

# Filletedge command

2744 GstarCAD MY /KW April 15, 2022 [3D CAD](#) 0 8730

## Function Description :

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

## Command Access :

**Icon :**

**Menu :** /

**Ribbon :** /

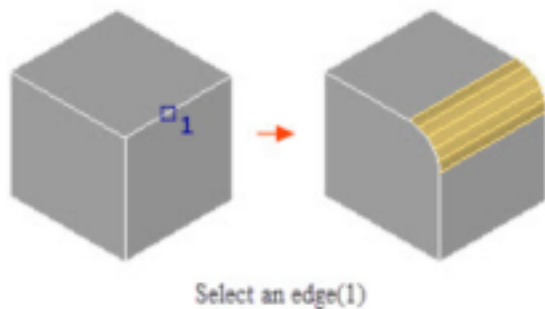
**Toolbar:** /

**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

---

Related tutorial video:

- [View command](#) 00:25
- [Menus and Shortcut Menus](#) 00:37
- [Unisolateobjects command](#) 00:37
- [Qnew command](#) 00:48
- [Status bar related setting](#) 00:52
- [Box command](#) 01:03
- [Union command](#) 01:50
- [Xedges command](#) 02:03
- [Filletedge command](#) 02:08
- [Vscurrent command](#) 02:45
- [Extrude command](#) 03:27
- [Rectang command](#) 04:22
- [Move command](#) 04:45
- [Line command](#) 05:06
- [ARC command](#) 05:16
- [Trim command](#) 05:23
- [Join command](#) 05:46
- [Convtsurface command](#) 06:35
- [Slice command](#) 06:51
- [Solidedit](#) 08:32

Above video sources from [TUTORIAL - GETTING STARTED WITH GSTARCAD](#), enrol now for free.

Online URL: <https://www.kb2.gstarcad.com.my/article.php?id=2744>