EGR 110
Engineering Graphics
File: N110IL7

Inventor Lecture #7

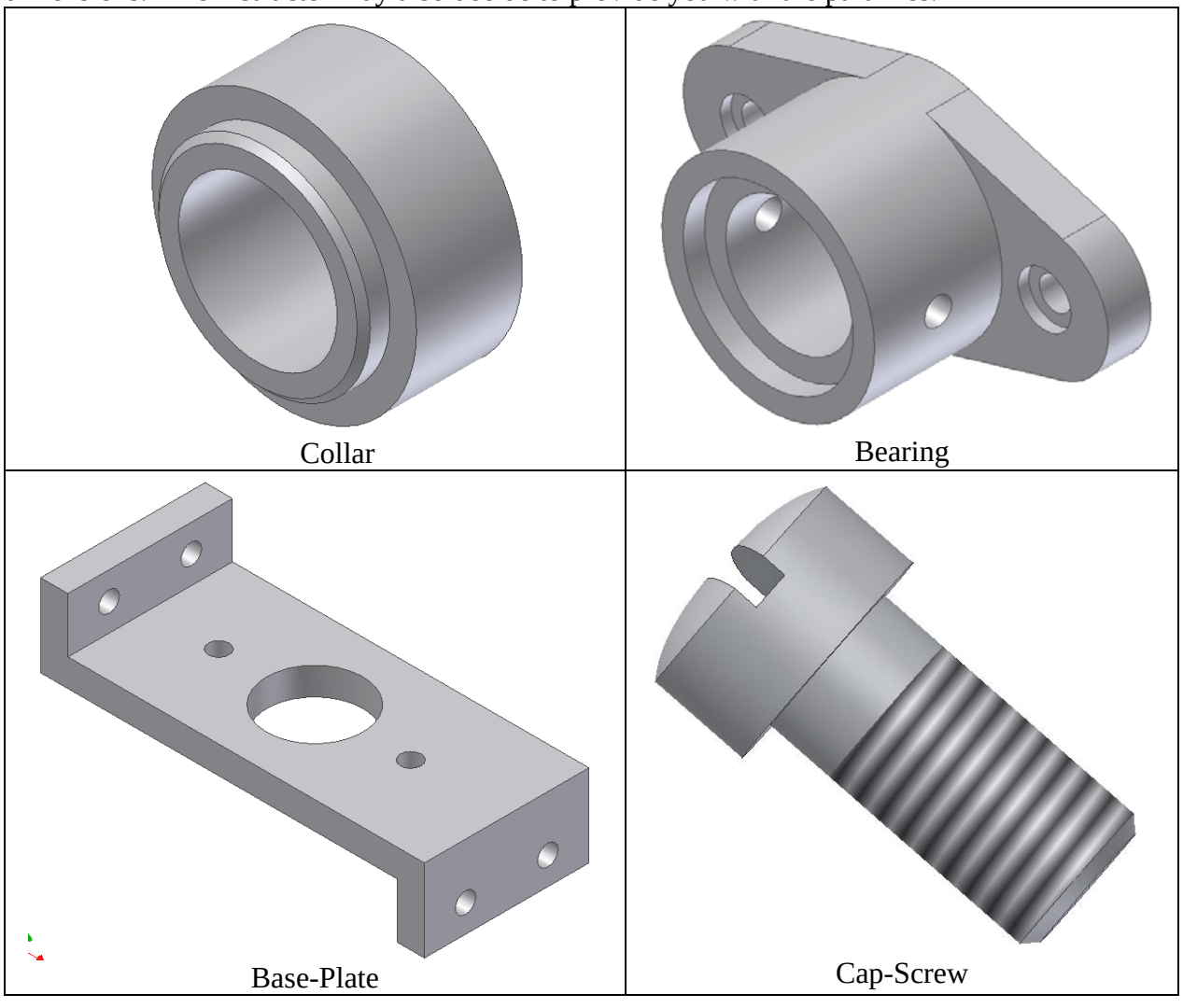
Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2009 by Randy Shih:
Chapter 12 – Assembly Modeling

Lecture Outline:

Assembly Modeling (Chapter 12)
Read the Introduction and section on Assembly Modeling Methodologies in the textbook.
As we will be creating assemblies consisting of only a few parts, we will use the "***Bottom Up***" approach as described in the text, where each of the parts are created first and then are pulled together in an assembly.
The text has an excellent example showing how to create an assembly consisting of 4 parts. This example will be used in class today.

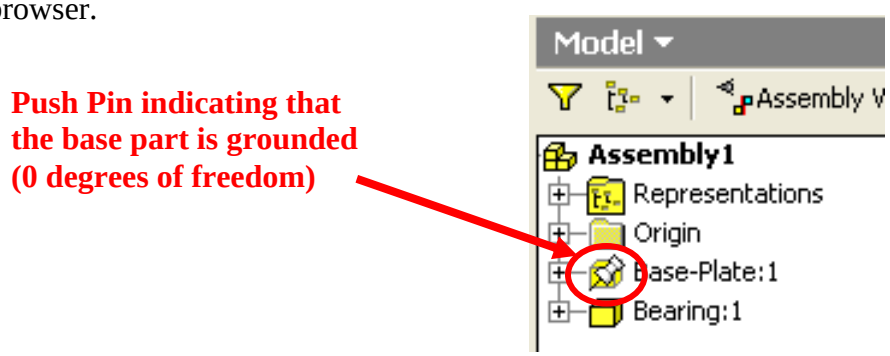
1) **Create the Parts**
   Create the four parts required for the assembly as shown below. Refer to the text for the dimensions. The instructor may also decide to provide you with the part files.



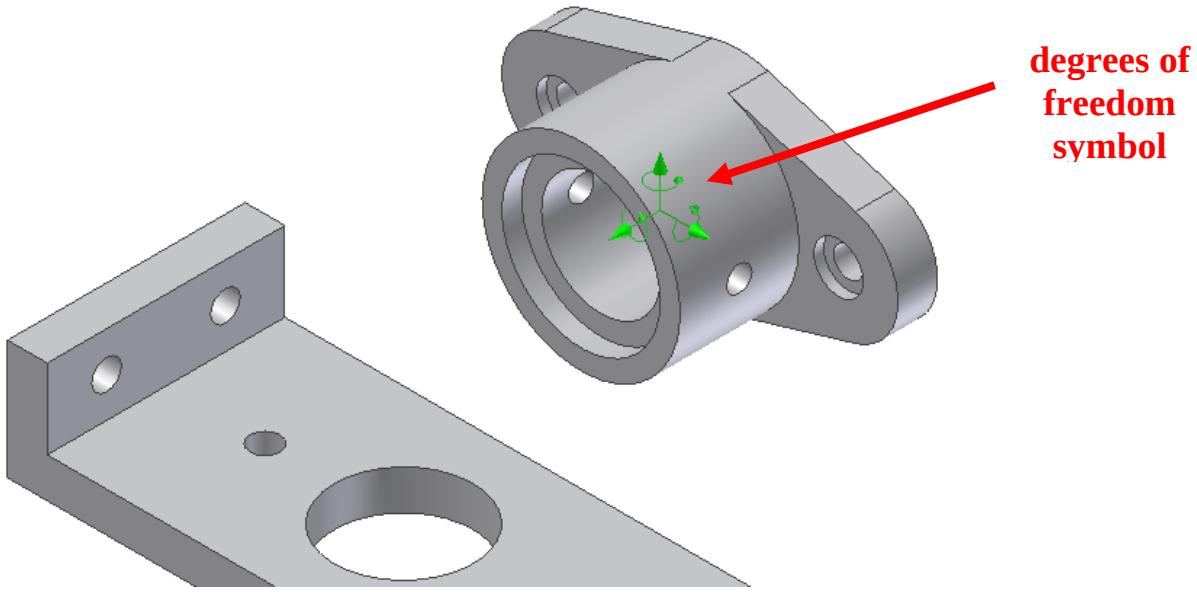
2) **Create a new Assembly**
   Select **File – New – Standard(in).iam**
3) **Place the Base Part**
   Each assembly will have a base part. It is important to carefully select which part to serve as the base, although it is often an obvious choice. Other parts will later be added to the assembly. Note that the base part will not be able to move (no degrees of freedom) whereas other parts can be moved in several ways (to be described shortly).
   Select **Place Component** from the Assembly Panel. Browse and pick the desired part (**Base-Plate**) to serve as a base (right-click and pick Done after inserting it as Inventor will otherwise let you insert multiple copies).
4) **Placing the Second Component**
   Select **Place Component** again and select the **Bearing** part. Place it somewhere near the base part. In general, components have 6 degrees of freedom, meaning that they can be moved in the assembly as follows:
   - Moved (translated) in the x, y, or z direction
   - Rotated about the x, y, or z axis

Degrees of Freedom

Recall that the base plate cannot be moved (has zero degrees of freedom) as indicated by the push pin in the browser.



A new component that has been inserted will initially have all 6 degrees of freedom.
Select **View – Degrees of Freedom** to show the degrees of freedom symbol on parts.

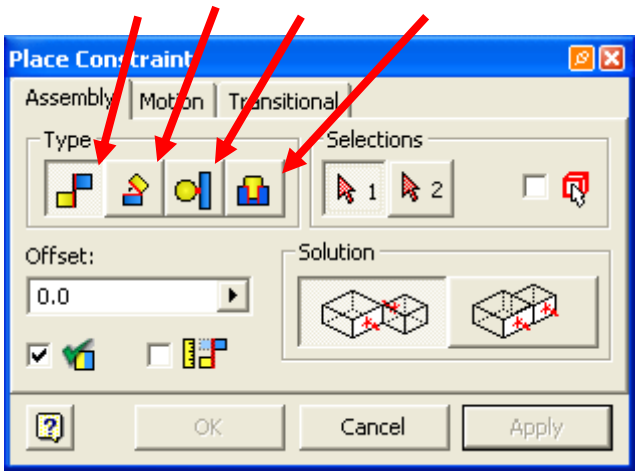


Assembly Constraints

Parts are put together in assemblies by placing **assembly constraints** on the parts. Each time a constraint is added, one or more degrees of freedom are eliminated.

Select **Constraint** from the Assembly Panel to place an assembly constraint. The Place Constraint dialogue box will appear. Note that four types of constraints are available:

Mate Angle Tangent Insert



Mate – Constrains features to be face-to-face or adjacent to one another. They can be in the same direction (flush) or in the opposite direction and they can be separated by an ***offset***.
Angle – Constrains features to be separated by a specified angle.
Tangent – Constrains features such as faces, planes, cylinders, spheres, and cones to make contact at a point of tangency.
Insert – Constrains circular features to share the same axis (become co-axial). The distance between the selected circular faces can be zero (the default) or given an ***offset*** value.

5) Applying Constraints to the Second Component

Apply the **Mate** constraint.

- Select the bottom face of the Bearing and the top face of the Base-Plate.
- Note that now only two degrees of freedom are left and the Bearing can be slid along the surface of the Base-Plate.
- Try repeating the last operation (or edit it in the browser) and using an offset. Return the offset to 0.0 when finished.

Apply the **Mate** constraint again.

- Select the center line of a small hole in the Bearing and the centerline of the corresponding small hole in the Base-Plate.
- Note that now only one degree of freedom is left (rotation about one axis) and that you can drag the part to rotate it.
- One more constraint can be used to fully constrain the part.

Apply the **Mate** constraint again.

- Select the center line of the other small hole (or the large hole) in the Bearing and the centerline of the corresponding hole in the Base-Plate. Note that the text uses a slightly different method of mating work planes. There are often many ways to apply constraints.
- Note that now the Bearing has zero degrees of freedom and cannot be moved.

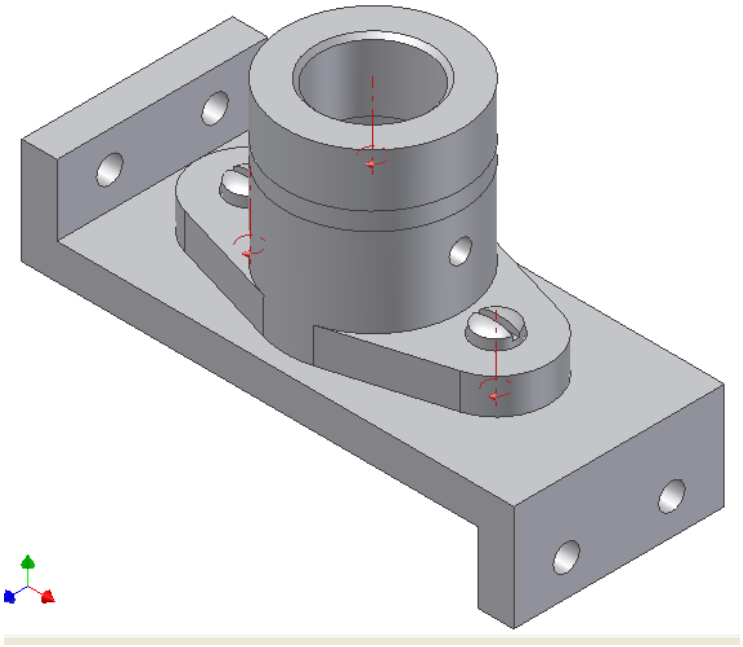
- **Placing the Third Component**

Select **Place Component** again and select the **Collar** part. Place it somewhere near the assembly.

Apply the **Insert** constraint.

- Select two circular surfaces that are coaxial and will also mate (be touching).
- Note that the collar now has one degree of freedom (rotation about its axis).
- It is not necessary to eliminate the last degree of freedom (especially since this part would naturally be free to rotate if you built it.)

- **Placing the Cap-Screws**
  Select **Place Component** again and select the **Cap-Screw** part. Place two screws somewhere near the assembly.
  Apply the **Insert** constraint and insert the two screws into the desired holes (see final assembly below). Note that the screws are still free to rotate.

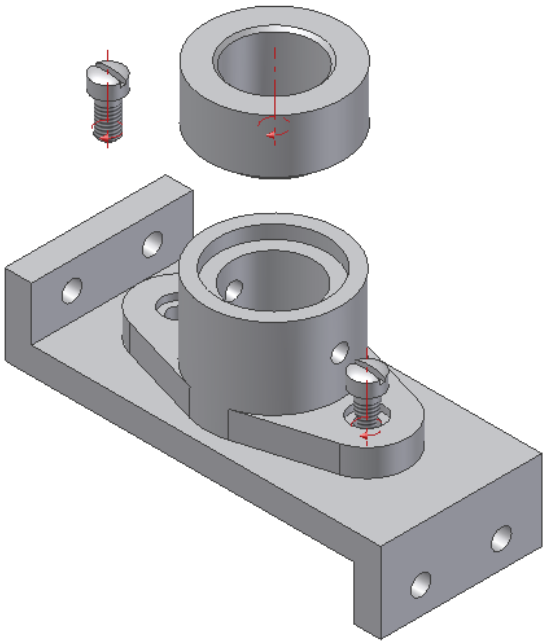


Exploded View Assemblies

Exploded assemblies are commonly used to illustrate how parts fit together in instructions, promotional literature, presentations, etc. Try using the following two commands to create the image shown below.

- Move Component
- Rotate Component

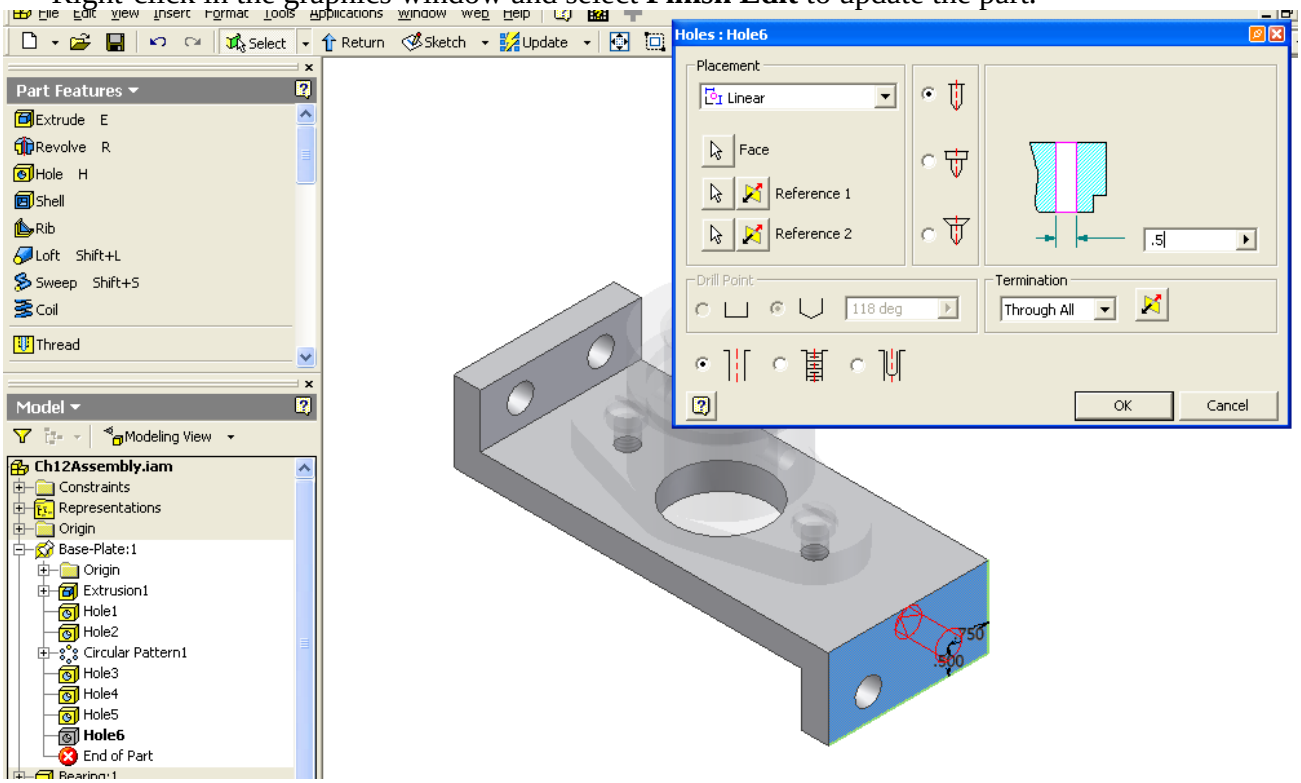
Select the Update tool on the standard toolbar to return the components to their original position.



Editing Parts used in Assemblies

Inventor will allow you to change designs at any level, so you can easily modify a part while in an assembly. Try the following:

- Right-click on Base-Plate in the browser and select **Edit**.
- Right-click on one of the small holes in the browser (the holes not used with the Cap-Screws) and select **Edit Feature**.
- Change the diameter of the hole from 3/8 to ½. Repeat for the other three small holes.
- Right-click in the graphics window and select **Finish Edit** to update the part.



Creating an Assembly Drawing

Create an assembly drawing (just like you would for a part).

- Select **File – New – Standard.idw**
- Change the sheet size to **A-size (portrait).**
- If a base view is automatically inserted, delete it and select Base View to insert a **Top Right Isometric** view for the assembly just created.
- Change the scale to 0.75

Adding a Parts List

- Pick the arrow next to **Drawing Views Panel** and select **Drawing Annotation Panel**.
- Select **Parts List** then select the desired view (we only inserted the base view so select it.)
- The Parts List – Item Numbering dialog box will appear. Select OK to use the default settings (since we have no sub-assemblies).
- Right-click on the Parts List and select Edit Parts List.
- Try adding a column for **Material** and removing the column for **Description**. Note that the material will appear as Default for each part.

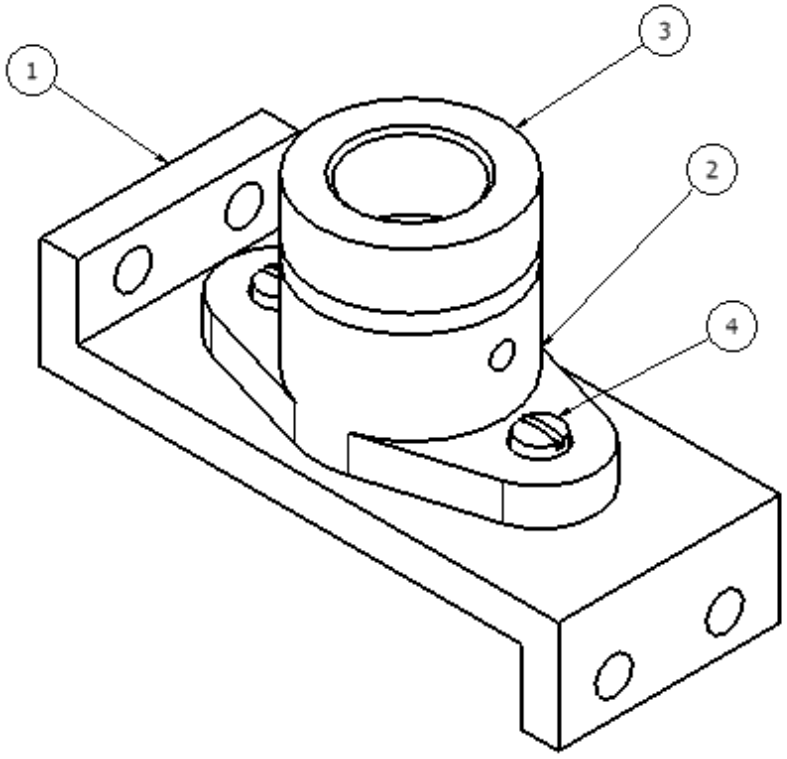
- Switch back to the assembly drawing.
- Right-click on Base-Plate in the browser and select **Edit**.
- Right-click on Base-Place again in the browser and select **iProperties**.
- Click on the **Physical** tab.
- Change the material type to **Steel, Mild** and then select **OK**.
- Right-click in the graphics area and select **Finish Edit**.
- Repeat for the remaining parts (only for one of the screws).
- Switch back to the drawing file, right-click on the parts list, and select Update.

Adding Balloons to the Parts List

Balloons are numbered labels inside circles with leaders that identify a part. The numbers are generated automatically from the Parts List.

From the Drawing Annotation Panel select **Balloon**

Pick a part, pick a point to place the balloon, and then right-click and pick **Continue** to pick the next part, etc. Right-click and pick **Done** when finished.



Parts List			
ITEM	QTY	PART NUMBER	MATERIAL
1	1	Base-Plate	Steel, Mild
2	1	Bearing	Steel, Mild
3	1	Collar	Steel, Mild
4	2	Cap-Screw	Steel, Mild