

## 2021 학년도 기계공학과 CADD (MEE1002)

CADD(Computer Aided Design and Drafting)

한양대학교  
터보기계연구실  
Prof. Keun Ryu

2021년 9월 30일



**HANYANG UNIVERSITY**



## CONTENTS

I 3D 실습 (Groove, Hole)

II 예제 도면을 통한 실습

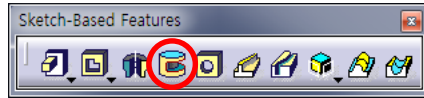
CADD (Computer Aided Design and Drafting)

# I 3D 실습 (Groove, Hole)

# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

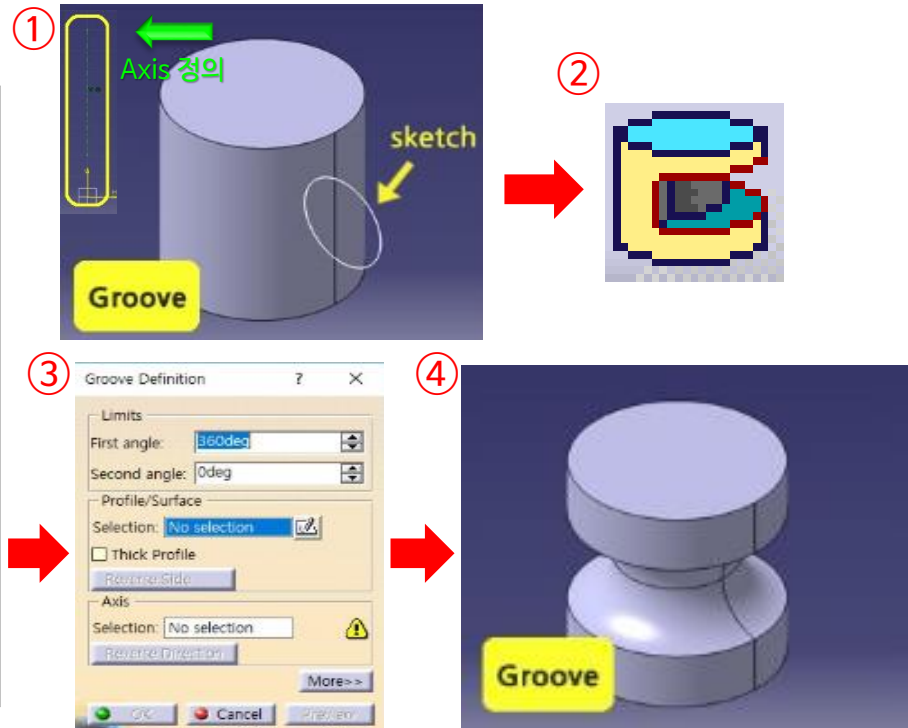
### Groove



#### 1-1 Groove

- 선택한 Sketch를 정의한 축 기준으로 회전시켜 형상을 제거

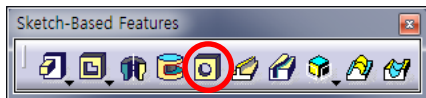
- ① 제거할 Sketch를 선택한다 [Sketch에 해당 축(Axis)를 정의하는게 일반적이다]
- ② Groove 아이콘을 선택한다
- ③ Groove Definition에 사용자가 원하는 항목을 정의한다
- ④ 형상 완료



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

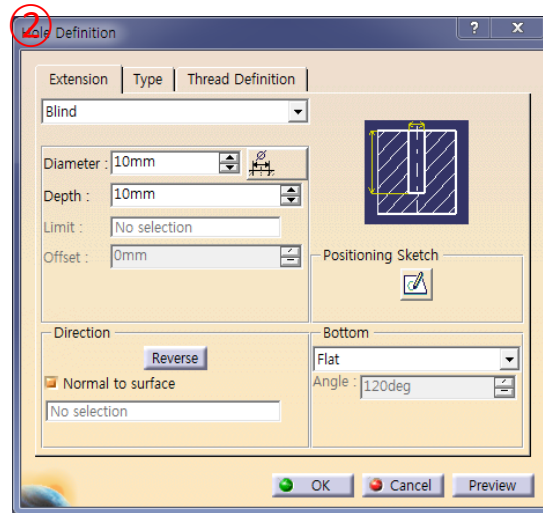
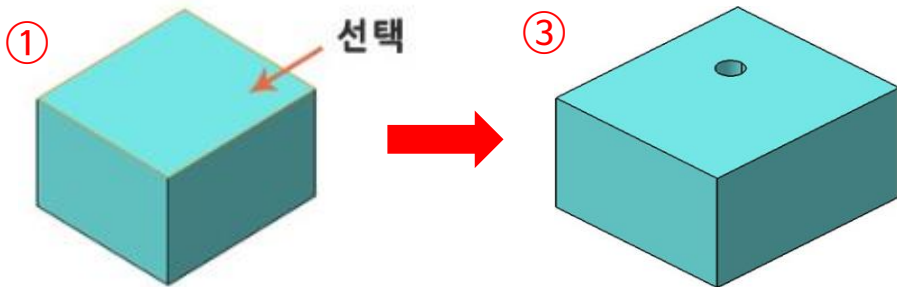
### Hole



### 2-1 Hole

- 3차원 Geometry에서 Profile 없이 Hole을 생성하는 기능

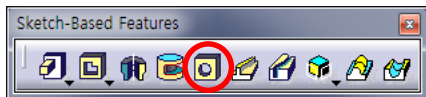
- Hole을 생성할 면 지정 후 아이콘 클릭
- Hole Definition Dialog Box를 통해 Hole의 크기 및 깊이 Type등을 정의한다
- 형상과 Tree에 Hole이 생성 및 추가된다



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

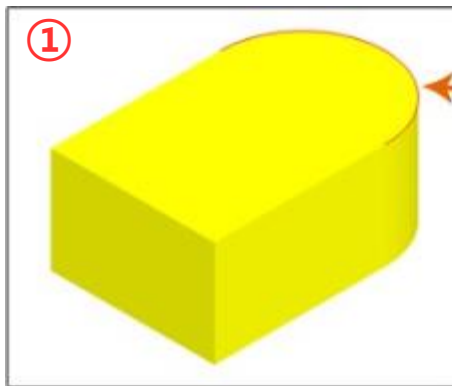
### Hole



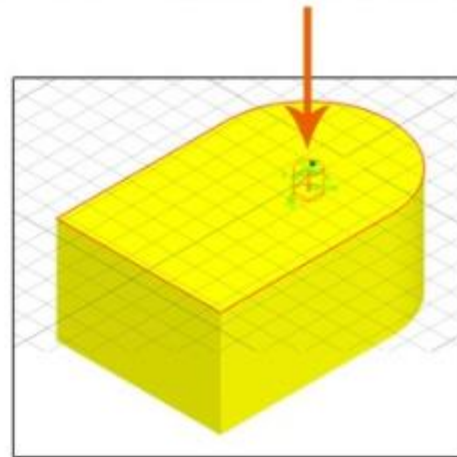
#### 2-1 Hole

##### • Hole 위치 지정 방법

- ① **Arc(Circle) 선택 시** : Arc(Circle) 선택 후 면을 클릭하면 Hole의 위치와 선택한 Arc가 동심을 이룬다
- ② **Edge 선택 시** : Edge 선택 후 면을 클릭하면 지정한 Edge에서 거리 값을 부여할 수 있다
- ③ **Sketch로 부여** : Point의 위치를 Sketch에서 지정해 준다



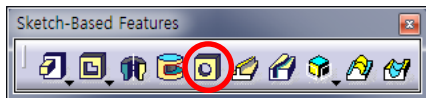
Arc(Circle) 선택 후 면을 클릭하면 Hole의 위치와 선택한 Arc가 동심을 이룬다.



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

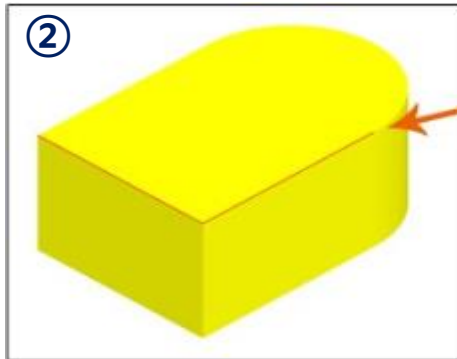
### Hole



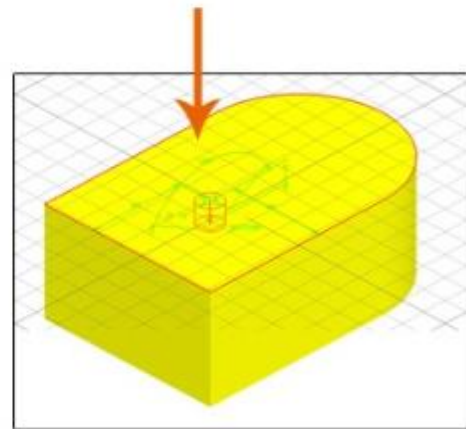
#### 2-1 Hole

##### • Hole 위치 지정 방법

- ① **Arc(Circle) 선택 시** : Arc(Circle) 선택 후 면을 클릭하면 Hole의 위치와 선택한 Arc가 동심을 이룬다
- ② **Edge 선택 시** : Edge 선택 후 면을 클릭하면 지정한 Edge에서 거리 값을 부여할 수 있다
- ③ **Sketch로 부여** : Point의 위치를 Sketch에서 지정해 준다



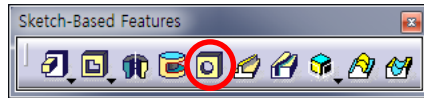
Edge 선택 후 면을 클릭하면 지정한 Edge에서 거리 값을 부여할 수 있다.



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

### Hole



### 2-1 Hole

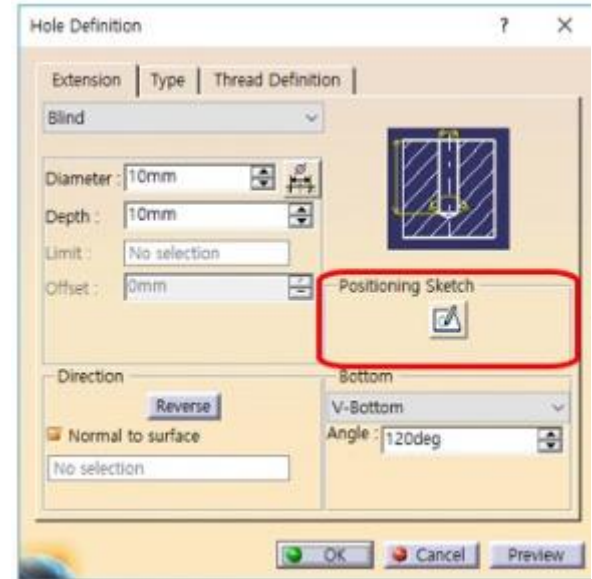
- Hole 위치 지정 방법

① **Arc(Circle) 선택 시** : Arc(Circle) 선택 후 면을 클릭하면 Hole의 위치와 선택한 Arc가 동심을 이룬다

② **Edge 선택 시** : Edge 선택 후 면을 클릭하면 지정한 Edge에서 거리 값을 부여할 수 있다

③ **Sketch로 부여** : Point의 위치를 Sketch에서 지정해 준다

③



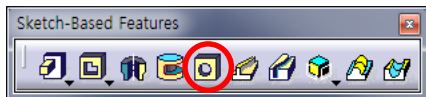


# 1

## CADD (Computer Aided Design and Drafting)

### 3D 실습 (Groove, Hole)

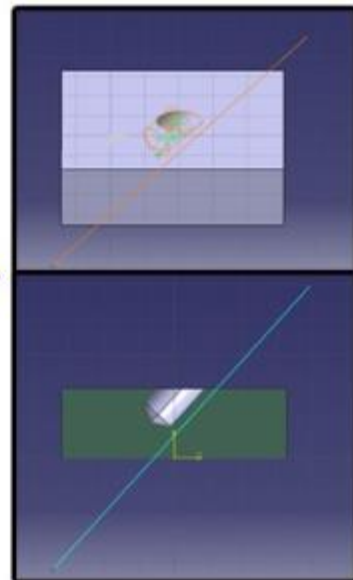
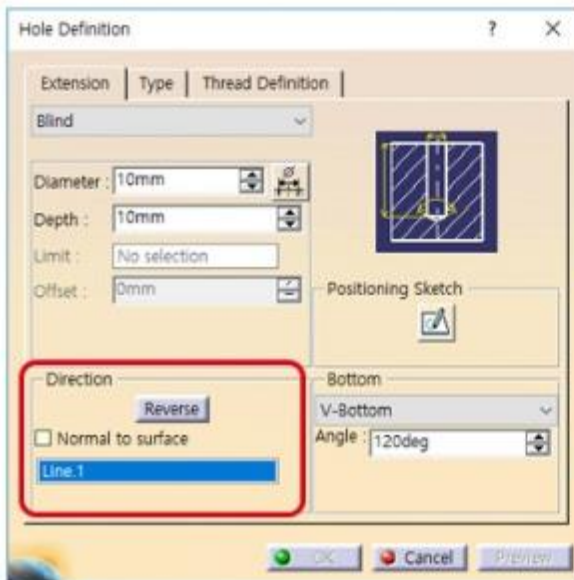
#### Hole



#### 2-1 Hole

- Normal to surface

: 사용자가 지정한 Reference 방향으로 Hole 생성

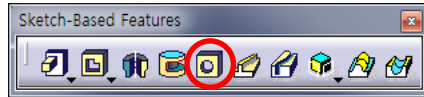


# 1

## CADD (Computer Aided Design and Drafting)

### 3D 실습 (Groove, Hole)

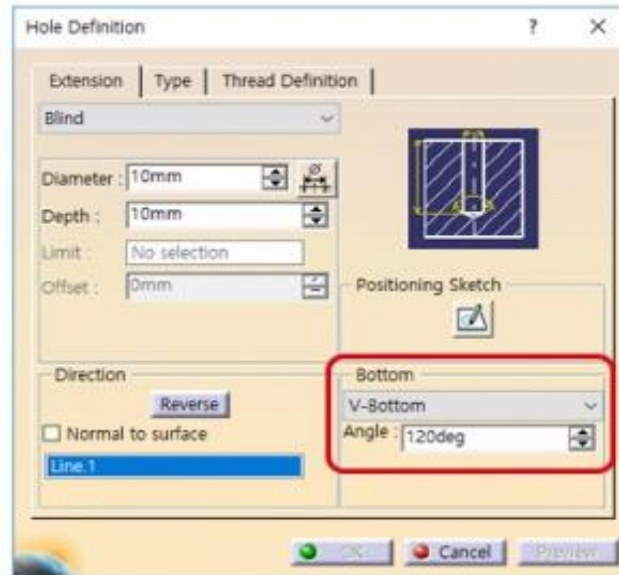
#### Hole



#### 2-1 Hole

- Hole Bottom

: Hole의 아래쪽 형상 정의



Flat



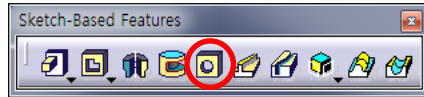
V-Bottom

# 1

## CADD (Computer Aided Design and Drafting)

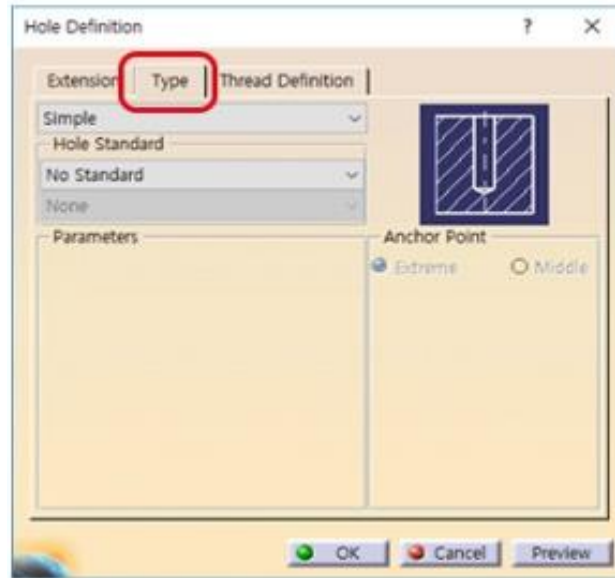
### 3D 실습 (Groove, Hole)

#### Hole



#### 2-1 Hole

- Hole Type  
: Hole의 전체 형상 정의



Simple



Tapered



Counterbored



Countsunk

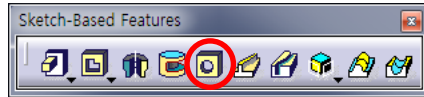


Counterdrilled

# 1 CADD (Computer Aided Design and Drafting)

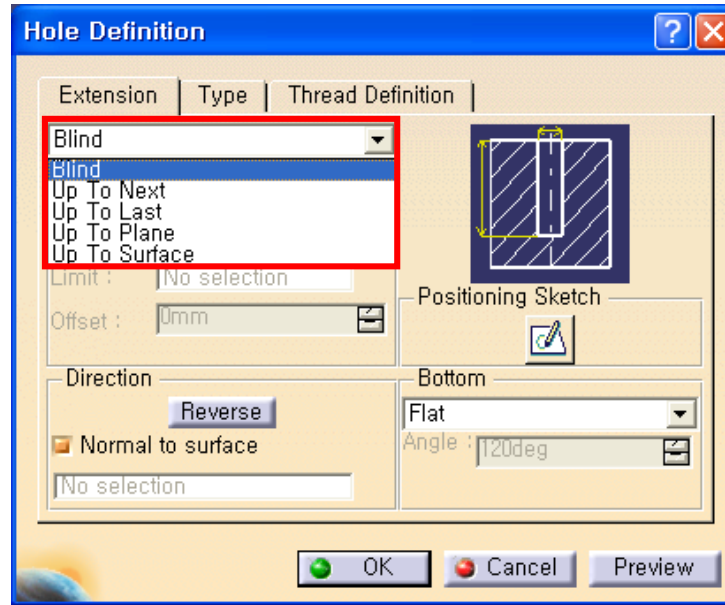
## 3D 실습 (Groove, Hole)

### Hole



### 2-1 Hole

- Extension Tap  
: Hole의 깊이와 지름 정의

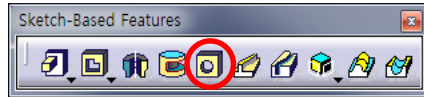


# 1

## CADD (Computer Aided Design and Drafting)

### 3D 실습 (Groove, Hole)

#### Hole



#### 2-1 Hole



① Blind type : 지름과 깊이의 값을 입력하여 Hole을 생성한다.



② Up to Next type : Hole을 생성하기 위하여 지정한 면으로부터 바로 다음 면까지 Hole을 생성한다.



③ Up to Last type : 모델의 마지막 면까지 Hole을 생성한다.



④ Up to Plane : 지정한 Plane까지 Hole을 생성한다.



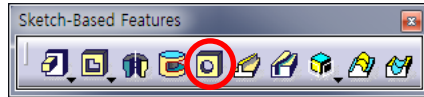
⑤ Up to Surface : 지정한 Surface까지 Hole을 생성한다.

# 1

## CADD (Computer Aided Design and Drafting)

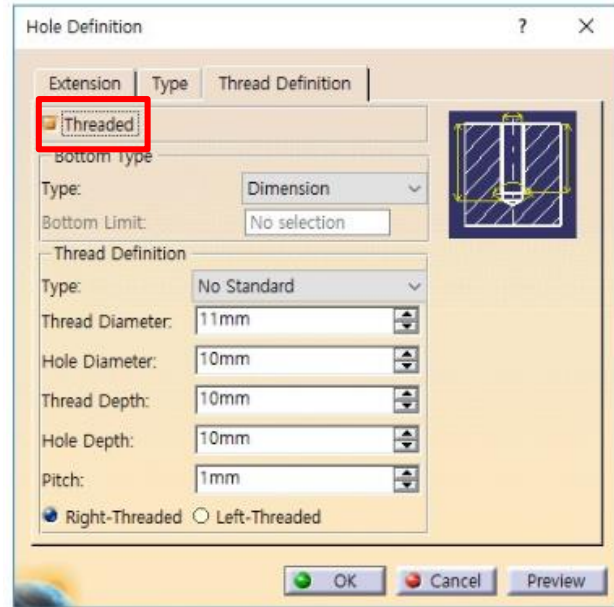
### 3D 실습 (Groove, Hole)

#### Hole



#### 2-1 Hole

- Thread Definition
  - Hole에 나사산을 정의한다
  - 모델링 형상에는 표현되지 않으며 도면 작업 시 활용된다

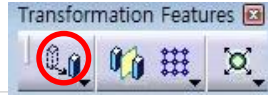


# 1

## CADD (Computer Aided Design and Drafting)

### 3D 실습 (Groove, Hole)

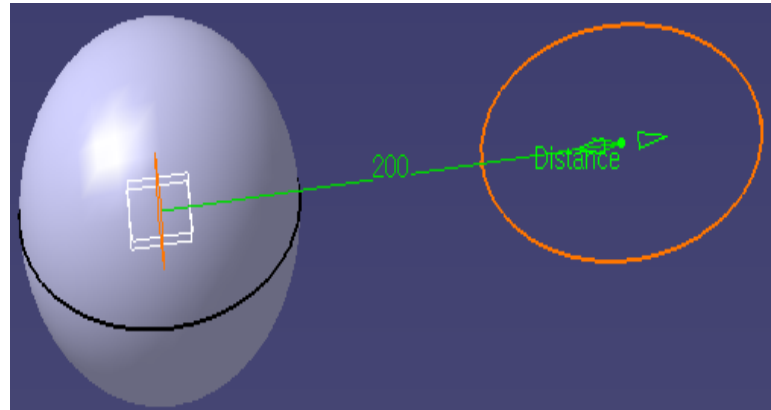
#### Transformation Features



#### 3-1 Translation



- Part를 원하는 위치로 이동시키는 명령어
- 방향과 거리를 입력한다



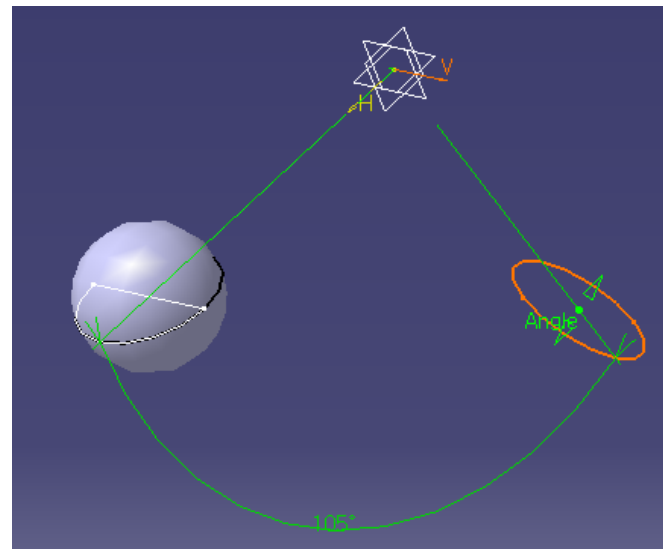
# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

### Transformation Features

#### 3-2 Rotation

- Part를 특정한 축을 중심으로 회전 시키는 명령어
- 축 지정과 각도를 입력한다

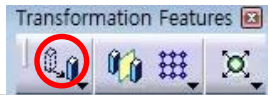




# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Groove, Hole)

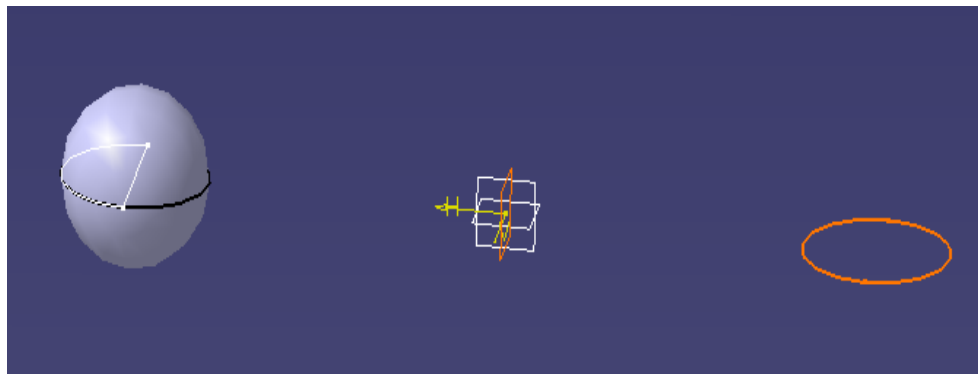
### Transformation Features



#### 3-2 Rotation



- Part를 특정한 면을 기준으로 대칭이동 시키는 명령어
- 면 지정



# 1 CADD (Computer Aided Design and Drafting)

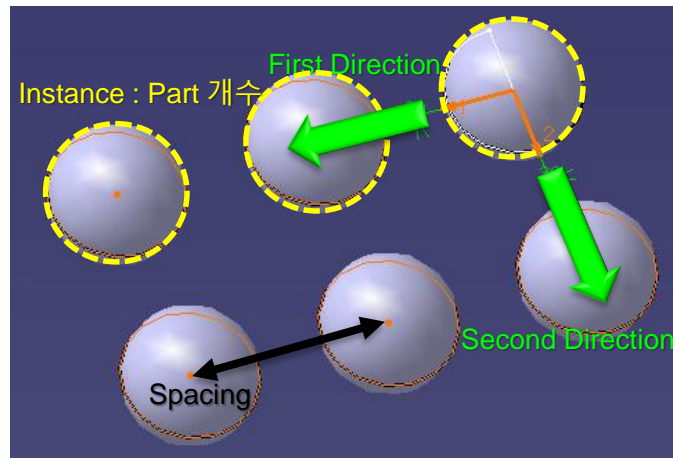
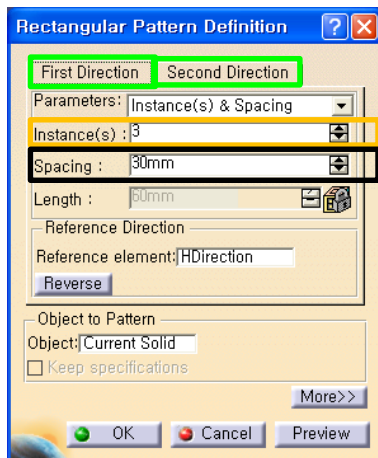
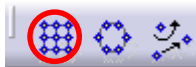
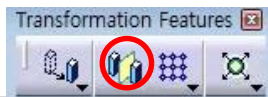
## 3D 실습 (Groove, Hole)

### Transformation Features

#### 3-4 Pattern

##### 1. Rectangular Pattern

- 직사각형 형태의 패턴을 생성하는 명령어
- Reference Direction 옵션에서 방향을 지정



# 1 CADD (Computer Aided Design and Drafting)

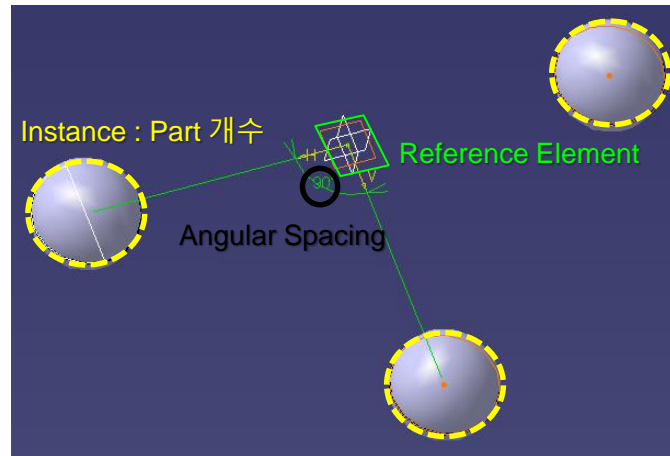
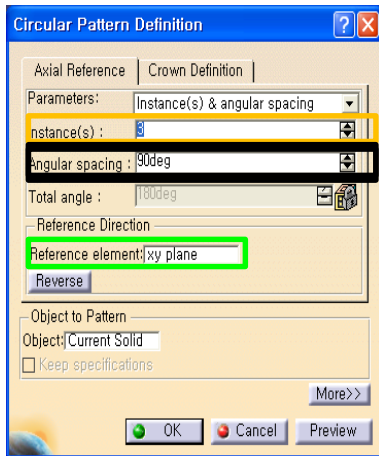
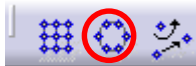
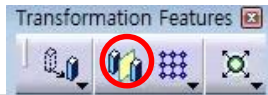
## 3D 실습 (Groove, Hole)

### Transformation Features

#### 3-4 Pattern

#### 2. Circular Pattern

- 원형 형태의 패턴을 생성하는 명령어
- Reference Direction 옵션에서 면을 지정



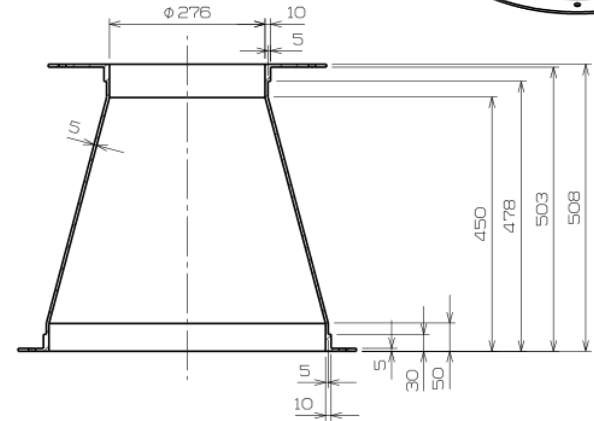
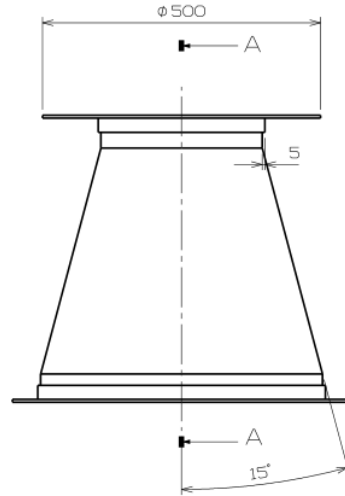
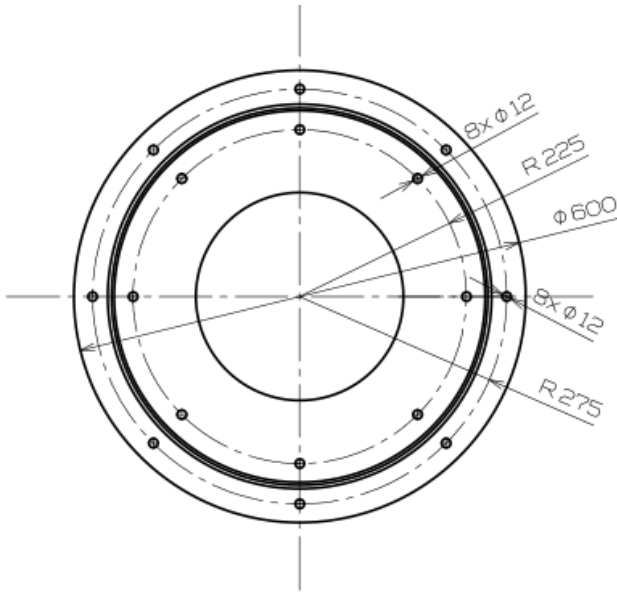
CADD (Computer Aided Design and Drafting)

## II 예제 도면을 통한 실습

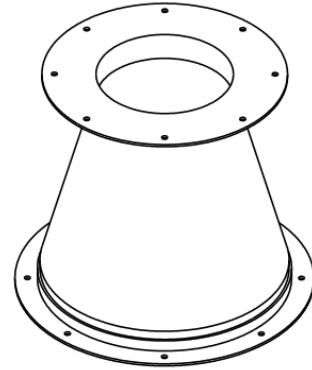
# 3

## CADD (Computer Aided Design and Drafting)

### 예제 도면을 통한 실습

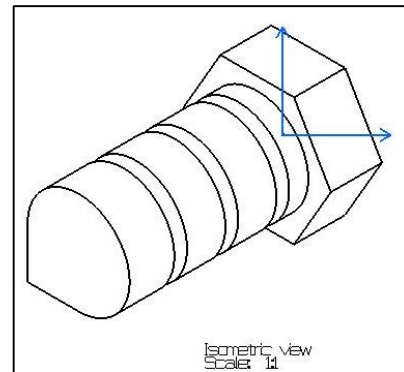
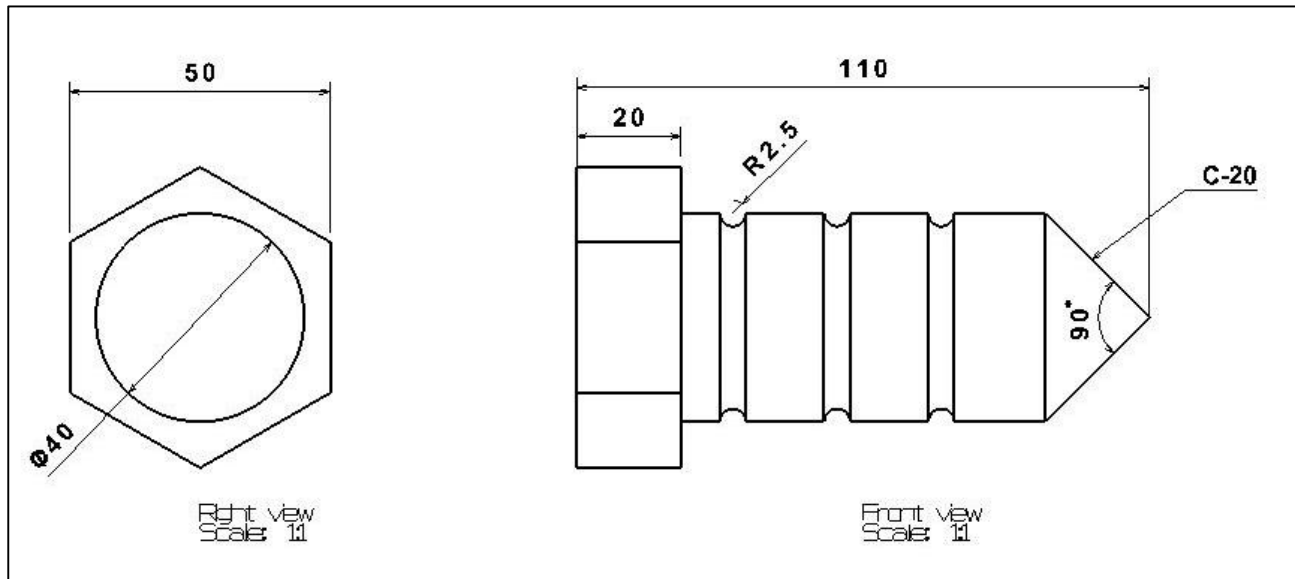


SECTION A-A



# 3 CADD (Computer Aided Design and Drafting)

## 예제 도면을 통한 실습



# 감사합니다