

Liceul „DIMITRIE NEGREANU”BOTOȘANI

FIȘĂ DE DOCUMENTARE

CLASA A XI A

Profesor : ALEXA CAMELIA

M2- Aplicații CAD

Denumirea temei:

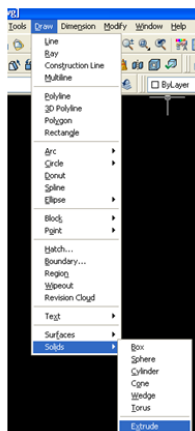
CREAREA MODELELOR 3D UTILIZÂND COMENZILE : EXTRUDE, SUBTRACT, UNION. APLICATII.

Obținerea obiectelor tridimensionale se poate realiza și pornind de la profile plane, cărora li se asociază o a treia dimensiune, utilizând comanda EXTRUDE. În programul AutoCad extrudarea prezintă o operație prin care se atribuie grosime unui profil plan în contur închis sau suprafața (polilinie, cerc, elipsa, regiune, curba etc.). Întodeauna, extrudarea are loc după o direcție în raport cu planul figurii plane considerate sau după o traiectorie (path). În cazul extrudării după o direcție , solidul generat are una din baze în coincidență în figura plană extrudată, iar cea de-a doua bază paralelă cu prima, la distanța specificată (pozitivă sau negativă). Cele două baze au arii egale dacă unghiul de extrudare este 0. Pentru un unghi de extrudare pozitiv, aria celeia de-a doua baze este mai mică decât a primei baze. Solidul obținut va avea, astfel, aspect ascuțit, respectiv largit, în zona celei de-a doua baze. Pentru ca extrudarea după o traiectorie să fie posibilă, este necesar ca profilul plan închis și traiectoria să nu fie coplanare. De asemenea, polilinia care formează traiectoria nu trebuie să prezinte schimbări bruște de direcție, sub unghuri foarte diferite. În acest caz, solidul se autointersectează, iar operația de extrudare nu mai este posibilă.

Comanda EXTRUDE are ca efect crearea de corpuri solide prin extrudarea poliliniilor închise, a cercurilor, elipselor, curbelor spline închise sau regiunilor.



Meniul derulant Draw » Solids » Extrude



Linia de comandă

Command: _extrude

Current wire frame density: ISOLINES=4

Select objects: (se selectează curba închisă)

Specify height of extrusion or [Path]: (se specifică înălțimea dorită sau calea de-a lungul căreia se face extrudarea)

Specify angle of taper for extrusion [Path]<0>: (se indică unghiul de înclinare dorit sau calea de-a lungul căreia se face extrudarea)



OBȚINEREA SOLIDELOR PRIN OPERAȚII LOGICE

Asupra obiectelor solide se pot aplica operații logice, obținându-se astfel obiecte solide cu o arhitectura complexă. Astfel, se pot folosi comenzile **Intersect**, **Subtract** și **Union**, prin acționarea pictogramelor caracteristice ale meniului grafic **Solids Editing**

UNION

Comanda **Union** permite crearea volumului rezultat prin reunirea unor solide, evidențiind și curbele de intersecție dintre suprafețe.

Dialogul comenzii este:

Command: Union

Select objects: Se selectează obiectele de editat (unit)

Select objects: <Enter>

SUBTRACT

Prin comanda **Subtract** se pot obține volumele rezultate din aplicarea operației logice de diferență asupra unor solide.

Dialogul comenzii este:



Command: Subtract

Select objects to subtract from: se selectează obiectele din care se face extracția multimedială de puncte

Select objects to subtract: se selectează obiectele care trebuie extrase


INTERSECT

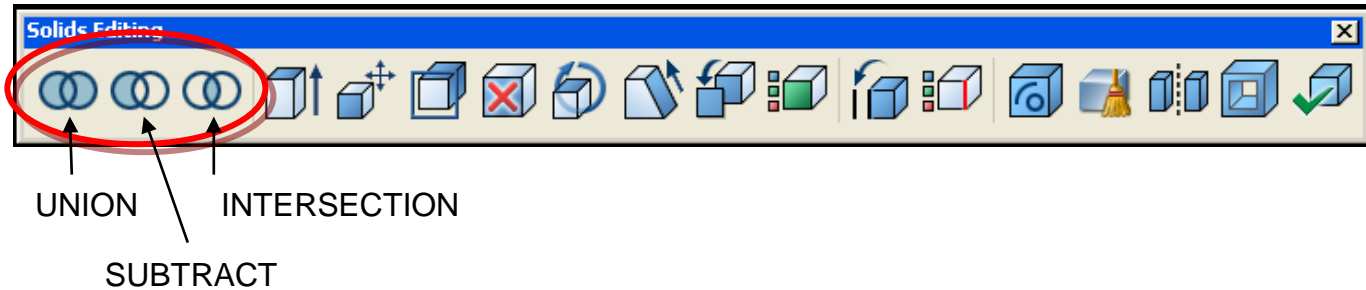
Prin comanda **Intersect** se poate obține volumul de intersecție dintre două sau mai multe solide.

Dialogul comenzii este:

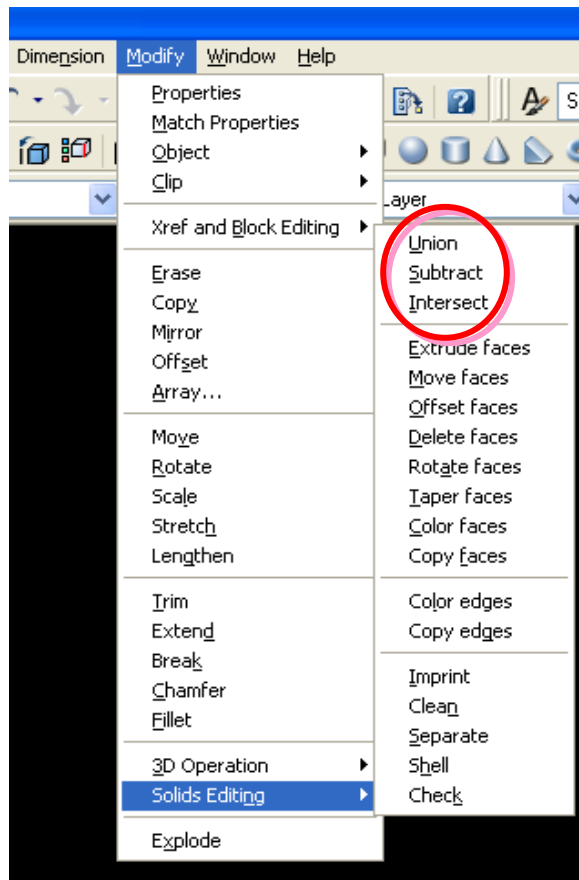
 **Command:** Intersect

Select objects: Selectați obiectele de intersectat. Zonele ramase vor fi indepartate din jurul solidului rezultat.

 Bara de instrumente **Solids Editing**



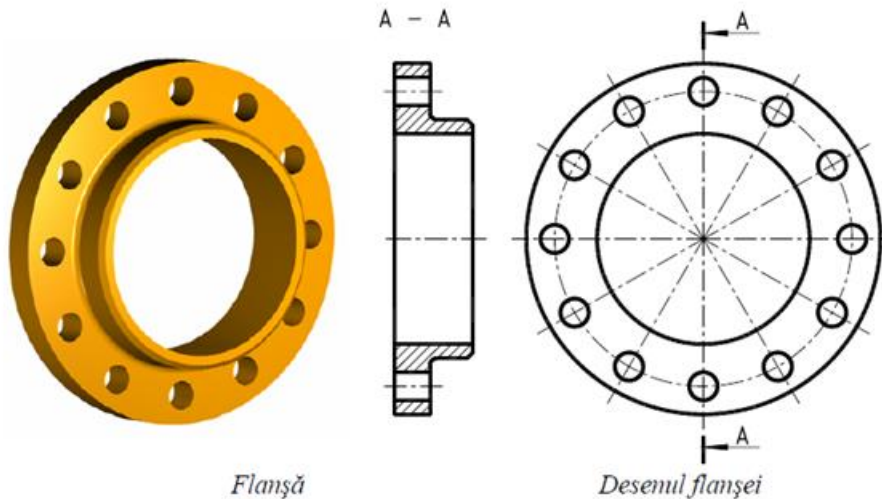
 Meniul derulant: **Modify » Solids Editing »**



APLICATII

REPREZENTATI O FLANSA ROTUNDA SI O FLANSA PATRATA FOLOSIND COMENZILE EXTRUDE, SUBTRACT, UNION:

EXEMPLUL I – MODELAREA FLANȘEI ROTUNDE



Pentru obținerea flanșei rotunde se vor folosi următoarele operații booleene:

- **UNION**
- **SUBTRACT**

Command: `_ucs`

Current ucs name: *WORLD*

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]

<World>: `_move`

Specify new origin point or [Zdepth]<0,0,0>:

Command: CIRCLE Specify center point for circle or [3P/2P/Tr (tan tan radius)]: 0,0

Specify radius of circle or [Diameter]: 100

Command: CIRCLE Specify center point for circle or [3P/2P/Tr (tan tan radius)]: 0,0

Specify radius of circle or [Diameter] <100.0000>: 50

Command: CIRCLE Specify center point for circle or [3P/2P/Tr (tan tan radius)]: 0,0

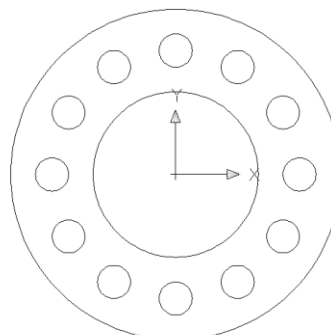
Specify radius of circle or [Diameter] <50.0000>: 75

Command: `_line` Specify first point: 0,0

COMMAND: ARRAY

Select objects: 1 found

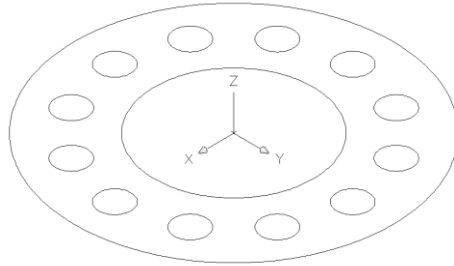
Select objects:



COMMAND: VPOINT

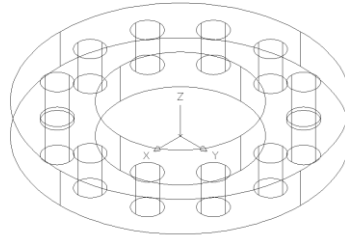
Current view direction: `VIEWDIR=0.0000,0.0000,1.0000`

Specify a view point or [Rotate] <display compass and tripod>: 1,1,1
Regenerating model.



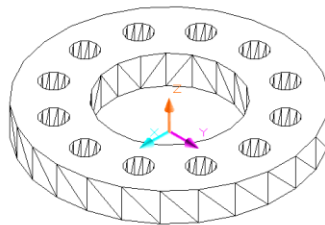
COMMAND: EXTRUDE

Current wire frame density: ISOLINES=4
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: Specify opposite corner: 14 found (2 duplicate), 14 total
Select objects:
Specify height of extrusion or [Path]: 25
Specify angle of taper for extrusion <0>:



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
Select objects:

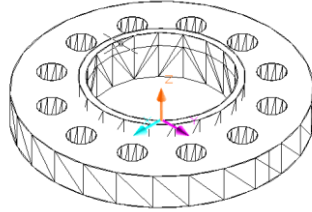
Command: shade

Command: hide

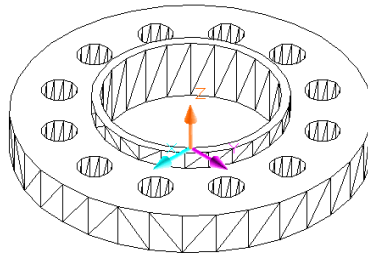
Command: CYLINDER

Current wire frame density: ISOLINES=4
Specify center point for base of cylinder or [Elliptical] <0,0,0>: 0,0,0
Specify radius for base of cylinder or [Diameter]: 55
Specify height of cylinder or [Center of other end]: 35

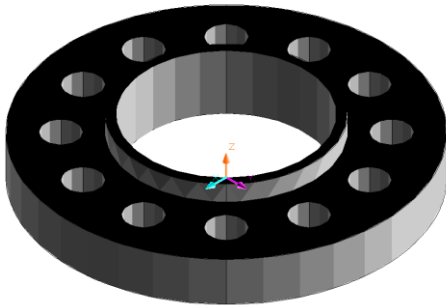
Command: CYLINDER
Current wire frame density: ISOLINES=4
Specify center point for base of cylinder or [Elliptical] <0,0,0>: 0,0,0
Specify radius for base of cylinder or [Diameter]: 50
Specify height of cylinder or [Center of other end]: 35
Command: **SUBTRACT**
Select solids and regions to subtract from ..
Select objects: 1 found

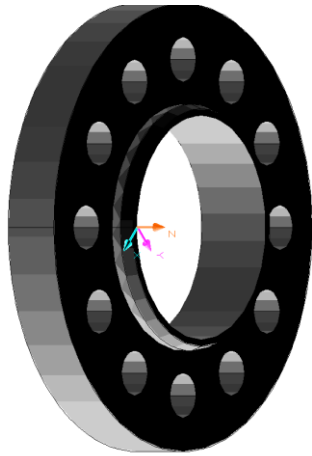


Select objects:
Command: **UNION**
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:

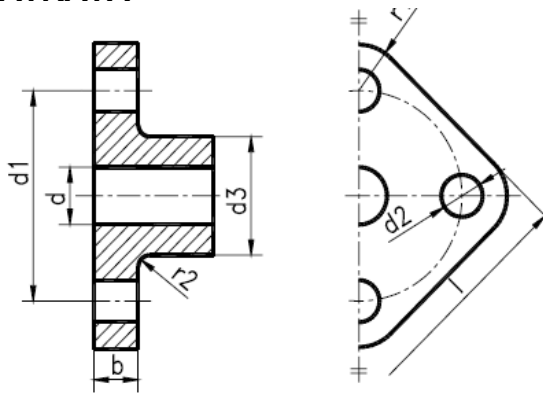


Command: **CHAMFER**
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Polyline/Distance/Angle/Trim/Method/mUltiple]:
Base surface selection...
Enter surface selection option [Next/OK (current)] <OK>:
Specify base surface chamfer distance: 2
Specify other surface chamfer distance <2.0000>: Specify second point: Select
an edge or [Loop]: Select an edge or [Loop]:





EXEMPLUL 2. FLANȘA PĂTRATĂ



d	d1	d2	d3	l	b	r1	r2
10	50	11	26	(59)	12	12	3
15	55	11	31	(64)	12	12	3
20	65	11	38	(79)	14	16	4
25	75	11	47	(86)	14	16	4
32	90	14	56	(100)	16	18	4
40	100	14	64	(108)	16	18	4
50	110	14	74	(114)	16	18	4
65	130	14	89	(130)	16	18	4

Pentru obținerea modelului flanșei pătrate s-au folosit următoarele comenzi:
AutoCAD menu utilities loaded.

COMMAND: _UCS

Current ucs name: *WORLD*

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]

<World>: _move

Specify new origin point or [Zdepth]<0,0,0>:

Command: CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 0,0

Specify radius of circle or [Diameter]: 6

Command: '_3dpan Press ESC or ENTER to exit, or right-click to display shortcut-menu.

Command: CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 0,0

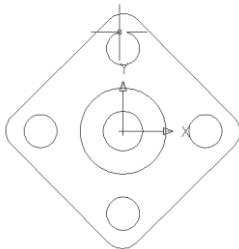
Specify radius of circle or [Diameter] <6.0000>: `3

Specify radius of circle or [Diameter] <6.0000>: 13

Command: CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 0,0

Specify radius of circle or [Diameter] <13.0000>: 50

Command: CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 0,0
Specify radius of circle or [Diameter] <50.0000>: 25
Command: Specify opposite corner:
Command: LINE Specify first point:
Specify next point or [Undo]:
Command: CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <25.0000>: 5
COMMAND: ARRAY
Select objects: Specify opposite corner: 1 found
Select objects:
COMMAND: POLYGON
Enter number of sides <4>: 4
Specify center of polygon or [Edge]: 0,0
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: 60
Specify radius of circle: 38
COMMAND: ROTATE
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0
Select objects: 1 found
Specify base point: 0,0
Specify rotation angle or [Reference]:
COMMAND: FILLET
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or [Polyline/Radius/Trim/mUltiple]: r
Specify fillet radius <0.0000>: 6
Select first object or [Polyline/Radius/Trim/mUltiple]:
Command: FILLET
Current settings: Mode = TRIM, Radius = 6.0000
Select first object or [Polyline/Radius/Trim/mUltiple]:
Command: FILLET
Current settings: Mode = TRIM, Radius = 6.0000
Select first object or [Polyline/Radius/Trim/mUltiple]:
Select first object or [Polyline/Radius/Trim/mUltiple]:
Command: FILLET
Current settings: Mode = TRIM, Radius = 6.0000
Select first object or [Polyline/Radius/Trim/mUltiple]:
Command: FILLET
Current settings: Mode = TRIM, Radius = 6.0000
Select first object or [Polyline/Radius/Trim/mUltiple]:
Select first object or [Polyline/Radius/Trim/mUltiple]:
Select first object or [Polyline/Radius/Trim/mUltiple]:

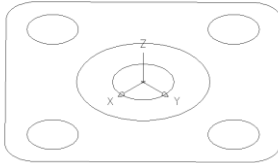


COMMAND: VPOINT
*** Switching to the WCS ***
Current view direction: VIEWDIR=0.0000,0.0000,1.0000

Specify a view point or [Rotate] <display compass and tripod>: 1,1,1

** Returning to the UCS ***

Regenerating model.



COMMAND: EXTRUDE

Current wire frame density: ISOLINES=4

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Specify height of extrusion or [Path]: 13

Specify angle of taper for extrusion <0>:

COMMAND: SUBTRACT

Select solids and regions to subtract from ..

Select objects: 1 found

Select objects:

Select solids and regions to subtract ..

Select objects: Specify opposite corner: 1 found

COMMAND: SUBTRACT

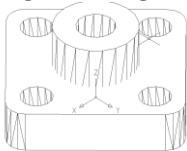
Select solids and regions to subtract from ..

Select objects: 1 found

Select objects: 1 found, 2 total

COMMAND: HIDE

Regenerating model.



COMMAND: UNION

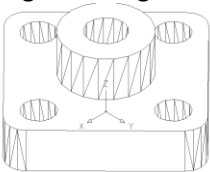
Select objects: 1 found

Select objects: Specify opposite corner: 1 found, 2 total

Select objects:

COMMAND: HIDE

Regenerating model



COMMAND: SHADE

Command: '_layer

Command: '_layer

