

Altium Designer 19

KeepOut객체 설계규칙 설정

2019.8.5

Altium Designer : KeepOut 객체 설계규칙 설정

객체 선택 » 마우스 오른쪽 버튼 » Find Similar Objects... 실행

유사 객체 찾기

| Section | Property | Value |
|-----------------|---------------------------|------------------------|
| Kind | Object Kind | Region Same |
| | Object Specific | |
| Object Specific | Restricted for Layer | BOTTOM Any |
| | Keepout | ✓ Same |
| | Keepout Restriction - Via | Any |
| | Keepout Restriction - Tr | Any |
| | Keepout Restriction - Co | Any |
| | Keepout Restriction - SM | Any |
| | Keepout Restriction - TH | Any |
| Graphical | Arc Approximation | 0.013mm Any |
| | Cavity Height | 0mm Any |
| | Locked | Any |
| Match | Select | Any |
| | 수식 생성 (X) | Mask ✓ Open Properties |

Object Kinde » Region : Same, Keepout » v(Check) Same 설정

수식 생성 v(Check) » 적용
** 유사 객체 찾기에서 설정한 조건에 부합한 수식을 만들어준다.

적용 (A) 확인 취소

Altium Designer : KeepOut 객체 설계규칙 설정

The image shows two overlapping dialog boxes in Altium Designer. On the left is the 'PCB Filter' dialog, and on the right is the '설계 규칙 종류 선택' (Design Rule Type Selection) dialog. An orange arrow points from the 'Create Rule' button in the PCB Filter dialog to the 'Electrical-Clearance Constraint' option in the Design Rule dialog.

PCB Filter Dialog:

- Object: Region
- Filter: (ObjectKind = 'Region')
- Buttons: Helper, Builder, History, Add To Favorites, **Create Rule**, Favorites

Design Rule Selection Dialog:

- Selected: **Electrical**
- Sub-selected: **Clearance Constraint**
- Callout: **Electrical-Clearance Constraint 선택 » 확인**
- Buttons: **확인**, 취소

Callout Box (White background, orange border):

PCB Filter에 생성된 수식을 복사하거나 Create Rule 버튼을 클릭하고, 수식을 적용시킬 규칙을 생성한다.

Altium Designer : KeepOut 객체 설계규칙 설정

PCB 규칙 및 조건 편집기 [mm]

이름: Clearance_KeepOut Area Unique ID: PWFCCSTL Test Queries

Where The First Object Matches
Custom Query: (ObjectKind = 'Region') And (Keepout = 'True')

Where The Second Object Matches
All

Constraints
Any Net
— 최소 이격 거리: 0mm

Ignore Pad to Pad clearances within a footprint:

Simple Advanced

규칙 우선권 편집

종류 (R): Clearance

| 순서 | 이름 | 조건 | 우선 |
|----|---------------------|--|----|
| 1 | Clearance_KeepOut A | (ObjectKind = 'Region' Clearance = 0mm | 적용 |
| 2 | Clearance_L2&L3 | (OnLayer('L2') OR OnLayer('L3')) AND Generic clearance = 0.04mm, and 9 value(s) for objects | |
| 3 | Clearance_TOP&BOT | (OnLayer('TOP') OR OnLayer('BOT')) AND Generic clearance = 0.035mm, and 9 value(s) for objects | |
| 4 | Clearance | All - All Generic clearance = 0.254mm, and 9 value(s) for objects | |

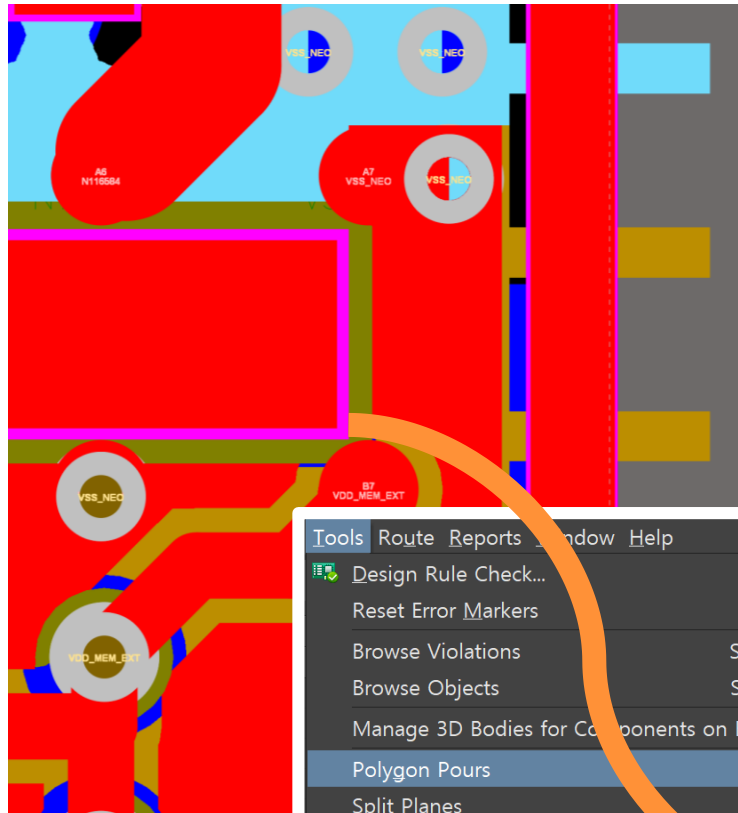
우선순위 (P)... 적용

우선순위가 높은 순으로 규칙 적용되므로, 규칙 생성 후 규칙 우선권을 설정한다.

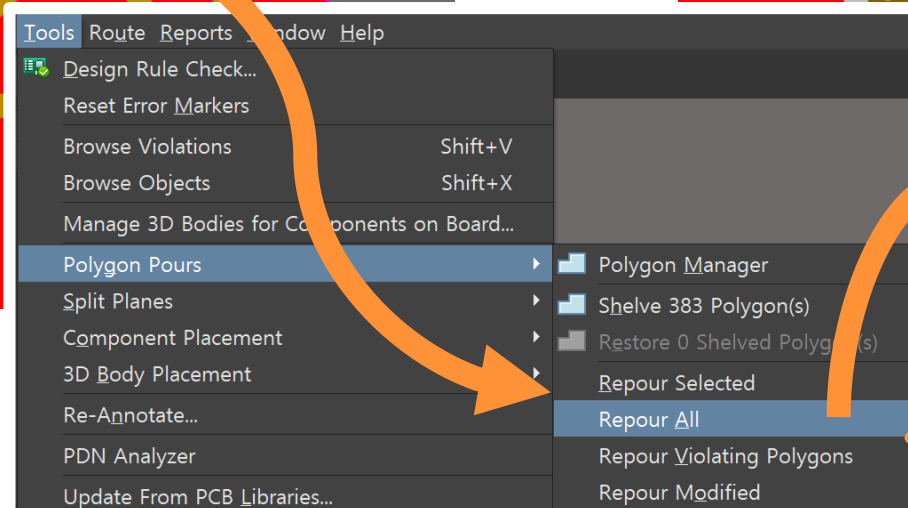
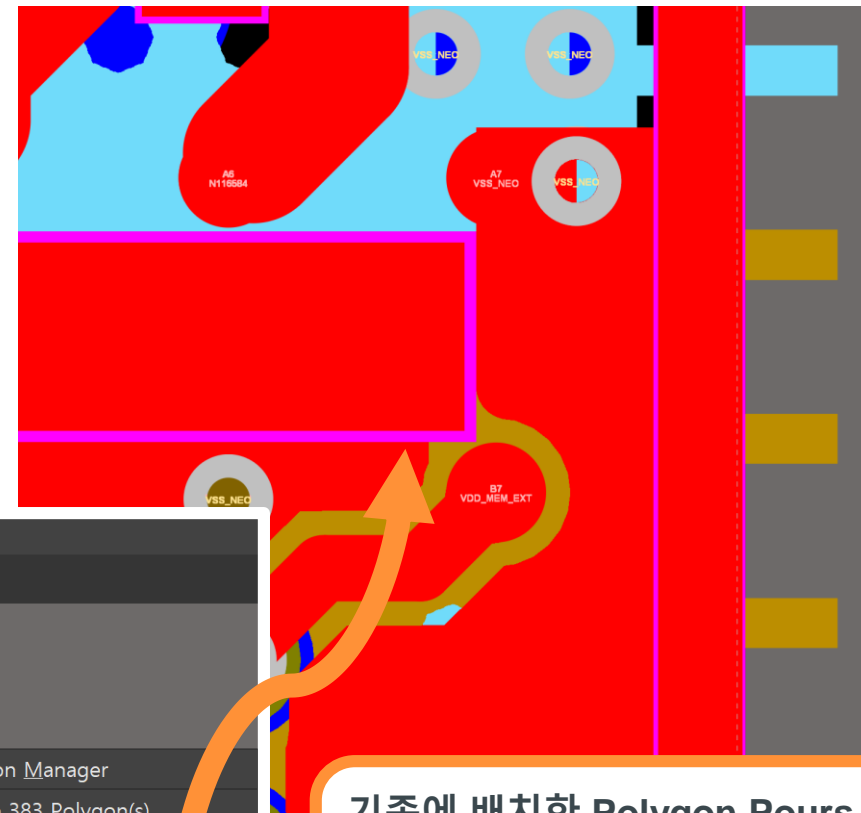
Constraint 설정
- Any Net
- 최소 이격 거리 : 0mm

Altium Designer : KeepOut 객체 설계규칙 설정

• Polygon Pours → Repour All 실행 후



• Polygon Pours → Repour All 실행 후



기존에 배치한 Polygon Pours에 변경된 설계규칙을 적용하기 위해 **T**ools » Polygon Pours » Repour **A**ll을 실행한다.



(주)한컴MDS altium.hancommds.com

031-600-5188 altium@hancommds.com

본사 13493 경기도 성남시 분당구 대왕판교로 644번길 49 한컴타워 3,4층 031-627-3000

연구소 13487 경기도 성남시 분당구 판교로 228번길 17 판교세븐벤처밸리 2단지 1동 9층 031-600-5000

HANCOM

Template Visual Guide, version 1.0

© Hancom Inc. / Pangyo, February 2019