Lesson 10
Hole Features

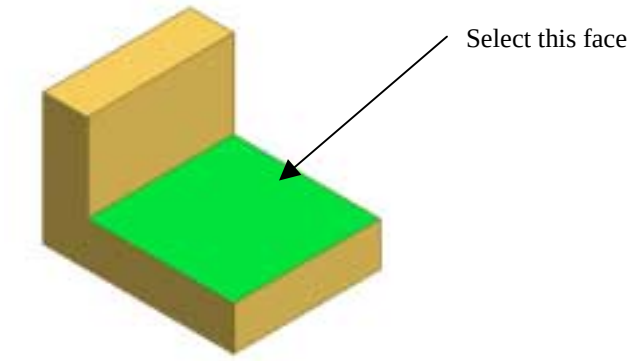
Learning Objectives

Upon completion of this lesson, the user will be comfortable using the Hole dialog box as well as placing and locating points to be used for the hole centers.

In this lesson, we will add a hole to the 'L' bracket created in Lesson 1.

Open the file we saved as 'lesson1.ipt.'

First, we must select the Sketch Plane where we will be locating the hole. Activate the 'Select' button in the Command Bar.

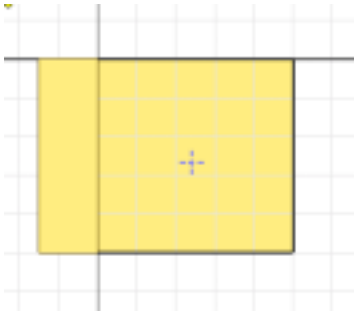


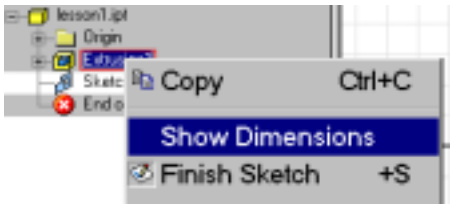
Place the mouse over the face indicated in the figure and left-click the mouse. The face will highlight and change color to indicate it is the active sketch plane.

Inventor will automatically turn on the Sketch toolbar.

Select the Hole Point tool from the Sketch toolbar and place the hole point anywhere on the desired plane. Right-click and select 'Done'. Don't be concerned about locating it exactly; remember we will be adding dimensions to constrain it.

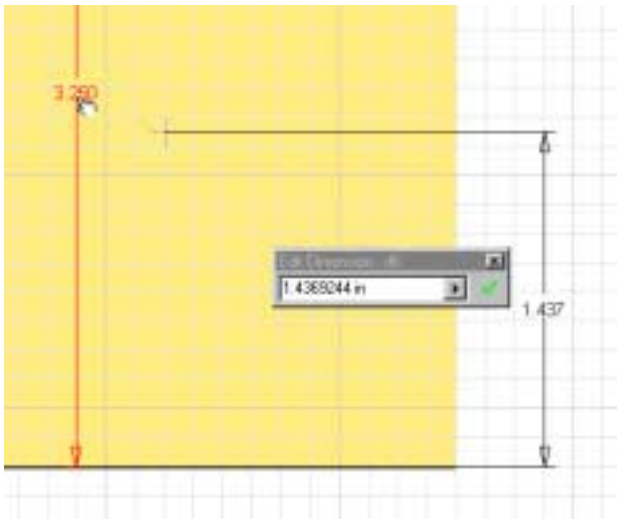
TIP: Use the Look At tool to orient our sketch plane to a PLAN view.





We can use the existing dimensions to help us locate our hole. Highlight the Extrusion in the browser. Right click and select 'Show Dimensions'.

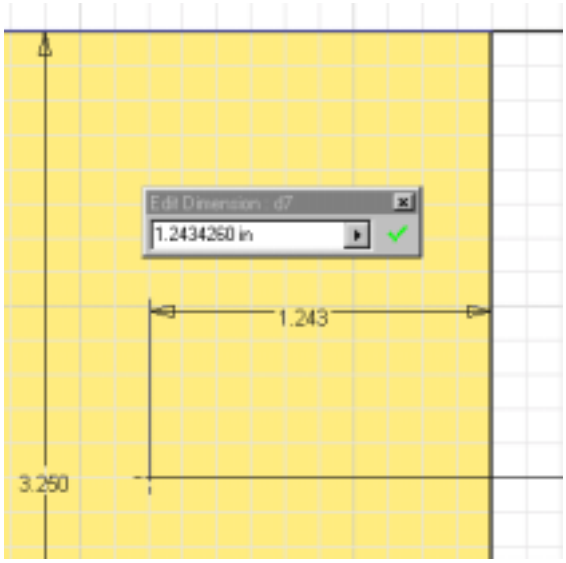
Use the General Dimension tool.



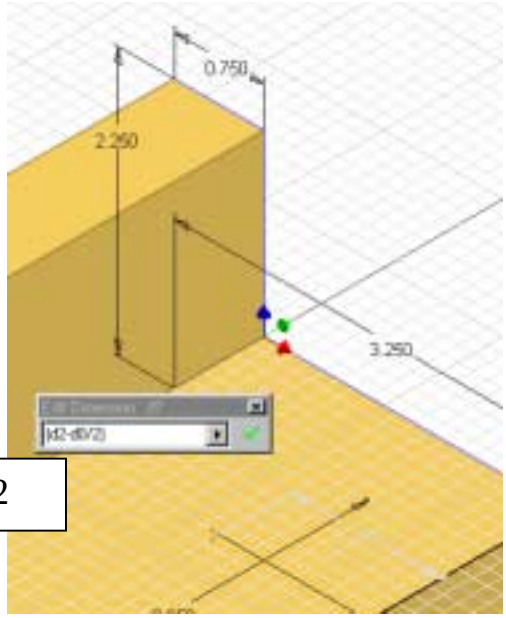
To place the vertical dimension, select the hole point and then select the bottom edge of the model by left-picking the mouse. The selected side will highlight in red. Next move the mouse away from the part and place the dimension.

Modify the dimension by left-picking on it.
Pick inside the edit box and then pick the 3.25 vertical dimension displayed.

The dimension id appears in the edit box. Add /2 to locate the hole in the middle of the part.
Press the green check mark.

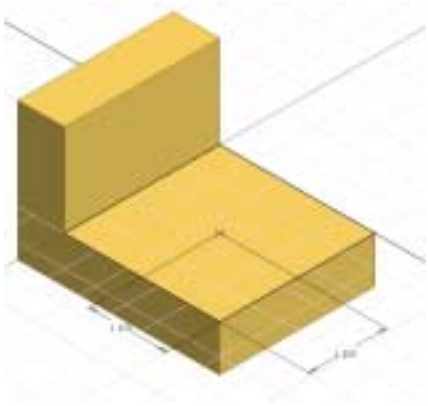


Apply the horizontal dimension by selecting the point and the right side edge.
We will use the existing dimensions again.
Select the 3.25 horizontal dimension, then type a -, then select the .75 horizontal dimension then divide by 2.

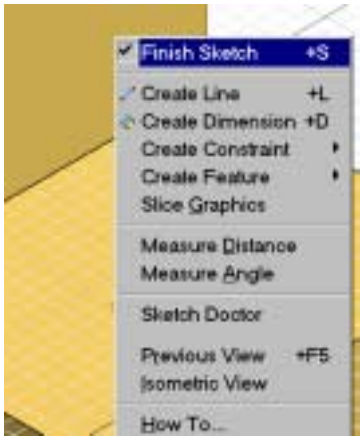


TIP: By using the existing dimensions of the part, we ensure that the hole will always be located at the midpoints; thus capturing design intent.

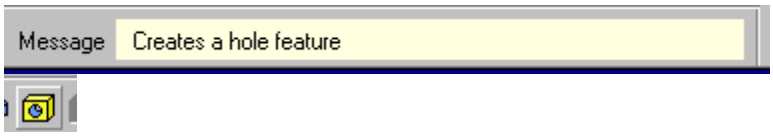
Once the hole point is located properly, switch to isometric view by right-clicking and selecting 'Isometric View.'



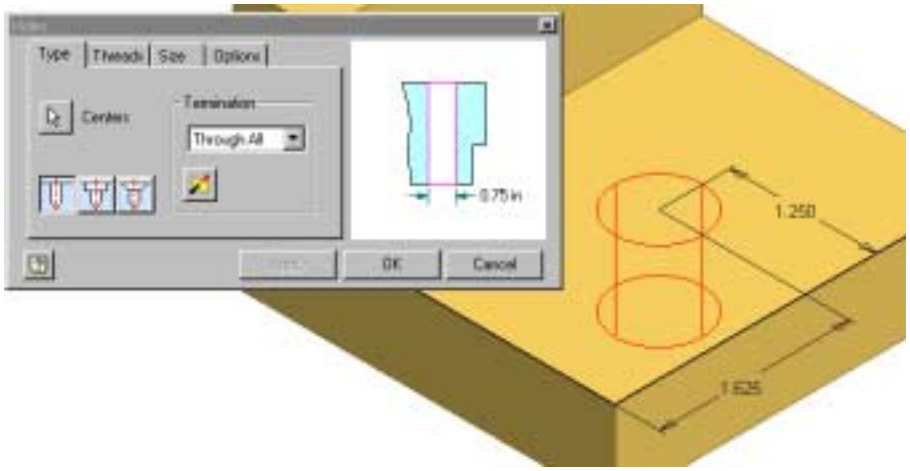
Note that Inventor automatically updates our formulas to show the actual values for our dimensions. Right-click and select 'Finish Sketch'.



When we select 'Finish Sketch', the Sketch tools are replaced by the Feature tools in the Panel Bar.

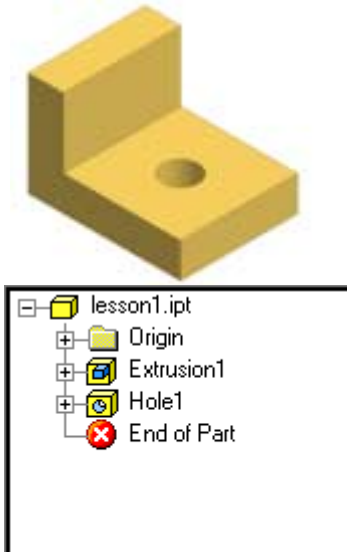


To place the hole, select the Hole tool from the Features toolbar. Note that when our mouse is over the Hole Icon, the Message in the Command Bar updates to indicate what the Hole tool does.

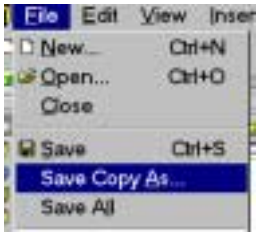


Select 'Through All' for the Termination type. To change the value of the diameter, left-click with the mouse over the diameter value and type '0.75'.

We see the hole previewed in our Drawing Screen area. Once the settings in our Holes dialog box are correct, press 'OK'.



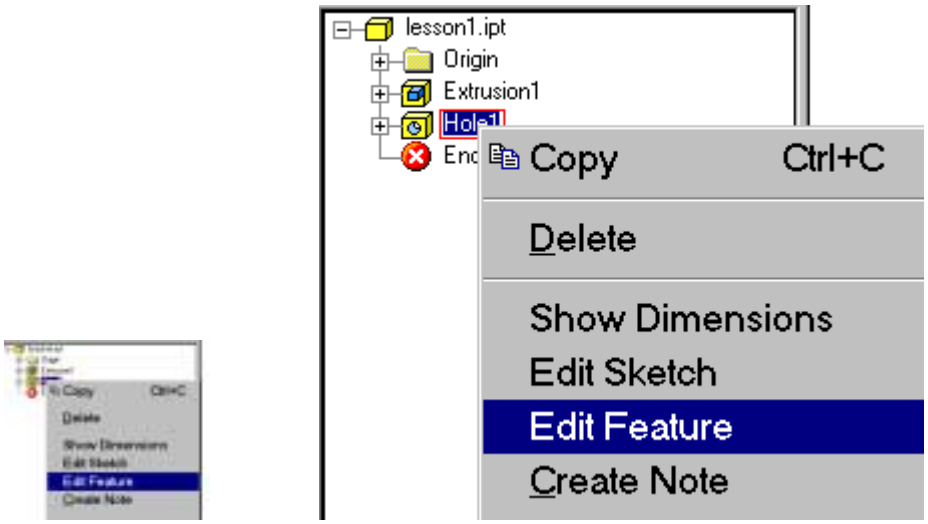
Notice that a Hole now appears in the browser.



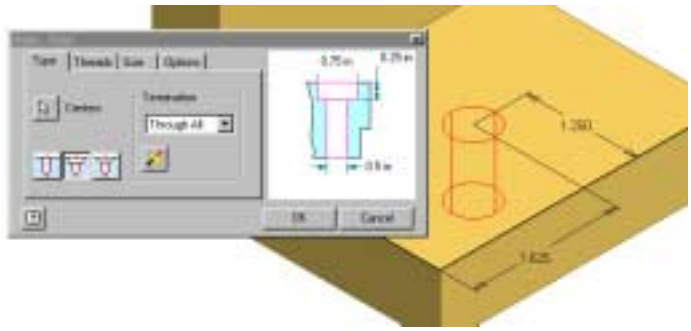
Select 'Save Copy As' under the File menu.
Type in 'L-bracket' for our part name and locate the directory where we wish to save the file.
Note that this does not change the name of the file we are currently working in. We are still working on the Lesson1.ipt file.

Press the 'Save' button.

Edit Hole



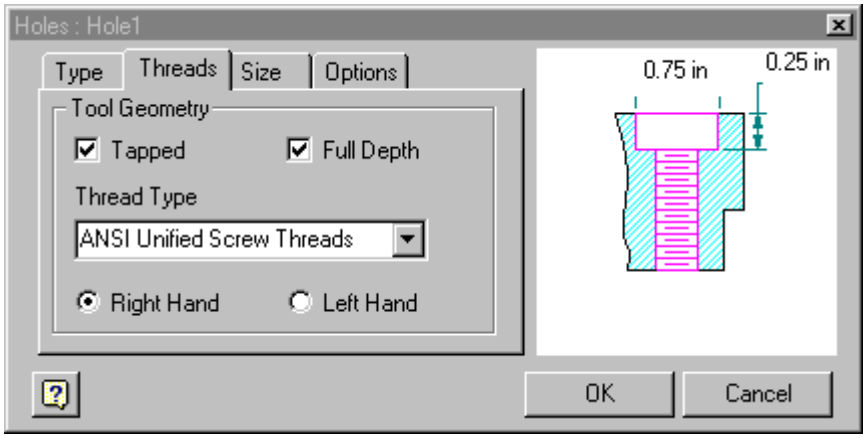
We can modify the hole by highlighting the hole in the browser, right click and select 'Edit Feature'



The Hole dialog box appears as well as the dimensions locating our hole.

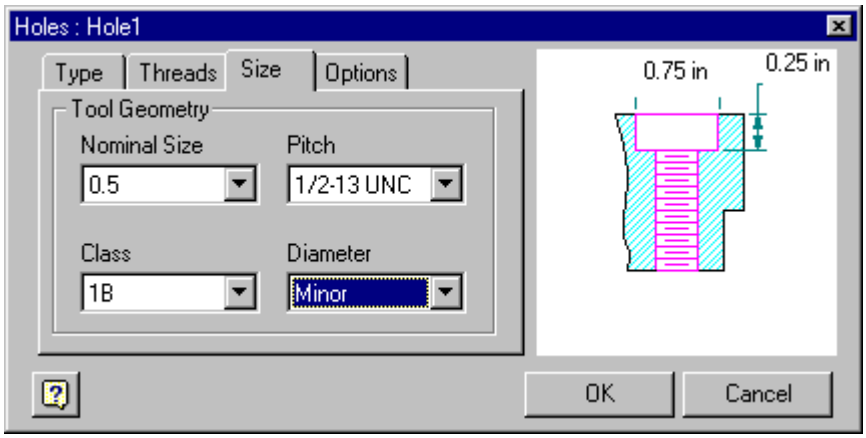
Select the counter bore option. Termination should be set to Through All. Counter bore depth should be 0.25 in. Counter bore diameter should be 0.75 and thread diameters set to 0.50.

Select the Threads tab.



Enable 'Tapped' and Full Depth.
Set the Thread Type to ANSI and the thread direction to Right Hand.

Select the Size Tab.



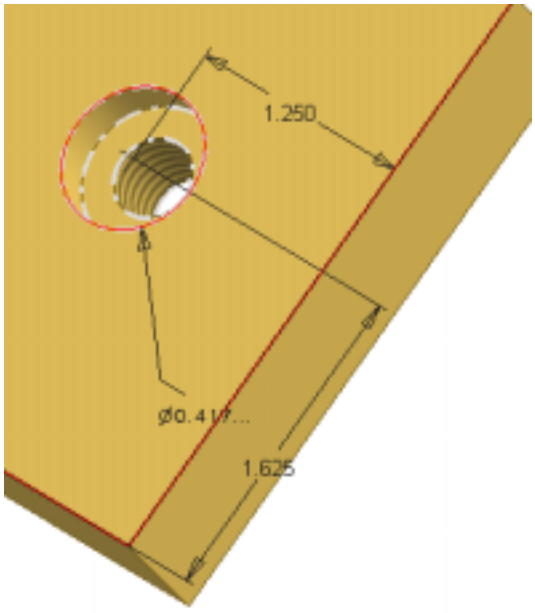
Set the Pitch to ½-13 UNC. Set the class to 1B and diameter to Minor.

Press 'OK'.

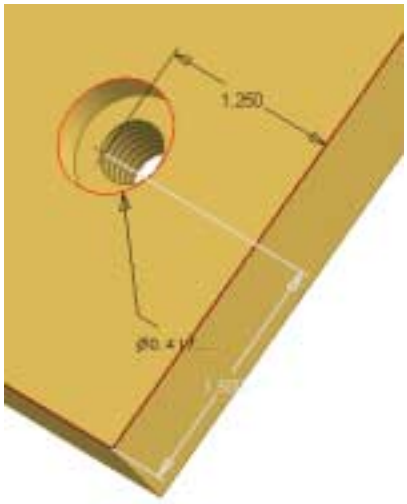
Notice how in R4, we can now see the threads assigned to our hole.

TIP: The Hole updates to the new definition immediately. The user does not need to select the Update tool to see the change. The hole notation we defined will be used when we create our drawing views.

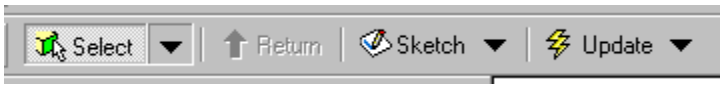
Go to the browser. Highlight the hole, right click and select 'Show Dimensions'.



The dimensions for the hole appear in the graphics window.
Select the 1.625 dimension and change it to 1.5.

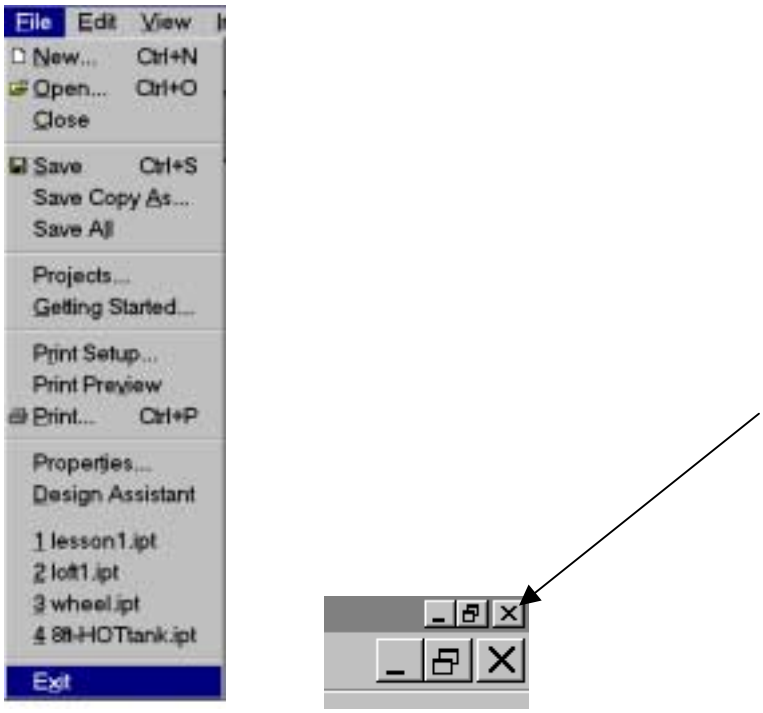


Notice that the dimension remains grayed in the graphics window and the hole location is unchanged.



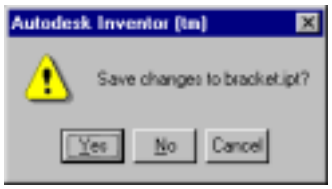
Select the ‘Update’ button located in the upper left of the screen.

The hole now updates to the new location.



To exit Inventor:

- Go to File->Exit
- Press the small ‘x’ on the upper right corner of the screen



The user will be prompted if he wants to save changes.

But – note we are saving changes to bracket.ipt, NOT L-bracket.ipt.
That is because Inventor did not automatically rename our file to L-bracket when we performed the ‘Save Copy As’, instead we remained in our original file.

TIP: Save Copy As does not rename your open file. It creates a copy of your file with a new name. If you wish to work in the new renamed file, you need to close your current file and open the new copy.

Review Questions

1. Before a hole can be created the user must first place and locate:

    A. A work point
    B. A Point, Hole Center
    C. A circle
    D. A drill

2. To modify a hole:

    A. Select the hole in the browser, right click and select 'Modify'
    B. Select the hole in the graphics window, right click and select 'Edit'
    C. Right click in the graphics window and select 'Edit Hole'
    D. Select the hole in the browser or in the graphics window, right click and select 'Edit Feature'

3. The menu option 'Save Copy As':

    A. Renames the active file and saves a copy with the previous name
    B. Saves a copy of the file with a different name and keeps the active file open with the same name and unsaved
    C. Saves a copy of the file with a different name and saves the active file. The user continues working on the active file.
    D. Renames the active file and saves it

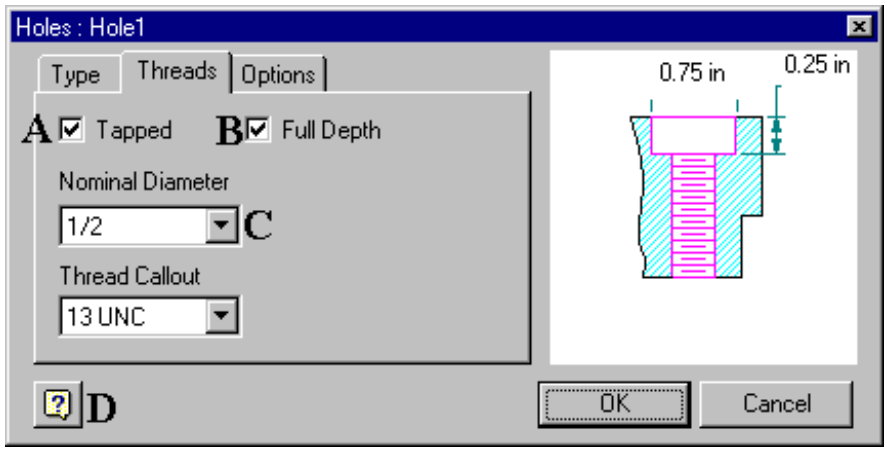
4. The three types of holes that can be placed are:

    A. Drill, Counter bore, and Counter sunk
    B. Through, blind, and mid-plane
    C. Drill, angled, and flat
    D. Threaded, unthreaded, and curled

5. True or False

In Inventor R4, we can see threads when they are applied to hole or cylindrical features.

ANSWERS: 1) B; 2) D; 3) B; 4) A; 5) T



6. Select here to add threads to a hole.
7. Select here to have the threads go the entire depth of the body part of the hole
8. Select here to bring up context-driven help

9. The Options tab is selected to control:

    A. The angle of countersunk holes
    B. The angle of the drill point
    C. The depth of the hole
    D. A & B, but not C

10. True-False

Threads are visible in shaded mode, but not in wire frame mode.

ANSWERS: 1) B; 2) D; 3) B; 4) A; 5) F; 6) A; 7) B; 8) D; 9) D; 10) D