Draw Ellipses

2507 GstarCAD MY /KW November 9, 2021 <u>Workflow</u> 0 756 You can draw an <u>ELLIPSE</u> by using two axes to specify its length and width .

You can choose one of the following methods to draw a ellipse:

- Center (Specify center of ellipse, Specify endpoint of axis, Specify distance to other axis or [Rotation])
- Axis (Specify axis endpoint of ellipse, Specify other endpoint of axis, Specify distance to other axis or [Rotation])



If you turn on Isometric Snap (<u>SNAPSTYL</u>) in the Drafting Settings dialog box, ELLIPSE has new option: Isometric. You can use ellipses to represent isometric circle's to simulate 3D on isometric plane.

Online URL: https://www.gstarcad.com.my/knowledge/article/draw-ellipses-2507.html