

# AUTODESK INVENTOR

## Getting to know Inventor



### Key Terms:





- **Inventor**
- **Task Bar**
- **Standard IPT**
- **2D Sketch**
- **Dimension**
- **Extrude**
- **Standard IAM**
- **Constrain**


### NOTES:

### Finding Inventor:

- Go to the start menu
  - All programs
    - Autodesk
      - Inventor
      - Once Inventor is open go to the bottom of your screen to find the “Task Bar”
      - Right click on the Inventor Logo
      - And click Pin to Task Bar
      - You can now find Inventor here when you log in to this computer


### Starting to work in Inventor:

- Once Inventor has opened (First time click Start Working)
  - File
    - New
      - Standard IPT (Part File)  **Standard.ipt**
        - You will need to create a new file for each “Part” you make even if they go together in the end
      - Create
    - Start a 2D Sketch   
**Start 2D Sketch** ▾
      - Pick a plane by clicking on it (common XY plane)
        - Create the part using shapes/line options from the top of the screen  
  
**Line**   **Circle**   **Arc**   **Rectangle**
      - Dimensions   
**Dimension**
        - Click on one side of shape and then on the other side then move your mouse out of the shape to add the dimension.
        - You can always click on the dimension to change it.

- In order to delete a dimension you need to finish sketch, right click on the number and delete.
- Once done drawing shape 



Finish Sketch

- Extrude to add depth

- Change dimension 

Extrude

- Adding a Hole


- Click on the Start 2D Sketch Button 
  - Select the plane you want to add the hole to
  - Create shape
    - Finish Sketch
  - Extrude (Change the direction of the extrude if needed)
    - Cut 

Start 2D Sketch

- Changing something that is already made
  - Left model box
    - Sketch 1 (Find the part)
      - Right Click
        - Edit Sketch

Finish 2D Sketch

### Using your view window:

- Use the view cube to navigate through your workspace
- Click on the magnified glass icon to see the whole shape while working 
- Scroll wheel to zoom in and out
- Click scroll wheel and drag to pan through the space



### How to create an Assembly:

- New
  - Standard IAM
    - Place
      - Open anchor file (Whichever part is the main or largest piece)
        - Click to place
          - Right click & OK
      - Constrain button (ICON)
        - Click on edge or face of one and repeat to the other
        - Apply \*
          - Repeat

\*Constrain = Glue

1<sup>st</sup>: Mate (Puts two parts together)

2<sup>nd</sup>: Flush (Makes them even)