

## 2021 학년도 기계공학과 CADD (MEE1002)

CADD(Computer Aided Design and Drafting)

한양대학교  
터보기계연구실  
Prof. Keun Ryu

2021년 9월 23일



**HANYANG UNIVERSITY**



## CONTENTS

**I** 3D 실습 (Pad, Pocket, Shaft)

**II** 예제 도면을 통한 실습

CADD (Computer Aided Design and Drafting)

# I 3D 실습 (Pad, Pocket, Shaft)

# 1 CADD(Computer Aided Design and Drafting)

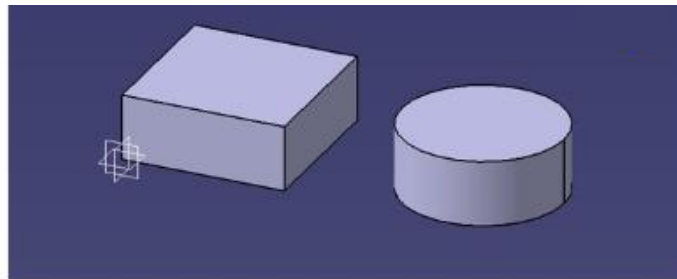
## 3D 실습(Pad, Pocket, Shaft)

### Pad

#### 1-1 Pad



- Sketch에 높이 값과 방향을 지정하여 Solid를 생성하는 기능
  - 기본적으로 하나의 Sketch에 작업된 Profile은 한번에 Pad로 생성된다
  - Pad 또는 화면상 Geometry를 더블 클릭하면 Pad의 값을 수정 할 수 있다



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

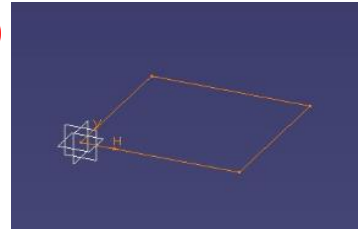
#### 1-1 Pad

#### • Pad 생성 과정

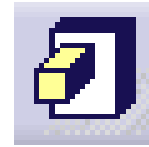
- ① Pad 작업을 수행할 Sketch Profile을 선택
- ② Pad Icon
- ③ Pad Definition으로 항목 수정
- ④ Pad 생성



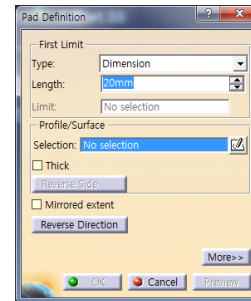
①



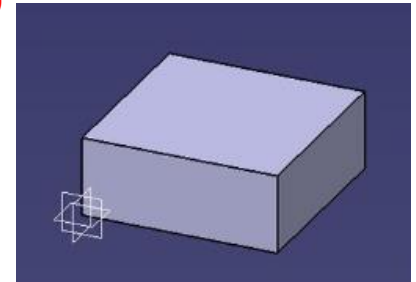
②



③



④



# 1 CADD(Computer Aided Design and Drafting)

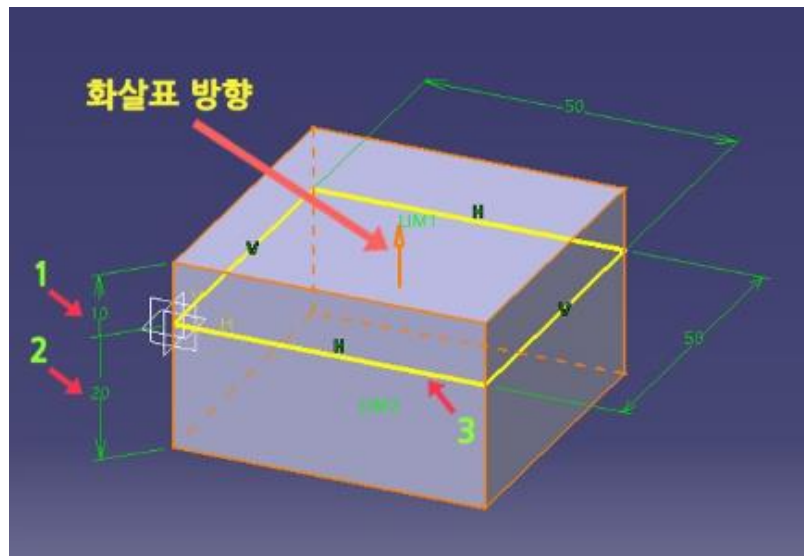
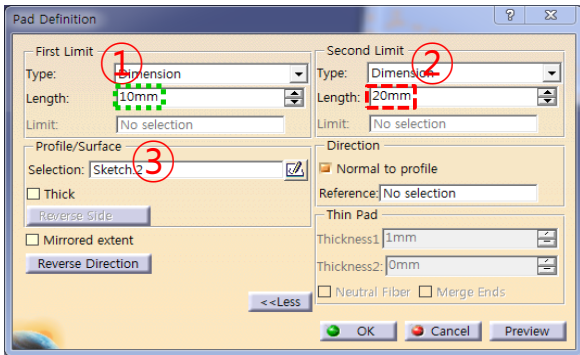
## 3D 실습(Pad, Pocket, Shaft)

### Pad

#### 1-1 Pad

- Pad Definition

- ① First Limit Length : 화살표 방향의 값을 설정한다
- ② Second Limit Length : 화살표 반대 방향의 값을 설정한다
- ③ Profile/Surface Selection : Profile 선택창



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

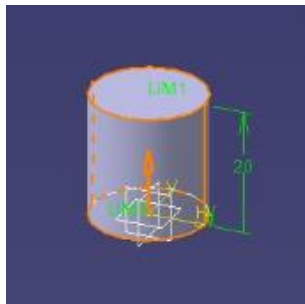
#### 1-1 Pad



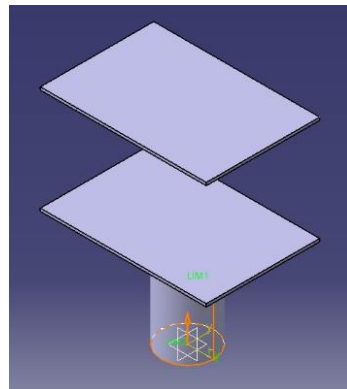
- Pad Type

- ① Dimension : Length 값을 입력하여 Pad 생성
- ② Up to next : Profile이 있는 위치에서부터 화살표 방향으로 첫번째 Solid면 까지 돌출
- ③ Up to last : Profile이 있는 위치에서부터 화살표 방향으로 마지막 Solid면 까지 돌출

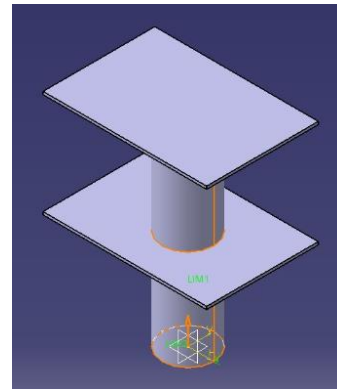
① Dimension



② Up to next



③ Up to last



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

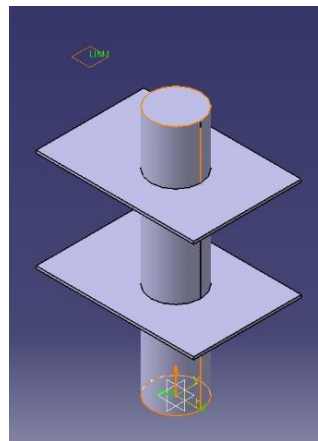
#### 1-1 Pad

- Pad Type

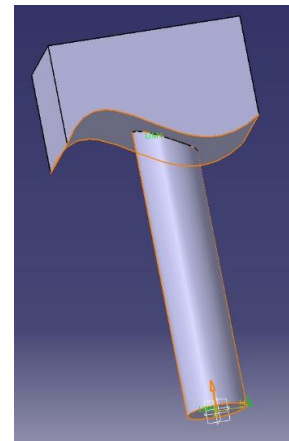
- ④ Up to plane : Limit를 면(Solid face 또는 Plane)으로 지정하여 해당 면까지 Solid 돌출
- ⑤ Up to Surface : Limit를 Surface(Solid의 곡면도 지정 가능)로 지정하여 해당 면까지 Solid 돌출



④ Up to plane



⑤ Up to Surface

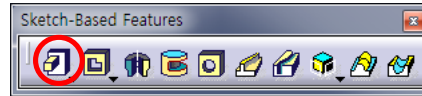




# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

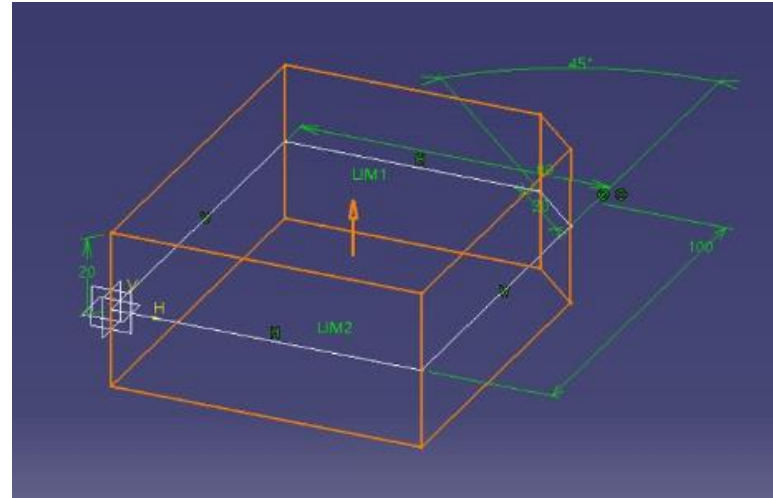
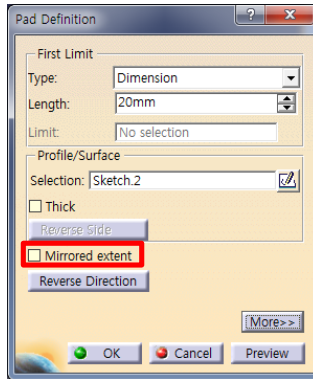
### Pad



### 1-1 Pad



- Mirrored Extent
- Profile을 기준으로 양쪽 방향으로 동일하게 돌출



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

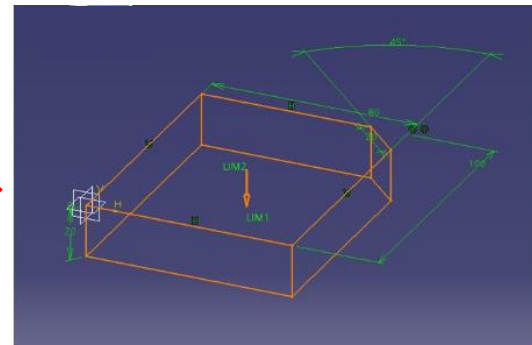
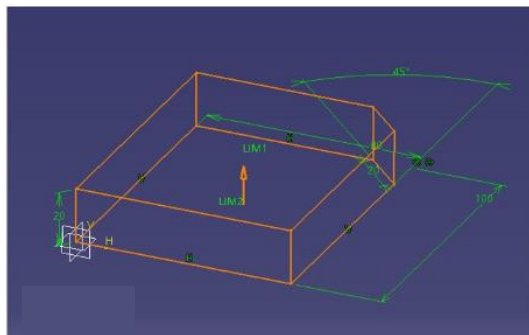
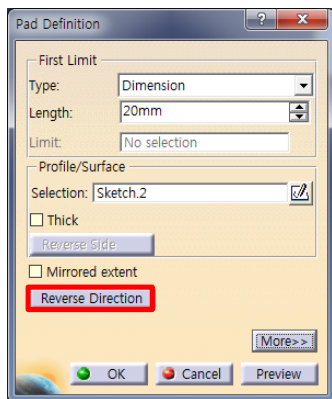


### 1-1 Pad



- **Reverse Direction**

➤ Profile을 기준으로 화살표의 방향을 전환



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

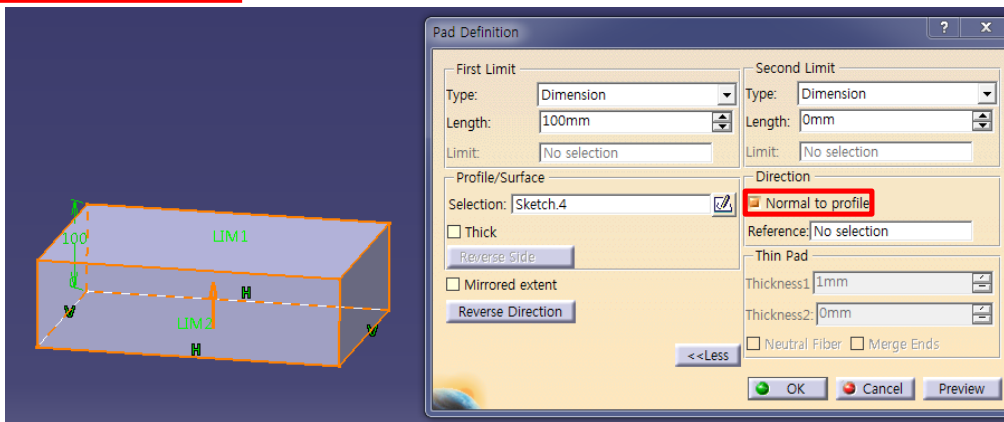
### Pad



### 1-1 Pad



- Direction (Normal to Profile)
- 활성화 시 Profile에 수직한 방향으로 돌출되며, 비 활성화 시 선택한 Reference 방향으로 돌출



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

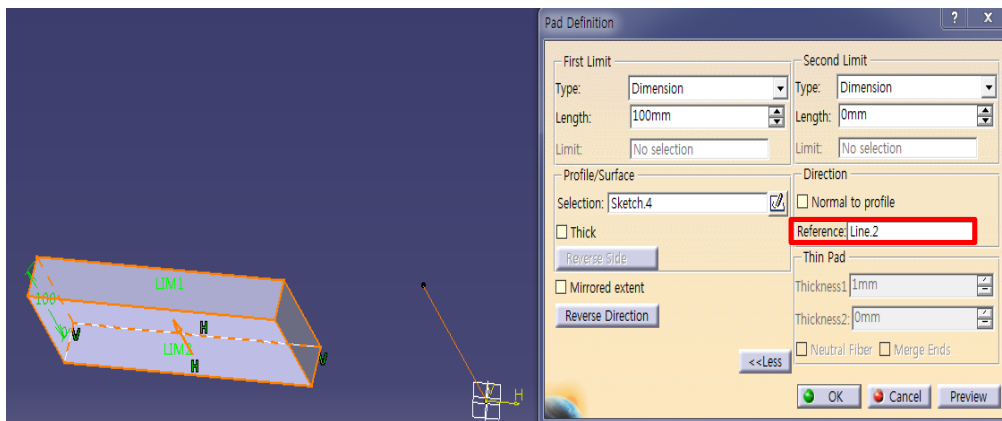
### Pad



### 1-1 Pad



- Direction (Normal to Profile)
- 활성화 시 Profile에 수직한 방향으로 돌출되며, 비 활성화 시 선택한 Reference 방향으로 돌출



# 1 CADD(Computer Aided Design and Drafting)

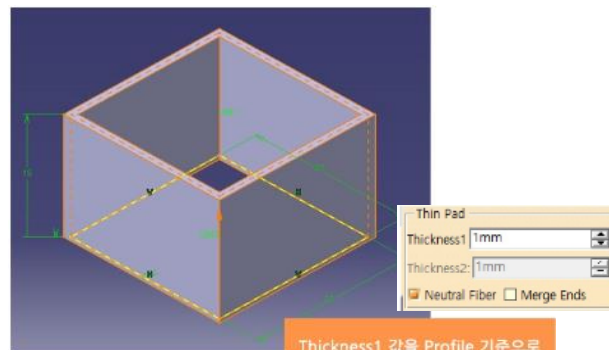
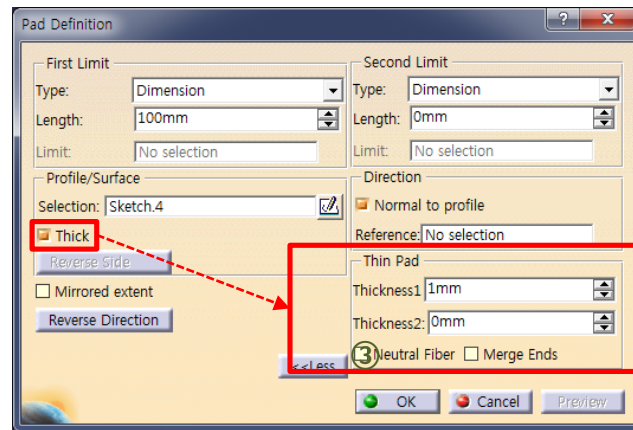
## 3D 실습(Pad, Pocket, Shaft)

### Pad

#### 1-1 Pad



- Thin Solid
  - Profile에 두께를 부여하여 Pad 생성
  - ① Thickness 1 : Profile 기준 안쪽
  - ② Thickness 2 : Profile 기준 바깥쪽
  - ③ Neutral Fiber : Thickness 1 값을 Profile 기준으로 절반씩 나눈다
  - ④ Merge Ends



Thickness1 값을 Profile 기준으로 절반씩 나눈다.

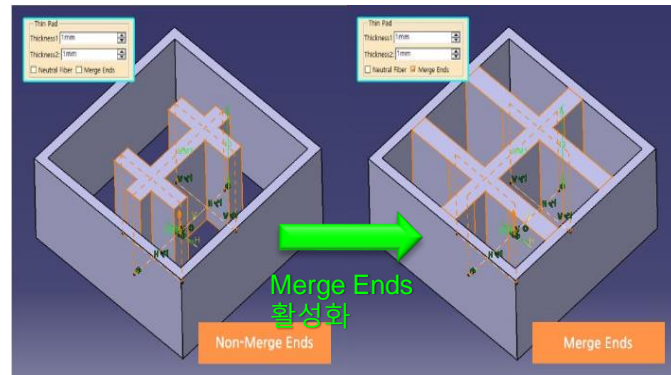
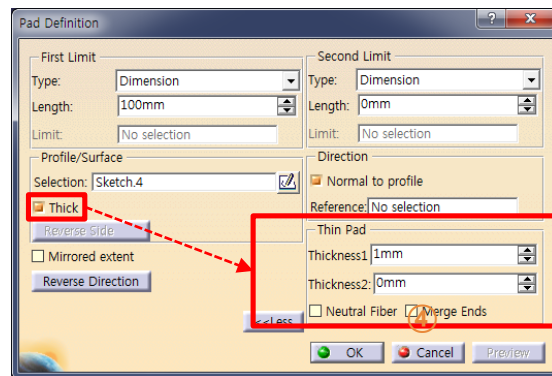
# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Pad, Pocket, Shaft)

### Pad

#### 1-1 Pad

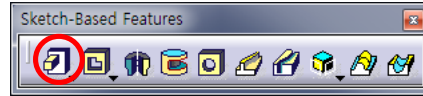
- Thin Solid
  - Profile에 두께를 부여하여 Pad 생성
    - ① Thickness 1 : Profile 기준 안쪽
    - ② Thickness 2 : Profile 기준 바깥쪽
    - ③ Neutral Fiber : Thickness 1 값을 Profile 기준으로 절반씩 나눈다
    - ④ Merge Ends



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

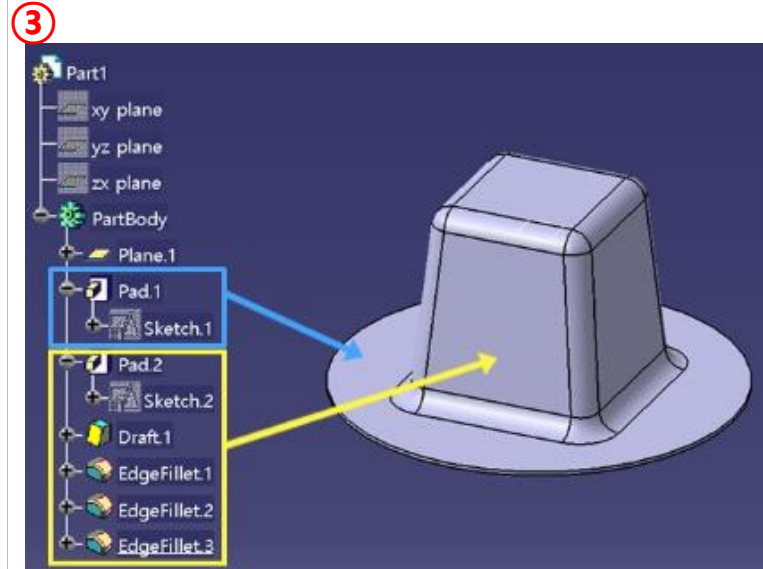
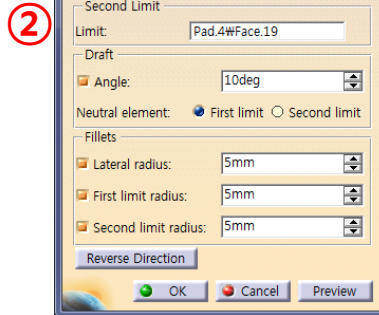
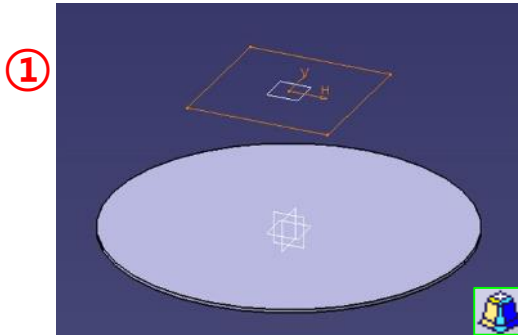


#### 1-2 Drafted Filleted Pad



- Profile을 이용하여 Pad, Draft, Fillet을 한번에 생성

- ① Drafted Filleted Pad 아이콘 클릭 후 Profile 선택
- ② Pad, Draft, Fillet 값 입력
- ③ 2번 항목에서 정의한 항목별로 Tree에 생성됨



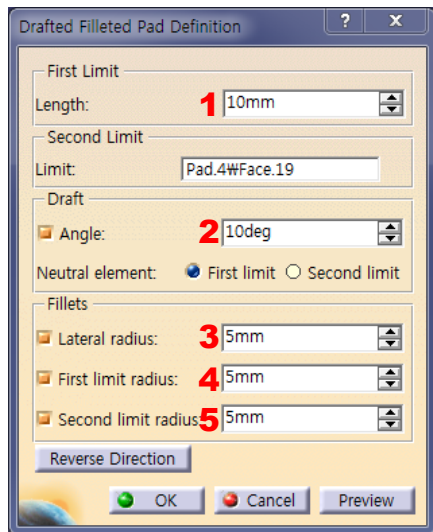
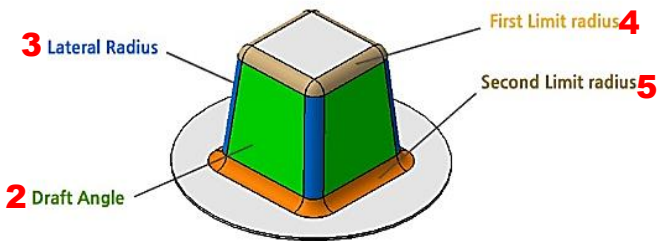
# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pad

#### 1-2 Drafted Filledted Pad

- Drafted Filleted Pad Definition



1. First / Second Limit를 이용하여 Pad값 지정
2. 기울기(구배) 지정
3. Pad 옆면 수직 모서리의 라운드 지정
4. First Limit 모서리의 라운드 지정
5. Second Limit 모서리의 라운드 지정



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Pad, Pocket, Shaft)

### Pad

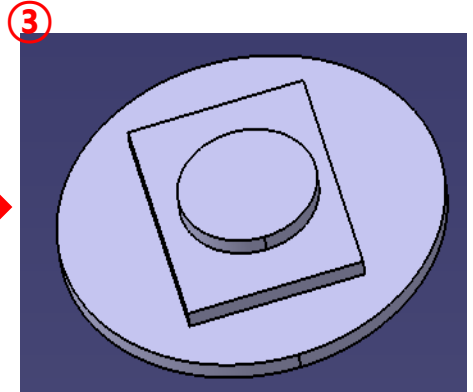
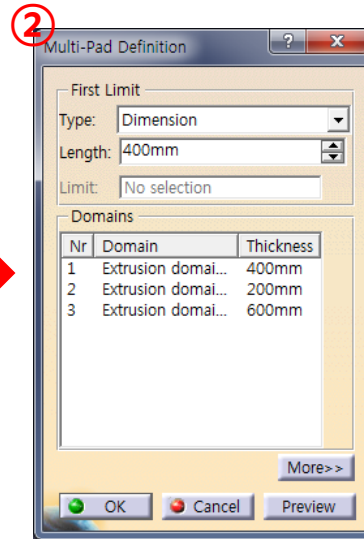
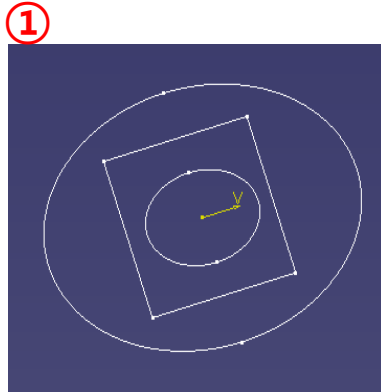


### 1-3 Multi-pad



- 한 개의 Sketch에 두께가 다른 Profile을 Pad로 생성

- ① Sketch 생성
- ② Multi-Pad 정의
- ③ Multi-Pad 생성



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

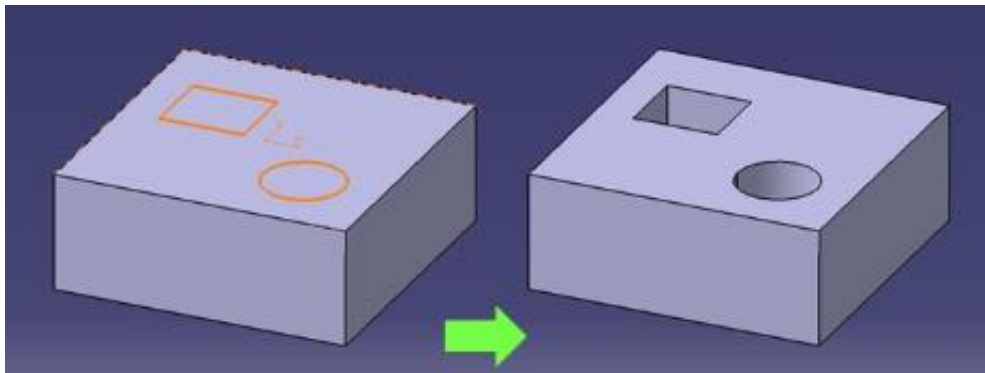
### Pocket



#### 2-1 Pocket



- Sketch에 높이 값과 방향을 지정하여 Solid를 제거하는 기능
  - 기본적으로 하나의 Sketch에 작업된 Profile은 한번에 Pocket으로 생성된다
  - Pocket 또는 화면상 Geometry를 더블클릭하면 Pocket의 값을 수정할 수 있다



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pocket

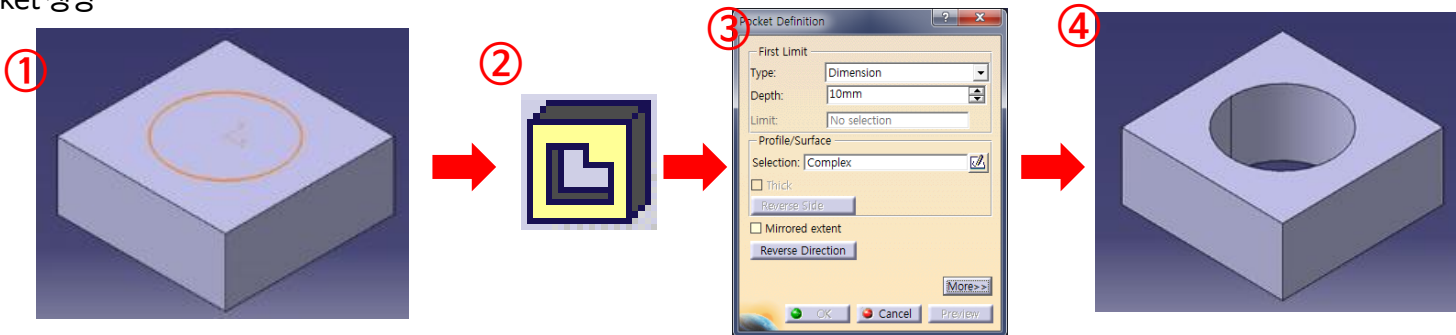


#### 2-1 Pocket



#### • Pocket 생성 과정

- ① Pocket 작업을 수행할 Sketch Profile을 선택
- ② Pocket Icon 클릭
- ③ Pocket Definition으로 항목 수정
- ④ Pocket 생성



# 1 CADD(Computer Aided Design and Drafting)

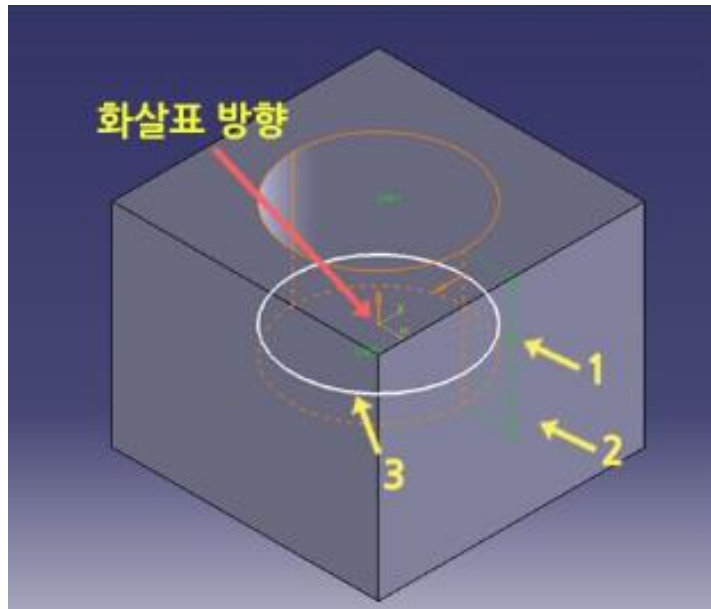
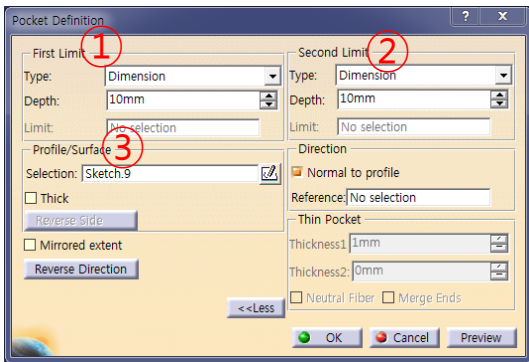
## 3D 실습(Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

- Pocket Definition

- ① First Limit Length : 화살표 방향의 값을 설정한다
- ② Second Limit Length : 화살표 반대 방향의 값을 설정한다
- ③ Profile/Surface Selection : Profile 선택창



# 1 CADD(Computer Aided Design and Drafting)

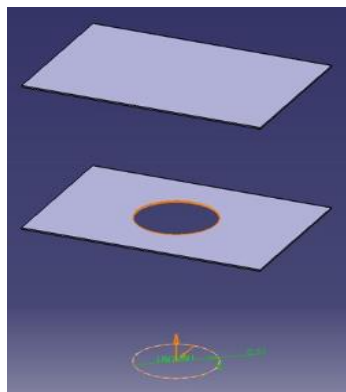
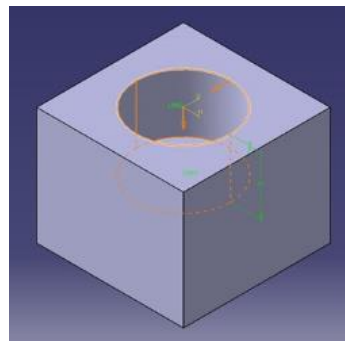
## 3D 실습(Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

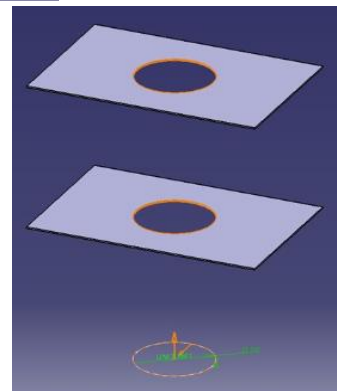
- Pocket Type

- ① **Dimension** : Length 값을 입력하여 Pocket 생성
- ② **Up to next** : Profile이 있는 위치에서부터 화살표 방향으로 첫번째 Solid면 까지 제거
- ③ **Up to last** : Profile이 있는 위치에서부터 화살표 방향으로 마지막 Solid면 까지 제거



② Up to next

① Dimension



③ Up to last

# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pocket



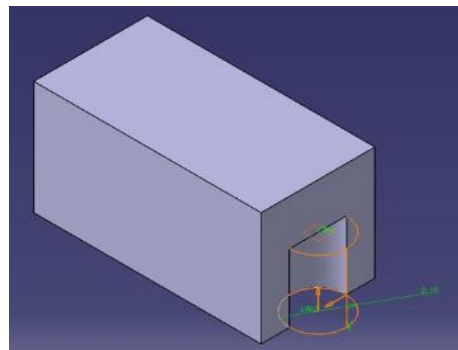
#### 2-1 Pocket



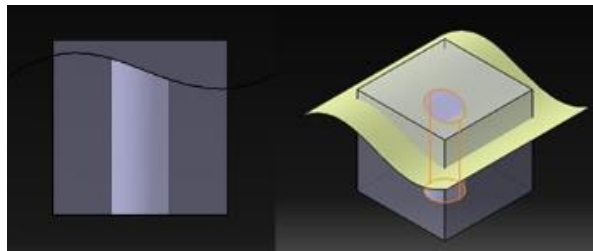
- Pocket Type

④ Up to plane : Limit를 면(Solid face 또는 Plane)으로 지정하여 해당 면까지 Solid 제거

⑤ Up to Surface : Limit를 Surface(Solid의 곡면도 지정 가능)로 지정하여 해당 면까지 Solid 제거



④ Up to plane



⑤ Up to Surface

# 1 CADD (Computer Aided Design and Drafting)

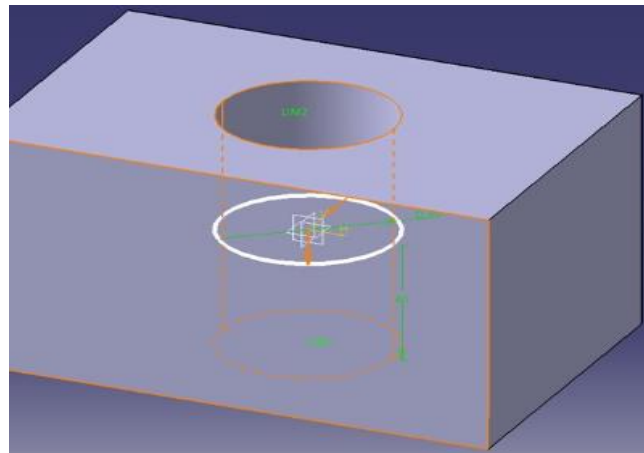
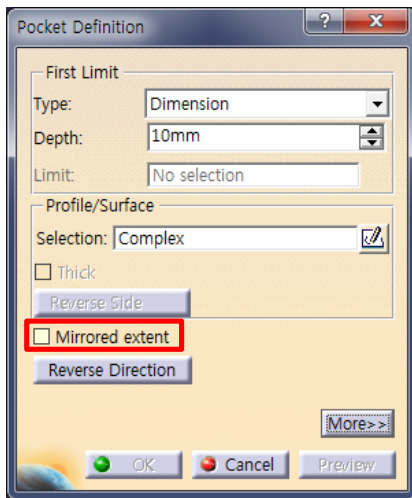
## 3D 실습 (Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

- **Mirrored Extent**

- Profile을 기준으로 양쪽 방향으로 동일하게 제거



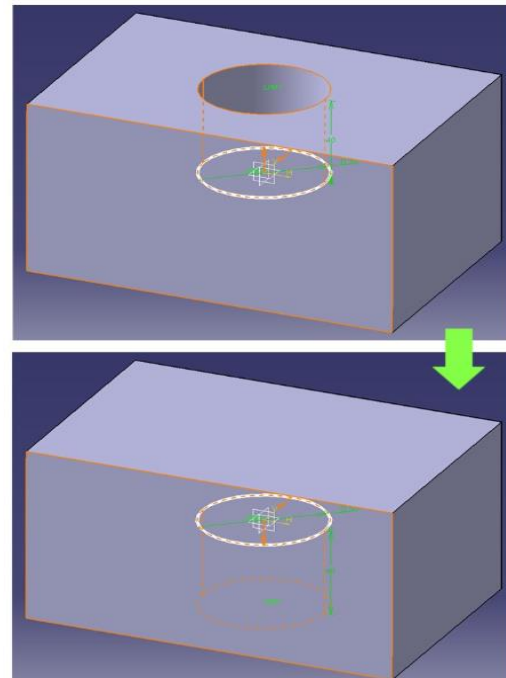
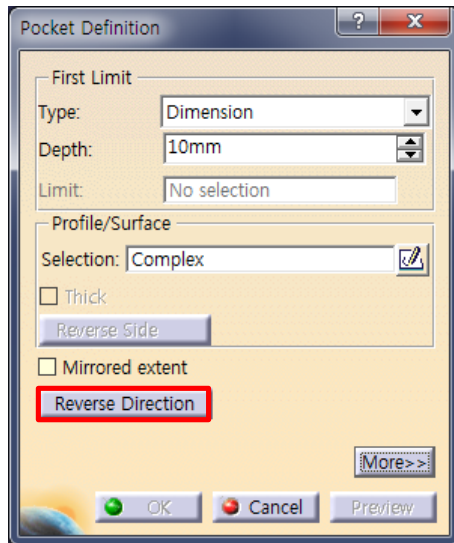
# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

- Reverse Direction
  - Profile을 기준으로 화살표의 방향을 전환





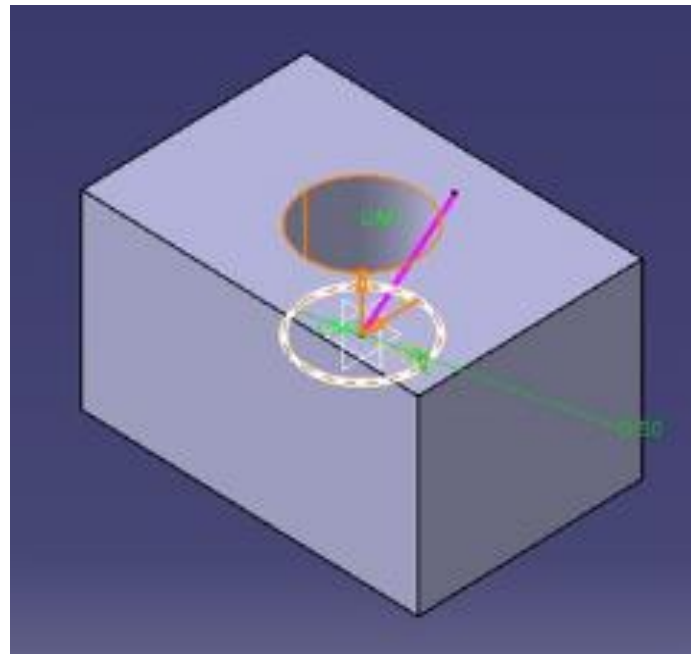
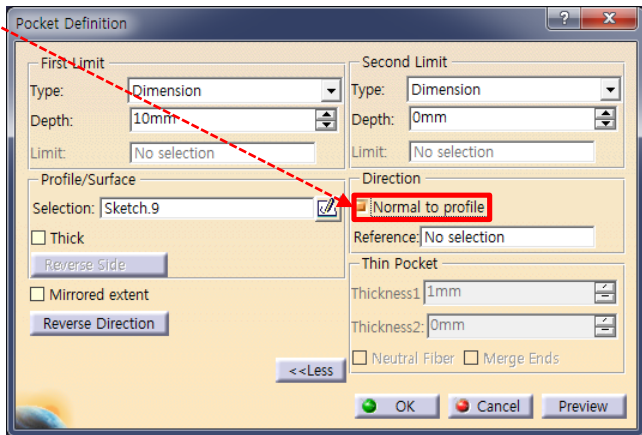
# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

- Direction (Normal to Profile)
- 활성화시 Profile에 수직인 방향으로 돌출되며, 비 활성화시 선택한 Reference 방향으로 돌출



# 1 CADD (Computer Aided Design and Drafting)

## 3D 실습 (Pad, Pocket, Shaft)

### Pocket

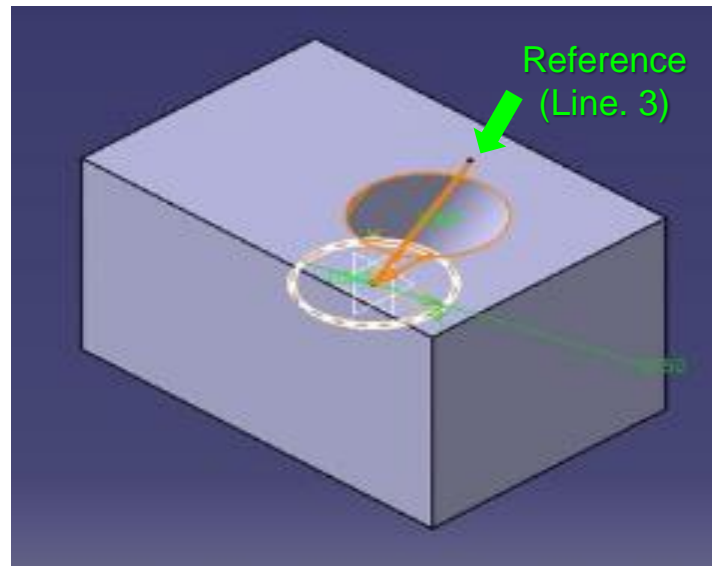
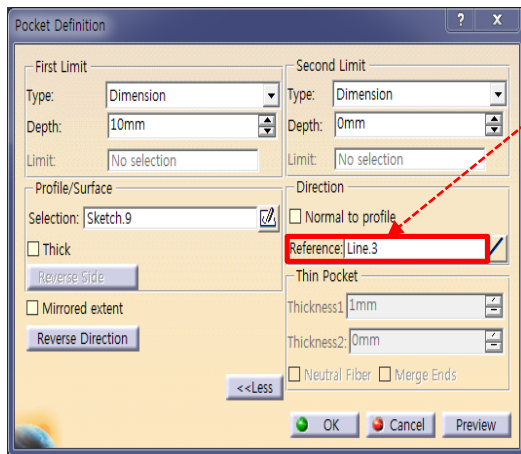


#### 2-1 Pocket



- Direction (Normal to Profile)

- 활성화 시 Profile에 수직인 방향으로 돌출되며, 비 활성화 시 선택한 Reference 방향으로 돌출



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### Pocket



#### 2-1 Pocket

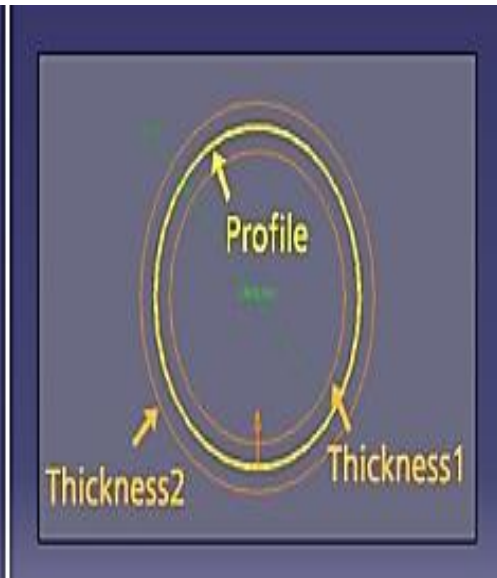
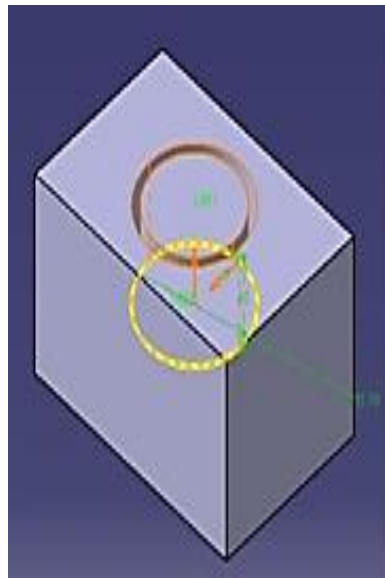
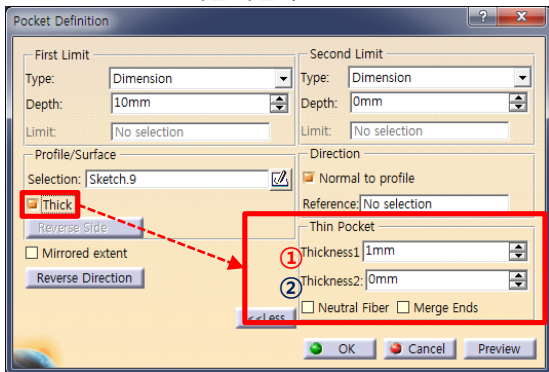


- **Thin Solid**

➢ Profile에 두께를 부여하여 Pad 제거

① Thickness 1 : Profile 기준 안쪽

② Thickness 2 : Profile 기준 바깥쪽



# 1 CADD(Computer Aided Design and Drafting)

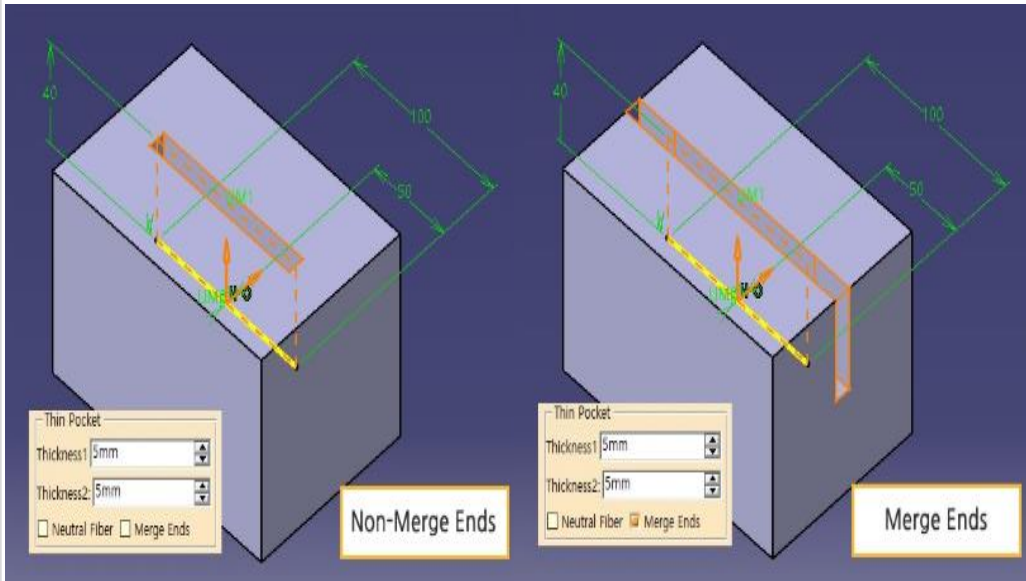
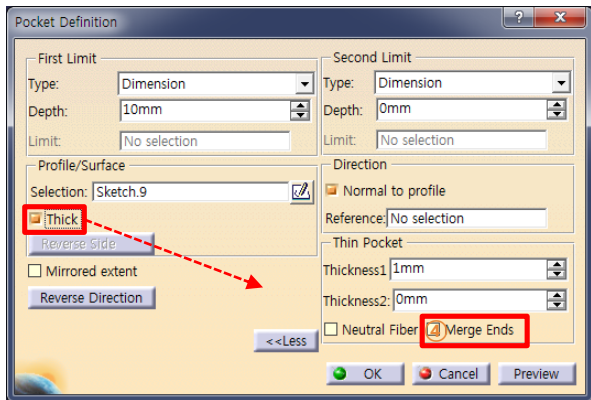
## 3D 실습(Pad, Pocket, Shaft)

### Pocket

#### 2-1 Pocket

- Thin Solid
- Profile에 두께를 부여하여 Pad 제거

#### ④ Merge Ends



# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

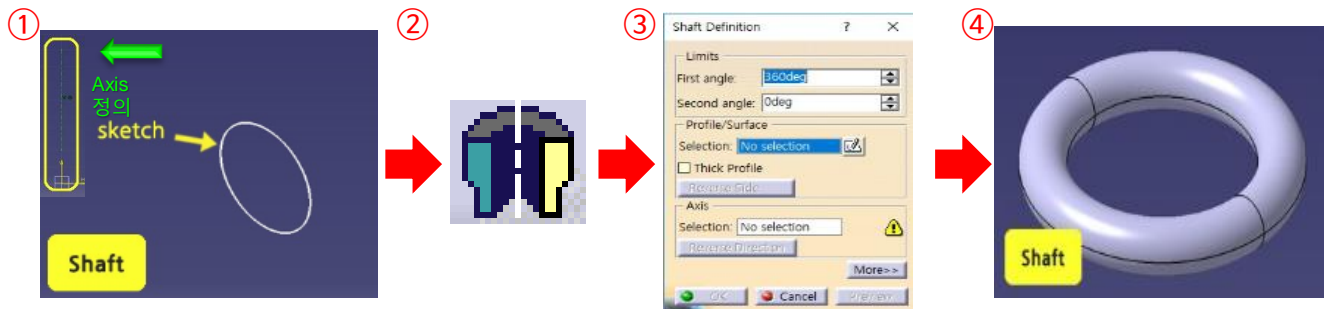
### Shaft



#### 3-1 Shaft

- 선택한 Sketch를 정의한 축 기준으로 회전시켜 형상을 생성

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

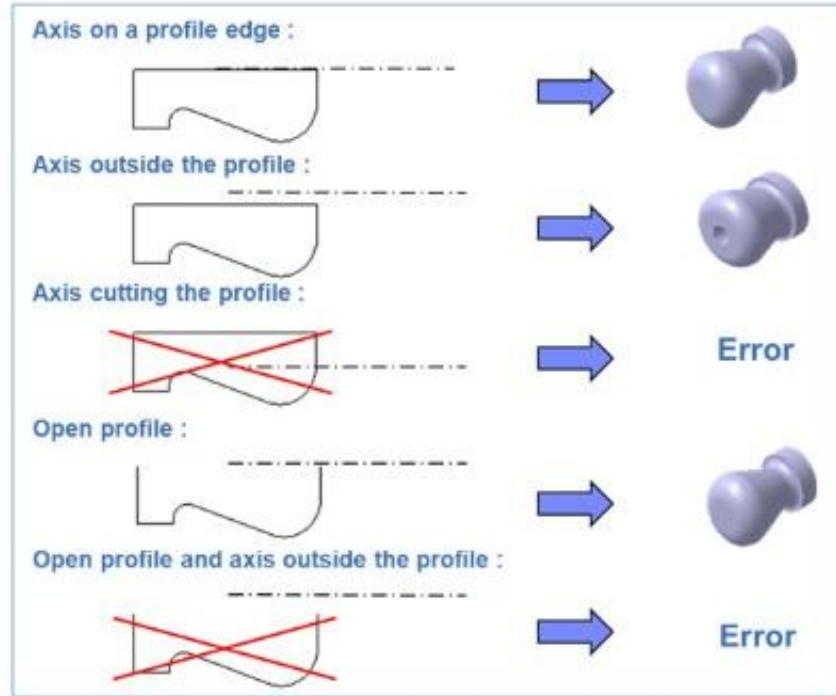


# 1 CADD(Computer Aided Design and Drafting)

## 3D 실습(Pad, Pocket, Shaft)

### 3-1 Shaft

- 생성 조건

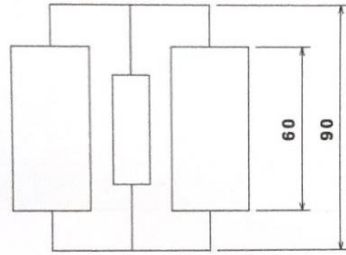


CADD (Computer Aided Design and Drafting)

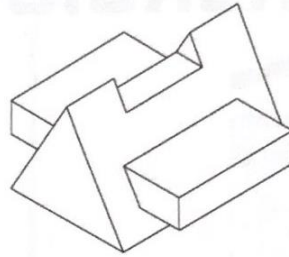
## 예제 도면을 통한 실습

# 3 CADD(Computer Aided Design and Drafting)

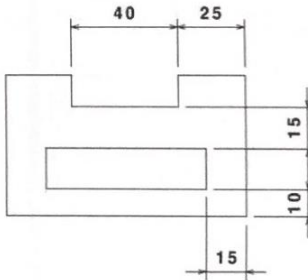
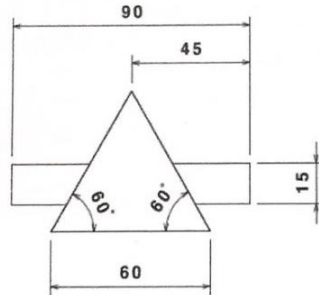
## 예제 도면을 통한 실습



Top view  
Scale: 2:3



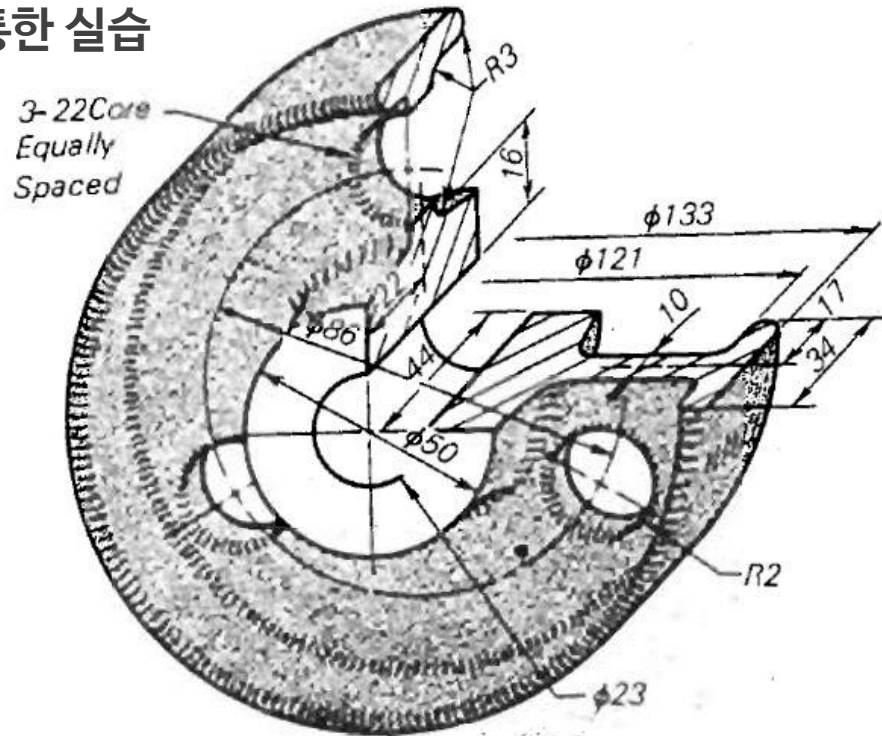
Isometric view  
Scale: 2:3





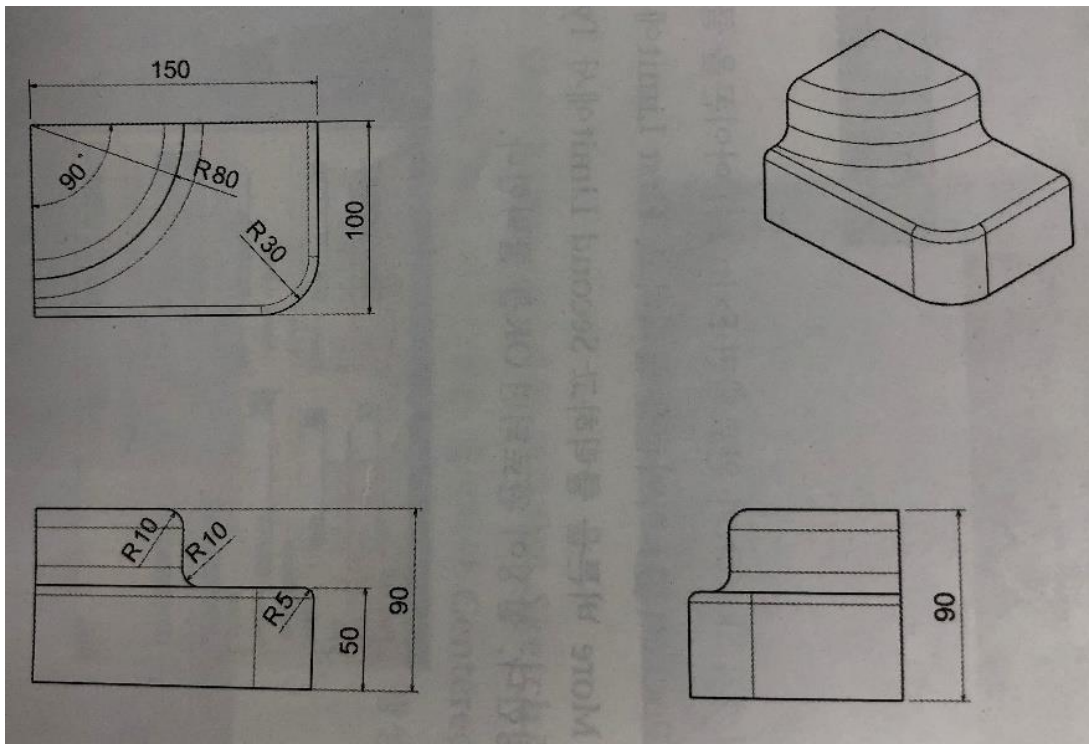
# 3 CADD (Computer Aided Design and Drafting)

## 예제 도면을 통한 실습



# 3 CADD(Computer Aided Design and Drafting)

## 예제 도면을 통한 실습



# 감사합니다