Part Modeling Intro

Using Autodesk Inventor
Basic Dimensions
General things to consider
Follow specific instruction for the class

1. **INITIALIZATION**
- Start Inventor
- Verify project
  - Selected Project
  - Workspace
  - Libraries
  - Folder Options

a. Startup
b. Application “Options”
Application Options:

- Options - General

Note:
- other possibilities.
- Carefully adjust (only if needed)

C. Options - General
Application Options:

- **Part**
- No new sketch

Note:
- Optional setting
- You may prefer another choice
- Other possibilities

d. Options - Part
e. Options - Sketch

Note:
- other possibilities
- Selectively change
- Remember what you change
• Application Options:
  - Colors
    - Millennium

Note: Other possibilities

f. Options - Colors
2. NEW PART MODEL
a. New Part Model
• Reference Planes
  ✓ Select: <Ctrl>+<lmb>
  ✓ Visibility

• View
  ✓ Home
  ✓ Other
    ✓ Navigation
    ✓ Pan
    ✓ Zoom
    ✓ Orbit
    ✓ View Face

b. Adjust Display
e. Save

- Workspace
- File name: FirstDemo.ipt
- Save as type: Autodesk Inventor Parts (*.ipt)
3. BASE FEATURE
Pick: XY Plane
Pick: Create Sketch

a. Sketch Plane
- Front
- Sketch
  - Line
  - Three Point Arc

b. Sketch
• Sketch

☑ General Dimension

Note: Fully Constrained

c. Dimension
- **<RMB> Show All Constraints**
- **Sketch**
  - ✓ Tangent
  - ✓ Vertical
  - ✓ Parallel
  - ✓ Perpendicular
  - ✓ Coincident

Note: Fully Constrained

d. Constraints
e. Finish Sketch
- Model
  - Extrude
  - Profile
  - Join
  - Direction
  - Distance
  - OK

f. Extrude
4. SECOND FEATURE
Pick: Front Face
Pick: Create Sketch

a. Initialize
b. Sketch

- Center Point Circle
  - Center Point
  - Diameter:
• Sketch
  ✔ General Dimension

Note: 2 dimensions needed

c. Dimension
- `<RMB>` Show all Constraints
- Sketch:
  - ✔ Equal
  - ✔ Horizontal

**d. Constraint**
- Home View
- Finish Sketch
- Model
  - Extrude
    - Join
    - Other-previous

e. Extrude
5. THIRD FEATURE
• Home View
• New Sketch

a. Initialize
- Front
- Sketch
  - Circle
    - Center point
    - Diameter .75
Important: Project Geometry allows the edges to be used in the sketch. This will be used to make new features concentric to the edges.
Important: Project Geometry allows the edges to be used in the sketch. This will be used to make new features concentric to the edges.
- No additional dimensions required

**e. Dimension**
• <RMB> Show All Constraints
• Sketch
  - Concentric
  - Equal

f. Constrain
- Home View
- Finish Sketch
- Model
  - Extrude
  - Cut
  - Through All
- g. Extrude
5. FINISH THE PART
a. Save the part