

# OFFSET command

2257 GstarCAD MY /KW April 14, 2022 [CAD Commands](#) 0 1708

The **OFFSET** command is used to create concentric circles, parallel lines and parallel curves.

## Command Access:

**Ribbon** : Home > Modify > Offset

**Menu** : Modify > Offset

**Command** : OFFSET

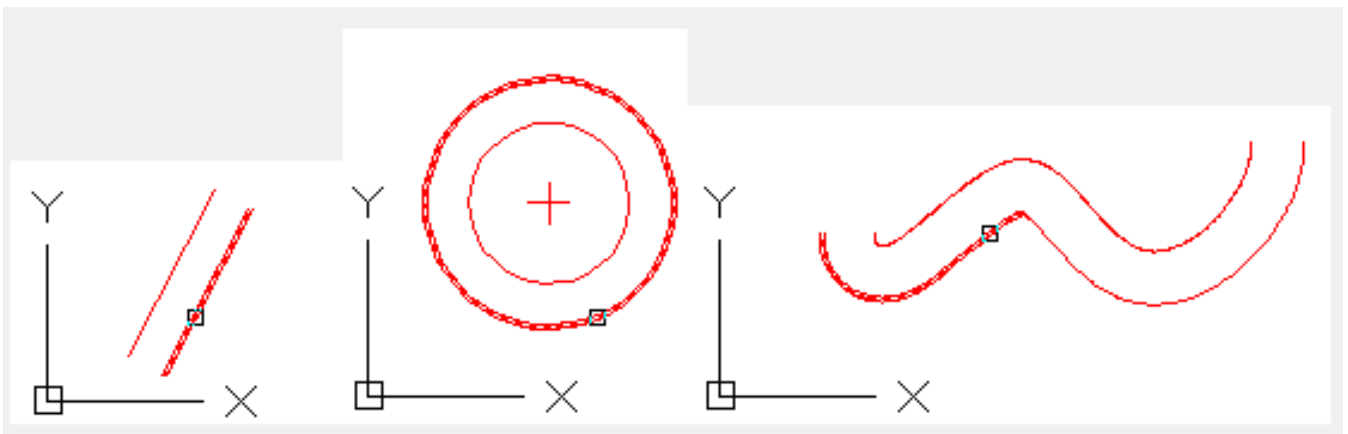
## Command Prompts:

Current settings: Erase source=No Layer=source

Specify offset distance or [Through/Erase/Layer] :

Specify through point or [Exit/Multiple/Undo] :

## Function Description:

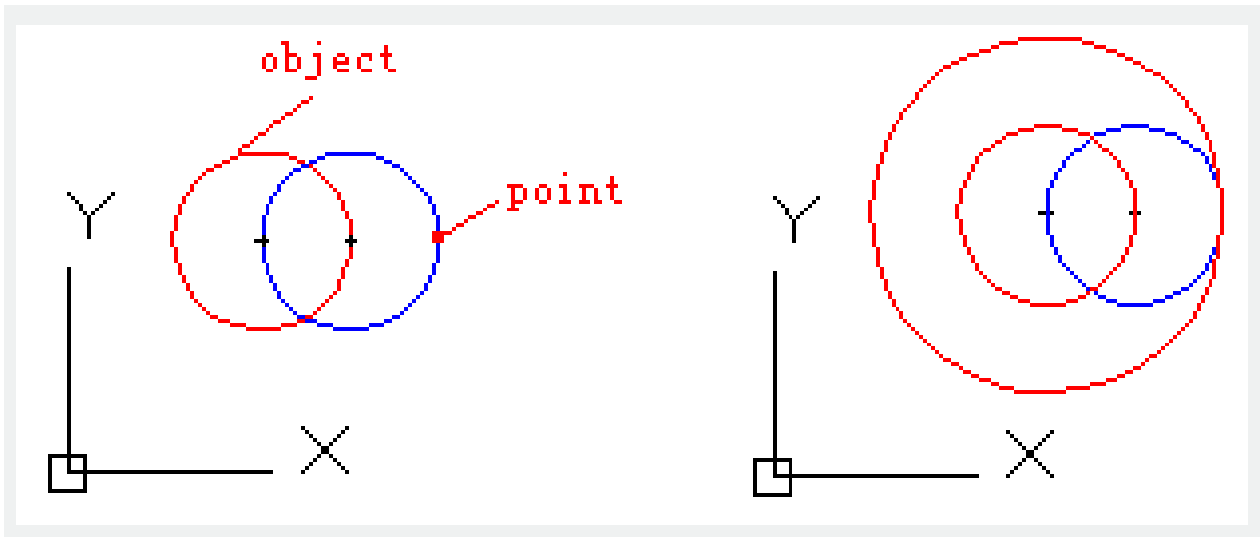


Users could offset selected objects by a specified distance or a point. After offsetting objects, users could use tools of "TRIM" or "EXTEND" to create graphics with several parallel lines and curves. For the convenience of use, the OFFSET command will not stop prompting until pressing ENTER to exit.

## Relative Glossary:

### Offset distance:

Create objects by specified distance from selected objects.

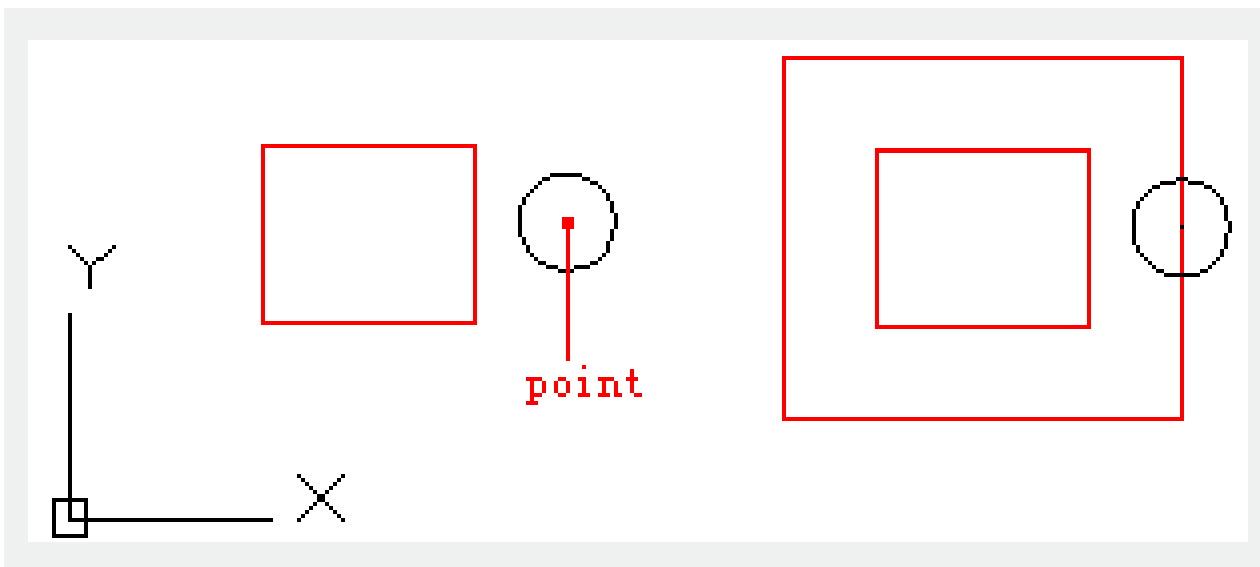


**Exit :** Exit the OFFSET command.

**Undo :** Restore last offset.

**Through:**

Create an object through specified point.



**Exit :** Exit the OFFSET command.

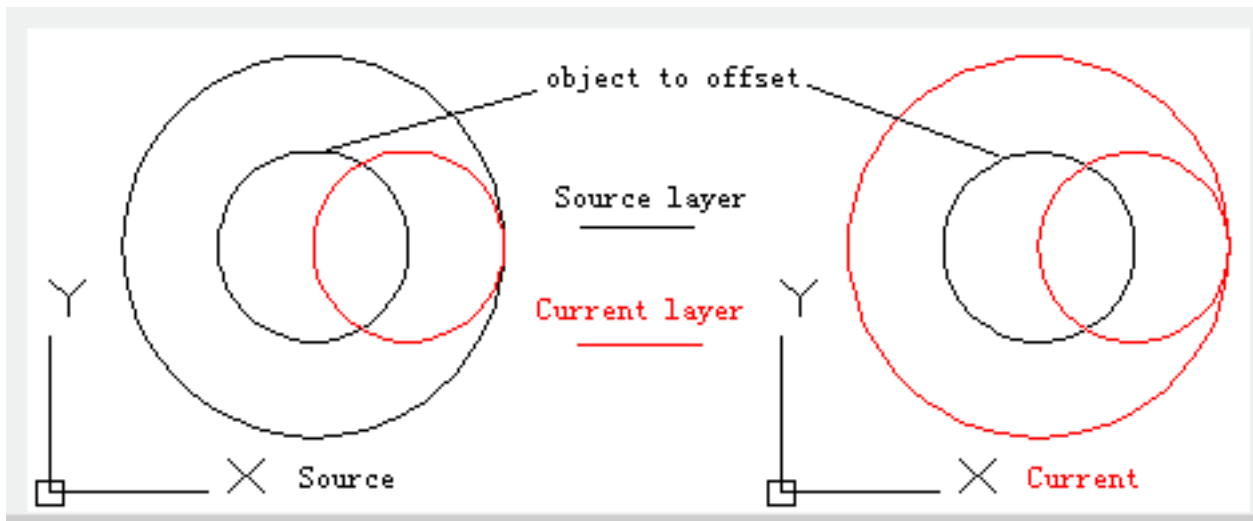
**Undo :** Restore last offset.

**Erase :** Erase source object after offsetting.

**Layer :** Apply current layer or source layer on object for offsetting.

**Current :** Apply current layer on selected object for offsetting.

**Source :** Apply source layer on selected object for offsetting.



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