

# LESION 3 ADVANCE ROTATION LAB 02 (WATER BOTTLE)

IDE 3301 CAID

DR. PALANG WONGTANASUPORN

LECTURE 4-5

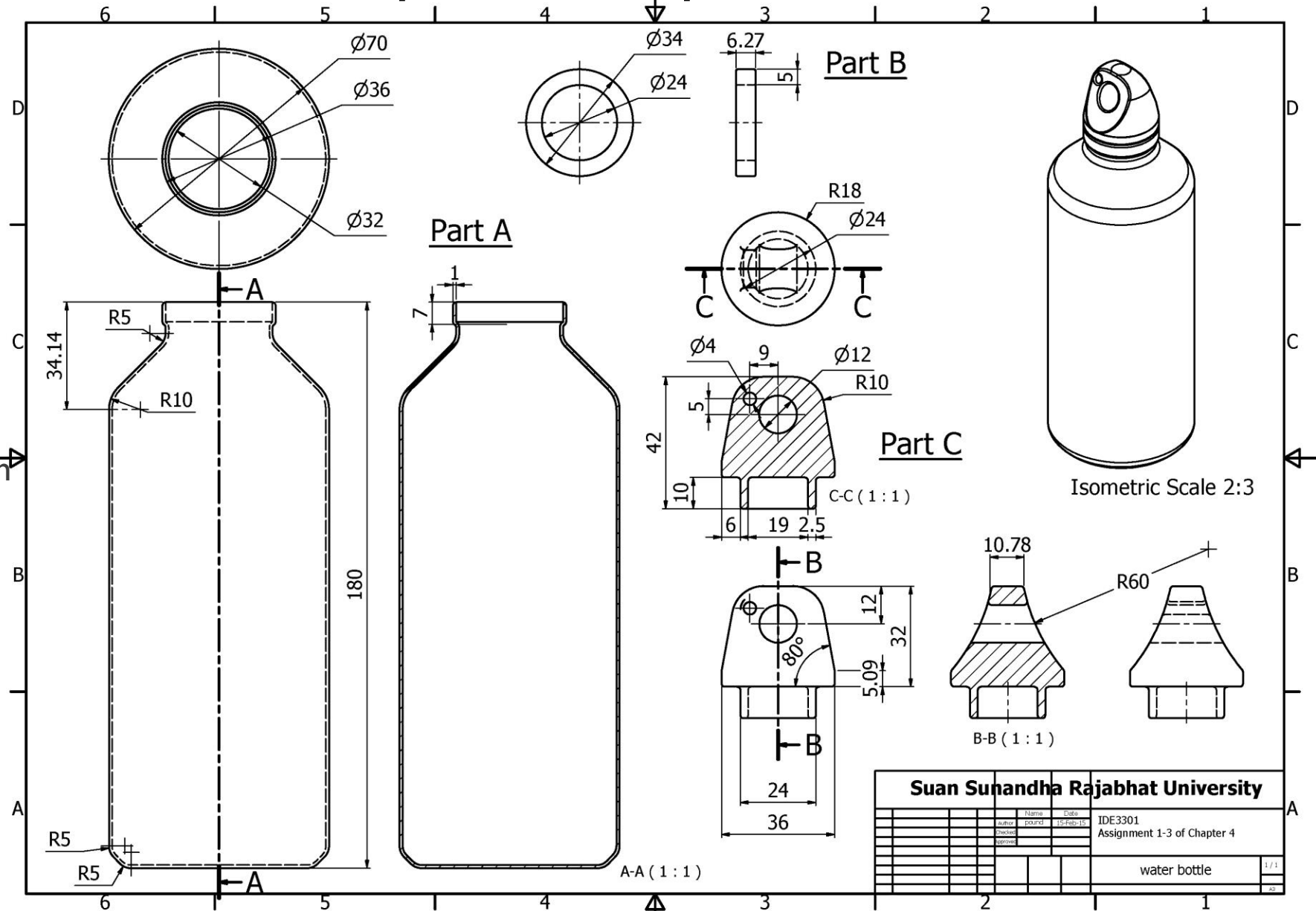
**water bottle**  
**Step by step**  
**reveal**



# This Model may be separated in three main parts **Bottle Cap and Washer**

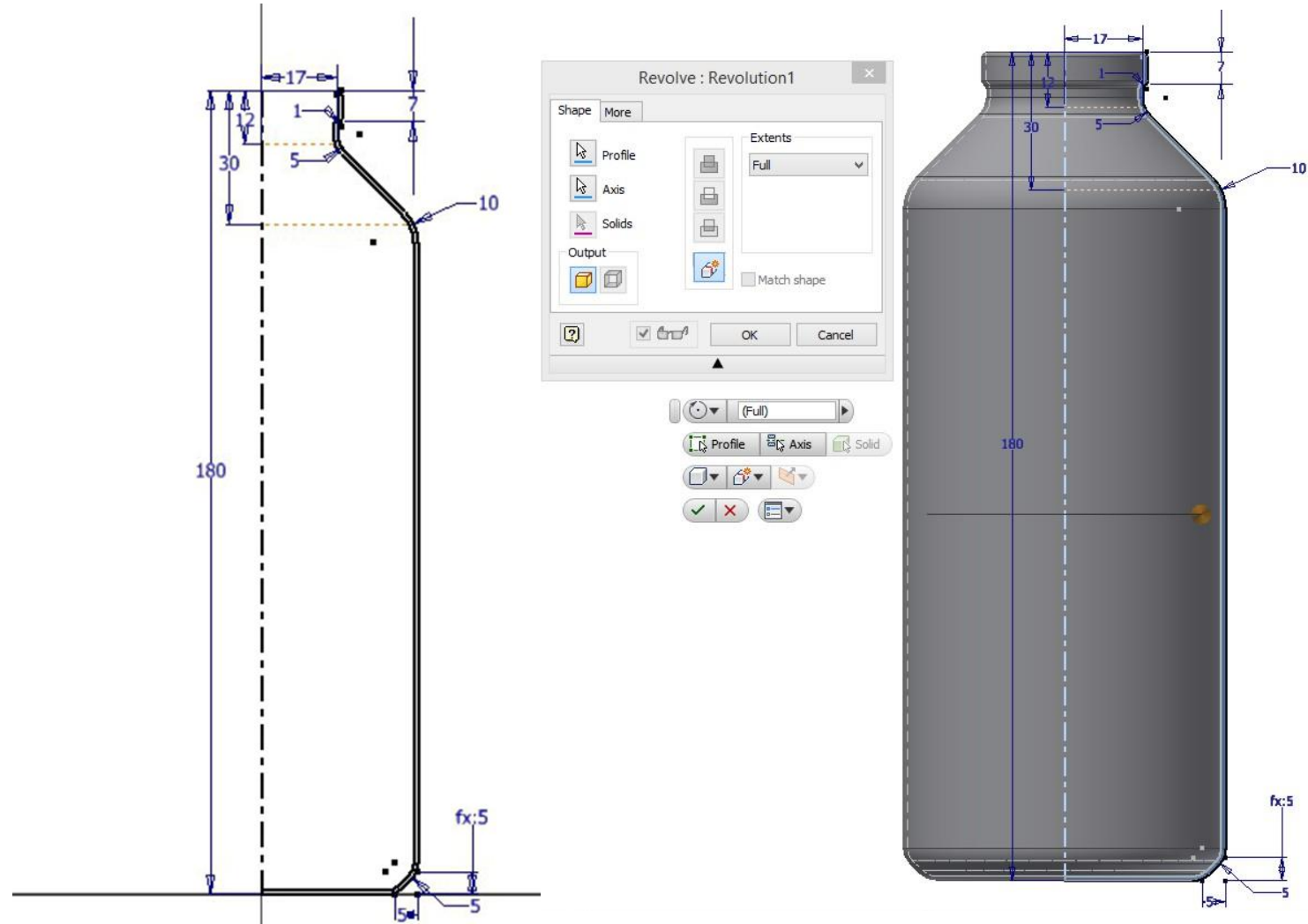
## Step of Preparation

1. Think about primitive form
2. How many component
3. How to combine
4. Reference position of each component
5. Plan for step of modeling



# MAKING A BOTTLE

- **Step 1 Draw a profile sketch on XY Plane**
- **Step 2 Create a center line for rotation axis**
- **Step 3 Select Profile**  
**Select axis**
- **Step 4 Revolve FULL + New Solid**



# MAKING A CAP

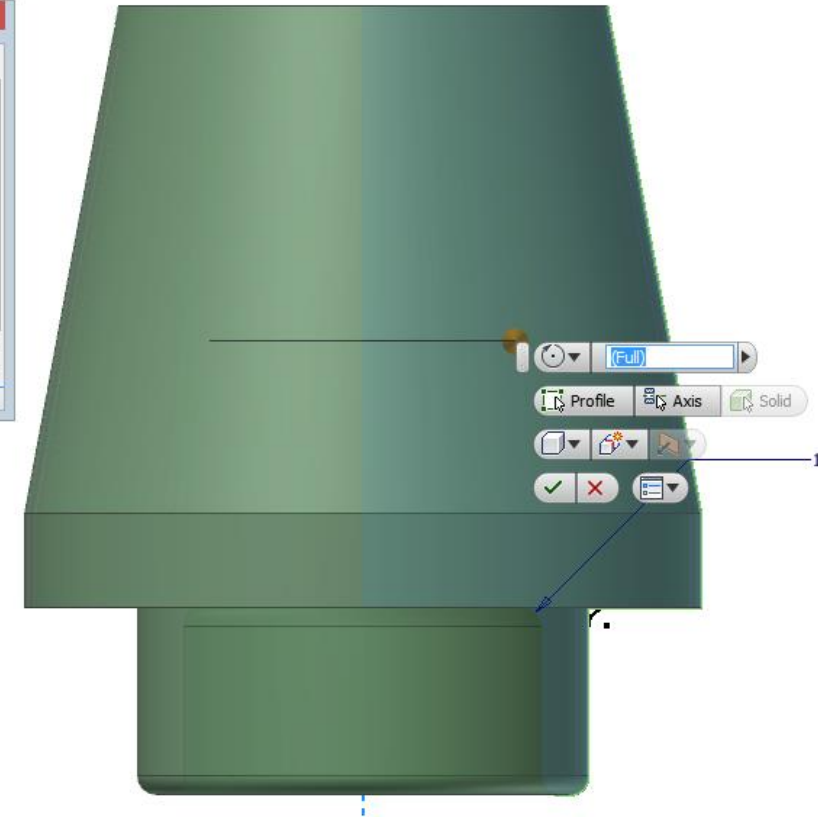
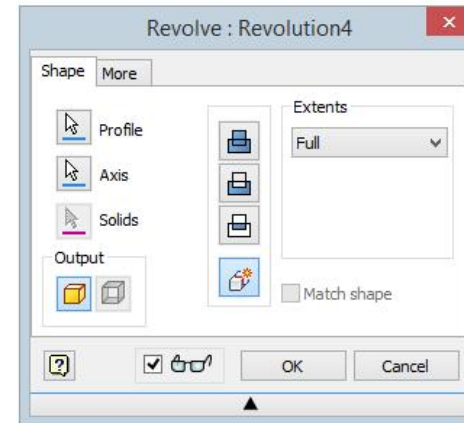
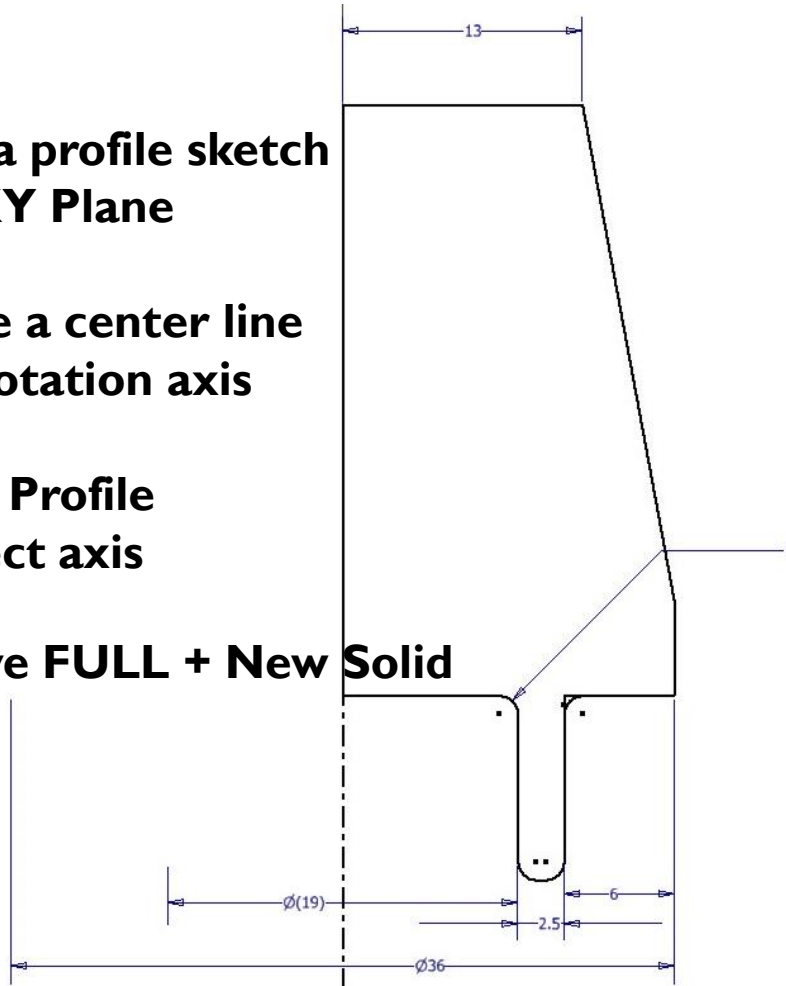
## Phase I Revolve

**Step 1 Draw a profile sketch on XY Plane**

**Step 2 Create a center line for rotation axis**

**Step 3 Select Profile  
Select axis**

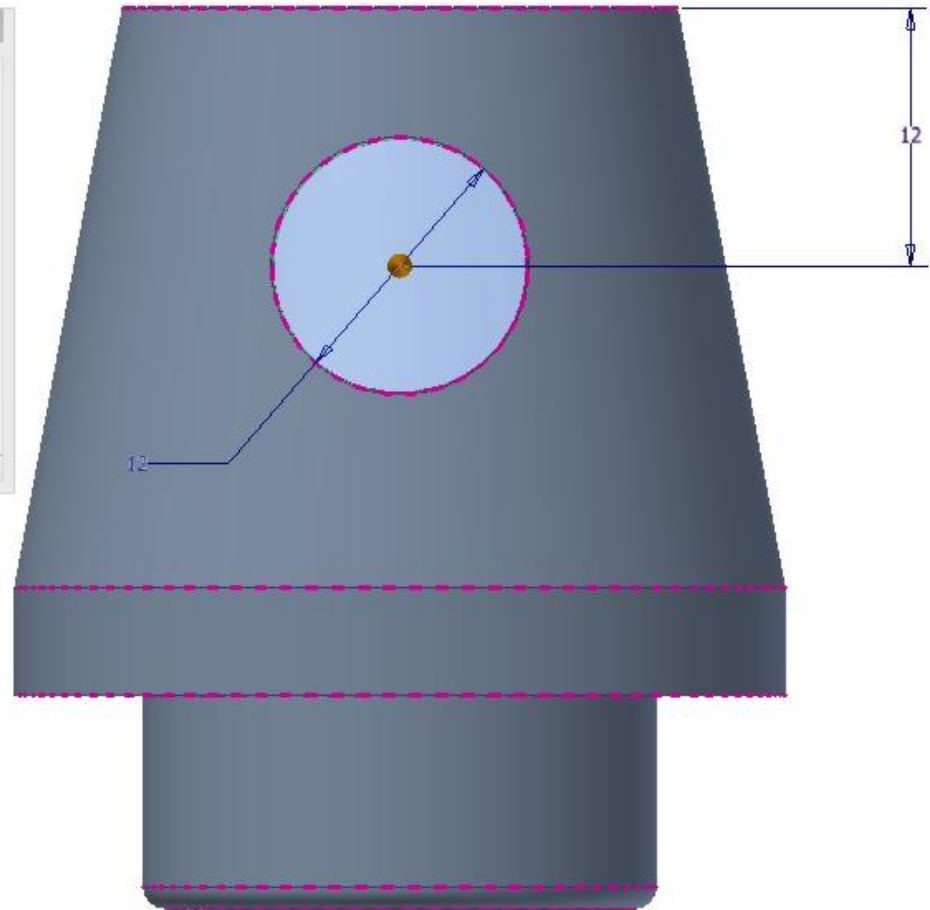
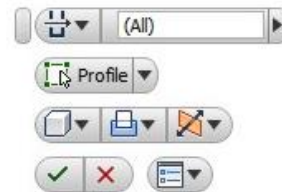
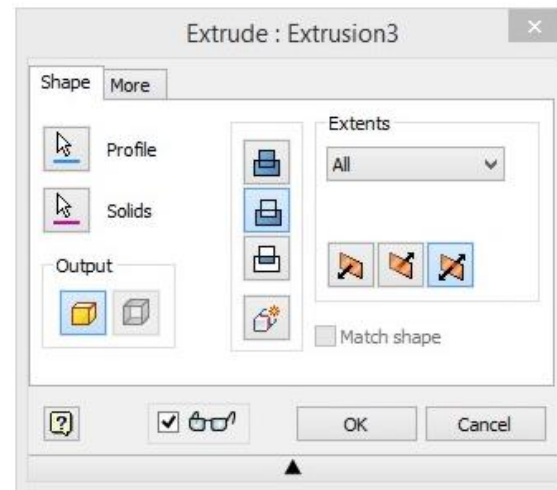
**Step 4 Revolve FULL + New Solid**



# MAKING A CAP

# PHASE 2 MAKING A BIG HOLE

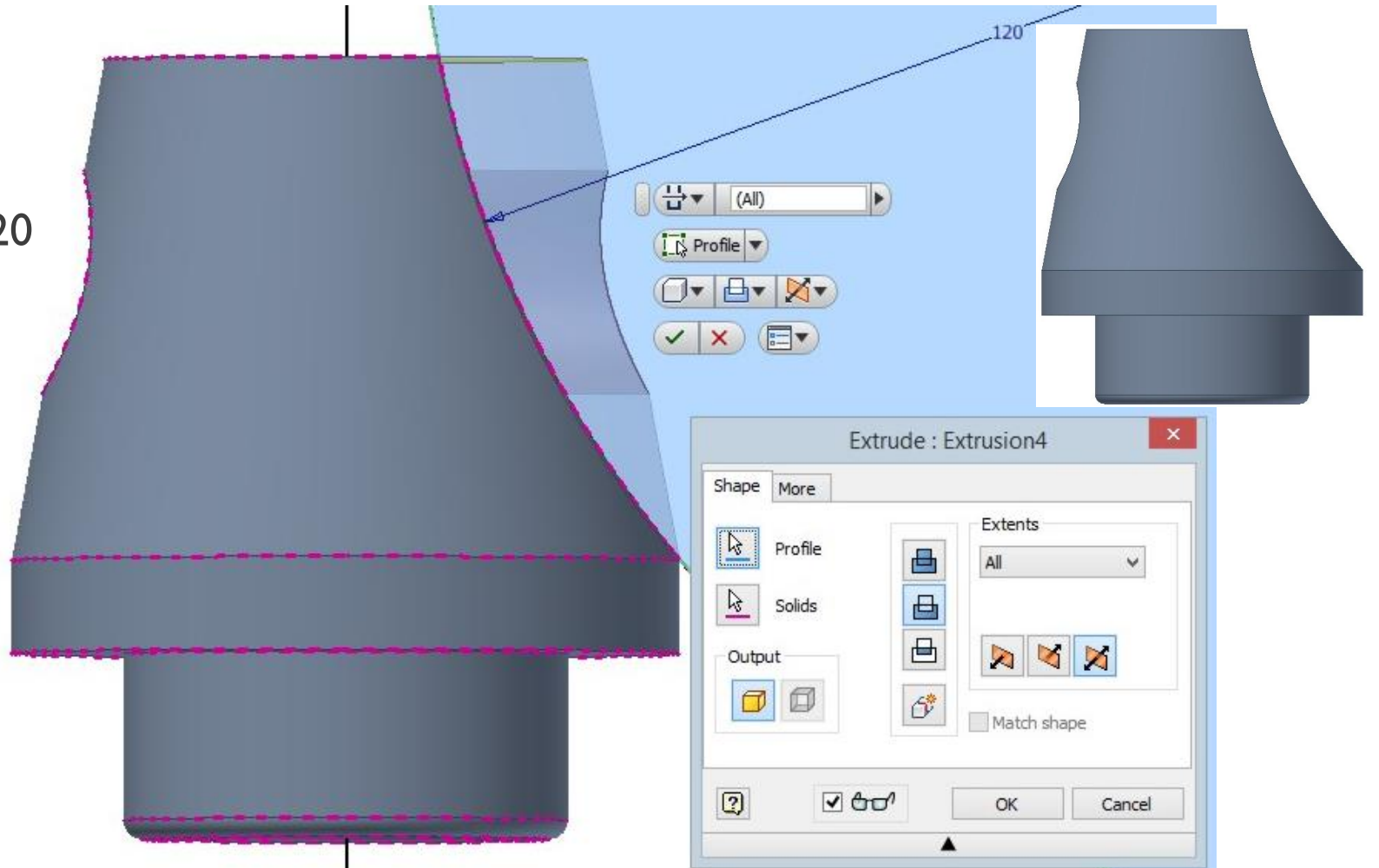
- Step1 On XY plane Draw a  $\varnothing 12$  mm circle
- Step2 Select profile sketch
- Step3 Extrude CUT + Both side



# MAKING A CAP

## PHASE 3 CURVE A FINGER TIP

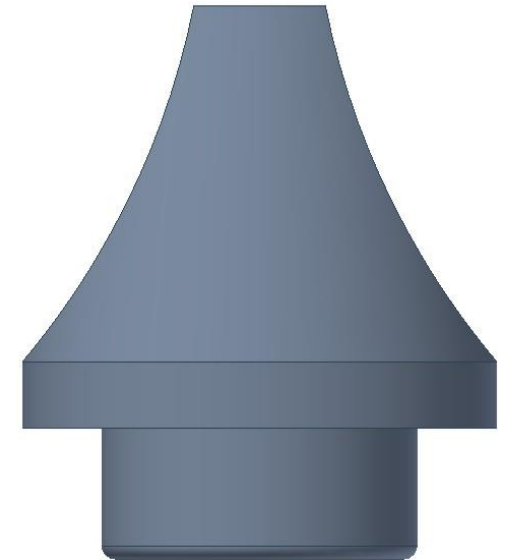
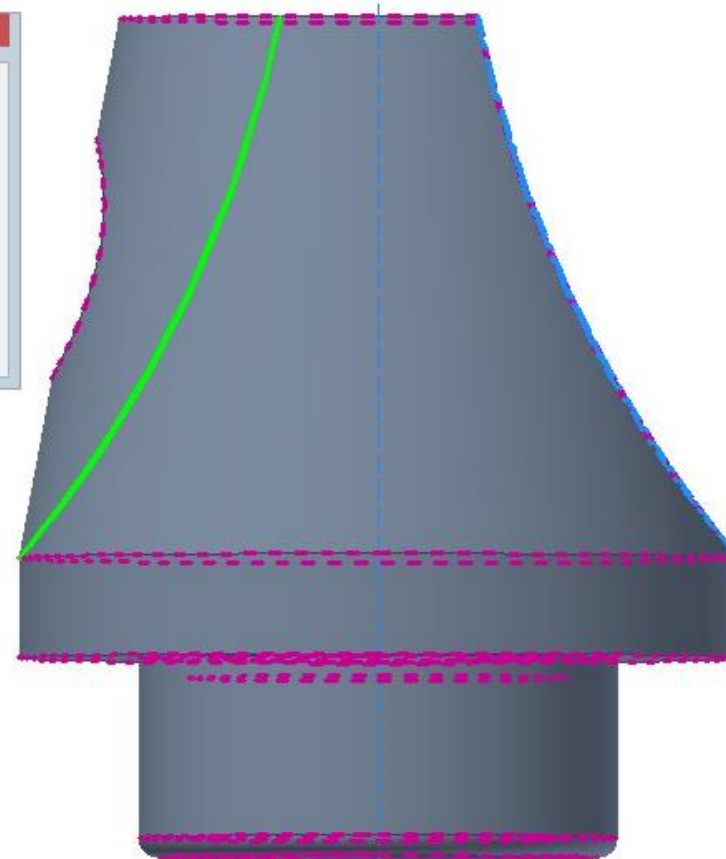
- Step1 On XY plane Draw a  $\varnothing 120$  mm circle
- Step2 Select profile sketch
- Step3 Extrude CUT + Both side



# MAKING A CAP

# PHASE 4 MIRROR ANOTHER FINGER TIP

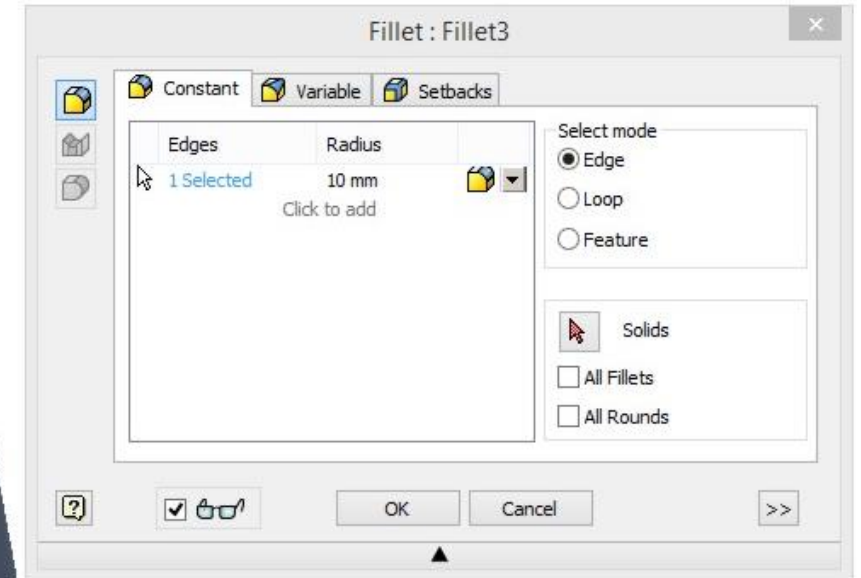
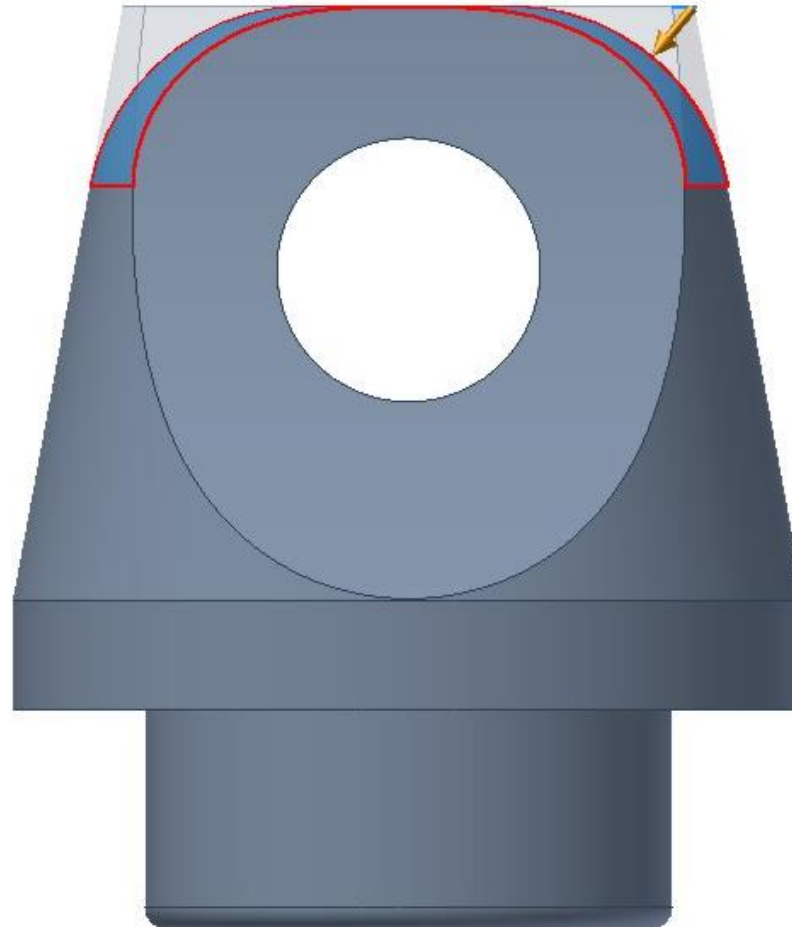
- Step1 On 3D Model Menu, Select Mirror command in Pattern Tab
- Step2 Select Features
- Step3 Select XY plane as a mirror plane
- Step4 Click OK



# MAKING A CAP

## PHASE 5 FILLET TOP EDGES

- Step 1 Select the both top edge of the cap
- Step 2 in Modify tab select fillet command
- Step3 set radius as 10 mm
- Step4 click OK

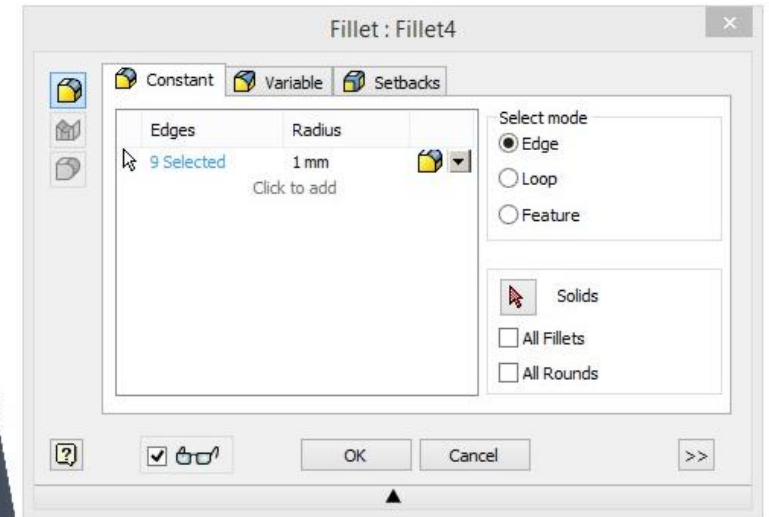
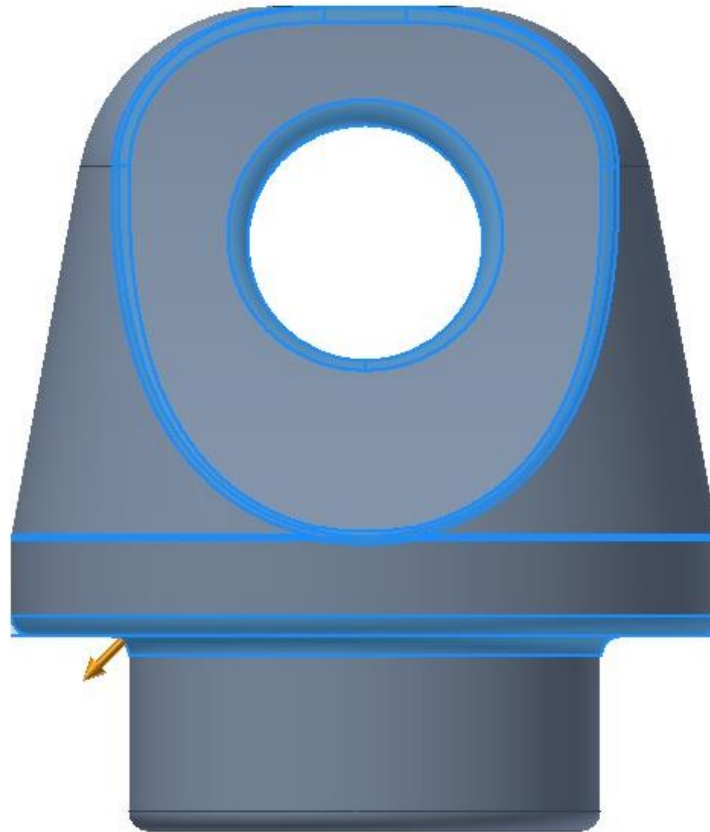


# MAKING A CAP

## PHASE 6

## FILLET OTHER SHARP EDGES

- Step 1 Select all sharp edges on the cap
- Step 2 in Modify tab select fillet command
- Step 3 set radius as 1 mm
- Step 4 click OK

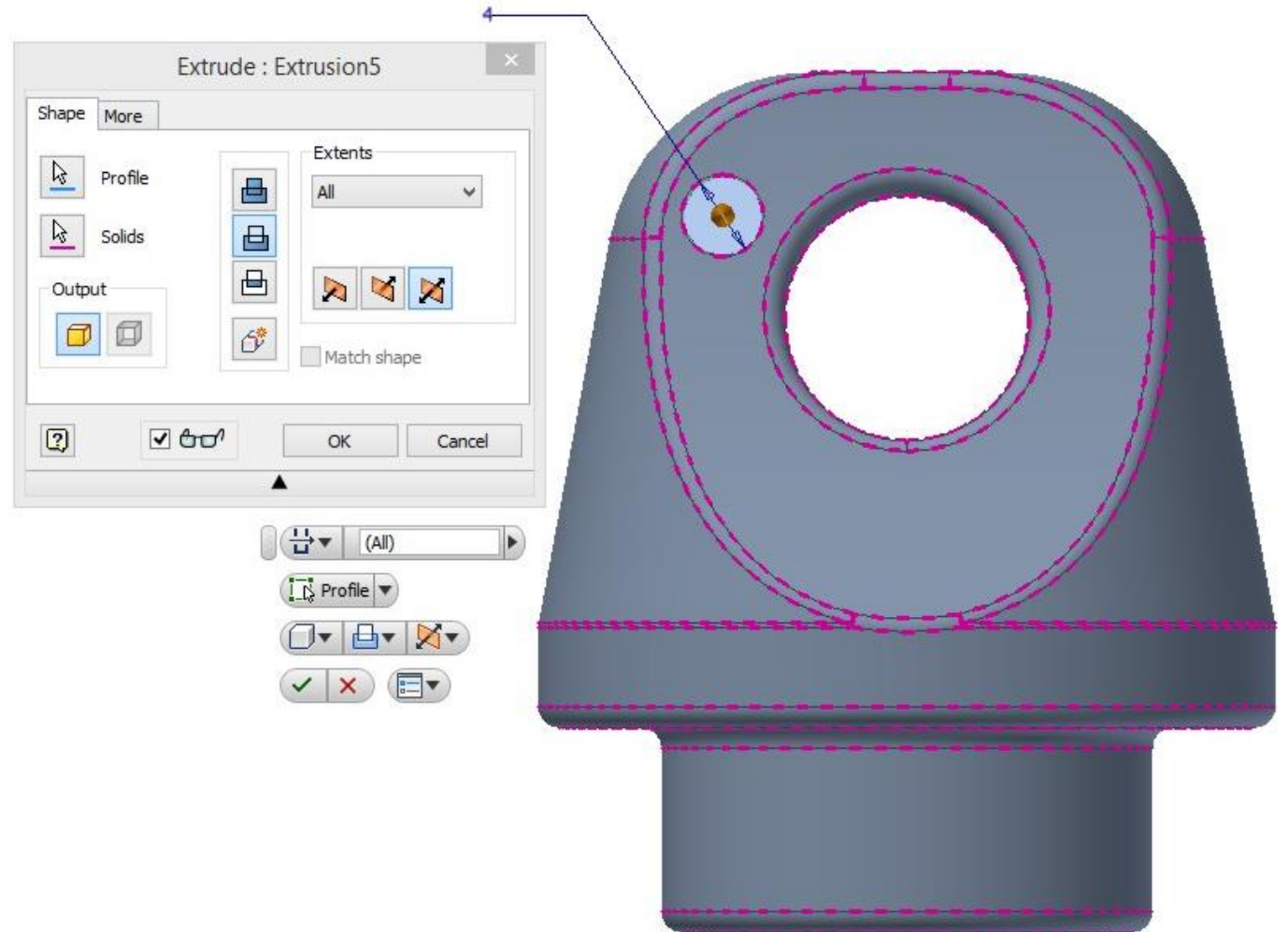


# MAKING A CAP

## PHASE 7

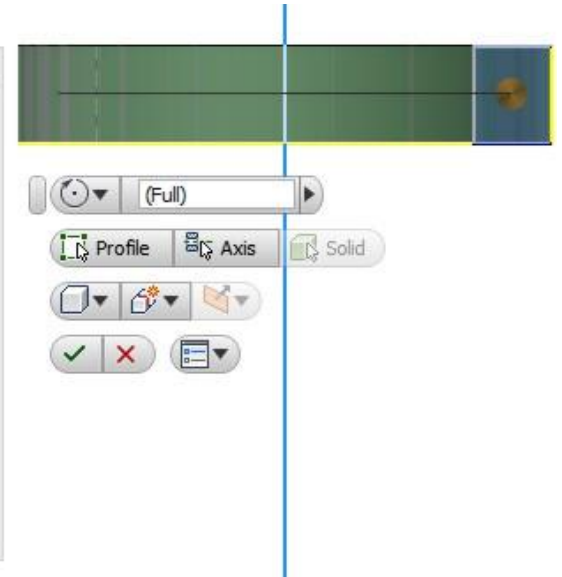
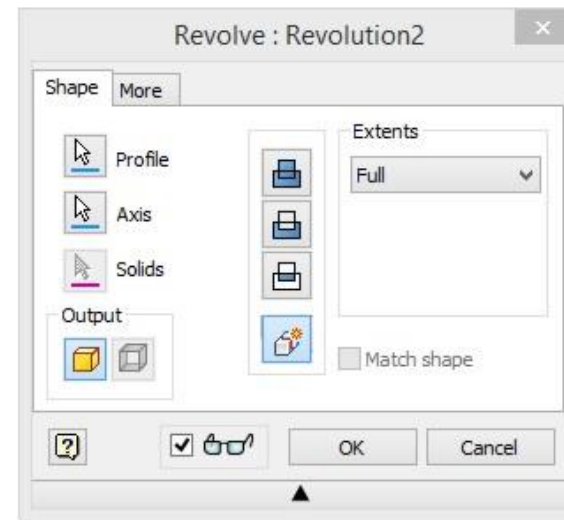
# MAKING A HOOKED HOLE

- Step1 On XY plane Draw a  $\varnothing 4$  mm circle
- Step2 Select profile sketch
- Step3 Extrude CUT + Both side
- Step4 Fillet Edge of the hole



# MAKING A WASHER

- **Step 1 Draw a profile sketch on XY Plane**
- **Step 2 Create a center line for rotation axis**
- **Step 3 Select Profile  
Select axis**
- **Step 4 Revolve FULL + New Solid**
- **Step5 Fillet all sharp edges**



# EMBOSS

- Step1 Create a new plane that offset 50 mm from XY plane then use it as the sketch plane
- Step2 Type in your name on the new plane
- Step3 On 3D Model Menu, Select Emboss command in Create Tab
- Step4 Select your name as profile
- Step5 select side for emboss and wrap to face

