SWEEP

Creates a 3D solid or surface by sweeping a 2D curve along a path

**Toolbar:** Modeling

**Menu:** At the Command prompt, enter sweep.

**Command entry:** `sweep`

**Dashboard:** 3D Make panel, Sweep

With the SWEEP command, you can create a new solid or surface by sweeping an open or closed planar curve (profile) along an open or closed 2D or 3D path. SWEEP draws a solid or surface in the shape of the specified profile along the specified path. You can sweep more than one object, but they all must lie on the same plane.

You can use the following objects and paths when creating a swept solid or surface:

<table>
<thead>
<tr>
<th>Objects that Can Be Swept</th>
<th>Objects that Can Be Used as a Sweep Path</th>
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</thead>
<tbody>
<tr>
<td>Line</td>
<td>Line</td>
</tr>
<tr>
<td>Arc</td>
<td>Arc</td>
</tr>
<tr>
<td>Elliptical arc</td>
<td>Elliptical arc</td>
</tr>
<tr>
<td>2D polyline</td>
<td>2D polyline</td>
</tr>
<tr>
<td>2D spline</td>
<td>2D spline</td>
</tr>
<tr>
<td>Circle</td>
<td>Circle</td>
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<tr>
<td>Ellipse</td>
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<tr>
<td>Planar 3D face</td>
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<tr>
<td>2D solid</td>
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<tr>
<td>Trace</td>
<td>Helix</td>
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<tr>
<td>Region</td>
<td>Edges of solids or surface</td>
</tr>
</tbody>
</table>
**Note** You can select faces and edges on solids or surfaces by pressing and holding CTRL, and then selecting these subobjects.

The **DELOBJ** system variable controls whether the profile(s) and sweep path are automatically deleted when the solid or surface is created or whether you are prompted to delete the profile(s) and path.

You can select the objects to sweep before you start the command.

- **Current wire frame density:** ISOLINES=4
- **Select objects to sweep:** *Use an object selection method and press ENTER when you finish*
- **Select sweep path or [Alignment/Base Point/Scale/Twist]:** *Select a 2D or 3D sweep path, or enter an option*

### Alignment

Specifies whether the profile is aligned to be normal to the tangent direction of the sweep path. By default, the profile is aligned.

- **Align sweep object perpendicular to path before sweep [Yes/No] <Yes>:** *Enter no to specify that the profile is not be aligned or press ENTER to specify that the profile is aligned*

**Note** If the profile curve is not perpendicular (normal) to the tangent of the start point of the path curve, then the profile curve automatically aligns. Enter No at the alignment prompt to prevent this.

### Base Point

Specifies a base point for the objects to be swept. If the specified point does not lie on the plane of the selected objects, it is projected onto the plane.

- **Specify base point:** *Specify a base point for the selection set*

### Scale

Specifies a scale factor for a sweep operation. The scale factor is uniformly applied to the objects that are swept from the start to the end of the sweep path.

- **Enter scale factor or [Reference] <1.0000>:** *Specify a scale factor, enter r for the reference option, or press ENTER to specify the default value*

### Reference

Scales the selected objects based on the length you reference by picking points or entering values.

- **Specify start reference length <1.0000>:** *Specify a beginning length from which to scale the selected objects*
- **Specify end reference length <1.0000>:** *Specify a final length to which to scale the selected objects*

### Twist

Sets a twist angle for the objects being swept. The twist angle specifies the amount of rotation along the entire length of the sweep path.

- **Enter twist angle or allow banking for a non-planar sweep path [Bank] <n>:** *Specify an angle value less than 360, enter b to turn on banking, or press ENTER to specify the default angle value*
- **Select sweep path [Alignment/Base point/Scale/Twist]:** *Select a sweep path or enter an option*

Banking specifies whether or not the curve(s) being swept will bank naturally (rotate) along a 3D sweep path.
SWEEP (Quick Reference)

(3D polyline, 3D spline, or helix).