

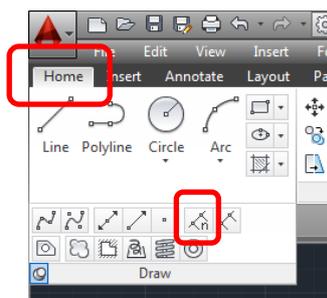
Using the Divide Command

AutoCAD

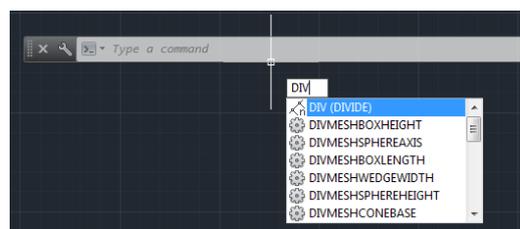
Using the Divide Command.

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 "DIVIDE".

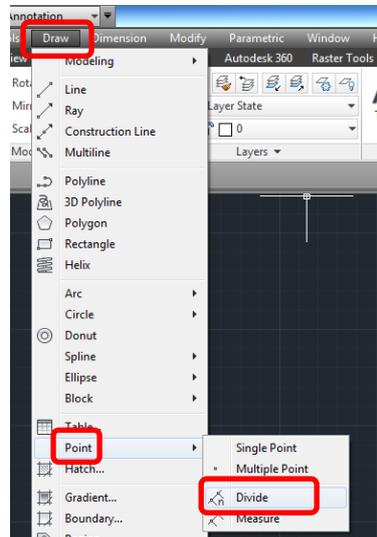
The "DIVIDE" command can be accessed in one of the following ways, expand the "DRAW" panel in the "HOME" tab and select "DIVIDE".



Start typing the word "DIVide" in the command line, then select the "DIVIDE" command from the displayed list.



Or if the Menu Bar is displayed, select “DIVIDE” from the “POINT” fly out in “DRAW” drop down.



The “DIVIDE” command can work with most objects including “LINE”, “POLYLINE”, “RECTANGLE”, “POLYGON”, “ARC”, and “CIRCLE”.

Using the “DIVIDE” Command to Equally place points on a Drawn Object.

After selecting the “DIVIDE” command, AutoCAD will prompt to select the object to be divided; select a “LINE”, “POLYLINE” etc.

Once the object has been selected, AutoCAD will prompt for a number by which the object will be divided by; type in the required value (this can be any numerical value between 2 and 32767).

Press “ENTER” to confirm the value.

Providing the “POINT” style has been set, “POINTS” are now displayed along the selected object.

In the example below the line has been divided by five displaying four “POINTS”.



Note:

1. The item selected has not been broken or split into segments, it still remains as one object but "POINTS" have now been placed along it.
2. In the case of dividing a "LINE" one point less is shown than the specified value, therefore dividing the object by the specified number of segments.

Using the Divide Command with "BLOCKS".

If none of the "POINT" styles are suitable a block may be used instead, the "DIVIDE" command can also place "BLOCKS" along an object instead of "POINTS".

The "BLOCK" must exist within the drawing either by creating or insertion.

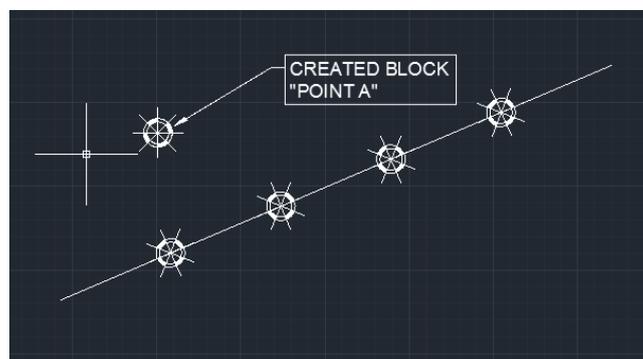
In this example the "BLOCK" "POINT A" has been created, placing blocks on the line using the "DIVIDE" command is a similar process as before.

But after the selecting the object instead of typing in a dividing value use the "BLOCK" sub command, either by selection from the command line, typing in "B" and "ENTER" or accessing "BLOCK" from the right click menu.

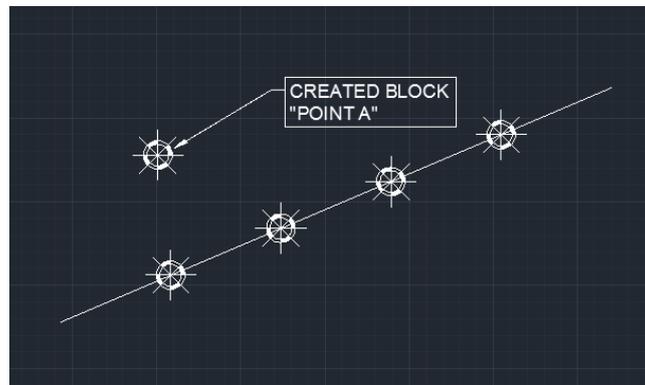
AutoCAD will then prompt for the "BLOCK" name, type in the block name and confirm by pressing "ENTER", there are now two available options either to "ALIGN" or "NOT TO ALIGN" the blocks with the selected object, the default option being align the "BLOCK" with the selected object.

Once the type of alignment has been decided, a dividing number can be entered as above.

The screen grabs below show both options "ALIGNING" the block "POINT A" with the selected object, and keeping the block in its original orientation, "NOT ALIGNED".



Blocks aligned



Blocks not aligned

Note:

Standard blocks have a fixed size and therefore will not automatically re-scale using the REGEN after a change of display as with the “POINTS”.

Recap Notes:-

1. If the “POINTS” are displayed they will print.
2. Only one style of “POINT” can be used in the current drawing.
3. To snap to “POINTS” use the “NODE” object snap.
4. Use the “REGEN” command to correctly display the “POINTS” after zooming in or out of the drawing.
5. “POINTS” take on the properties of the layer they are placed on.
6. To stop the “POINTS” from prints 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).