

SWEEP Command — GstarCAD 2027

4099 kbadmin September 25, 2026 [Command Reference](#) 0 15

SWEEP Command

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

Commands > S Command > SWEEP

Function Description:

This command is used to create a 3D solid or surface by sweeping a 2D object, 3D object or subobject. Users can create a solid or surface by sweeping closed or open surface (planar or non-planar) along a closed or open path. They can select open surface to create another surface and select closed surface to create solid or another surface (depending on specified mode).

The following objects can be used to sweep:

2D or 3D splines, 2D polylines, 2D solids, edge subobjects of 3D solid, arcs, circles, ellipses, elliptical arcs, lines, regions, solids, edge subobjects of solid, surfaces and meshes, wide lines.

The following objects can be used as a sweep path:

2D or 3D splines, 2D or 3D polylines, edge subobjects of 3D solid, helixes, arcs, circles, ellipses, elliptical arcs, lines.

Command Access:

Menu: Draw > Modeling > Sweep

Ribbon: 3D > Modeling > Extrude > Sweep

Toolbar: Modeling > Sweep
Command: SWEEP
Alias: /

Command Prompts:

Select objects to sweep or
[MOde]:

Related References?

- [Commands about Constructing 3D Solids and Surfaces from 2D Geometry](#)
- [Commands about Creating Procedural Surfaces](#)
- [Commands about Working with 3D Models](#)

Related Tasks?

- [To Create a Solid by Sweeping an Object Along a Path](#)
- [To Create a Surface by Sweeping an Object Along a Path](#)

Other versions with matching text

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

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