

SOLDRAW Command — GstarCAD 2027

4075 kbadmin September 25, 2026 [Command Reference](#) 0 1

SOLDRAW Command

[GstarCAD 2027](#) · [Command Reference](#)

Commands > S Command > SOLDRAW

Function Description:

This command is used to generate profiles and sections in layout viewports.

The SOLDRAW command can only be used in viewports created by the SOLVIEW command. Users can use the SOLDRAW command to show solid profiles and visible edges and hide lines in new viewports, and then project them onto the plane that perpendicular to the viewing direction. The slicing plane creates

profiles and edges for solids. According to sectional views, users can create section hatched by setting the current values of the HPNAME, HPSCALE, HPANG system variables.

The profile drawings and sections in the selected viewport are removed, it automatically regenerates new profile drawings

and sections. All layers are frozen except layers for displaying profiles and sections.

Warning: Please do not place permanent drawings on the layer named by "-VIS", "-HID", "-HAT". The information of these

drawings is changed when running the SOLDRAW command. Users can quit the viewport created by the SOLDRAW command by UNDO command.

Command Access:

Menu: Draw > Modeling > Setup > Drawing

Ribbon: 3D > Settings > Drawing

Toolbar: /

Command: SOLDRAW

Alias: /

Command Prompts:

Select object:

Related References?

- [Commands about Creating and Modifying Flattened Views](#)

Related Tasks?

- [To Create a Flattened View of a 3D Model](#)

Other versions with matching text

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

Online URL: <https://www.kb2.gstarcad.com.my/article.php?id=4075>