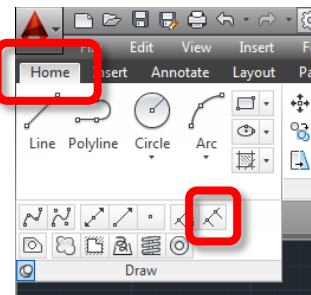# Using the Measure Command
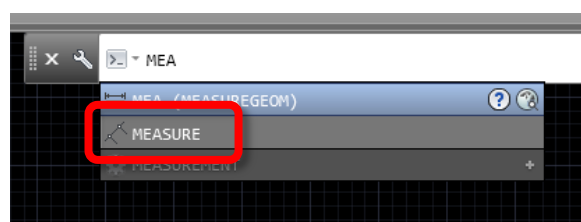
AutoCAD

Once the "POINT" style has been set (see "2013-11-19_MJP_WP_NODES AND POINTS.PDF" on Cadline Community web site), not only can "POINTS" be placed in the drawing, but they may be used with a couple of very useful commands these are "DIVIDE" and "MEASURE". The text below looks at "MEASURE".
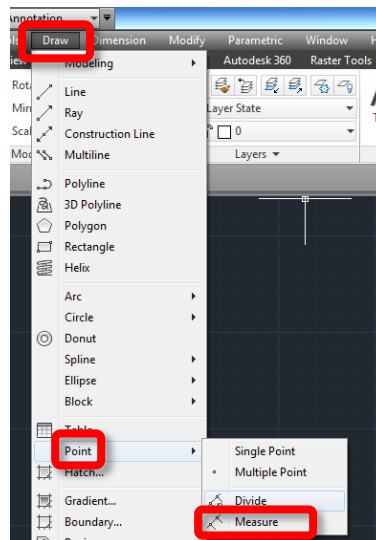
The "MEASURE" command can be accessed in one of the following ways, expand the "DRAW" panel in the "HOME" tab and select "MEASURE".

Or start typing the word "MEASure" at the command line, then select the "MEASURE" command from the displayed list.

Or if the Menu Bar is displayed, select "MEASURE" from the "POINT" fly out in "DRAW" drop down.

The "MEASURE" command can work with most objects including "LINE", "POLYLINE", "RECTANGLE", "POLYGON", "ARC", and "CIRCLE".

Using the "MEASURE" Command to Place Points at a Specified Distance Along a Drawn Object

Prior to using the "MEASURE" command it is advisable to set the point style (see "2013-11-19_MJP_WP_NODES AND POINTS.PDF" on Cadline Community web site).
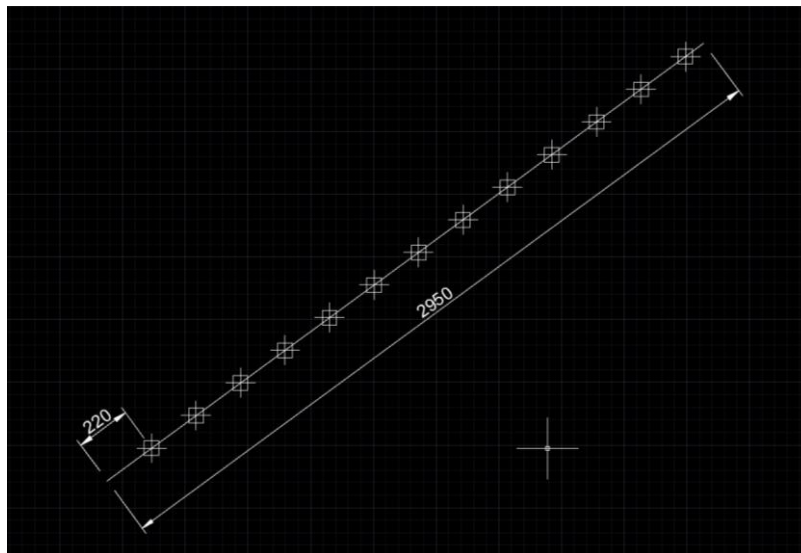
After selecting the "MEASURE" command, AutoCAD will prompt to select an object to be measured; select a "LINE", "POLYLINE" etc.

Once the object has been selected, AutoCAD will prompt for a measureable distance, type in a numerical value.
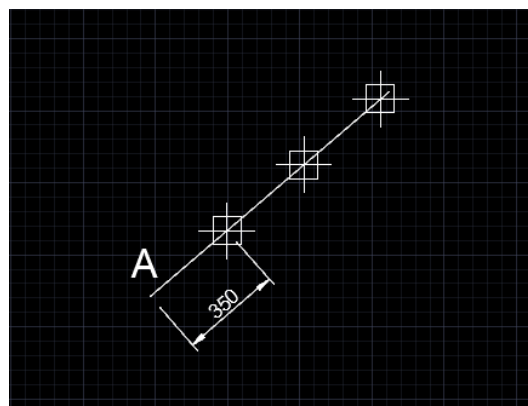
Press "ENTER" to confirm the value.

"POINTS" are now displayed along the selected object and are snappable using the "NODE" object snap.

In the example below the length of the line is 2950 units, and "POINTS" are placed along the line every 220 units.
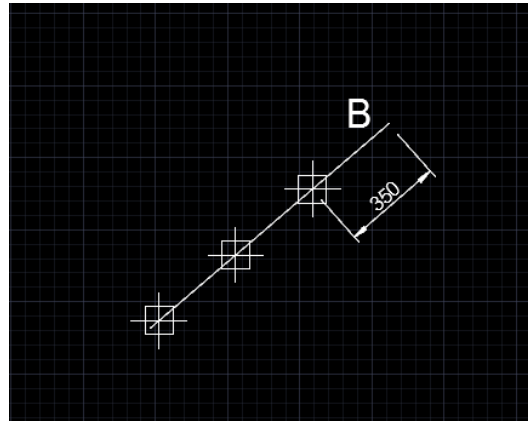

NOTE:

When selecting the object to be measured, the location picked on the object will effect on the way POINTS are placed on the object. AutoCAD will measure from the nearest end point to the point picked on the object.

Object selected closer to point "A"

Object selected closer to point "B"

<u>ALSO</u>

1. If the "POINTS" are displayed they will print.
2. Only one style of "POINT" can be used in the current drawing.
3. "POINTS" take on the properties of the layer they are placed on.
4. To stop the "POINTS" from printing use one of the following methods.
    1. Place the "POINTS" on a separate layer which can be set to either "NON PLOT" or "OFF"
    2. Reset the point style using the "DDPMODE" dialogue to none (second style option from the left on the top row of the point style dialogue).
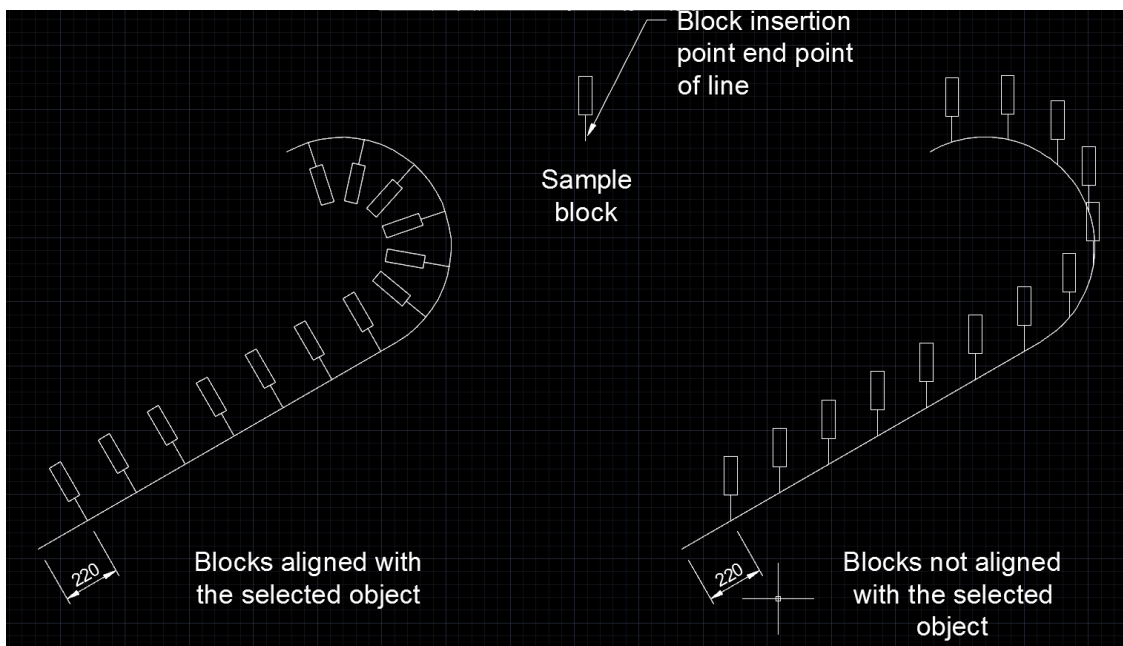
Using the "MEASURE" Command with "BLOCKS"

As with the "DIVIDE" command "BLOCKS" may also be used with the "MEASURE" command, and as with the "DIVIDE" command the "BLOCK" must exist within the drawing either by creation or insertion.

To "MEASURE" along an object placing "BLOCKS" instead of "POINTS".

After selecting to object to measure, instead of specifying the length of segment either type in "B" for "BLOCK" select "BLOCK" from the command line, or right mouse click and select "BLOCK" from the displayed menu. AutoCAD now prompts for the block name, type in the relevant "BLOCK" name, confirm by pressing enter.
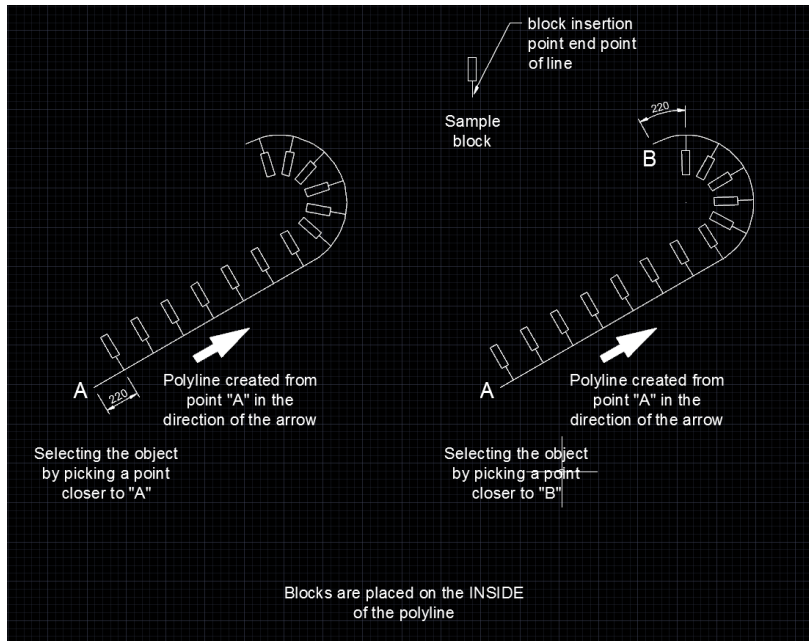
The block can be aligned with the selected object which is the default, ("YES") press enter or not aligned with the selected object enter "NO", the block will keep its original orientation.
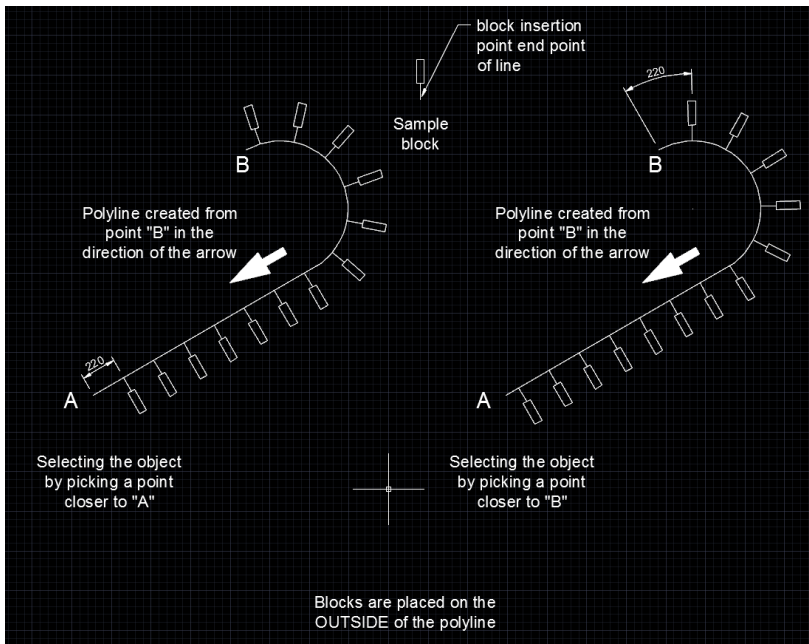

The final step is to specify the distance between the "BLOCKS", type in the required numerical value and confirm by pressing enter.

NOTES AND SUMMARY:-

Once the object to measure has been selected the measure whether it's using points or blocks will always start from the end point nearest to the point picked on the object as below.
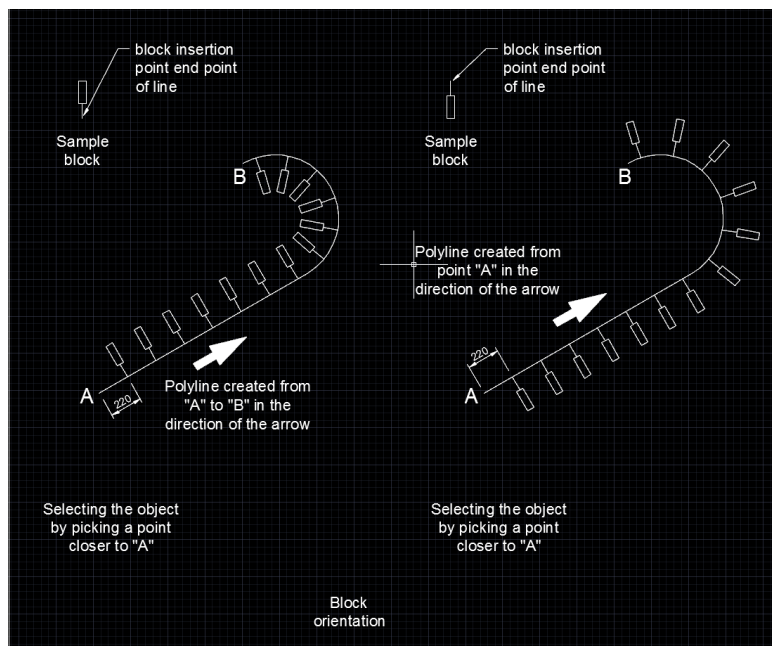

In the case of measuring along a polyline the direction of creation of the polyline can have an influence.

The direction of a "POLYLINE" can be reversed by selecting the polyline, right mouse click and from the displayed menu select "REVERSE"; this saves time redrawing the polyline. This function is only available in the later versions of AutoCAD.

When using the "MEASURE" command with the "BLOCK" option, the "BLOCK" orientation will affect the way the blocks are displayed along the selected object as below

Factors that can affect the way using "BLOCKS" are displayed within the "MEASURE" command.

1. The orientation of the created block, i.e.; - if the block was created through 180 degrees the results would be reversed.

2. If using the "MEASURE" command to place "BLOCKS" along a polyline the direction of the polyline may affect the display, but this is also dependent on the insertion point of the "BLOCK". (The direction of the "POLYLINE" may be changed by selecting the polyline, right mouse click and from the displayed menu select "REVERSE").

ADDITIONAL INFORMATION

2013-11-19_MJP_WP_NODES AND POINTS.PDF"

2013-12-02_MJP_WP_USING_THE_DIVIDE_COMMAND.PDF

On the Cadline Community web site.