

OFFSET Command — GstarCAD 2027

3291 kbadmin September 25, 2026 [Command Reference](#) 0 22

OFFSET Command

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Commands > O Command > OFFSET

Function Description:

This command is used to create concentric circles, parallel lines and parallel curves. Users can offset a selected object by a specified distance or a point. After offsetting objects, users can use the TRIM or EXTEND command to create drawings with several parallel lines and curves. For the convenience of use, the OFFSET command will not stop prompting until you press Enter to exit.

Command Access:

Menu: Modify > Offset

Ribbon: Home > Modify > Offset

Toolbar: Modify > Offset

Command: OFFSET

Alias: O

Command Prompts:

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

Offset distance

Creates an object by a specified distance from the selected object.

- **Exit:**
Exits the OFFSET command.
- **Undo:**
Restores the last offset.

Through

Creates an object through a specified point.

- **Exit:**
Exits the OFFSET command.
- **Undo:**
Restores the last offset.

Erase

Erases the source object after offsetting.

Layer

Applies the current layer or source layer on an object for offsetting.

- **Current:**
Applies current layer on the selected object for offsetting.
- **Source:**
Applies source layer on the selected object for offsetting.

Related References?

- [Commands about Curved Objects](#)

Related Concepts?

- [About Offsetting Objects](#)

Related Tasks?

- [To Offset Objects](#)

Other versions with matching text

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

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