

# WEDGE command

2440 GstarCAD MY /KW August 25, 2021 [CAD Commands](#) 0 1135

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:

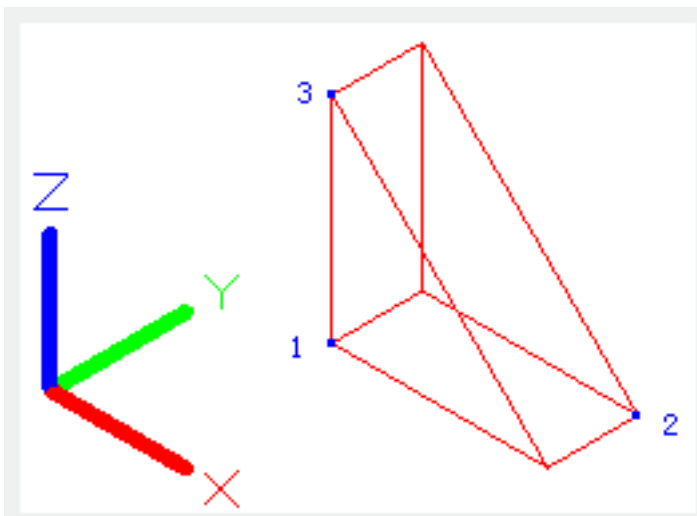
Specify first corner or [Center]:

Specify other corner or [Cube/Length]:

Specify height or [2Point]:

## Function Description:

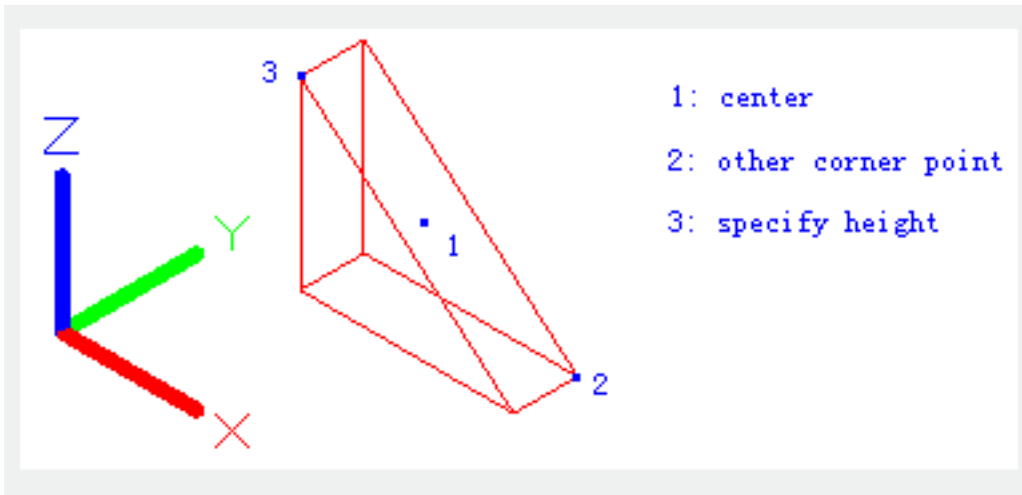
The incline direction is always along the positive direction of X axis in UCS



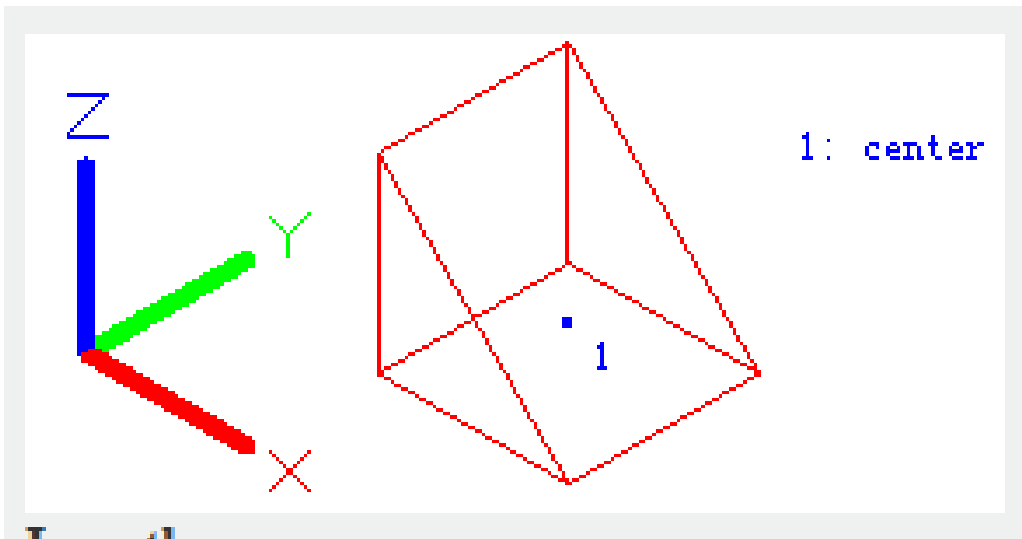
If the Z value of the specified other corner point is different to the first specified corner point, it will not display height prompt. When specifying height, inputting a positive value, the wedge height will be created along the positive direction of Z axis in current UCS, on the contrary, it will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

## Relative Glossary:

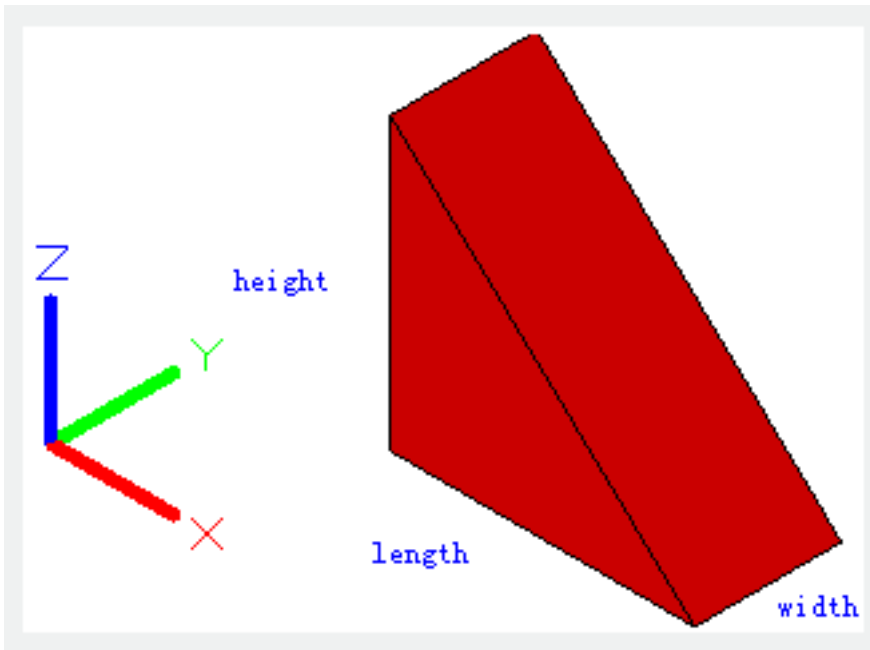
**Center** : Specify center to create a wedge.



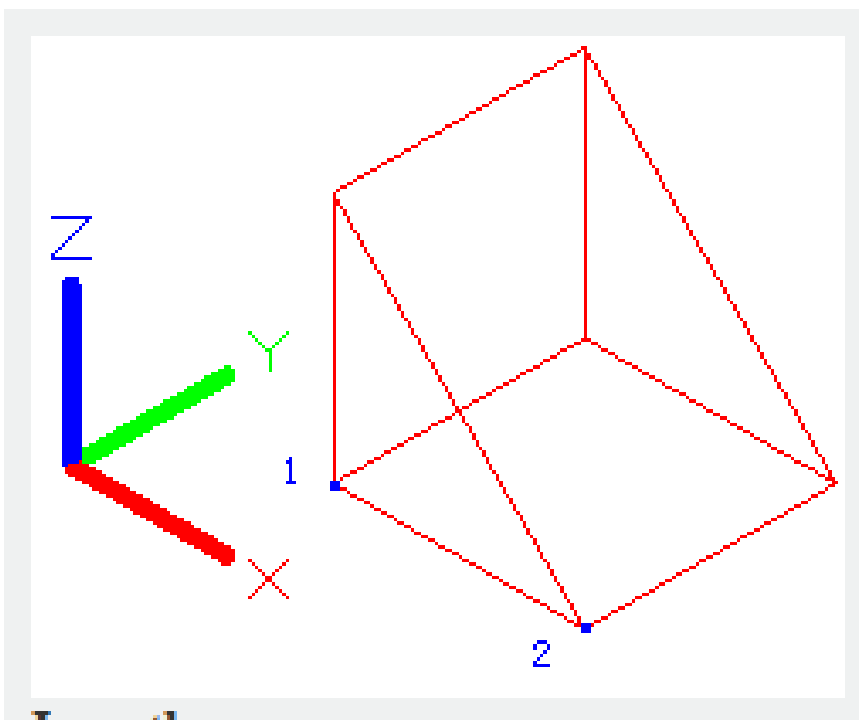
**Cube :** Create an equilateral wedge.



**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