

FILLETEDGE Command — GstarCAD 2027

3064 kbadmin September 25, 2026 [Command Reference](#) 0 9

FILLETEDGE Command

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Function Description:

This command is used to round and fillet the edges of solid objects.

Command Access:

Menu: /

Ribbon: /

Toolbar: /

Command: FILLETEDGE

Alias: /

Command Prompts:

Select an edge or
[Chain/Loop/Radius]:

Select an edge

Specifies one or more edges on the same solid body to fillet.
After pressing Enter, you can drag the fillet grip to specify a radius, or use the Radius option.

Chain

Specifies more than one edge when the edges are tangent to one another.

Loop

Specifies a loop of edges on the face of a solid. For any edge, there are two possible loops. After selecting a loop edge you are prompted to *Accept* the current selection or choose the *Next* loop.

Radius

Specifies a radius value.

Other versions with matching text

[2022](#) · [2023](#) · [2024](#) · [2025](#) · [2026](#)

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