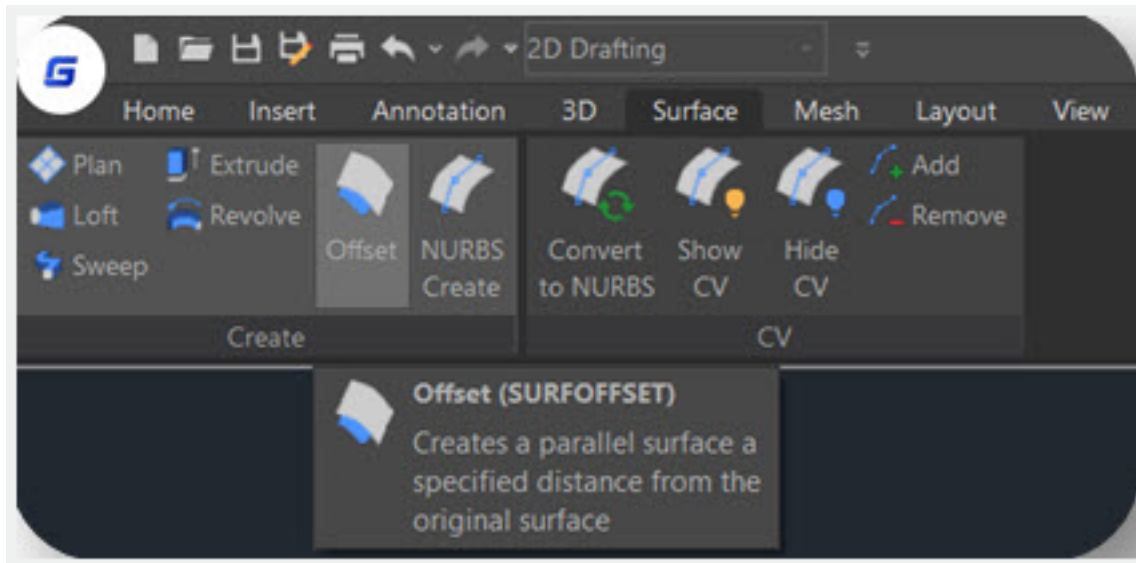# SURFOFFSET command

2373  GstarCAD MY /KW  August 23, 2021  [CAD Commands](#)   0  1056

The **SURFOFFSET** command or click Ribbon – Surface -SURFOFFSET, to create a parallel surface at a specified distance from the original surface.
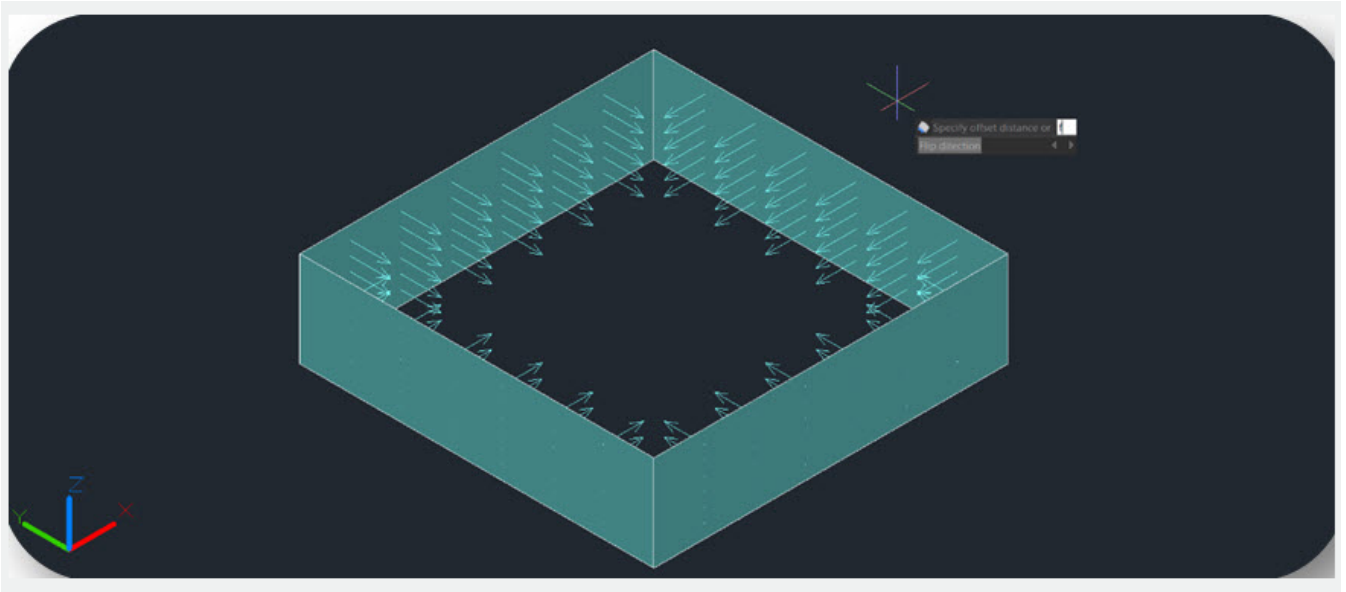
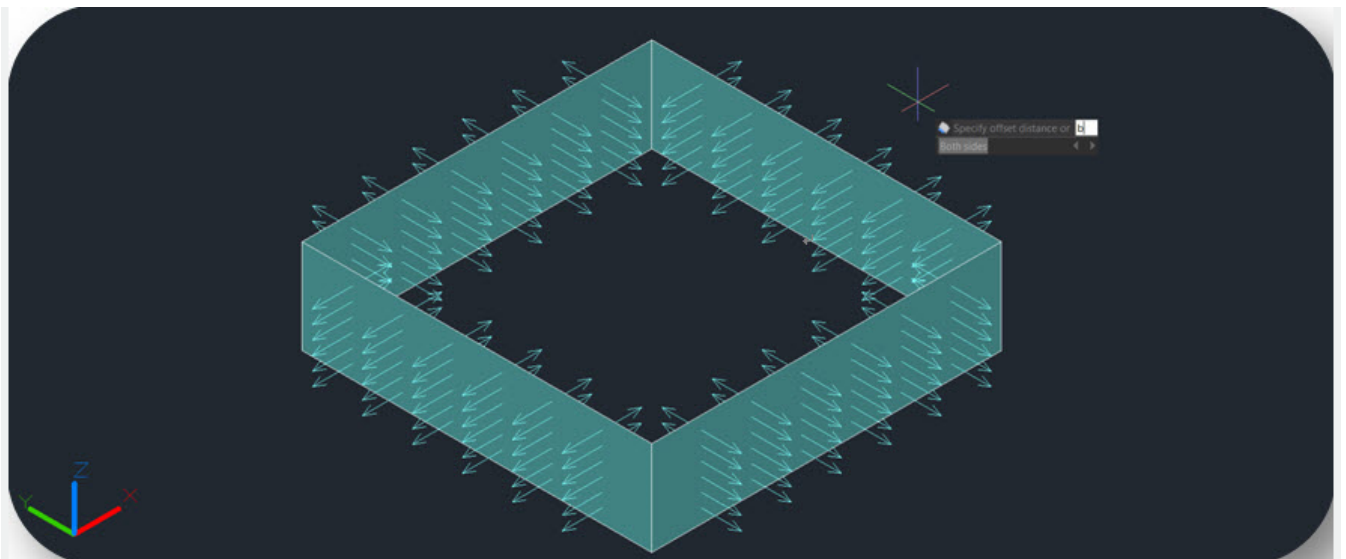**Command Access** :

**Command** : SURFOFFSET



After selecting the surface and pressing enter, you will get 4 options (Flip direction, Both sides, Solid, Expression) in the command line and dynamic input box.

You can **Offset the surface in one direction** (the arrows will show on one side of the surface) by entering a distance or select the point on the drawing
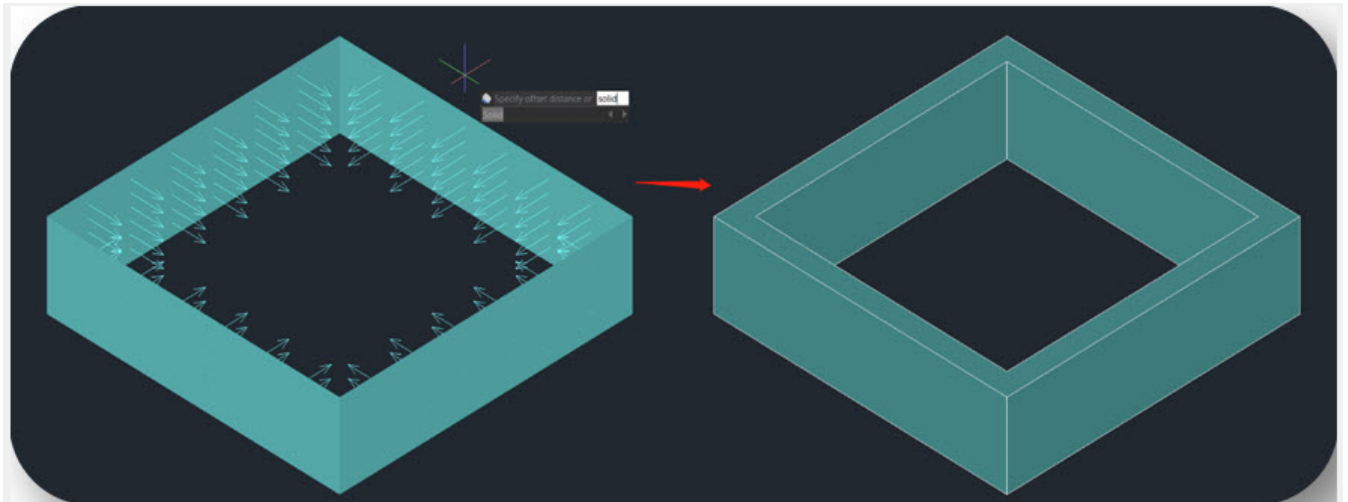
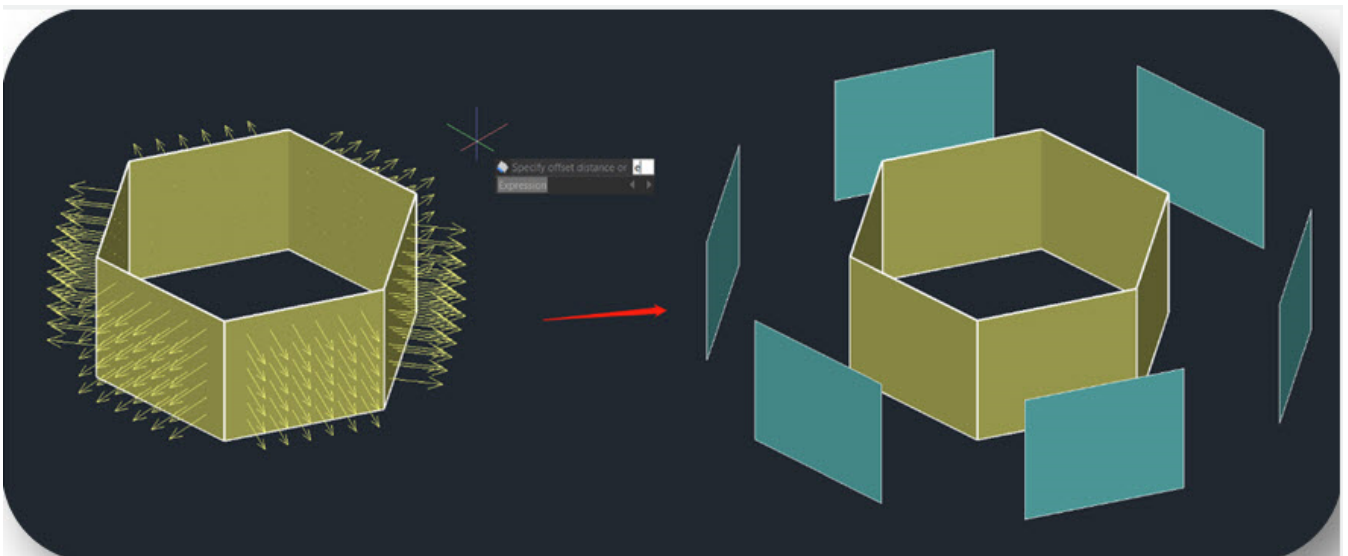If you type "**Flip direction"**, the arrows will flip to another side.

If you type "**Both sides**", you can **Offset the surface in both directions**(the arrows will show on both sides of the surfaces)



If you type "**Solid**", you can Generate the surface to a solid after the offset

If you type "**Expression**", you can enter a formula or equation to specify the distance of the surface offset.



**SURFACEASSOCIATICITY**: System Variable, when the value is 1, Expression option will be displayed. This option does not update the offset distance default value the next time you enter the command.

**SURFACEMODELINGMODE**: System Variable, when the value is 1, the surface will be a NURBS surface.

Online URL: https://www.gstarcad.com.my/knowledge/article/surfoffset-command-2373.html