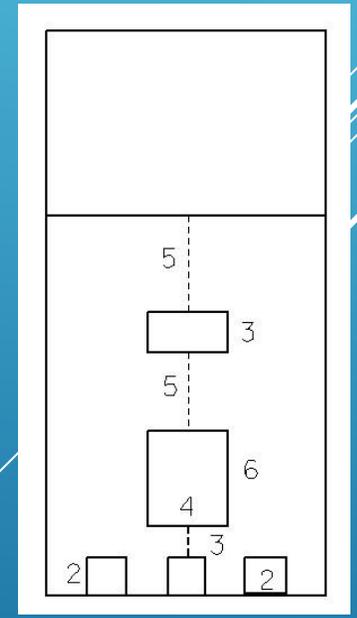
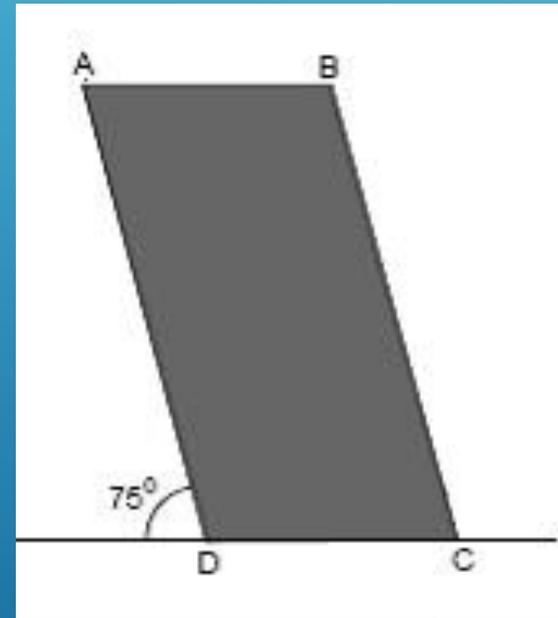
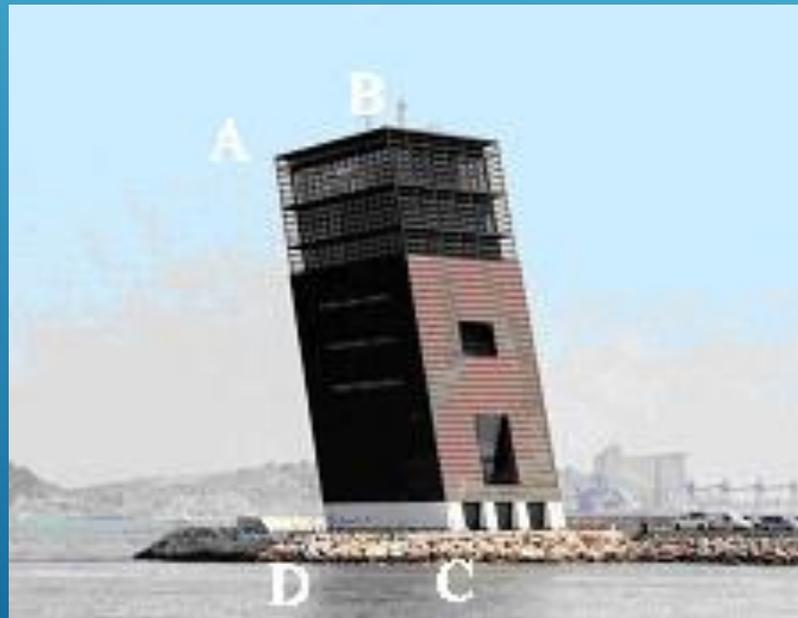
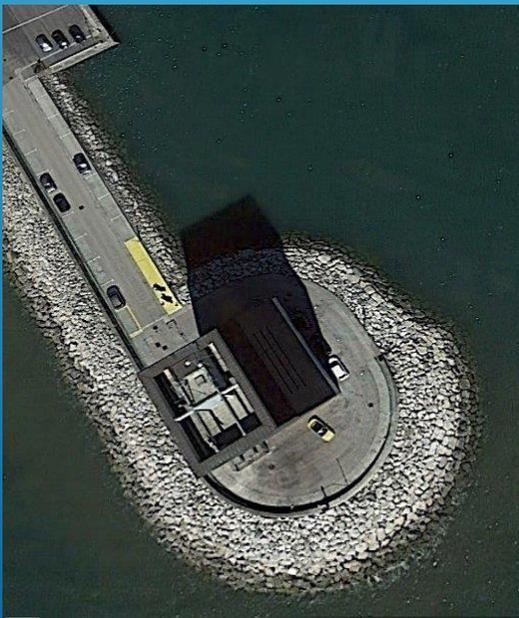
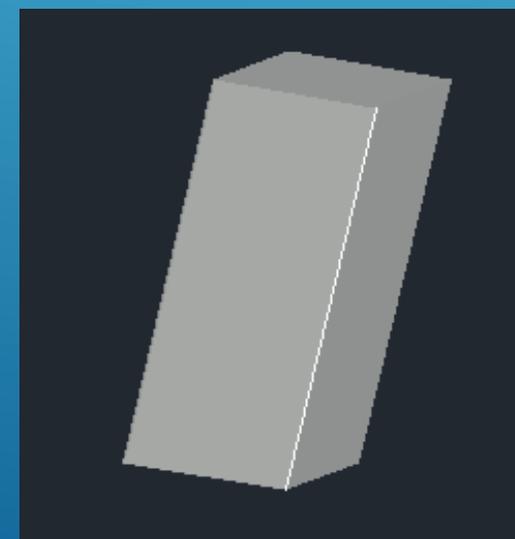
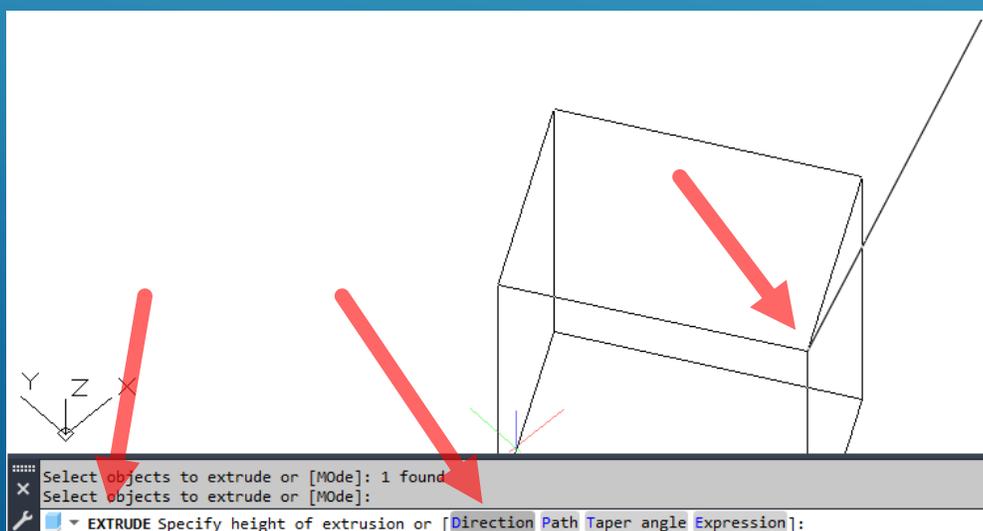
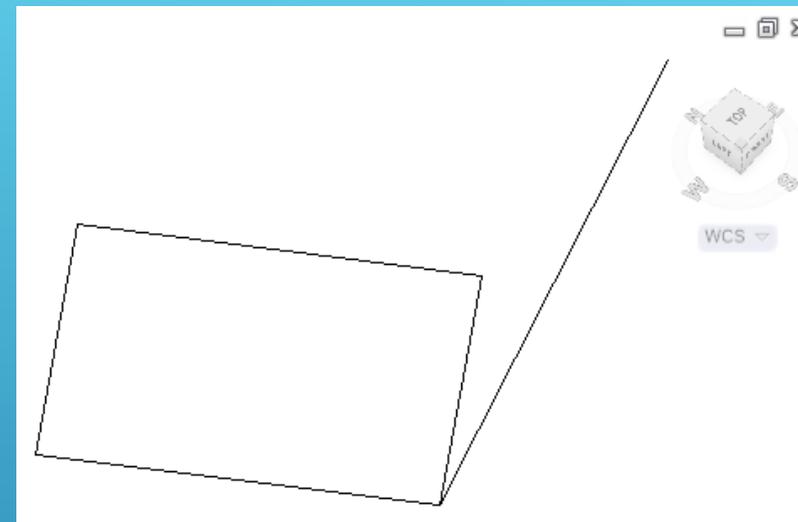
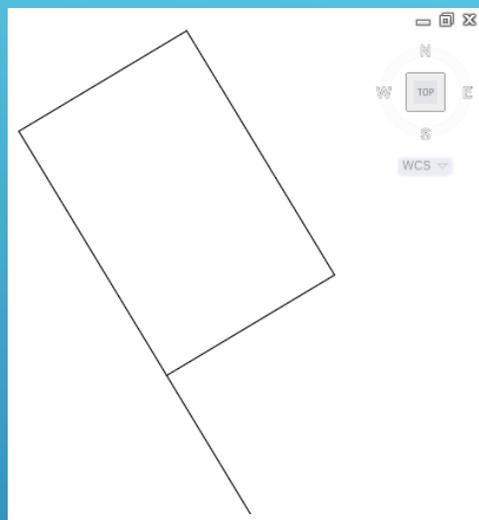


AULA 6 Desenho Técnico Assistido por Computador

A Torre de Controlo de Tráfego Marítimo do Tejo, em Algés, tem a forma de um prisma rectangular oblíquo, com 40 metros de altura, base com 13 metros por 19 metros e uma inclinação de 75° em relação à horizontal (o rumo da direcção do lado menor é igual a 59°).



AULA 6 Desenho Técnico Assistido por Computador



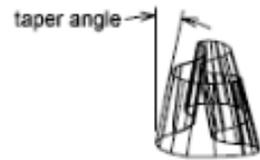
Entre o Cairo e Assuão, a margem ocidental do Nilo conserva mais de 60 pirâmides construídas entre 2700 e 1750 A.C.; a 10 km de Saqqara, o faraó Snefru mandou erguer uma enorme pirâmide de base quadrangular com 189 m de lado, cuja altura deve ter atingido os 102 m, conhecida actualmente como **pirâmide romboidal**.



Os arqueólogos têm sugerido que durante a construção, ao ser alcançada metade da altura prevista para o caso de uma pirâmide regular (todas as 8 arestas com igual comprimento), o ângulo de inclinação da pirâmide tenha sido reduzido pelo arquitecto para tentar diminuir o volume imenso de esforço sobre as paredes das câmaras internas que, acredita-se, poderiam estar a apresentar rachaduras.

Taper angle

Specifies the taper angle for the extrusion.



Positive angles taper in from the base object. Negative angles taper out. The default angle, 0, extrudes a 2D object perpendicular to its 2D plane. All selected objects and loops are tapered to the same value.

Specifying a large taper angle or a long extrusion height can cause the object or portions of the object to taper to a point before reaching the extrusion height.

Individual loops of a region are always extruded to the same height.

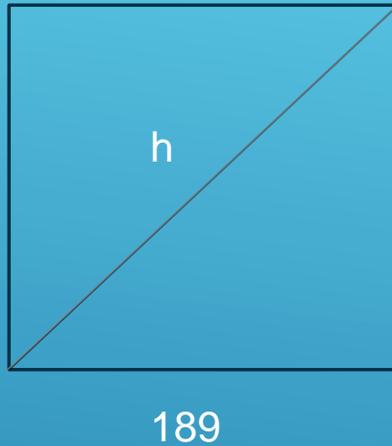
When an arc is part of a tapered extrusion, the angle of the arc remains constant, and the radius of the arc changes.

- **Angle of taper.** Specifies the taper between -90 and +90 degrees.
- **Specify two points.** Specifies the taper angle based on two specified points. The taper angle is the distance between the two specified points.

Drag the cursor horizontally to specify and preview the taper angle. You can also drag the cursor to adjust and preview the height of the extrusion. The dynamic input origin should be placed on the extruded shape, on the projection of the point to the shape.

When you select the extruded object, the position of the taper grip is the correspondent point of the dynamic input origin on the top face of the extrusion.

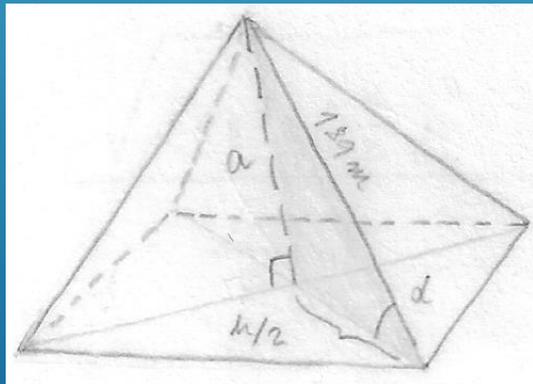
em planta



$$h^2 = \sqrt{189^2 + 189^2} \Rightarrow h = \sqrt{2 \times 189^2} = 267.2864 \text{ m} \Rightarrow \frac{h}{2} = 133.6431 \text{ m}$$

Altura do vértice da pirâmide regular

$$\left(\frac{h}{2}\right)^2 + a^2 = 189^2 \Rightarrow a = \sqrt{189^2 - \left(\frac{h}{2}\right)^2} = 133.6431 \text{ m}$$



corte vertical

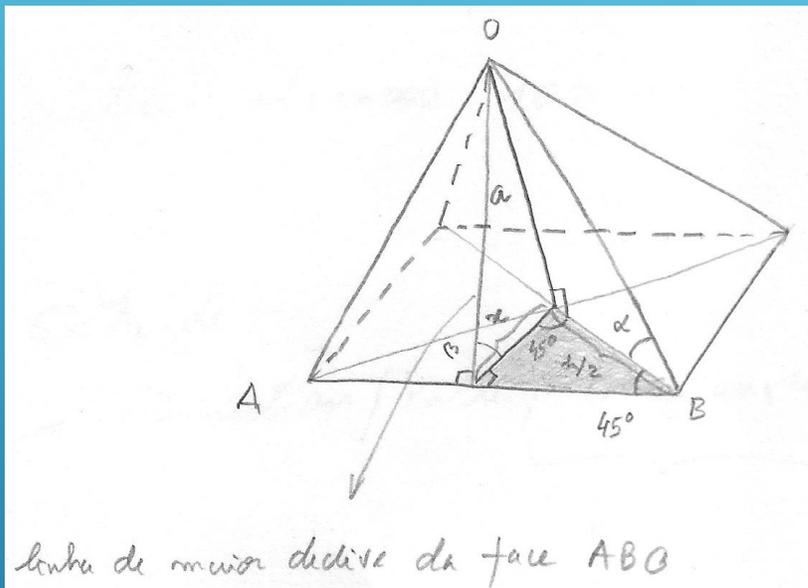


Inclinação da aresta da pirâmide regular

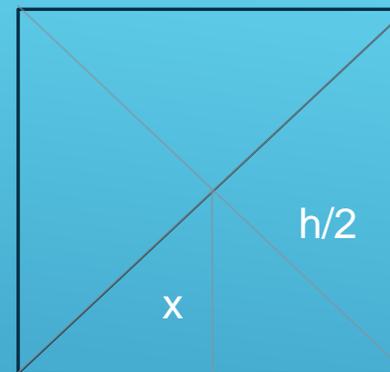
$$\tan \alpha = \frac{a}{\frac{h}{2}} = \frac{133.6431}{133.643} \Rightarrow \alpha = 45^\circ$$

a

α = inclinação das arestas que ligam a base ao topo da pirâmide



em planta

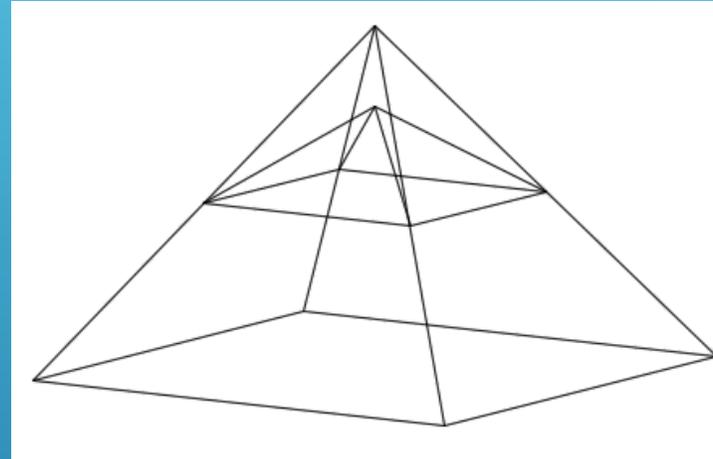
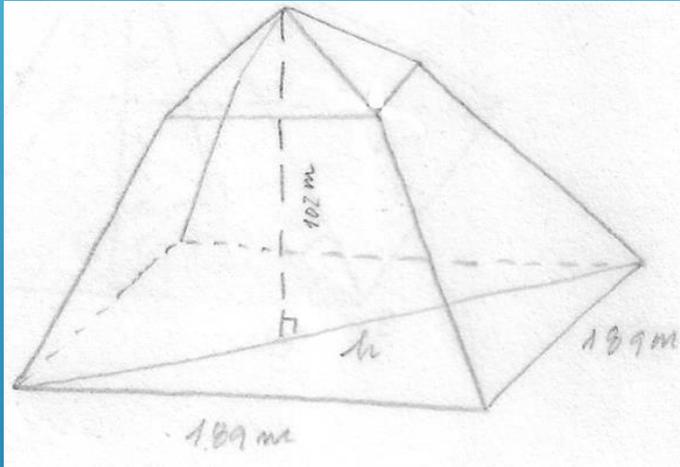


189

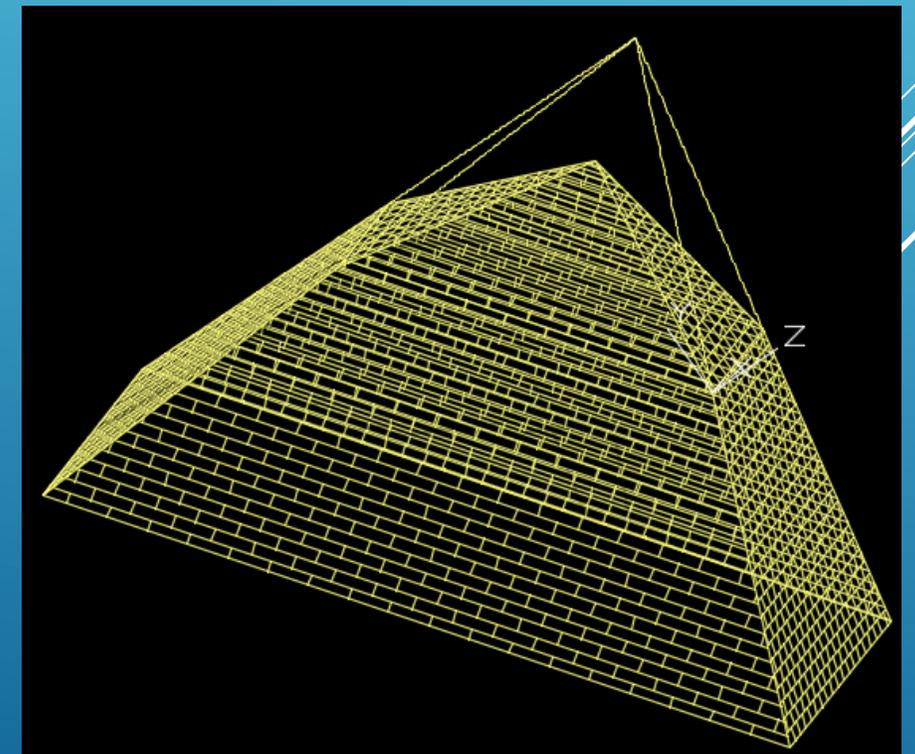
$$\frac{\sin 90^\circ}{\frac{h}{2}} = \frac{\sin 45^\circ}{x} \Rightarrow x = \frac{\sqrt{2}}{2} \frac{h}{2} = 94.5000 \text{ m}$$

Inclinação da linha de maior declive da pirâmide regular

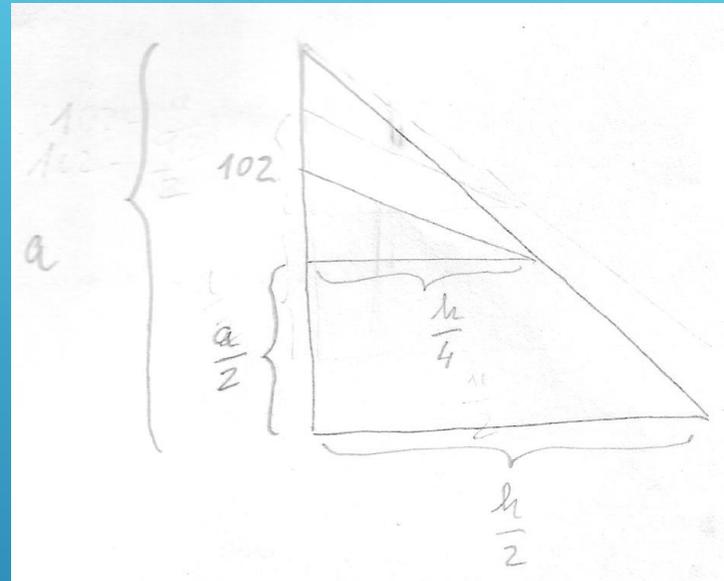
$$\begin{aligned} \tan \beta &= \frac{a}{x} = \frac{\sqrt{189^2 - \left(\frac{h}{2}\right)^2}}{\frac{\sqrt{2}}{2} \frac{h}{2}} = \sqrt{\frac{189^2 - \frac{h^2}{4}}{\frac{1}{2} \frac{h^2}{4}}} = \sqrt{\frac{189^2 - \frac{189^2 + 189^2}{4}}{\frac{1}{2} \frac{189^2 + 189^2}{4}}} = \sqrt{\frac{4 \times 189^2 - 2 \times 189^2}{\frac{2 \times 189^2}{8}}} = \sqrt{\frac{2 \times 189^2}{\frac{189^2}{4}}} = \sqrt{2} \\ \Rightarrow \beta &= 54.735614^\circ \end{aligned}$$



Quando a pirâmide alcançou metade da altura prevista: **$a/2=66.8216$ m**, a inclinação da pirâmide diminuiu, de tal forma que o vértice alcançou a altura 102 m.



$$\frac{a}{\frac{h}{2}} = \frac{\frac{a}{2}}{\frac{h}{4}}$$



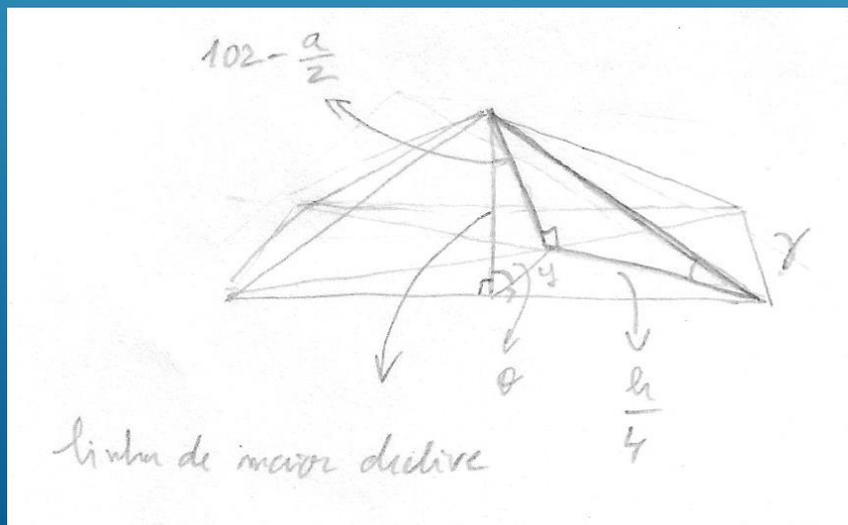
Inclinação da aresta

$$\gamma = \text{atan} \frac{102 - \frac{a}{2}}{\frac{h}{4}} = \text{atan} \frac{35.17845}{66.80655} = 27^{\circ}.7702$$

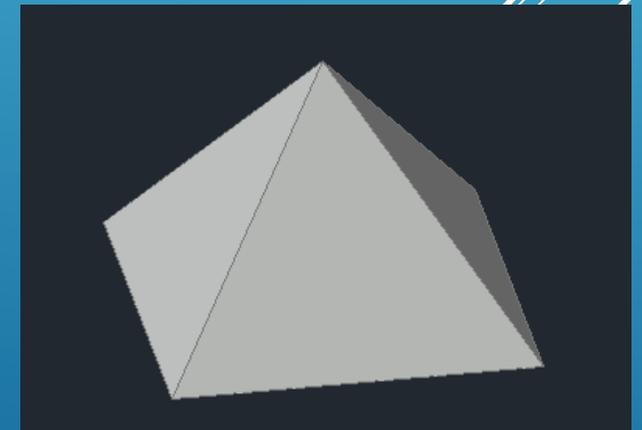
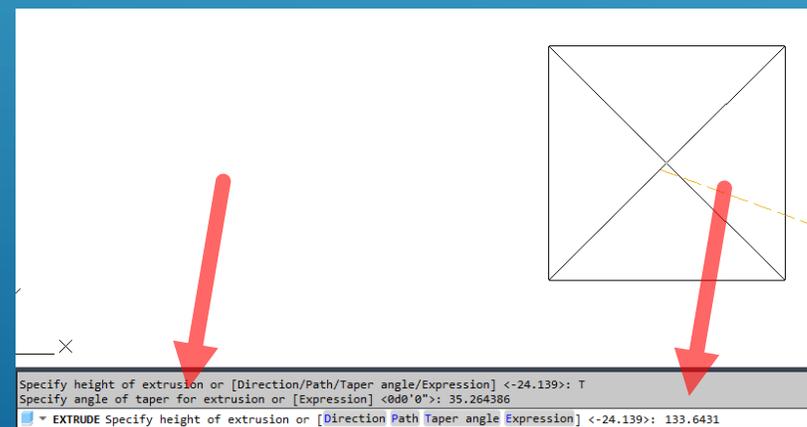
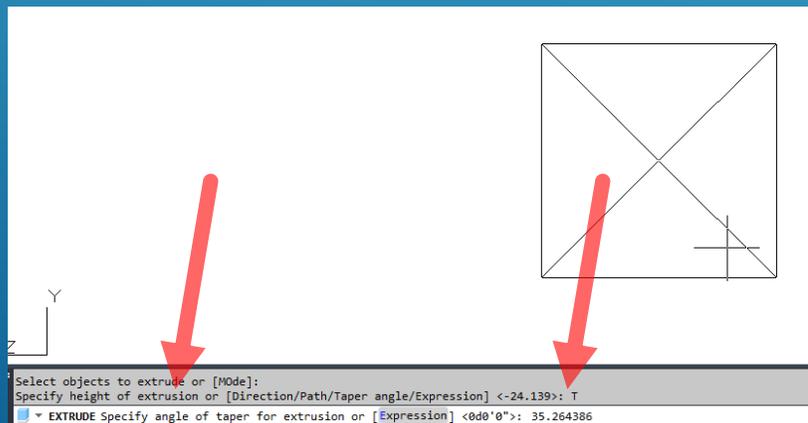
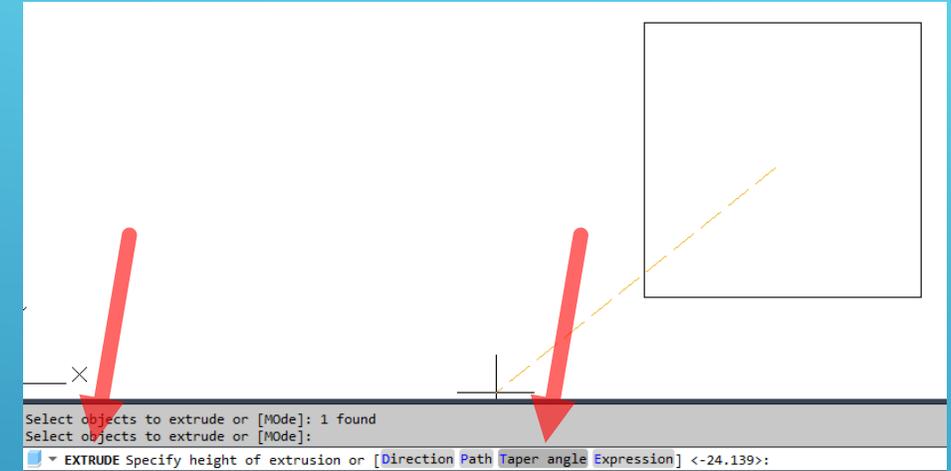
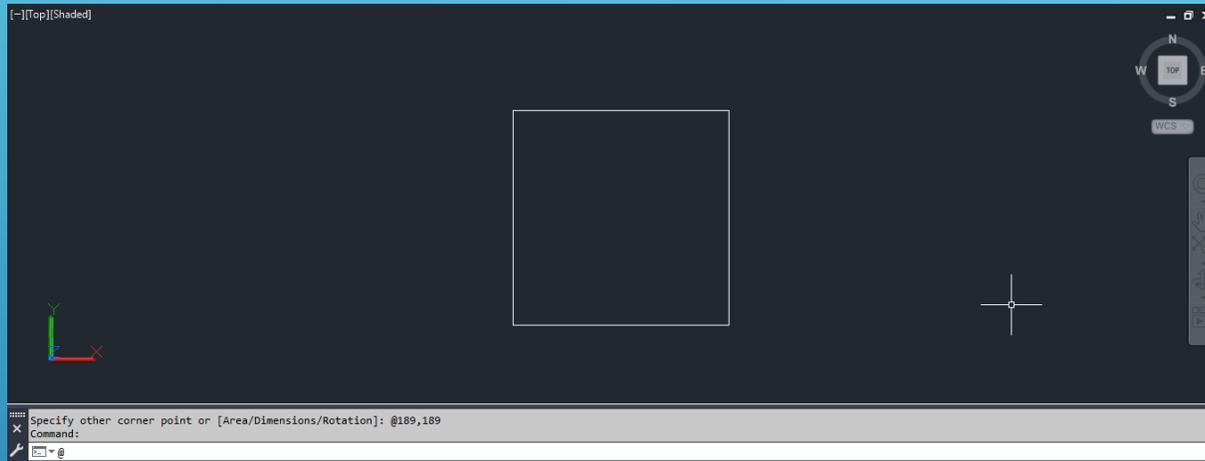
Inclinação da linha de maior declive

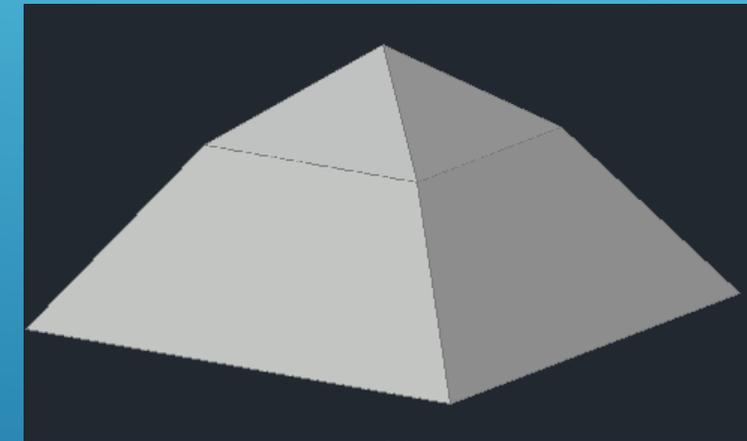
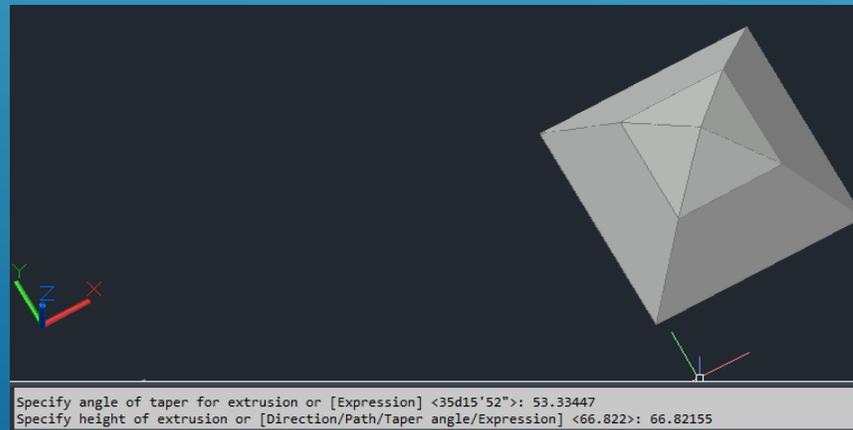
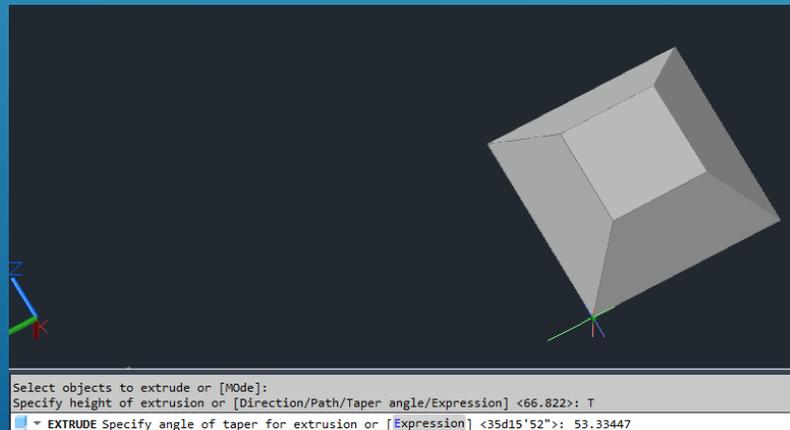
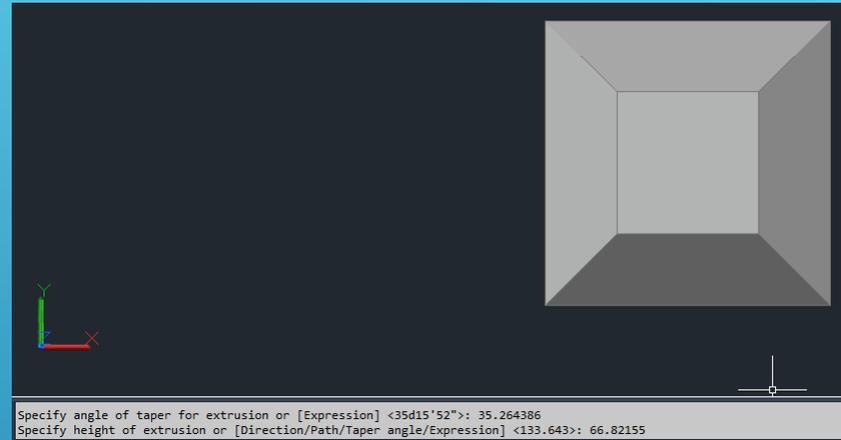
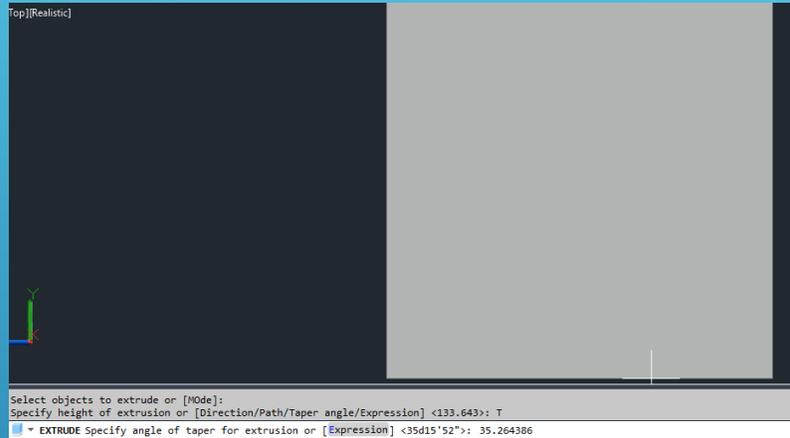
$$\frac{\sin 90^{\circ}}{\frac{h}{4}} = \frac{\sin 45^{\circ}}{y} \Rightarrow y = \frac{\sqrt{2} h}{2 \cdot 4}$$

$$\tan \theta = \frac{102 - \frac{a}{2}}{\frac{\sqrt{2} h}{2 \cdot 4}} = \frac{35.17485}{47.24997} \Rightarrow \theta = 36^{\circ}.66553$$

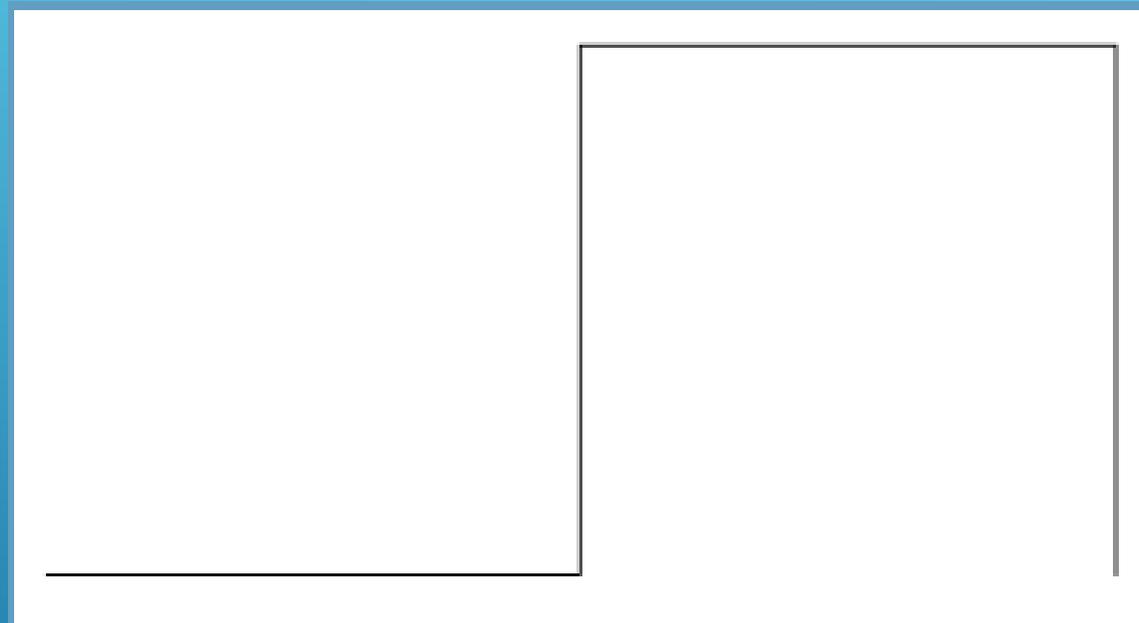


AULA 6 Desenho Técnico Assistido por Computador

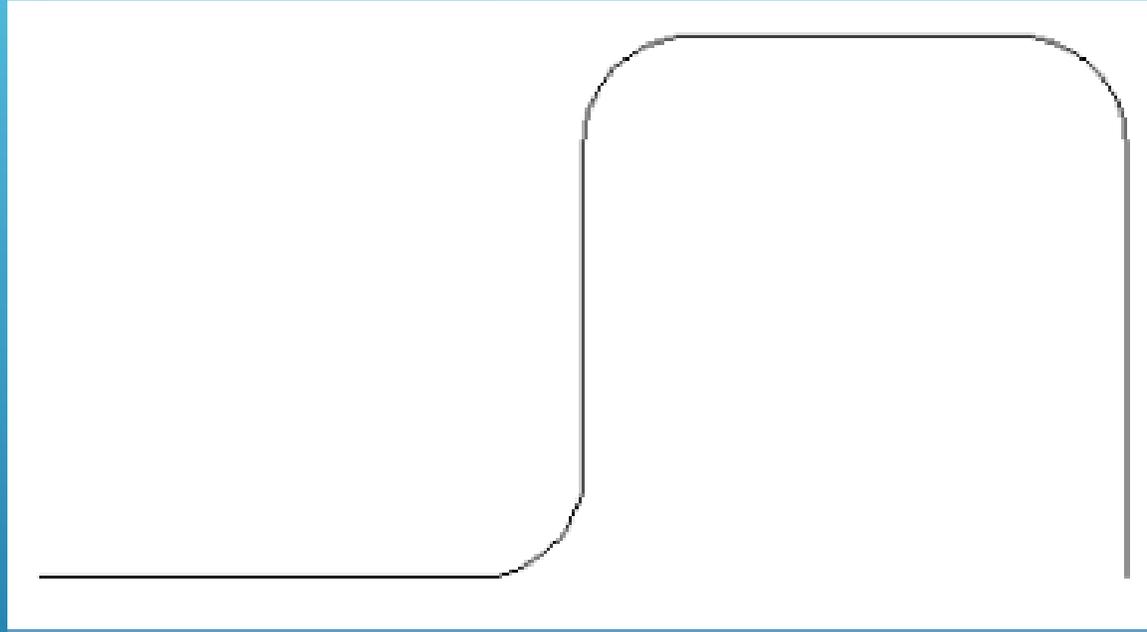




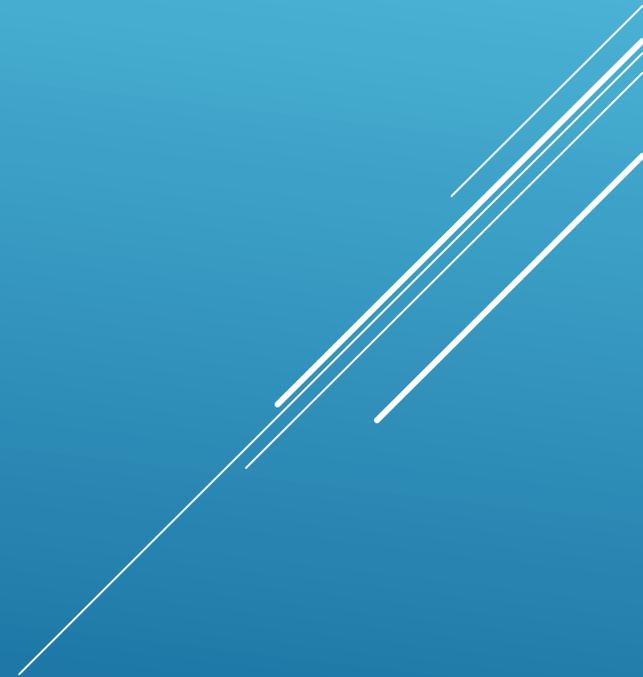
Extrude ao longo
de um caminho

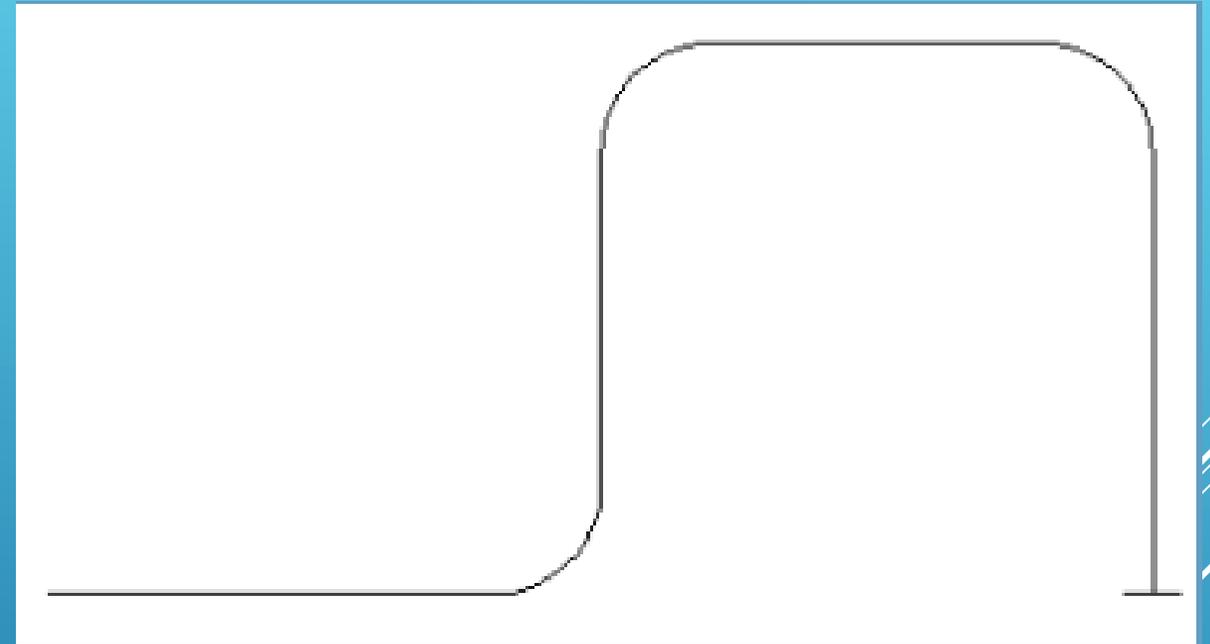


Desenhar uma Polyline entre os pontos $(0,0)$,
 $(120,0)$, $(120,120)$, $(240,120)$, $(240,0)$



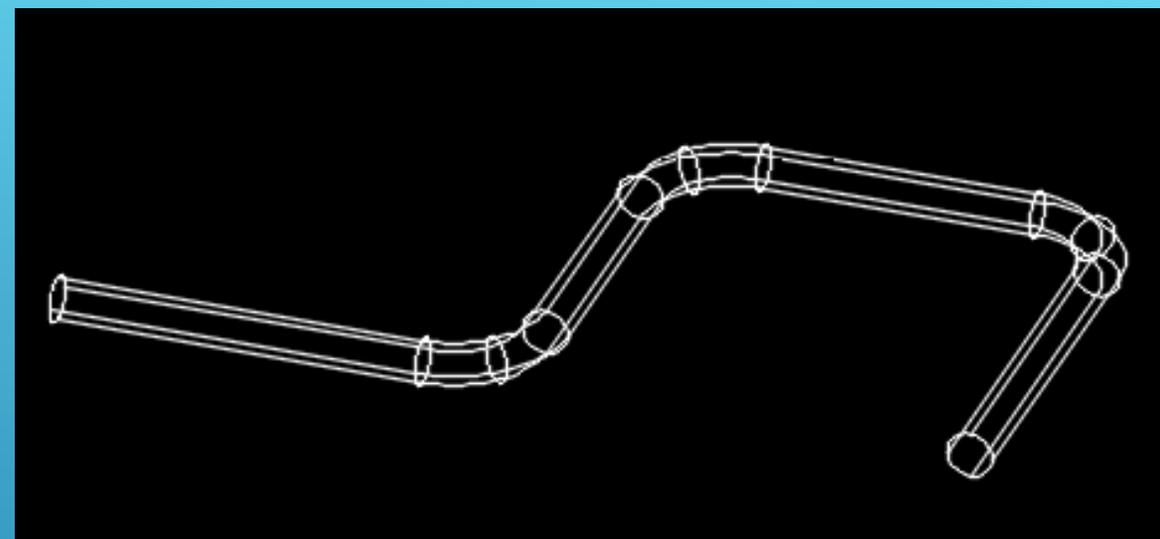
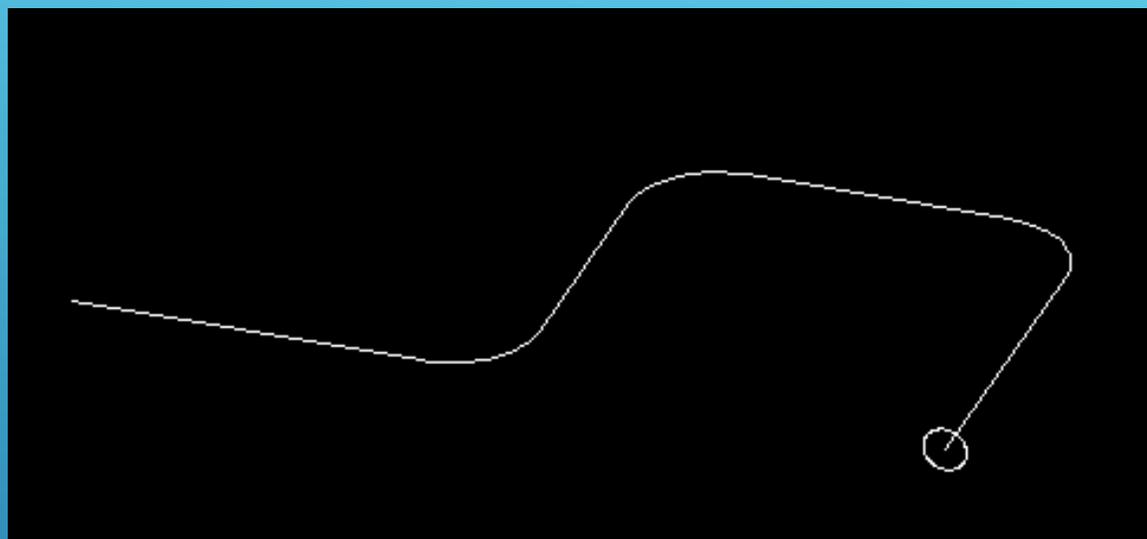
Utilizar o comando Fillet com raio 24



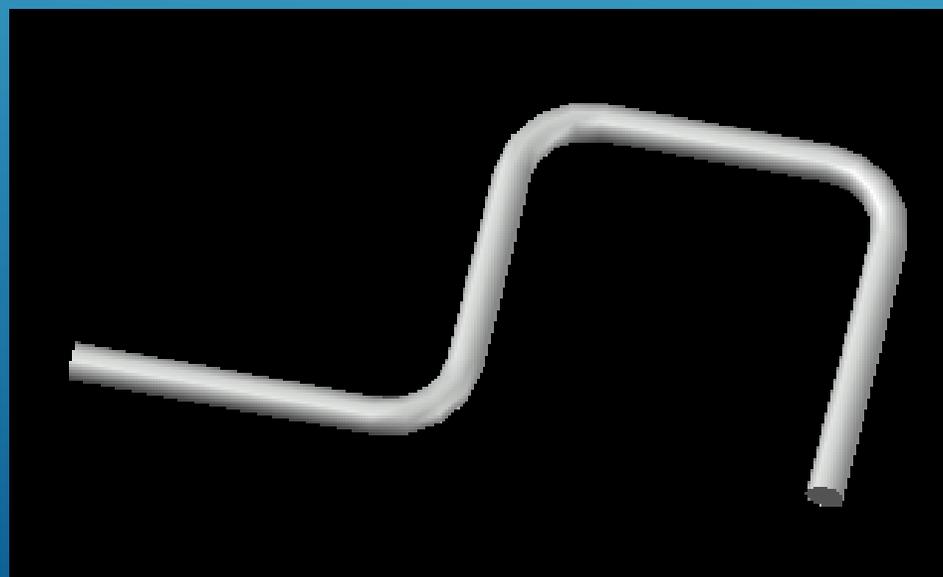


Vista Top

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.

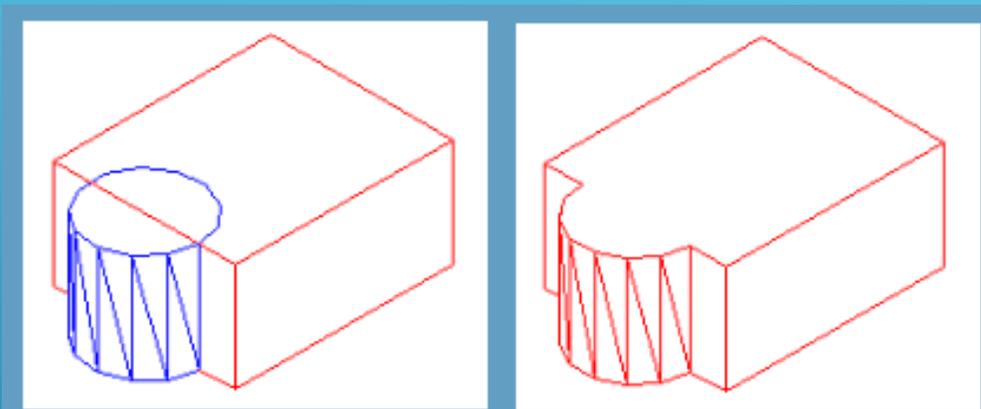


Utilizar o comando Extrude, seleccionar a circunferência, seleccionar **Path** e indicar a polyline

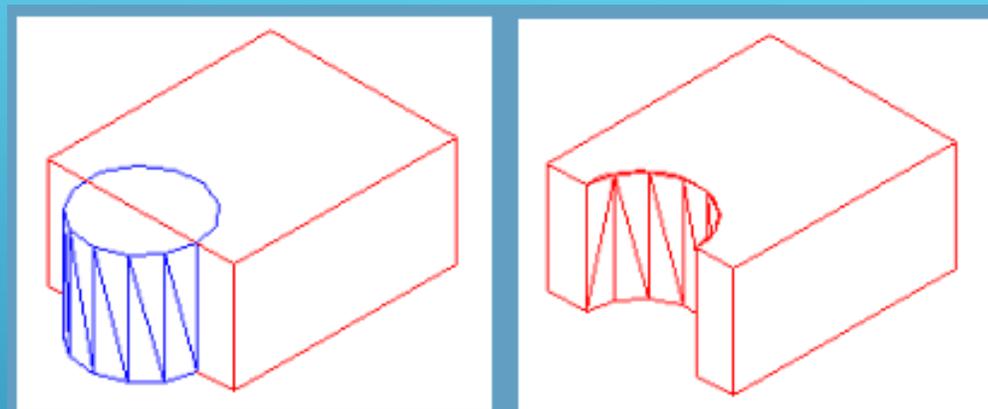


Boolean Operations

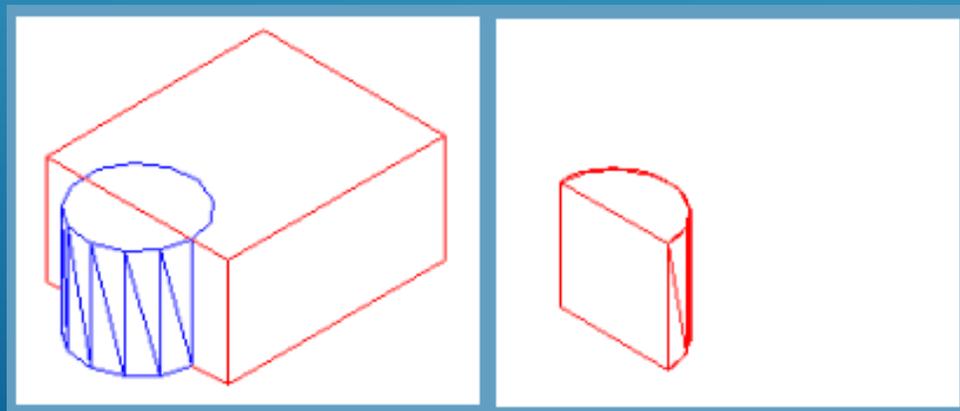
COMMAND	INPUT	ICON	DESCRIPTION
UNION (Boolean)	UNION / UNI		Joins two or more solids into creating one based on the total geometry of all.
SUBTRACT (Boolean)	SUBTRACT / SU		Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT (Boolean)	INTERSECT / IN		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.



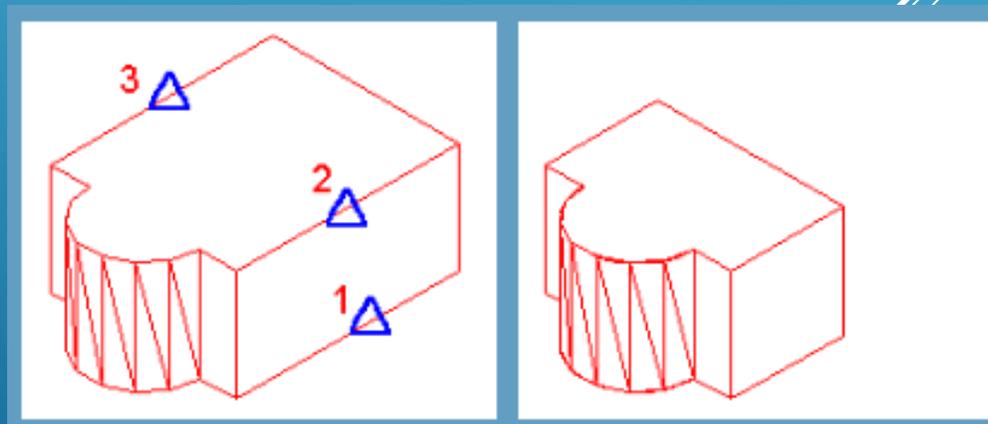
Union



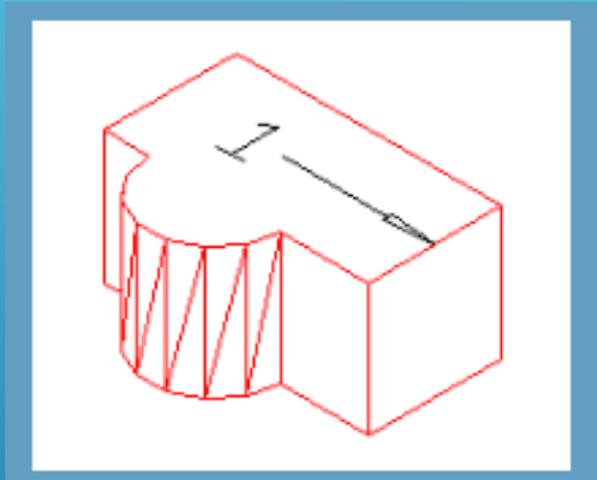
Subtract



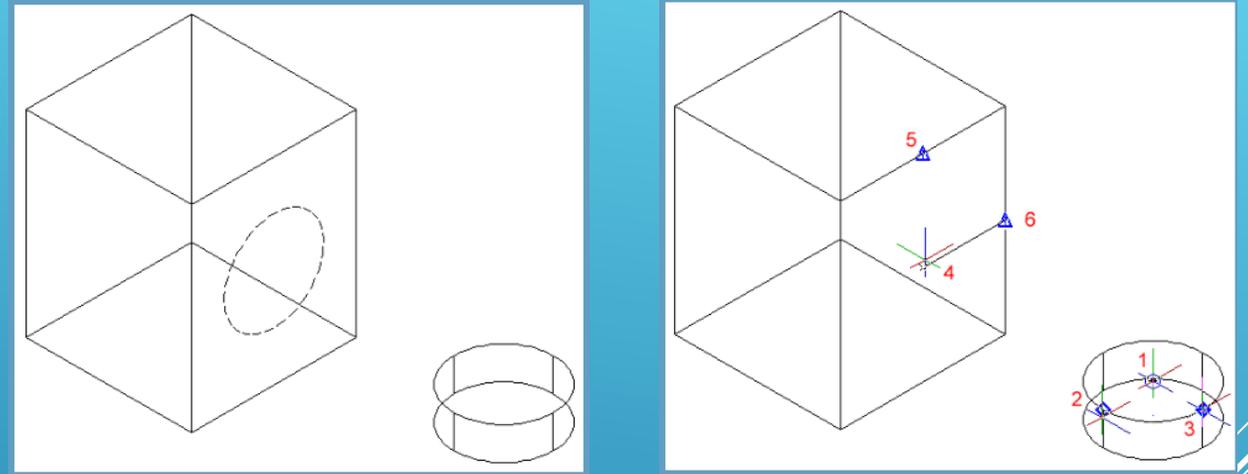
Intersect



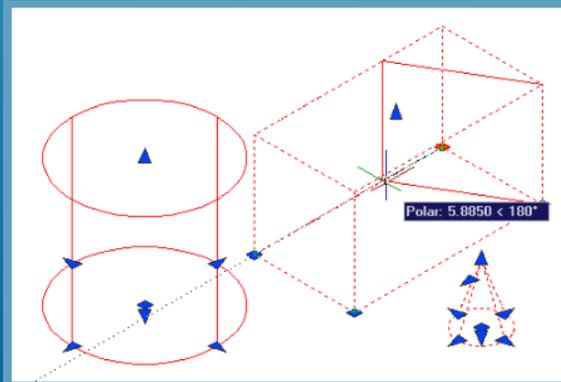
Slice



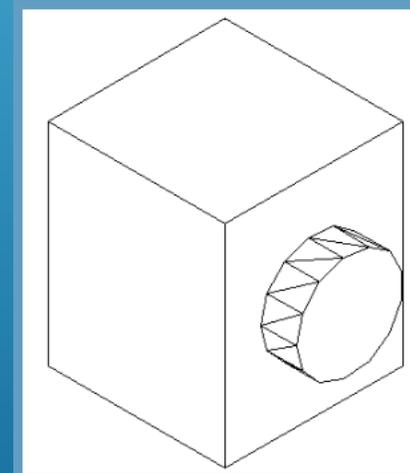
Extrude face

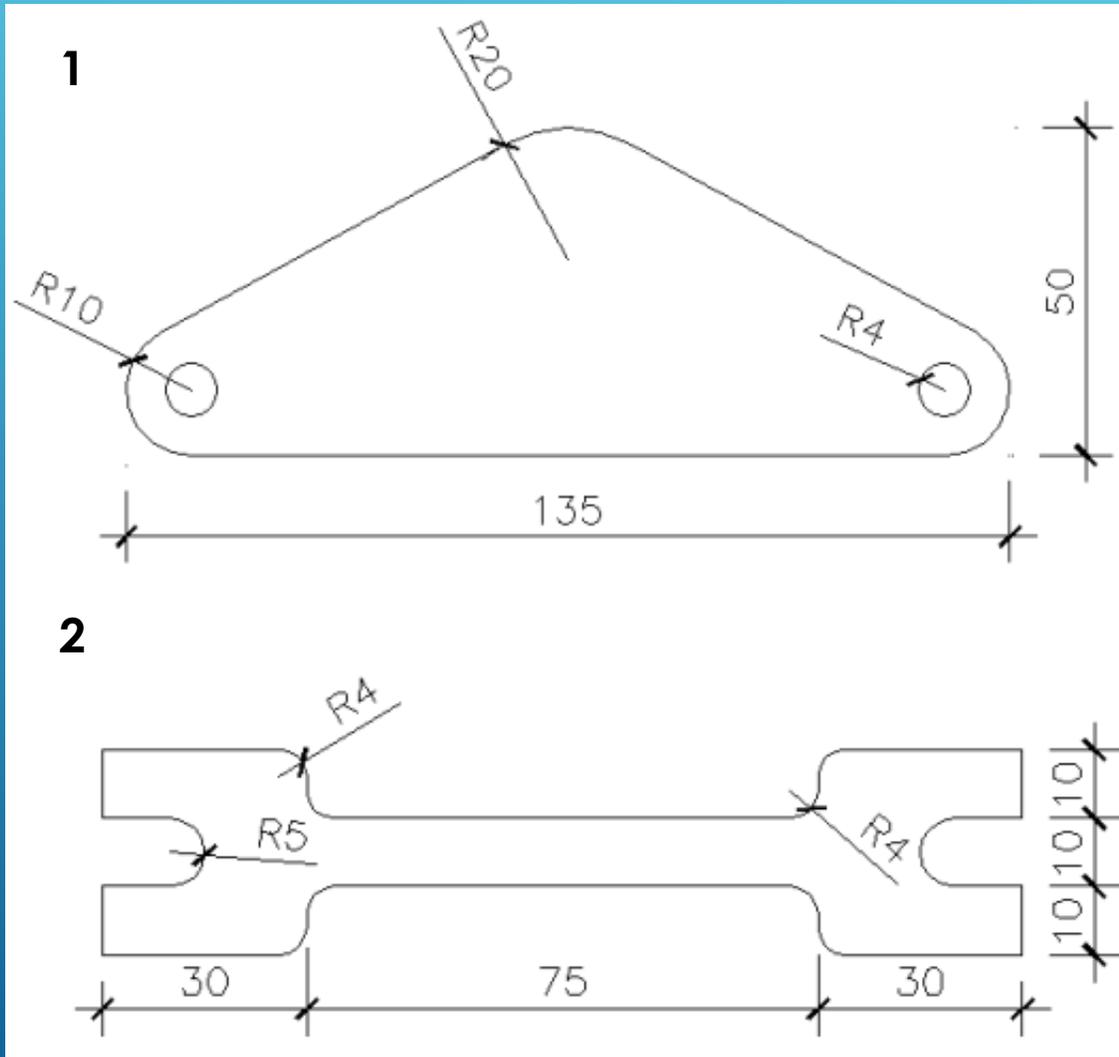


3D Align



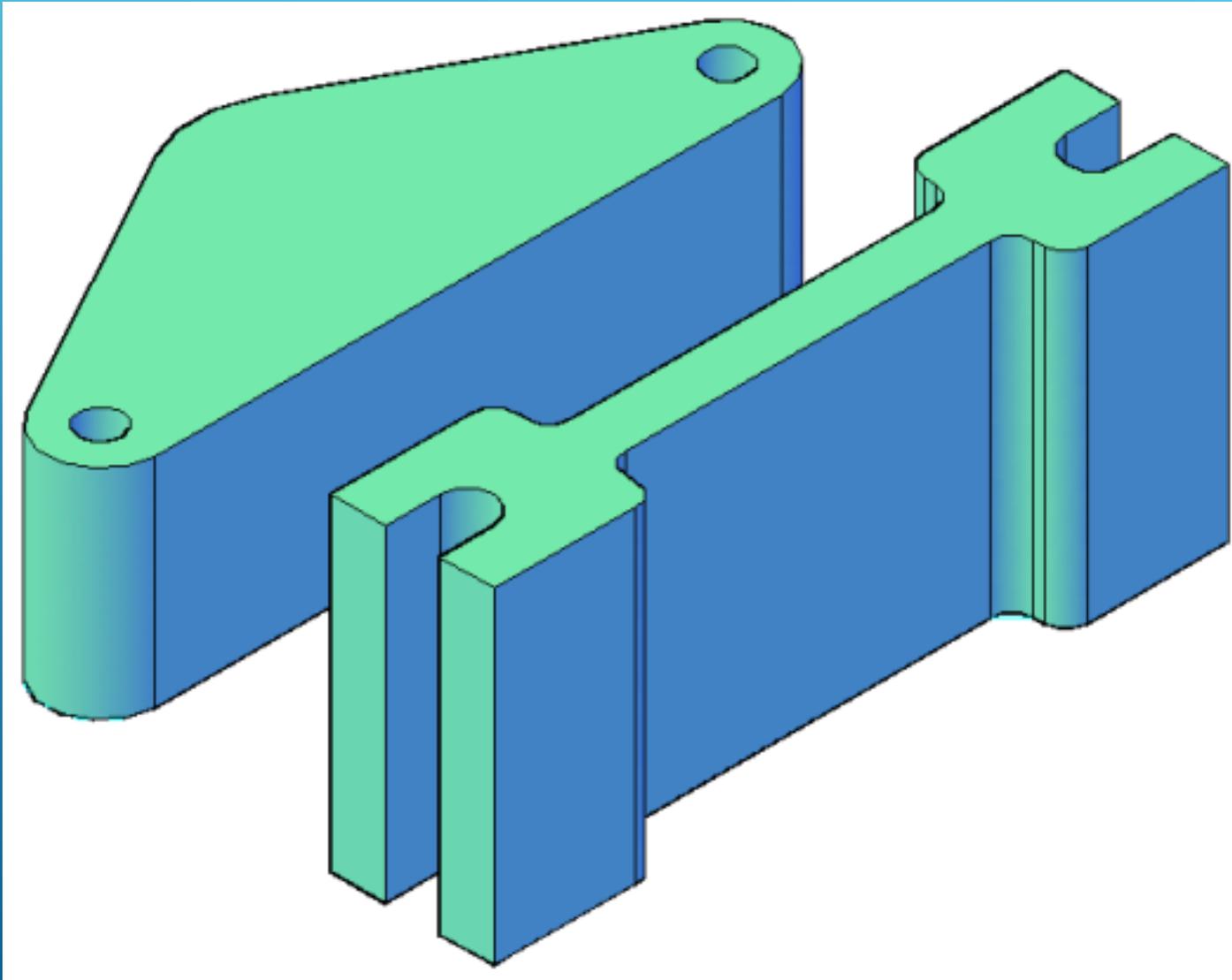
Extrude using grips





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

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).

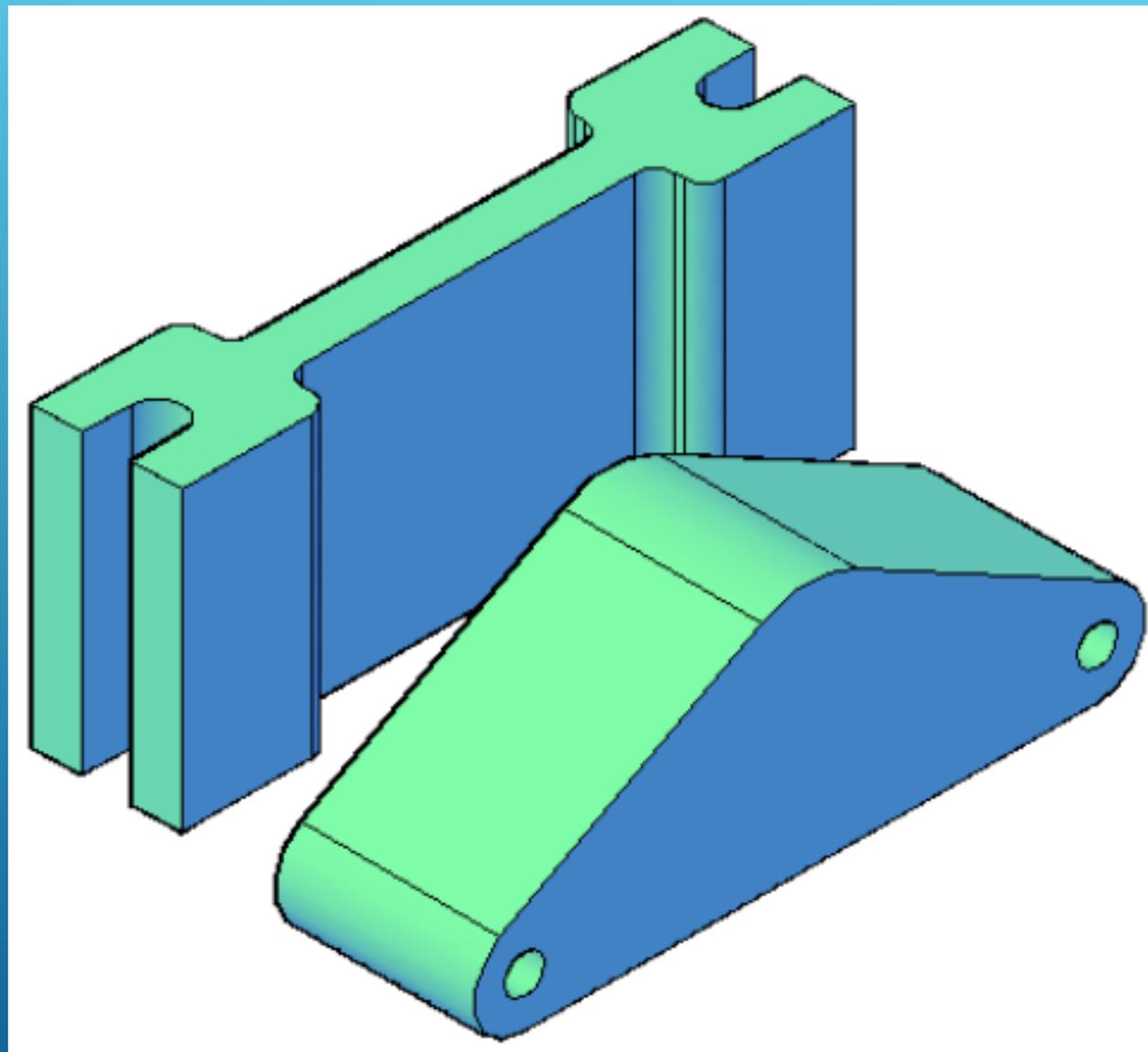
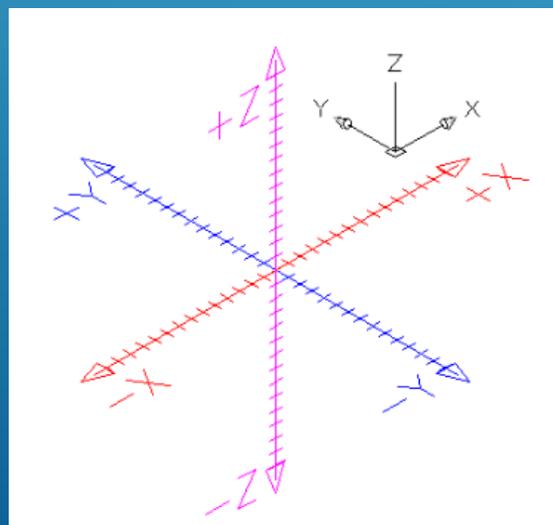
View > NE ISO

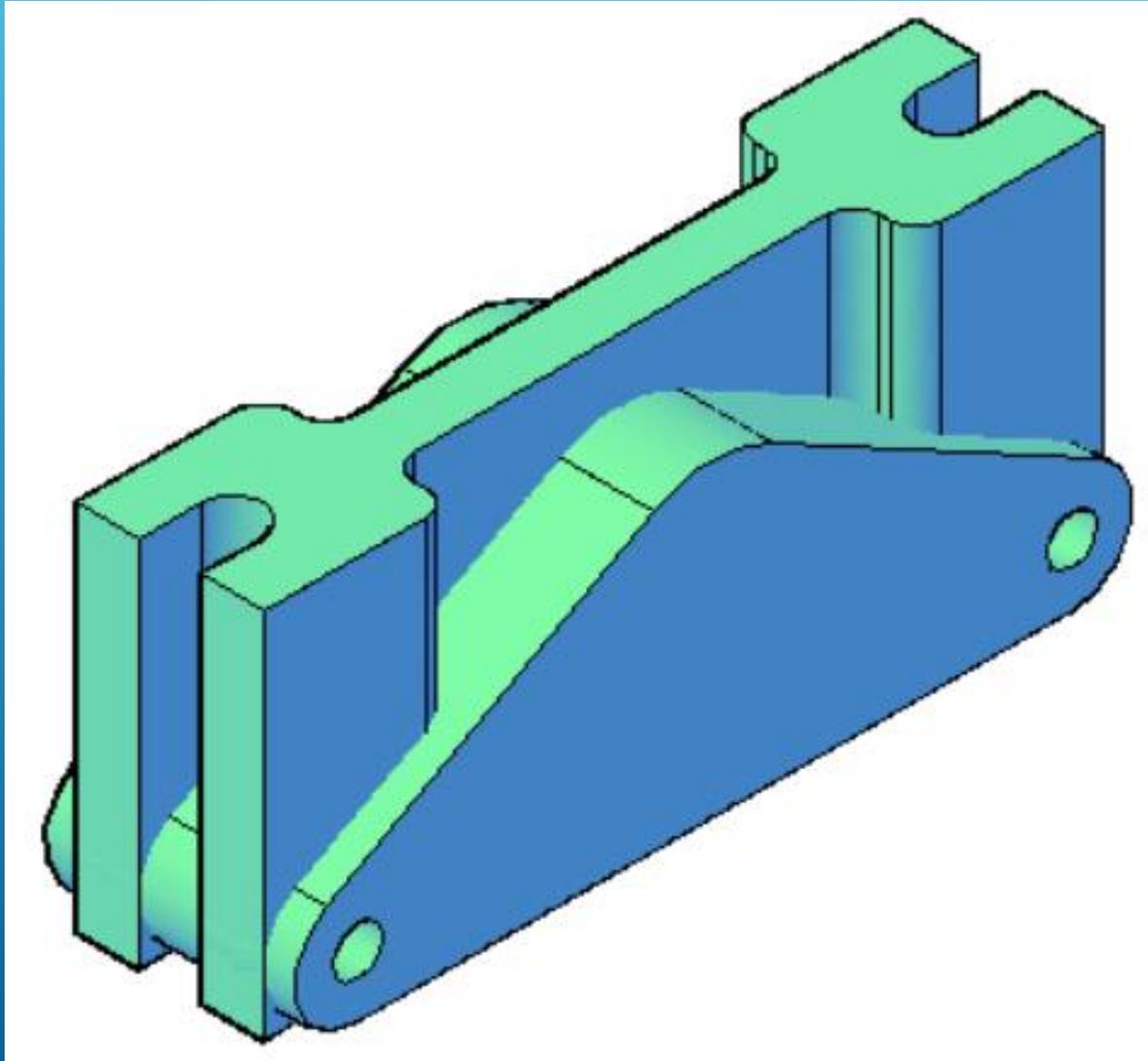
Modify > Rotate 3D



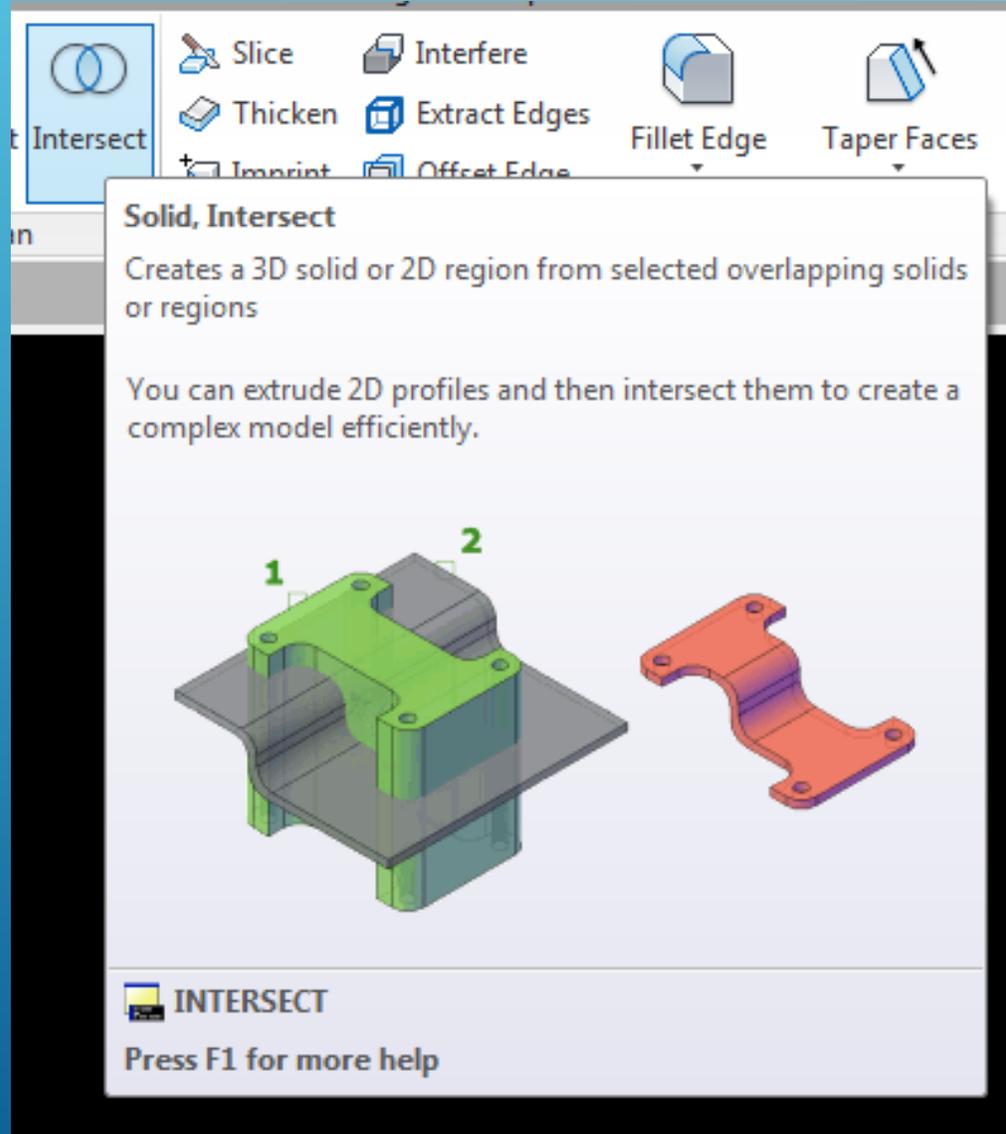
Seleccionar o desenho 1

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





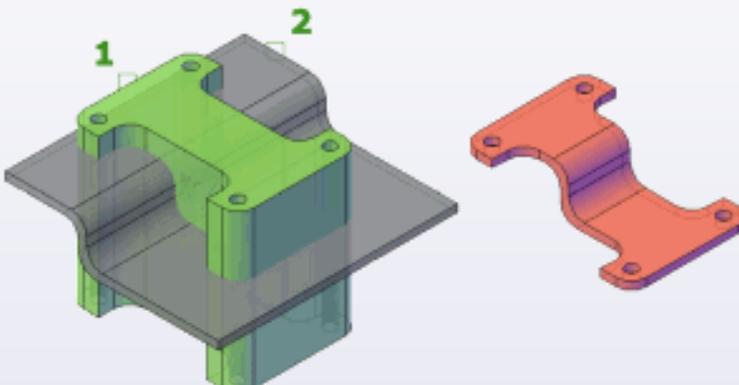
Posicionar o centro do desenho1 no
centro do desenho 2



Solid, Intersect

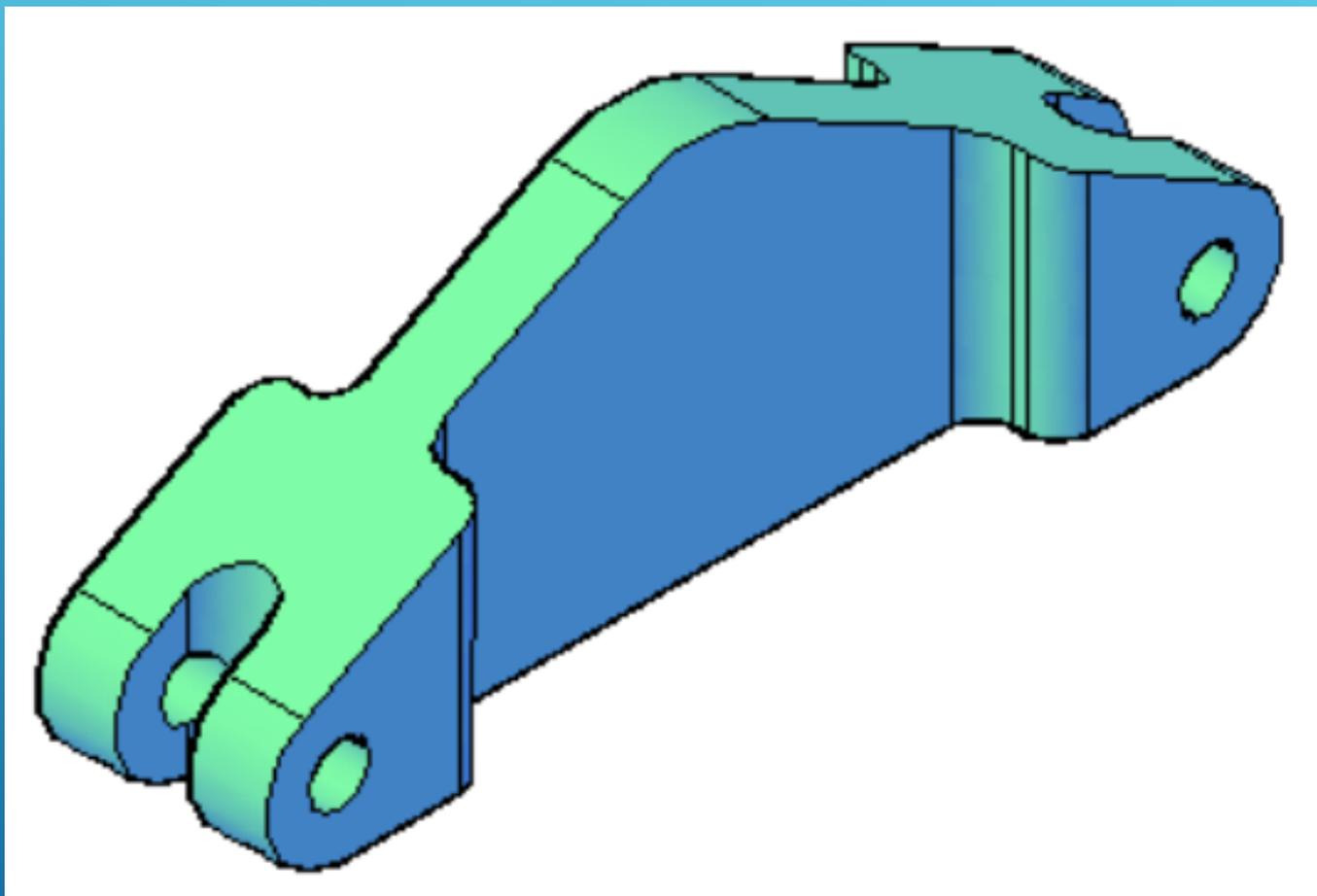
Creates a 3D solid or 2D region from selected overlapping solids or regions

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



INTERSECT

Press F1 for more help



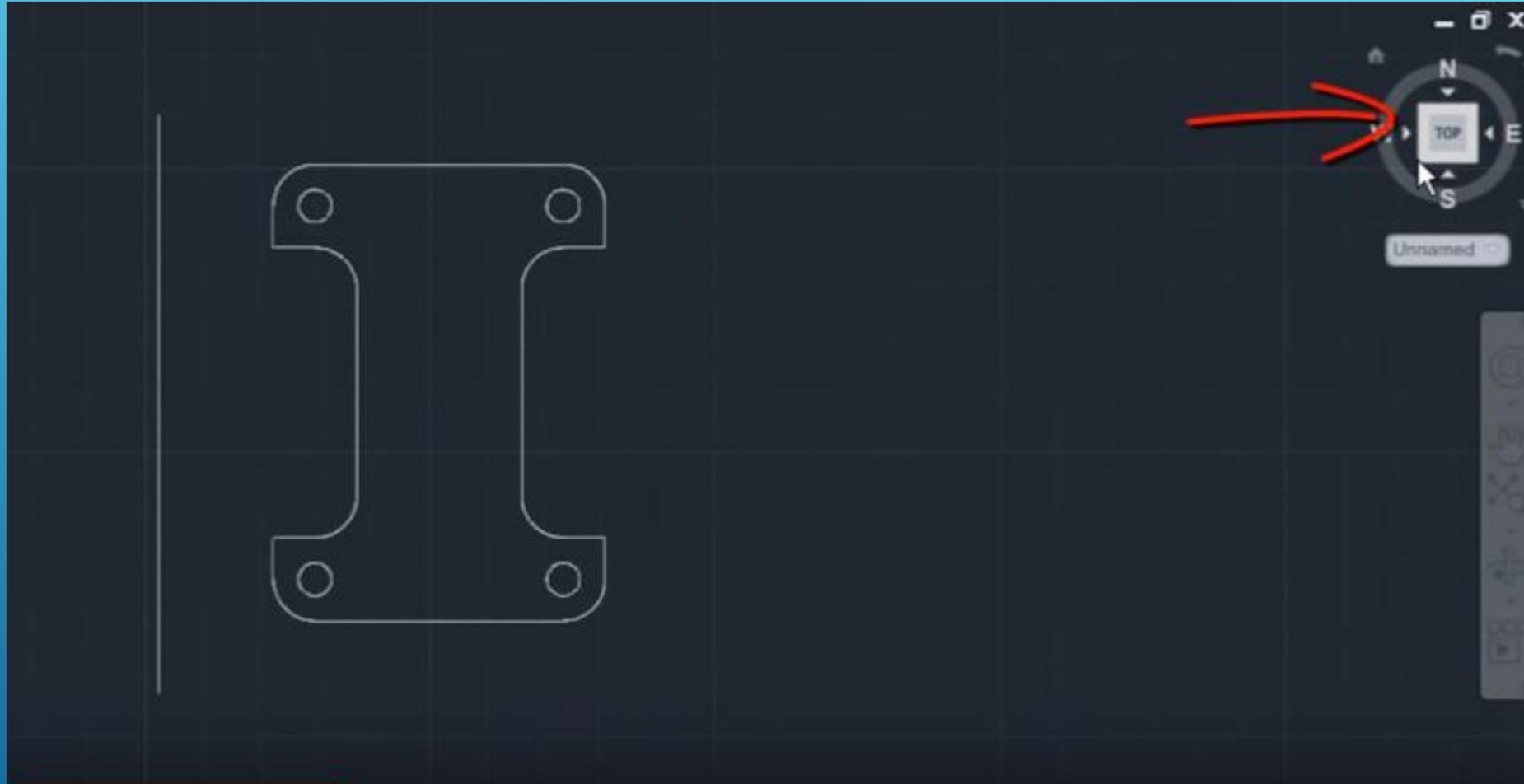
Efectuar a intercessão das duas figuras
(seleccionar as 2 figuras com shift + enter)

AULA 5 Desenho Técnico Assistido por Computador



Ciências
ULisboa

Engenharia Geográfica,
Geofísica e Energia



Ver intersect.dwg

AULA 5 Desenho Técnico Assistido por Computador



Ciências
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Engenharia Geográfica,
Geofísica e Energia

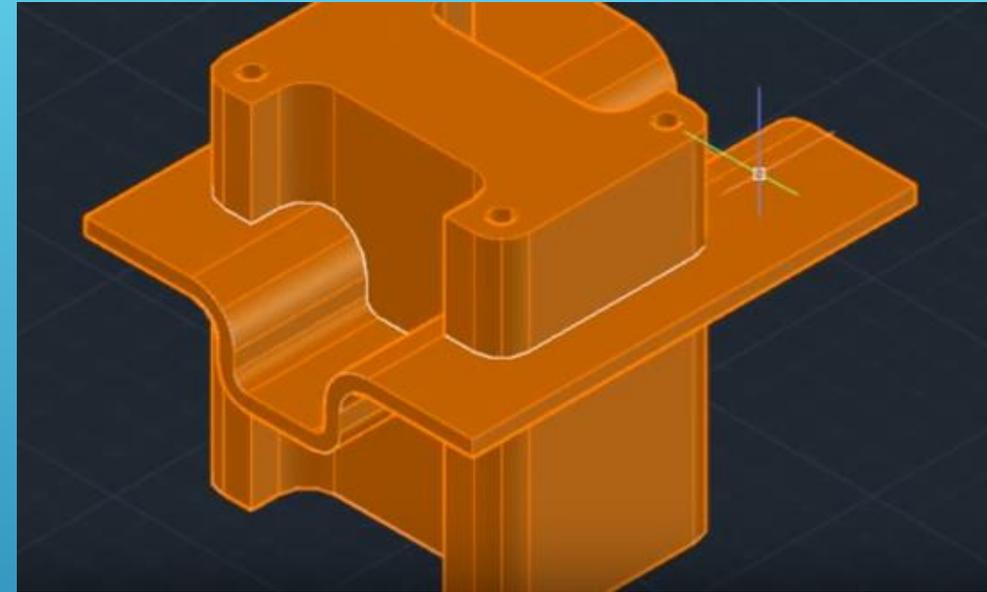
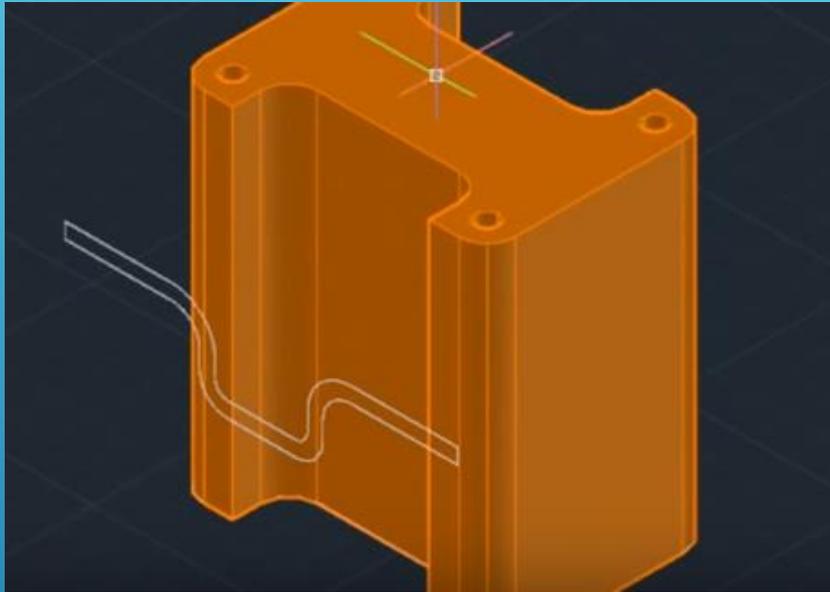


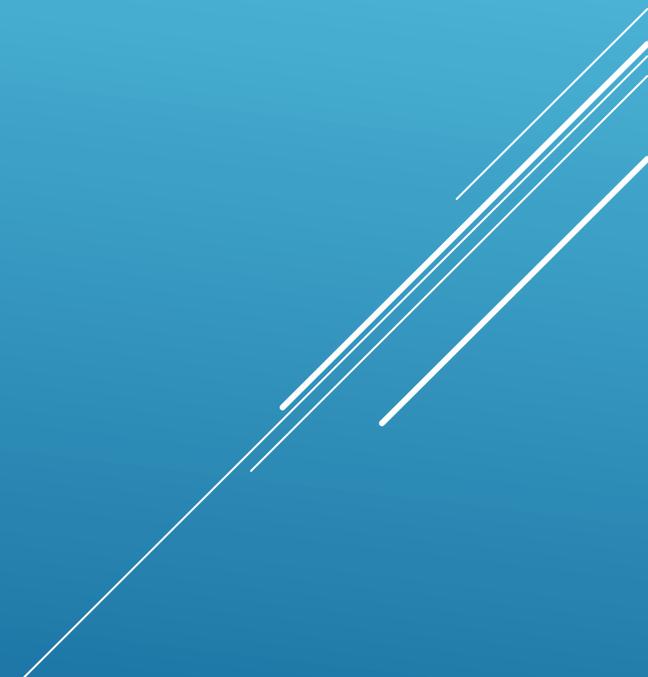
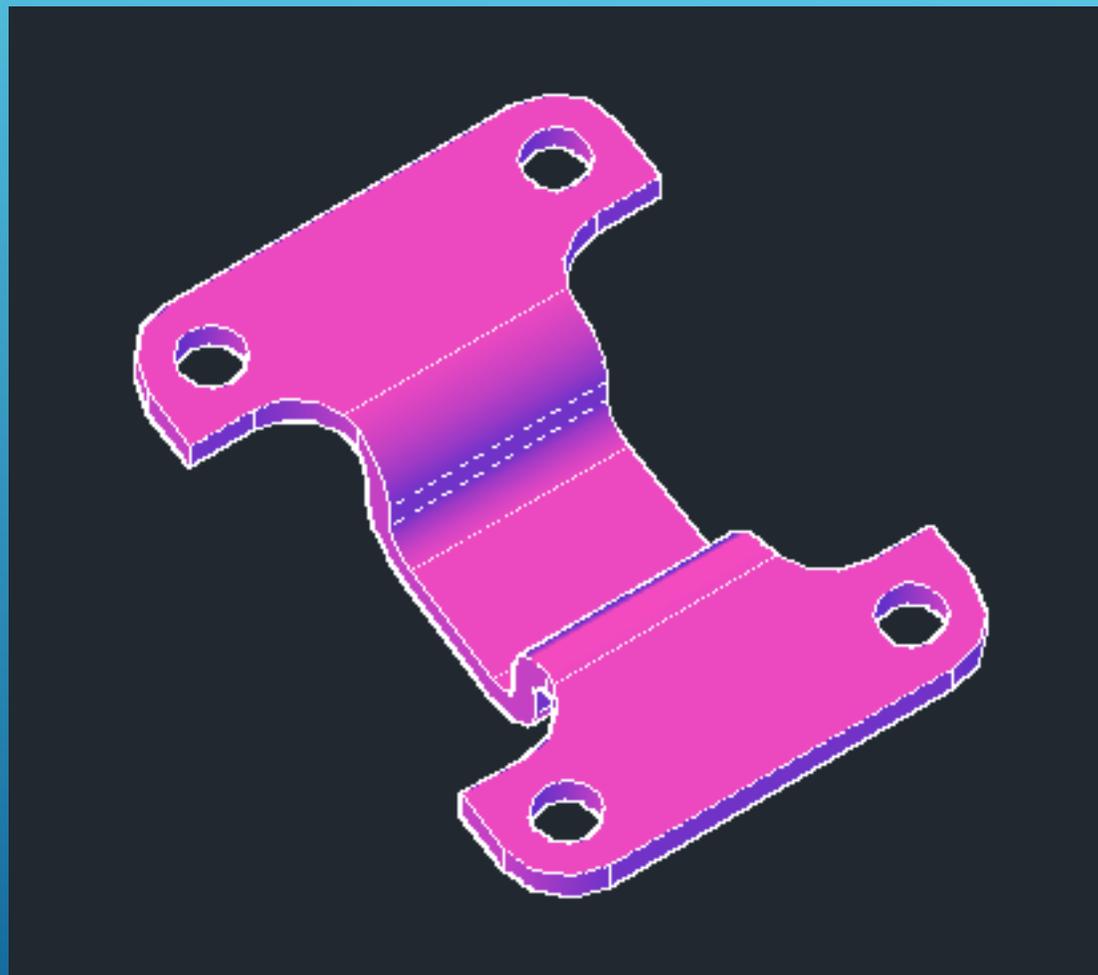
AULA 5 Desenho Técnico Assistido por Computador

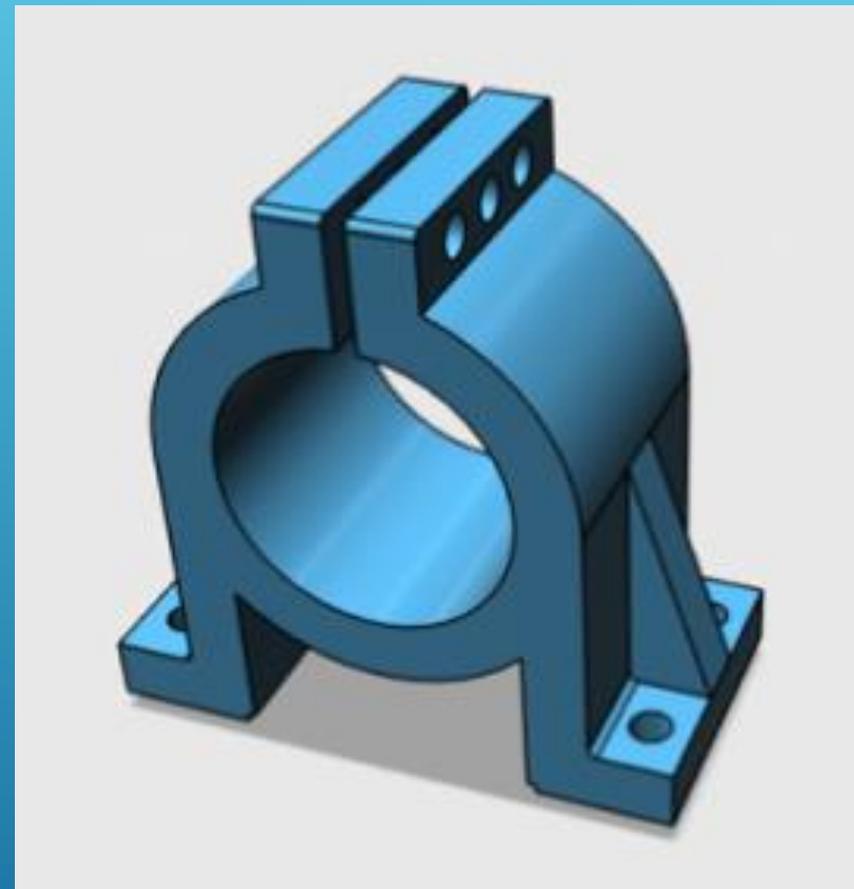
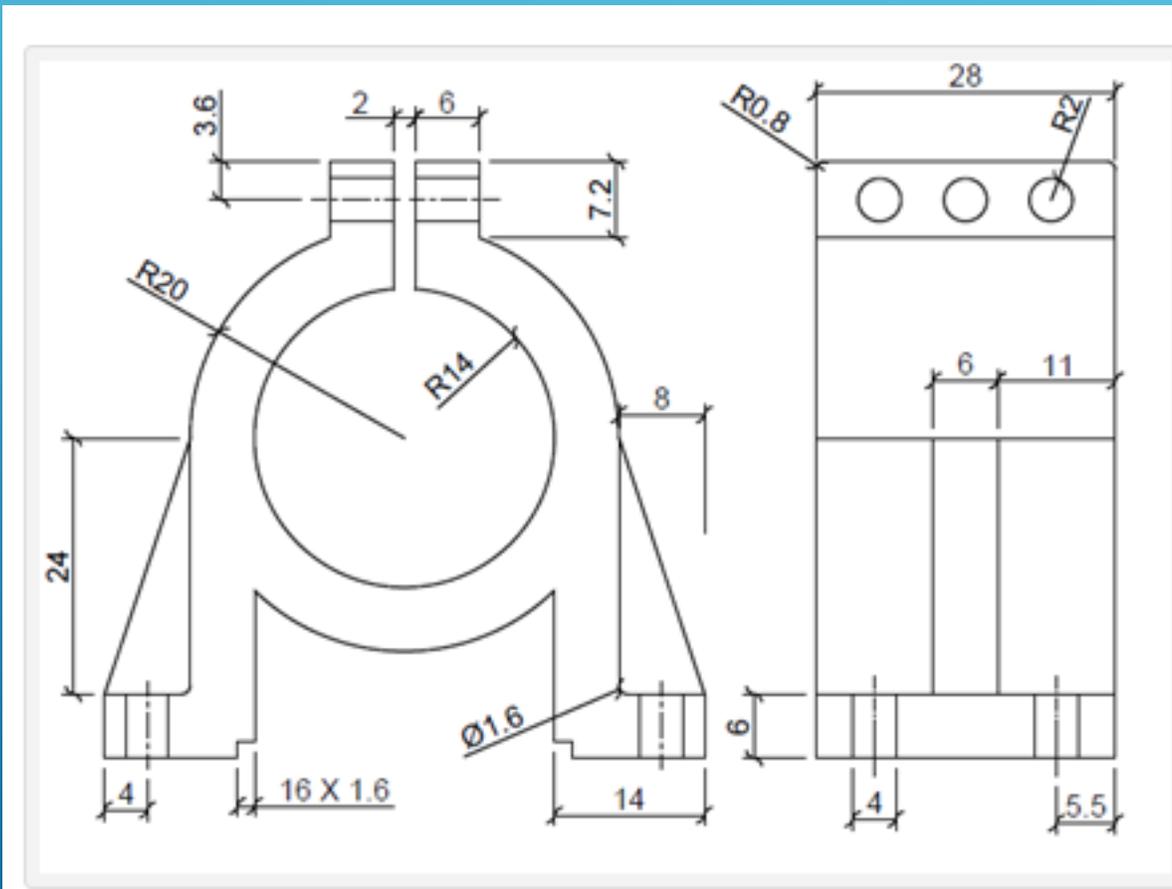


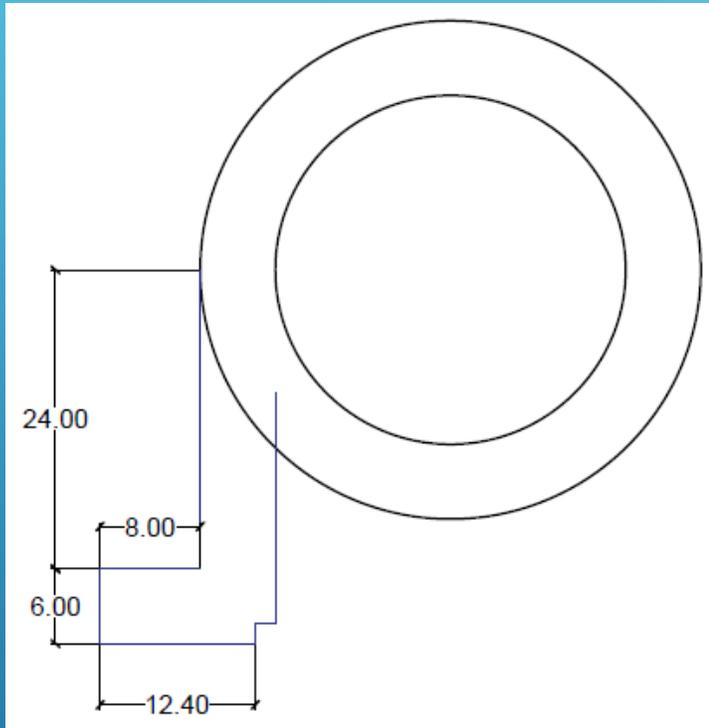
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Geofísica e Energia



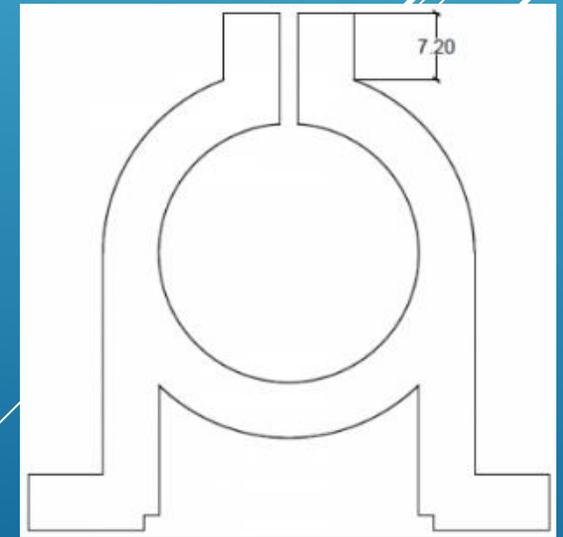
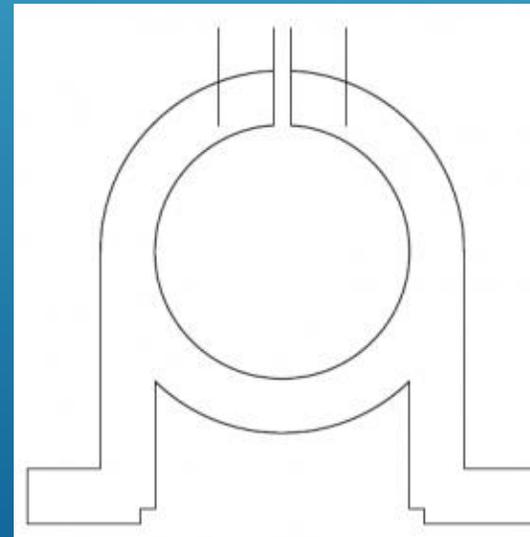
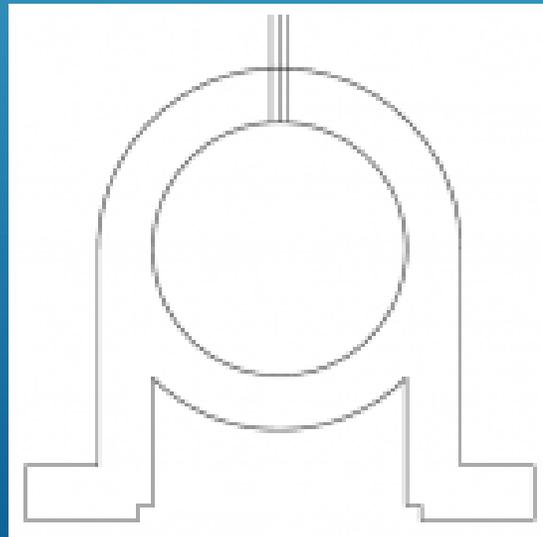
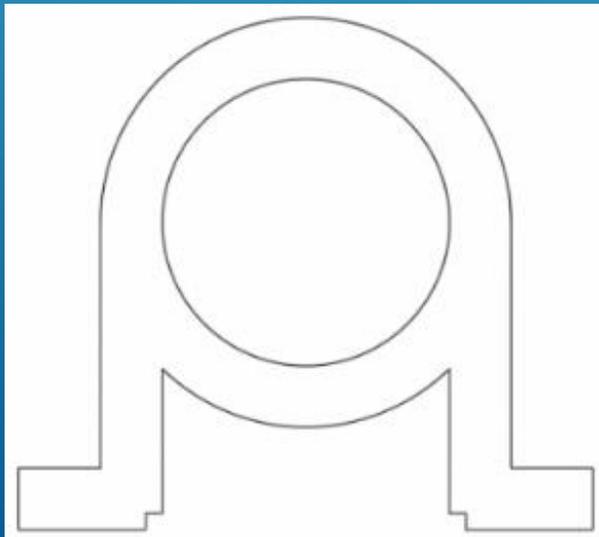


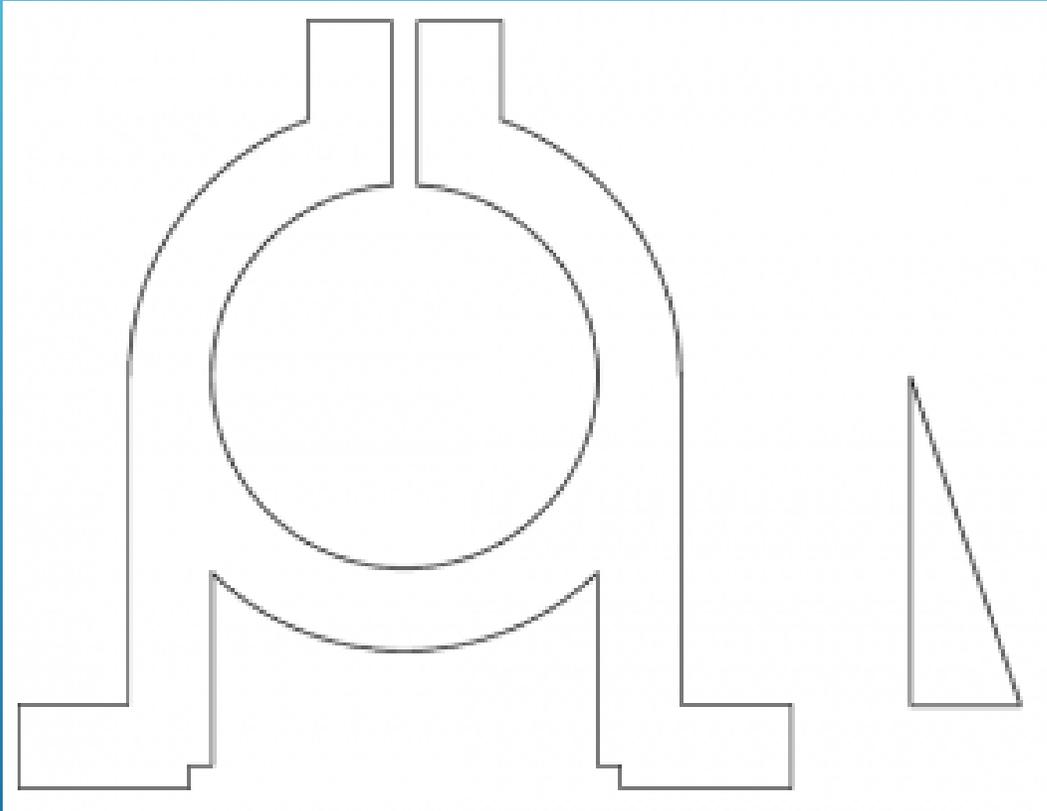




1. Desenhar 2 circunferências concêntricas de raios $R20$ e $R14$.
2. Desenhar uma linha vertical para baixo partindo do quadrante esquerdo da circunferência de raio maior com 24 unidades de comprimento.
3. Desenhar uma linha horizontal com 8 unidades de comprimento para a esquerda a partir da extremidade 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.

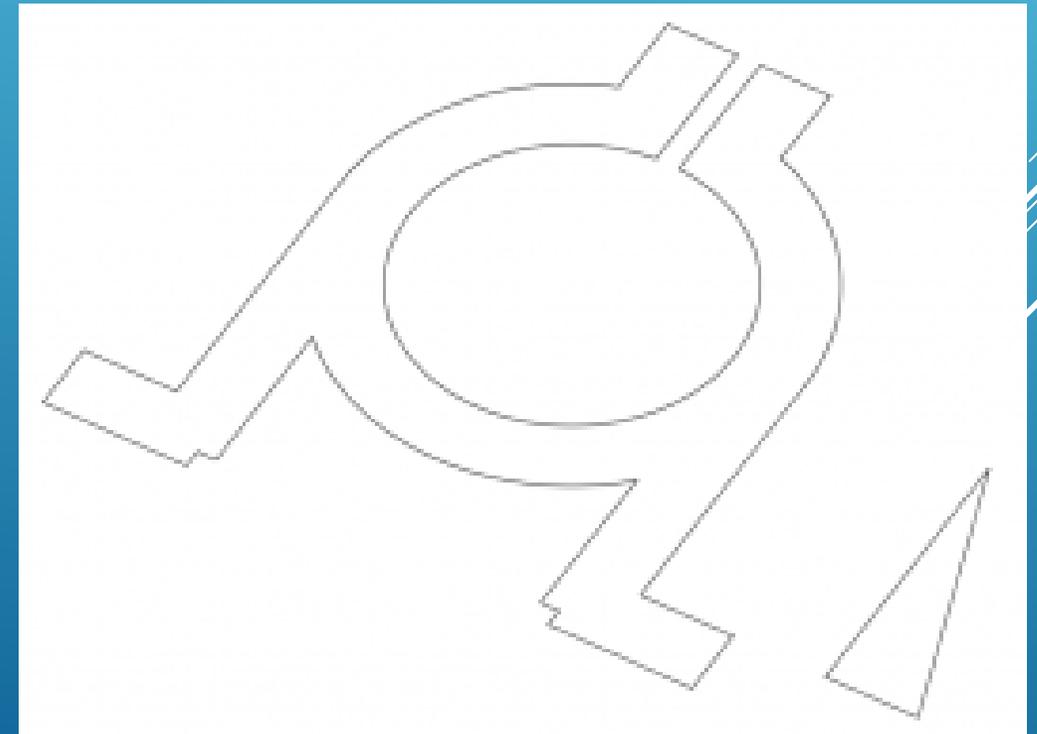


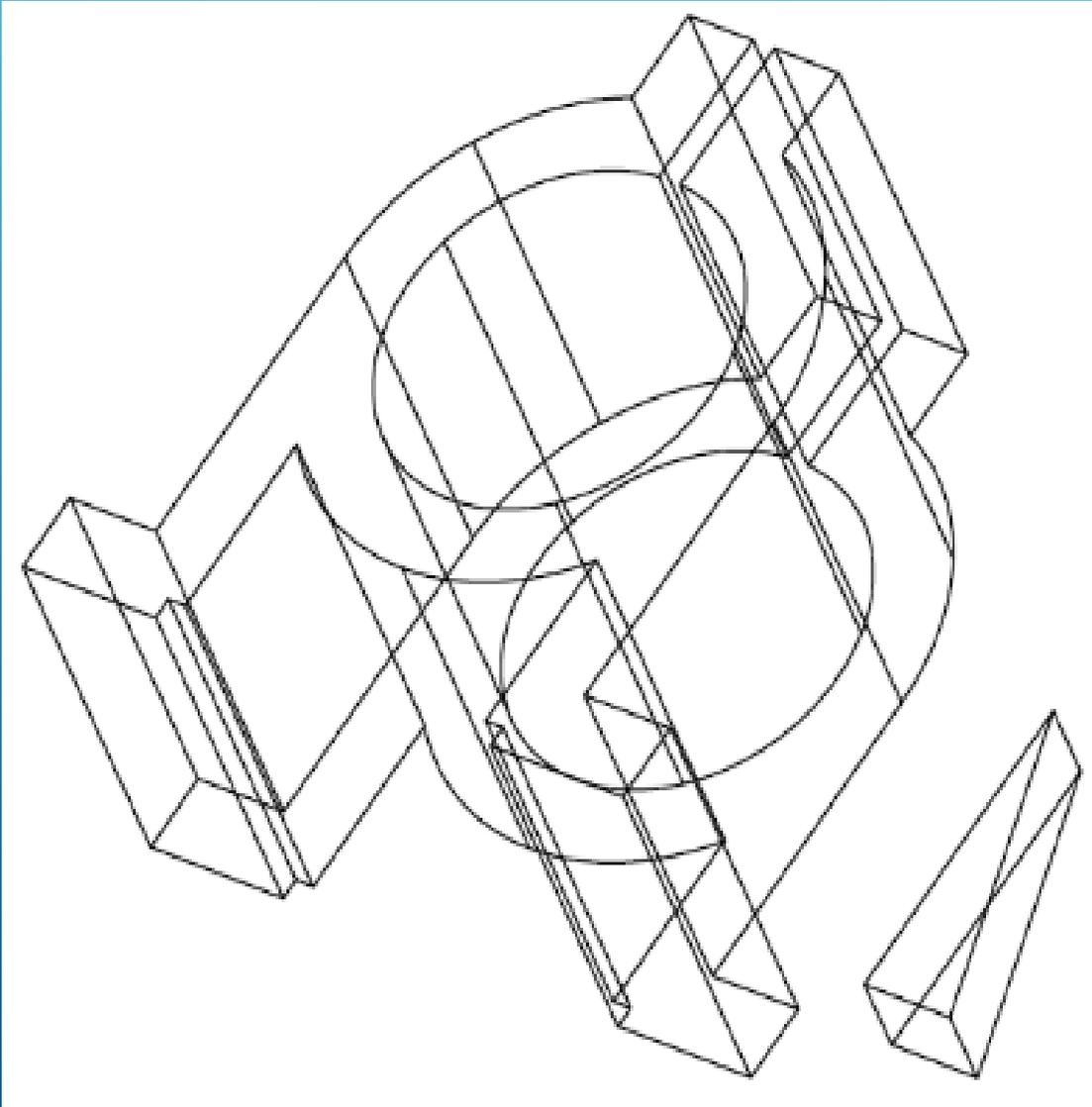


18. Desenhar o triângulo lateral, que tem o lado vertical com 24 unidades de comprimento e o lado horizontal com 8 unidades de comprimento.

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.

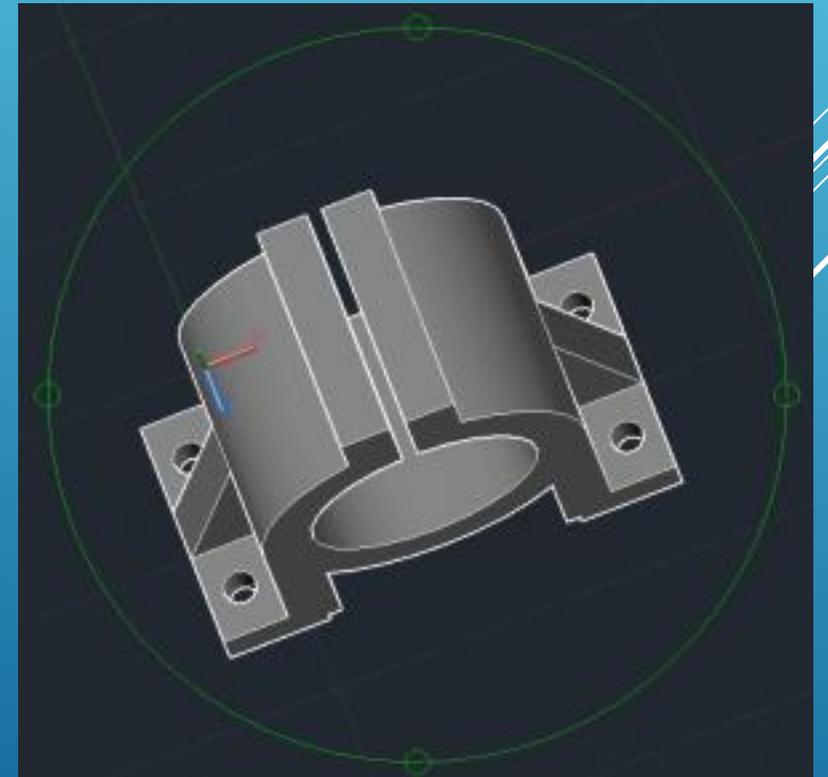
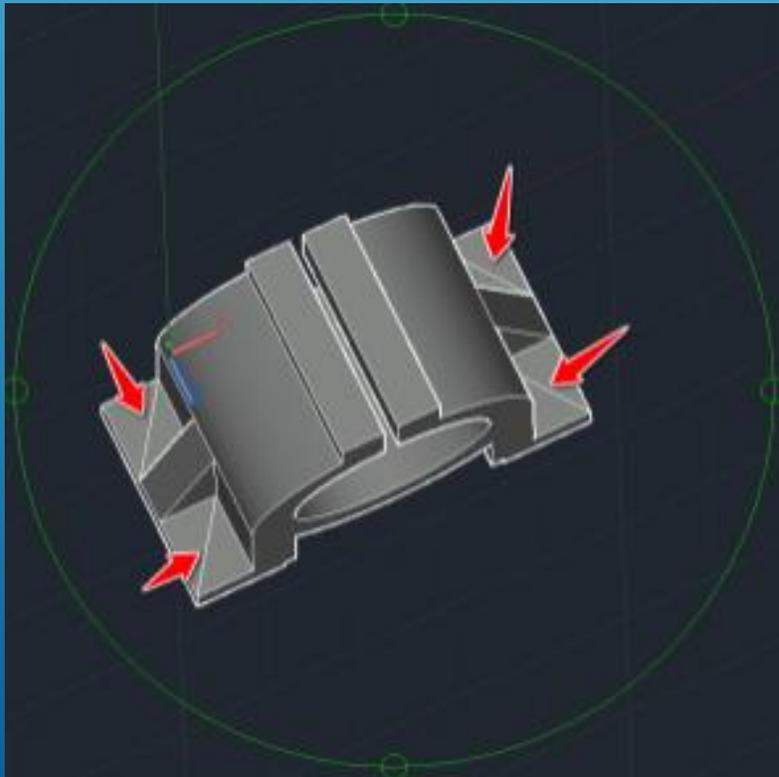
22. Na vista Top, utilize o comando Mirror para copiar o triângulo para o lado esquerdo do corpo principal e volte à vista anterior.

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.

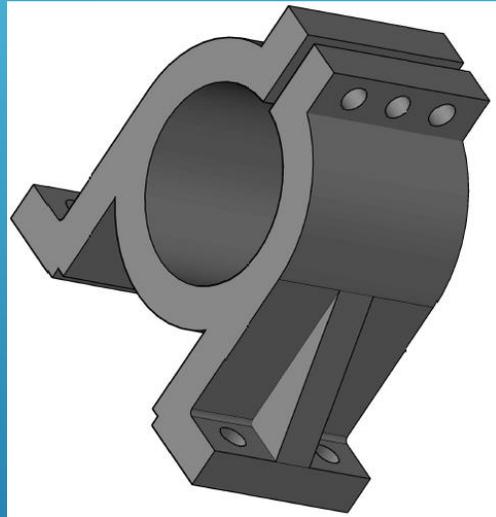
24. Utilize o comando Union, seleccione os 3 objectos e carregue em Enter.



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.



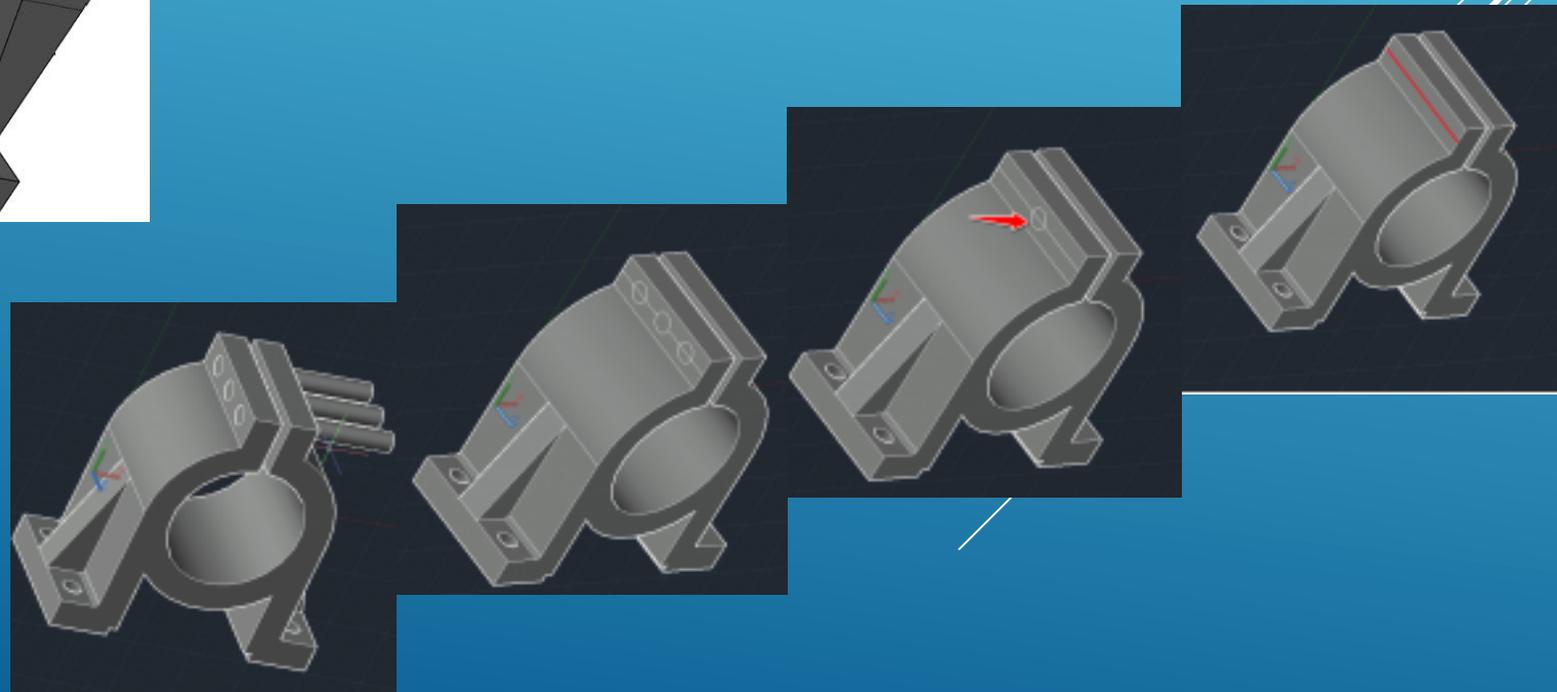
30. Desenhe a linha indicada na figura com snap Midpoint.

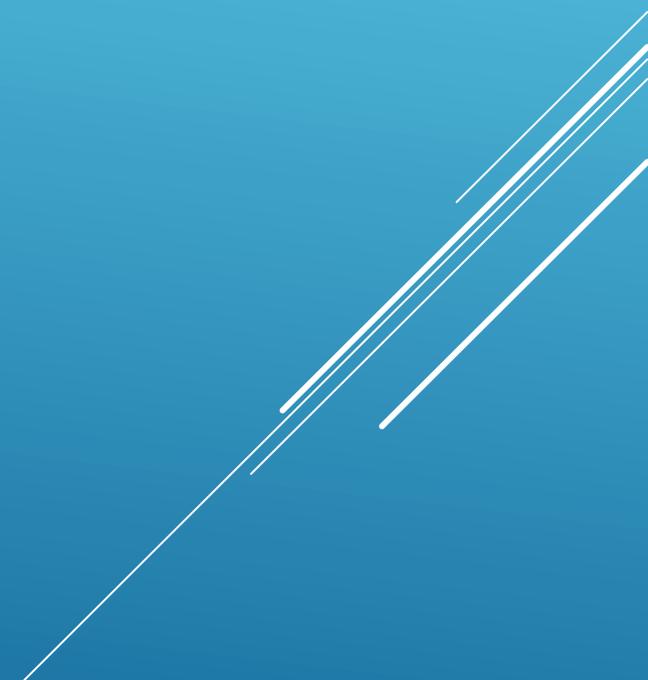
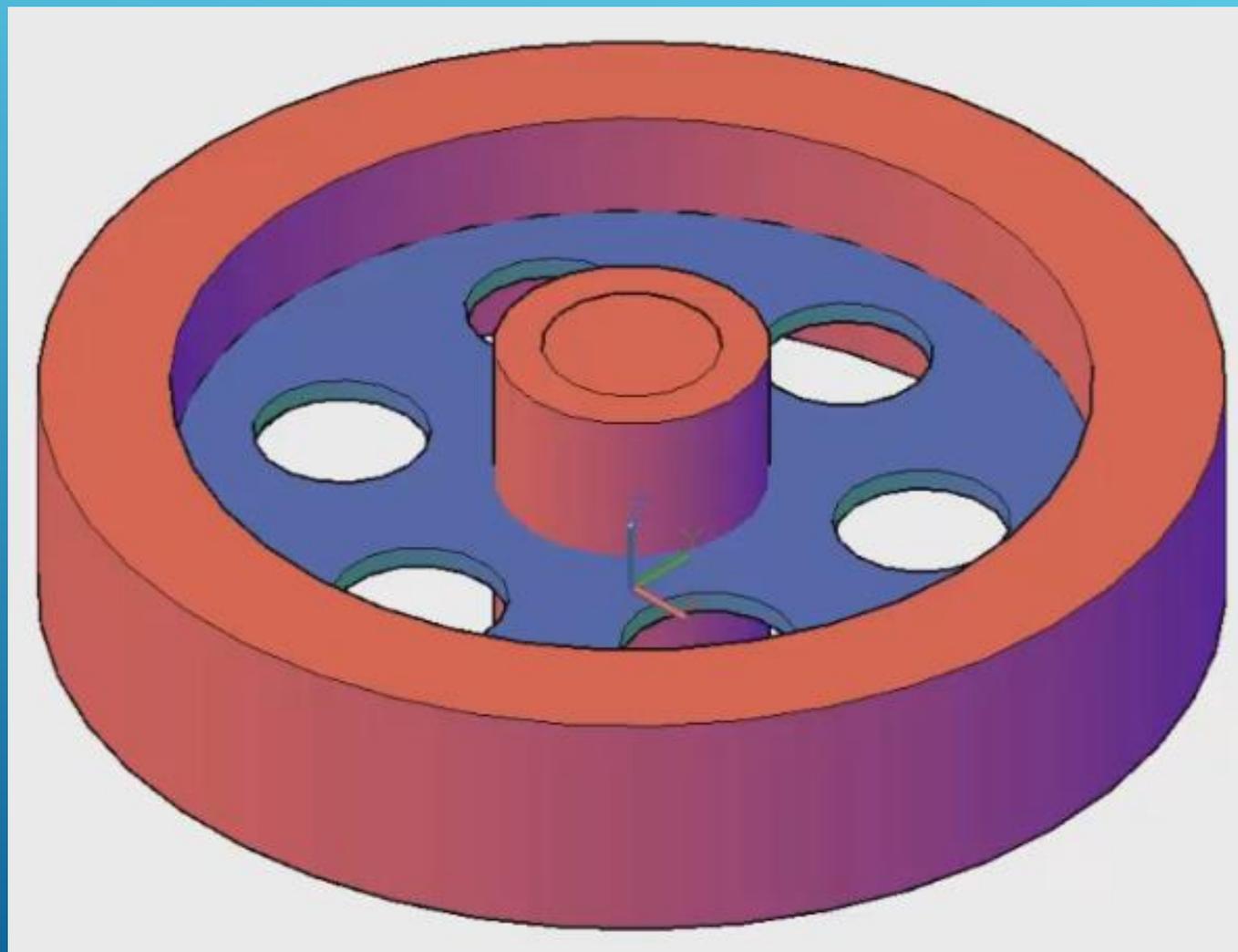
31. Criar circunferências de raio 2 unidades usando o centro do segmento anterior.

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.



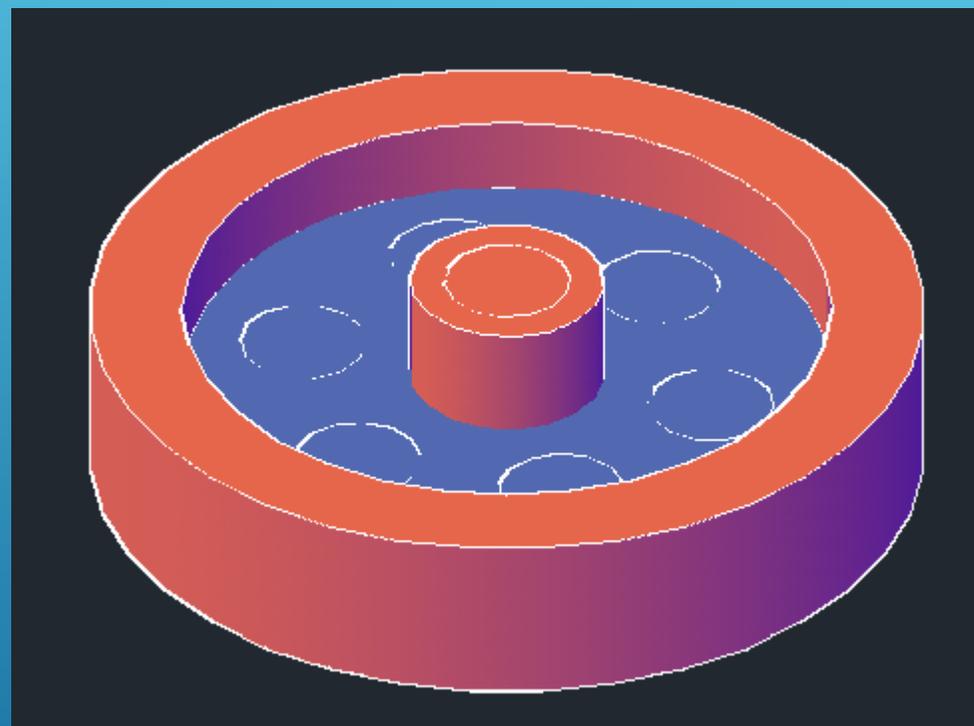
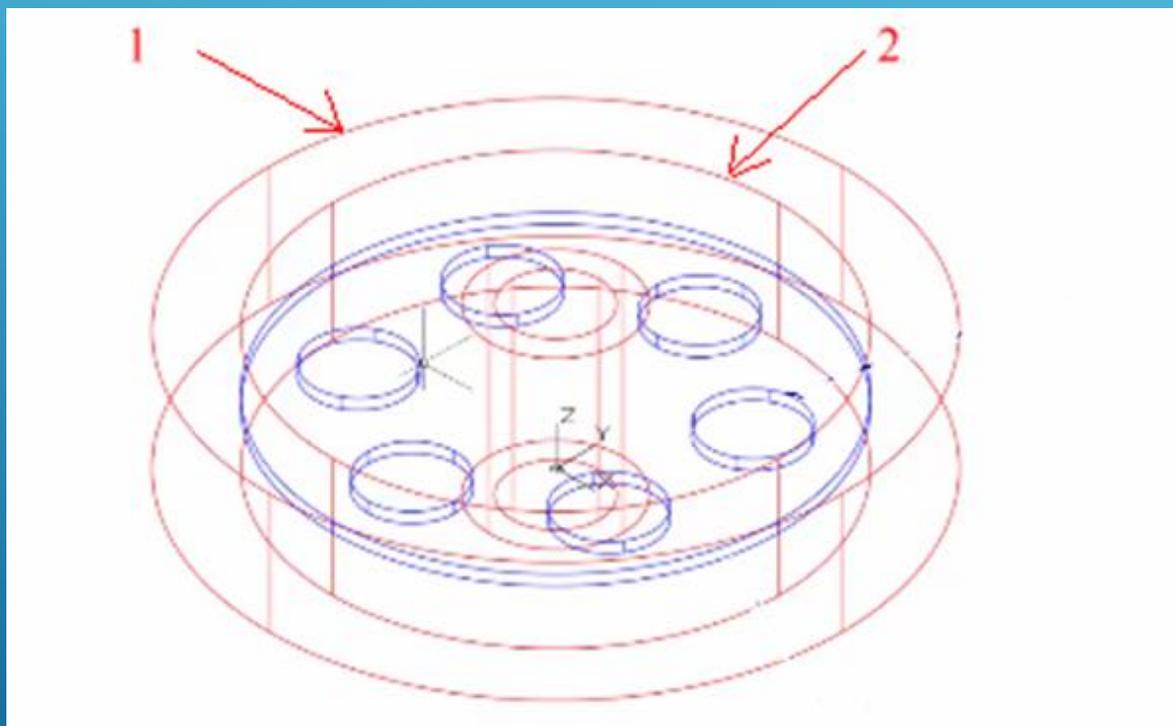


1. New Drawing
2. Criar 2 layers: Encarnado e Azul
3. Ambiente de trabalho: 3D Modeling
4. 3dWireframe
5. Solid Tab > Cylinder > Center point: **(0,0,0)** + enter
Radius: **430** + enter
Height: **180** + enter
6. Home Tab > View Panel > 3D Navigation > Views > SE Isometric
7. Snap Center
8. Solid Tab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

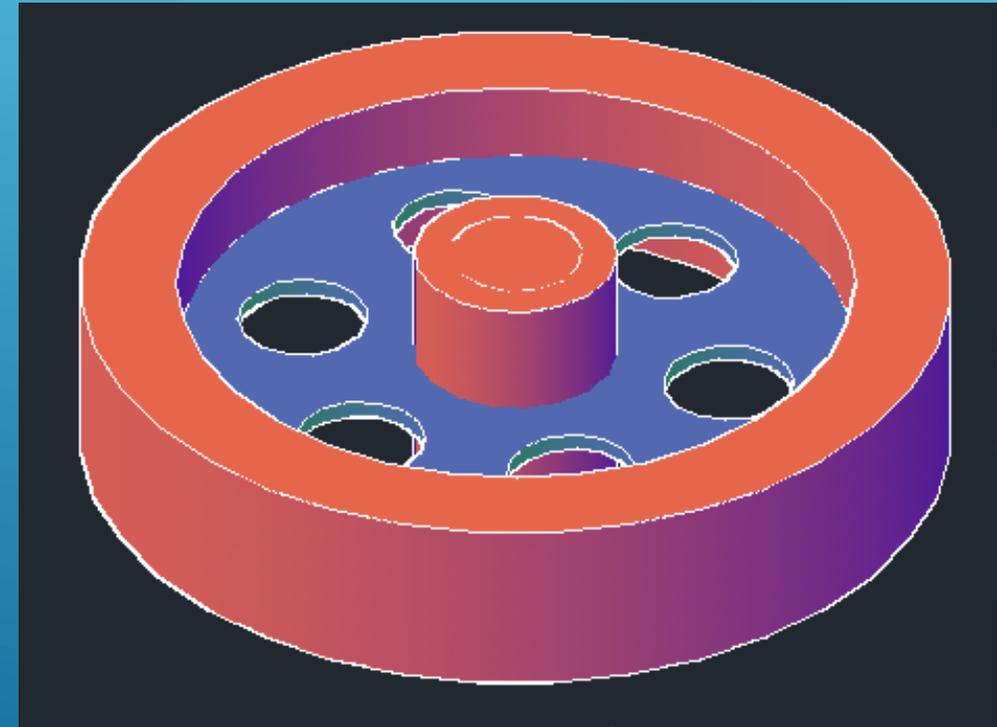
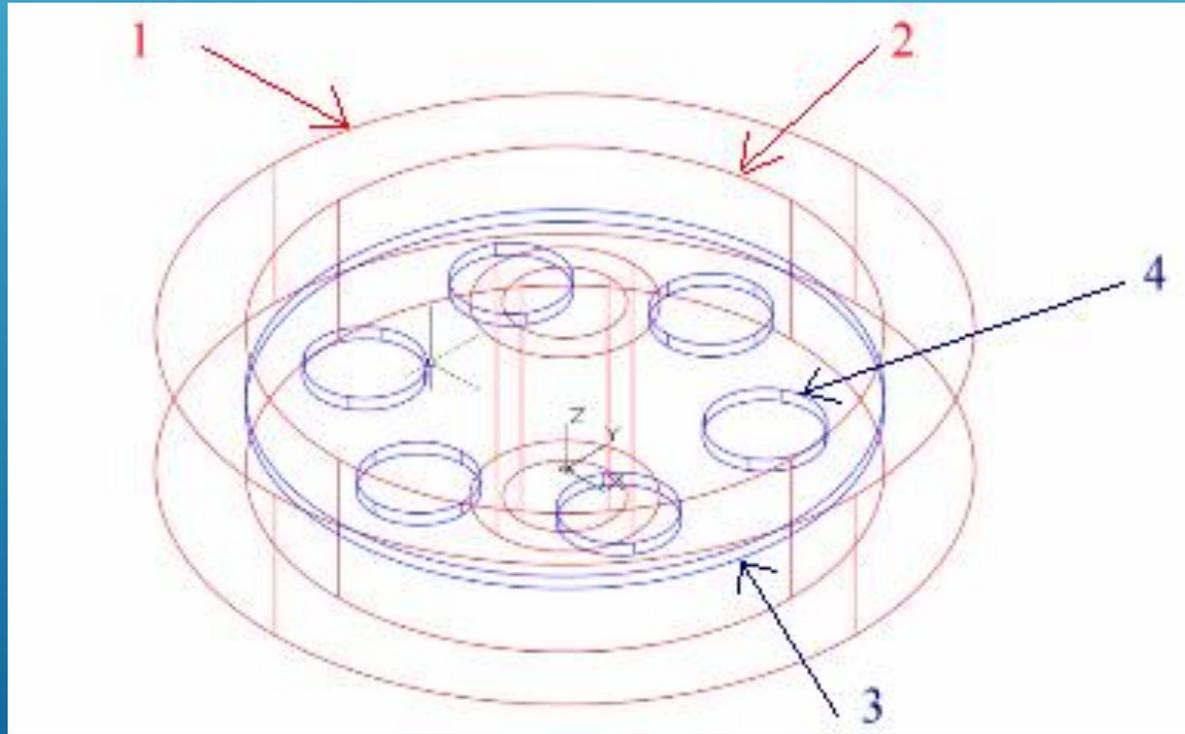
1

2

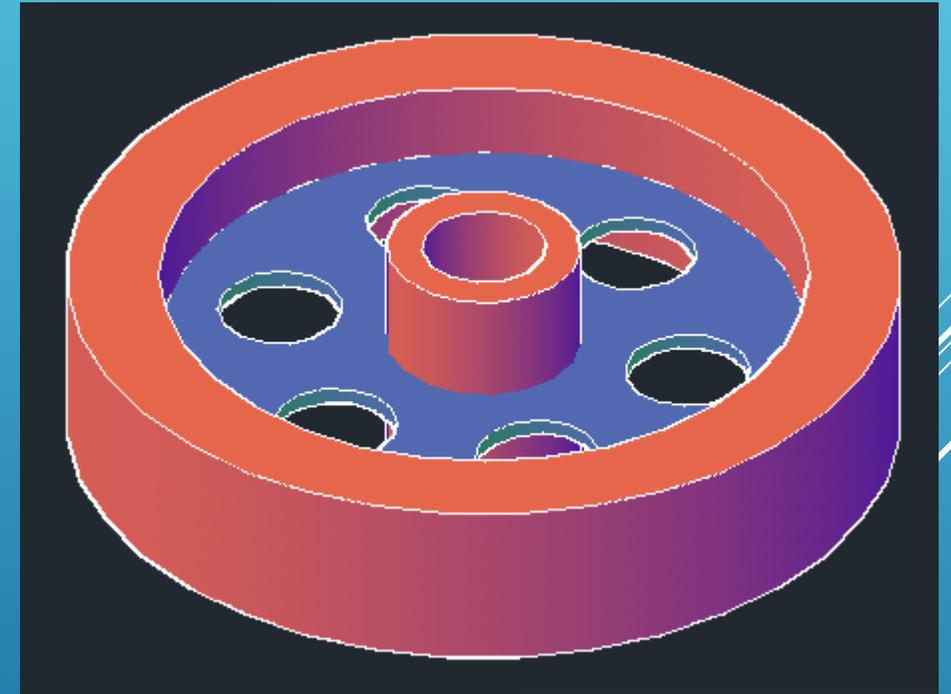
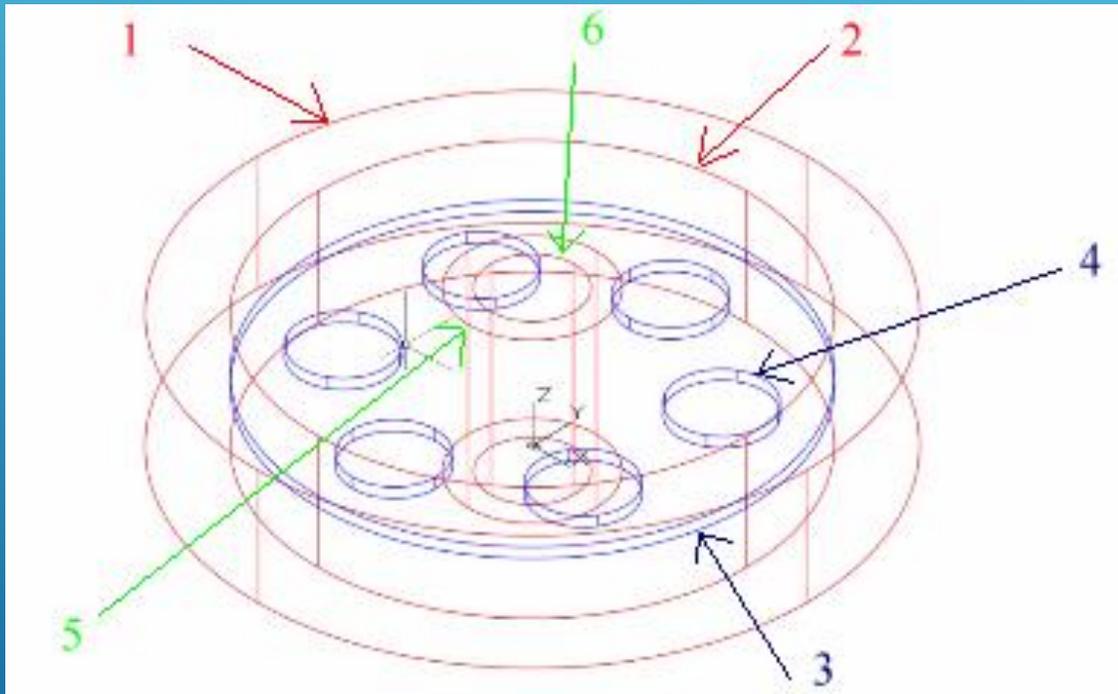
18. Solid Tab > Boolean > Subtract



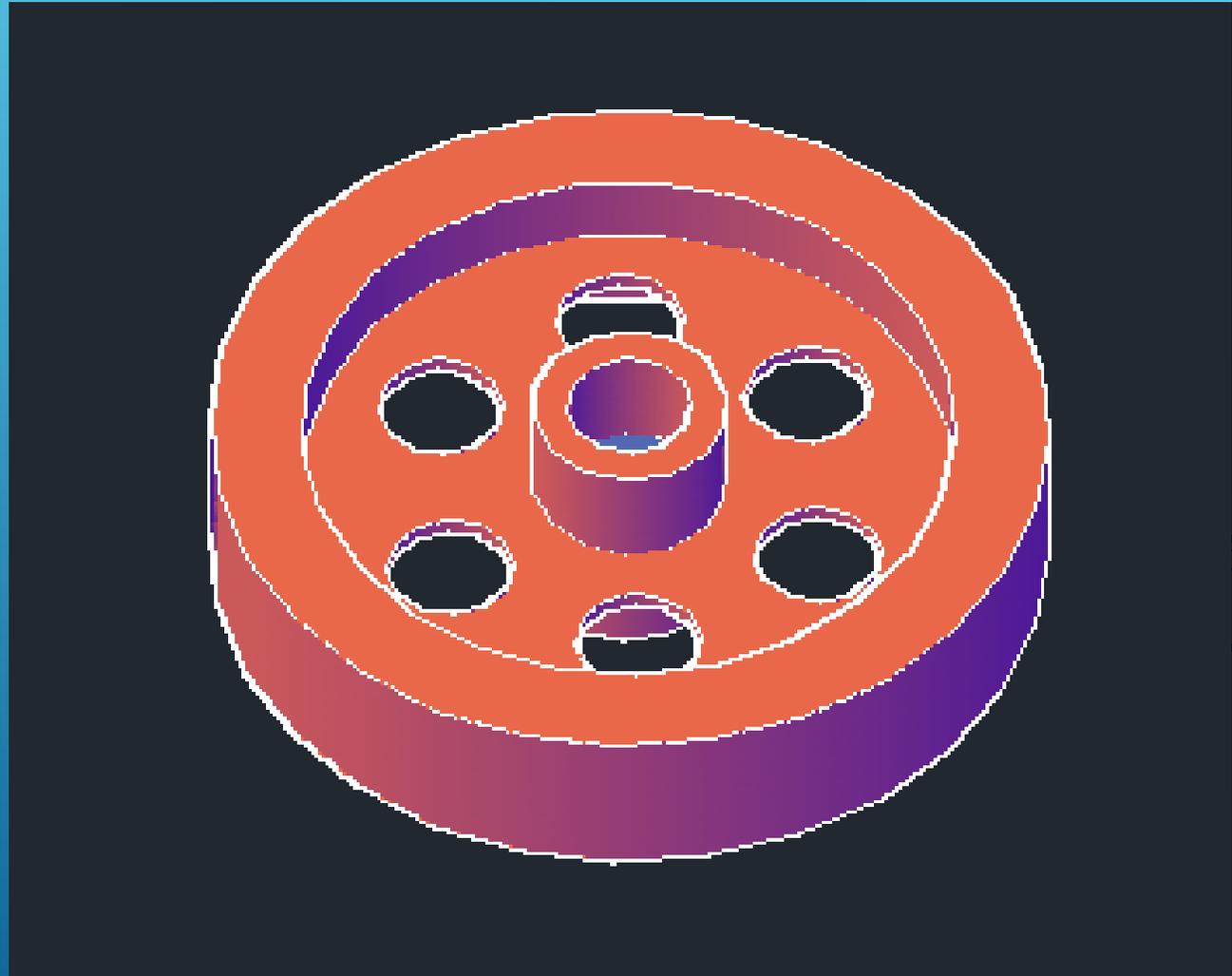
19. Solid Tab > Boolean > Subtract

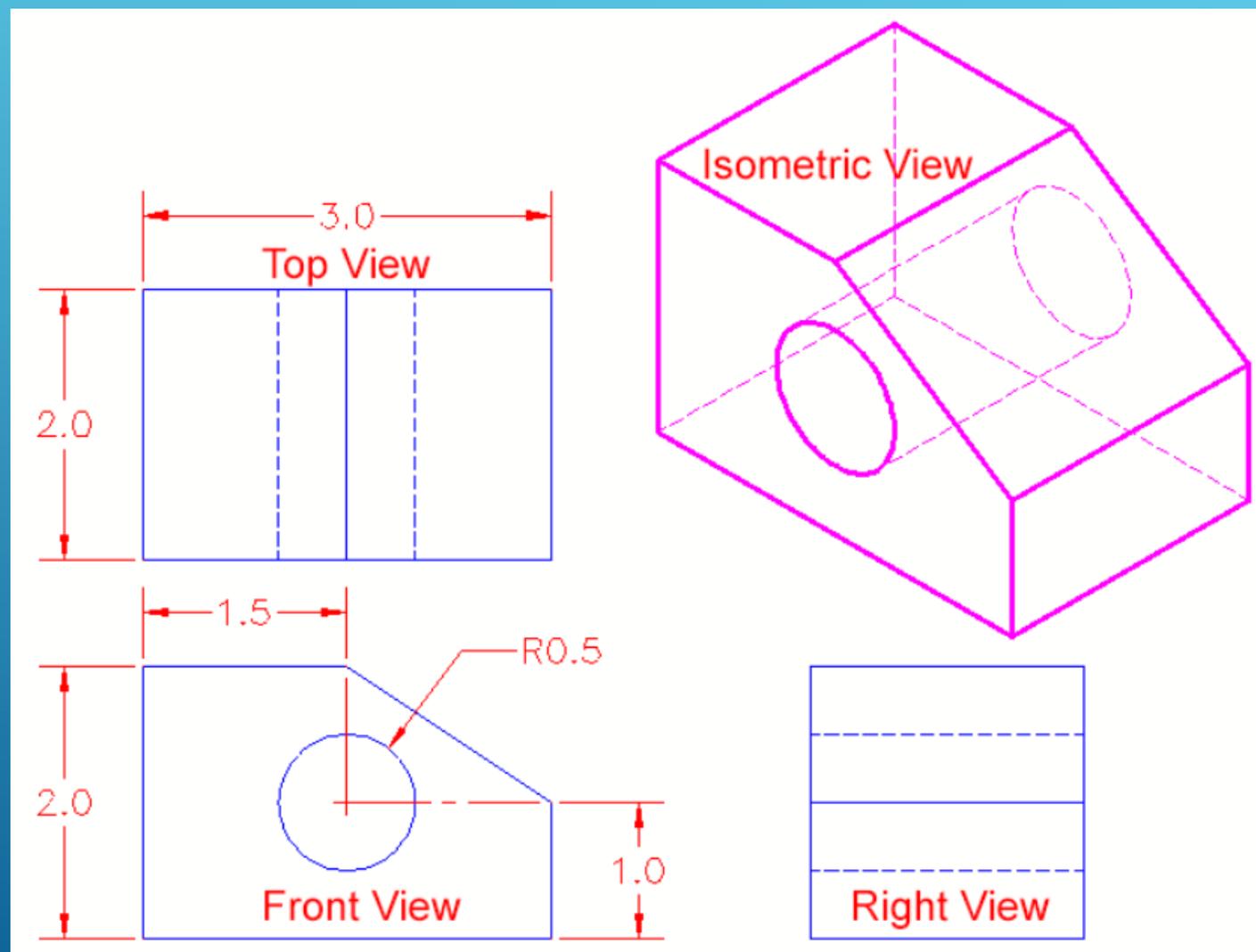


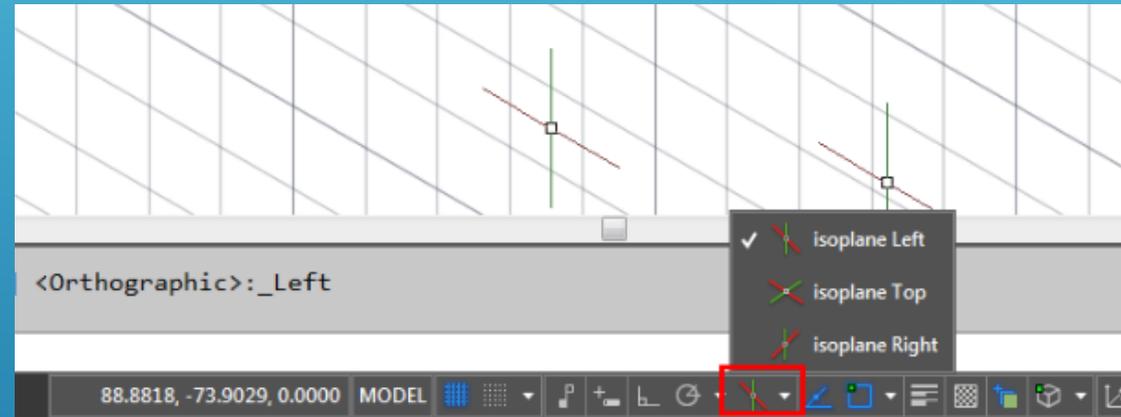
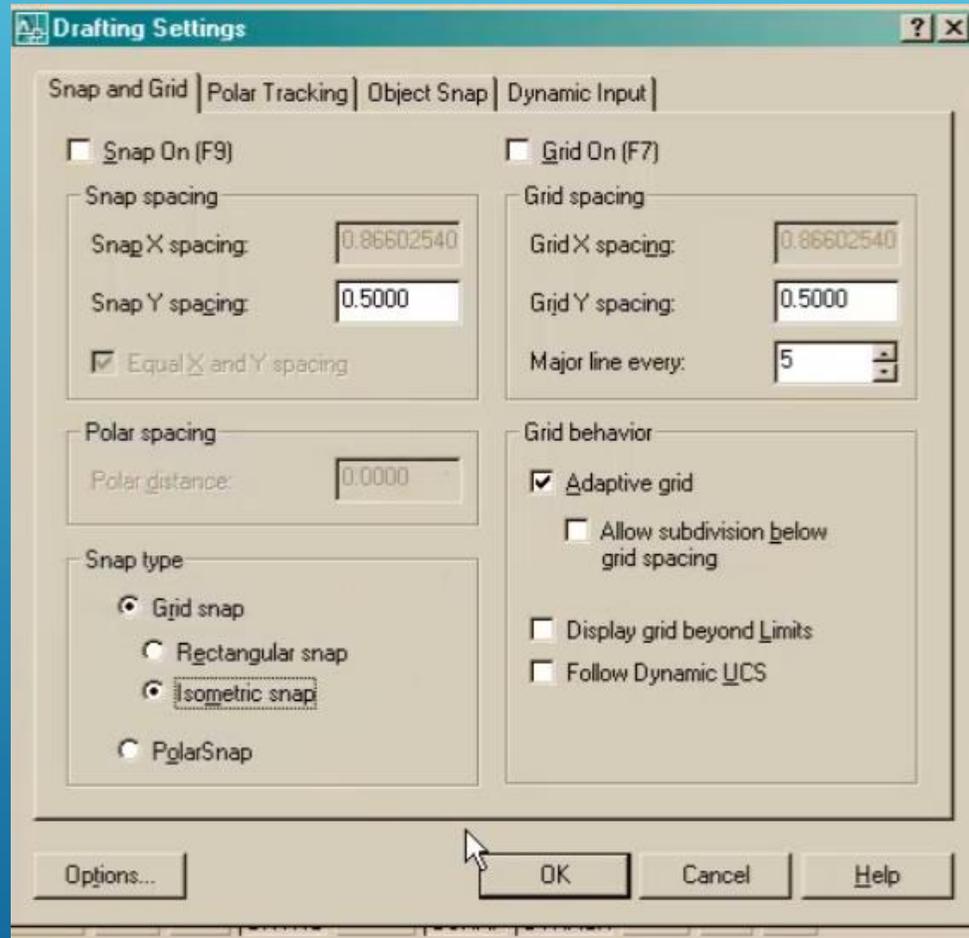
20. Solid Tab > Boolean > Subtract



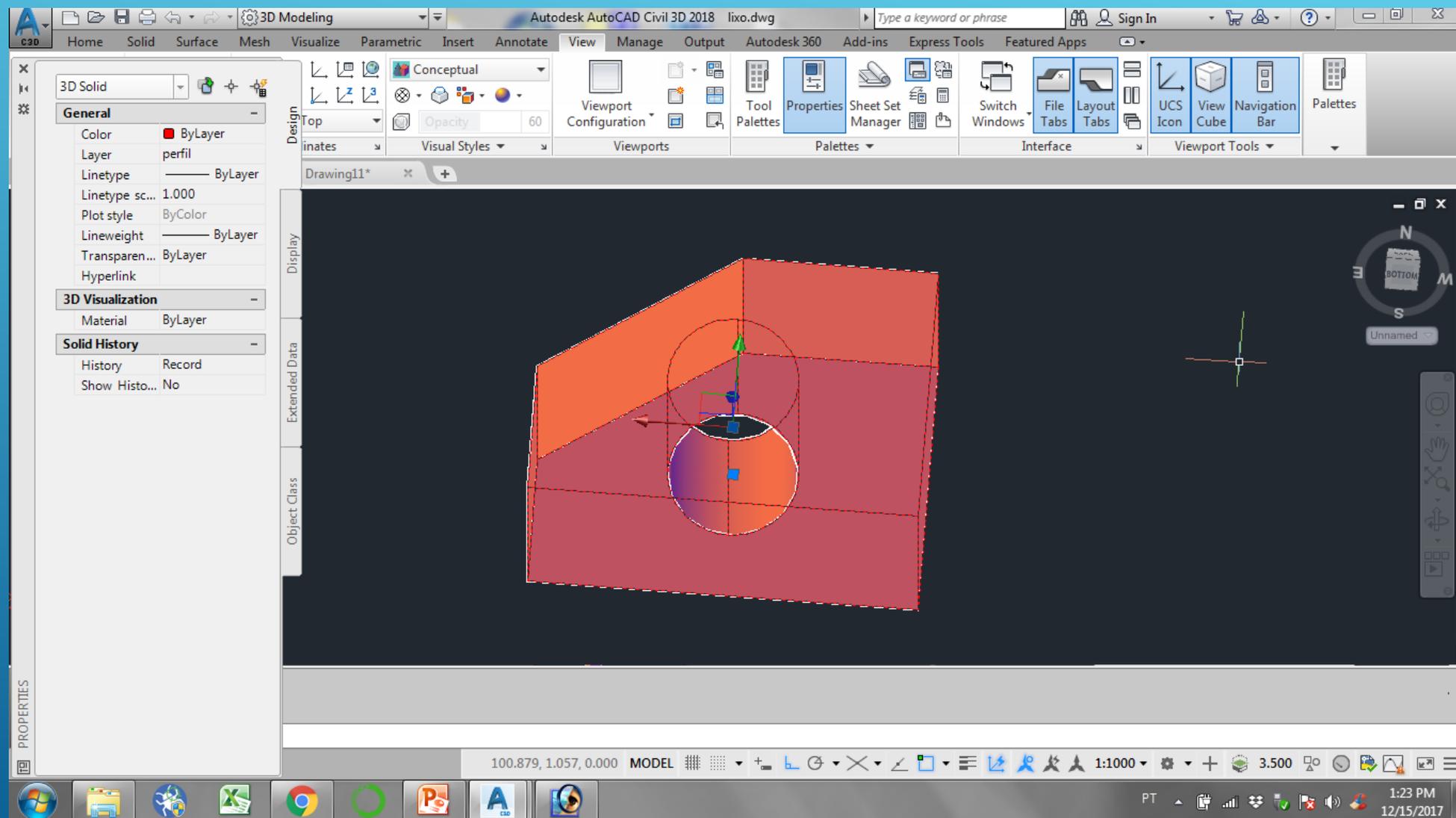
20. Solid Tab > Boolean > Union

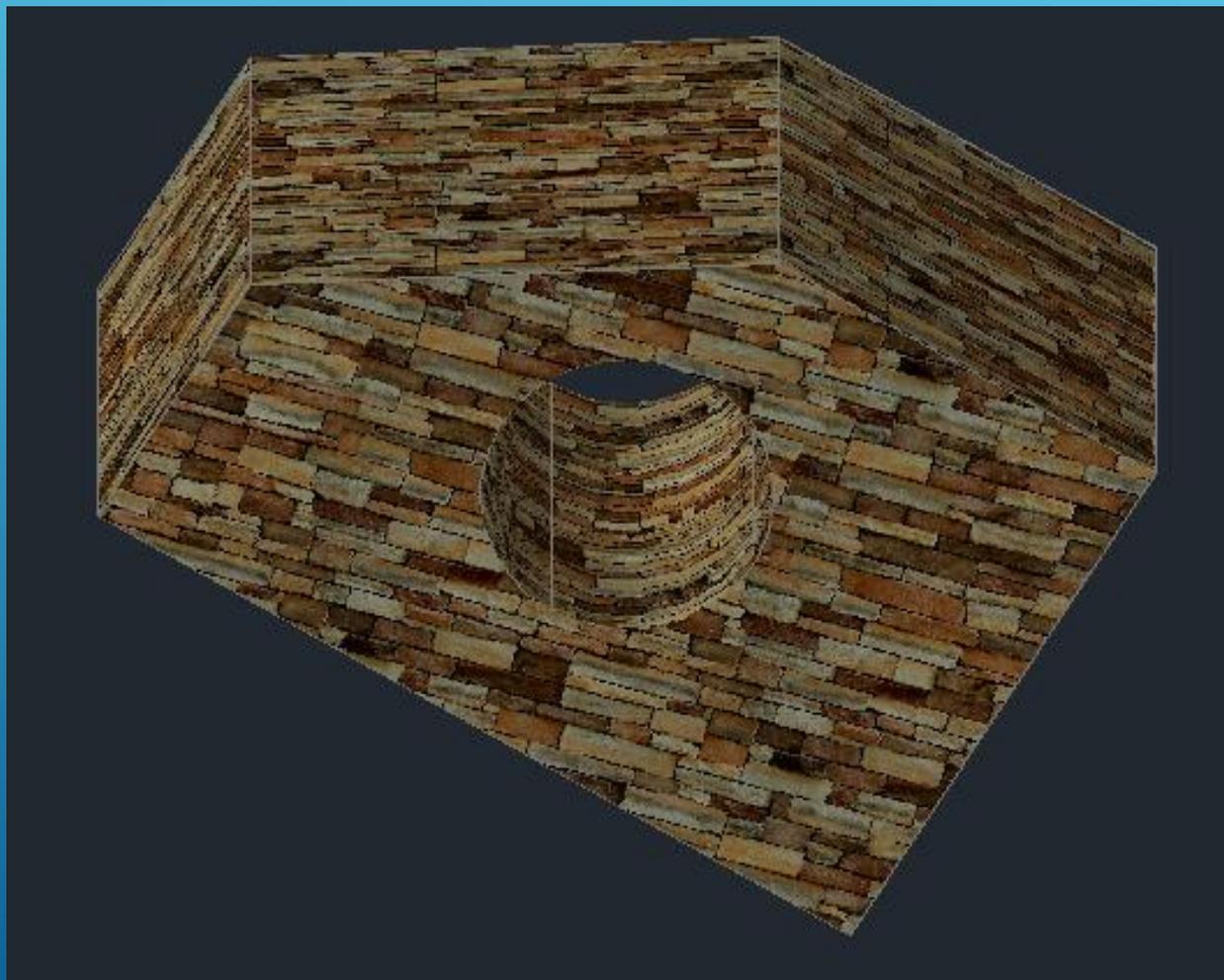




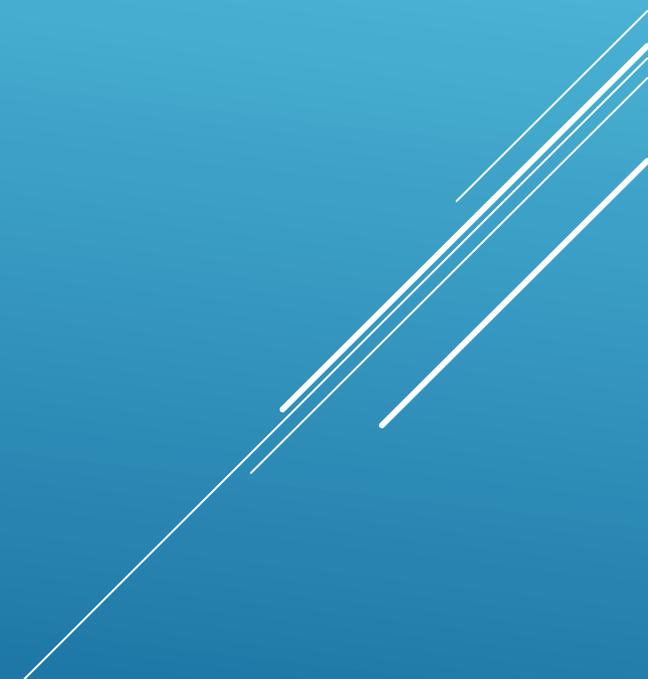
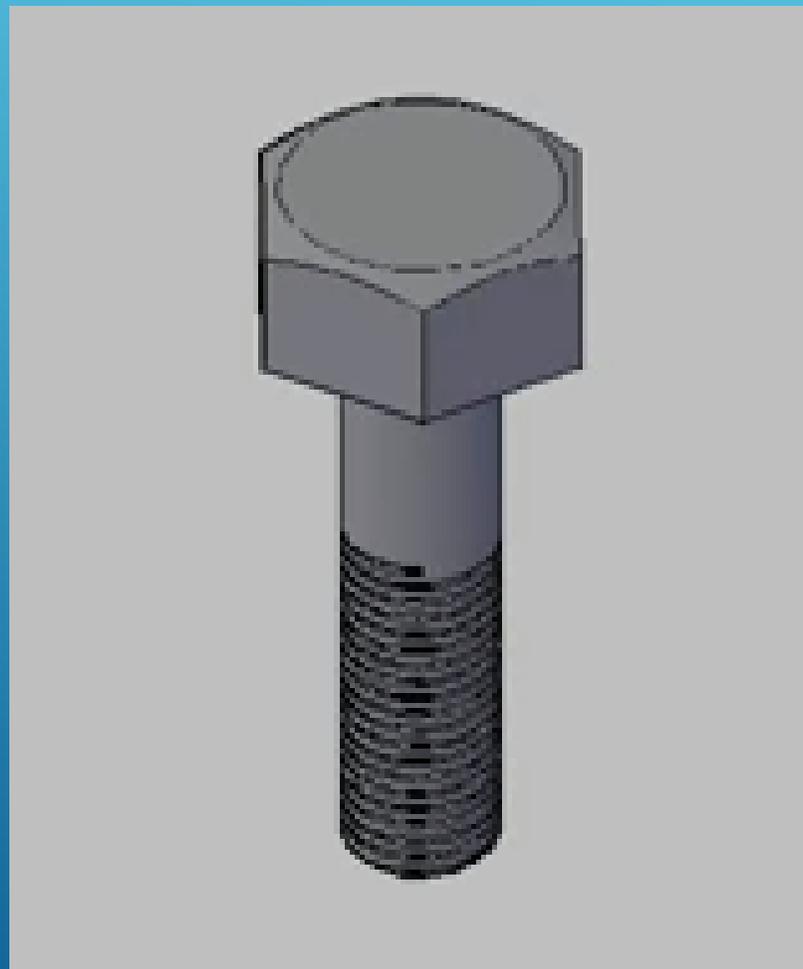


AULA 5 Desenho Técnico Assistido por Computador





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



The screenshot shows the AutoCAD software interface with the 'Draw' tab selected. A red box highlights the 'Polygon' icon in the 'Draw' panel. An arrow points from this icon to the 'Polygon' help window. The help window contains the following text:

Polygon
Creates an equilateral closed polyline

You can specify the different parameters of the polygon including the number of sides. The difference between the inscribed and circumscribed options is shown.

The diagrams show two hexagons. The left one is inscribed in a circle, with a green asterisk labeled '*1' at the center and a green 'x' labeled '2' at a vertex. The right one is circumscribed about a circle, with a green asterisk labeled '*1' at the center and a green 'x' labeled '2' at a vertex.

POLYGON
Press F1 for more help

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

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

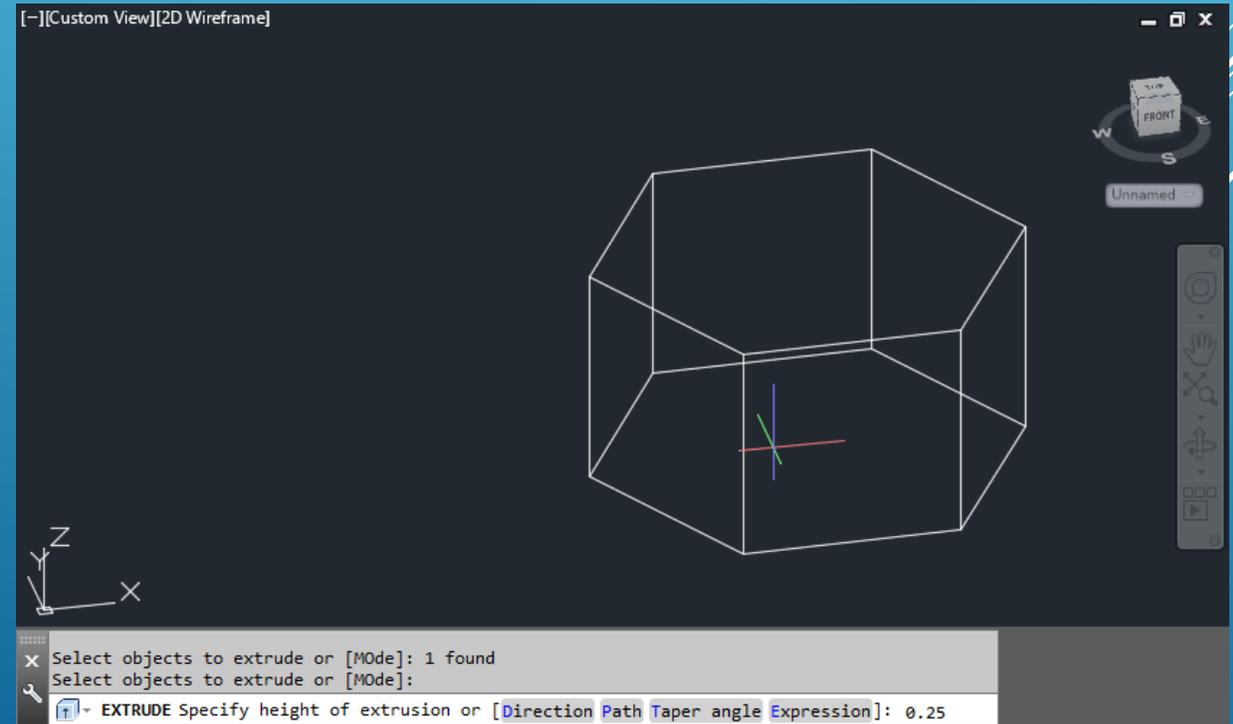
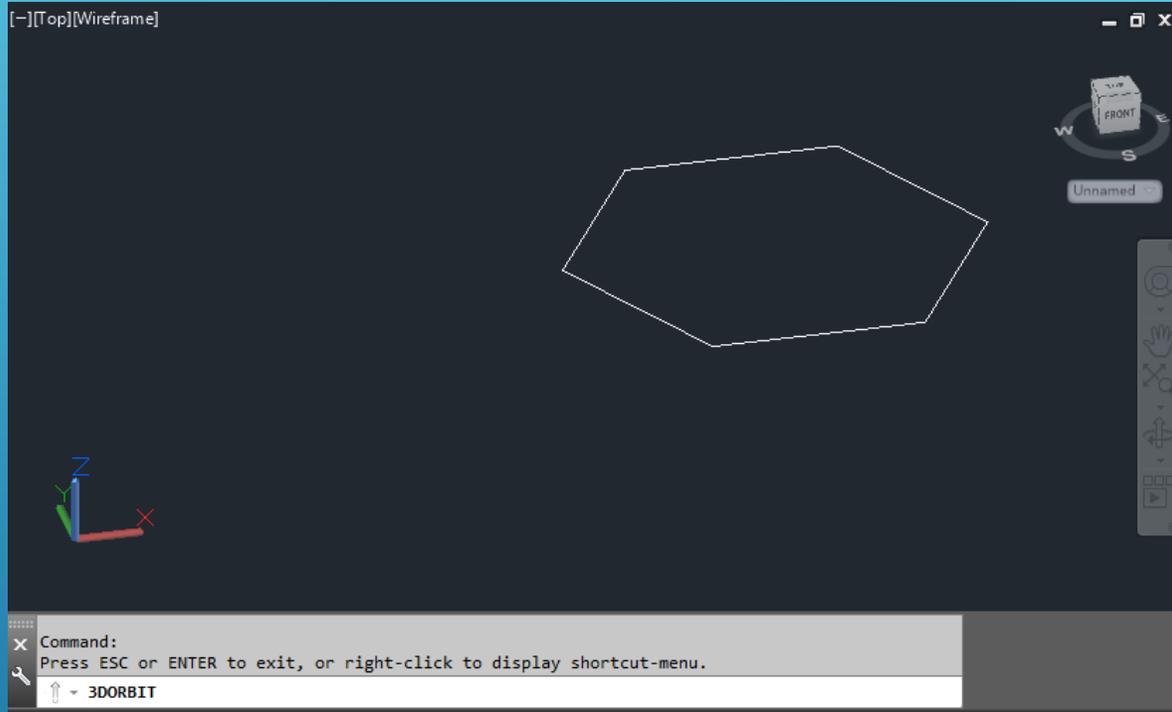
```
Specify center of polygon or [Edge]:
Point or option keyword required.
POLYGON Specify center of polygon or [Edge]:
```

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

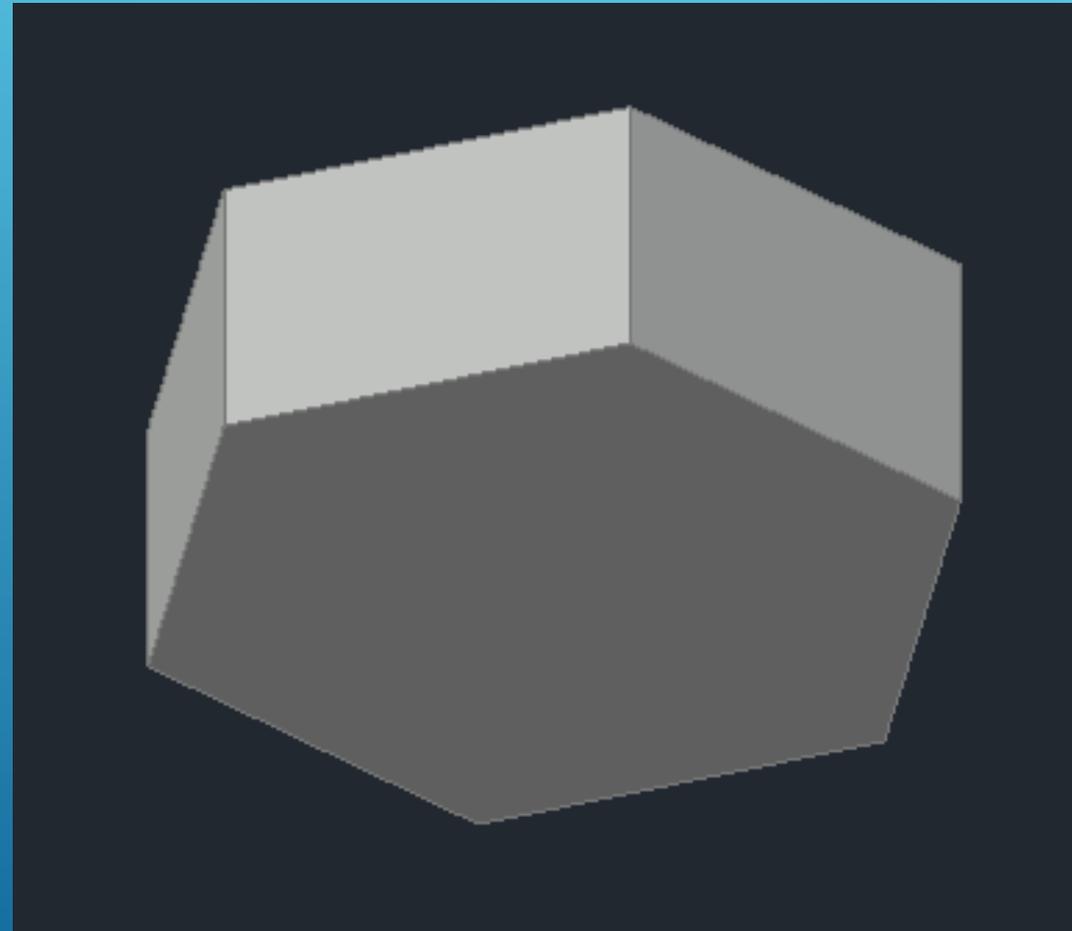
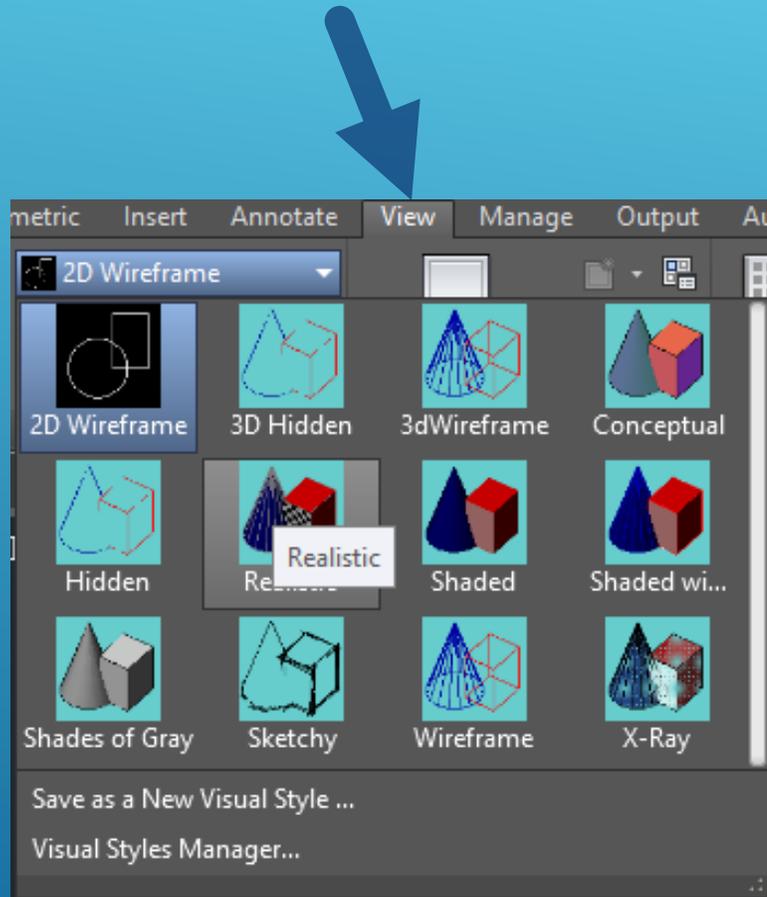
```
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <I>:
POLYGON Specify radius of circle: 0.375|
```



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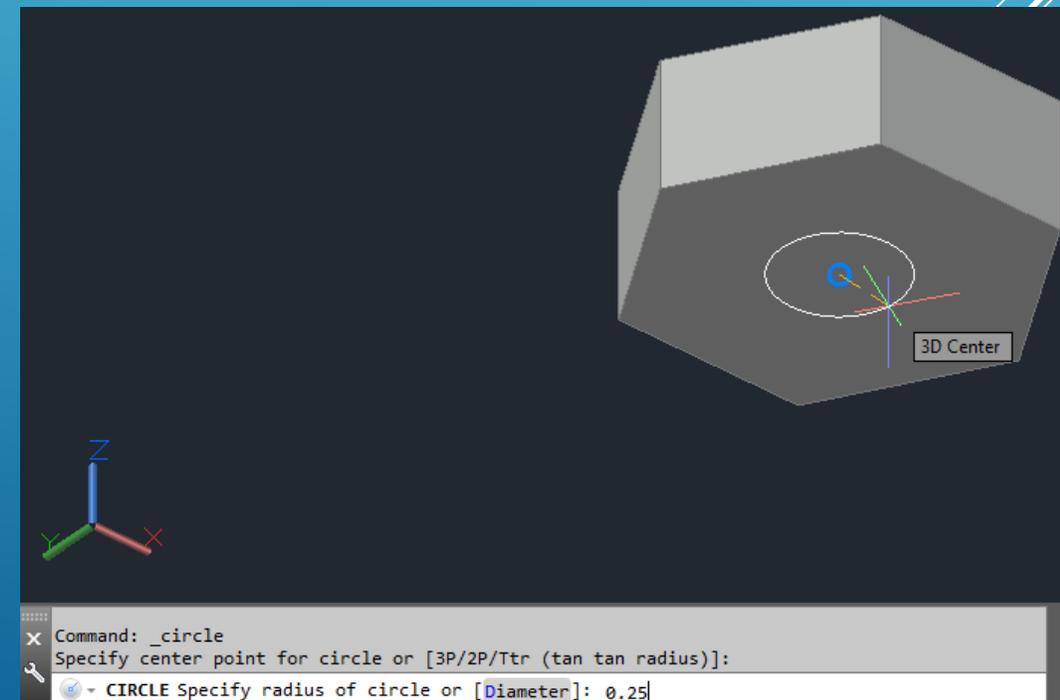
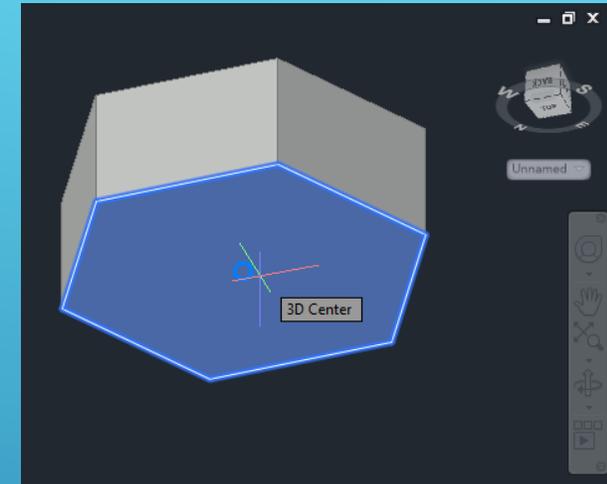
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- ✓ Dynamic Input
- ✓ Ortho Mode
- ✓ Polar Tracking
- ✓ Isometric Drafting
- ✓ Object Snap Tracking
- ✓ 2D Object Snap
- ✓ LineWeight
- Transparency
- Selection Cycling
- 3D Object Snap**
- Dynamic UCS
- Selection Filtering
- Gizmo
- ✓ Annotation Visibility
- ✓ 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

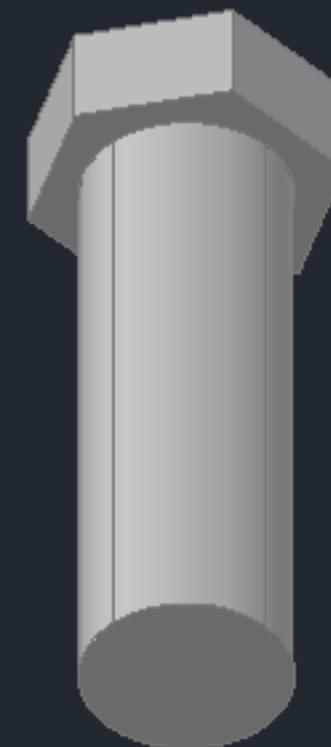
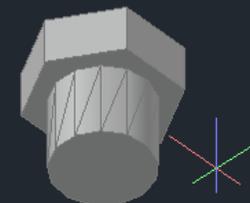
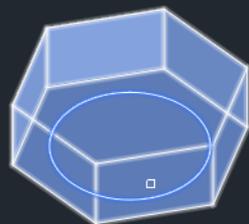


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```
.....  
x Select objects to extrude or [MOde]: _MO Closed profiles creation mode [Solid/SURface] <Solid>: _SO  
Select objects to extrude or [MOde]: 1 found
```

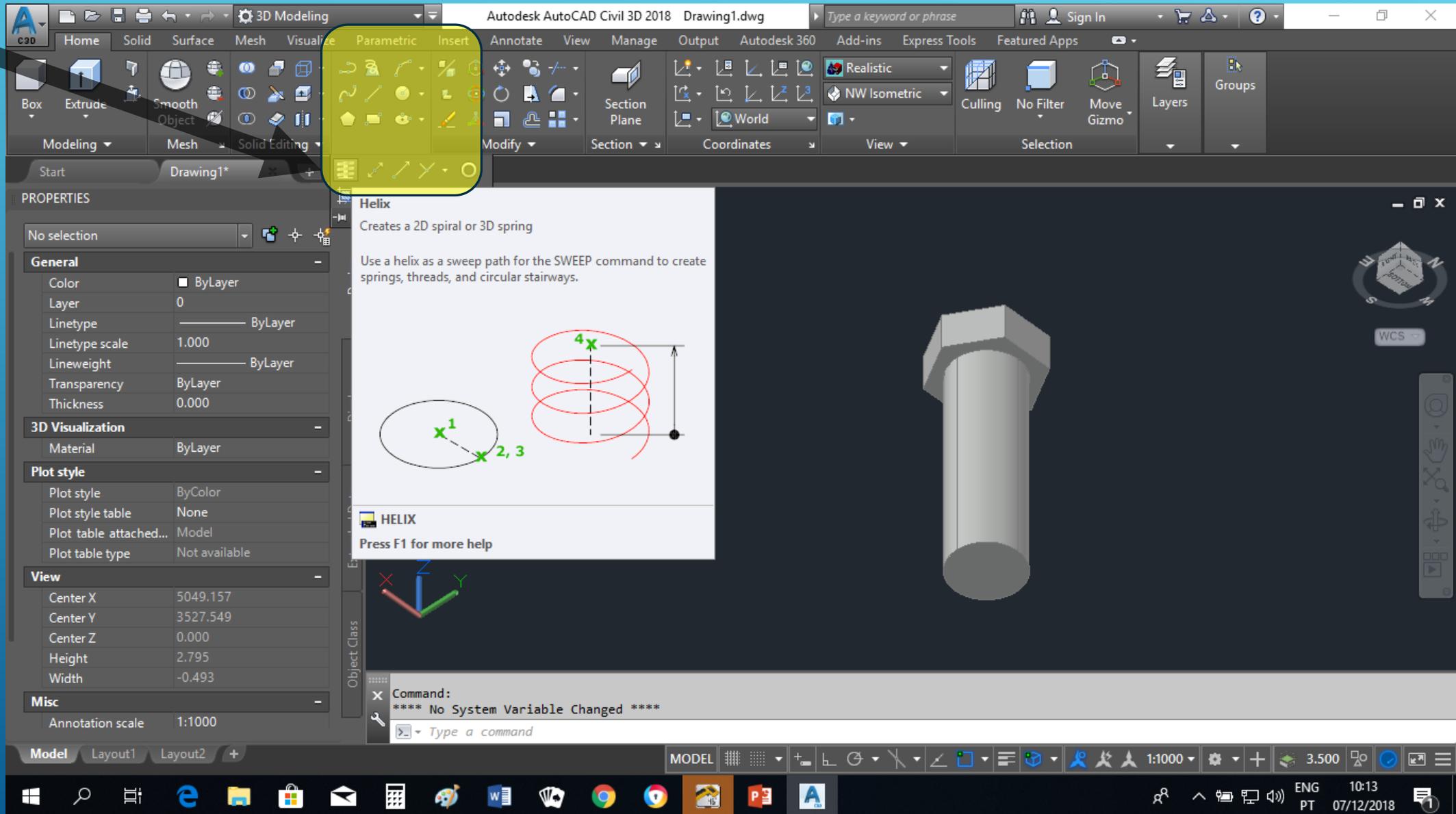
```
f EXTRUDE Select objects to extrude or [MOde]:
```



```
.....  
x Select objects to extrude or [MOde]: 1 found  
Select objects to extrude or [MOde]:
```

```
f EXTRUDE Specify height of extrusion or [Direction Path Taper angle Expression] <-0.150>: -1.5
```

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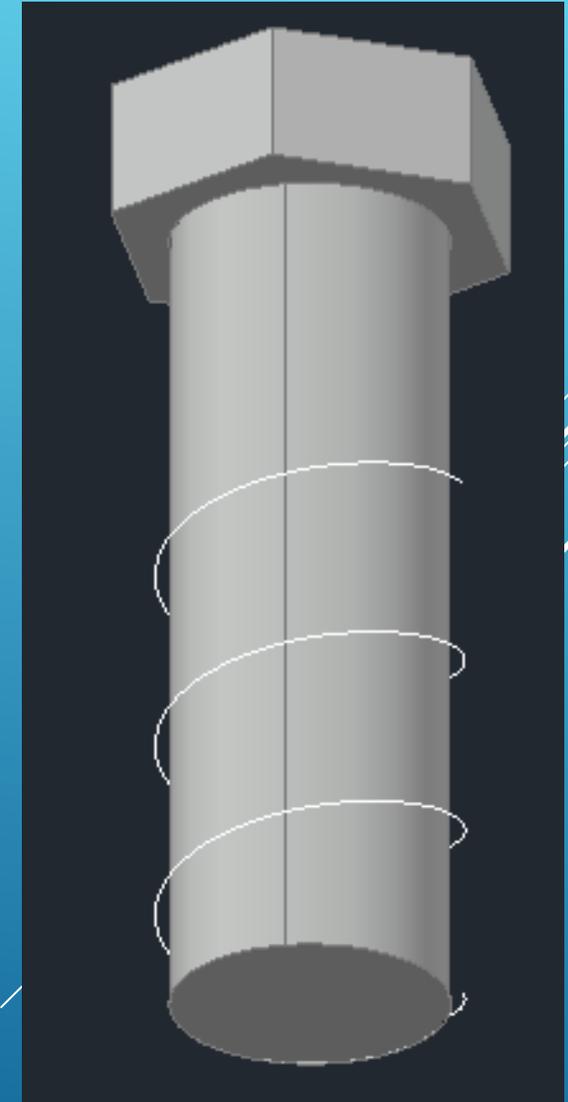
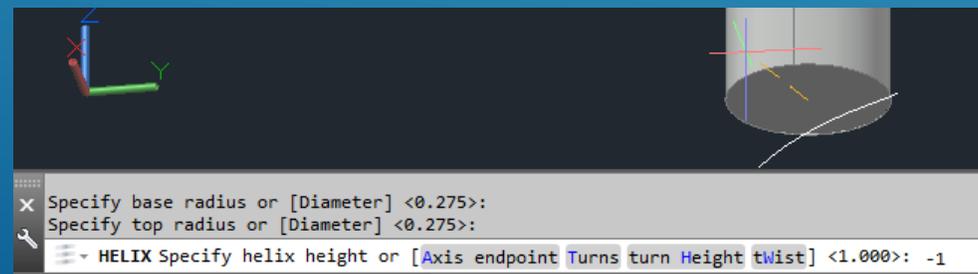
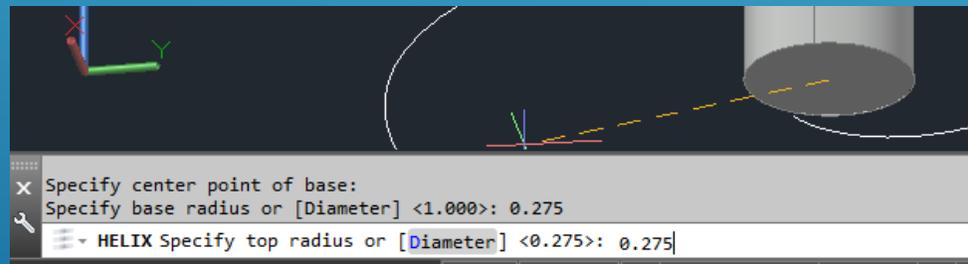
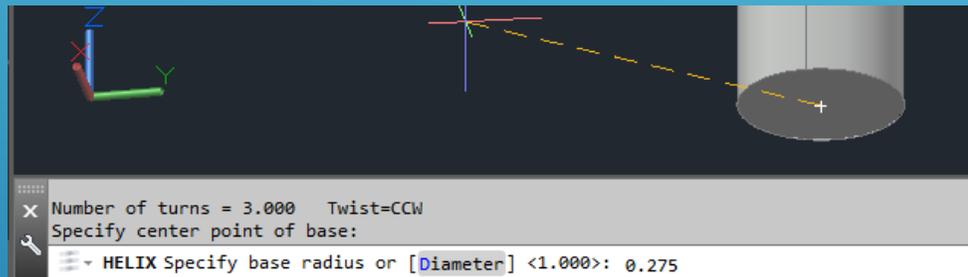
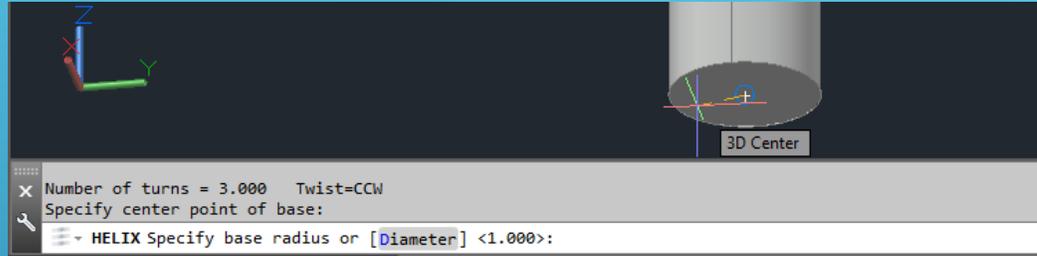
The screenshot displays the Autodesk AutoCAD Civil 3D 2018 software interface. The ribbon is set to the '3D Modeling' tab, with the 'Helix' command highlighted in the 'Parametric' panel. A help window for the 'Helix' command is open, providing instructions and a diagram. The diagram shows a 2D circle with a center point '1' and a radius line '2, 3', and a 3D helix with a height dimension '4'. A 3D model of a hexagonal bolt is visible in the background. The command line at the bottom shows the command being executed.

Helix
Creates a 2D spiral or 3D spring

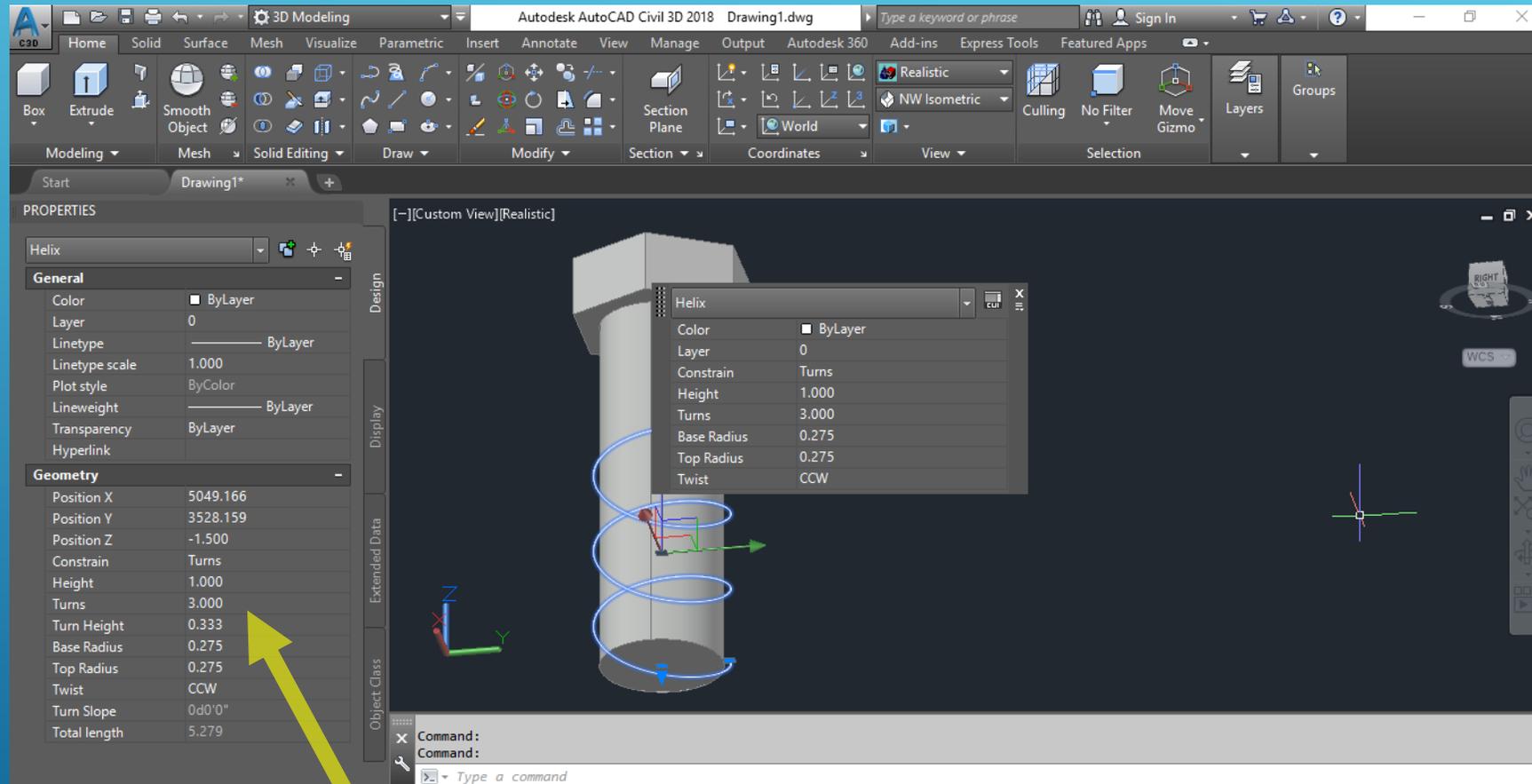
Use a helix as a sweep path for the SWEEP command to create springs, threads, and circular stairways.

HELIX
Press F1 for more help

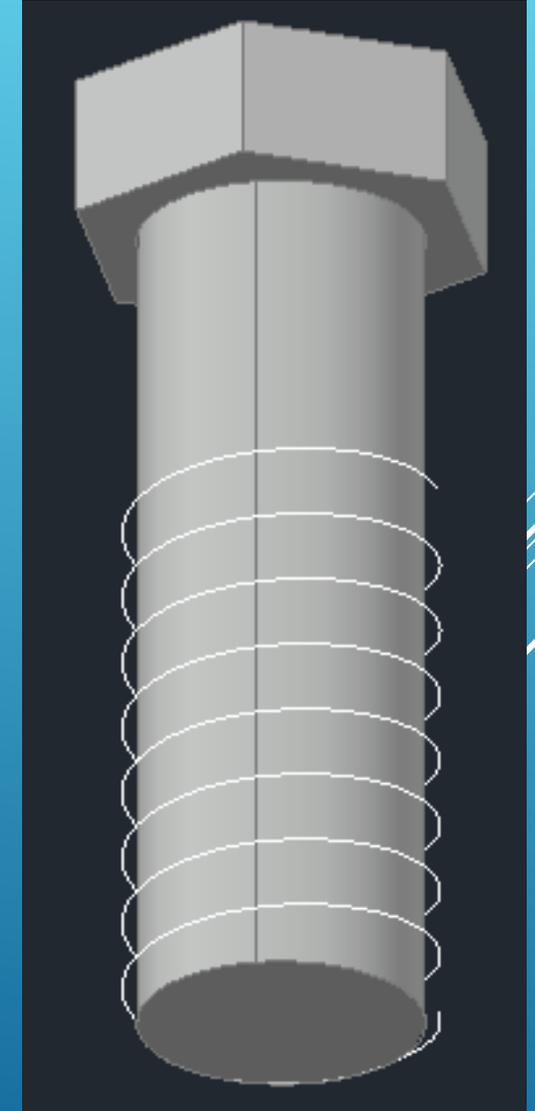
Command:
**** No System Variable Changed ****



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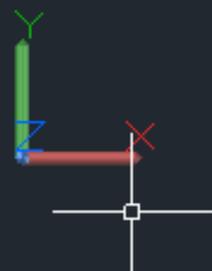
Turns: 8



[-] [Top] [Realistic]

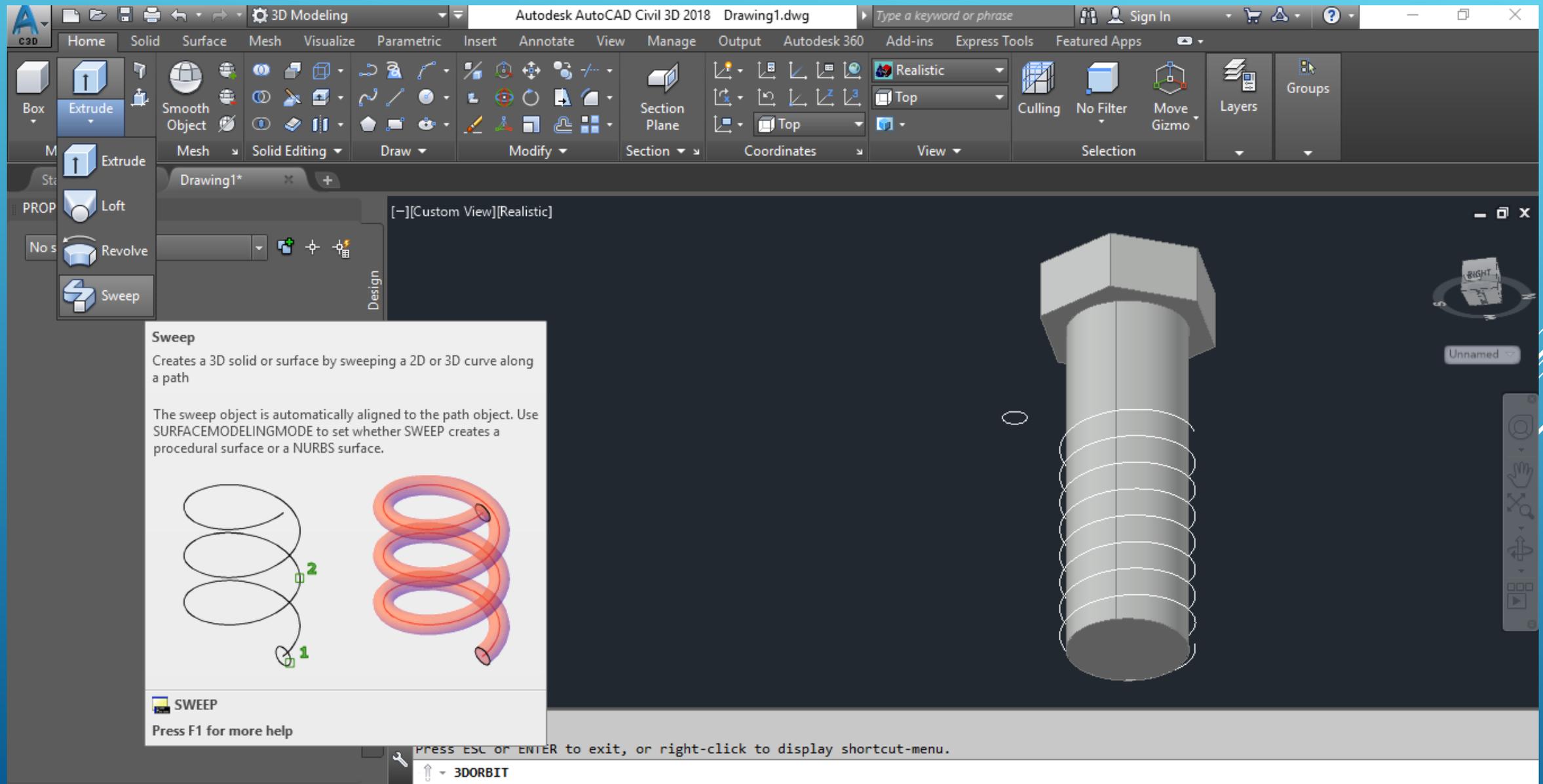


Unnamed



X Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <0.250>: 0.05

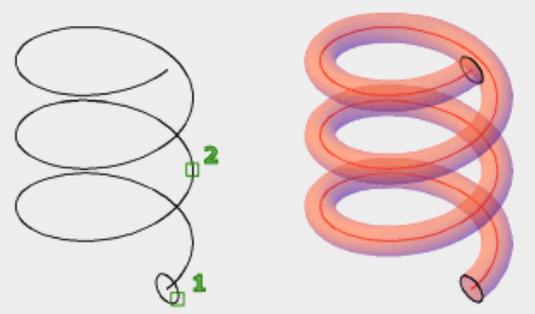
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The screenshot displays the Autodesk AutoCAD Civil 3D 2018 software interface. The title bar shows the application name and the current drawing file, "Drawing1.dwg". The ribbon is set to the "Solid" tab, with the "Sweep" command selected in the "Solid" panel. A help window for the "Sweep" command is open, providing a definition and usage instructions. The main 3D viewport shows a realistic rendering of a bolt, illustrating the result of a sweep operation. The bolt has a hexagonal head and a cylindrical shaft with a helical thread pattern. The interface includes various toolbars, a command line at the bottom, and a status bar.

Sweep
Creates a 3D solid or surface by sweeping a 2D or 3D curve along a path

The sweep object is automatically aligned to the path object. Use SURFACEMODELINGMODE to set whether SWEEP creates a procedural surface or a NURBS surface.



SWEEP
Press F1 for more help

Press ESC or ENTER to exit, or right-click to display shortcut-menu.

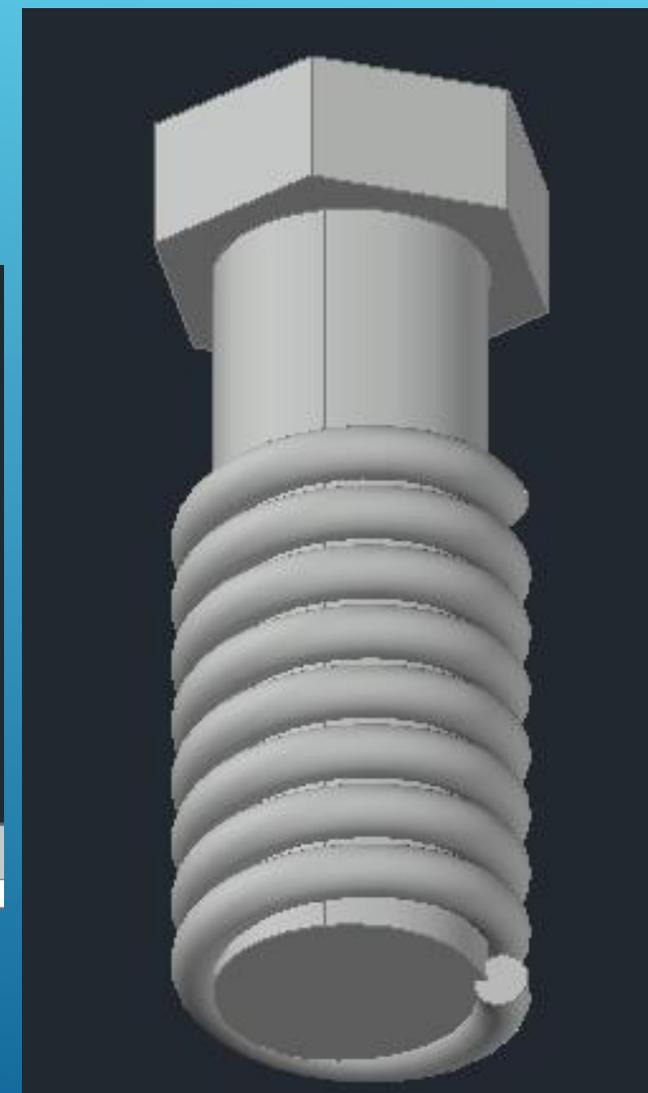
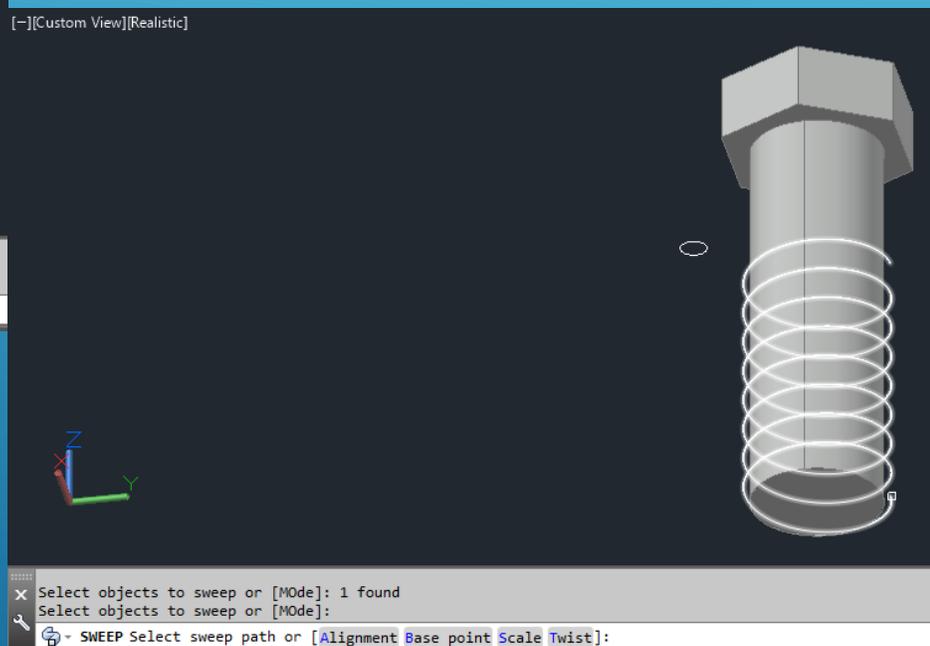
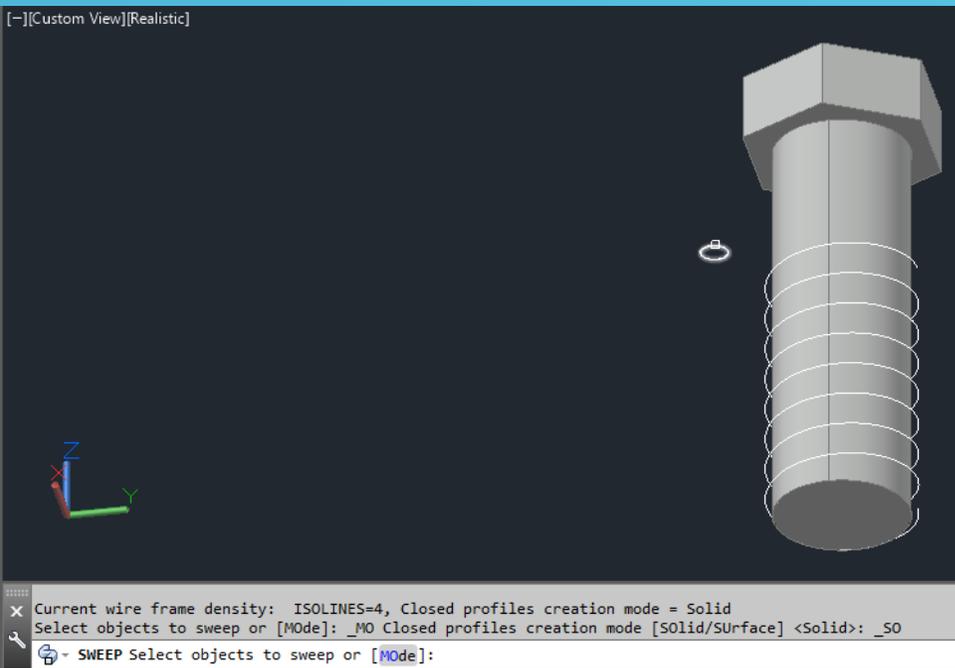
3DORBIT

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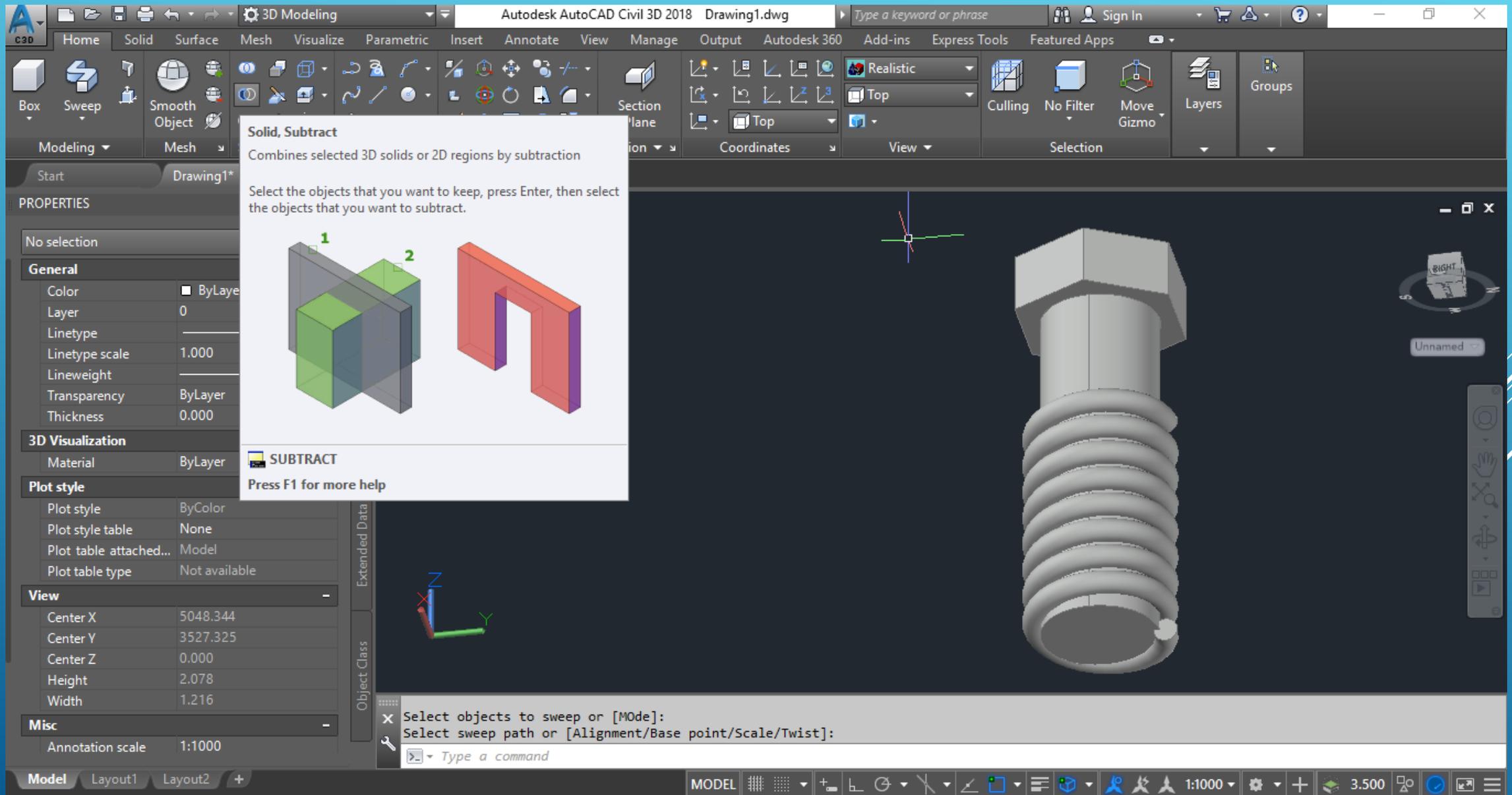


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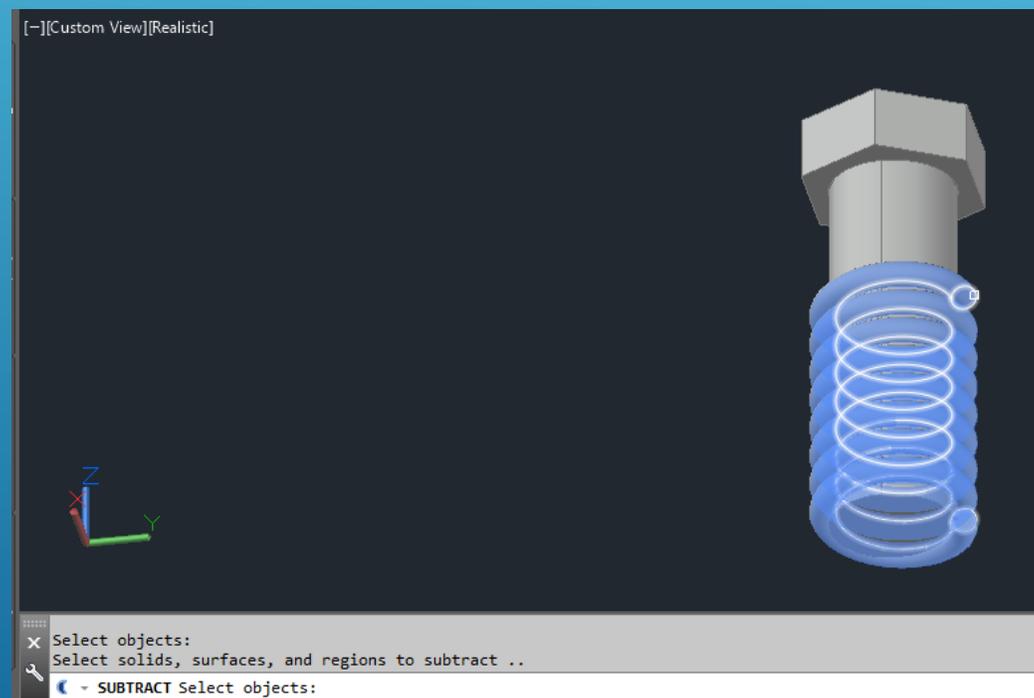
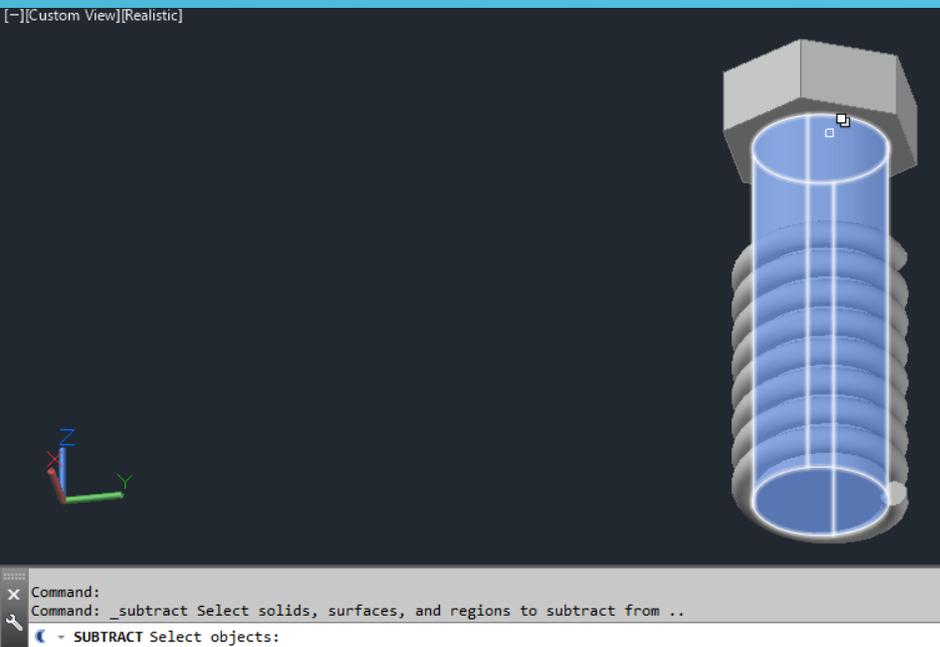
The screenshot displays the Autodesk AutoCAD Civil 3D 2018 software interface. The main window shows a 3D model of a bolt with a hexagonal head and a threaded shaft. A help window for the 'Solid, Subtract' command is open, providing instructions and a visual diagram. The diagram shows two overlapping 3D solids, one green and one blue, with the resulting subtracted shape shown in red. The help text reads: 'Solid, Subtract' 'Combines selected 3D solids or 2D regions by subtraction' 'Select the objects that you want to keep, press Enter, then select the objects that you want to subtract.' Below the diagram, there is a 'SUBTRACT' icon and the text 'Press F1 for more help'. The software interface includes a ribbon with various toolsets (Home, Solid, Surface, Mesh, Visualize, Parametric, Insert, Annotate, View, Manage, Output, Autodesk 360, Add-ins, Express Tools, Featured Apps), a Properties panel on the left, and a Command Line at the bottom. The Command Line shows the current command: 'Select objects to sweep or [MODE]:' followed by 'Select sweep path or [Alignment/Base point/Scale/Twist:]'. The status bar at the bottom indicates the current view is 'MODEL' and the scale is '1:1000'.

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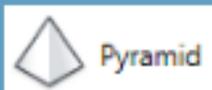
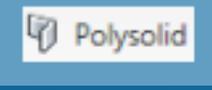


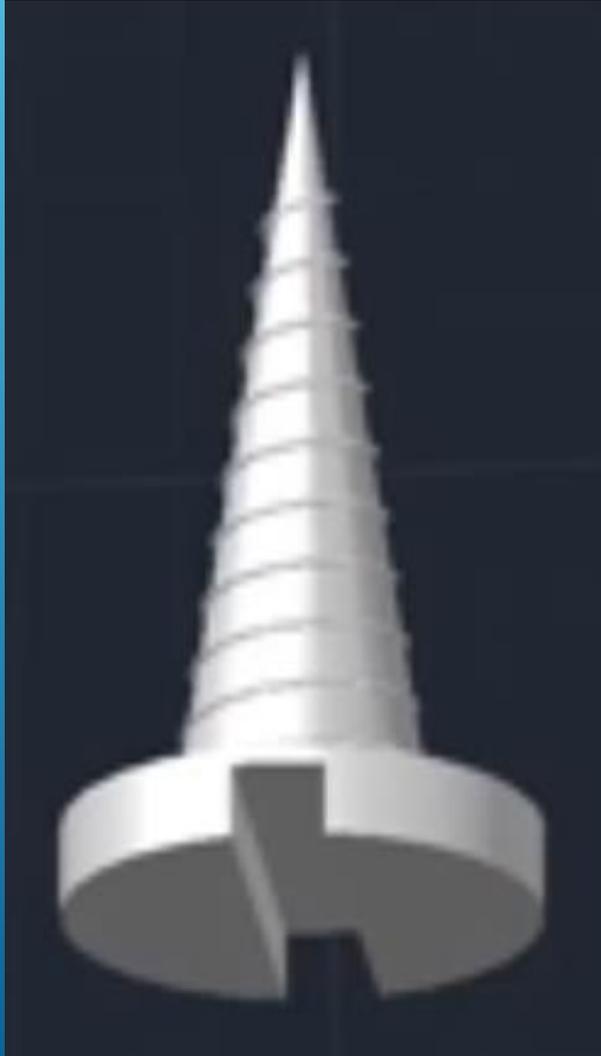
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Primitive Solids

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.

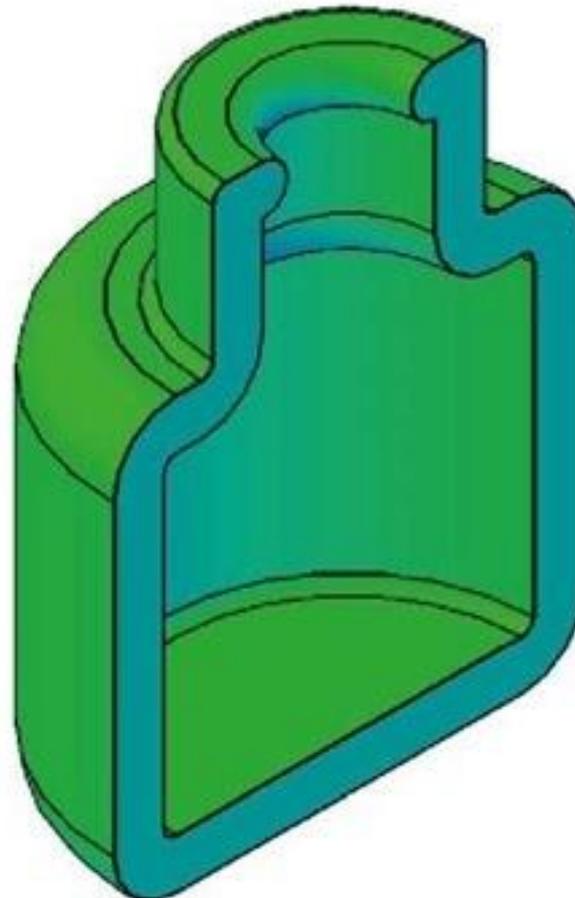
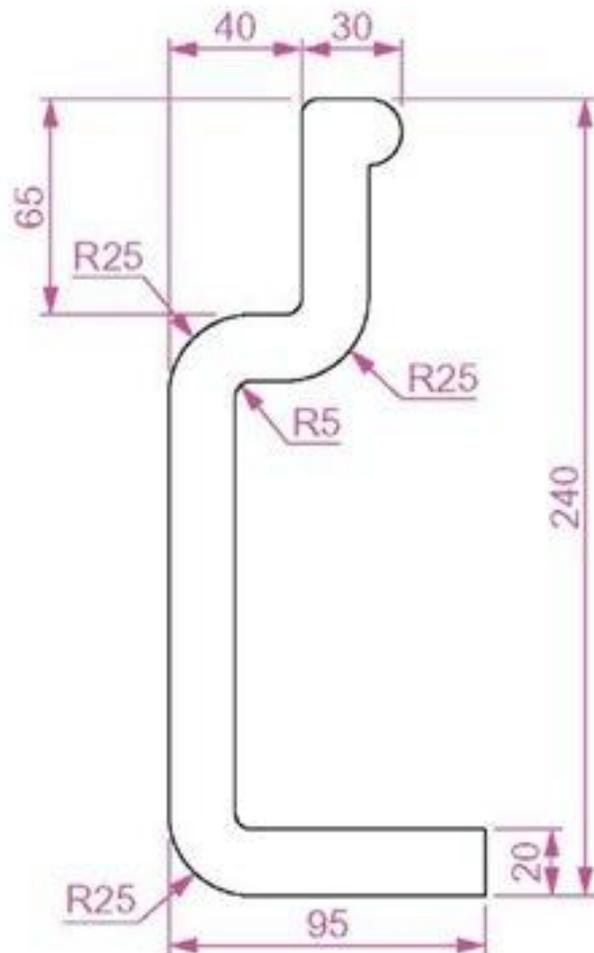


Desenhe como um sólido o parafuso com as características seguintes:

- raio da cabeça=0.375
- altura da cabeça=0.25
- ranhura do parafuso: largura=0.10, altura=0.15
- raio da base do cone=0.25
- altura do cone=1.5
- hélice: diâmetro topo=0.275, diâmetro base=0; 8 voltas
- rosca do parafuso: diâmetro=0.05

Apresente o desenho de forma a que o eixo longitudinal do parafuso coincida com o eixo X do referencial, e de tal forma a que a ponta do parafuso tenha coordenadas $(a, b, 0)$.

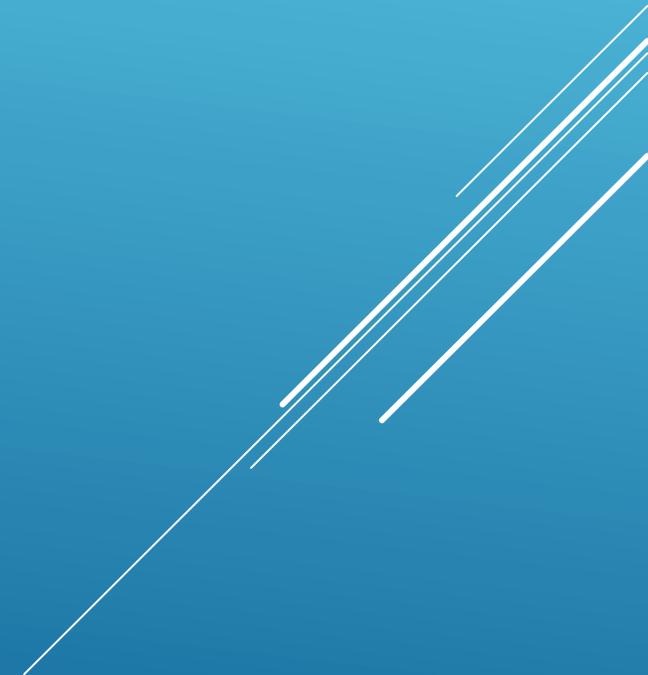
Superfícies de revolução



Represente a 3D, de acordo com as indicações, o sólido da figura.



Desenhar o perfil do copo usando
Lines e Arcs

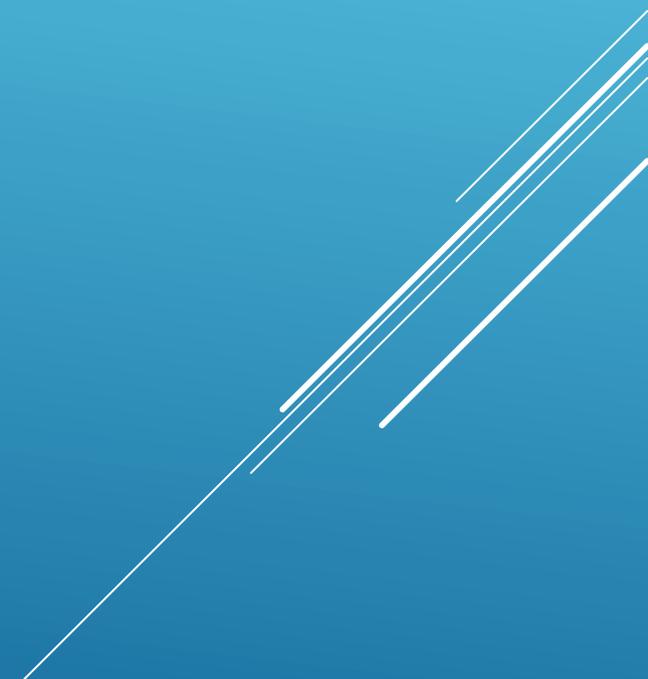


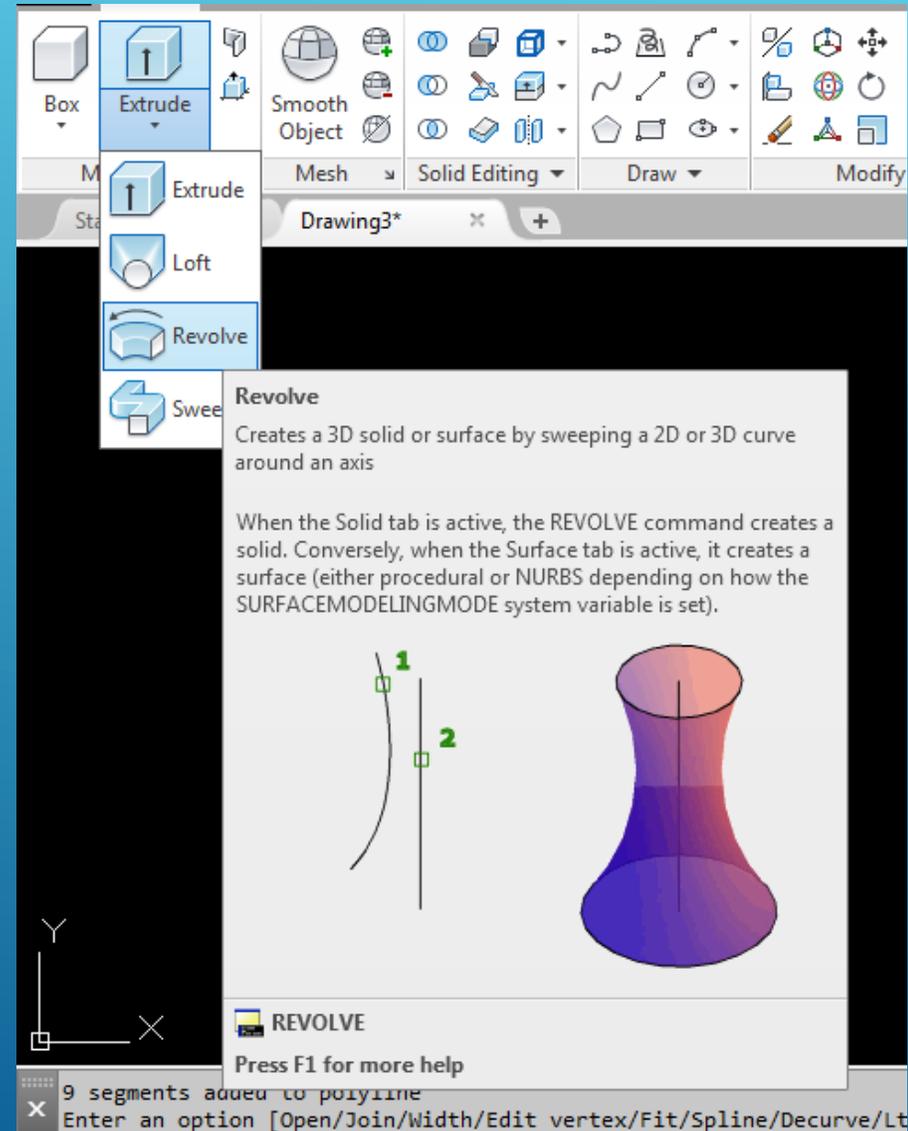
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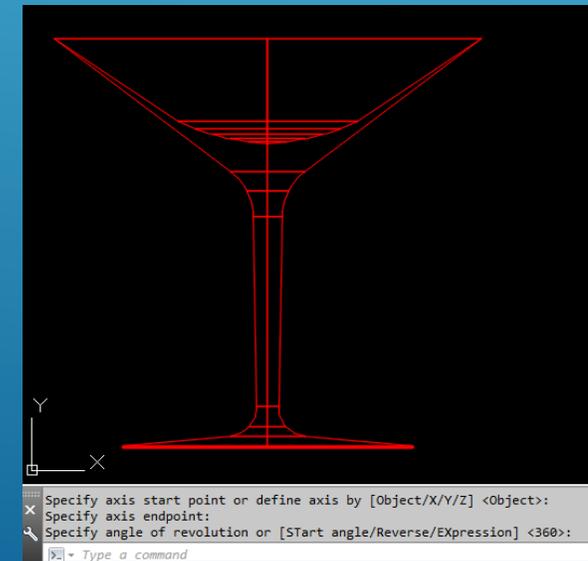
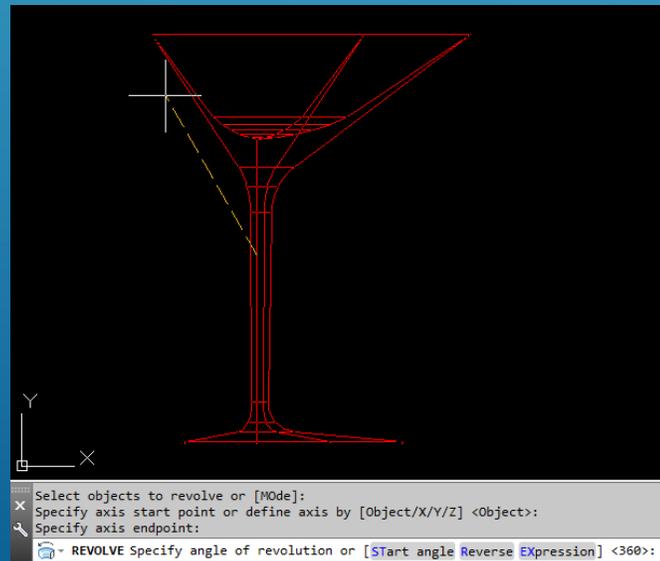
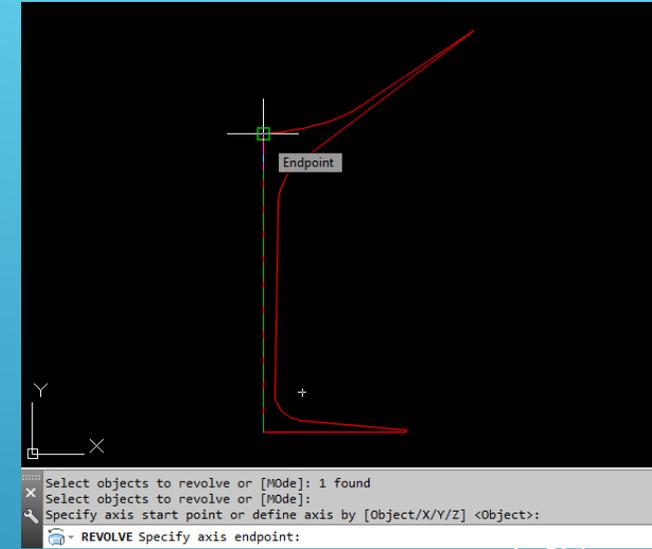
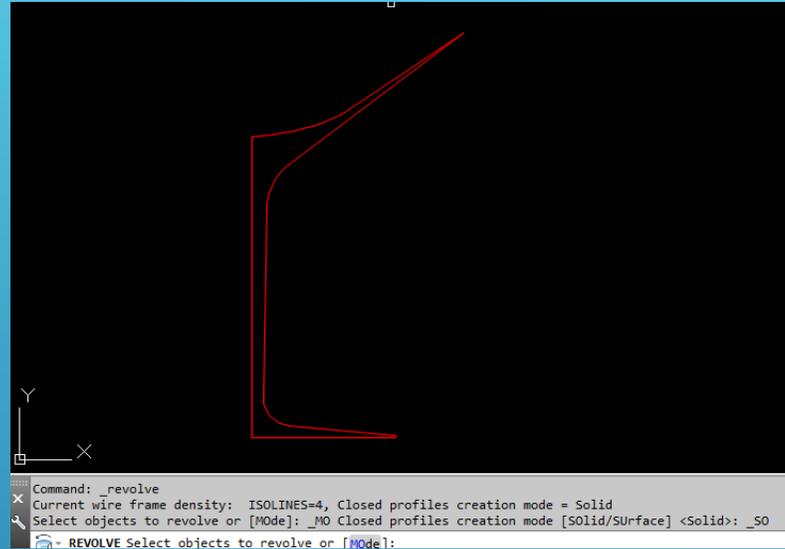
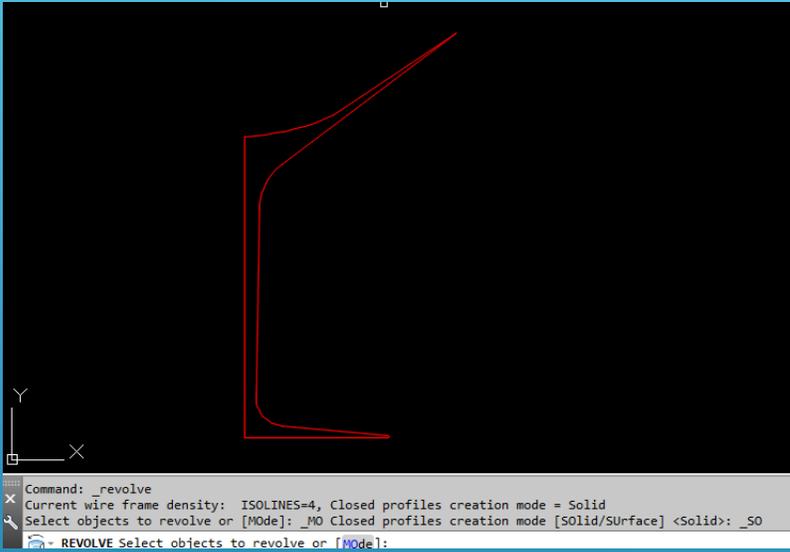
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Usar o comando PEDIT para transformar os elementos LINES e ARCS numa POLYLINE



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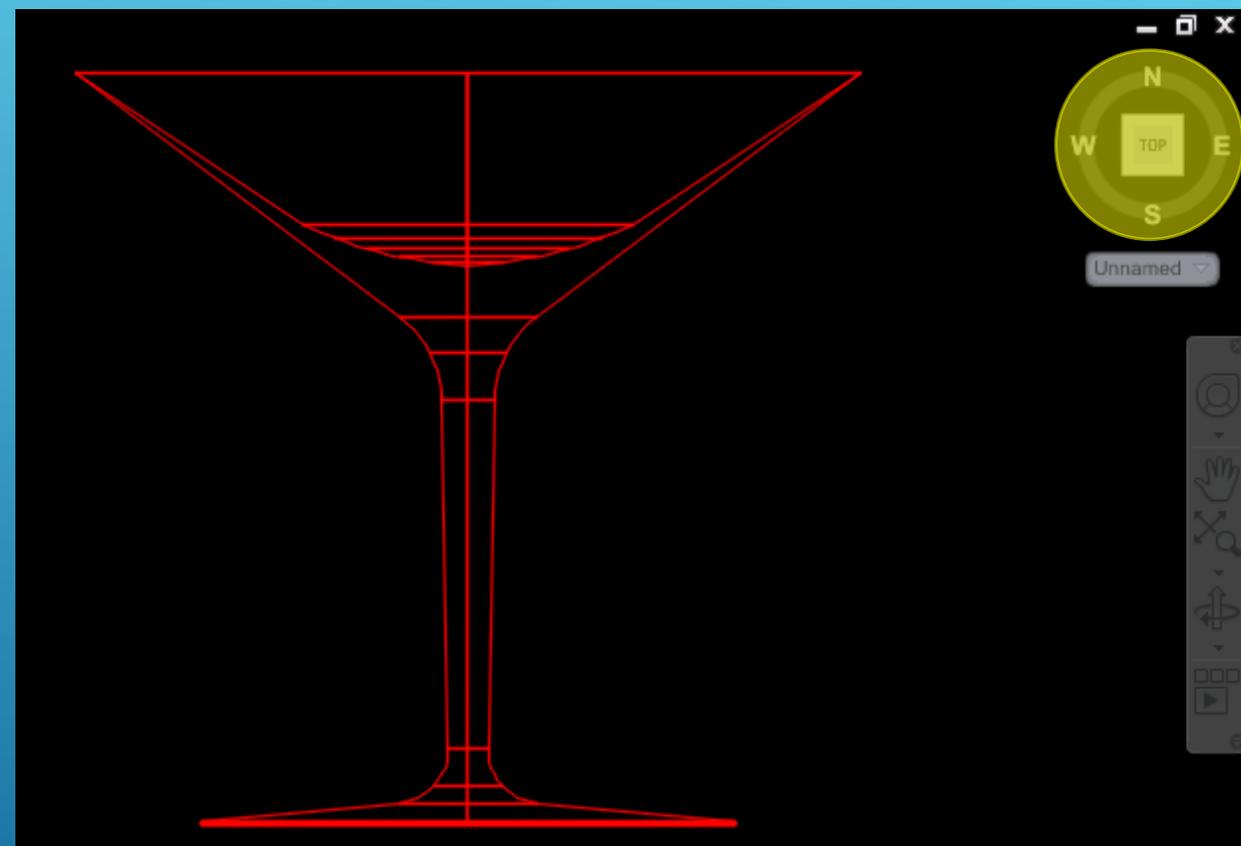
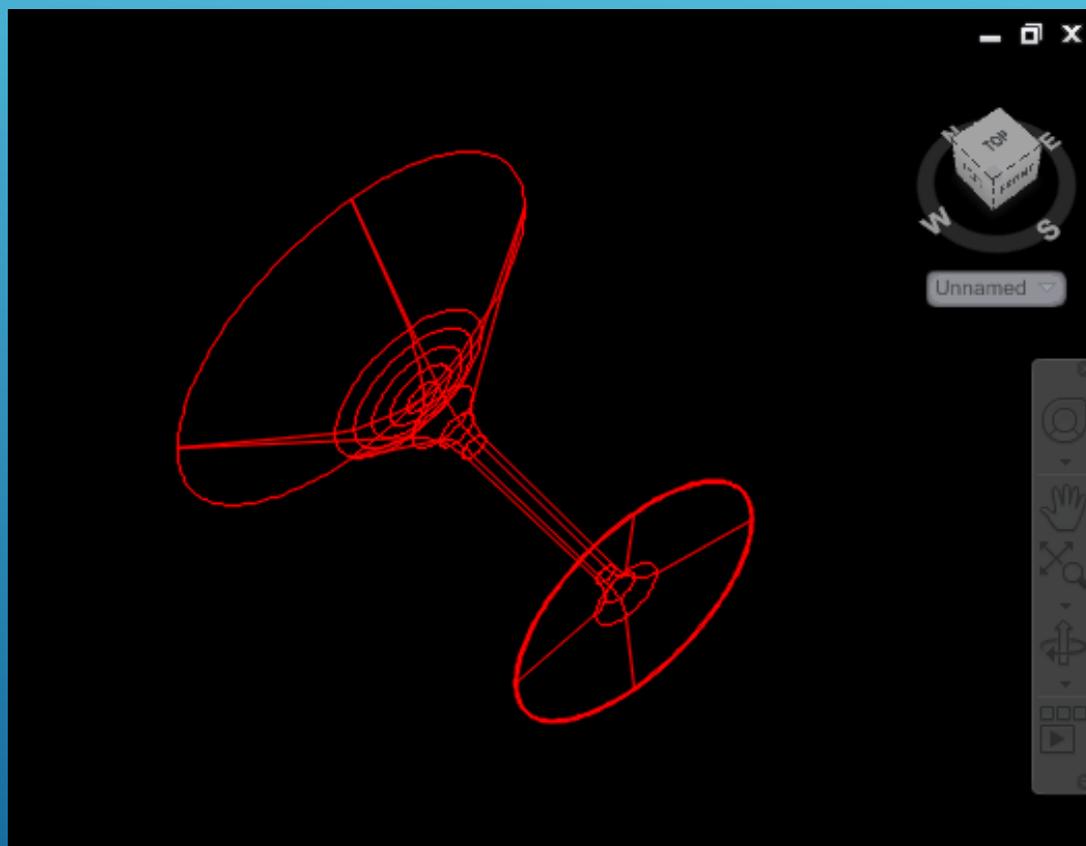


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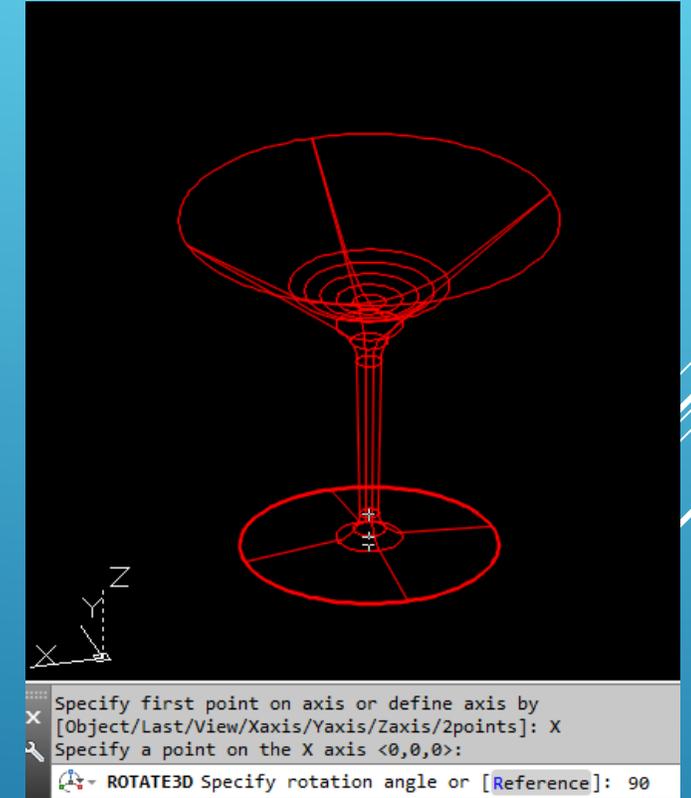
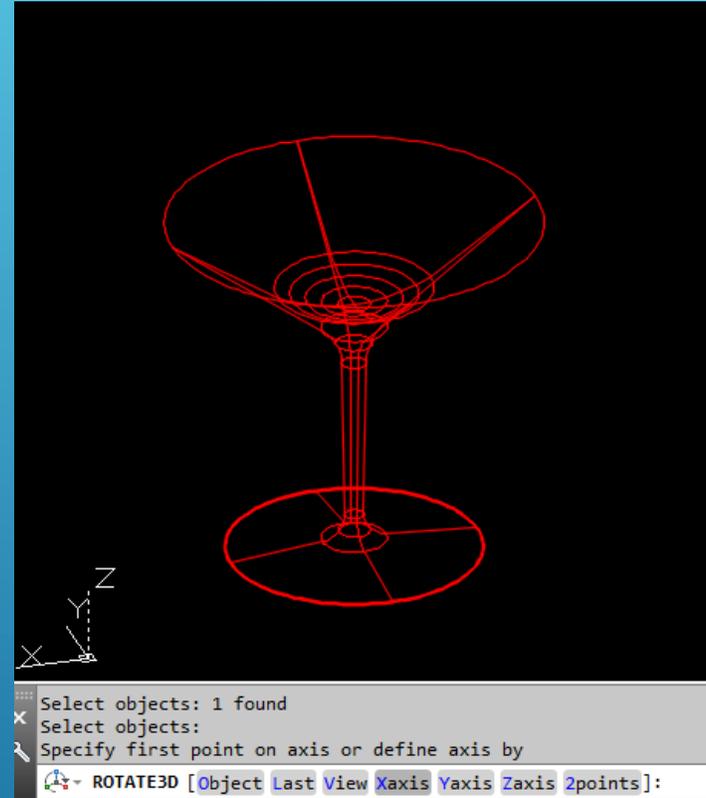
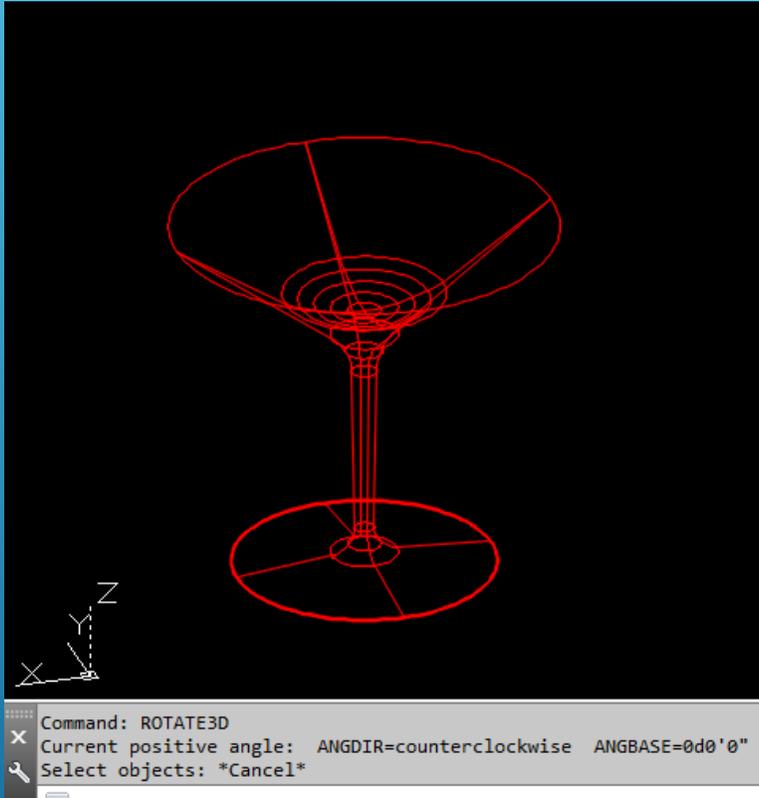
3D Rotate

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



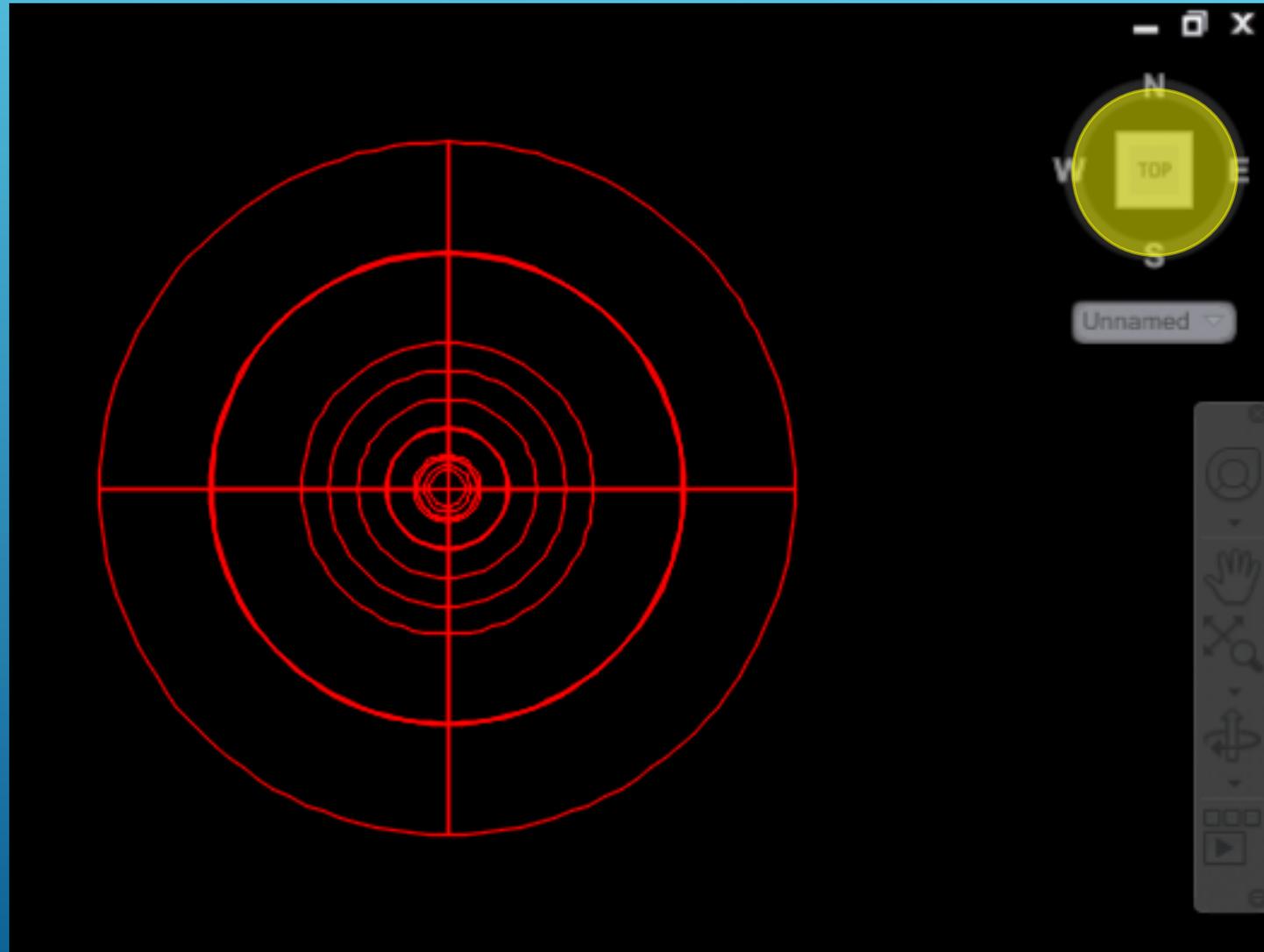
Rodar o copo em torno do eixo X para que na vista Top o copo apareça em pé

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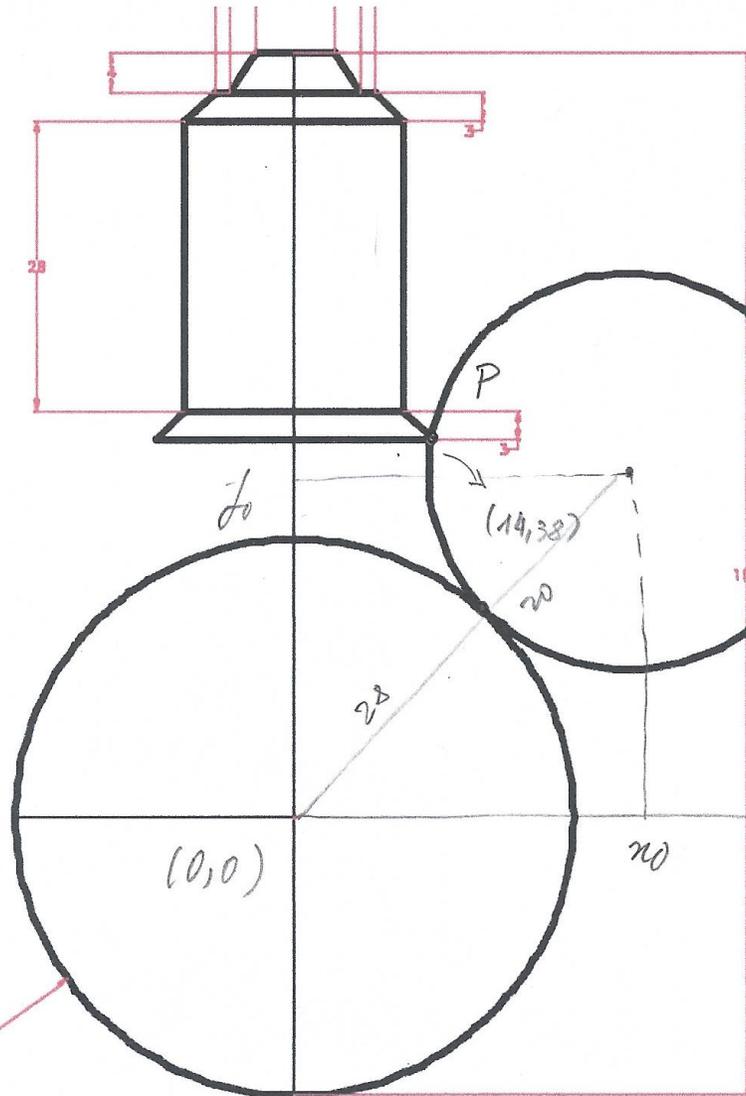


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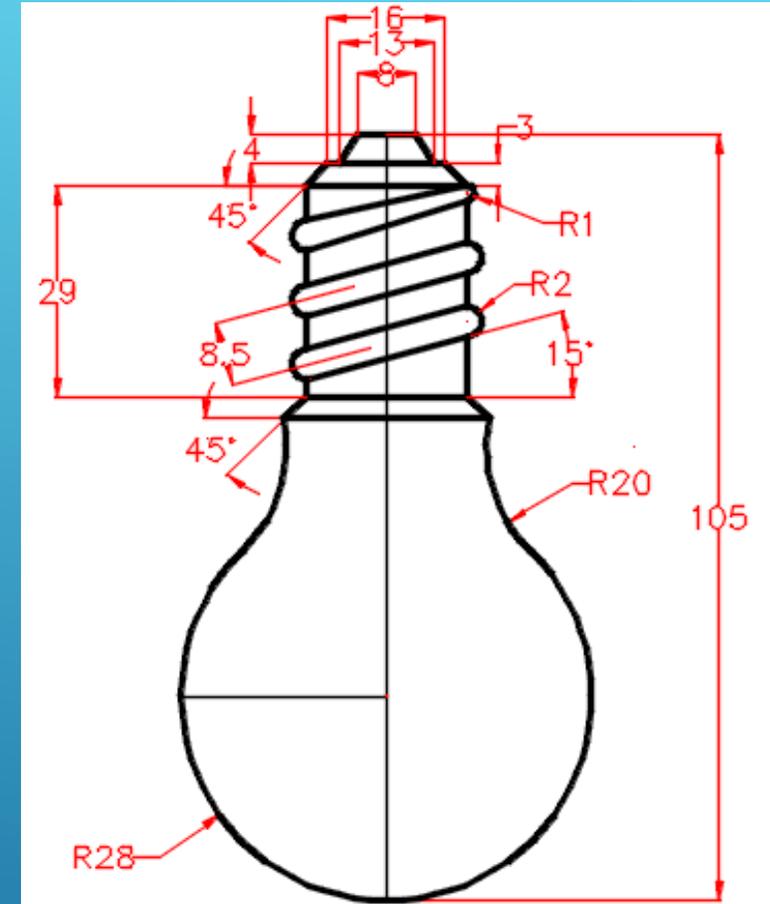
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$$(x-x_0)^2 + (y-y_0)^2 = 20^2$$

em P: $(14-x_0)^2 + (38-y_0)^2 = 20^2$

$$x_0^2 + y_0^2 = (28+20)^2$$



Maple 6 - [lampada.mws]

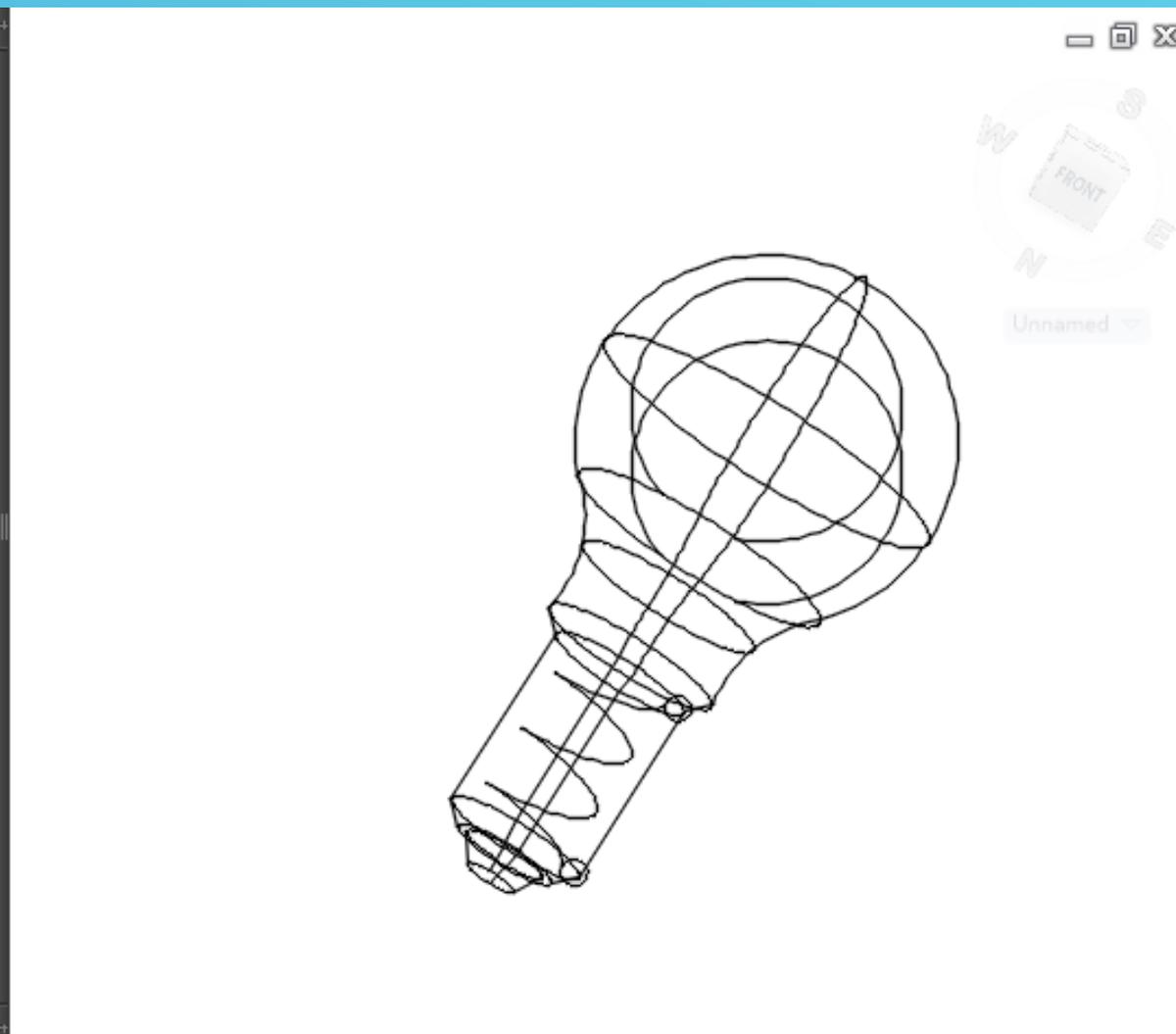
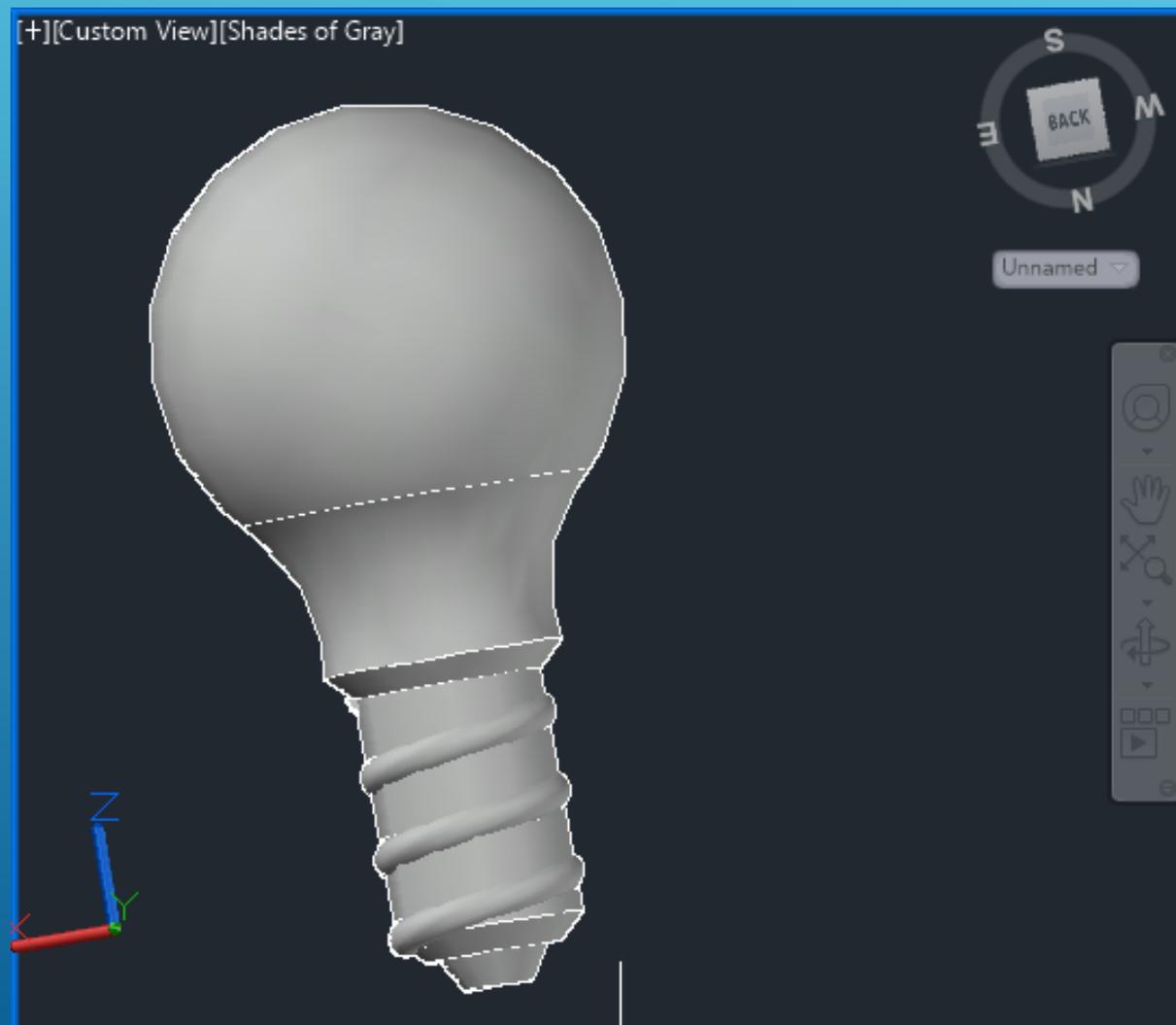
File Edit View Insert Format Spreadsheet Options Window Help



```
> fsolve({ (14-x0)^2+(38-y0)^2=20^2, x0^2+y0^2=(28+20)^2 }, {x0,y0});
```

{x0 = 33.64278120, y0 = 34.23687008}

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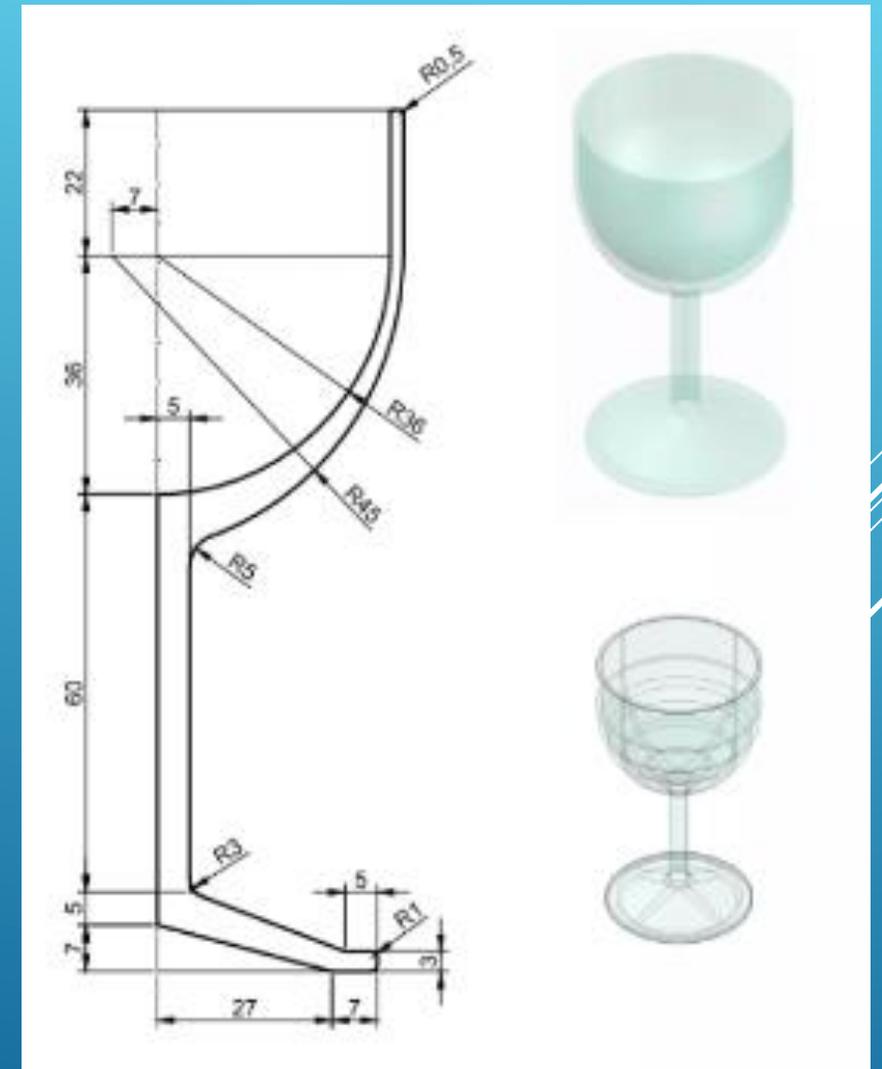
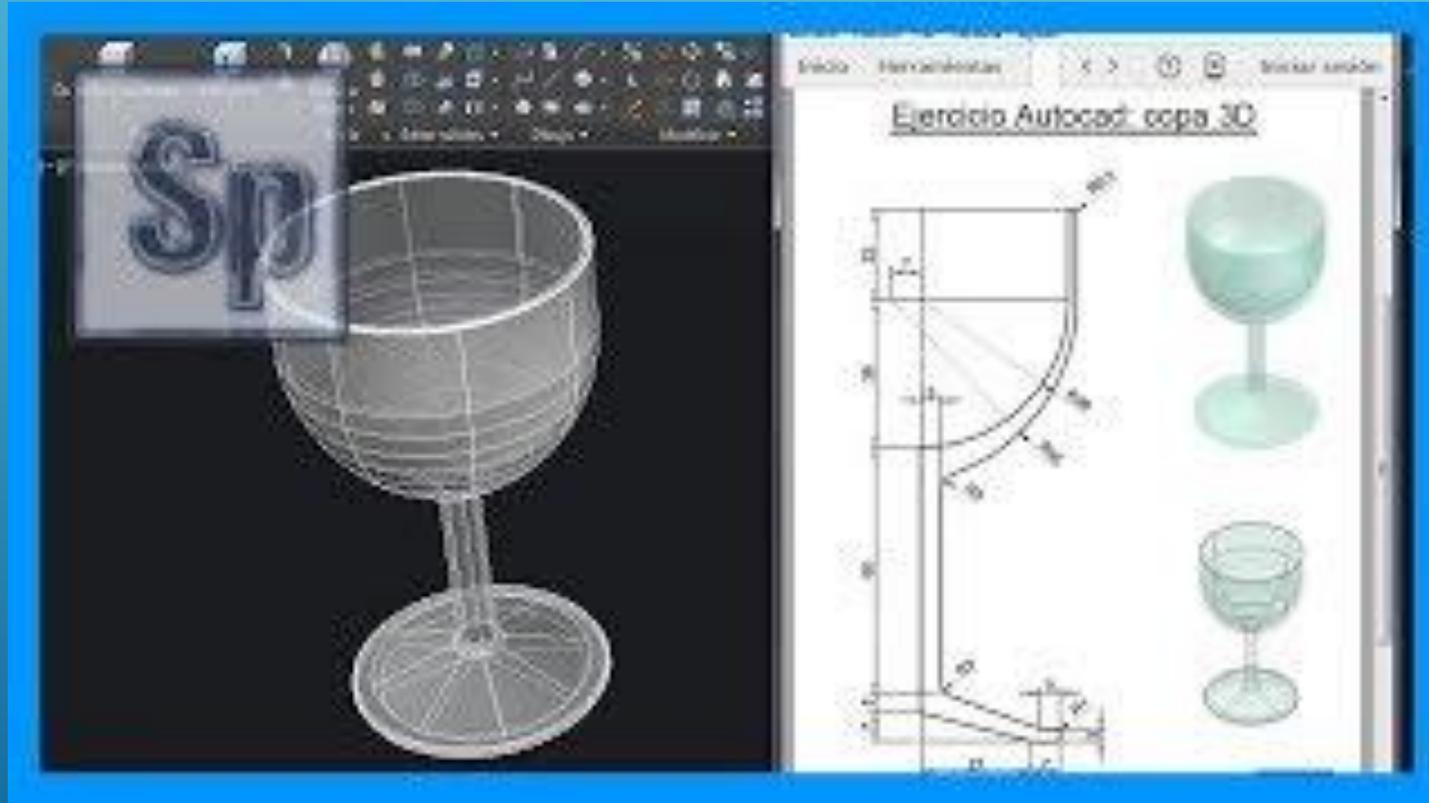


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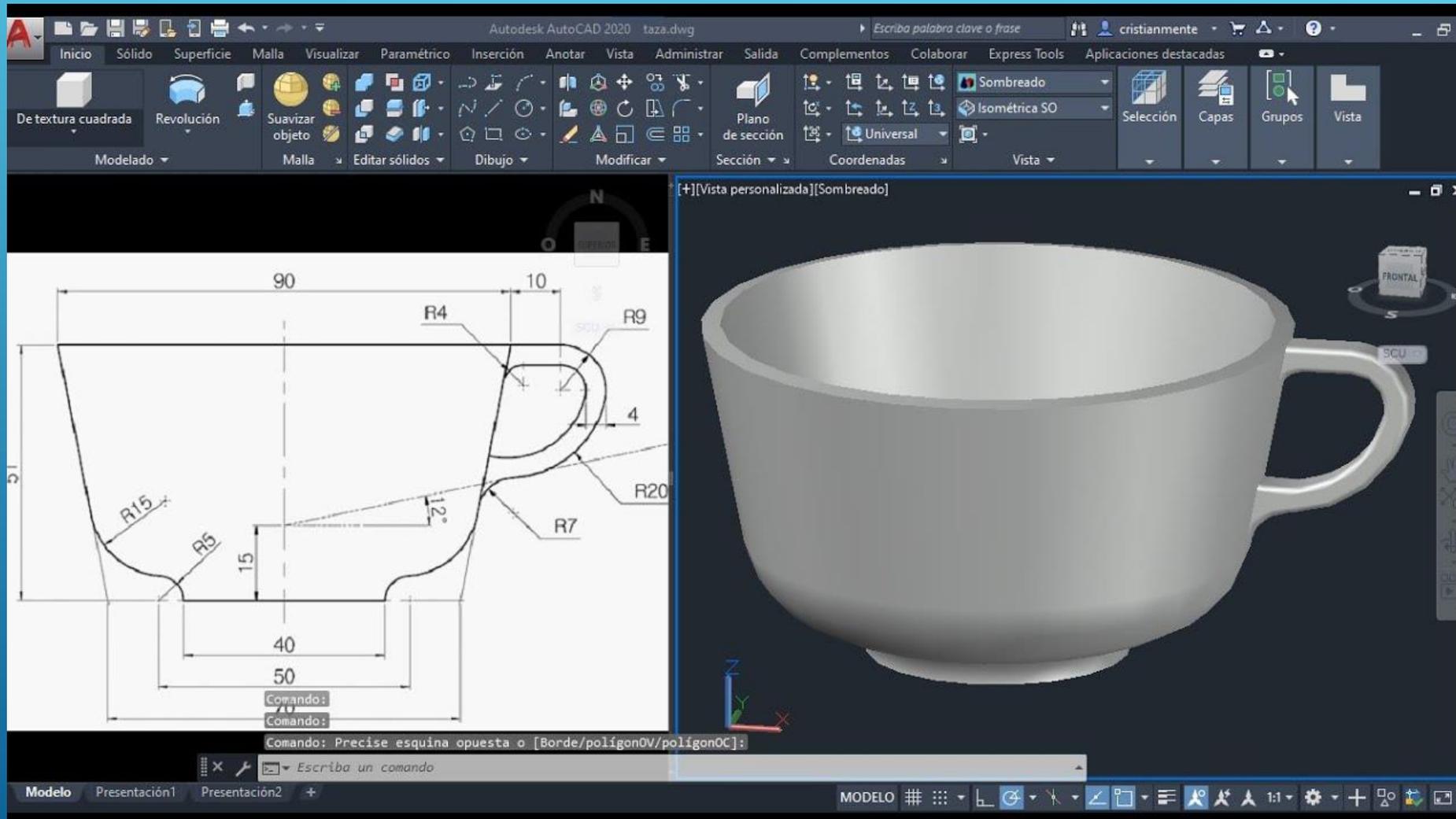


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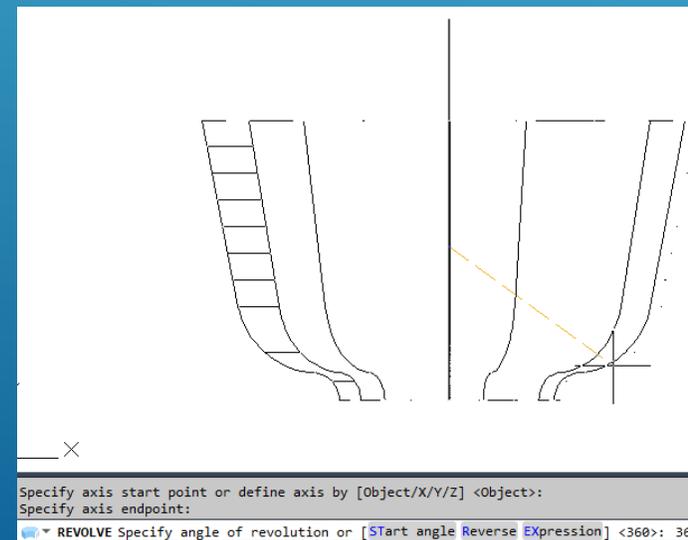
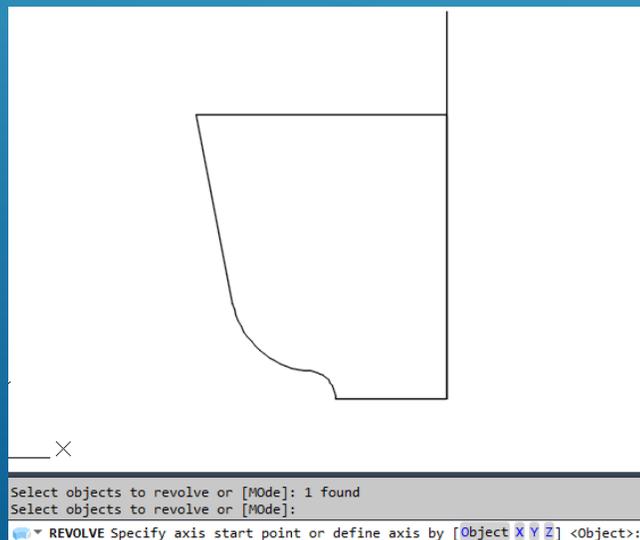
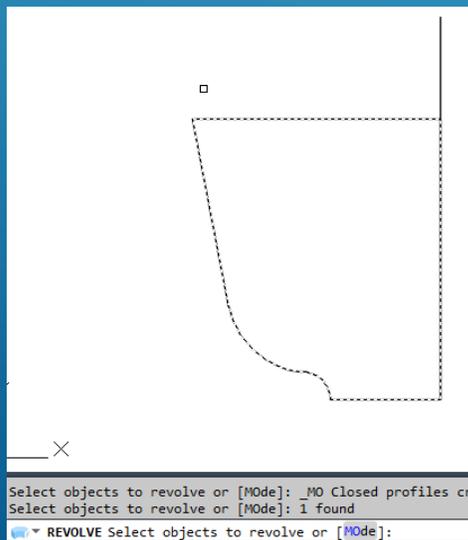
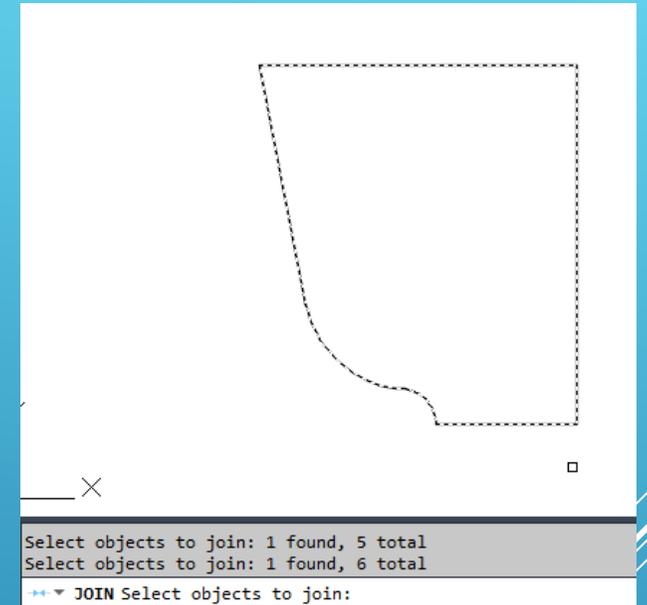
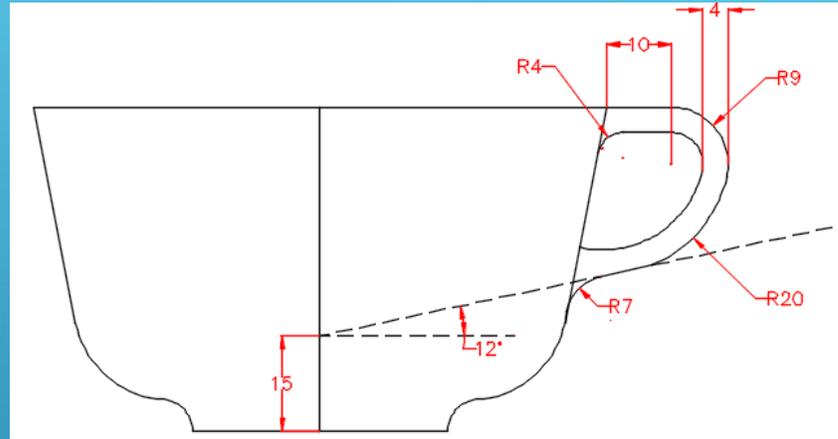
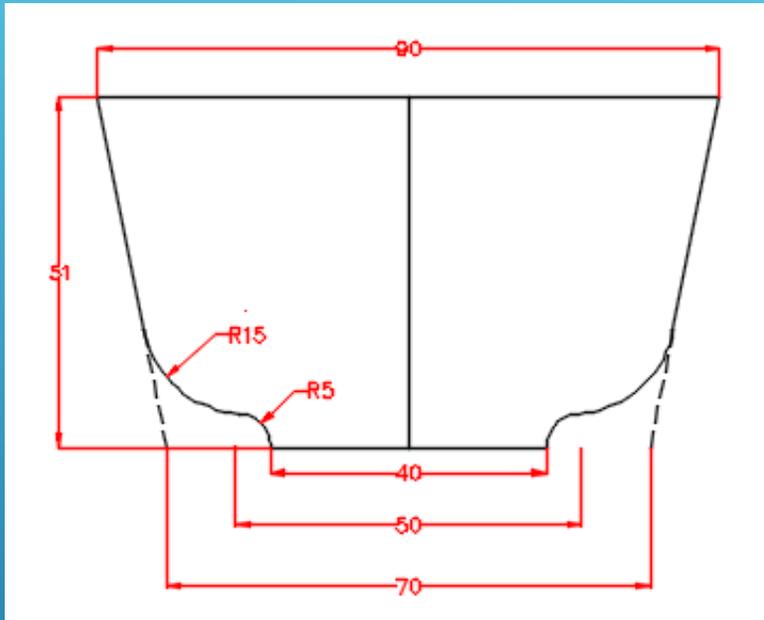


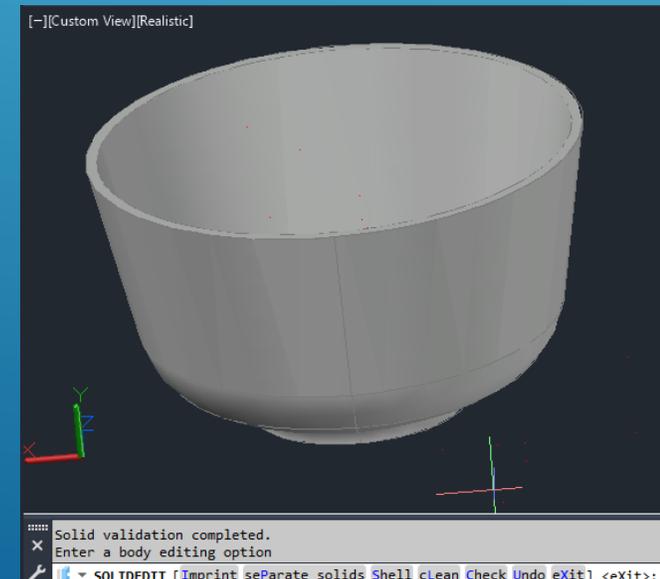
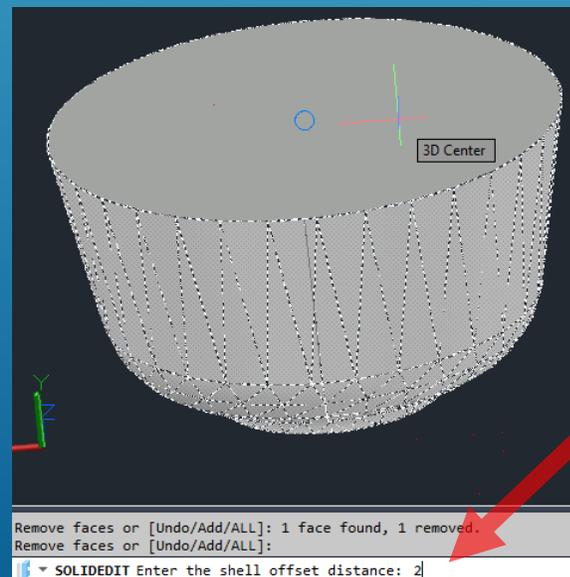
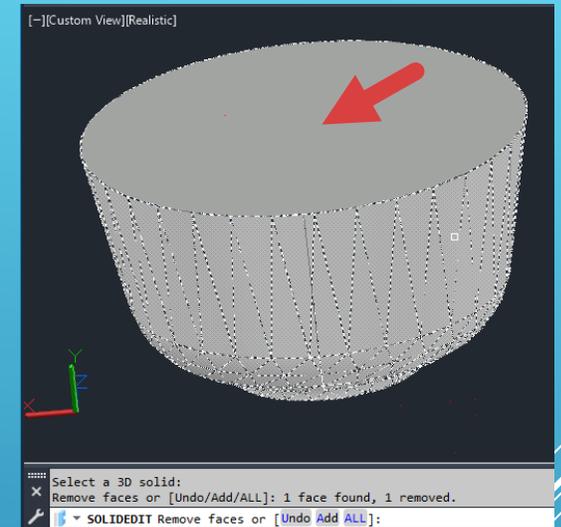
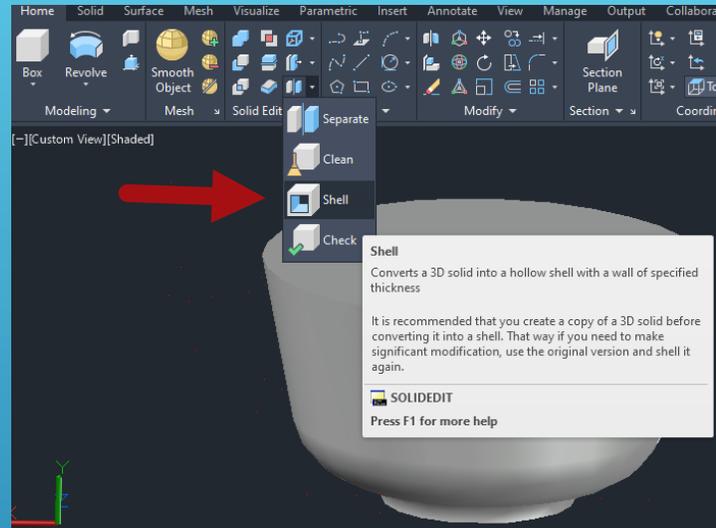
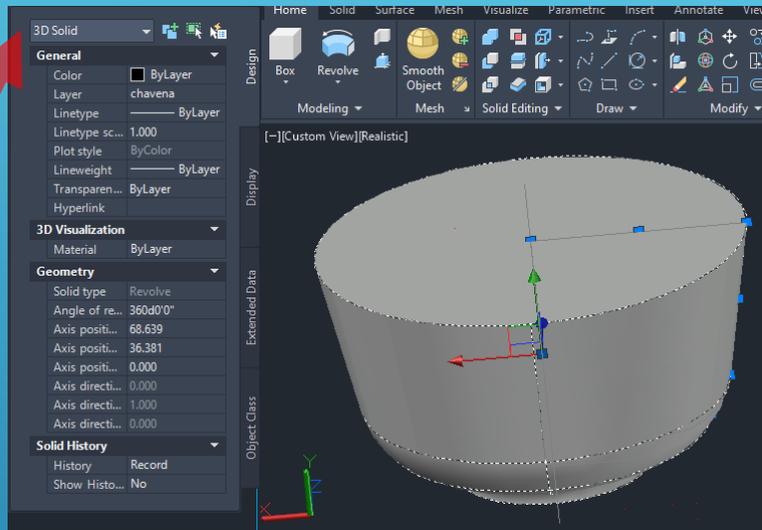
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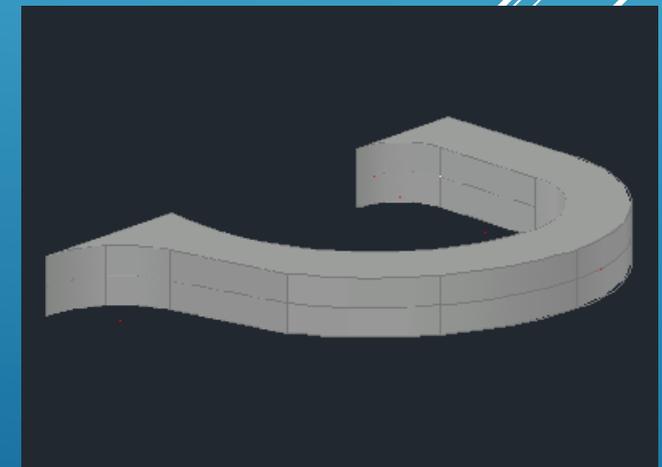
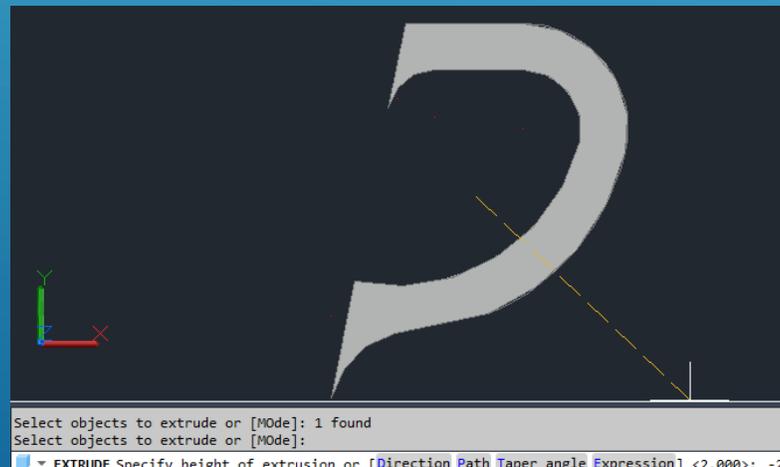
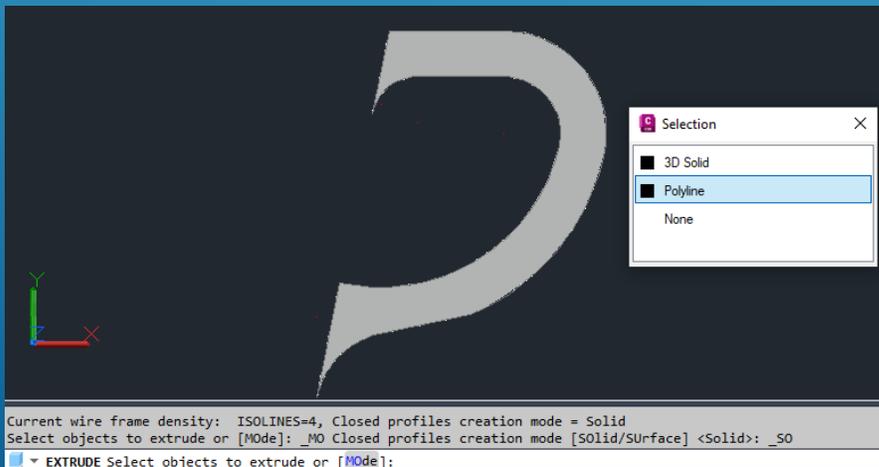
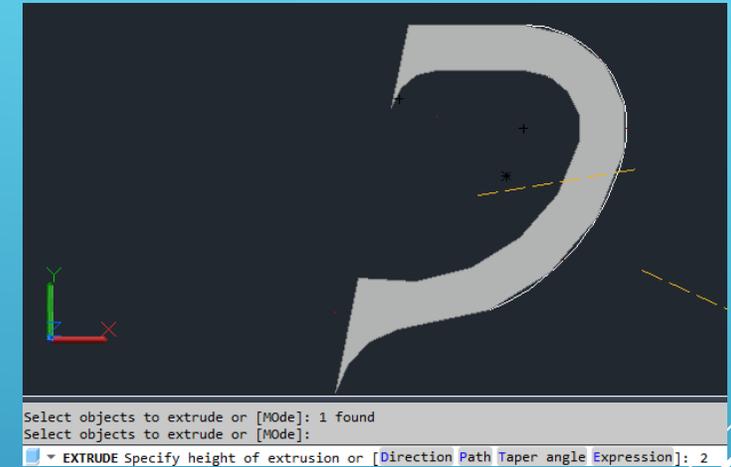
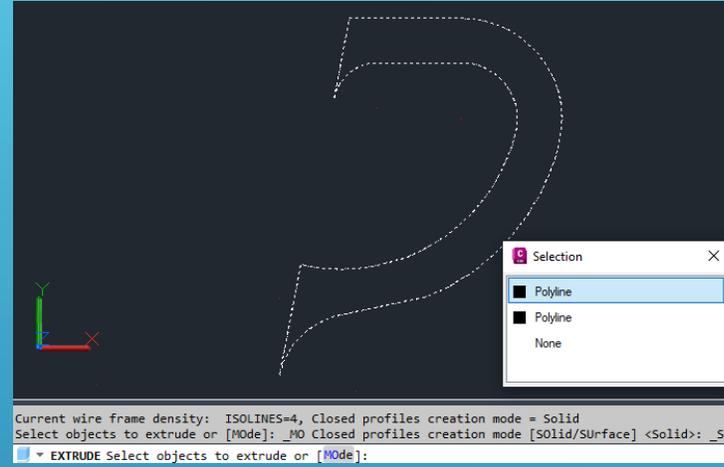
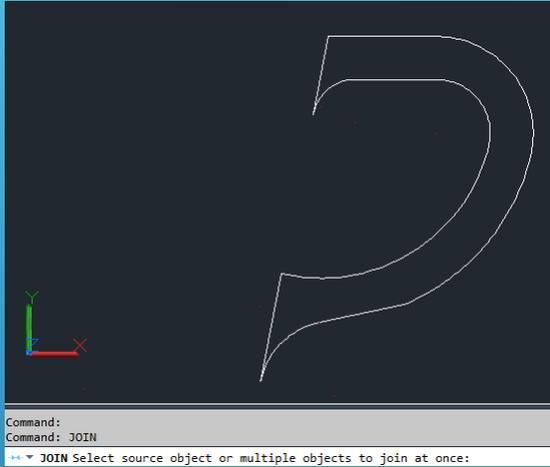


Desenhe a chávena como um sólido, supondo que a asa tem 4 unidades de largura. Após ter construído o sólido, use o comando Home>Solid Editing>Shell para criar um “buraco” no interior da chávena, identificando a superfície de topo da chávena e indicando duas unidades para a espessura da chávena. Apresente o desenho na vista NE Isometric.

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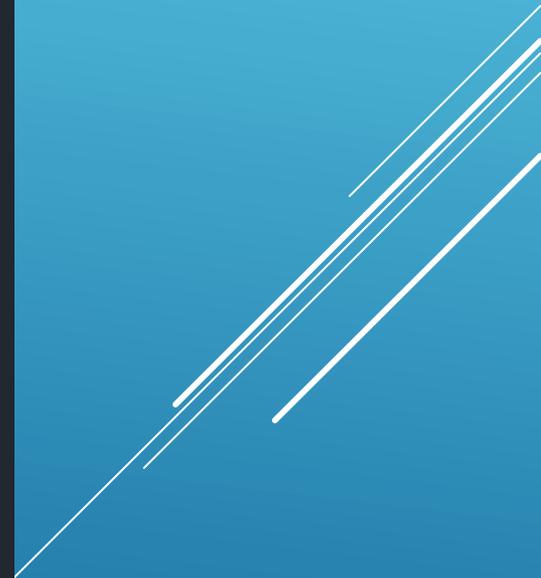


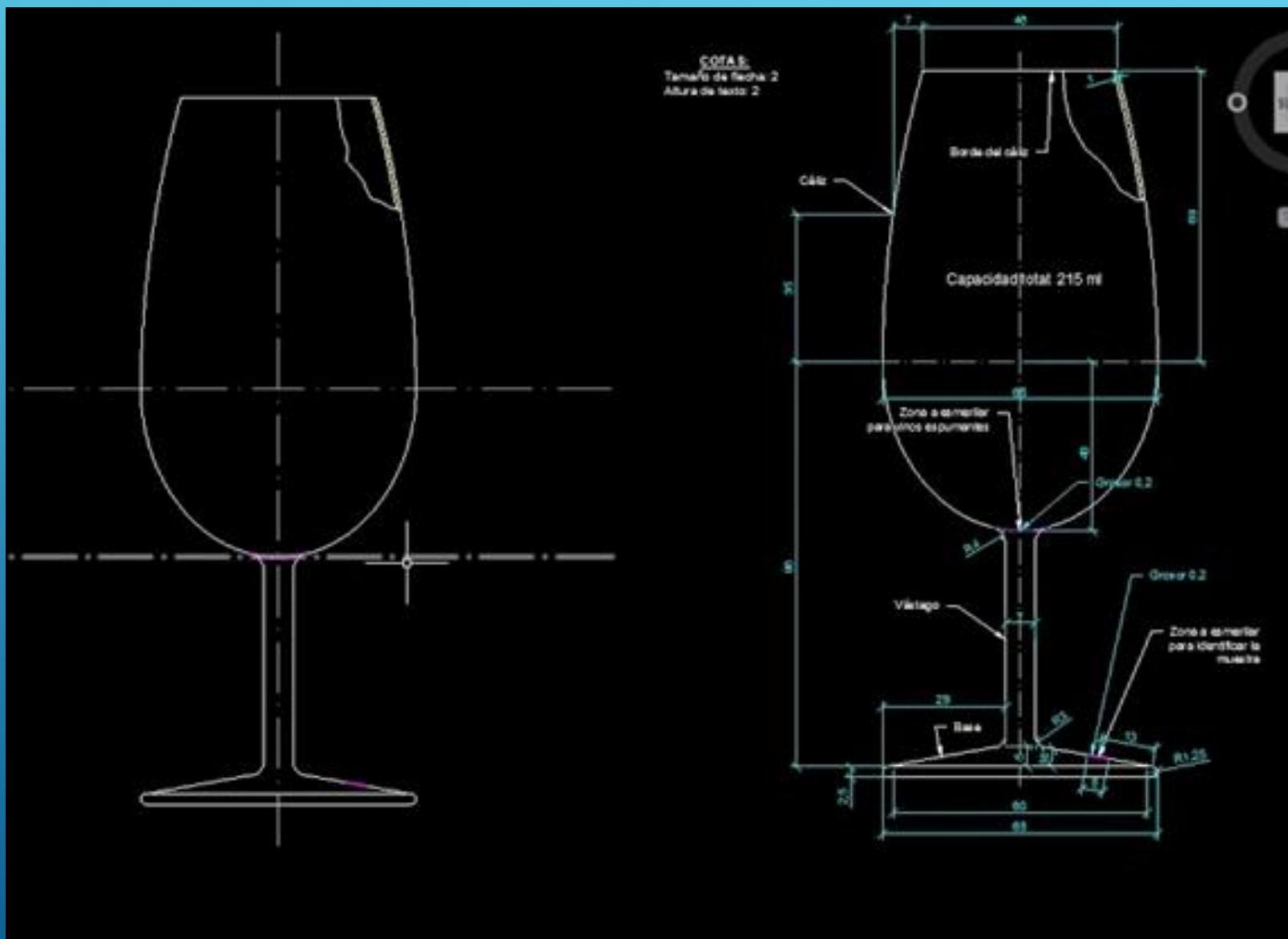
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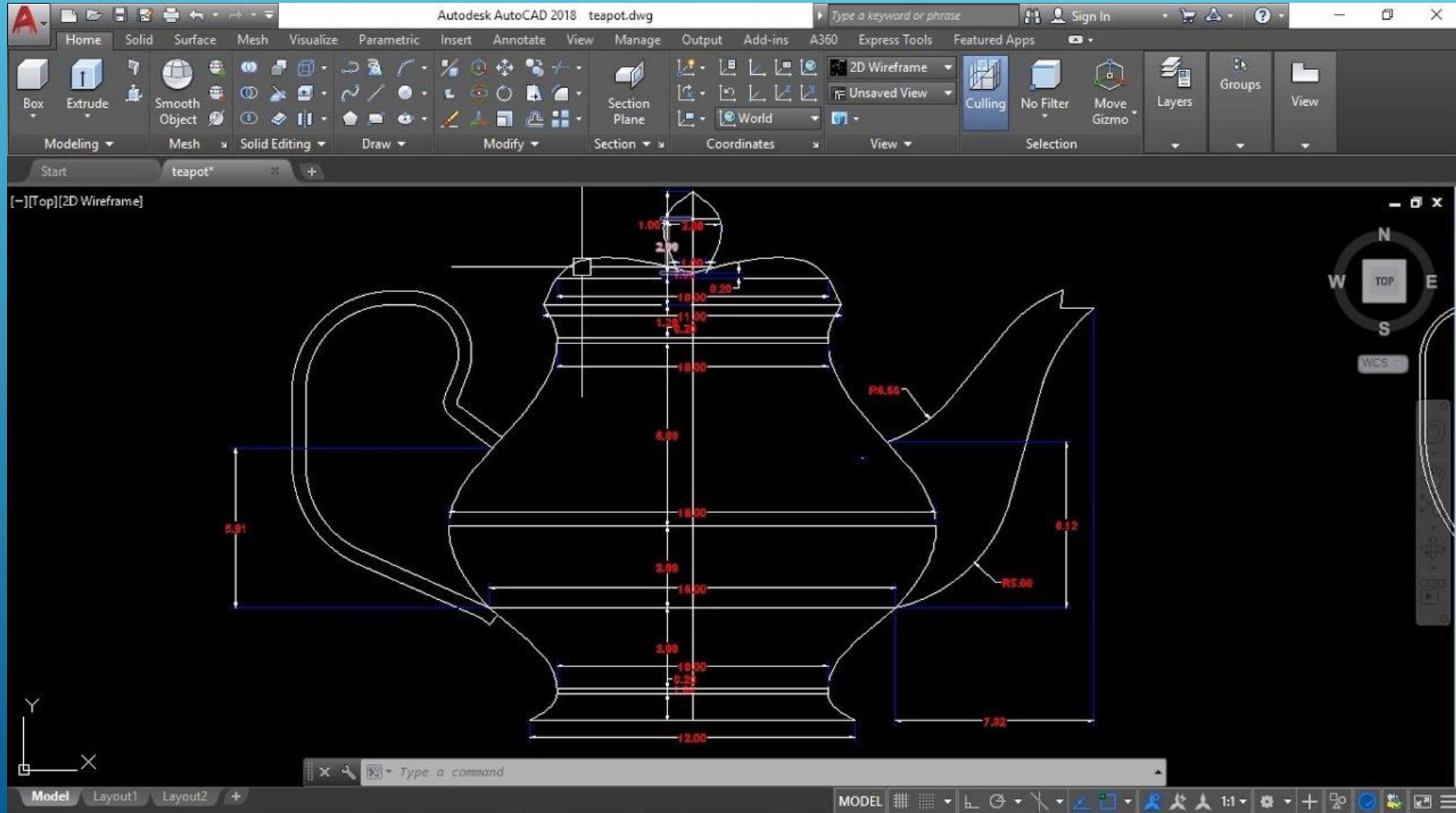


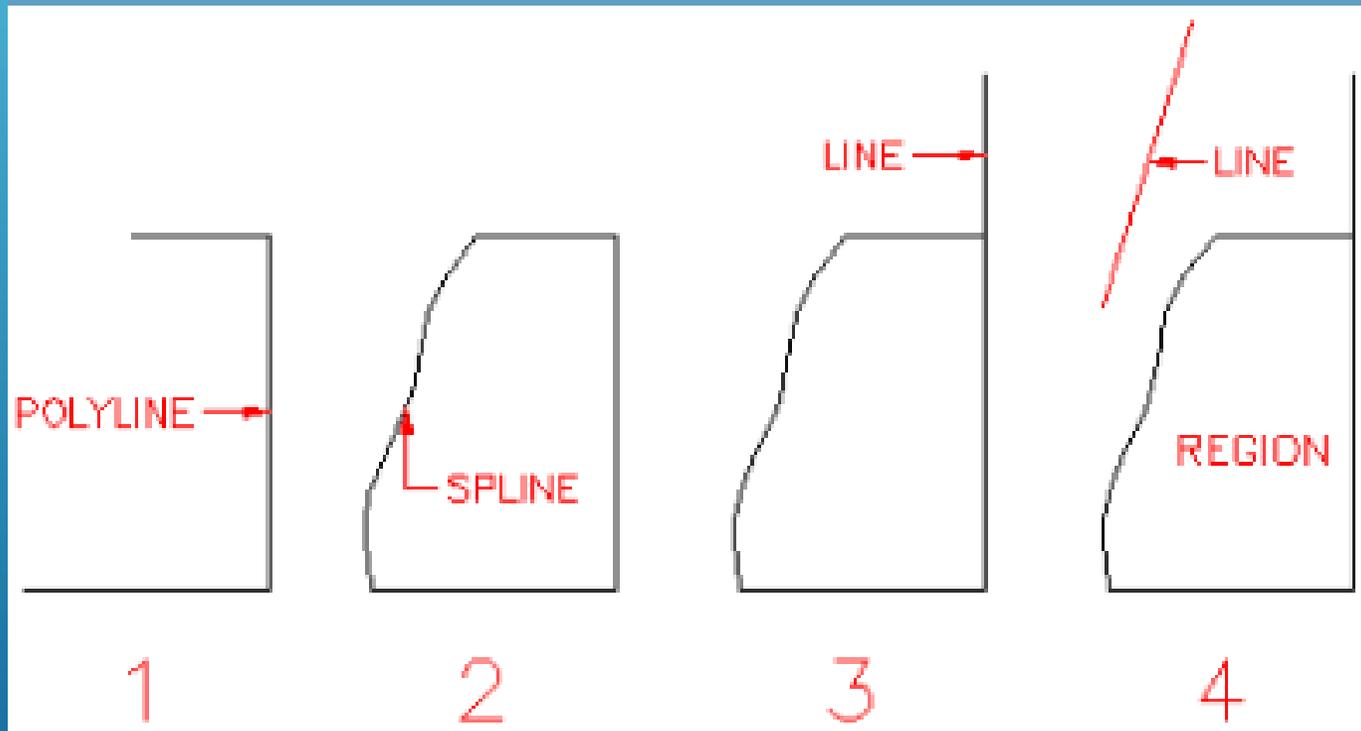
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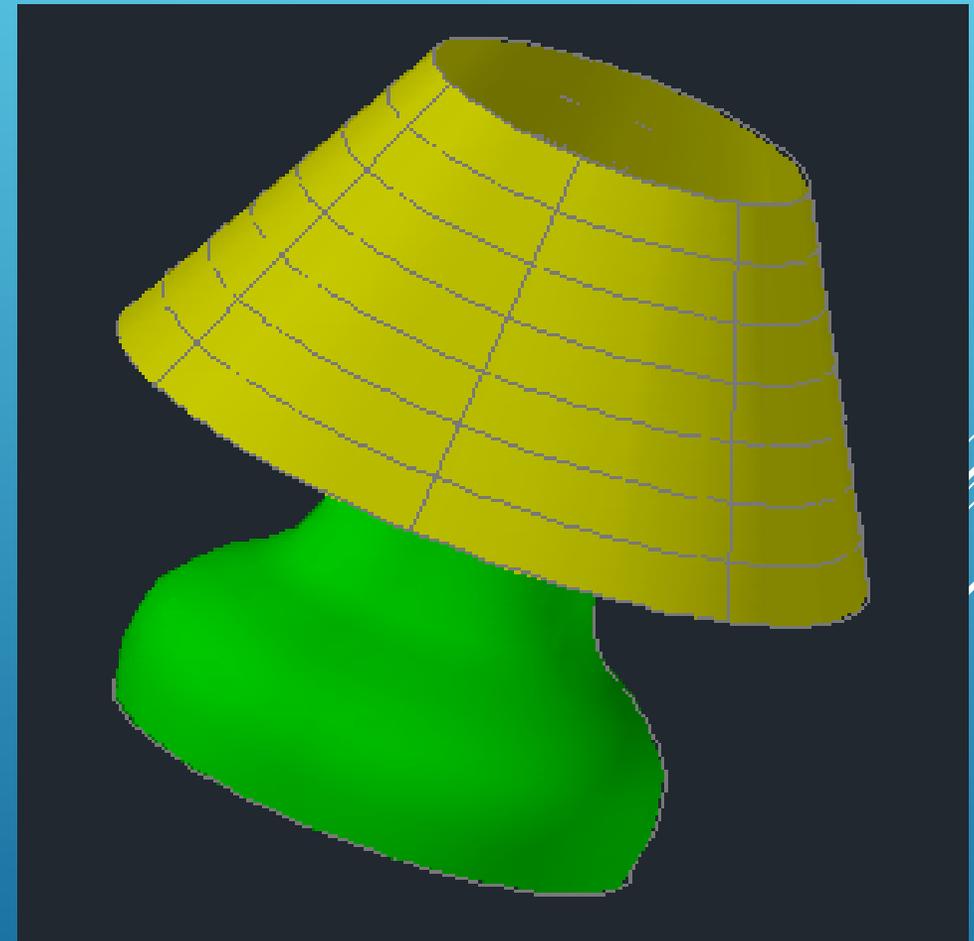
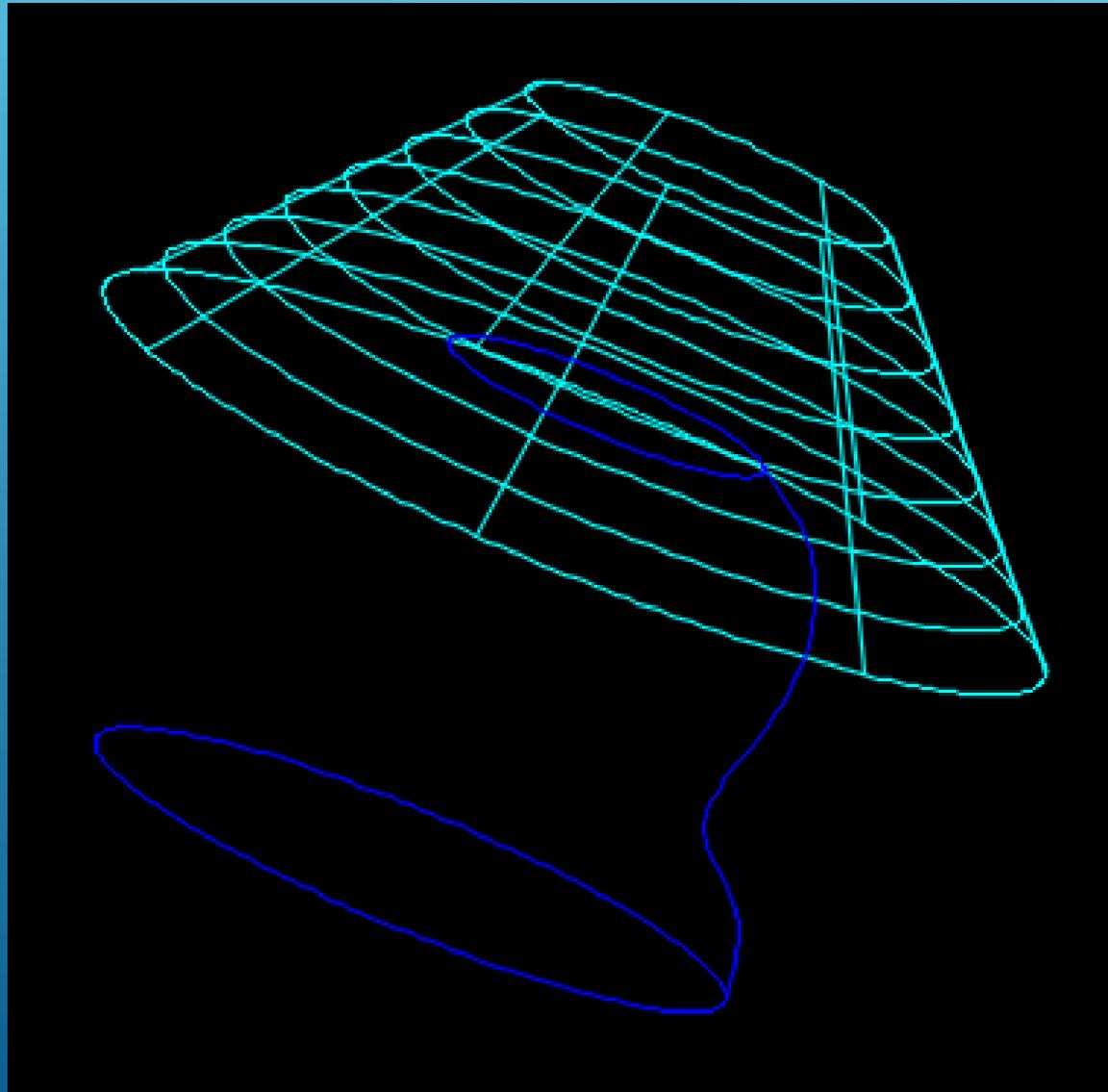




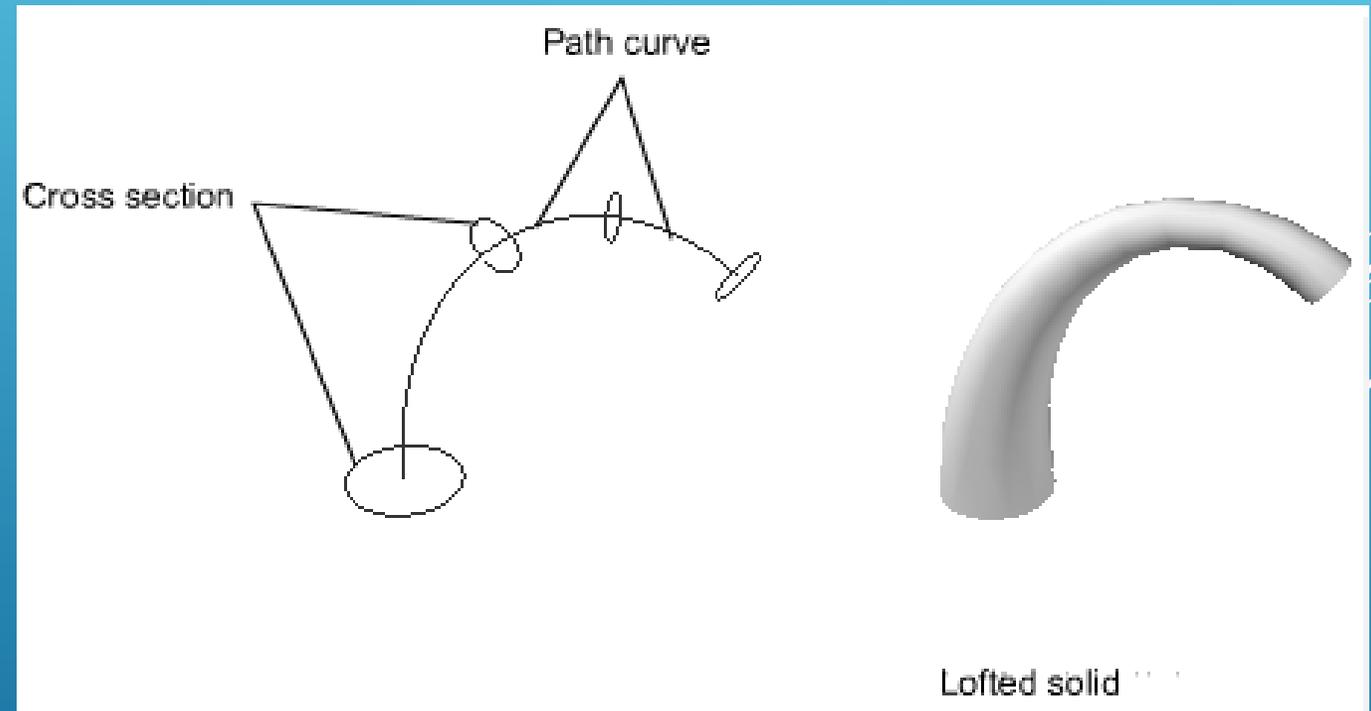




1. Criar 2 layers: base, abat-jour
2. Desenhar Polyline (layer base)
3. Desenhar Spline (layer base)
4. Desenhar Line vertical
5. Desenhar Line (layer abat-jout)
6. Definir Region (seleccionar Polyline e Spline, com Shift)
7. Revolve
8. Explode Region



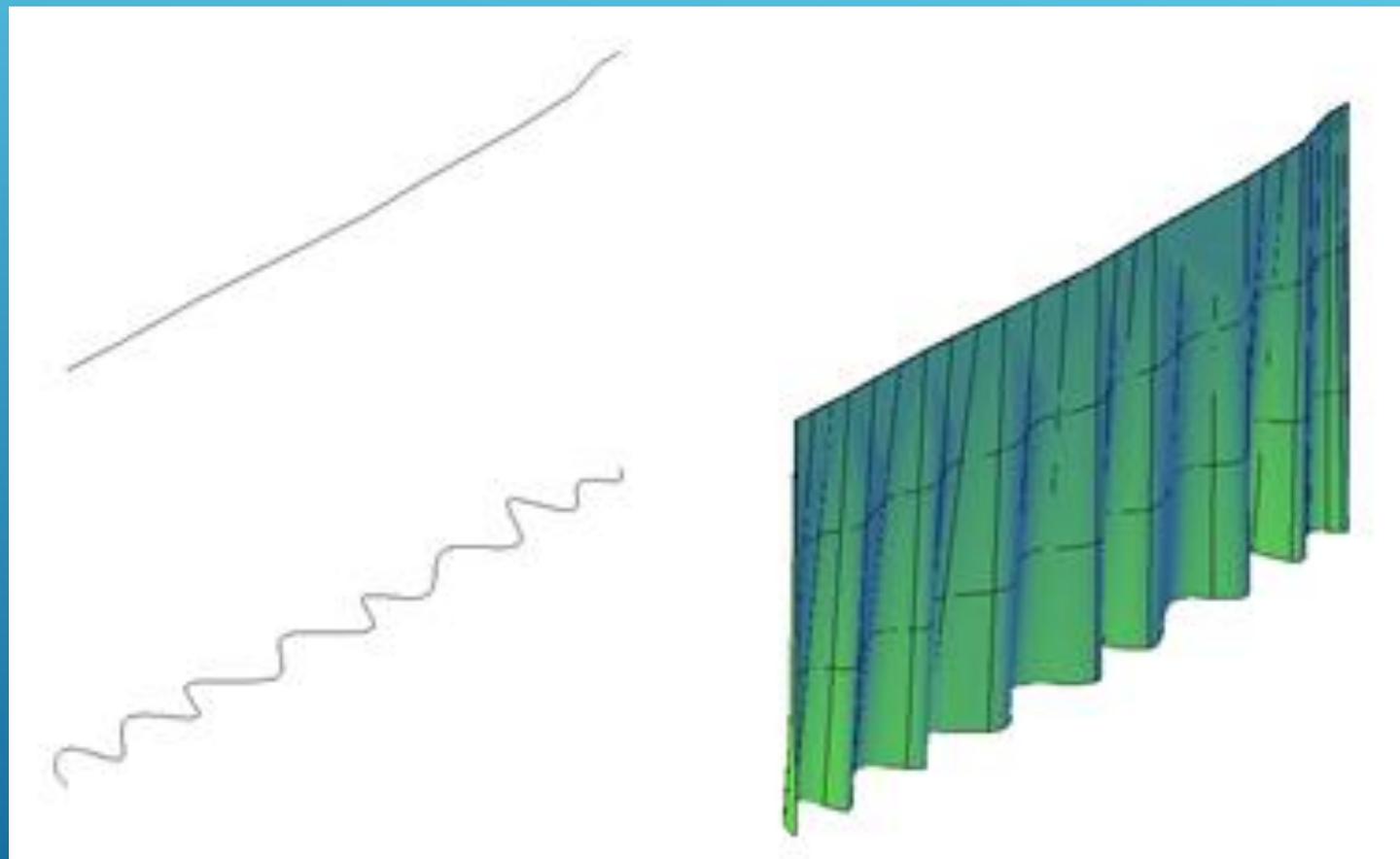
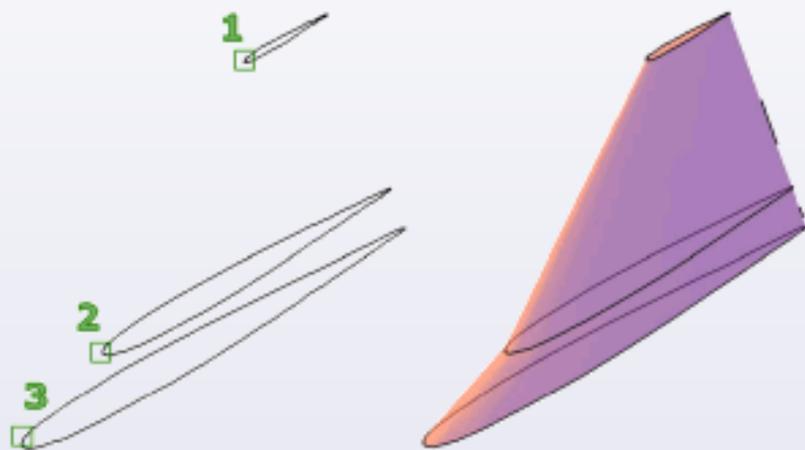
O comando **LOFT** é similar ao comando EXTRUDE mas é muito mais versátil: ao contrário do primeiro, onde a extrusão é efectuada sobre uma polyline fechada, neste caso a extrusão pode ser efectuada sobre diversas polylines fechadas e torna-las um objecto contínuo.

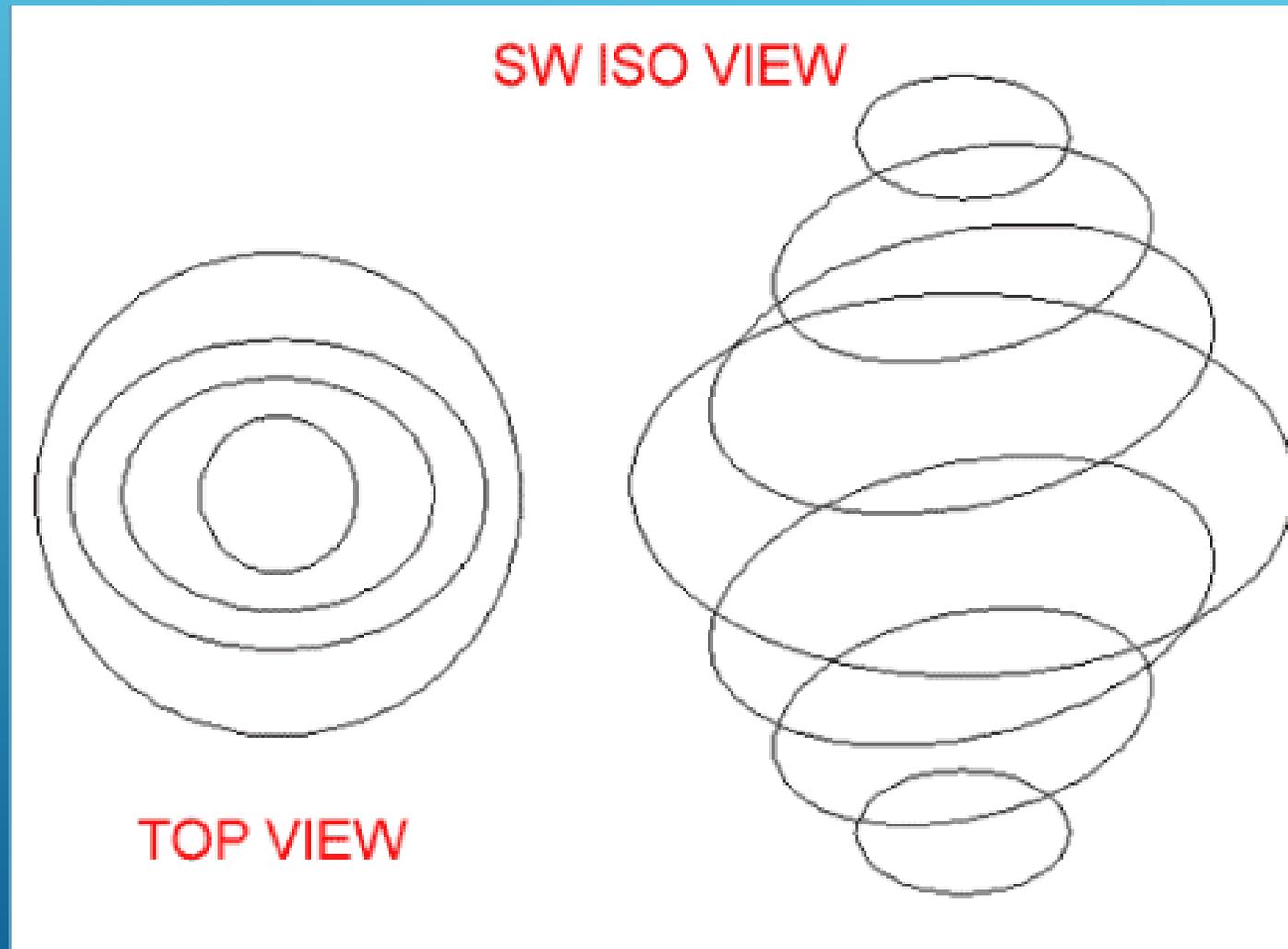


Loft

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.



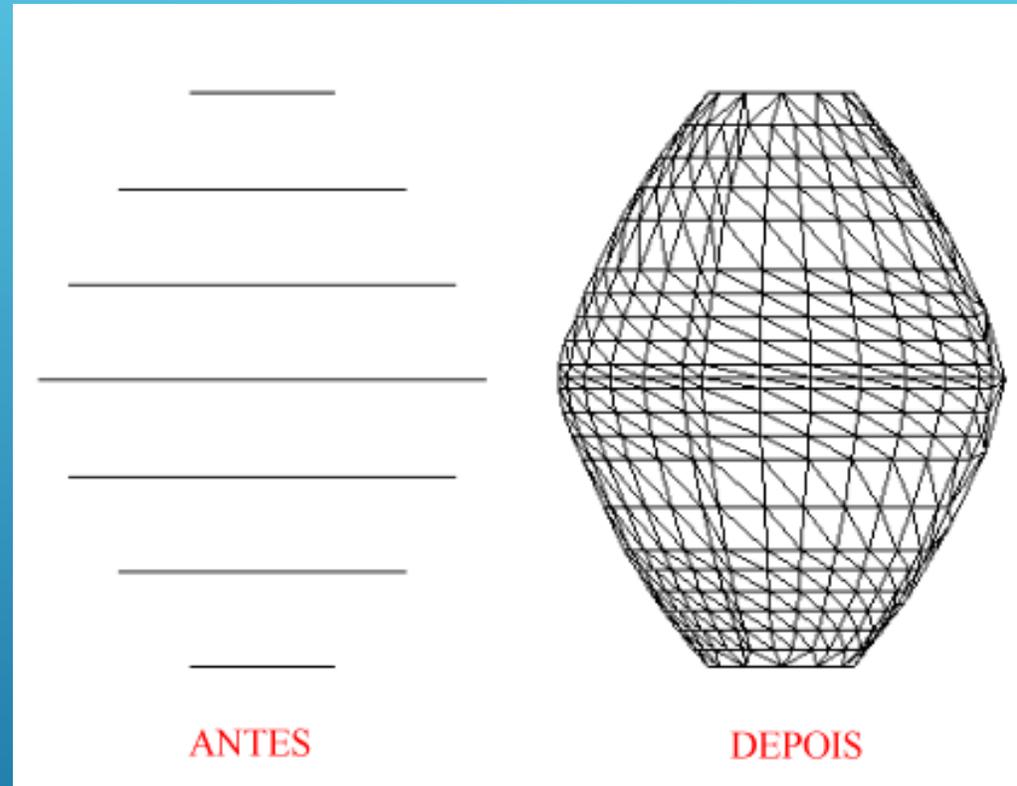


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<https://www.youtube.com/watch?v=-csG9hukV0w>

<https://www.youtube.com/watch?v=mwZcCmnbRIA>

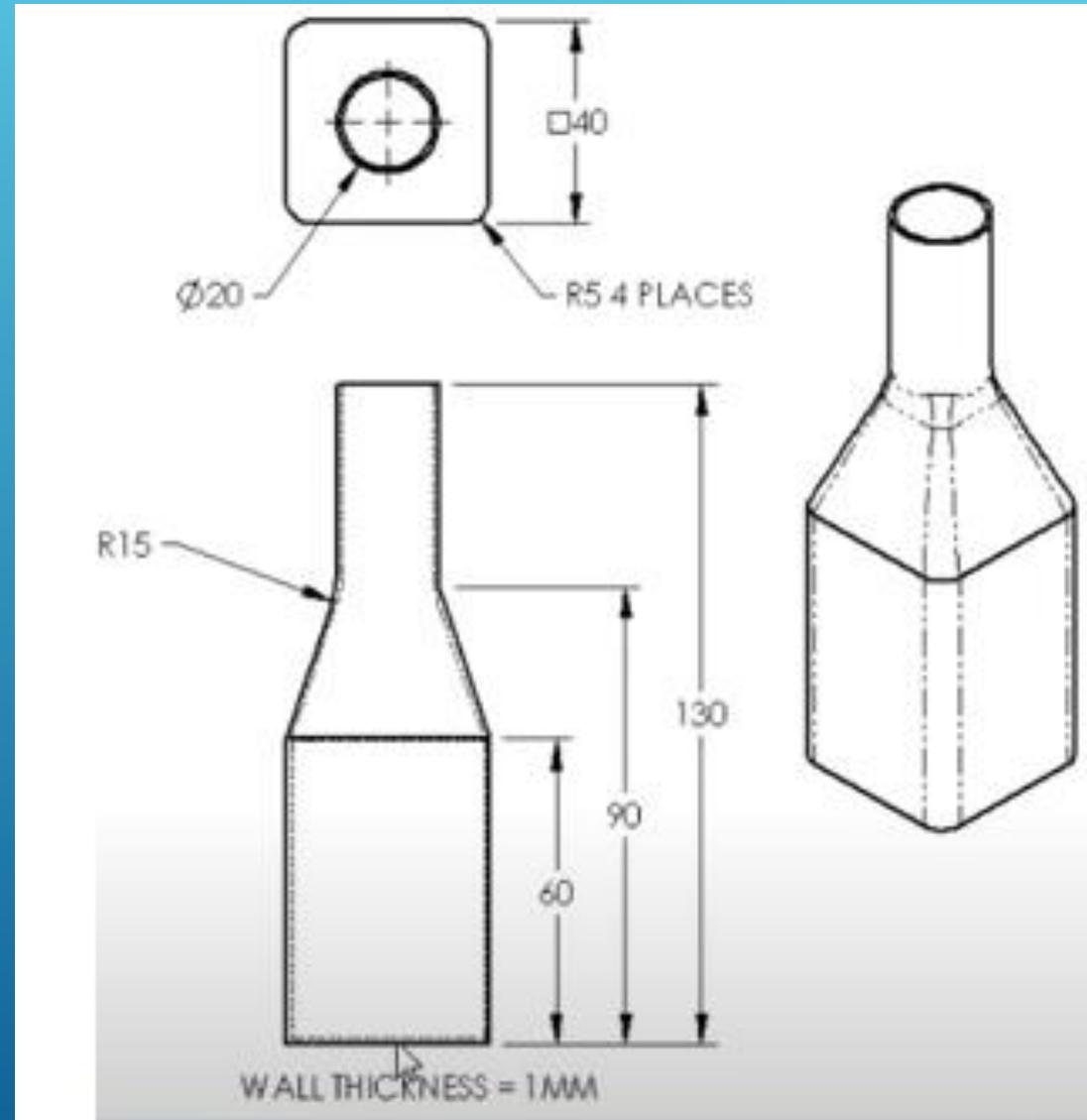
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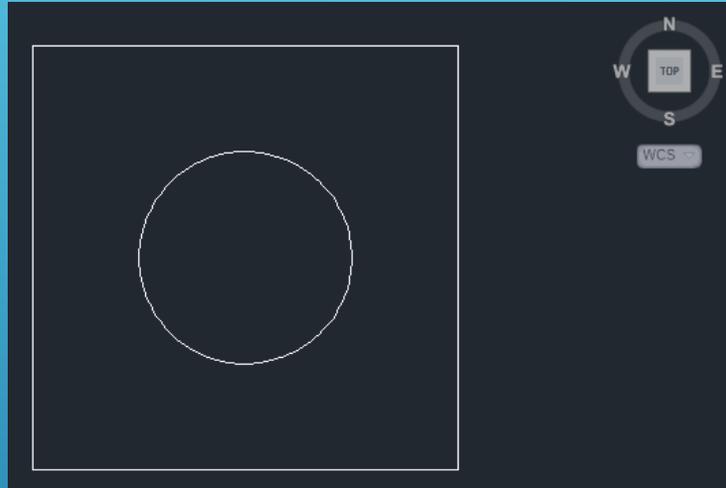


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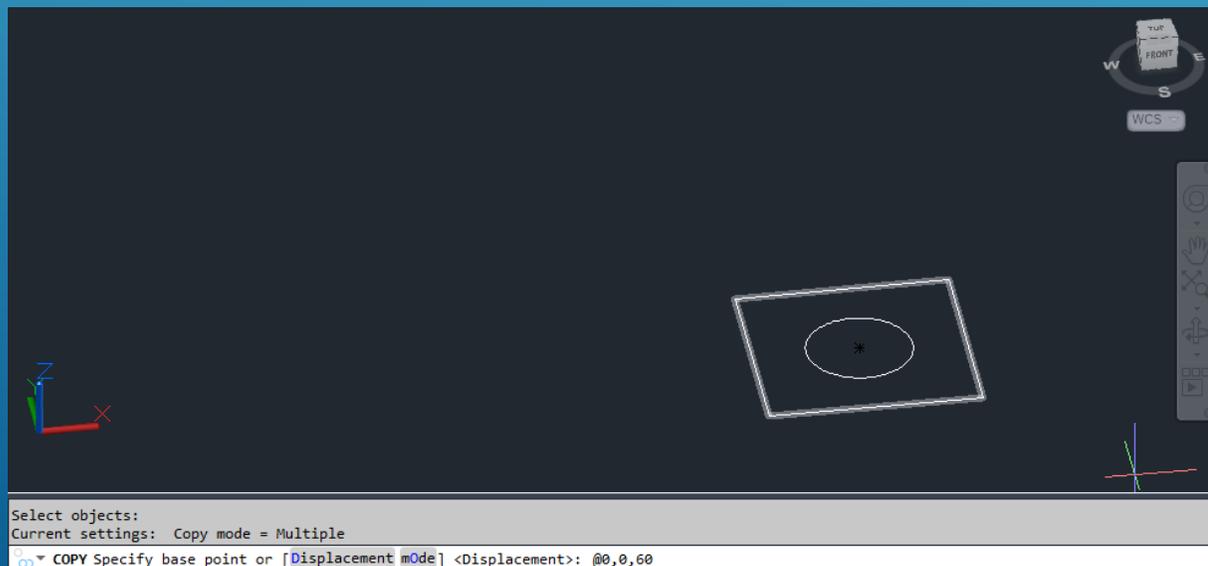
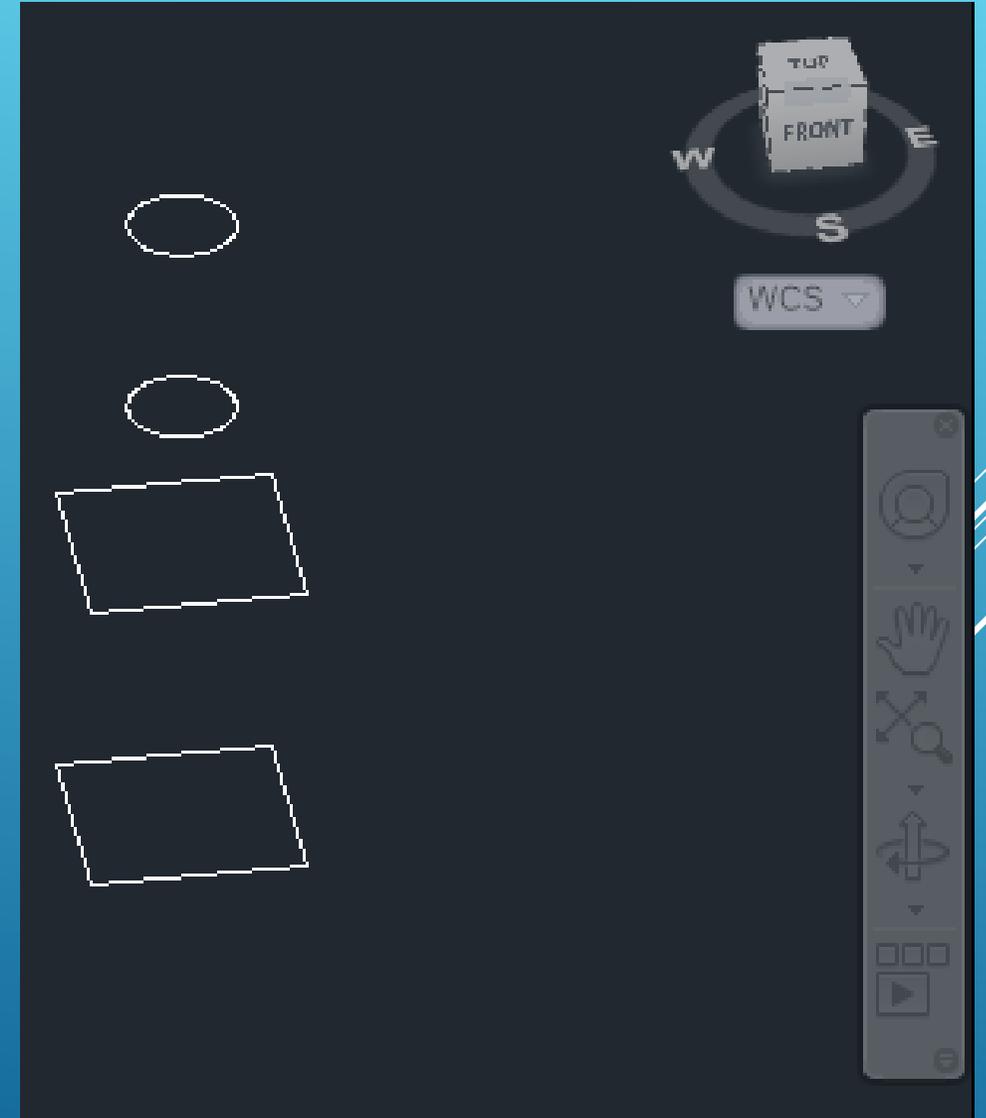


Rectangle
first point
dimensions

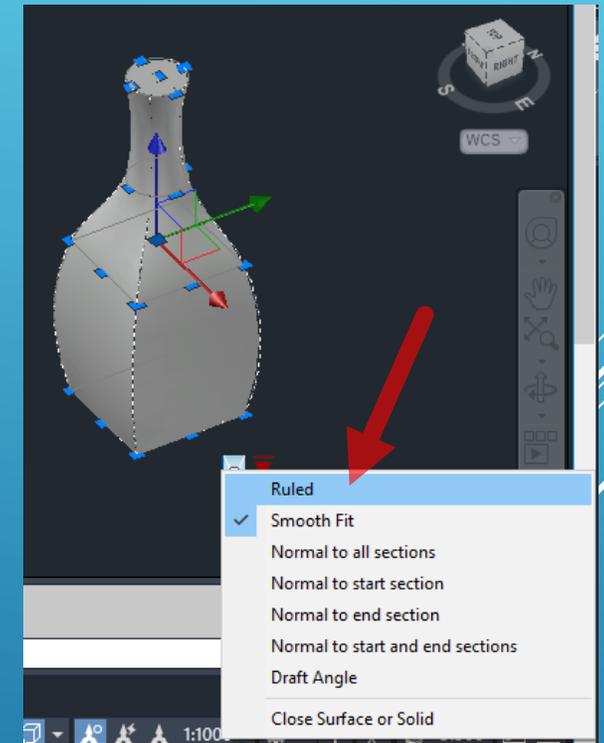
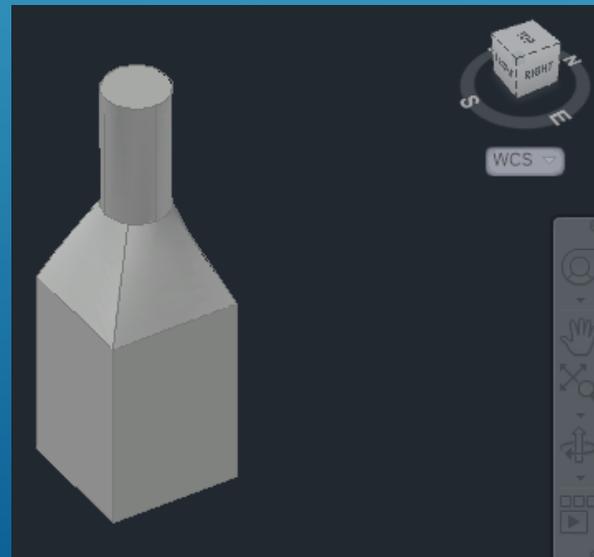
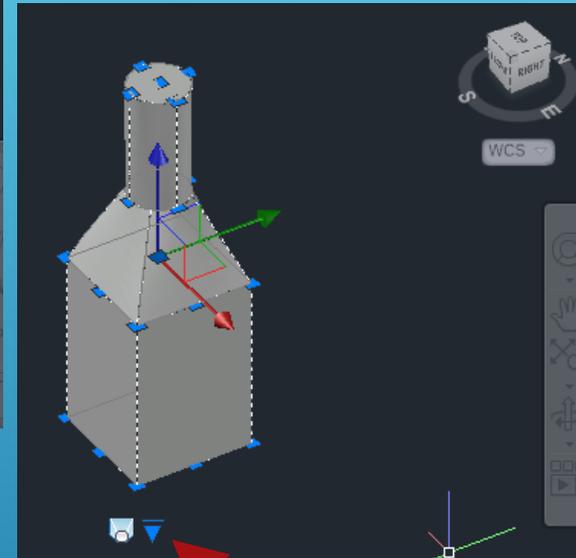
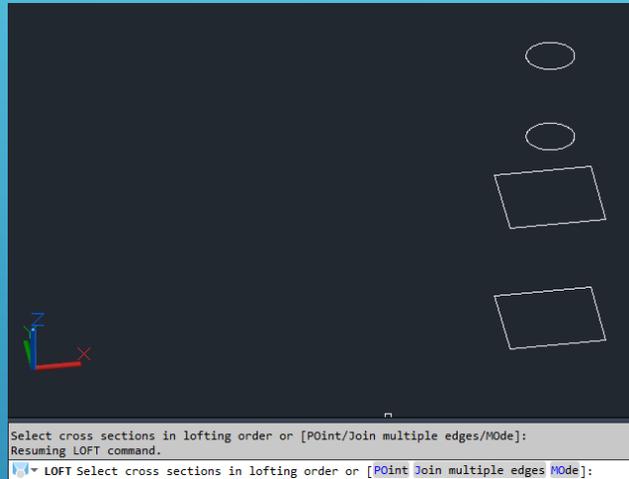
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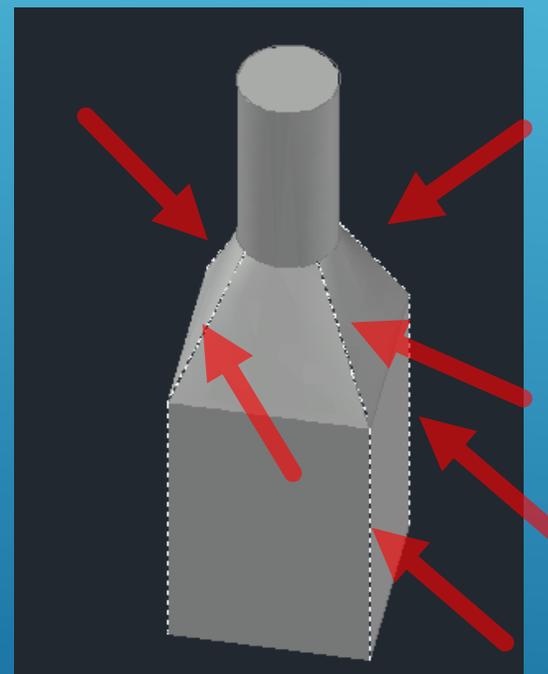
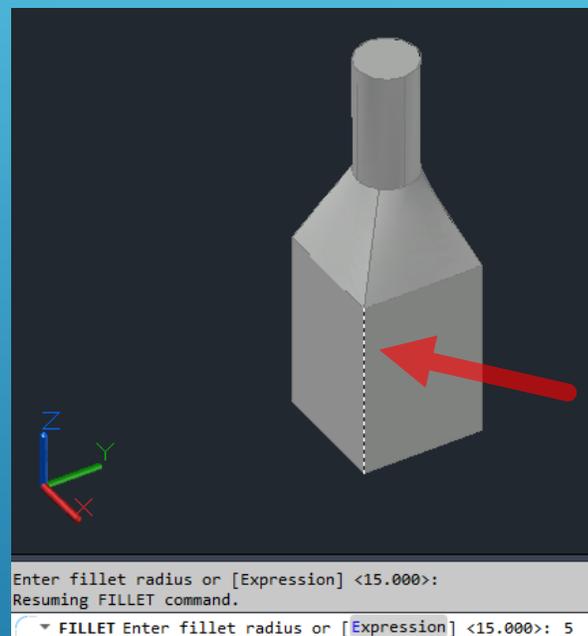
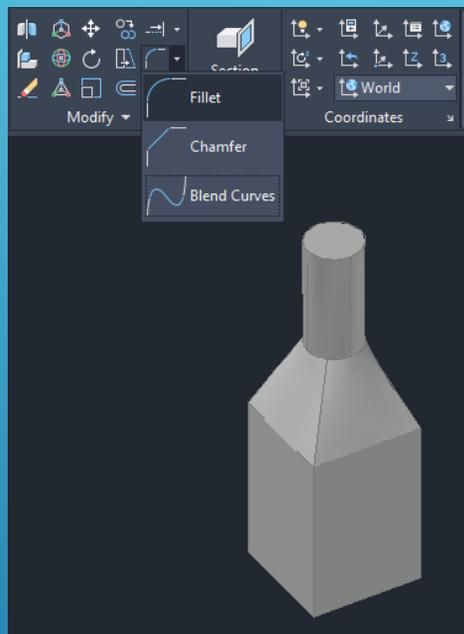
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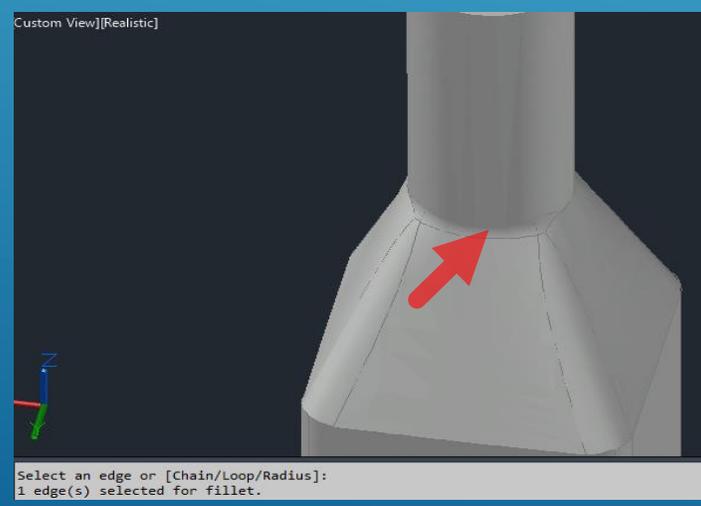
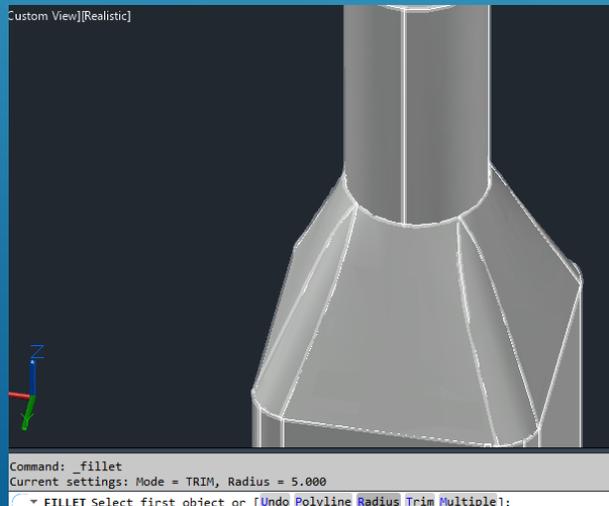
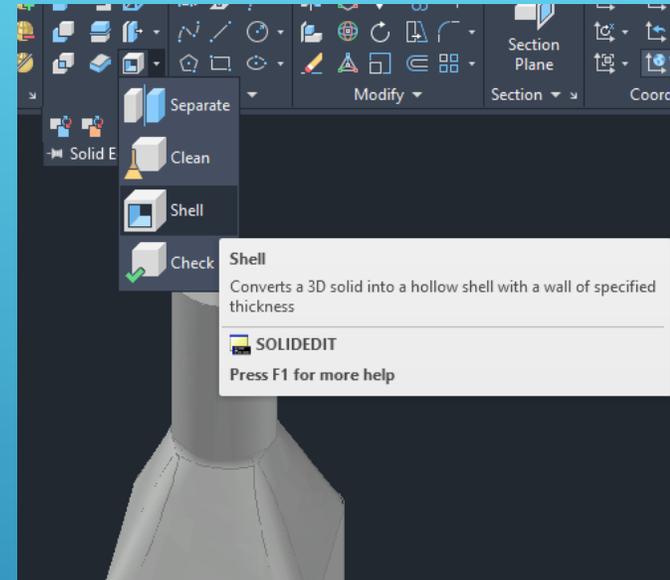
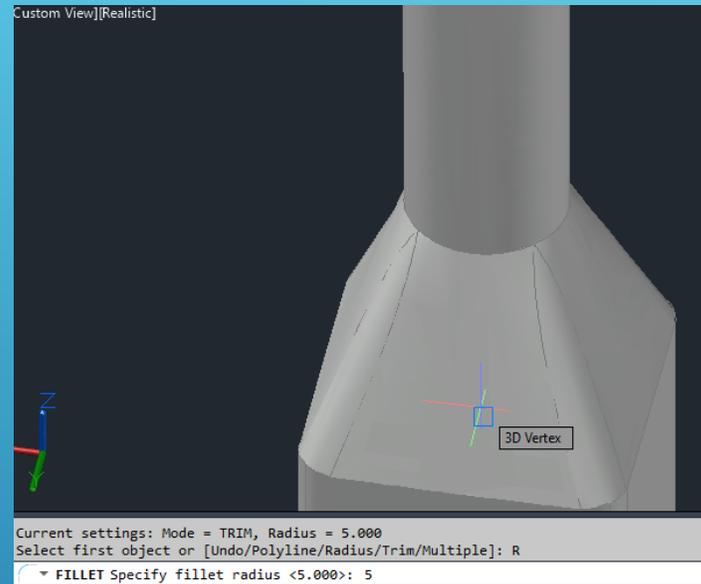
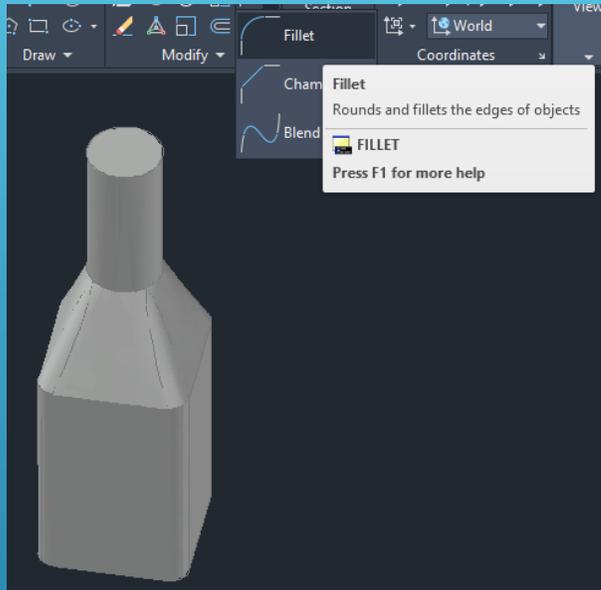
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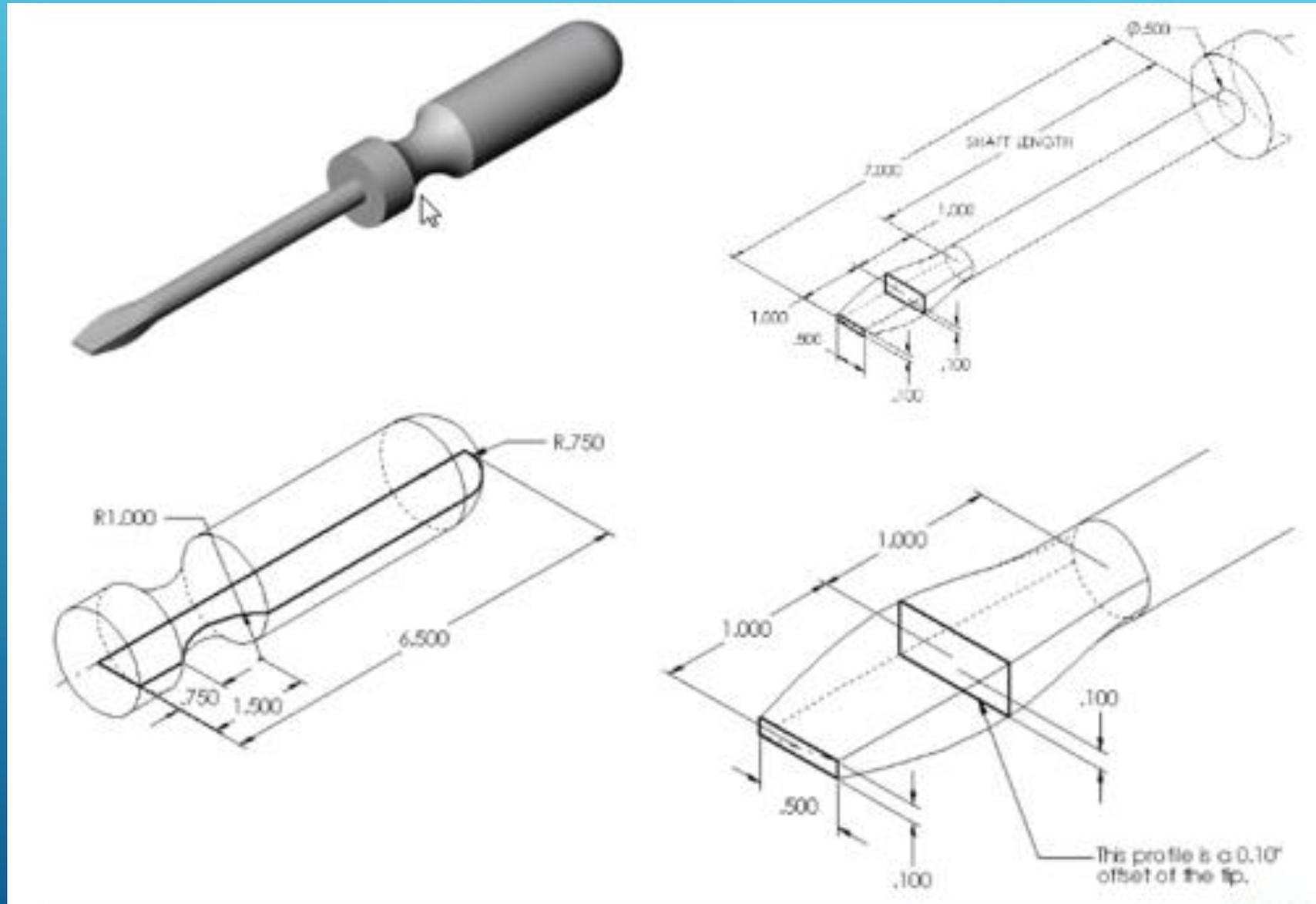


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SOLIDEDIT Enter the shell offset distance: 1



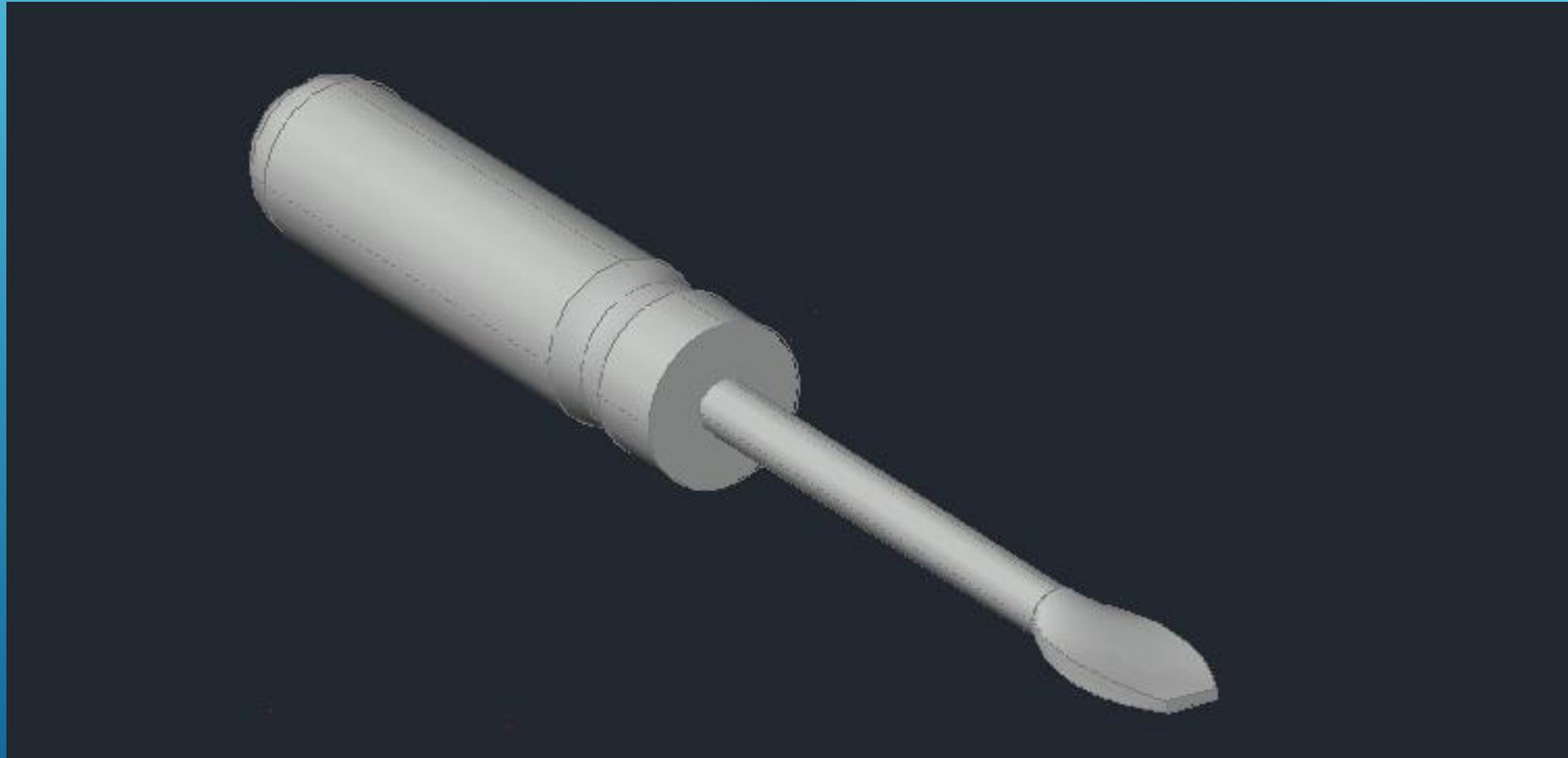


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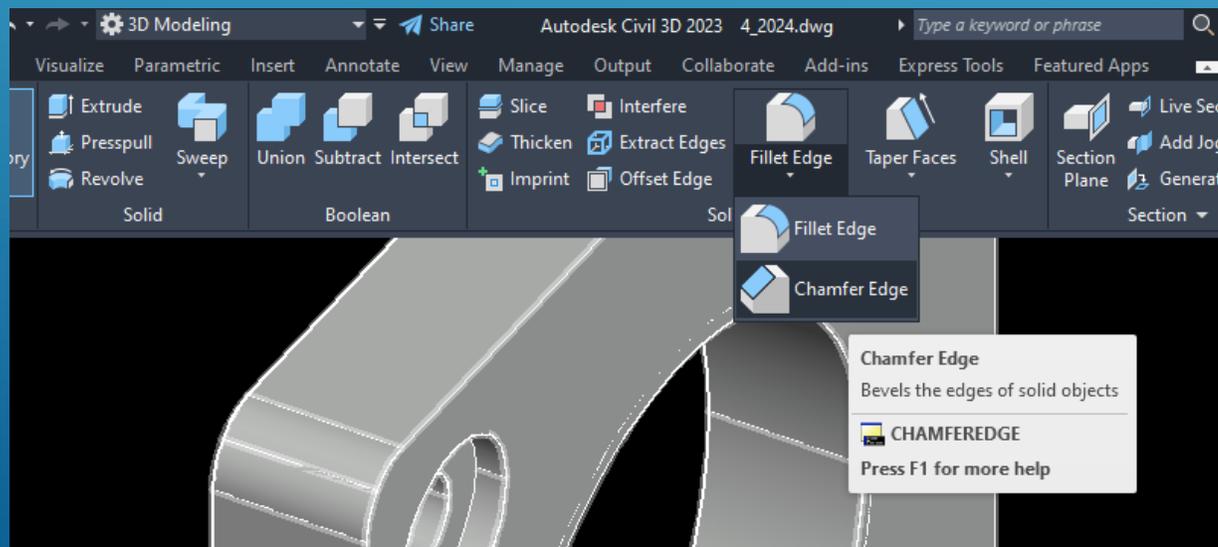
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To Chamfer a 3D Solid

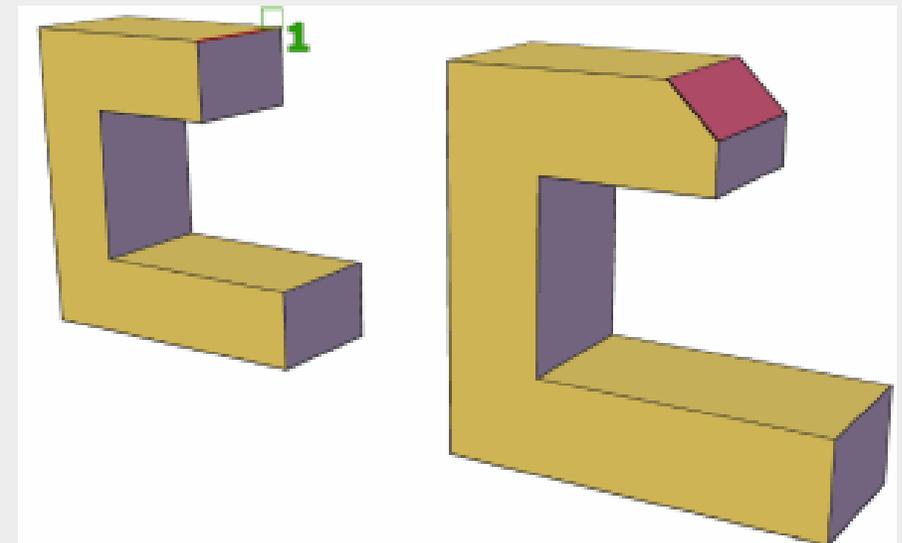
1. Click Solid tab Solid Editing panel Chamfer Edge. Find.
2. Select the edge to chamfer. A preview of the chamfer is displayed.
3. Do one of the following: ...
4. Specify the distances for the base surface and the other, adjacent surface. ...
5. Do one of the following: ...
6. To complete the chamfer, press Enter.



Chamfer Edge

Bevels the edges of solid objects

You can select more than one edge at a time, as long as they belong to the same face. Enter a value for the chamfer distance or click and drag the chamfer grips.

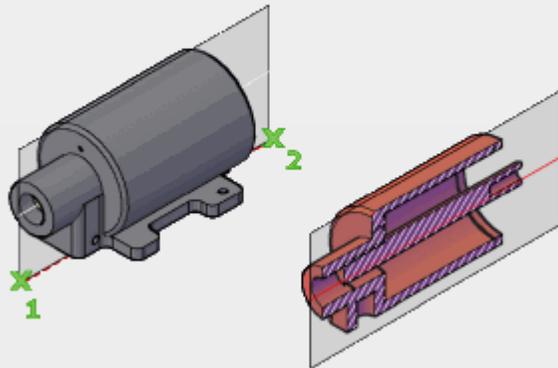


 CHAMFEREDGE

Press F1 for more help

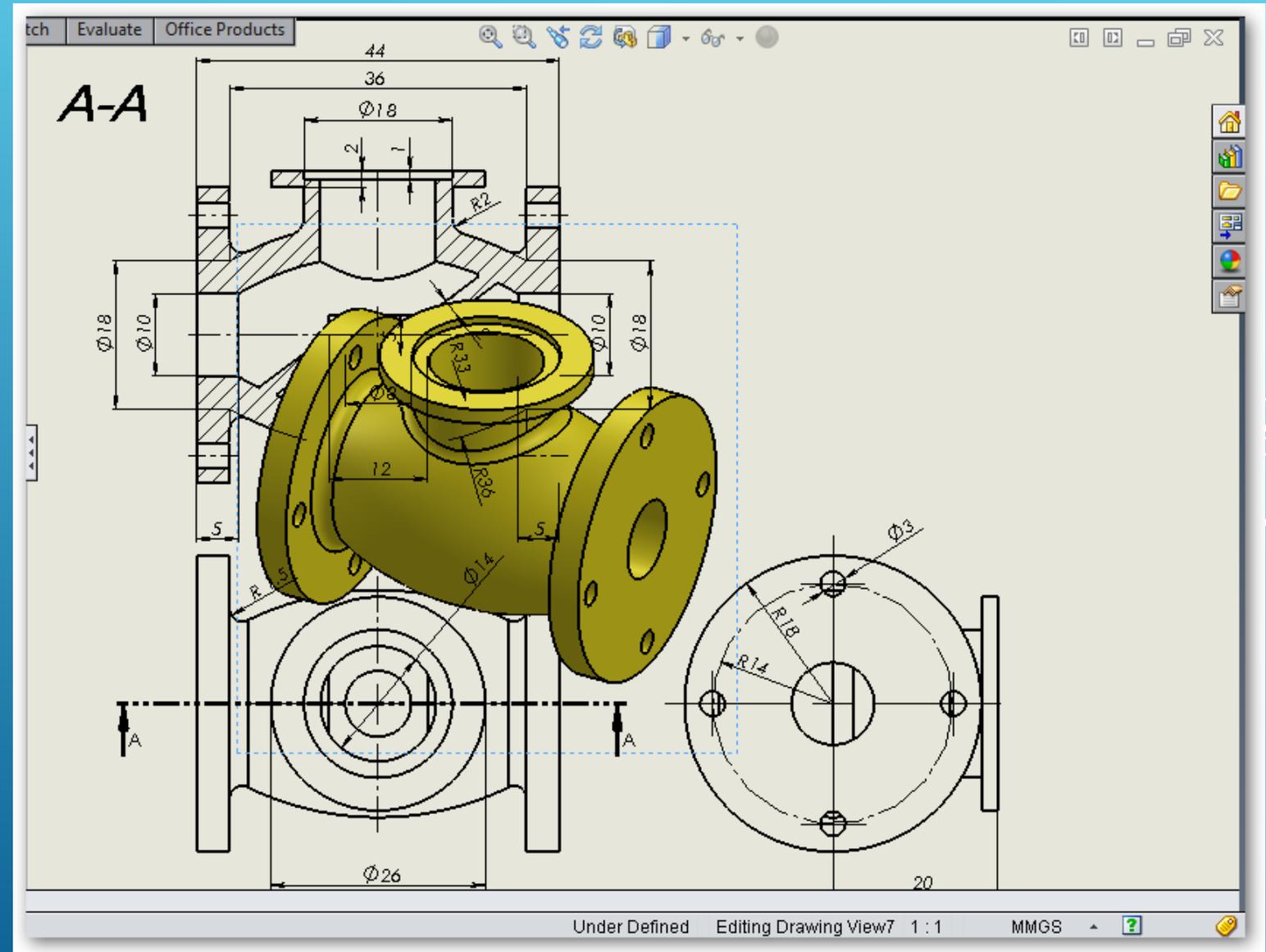
Creates a section object that acts as a cutting plane through 3D objects

Section plane objects create sections of 3D solids, surfaces, and meshes. Use live sectioning with section plane objects to analyze a model, and save sections as blocks for use in layouts.



SECTIONPLANE

Press F1 for more help

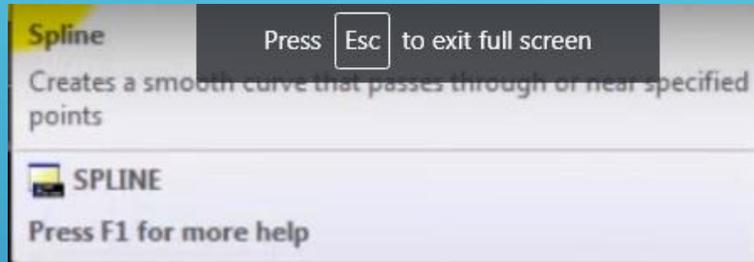


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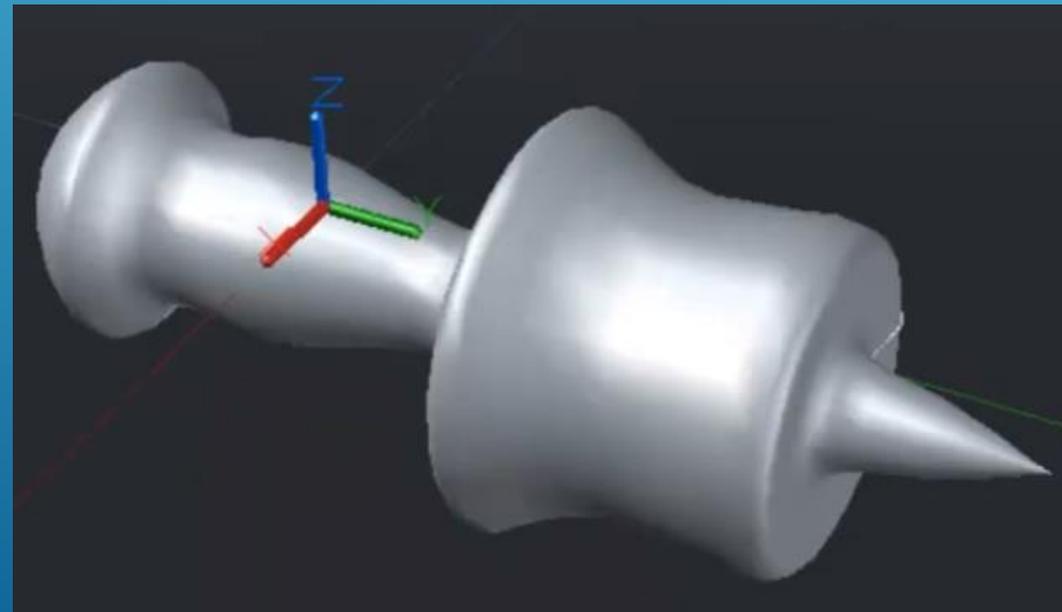
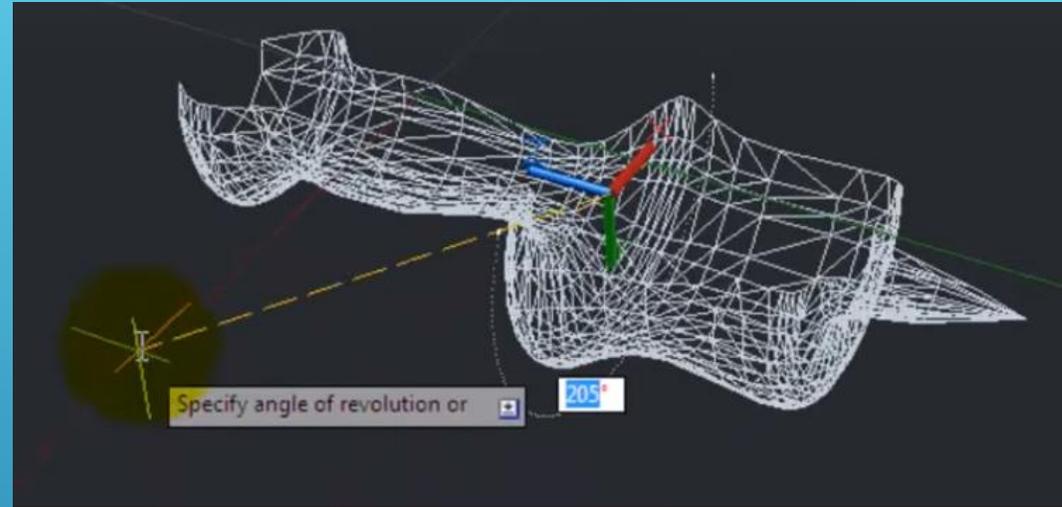
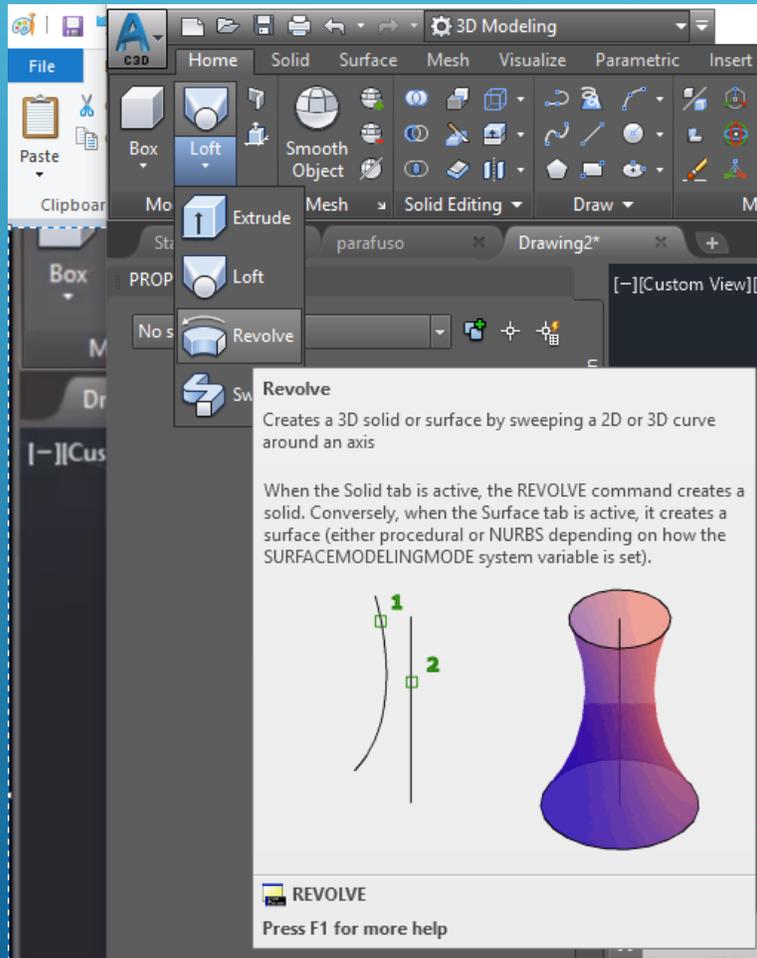


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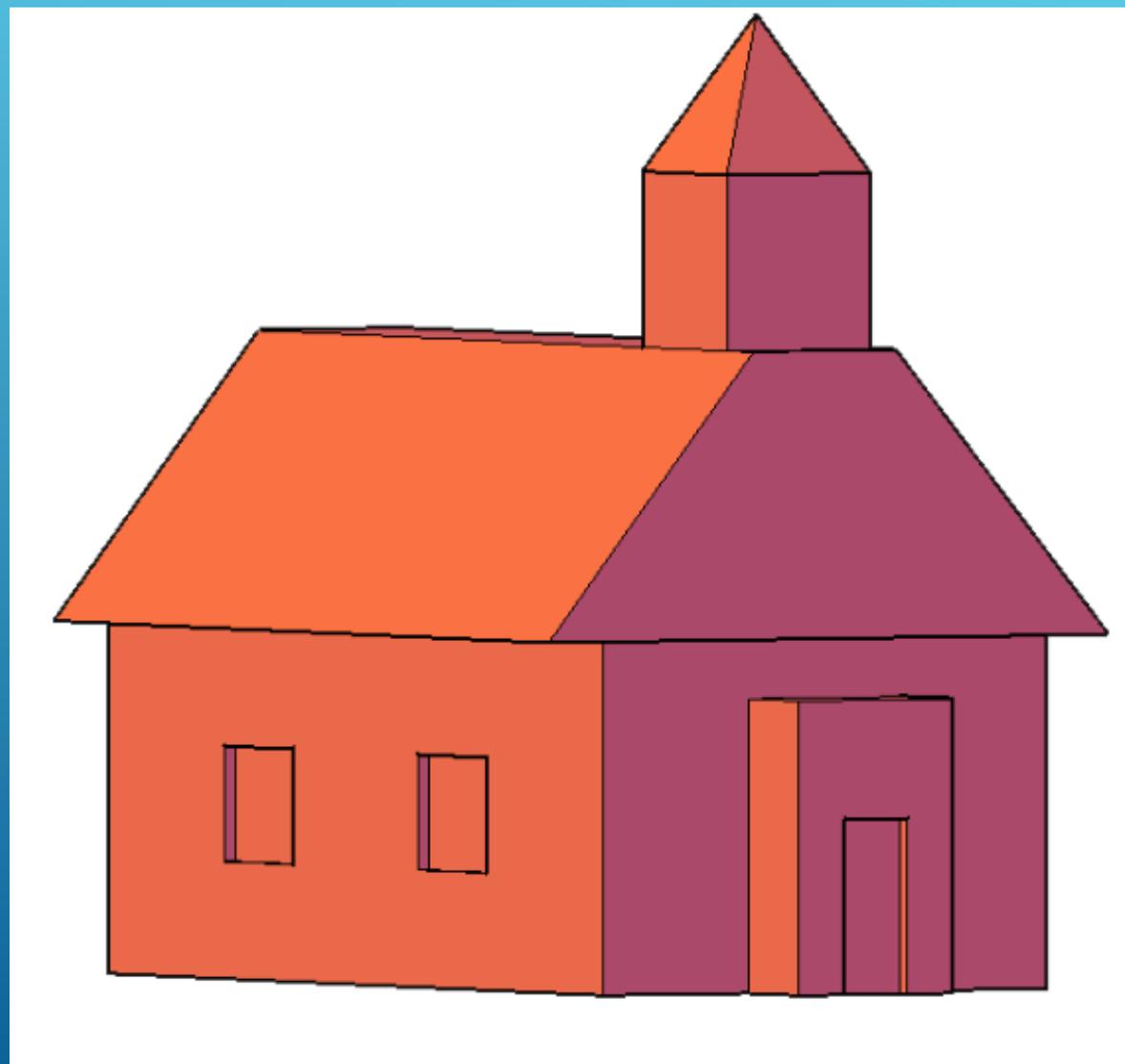


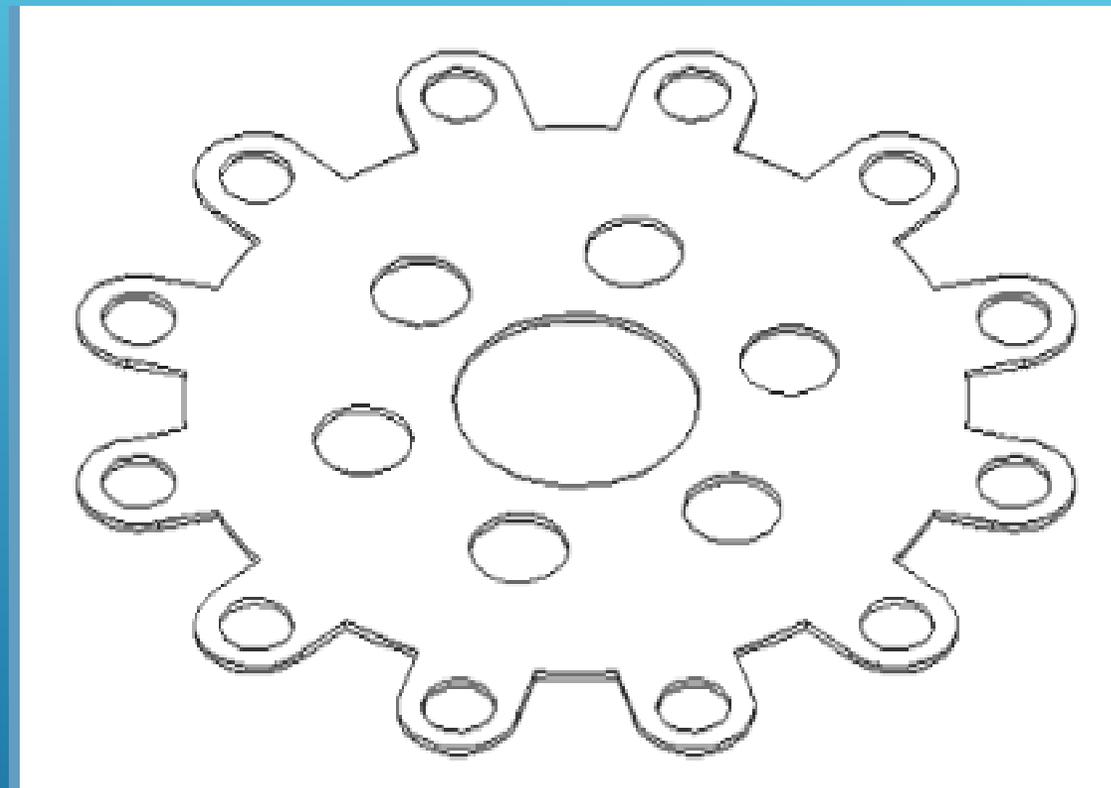
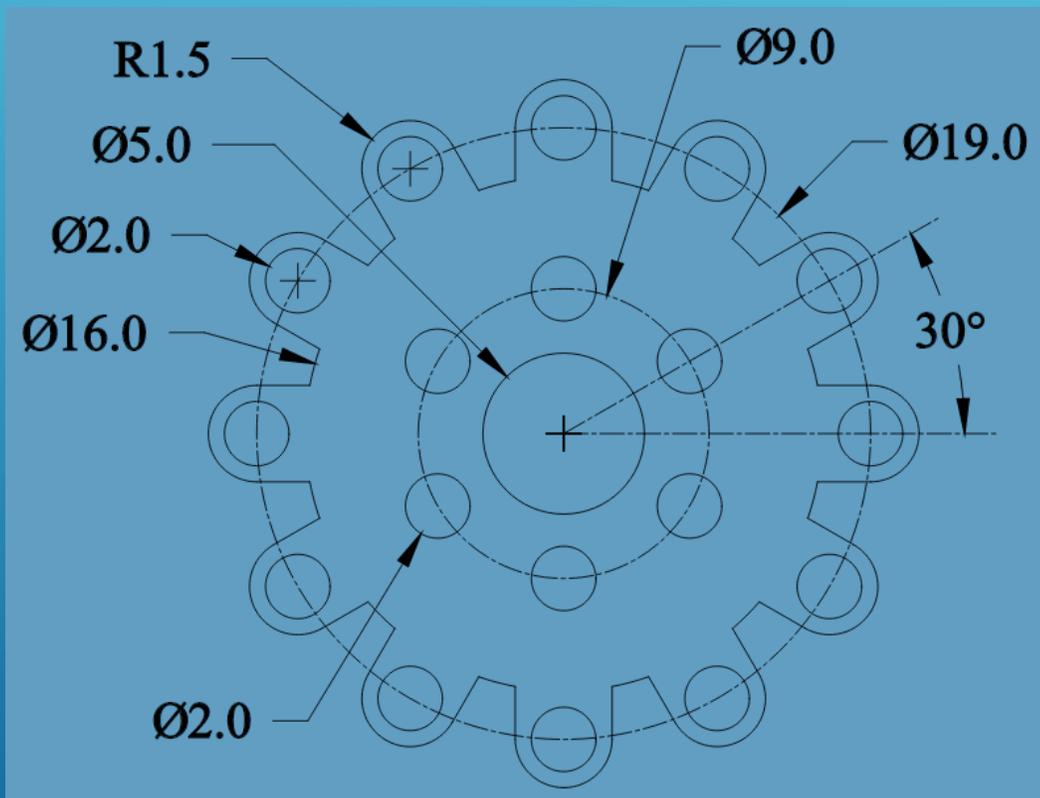
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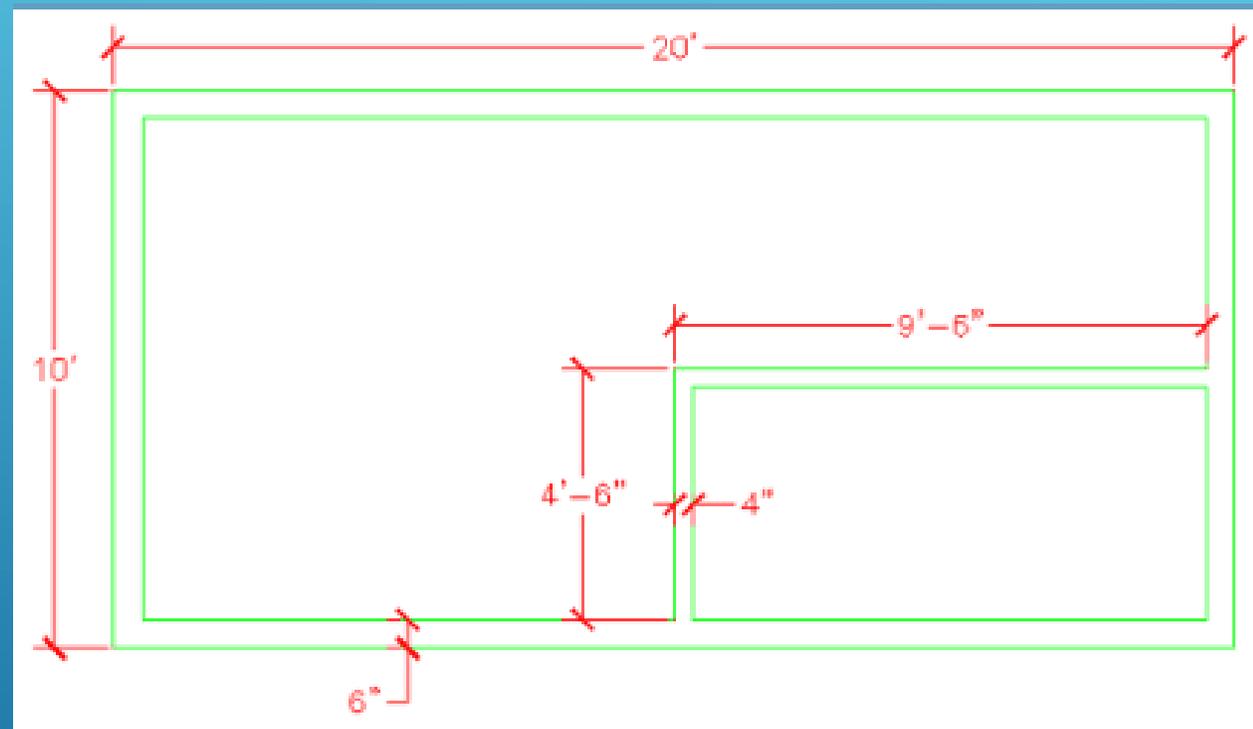
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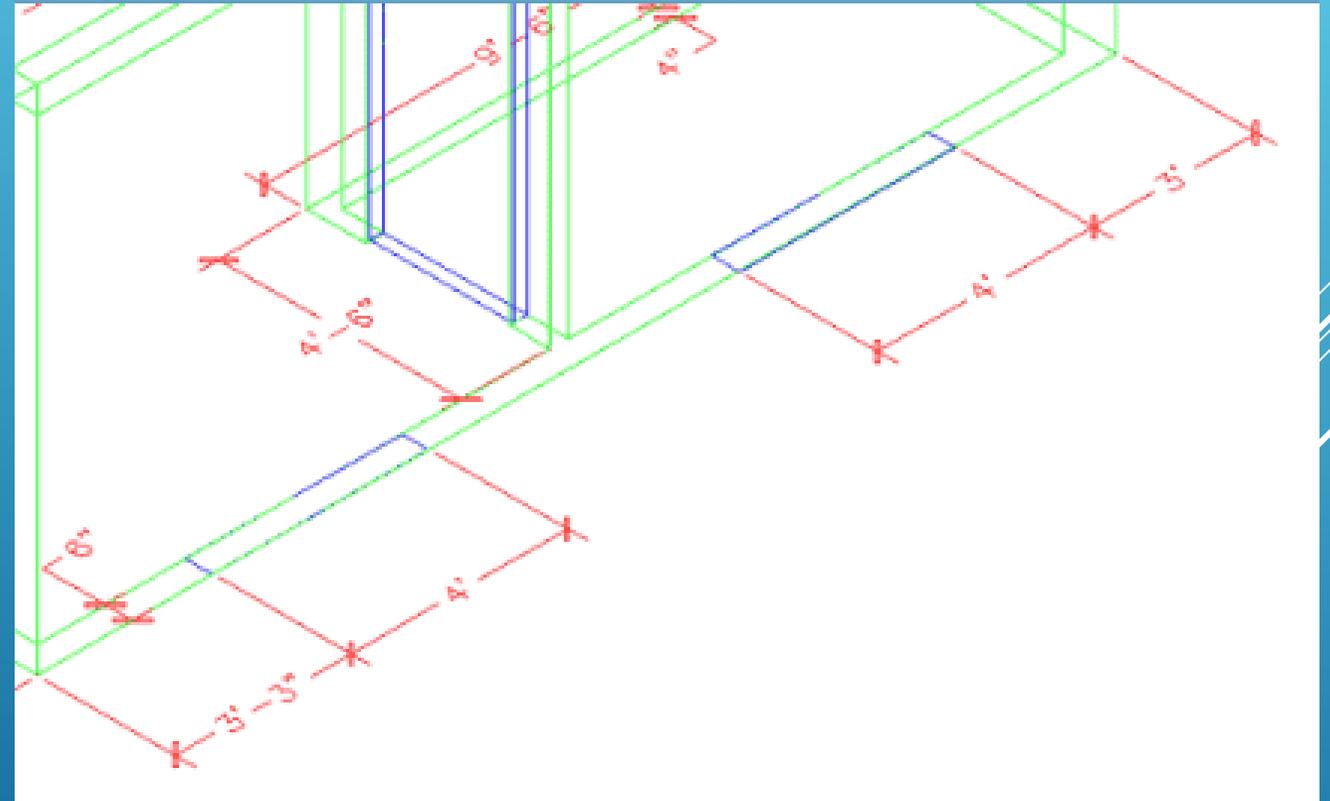
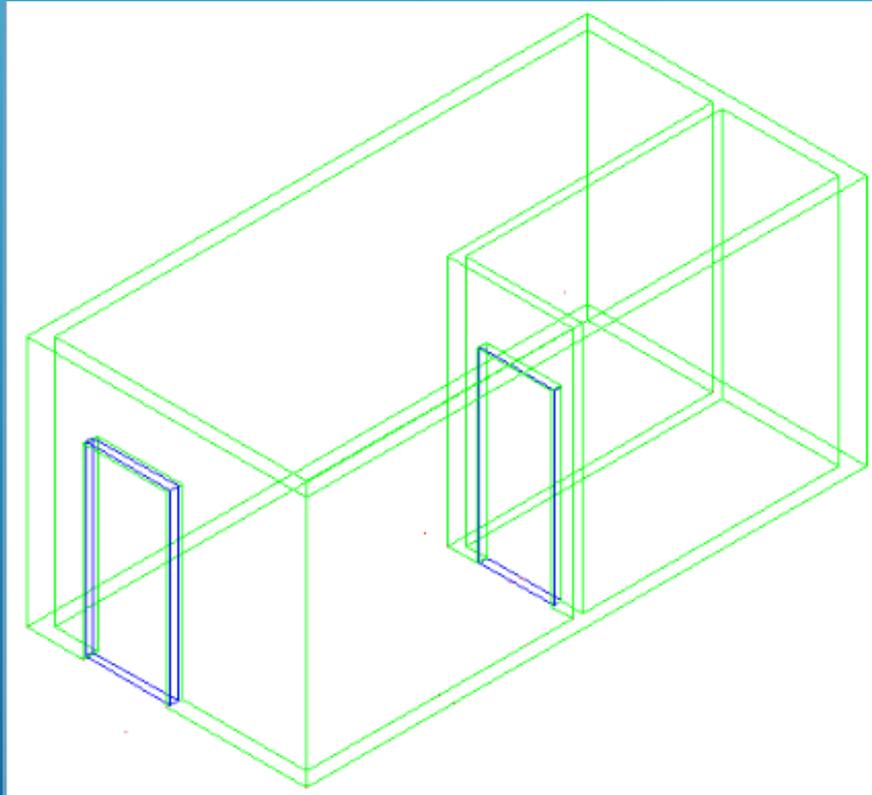
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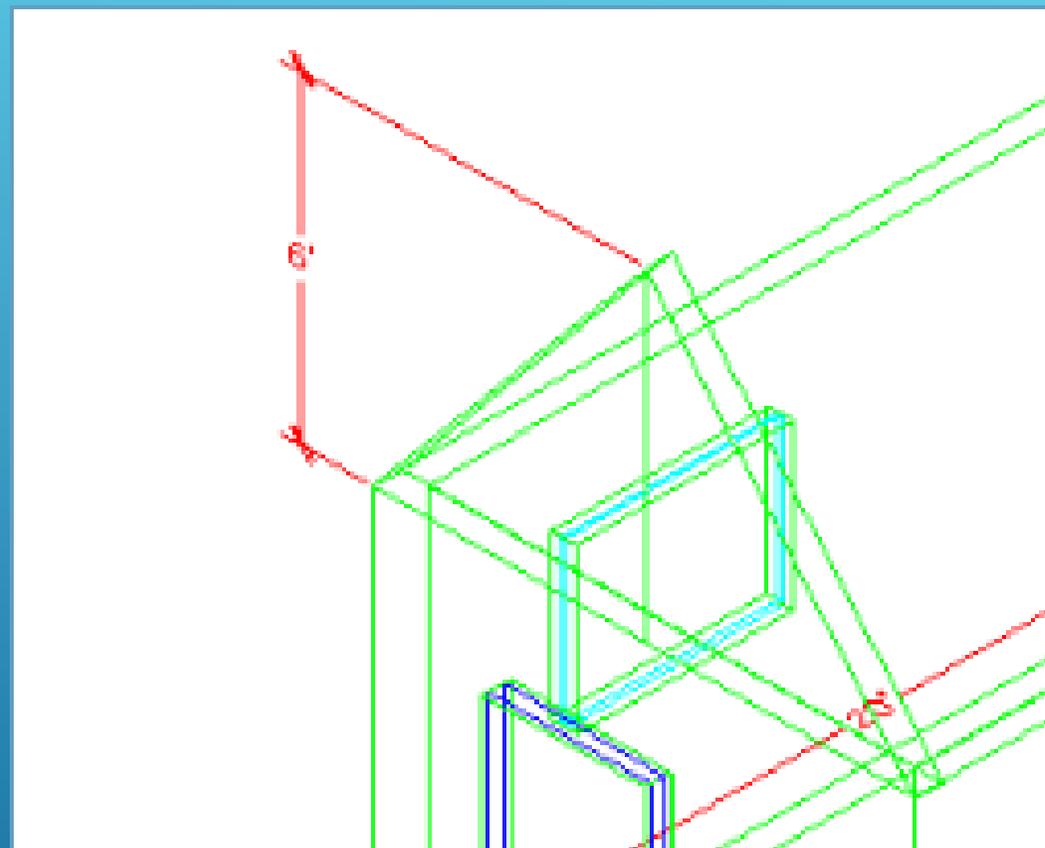
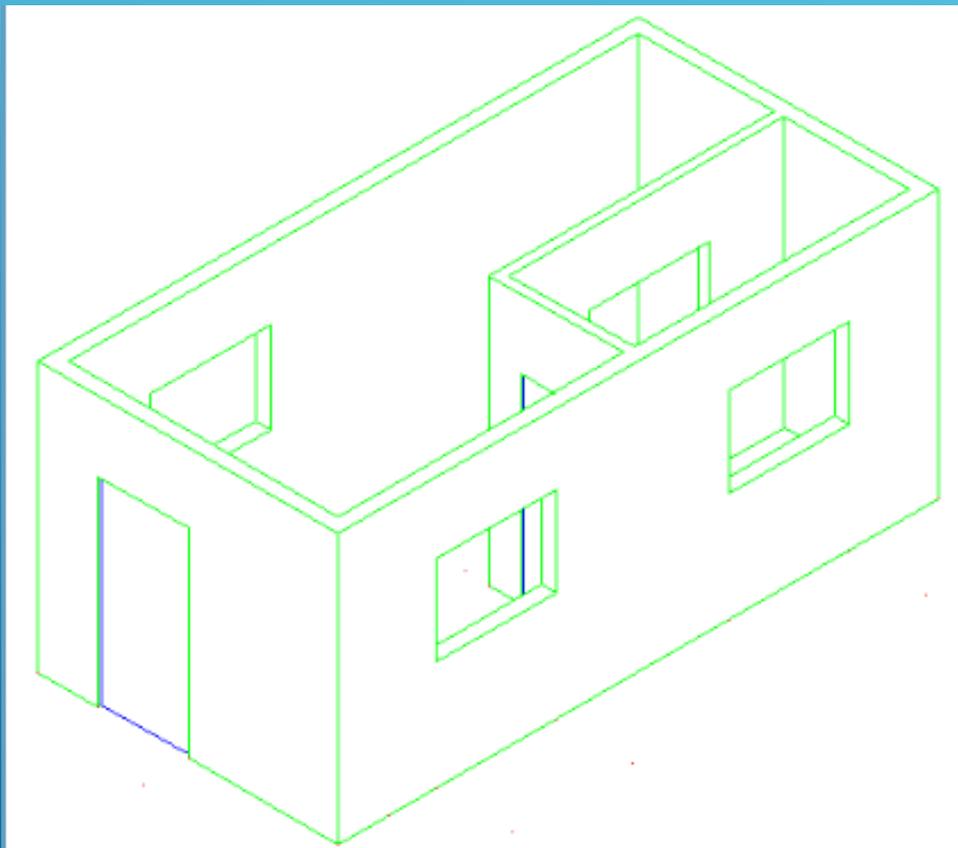




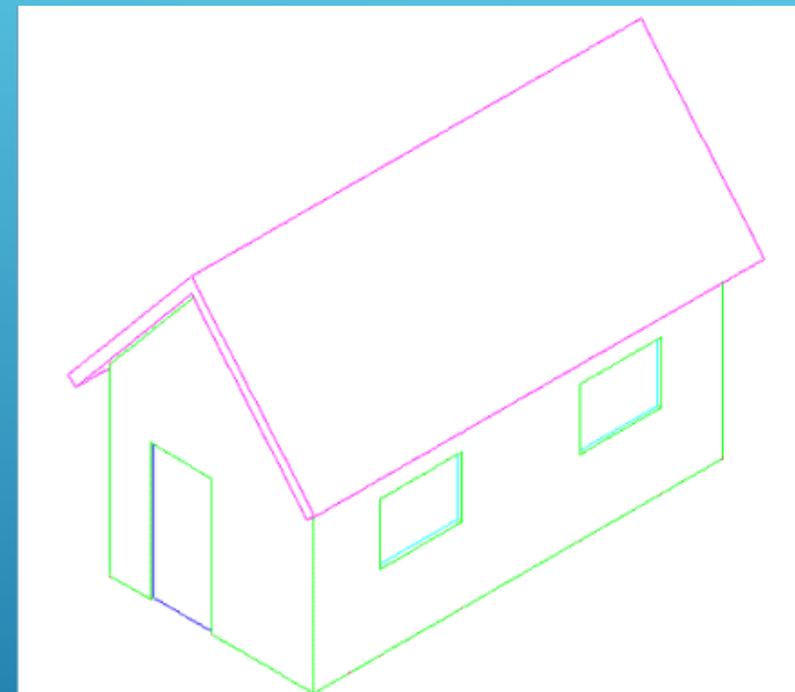
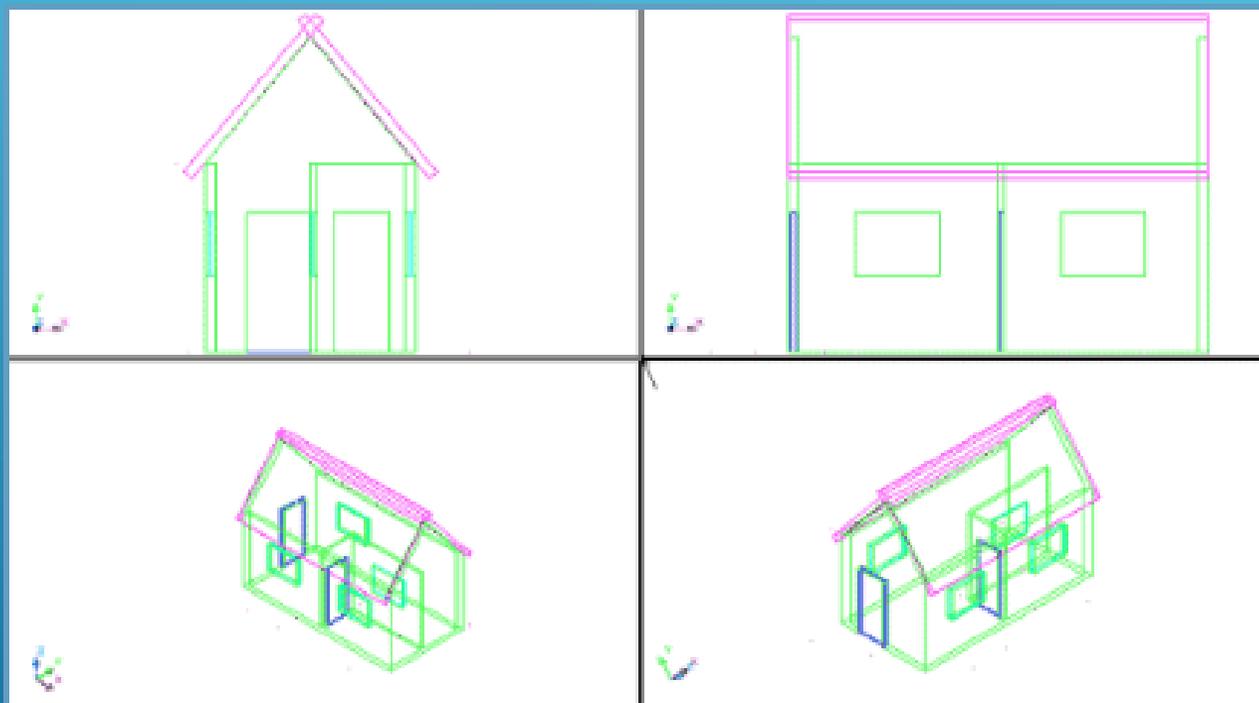
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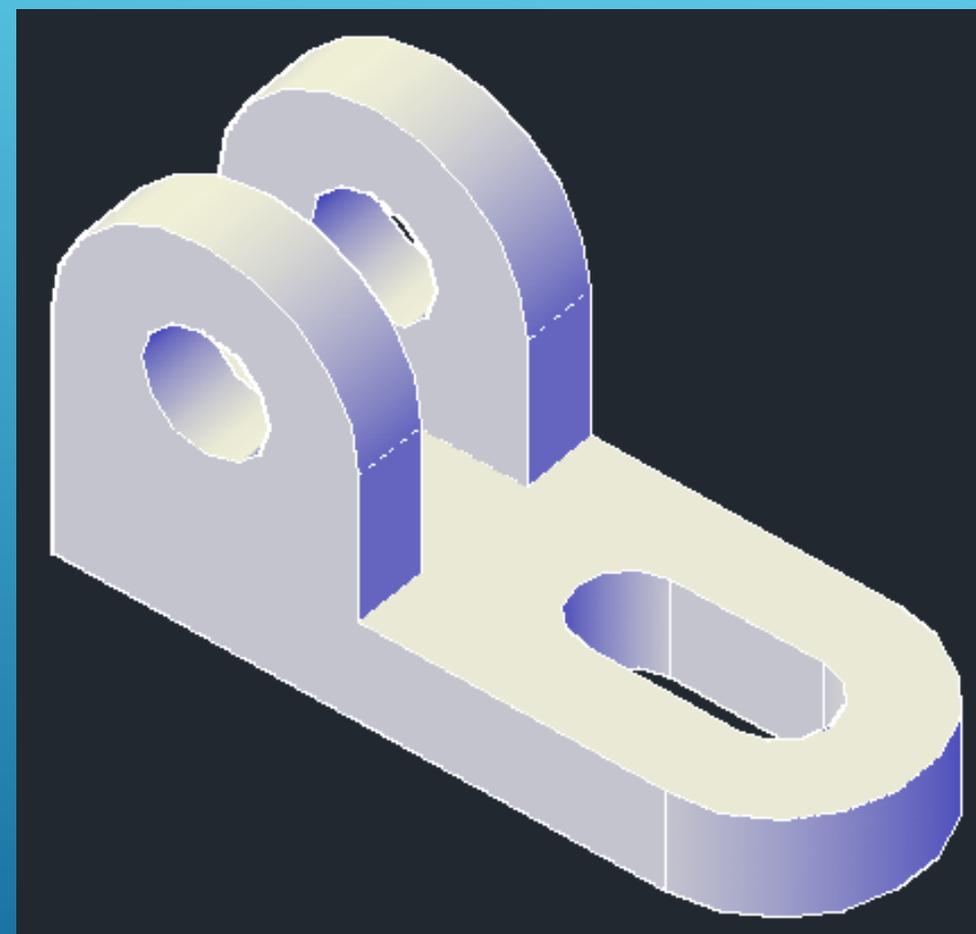
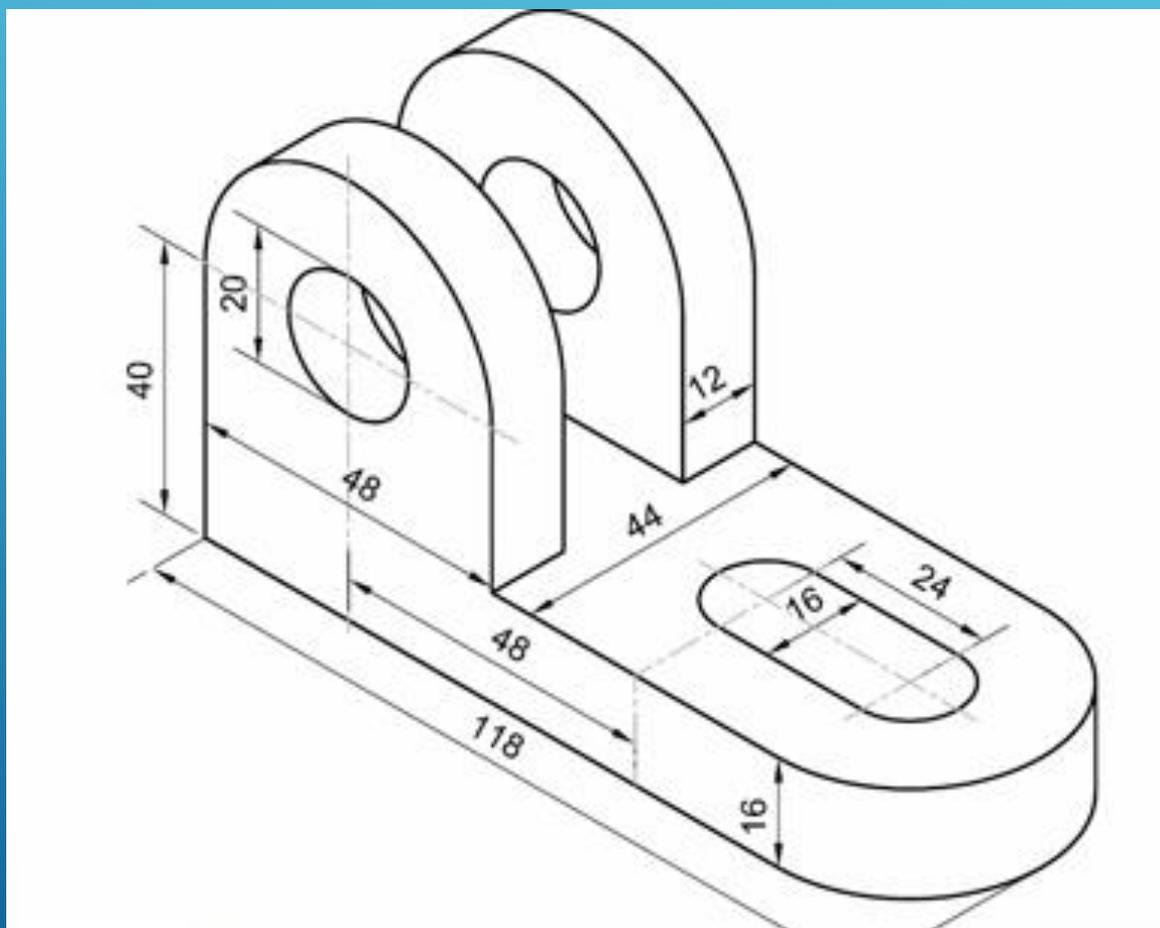


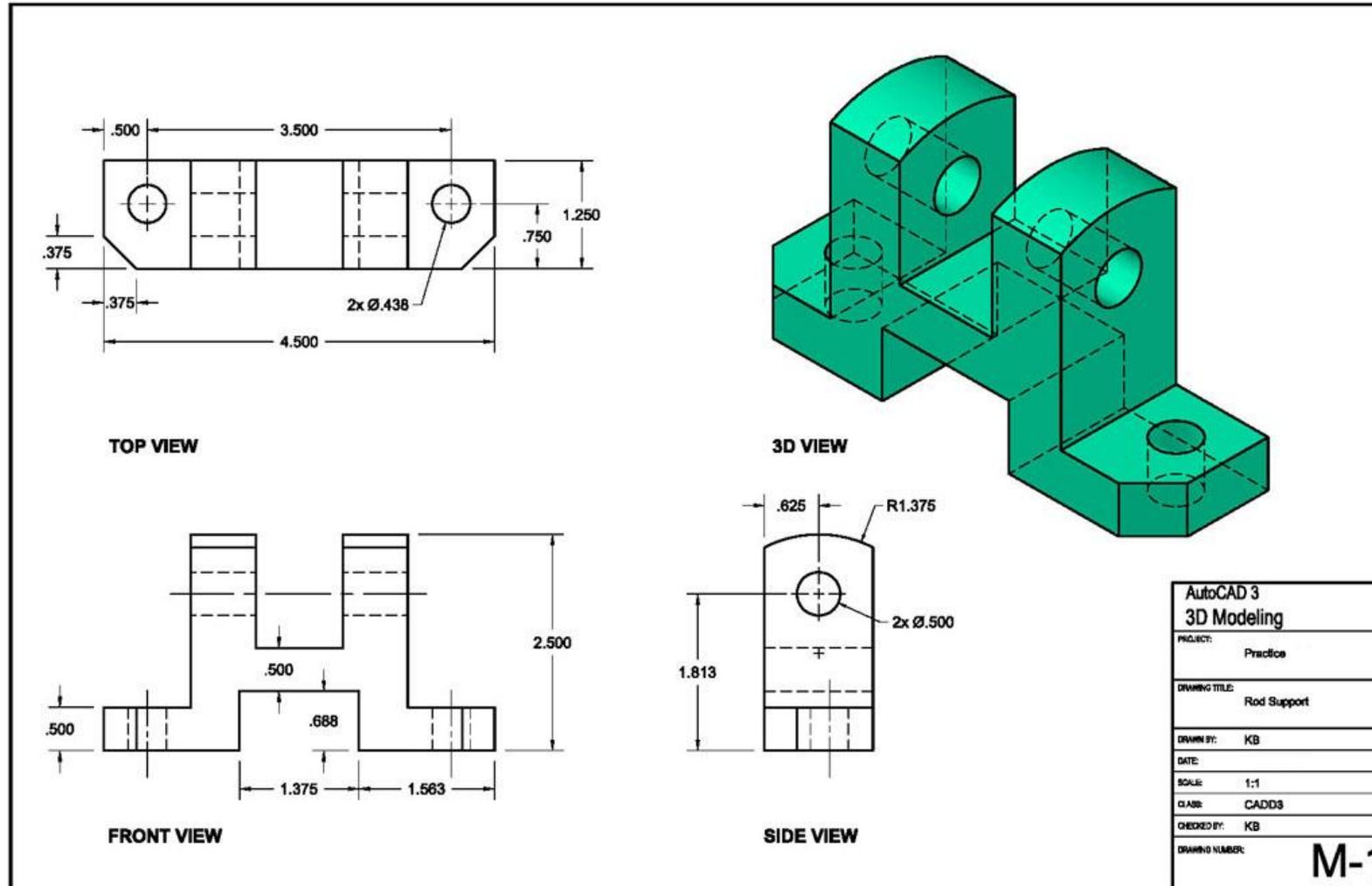




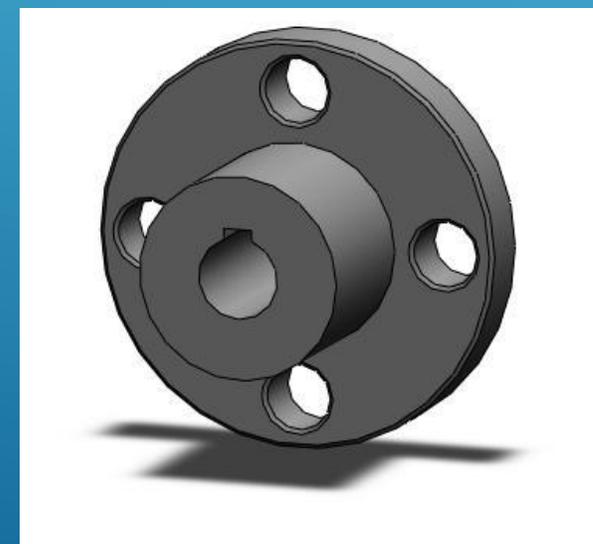
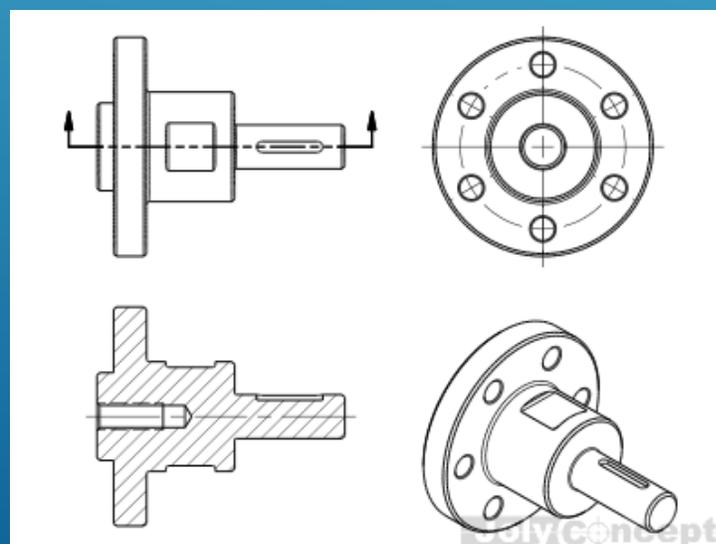
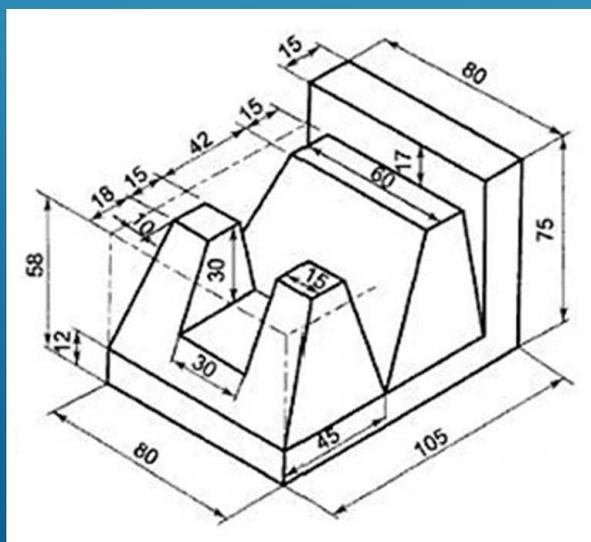
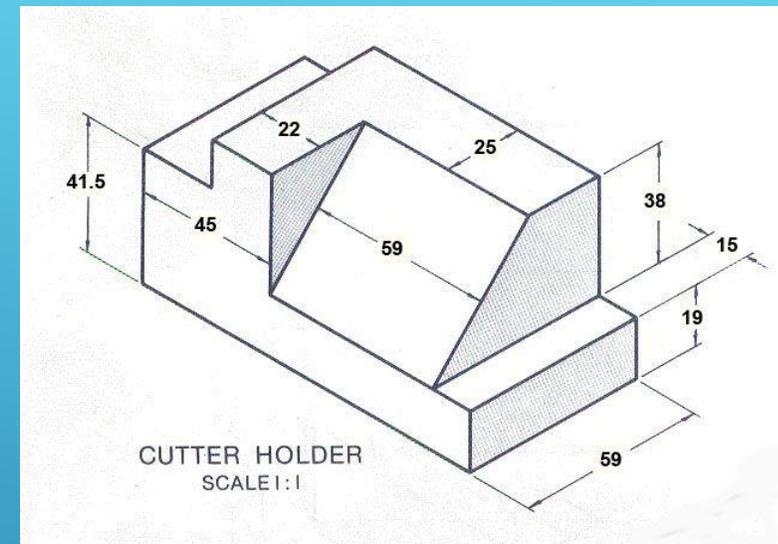
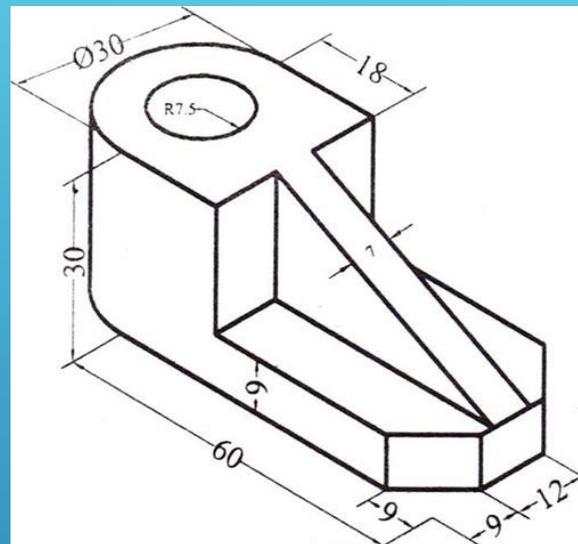
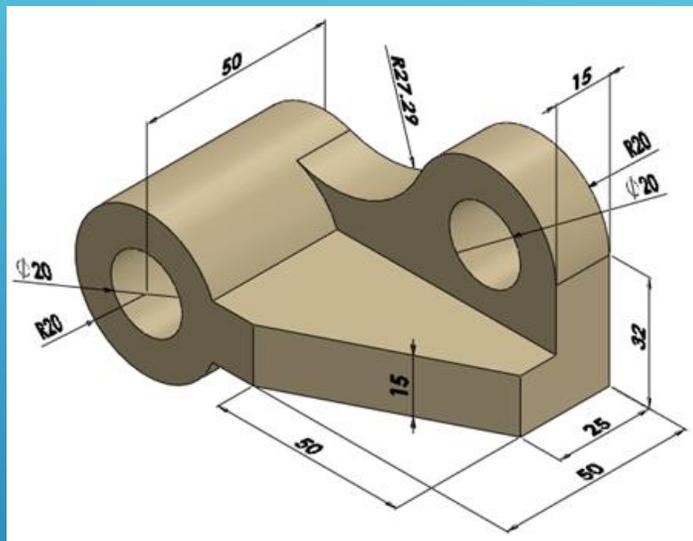
AULA 5 Desenho Técnico Assistido por Computador







AULA 5 Desenho Técnico Assistido por Computador



AULA 5 Desenho Técnico Assistido por Computador

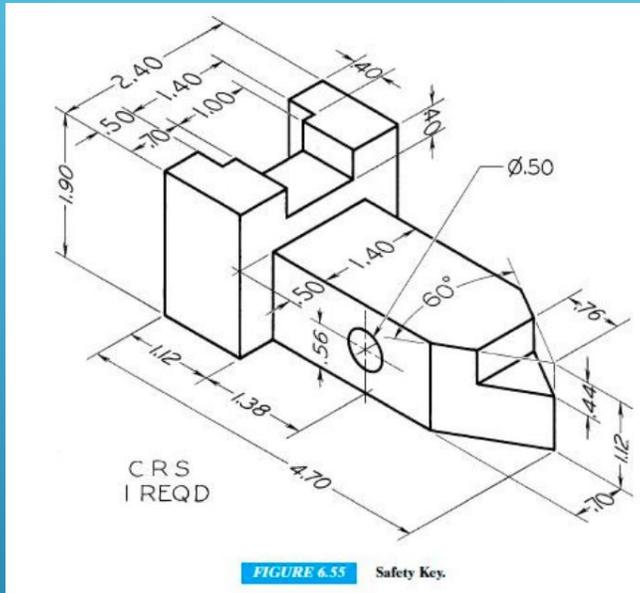
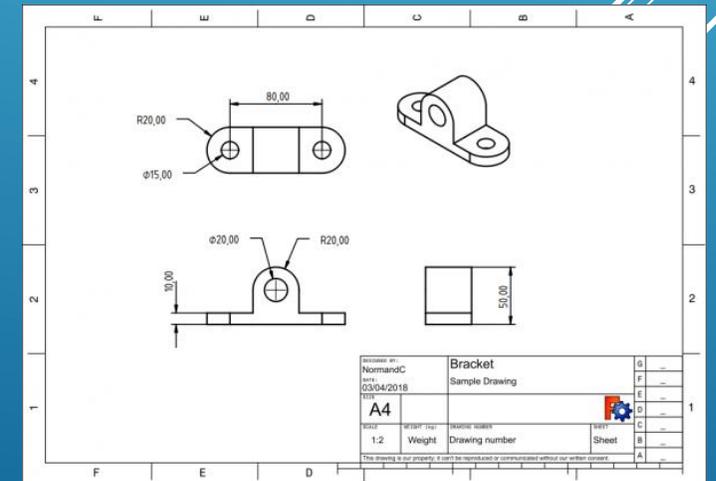
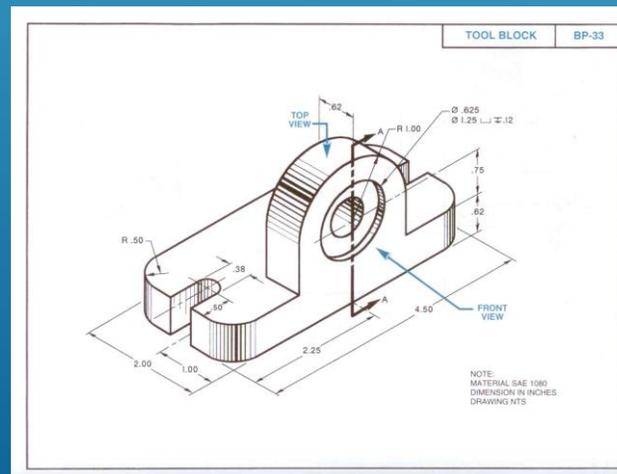
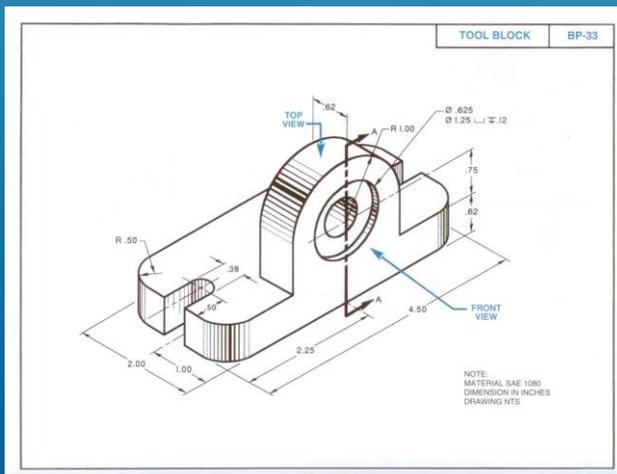
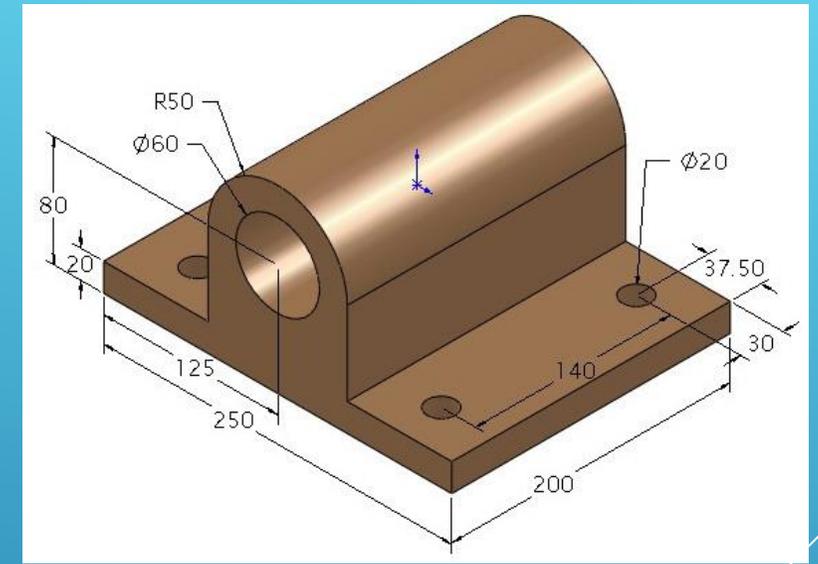
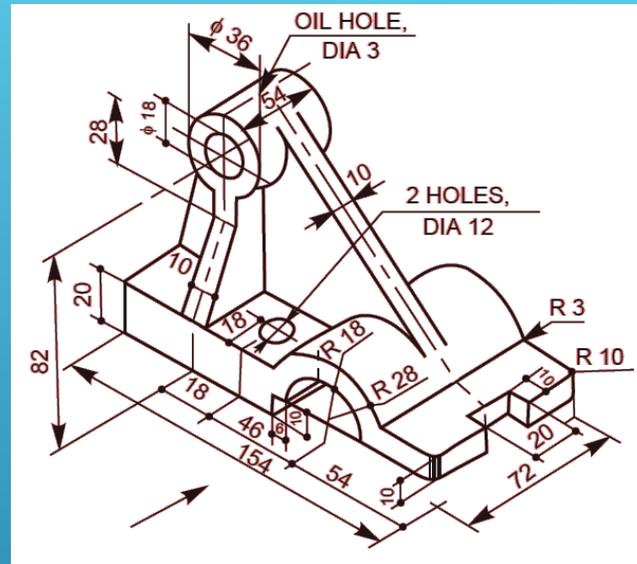
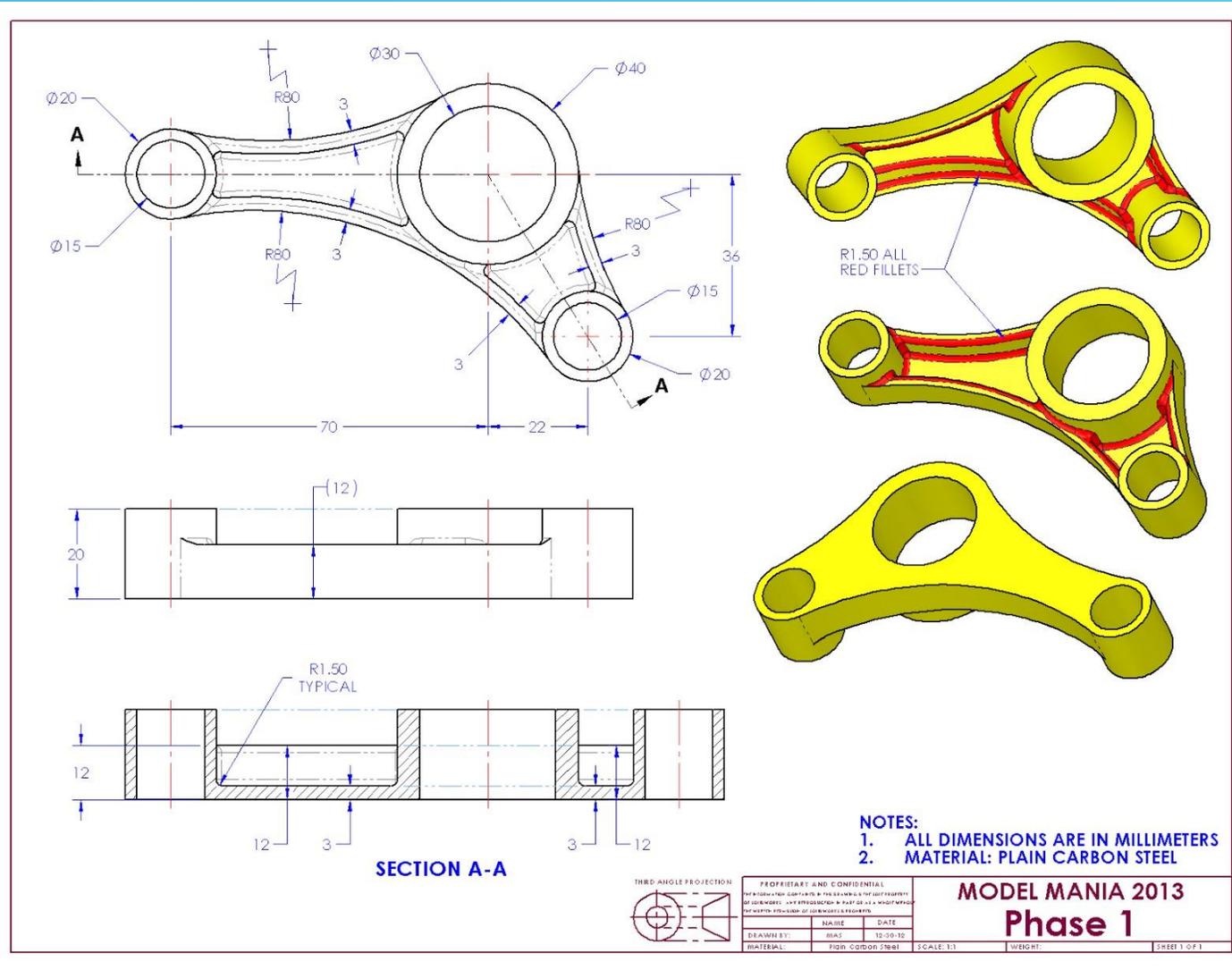


FIGURE 6.55 Safety Key.



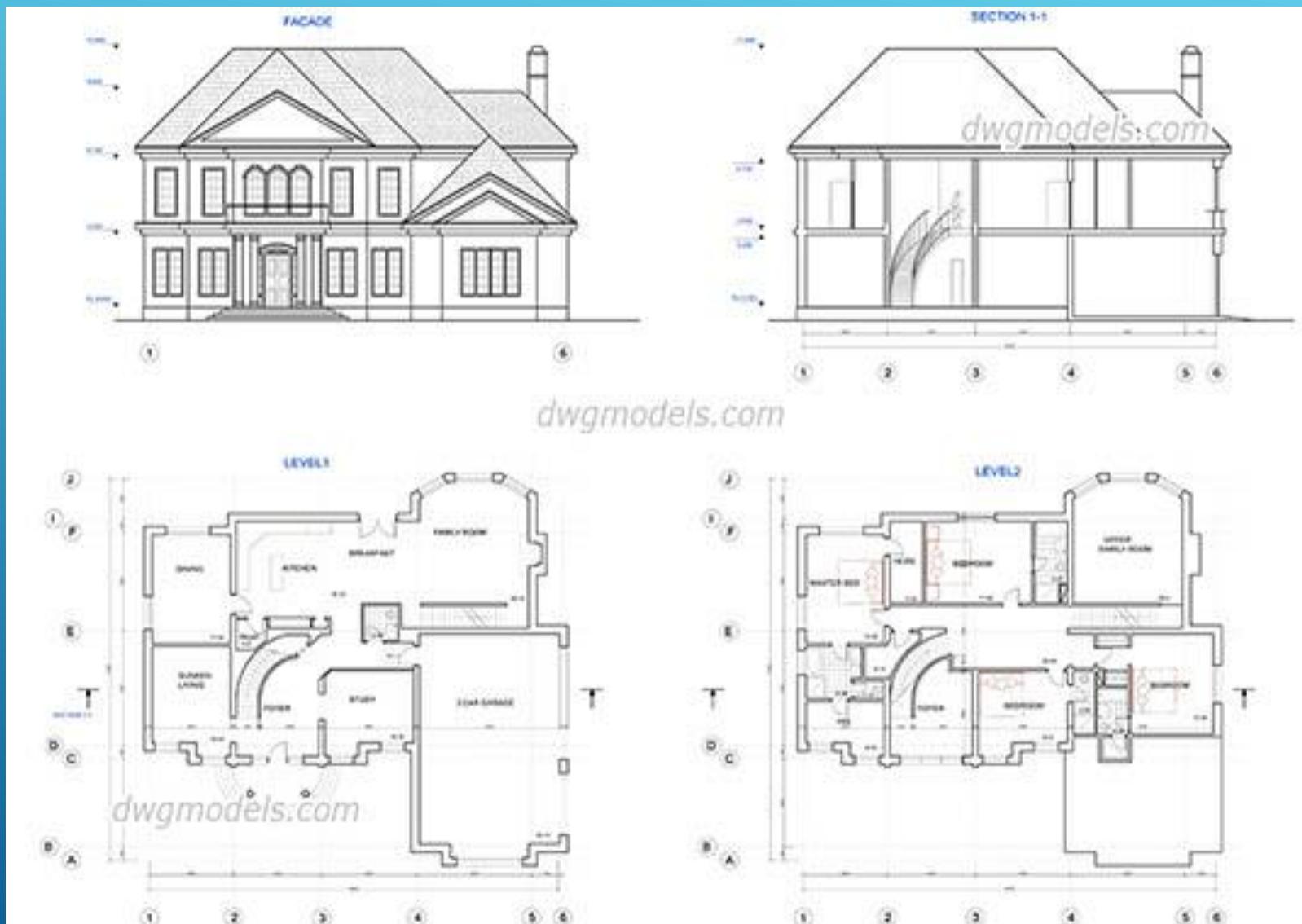


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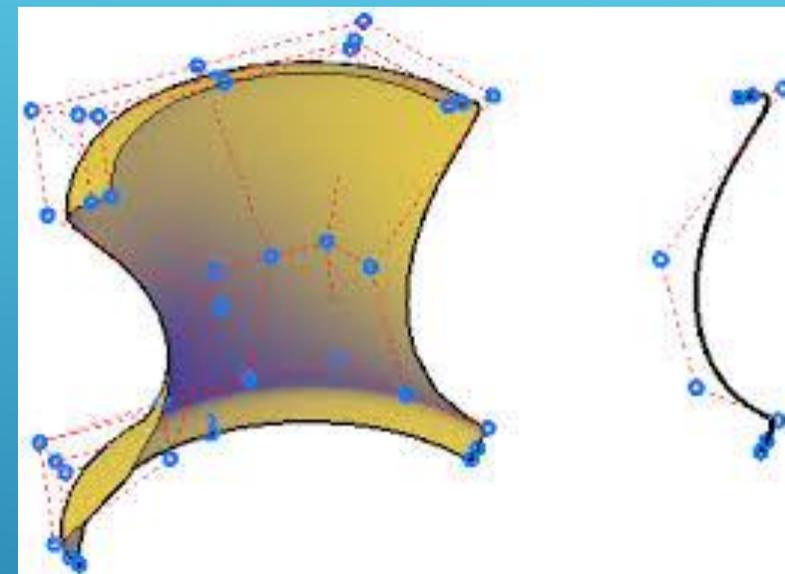
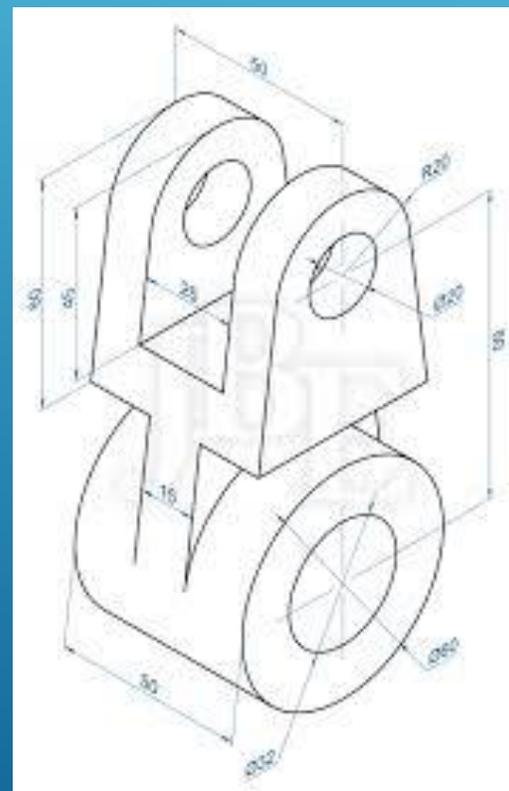
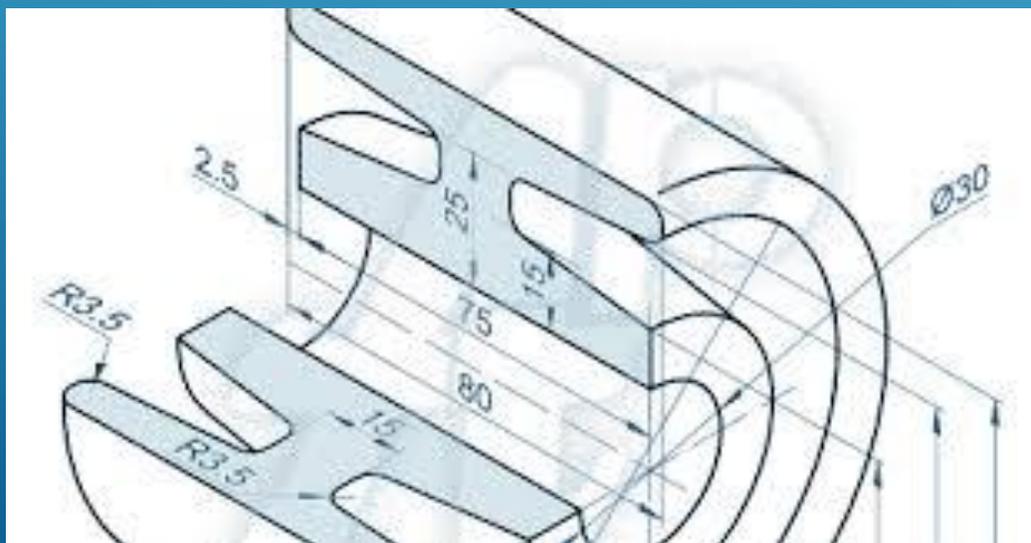
Figura 9.74_Mensula de contra cojinete - Drawing *

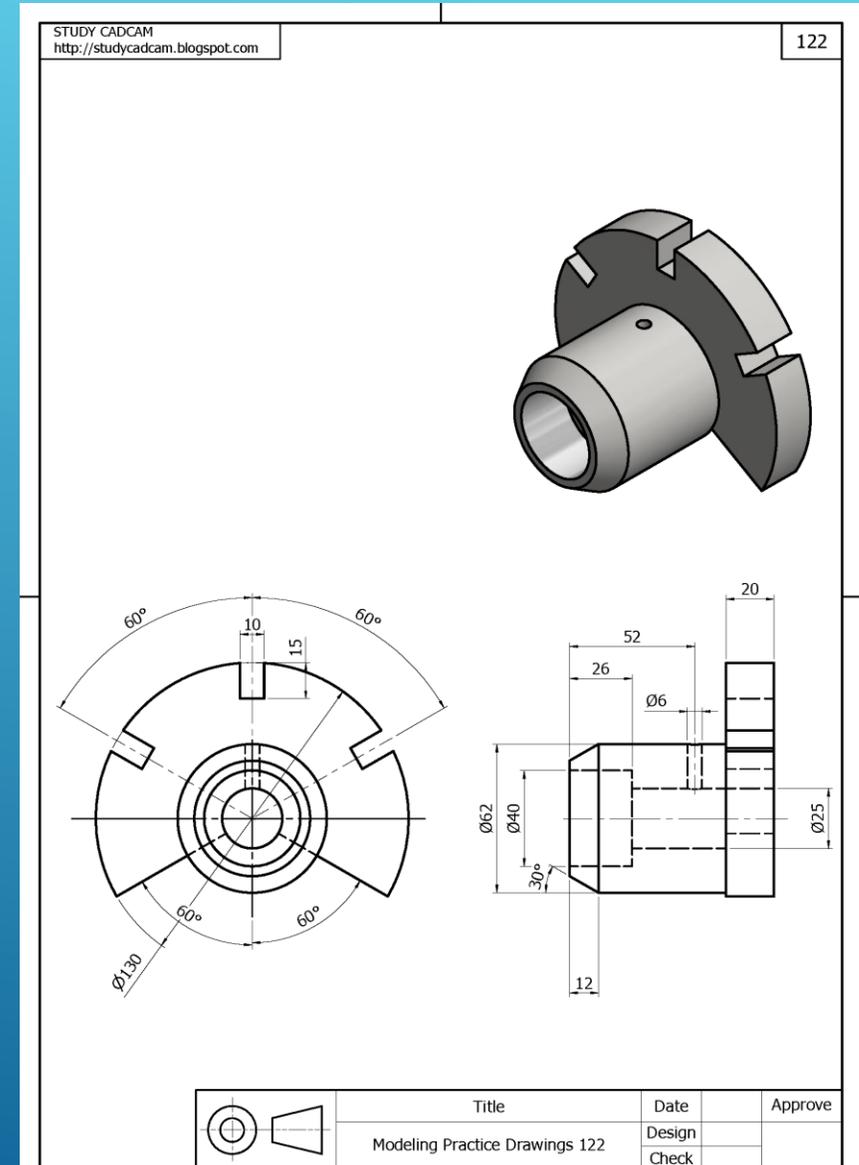
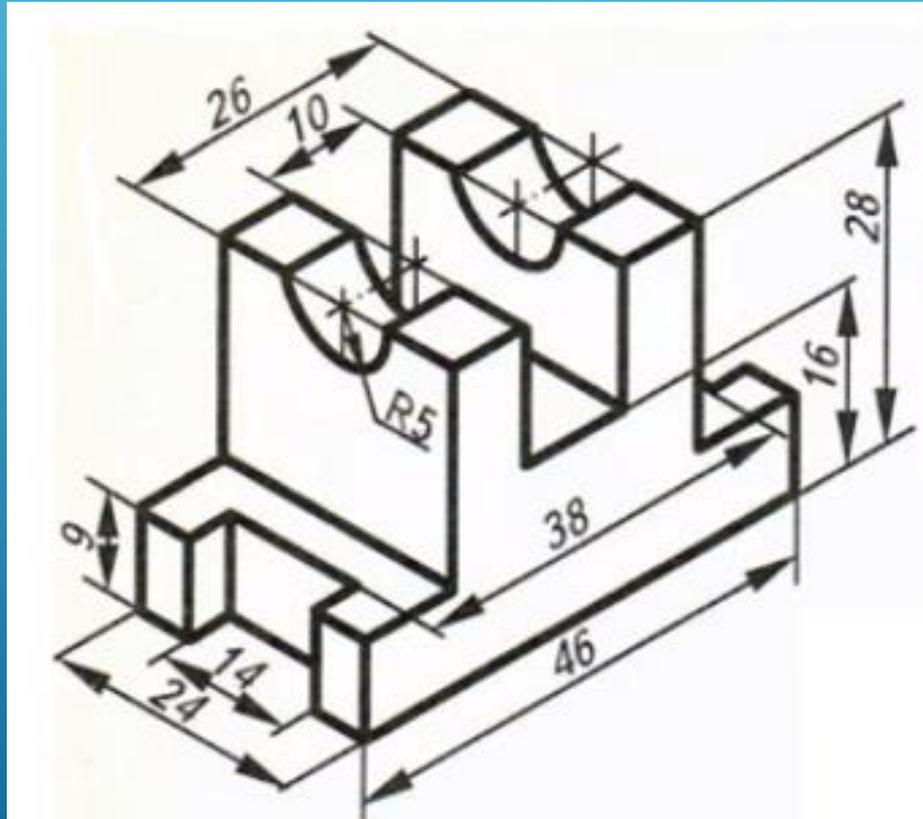
ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Figura 9.74_Mensula de contra cojinete		1

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

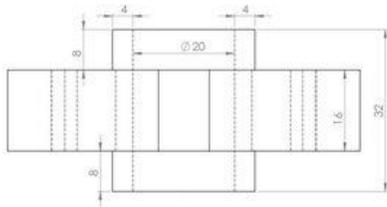
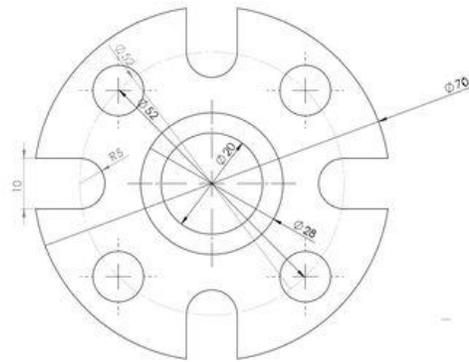
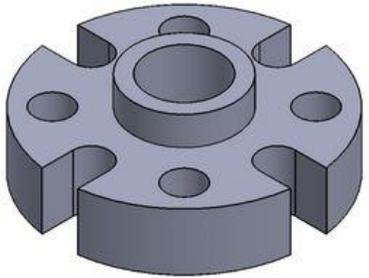


AULA 5 Desenho Técnico Assistido por Computador

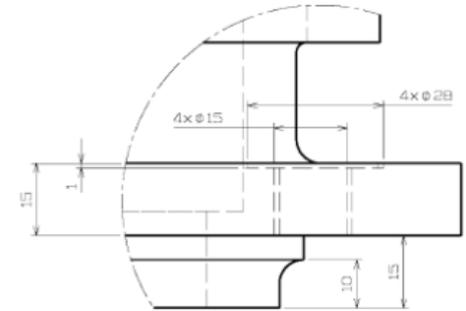
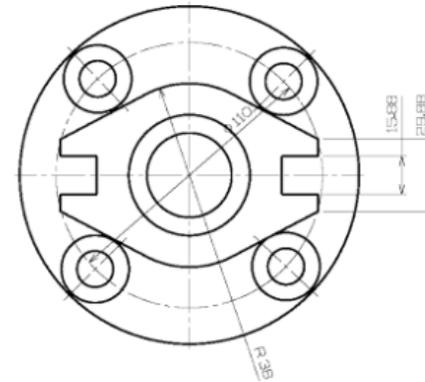




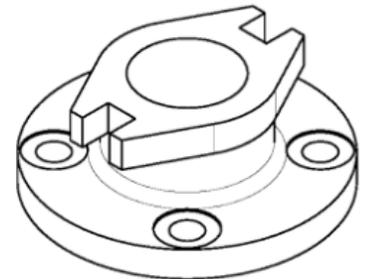
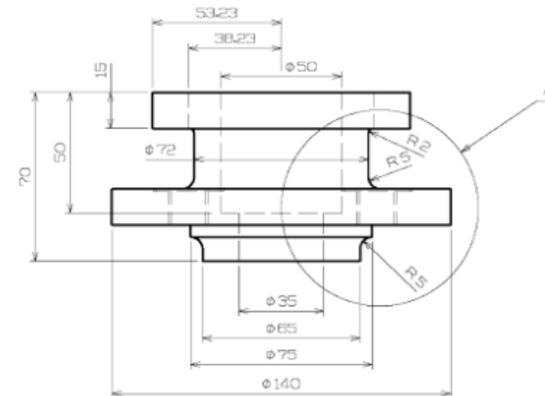
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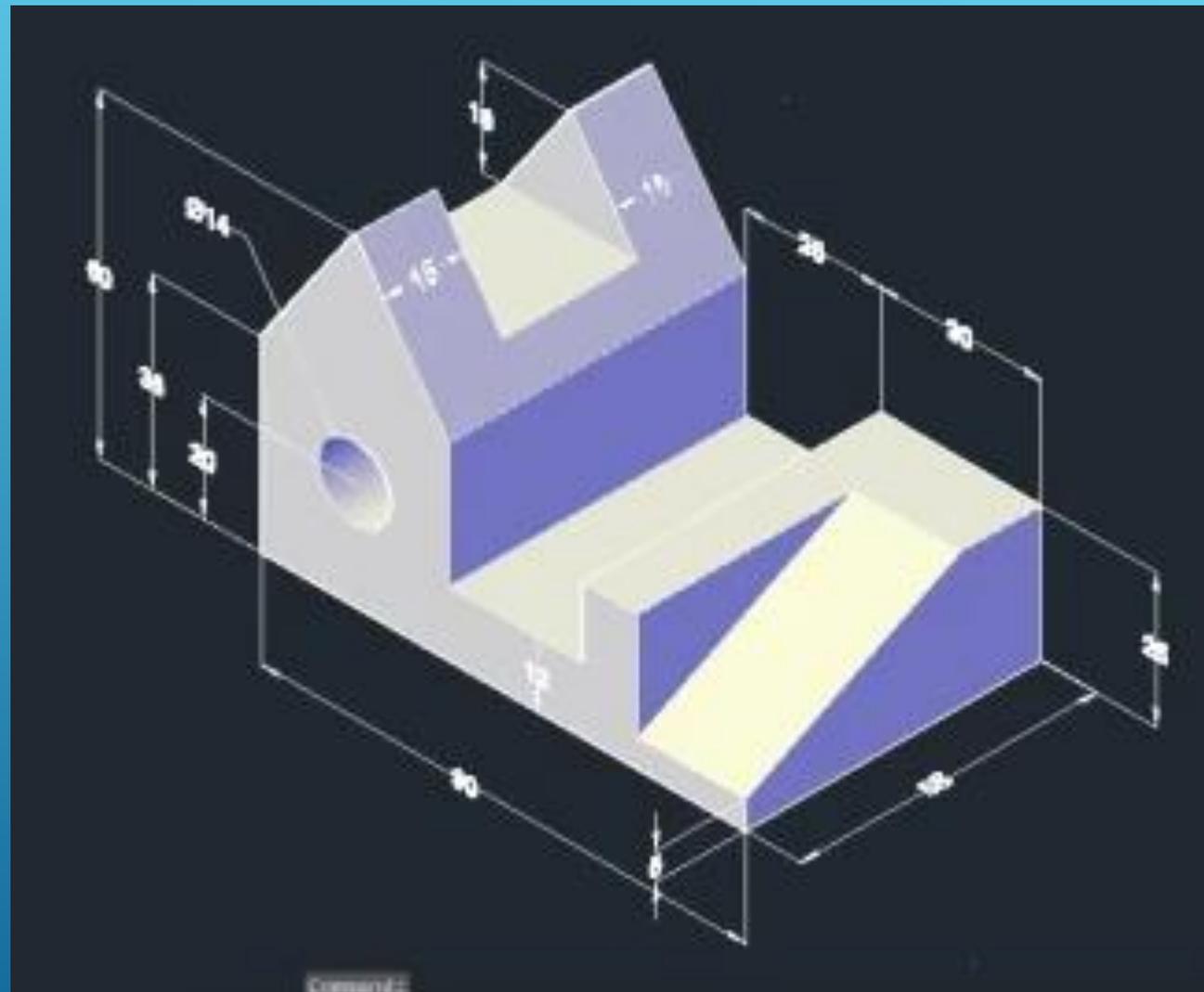
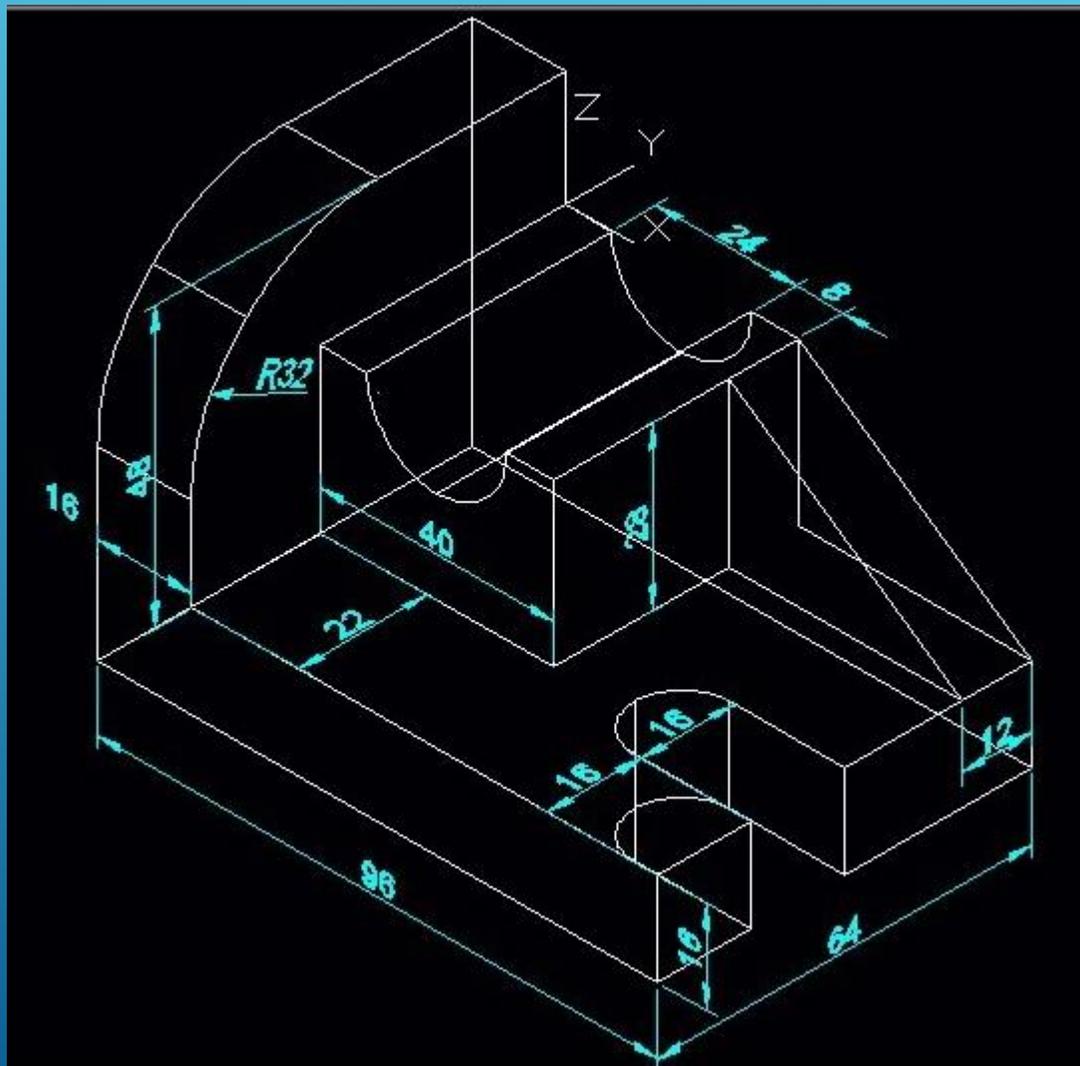


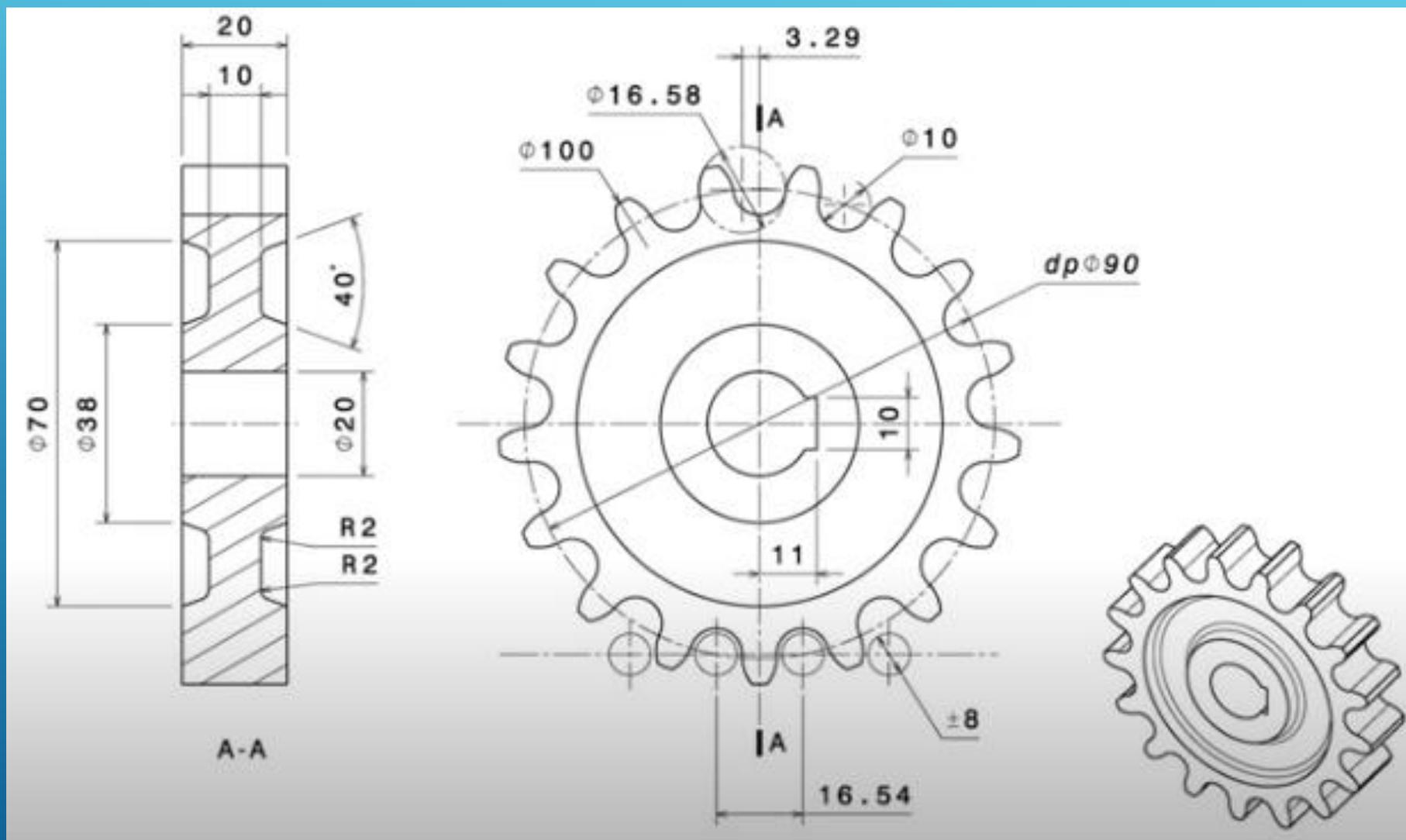
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DETAIL A
SCALE 2:1





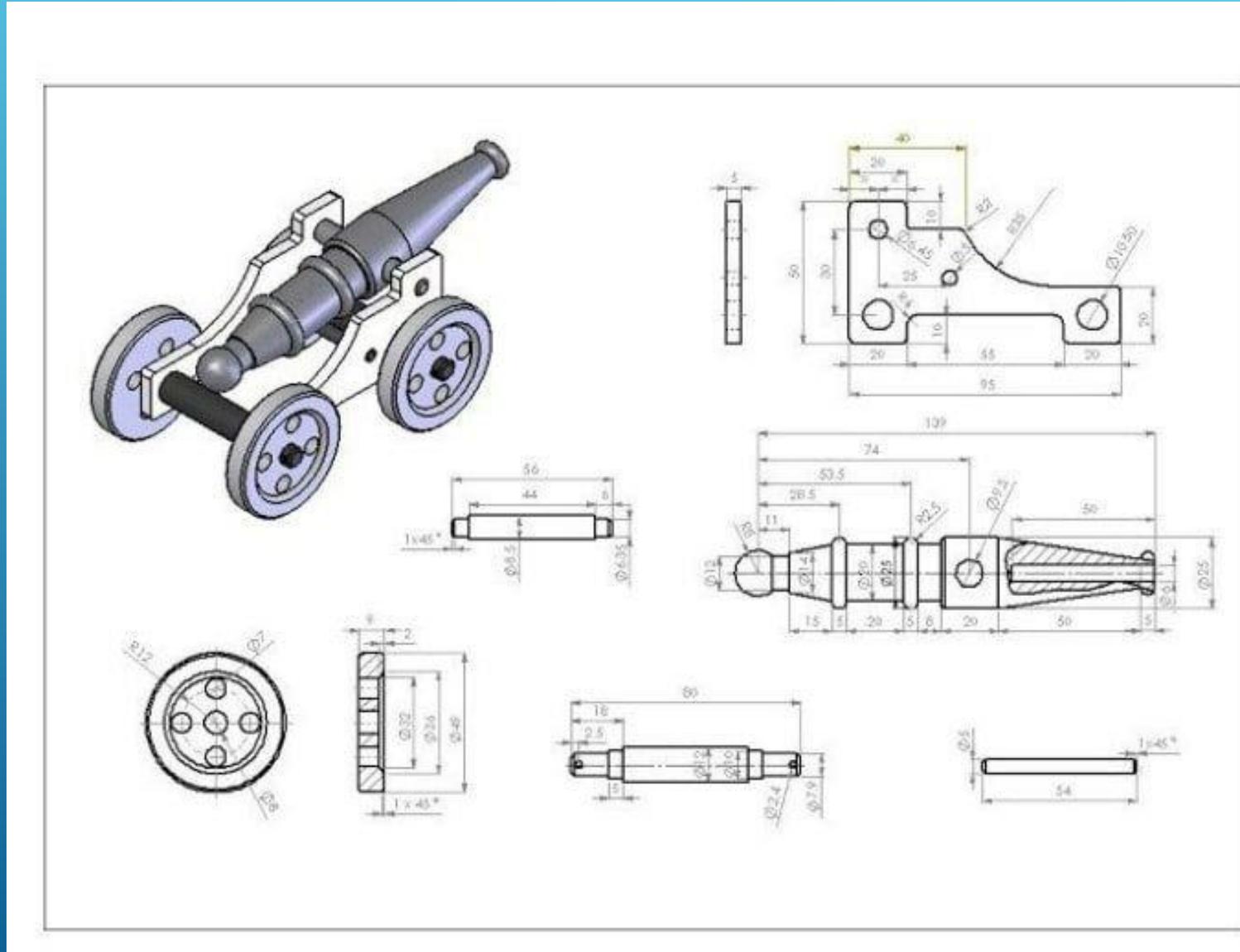


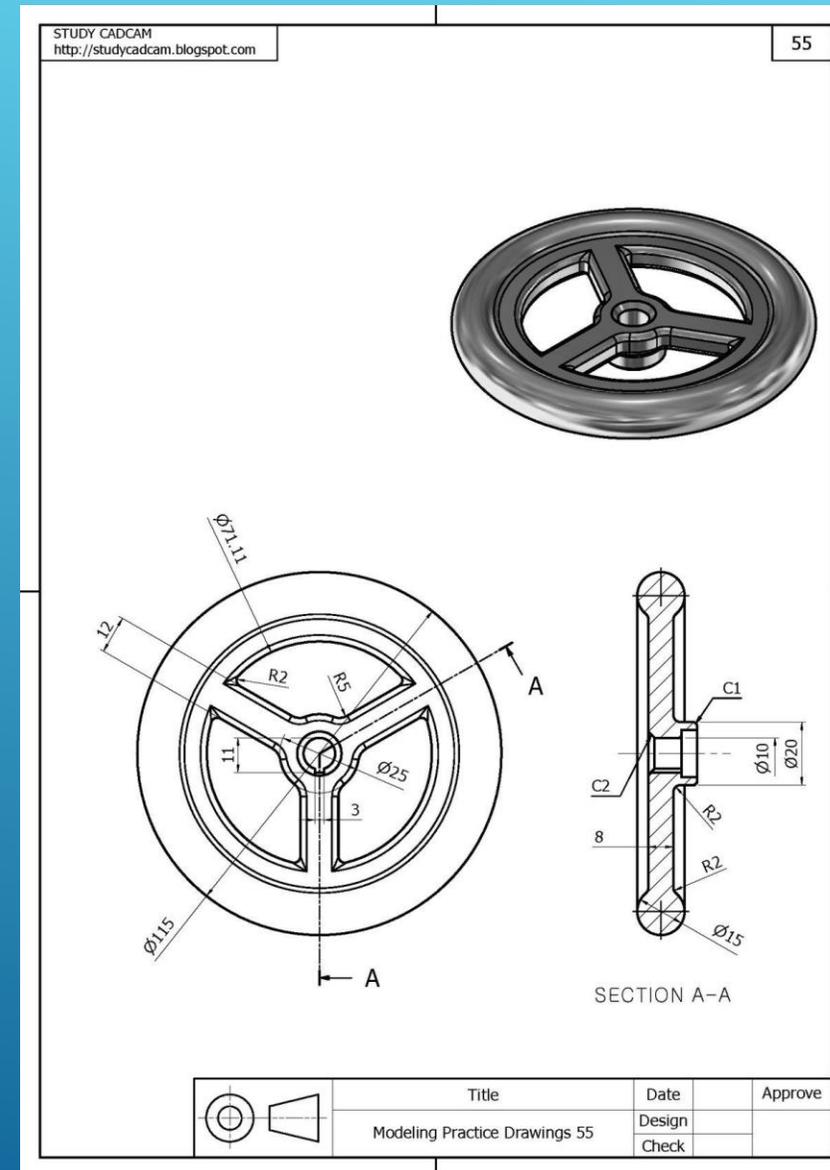
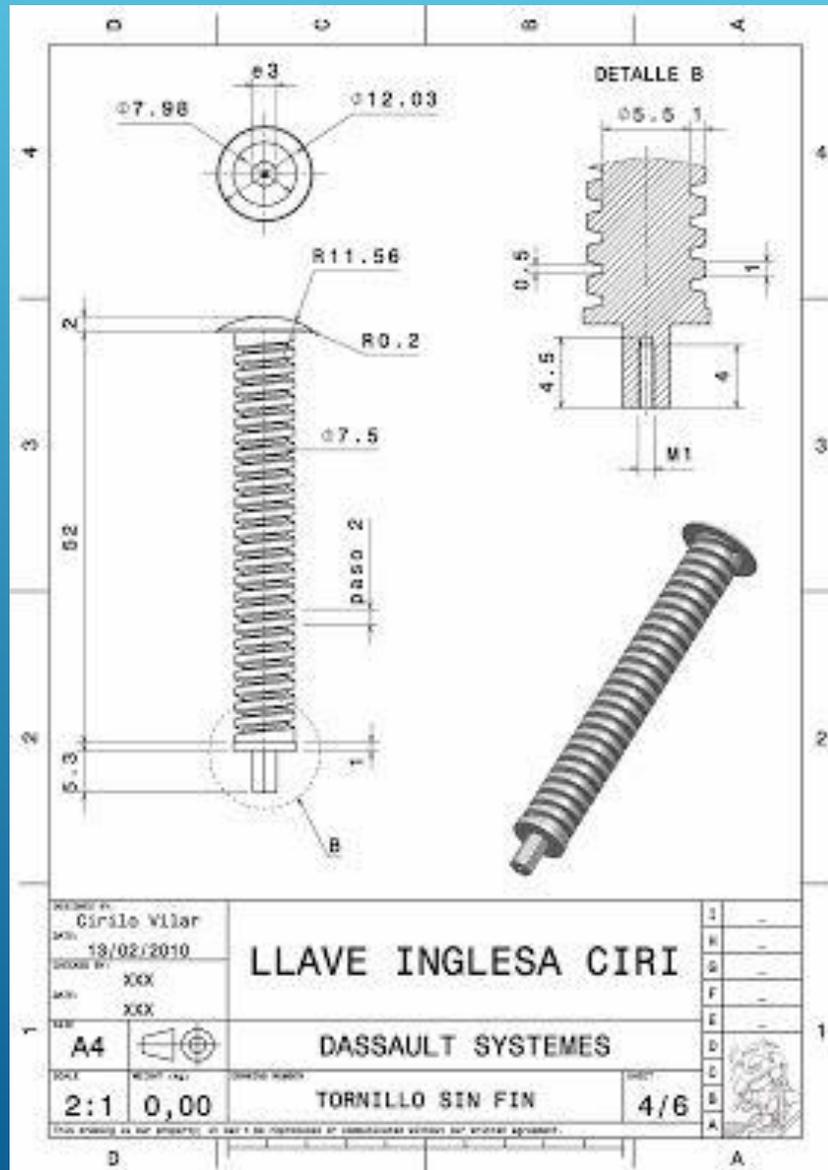
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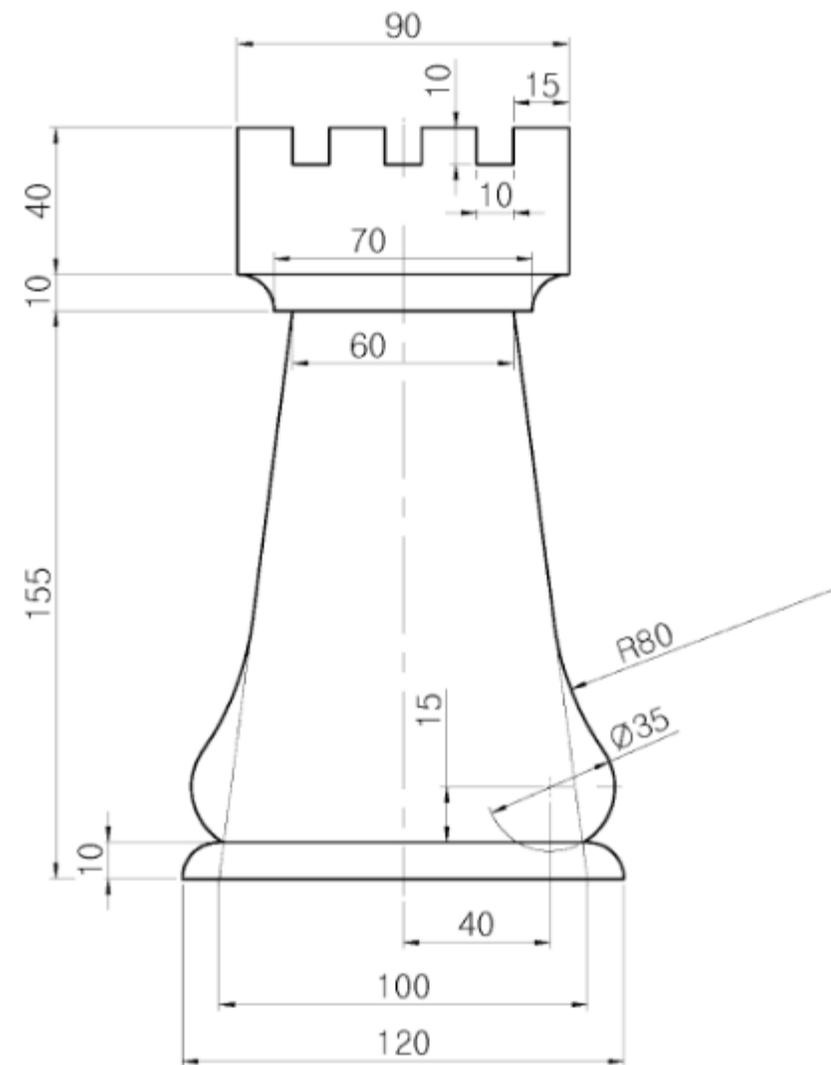
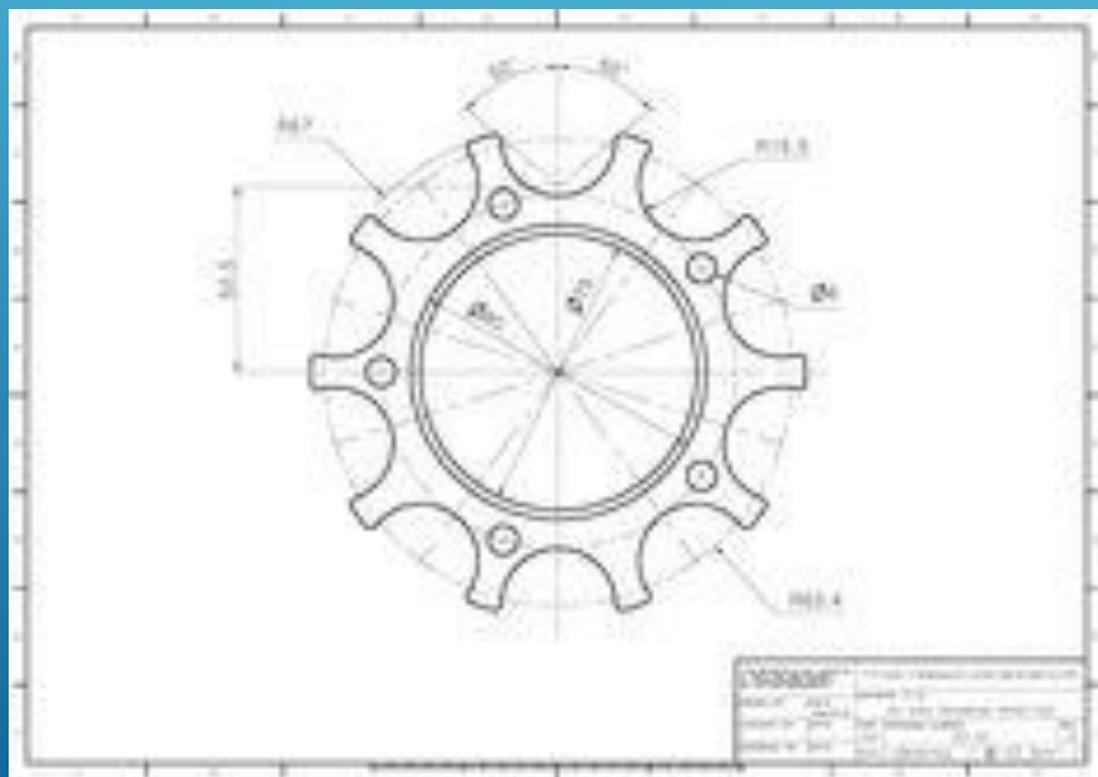


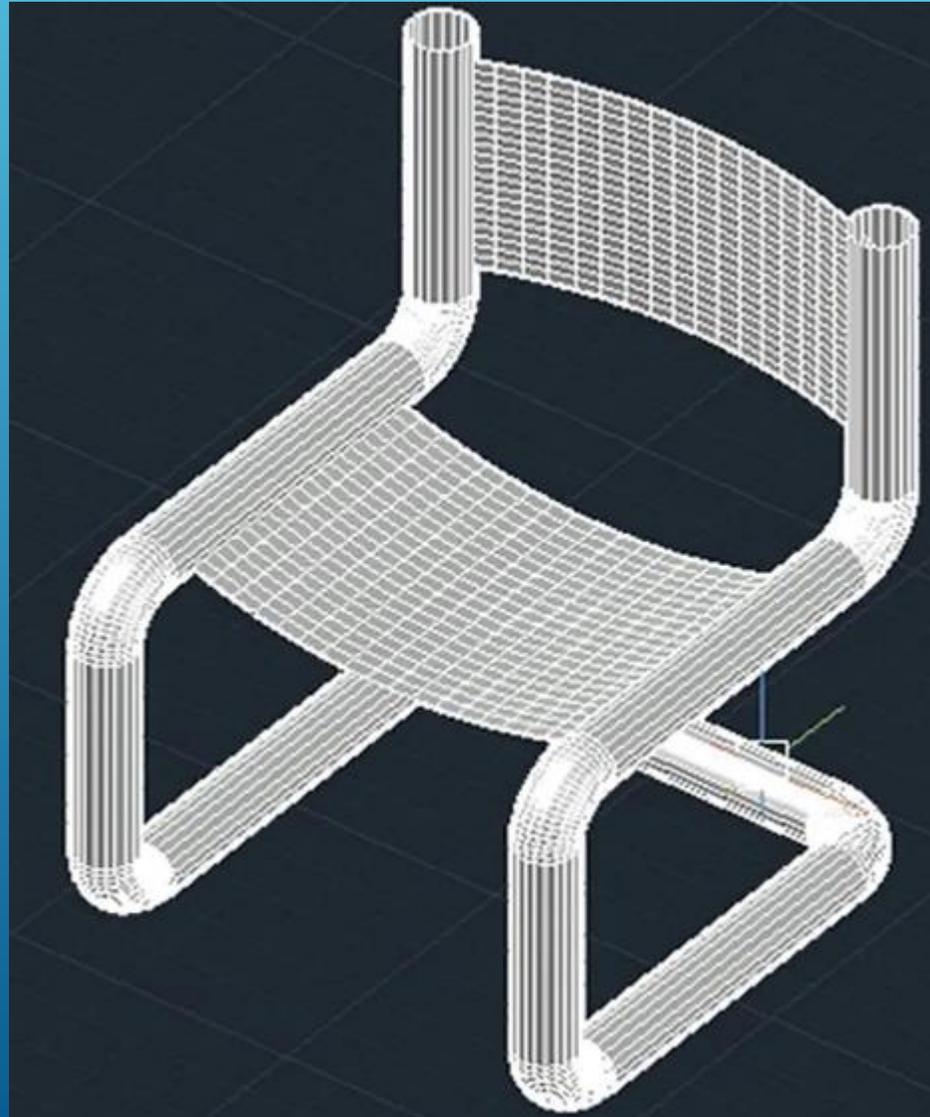
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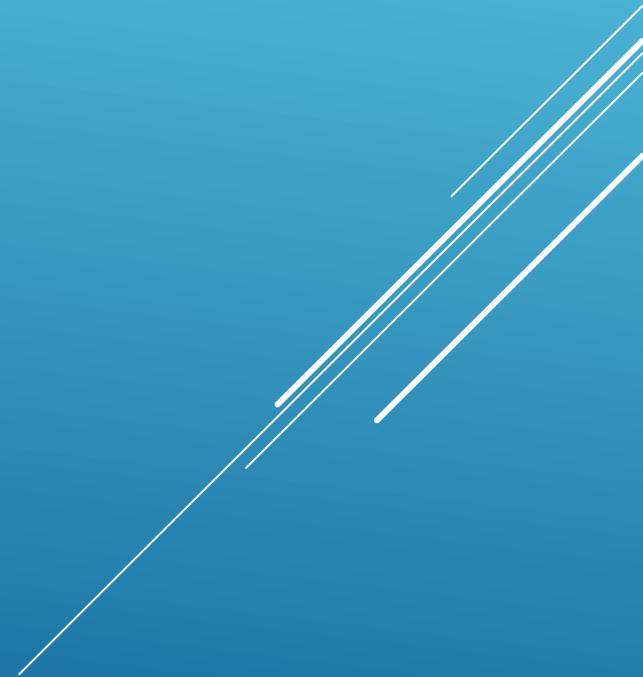


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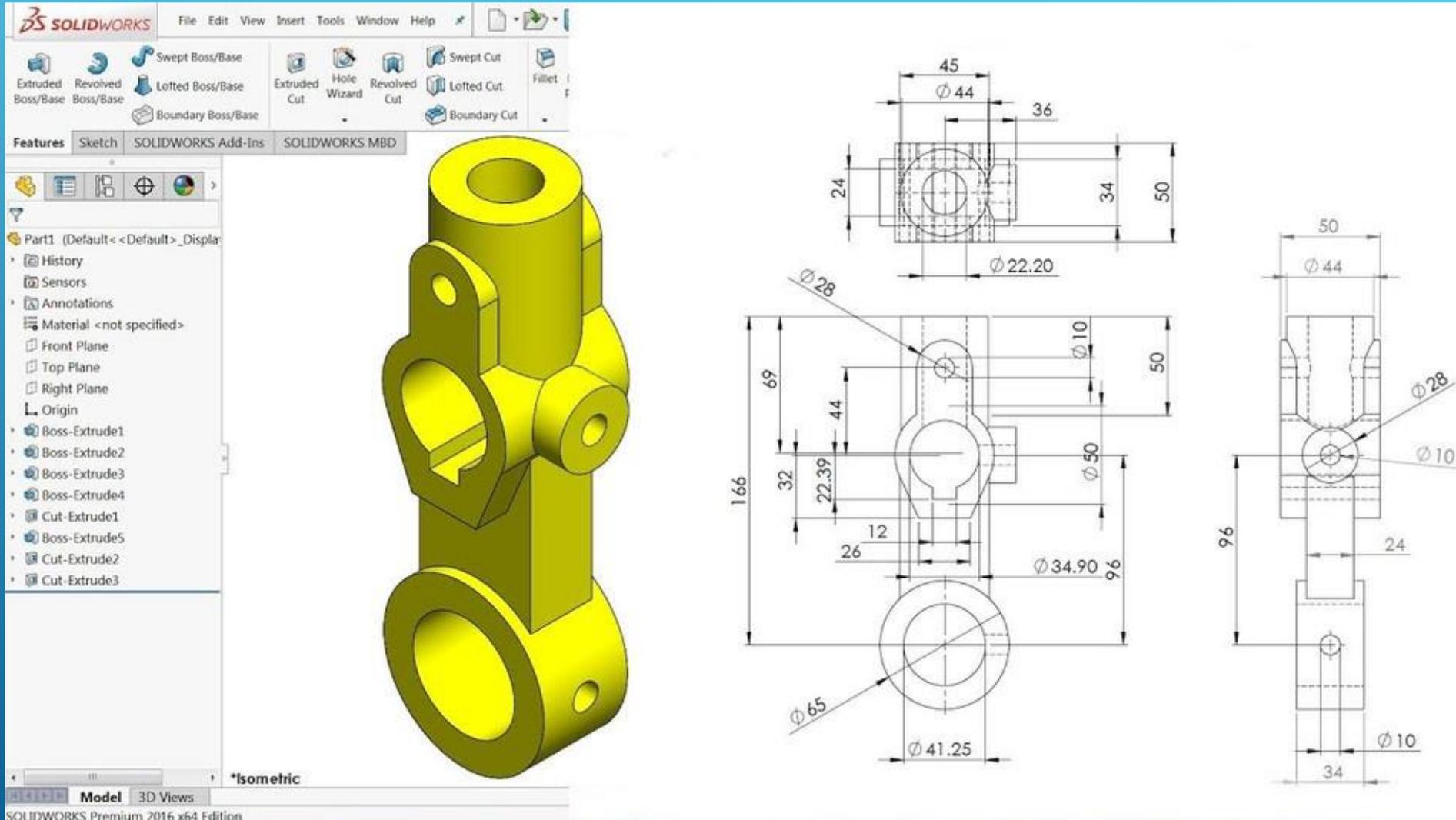


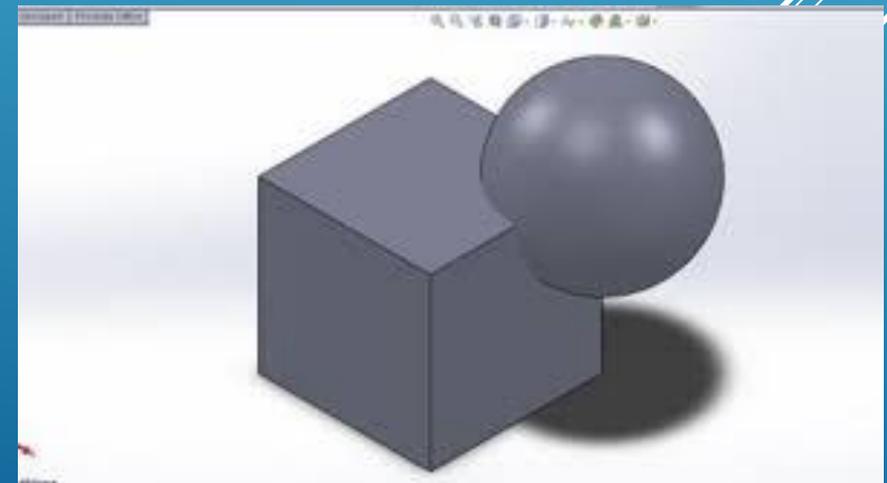
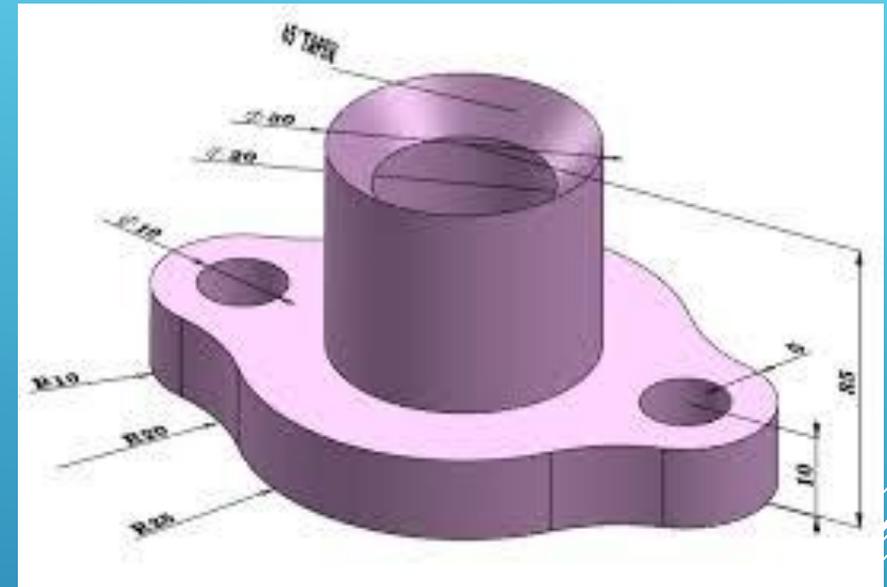
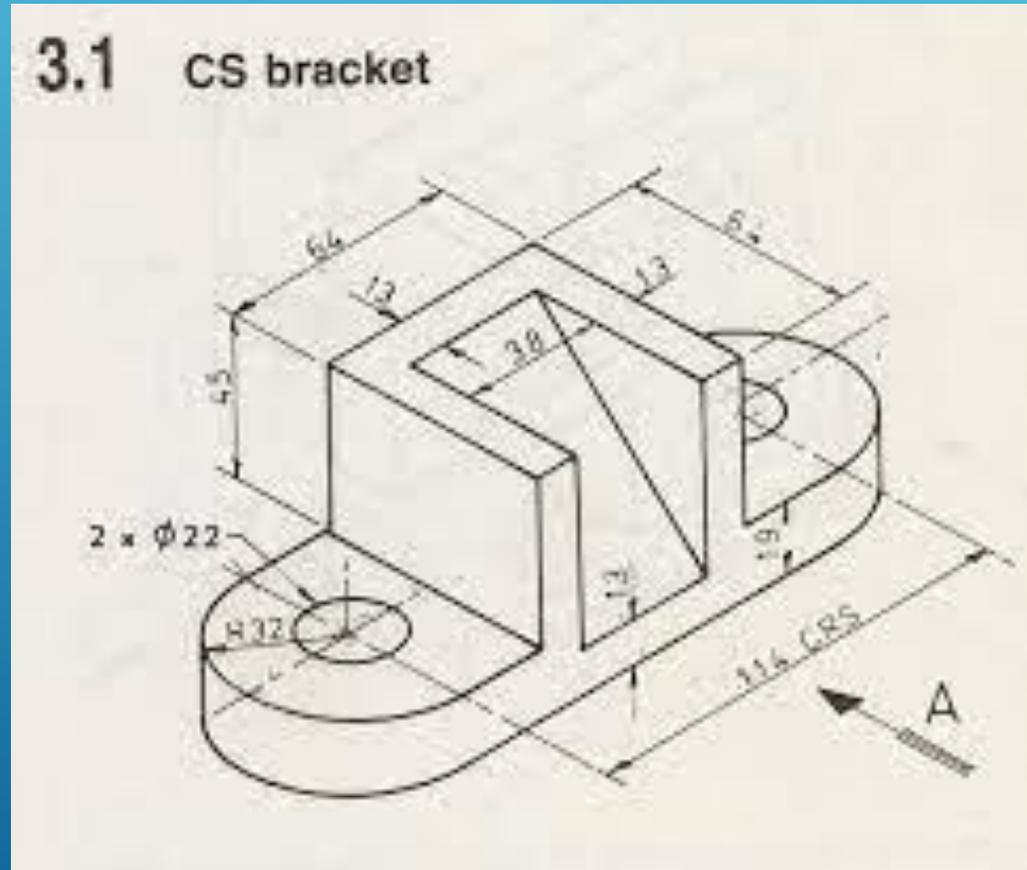
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ULisboa

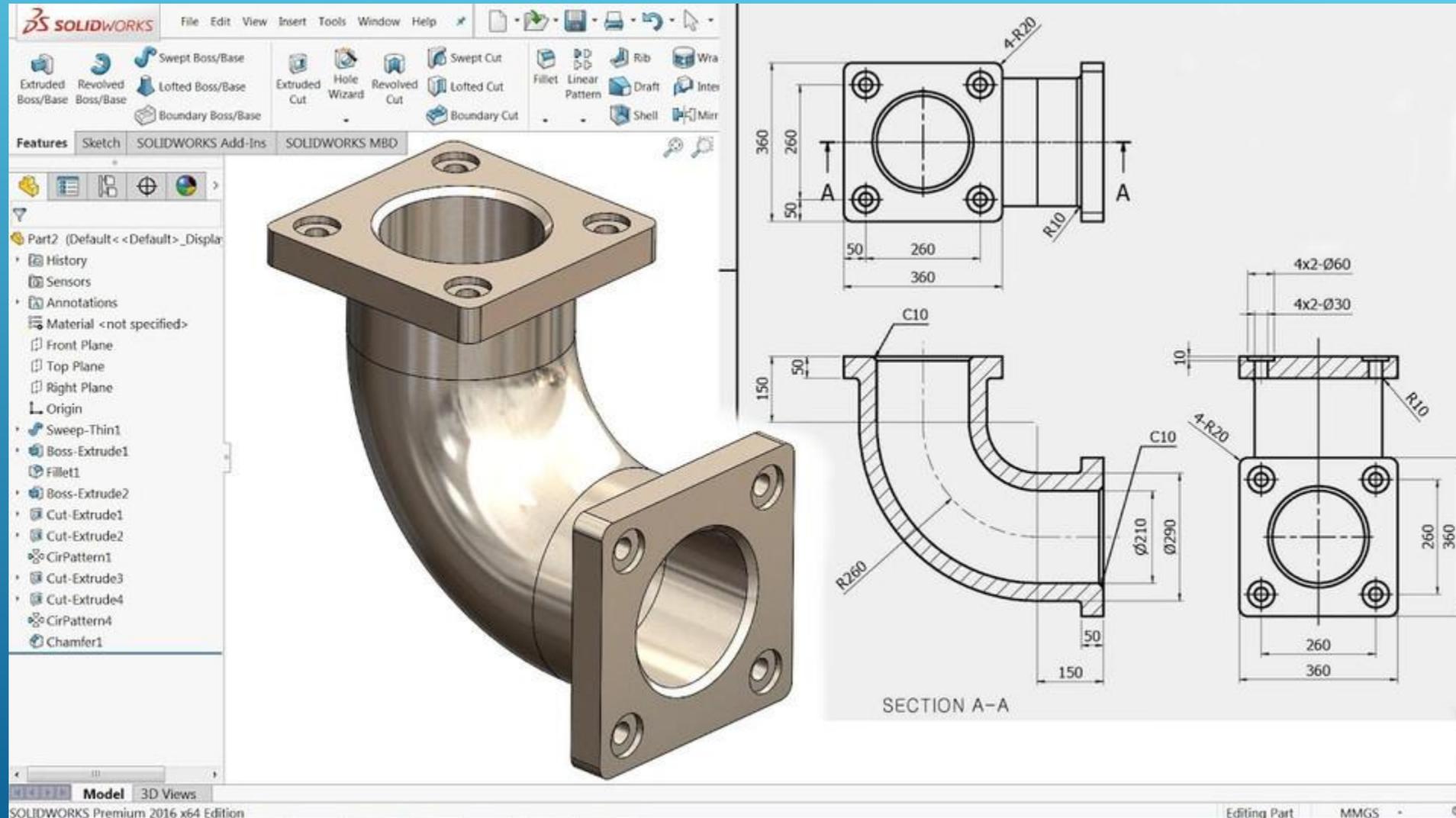
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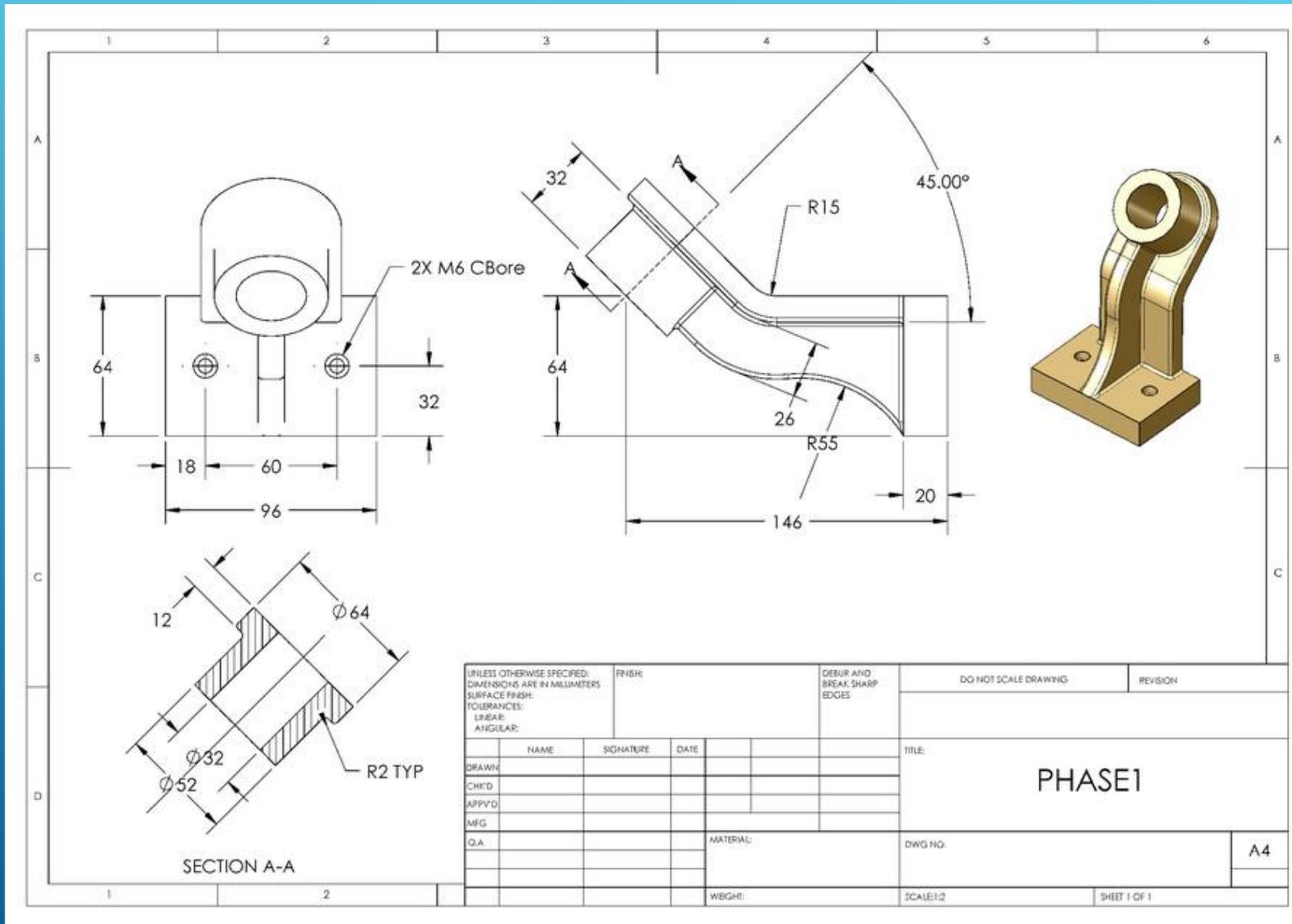


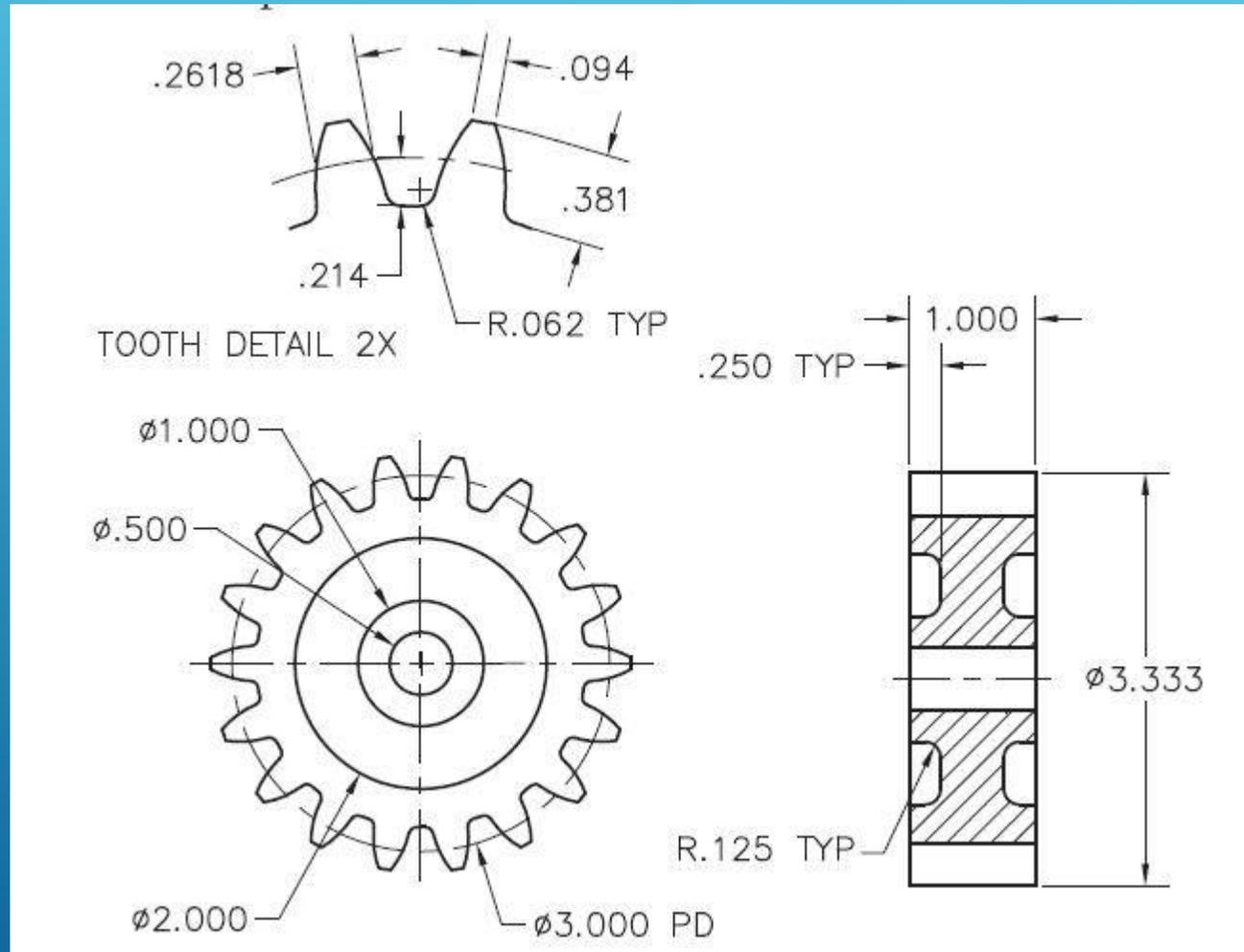
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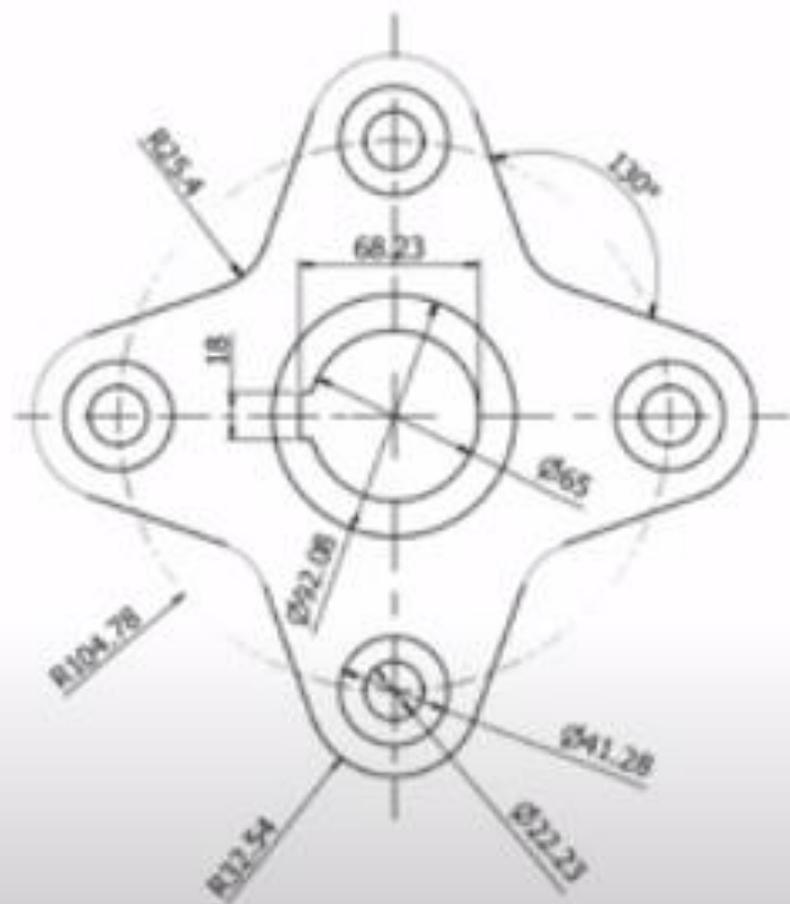
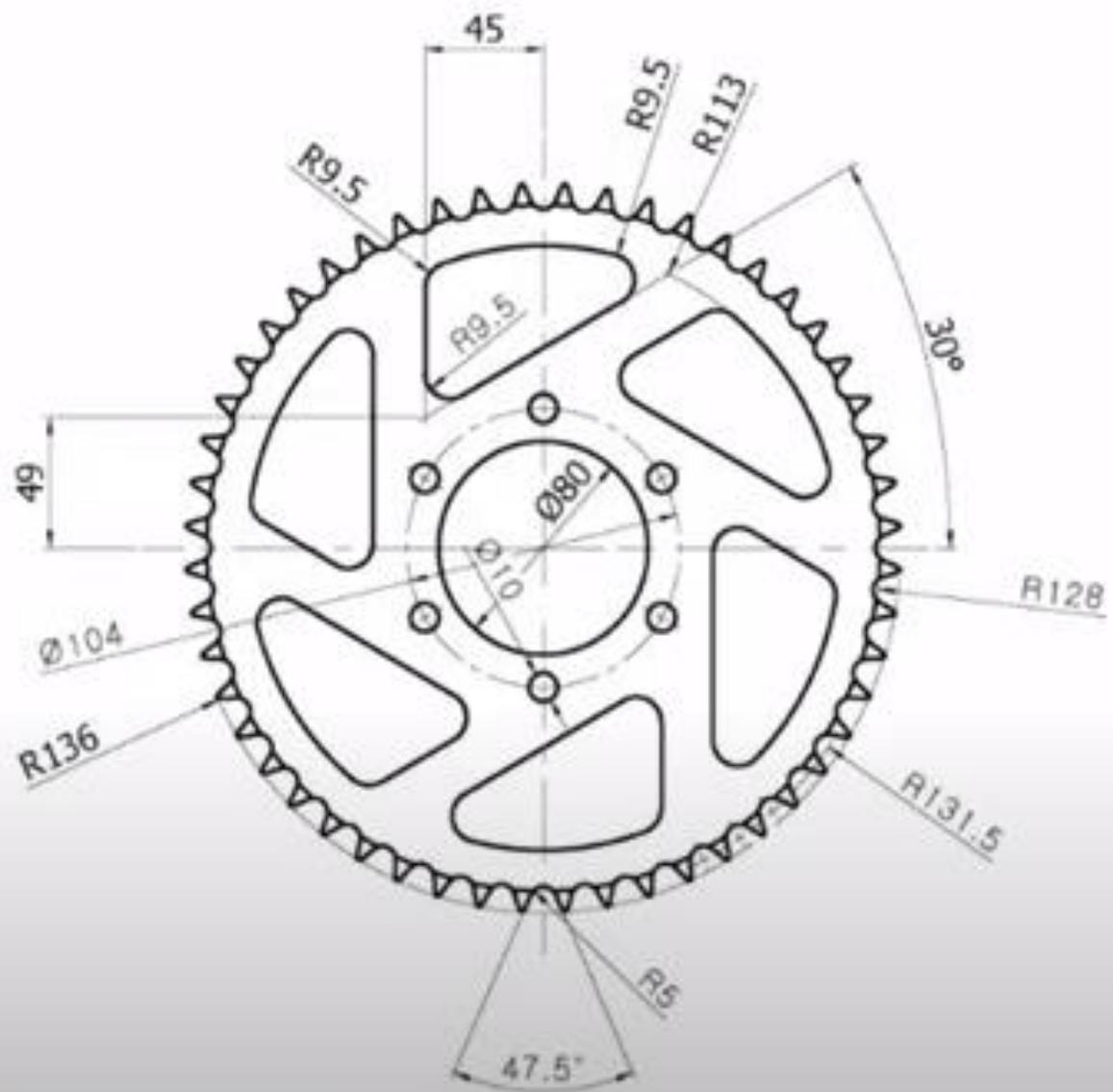


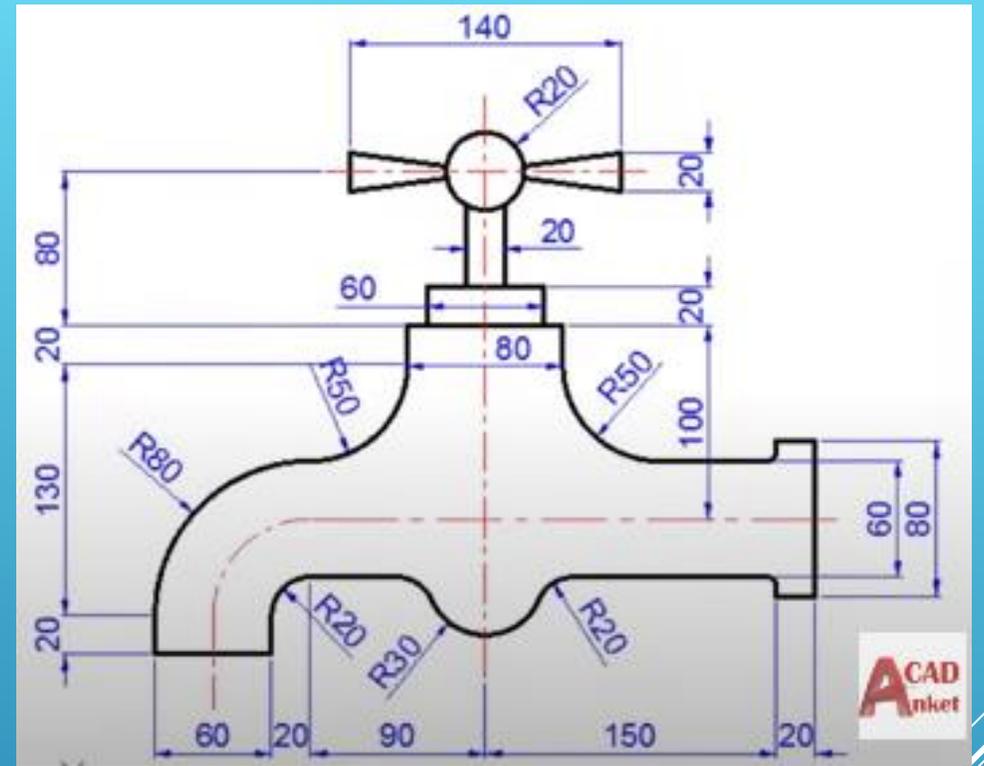
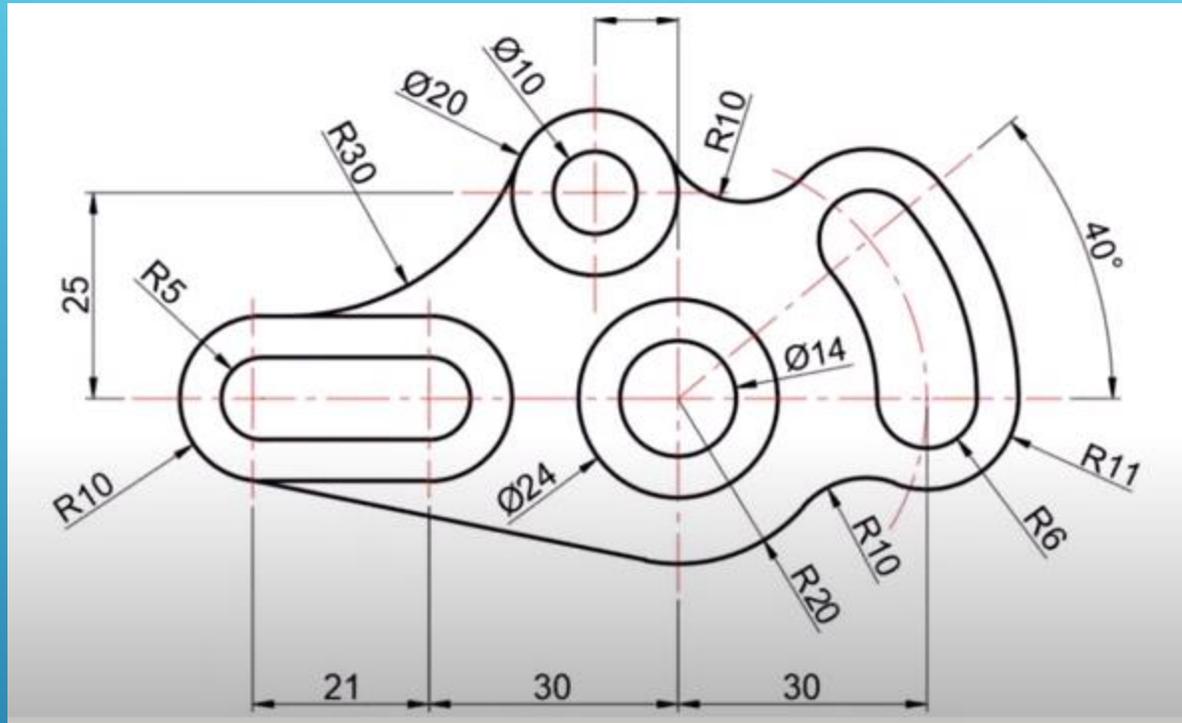


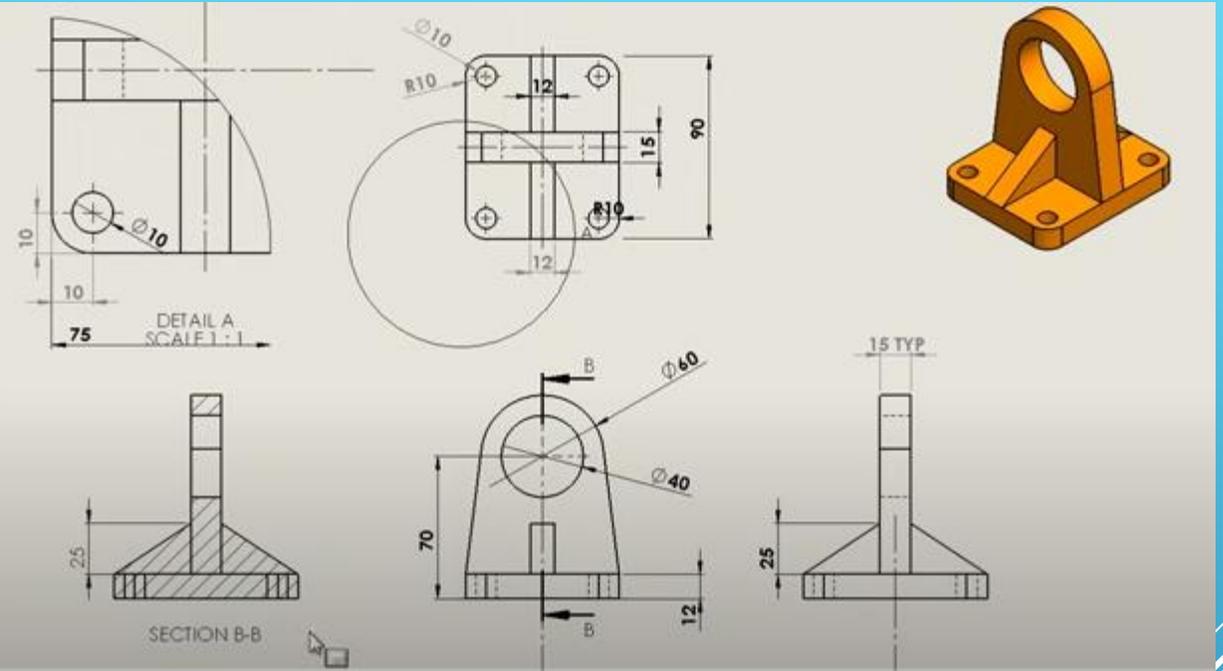
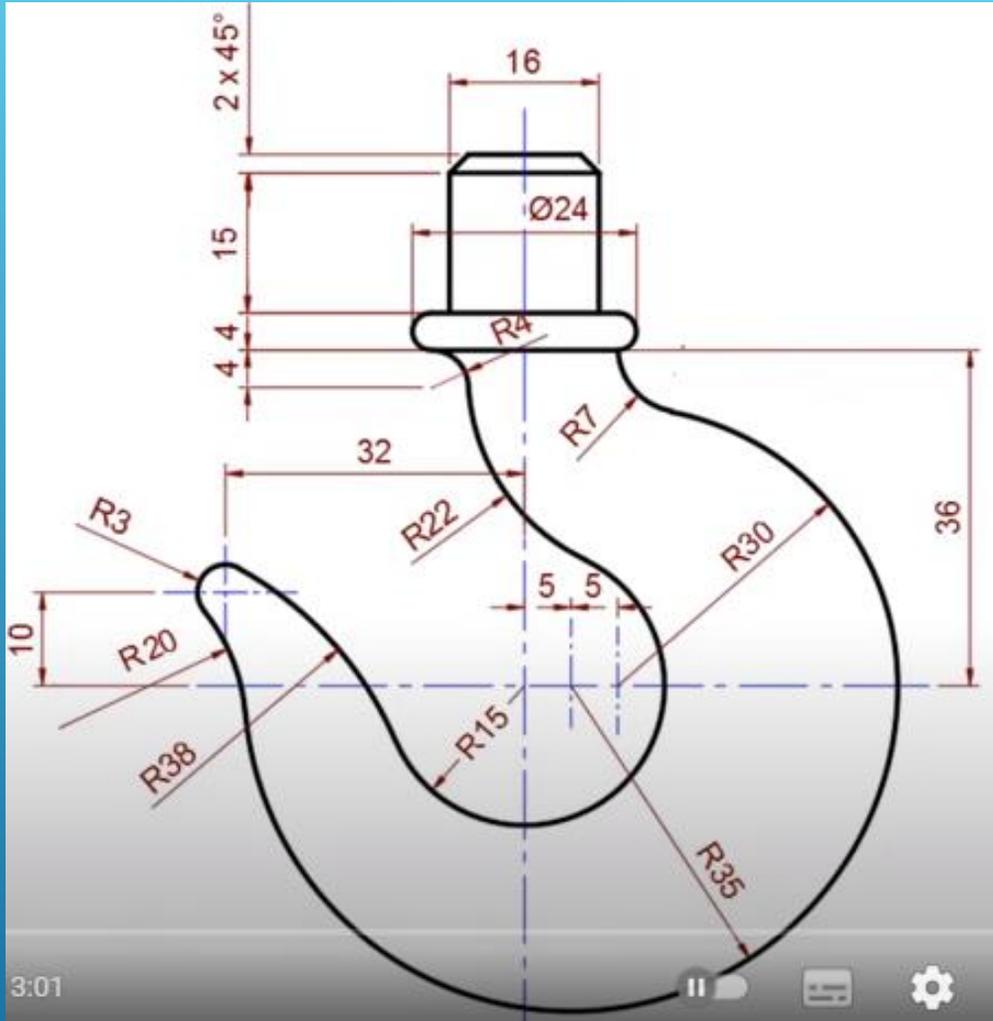












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