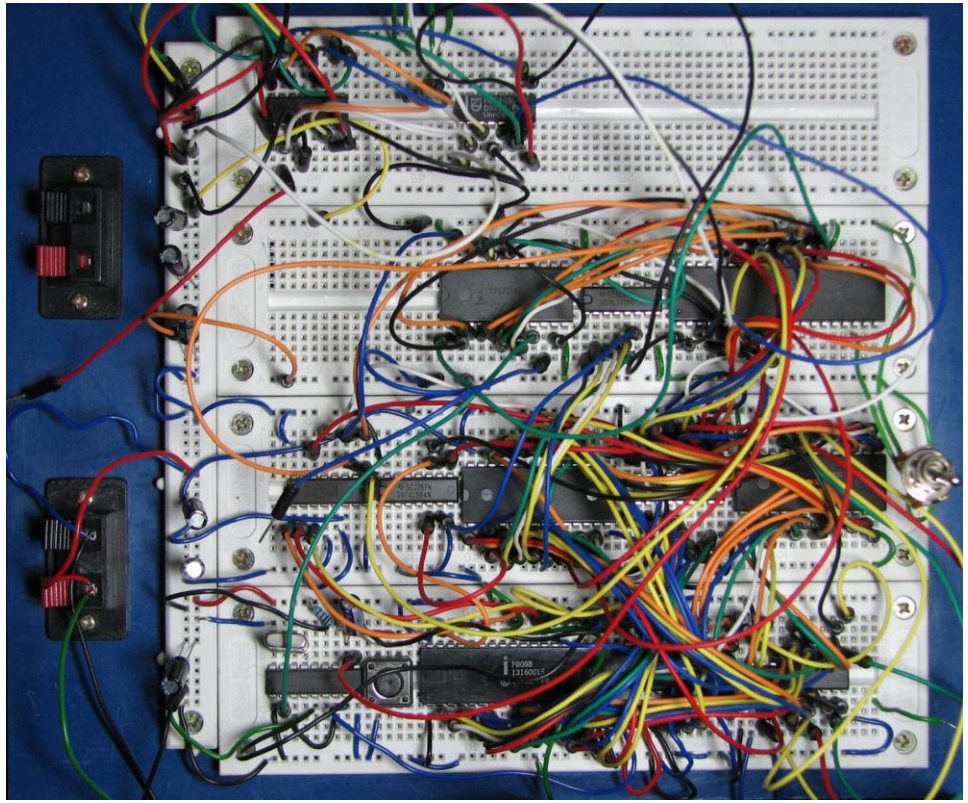


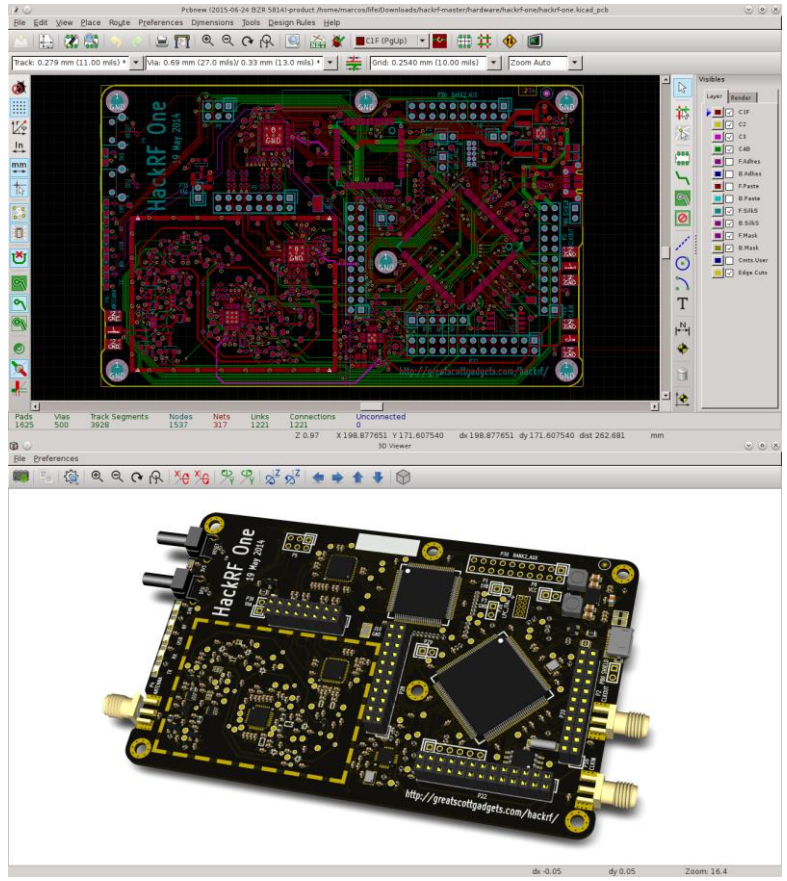
1. PCB

1. PCB

- **PCB (Printed Circuit Board)**

- **PCB**는 인쇄회로기판을 의미하며, PCB 제작을 통해 더 작고, 신뢰성 있는 제품을 만들 수 있다.





1. PCB

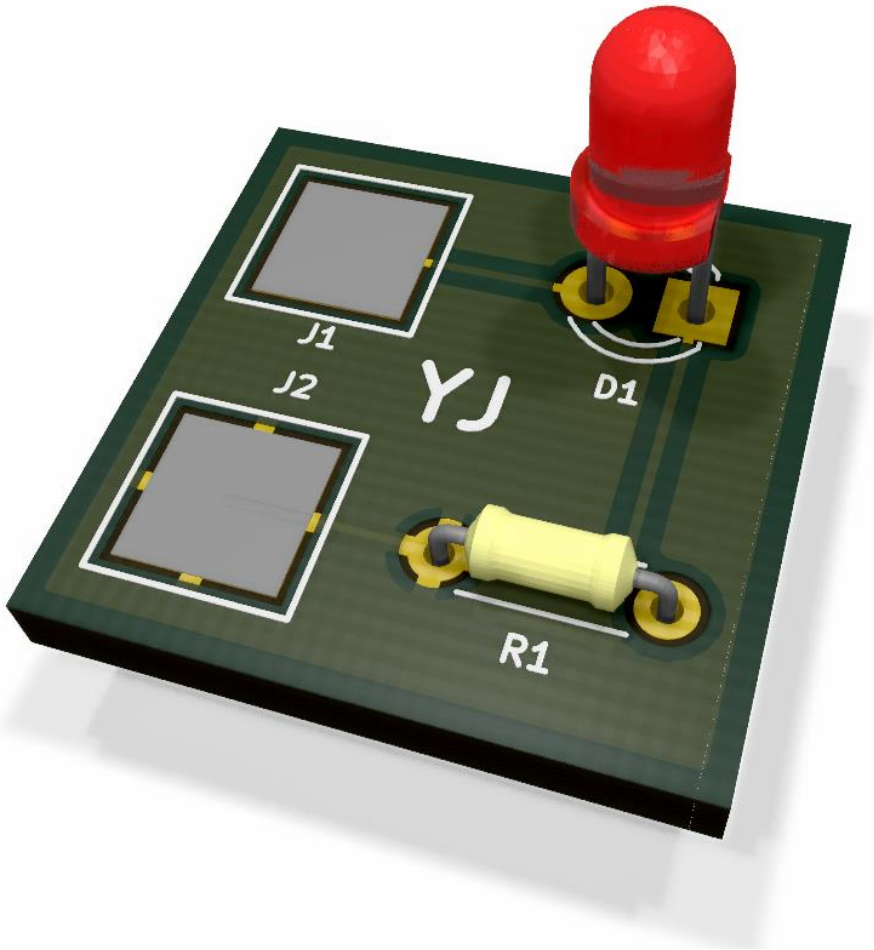
• Artwork Program

- 회로 설계를 위한 다양한 프로그램이 존재하며, 본 강의에서는 **KiCad**를 사용한다.
- KiCad의 특징은 다음과 같다.

  - ➢ Open Source
  - ➢ No functional limitation
  - ➢ Managed by CERN

1. PCB

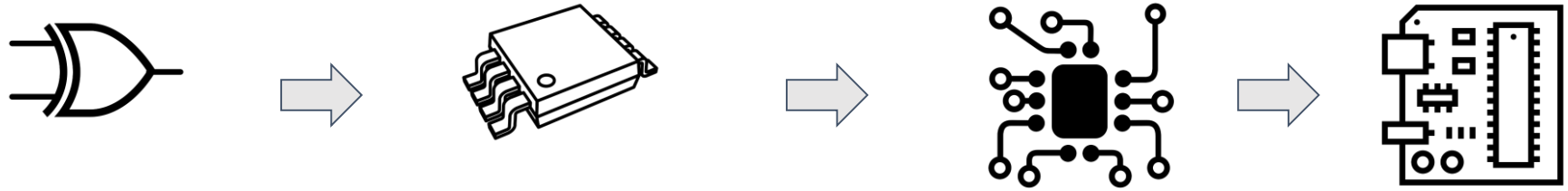
오늘의 목표



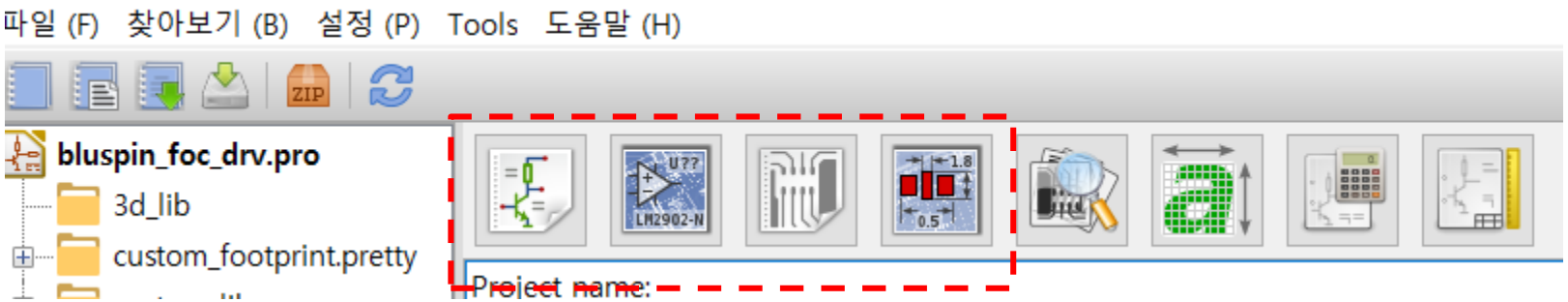
BLUESINK

1. PCB

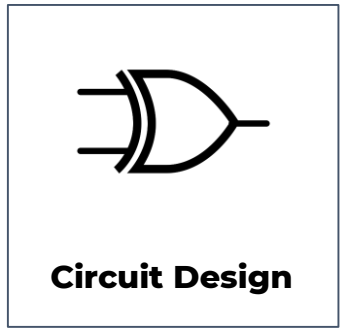
PCB DESIGN PROCESS



Circuit Design **Specifying Parts** **Arrange & Routing** **Order**



1. PCB

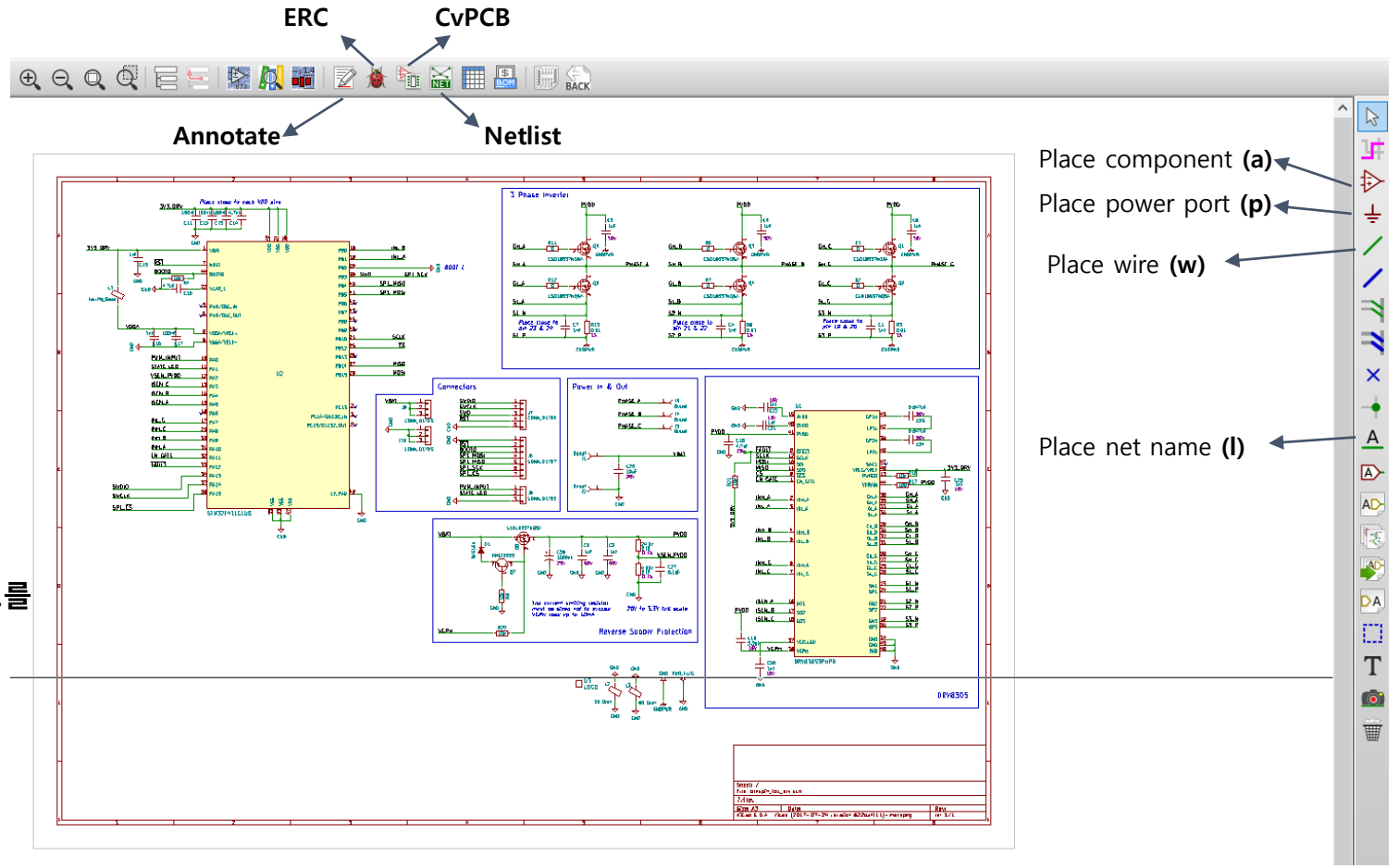


회로도 그리기

주요 기능 소개

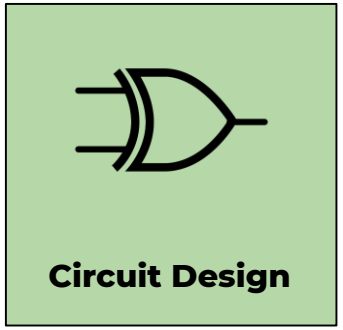
*단축키를 익히면 작업 속도를 매우 증가시킬 수 있다!

Component **m**ove (**m**)
Component **c**opy (**c**)
Component **r**otate (**r**)
Component property (**e**)



BLUESINK

1. PCB

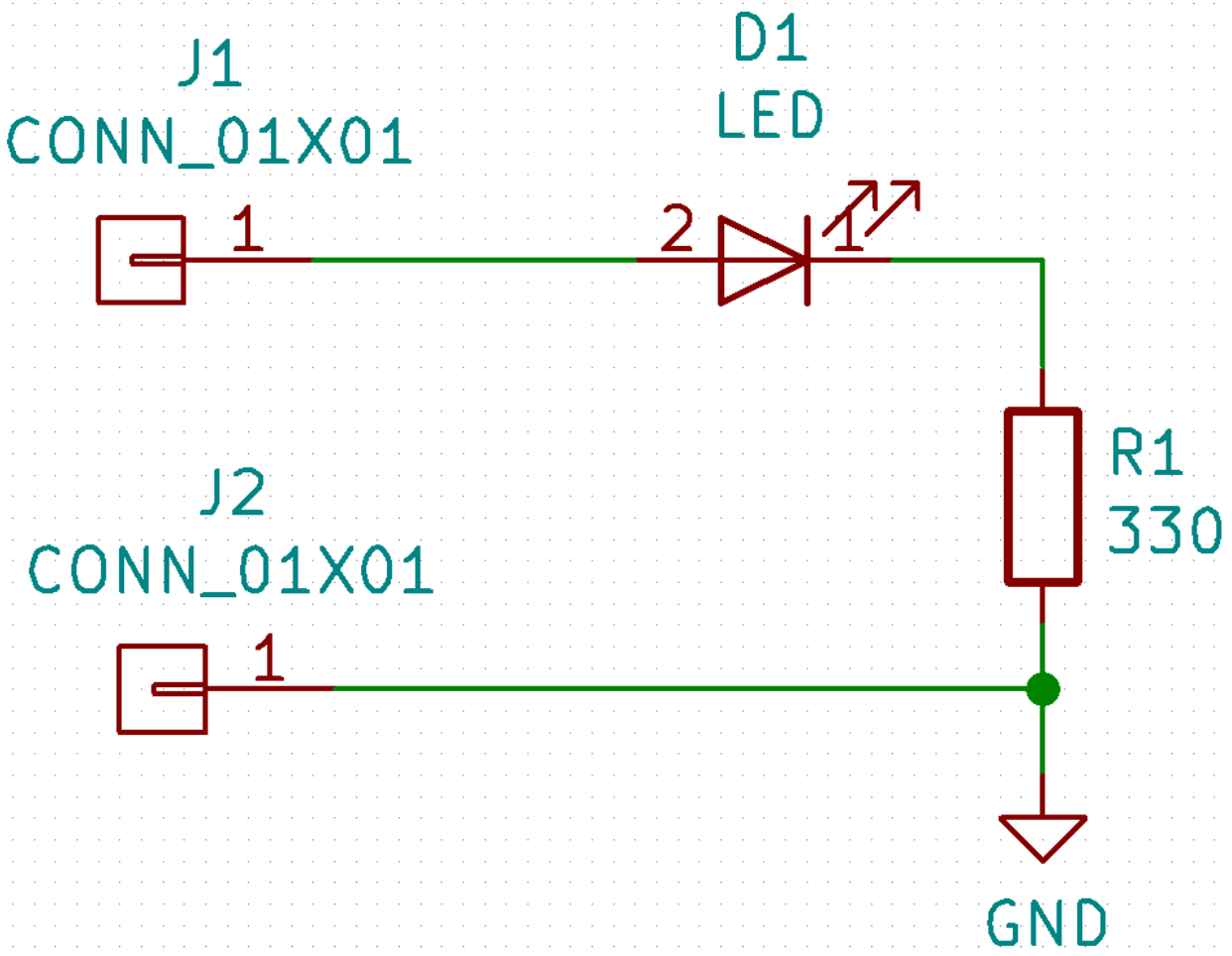


회로도 그리기

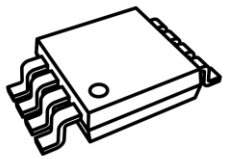
주요 기능 소개

*단축키를 익히면 작업 속도를
매우 증가시킬 수 있다!

Component **m**ove (**m**)
Component **c**opy (**c**)
Component **r**otate (**r**)
Component property (**e**)



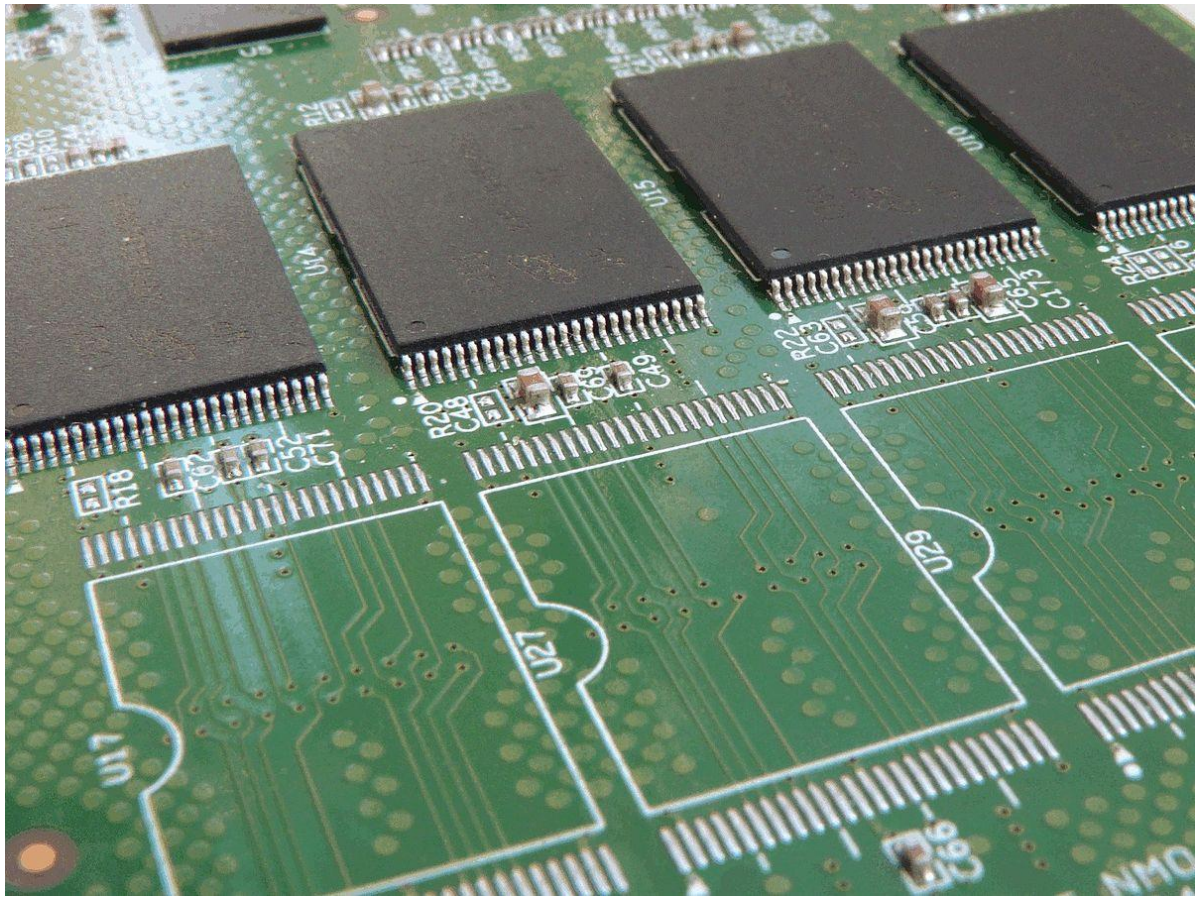
1. PCB



Specifying Parts

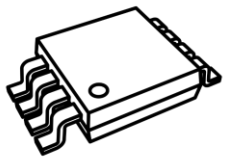
FOOTPRINT 란?

전자 부품이 PCB에 놓이게 될 때 차지하는 영역



BLUESINK

1. PCB



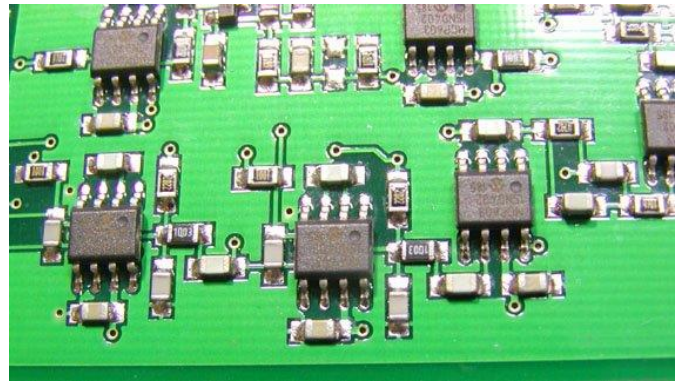
Specifying Parts

FOOTPRINT 의 종류

- **SMD :** PCB 표면에 부착되는 부품
- **THT :** PCB를 관통해, 구멍을 통해 부착되는 부품

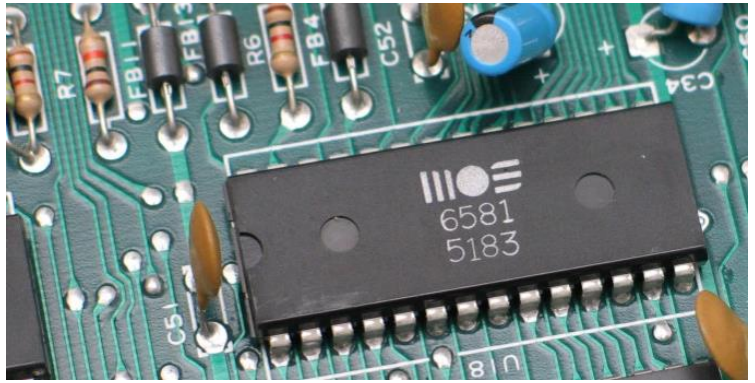
SMD

Surface Mount Devices

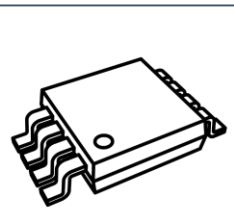


THT

Through Hole

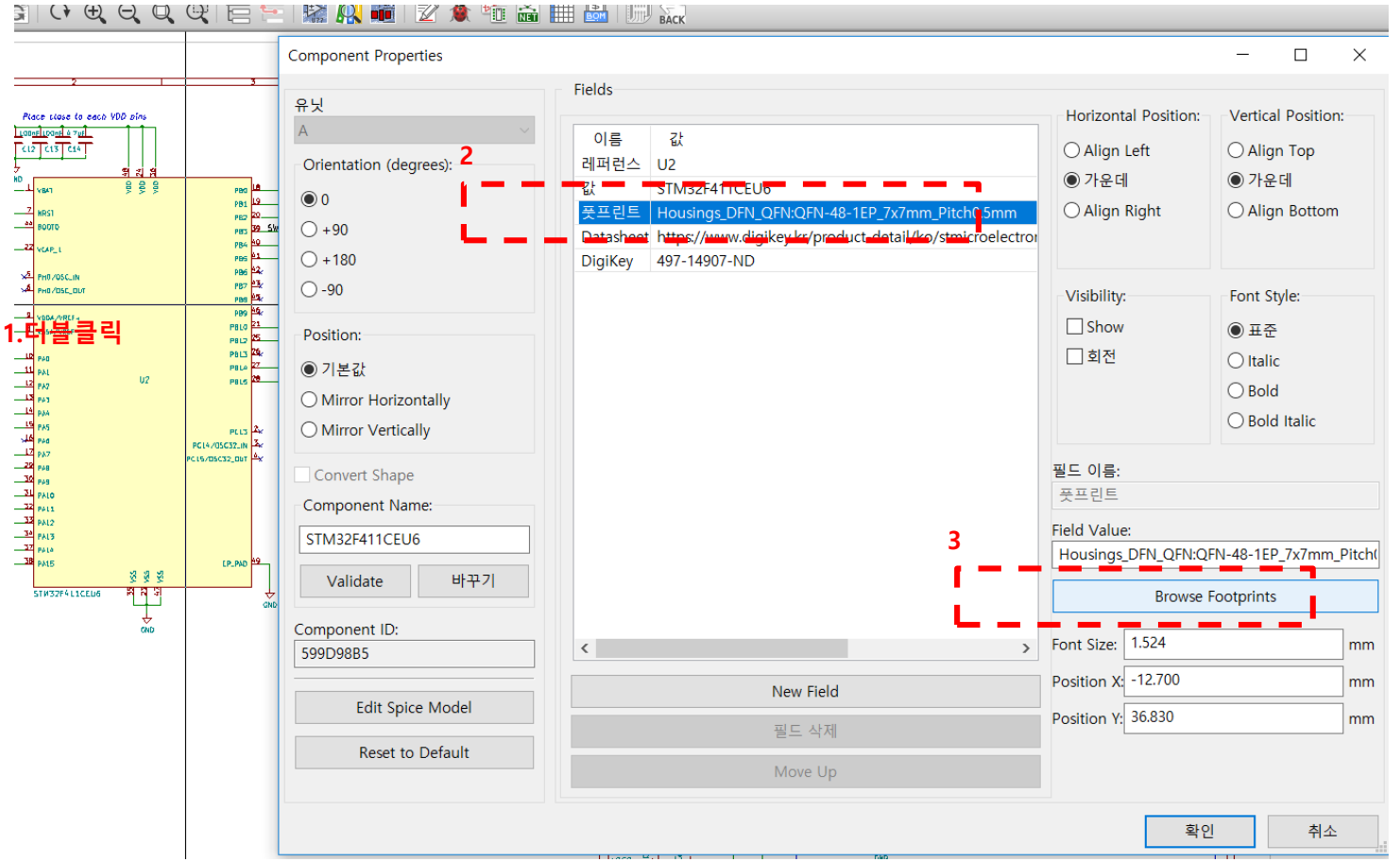


1. PCB



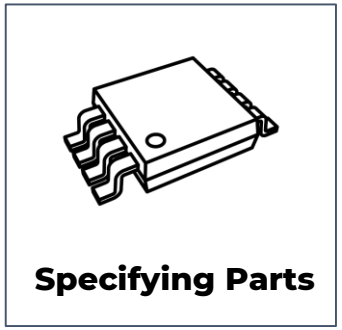
Specifying Parts

FOOTPRINT 지정하기

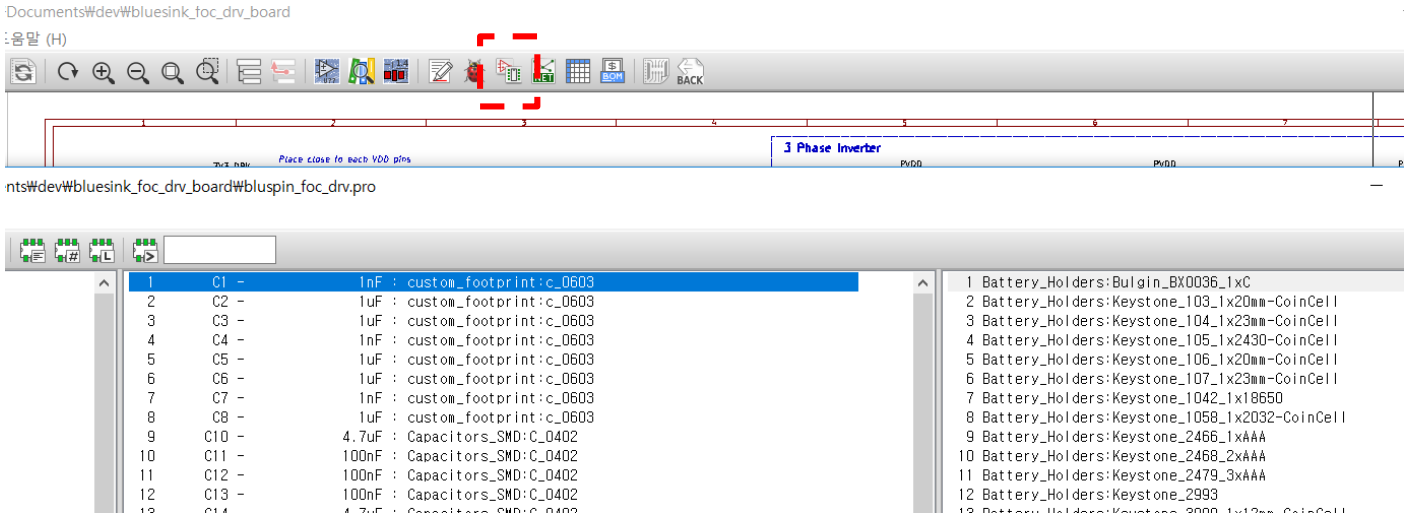


<방법 1>
부품 Property 탭에서 지정

1. PCB



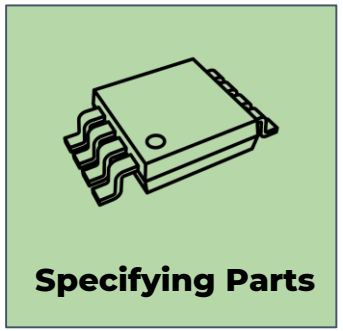
FOOTPRINT 지정하기

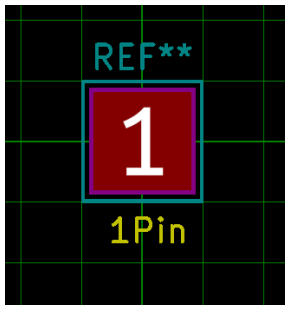


<방법 2>
CvPCB를 통해 지정

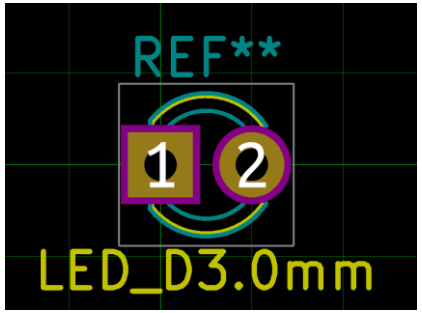
=> 각 부품 property에서 footprint를 지정하고, 최종적으로 CvPCB 에서 확인하는것을 권장

1. PCB

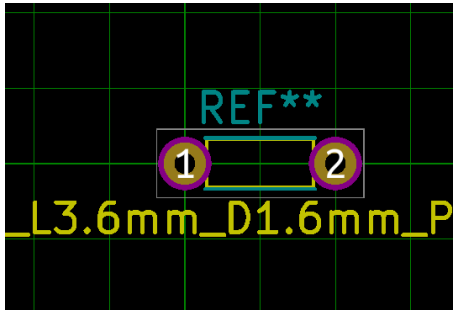




SMD_Packages -> 1Pin



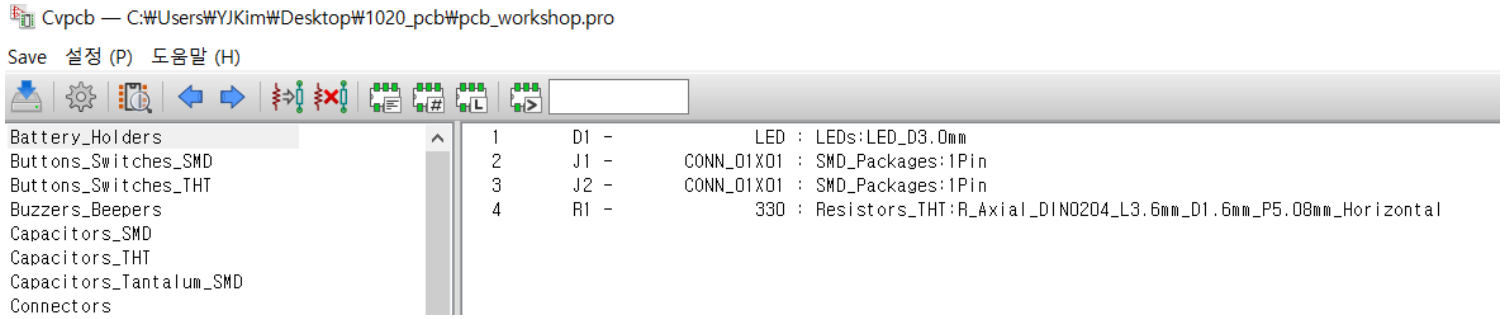
LEDs -> LED_D3.0mm



Resistors_THT

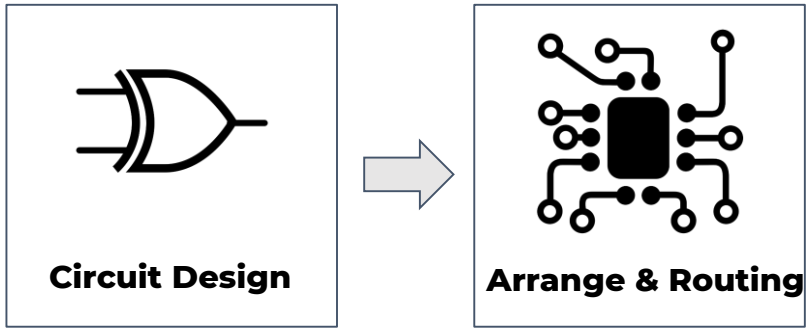
FOOTPRINT 지정하기

실습



=> 각 부품 property에서 footprint를 지정하고, 최종적으로 CvPCB 에서 확인하는것을 권장

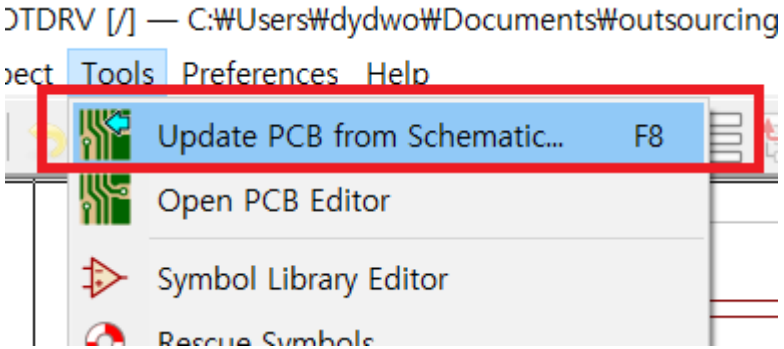
1. PCB



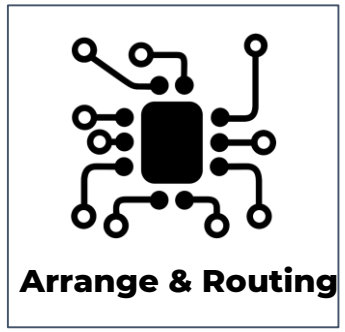
연결 관계 전달

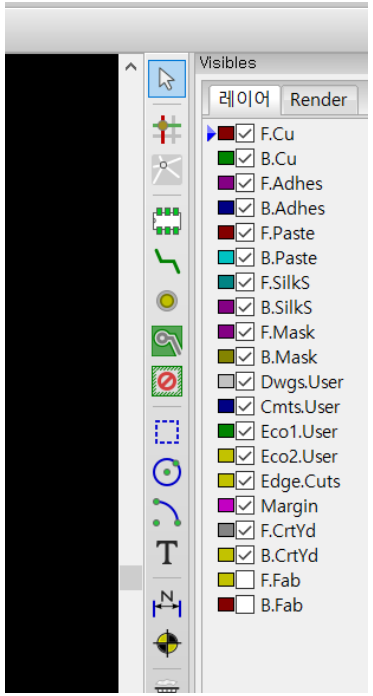
Tools -> Update PCB from Schematic... 클릭

혹은 **F8 단축키** 사용



1. PCB



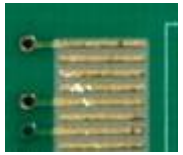


Layer의 이해

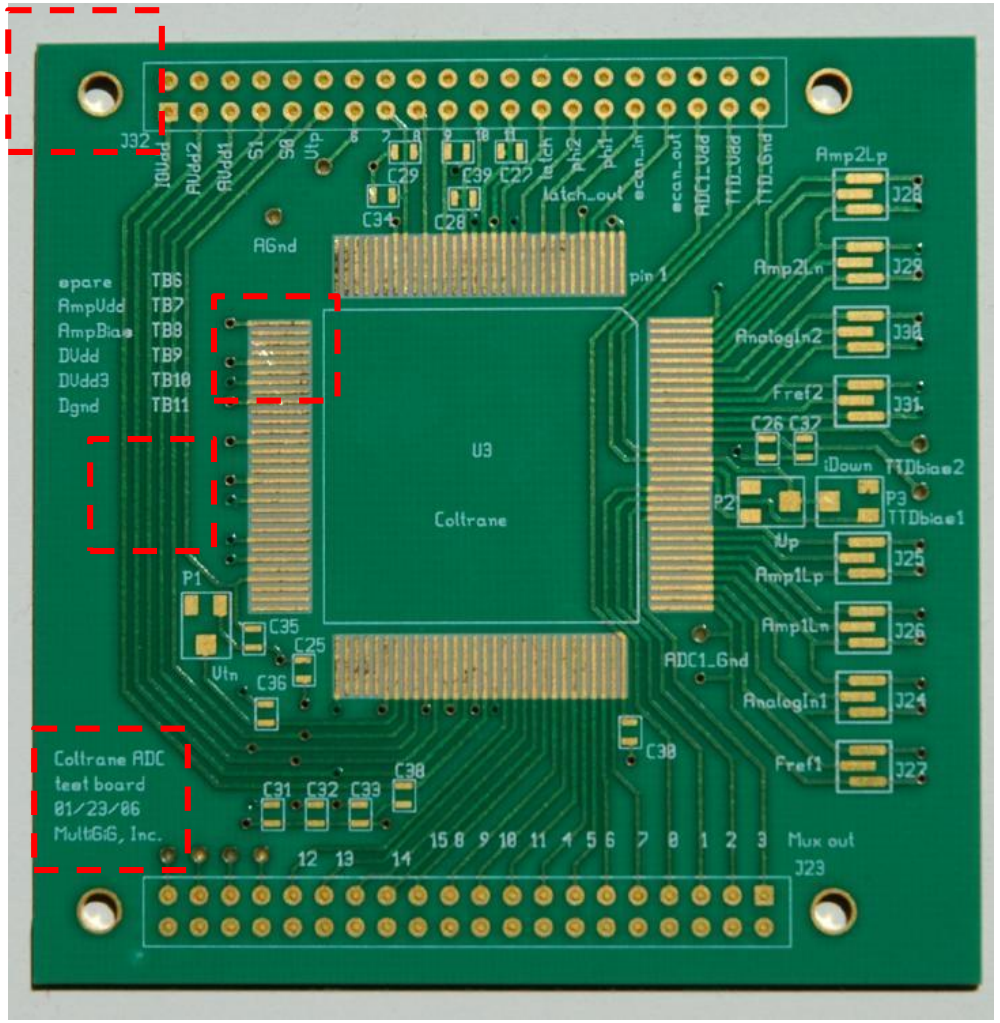
Edge Cuts

Copper Layer

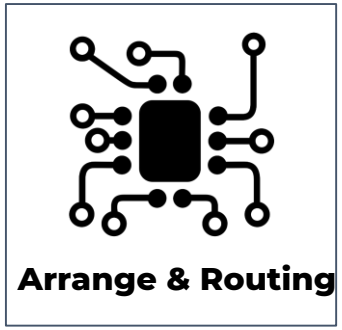
Mask Layer



Silk Layer



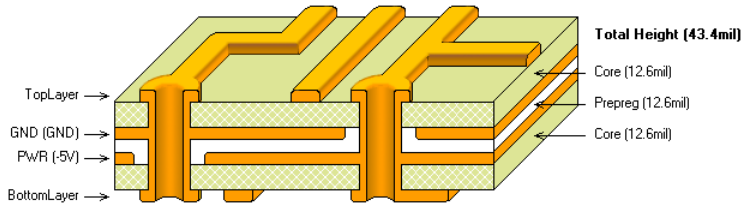
1. PCB

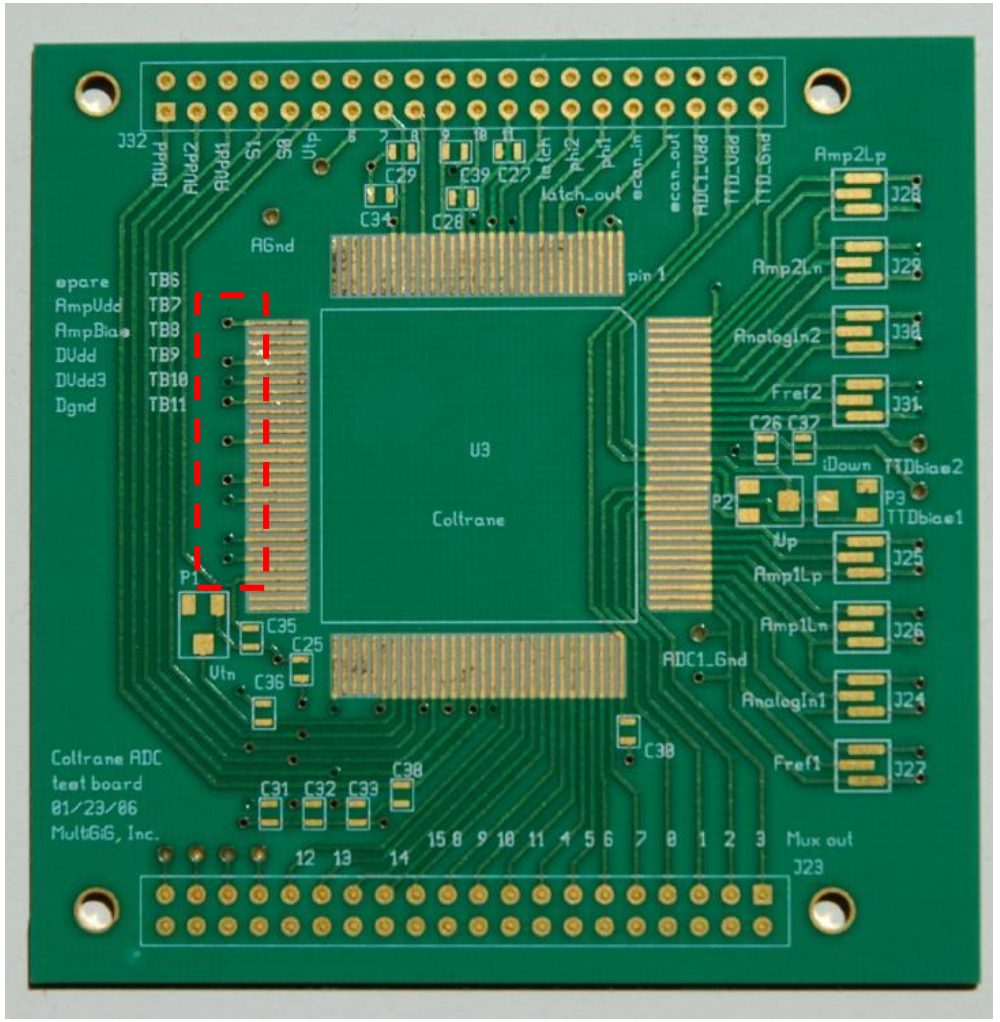


PCB 의 이해

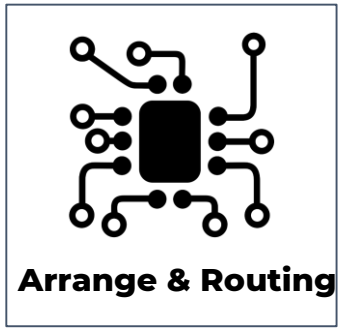
Via (비아)

기판의 회로 선이 다른 면으로 가기 위해 뚫리는 구멍

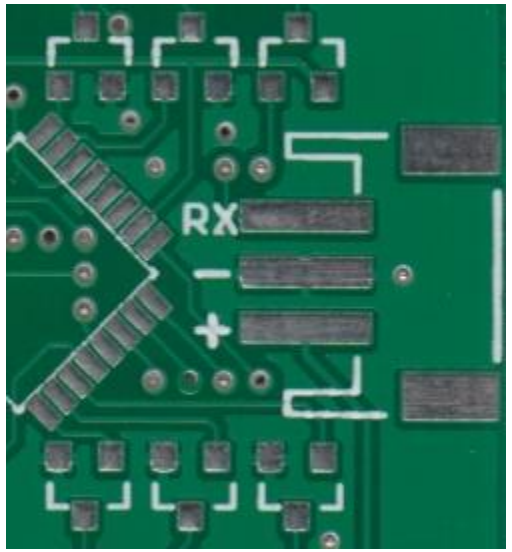




1. PCB



PCB 의 이해

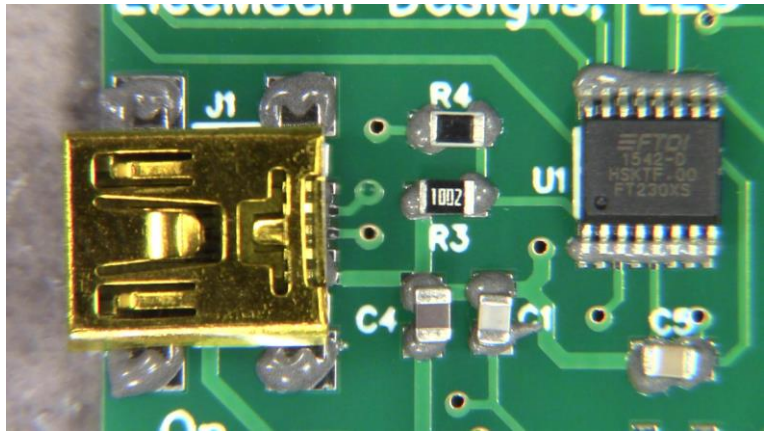


Ground Zone

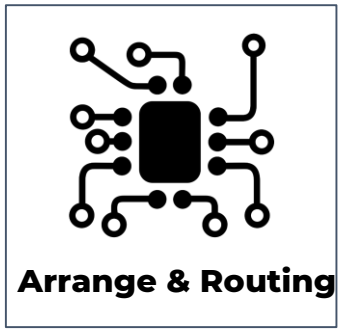
ZONE (존)

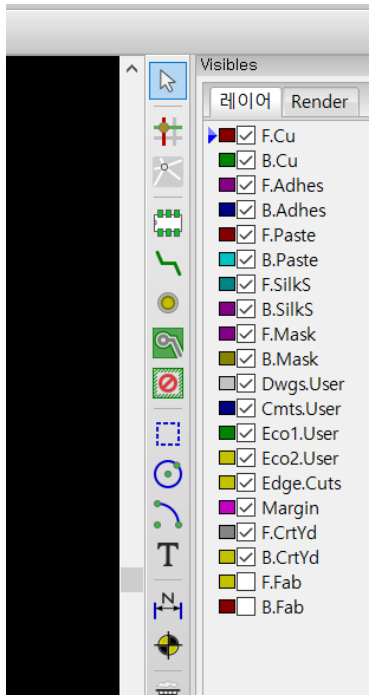
회로 기판에서 면으로 깔리는 영역

No Zone

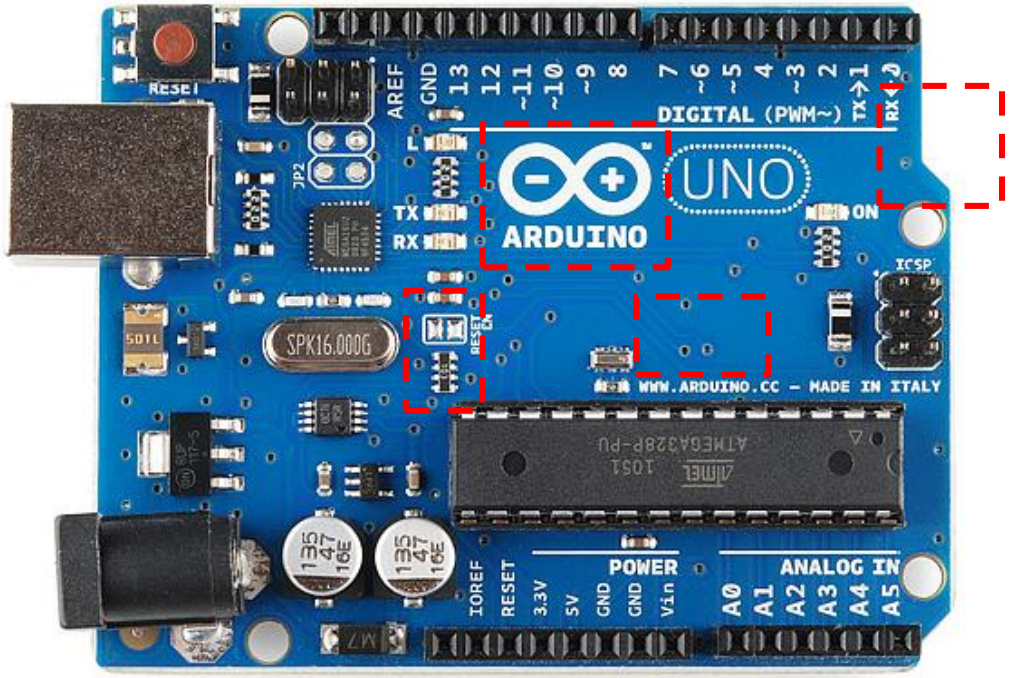


1. PCB



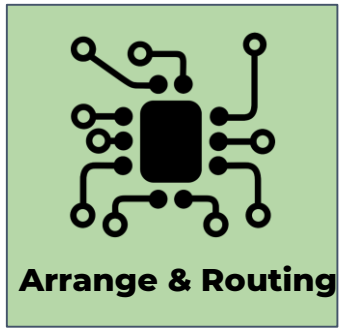


다시보는 아두이노



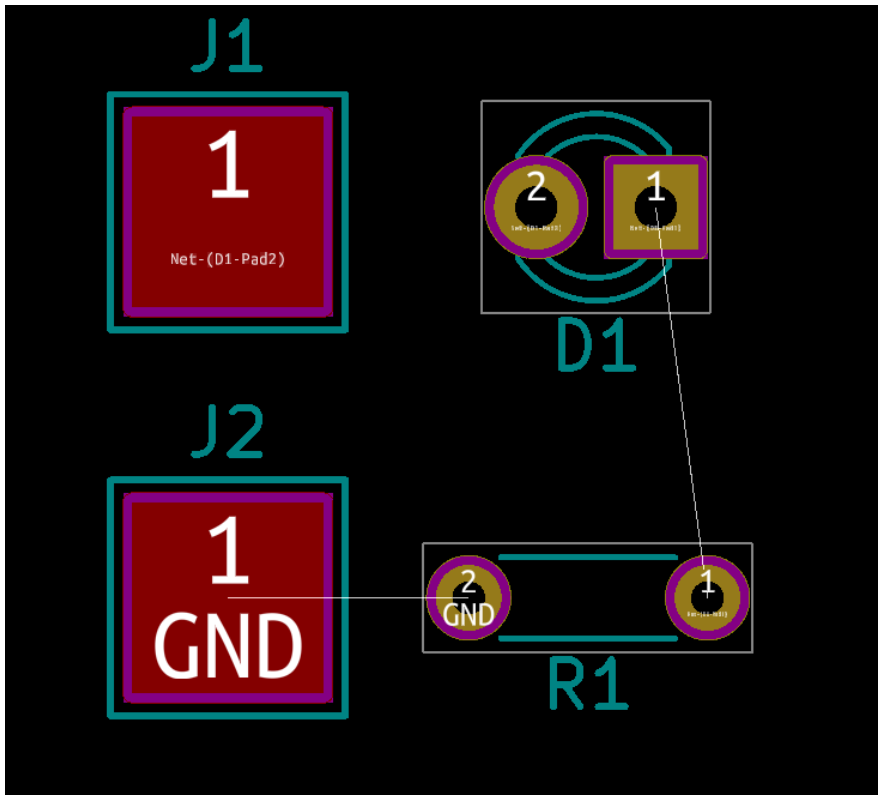
BLUESINK

1. PCB

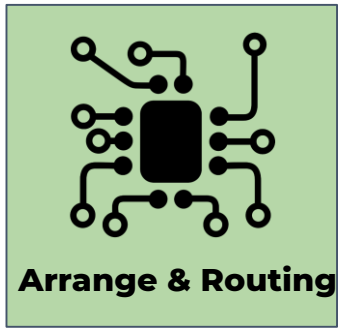


실습

1. 부품 배치

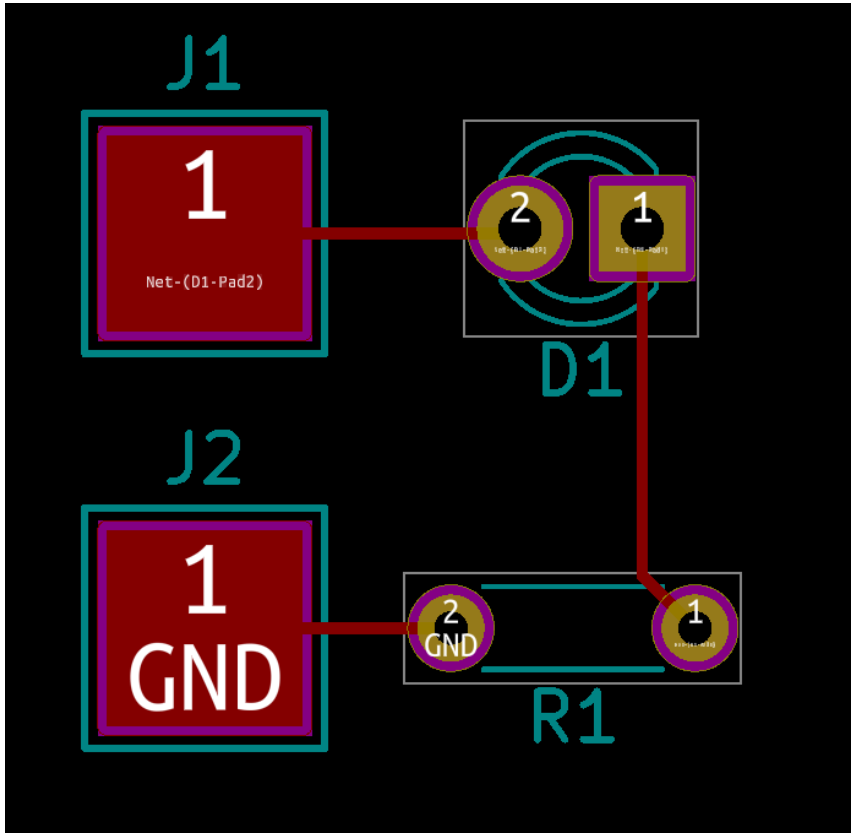


1. PCB

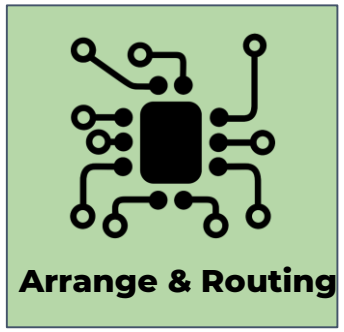


실습

2. 라우팅

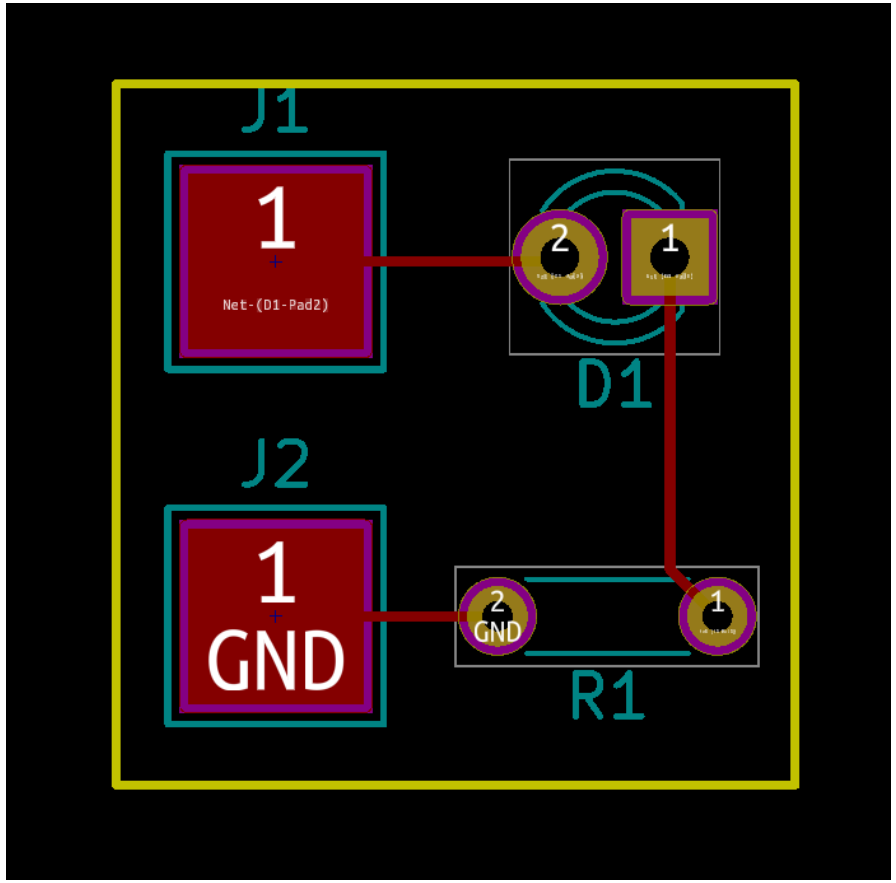


1. PCB

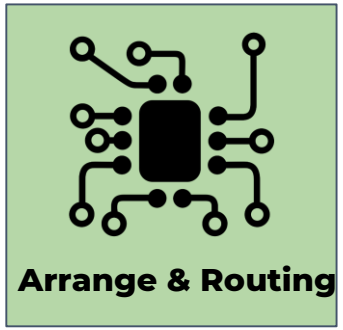


실습

3. Edge Cuts 그리기

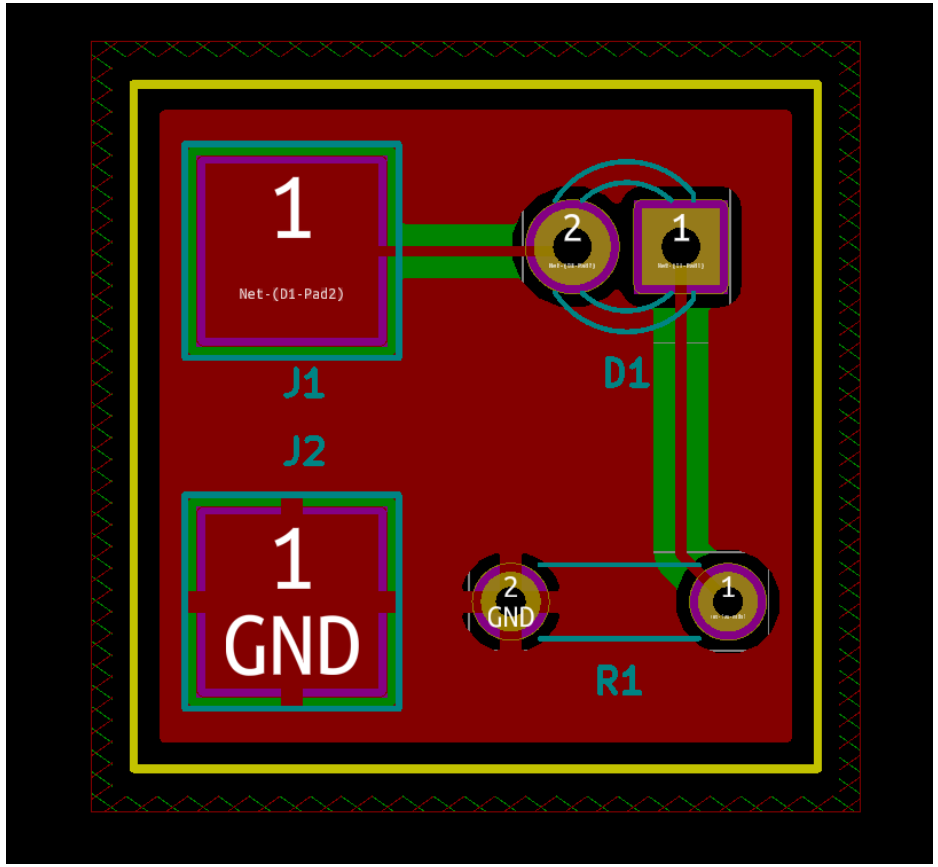


1. PCB

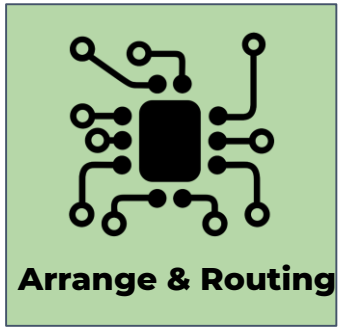


실습

4. Zone 깔기

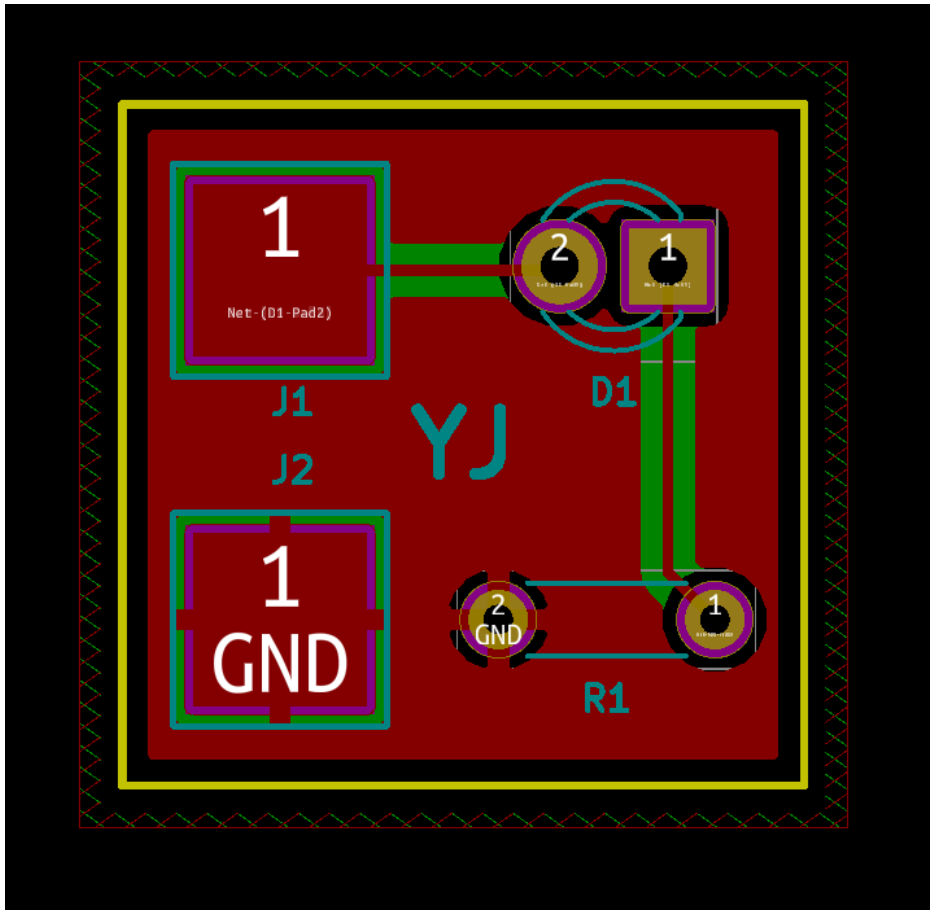


1. PCB

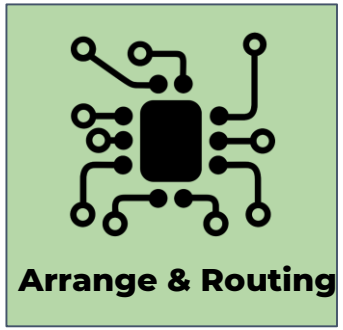


실습

5. Silks 레이어에 이름 추가

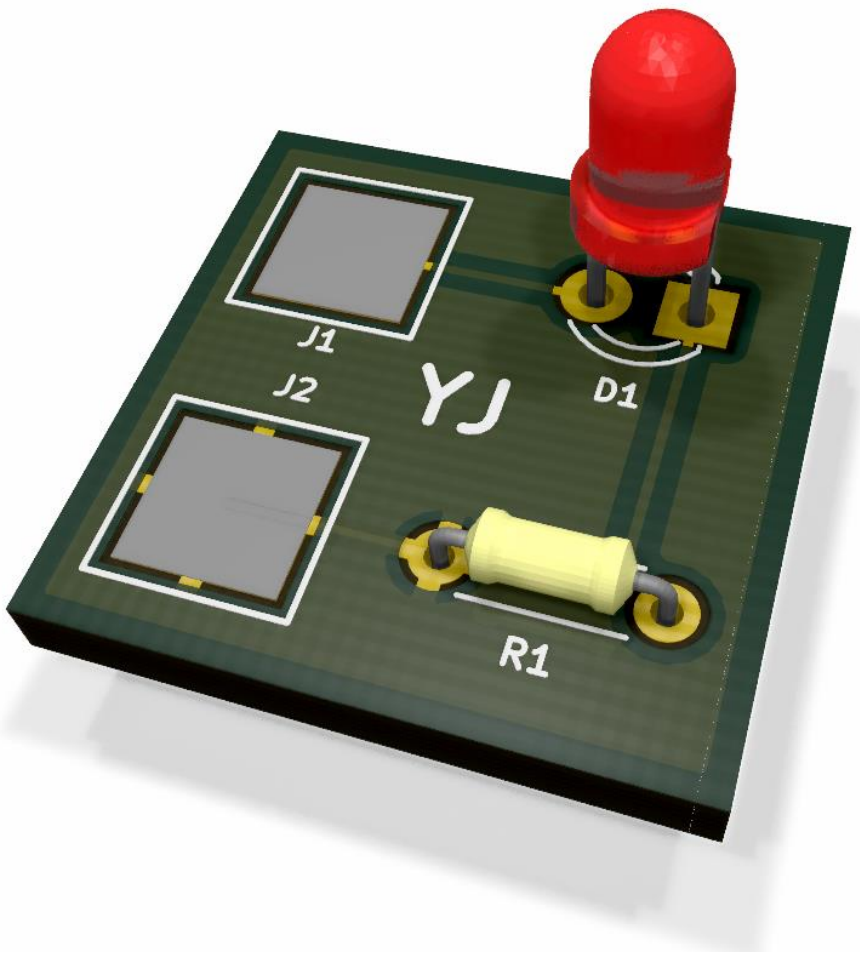


1. PCB

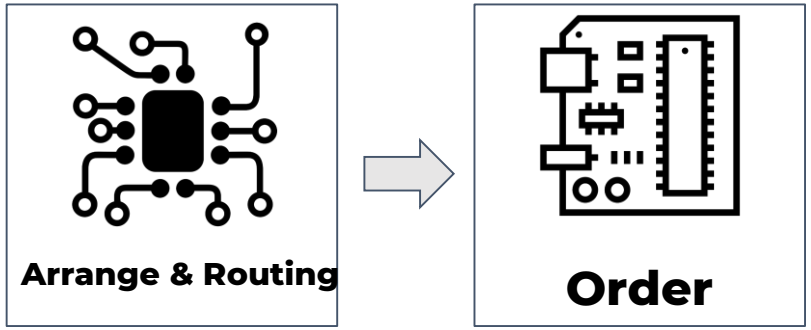


실습

6. 완료



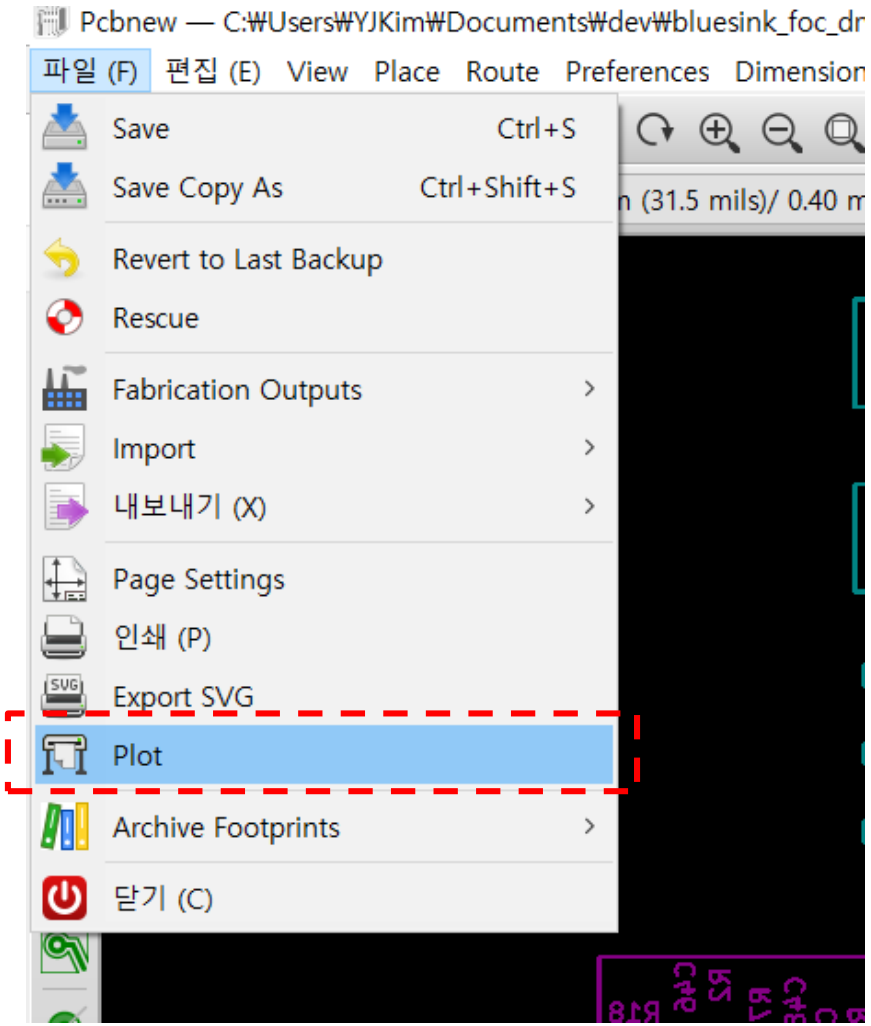
1. PCB



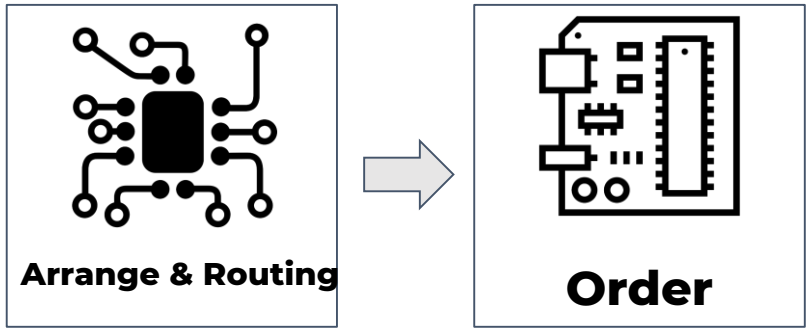
Gerber (거버) 생성하기

*Gerber : 실제 PCB 제조 공장에서 작업할 때 필요한 파일 포맷

1. 파일 -> Plot 클릭



1. PCB

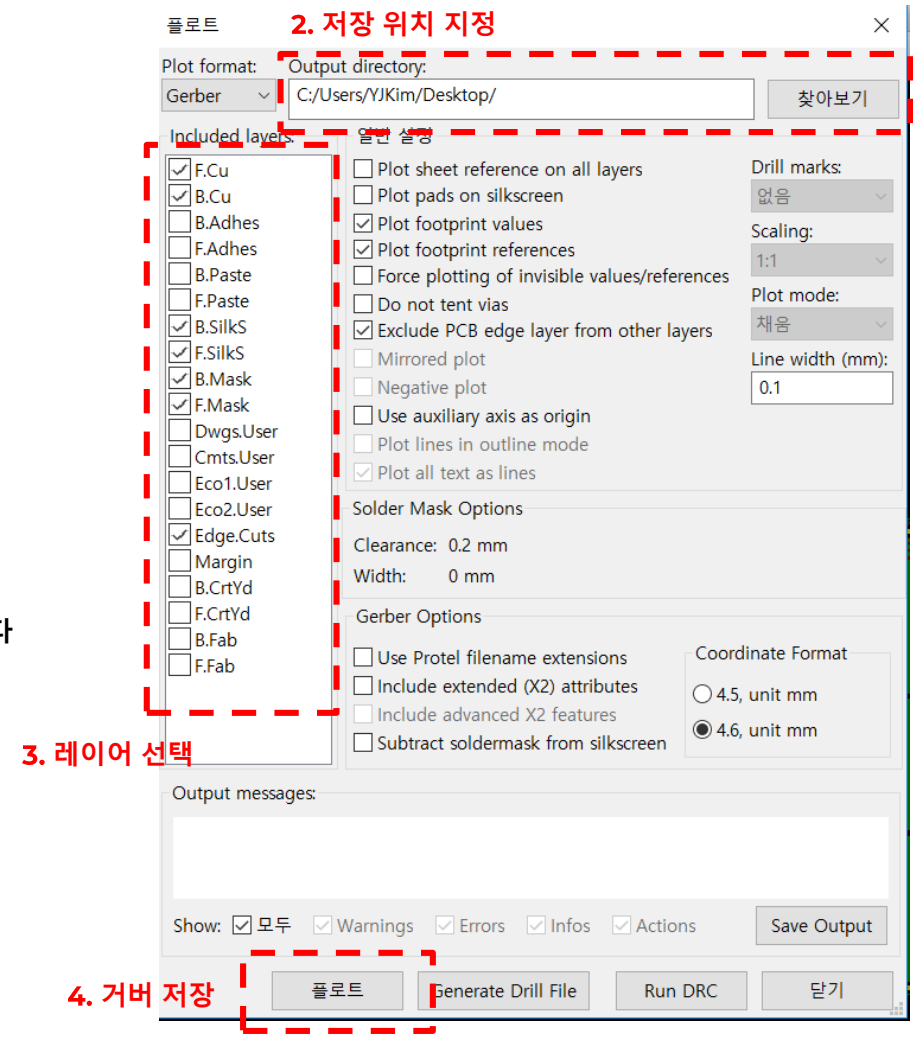


Gerber (거버) 생성하기

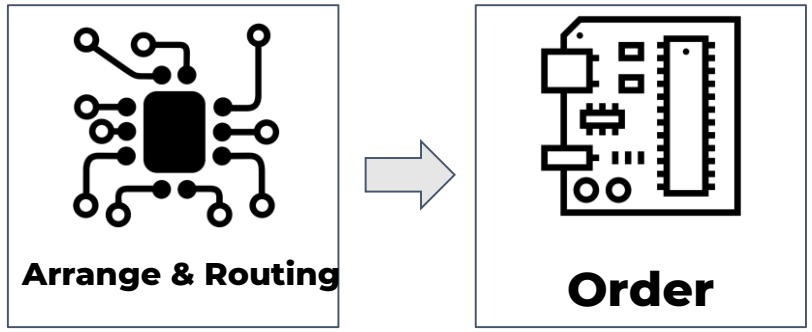
*Gerber : 실제 PCB 제조 공장에서 작업할 때 필요한 파일 포맷

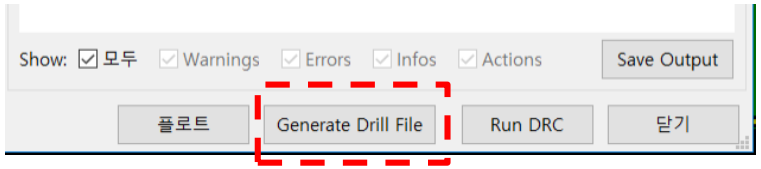
*레이어를 선택할 때, 일반적으로 다음과 같은 레이어를 선택하면 된다
.

- F.Cu
- B.Cu
- B.Silks
- F.Silks
- B.Mask
- F.Mask
- Edge.Cuts



1. PCB

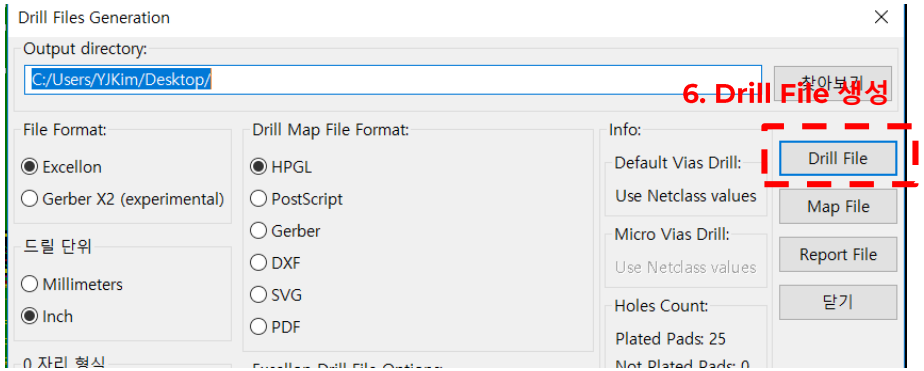




5. Generate Drill File 클릭

Gerber (거버) 생성하기

*Gerber : 실제 PCB 제조 공장에서 작업할 때 필요한 파일 포맷

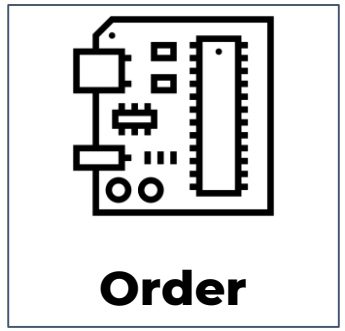


6. Drill File 생성

*최종적으로 다음과 같은 파일이 생성되면, 이제 업체에 넘겨 실제 PCB제작에 들어갈 수 있다.

- F.Cu.gbr
- B.Cu.gbr
- B.Silks.gbr
- F.Silks.gbr
- B.Mask.gbr
- F.Mask.gbr
- Edge.Cuts.gbr
- NPTH.drl
- PTH.drl

1. PCB



[한샘 디지텍] - https://www.hsdgt.com

주문서작성 온라인에서 간편하게 주문 하실 수 있습니다.

기본사양정보

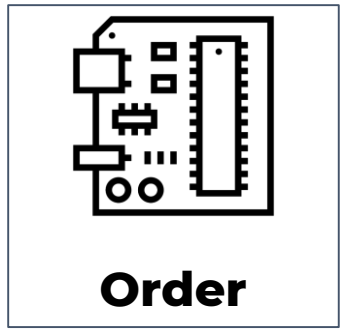
레이어	2	
납기	선택	※수량 단위를 고려하여 납기를 선택하십시오. (샘플 / 양산)
수량	PCS 매(원판)	
사이즈	mm × mm	※최소 사이즈 5mm ×5mm 최대 450mm ×550mm까지 작업이 가능합니다.
재질	FR-4	※기본 재질은 FR-4입니다.
두께	1.6T	※기본 두께는 1.6T입니다.
동박	외층 1oz	※1, 2 Layer은 1oz가 기본입니다.
솔더색	녹색유광	※기본 녹색 유광
실크색	흰색	※기본 흰색
실크	데이터기준 UL로고 주기 없음	※선택 없을 시 데이터 기준입니다.
표면처리	HASL 카본	
단자금도금	EA 면취	※기본핀 사이즈는 1mm ×10mm입니다.
파일	파일 선택 선택된 파일 없음	
주문확인전화	예 아니오	※사양 확인 전화 없이 진행을 원하실 경우 "아니오" 를 선택하시면 됩니다.

생산가격을 결정하는 요소

1. 납기
2. 크기
3. 레이어

50mm x 50mm 5장 2박 3일 ~ 약 8만원

1. PCB



Order

[IcbanQ] - http://www.icbanq.com

생산가격을 결정하는 요소

1. 납기
2. 크기
3. 레이어

50mm x 50mm 5장 2박 3일 ~ 약 8만원