

CHAMFER Command — GstarCAD 2027

4180 kbadmin September 25, 2026 [Command Reference](#) 0 9

CHAMFER Command

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Commands > C Command > CHAMFER

Function Description:

This command is used to create chamfers for selected objects. When creating a chamfer, the specified distance and angle are applied on it by selection order.

Users can chamfer lines, polylines, rays and Xlines. They can also chamfer 3D solids and surfaces. To create a chamfer for mesh objects, they should be firstly converted to solid or surface.

Command Access:

Menu: Modify > Chamfer

Ribbon: Home > Modify > Chamfer

Toolbar:

Modify > Chamfer

Command: CHAMFER

Alias: CHA

Command Prompts:

Select first line or [Undo/Polyline/Distance/Angle/Trim/Method/Multiple]:

First line

Specify the first of two lines required to define a 2D chamfer, or the edge of 3D solid to chamfer.

If selected straight lines or polylines, they will adjust their length to form

chamfer edge. When selecting objects by pressing SHIFT key, users can use 0 to instead current chamfer distance.

If selected 2D polylines, they must be adjacency or separated by at most one line. If they are separated by another polyline, the CHAMFER command will create a chamfer and delete the separation line.

- Enter surface selection option [Next/OK (current)] <OK>:
Enter O
or press ENTER to specify the selected surface to be base. Enter N to select an adjacent surface. After selecting base and chamfer distance, users should select edges on the base to be chamfered. They can also select all edges at one time.
- Select second line or shift-select to apply corner:
Select the second object or line segment of a 2D polyline to define the chamfer. You can also hold down the Shift key before selecting the second object or line segment of a 2D polyline to extend or trim the selected objects to form a sharp corner. While Shift is held down, a temporary value of zero is assigned to the current chamfer distance and angle values.

Undo

Restore the last operation performed in command.

Polyline

Chamfer the whole 2D polyline.

When polylines are intersecting, selecting

“Polyline” option can chamfer each polylines at each vertex. Chamfering segment and previous Multiple lines compose a new polyline. If the component line of polyline is too short to chamfer, it will be not chamfered.

Distance

Specify the distance from chamfer to specified endpoint. If the two distances are all specified to 0, it will extend or trim the selected two lines and join the two lines at one point.

Angle

Specify chamfer distance by chamfer distance of the first line and angle of the second line.

Trim

This option is used to control whether to trim specified line to the end point of chamfer.

Note:

the “Trim” option can specify TRIMMODE system variable to 1; the “No trim” option can specify TRIMMODE system variable to 0. If the TRIMMODE system variable is specified to 1, the CHAMFER command will trim intersection line to the end point of chamfer. If the second selected line is not intersected with the first one, it will be extended. If the TRIMMODE system variable is specified to 0, it will create chamfer rather than trim selected line.

mEthod

This option is used to control methods of creating chamfer, by two distances or one distance and one angle. **## Multiple**

Create chamfers for more than one set of objects.

Related References?

- [Commands about Creating Fillets and Chamfers](#)

Related Concepts?

- [About Chamfers](#)

Related Tasks?

- [To Work With Chamfers](#)

Other versions with matching text

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

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