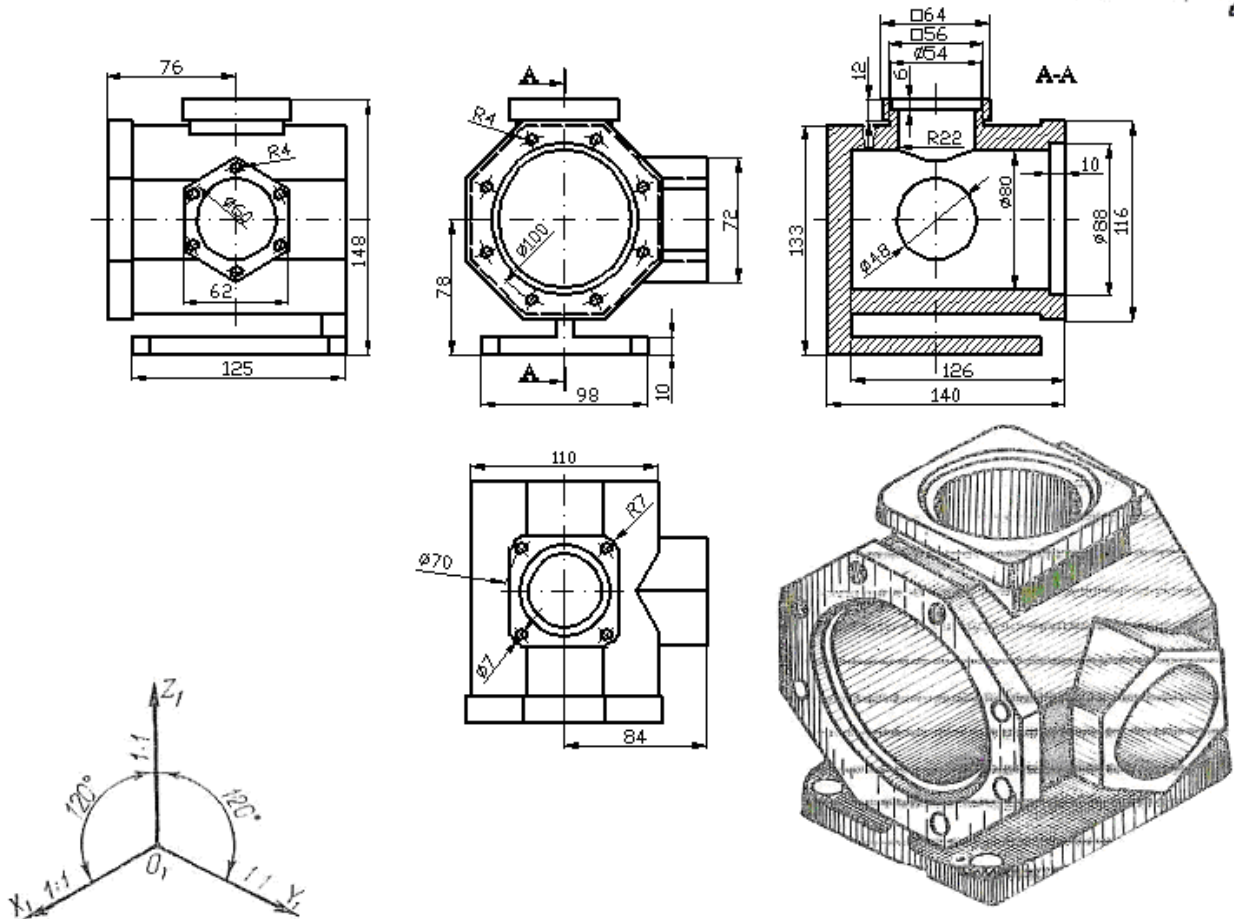


STUDIU PRIVIND MODELAREA 3D CU AJUTORUL PROGRAMULUI AutoCAD

APLICAȚIA 2

PLANȘA 27



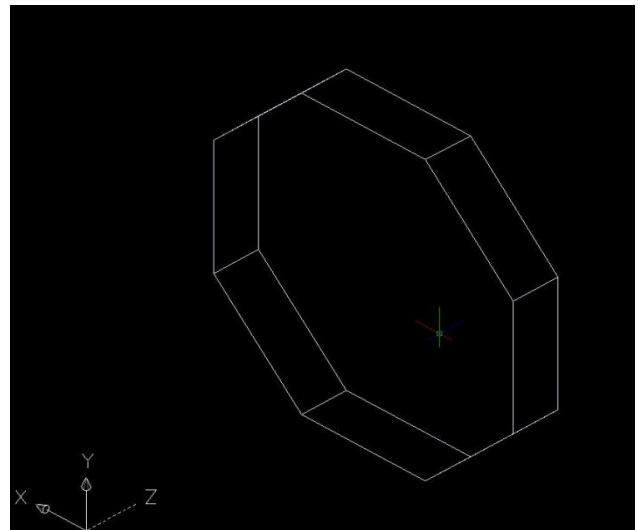
MODELAREA 3D

PASUL 1

Pentru reprezentarea reperului în 3D am trasat un poligon cu 8 laturi circumscris cercului de rază de 60mm.

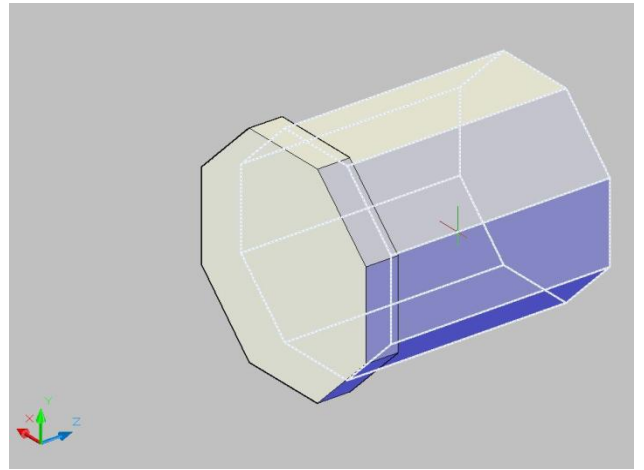
Prin comanda extrude vom trece în cea de a treia dimensiune de lungul axei Z. Înălțimea de extrudare va fi de 18 mm, așa cum este cotat în desenul de execuție.

>>>>>>>>>>>>>>



PASUL 2

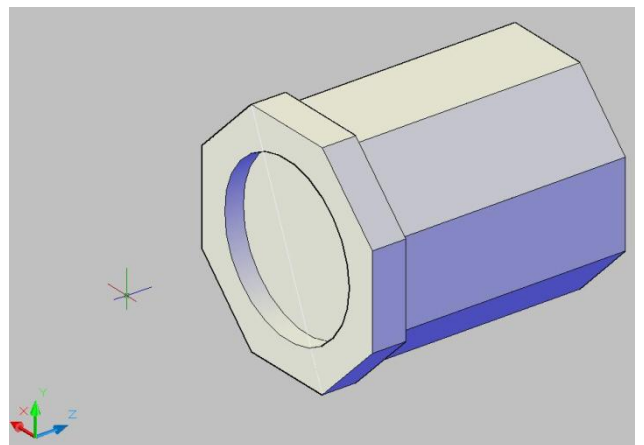
Se continuă cu un alt octogon circumscris, dar de data aceasta cercului cu raza de 55 mm și va fi extrudat de-a lungul axei Z pe înălțimea de 122 mm
>>>>>>>>>>



PASUL 3

Prin intermediul comenzii line am aflat centrul octogonului mare, pentru a putea trasa cercul ce trebuie extrudat pentru a găuri piesa. Cercul are diametrul de 88 mm, iar pentru găurire mai întâi am extrudat cercul cu 10 mm și am folosit comanda subtract.

Găurirea prin comanda subtract se face astfel : se selectează mai întâi piesa pe care vrem să o găurim (în cazul nostru octogonul mare) după care apăsăm Enter, apoi selectăm cilindrul, după care Enter. Se va obține imaginea : >>>>>>>

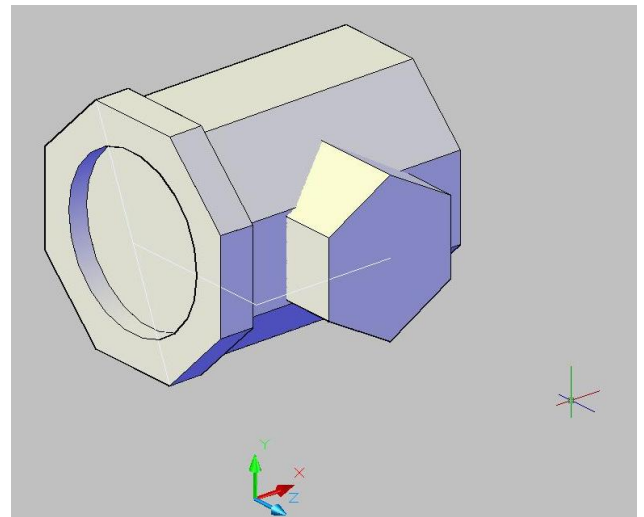


PASUL 4

De la mijlocul găurii executate vom desena cu line o linie de 84 mm de-a lungul axei Z și apoi în continuarea liniei de-a lungul axei X încă o linie de 76 mm, pentru a putea găsi centrul hexagonului ce va fi construit pe partea laterală a piesei.

Hexagonul este înscris într-un cerc cu raza de 40 mm și se construiește cu comanda polygon.

>>>>>

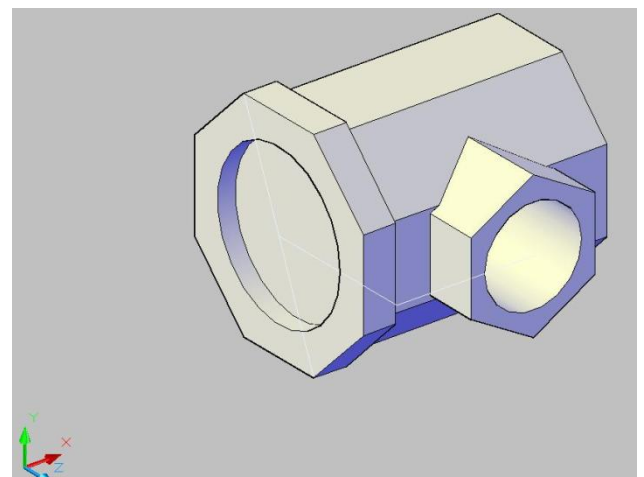


PASUL 5

În pasul următor vom construi un cerc cu diametrul de 54 mm și îl vom plasa în centrul poligonului cu 6 laturi și după aceea îl vom extruda cu o lungime de 84 mm.

Prin comanda subtract vom găuri hexagonul

>>>>>

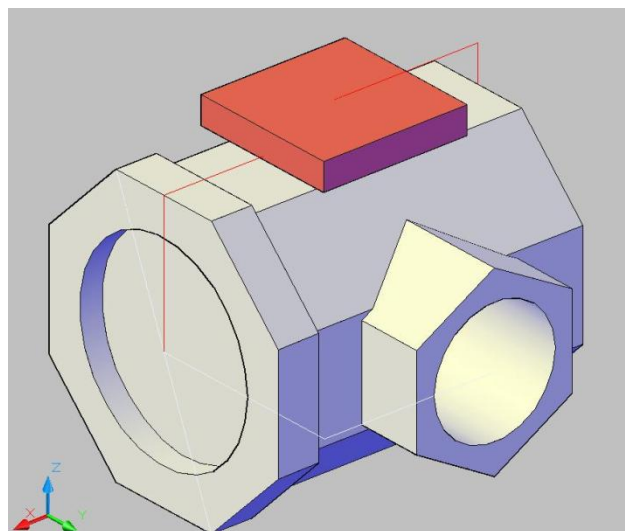


PASUL 6

Vom trece la partea superioară a piesei. Pentru a afla centrul pătratului, cu comanda *line* vom trasa linia de reper. Acum, când cunoaștem unde este centrul, vom construi un pătrat cu latura de 64 mm cu comanda *rectang*.

Pătratul este extrudat de-a lungul axei Z cu o înălțime de -12mm (valoarea este negativă pentru a putea extruda în partea opusă axei OZ)

>>>>>>

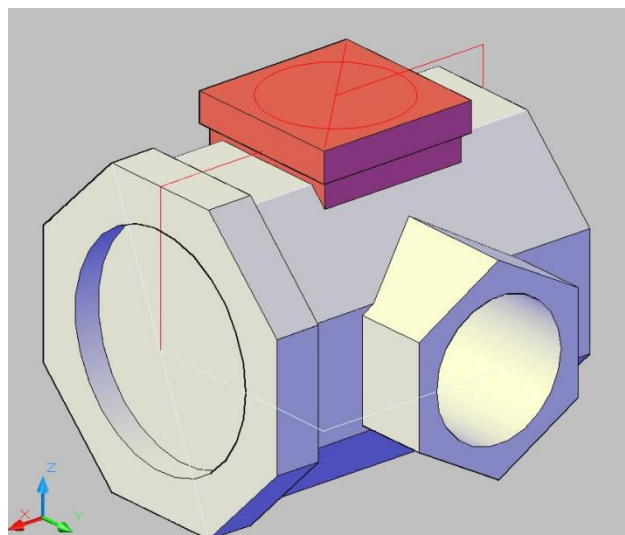


PASUL 7

În această etapă se va construi un pătrat cu latura de 59mm și va fi situat în centrul pătratului construit anterior, apoi se va extruda cu dimensiunea -40 mm.

Se va construi un cerc cu diametrul de 54 mm.

>>>>>>>>

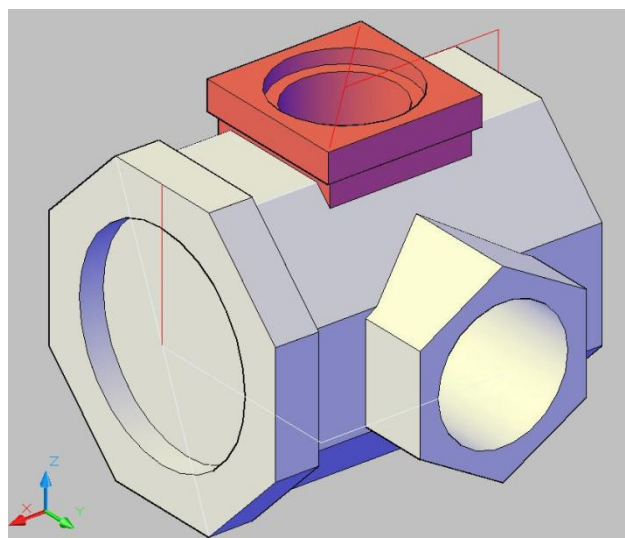


PASUL 8

Se va selecta cercul de 54 mm și va fi extrudat de-a lungul axei Z cu valoarea de -6mm, după care prin comanda *subtract* vom găuri piesa.

Următoarea etapă va fi construirea unui cerc cu diametrul de 44 mm, concentric găurii de 54mm, după care se va extruda cu o lungime de -35mm de-a lungul axei Z. Prin comanda *subtract* se va face gaura de diametrul 44 mm.

>>>>>>>>>>



PASUL 9

Vom construi octoconul mare de 120mm pentru a putea găuri restul piesei.

În centrul găurii cu diametrul de 88 mm vom construi un cerc cu raza de 40 mm, iar apoi îl vom extruda de-a lungul axei X cu -126 mm.

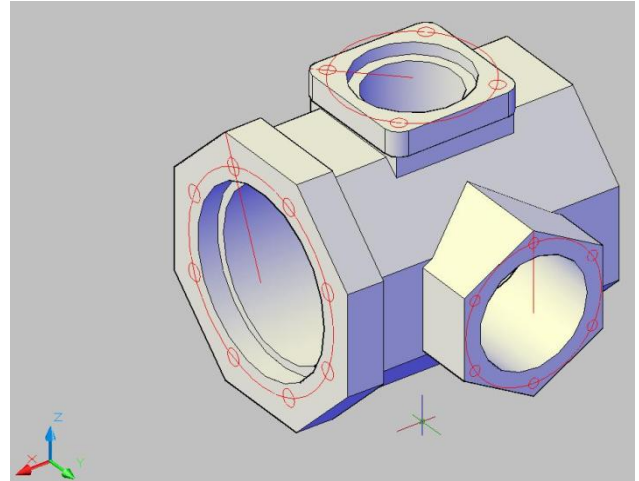
Folosim comanda subtract : vom selecta toate piesele componente în afară de cilindrul cu diametrul de 80 mm, apăsăm Enter și acum selectăm cilindrul după care Enter din nou.

După realizarea găurii vom duce cercuri de referință : prima dată pe octogon un cerc de rază 50, după care vom construi 8 cercuri, fiecare cu diametrul de 8mm.

Se va construi un alt cerc, cu diametrul de 68mm , pe hexagon după care încă 6 cercuri de rază 3mm.

Se construiește pe partea superioară un cerc cu diametrul de 70 mm, după care 4 cercuri cu raza de 3.5mm dispuse radial.(pentru dispunerea cercurilor radial am folosit comanda array).

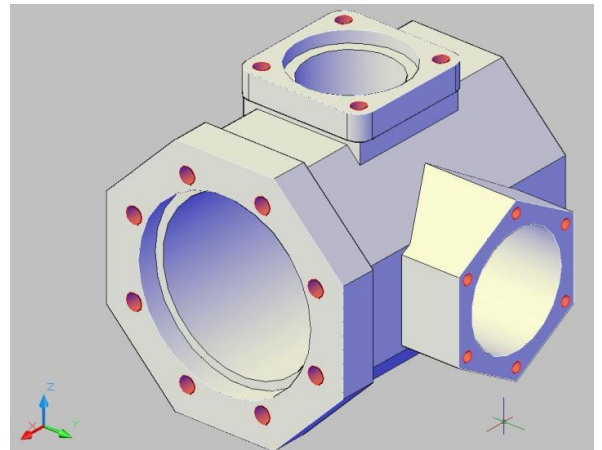
Rotunjirea colturilor cu raza de 7mm s-a fost realizat cu comanda fillet .



PASUL 10

În această etapă se modelează găurile.

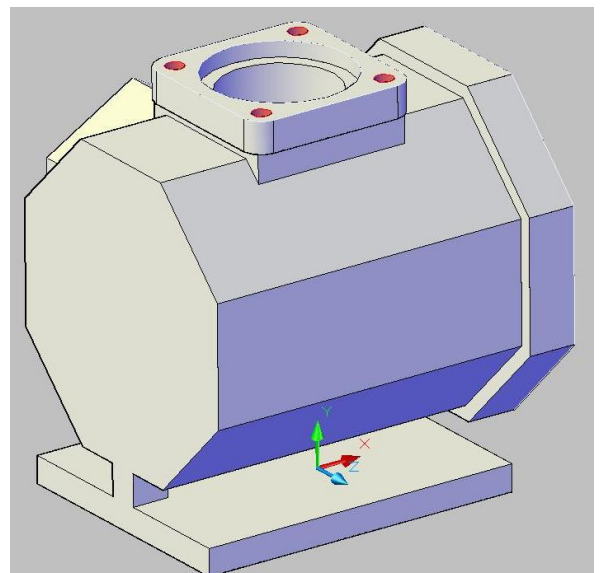
Se vor selecta toate cercurile reprezentate cu culoarea roșie în desen și se vor extruda la o înălțime de 10mm, după care se va folosi comanda subtract . >>>>>>



PASUL 11

Vom construi, conform desenului de execuție, piciorul care este format dintr-un dreptunghi cu lățimea de 98 mm și lungimea de 122 mm.

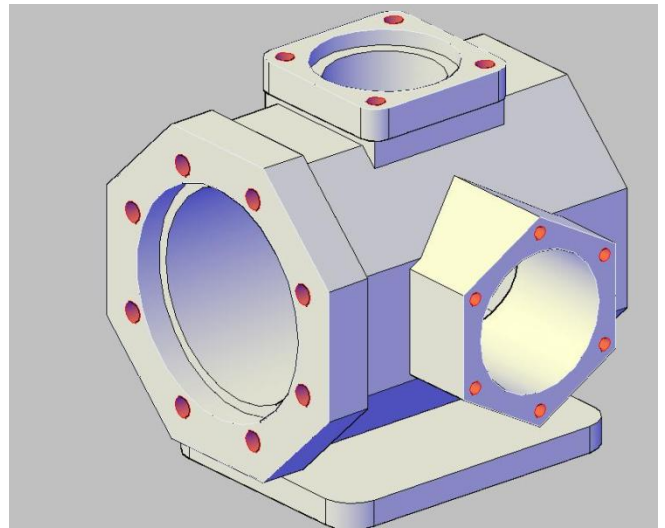
>>>>>>



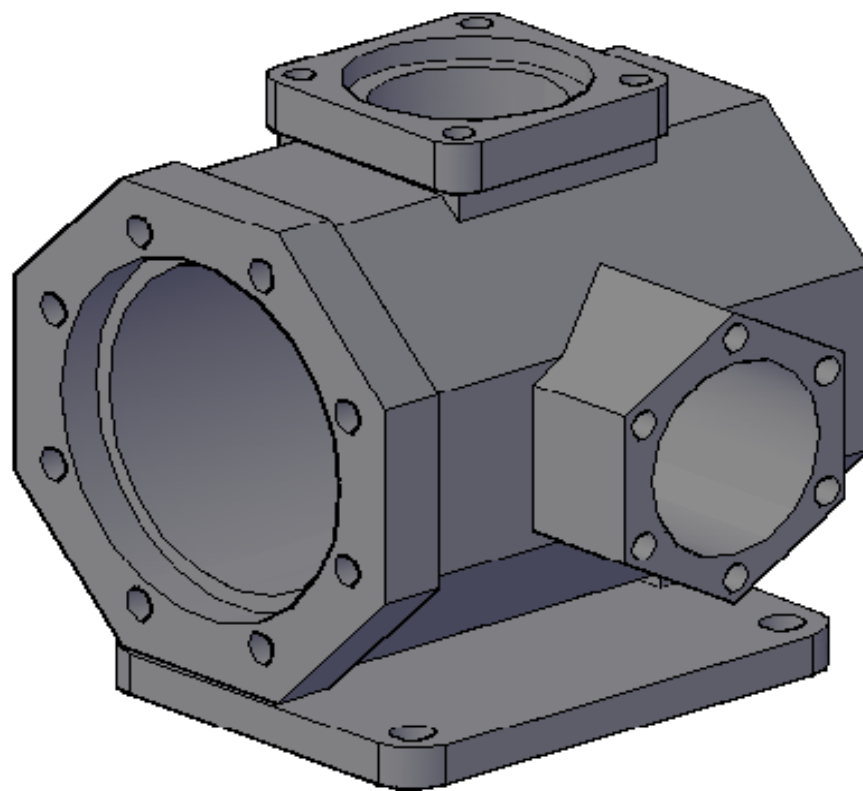
PASUL 12

Se va finaliza piesa rotunjind colțurile suportului cu raza de 12 mm și comanda fillet.

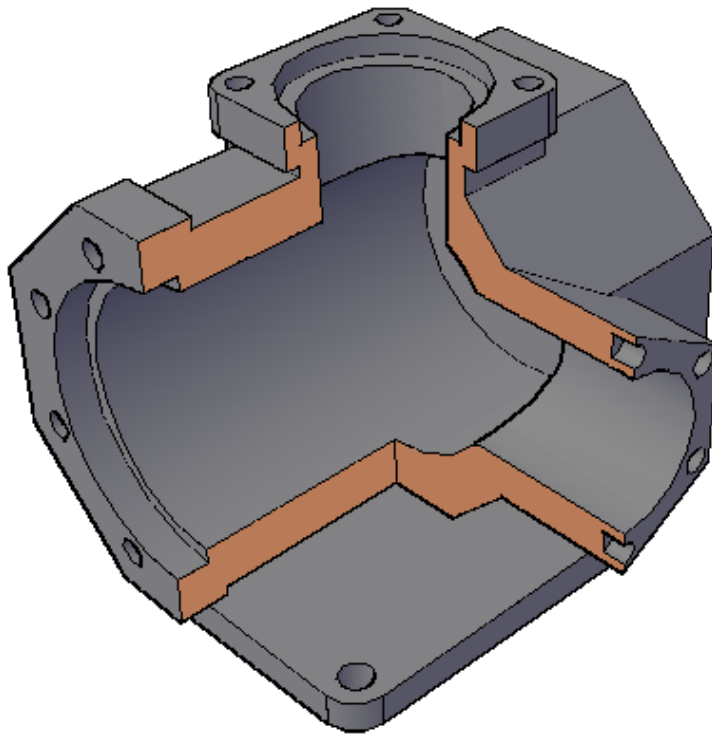
>>>>>>>



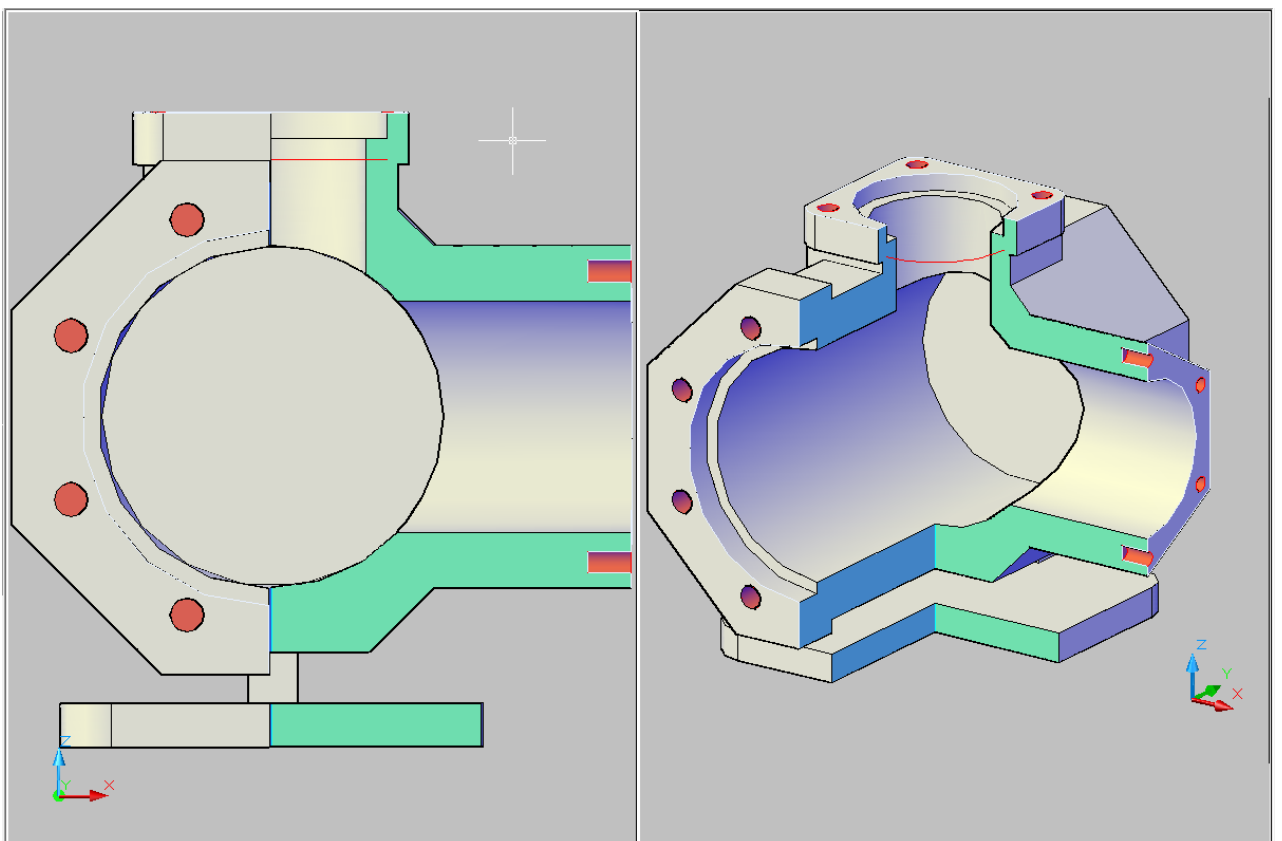
REPREZENTAREA REPERULUI MODELAT BRUT



SECȚIUNEA MODELULUI

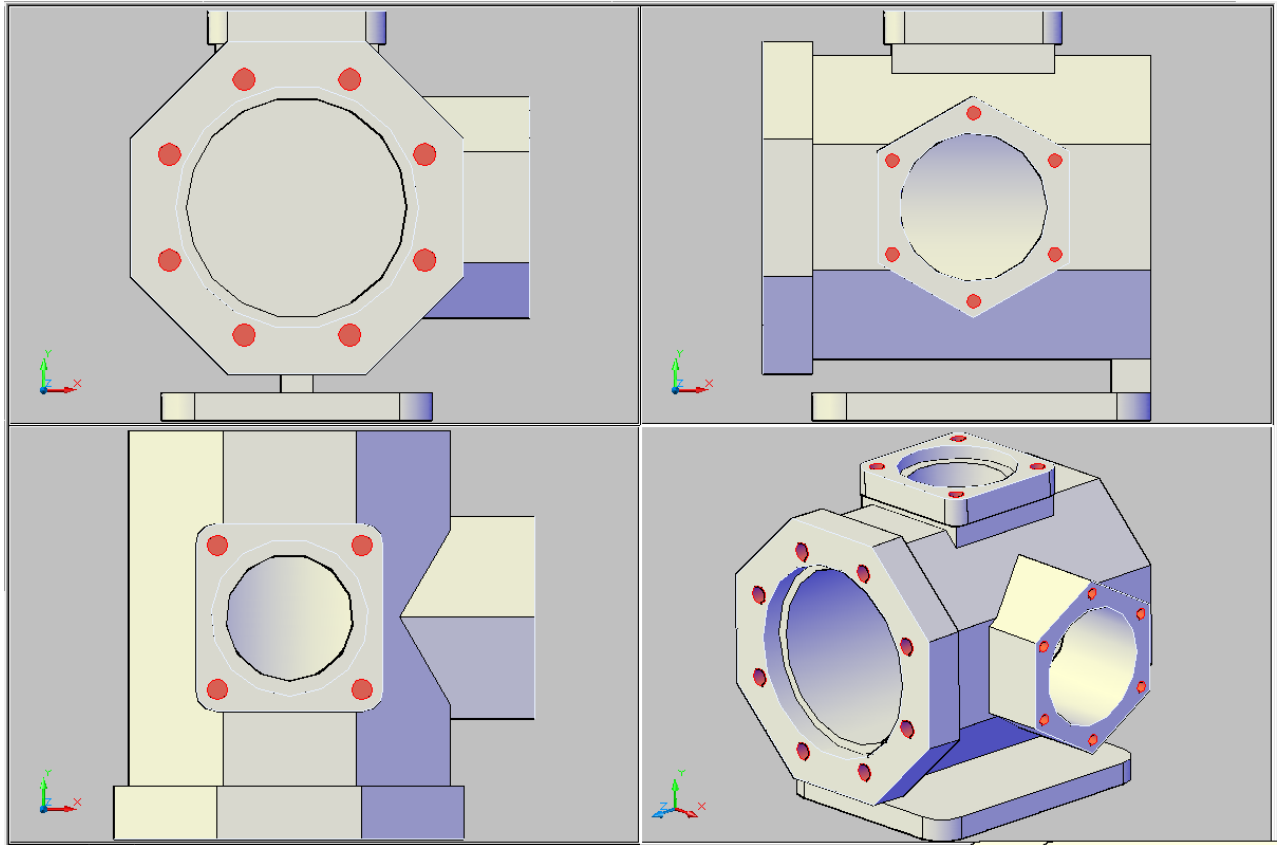


REPREZENTAREA ÎN DOUĂ VIEWPORTS

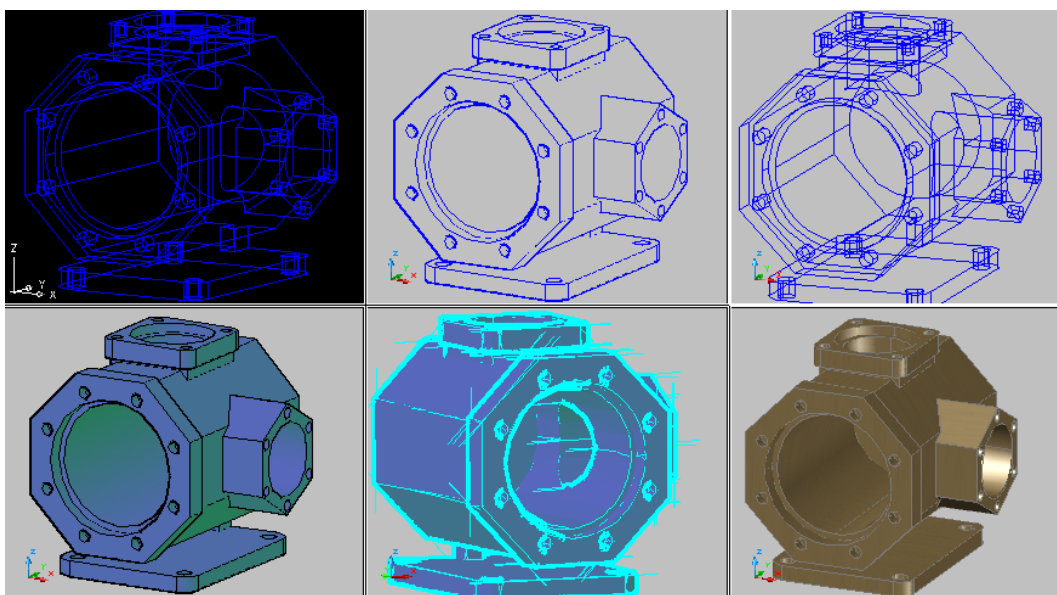


Rezentarea in 4 Viewports
ne permite vizualizarea piesei din toate unghiurile.

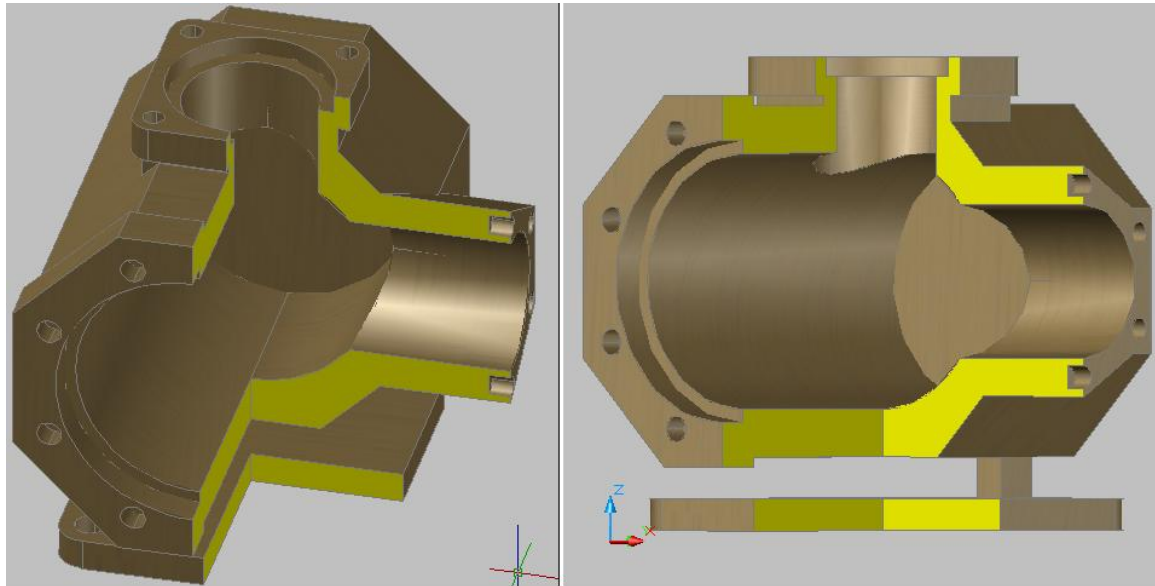
VI
Z
U
A
L
I
Z
A
R
E
A
C
A
R
E
Ț
E
A
W



WIREFRAME
(cu toate muchiile vizibile și cu muchiile acoperite ascunse)



APLICAREA MATERIALULUI METALIC ȘI A LUMINIILOR



Comenzi folosite:

AutoCAD menu utilities loaded.

Command: COMMANDLINE

Command: DASHBOARD

Command: units

Command: l

LINE Specify first point:

Specify next point or [Undo]: <Ortho on> 140

Command: '_zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: _e

Regenerating model.

Command: Regenerating model.

Command: ucs

Current ucs name: *TOP*

Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis]

<World>: y

Specify rotation angle about Y axis <90>:

Command: _polygon Enter number of sides <4>: 8

Specify center of polygon or [Edge]:

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: C

Specify radius of circle: 60

Command: _extrude

Current wire frame density: ISOLINES=41 found

Specify height of extrusion or [Direction/Path/Taper angle]: 15

Command: _cylinder

Specify center point of base or [3P/2P/Ttr/Elliptical]:

Specify base radius or [Diameter]: d

Specify diameter: 88

Specify height or [2Point/Axis endpoint] <15.0000>: 10

Command:

CYLINDER

Specify center point of base or [3P/2P/Ttr/Elliptical]:
Specify base radius or [Diameter] <44.0000>: d
Specify diameter <88.0000>: 80
Specify height or [2Point/Axis endpoint] <10.0000>: 116
Command: _polygon Enter number of sides <8>:
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <C>: C

Specify radius of circle: 55
Command: _extrude
Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <116.0000>: 125
Command: Regenerating model.

Command: _qsave
Command: _subtract Select solids and regions to subtract from ..
Select objects: 1 found
Select objects: 1 found, 2 total
Command: _subtract Select solids and regions to subtract from ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:

Select solids and regions to subtract ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Select objects: 1 found, 5 total
Select objects: 1 found, 6 total
Select objects: 1 found, 7 total
Select objects: 1 found, 8 total
Select objects: 1 found, 9 total
Select objects: 1 found, 10 total
Command: _vscurrent
Enter an option [2dwireframe/3dwireframe/3dHidden/Realistic/Conceptual/Other]
<2dwireframe>: _C
Command: '_3dorbit Press ESC or ENTER to exit, or right-click to display
shortcut-menu.

** STRETCH **
Specify stretch point or [Base point/Copy/Undo/eXit]:
Command: _vscurrent
Enter an option [2dwireframe/3dwireframe/3dHidden/Realistic/Conceptual/Other]
<Conceptual>: _C
Command: '_3dorbit Press ESC or ENTER to exit, or right-click to display
shortcut-menu.

Command: _polygon Enter number of sides <6>:
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: I
Specify radius of circle: 38
Command: l
LINE Specify first point:
Specify next point or [Undo]: 40
Command: _extrude

Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <-84.0000>: 44
Command: <Osnap off>
** STRETCH **
Specify stretch point or [Base point/Copy/Undo/eXit]:
Command: _erase 1 found
Command: _polygon Enter number of sides <6>: <Osnap on>
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: I
Specify radius of circle: 36
Command: _extrude
Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <-44.0000>: 44
Command: '_3dorbit Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: l
LINE Specify first point:
Specify next point or [Undo]: 35
Command: c
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <27.0000>: d
Specify diameter of circle <54.0000>: 54
Command: c
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <27.0000>: d
Specify diameter of circle <54.0000>: 68
Command: _polygon Enter number of sides <6>:
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: I
Specify radius of circle: <Osnap off>
Automatic save to C:\DOCUME~1\cata\LOCALS~1\Temp\cataaa_1_2_2449.sv\$...
Command: _extrude
Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <-44.0000>: 40
Command: c
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <34.0000>: <Osnap off>
Command: l
LINE Specify first point: <Osnap on>
Specify next point or [Undo]: *Cancel*
Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan
radius)]:
Specify radius of circle or [Diameter] <30.4335>: d
Specify diameter of circle <60.8671>: 60
Command: _cylinder
Specify center point of base or [3P/2P/Ttr/Elliptical]:
Specify base radius or [Diameter] <4.0000>: d

Specify diameter <8.0000>: <Osnap off>
Resuming CYLINDER command.
Specify diameter <8.0000>: 7
Specify height or [2Point/Axis endpoint] <-40.0000>: 7

Command: _array 1 found
Specify center point of array: <Osnap on>
Command: c
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <30.0000>: 24
Command: '_3dorbit Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: _extrude
Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <-7.0000>: 84
Command: _subtract Select solids and regions to subtract from ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:
Select solids and regions to subtract ..
Select objects: 1 found
Command:
SUBTRACT Select solids and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids and regions to subtract ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Select objects: 1 found, 5 total
Select objects: 1 found, 6 total
Command: l
LINE Specify first point:
Specify next point or [Undo]: 70
Automatic save to C:\DOCUME~1\cata\LOCALS~1\Temp\cataaa_1_2_2449.sv\$...
Command: Specify opposite corner:
Command: _extrude
Current wire frame density: ISOLINES=41 found
Specify height of extrusion or [Direction/Path/Taper angle] <26.9694>:
Command: _vscurrent
Enter an option [2dwireframe/3dwireframe/3dHidden/Realistic/Conceptual/Other]
<2dwireframe>: _C
Command: <Osnap off>
Command: _cylinder
Specify center point of base or [3P/2P/Ttr/Elliptical]:
Specify base radius or [Diameter] <22.0000>: d
Specify diameter <44.0000>: 7
Specify height or [2Point/Axis endpoint] <31.0604>:
Resuming CYLINDER command.
Specify height or [2Point/Axis endpoint] <31.0604>: 7
Command: _array 1 found
Command: _copy 1 found
Current settings: Copy mode = Multiple
Specify base point or [Displacement/mOde] <Displacement>:
Checking 1081 intersections... Specify second point or <use first point as
displacement>:

Specify second point or [Exit/Undo] <Exit>:
Command: _subtract Select solids and regions to subtract from ..
Select objects:
Resuming SUBTRACT command.
Select objects: 1 found
Select objects:
Select solids and regions to subtract ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Command: _subtract Select solids and regions to subtract from ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Command: l
LINE Specify first point:
Specify next point or [Undo]: 78
Specify next point or [Undo]:
Command: l
LINE Specify first point:
Specify next point or [Undo]: 49
Specify next point or [Undo]: 125
Specify next point or [Close/Undo]:

Command: _box
Specify first corner or [Center]:
Specify other corner or [Cube/Length]: 125
Specify height or [2Point] <-7.0000>: 10

Command: _fillet
Current settings: Mode = TRIM, Radius = 7.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Command:
FILLET
Current settings: Mode = TRIM, Radius = 7.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: r
Specify fillet radius <7.0000>: 10
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <10.0000>:
Select an edge or [Chain/Radius]:
1 edge(s) selected for fillet.
Command:
FILLET
Current settings: Mode = TRIM, Radius = 10.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <10.0000>:
Select an edge or [Chain/Radius]:
1 edge(s) selected for fillet.
Command:
FILLET
Current settings: Mode = TRIM, Radius = 10.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <10.0000>:
Select an edge or [Chain/Radius]:
1 edge(s) selected for fillet.
Command:
FILLET
Current settings: Mode = TRIM, Radius = 10.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Resuming FILLET command.
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <10.0000>:
Select an edge or [Chain/Radius]:
1 edge(s) selected for fillet.
Command: _cylinder
Specify center point of base or [3P/2P/Ttr/Elliptical]:
Specify base radius or [Diameter] <3.5000>: d
Specify diameter <7.0000>: 10
Specify height or [2Point/Axis endpoint] <-10.0000>: 10
Command: _copy 1 found
Current settings: Copy mode = Multiple
Specify base point or [Displacement/mOde] <Displacement>:
Checking 946 intersections... Specify second point or <use first point as displacement>:
Specify second point or [Exit/Undo] <Exit>:
Command: _subtract Select solids and regions to subtract from ..
Select objects:
Command:
SUBTRACT Select solids and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids and regions to subtract ..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:
Resuming SUBTRACT command.
Select objects: 1 found, 3 total
Select objects:
Resuming SUBTRACT command.
Select objects: 1 found, 4 total
Command: _box
Specify first corner or [Center]:
Specify other corner or [Cube/Length]: 14
Specify height or [2Point] <10.0000>:
Command: _erase 1 found