This tutorial is written in a way that the user is familiar with 2D sketching and is a beginner with 3D modeling

AutoDesk Inventor

EXTRUDE:

In this session we shall discuss about the [EXTRUDE COMMAND] in AutoDesk Inventor. The process can be easily understood using the flow chart shown at [page 19]*

One of the most common and simplest method of creating a part in AutoDesk Inventor is to use extrude command for the given sketch along the Z direction. Generally it is essential for the 2D sketch to be in isometric view before the model is extruded as this can facilitate the user to analyze about model extrusion depth and direction (+Z or –Z direction)

**NOTE:**
To extrude a base feature (part) it is essential for the user to start from 2D drawing this can be achieved in 2 ways

- By selecting the 2D sketch which is drawn in AutoDesk Inventor (2D Sketch panel)
- By using an AutoCAD drawing and importing it into AutoDesk Inventor by using a wizard.

In this session we shall deal with 2D sketches drawn in AutoDesk Inventor for the process of extrusion.

**Problem definition:**
We shall take up the example of creating a simple circle and extruding it.

**Solution:**
The solution is divided into 2 stages

- Creation of 2D geometry
- Extruding the 2D profile to create a 3D geometry

**STAGE – 1:**
The circle is drawn in the following steps

Theoretical Approach

Step –1: Click start > Program Files > AutoDesk Inventor > AutoDesk Inventor Professional
Step – 2: Click File > New (Ctrl+N)* > select standard.ipt (NEW) > OK

Step – 3: 2D Sketch Panel > Center Point Circle > Origin (0,0) > Draw Arbitrary Circle

Step – 4: Click on Dimension (D)* > Click on the circle > Drag out to dimension > Double click the dimension to change its value.

Step – 5: Right Click Mouse on the graphics area > Finish Sketch > Right Click Mouse on the graphics area > Isometric view.

Graphical Approach

Step – 1
Step – 3:

Draw the Circle at Origin (0,0)

Select Circle Center Point Circle

Step – 4:

Select Circle Center Point Circle

Draw the Circle at Origin (0,0)
Step – 5:

Step – 6:
Step – 7:

![Image of a circle with a dimension dialog box showing 50 mm]

Step – 8:

![Image of a menu with various options including 'Finish Sketch']

AutoDesk Inventor Tutorials prepared by Sunith Babu. L
Step –9:

Step –10: (Model ready for Extrusion)
Stage –2:  
The process of Extrusion begins here

The Extrusion can be obtained in three ways

A) Press (select) Extrude in [Part Feature]

B) Press Hot Key (E) in the Key Board

C) Right Click on the graphics area and from the pop-up select
Create Feature > Extrude (only available above AI 7.0)*

Once you select the extrude command by any of the methods, a dialogue box appears as shown below

![Extrude Dialogue Box]

The extrude command has 2 Main parts

a) Shape
b) More

Shape: This can be sub divided into 4 main categories

a) Profile
b) Output
c) Operation type
d) Extents

The details will be discussed below
PROFILE:

The profile button [the arrow icon] helps in selecting the sketch to be extruded. The advantage with AutoDesk Inventor is that if the profile is single area the user need not select the area, instead AutoDesk Inventor selects and provides an immediate preview of the solid model as shown in the figure below. Irrespective of the profile shape this feature can be achieved.

Note:

Before you can press the profile button the preview is immediately shown

On the other hand if the profile is not an area or has multiple areas, then you need to select the profile of interest and carry out the process of extrusion as shown in the figure below.
NOTE:
You can select the profile only after depressing the profile button and it is the same case when done for an open loop sketch, which results a surface model.

TIPS:
In case you select the wrong profile, you can select the profile button once again and select the area of interest.

OUTPUT:
This is a preview tool to help the user know whether the model extruded is a Solid or Surface Model. As explained above, the output can be explained in two ways:

a) Solid Model
b) Surface Model

Solid model:
The output will show the icon solid depressed when the profile has single area or when you select a particular area of interest, which is closed as shown below.
**Surface Model:**

The output will show the icon Surface depressed when the profile has open loop as shown below.

![Surface Model Diagram]

**OPERATION TYPE:**

This is divided into 3 types.

a) **Join** (Extrude)
b) **Cut** (Cutout)
c) **Intersect** (A\(\cap\)B) \{Model A intersect Model B\}

**Note:**

- During the process of extrusion, the join button is by default selected and the part created is called as base feature meanwhile the other buttons are grayed (Cut and Intersect).
- The green frame [preview] represents material addition and red frame material deletion.
JOIN: This is an additive process where the material is added to the profile selected.

CUT: This is a subtractive process where the material is removed from an extruded model.

INTERSECT: This is an Intersecting process where the material is removed from the intersection of two objects chosen or area chosen.

The above types are clearly shown in the following figures:

JOIN: [PART 1]

![Join Process Example](image1)

CUT: [PART 2]

![Cut Process Example](image2)
The cut process is also accompanied by flipping of direction to cut. This will be dealt separately when we deal with [EXTENT]

**INTERSECT:**

![Intersection Process](image)

When you apply the intersection process A – B the resultant object is shown below

![Resultant Object](image)

**EXTENTS:**

The Extents determine the distance by which the extrusion process will be carried out by defining the direction. This can be achieved by knowing 3 factors associated with it.

- a) Termination criteria
- b) Distance criteria
- c) Direction criteria

![Extents Control Panel](image)
TERMINATION CRITERIA: This determines based on what criteria the model will be extruded. The basic criteria are as follows

- a) Distance
- b) To Next
- c) To
- d) From To
- e) All

Note: During the initial process of extrusion (base feature creation) only [Distance, To, From To] are activated while others are activated when sufficient conditions are achieved.

All the parameters are explained in a simple way

- **Distance:** If this is selected the sketch will be extruded based on the values entered in the Distance Dialogue Box.
- **To Next:** If this is selected the sketch will be extruded if it immediately reaches an Plane or Face
- **To:** If this is selected the sketch will be extruded to the Plane or Face you select.
- **From To:** As the name says, If this is selected the extrusion will start at the Plane or Face [First selected] and end at another Plane or Face [Second selected].
- **All:** If this is selected the sketch will be extruded in the direction specified.

DISTANCE: If distance is the Termination Criteria you must carry out any one of the following process

The preview can be dragged to get different views of the model without typing the values.

- a) Enter the Distance value by which the sketch will be extruded

![Distance Dialogue Box](Image)

- b) Measure the distance [select 2 points and measure]
c) Select Dimension of previously used

Note:

If the display dimension appears in red then an immediate correction is required for example

a) 45 mm typed as 45mmm then:
b) Two decimal values are typed by mistake

Once the correct distance is typed the preview of the model is show immediately showing how the extrusion process will be carried out. To alter the direction separate tool is used which will be discussed as show below.

**DIRECTION:**

The direction of extrusion of a model can be easily understood by using the direction buttons

Selecting the First two options can lead to changing direction of the model or selecting the last leads to flipping of the model in both [\(+Z\) and \(\text{–}Z\)] direction.

The different process are shown below
Note:

The Final Model created with dimension of the 2D profile as 50mm and Extruded to a height of 10mm.

MORE:

This is used to provide a tapering effect to the model

This can be achieved by 3 ways

a) By entering the Taper angle
b) By measuring
c) By selecting a value which was previously used
Note:

For negative taper

For positive taper
2D Sketch [XY plane]

Right Click
And select
Finish Sketch

Right Click
And select
Isometric View

CLOSED LOOP
OPEN LOOP

SOLID / SURFACE MODEL
SURFACE MODEL [ONLY]

[GENERALLY]
The solid and surface model can be created from 3 different ways.
1) Extrude in [+] Z direction
2) Extrude in [-] Z direction
3) Extrude in both [+ and -] direction
4) Taper Effect can be given if required
CONCLUSION:

To make an extruded feature, you need to make a 2D sketch on a plane [XY axis] and extrude the sketch perpendicular to the plane in which the profile is drawn.

- If you extrude an closed loop sketch, you can extrude an solid model or surface model
- If you extrude an open loop sketch, you can extrude an surface model only
- Green preview [during extrusion join] represents material addition
- Red preview [during extrusion cut] represents material removal
- Intersection process can also be achieved during extrusion of multiple solids
- Desired distance of extrusion can be achieved by using
  a) Distance
  b) Termination
  c) Direction