

### Inventor Lecture #4

#### Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2011 by Randy Shih:

Chapter 3 – Grid and Snap Settings, Holes

Chapter 4 – Chamfer and Fillet

Chapter 10 – Revolved Features, Circular Patterns

#### Lecture Outline:

##### GRID and SNAP Intervals Setup (Chapter 3)

AutoCAD users may be familiar with setting up grids and enabling the “snap to grid” feature. Snapping to a grid is possible, but is not recommended in Inventor. However, having a grid in the background might be useful for approximating the sizes of objects as they are drawn.

SNAP and GRID options can be set in Inventor using Tools – Document Settings – Sketch.

Note that the “Snap Spacing” actually sets the grid spacing.

Try various settings. Try zooming in and out and note how the grid spacing changes.

Note in the following screen capture:

- 8 minor divisions for every major
- Minor Grid Spacing = Snap Spacing \* Grid Display  
= (0.125 in/snap)(2 snaps/minor grid) = 0.25 in/minor grid

Be sure to letter your name neatly (between the guidelines provided) on all worksheets.

The image shows a screenshot of the Autodesk Inventor software interface. On the left, a grid is displayed with a 2.000 inch wide section highlighted. Below this section, a dimension line indicates "8 minors at 0.25 each". On the right, the "Part1 Document Settings" dialog box is open, showing the "Sketch" tab. Under "2D Sketch", the "Snap Spacing" is set to 0.125 in for both X and Y. The "Grid Display" is set to 2 snaps per minor, and "Major every" is set to 8 minor lines. Under "3D Sketch", the "Auto-Bend Radius" is set to 0.125 in.

## Features in Inventor

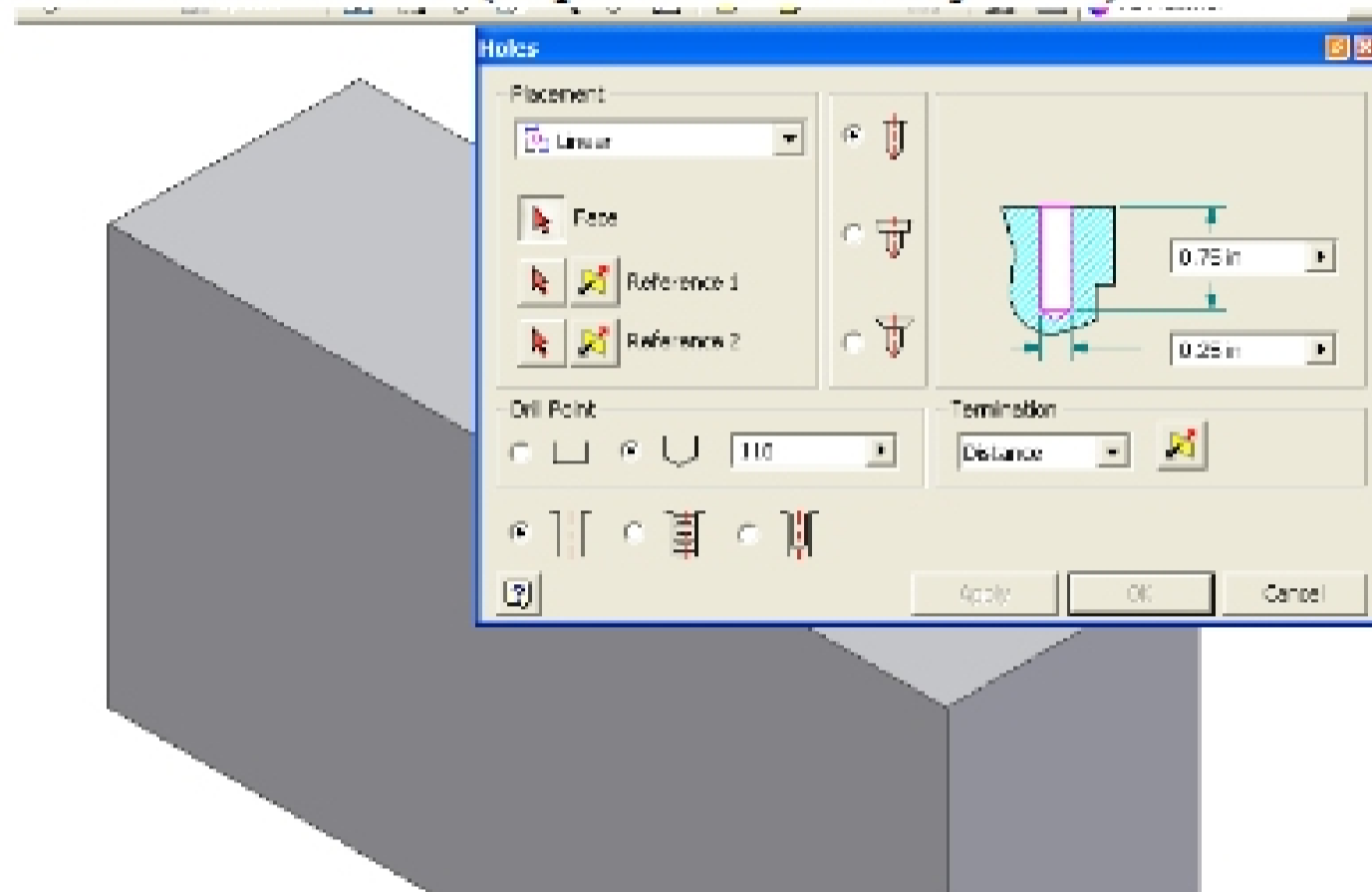
Inventor includes two types of geometric features:

- 1) Sketched features (including extrusions, revolutions, and sweeps)
- 2) Placed features (including holes, fillets, chamfers, and shells)

So far we have only used a sketched feature (extrusion) to create solid models

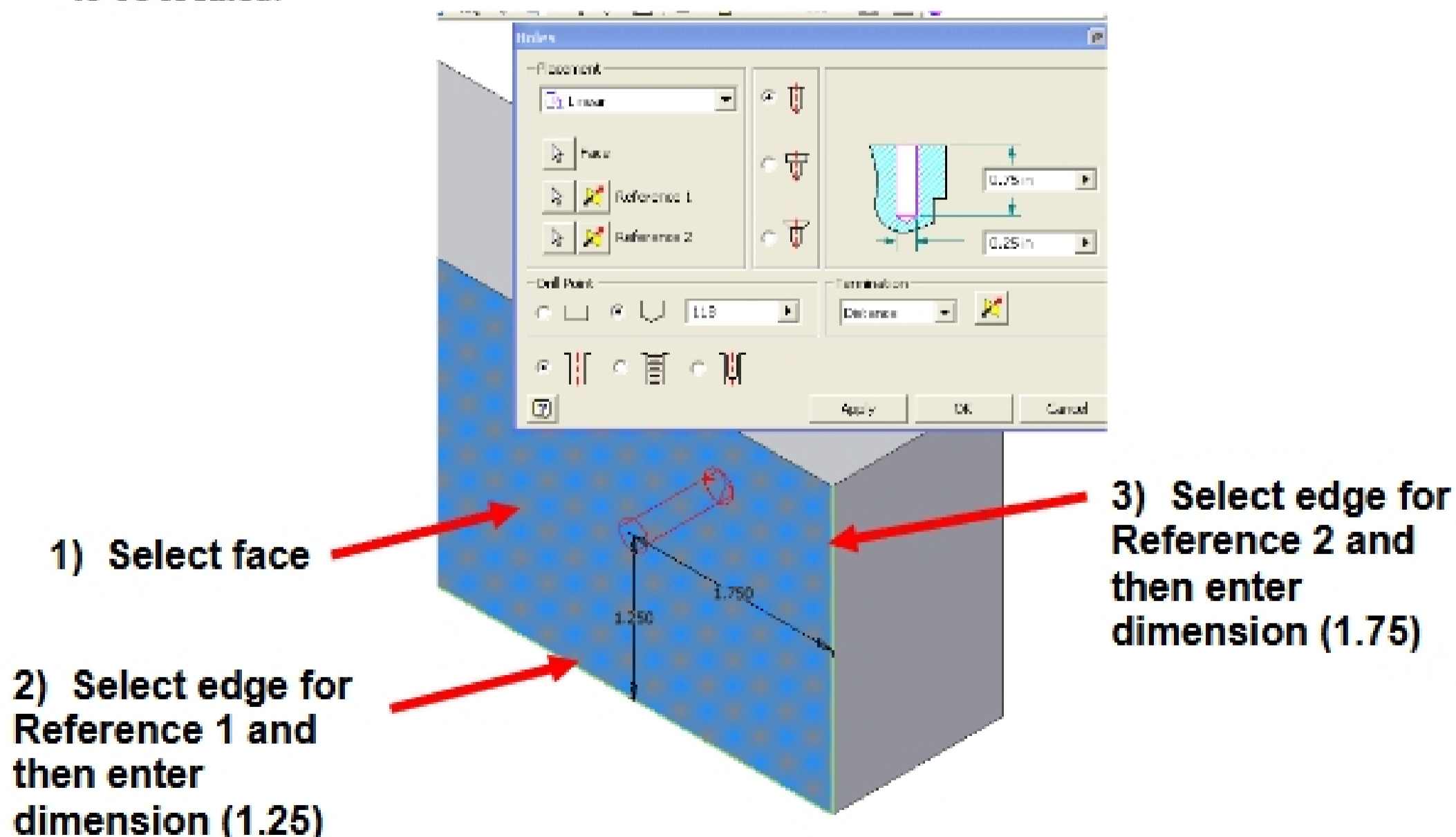
### Holes (Chapter 3)

Draw a simple solid block (extrude a rectangle) and switch to an isometric view. Select Hole from the Part Features Menu (or press H on the keyboard).



Note the three arrows on the **Hole** dialog box above. These represent the first three steps in creating a hole.

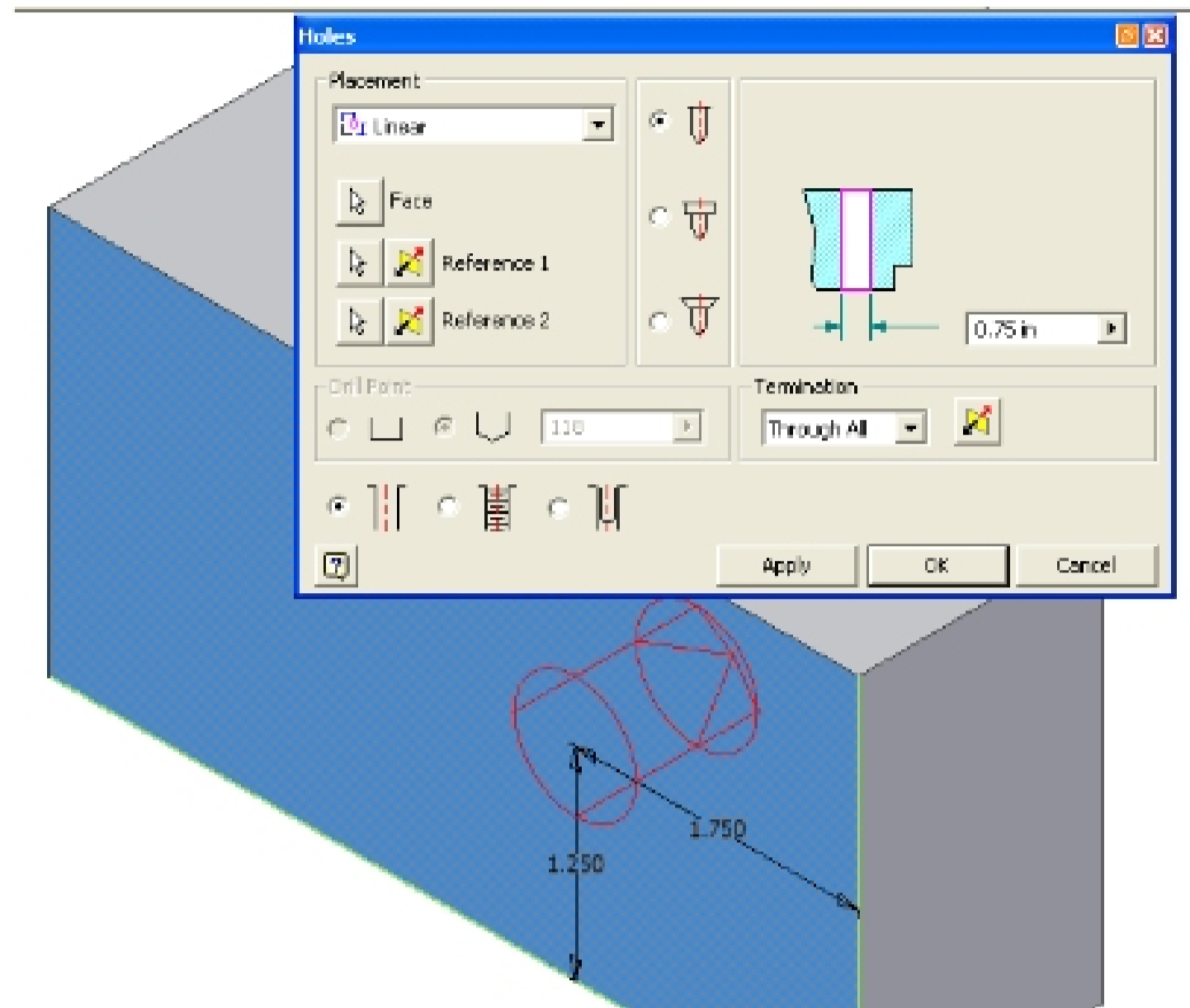
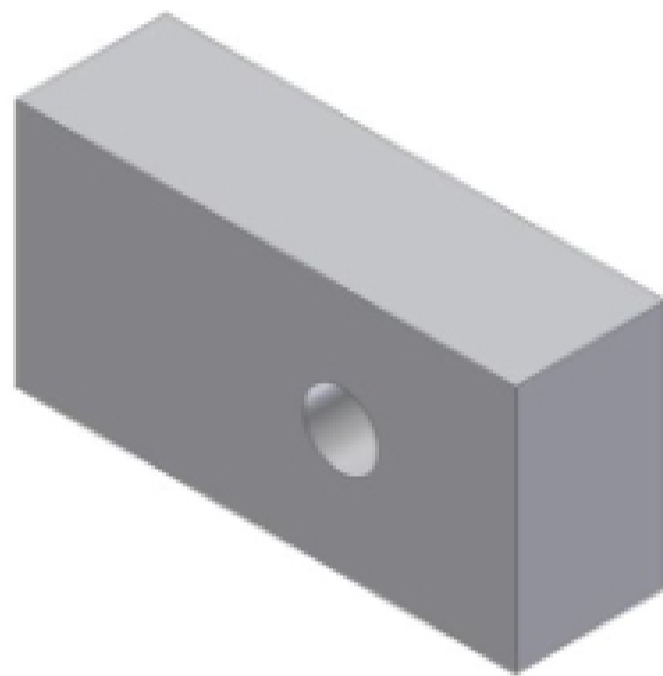
- 1) Face – Select the face where the hole is to appear
- 2) Reference 1 – Select an edge on the object and specify how far from this edge the hole is to be located (a dimension box will appear after you select the edge).
- 3) Reference 2 – Select another edge on the object and specify how far from this edge the hole is to be located.



As shown to the right:

- Change the **Drill Size** (diameter) - 0.75 shown
- Change the **Termination** to *Through All*
- Click **OK**.

The resulting solid is shown below:



### Editing a Placed Feature

- Note that Hole1 now appears in the Browser. Right click on **Hole1** and select **Edit Feature**.
- Try editing various features, including:
- Changing the diameter of the hole
- Changing the location of the hole
- Adding threads to the hole
- Making the hole countersunk or counterbored
- Changing the Through All option to Distance and set the hole depth.

