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AUTOCAD 3D

Lab

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CONTENTS

IntRO	1
Working with Dynamic UCS	2
UCSDETECT	2
UCS	2
Box	2
Exercise 1	3
Mesh.....	3
Exercise 2	3
Exercise 3	4
Exercise 4	6
Exercise 5	6
Exercise 6	7
TABSURF	8
REVSURF.....	9
RULESURF	10
EXTRUDE.....	10
EXERCITIU 7	11
UNION	11
INTERSECT	12
SUBTRACT	12

AUTOCAD 3D

INTRO

AutoCAD uses by default three values to specify the drawing, but working in the the bi-dimensional space defines the z coordinate equal to zero.

Adding depth and working in AutoCAD 3D means adding a non zero value for the z coordinate.

Let's try to recap some of the most useful 3D commands:

- Box, cone, mesh, pyramid, sphere, torus, wedge

- Mesh and several of its options: box, cone, cylinder, pyramid, sphere, wedge, torus
- 3Dorbit, shademode, etc.

WORKING WITH DYNAMIC UCS

When creating 3D objects, to align the UCS with the active face is an advantage. This will allow objects to be draw on that specific plane.

UCSDETECT

Specifies id the Dynamic UCS is active or not. The value 0 means that the option is inactive, 1 the opposite.

To activate/deactivate the dynamic UCS, one option is to press F6 or use the UCSDETECT command.

UCS

The command has several options to be able to define the position of the new UCS.

Next, several options will be presented.

The default option is

Specify origin of UCS:

- specifying one point, the base point of the coordinate system moves to the new position, maintaining the previous orientations;
- specifying two points, the OX will pass from the first point to the second one,
- specifying three points, determines the orientation of the OY axis.

Next

positions the UCS on an adjacent or back face of the selected edge.

Xflip

Rotates the UCS 180 degrees around the X axis.

Yflip

Rotates the UCS 180 degrees around the Y axis.

Accept

Accepts the changes and places the UCS.

Face

To align the UCS with a specific face of a 3D object

Named

Shifts the ucs orientation and position to a previous named UCS. One of the sub-options of named is to save a current UCS.

Object

Aligns the UCS to a selected 2D or 3D object. The UCS can be aligned with any object except xlines and 3D polylines.

Z Axis

Aligns the UCS to a specified positive Z axis.

The UCS origin is moved to the first point and its positive Z axis passes through the second point.

BOX

The Box can be created by providing the insertion point or the center point, followed by the options: to provide the opposite corner, to draw a cube or to provide the length, width and height.

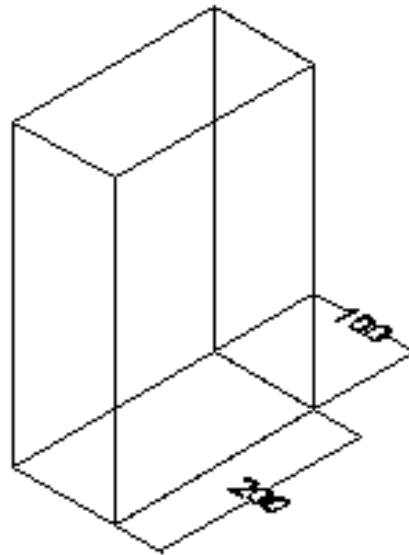


Figure 1 First box

EXERCISE 1

- Create the following box: length =200mm, width=100mm, height=300mm.
- Align the UCS with 3 of the faces.

MESH

As the name suggests, the options allow the generation of boxes, cones, cylinders, sphere, wedges and toruses.

The Torus options will be detailed because is an unusual geometrical form, as the next figure presents: one needs to define the center point, the radius or diameter and the radius of the tube.

EXERCISE 2

Recreate your one torus.

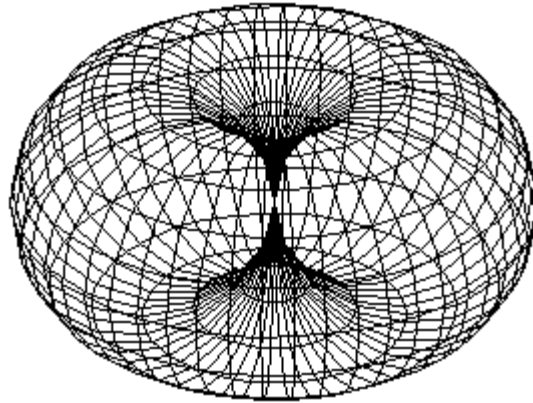


Figure 2 Torus

Using the option Settings, the smoothness of the surfaces could be adjusted.

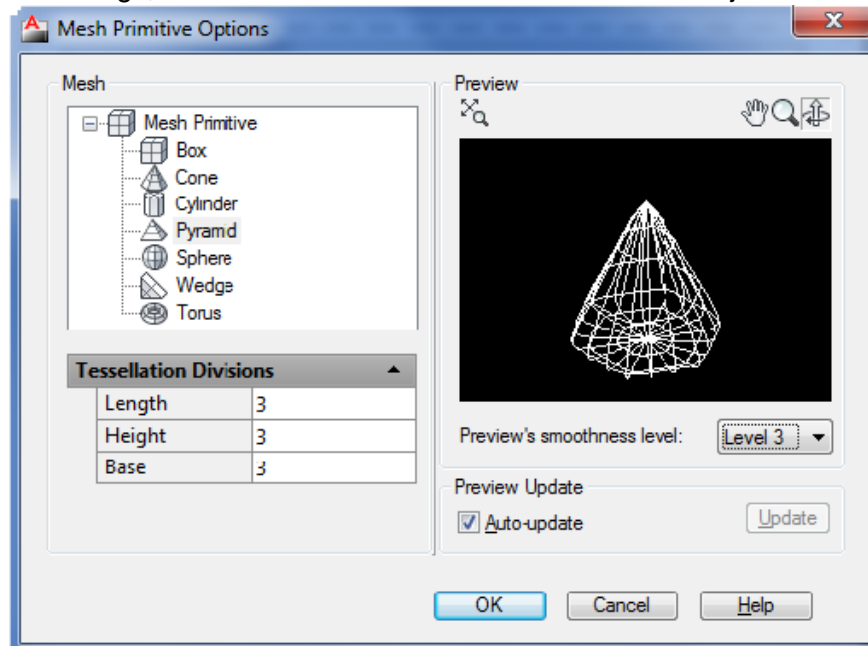


Figure 3 Mesh and Tessellation

EXERCISE 3

Try the option Settings and Smoothness level in order to change the smoothness from 0 to level 4. Create a new Torus.

To change the Tessellation division for each mesh, select the mesh and edit the fields: Length, Height, Base.

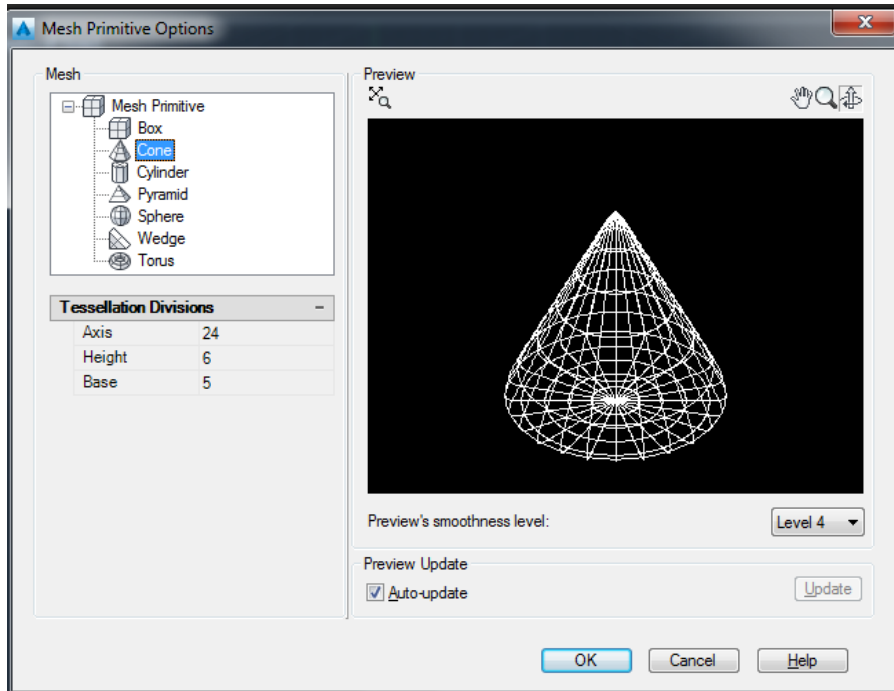
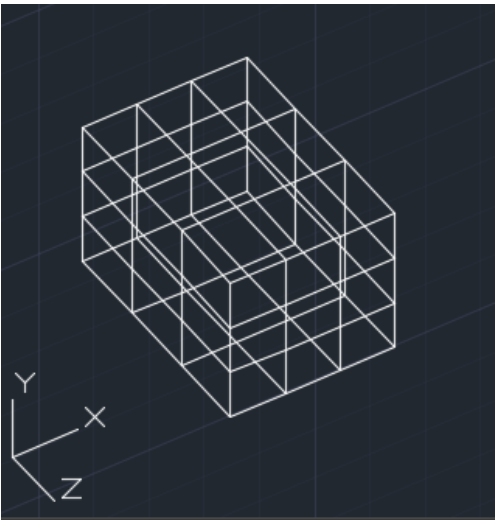


Figure 4 Tessellation settings

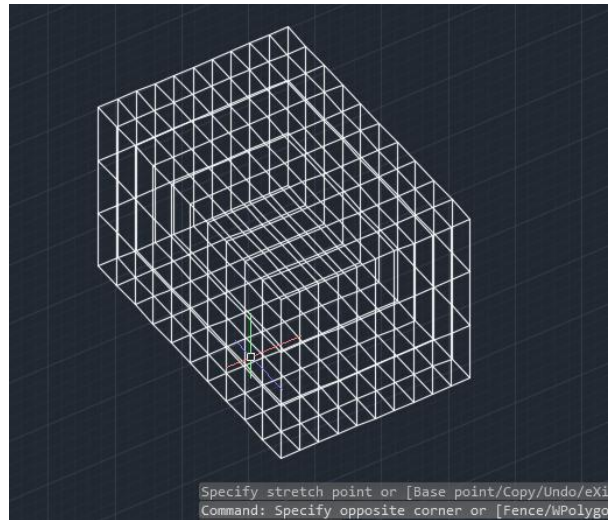
In the case of Box option, there is a second option: the system variables to control the number of surfaces in a mesh box object for each axis are:

- DIVMESHBOXHEIGHT for Oz
- DIVMESHBOXLENGTH for Ox,
- DIVMESHBOXWIDTH for Oy.



a) Before –

DIVMESHBOXHEIGHT set to 3
 DIVMESHBOXLENGTH set to 3
 DIVMESHBOXWIDTH set to 3



b) After

DIVMESHBOXHEIGHT set to 10
 DIVMESHBOXLENGTH set to 10
 DIVMESHBOXWIDTH set to 10

Figure 5 Controlling the surfaces dividing

Using **EXPLODE** command for the mesh, each part of the mesh could be modified independently.

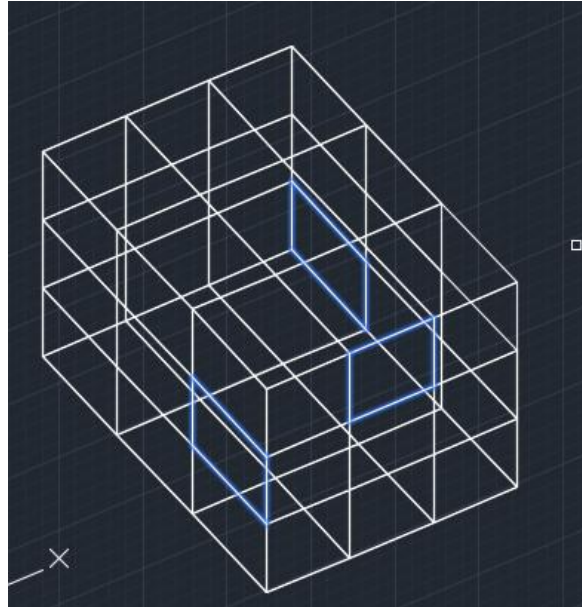


Figure 6 Mesh parts selected separately

In order to access their current value, use the LISP call (getvar "DIVMESHBOXHEIGHT").

EXERCISE 4

- Try to set the variables as presented in Figure 6 b). Recreate the cube and apply extrude. Try to access the parts.
- Use the command move to move 4 of the selected faces.
- Apply the command SHADEMODE, shaded with grey.

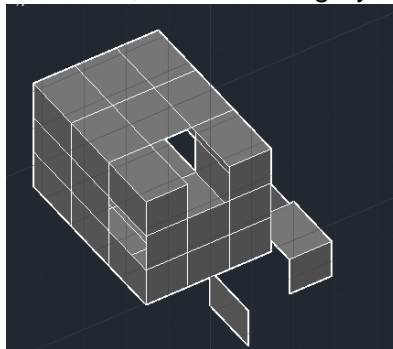
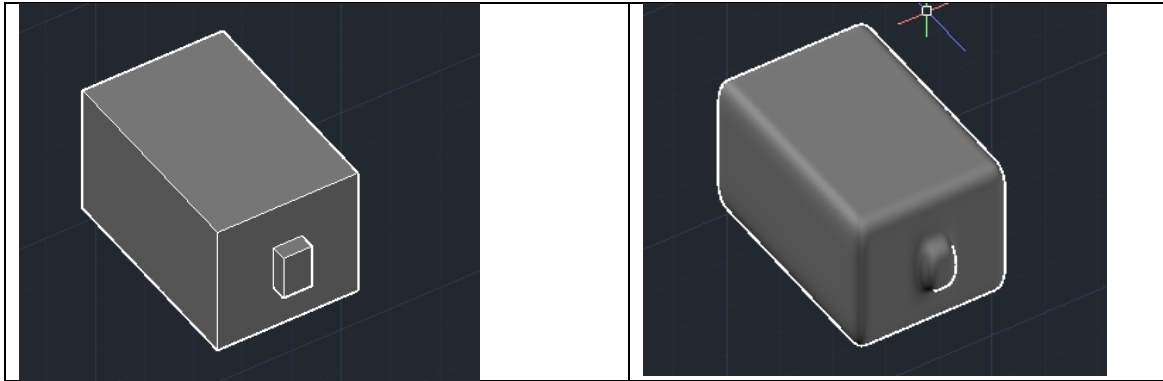


Figure 7 Modifying the object by accessing parts of the mesh

EXERCISE 5

Create a new cube mesh.

Use the command MESHEXTRUDE to select one or two surfaces and extrude them by 100mm. Apply smoothness with level 4(right click – Properties).



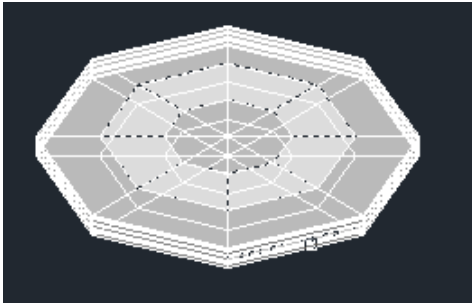
a) Before the smoothness

b) After applying smoothness level 4 sm

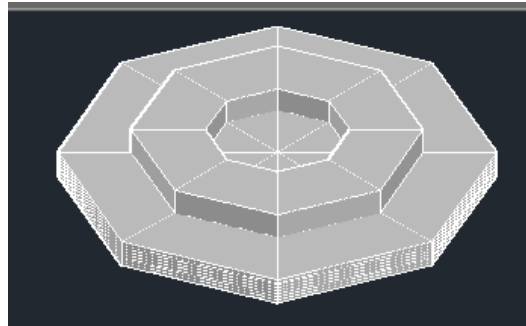
Figure 8 Extrude the mesh with MESHEXTRUDE

EXERCISE 6

Identify the steps in order to obtain the following:



a) Step 1 – The object



b) Step 2 – the extruded parts



c) Add Smoothness

Figure 9 Identify the actions to obtain the image

There are several commands to be applied for surfaces:

MESHREFINE – divides a selected surfaces from the mesh to the GRID settings in order to be able to manipulate the areas of the mesh by small surfaces.

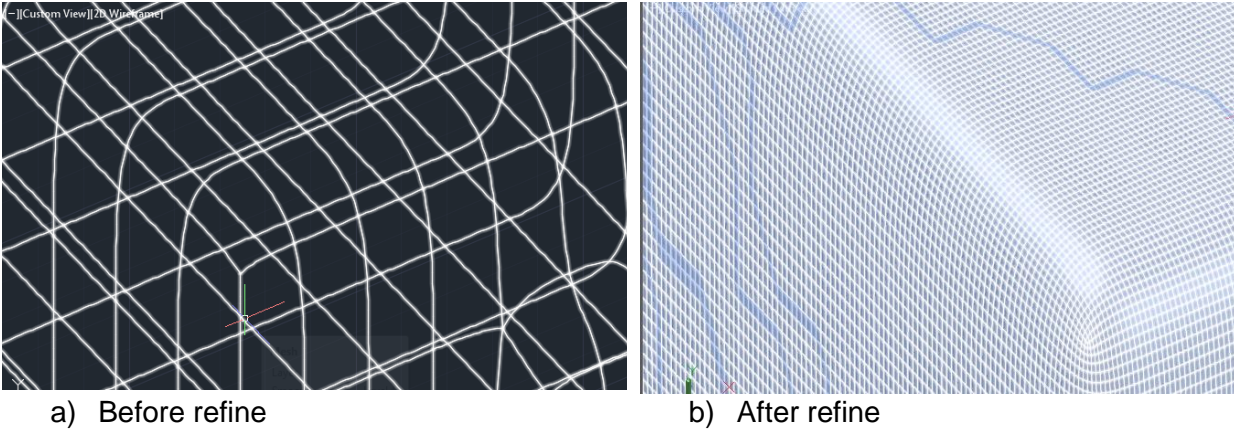


Figure 10 Using MESHREFINE

MESHCREASE- disables the settings related to SMOOTHNESS

MESHSPILT- divides the surfaces of the message based on a template.

MESHMERGE- unifies the surfaces

MESHCOLLAPSE - the surfaces having a common side will be replaced by a point.

MESHCAP - generates a unifying surfaces between several sides.

TABSURF

Follow the steps:

Command: VIEW Choose Top View.

Command: CIRCLE

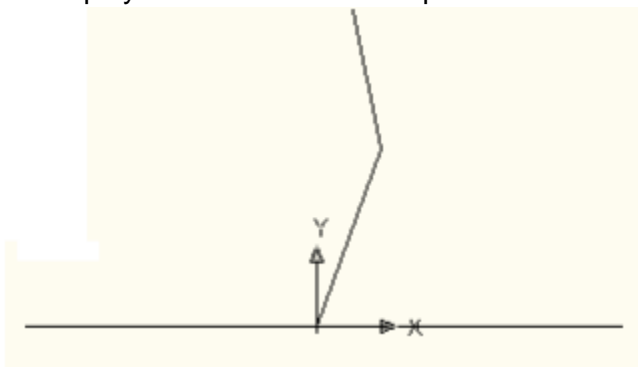
Start point: 0,0,0

Radius: 100.

Command: VIEW

Choose Front View.

Create a polyline similar to the one presented next:

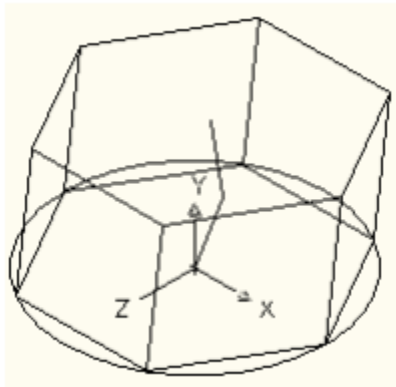


By selecting the polyline inside the command TABSURF, it will determine the direction for TABSURF

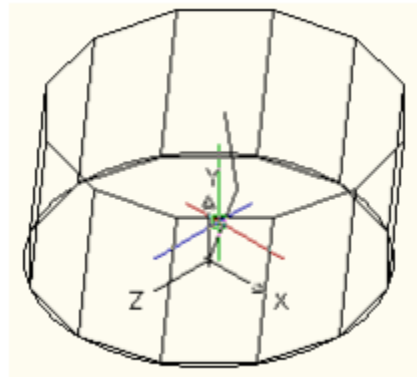
Command VIEW CHOOSE SE Isometric View.

Command: TABSURF

The circle will be chosen for the contour to follow and the polyline for the direction. The result is presented next:



a) The result with system variable SURFTAB1=6



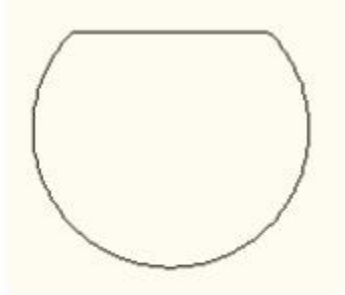
b) The result with system variable SURFTAB1=12

Figure 11 TABSURF

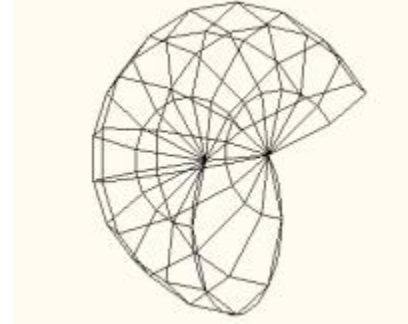
REVSURF

Choose top view and generate the profile presented by the **Error! Reference source not found.** a) where two entities are visible: a line segment and an arc.

The object to execute a revolution rotation is the arc, and the object considered as the rotation axis is the line. The start angle has the value 120 degrees and the final one, 240 degrees.



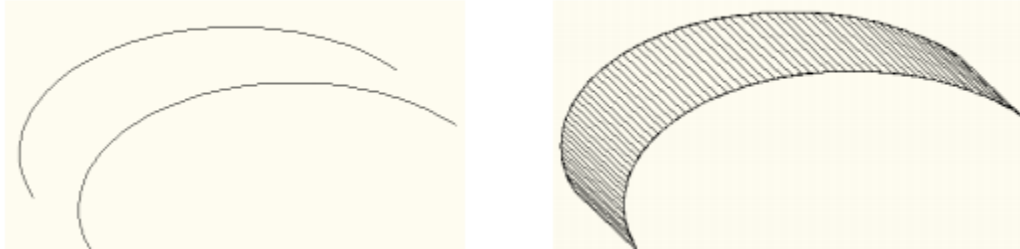
a)



b)

RULESURF

First create an arc in 3d coordinates, then create a copy paralel to it, at a distance large enough for both to be visible.



Execute RULESURF using as curves the two arcs. SurfTab =70

EXTRUDE

The command allows the #D object generation based on their orthogonal projection. Generate the entities from the next figure using ARRAY with the POLAR option, as rotation point, the center of the circles and 120 multiplications under a 360-degree angle. Starting with the red colored entity and using ARRAY, add 14 entities.

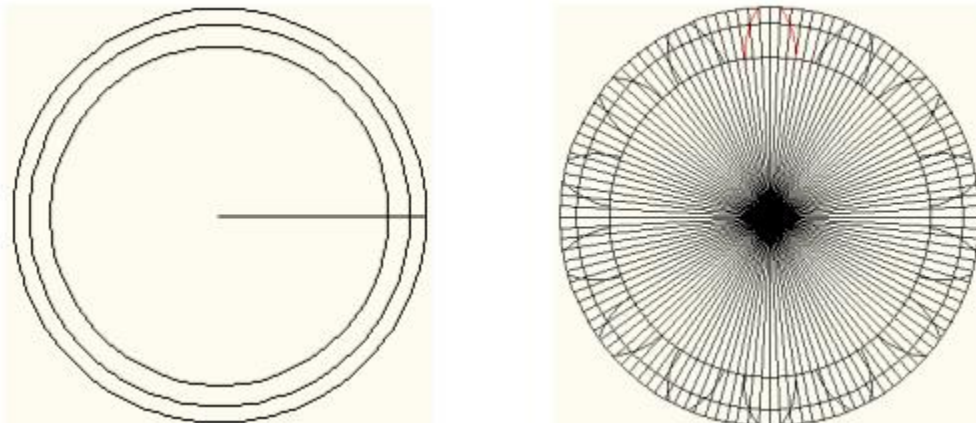


Figure 12 The base entities

To ease the TRIM usage and the elimination of all aiding elements, the line must be visible longer than the larger radius. After obtaining Figure 13, apply the command BLOCK. The effect of TRIM is visible next:

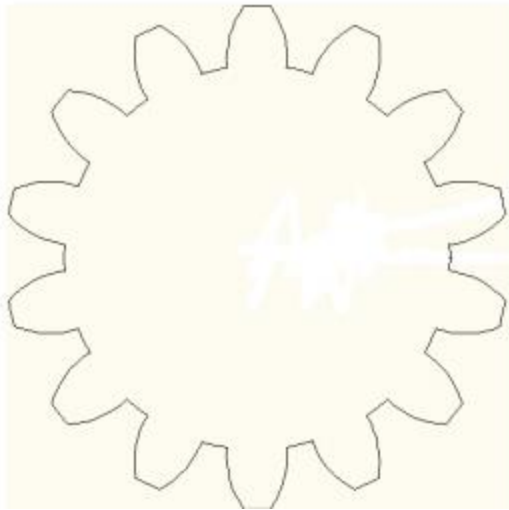


Figure 13 TRIM effect

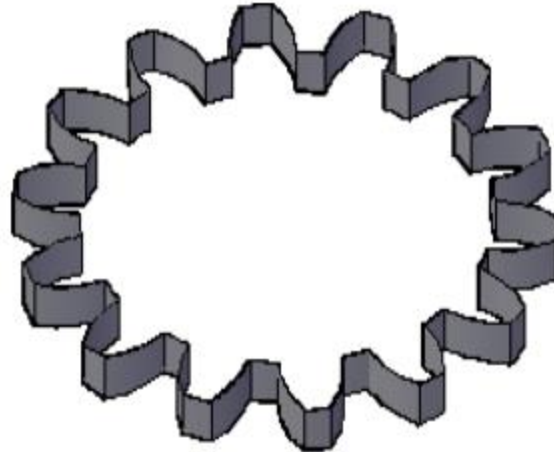


Figure 14 EXTRUDE effect with 200 mm

The command can be applied to different type of entities as: mesh, solids, curves, etc.

EXERCITIU 7

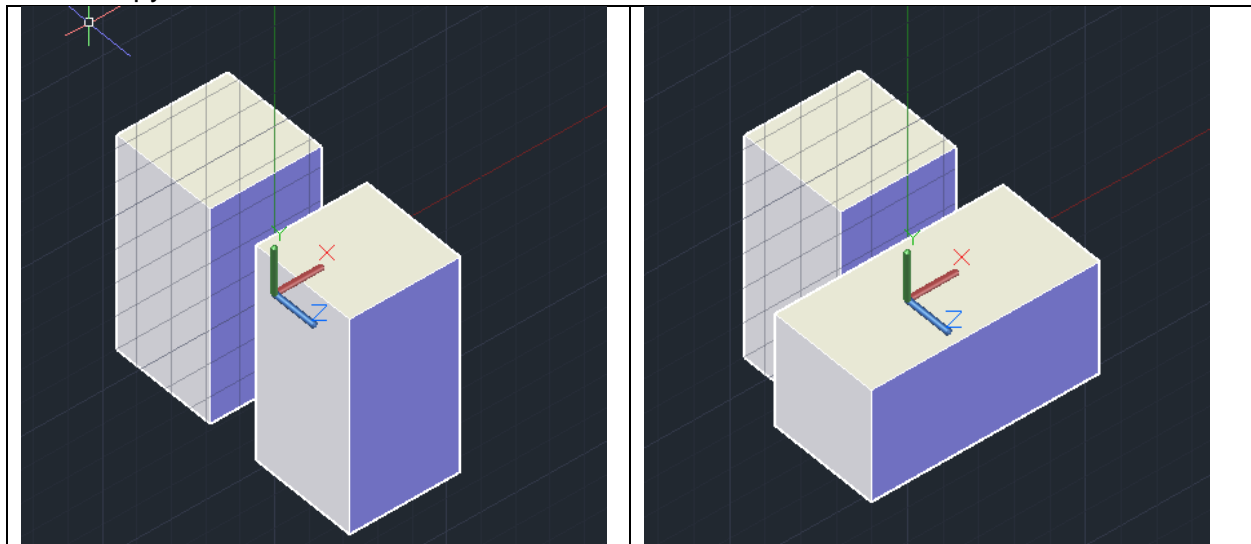
RECREATE the first project's object in 3D! The best one will be printed on a 3D Printer.

UNION

Combines two or more 3D solids, surfaces, or 2D regions into a single, composite 3D solid, surface, or region.

Create the following boxes: length - 200mm, width - 200mm, height - 300mm

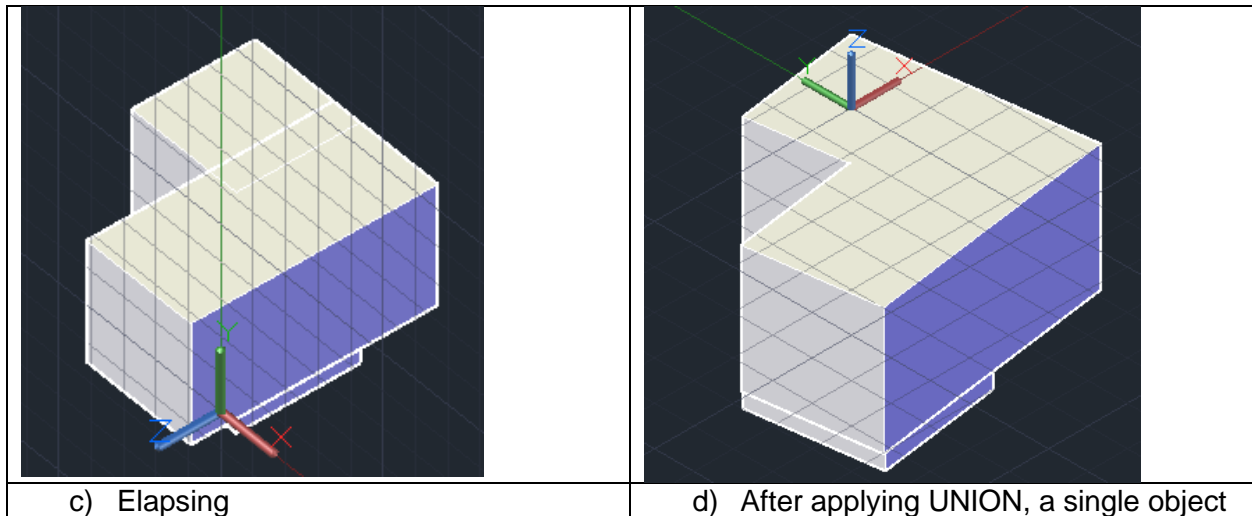
Create copy of the first and rotated as in shown bellow:



a) 2 boxe

b) The second is rotated

Move the copy towards the first entity by 150mm and elapsing this first one. Do not forget to change the UCS.



The surface editing commands available are:

- surfblend
- surffillet
- surfpatch

INTERSECT

Obtains the intersection of solids, surfaces, regions as a 3D SOLID , surface or 2D Region.

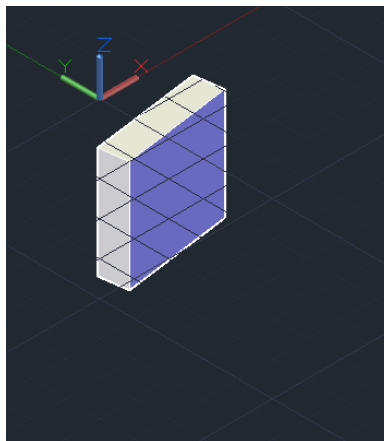


Figure 15 After applying INTERSECT

SUBTRACT

Creates as a new object by subtracting one overlapping region or 3D solid from another.

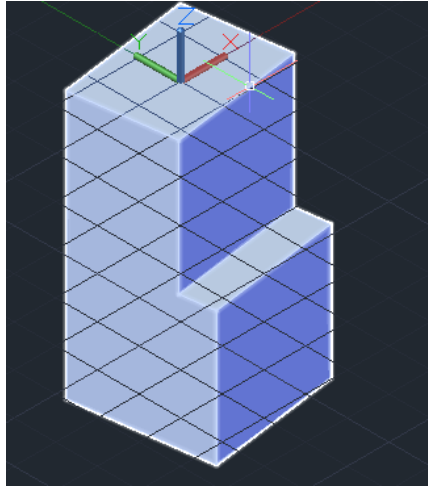


Figure 16 After applying SUBTRACT

NOTE SOLIDEDIT – To be considered.

The end.:)