

# Filletedge command

2744 GstarCAD MY /KW April 15, 2022 [3D CAD](#) 0 1711

## Function Description :

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

## Command Access :

**Icon :**

**Menu :** /

**Ribbon :** /

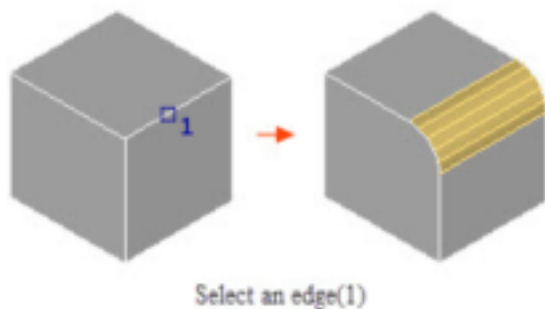
**Toolbar:** /

**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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