

CIRCLE command

2032 GstarCAD MY /KW April 14, 2022 [CAD Commands](#) 0 1527

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]:

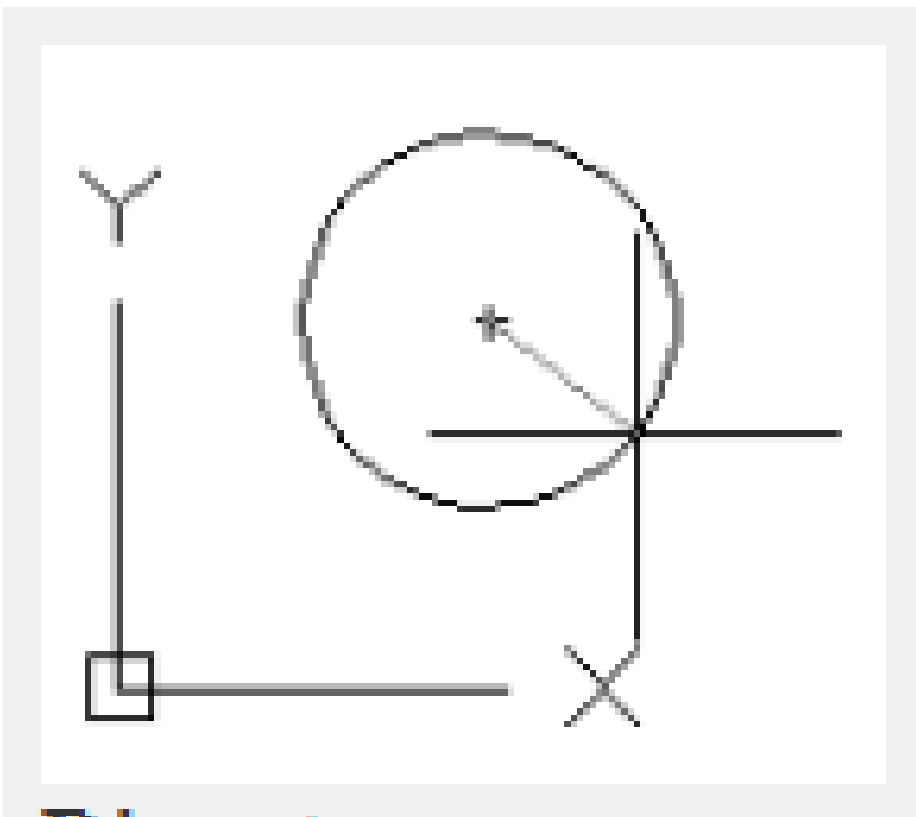
Relative Glossary:

Center point:

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

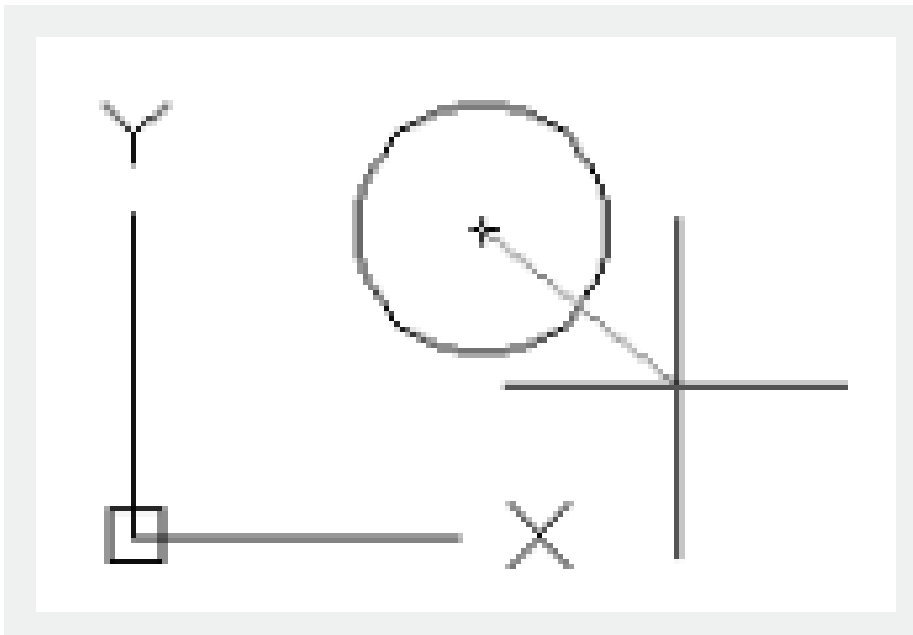
Radius:

Define circle radius by inputting value or specifying a point.

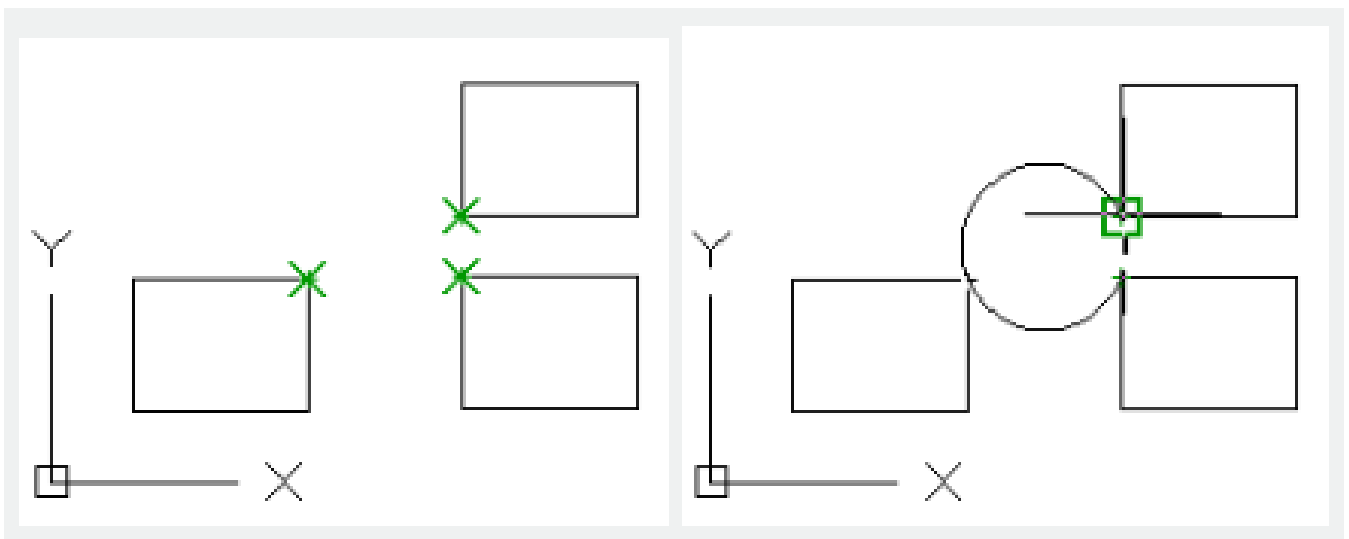


Diameter:

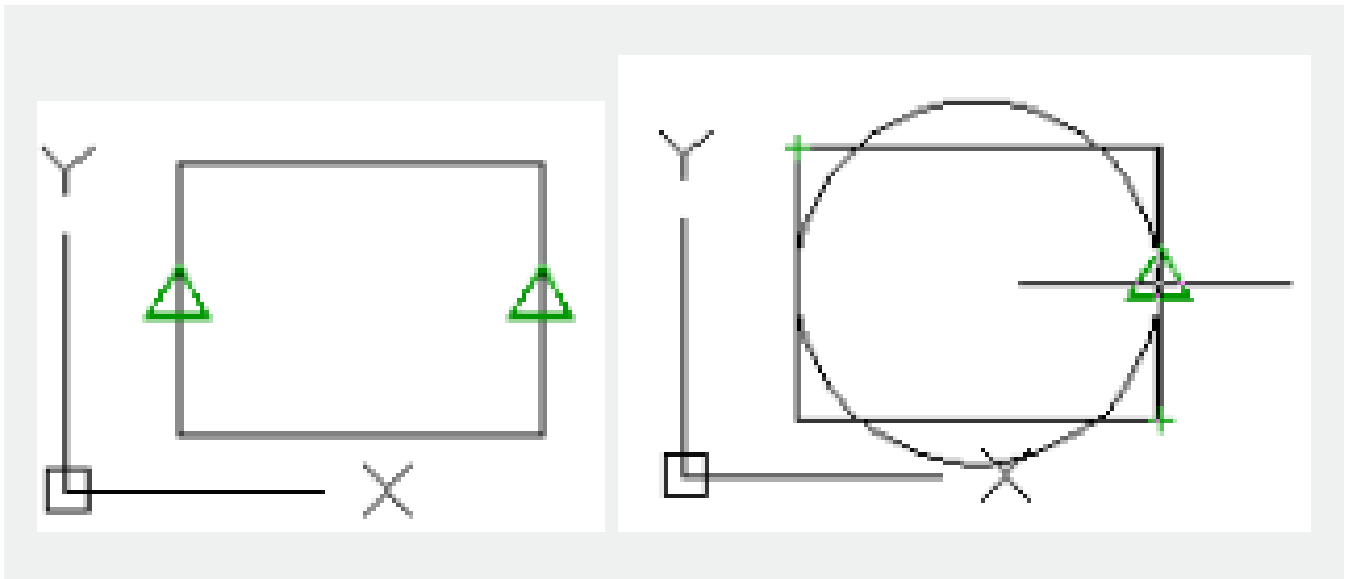
Define circle diameter by inputting value or specifying the second point.



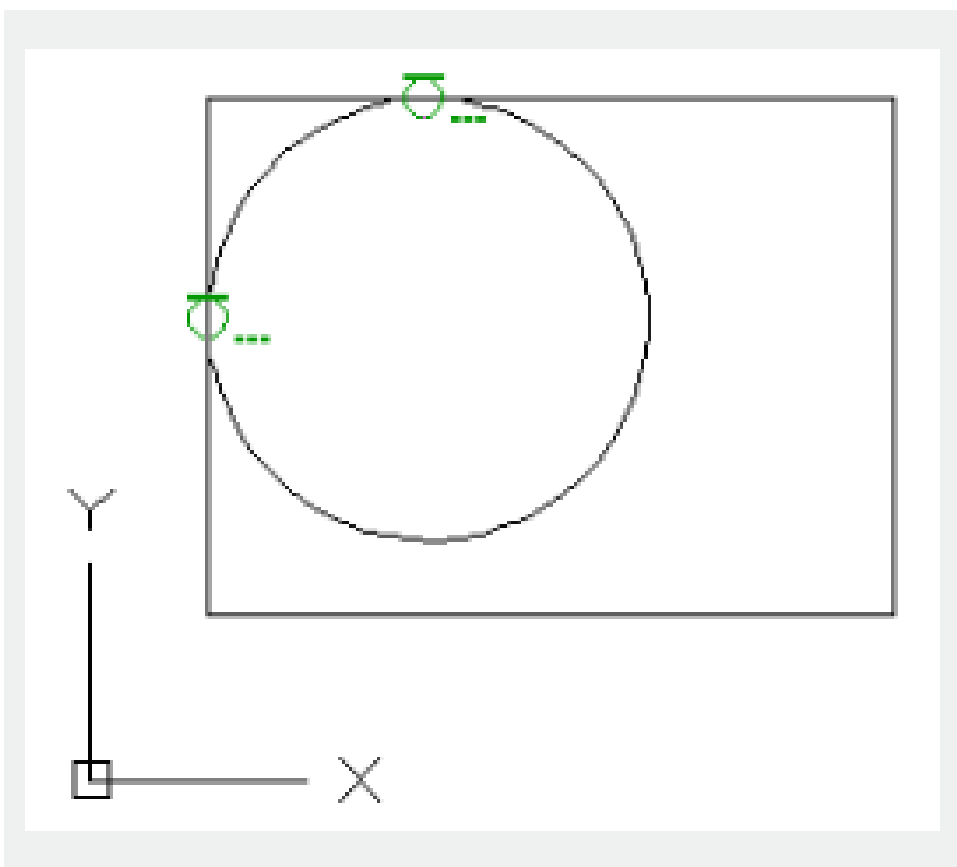
3P:
Create a circle by specifying 3 points on it.



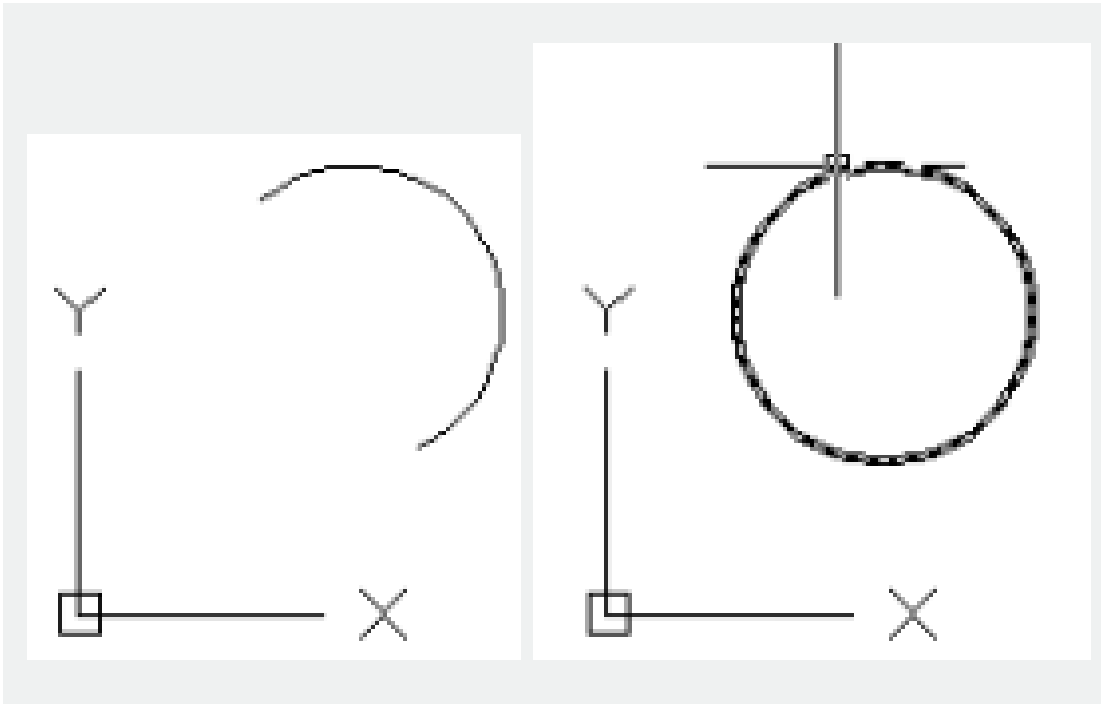
2P:
Create a circle by 2 endpoints of its diameter.



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

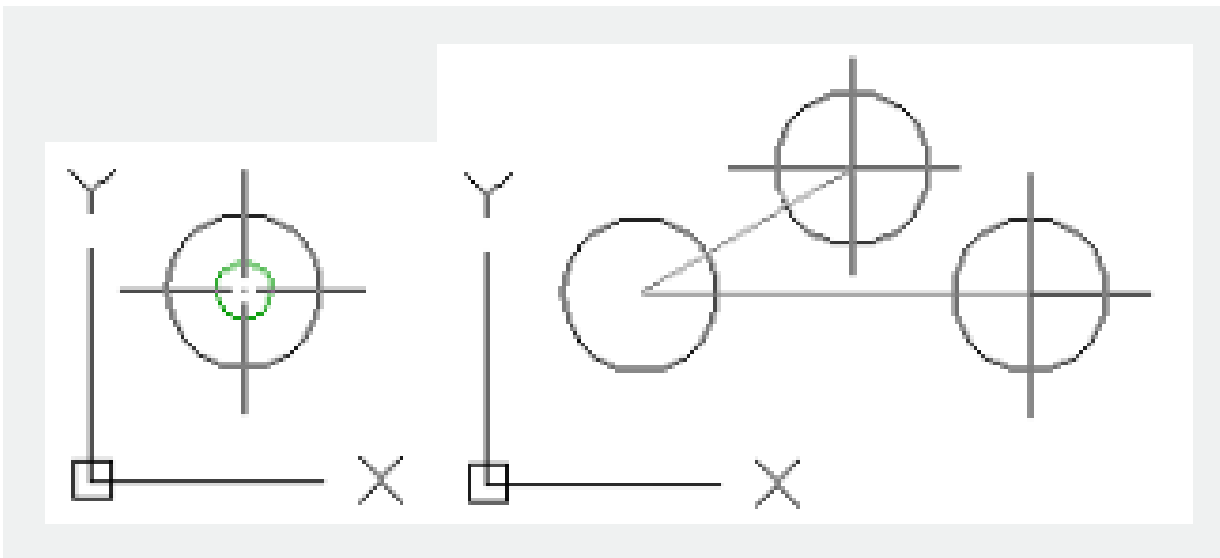


Arc:
Create a circle by an existing arc.



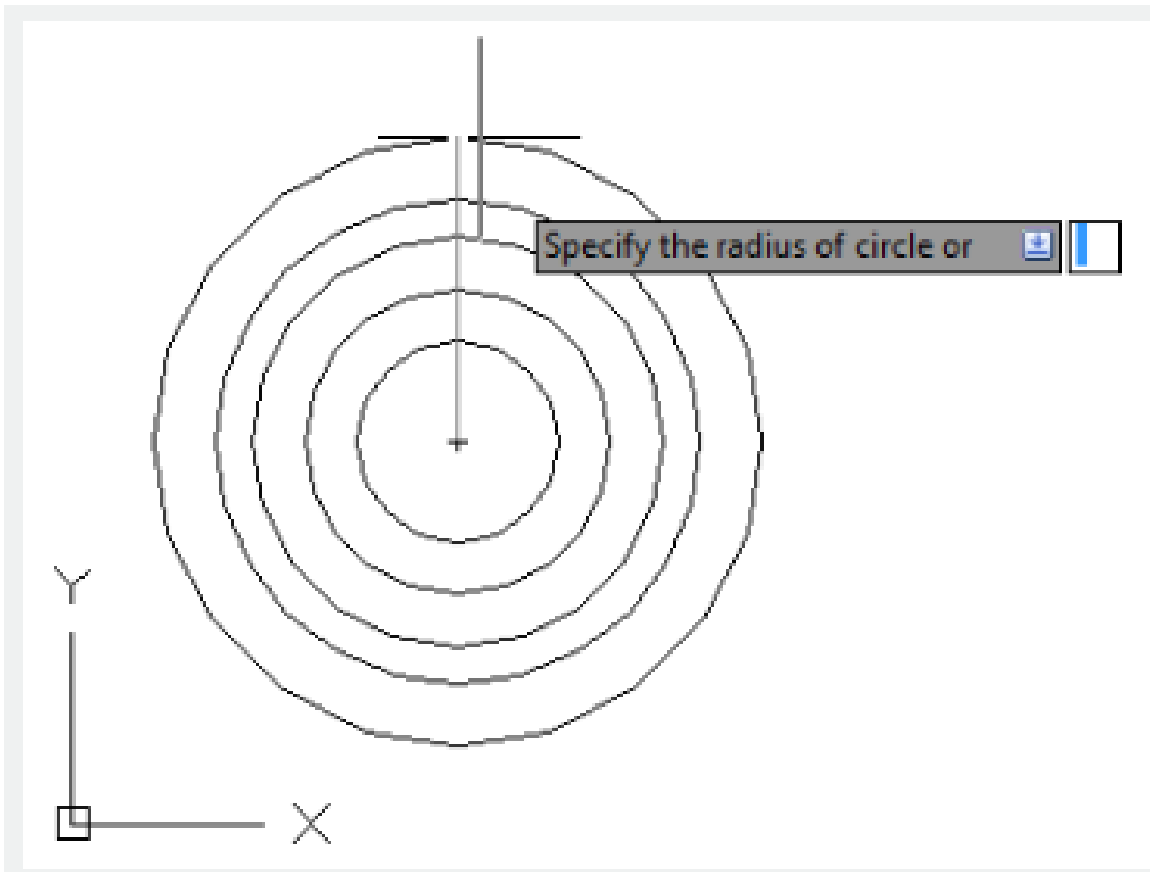
Multiple:

Create more than one circle:



Concentric:

Create more than one concentric circle.



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