

# DONUT command

2097 GstarCAD MY /KW April 14, 2022 [CAD Commands](#) 0 2380

The **DONUT** command is used to create a solid circle or a wide ring.

## Command Access:

**Ribbon** : Home > Draw > Polyline > Donut

**Menu** : Draw > Donut

**Command** : DONUT

## Command Prompts:

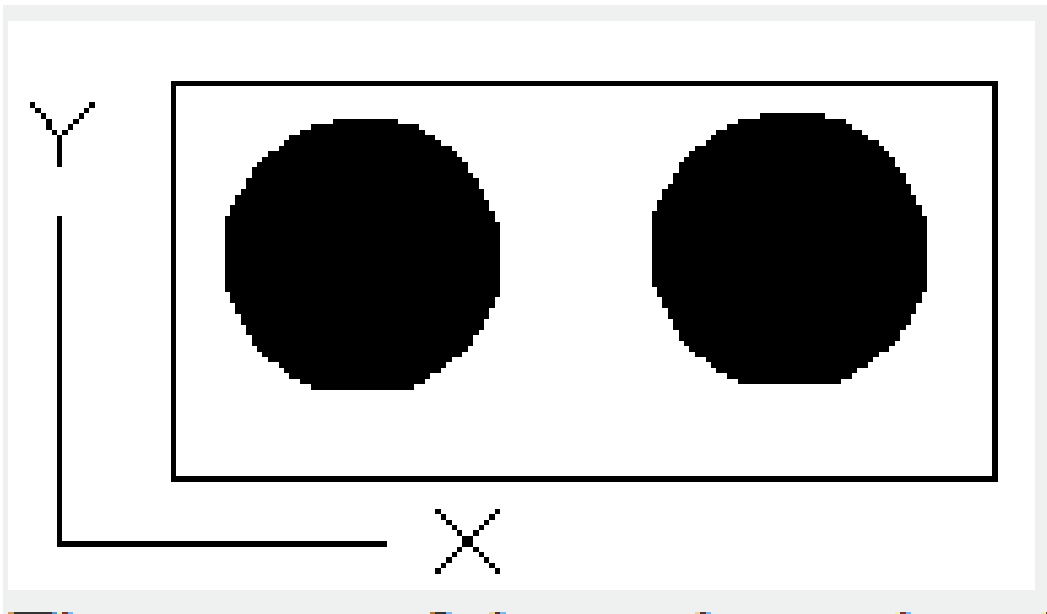
Specify inside diameter of donut <0.5000>:

Specify outside diameter of donut<1.000>:

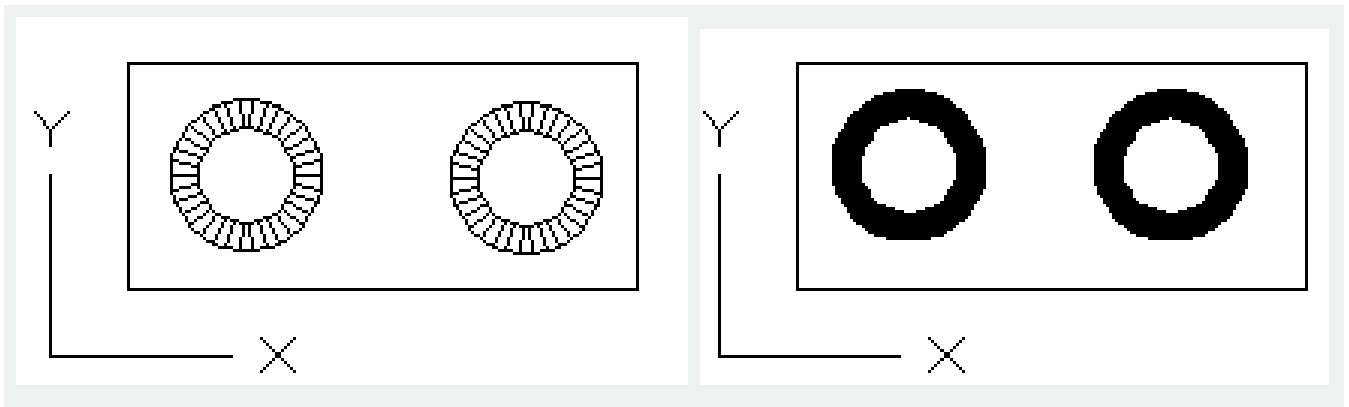
Specify center of donut or:

## Function Description:

The ring is composed by two arc polylines; they connect end to end and form a circle. The width of ring is determined by the inside diameter and outside diameter. To create a solid circle, users could specify the inside diameter to 0.



The center of donut determines its location. After specifying diameter of donut, it will prompt to specify location of donut. Users could use **FILL** command to decide the current hatch style of donut.



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