

WEDGE command

2440 GstarCAD MY /KW August 25, 2021 [CAD Commands](#) 0 1122

The **WEDGE** command is used to create 3D solid wedges.

Command Access:

Ribbon : 3D > Modeling > Box Wedge

Menu : Draw> Modeling> Wedge

Command : WEDGE

Command Prompts:

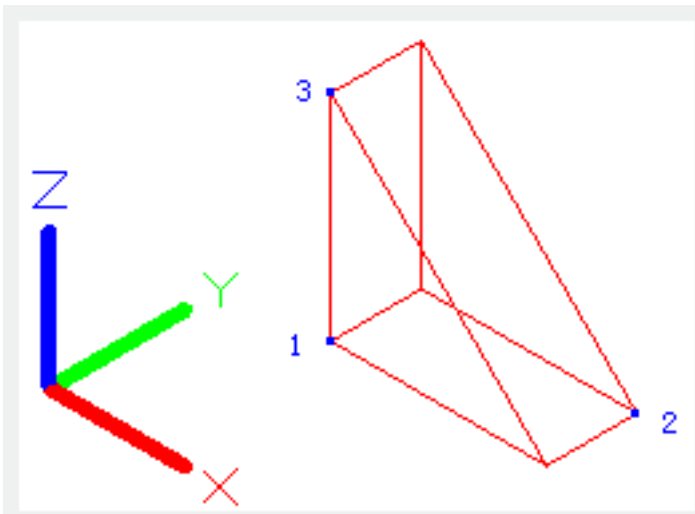
Specify first corner or [Center]:

Specify other corner or [Cube/Length]:

Specify height or [2Point]:

Function Description:

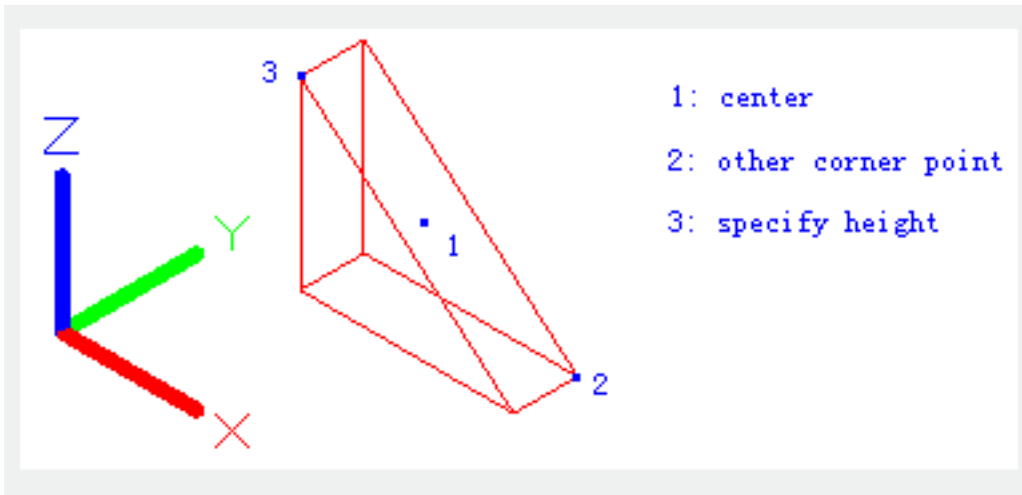
The incline direction is always along the positive direction of X axis in UCS



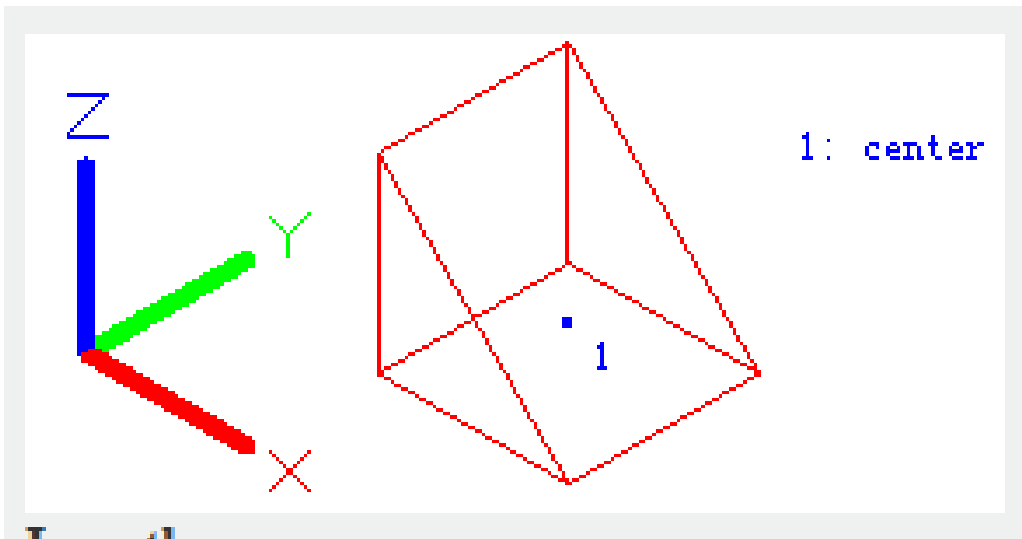
If the Z value of the specified other corner point is different to the first specified corner point, it will not display height prompt. When specifying height, inputting a positive value, the wedge height will be created along the positive direction of Z axis in current UCS, on the contrary, it will be along the negative direction.

Relative Glossary:

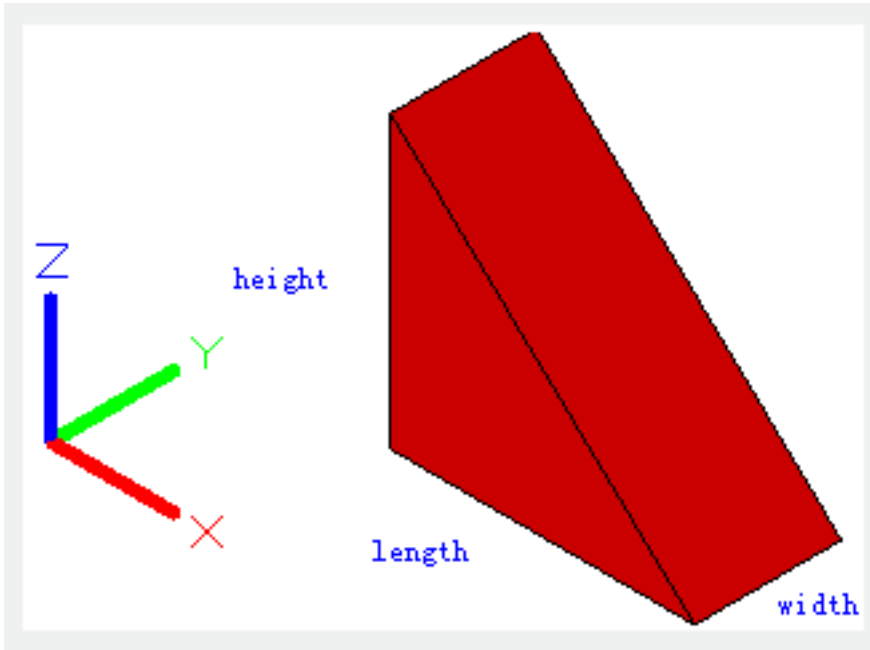
Center : Specify center to create a wedge.



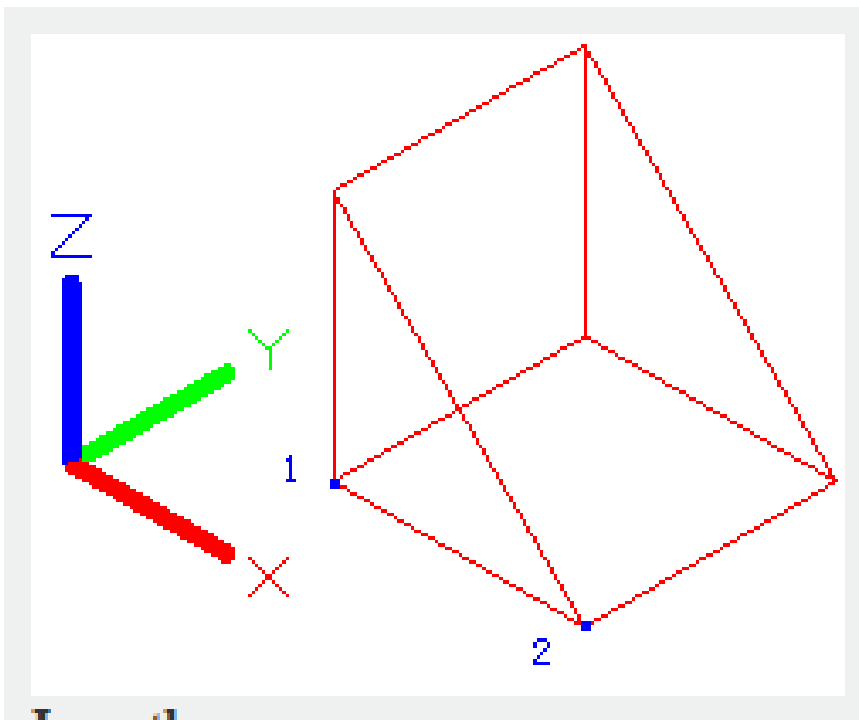
Cube : Create an equilateral wedge.



Length : Create a wedge by its length, width and height. The length is along the X axis direction, the width is along the Y axis direction and the height is along the Z axis direction.



Cube : Create an equilateral wedge.



Length : Create a wedge by its length, width and height. The length is along the X axis direction, the width is along the Y axis direction and the height is along the Z axis direction.

2Point : Specify wedge height by distance between two specified points