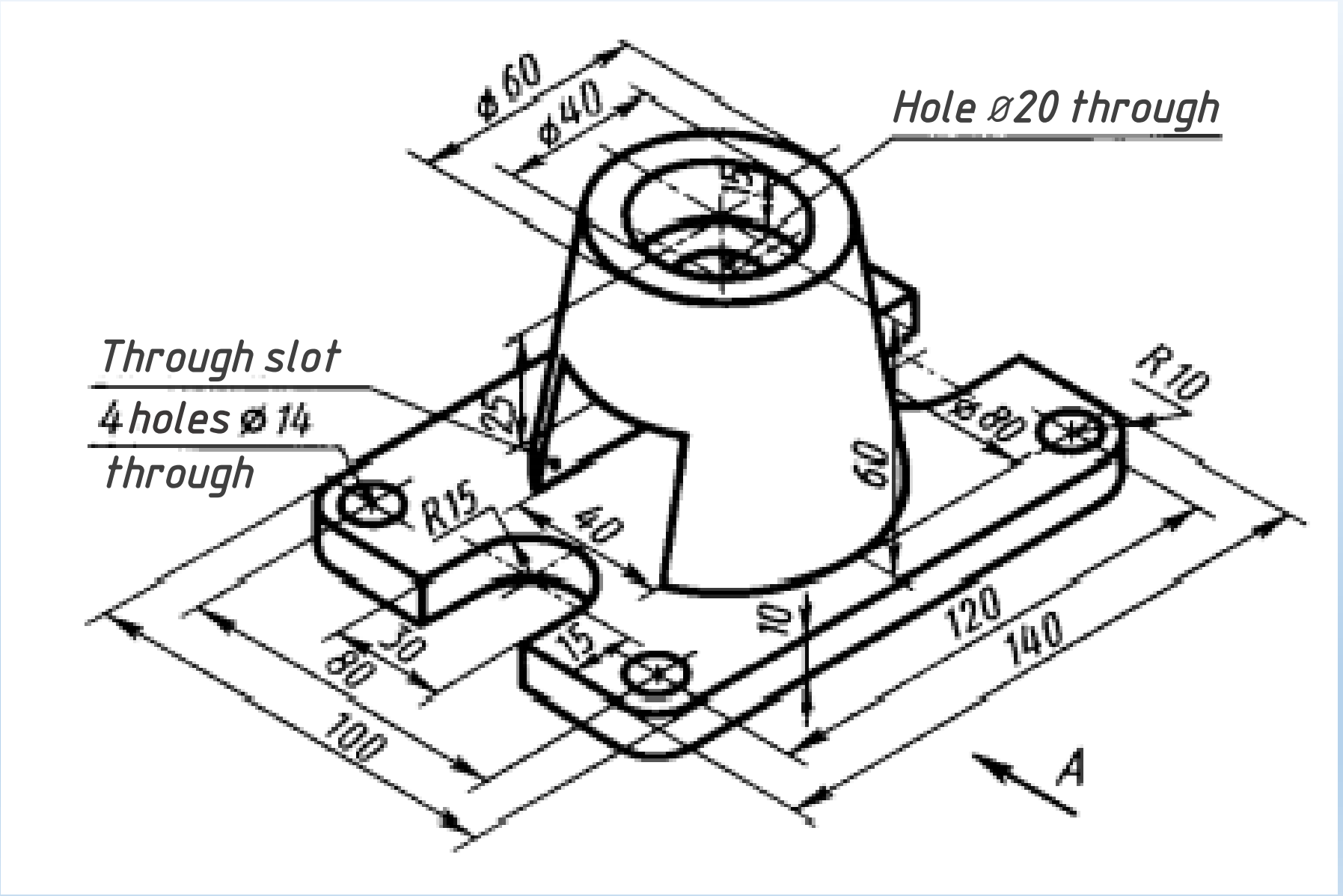
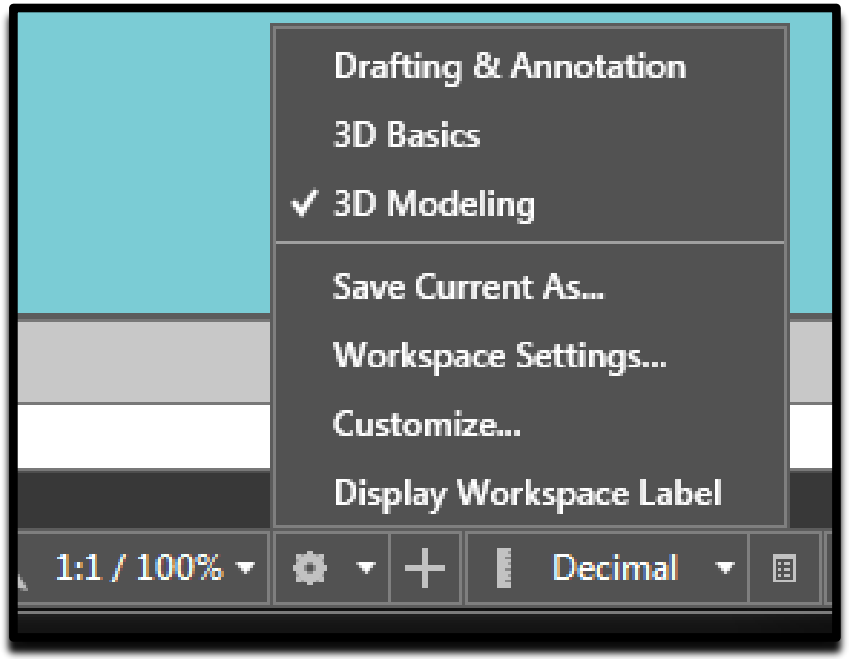
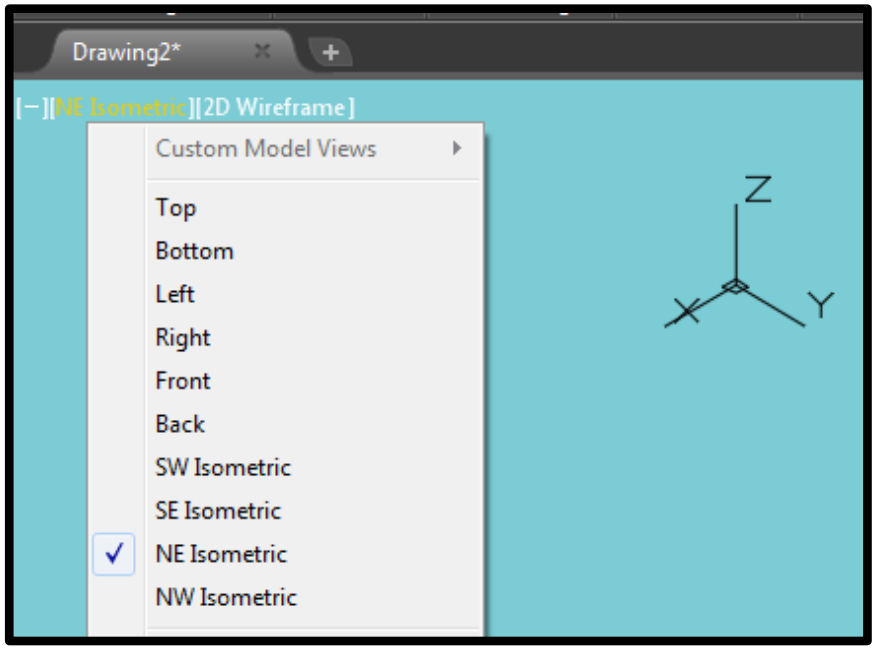
Creating 3D model in AutoCad



Preparing for modeling (setting up the workspace)

1. Select the workspace **3D modeling** in the status bar





2. Select in left top corner of a screen the axonometric orientation of the axes (**NE Isometric**)

Creation the base of a detail

1. Command ‘’Rectangle’’ 

Specify first corner point: select in command line **Fillet**

```
Command: _rectang
RECTANG Specify first corner point or [Chamfer Elevation Fillet Thickness Width]:
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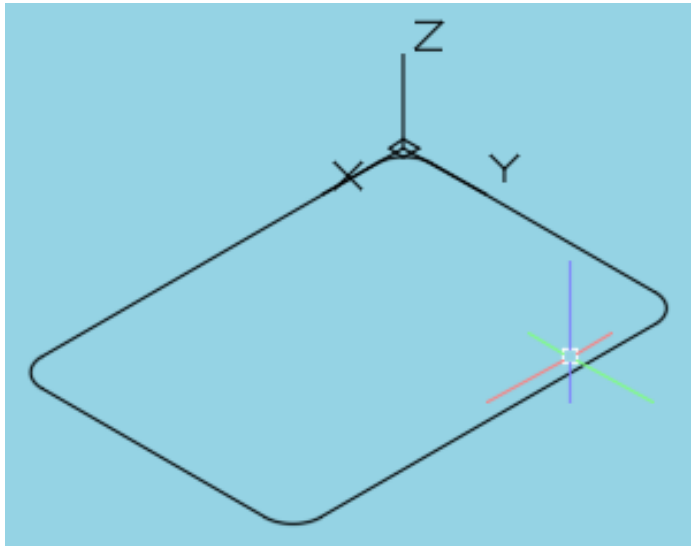
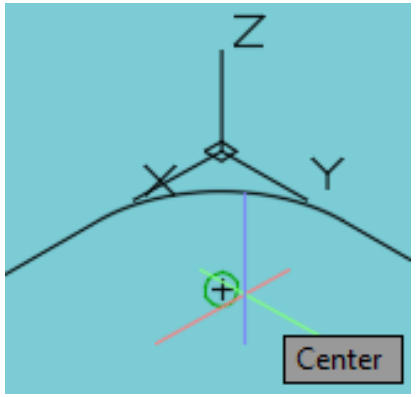
Specify fillet_radius :**10**
Specify first corner: **0, 0**
Specify other corner: **140, 100**

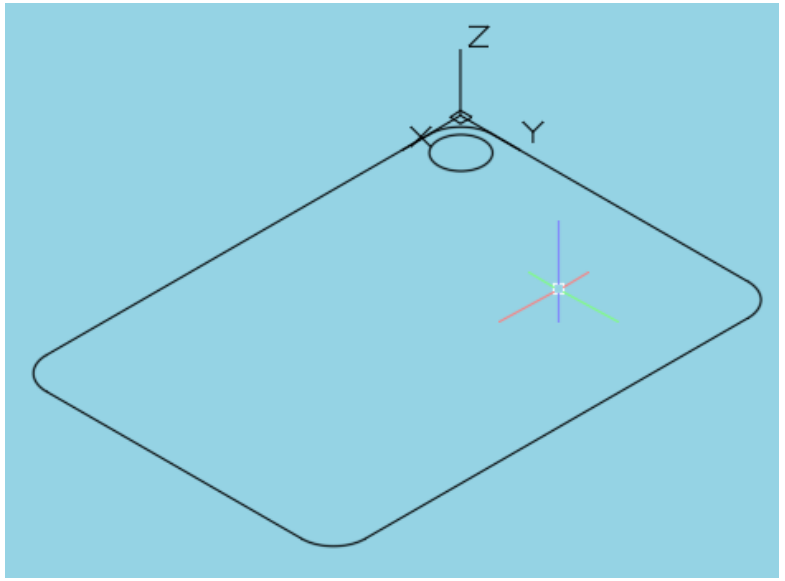


2. Command ‘’Circle’’ 

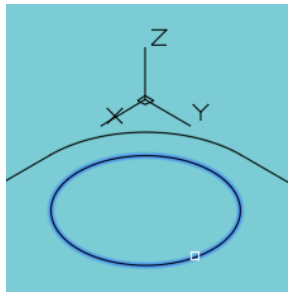
Center point: **10,10** or you can use the snap Center

Radius: **7**

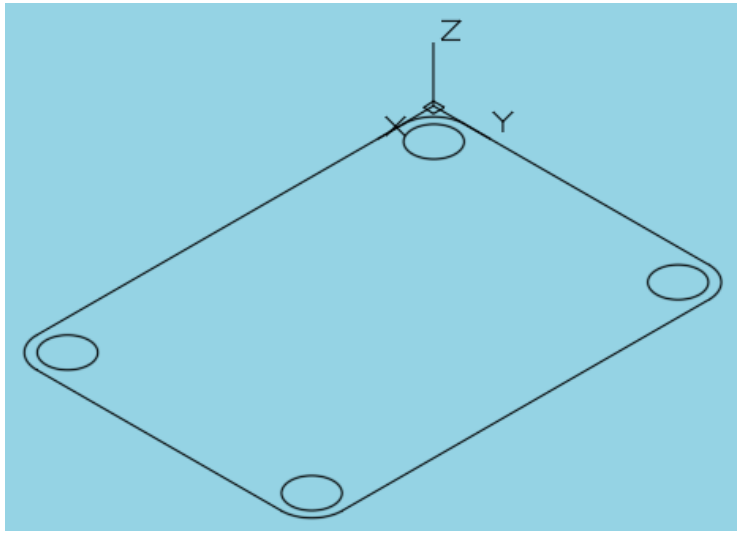




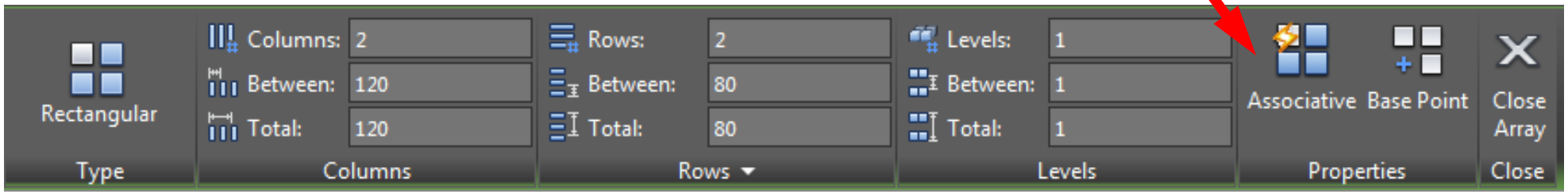
3. Command ‘’Rectangular Array’’



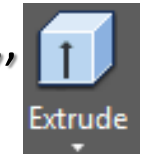
Select circle by mouse - Enter

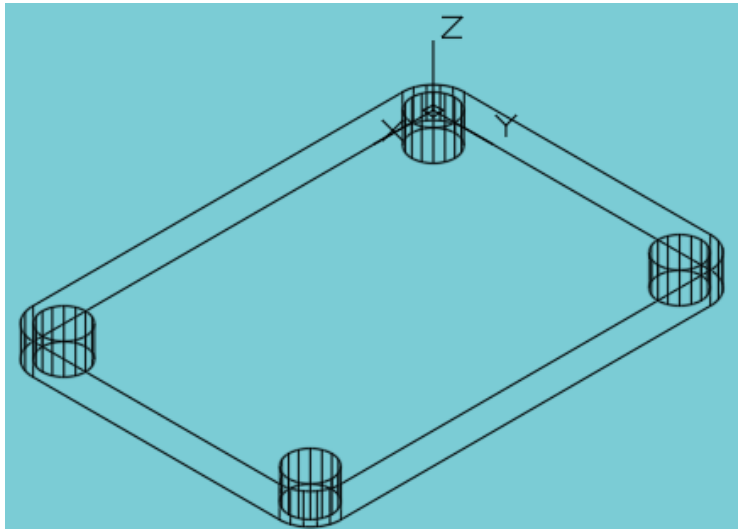


Enter the array parameters into the table as in the picture and turn off Associative mode in Properties!



4. Command ‘’Extrude’’

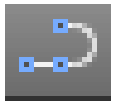




Select objects to extrude: select rectangle and all circles by mouse

Specify height of extrusion: **10**

Creation of grooves in the base

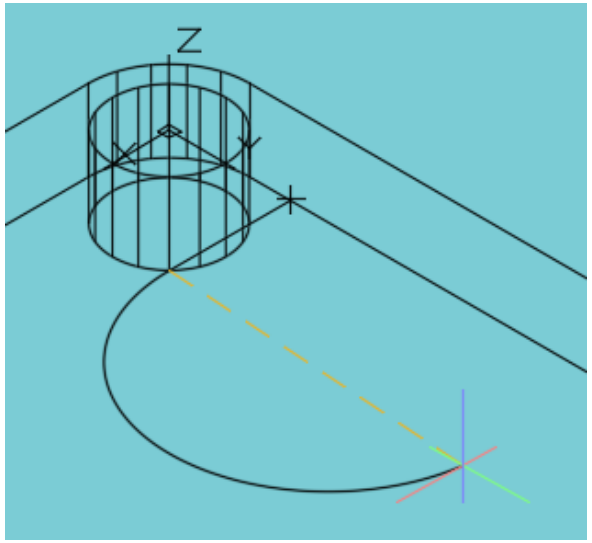
1. Command ‘’Polyline’’ 

Specify start point: **0, 35**

Specify next point: **15, 35**

Specify next point or [Arc Close Halfwidth Length Undo Width]: select **Arc**

Arrange the arc as shown using Orthomode  **: 30** Enter



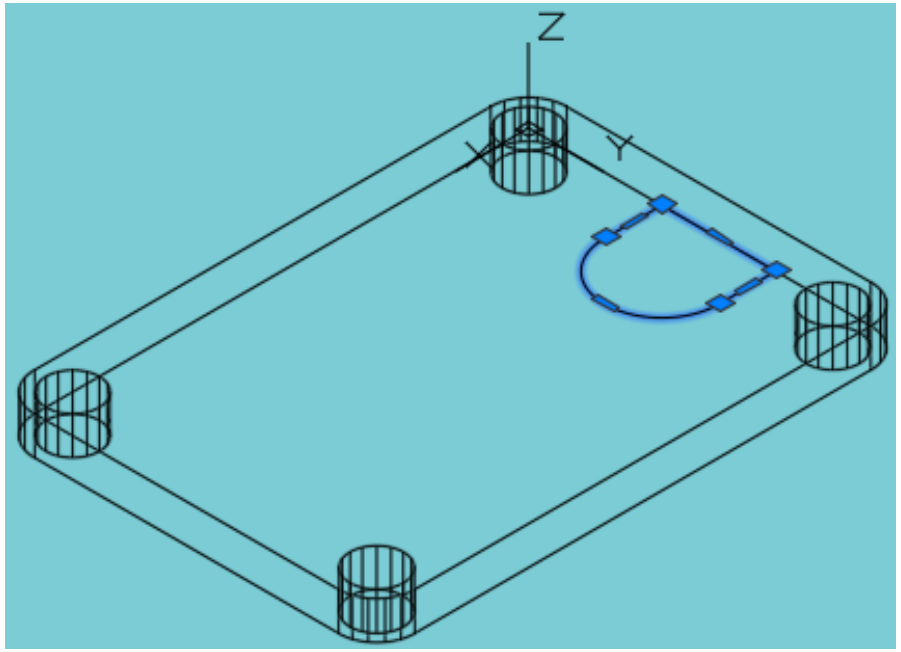
PLINE [Angle CEnter CLose Direction Halfwidth Line Radius Second pt Undo Width]:

select **Line**

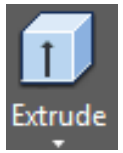
Specify next length of line: **0**, **65**

Specify next point or [Arc Close Halfwidth Length Undo Width]

select **Close**

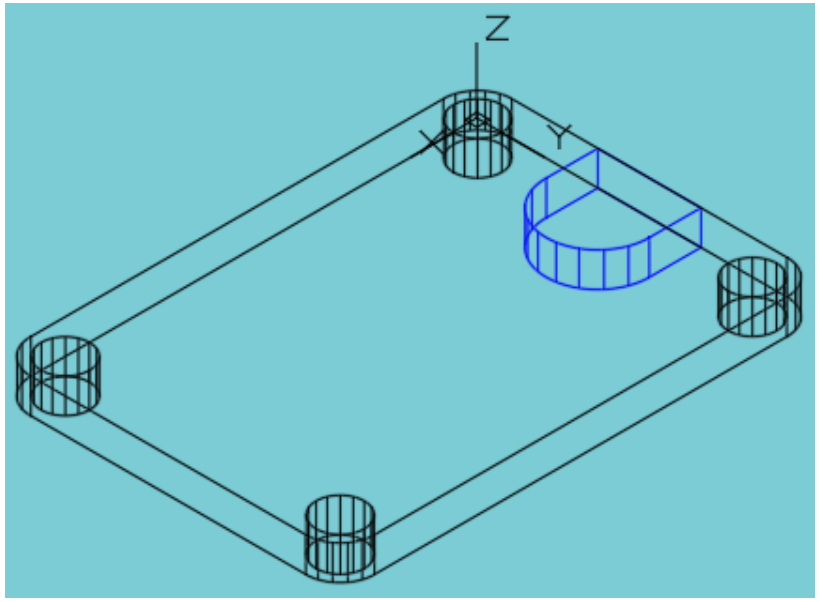


2. Command ‘’Extrude’’



Select objects to extrude: select rectangle and all circles by mouse

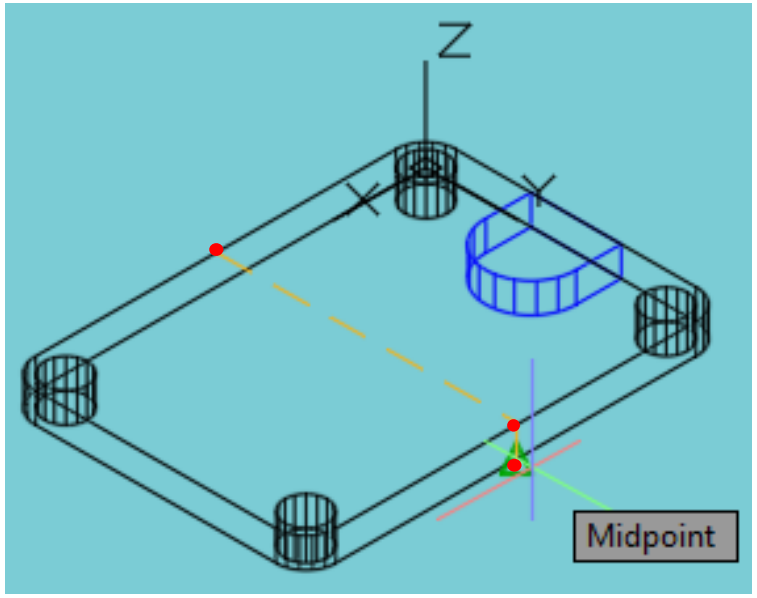
Specify height of extrusion: **10**

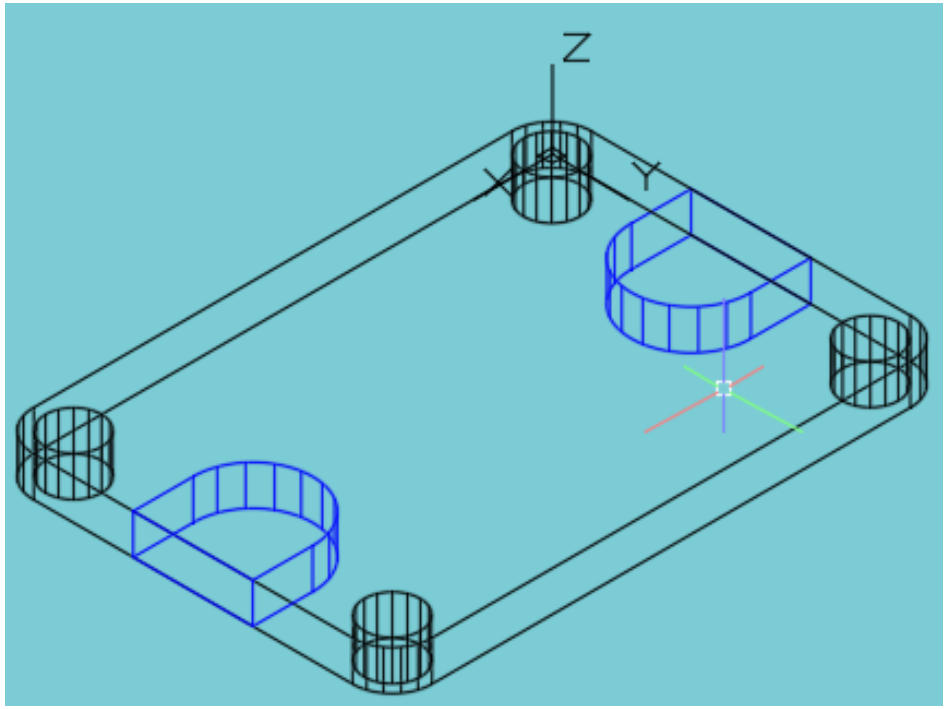


3. Command ‘’3D Mirror’’

Select objects: select Groove - **Enter**

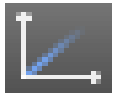
<3 points>: using snap **Midpoint** select middle of a base sides as in the picture



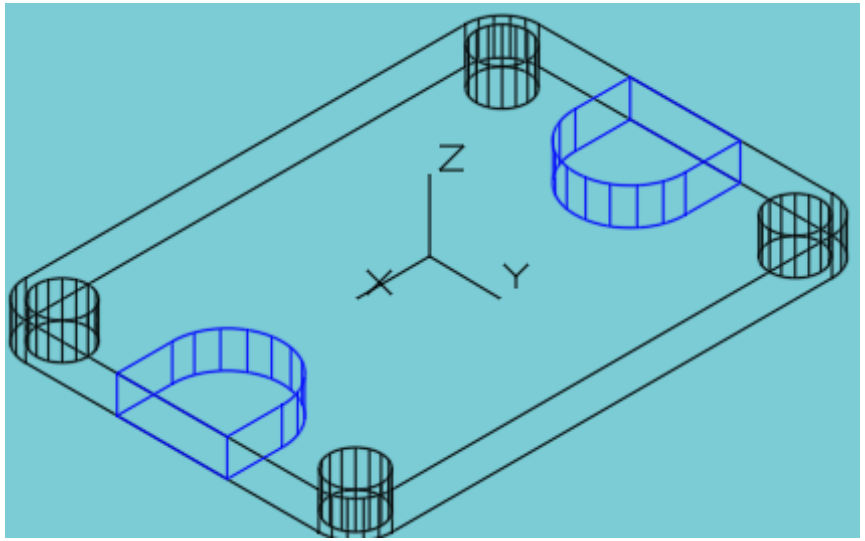


Delete source objects? <N>: **Enter**

Creating a 3D cone model

1. Command ‘’Origin’’ 

Specify new origin point <0,0,0>: **70, 50, 10**



2. Command ‘’Cone’’ 

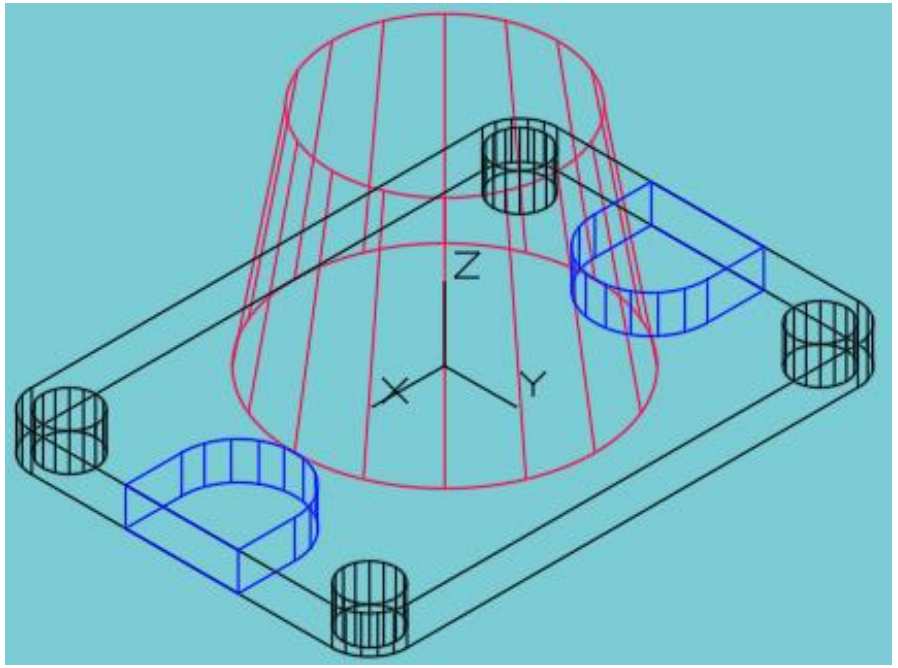
Specify center point of a base : **0, 0**

Specify base radius : **40**

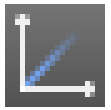
Specify height or [2Point Axis endpoint Top radius]: **T**

Specify top radius : **30**

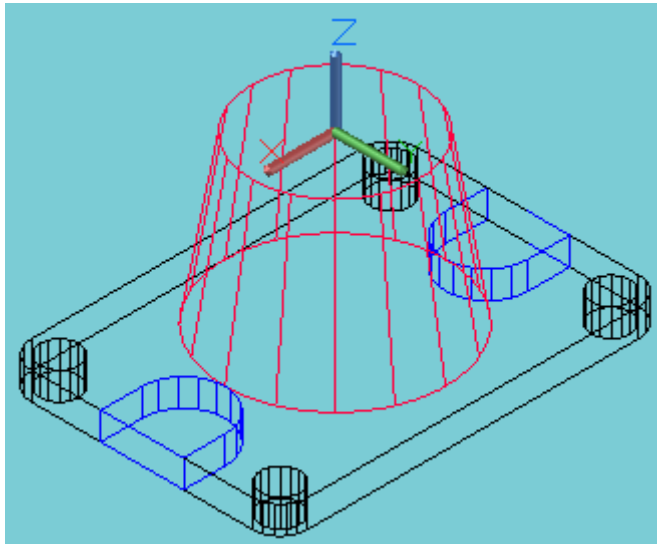
Specify height : **60**



Create cylindrical holes in a cone

1. Command ‘’Origin’’ 

Specify new origin point <0,0,0>: **0, 0, 60**

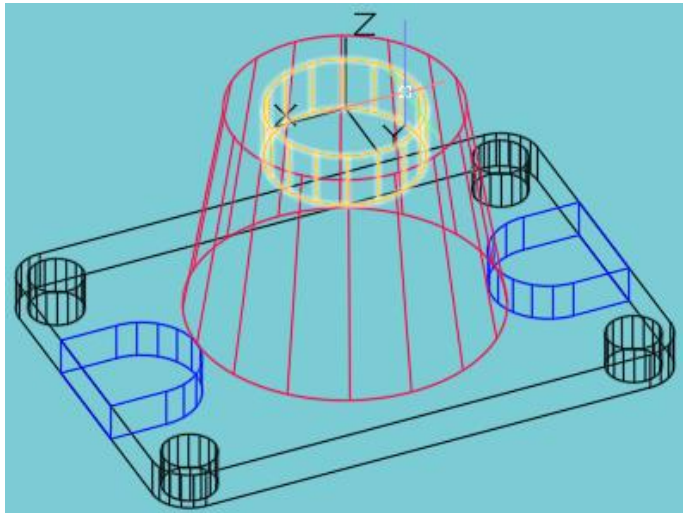


2. Command ‘’Cylinder’’ 

Specify center point of a base : **0, 0**

Specify base radius : **20**

Specify height : **- 15**

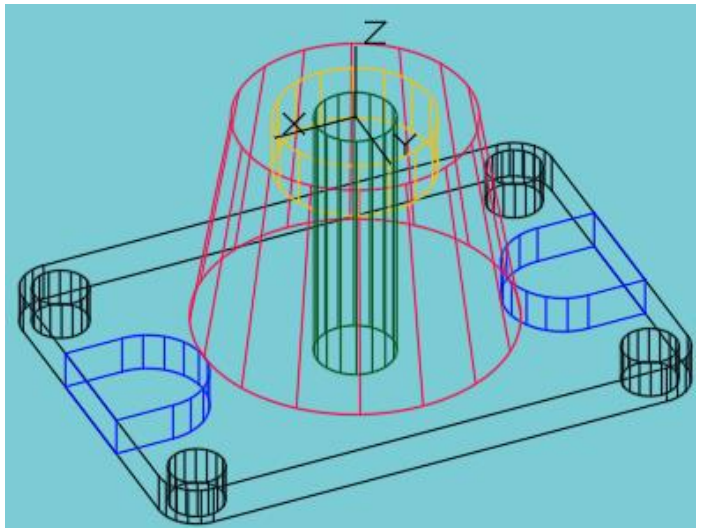


3. Command ‘’Cylinder’’ 

Specify center point of a base : **0, 0**

Specify base radius : **10**

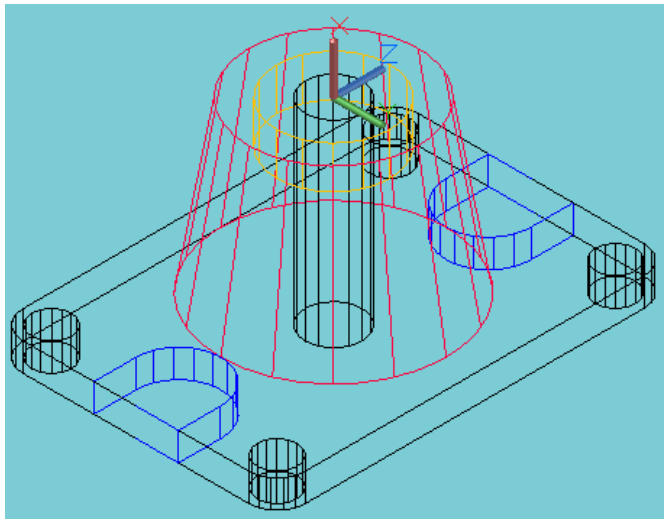
Specify height : **- 70 (or more)**



Create prismatic hole in a cone

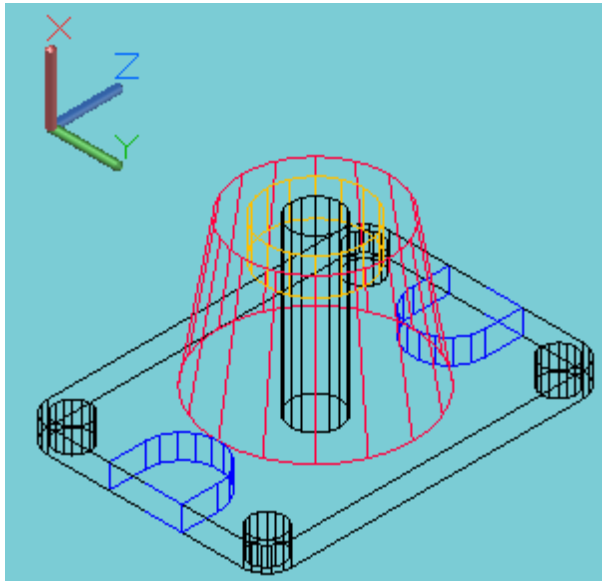
1. Command ‘’Rotates’’ 

Specify rotation angle about Y axis <90>: **-90**



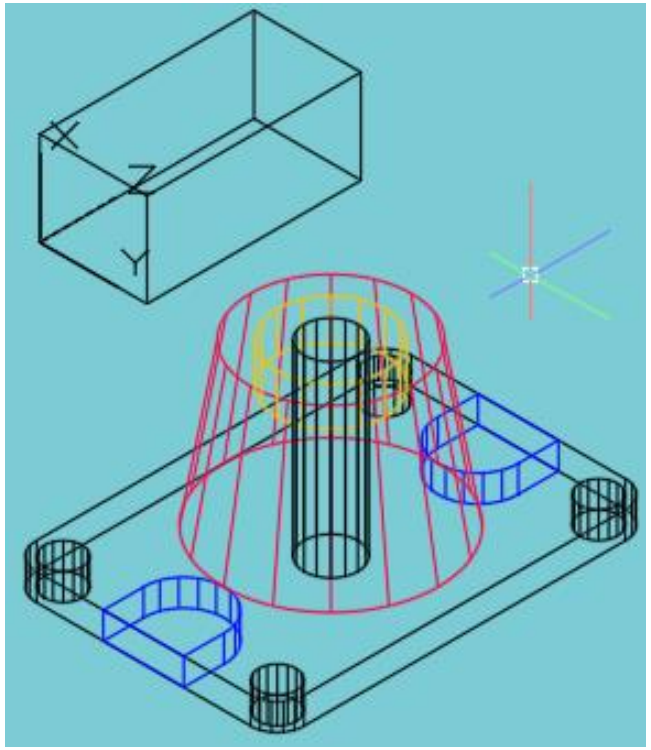
2. Command ‘’Origin’’ 

Move the coordinate system anywhere in the drawing field



3. Command ‘’Box’’ 

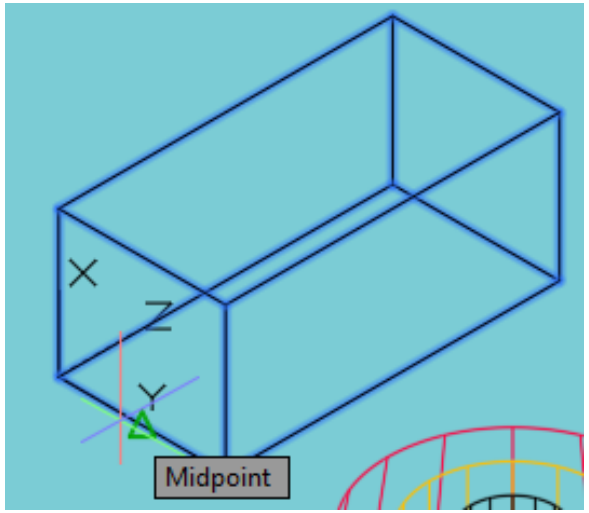
Specify first corner: 0,0
Specify other corner: 35, 40
Specify height: 80



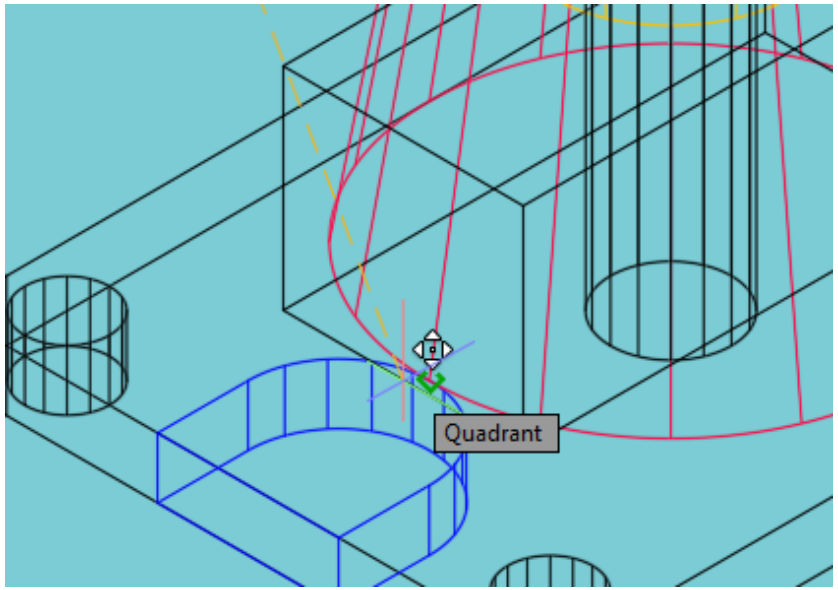
4. Command ‘‘Move’’

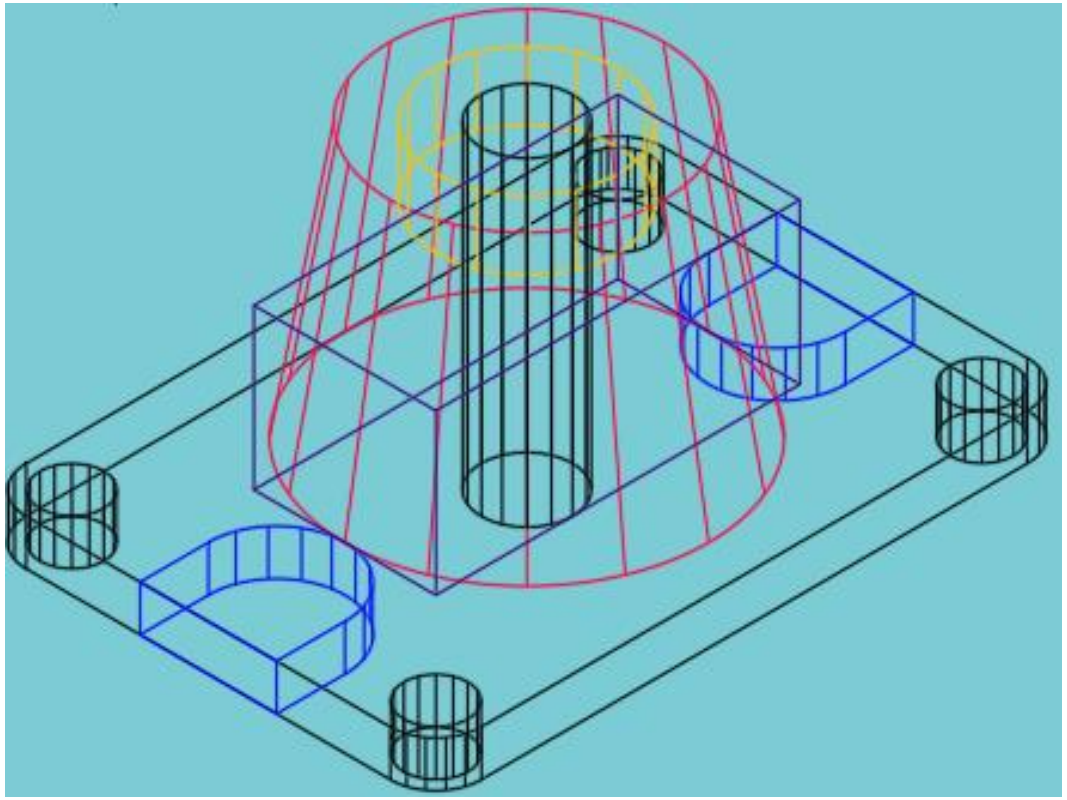
Select objects: select box by mouse

Specify base point: using snap **Midpoint** select middle of a box bottom edge as in the picture



Specify second point: using snap **Quadrant** select quadrant point of a cone base edge as in the picture

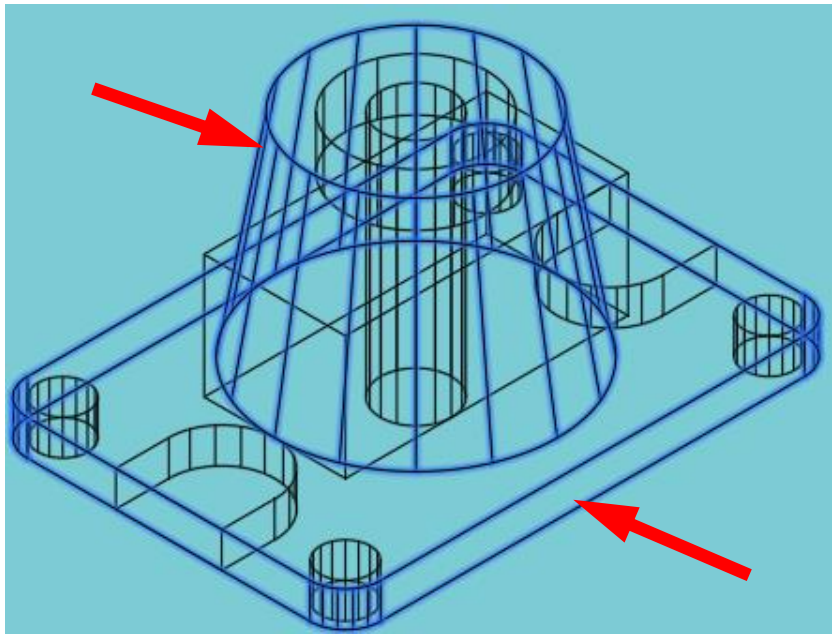




Logical operations when creating 3D models

1. Command ‘’Solid, Union’’ 

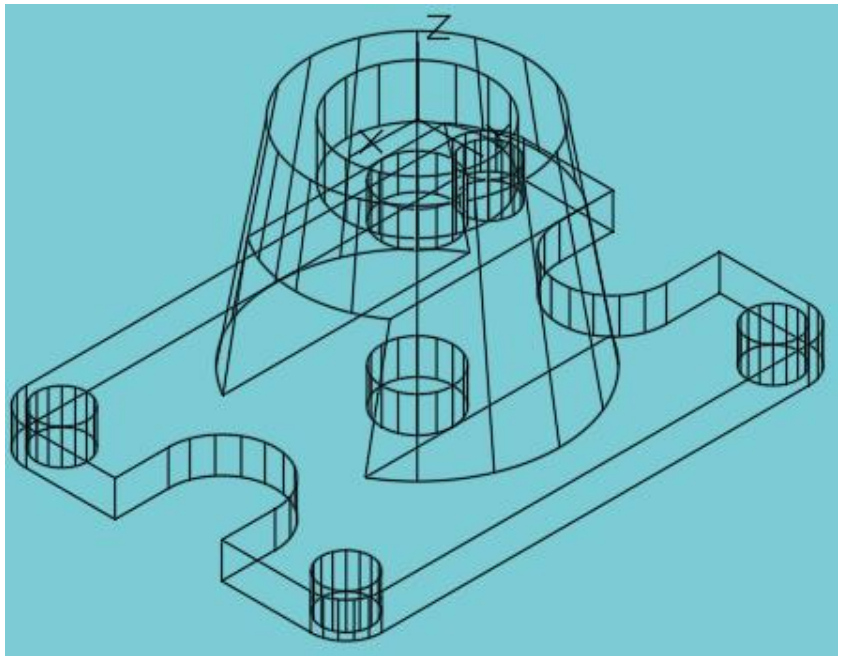
Select objects: select by mouse base and cone parts of the detail



2. Command ‘’Solid, Subtract’’ 

Select objects: select by mouse base or cone of the detail (these are already united bodies) Enter

Select objects: select by mouse all subtraction elements (cylinders, grooves and prism)



Modeling result