

SPHERE Command — GstarCAD 2027

4081 kbadmin September 25, 2026 [Command Reference](#) 0 7

SPHERE Command

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Commands > S Command > SPHERE

Function Description:

This command is used to create a 3D solid sphere.

Users can create a sphere by specifying its center and radius. They can also control its 3D smoothness of colored or shaded visual style by setting the FACETRES system variable.

Command Access:

Menu: Draw > Modeling > Sphere

Ribbon: 3D > Modeling > Sphere

Toolbar: Modeling > Sphere

Command: SPHERE

Alias: /

Command Prompts:

Specify center point or [3P/2P/Ttr]:

Center point

Specifies the sphere center point.

After specifying the sphere center point, the central axis of the sphere is parallel to the Z axis of the current UCS and the latitude lines are parallel to the XY plane.

- **Radius:**
Define the sphere radius.
- **Diameter:**
Define the sphere diameter.

3p

Defines the circumference of a sphere by 3 points at any position in 3D space, these 3 points can also define a circumferential plane.

2p

Defines the circumference of a sphere by 2 points at any position in 3D space and the Z value of the first point defines the local plane.

Ttr

Defines a sphere by the radius and two objects that tangent with it. The specified tangent points are projected onto the current UCS. The selected objects to be tangent must be arcs, circles, or lines.

Related References?

- [Commands about Creating 3D Solids](#)
- [Commands about Working with 3D Models](#)

Related Tasks?

- [To Create Boxes](#)
- [To Create Cones](#)
- [To Create Cylinders](#)
- [To Create Spheres](#)
- [To Create Pyramids](#)

Other versions with matching text

[2023](#) · [2024](#) · [2025](#) · [2026](#)

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