

# CIRCLE command

2032 GstarCAD MY /KW April 14, 2022 [CAD Commands](#) 0 1742

The **CIRCLE** command is used to create a circle.

## Command Access:

**Ribbon** : Home > Draw > Circle

**Menu** : Draw > Circle

**Command** : CIRCLE

## Command Prompts:

Specify center point for circle or [3P/2P/Ttr/Arc/Multiple/Concentric]:

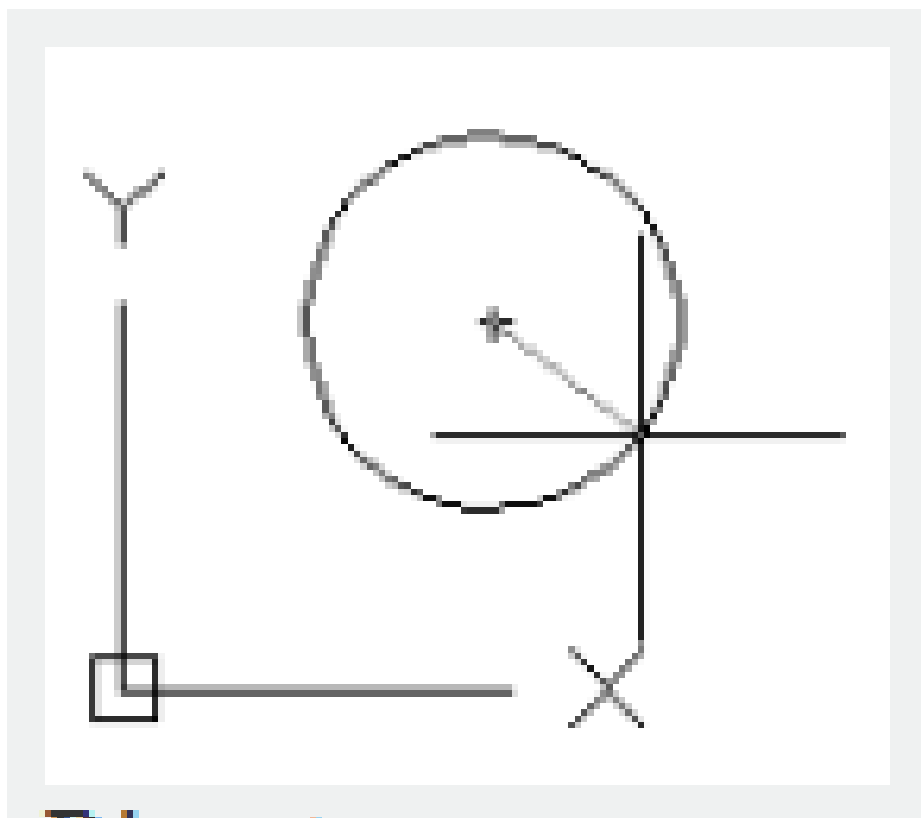
## Relative Glossary:

### Center point:

Create a circle by specifying its center point and diameter (or radius).

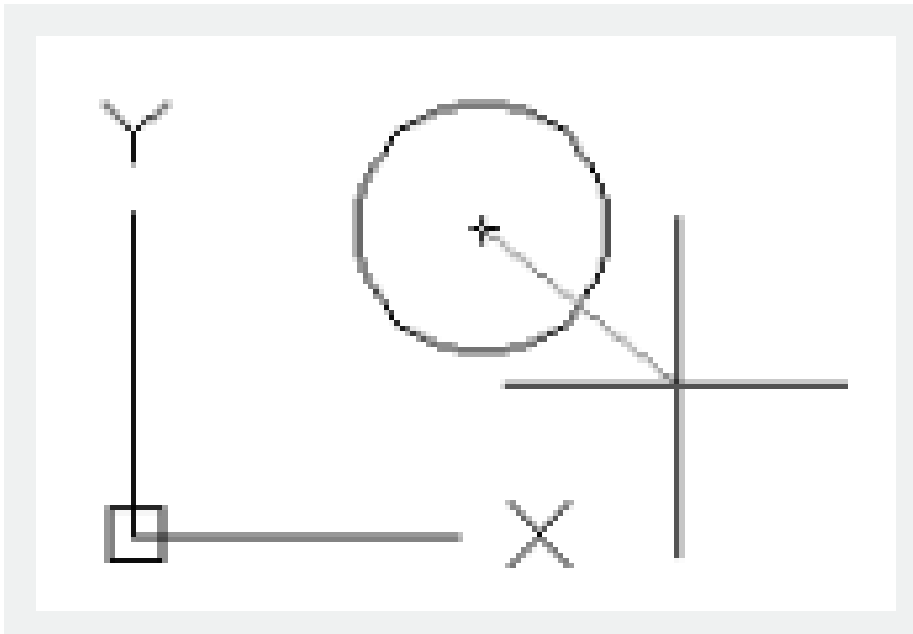
### Radius:

Define circle radius by inputting value or specifying a point.



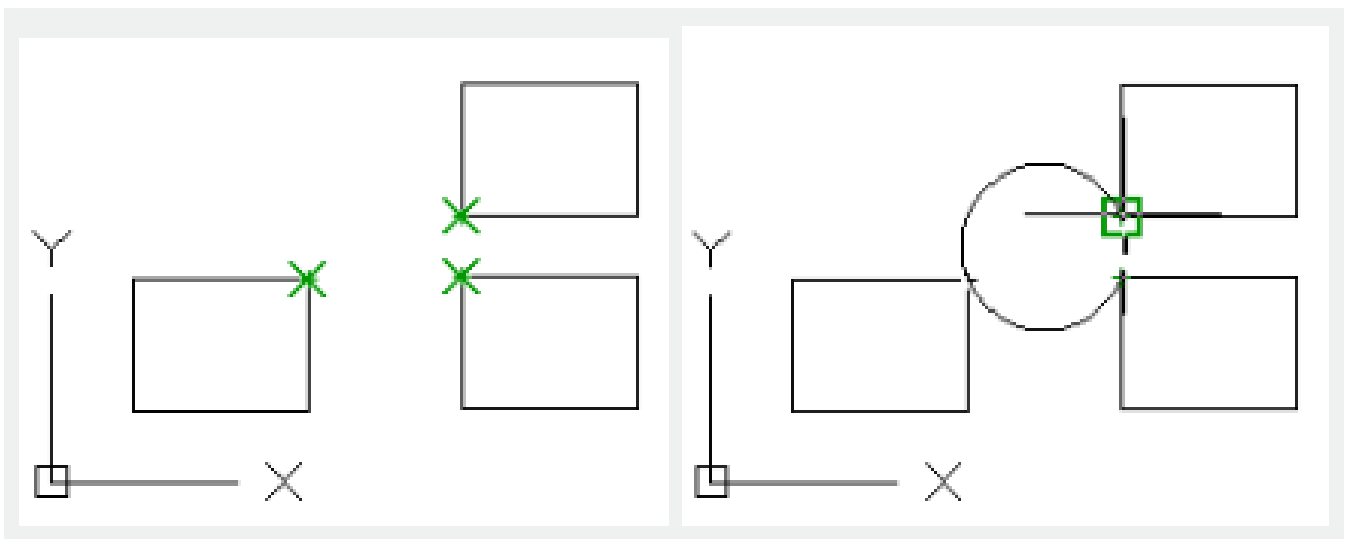
### Diameter:

Define circle diameter by inputting value or specifying the second point.



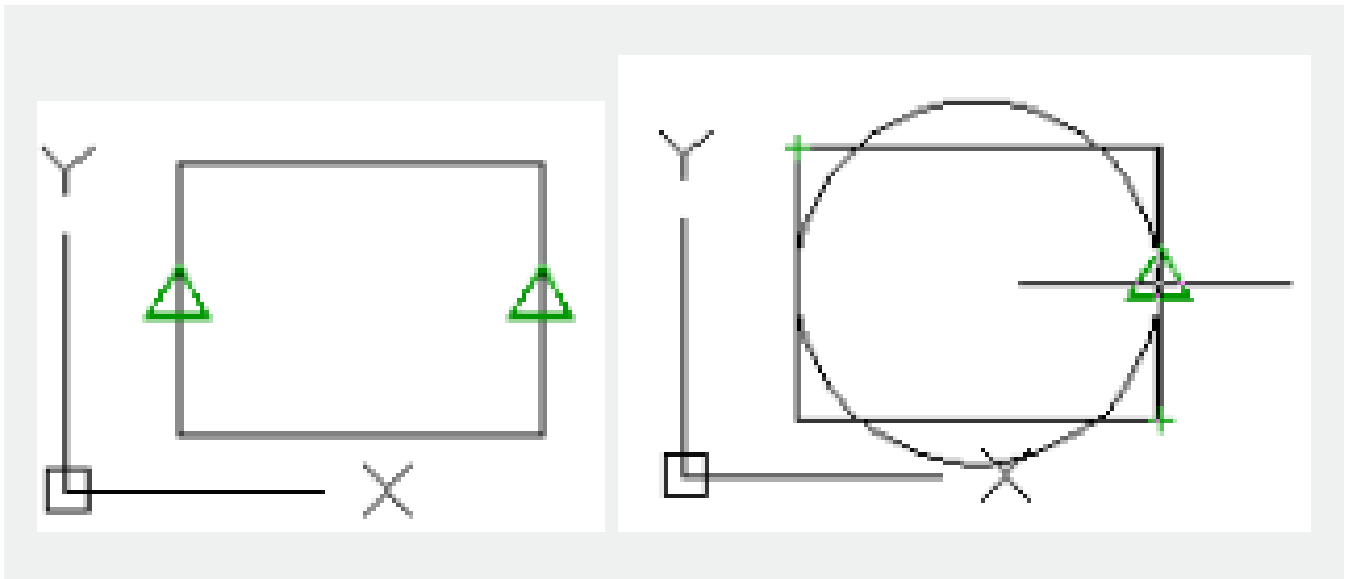
**3P:**

Create a circle by specifying 3 points on it.

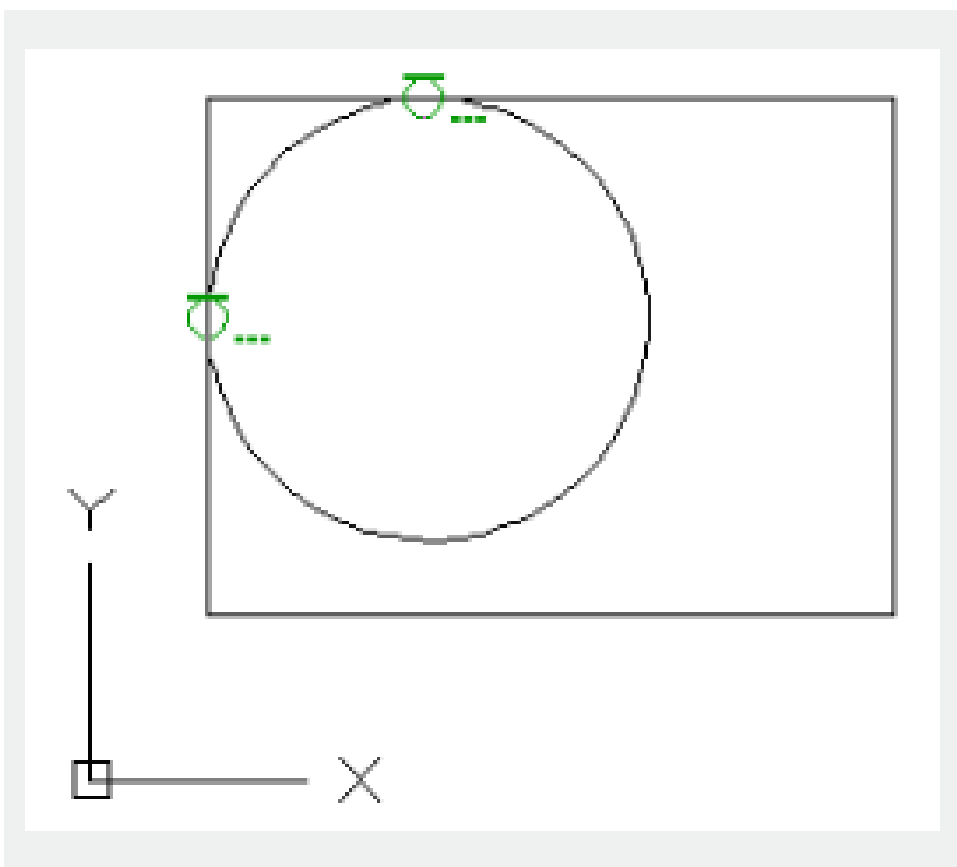


**2P:**

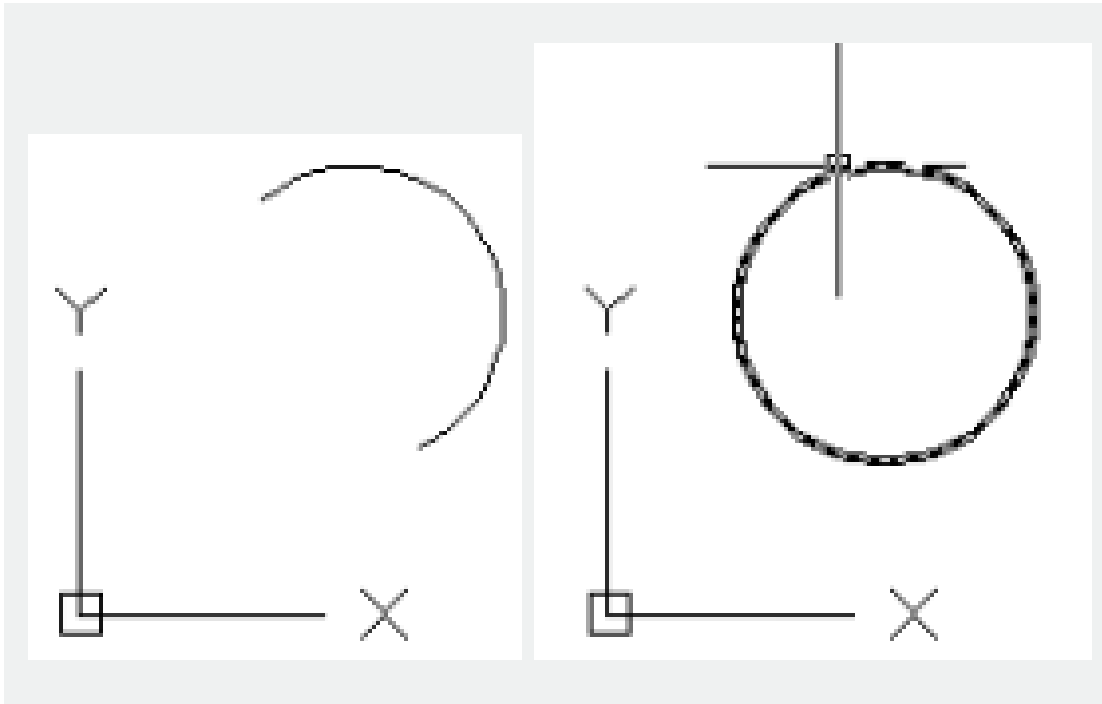
Create a circle by 2 endpoints of its diameter.



**Ttr:**  
Create a circle by specified diameter and two tangent objects.

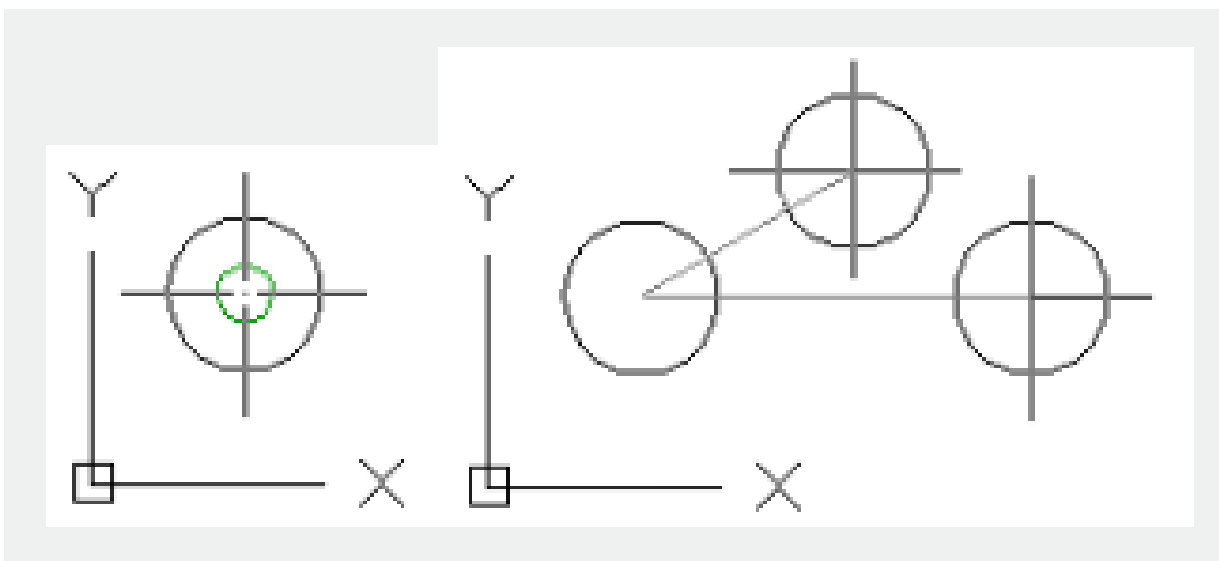


**Arc:**  
Create a circle by an existing arc.



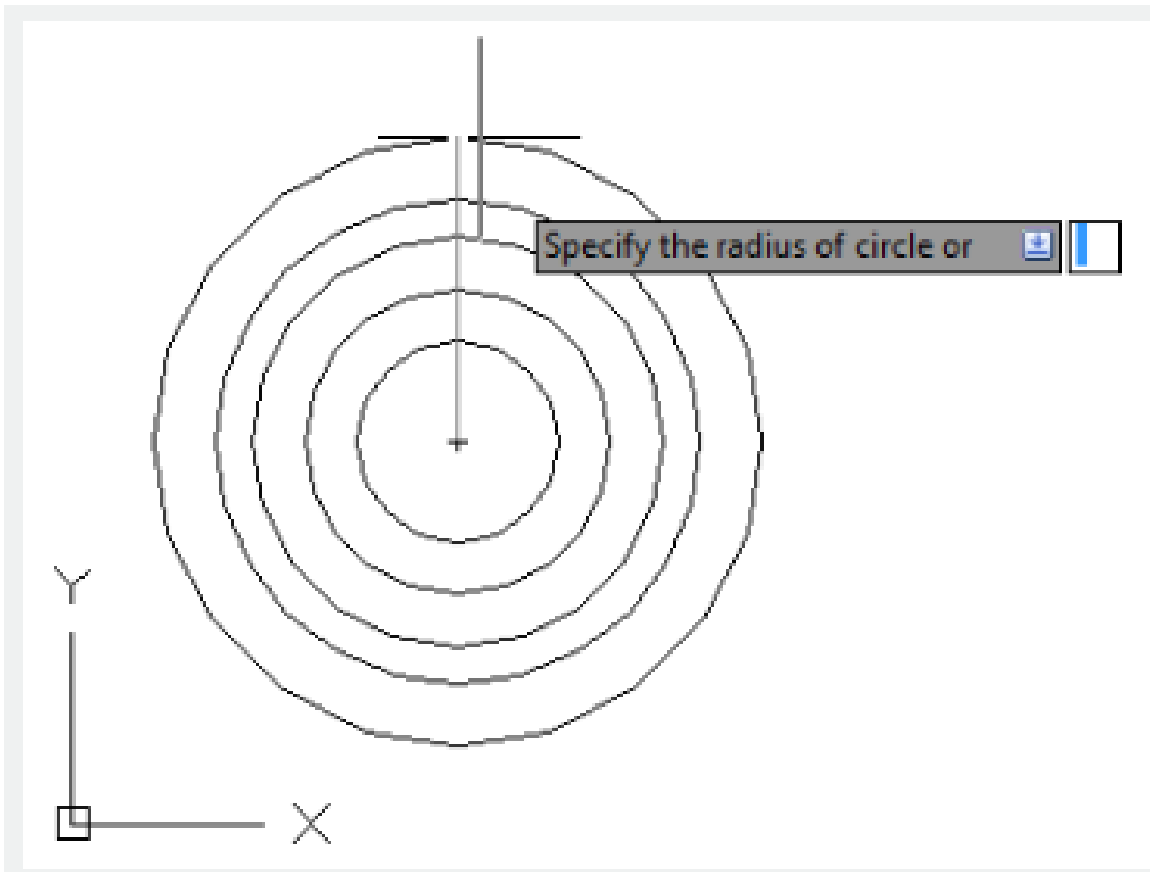
**Multiple:**

Create more than one circle:



**Concentric:**

Create more than one concentric circle.



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