









Primitive Solids

SHAPE	COMMAND	ICON	DESCRIPTION
BOX	BOX	 Box	Creates a solid box after you provide 2 opposite corners and a height.
SPHERE	SPHERE / SPH	 Sphere	Creates a solid sphere from a center point and radius.
CYLINDER	CYLINDER / CYL	 Cylinder	Creates a straight cylinder from a center point, radius and height.
CONE	CONE	 Cone	Creates a tapered cone from a center point, radius and height.
WEDGE	WEDGE / WED	 Wedge	Creates a triangular wedge from 2 opposite points.
TORUS	TORUS / TOR	 Torus	Creates a torus (donut shape) based on center point, radius and tube radius.
PYRAMID	PYRAMID / PYR	 Pyramid	Draws a solid object with a polygon (3-32 sides) base that rises to a central point.
POLYSOLID	PSOLID	 Polysolid	Draws a solid object with width and height as you would draw a polyline.



Desenhar o perfil do copo usando
Lines e Arcs

Superfícies de revolução

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The screenshot shows the AutoCAD software interface with the Revolve command help window open. The help window contains the following text:

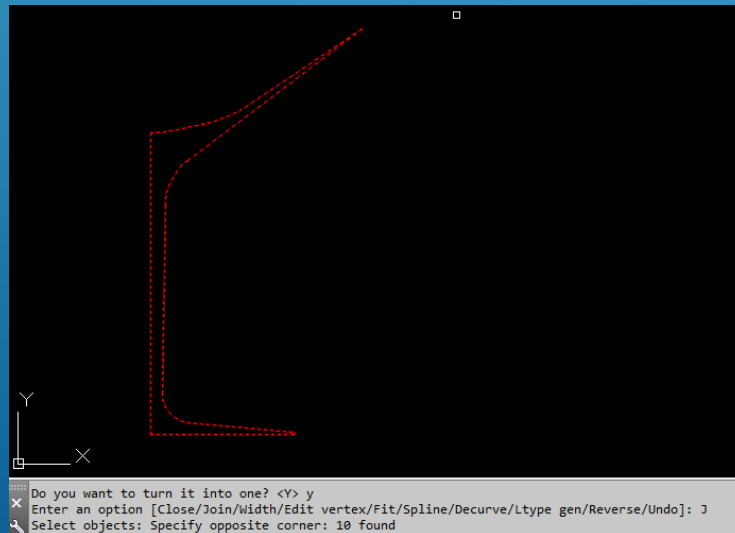
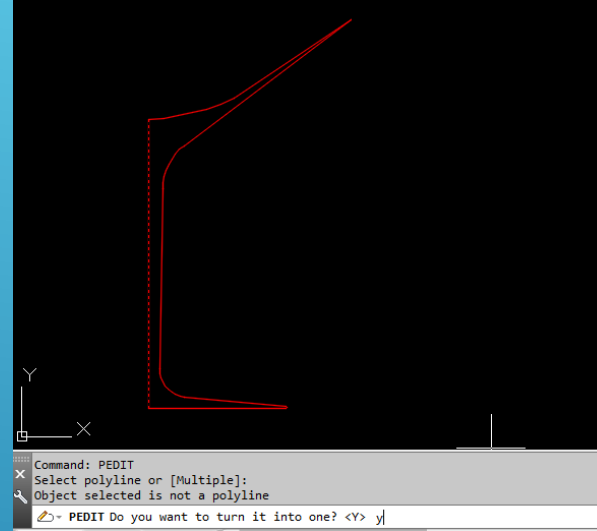
Revolve
Creates a 3D solid or surface by sweeping a 2D or 3D curve around an axis

When the Solid tab is active, the REVOLVE command creates a solid. Conversely, when the Surface tab is active, it creates a surface (either procedural or NURBS depending on how the SURFACEMODELINGMODE system variable is set).

REVOLVE
Press F1 for more help

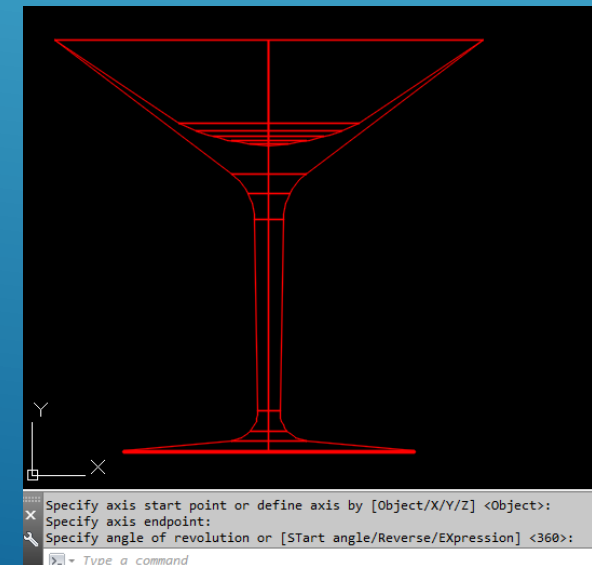
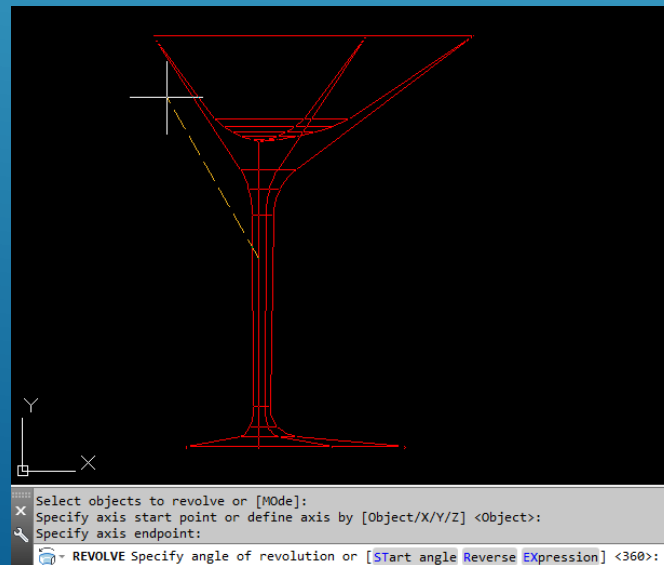
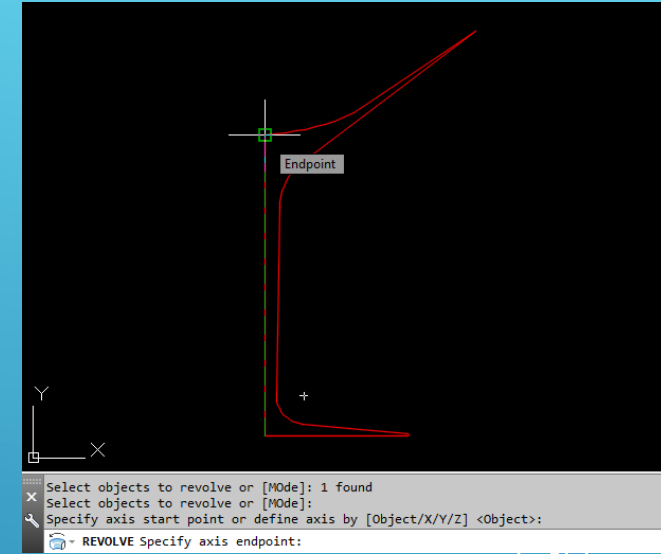
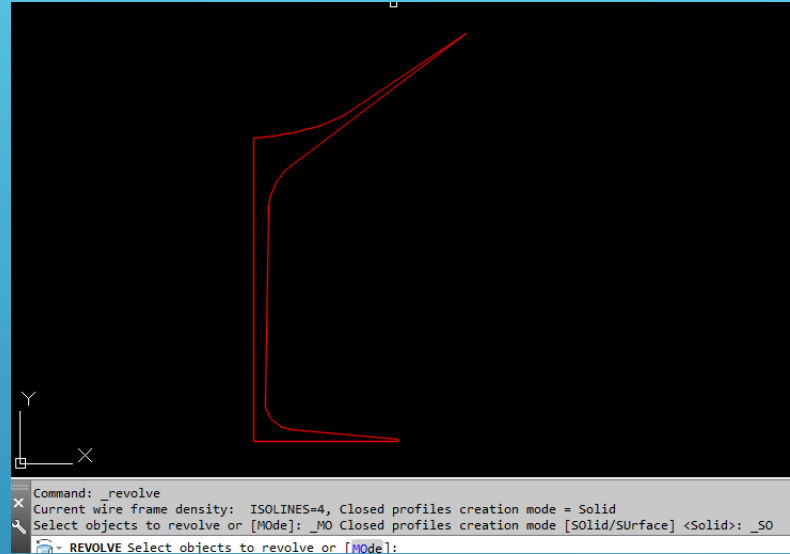
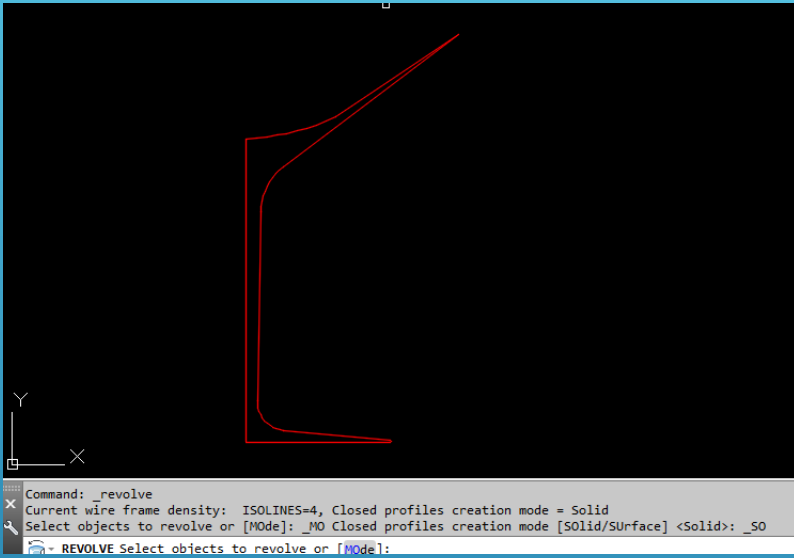
9 segments added to polyline
Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Lt

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Usar o comando PEDIT para transformar os elementos LINES e ARCS numa POLYLINE

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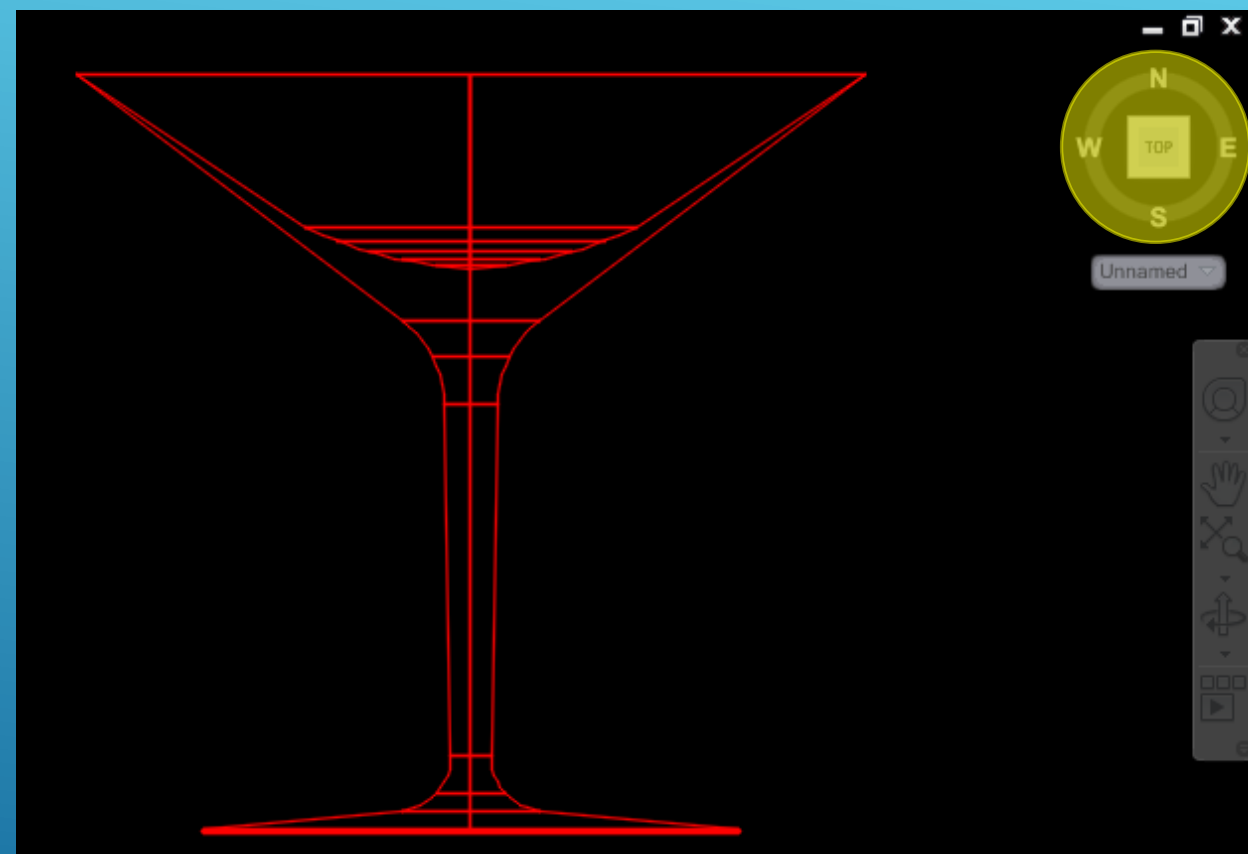
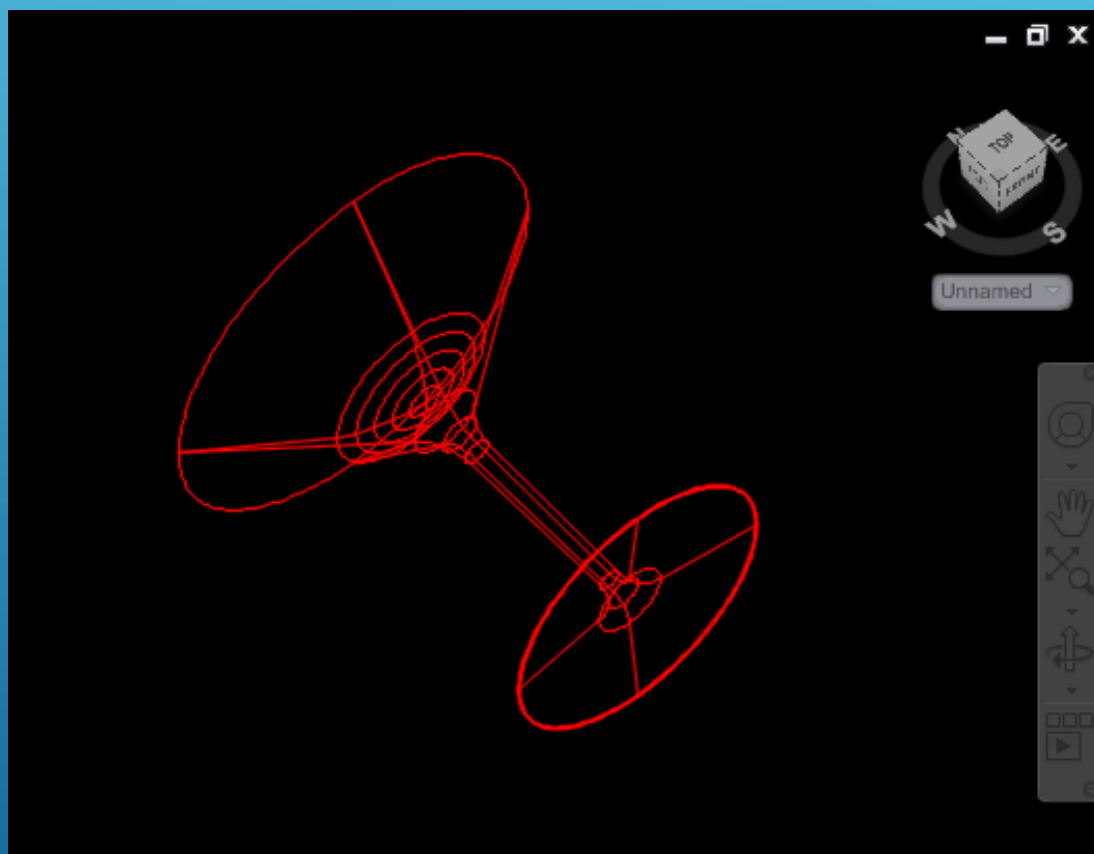


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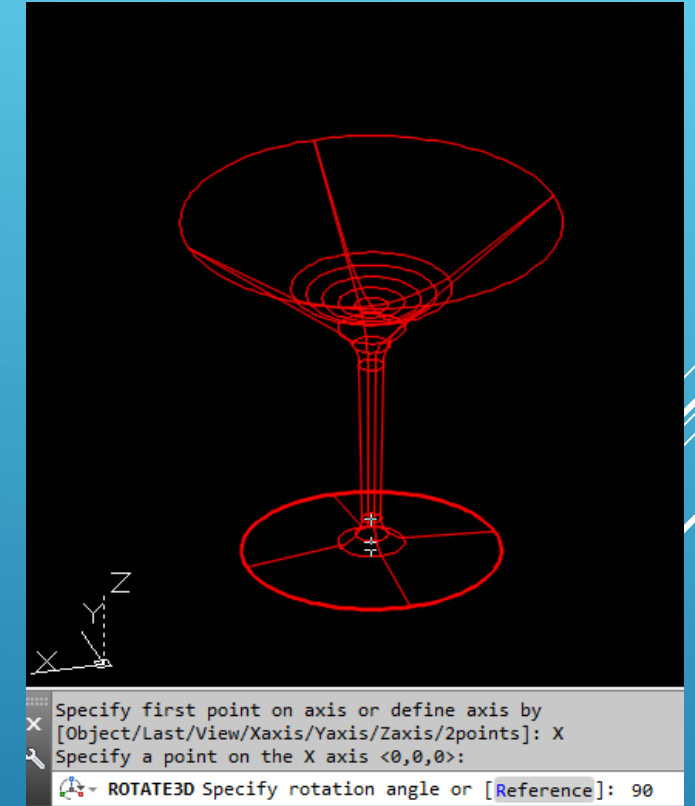
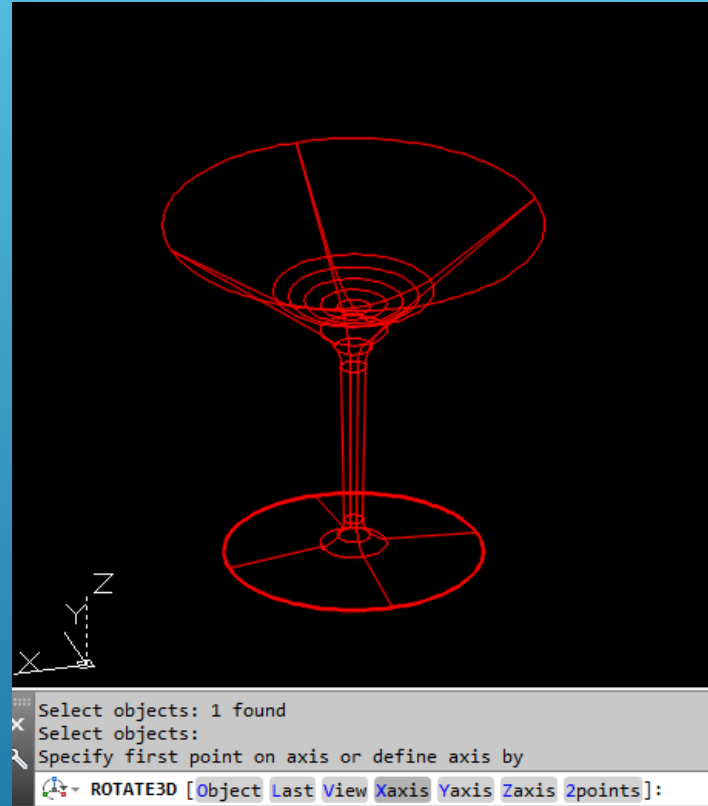
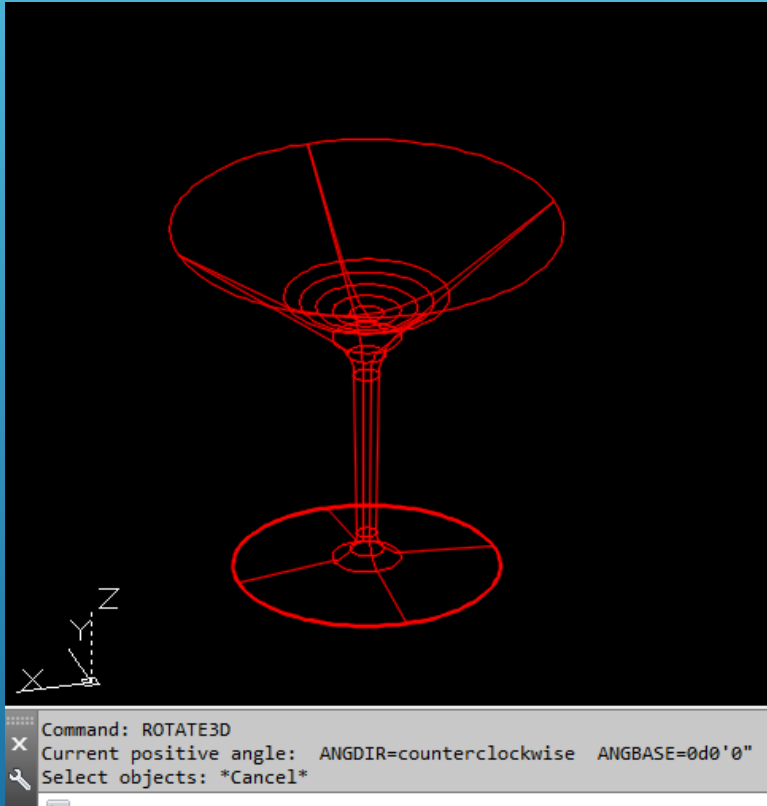


3D Rotate

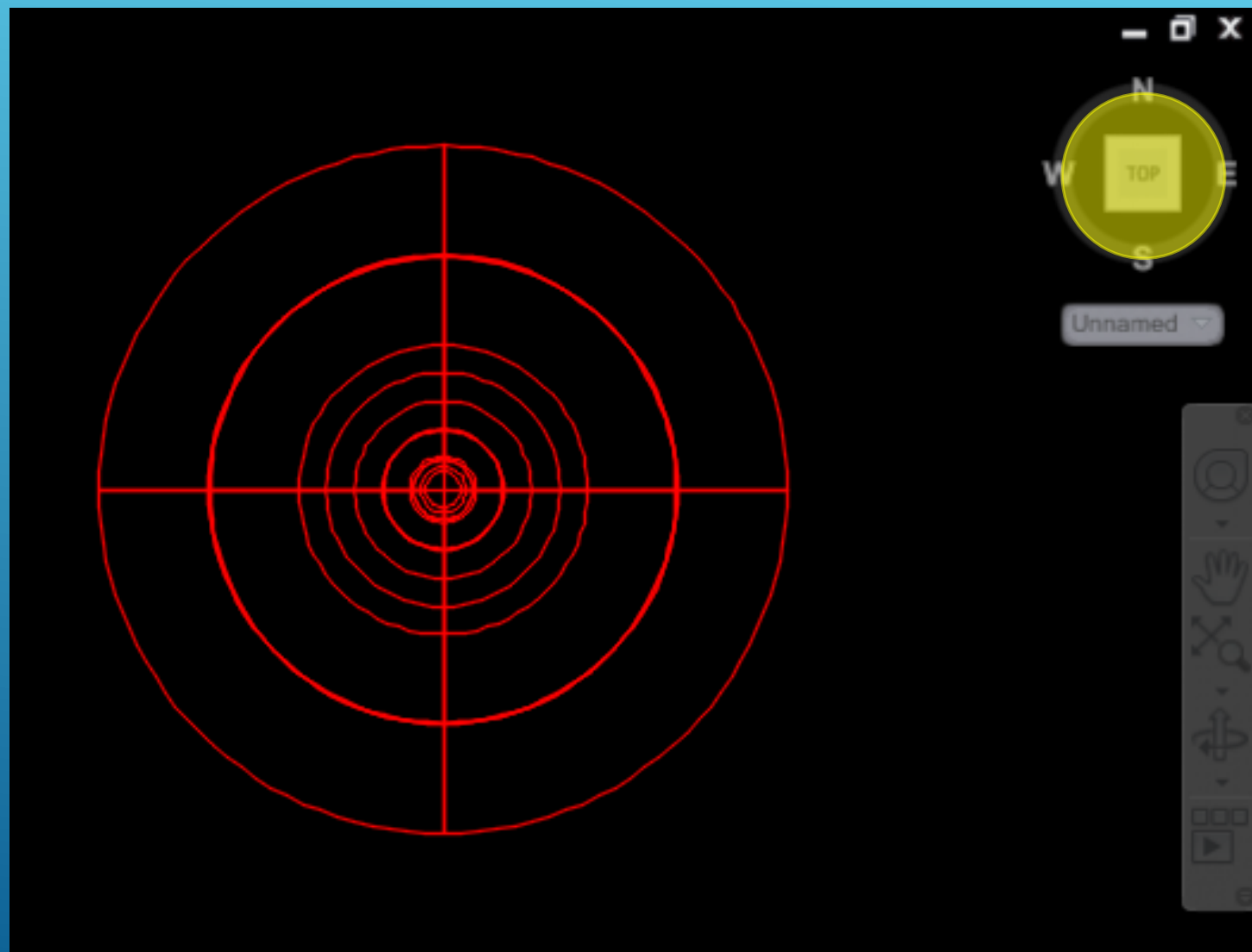
In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point

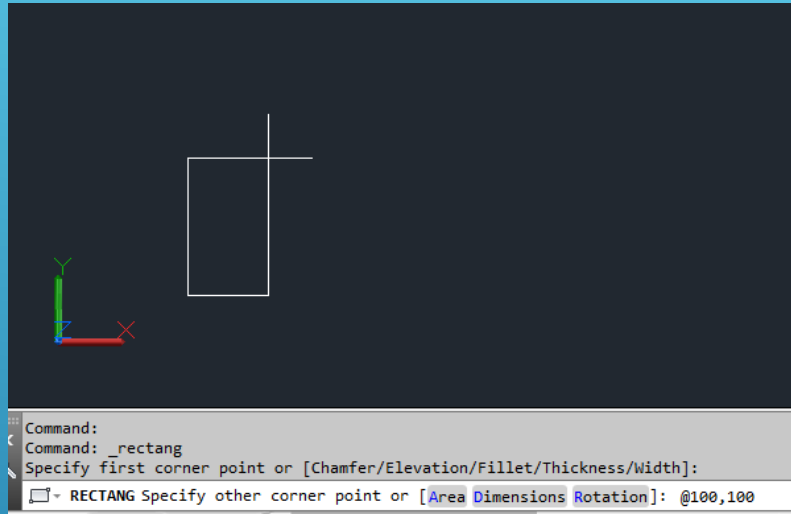
With the 3D Rotate gizmo, you can rotate selected objects and subobjects freely or constrain the rotation to an axis.

3DROTATE
Press F1 for more help



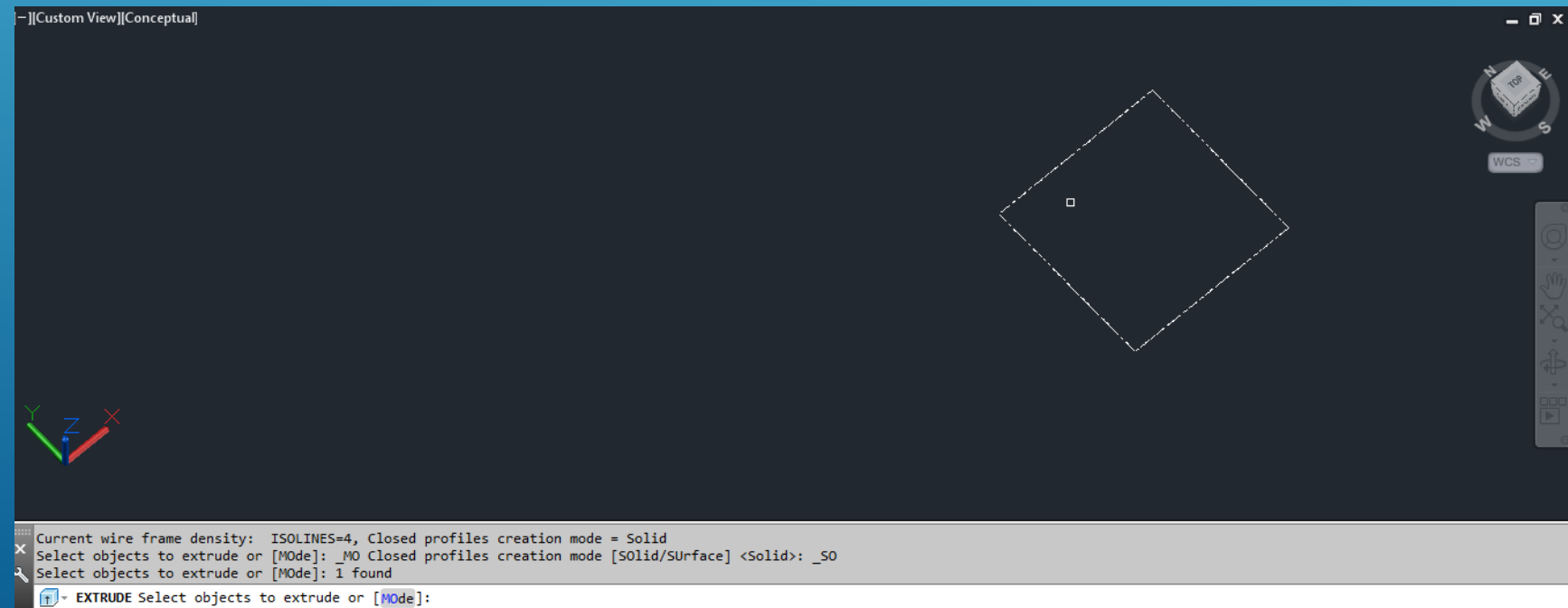
Rodar o copo em torno do eixo X para que na vista Top o copo apareça em pé





Extrude com opção Taper

1. Desenhar um quadrado com 100 unidades de lado
2. Efectuar o Extrude

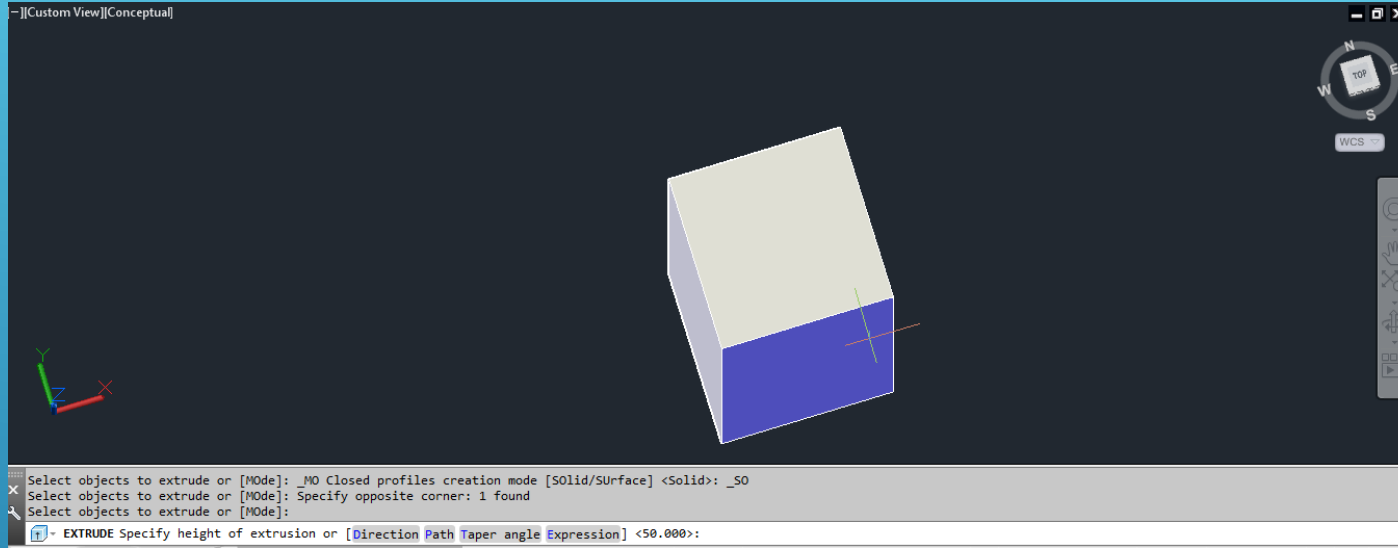


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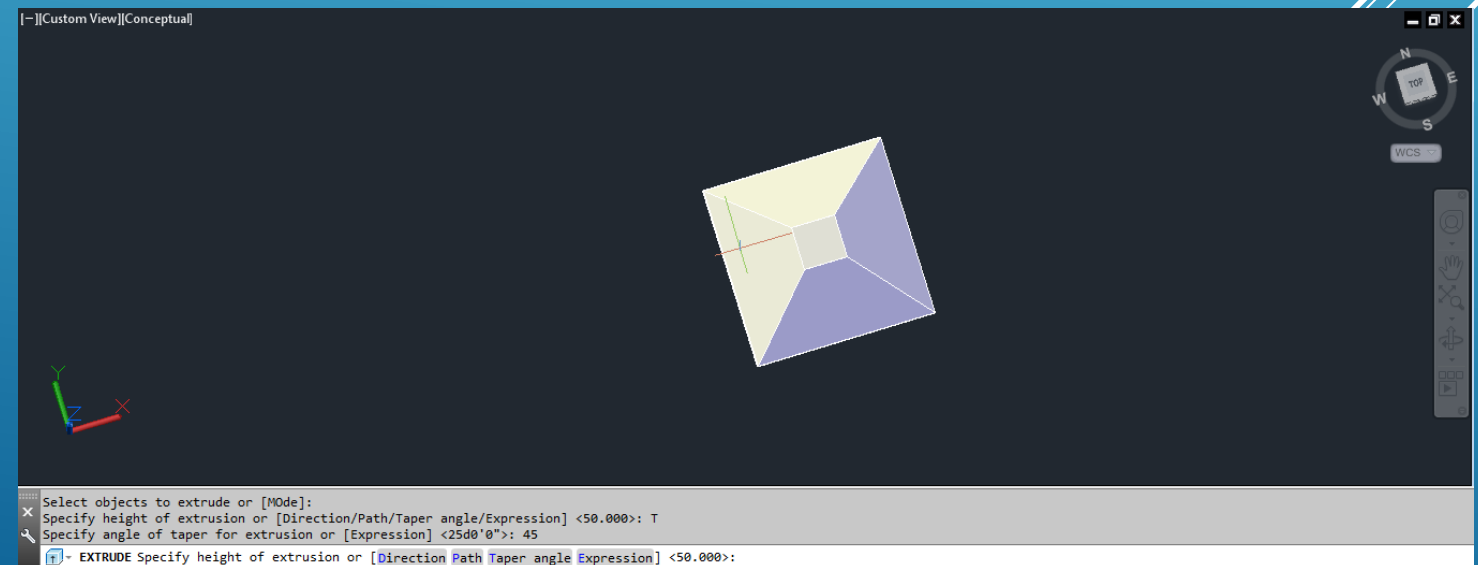


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angle=45°
height=30

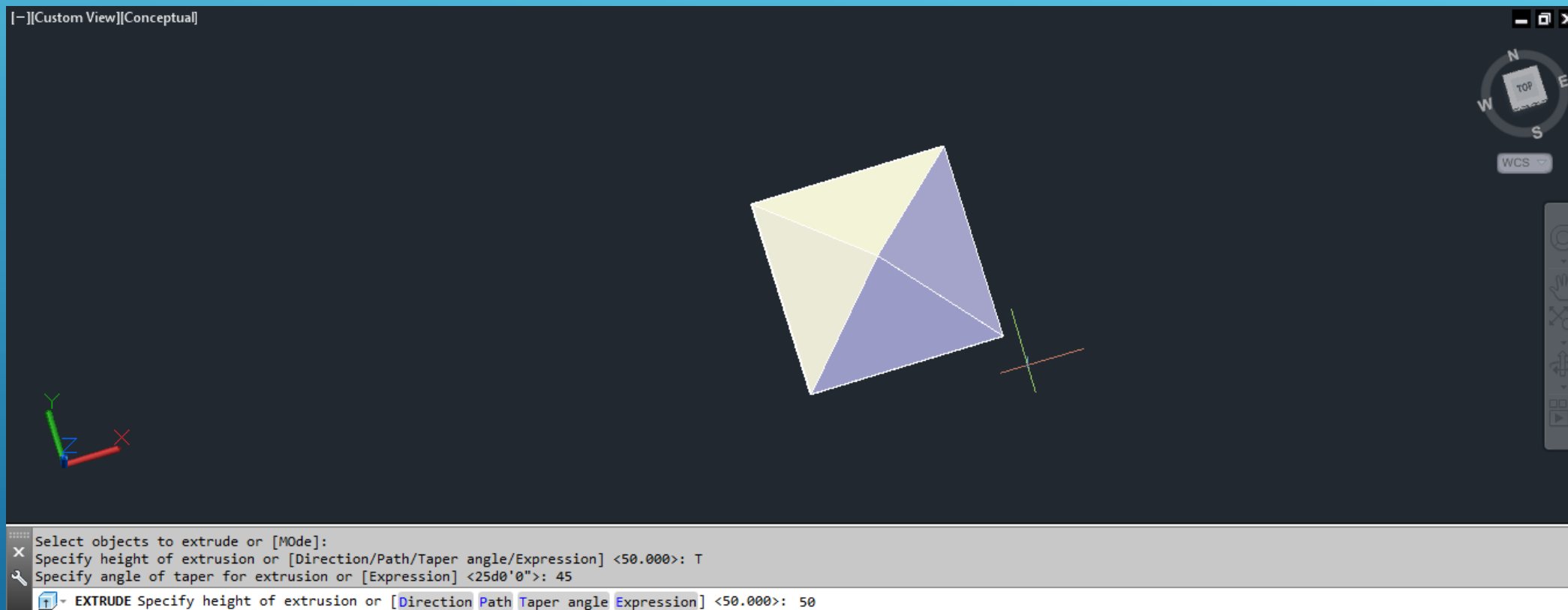


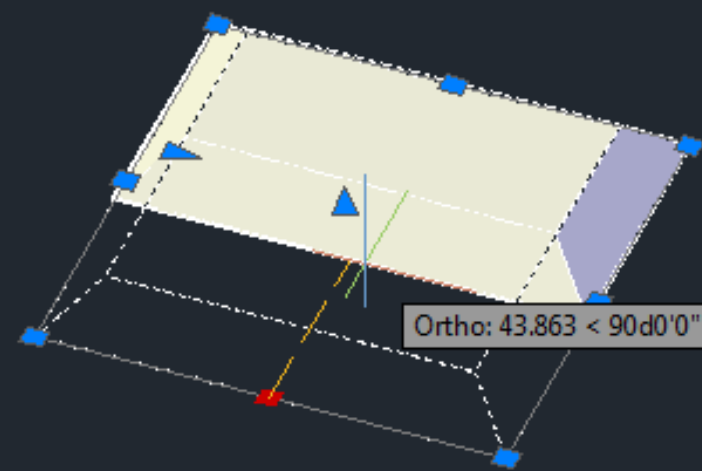
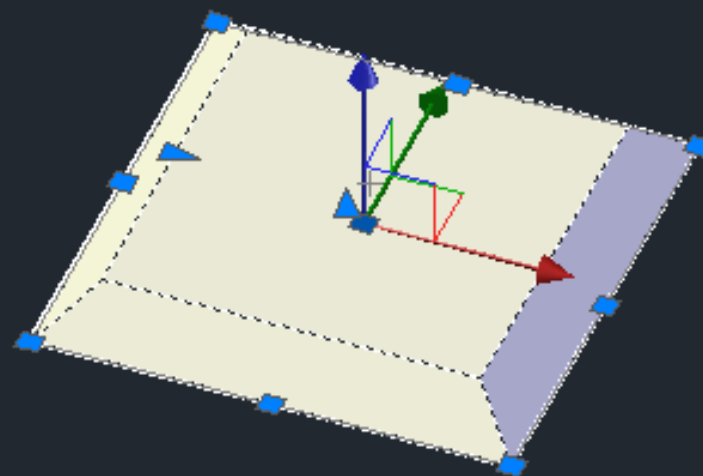
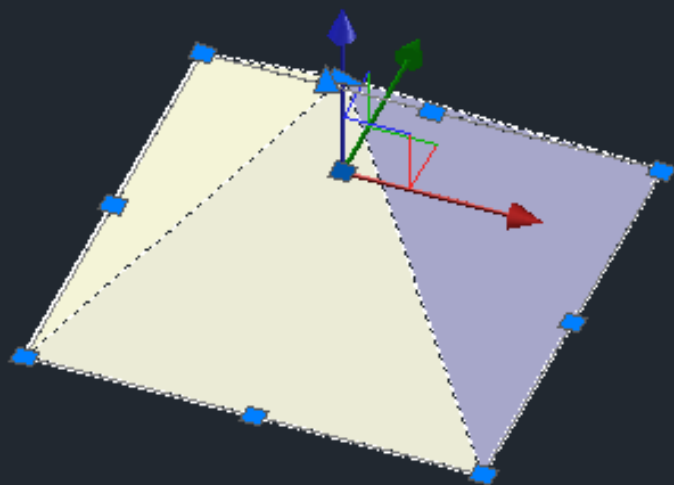
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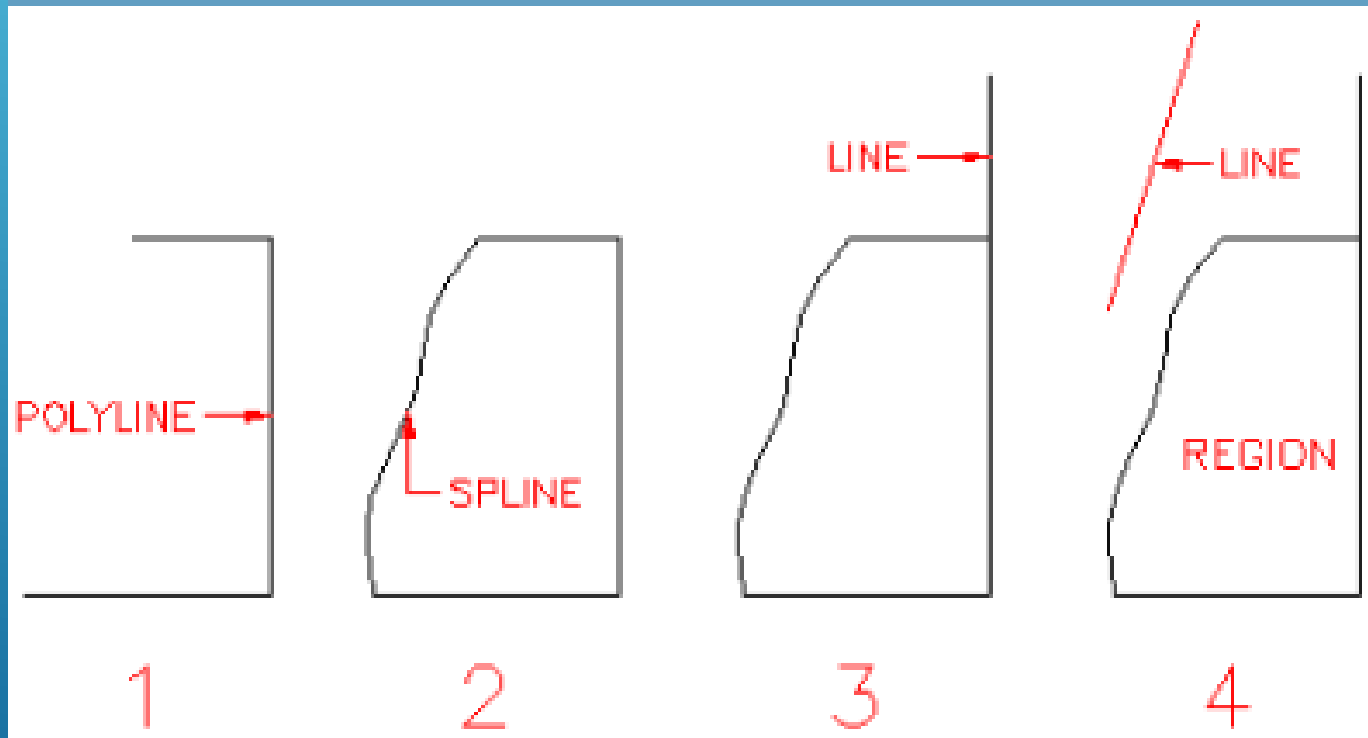


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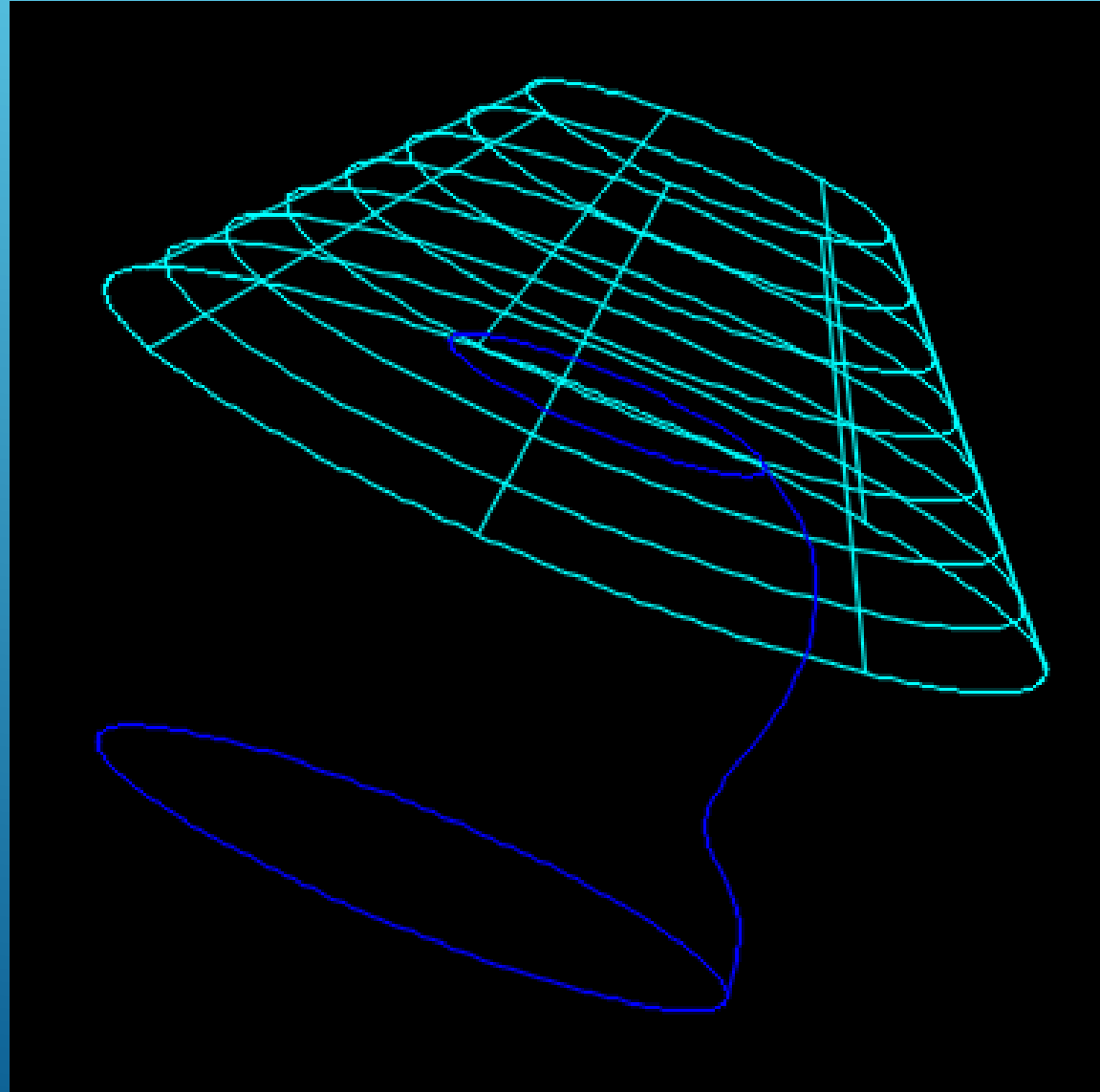
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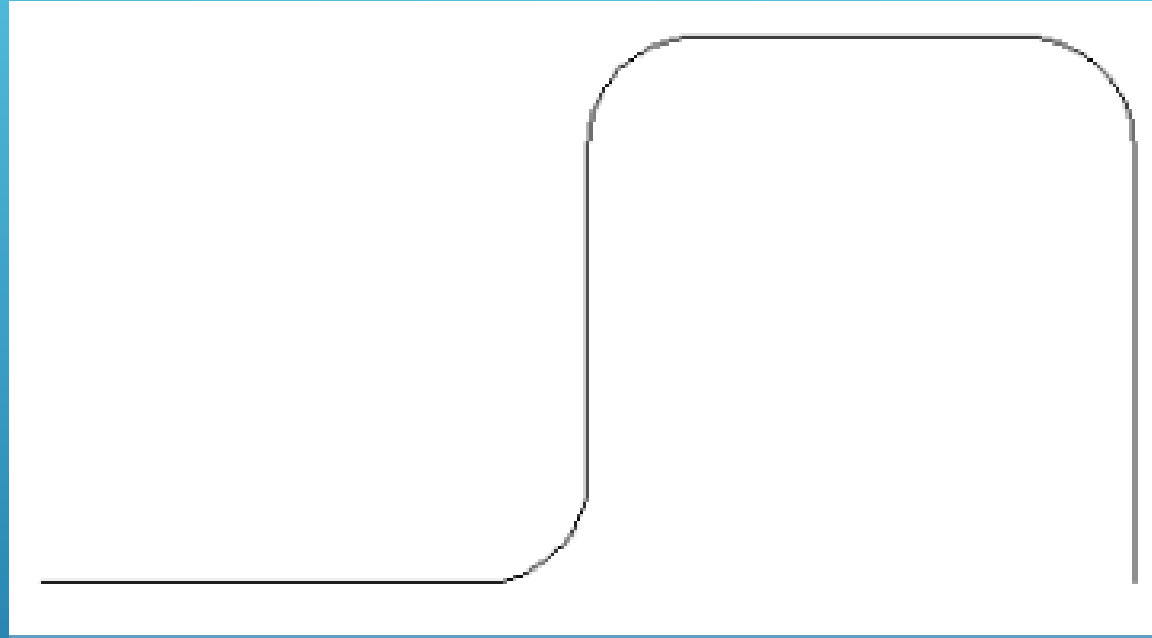
1. Criar 2 layers: base, abat-jour
2. Desenhar Polyline (layer base)
3. Desenhar Spline (layer base)
4. Desenhar Line vertical
5. Desenhar Line (layer abat-jout)
6. Definir Region (seleccionar Polyline e Spline, com Shift)
7. Revolve
8. Explode Region



Extrude ao longo
de um caminho



Desenhar uma Polyline entre os pontos $(0,0)$,
 $(120,0)$, $(120,120)$, $(240,120)$, $(240,0)$

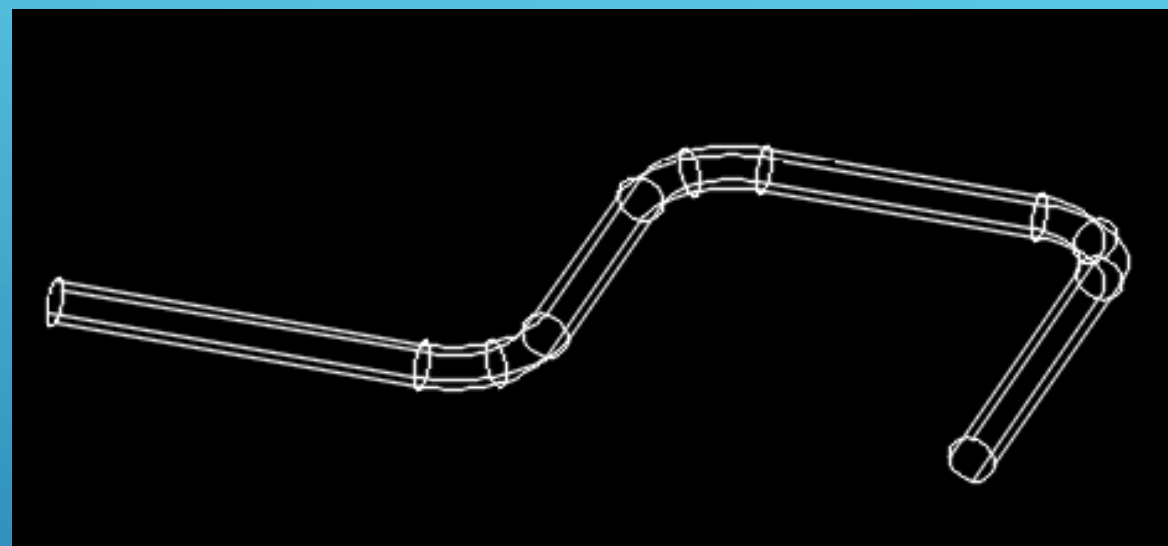
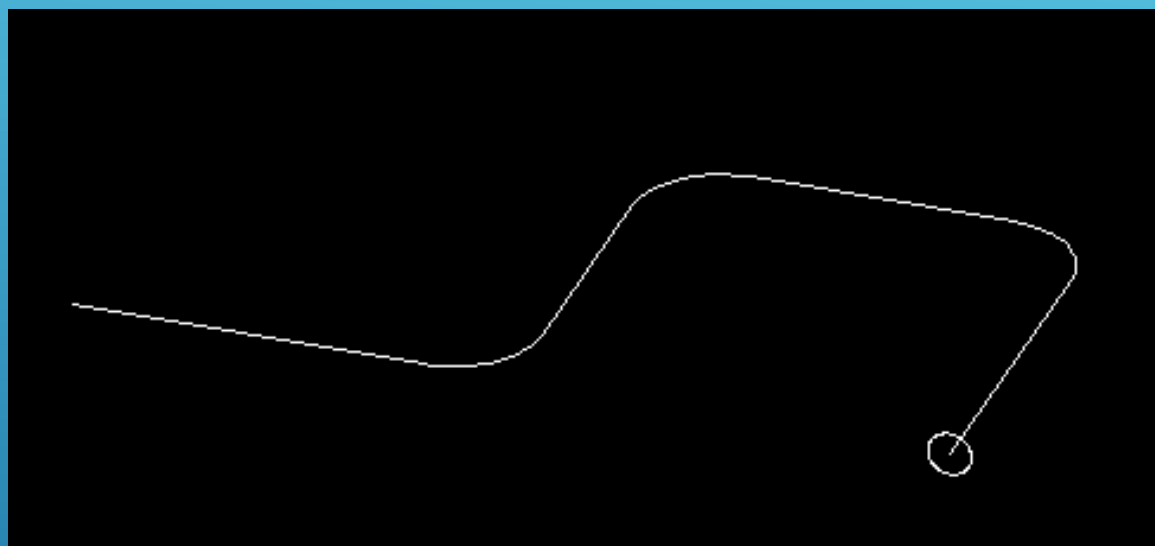


Utilizar o comando Fillet com raio 24

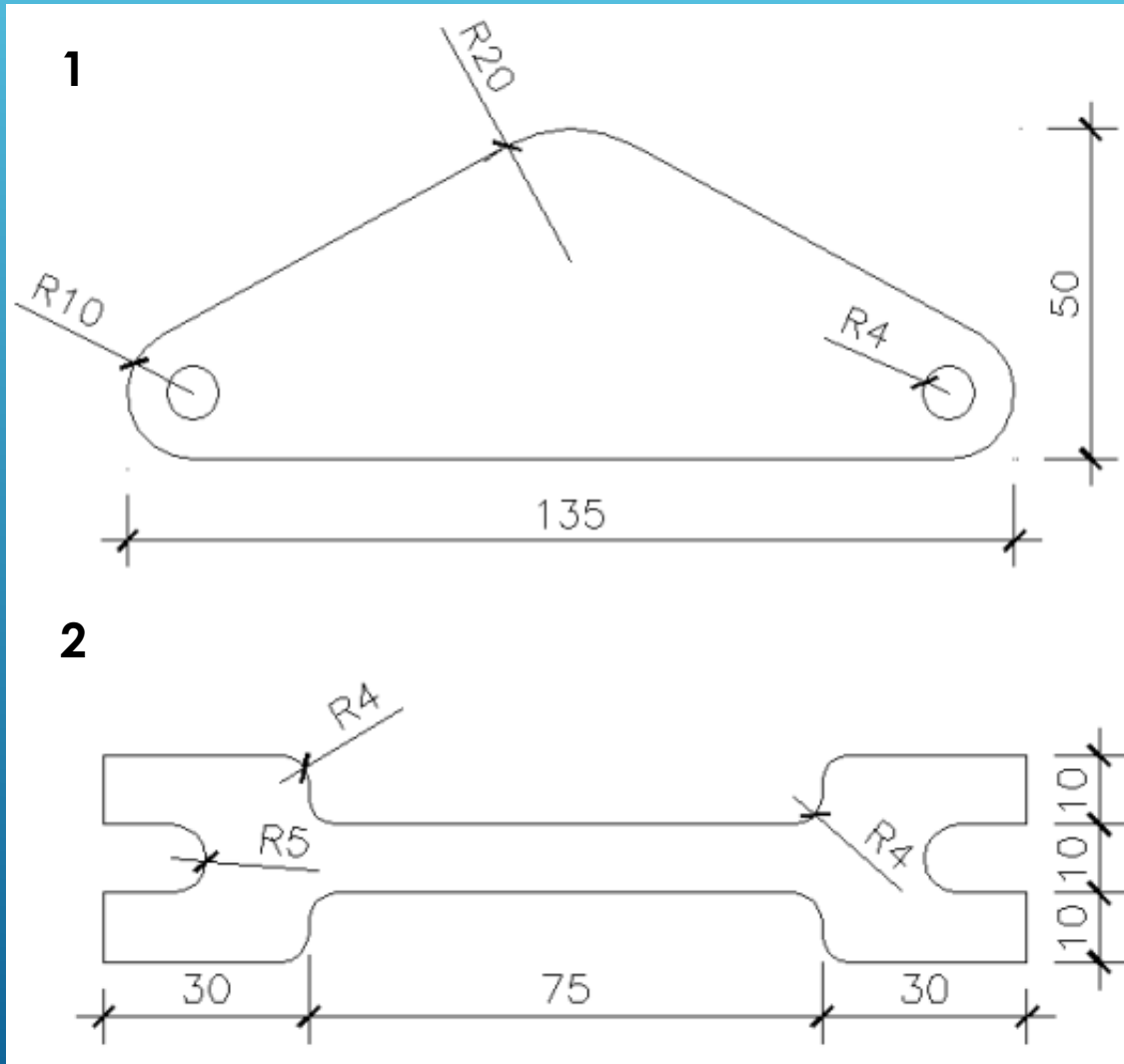


Vista Top

Desenhar uma circunferência na extremidade do lado direito da polyline, com diâmetro 12 . Numa vista isométrica, usar o comando Rotate3d para rodar a circunferência 90° em torno do eixo X.

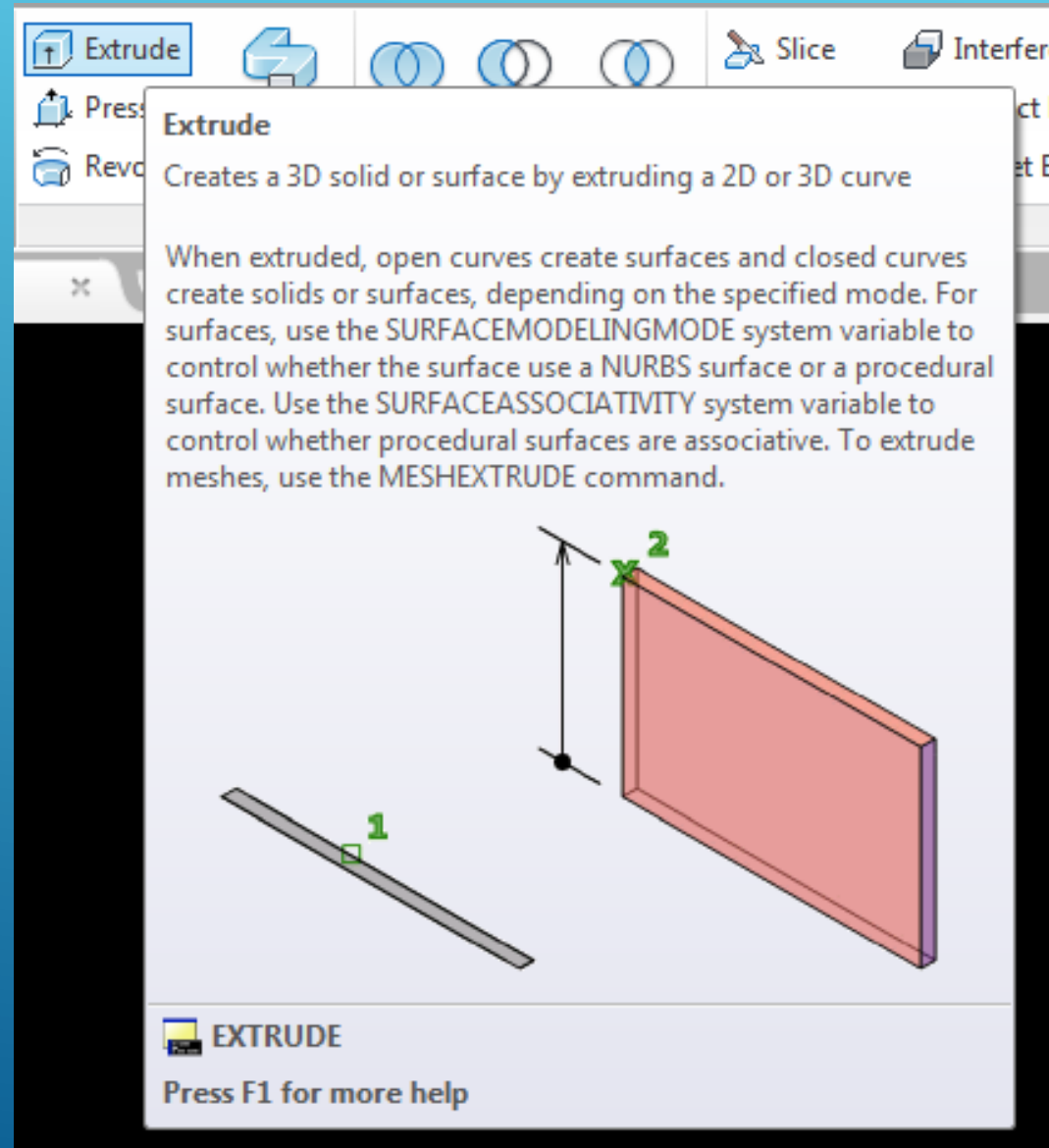


Utilizar o comando Extrude, seleccionar a circunferência, seleccionar Path e indicar a polyline



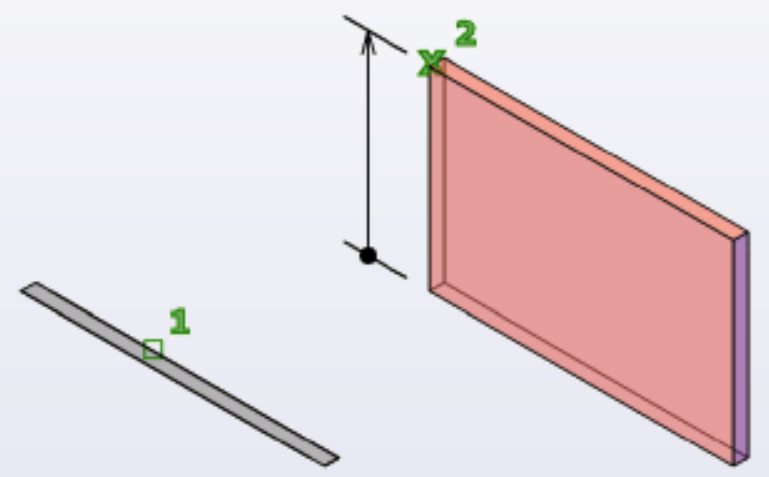
Após desenhar cada um dos desenhos, utilizar a função `home > Modify > Join` para transformar os vários elementos numa polyline única.

Desenhar na vista Top os desenhos 1 e 2

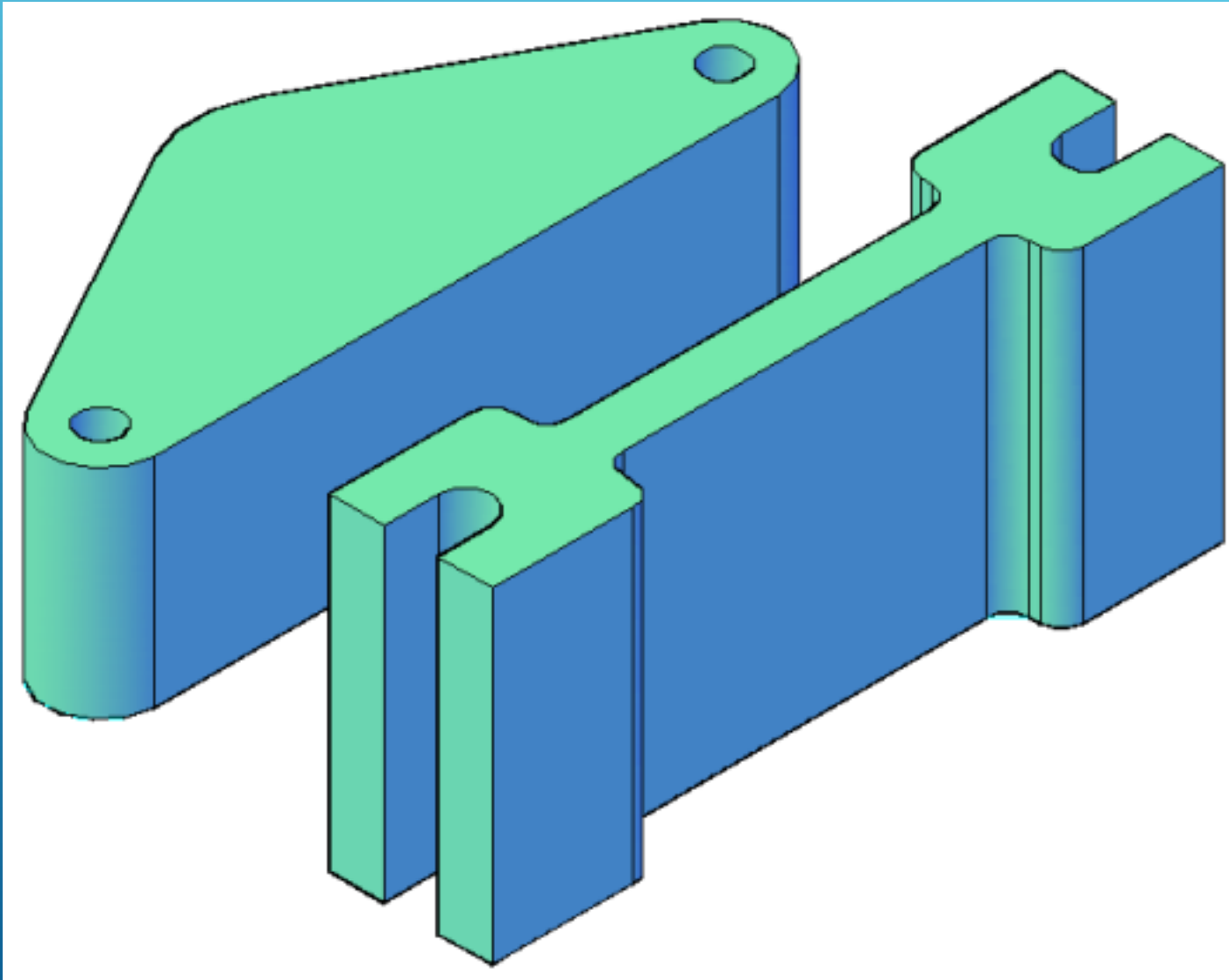


Extrude
Creates a 3D solid or surface by extruding a 2D or 3D curve

When extruded, open curves create surfaces and closed curves create solids or surfaces, depending on the specified mode. For surfaces, use the SURFACEMODELINGMODE system variable to control whether the surface use a NURBS surface or a procedural surface. Use the SURFACEASSOCIATIVITY system variable to control whether procedural surfaces are associative. To extrude meshes, use the MESH EXTRUDE command.



EXTRUDE
Press F1 for more help



Extrude

Seleccionar todos os elementos do desenho 1, 40.

Extrude

Seleccionar todos os elementos do desenho 2, 60.

No caso do desenho 1, utilizar a função Solid > Subtract para fazer os 2 furos, seleccionando primeiro o contorno exterior e depois o contorno correspondente dos furos (com Shift).

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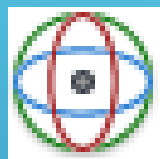


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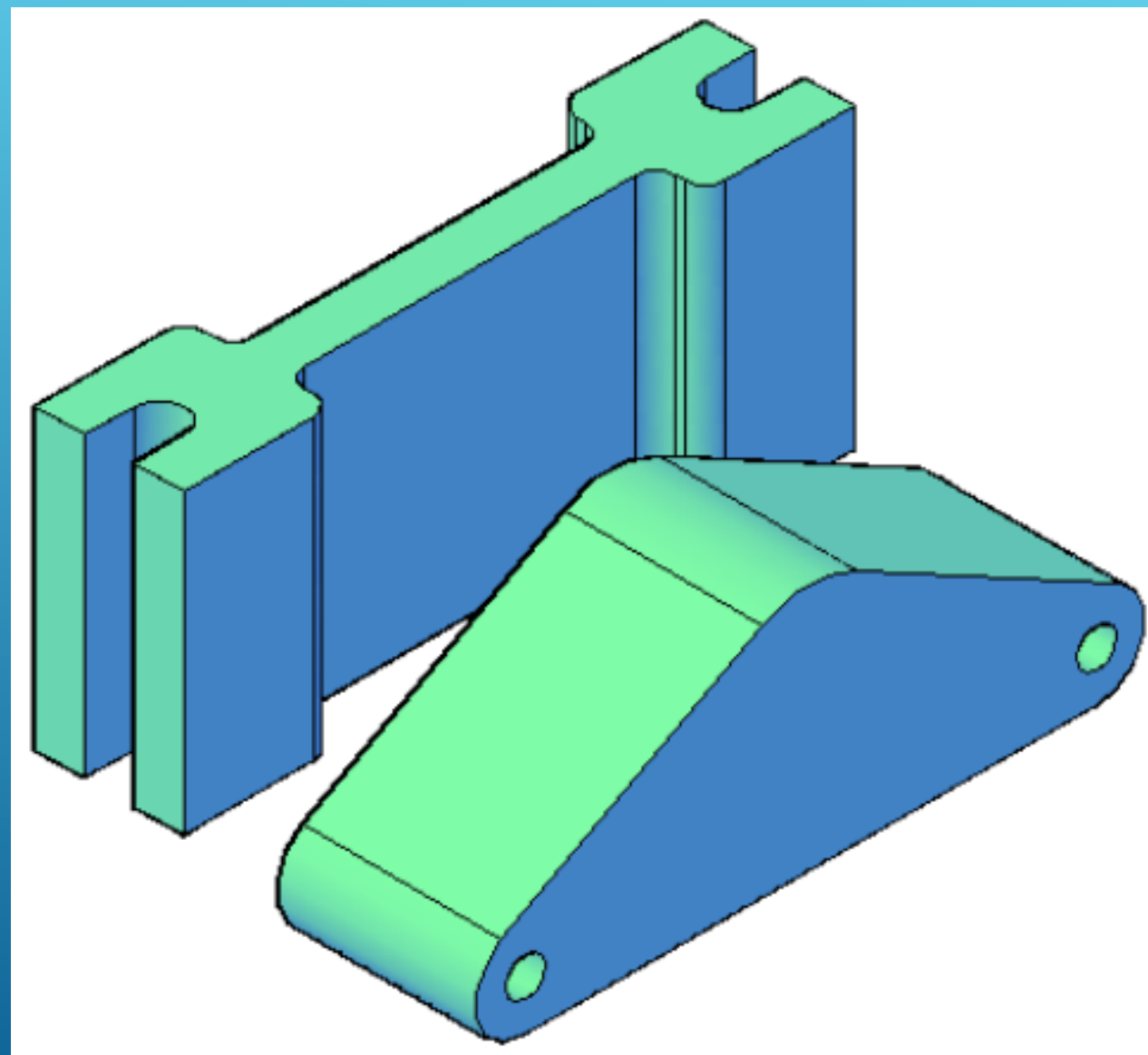
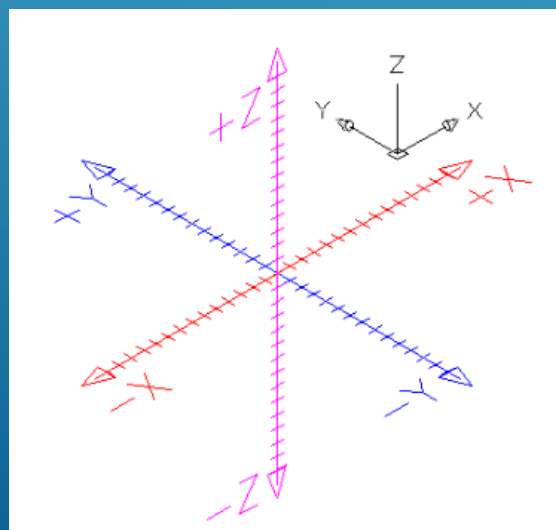
View > NE ISO

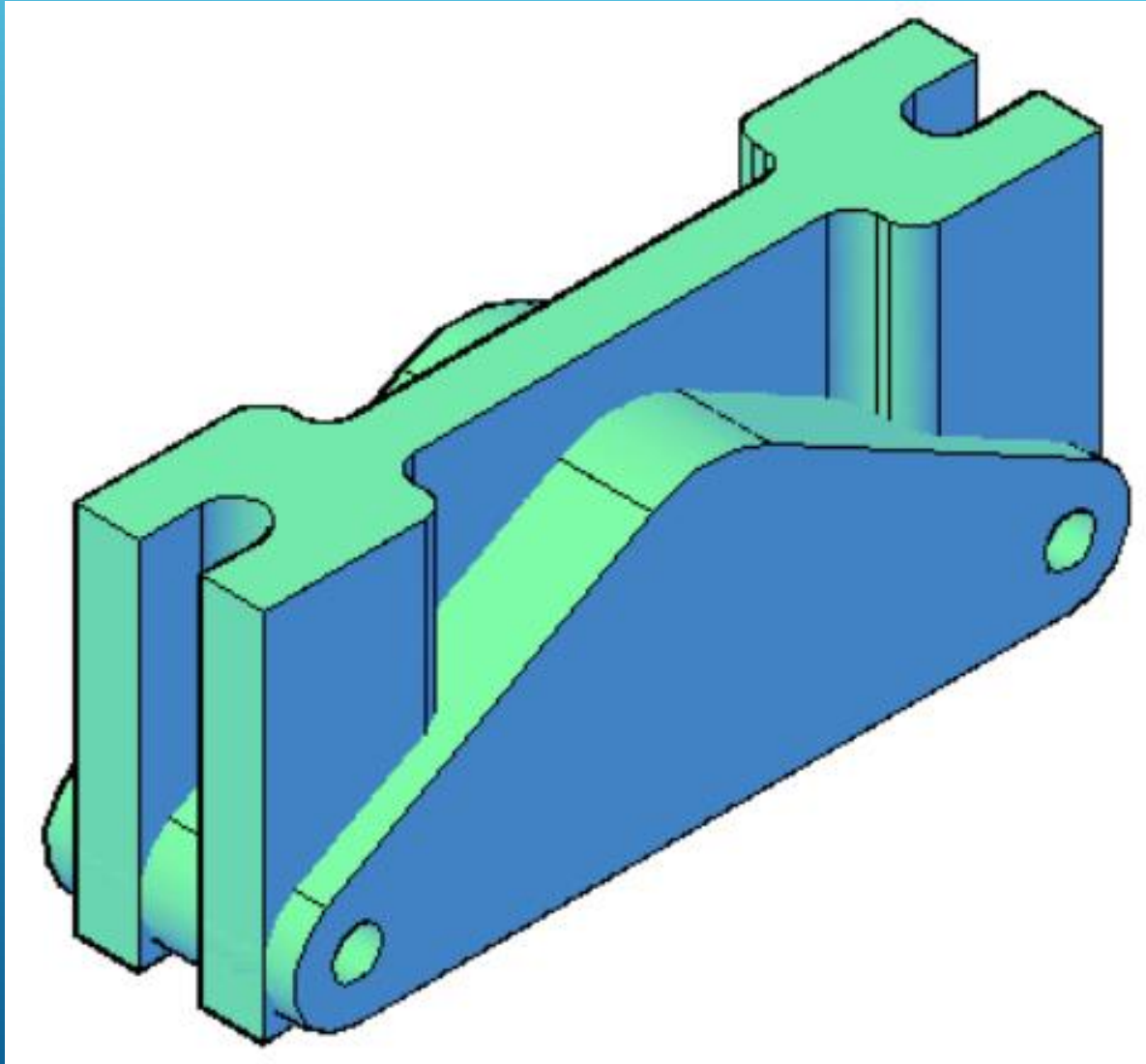
Modify > Rotate 3D



Seleccionar o desenho 1

Rodar o desenho 1 90° segundo o eixo X
(encarnado)





Posicionar o centro do desenho1 no
centro do desenho 2

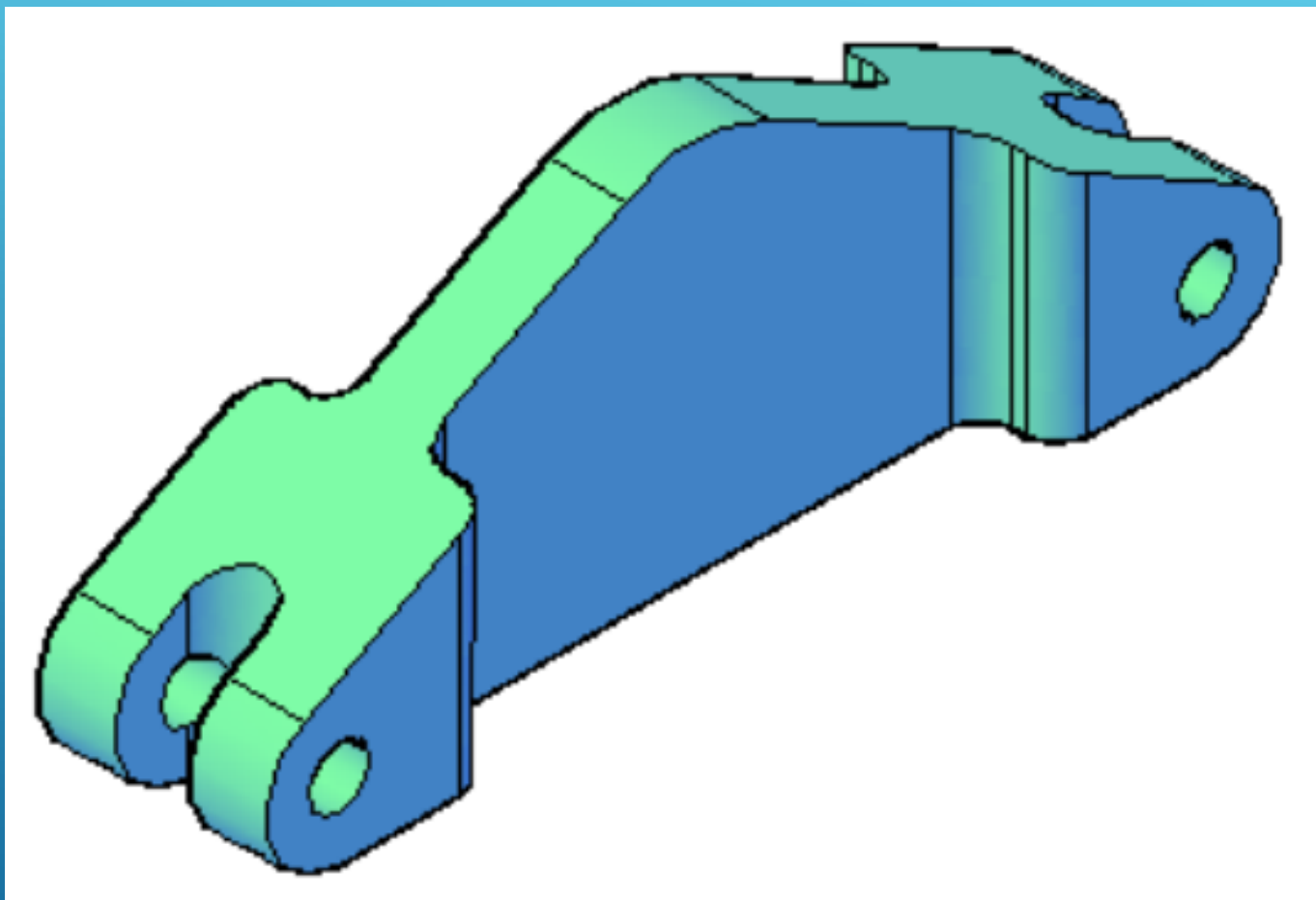
Solid, Intersect

Creates a 3D solid or 2D region from selected overlapping solids or regions

You can extrude 2D profiles and then intersect them to create a complex model efficiently.

INTERSECT

Press F1 for more help



Efectuar a intercessão das duas figuras
(seleccionar as 2 figuras com shift + enter)



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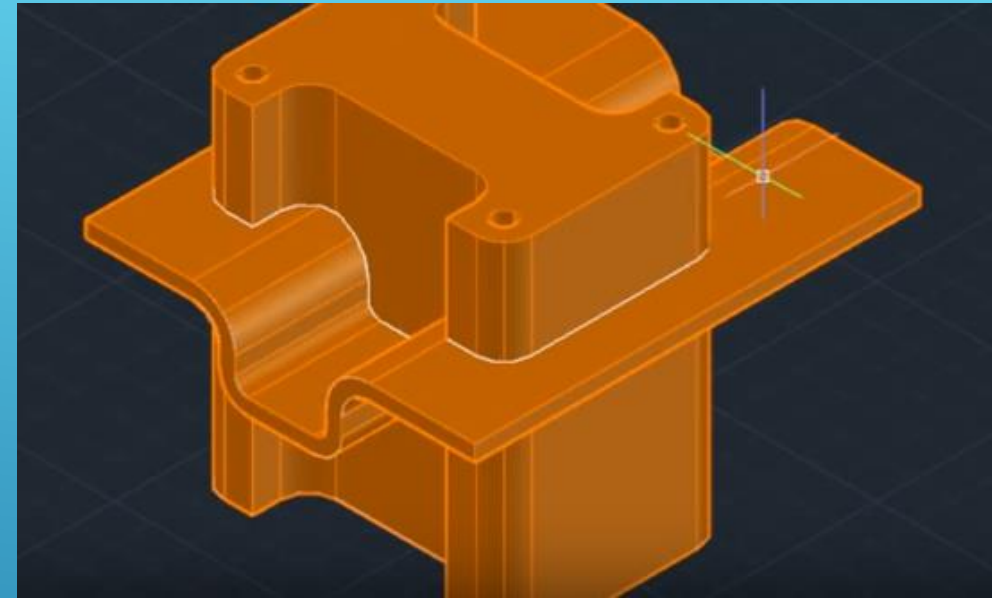
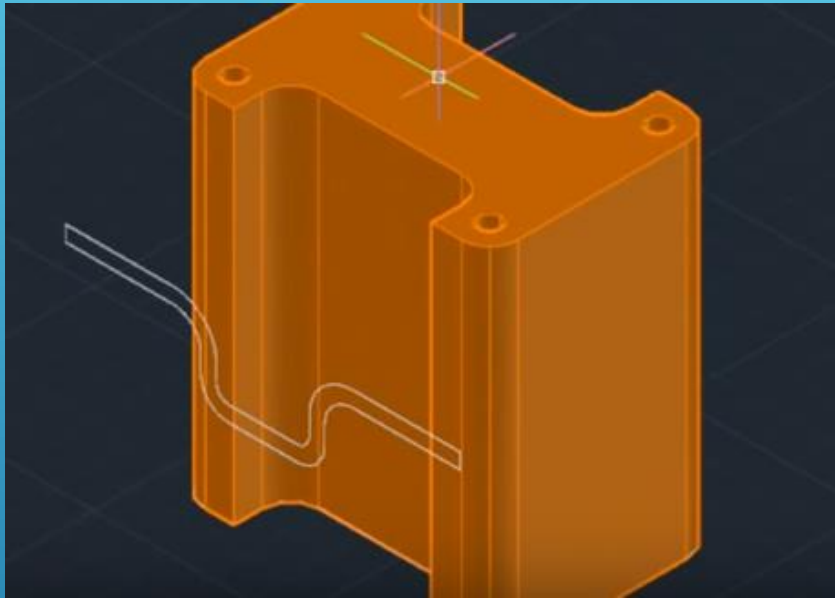


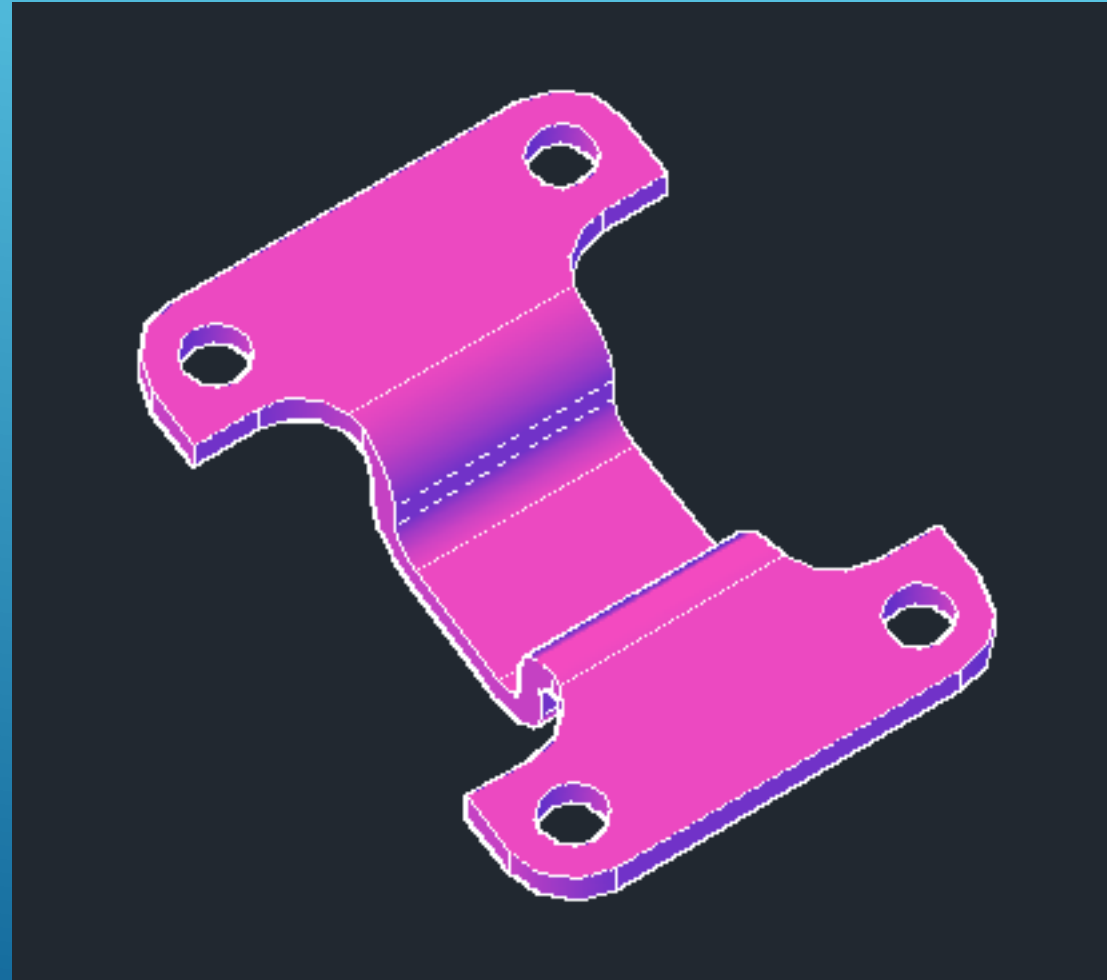
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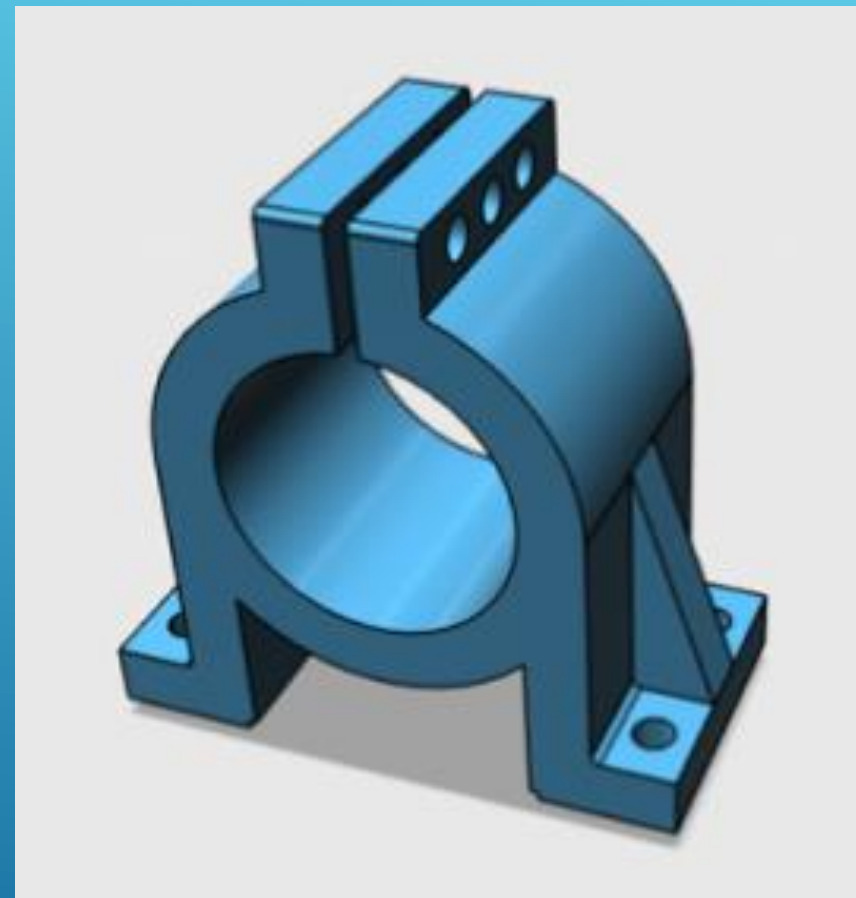
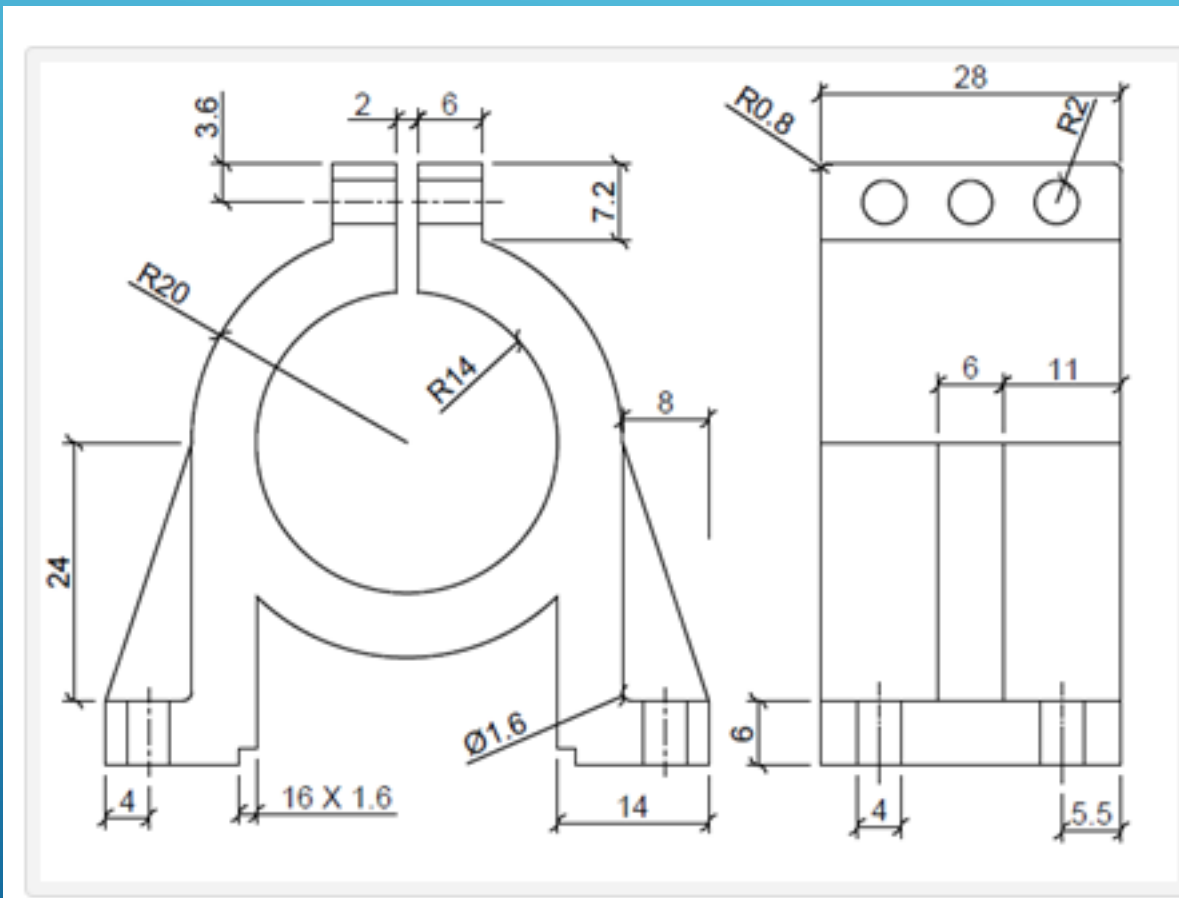


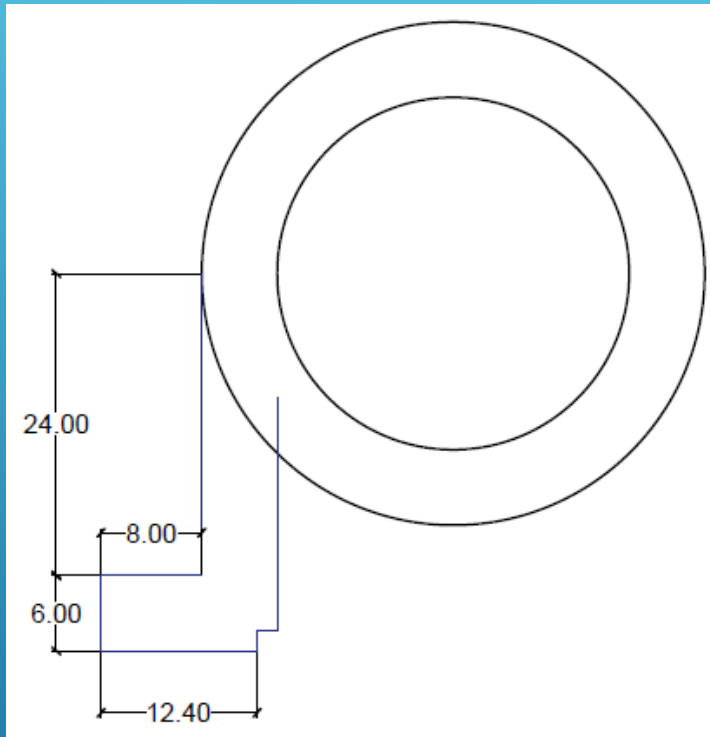
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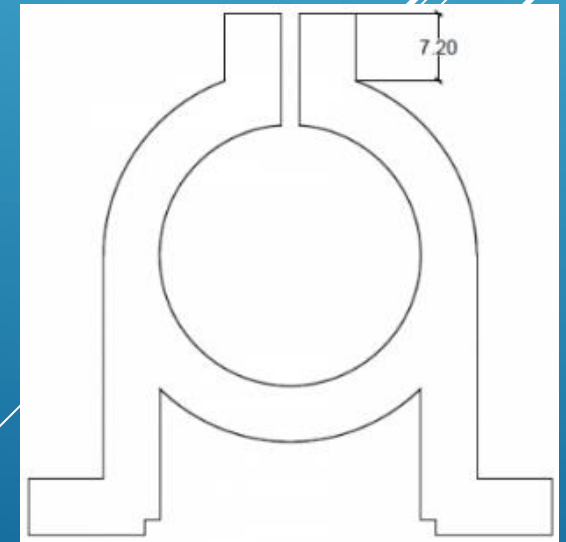
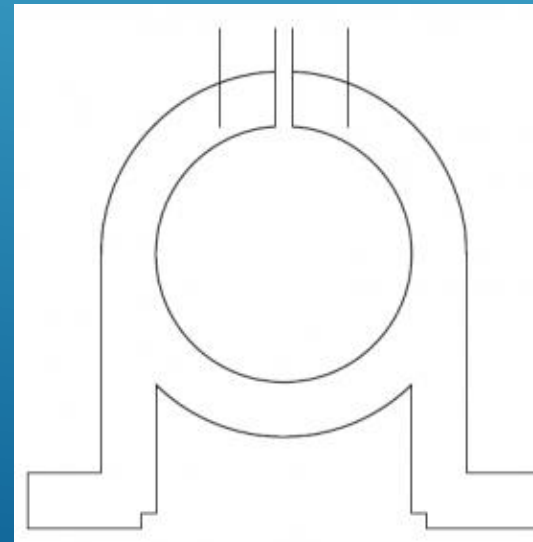
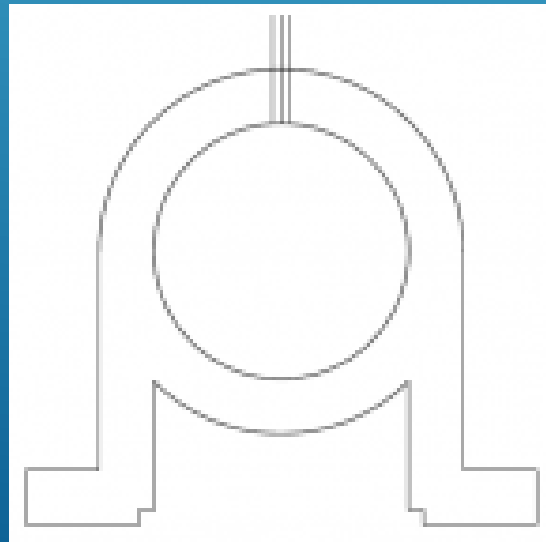
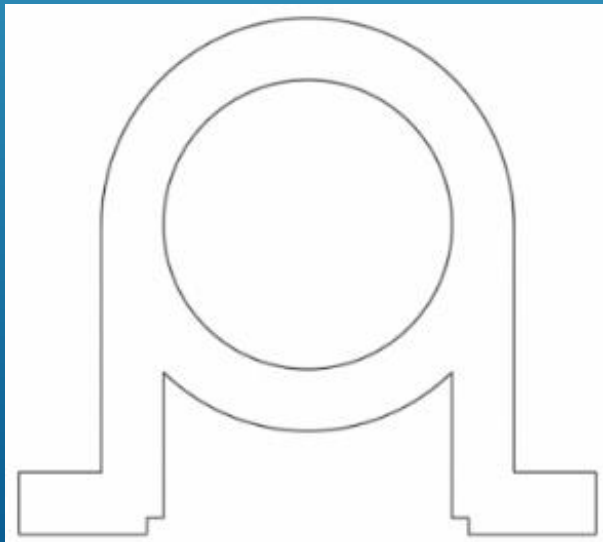


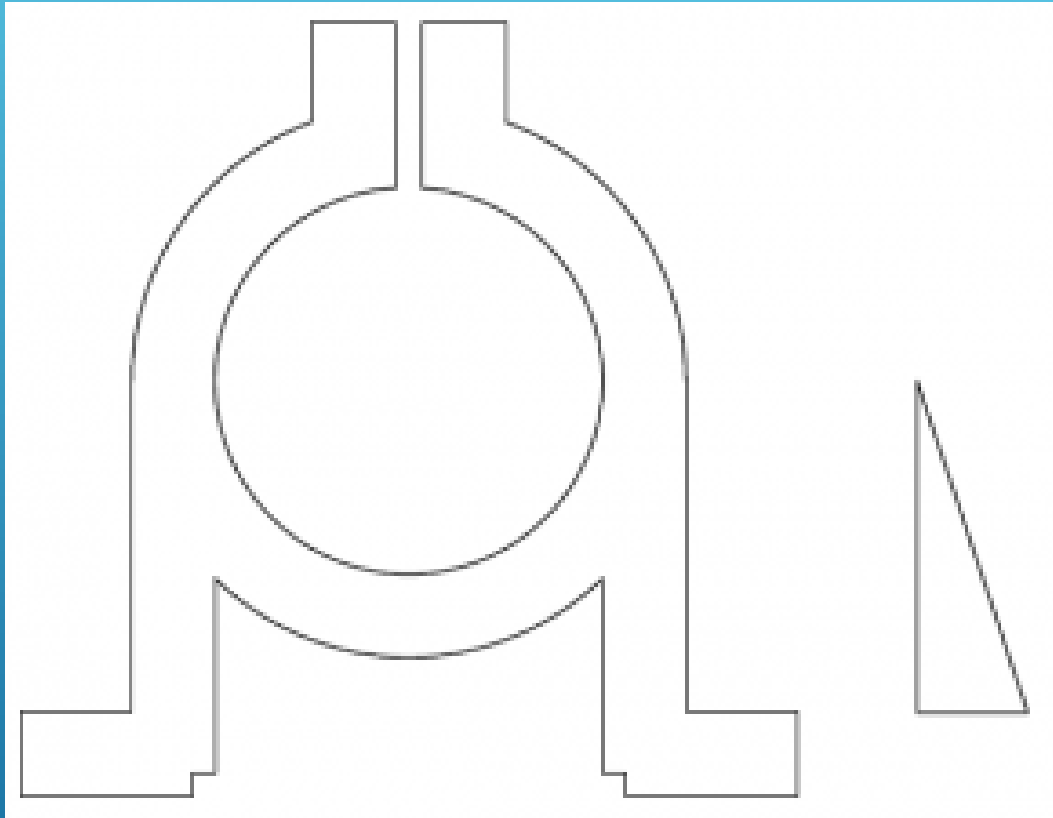




1. Desenhar 2 circunferências concêntricas de raios $R20$ e $R14$.
2. Desenhar uma linha vertical para baixo partindo do quadrante esquerdo da circunferência de raio maior com 24 unidades de comprimento.
3. Desenhar uma linha horizontal com 8 unidades de comprimento para a esquerda a partir da extremidade inferior do segmento anterior.
4. Desenhar uma linha vertical para baixo com 6 unidades de comprimento para baixo a partir da extremidade do segmento anterior.
5. Desenhar uma linha horizontal de 12.4 unidades de comprimento para a direita da extremidade do segmento anterior.
6. Desenhar uma linha vertical com 1.6 unidades de comprimento para cima a partir da extremidade do segmento anterior.
7. Desenhar uma linha horizontal com 1.6 unidades de comprimento para a direita a partir da extremidade do segmento anterior.
8. Desenhar uma linha vertical para cima que ligue à circunferência exterior.
9. Utilizar o comando Trim para apagar a parte desnecessária da circunferência exterior.

10. Utilize o comando Mirror para replicar a construção anterior para o lado direito do objecto.
11. Complete o desenho com o comando Trim.
12. Utilize o quadrante superior da circunferência interior para desenhar uma linha vertical que atravesse a circunferência exterior.
13. Utilize o comando Offset para desenhar 2 linhas paralelas à linha anterior à distância 1 unidade e apague a linha central.
14. Utilize o comando Extend para ligar as extremidades inferiores das 2 linhas anteriores à circunferência interior.
15. Utilize o comando Trim para completar o desenho conforme a figura.
16. Utilize o comando Offset com valor 6 unidades para obter 2 paralelas às 2 linhas anteriores.
17. Complete conforme a figura.

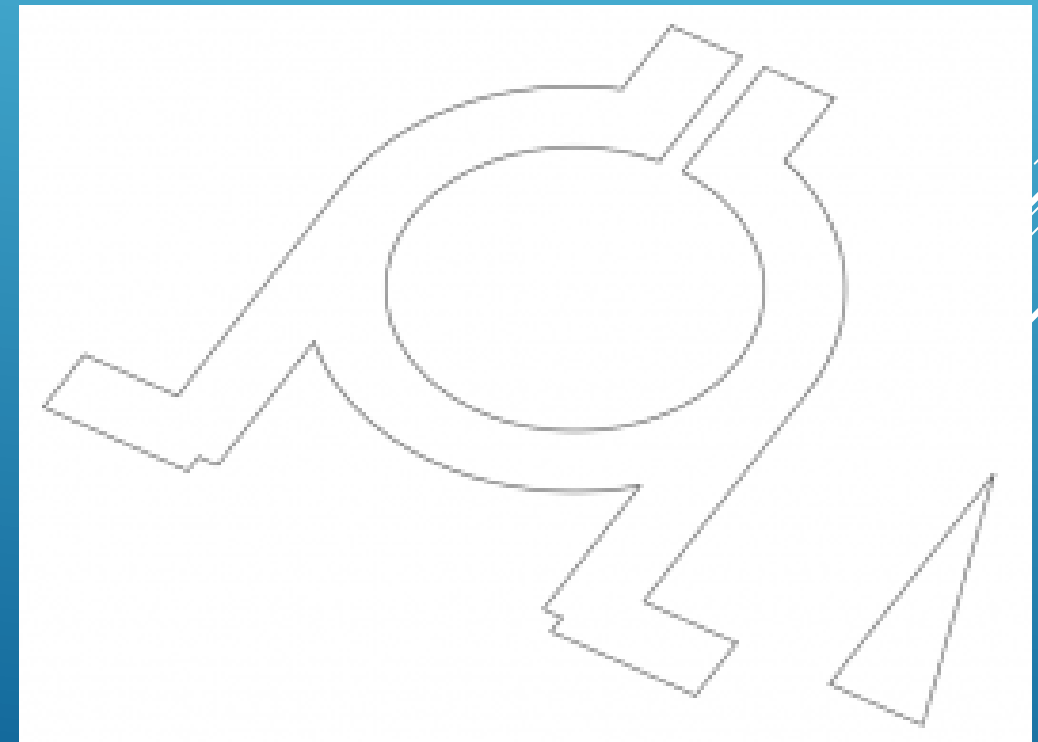


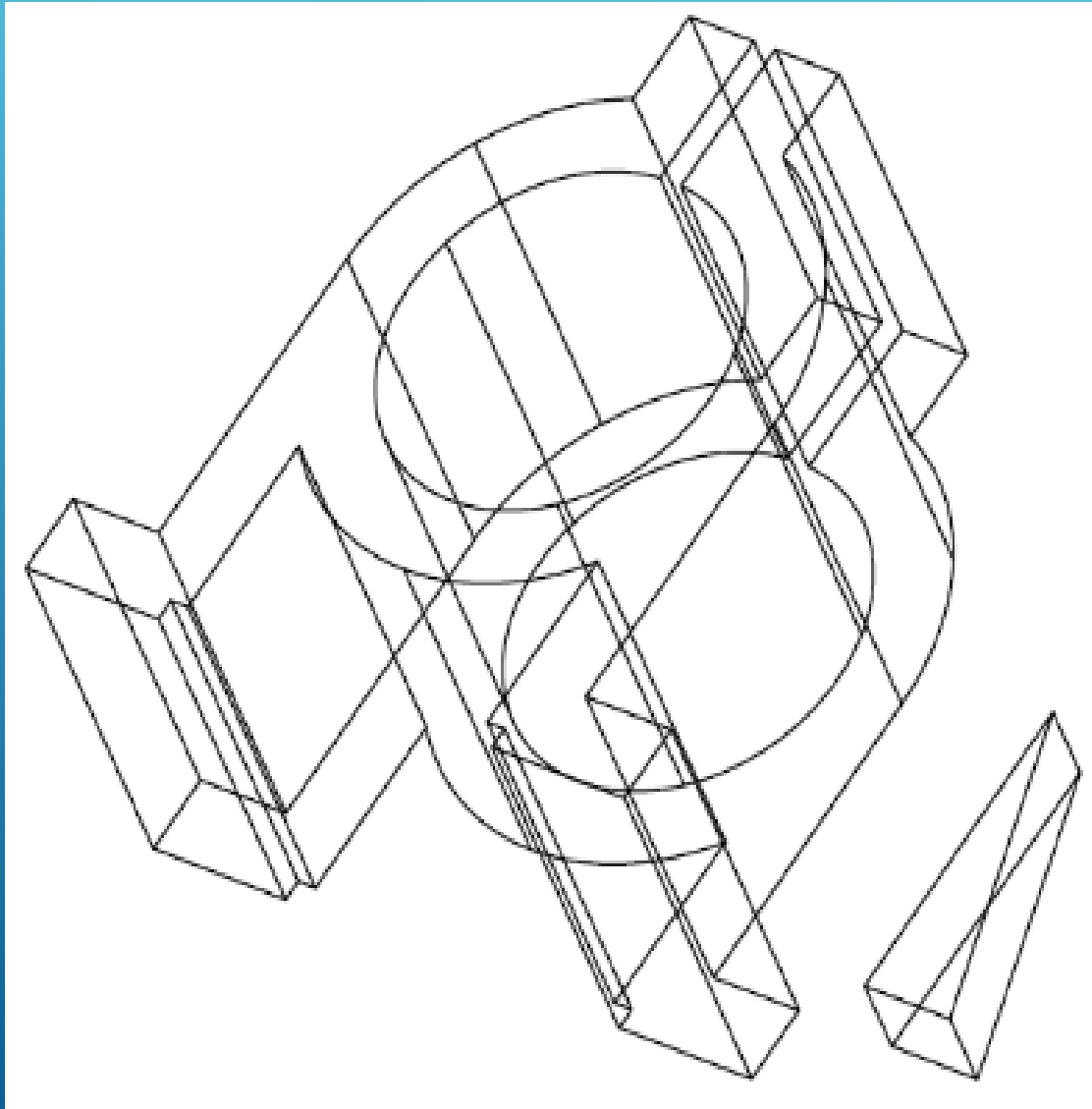


18. Desenhar o triângulo lateral, que tem o lado vertical com 24 unidades de comprimento e o lado horizontal com 8 unidades de comprimento.

19. Utilize o comando Region e forme 2 regiões.

20. Utilize o comando Free Orbit até obter uma vista semelhante à indicada na figura.





21. Utilize o comando Extrude com 28 unidades para o corpo principal e 6 unidades para o triângulo. Visualize o modelo com o Visual Style Shades of Gray.

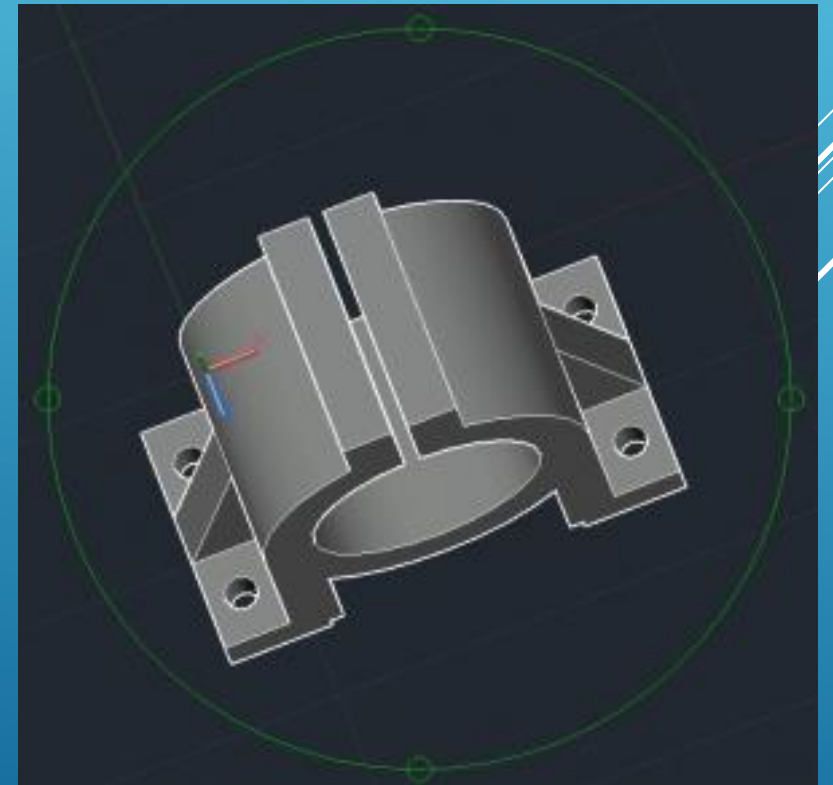
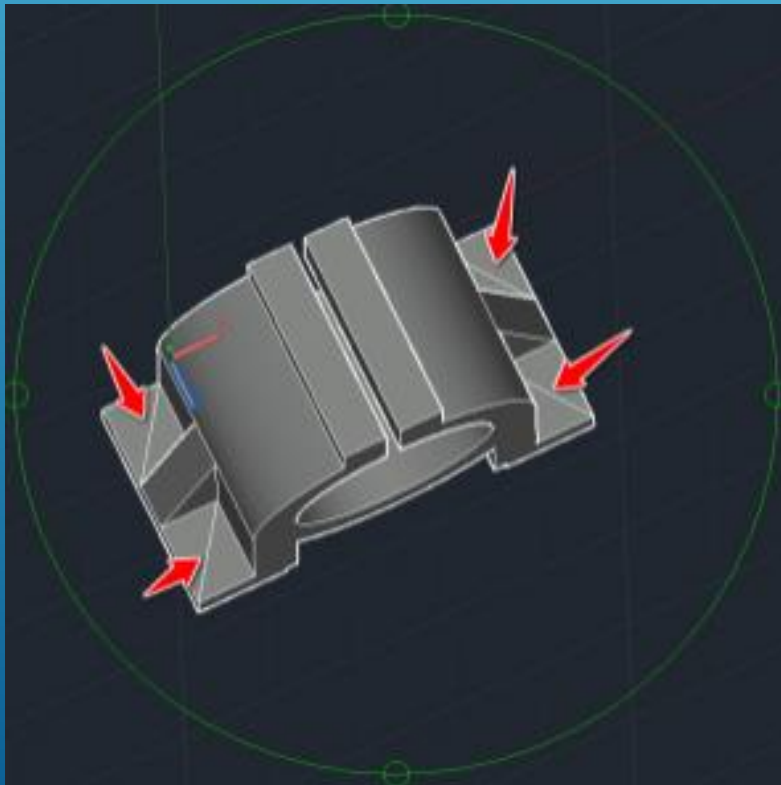
22. Na vista Top, utilize o comando Mirror para copiar o triângulo para o lado esquerdo do corpo principal e volte à vista anterior.

23. Utilize o comando Move para efectuar a translação de cada triângulo, seleccionando o ponto médio do segmento superior para o ponto médio dos quadrantes esquerdo e direito do corpo principal.

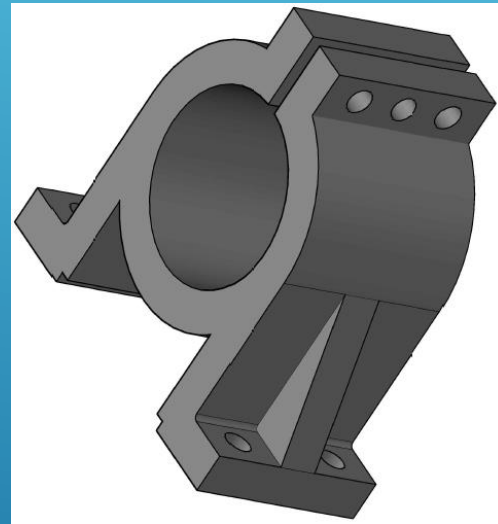
24. Utilize o comando Union, seleccione os 3 objectos e carregue em Enter.



24. Visualize o modelo de forma a que consiga acrescentar as linhas diagonais indicadas na figura.
25. Crie circunferências com snap Midpoint em cada uma das linhas com raio 2 unidades
26. Efectue o Extrude dessas circunferências para baixo de tal forma que alcancem a outra superfície da peça.
27. Efectue o Subtract dos cilindros obtidos do modelo.
28. Apague as diagonais auxiliares.



29. Utilize o comando Fillet para arredondar os vértices do modelo com $R=0.8$ unidades.



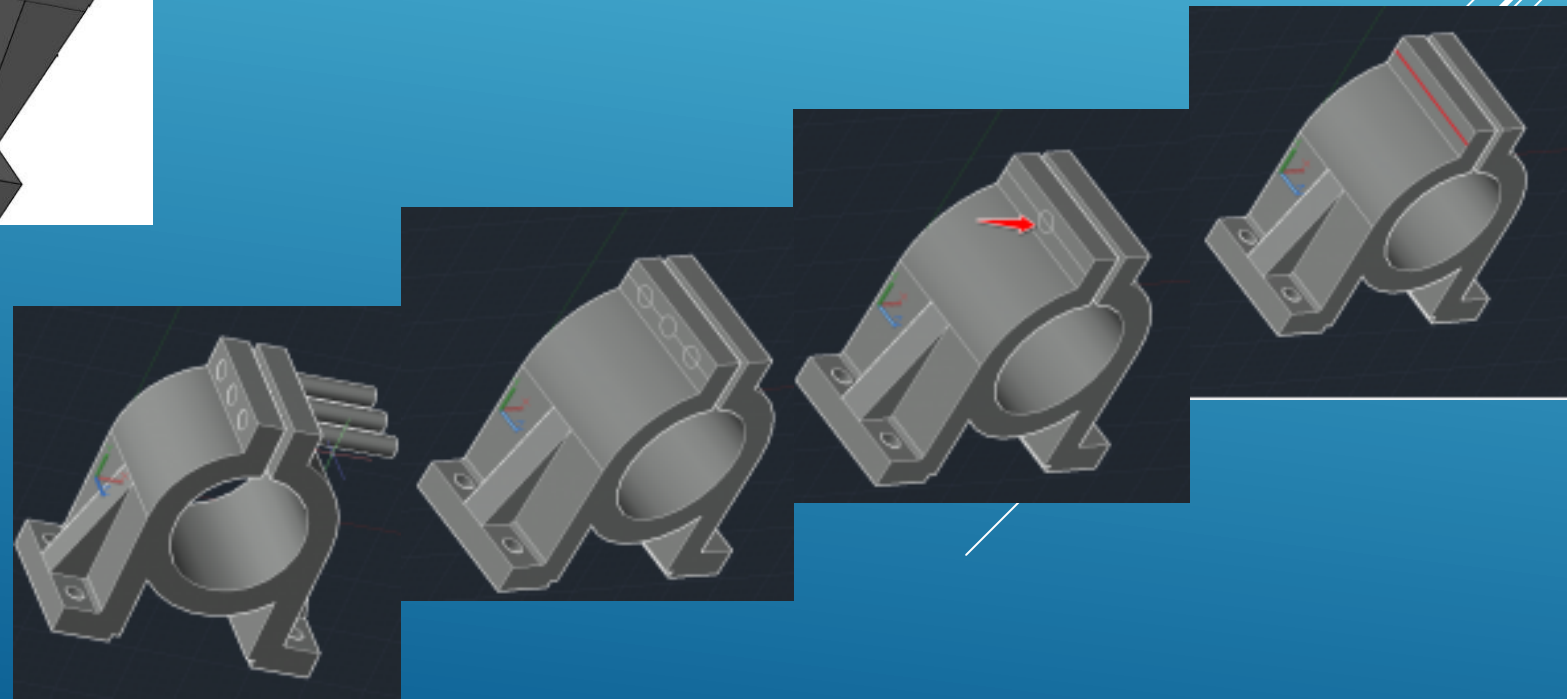
30. Desenhe a linha indicada na figura com snap Midpoint.

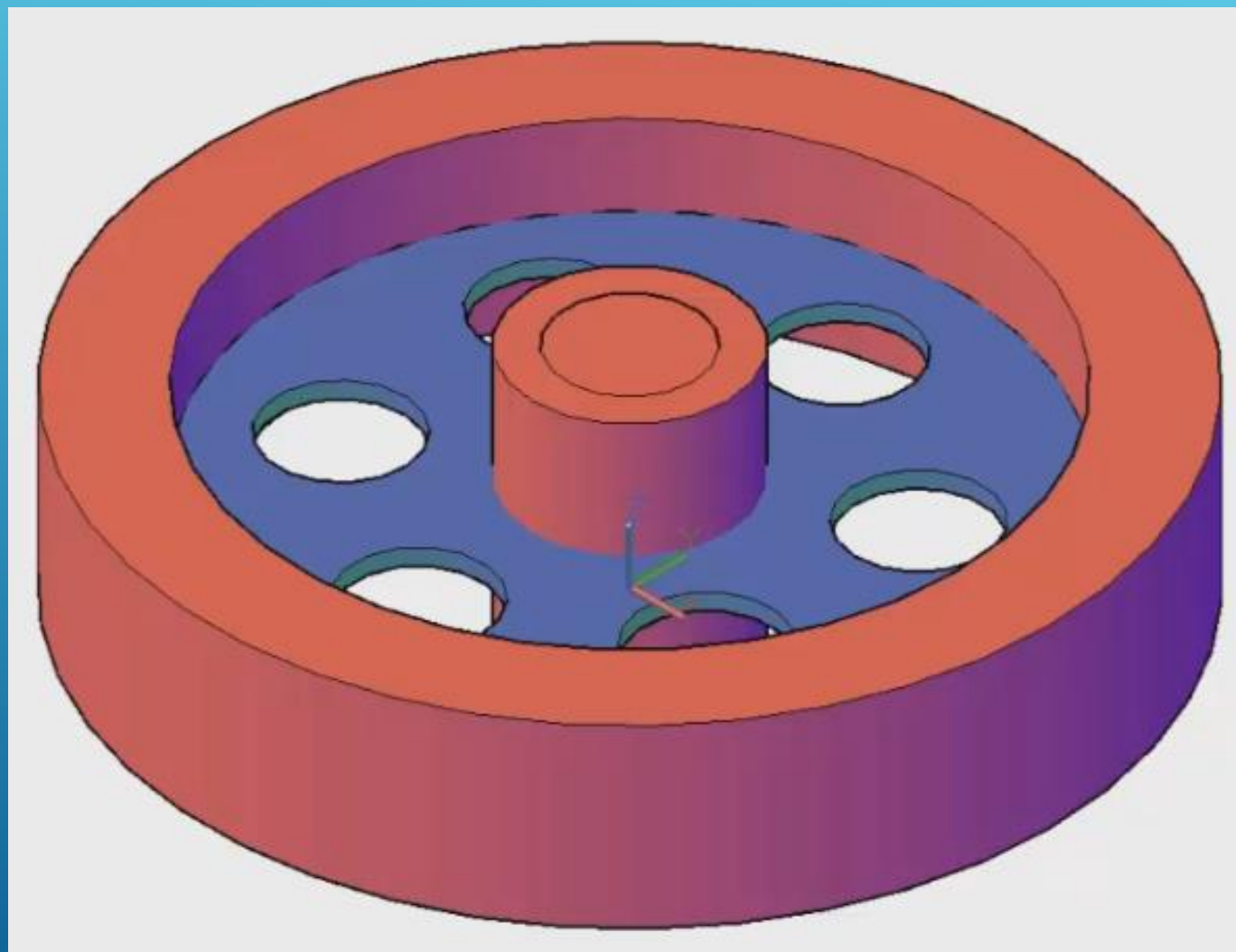
31. Criar circunferências de raio 2 unidades usando o centro do segmento anterior.

32. Acrescente as 2 circunferências restantes com o mesmo raio.

33. Apague as linhas auxiliares e efectue o Extrude das circunferências.

34. Subtraia os cilindros obtidos do modelo.





1. New Drawing
2. Criar 2 layers: Encarnado e Azul
3. Ambiente de trabalho: 3D Modeling
4. 3dWireframe
5. Solid Tab > Cylinder > Center point: **(0,0,0)** + enter
Radius: **430** + enter
Height: **180** + enter
6. Home Tab > View Panel > 3D Navigation > Views > SE Isometric
7. Snap Center
8. Solid Tab > Cylinder > Center point: **(0,0,0)** + enter
Radius: **335** + enter
Height: **180** + enter
9. Solid Tab > Cylinder > Center point: **(0,0,-35)** + enter
Radius: **100** + enter
Height: **250** + enter
10. Solid Tab > Cylinder > Center point: **(0,0,-35)** + enter
Radius: **65** + enter
Height: **250** + enter

1

2

11. Home > Layers > Azul

12. Solid Tab > Cylinder > Center point: **(0,0,82)** + enter 3
Radius: **335** + enter
Height: **16**+ enter

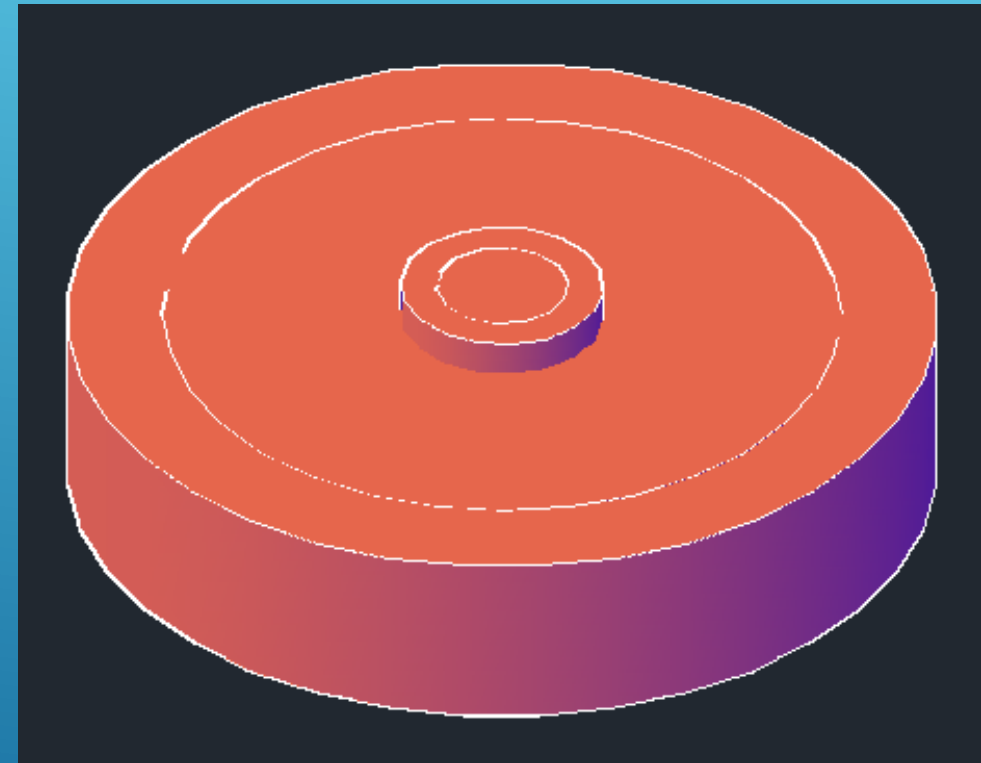
13. Solid Tab > Cylinder > Center point: **(0,-217.5,82)** + enter
Radius: **65** + enter
Height: **16** + enter

14. Home > Modify > Polar Array
(0,0,82)
6 4
60
360

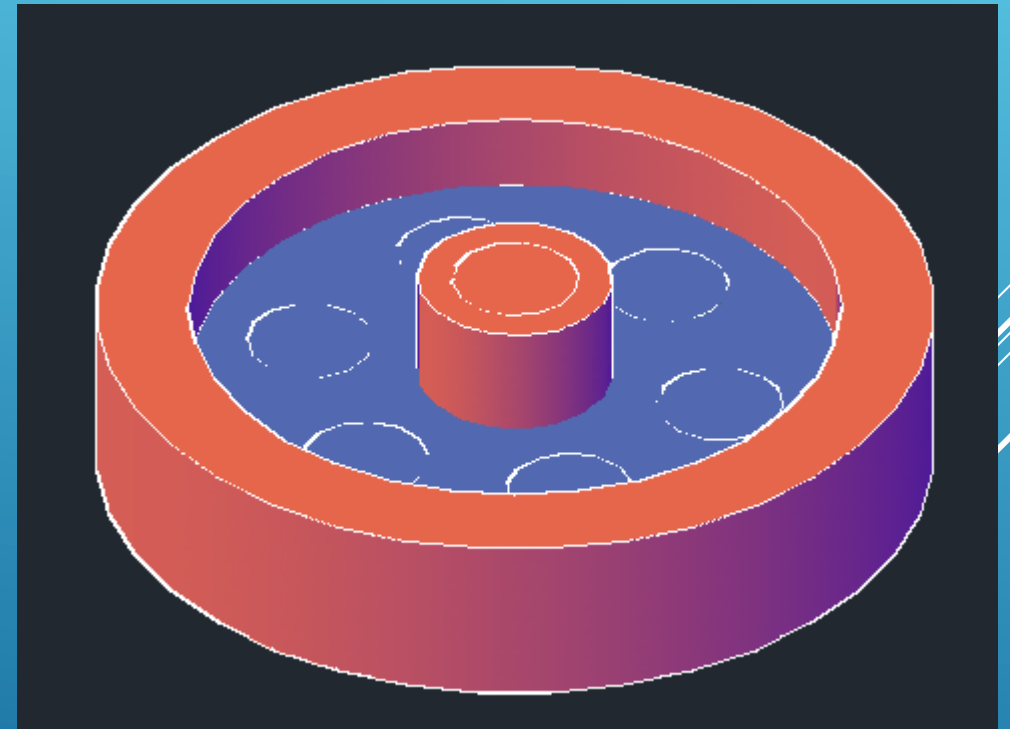
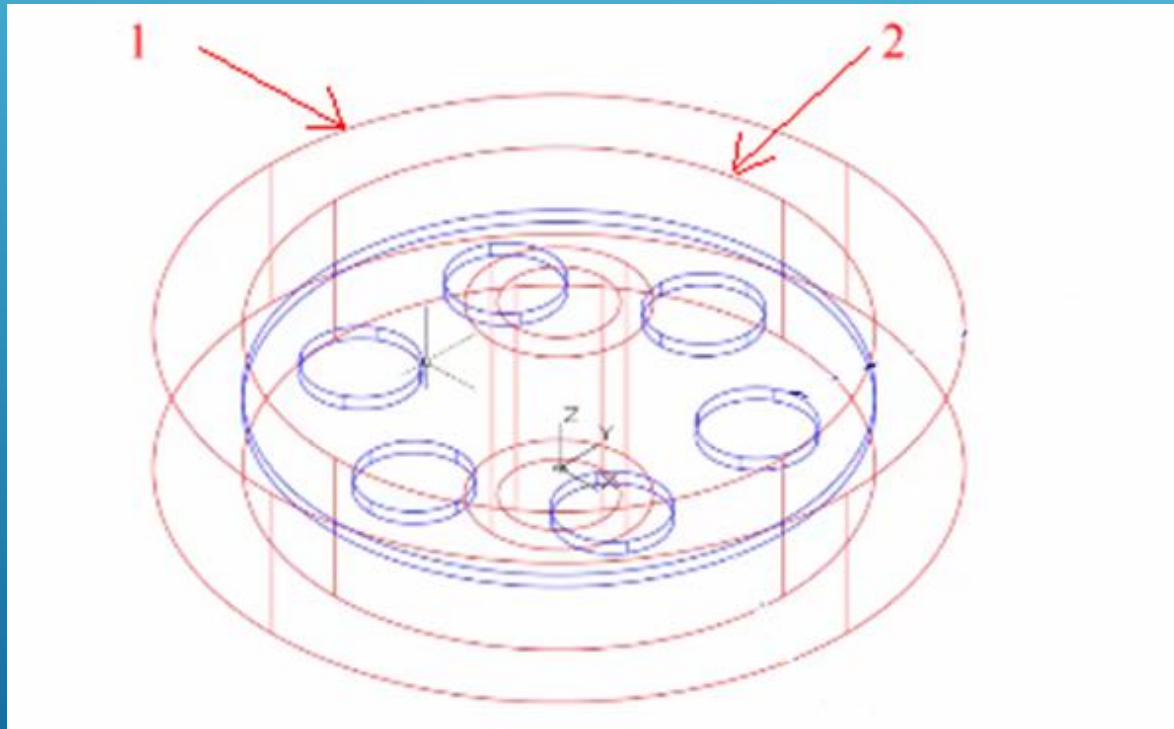
15. Home > Modify > Explode (the polar array)

16. Solid Tab > Cylinder > Center point: **(0,0,82)** + enter
Radius: **65** + enter
Height: **16**+ enter

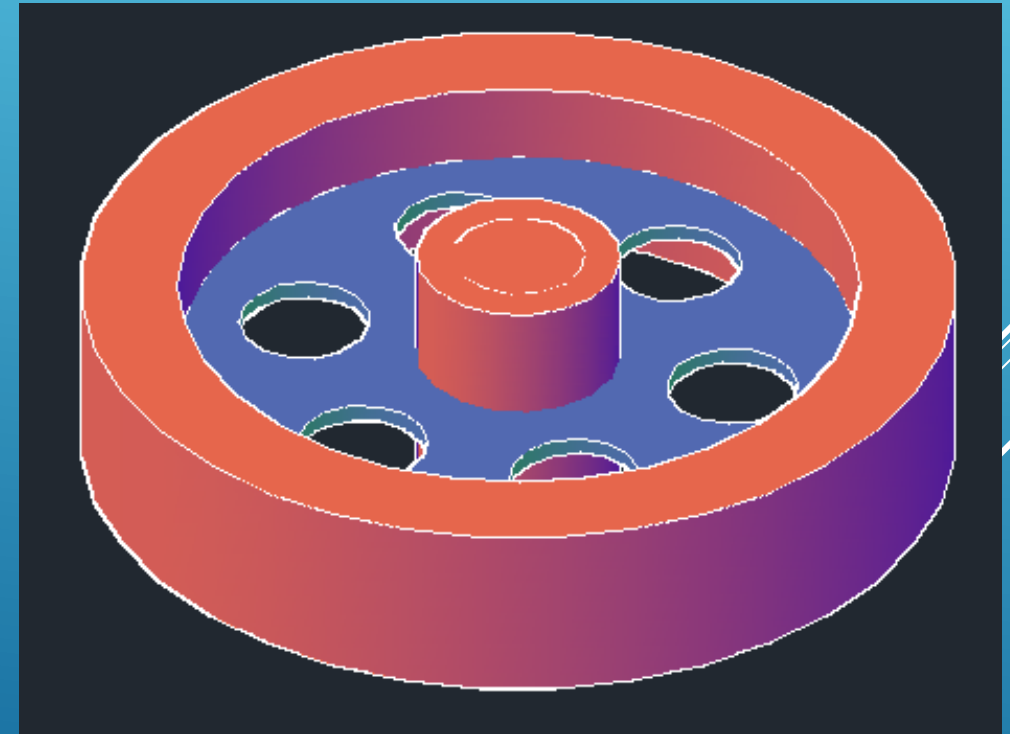
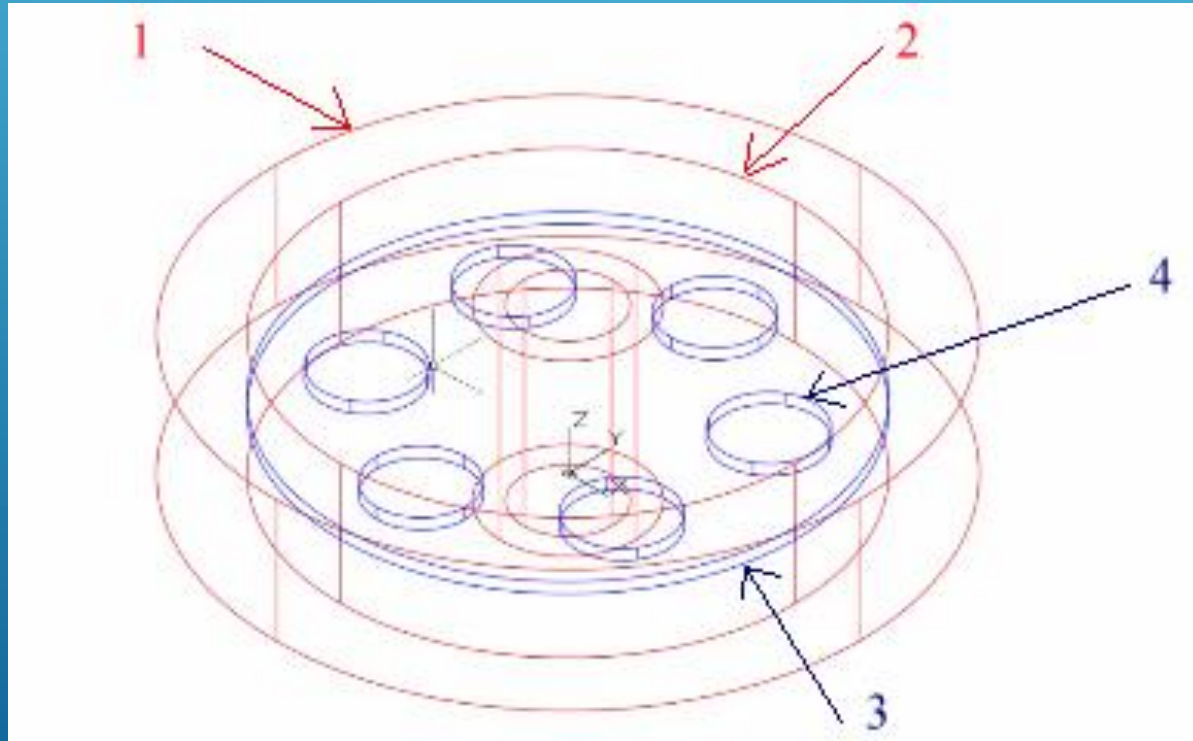
17. Save



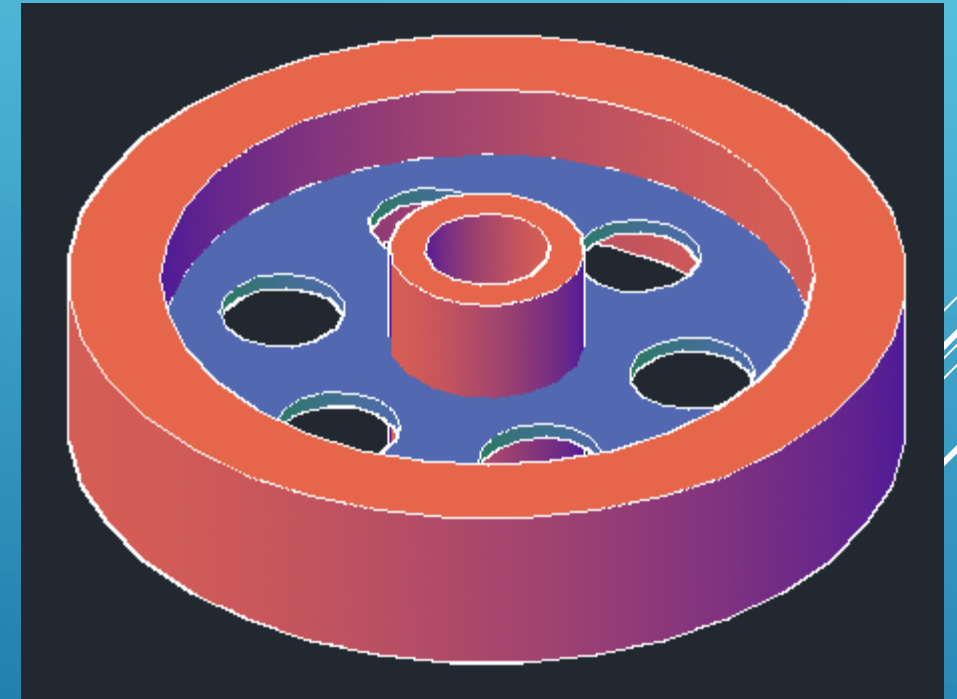
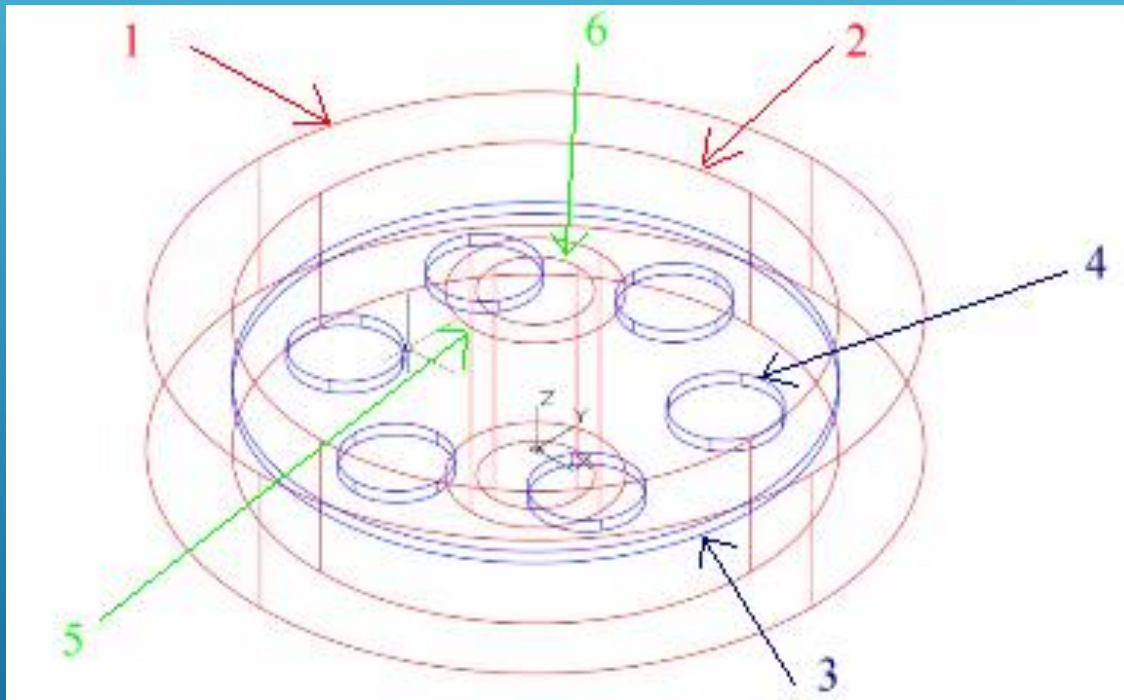
18. Solid Tab > Boolean > Subtract



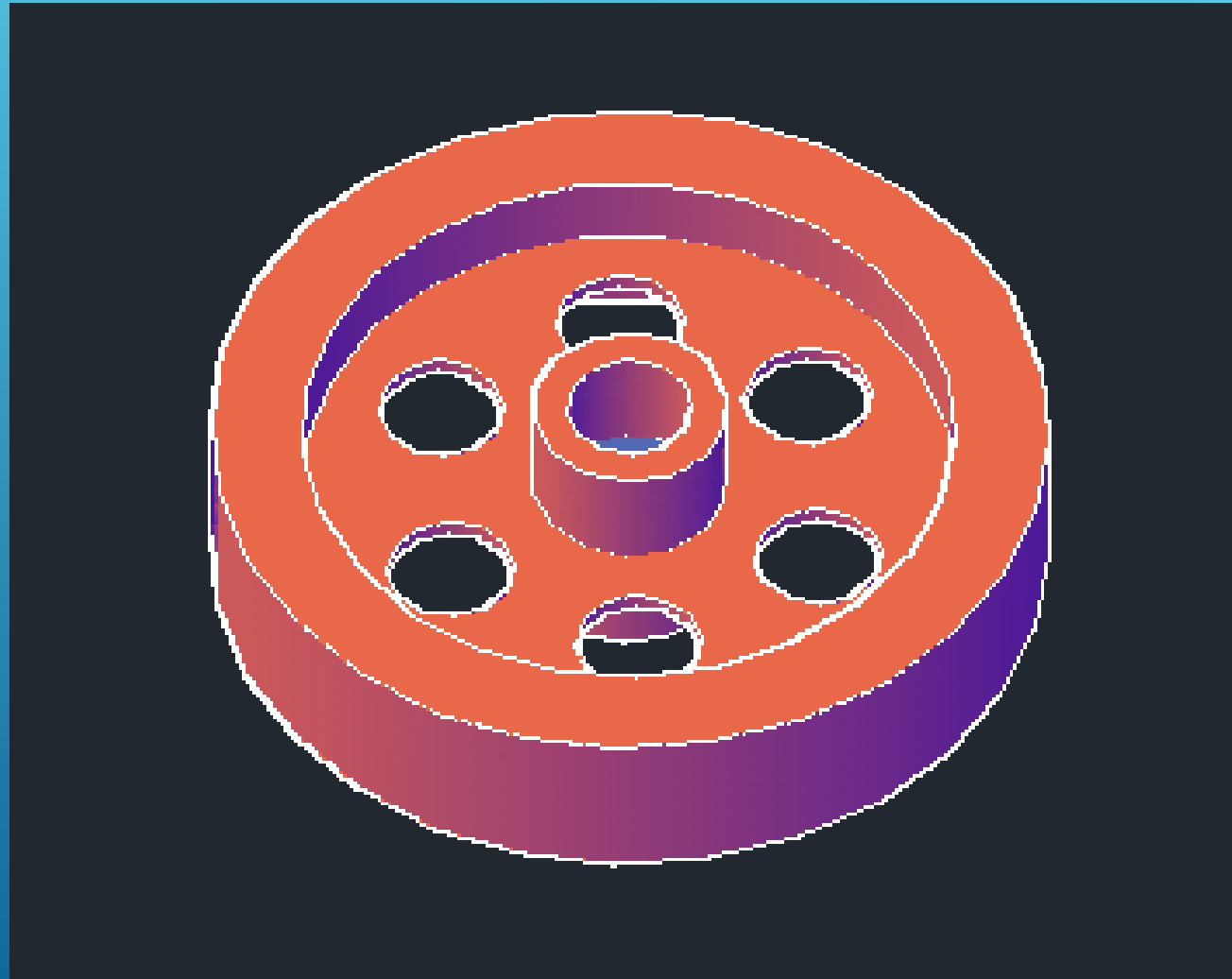
19. Solid Tab > Boolean > Subtract

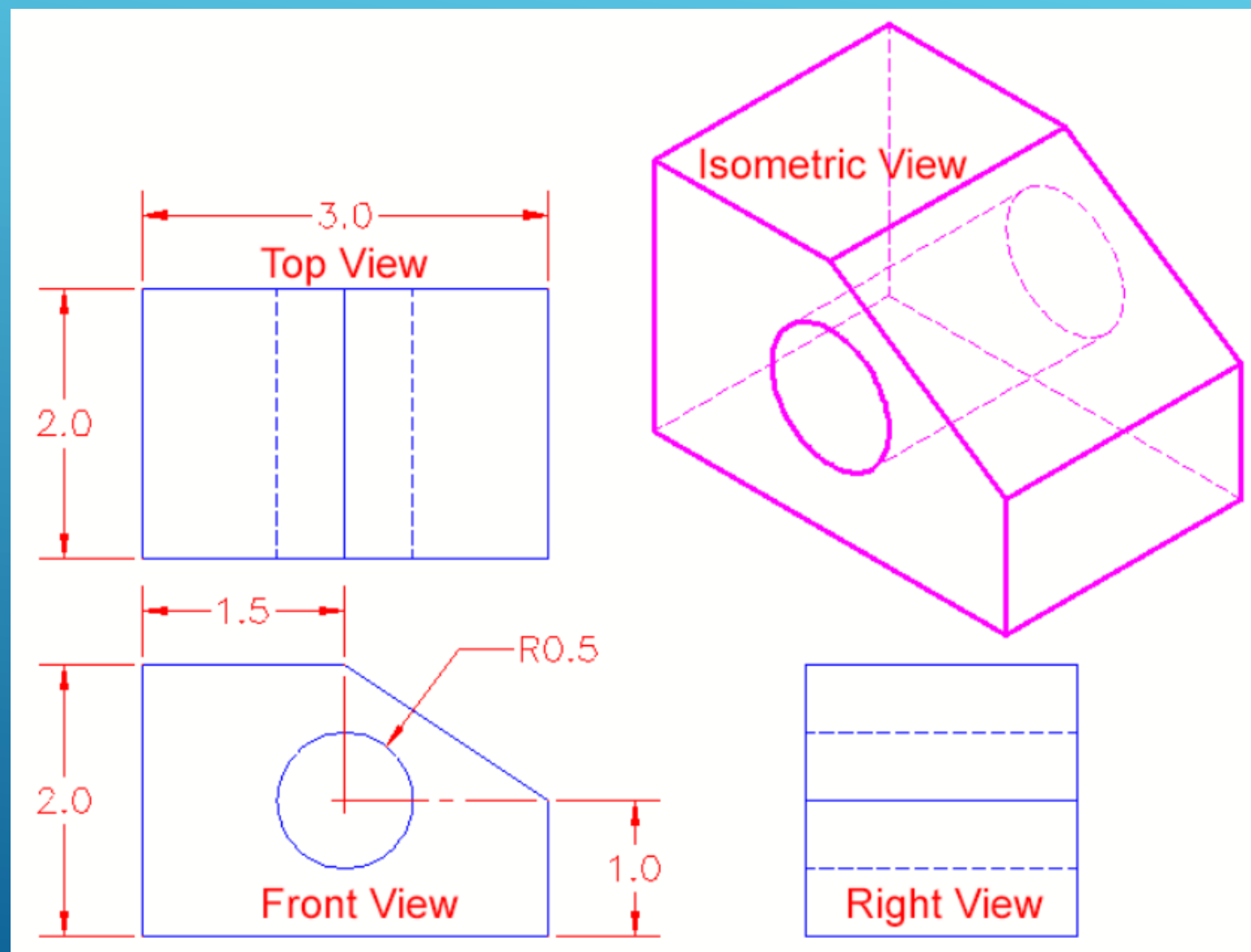


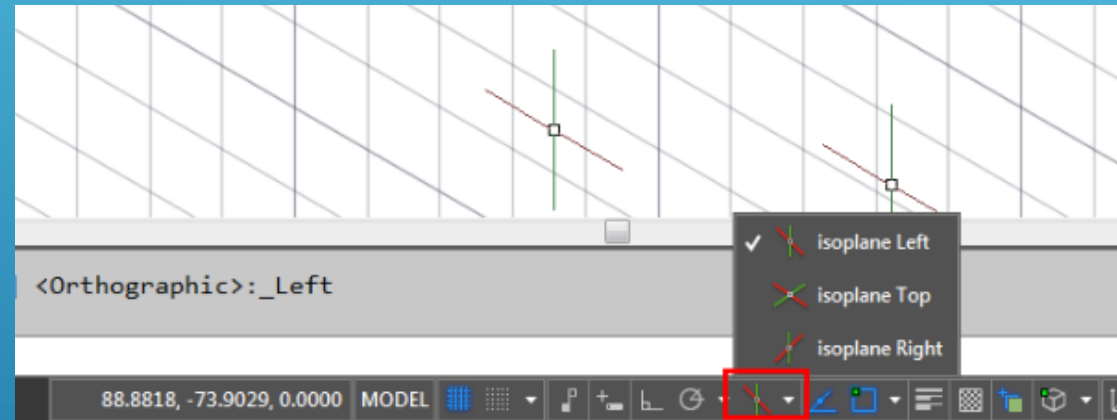
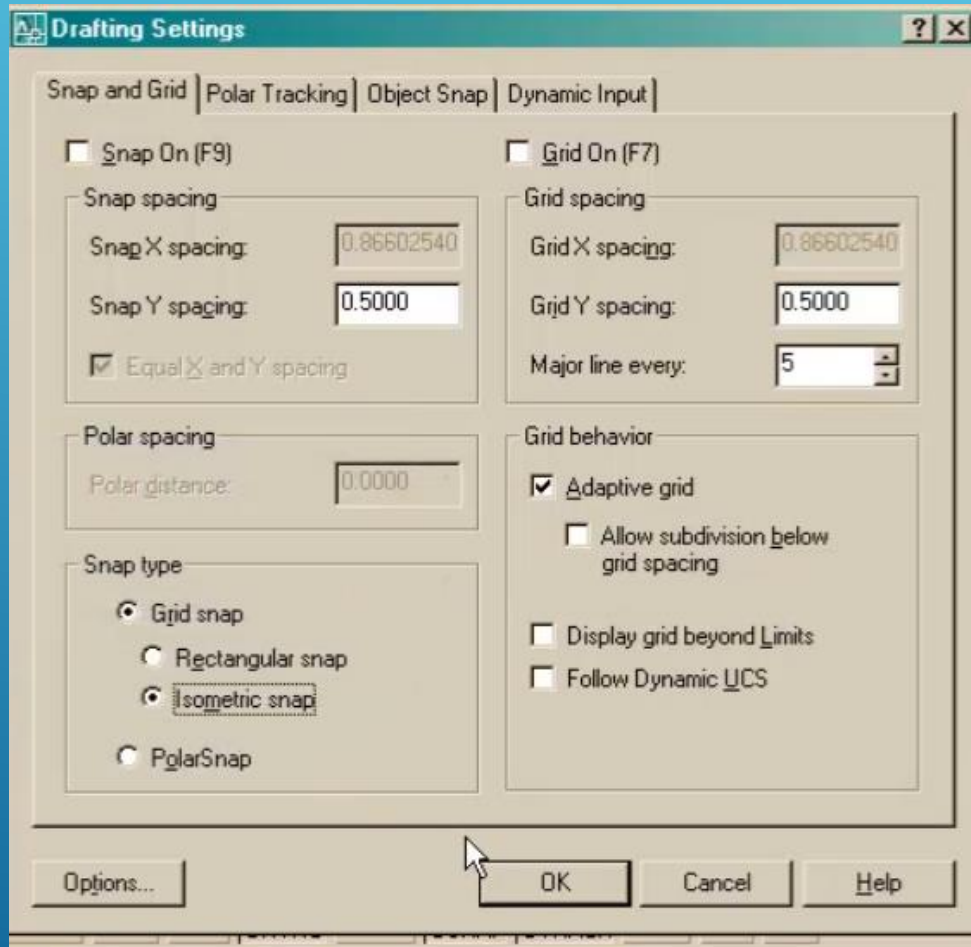
20. Solid Tab > Boolean > Subtract



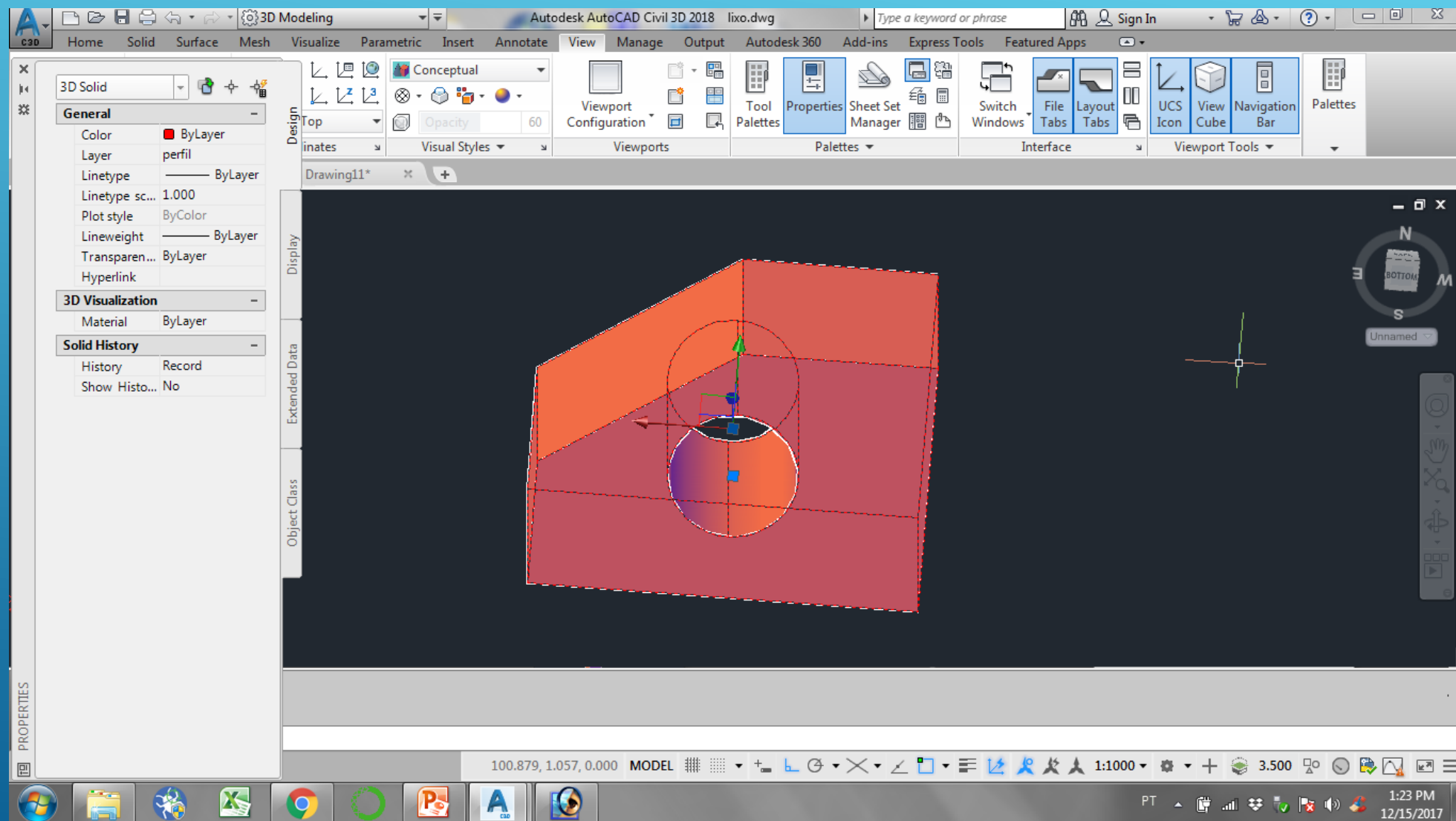
20. Solid Tab > Boolean > Union

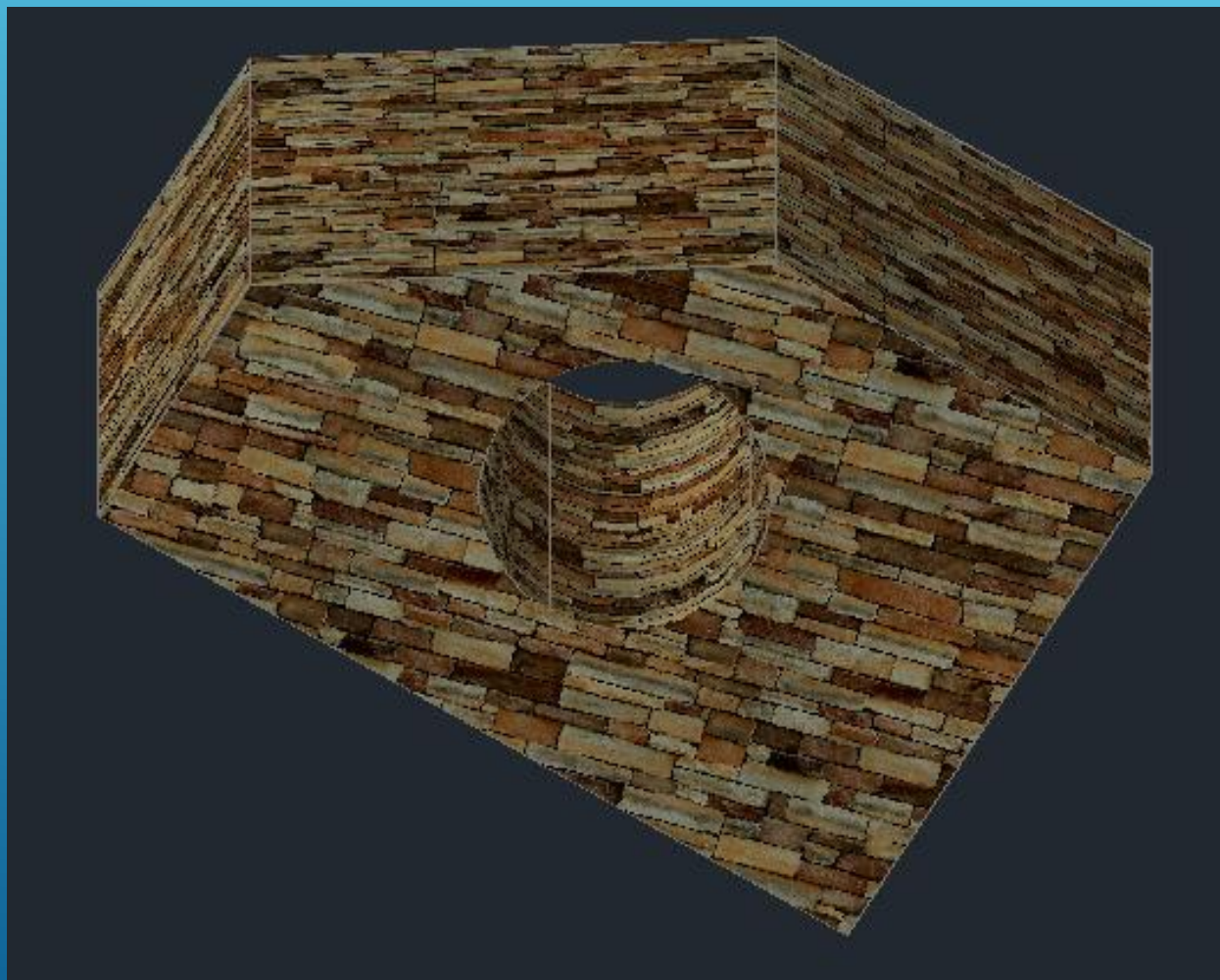




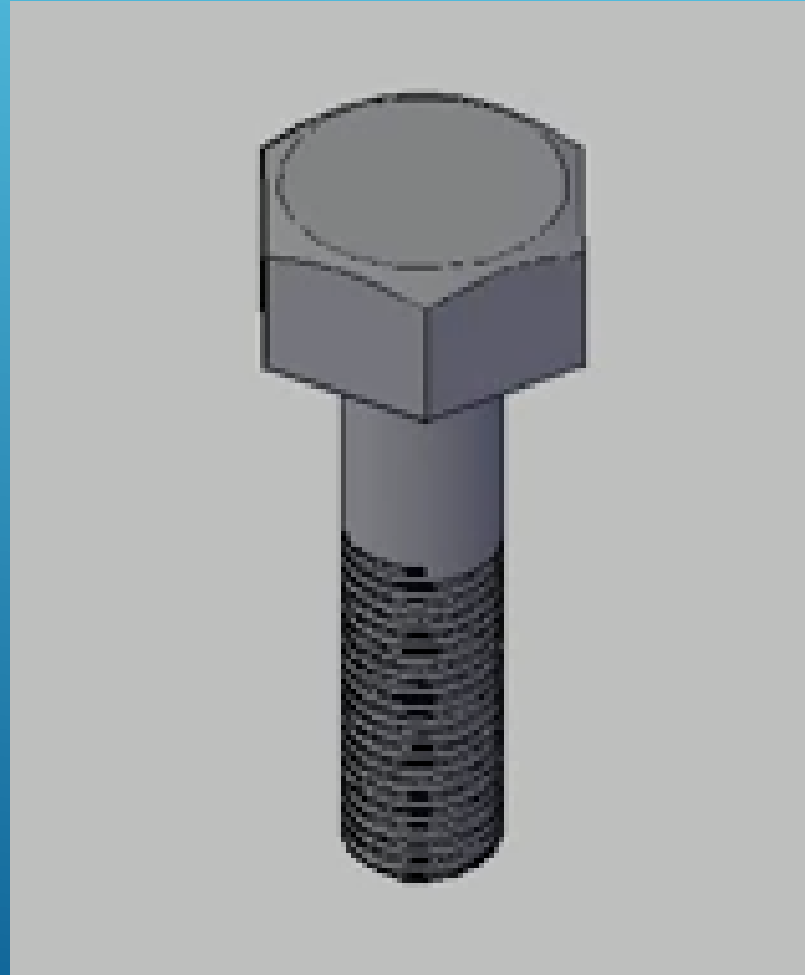


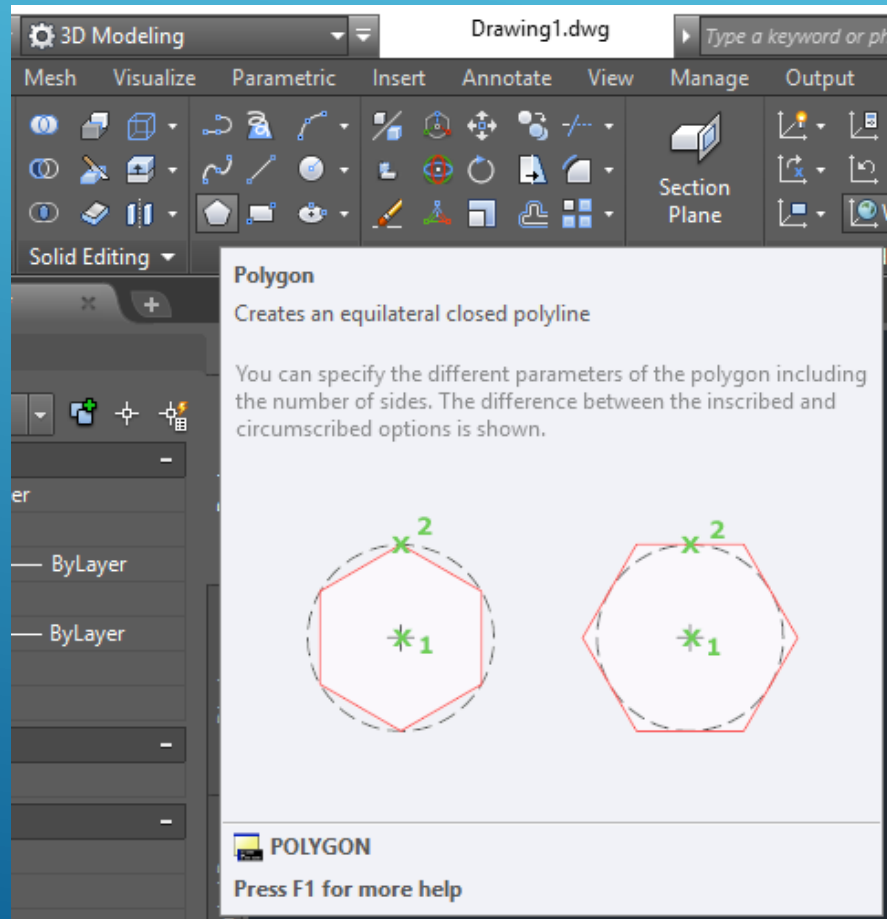
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View > Realistic
RMAT (material Browser)
Seleccionar um material e arrasta-lo
para o sólido





```

Command:
Command:
POLYGON _polygon Enter number of sides <6>:
    
```

```

Command:
Command: _polygon Enter number of sides <6>:
POLYGON Specify center of polygon or [Edge]:
    
```

```

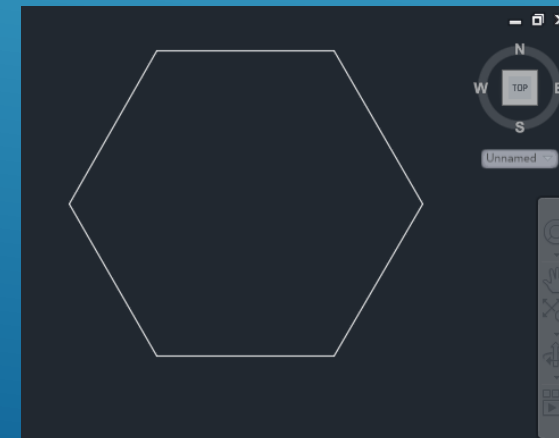
Specify center of polygon or [Edge]:
Point or option keyword required.
POLYGON Specify center of polygon or [Edge]:
    
```

```

Point or option keyword required.
Specify center of polygon or [Edge]:
POLYGON Enter an option [Inscribed in circle Circumscribed about circle] <I>:
    
```

```

Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <I>:
POLYGON Specify radius of circle: 0.375|
    
```

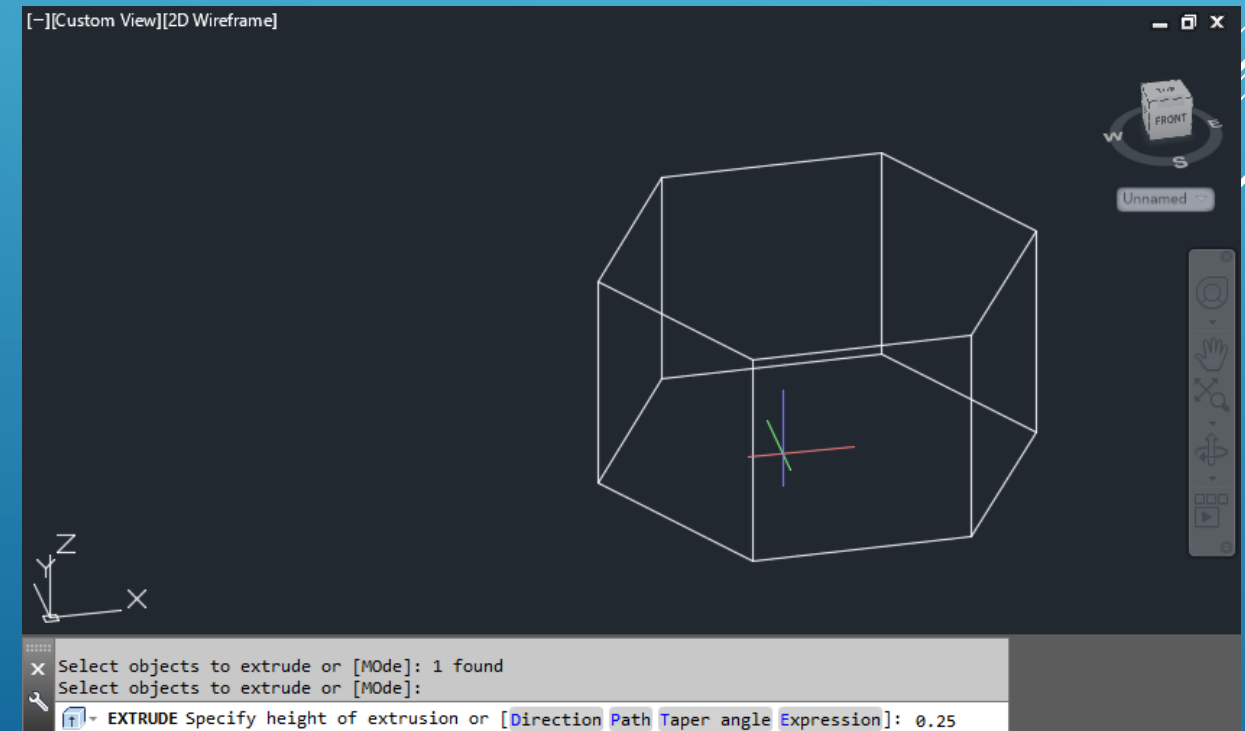
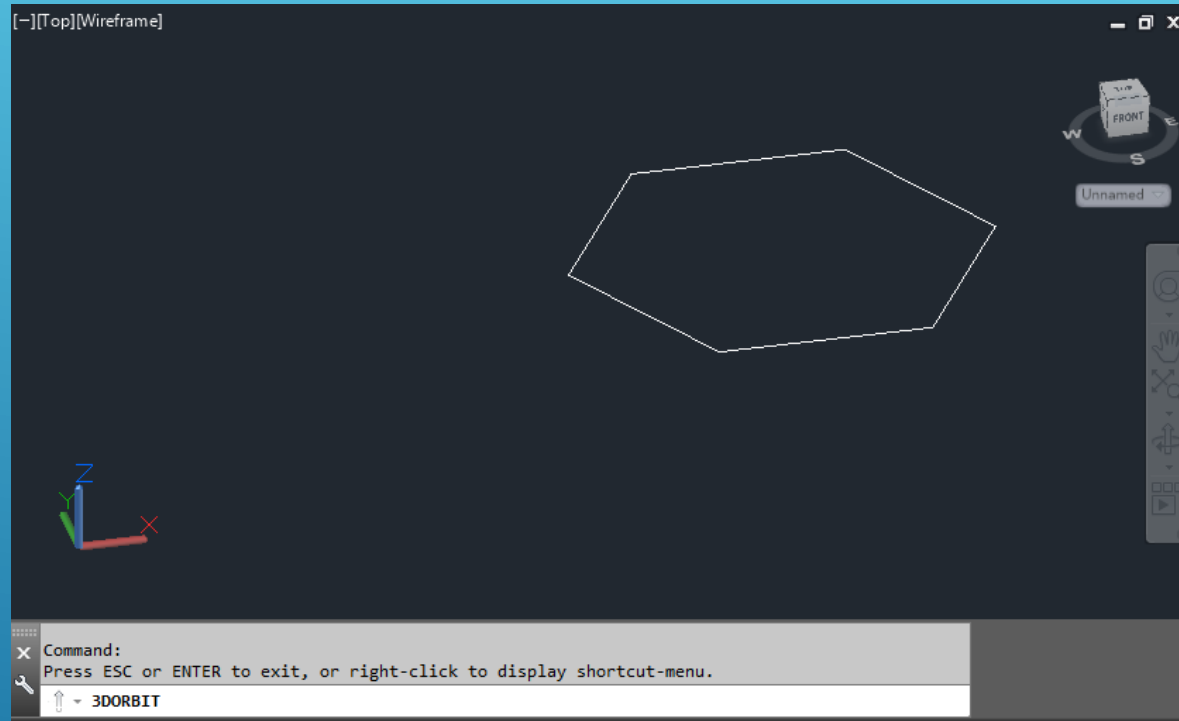


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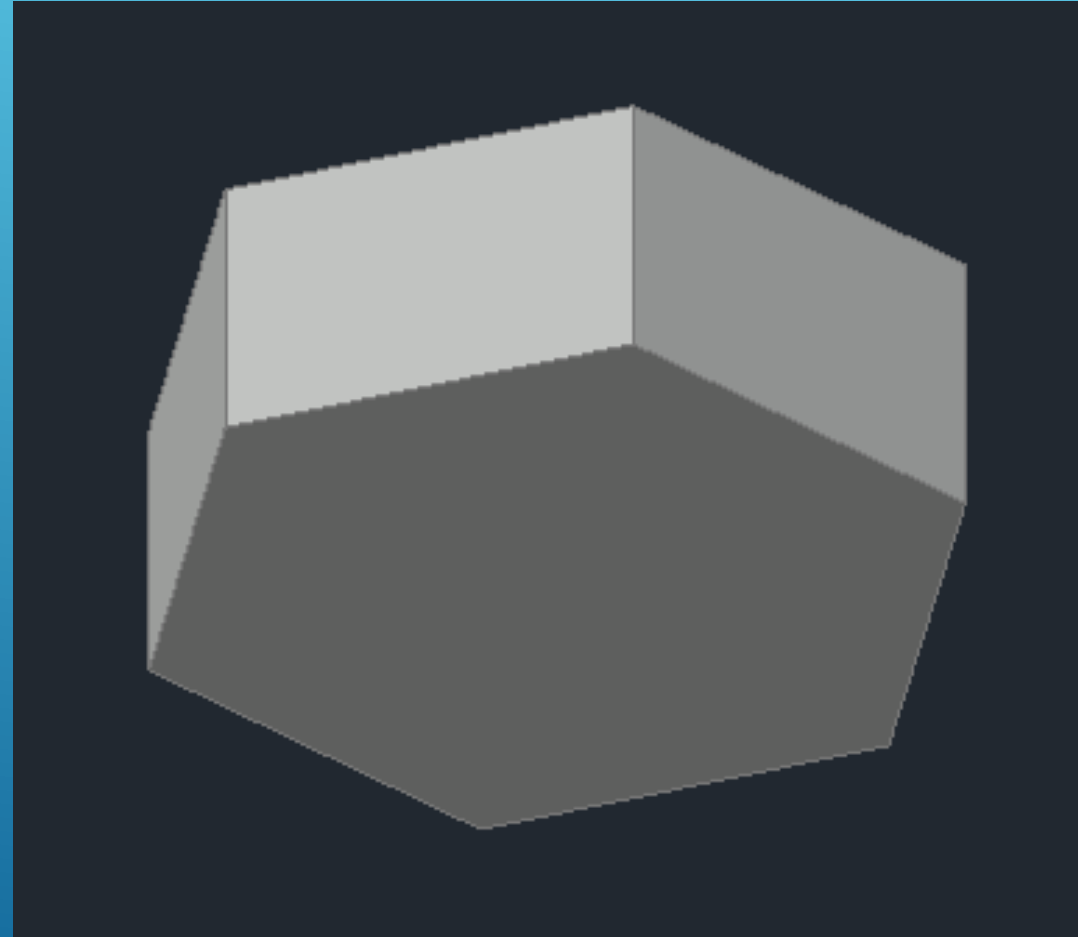
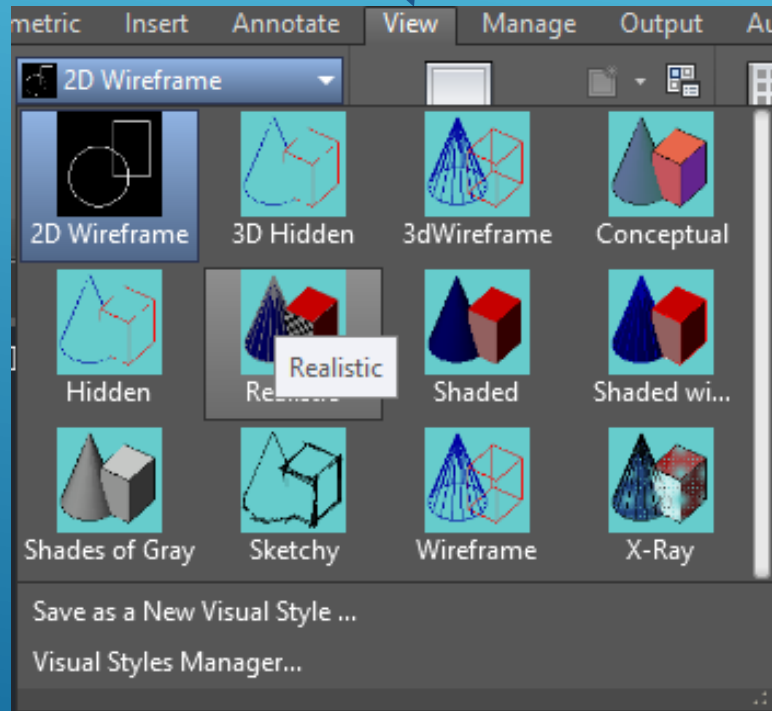


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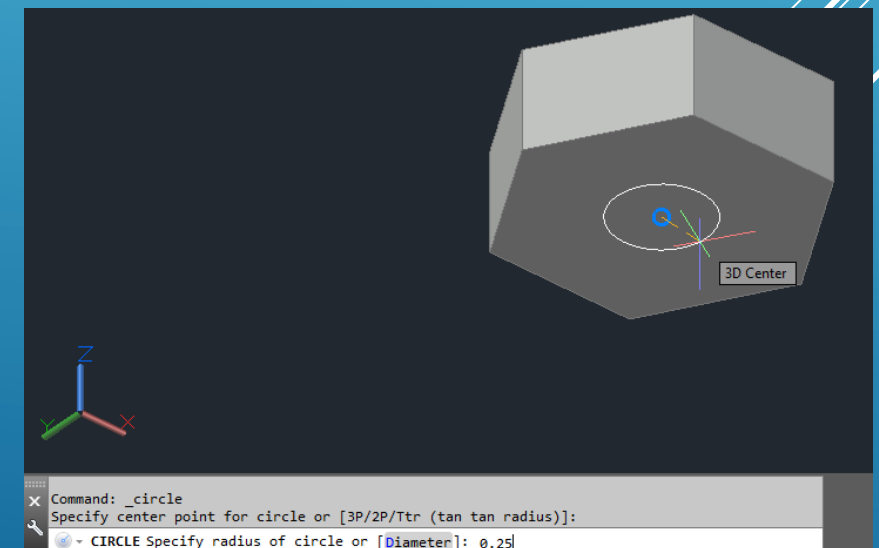
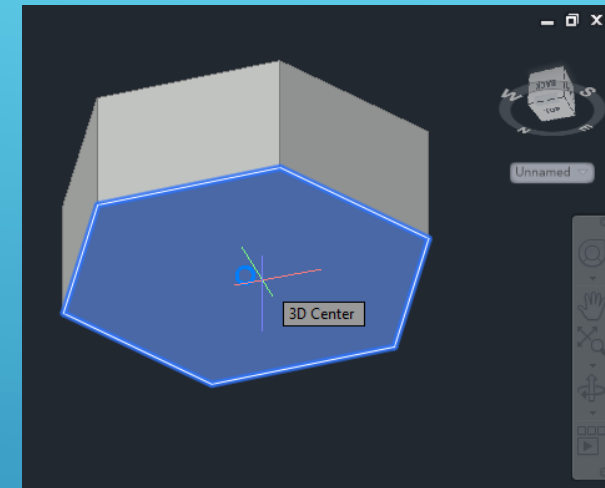


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- ✓ Dynamic Input
- ✓ Ortho Mode
- ✓ Polar Tracking
- ✓ Isometric Drafting
- ✓ Object Snap Tracking
- ✓ 2D Object Snap
- ✓ LineWeight
- Transparency
- Selection Cycling
- 3D Object Snap**
- Dynamic UCS
- Selection Filtering
- Gizmo
- ✓ Annotation Visibility
- ✓ AutoScale
- ✓ Annotation Scale
- ✓ Workspace Switching
- ✓ Annotation Monitor
- Units
- Quick Properties

- ✓ Vertex
 - Midpoint on edge
 - ✓ Center of face
 - Knot
 - Perpendicular
 - Nearest to face
-
- ✓ Node of point cloud
 - Nearest plane of point cloud
 - Perpendicular to point cloud
 - Intersection of point cloud
 - Nearest to edge of point cloud
 - Perpendicular to edge of point cloud
 - Nearest to centerline of cylinder on point cloud
 - Corner of point cloud
- Object Snap Settings...
- 1:1000 3.500

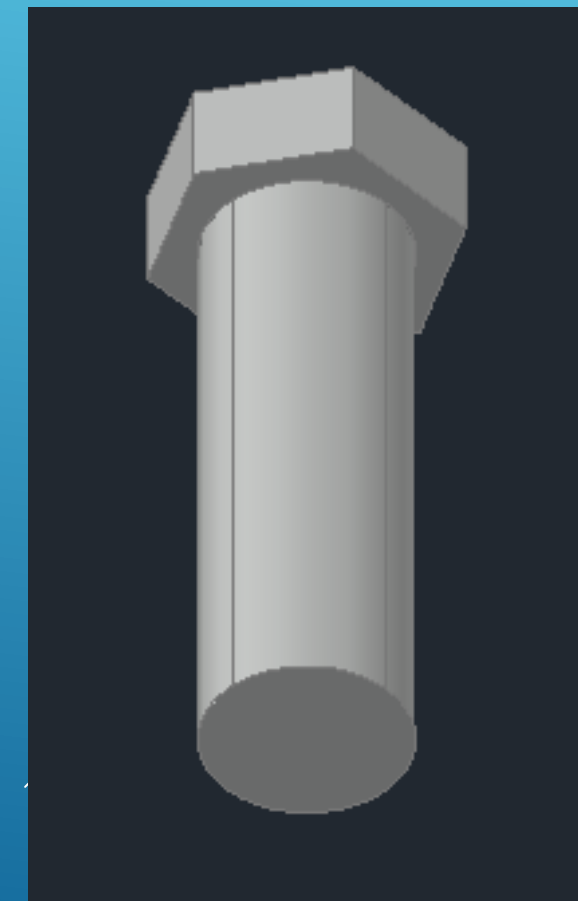
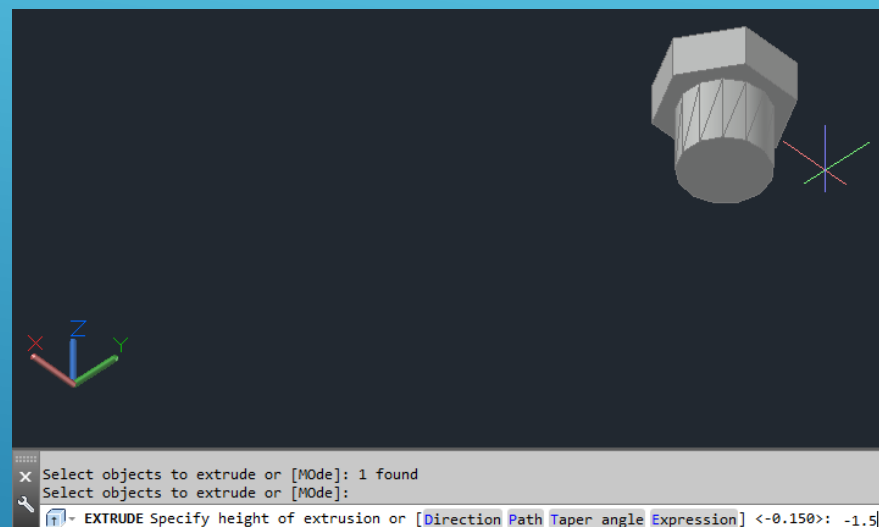
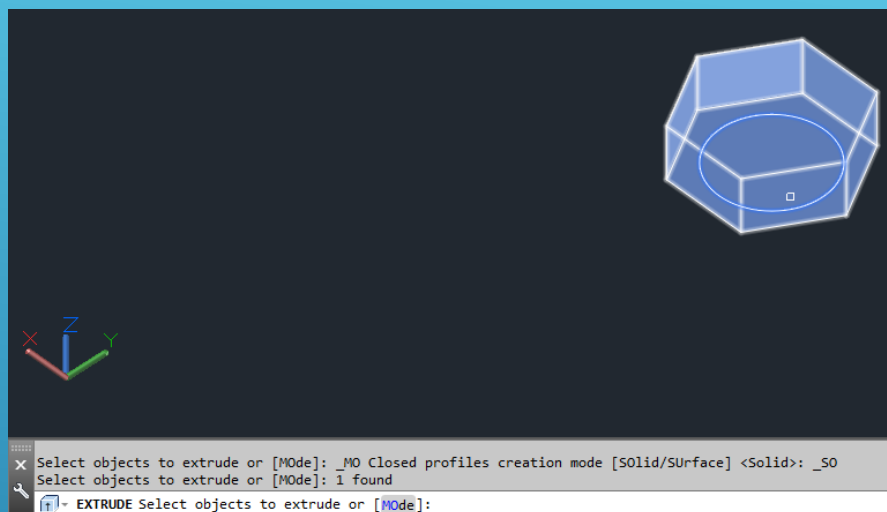


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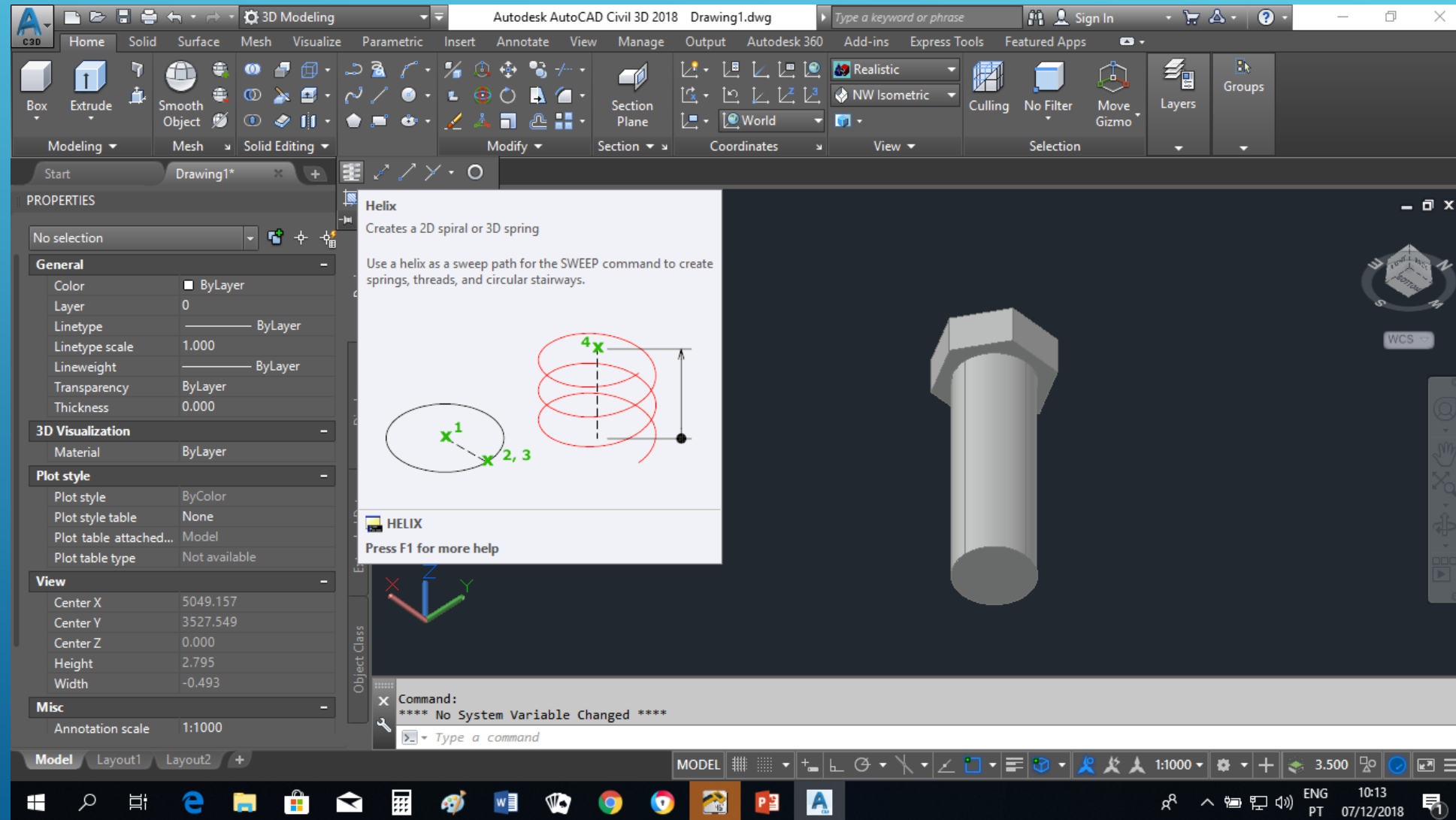


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The screenshot displays the Autodesk AutoCAD Civil 3D 2018 software interface. The main window shows a 3D model of a hexagonal bolt. A help window for the HELIX command is open, providing instructions and a diagram. The Properties palette on the left shows the current object's settings. The Command line at the bottom shows the command prompt.

Autodesk AutoCAD Civil 3D 2018 Drawing1.dwg

Modeling | **Mesh** | **Solid Editing** | **Modify** | **Section** | **Coordinates** | **View** | **Selection**

Start | **Drawing1***

PROPERTIES

No selection

General

- Color: ByLayer
- Layer: 0
- Linetype: ByLayer
- Linetype scale: 1.000
- Lineweight: ByLayer
- Transparency: ByLayer
- Thickness: 0.000

3D Visualization

- Material: ByLayer

Plot style

- Plot style: ByColor
- Plot style table: None
- Plot table attached...: Model
- Plot table type: Not available

View

- Center X: 5049.157
- Center Y: 3527.549
- Center Z: 0.000
- Height: 2.795
- Width: -0.493

Misc

- Annotation scale: 1:1000

Helix

Creates a 2D spiral or 3D spring

Use a helix as a sweep path for the SWEEP command to create springs, threads, and circular stairways.

HELIX

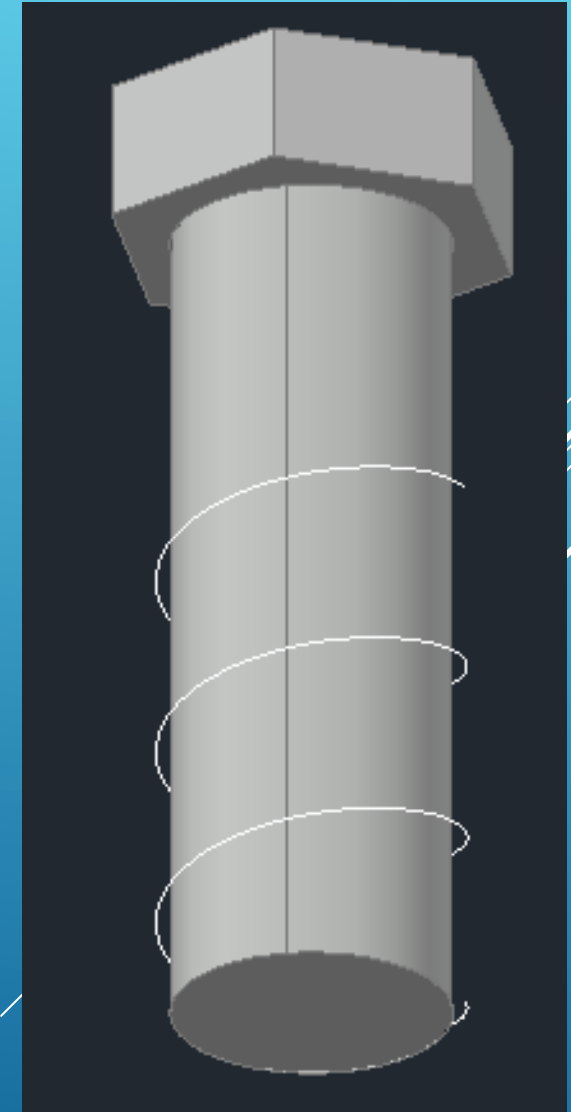
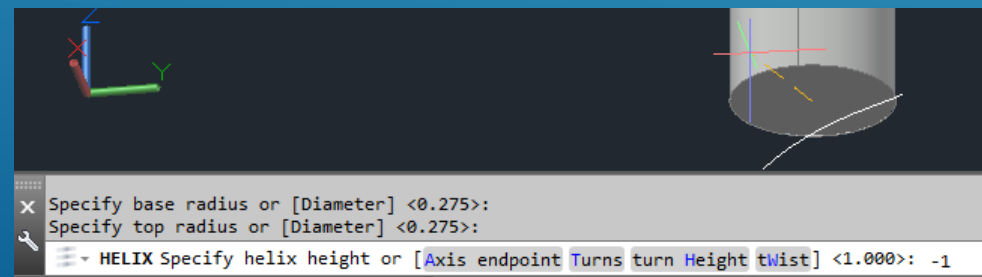
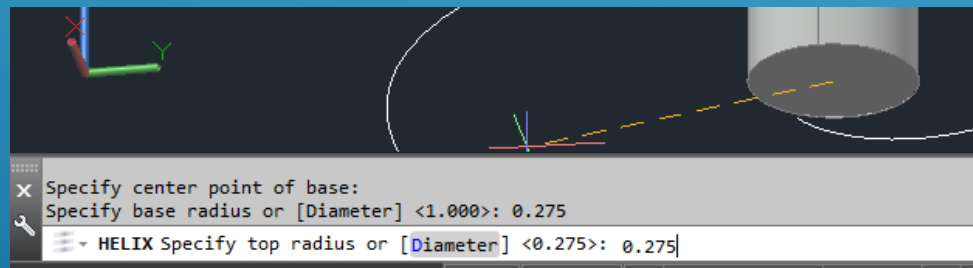
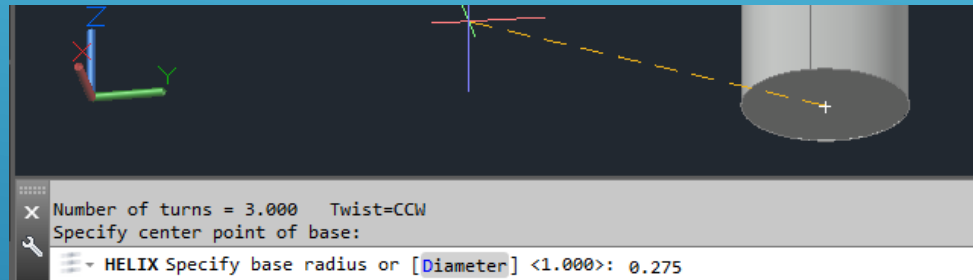
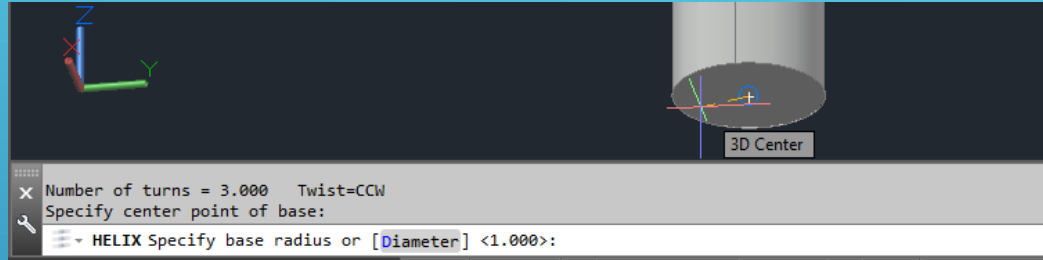
Press F1 for more help

Command:

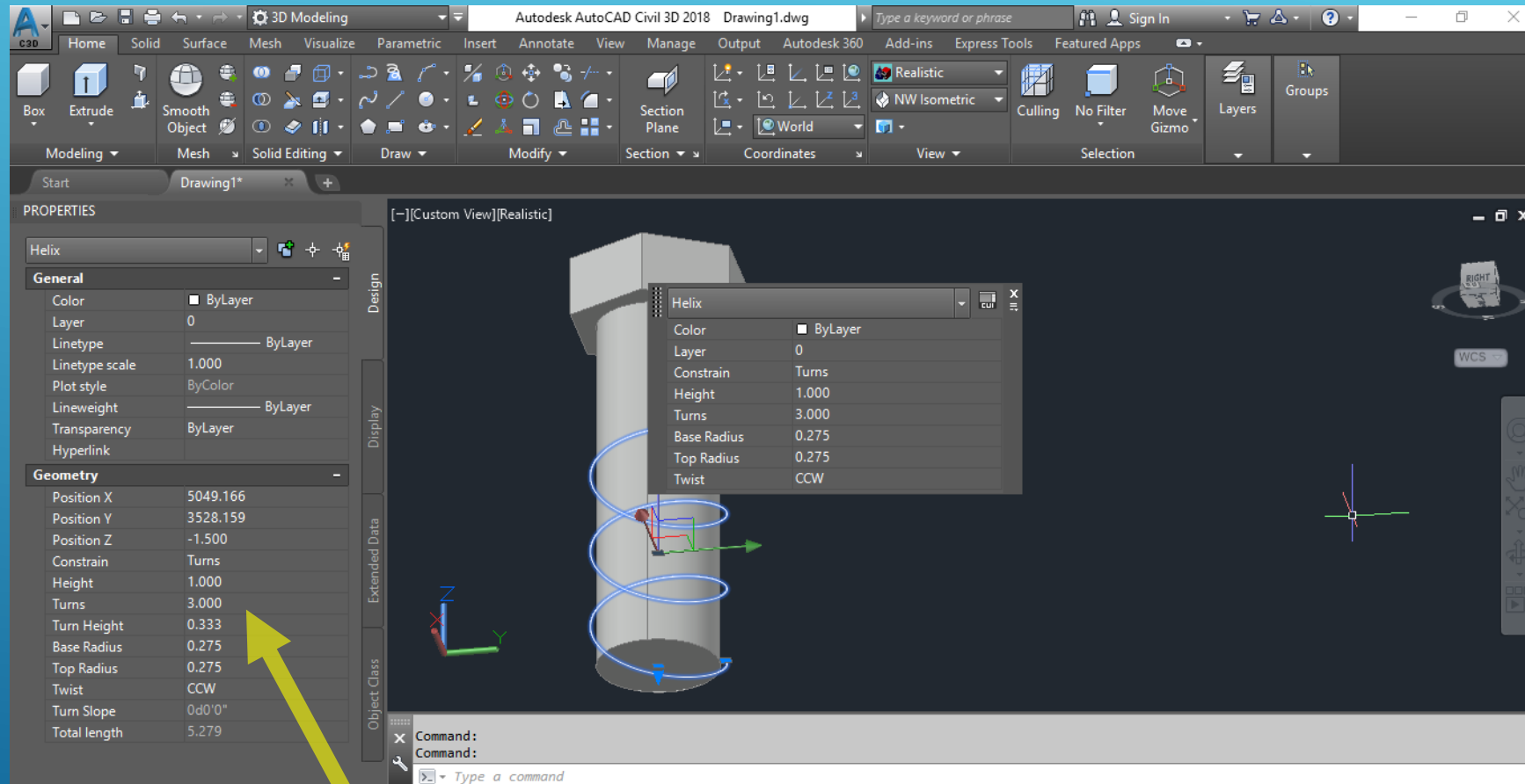
**** No System Variable Changed ****

Type a command

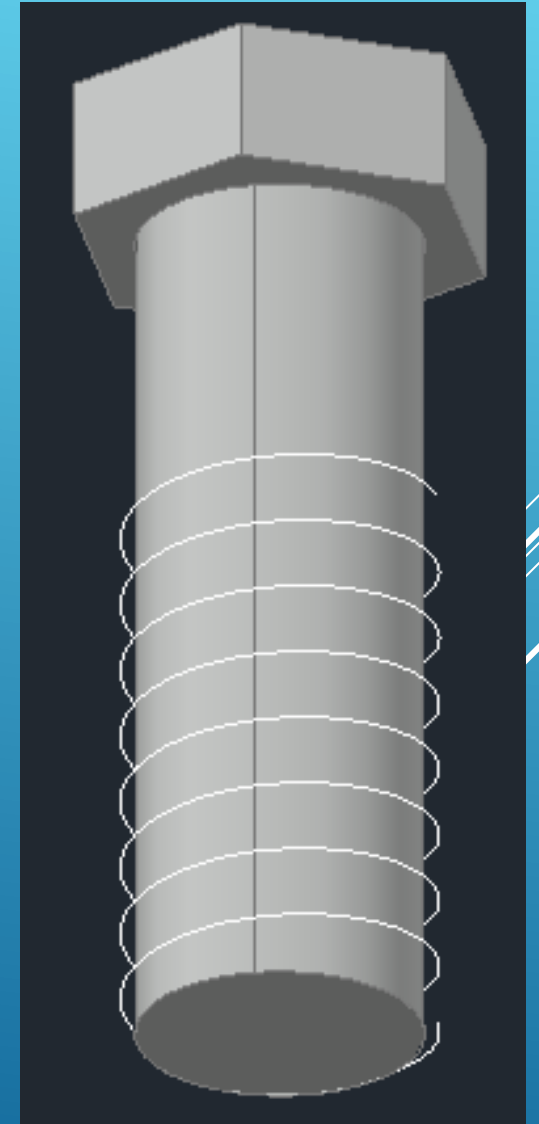
MODEL | 1:1000 | 3.500 | ENG PT | 10:13 | 07/12/2018



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Turns: 8

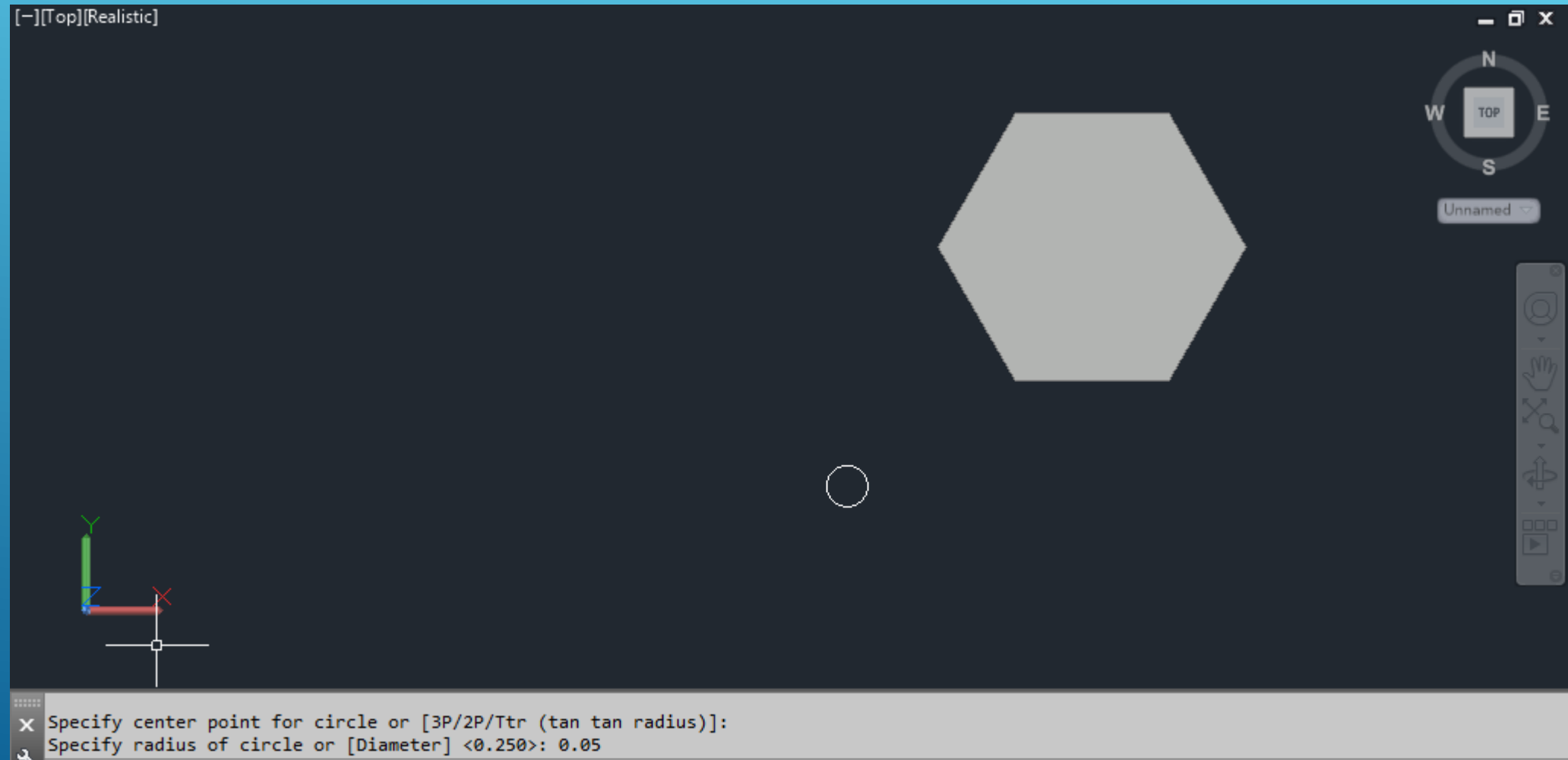


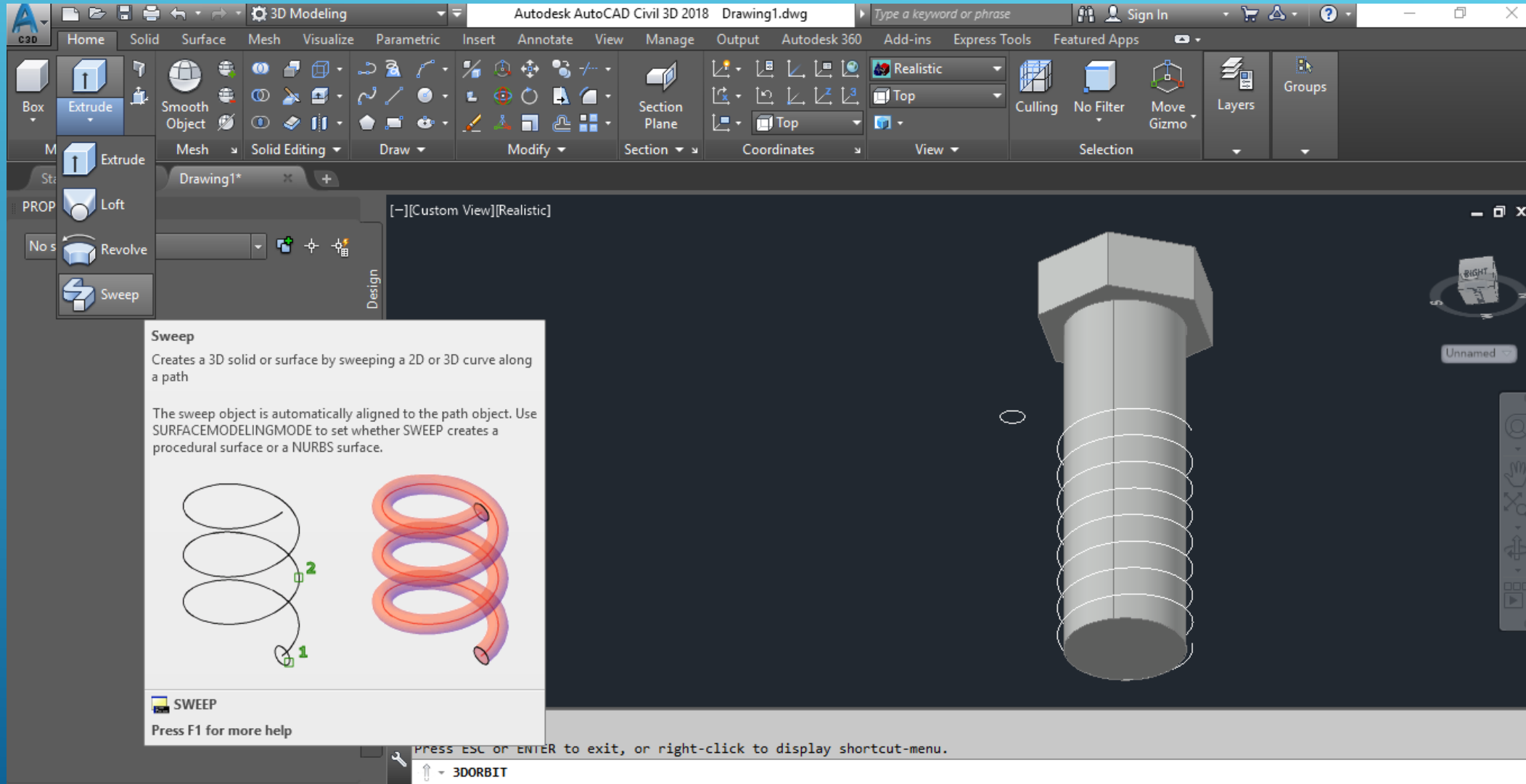
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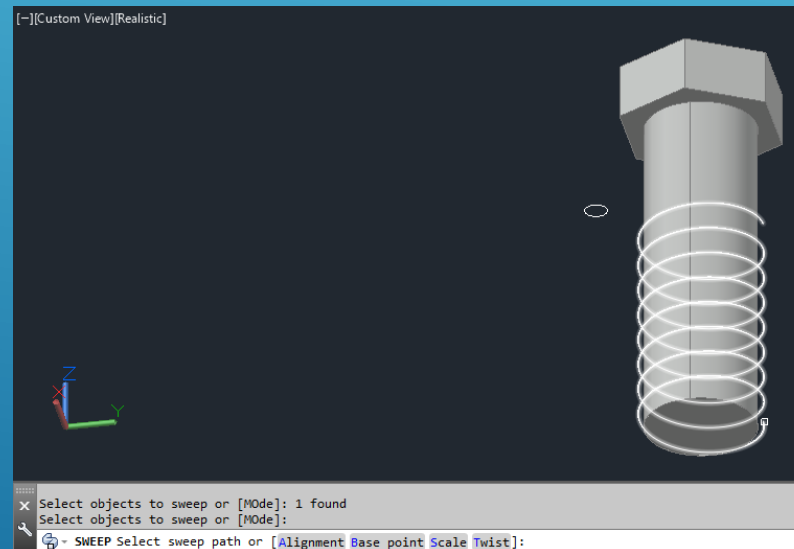
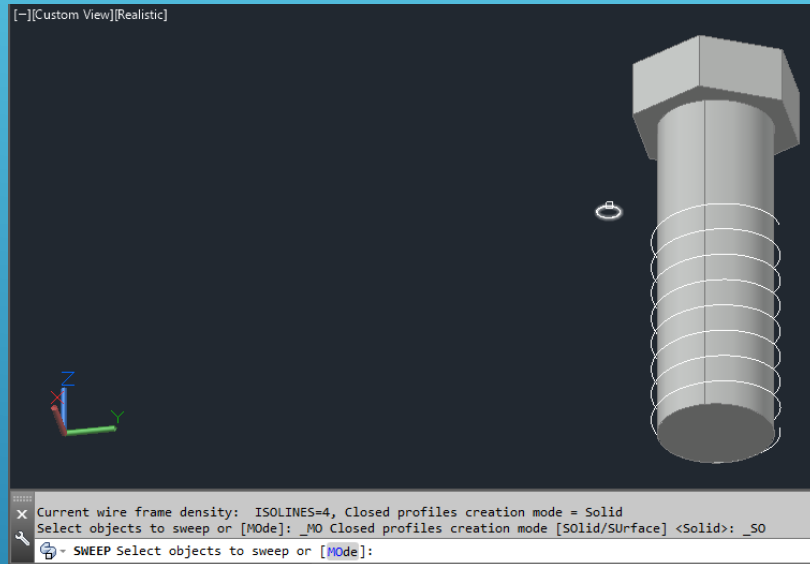


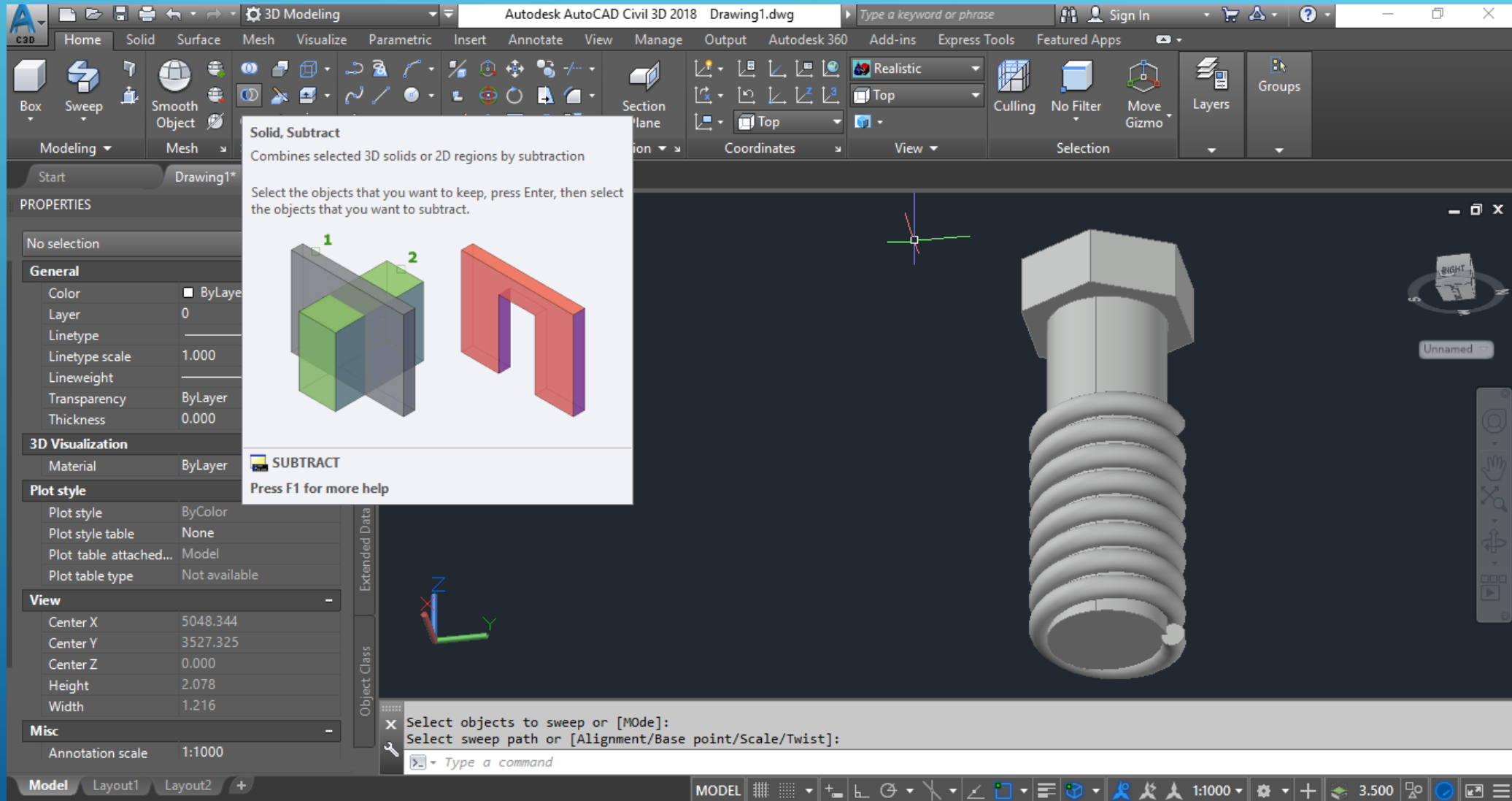
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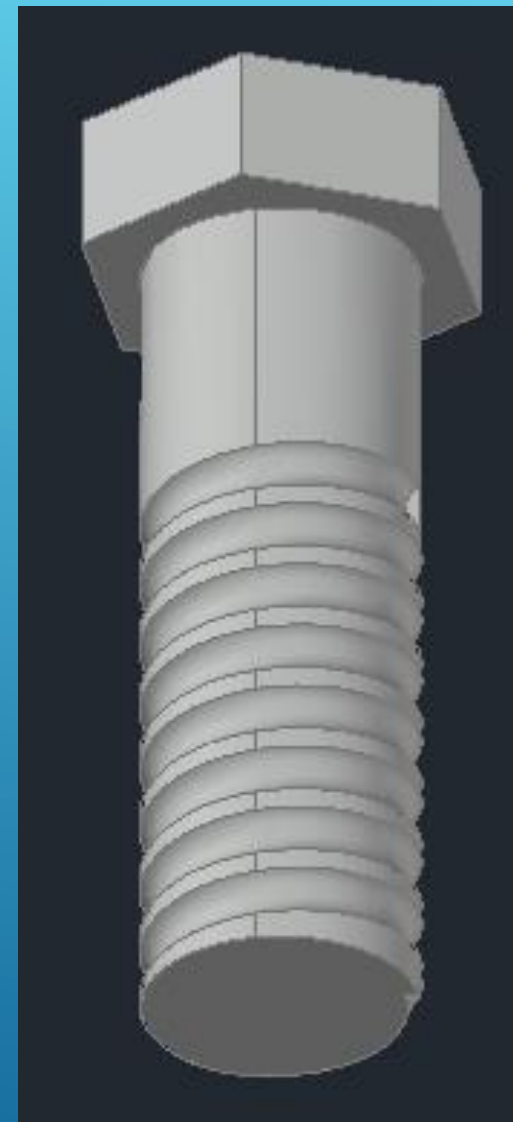
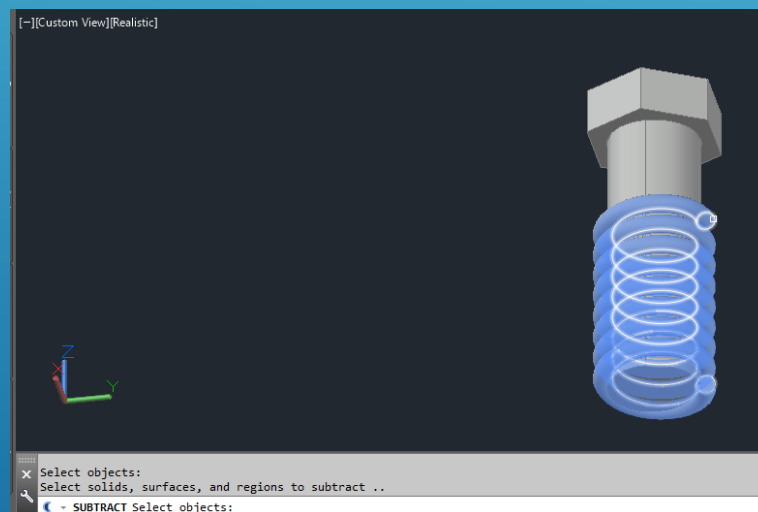
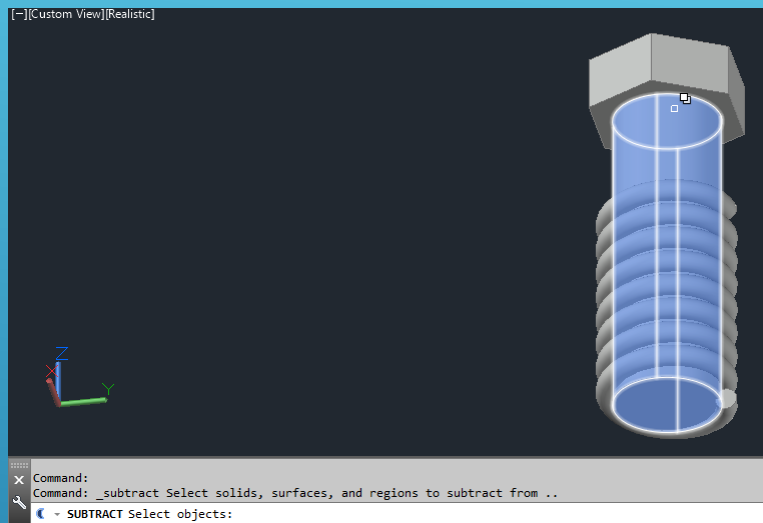
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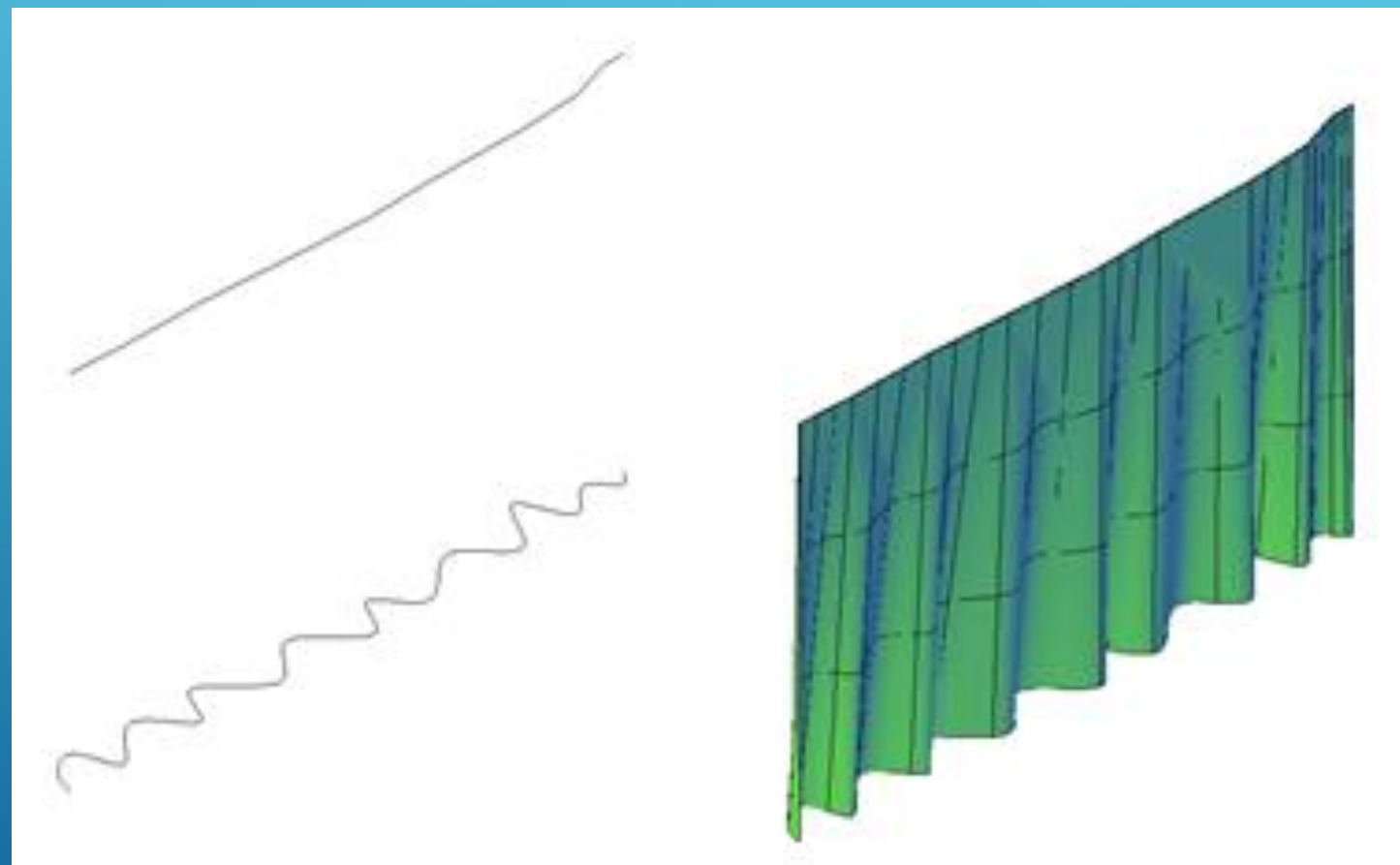
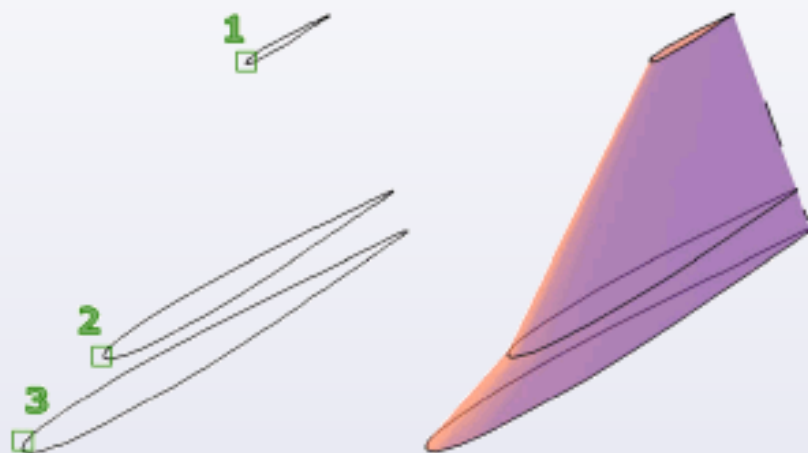


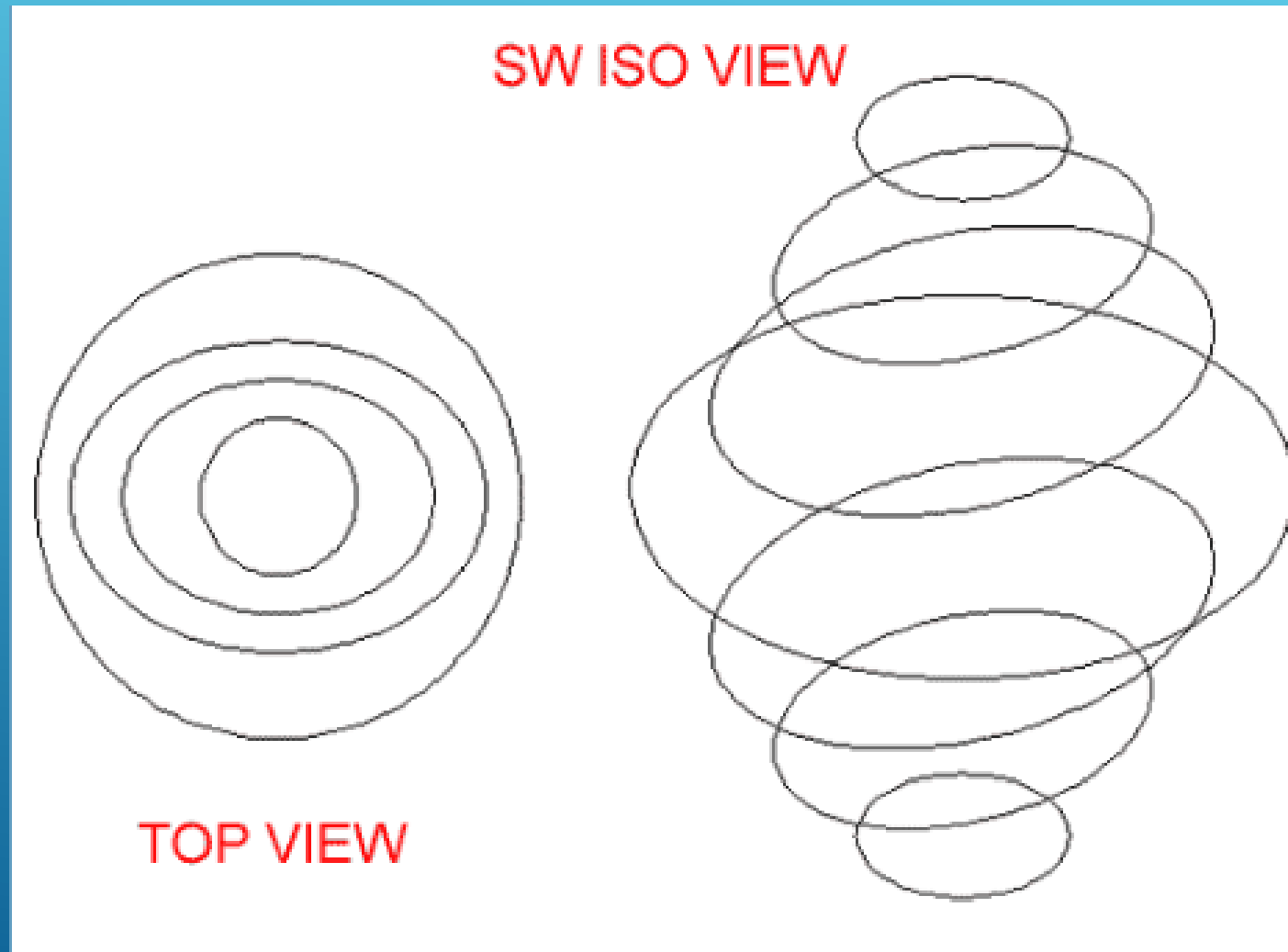
O comando **LOFT** é similar ao comando EXTRUDE mas é muito mais versátil: ao contrário do primeiro, onde a extrusão é efectuada sobre uma polyline fechada, neste caso a extrusão pode ser efectuada sobre diversas polylines fechadas e torna-las um objecto contínuo.

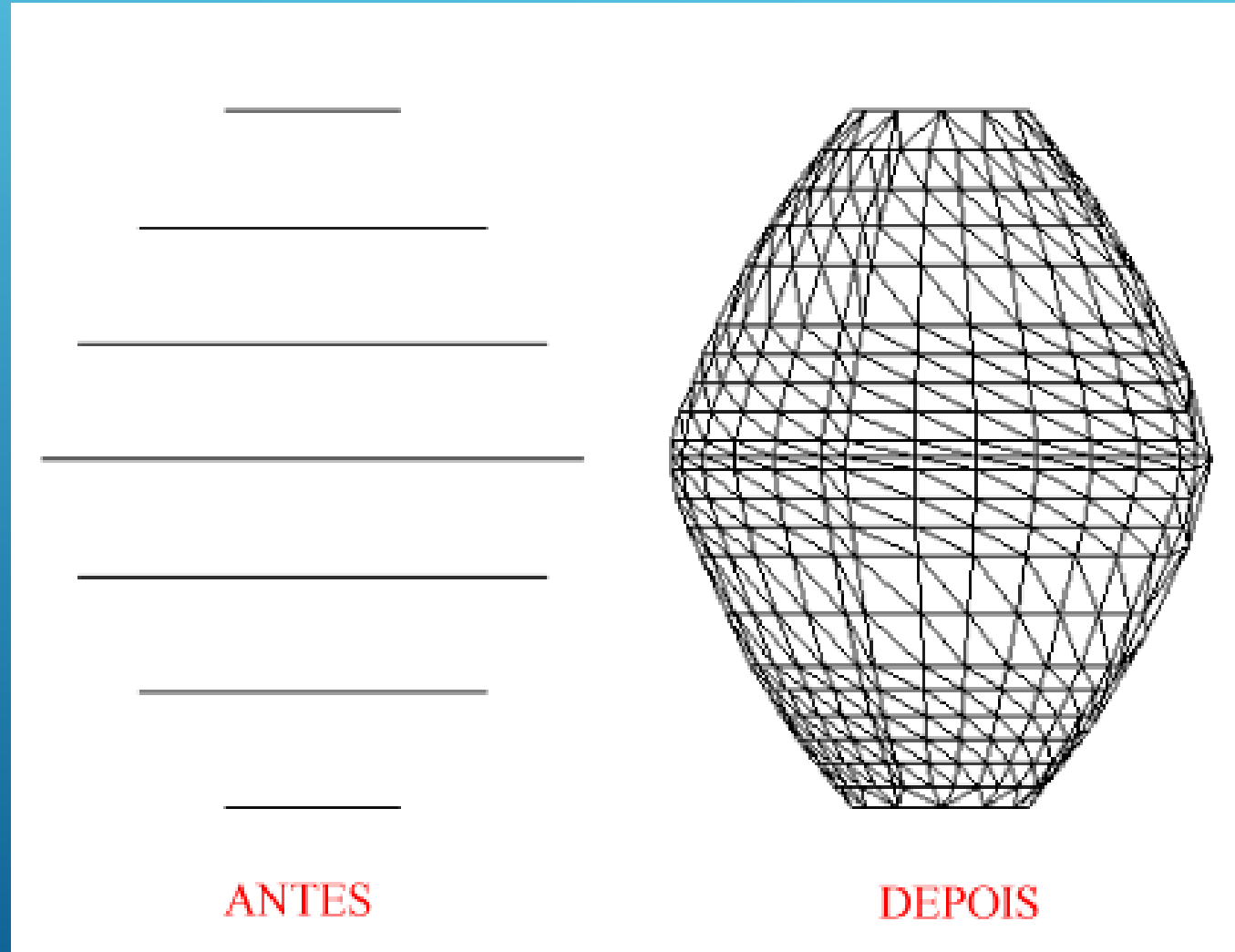
Loft

Creates a 3D solid or surface in the space between several cross sections

Loft cross sections can be open or closed, planar or non-planar, and can also be edge subobjects. Open cross sections create surfaces and closed cross sections create solids or surfaces, depending on the specified mode.







ANTES

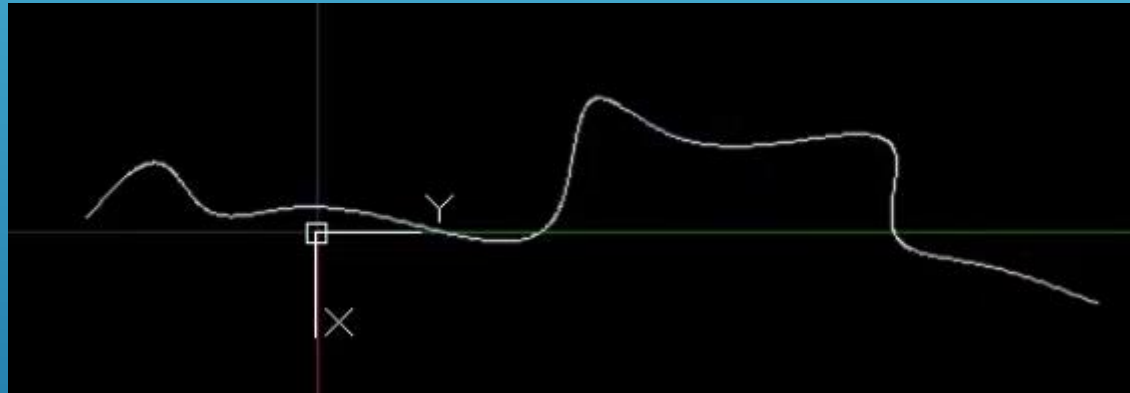
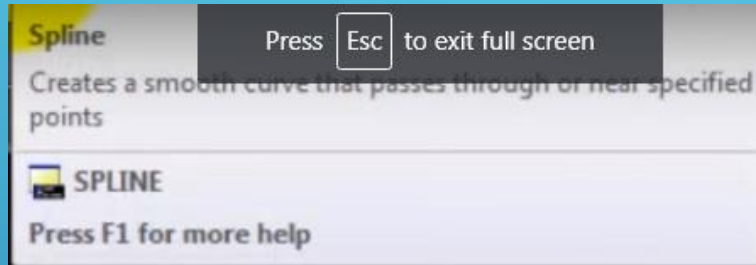
DEPOIS

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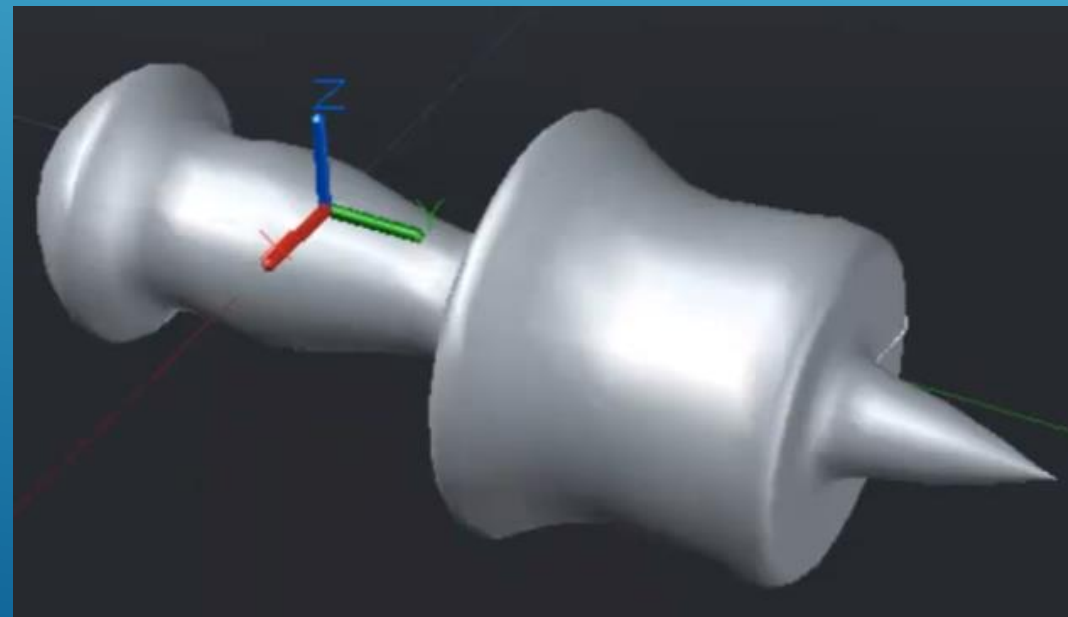
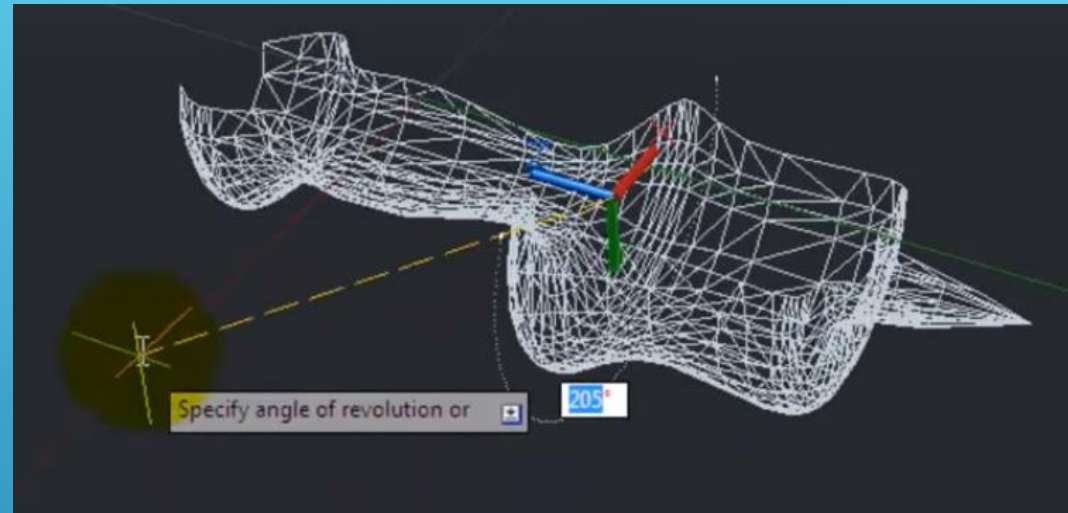
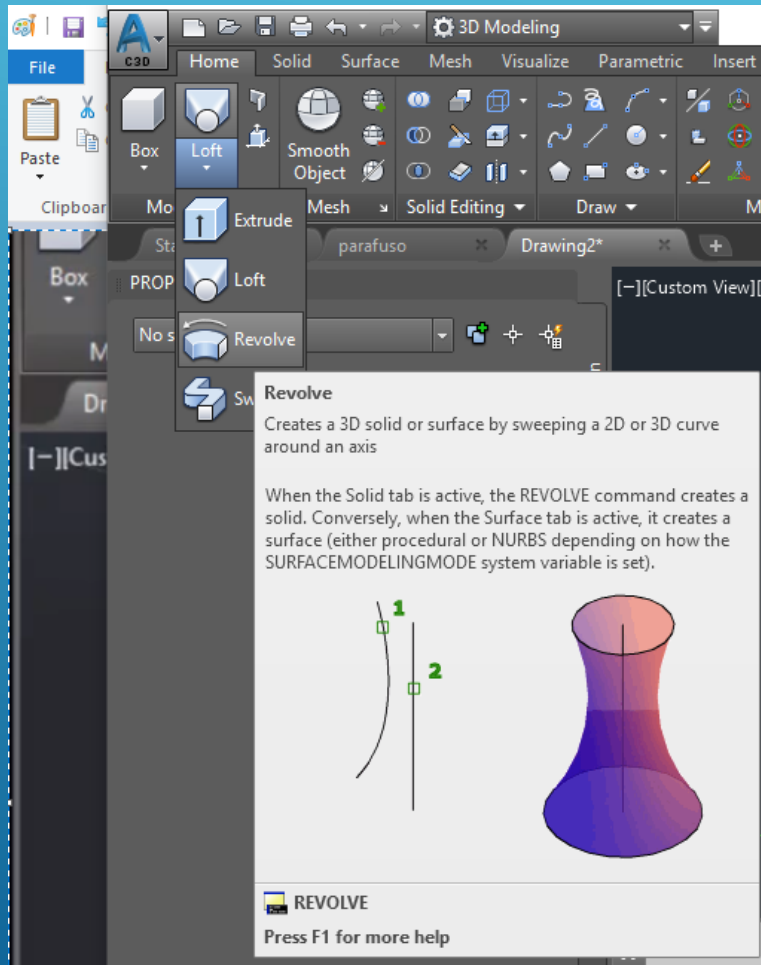


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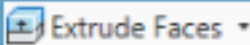


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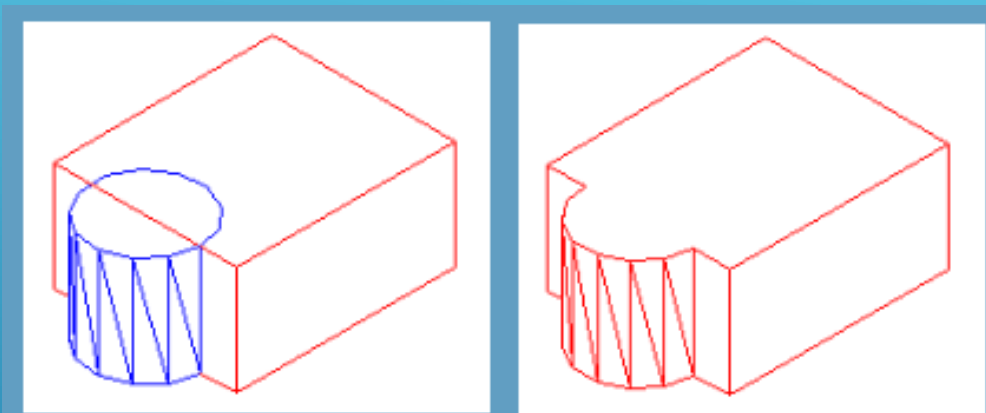


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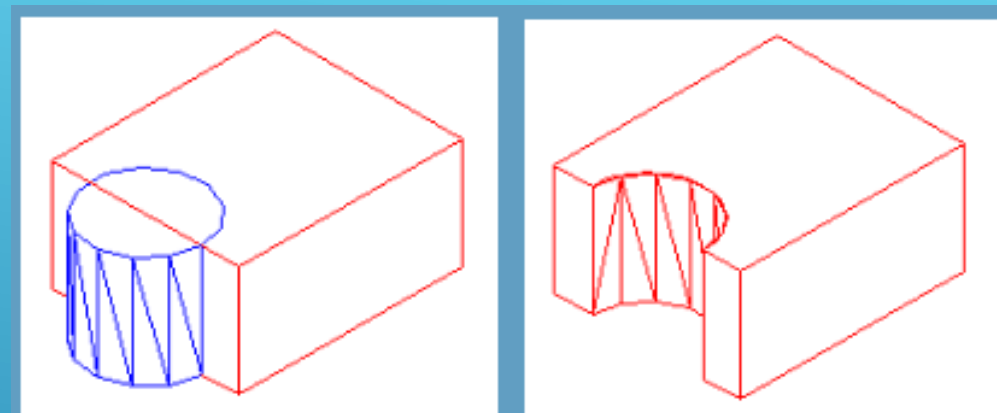


Boolean Operations

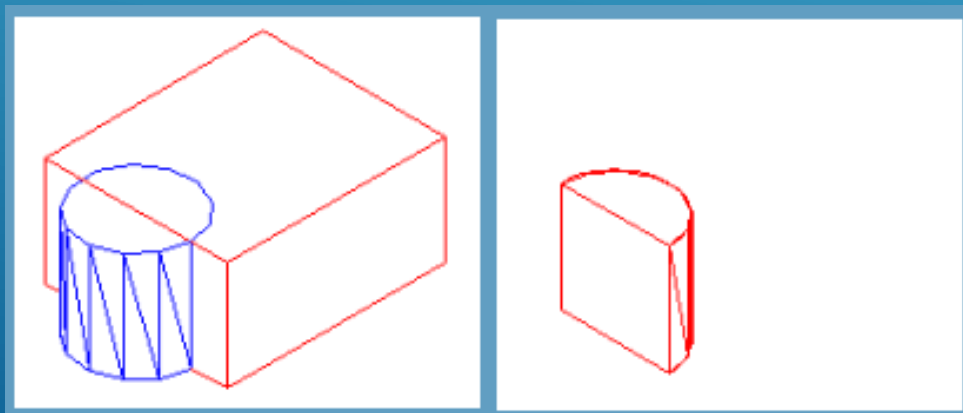
COMMAND	INPUT	ICON	DESCRIPTION
UNION (Boolean)	UNION / UNI		Joins two or more solids into creating one based on the total geometry of all.
SUBTRACT (Boolean)	SUBTRACT / SU		Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT (Boolean)	INTERSECT / IN		Creates a single solid from one more solids based on the intersected geometry.
EXTRUDE FACE	SOLIDEDIT		Allows you to increase the size of a solid by extruding out one of its faces.
SLICE	SLICE		Slices a solid along a cutting plane.
3D ALIGN	3DALIGN		Aligns 2 3D Objects in 3D space.



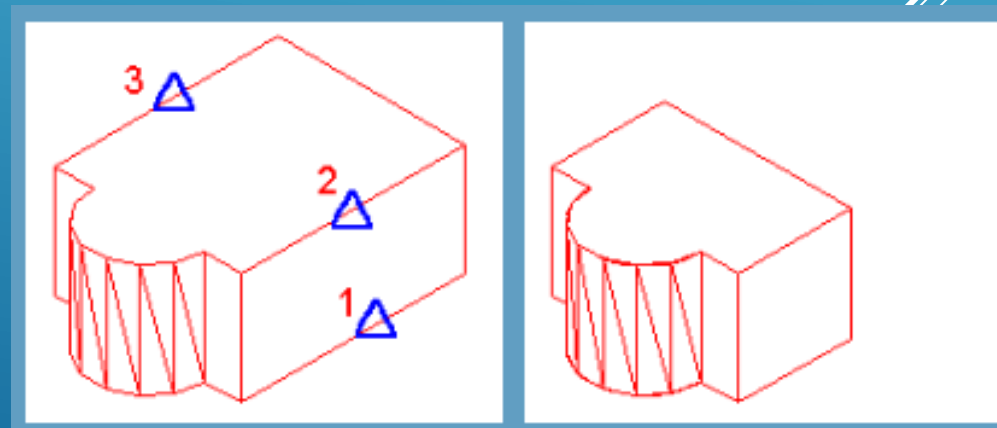
Union



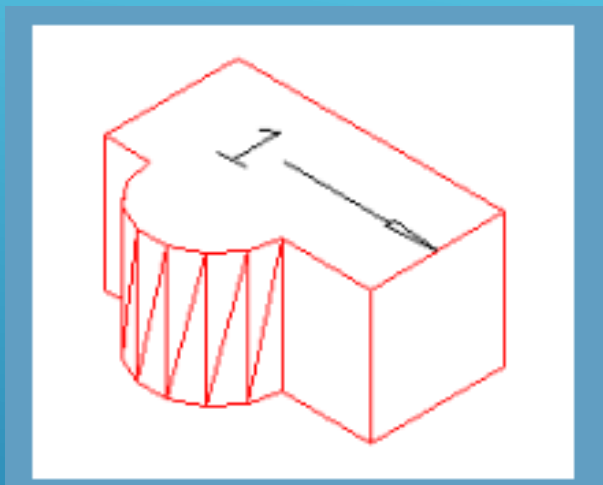
Subtract



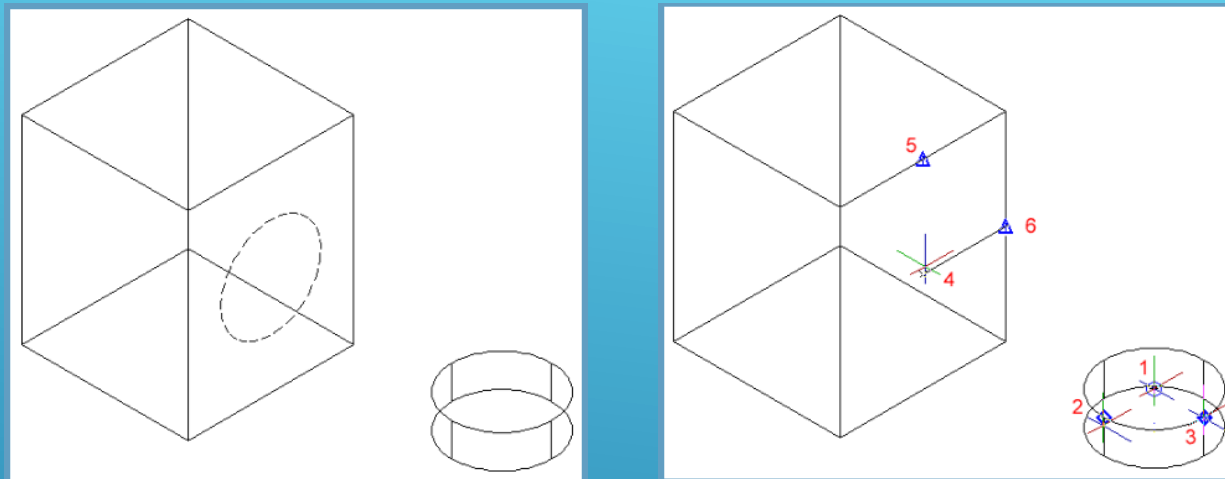
Intersect



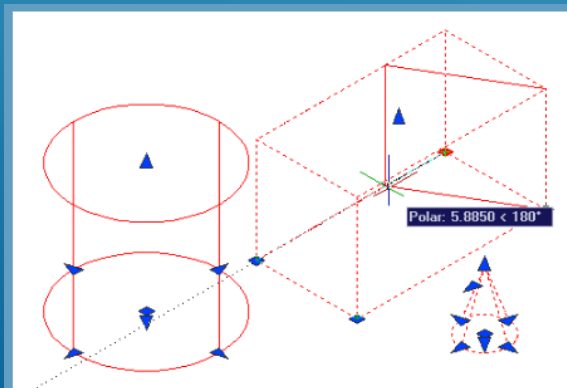
Slice



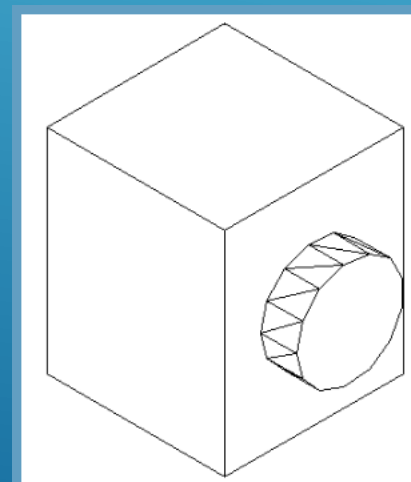
Extrude face

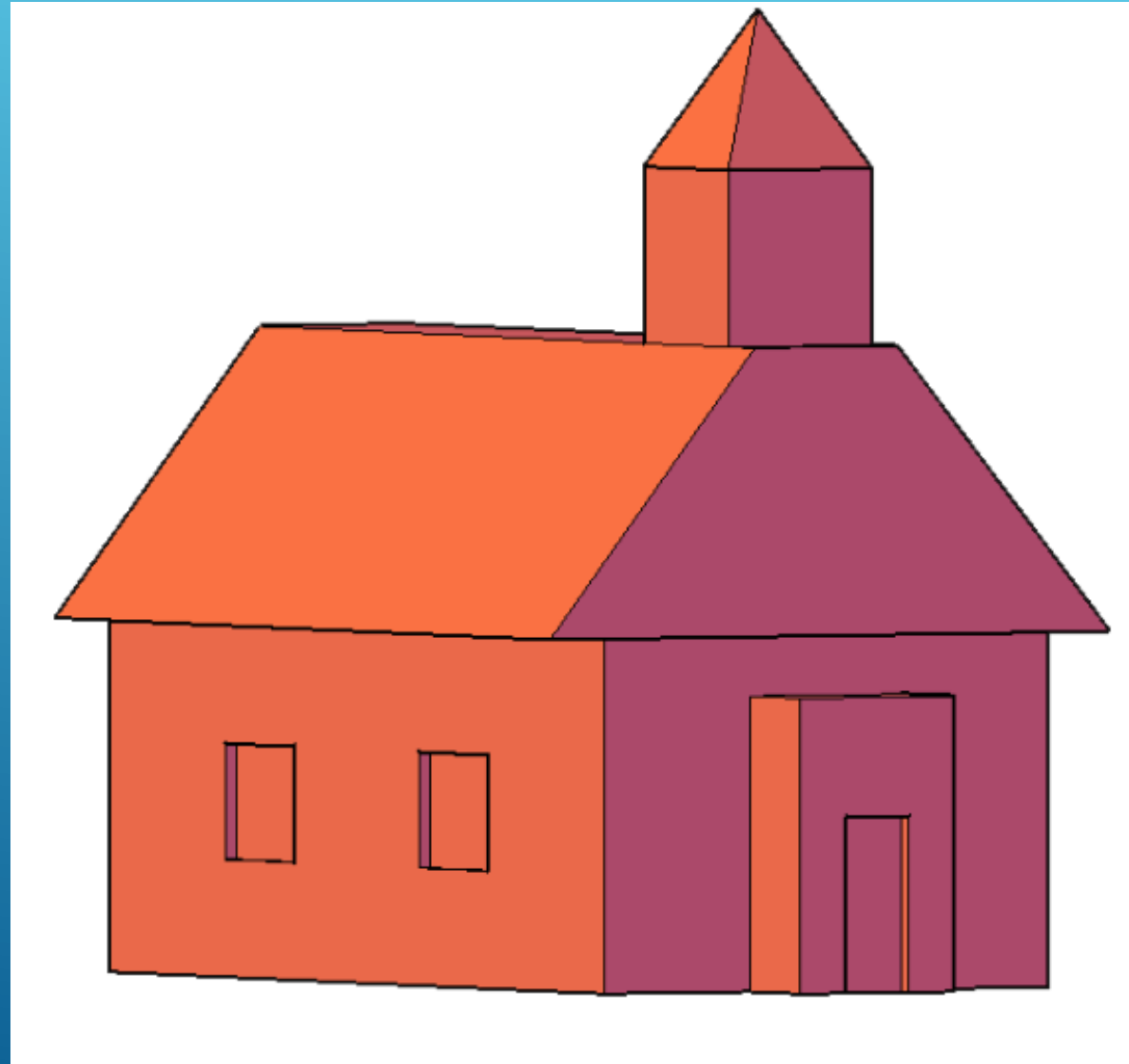


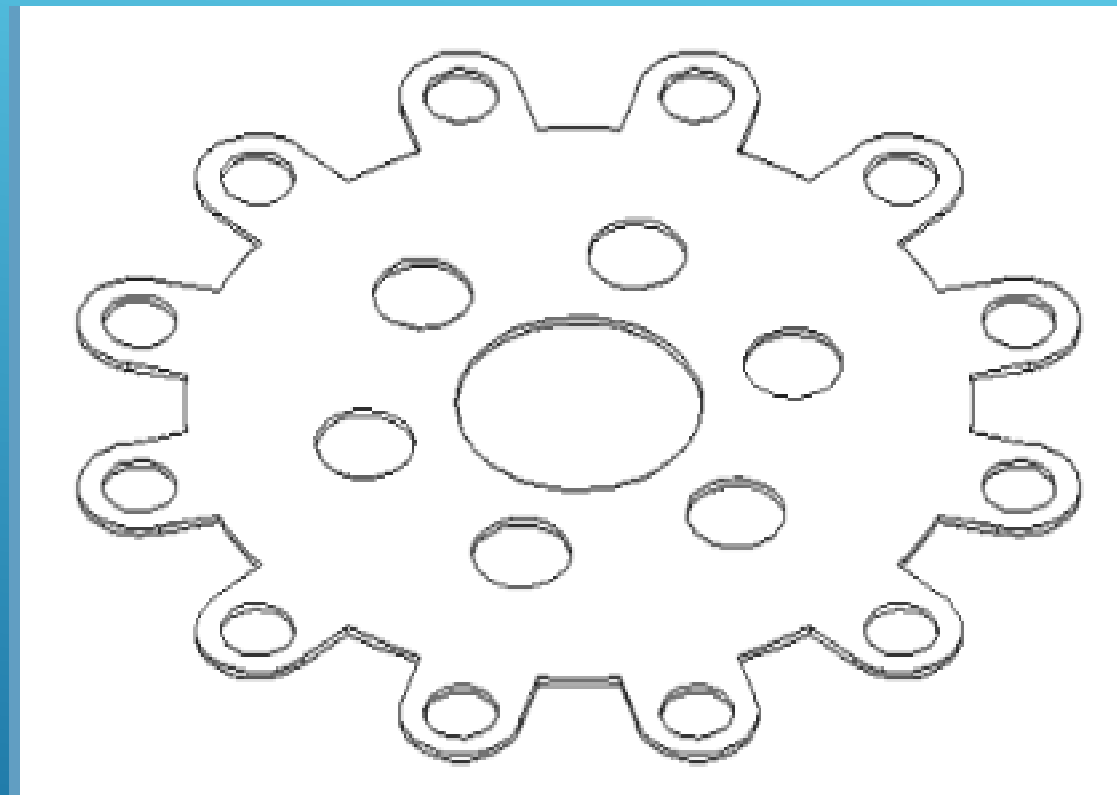
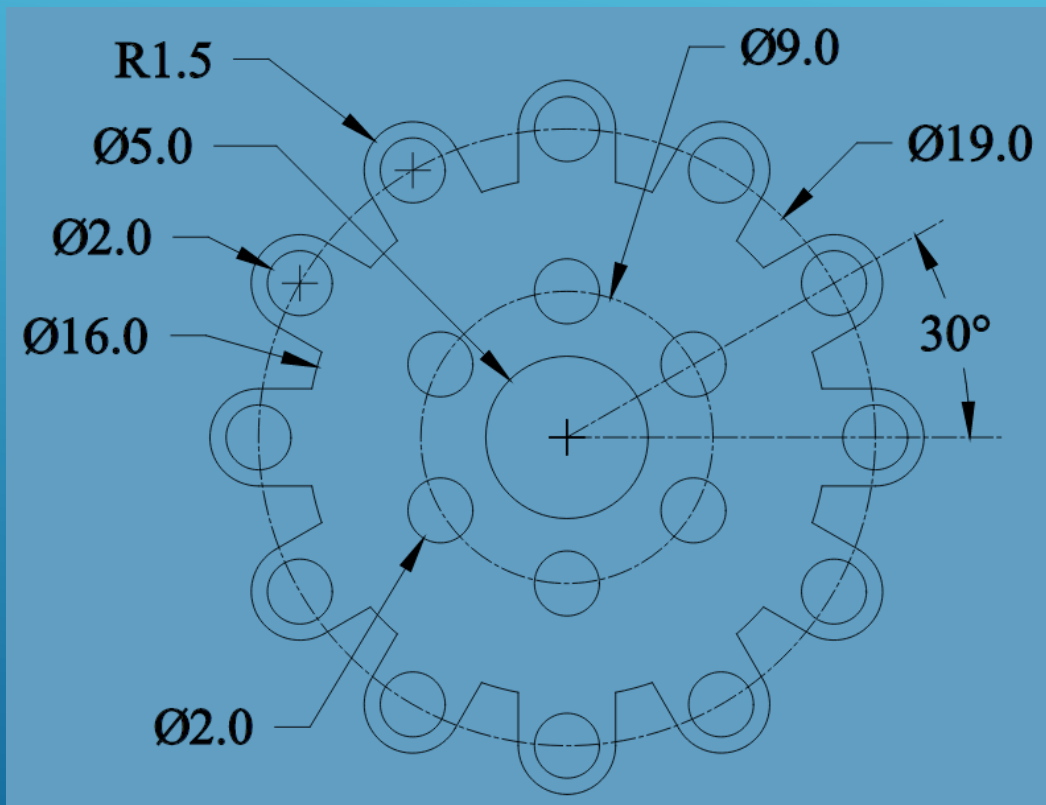
3D Align



Extrude using grips





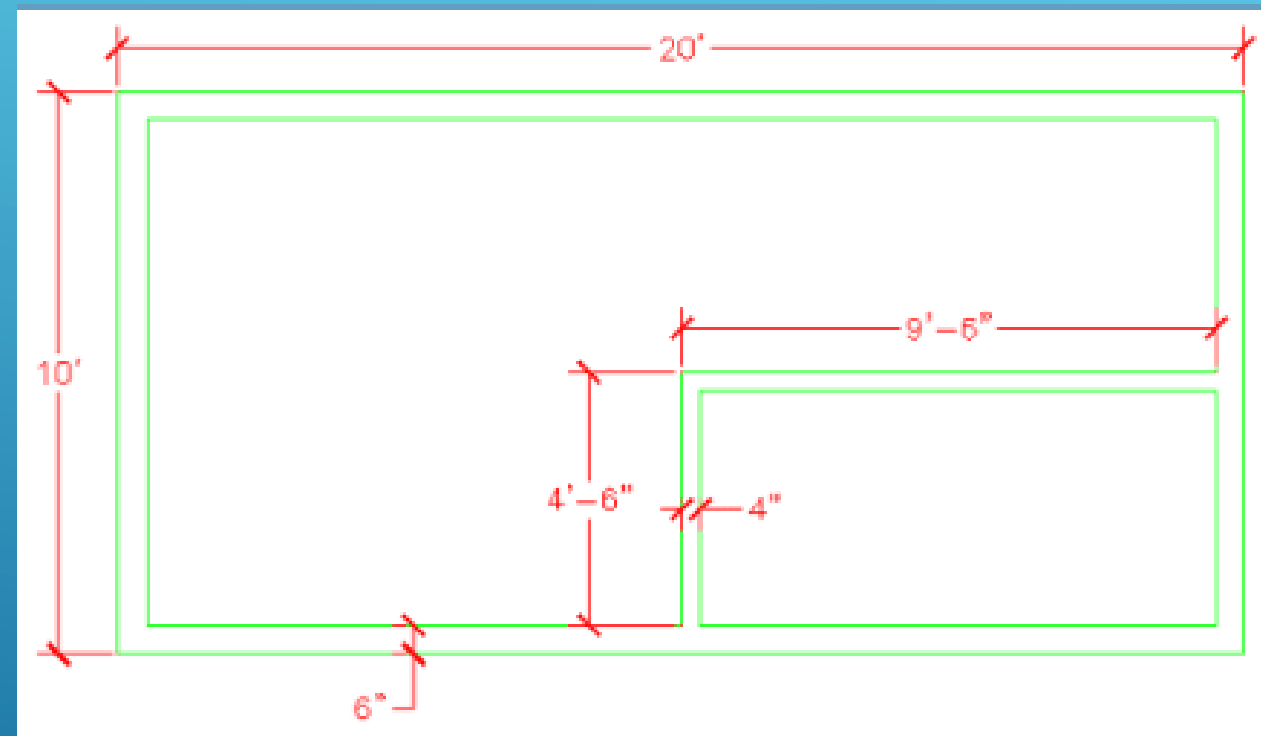


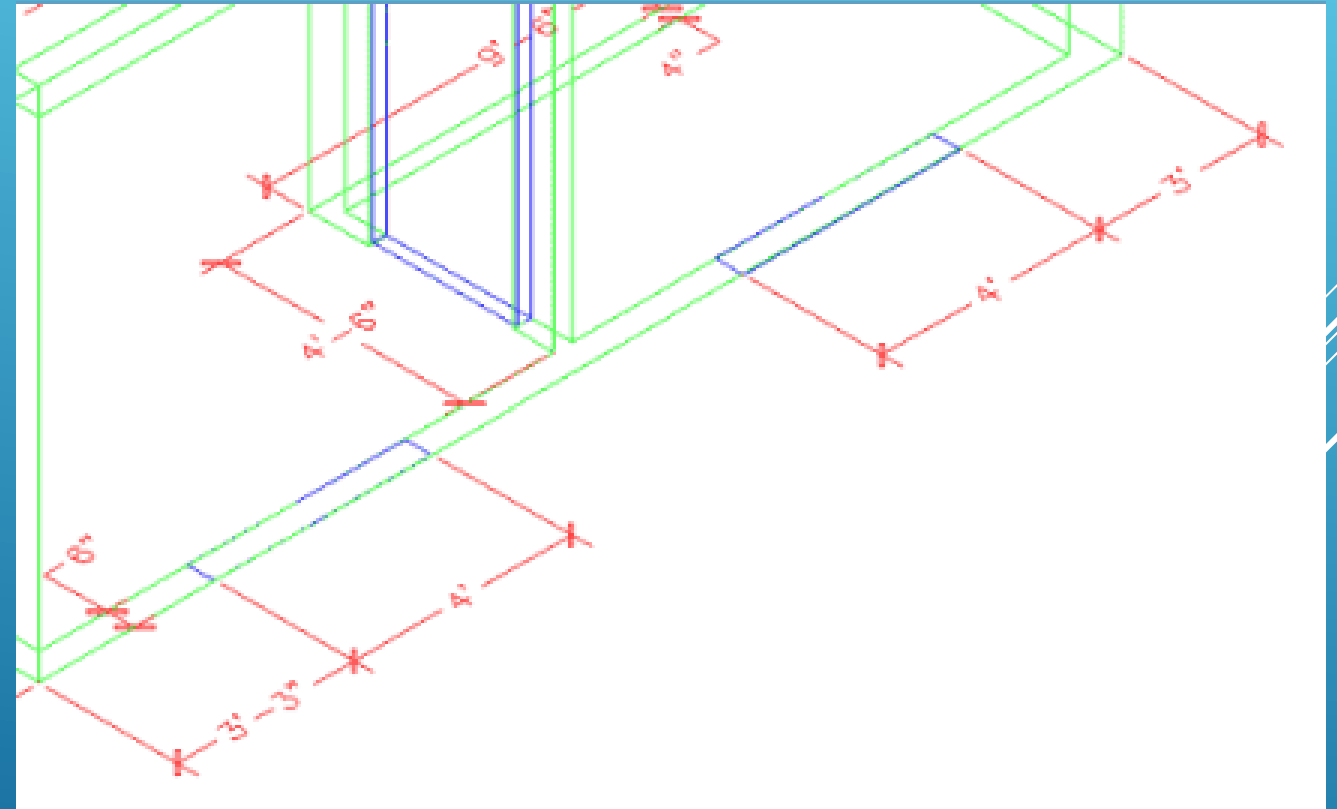
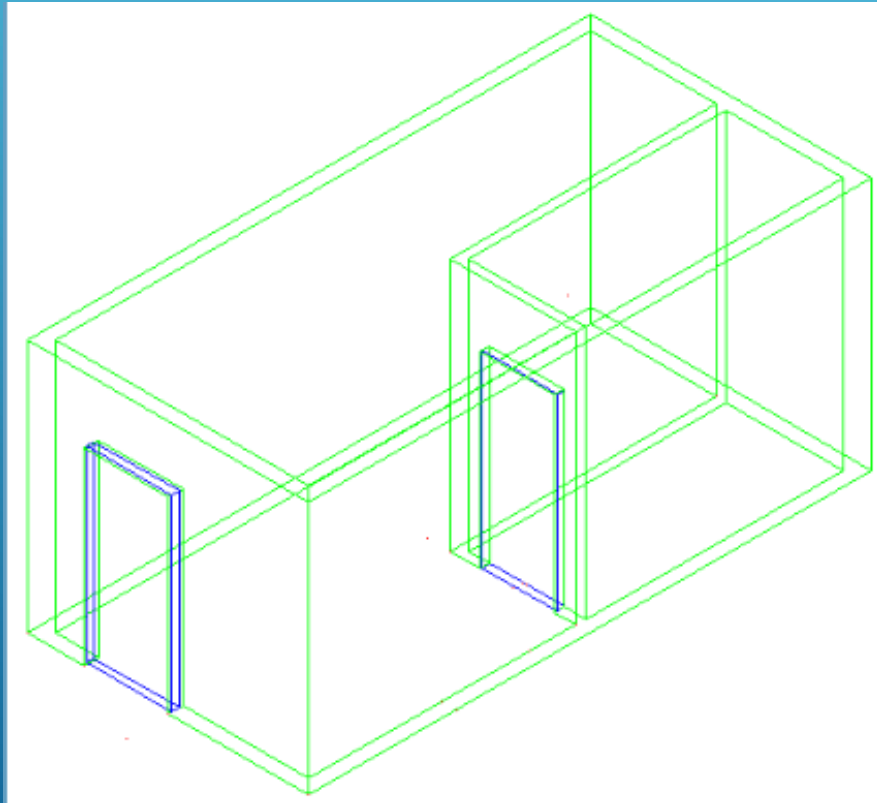
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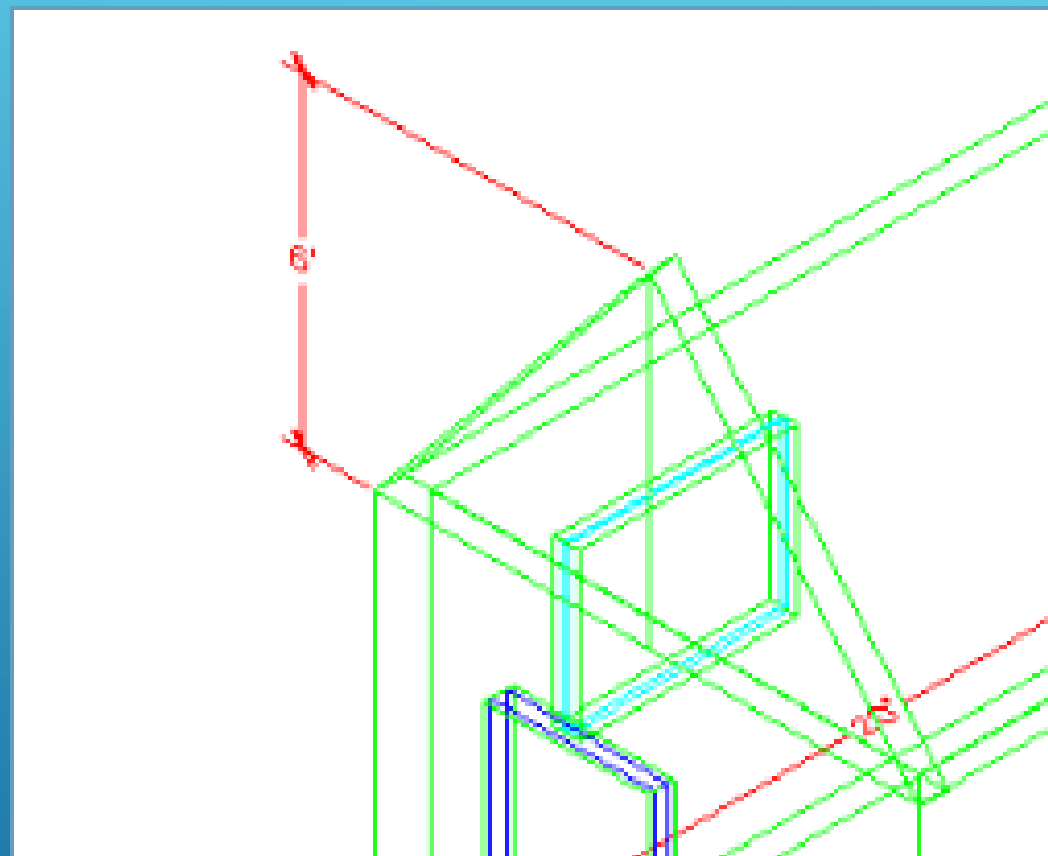
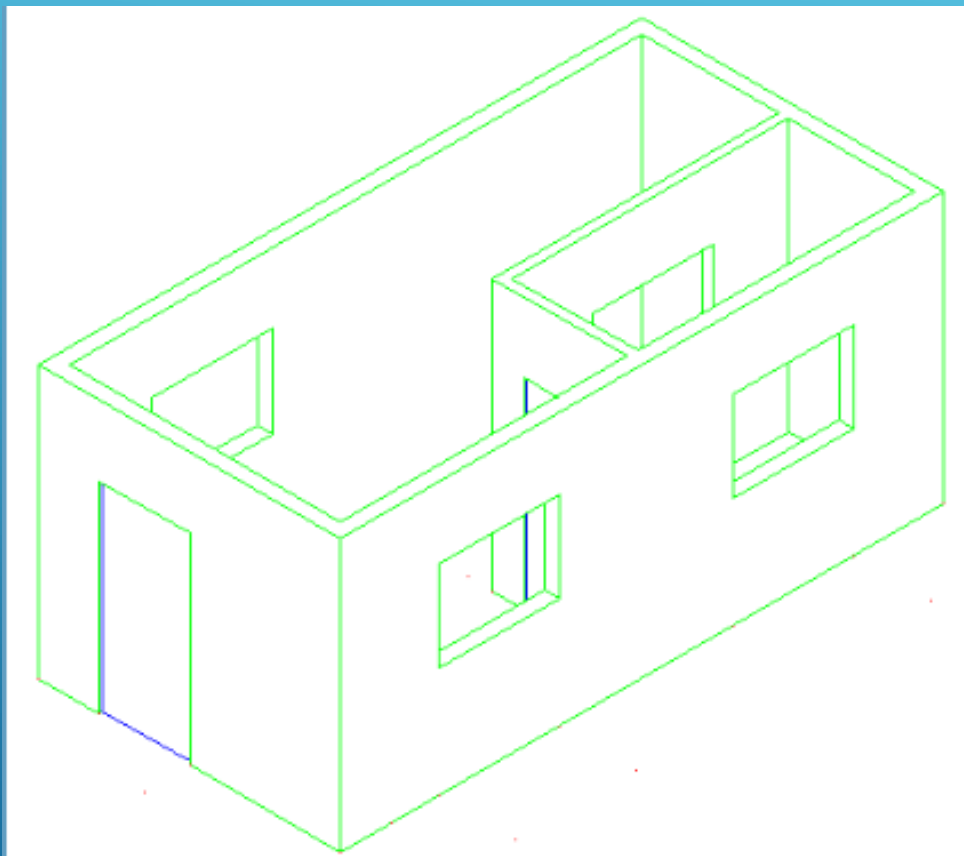


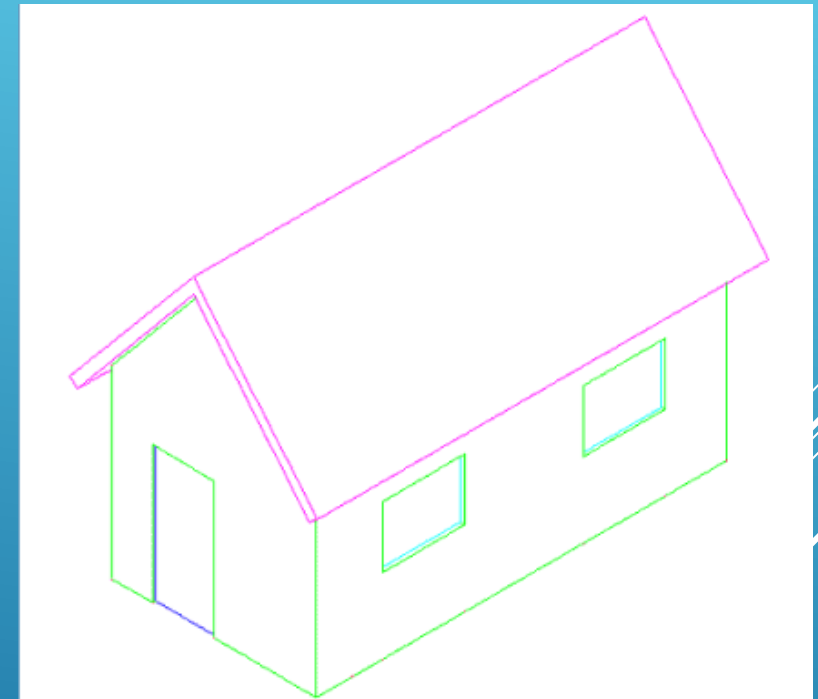
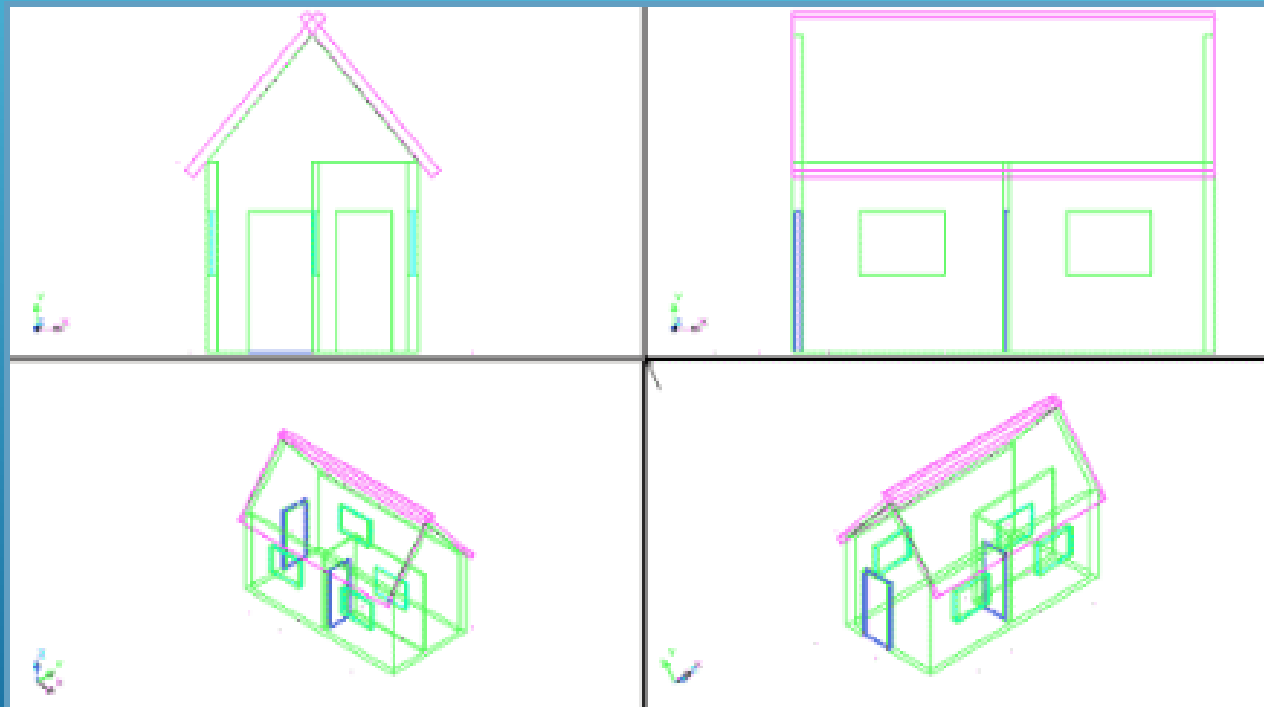
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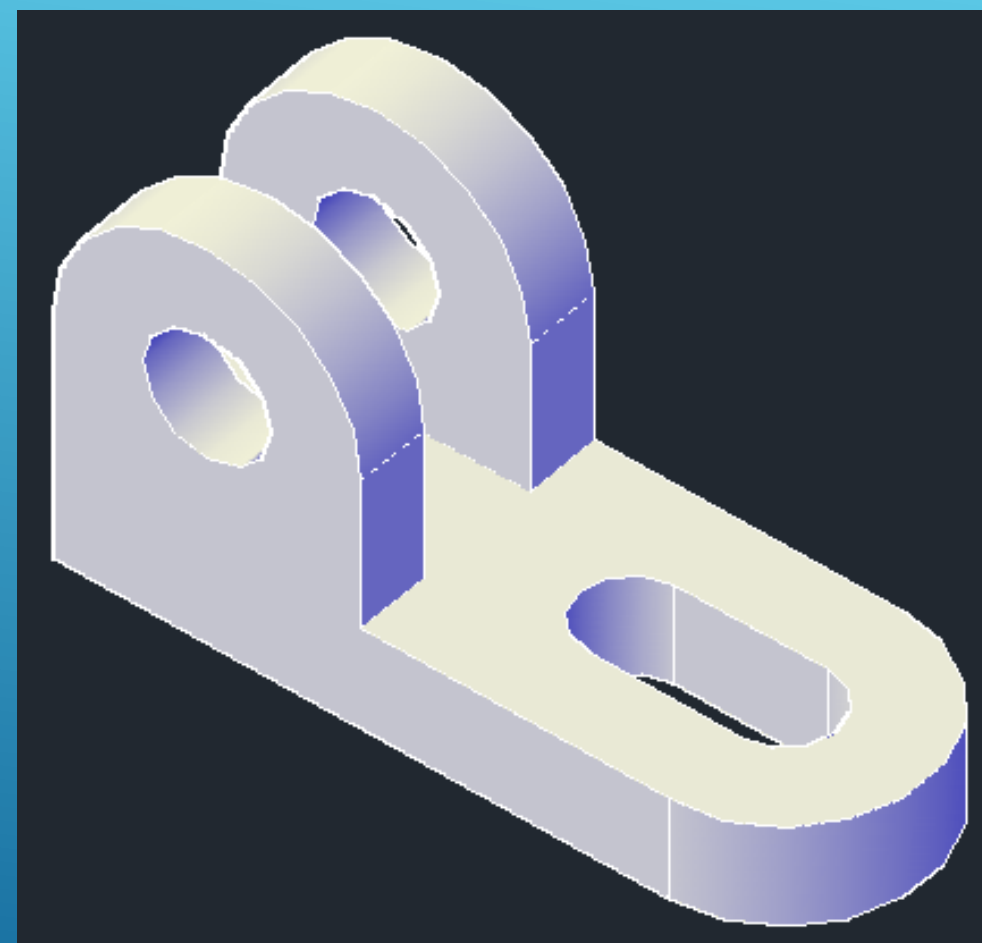
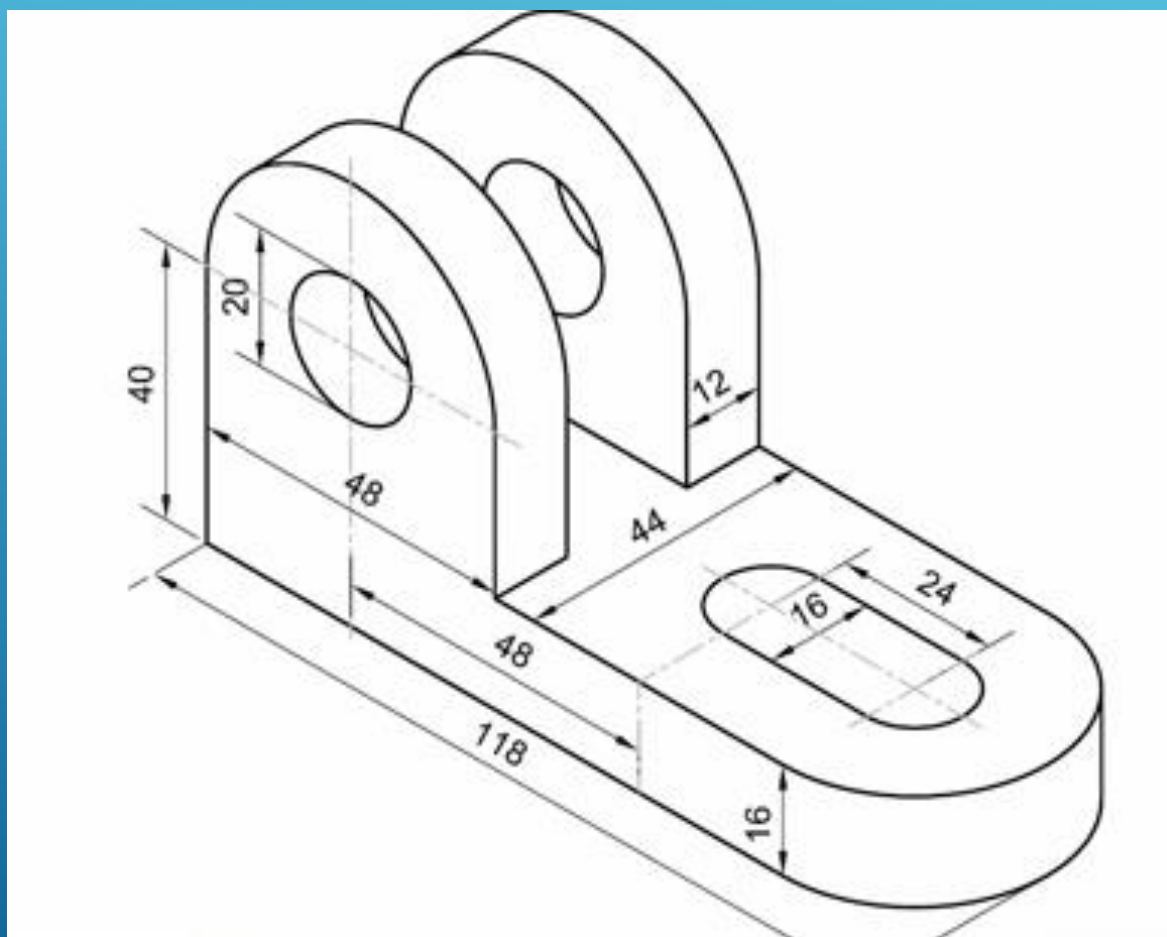
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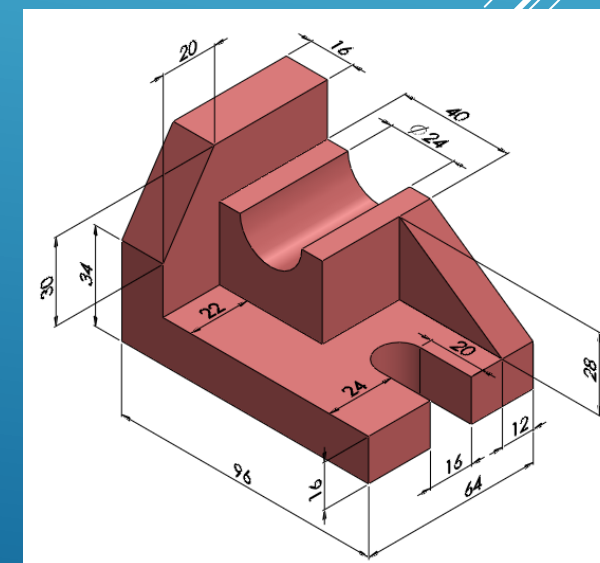
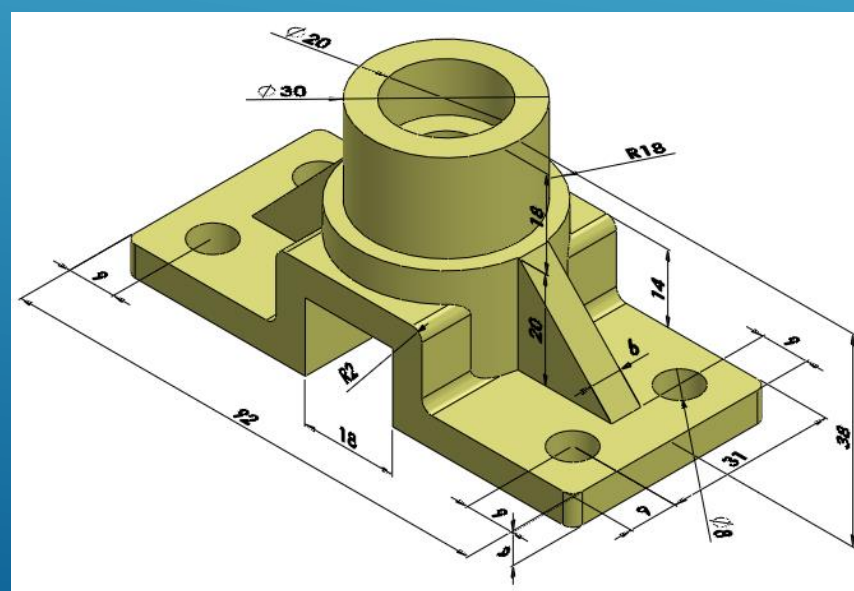
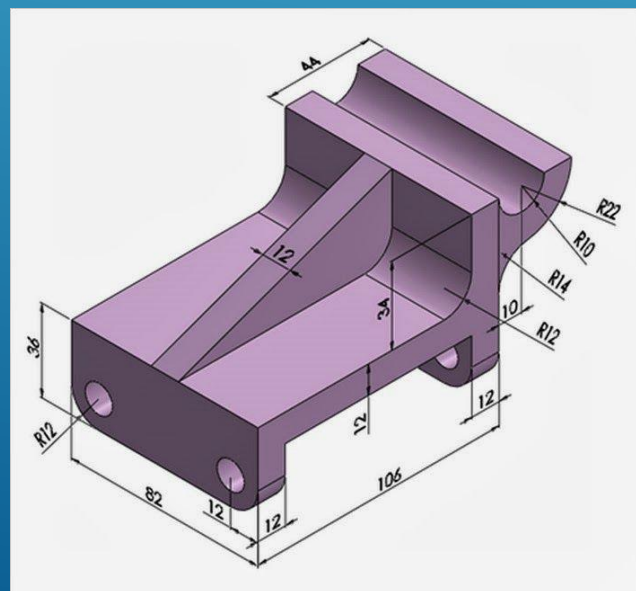
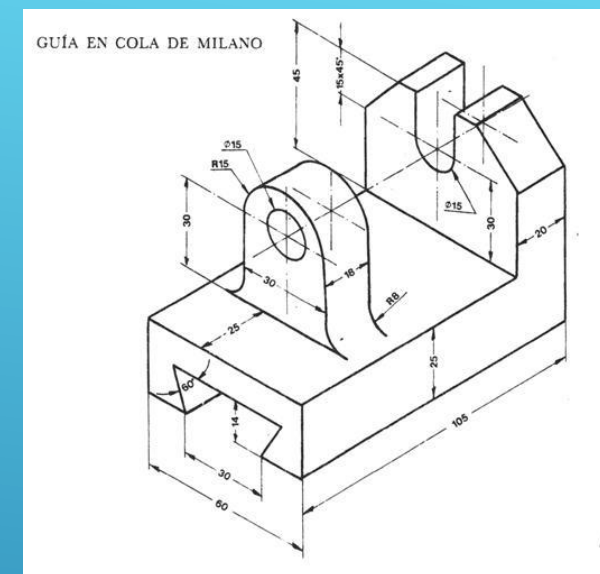
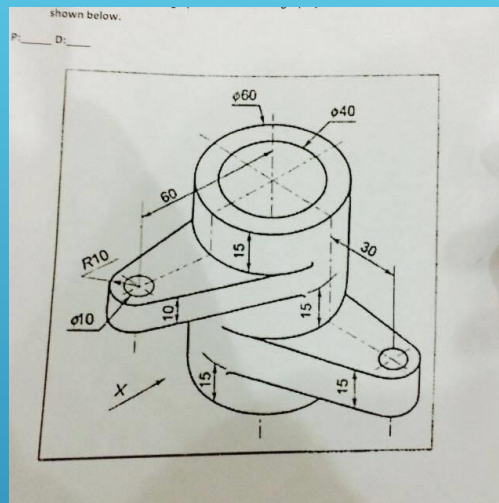
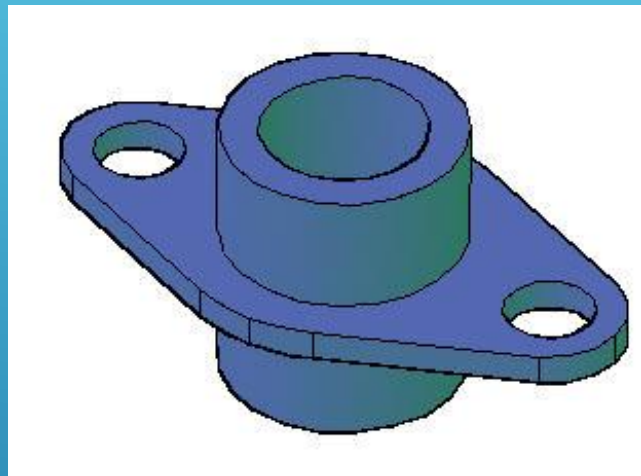


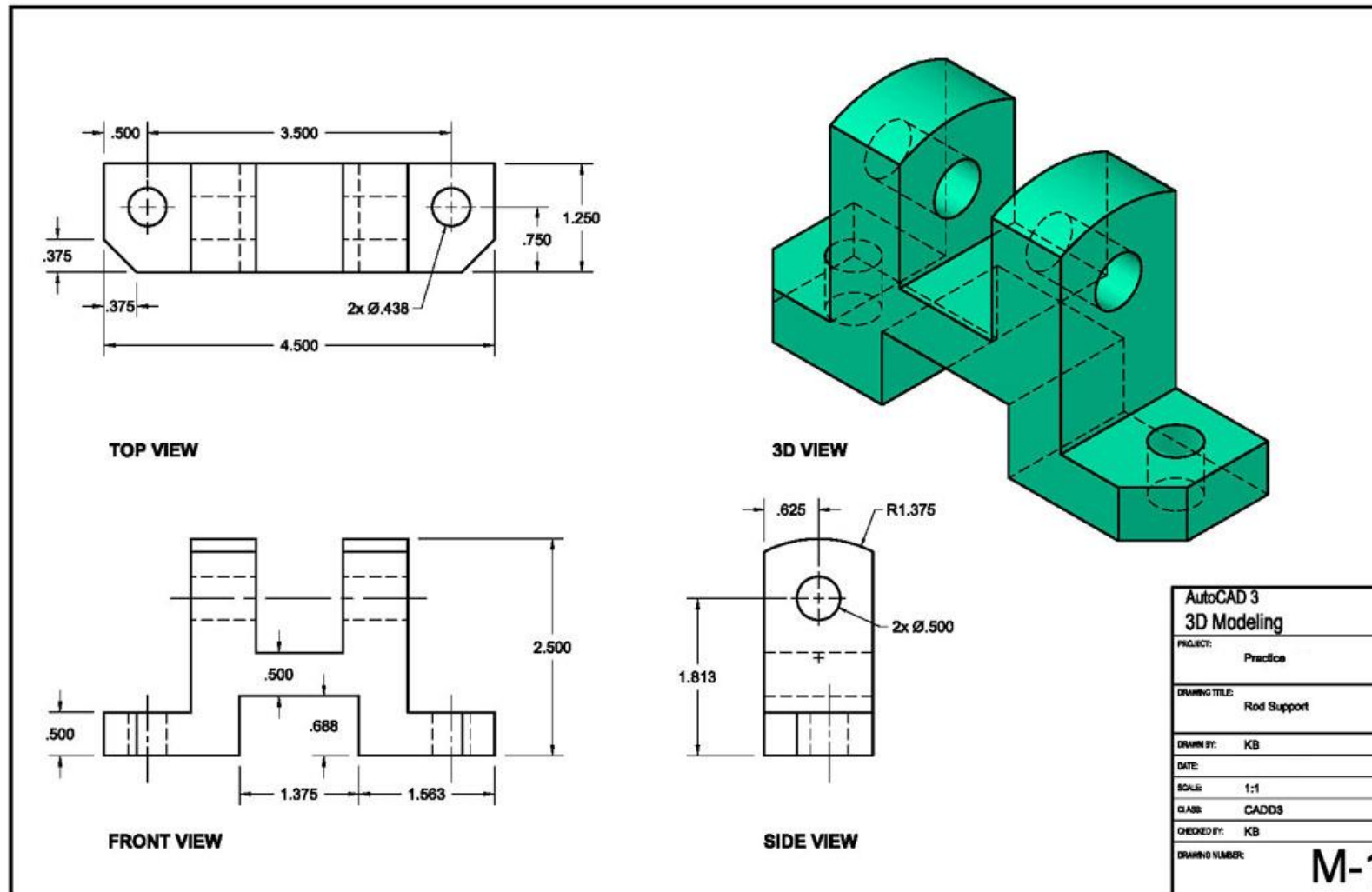




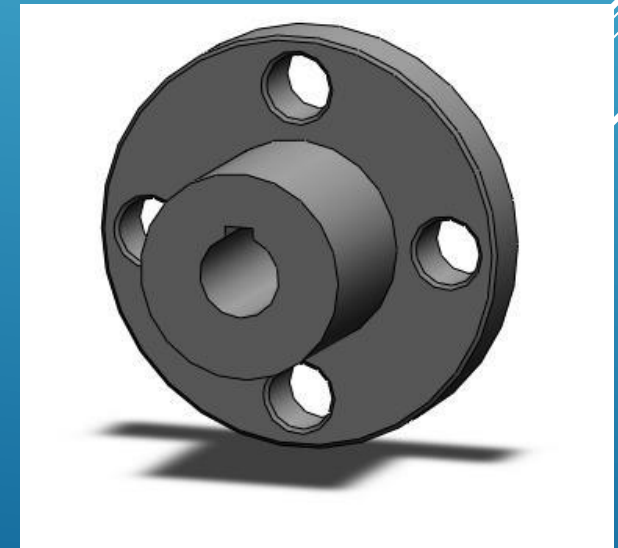
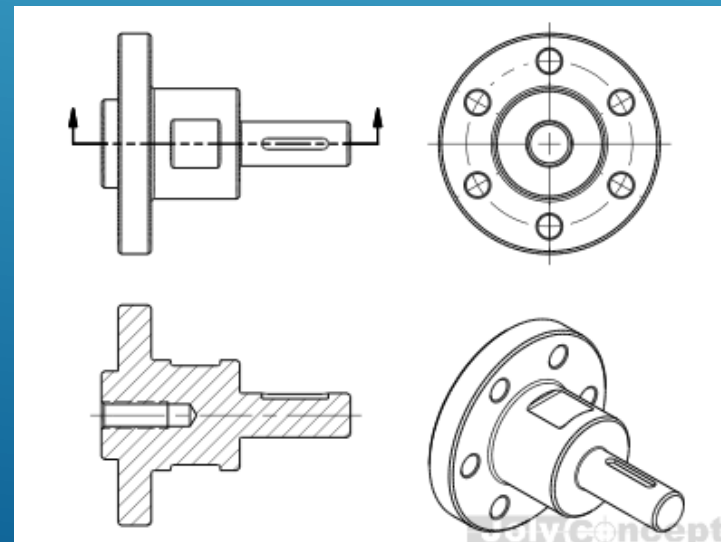
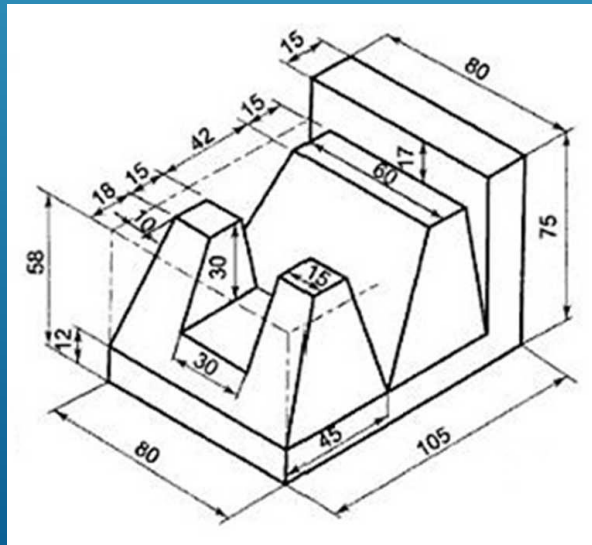
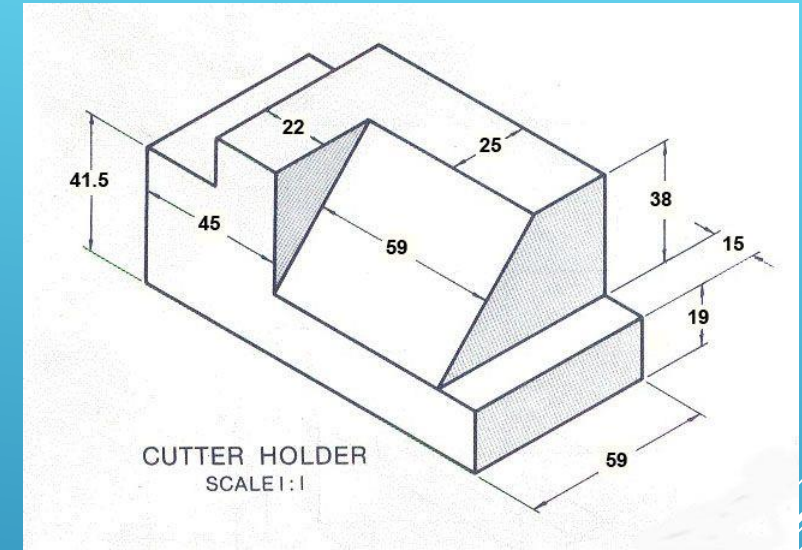
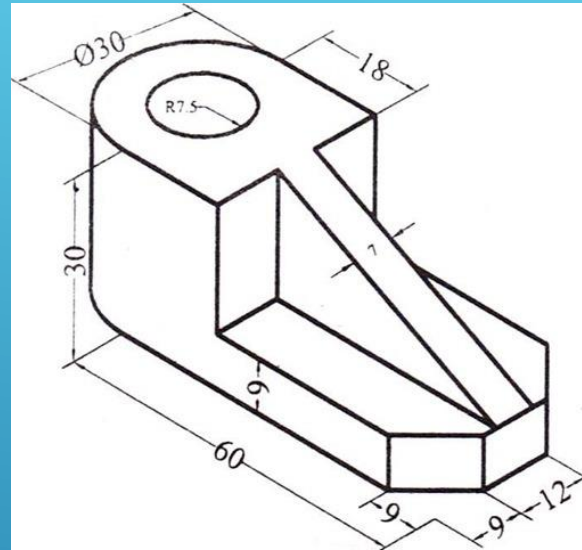
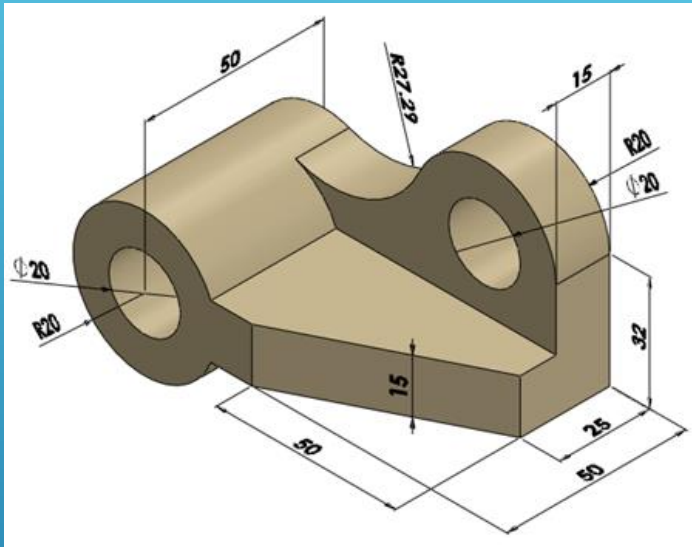


AULA 6 Desenho Técnico Assistido por Computador





AULA 6 Desenho Técnico Assistido por Computador



AULA 6 Desenho Técnico Assistido por Computador

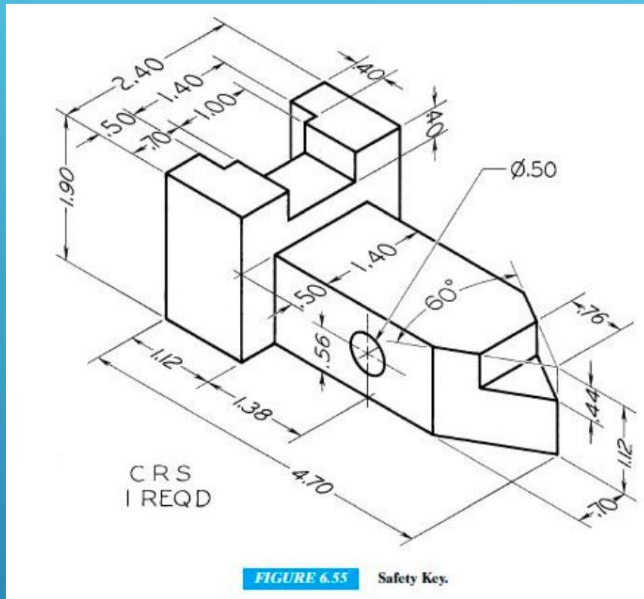
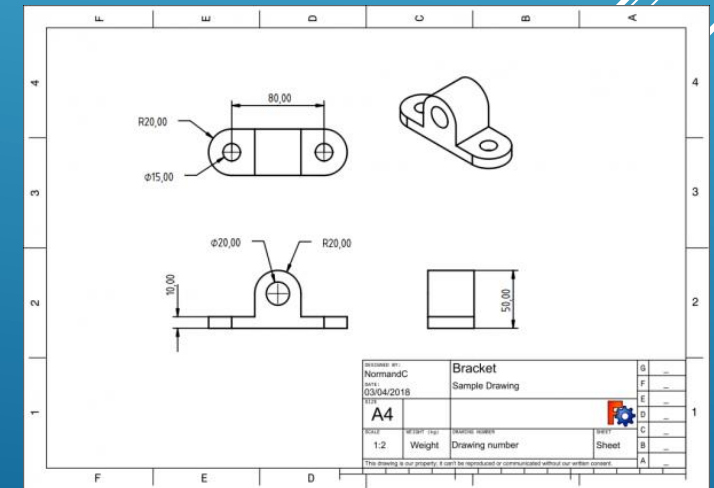
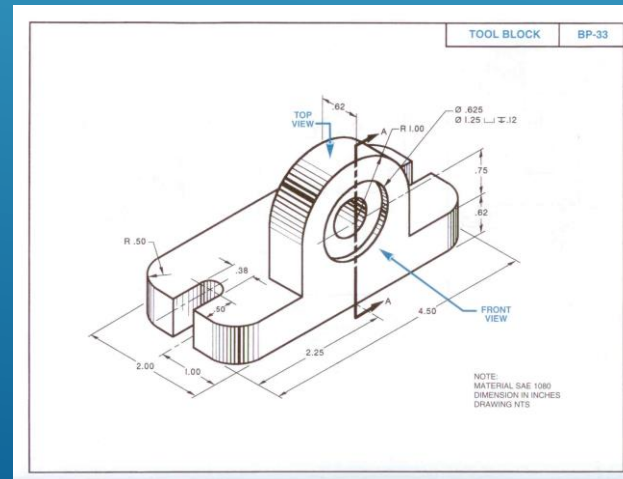
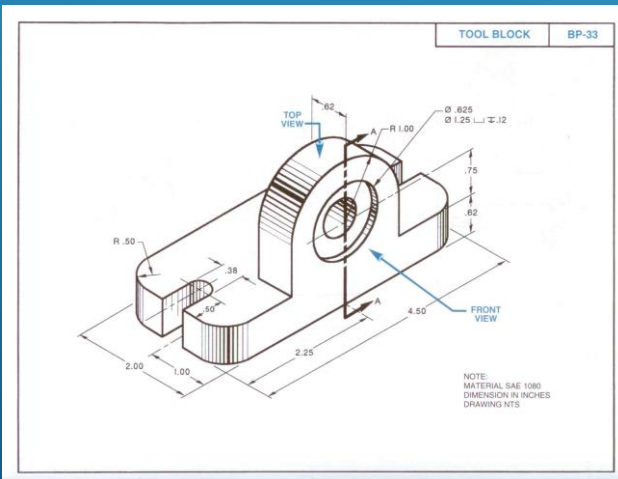
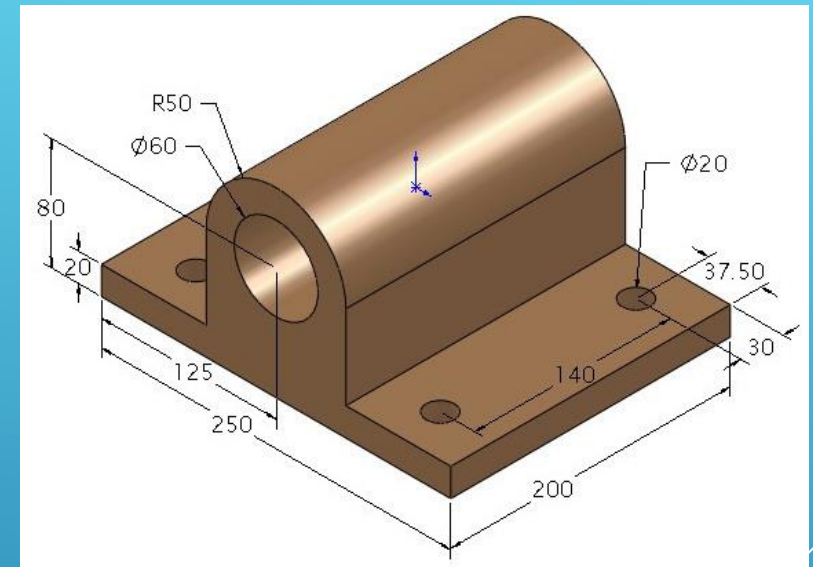
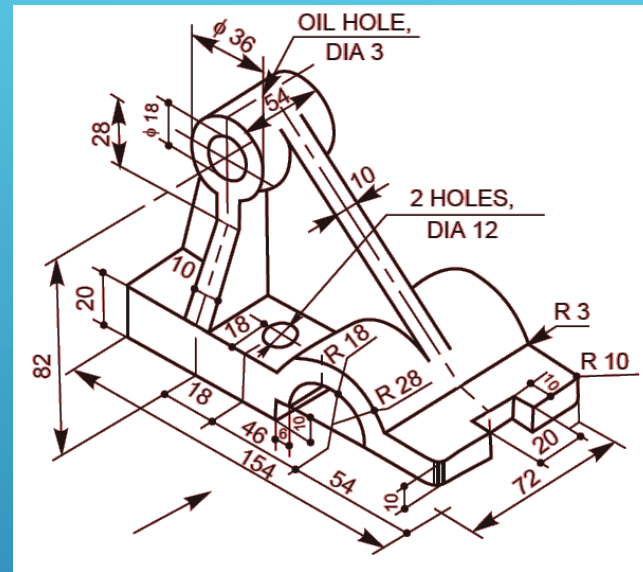
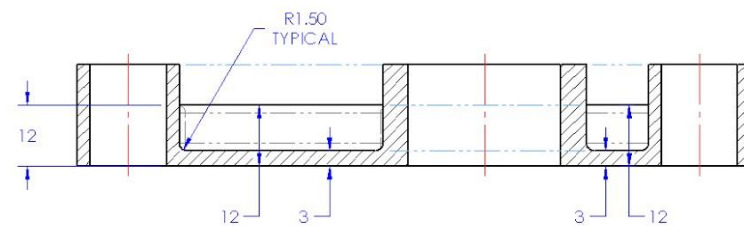
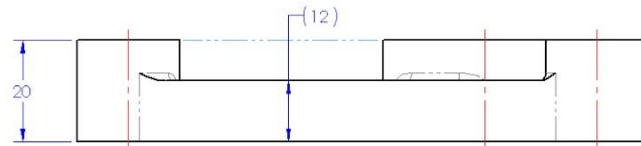
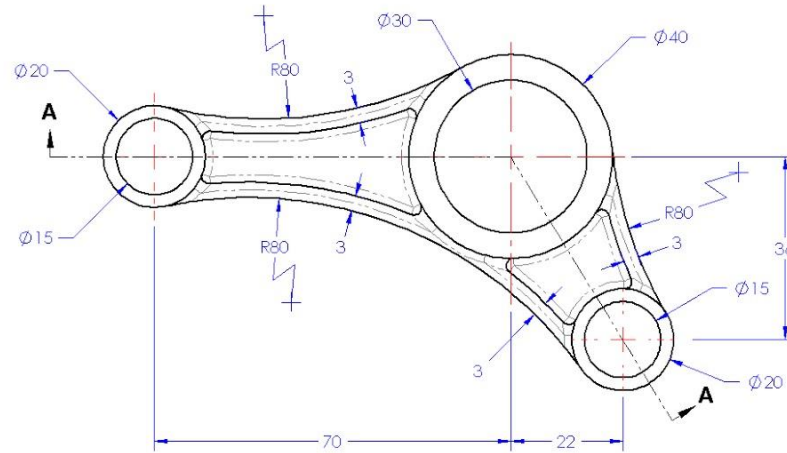
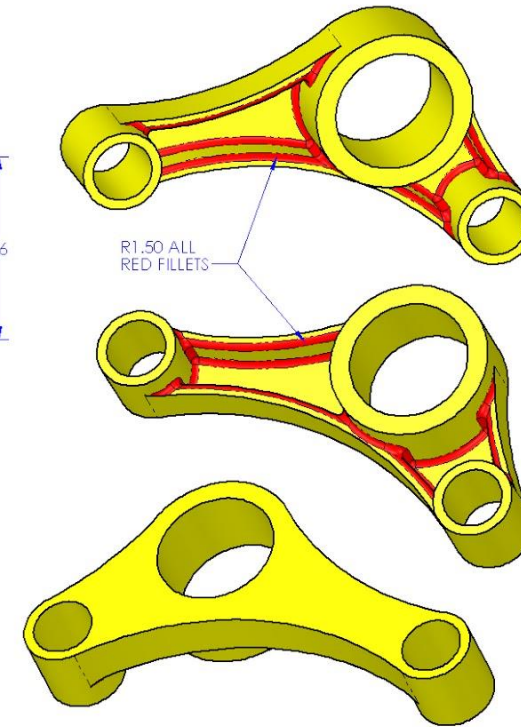


FIGURE 6.55 Safety Key.





SECTION A-A



- NOTES:
1. ALL DIMENSIONS ARE IN MILLIMETERS
 2. MATERIAL: PLAIN CARBON STEEL

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NAME	DATE		
DRAWN BY: IRAS	12/09/12		
MATERIAL: POIR-CODON 5881	SCALE: 1:1	WEIGHT:	SHEET 1 OF 1

MODEL MANIA 2013
Phase 1

AULA 6 Desenho Técnico Assistido por Computador

Figura 9.74_Mensula de contra cojinete - Drawing *

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Figura 9.74_Mensula de contra cojinete		1

SOLIDWORKS Premium 2016 x64 Edition | 7.1mm | 193.27mm | 0mm | Fully Defined | Editing Drawing | 1:1.5 | MMGS

AULA 6 Desenho Técnico Assistido por Computador

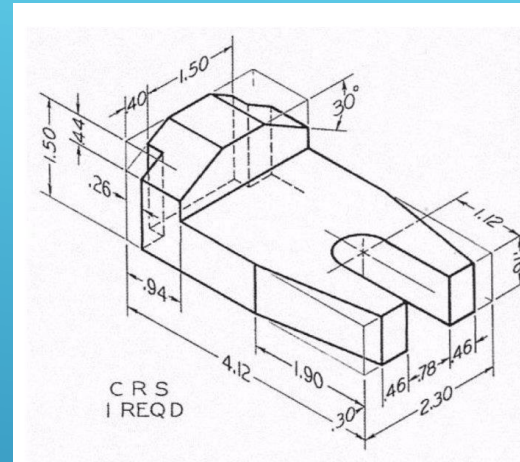
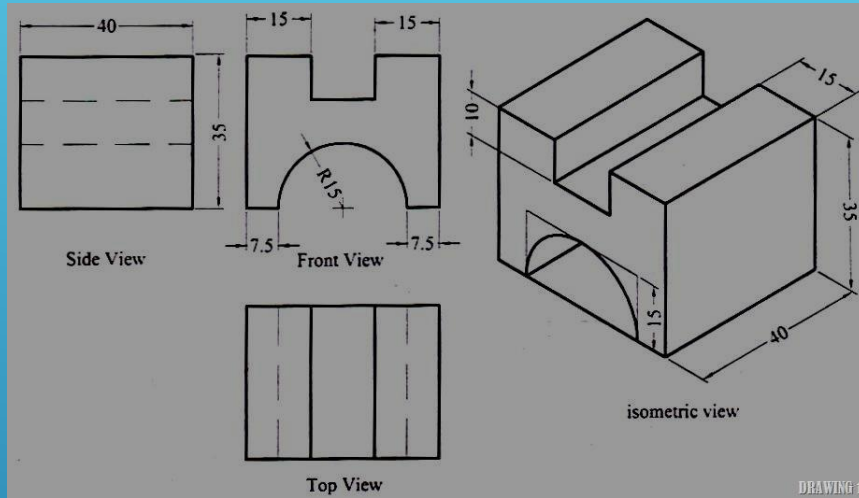
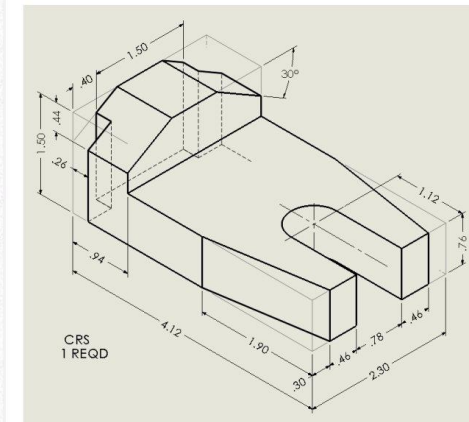


Fig. 6.55 Finger Guide



Nailed it.

