

# MEASURE Command — GstarCAD 2027

3621 kbadmin September 25, 2026 [Command Reference](#) 0 11

## MEASURE Command

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### Function Description:

This command is used to create points or blocks at measured intervals along the length or perimeter of an object.

Points or blocks are located on the selected objects and their orientation is determined by

the XY plane of the UCS. Users can use the DDPTYPE command to set the style and size of all point objects in a drawing.

### Command Access:

**Menu:** Draw > Point > Measure

**Ribbon:** Home > Draw > Measure

**Toolbar:** /

**Command:** MEASURE

**Alias:** ME

## Command Prompts:

Select object to measure:

Specify length of segment or [Block]:

Enter name of block to insert:

Align block with object? [Yes/No] <Y>:

## Length of segment

Places point objects along the selected object with the specified interval and starting at the endpoint closest to the point you used to select the object.

When selecting a closed polyline, the specified length is measured from the first point.

When selecting a circle, the specified length is measured from the current snap rotation angle. If the current snap rotation angle is

0, the specified length is measured from the right of center.

## Block

Places blocks along the selected object at a specified interval.

### Align block with object

- **Yes:**  
The inserted block is rotated around its insertion point and its horizontal lines are aligned to tangent to the selected object.
- **No:**  
The block is always inserted by a rotation angle of 0.

After specifying the length of segment, blocks are inserted at the specified interval. If the inserted block contains variable attributes, those attributes are abandoned.

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### Related References?

- [Commands about Measuring and Dividing](#)

### Related Tasks?

- [To Insert Blocks at a Specified Interval Along an Object](#)
- [To Create Points at a Specified Interval Along an Object](#)

Other versions with matching text

[2023](#) · [2024](#) · [2025](#) · [2026](#)

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