

Fillet command :This command connects two elements with an arc that touches the two elements and is defined by a radius . With this command, you can rotate the corners of the graphic elements.



Execution method:

-From the drop-down menu bar **Modify >Fillet**

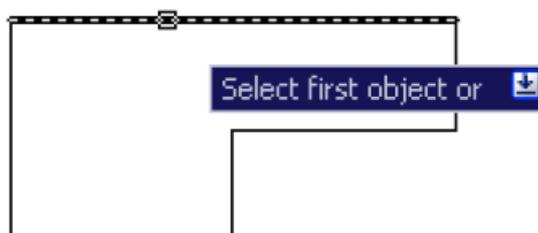
-From the modification toolbar by clicking on the command icon 

-Or by typing the command or its abbreviation **F** on the command bar

When requesting the command, the program message appears, which to select the first line

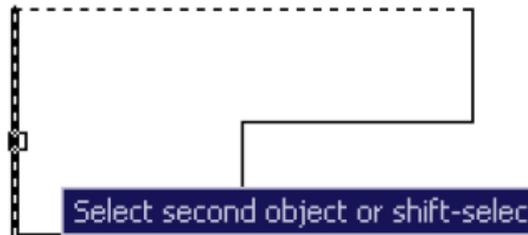
Select first line or[undo/polyline/Radius /trim/ multiple]

You can see in the command line that the program has shown you the radius of the current arc that you are from through it, the ends will be rotated. If you want to change the value, from the keyboard, type the letter **R**, then **Enter**, then enter the new value for the radius, let it be 5 units .Now select the first element you want to rotate by clicking on it with the left mouse bottom

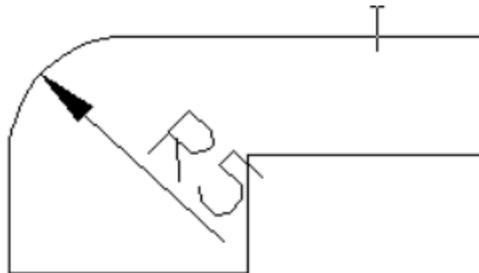


The second AutoCAD message appears asking to specify the second line to be rotated

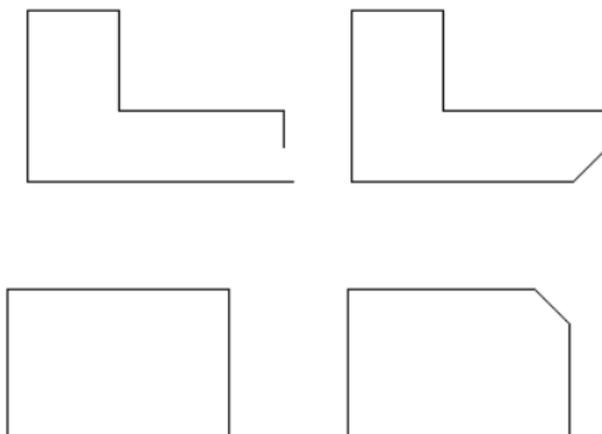
Select second line or shift-select to apply corner



Select the second line by clicking on it, and you will find that the program has rounded the angle where the two lines meet directly. An arc with a radius of 5 units



Chamfer command: This command works to achieve the connection of two elements with each other with an oblique bevel or to convert their connection into an oblique bevel



Execution method:

-From the drop-down menu bar **Modify >chamfer**

-From the modification toolbar by clicking on the command icon 

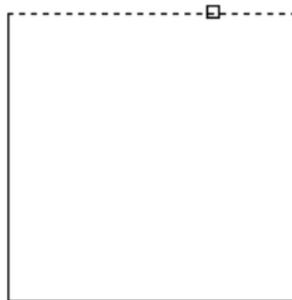
-Or by typing the command or its abbreviation **cha** on the command bar

When the command is requested, the program message appears, which asks to specify the first line to be chamfered

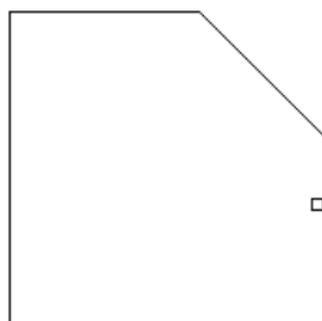
Select first line or[undo/polyline/distance/angle/trim/mEthod/ multiple]

You can see in the command line that the program has shown you the current first and second chamfer distance and the history in the system variable ChamferA for the first chamfer distance and ChamferB for the second chamfer distance. In case you want to change the chamfer distance, type d and then enter the first and second chamfer distance as required . The program, knowing that the distances will remain the default until you change them again.

Now that you have defined the chamfer distances, select the first line to be chamfered

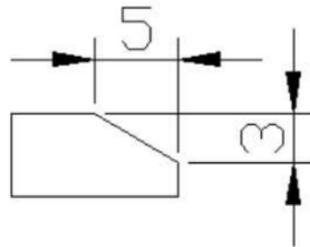


A second AutoCAD message appears asking to specify the second line to be chamfered

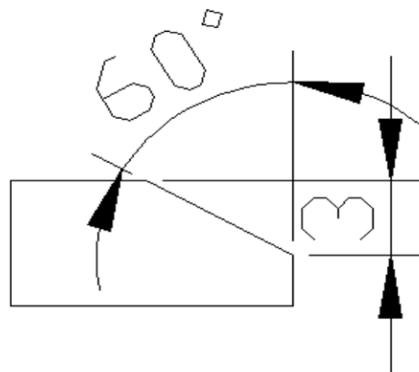
Select second line or shift-select to apply corner

Select the second line by clicking on it, and you will find that the program has chamfered the angle of the two lines meeting directly

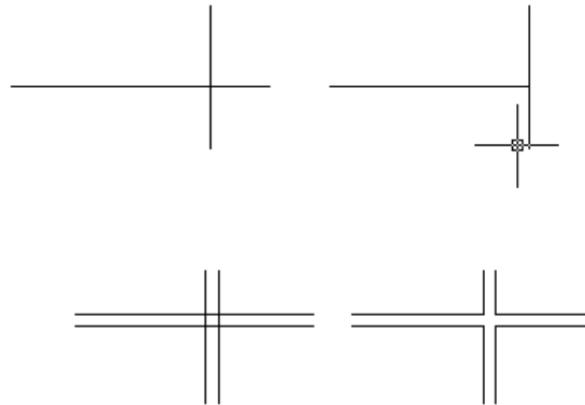
In case, you want to change the chamfer distance, type **d** then enter the first and second chamfer distance



Angle: To make an oblique chamfer in terms of the first chamfer distance for the first line and a certain angle for the second line. This option is activated by typing the letter **A**. Then the program asks you to specify the first chamfer distance, after entering the chamfered distance, the program asks you for the chamfer angle for the second line, then the program asks you to specify the first line and the second line, select them as previously explained, and you get the required drawing.



Trim command: This command is considered one of the important commands in the Modify menu in terms of use, as this command cuts excess parts of the element in reference to another element that forms the borders of the crop



Execution method:

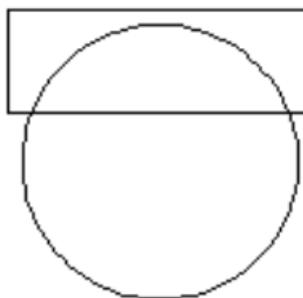
-From the drop-down menu bar **Modify >Trim**

-From the modification toolbar by clicking on the command icon

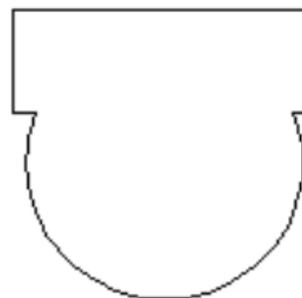


-Or by typing the command or its abbreviation **TR** on the command bar

When writing the command, an AutoCAD **select object** message appears, which means choosing the element that defines the cutting edge. We will work to clarify the matter through the following example

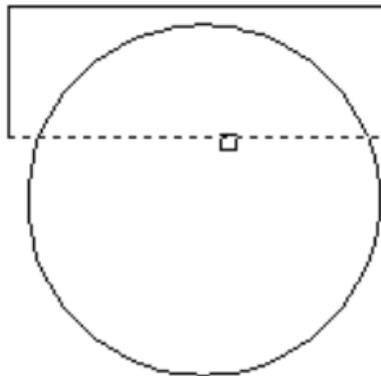


Before Trim

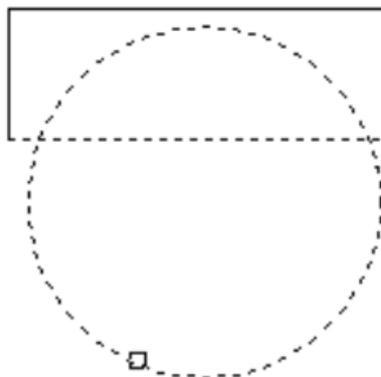


After Trim

Since it is required to cut the upper part of the circle inside the rectangle, we will define the lower rectangle side as the cutting edge



And since we also want to cut the lower side part of the rectangle located within the borders of the circle, then we define the circle as a border for cutting the side of the rectangle, and this means that the chosen element can be a border for cutting and at the same time an element to be cut

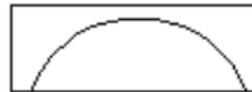
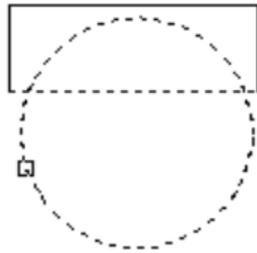
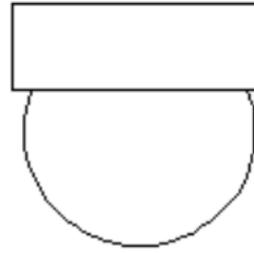
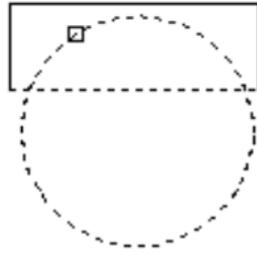


After completing the selection of the elements that represent the cutting borders, we press the Enter key, indicating the end of the selection. Knowing that you can choose the elements in any of the methods of selecting the elements previously explained in previous lessons. After that, the program asks you to select the item(s) that you want to cut

Select object to trim or shift-select to extend Or [fence/crossing/project/edge/erase/undo]

In order to determine the part to be cut, attention must be paid to the direction. For example, in order to cut the part of the circle located above the side of the rectangle, we must click the left mouse button on the upper part of the circle, above the side of the rectangle, and if you click the left mouse button on the circle from the bottom side of the side of the square, the program will understand that you want to cut out the

bottom part of the circle.

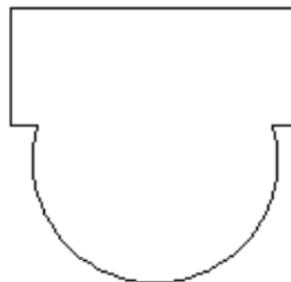


Selection of the element

The resulting figure

Select the item to be cut from the right side. You will notice that the program cuts the item immediately after selecting it without pressing the Enter button. The command remains effective for selecting more items until you press the Enter button

Ro



Rotate Command: This command is used to rotate the element at a specific angle around a specific basis point

Execution method:

-From the drop-down menu bar **Modify >Rotate.**

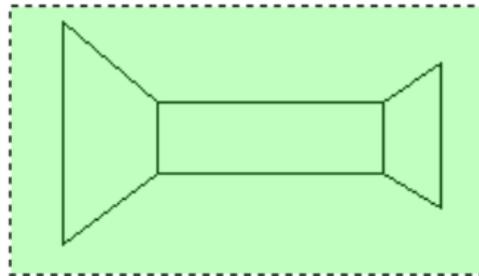


-From the modification toolbar by clicking on the command icon .

-Or by typing the command or its abbreviation **RO** on the command bar.

When executing the command, the program's usual message appears, **select object** ,select the items to be rotated.

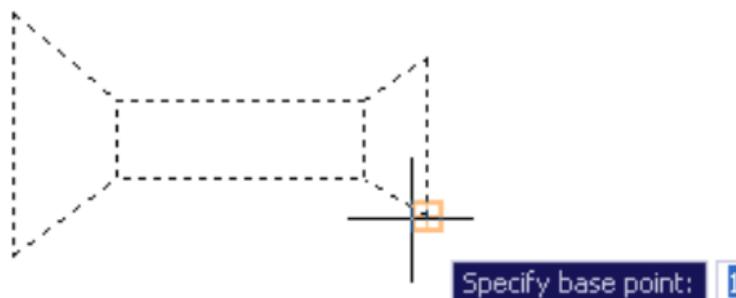
Select the item or items in one of the ways you know, then press **Enter.**



After selecting the element, the program asks you for the base point around which the element will rotate

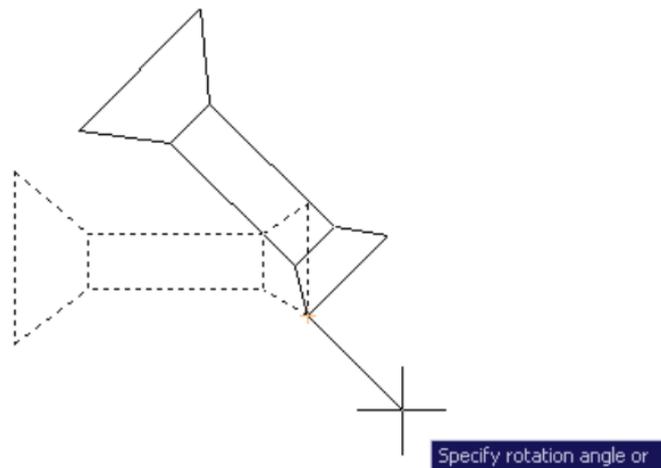
Specify base point

Select the point with the help of **Object snap** and capture the point with the left mouse button, or enter the coordinates of the base point if you have it available .

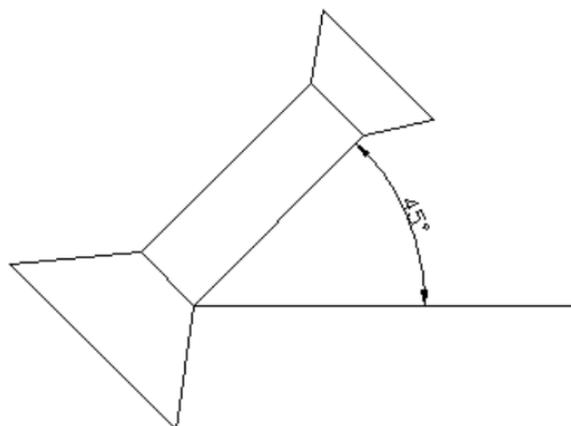


After selecting the rotation point, you will find that the element has started to move from the base point in a circular motion and in all directions according to the movement of the mouse. With the appearance of the command Determine the angle of rotation

Specify rotation angle or [copy/reference]



Specify the angle of rotation you want by typing it on the command line, let it be 45 degrees. You find that the element has been rotated at an angle 45 degrees around the base point and in a counterclockwise direction, because the value of the angle is positive, and if you want to rotate the element clockwise, the angle is entered with the negative value - 45 .



Scale command:(Enlarged or Reduced of the graphics)

This command is used to real change the dimensions of the graphics by a certain percentage of the original drawing. For example, to enlarge a rectangle whose dimensions are 3×4 twice, the output of the process is a rectangle whose dimensions are 6×8.

Execution method:

-From the drop-down menu bar **Modify >Scale**.

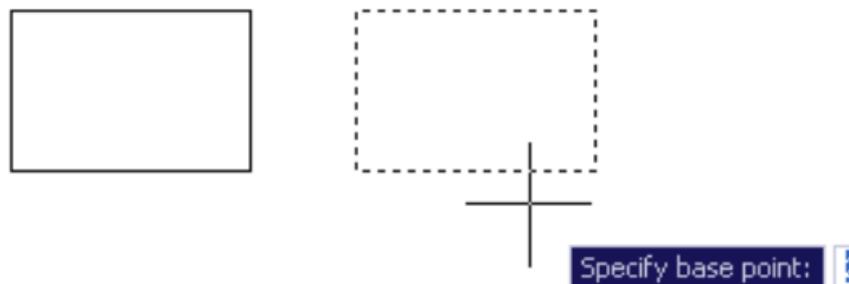
-From the modification toolbar by clicking on the command icon  .

-Or by typing the command or its abbreviation **SC** on the command bar.

When executing the command in the previous ways, the usual AutoCAD message appears, select the required item enlarged or reduced **Select Object**

After you choose the element by one of the elements selection methods, the program asks you to specify the base point from which the element will be enlarged or reduced

Specify base point



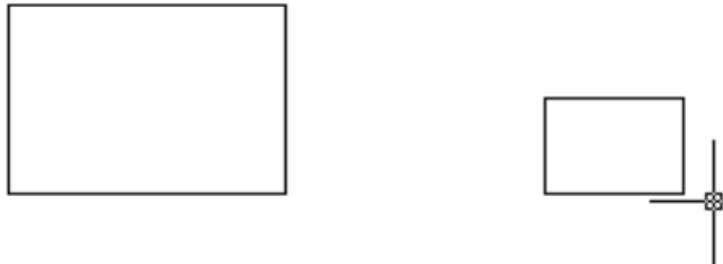
Determine the base point with the help of Osnap or by entering the coordinates of the point if it is known to you. The program directly asks you to specify the factor of reduction or enlargement

Specify scale factor or [copy/reference]

To enlarge the element, enter a value greater than 1, and to reduce it, enter a value less than 1. When the value is 1, the element remains without enlarging or reducing.

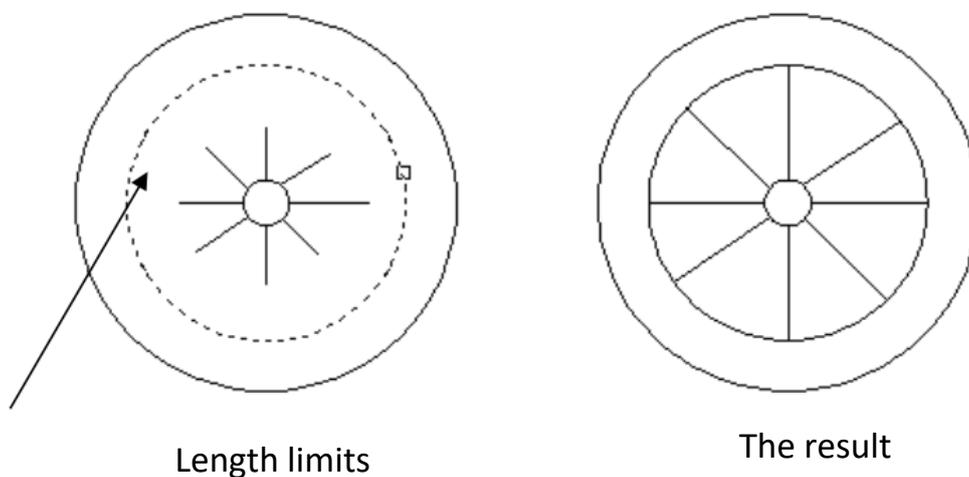
You must consider entering real numbers and not negative numbers in the case of zooming out (for values less than 1), for example 0.8..... 0.05 and so on.

Enter the value you want, and here we will enter a value of 0.5 to reduce the element in half.



Extend command (Extend items to border):

This command works to lengthen the elements towards other elements that represent the limits of the lengthening. It is quite similar to the Trim command in terms of the method of dealing with it and reverses it in the function. The program asks you to specify two elements, the first element, which is the element that represents the limits of lengthening, and the second element, which is the element to be lengthened .



Execution method:

-From the drop-down menu bar **Modify >Extend**.

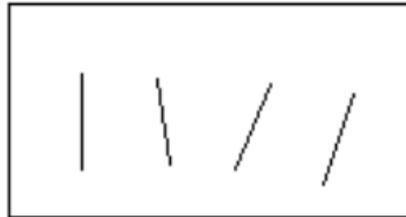
-From the modification toolbar by clicking on the command icon .

-Or by typing the command or its abbreviation **EX** on the command bar.

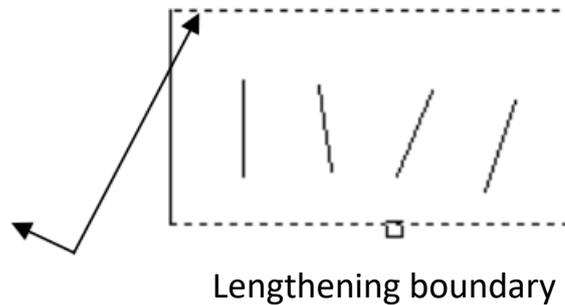
When executing the command in one of the ways described above, the program asks you through the command bar to specify the lengthening boundary

Select boundary edges

In our next example, we will extend the straight lines to the borders of the rectangle from the upper and lower sides



Select the top and bottom sides of the rectangle as the lengthening boundary, and press **Enter**.



The program now asks you to select the element or elements to be extended

Select object to extend or shift-select to trim Or [fence/crossing/project/edge/erase/undo]

Select the items to be extended by clicking on them one by one or by selecting them once. You will notice that in order to lengthen the element from both sides, you must click on it twice when selecting it individually, each time from the side you want to lengthen.

