

Filletedge command

2744 GstarCAD MY /KW April 15, 2022 [3D CAD](#) 0 1492

Function Description :

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

Command Access :

Icon :

Menu : /

Ribbon : /

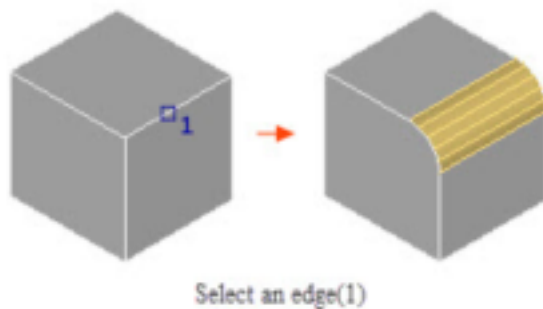
Toolbar: /

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

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