Footprint 생성 방법 (v16.6)

NINEPLUS IT

나인플러스 IT (주)

cadence

Overview

- **Notice**

1. Through-Hole Pin Device 생성

2. Package Symbol 생성
   - Package Symbol
   - Package Symbol(Wizard)

Notice

- 생성한 Pad, Footprint를 이용하기 위해서는 다음의 경로중 하나에 저장이 되어 있어야 함.
- **C:\spb_data**172_PCB_180227

순위	경로	BRD 파일이 아래의 경로에 있는 경우 C:\spb_data\172_PCB_180227\allegro
1	.\	C:\spb_data\172_PCB_180227\allegro
2	.\symbols	C:\spb_data\172_PCB_180227\allegro\symbols
3	..\	C:\spb_data\172_PCB_180227
4	..\symbols	C:\spb_data\172_PCB_180227\symbols
5	C:\Cadence\SPB_17.2\share\pcb\pcb_lib\symbols	기본 제공 PCB footprint library
6	C:\Cadence\SPB_17.2\tools\capture\library	기본 제공 capture symbol library

- spb_data는 환경변수 Home에 설정 되어 있는 기본 경로
  (spb_data가 아닌 경우 환경 변수에서 home에 설정 되어 있는 경로 확인)
- 172_PCB_180227는 폴더 예시 (이름 변경 가능)

1. Through-Hole Pin Device 생성(1/3)

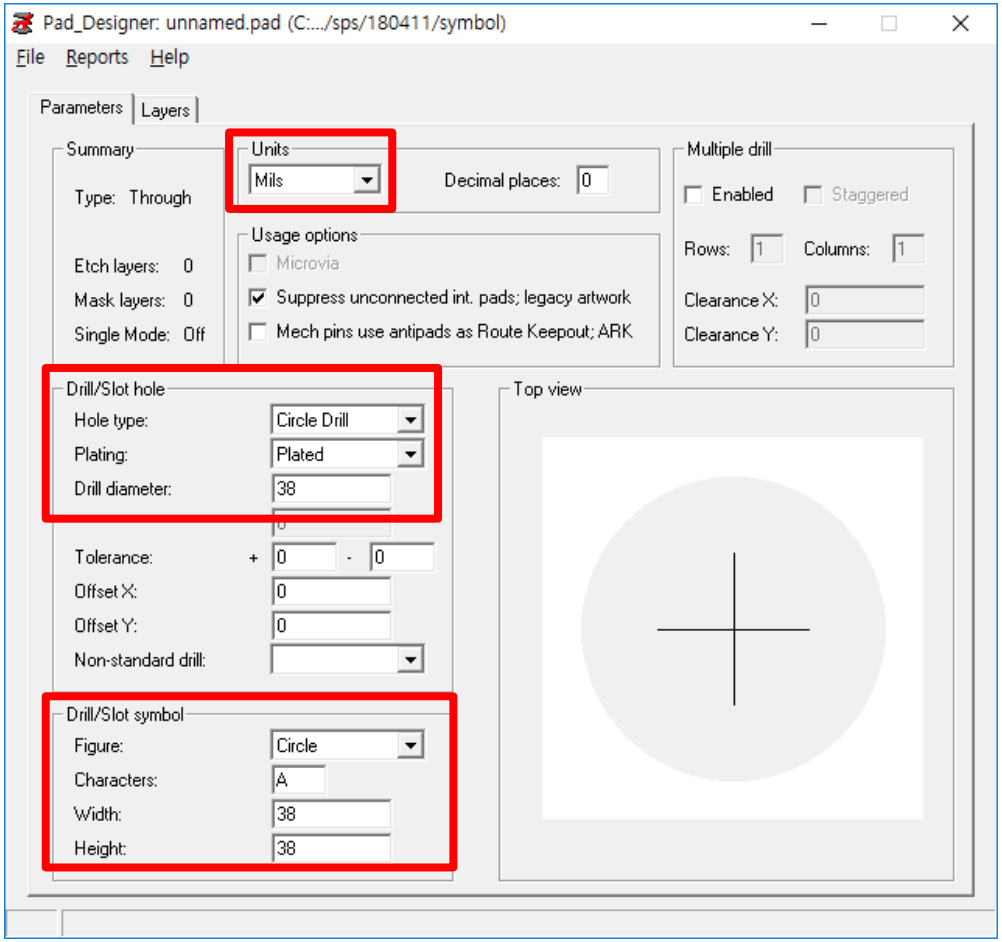
- 60c38d.pad

◆ **Pad Designer**

1. Pad Designer 실행
2. 각각의 tap에 수치 입력

- Drill/Slot hole :
  - » Hole type : Circle Drill
  - » Plating : Plated
  - » Drill diameter : 38

- Drill/Slot symbol :
  - » Figure : Circle
  - » Characters : A
  - » Width/Height : 38



1. Through-Hole Pin Device 생성(2/3)

- 60c38d.pad

3. Design Layers Tab 설정

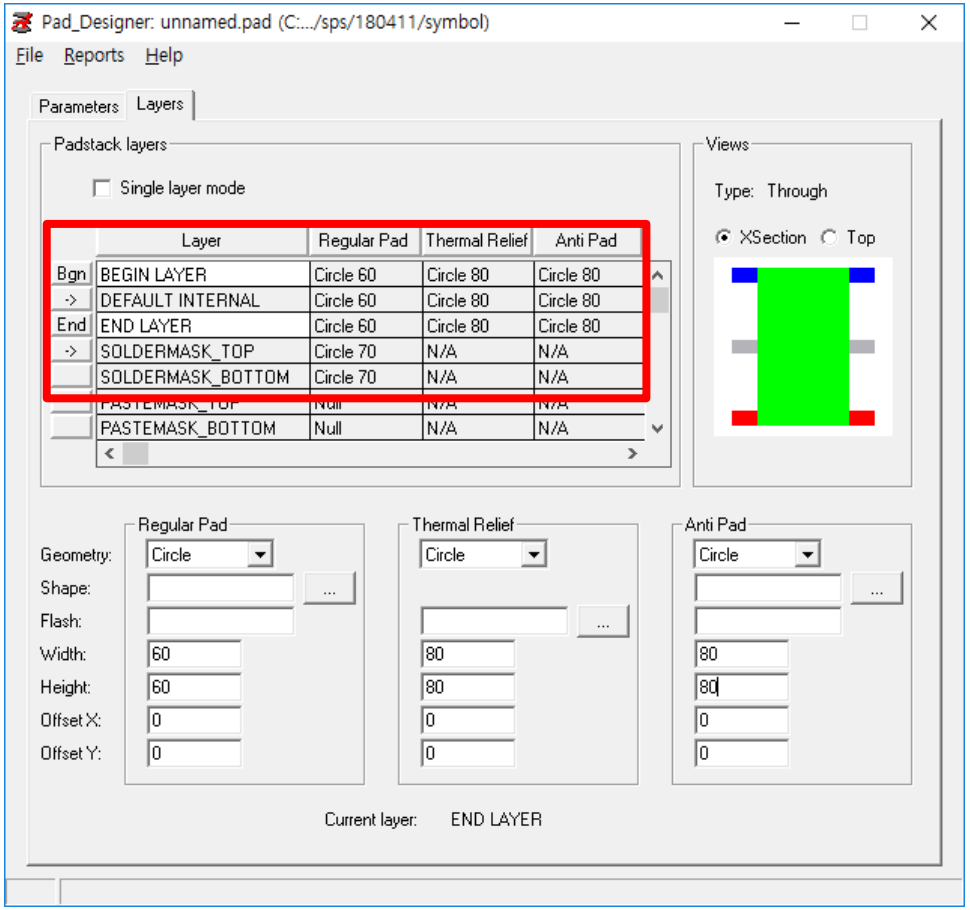
- Regular Pad :
  - » Geometry : Circle
  - » Width/Height : 60
- Thermal Relief, Anti Pad :
  - » Geometry : Circle
  - » Width/Height : 80

4. Mask Layers 설정

Soldermask_TOP, BOTTOM :

- » Geometry : Circle
- » Width/Height : 70

5. File – Save As.. – “60c38d.pad”



1. Through-Hole Pin Device 생성(3/3)

- 60s38d.pad

6. Design Layers Tab 변경

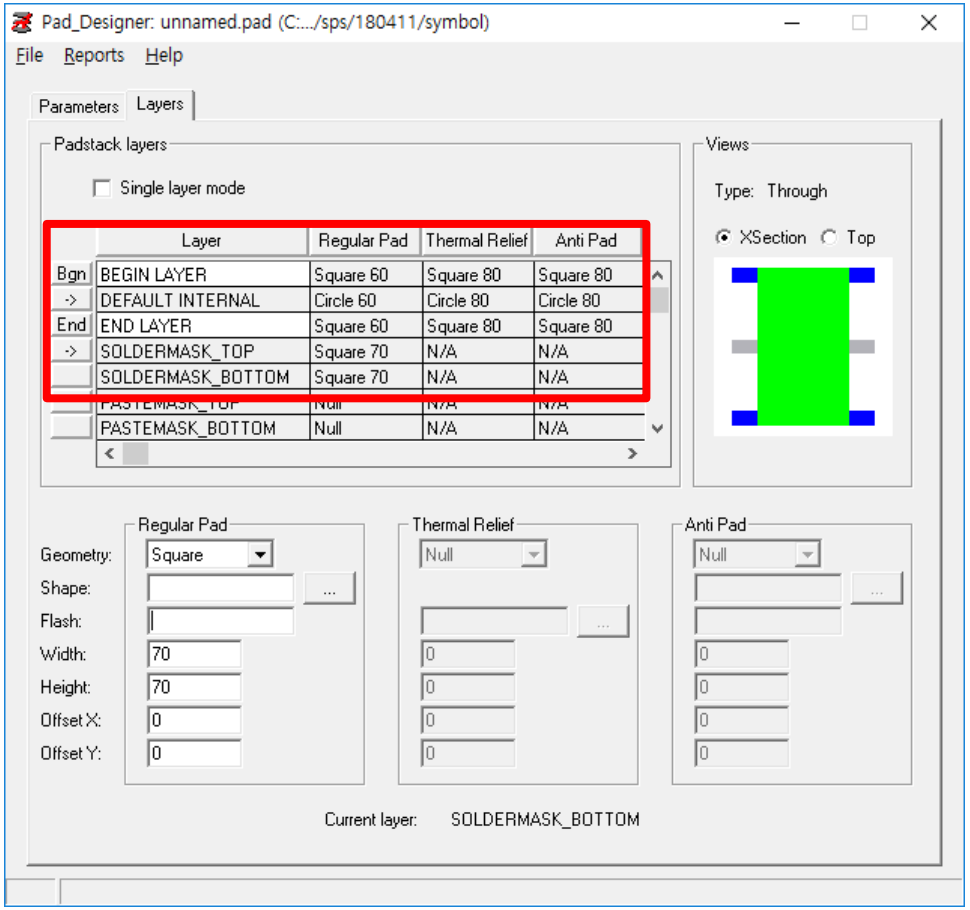
Begin, End layer

- Regular Pad :
  - » Geometry : Circle ➔ Square
- Thermal Relief, Anti Pad :
  - » Geometry : Circle ➔ Square

Soldermask_TOP, BOTTOM :

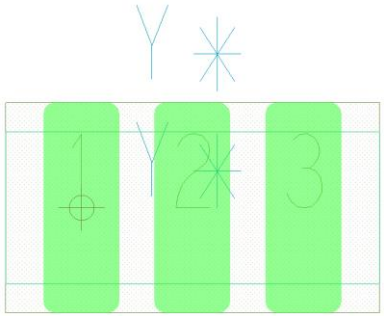
- Regular Pad :
  - » Geometry : Circle ➔ Square

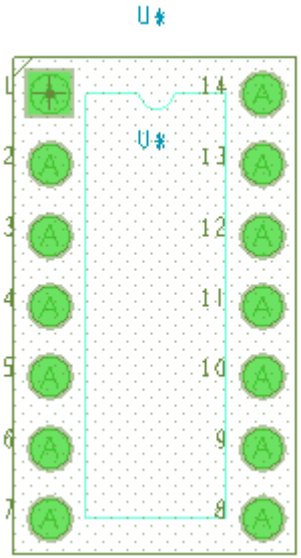
7. File - Save as… - “60s38d.pad”



2. Component Symbols (Package Symbols)

- Footprint?
  - 부품을 PCB에 실장 하기 위한 부품의 물리적인 모양의 정의
  - 다음의 3가지 정보를 가진다
    1. Pad 정보
    2. Assembly outline, silkscreen outline, Package boundary 등의 외각 정보
    3. 부품명 등의 참조번호 등을 나타내는 문자 정보



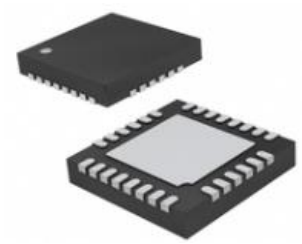


2. Package Symbol 생성

- OrCAD PCB Editor를 이용하여 생성

- Package Symbol

QFN



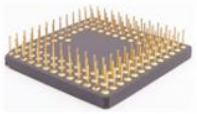
- Package Symbol(Wizard)

DIP

SOIC

PLCC/QFP

PGA/BGA

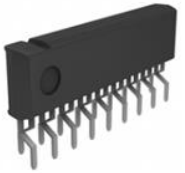


TH DISCRETE

SMD DISCRETE

SIP

ZIP

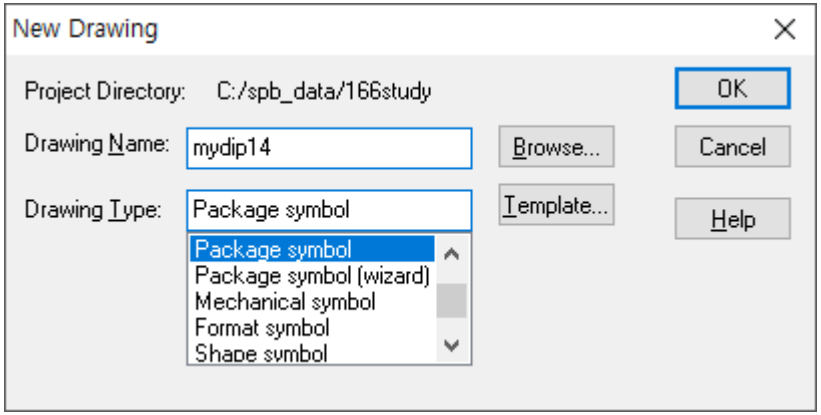


2. Package Symbol(1/10)

- mydip14 – pin 배치

1. OrCAD PCB Editor 실행
2. 메뉴 File – New

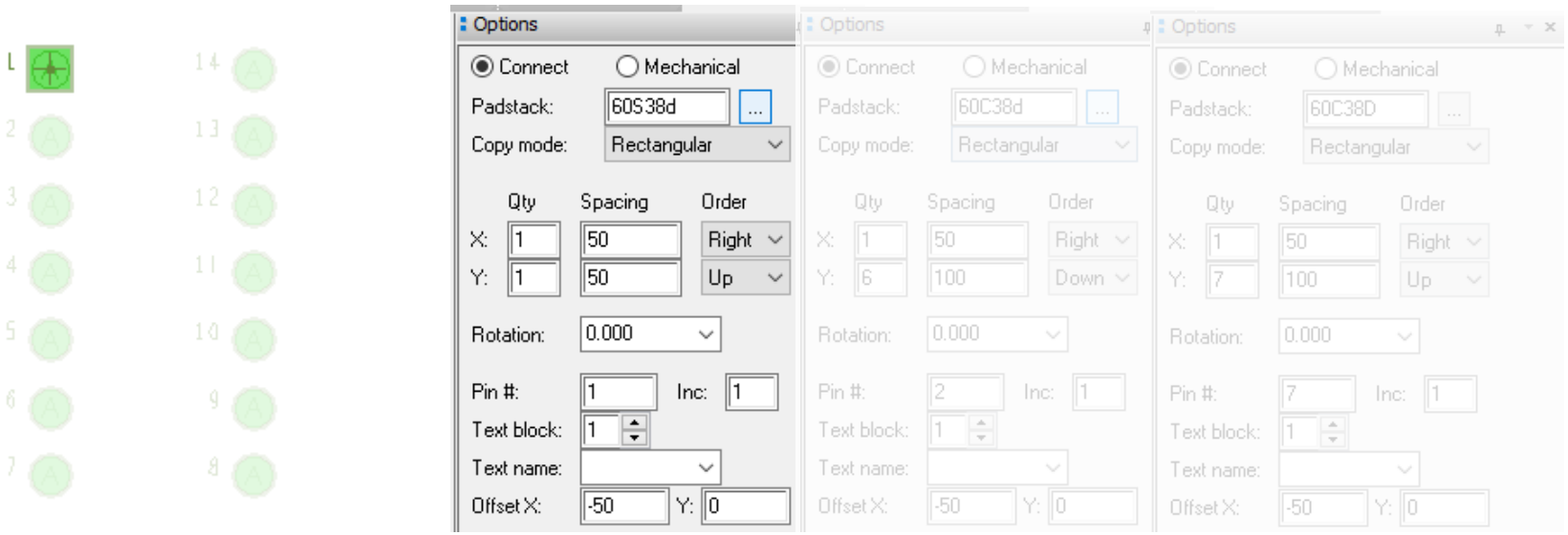
- Drawing Type : Package Symbol
- Drawing Name : mydip14



2. Package Symbol(2/10)

- mydip14 – pin 배치

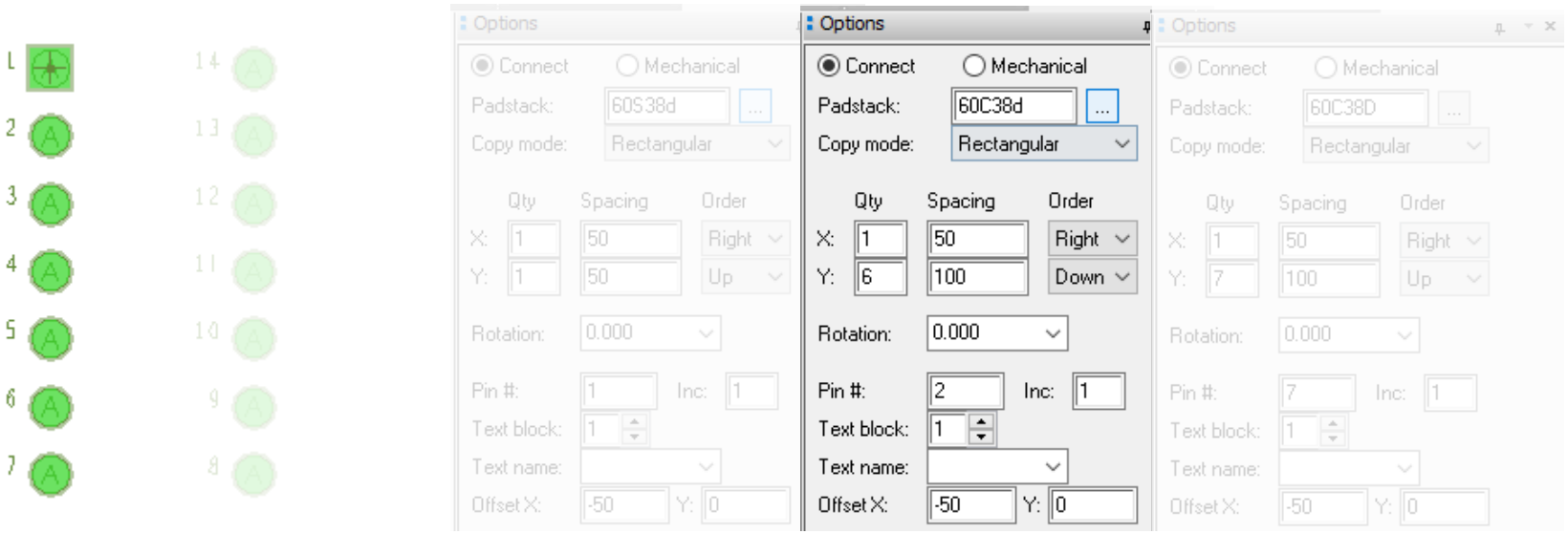
3. Layout – pins
4. Option panel에서 Padstack란 의 '…' 클릭
5. 1번 pin으로 사용할 '60s38d' 선택 후 OK
6. Command window에 x 0 0 입력 후 Enter



2. Package Symbol(3/10)

- mydip14 – pin 배치

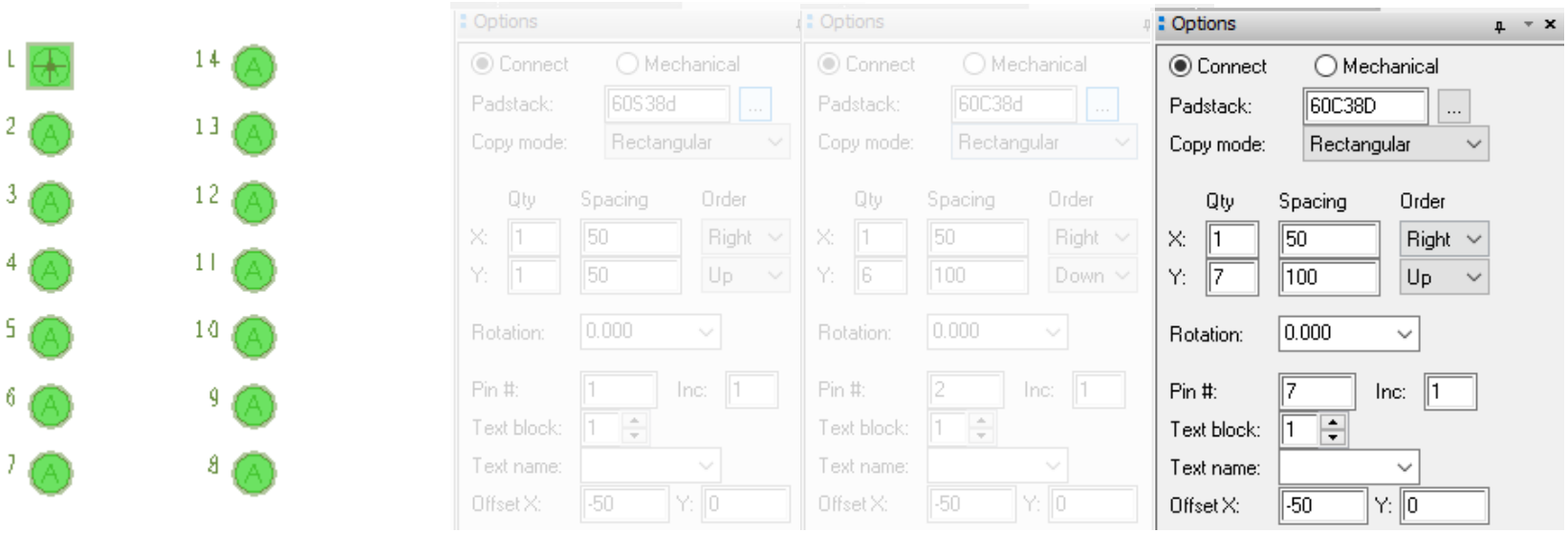
7. Layout – pins
8. Option panel에서 Padstack란 의 '…' 클릭
9. '60s38d' 선택 후 OK
10. Option panel에서 Y의 Qty를 6, Spacing은 100, Order를 Down, Pin #을 2로 설정
11. Command window에 x 0 -100 입력 후 Enter



2. Package Symbol(4/10)

- mydip14 – pin 배치

12. Layout – pins
13. Option panel에서 Padstack란 의 '…' 클릭
14. '60c38d' 선택 후 OK
15. Option panel에서 Y의 Qty를 7, Spacing은 100, Order를 Up, Pin #을 7로 설정
16. Command window에 x 300 -600 입력 후 Enter

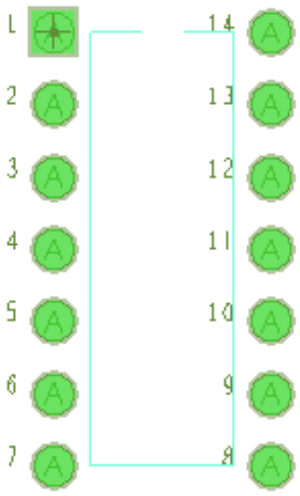


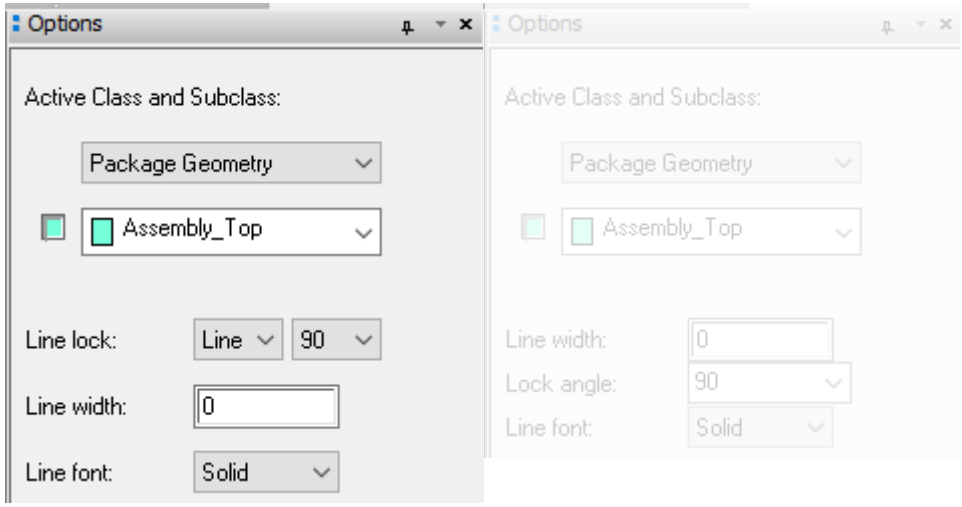
2. Package Symbol(5/10)

- mydip14 – Assembly outline / Silkscreen outline 생성

1. 메뉴 Setup – Grids..
2. Non-Etch의 x,y의 Spacing에 25을 입력
3. 메뉴 Add – Line
4. Option panel에서 Active Class에 Package Geometry, Subclass에 Assembly_Top으로 설정
5. Command window에 아래의 좌표를 입력 후 Enter

(125, 0), (50, 0), (50, -600), (250, -600), (250, 0), (175, 0)





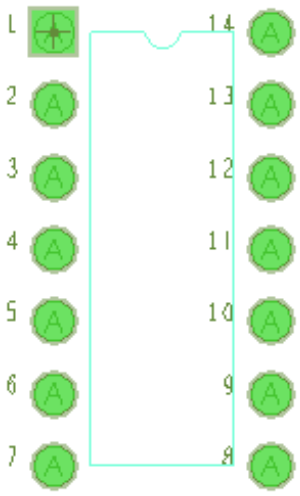
2. Package Symbol(6/10)

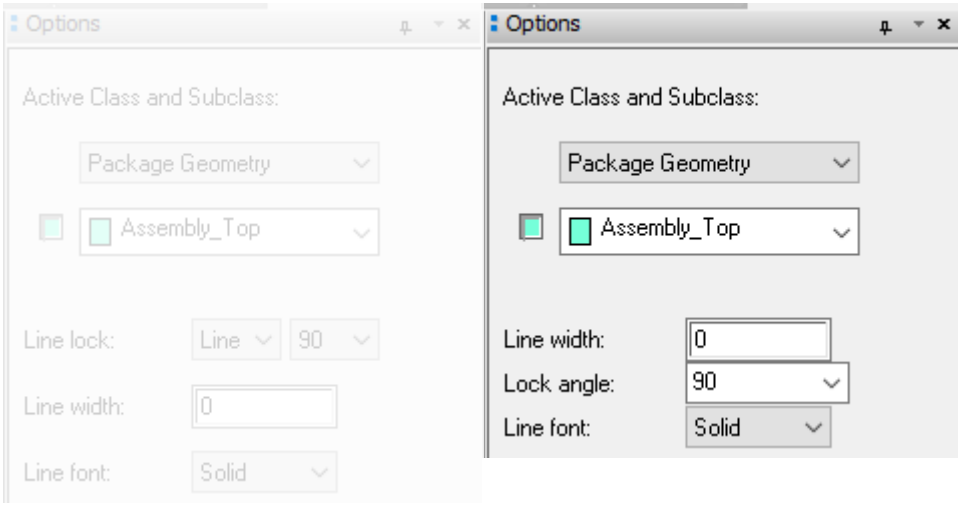
- mydip14 – Assembly outline / Silkscreen outline 생성

6. 메뉴 Add – Arc w/Radius

7. 아래의 좌표 순서대로 클릭

   (150, 0), (175, 0), (125, 0)





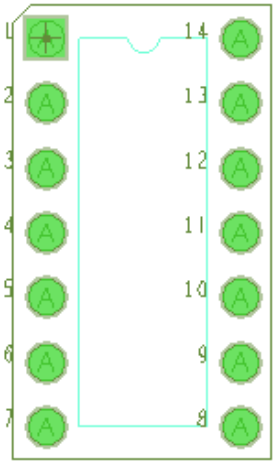
2. Package Symbol(7/10)

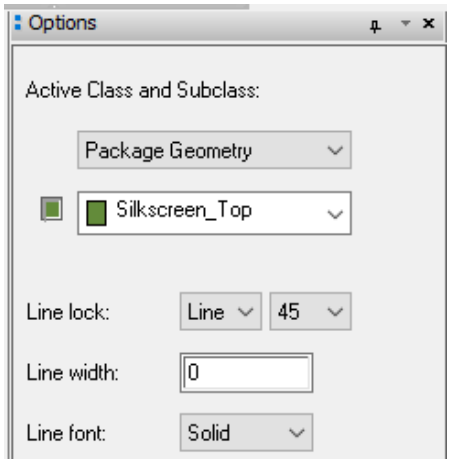
- mydip14 – Assembly outline / Silkscreen outline 생성

8. Option panel에서 Active Class에 Package Geometry, Subclass에 Silkscreen_Top으로 설정

9. Command window에 아래의 좌표를 입력 후 Enter

(-25, 50), (-50, 25), (-50, -650), (350, -650), (350, 50), (-25, 50)

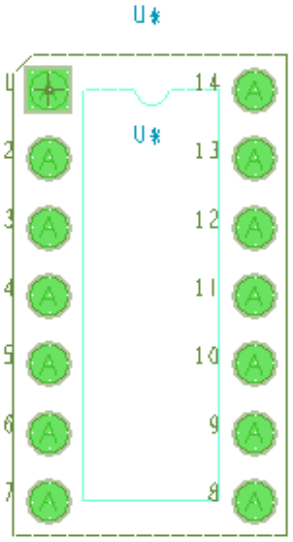


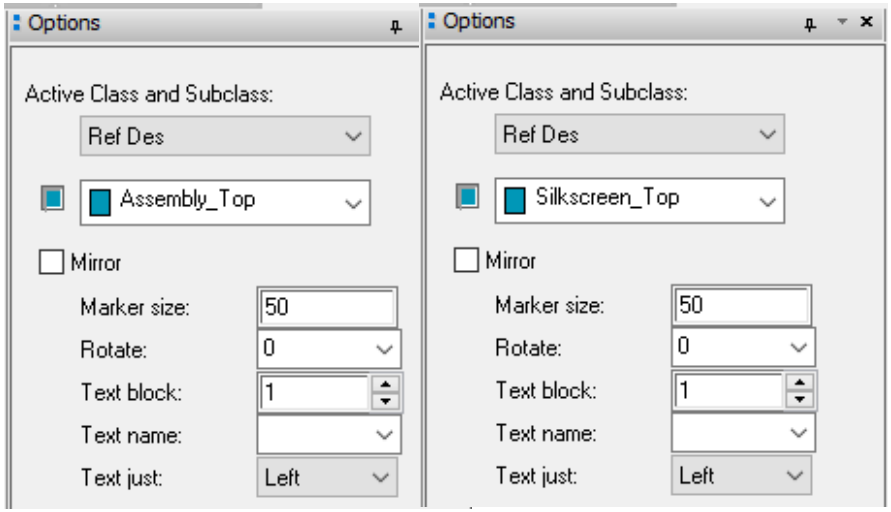


2. Package Symbol(8/10)

- mydip14 – RefDes(Reference Designator) 생성

1. 메뉴 Layout – Labels - RefDes
2. Option panel에서 Active Class에 Ref Des, Subclass에 Assembly_Top으로 설정
3. Assembly outline 내부를 클릭, ‘U*’를 입력
4. Option panel에서 Active Class에 Ref Des, Subclass에 Silkscreen_Top으로 설정
5. Silkscreen outline 내부를 클릭, ‘U*’를 입력



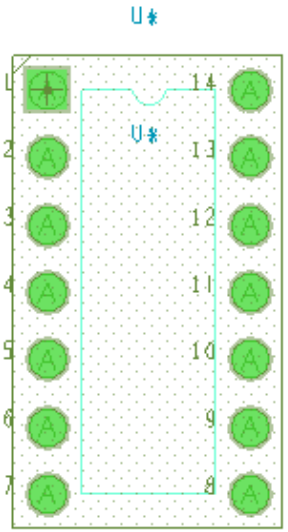


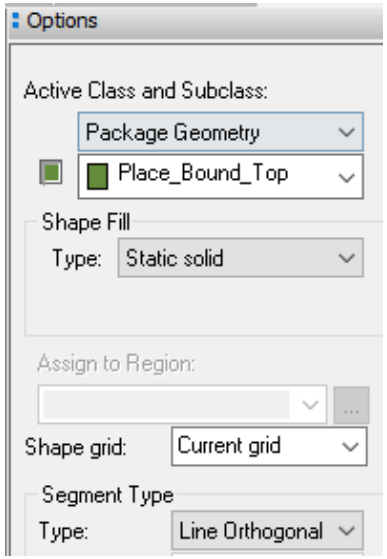
2. Package Symbol(9/10)

- mydip14 – Package Boundary 생성

1. 메뉴 Setup – Area – Package Boundary
2. Option panel에서 Active Class에 Package Geometry, Subclass에 Place_Bound_Top, Type에 Line Orthogonal을 선택
3. Command를 이용하여 아래의 좌표대로 Line 생성

   (-50, 50), (-50, -650), (350, -650), (350, 50)
4. RMB - Done



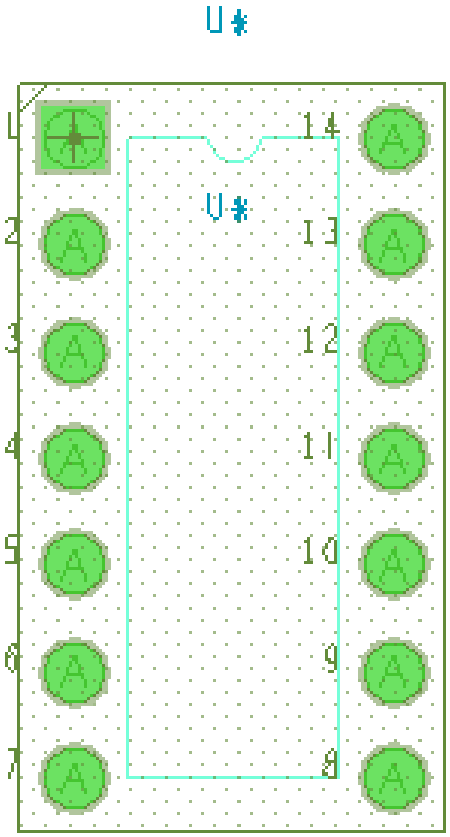


2. Package Symbol(10/10)

- mydip14 – Save

1. 메뉴 File – Save

2. mpdip14.dra, mpdip14.psm 생성

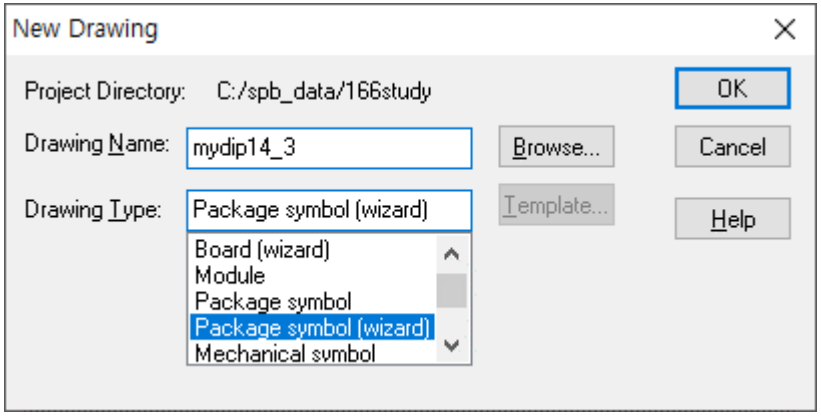


2. Package Symbol(Wizard)(1/5)

- mydip14_3

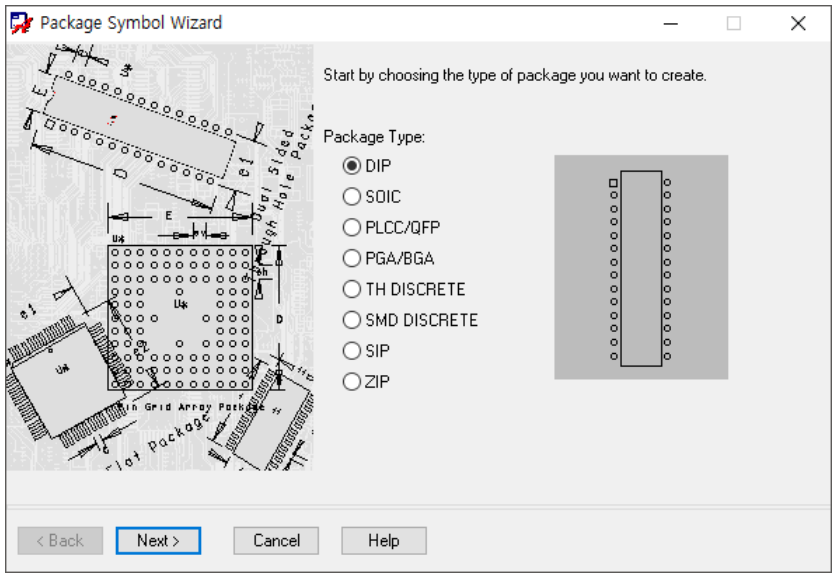
1. OrCAD PCB Editor 실행
2. 메뉴 File – New

- Drawing Type : Package Symbol(wizard)
- Drawing Name : mydip14_3



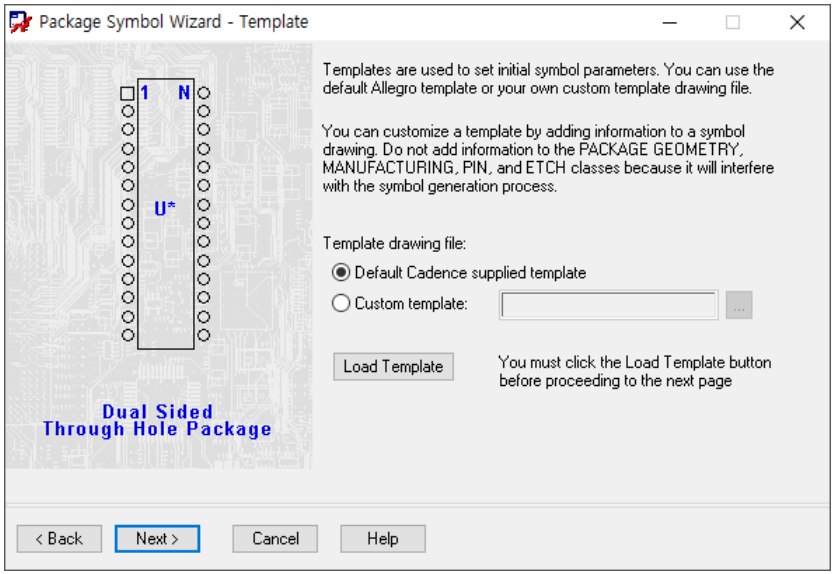
2. Package Symbols(Wizard)(2/5)

- mydip14_3



← Package Type 설정

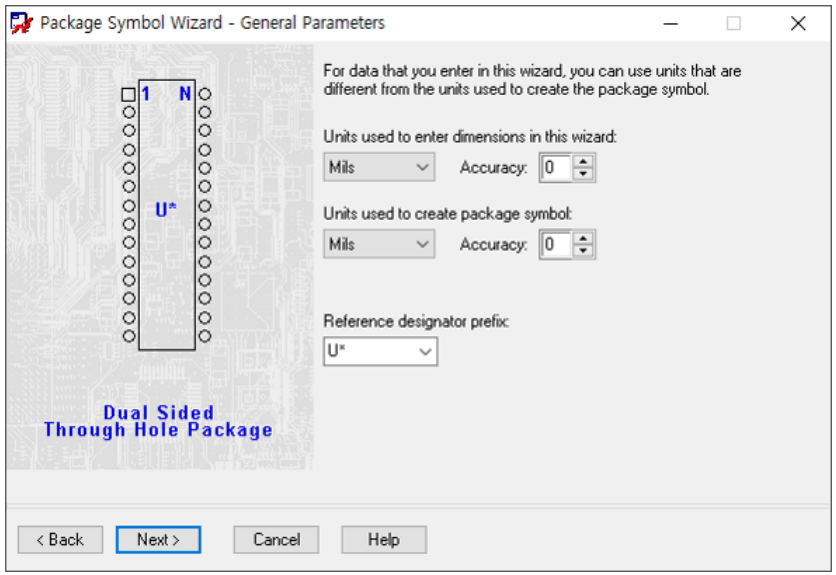
Template 설정➔



Template : Package symbol 에 대한 기본 정보(심볼 색상, 텍스트 크기 등)가 들어있는 .dra file

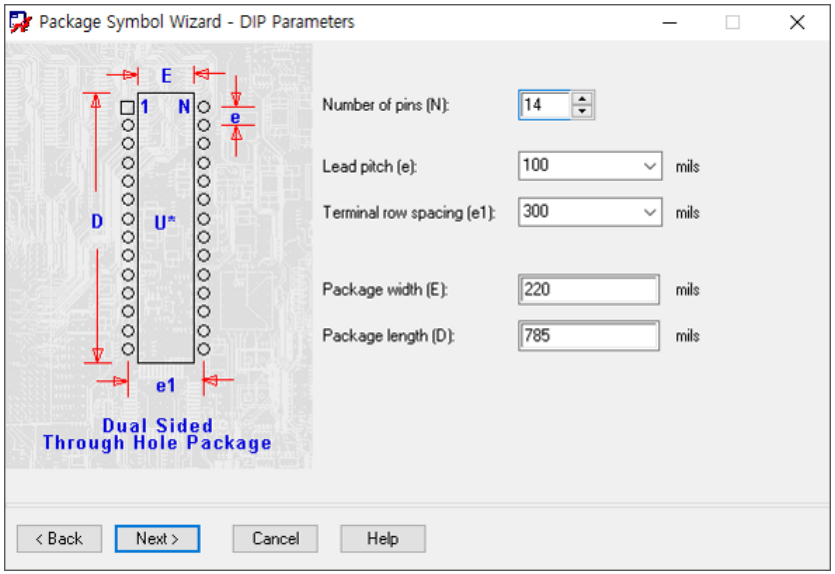
2. Package Symbols(Wizard)(3/5)

- mydip14_3



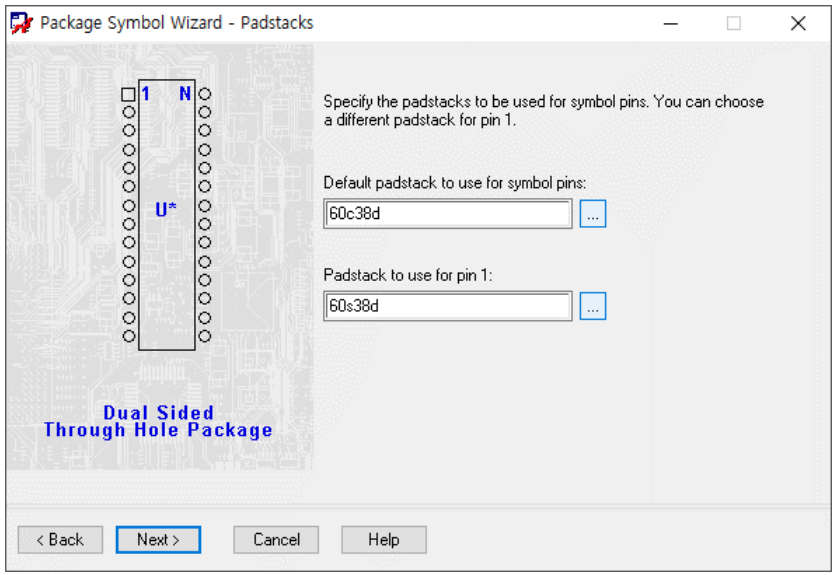
← 단위 및 RefDes 설정

Symbol의 dimension parameter 입력➔



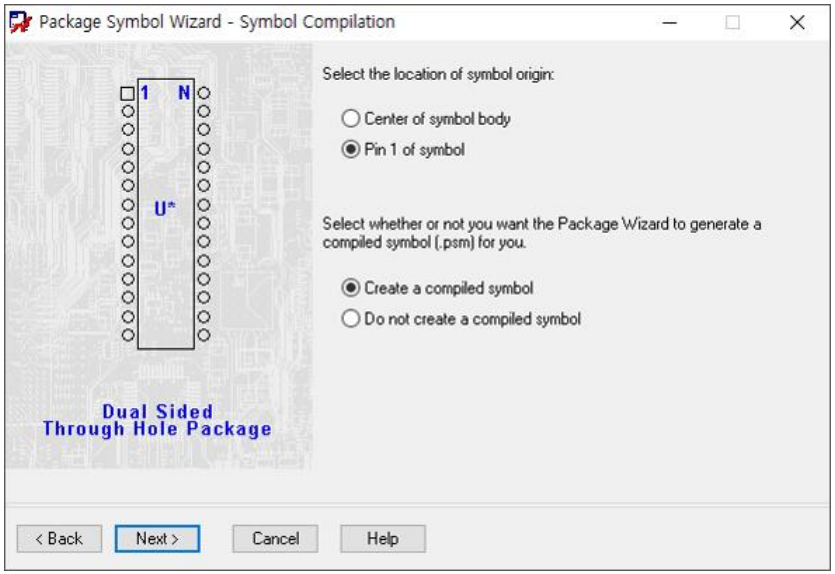
2. Package Symbols(Wizard)(4/5)

- mydip14_3



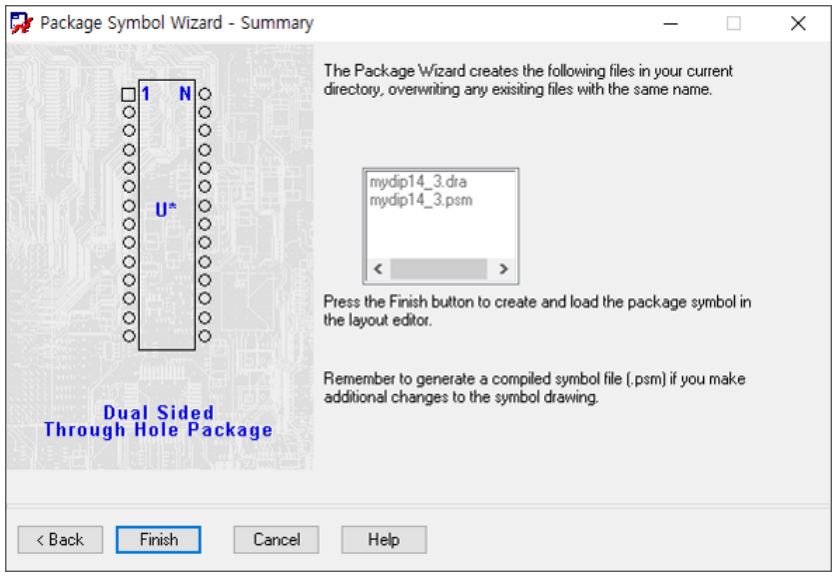
← Pad 설정 (Default, 1번)

Origin 설정 및 symbol compile →



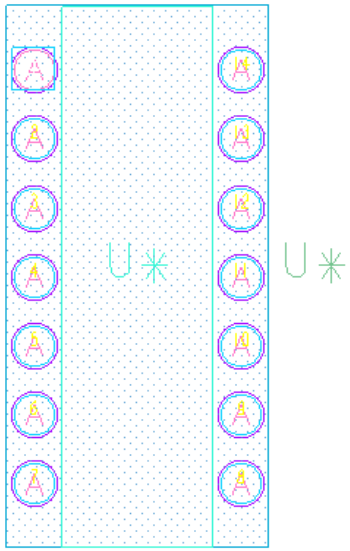
2. Package Symbols(Wizard)(5/5)

- mydip14_3



← Wizard summary (*.dra, *.psm 생성)

생성된 package symbol→



Thank you!