

## LAORT CADD LECTURE NOTES      WEDNESDAY 11/13

### CREATING OBJECTS TO DEFINE REVOLVE/EXTRUDE SOLIDS

#### **READING**

REGION pgs. 638-642

BOUNDARY pgs. 651-652

#### **OVERVIEW**

REGION objects are the most powerful of the objects that may be used to create solid objects in Autocad. Creating them can usually be accomplished easily by using the

REGION  command.


Regions are 2D enclosed areas you create from closed shapes called loops. A loop is a curve or a sequence of connected curves that defines an area on a plane with a boundary that does not intersect itself. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, splines, 3D faces, traces, and solids. The objects that make up the loops must either be closed or form closed areas by sharing endpoints with other objects. The objects must also be coplanar (on the same plane).

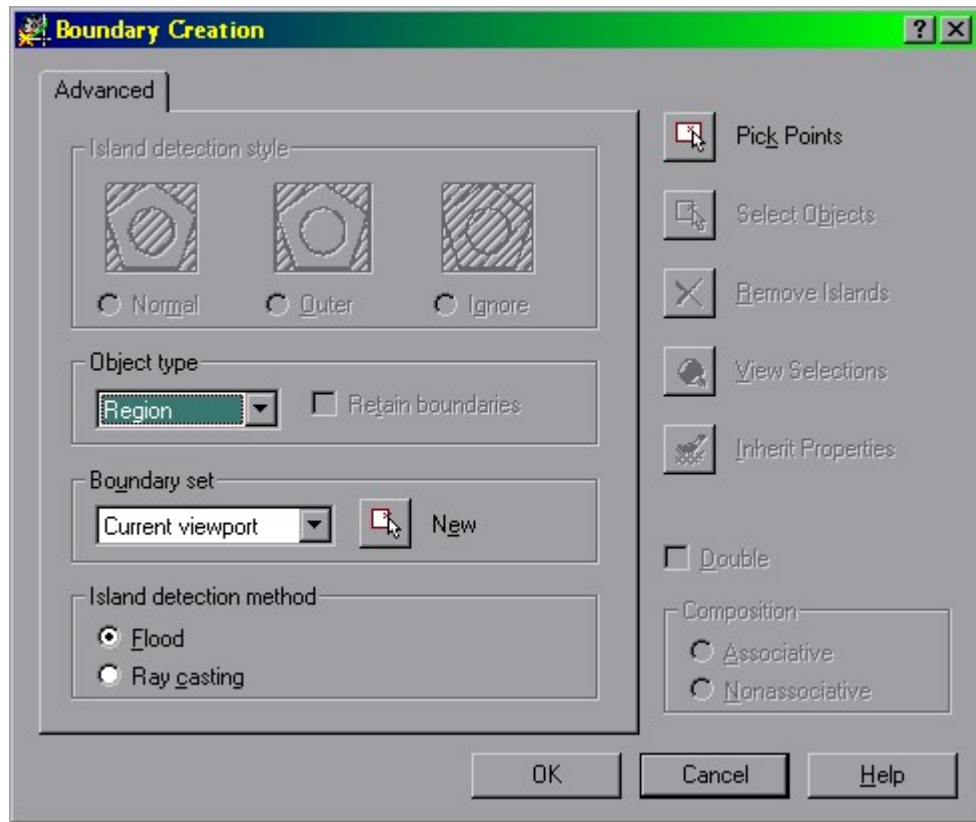
You can create regions out of multiple loops and out of open curves whose endpoints are connected and form loops. If the open curves intersect in their interior, they cannot form a region. Objects such as 3D polylines and face meshes can be converted to regions by being exploded. You cannot form regions from open objects that intersect to form a closed area: for example, intersecting arcs or self-intersecting curves.

The REGION command converts closed 2D and exploded planar 3D polylines in the selection set to separate regions and then converts polylines, lines, and curves to form closed planar loops (outer boundaries and holes of a region). If more than two curves share an endpoint, the resulting region might be arbitrary.

In some cases it may be easier to use the BOUNDARY command typically used for defining HATCH boundaries to quickly create one or several REGION loops.

To create a REGION using the BOUNDARY command dialog:

- *Type in BO from the command line, select the BOUNDARY  button from the toolbar, or use DRAW/BOUNDARY from the menu*
- *Set 'OBJECT TYPE' to Region*
- *Click on the 'New' button and select your geometry by window*
- *Click on the 'Pick Points' button and indicate points on the inside of each of your loops.*
- *Hit enter to exit*



### BOUNDARY CREATION DIALOG

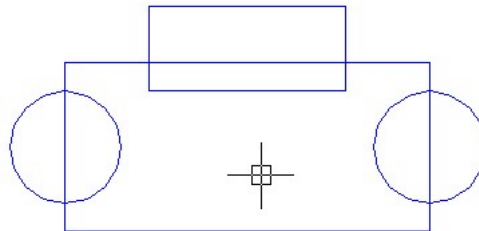
The **REGION** that is created is an entirely new object and the original objects are left unchanged. To make things more manageable you should make a new layer for your **REGION** and set it current before executing the **BOUNDARY** command.

You should use the 'New' button to avoid problems unless your drawing does not have a lot of objects in it.

Endpoints of objects do not have to be coincident but the objects may cross each other.

The **BOUNDARY** command can make a region out of geometry that has not been trimmed and otherwise cleaned up to get your final result. Try the following example.

Draw two pairs of rectangles and two circles as shown in the following picture.



Execute the **BOUNDARY** command and use the cursor location in the picture above for your 'Pick Point.'

The resulting REGION will look like the following.



## **SYSTEM VARIABLES**

Object Type pulldown in BOUNDARY dialog sets system variable HPBOUND. HPBOUND = 0 creates a region, HPBOUND = 1 creates a polyline.

For the REGION command DELOBJ = 1 deletes the original objects, default is DELOBJ = 0 which leaves the original objects untouched. This has no effect when REGION is created using the BOUNDARY command.

## **MODIFYING SOLIDS USING SOLIDEDIT**


### **READING**

SOLIDEDIT  
FACE/MOVE  
FACE/OFFSET

### **OVERVIEW**

SOLIDEDIT is a feature that allows you to modify solids without redefining your solid. The methods available include modifying by FACE, EDGE and BODY. We are going to first look at two of the FACE editing functions MOVE and OFFSET.

The MOVE FACE command is most useful for moving internal features such as holes. You cannot move a feature in such a manner that a negative volume (i.e. a hole) becomes a positive volume (i.e. a boss)

To use the command you either type in SOLDEEDIT from the command line, select the 'MOVE FACE'  icon from the 'Solids Editing' toolbar, or from the menu select 'Modify/Solids Editing/Move faces.'

Moves the selected face on a 3D solid object to a specified height or distance. You can select multiple faces at one time.

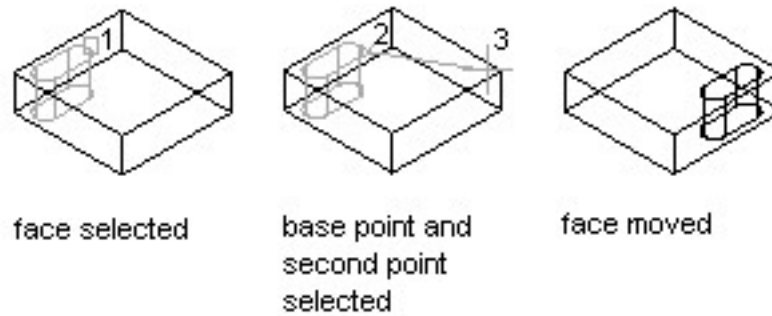
Select faces or [Undo/Remove]: Select one or more faces or enter an option

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, AutoCAD prompts as follows:

Select faces or [Undo/Remove/ALL]: Select one or more faces (1), enter an option, or press ENTER

Specify a base point or displacement: Specify a base point (2)

Specify a second point of displacement: Specify a point (3) or press ENTER



The two points you specify define a displacement vector that indicates how far AutoCAD moves the selected face and in what direction. AutoCAD uses the first point as a base point and places a single copy relative to the base point. If you specify a single point, usually entered as a coordinate, and then press ENTER, AutoCAD uses the coordinate as the new location.

The 'OFFSET FACE' command is used most typically for resizing holes or changing a major dimension of one feature such height or width.

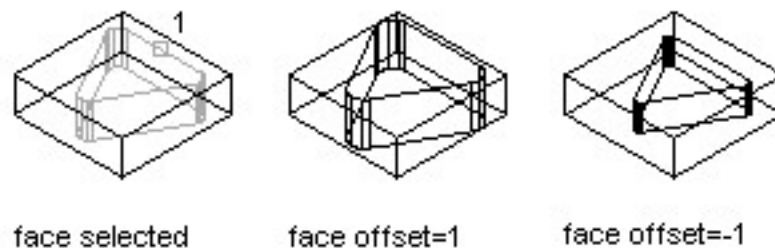
Offsets faces equally by a specified distance or through a specified point. A positive value increases the size or volume of the solid; a negative value decreases the size or volume of the solid.

Select faces or [Undo/Remove]: Select one or more faces or enter an option (1)

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, AutoCAD prompts as follows:

Select faces or [Undo/Remove/ALL]: Select one or more faces, enter an option, or press ENTER

Specify the offset distance: Specify a distance

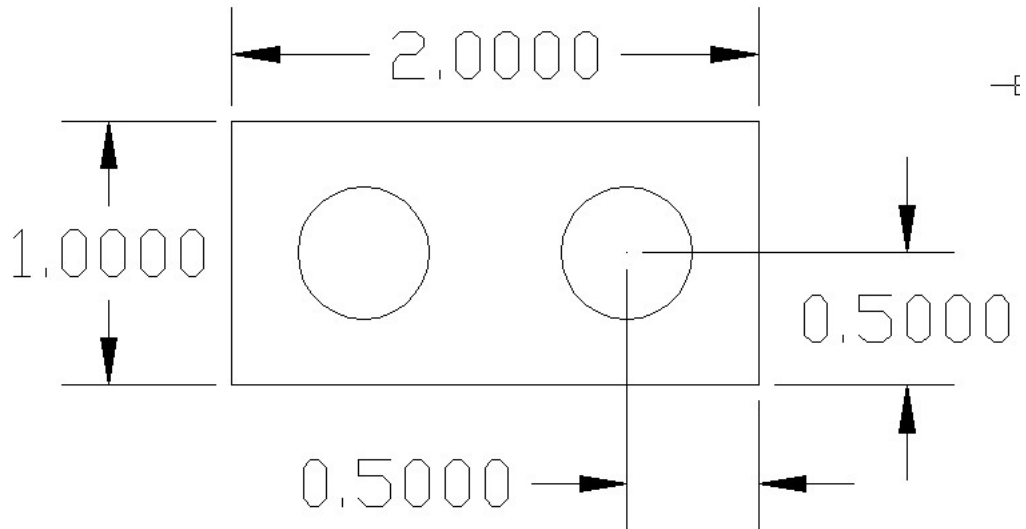


Specify a positive value to increase the size of the solid or a negative value to decrease the size of the solid.

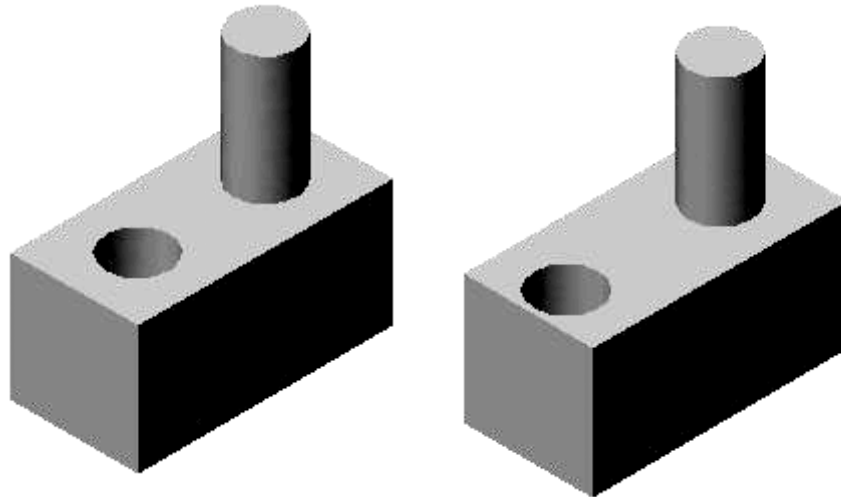
**NOTE** Holes inside a solid object offset smaller as the volume of the solid gets larger. To better understand these two editing functions try the following exercise.

**EXERSIZE: SOLIDEDIT FACE MOVE and OFFSET**

- Draw a **RECTANGLE** and two **CIRCLES** as shown. Use **.25"** radius for circles.

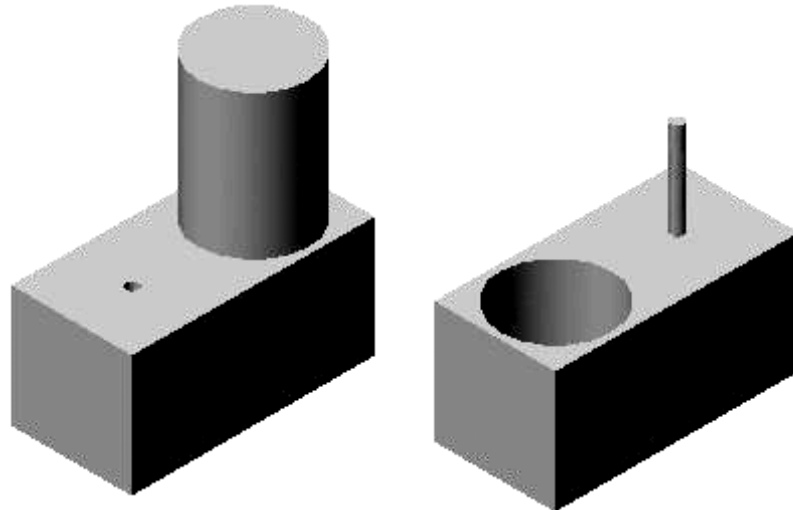


- Extrude the **RECTANGLE 1"** and the **CIRCLES 2"**. Subtract one cylinder from the box and union the other.

**BEFORE MOVE (LEFT) AND AFTER MOVE (RIGHT)**

- Move the hole **-.2"** in the **WCS X** direction. You should get the result shown. Move try moving it another **.05"** in the same direction. What happens? Why?
- Move the hole back to its original position

- Try the same thing with the protruding cylinder in the +X direction. What happens when you try to move it .25" or more.
- Move the protruding cylinder back to its original position
- Now use the **OFFSET** option. Select the Hole and the **CYLINDER**. Enter an offset distance of .2". You should get the following:



**OFFSET .2" (LEFT) AND -.2" (RIGHT)**

- For more practice try moving and offsetting the flat sides of the box and the top of the box. How are **MOVE** and **OFFSET** different?