

FILLET Command — GstarCAD 2027

3063 kbadmin September 25, 2026 [Command Reference](#) 0 11

FILLET Command

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Commands > F Command > FILLET

Function Description:

This command is used to fillet or round the edges of two 2D objects, or the adjacent faces of a 3D solid. The fillet is tangent to the selected two lines and the two lines are trimmed to meet the endpoints of the arc. Users can specify the radius to 0 to create a fillet for a sharp angle. Users can create fillets for arcs, circles, ellipses, elliptical arcs, lines, polylines, splines, Xlines and for 3D solids and surfaces. If users want to create fillets for mesh objects, they should firstly convert them into solids or surfaces.

Command Access:

Menu: Modify > Fillet

Ribbon: Home > Modify > Fillet

Toolbar: Modify > Fillet

Command: FILLET

Alias: F

Command Prompts:

Select first object or
[Undo/Polyline/Radius/Trim/Invert/Multiple]:

Select first object

Selects the first of two objects to define a fillet or selects one edge of a 3D solid or surface.
The length of objects will be adjusted to create fillets when selecting lines, arcs, or polylines.
If you press ENTER when selecting an object, the current radius of fillet will be 0.

Users can select two line segments on a polyline to create a fillet. Those two lines can be adjacent or separated by another line. If they are separated by another line, the created fillet will be created and the separating line will be deleted.

When creating fillets between circle and arc, there may be many kinds of fillets. When a circle is selected, the circle is not trimmed, and the fillet drawn meets the circle smoothly.

Users can separately select edges of 3D solid to create fillets.

Undo

Reverses the previous action in the command.

Polyline

Inserts a fillet at each vertex of a 2D polyline where two straight line segments meet.

If the selected two lines are separated by an arc segment, the FILLET command will create a fillet and delete the arc.

Radius

Specifies the radius for fillet, changing this value does not have effect on existing fillets.

Trim

Controls whether the selected objects are trimmed to meet the endpoints of the fillet.

Multiple

Creates fillets for more than one set of objects.

Edge

When the edge of a 3D solid is selected, you can select multiple edges to fillet. Press Enter to end selection.

If users select three or more edges that converge at a vertex to form the corner of a box, the vertices are blended to form part of a sphere if the three incident fillets have the same radius.

Chain

Changes selection mode from single selection to chain selection. Chain selection is to continuously select tangency edges.

Radius

Sets the radius of the fillet.

Related References?

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

Related Tasks?

- [To Work With Fillets](#)

Other versions with matching text

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

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