AULA 5 Desenho Técnico Assistido por Computador

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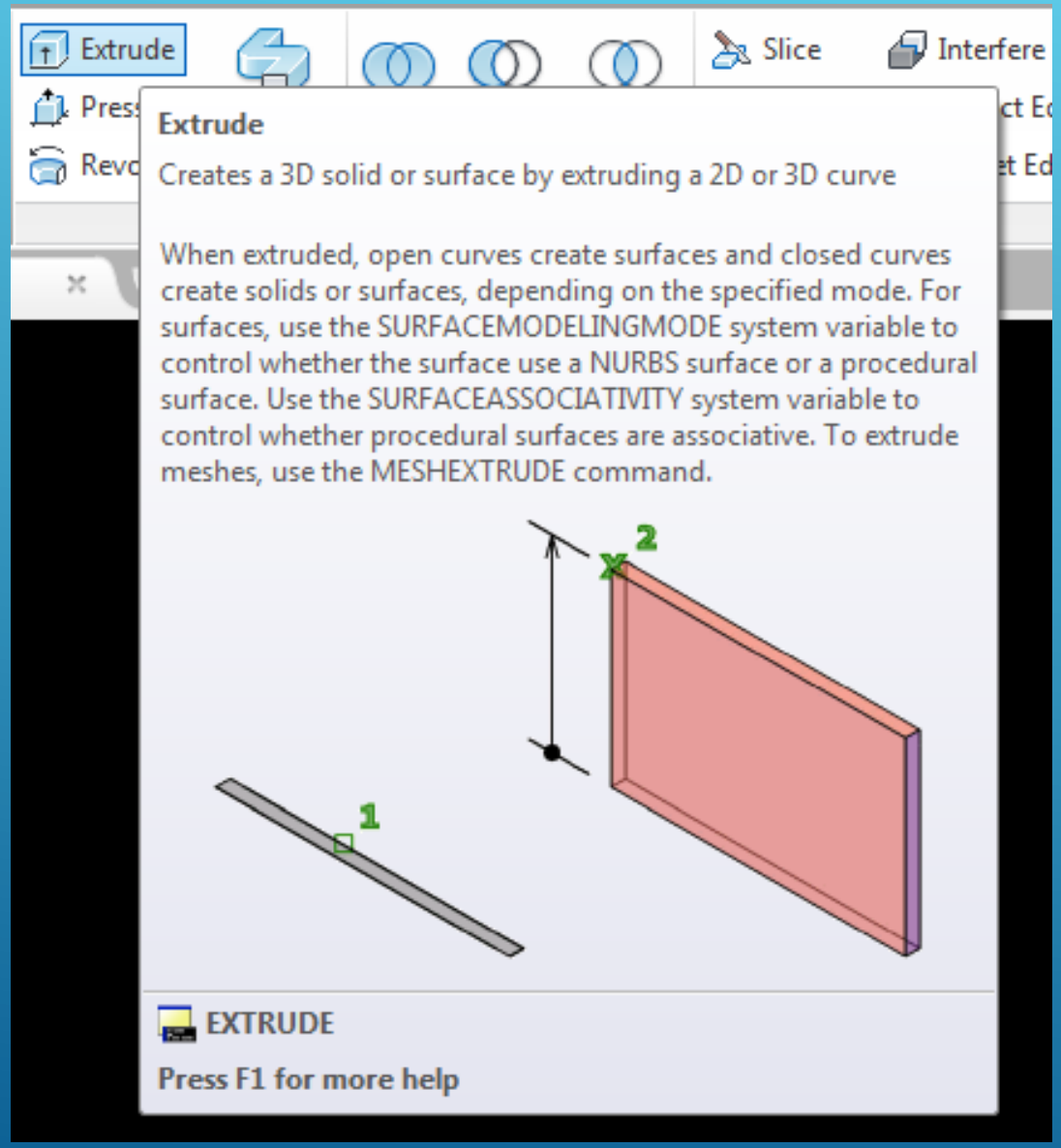

AULA 6 Desenho Técnico Assistido por Computador

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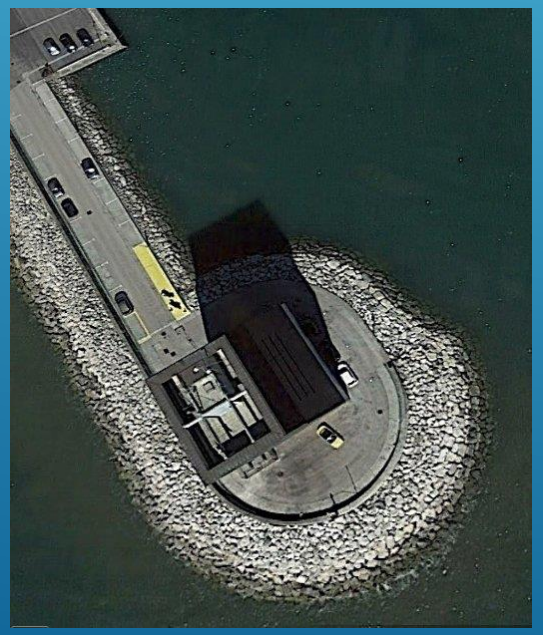
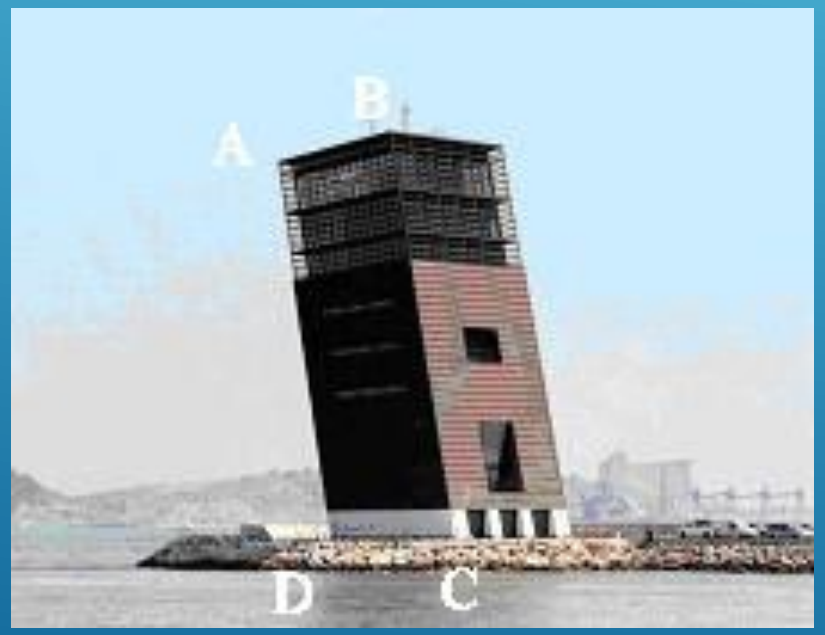
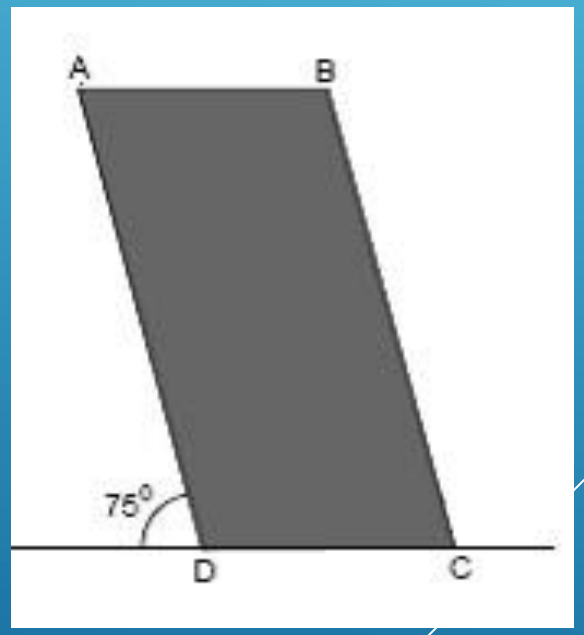
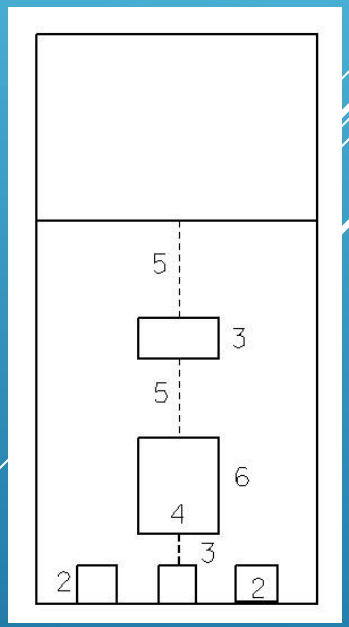
A Torre de Controlo de Tráfego Marítimo do Tejo, em Algés, tem a forma de um prisma rectangular oblíquo, com 40 metros de altura, base com 13 metros por 19 metros e uma inclinação de 75º em relação à horizontal (o rumo da direcção do lado menor é igual a 59 º) .









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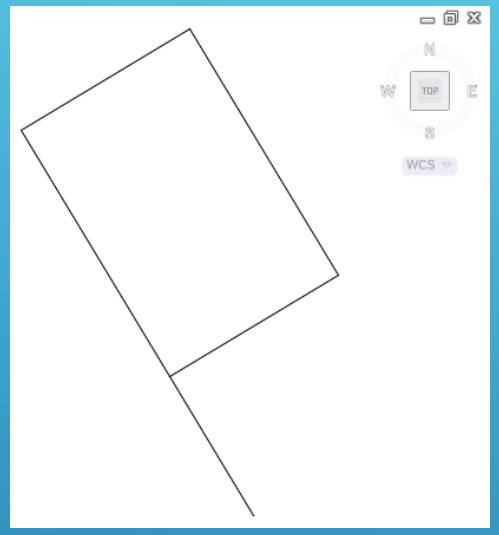

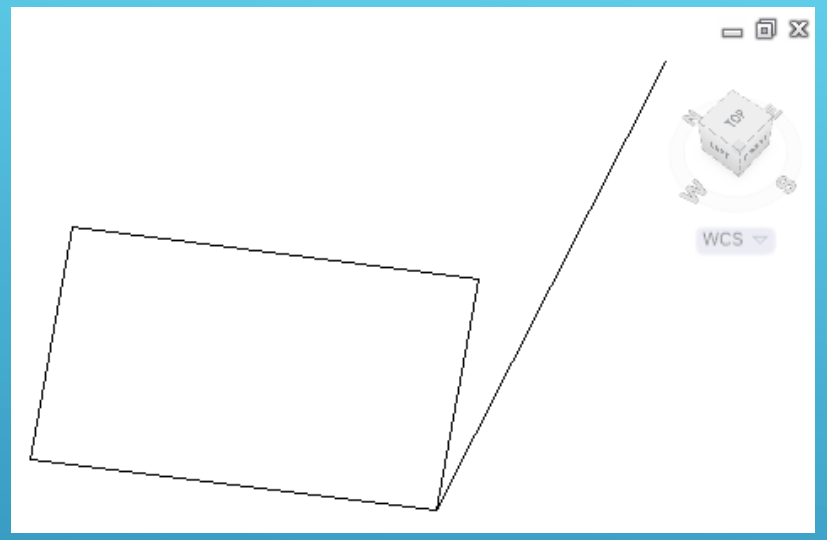

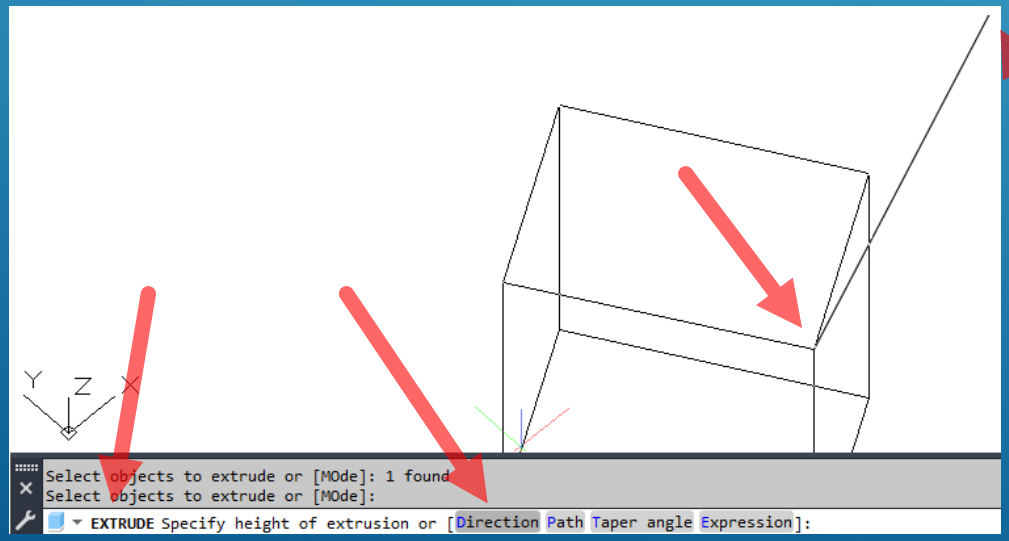

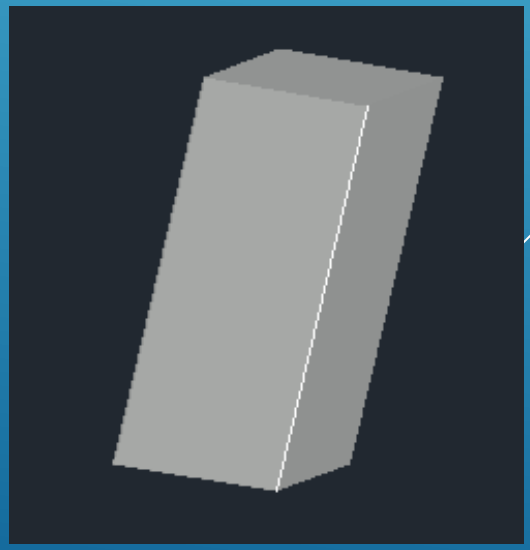

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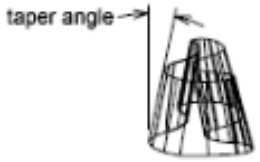
Entre o Cairo e Assuão, a margem ocidental do Nilo conserva mais de 60 pirâmides construídas entre 2700 e 1750 A.C.; a 10 km de Saqqara, o faraó Snefru mandou erguer uma enorme **pirâmide de base quadrangular com 189 m de lado**, **cuja altura deve ter atingido os 102 m**, conhecida actualmente como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

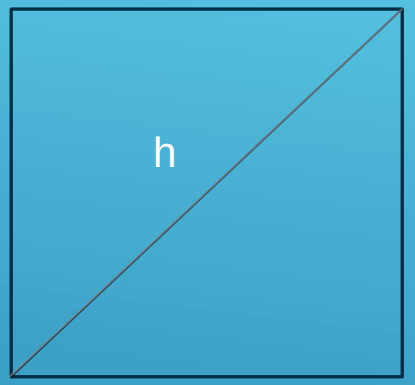
- **Angle of taper.** Specifies the taper between -90 and +90 degrees.
- **Specify two points.** Specifies the taper angle based on two specified points. The taper angle is the distance between the two specified points.

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.

AULA 6 Desenho Técnico Assistido por Computador

em planta

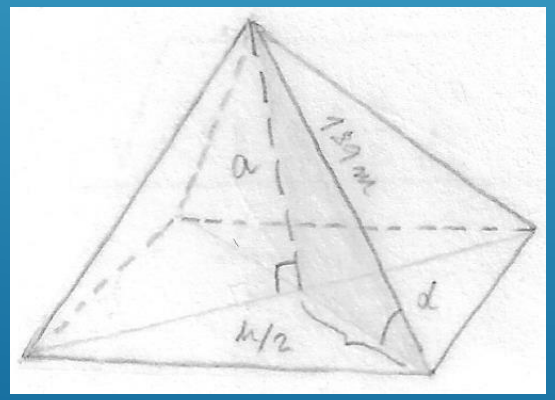


189

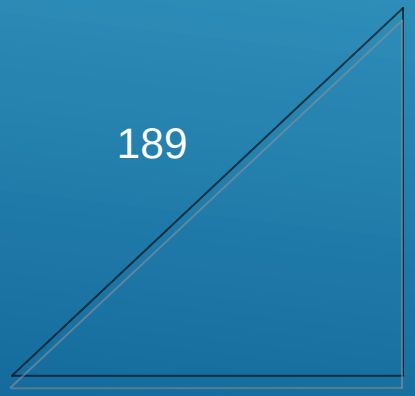
$$h^2 = \sqrt{189^2 + 189^2} => h = \sqrt{2x189^2} = 267.2864\ m => \frac{h}{2} = 133.6431\ m$$

Altura do vértice da pirâmide regular

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



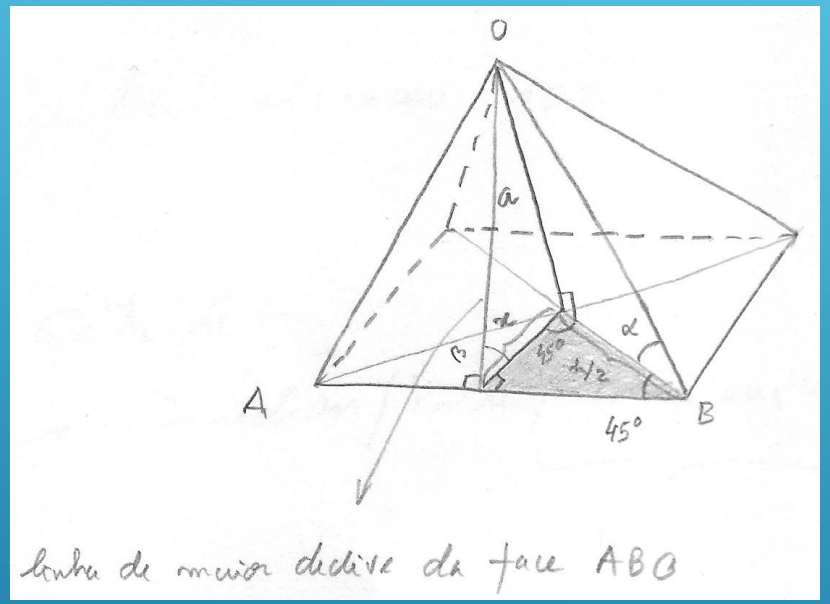
corte vertical



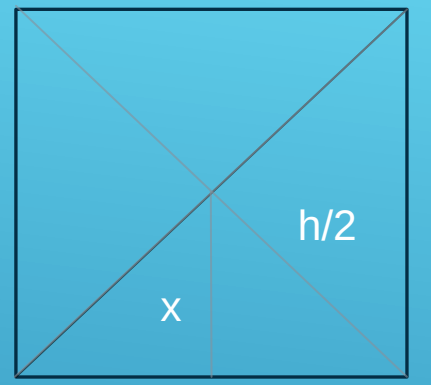
Inclinação da aresta da pirâmide regular

$$\tan\alpha = \frac{a}{\frac{h}{2}} = \frac{133.6431}{133.643} => \alpha = 45^o$$

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



em planta



189

$$\frac{\sin 90^0}{\frac{h}{2}} = \frac{\sin 45^0}{x} \Rightarrow x = \frac{\sqrt{2}}{2}\frac{h}{2} = 94.5000\ \text{m}$$

Inclinação da linha de maior declive da pirâmide regular

$$\tan\beta = \frac{a}{x} = \frac{\sqrt{189^2 - \left(\frac{h}{2}\right)^2}}{\frac{\sqrt{2}}{2}\frac{h}{2}} = \sqrt{\frac{189^2 - \frac{h^2}{4}}{\frac{1}{2}\frac{h^2}{4}}} = \sqrt{\frac{189^2 - \frac{189^2 + 189^2}{4}}{\frac{1}{2}\frac{189^2 + 189^2}{4}}} = \sqrt{\frac{4x189^2 - 2x189^2}{2x189^2}} = \sqrt{\frac{\frac{2x189^2}{4}}{\frac{189^2}{4}}} = \sqrt{2}$$

$$\Rightarrow \beta = 54.735614^0$$

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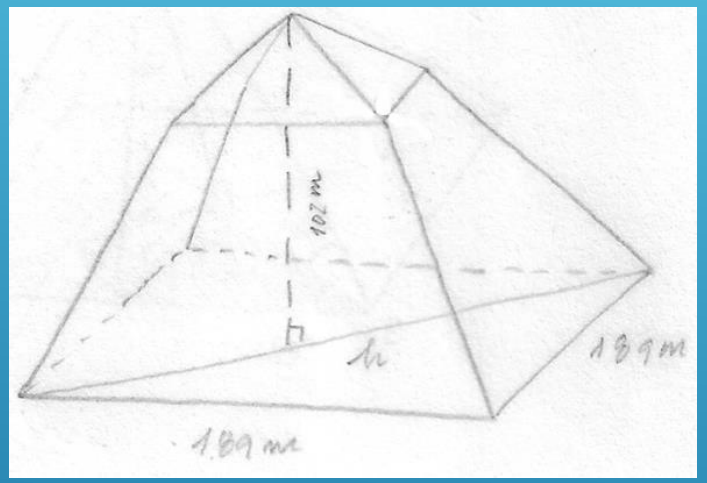
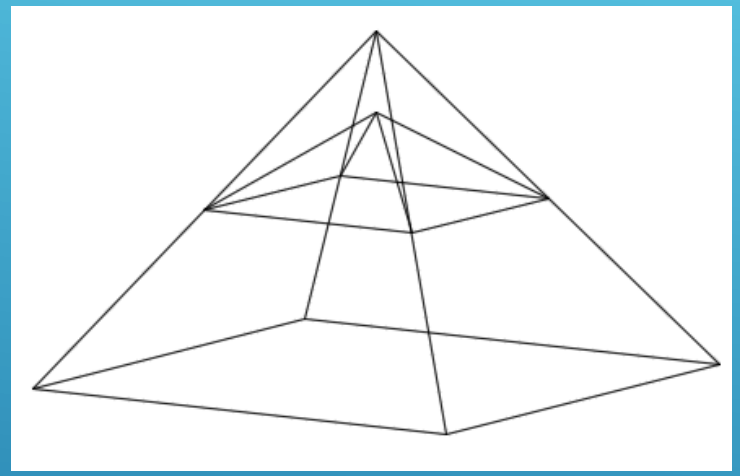
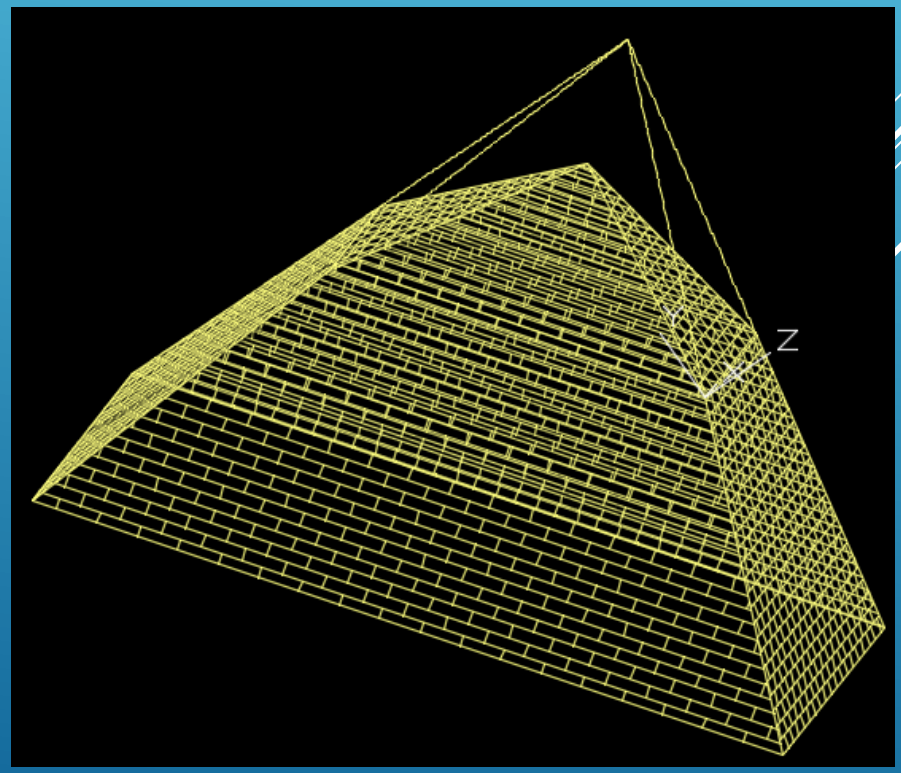
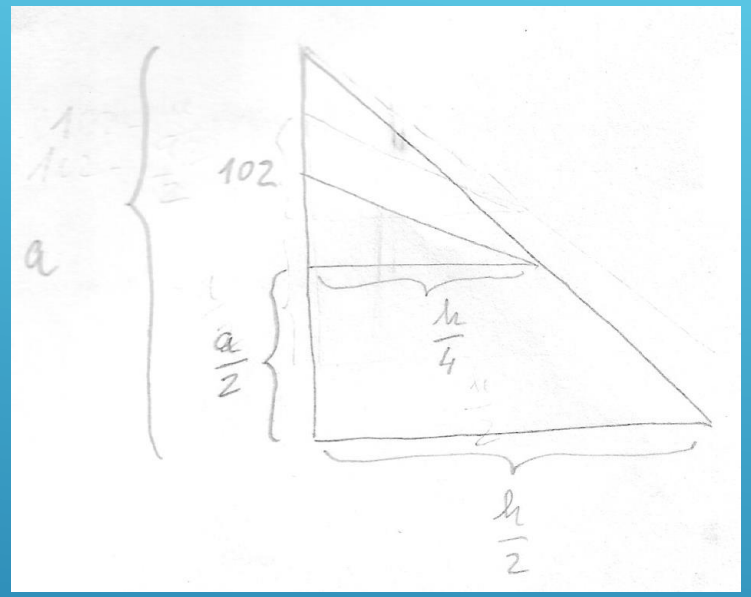






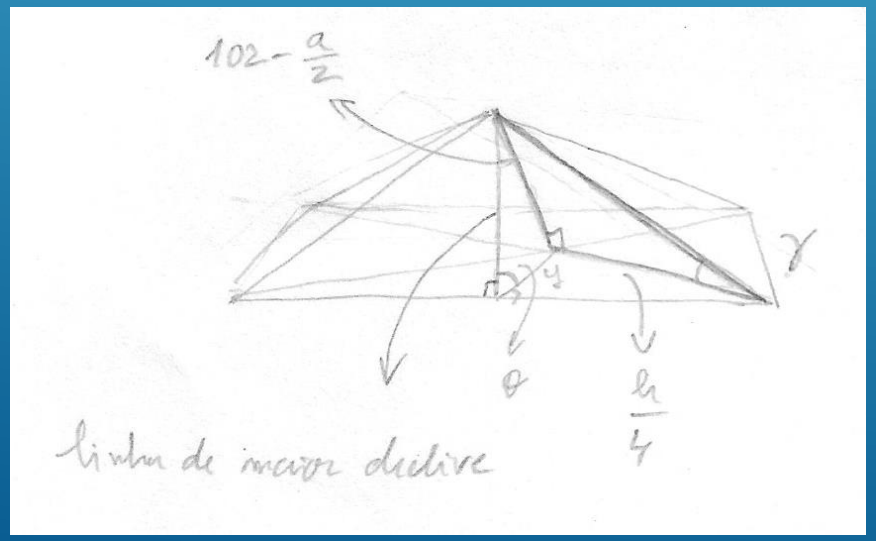
Quando a pirâmide alcançou metade da altura prevista: **a/2=66.8216 m**, a inclinação da pirâmide diminuiu, de tal forma que o vértice alcançou a altura 102 m.

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



Inclinação da aresta

$$\gamma = \operatorname{atan}\frac{102 - \frac{a}{2}}{\frac{h}{4}} = \operatorname{atan}\frac{35.17845}{66.80655} = 27^0.7702$$



Inclinação da linha de maior declive

$$\frac{\sin 90^0}{\frac{h}{4}} = \frac{\sin 45^0}{y} \Rightarrow y = \frac{\sqrt{2}}{2}\frac{h}{4}$$

$$\tan\theta = \frac{102 - \frac{a}{2}}{\frac{\sqrt{2}}{2}\frac{h}{4}} = \frac{35.17485}{47.24997} \Rightarrow \theta = 36^0.66553$$

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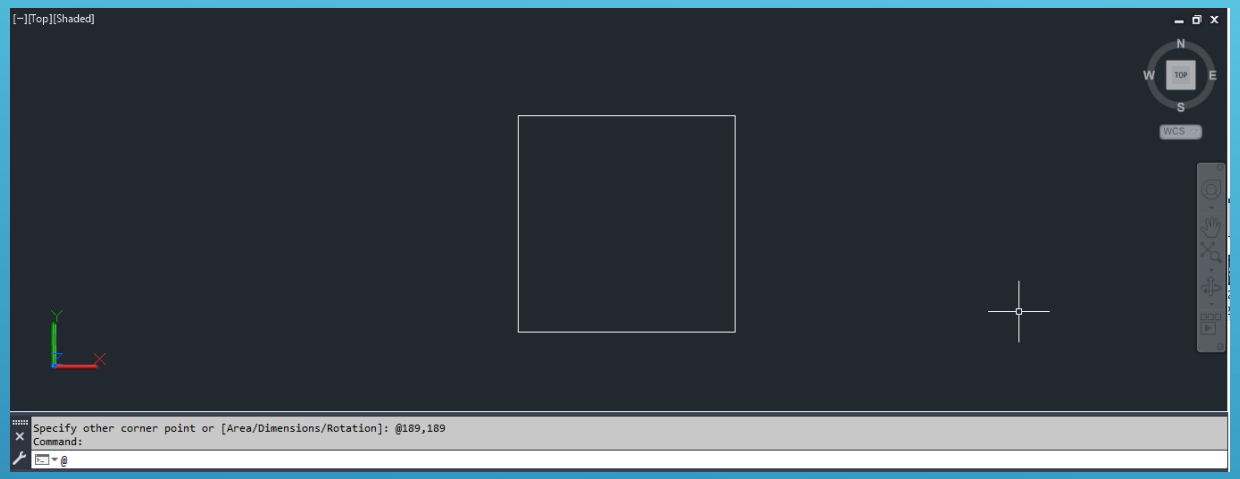

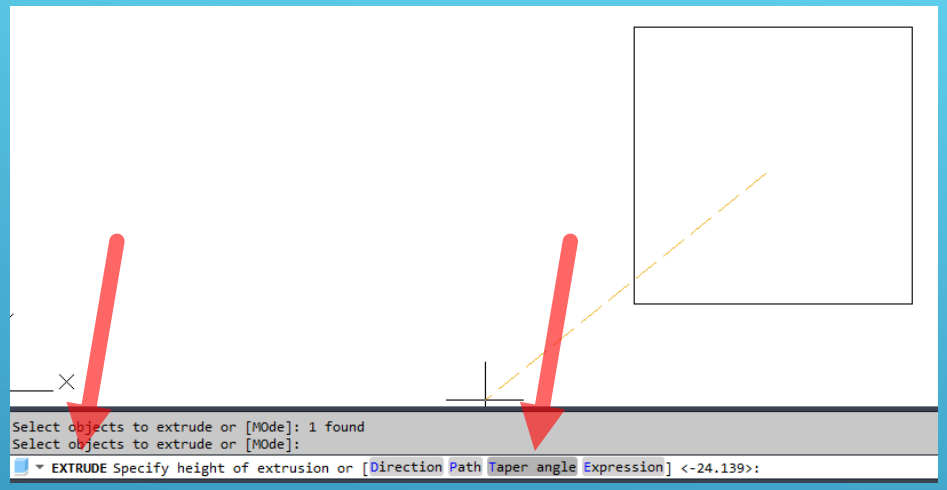

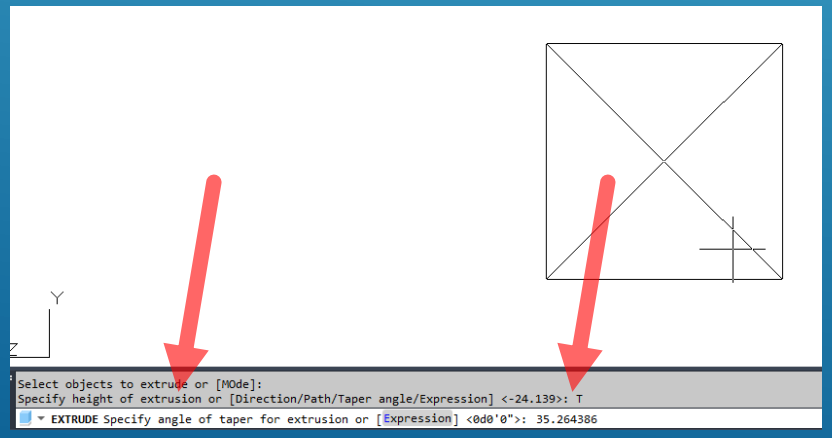

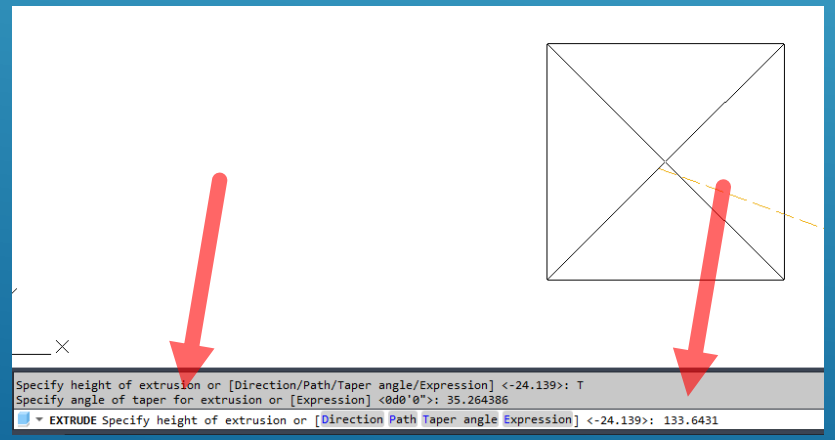

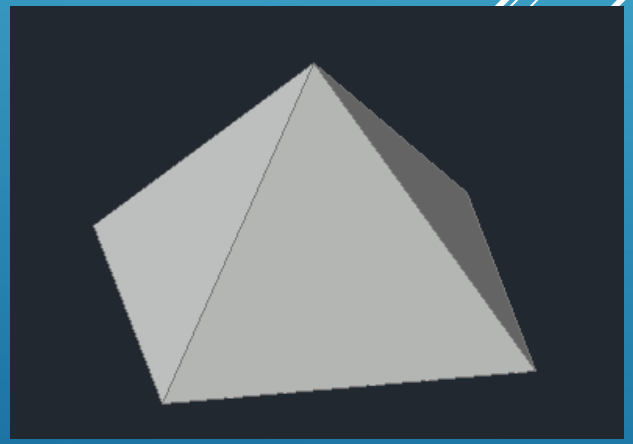

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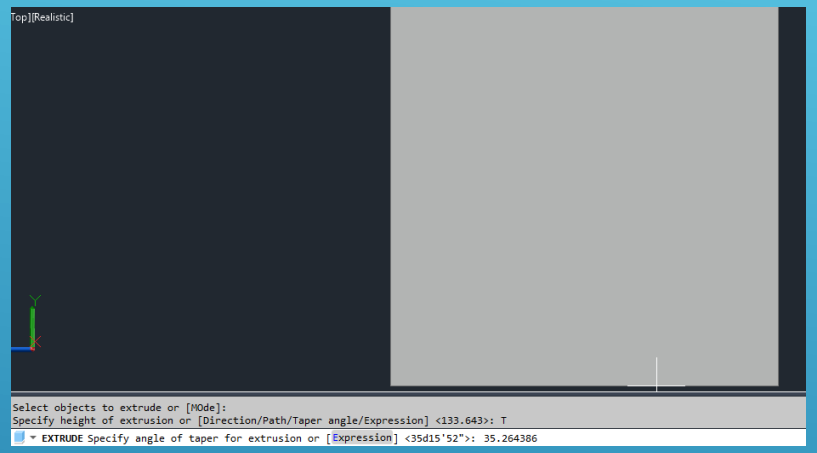
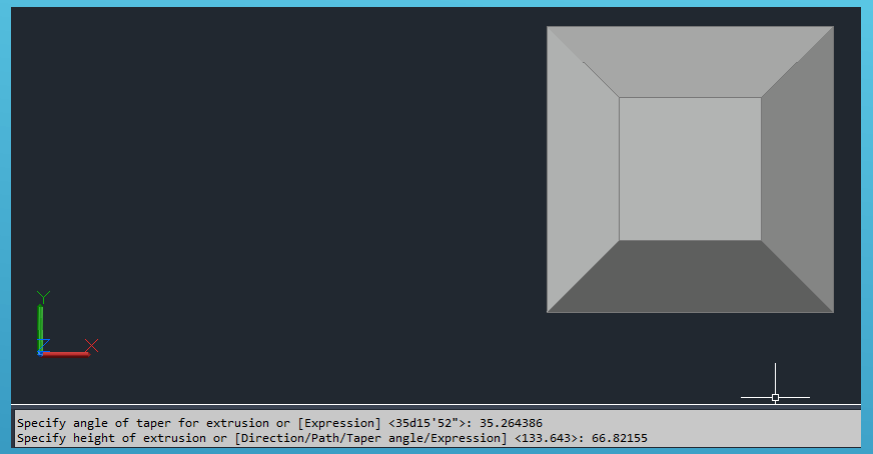
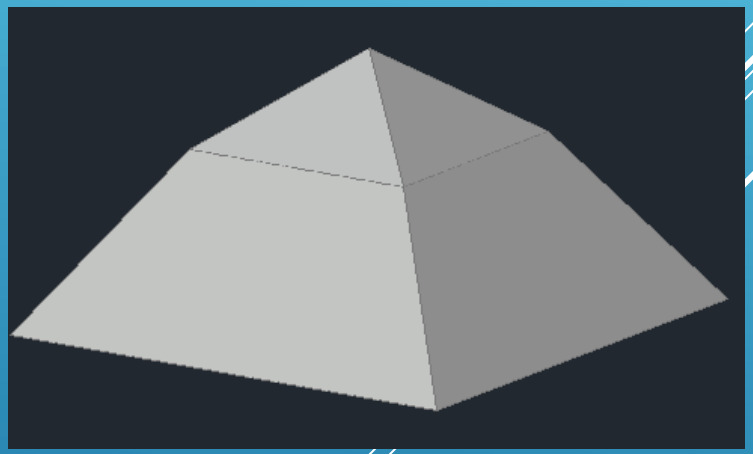
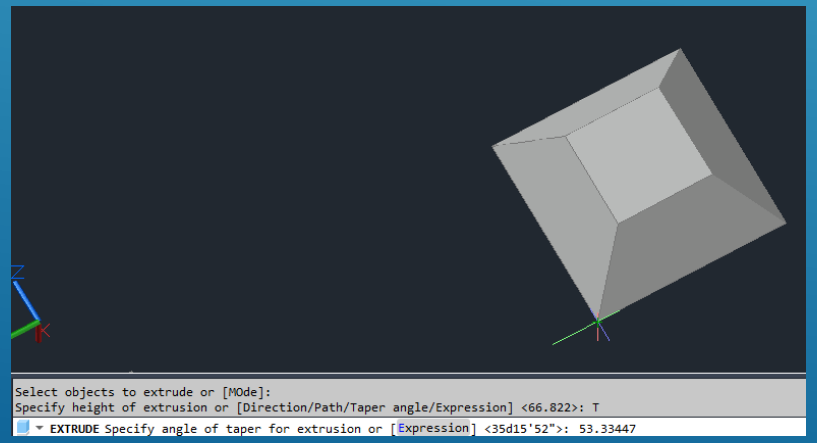
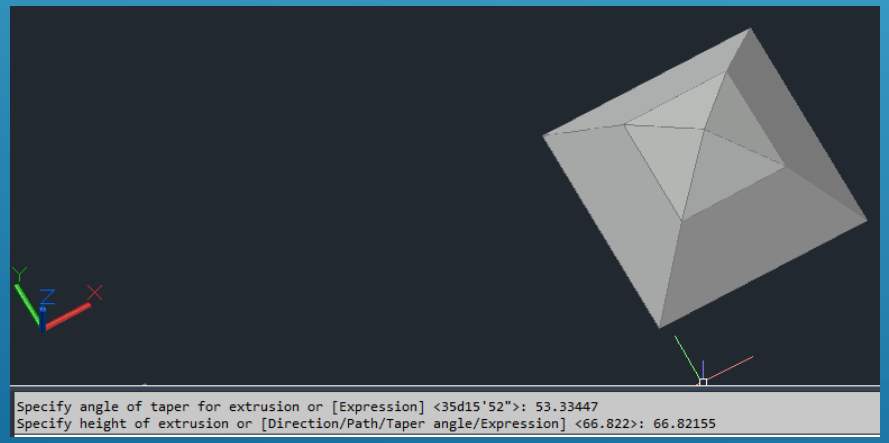


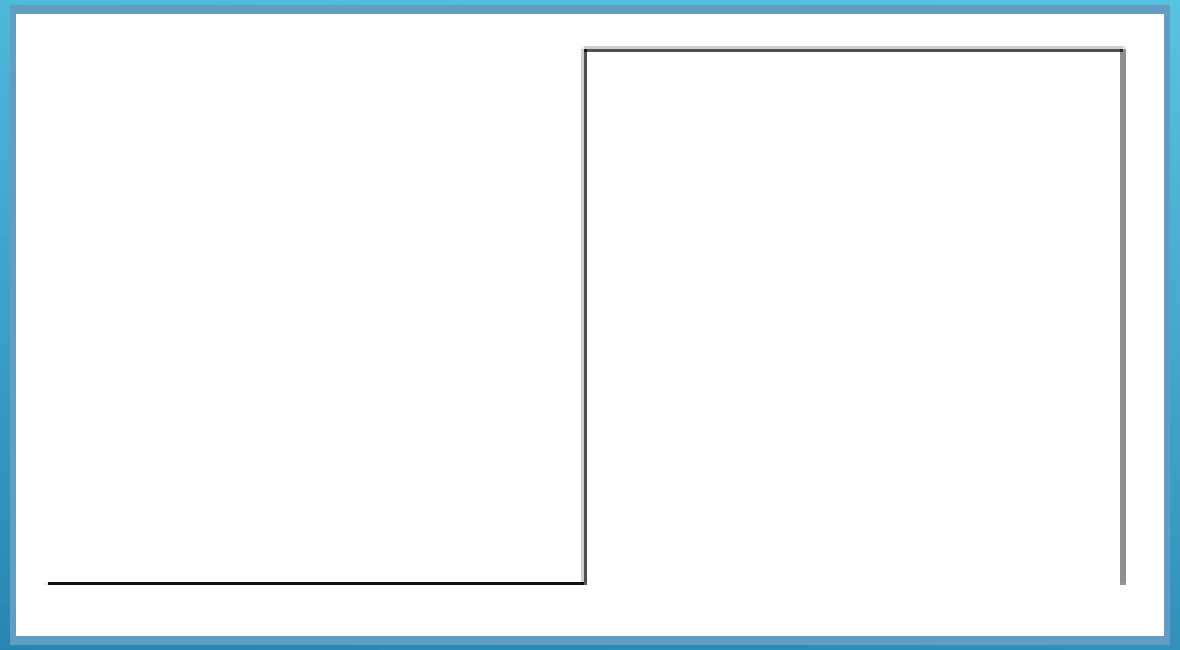




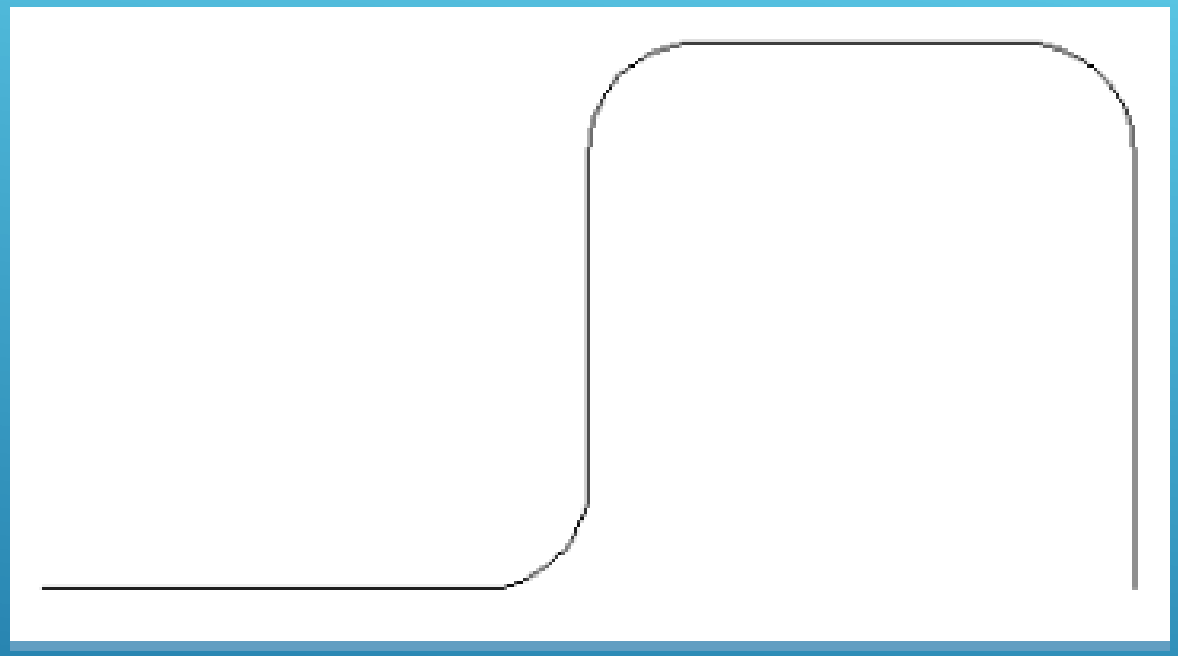




Extrude ao longo de um caminho



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



Utilizar o comando Fillet com raio 24

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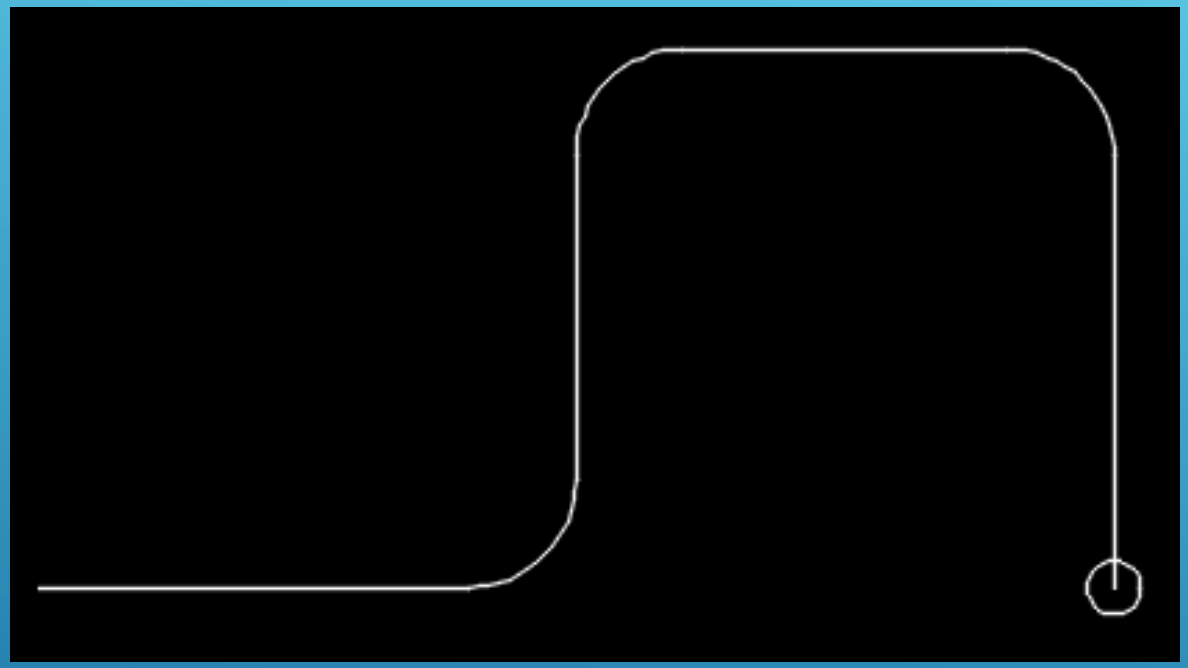

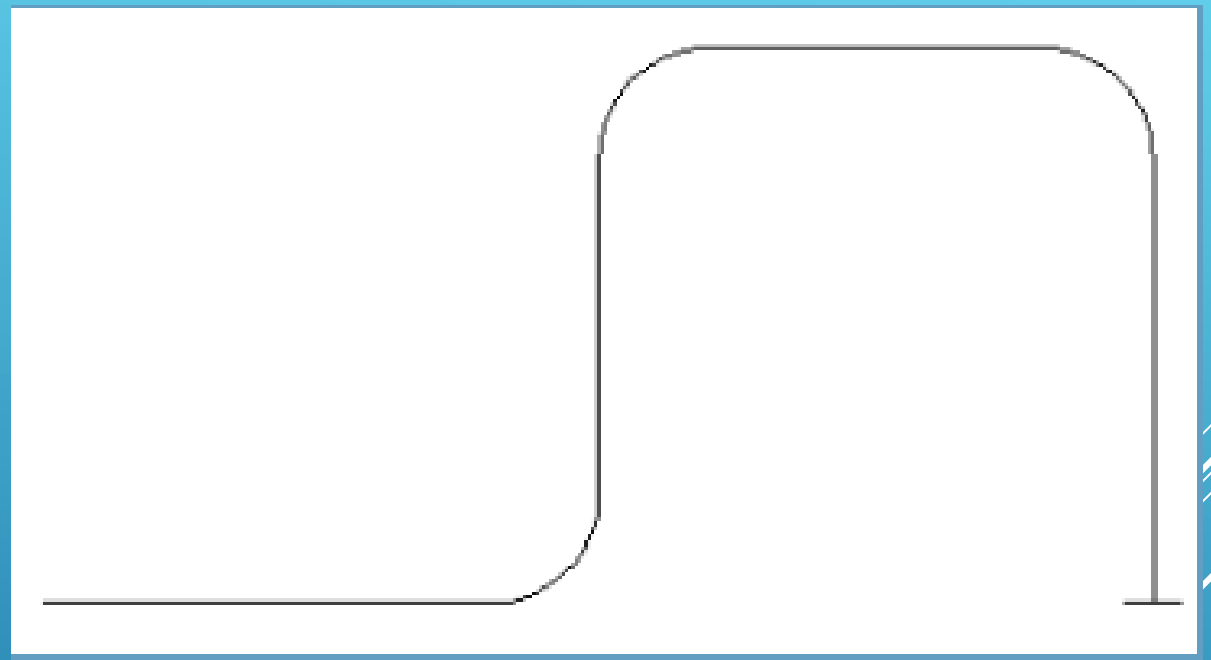

Vista Top

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

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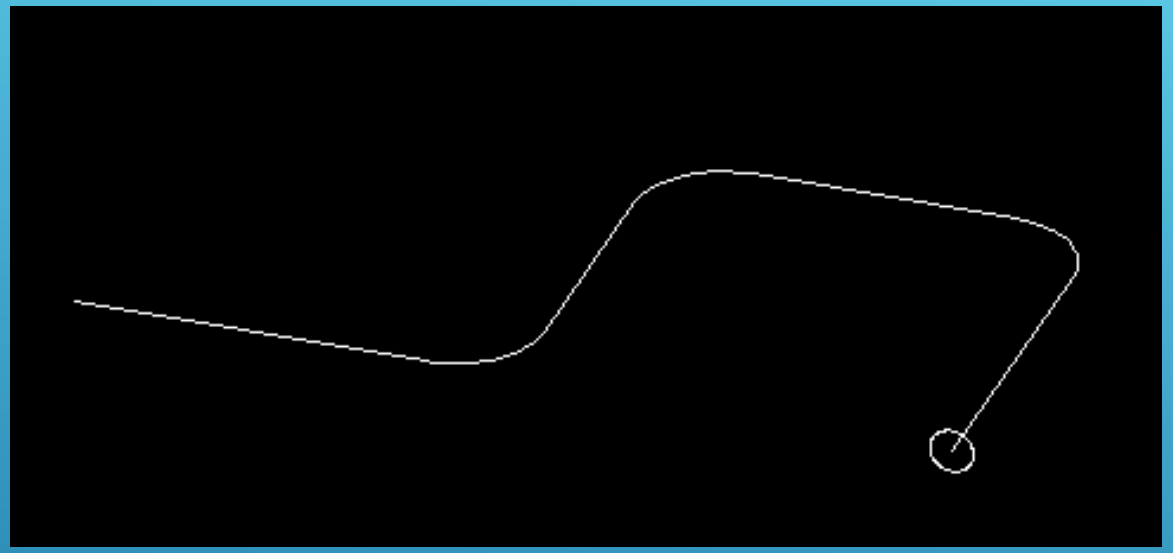
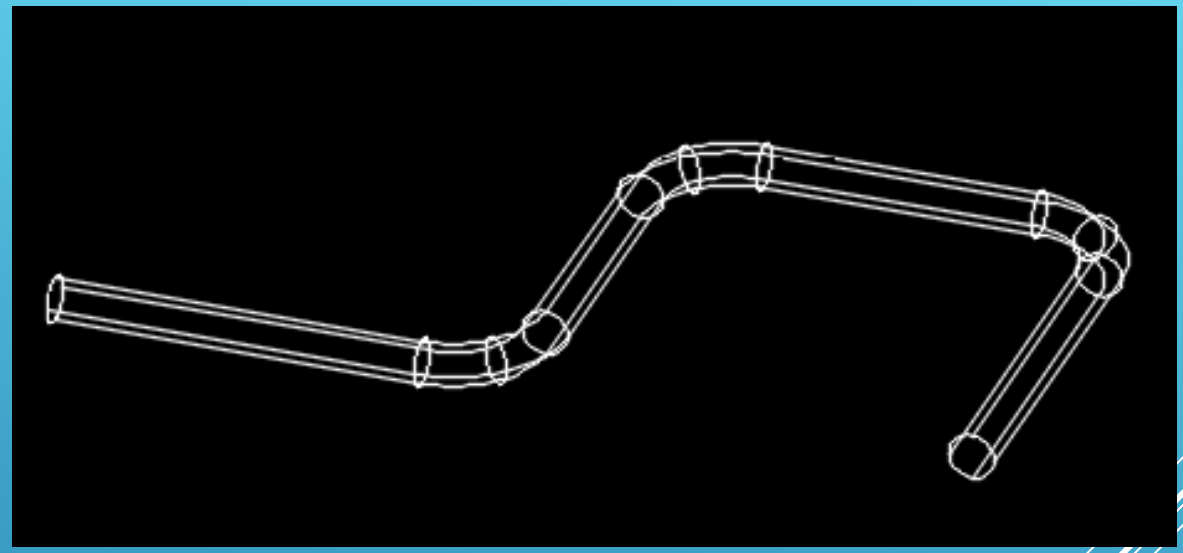
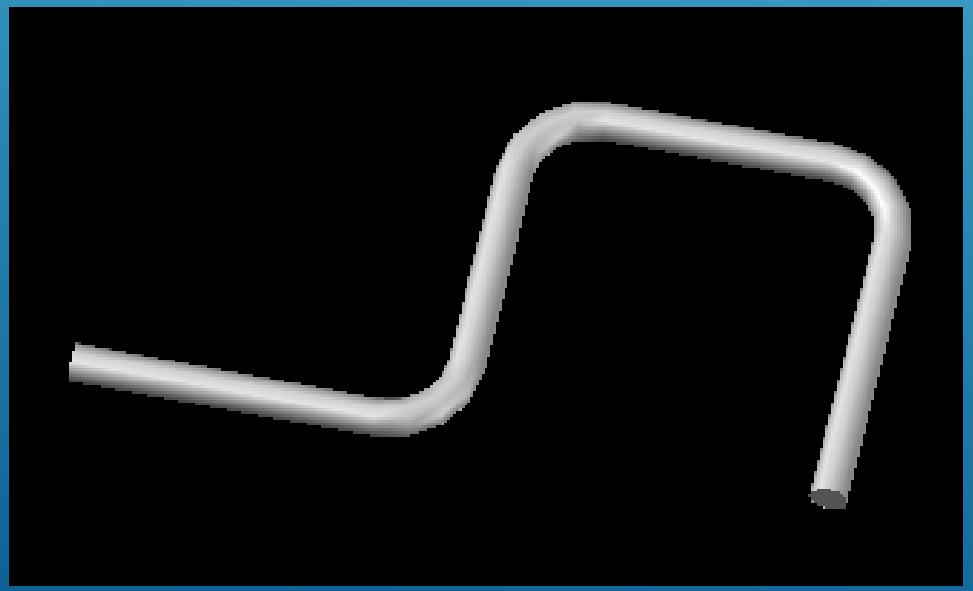
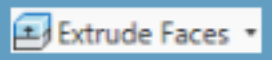




Utilizar o camando Extrude, seleccionar a circunferência, seleccionar **Path** e indicar a polyline



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.

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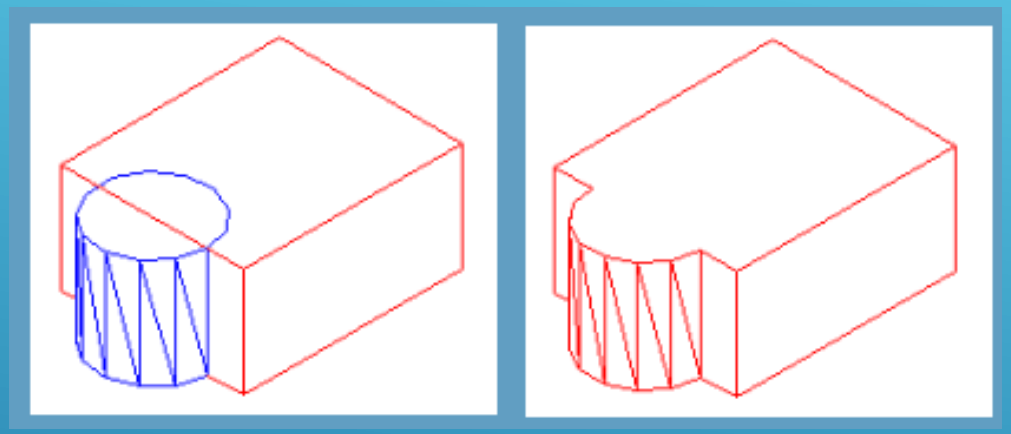
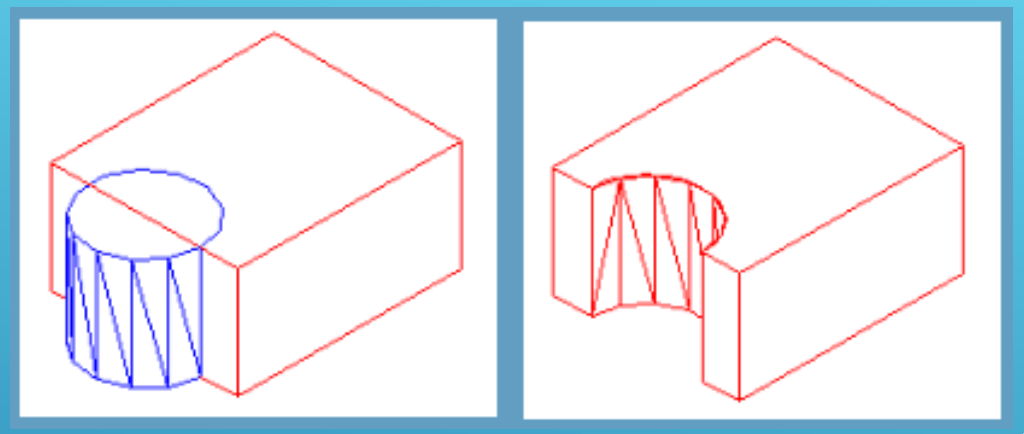
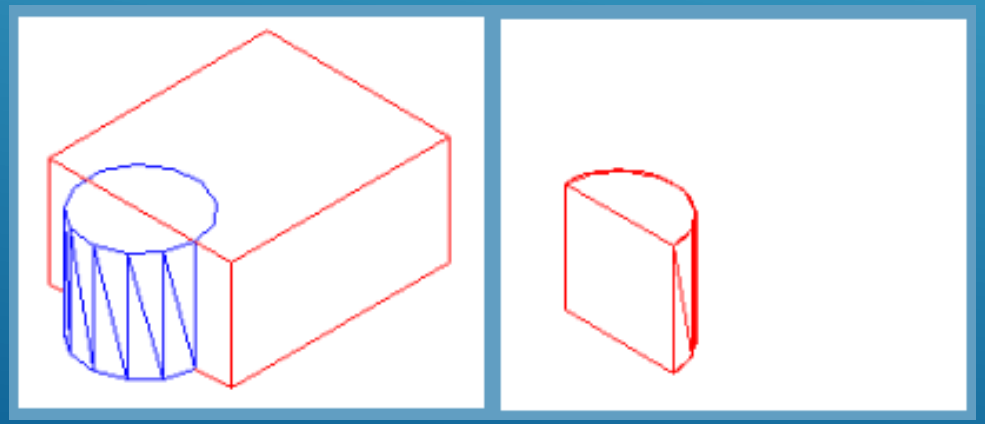


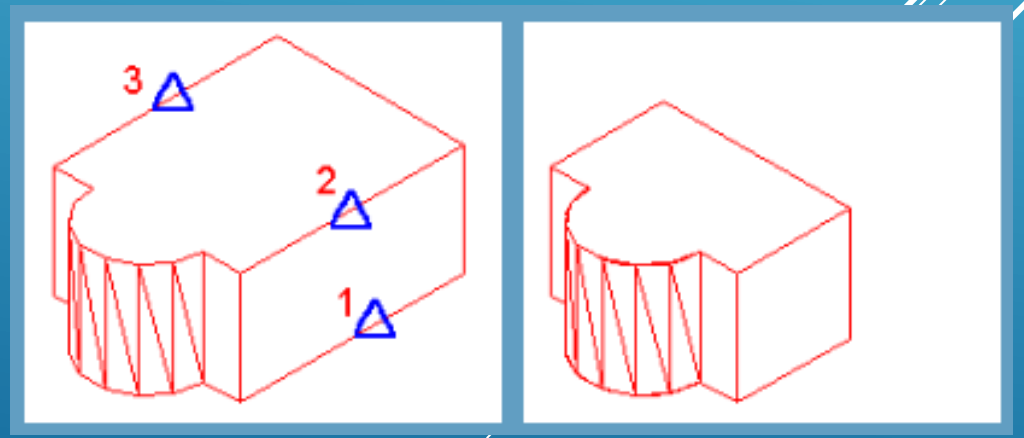
Union



Subtract



Intersect



Slice

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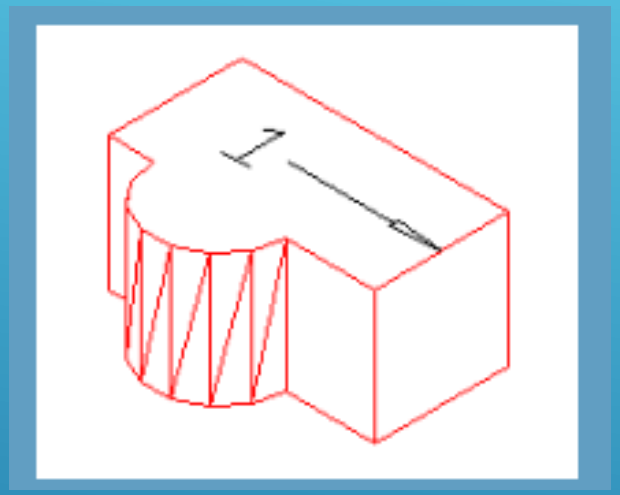
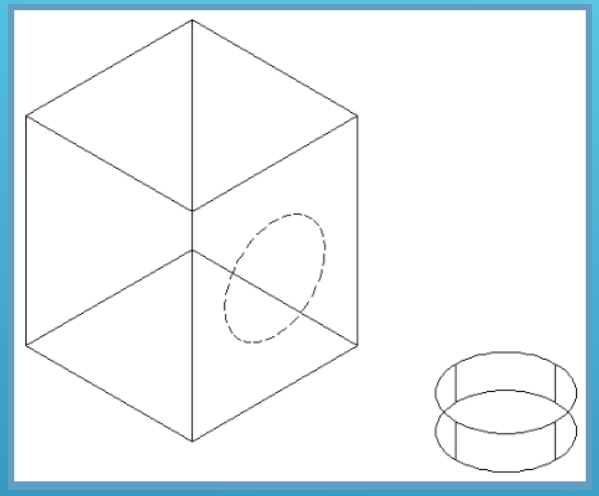
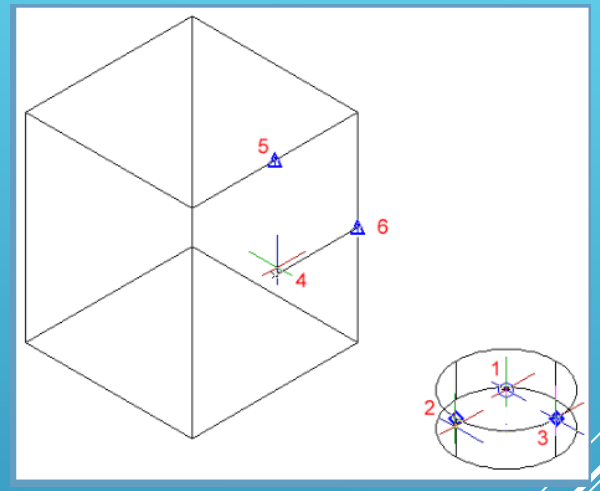
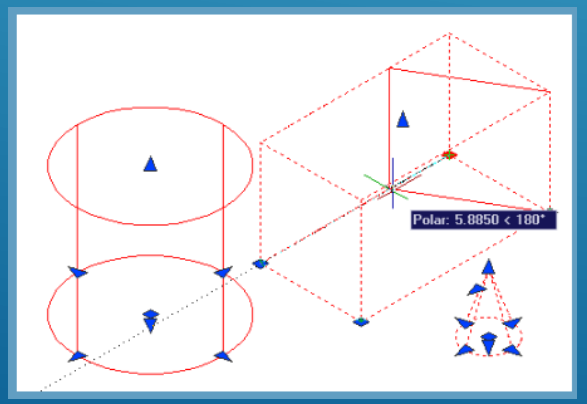


Extrude face

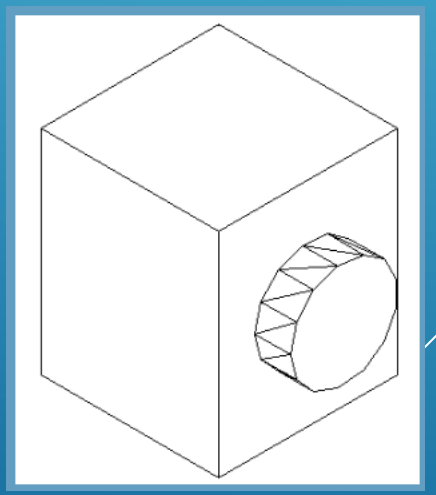






Extrude using grips

3D Align



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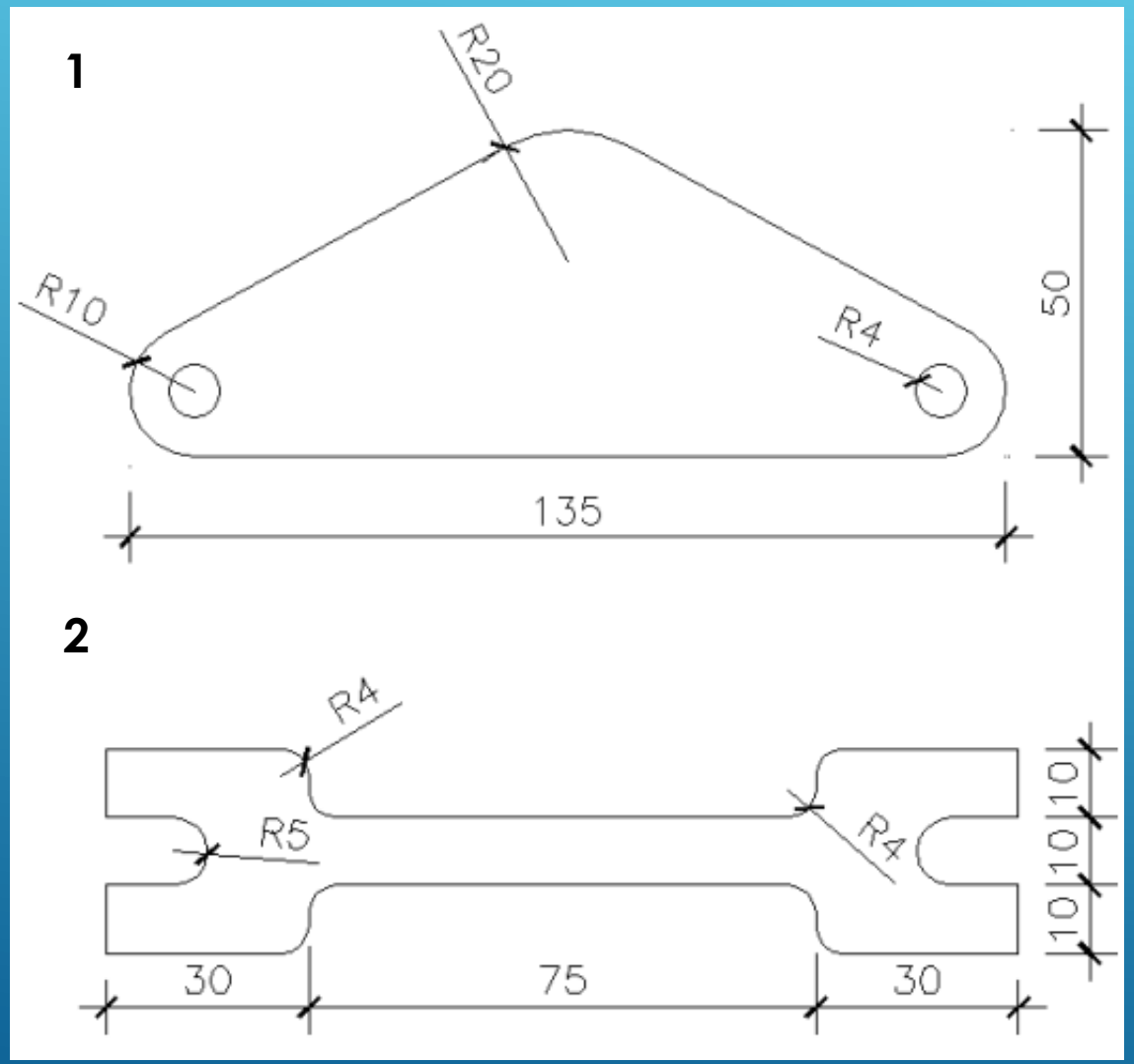

Após desenhar cada um dos desenhos, utilizar a função home > **Modify > Join** para transformar os vários elementos numa polyline única.

Desenhar na vista Top os desenhos 1 e 2

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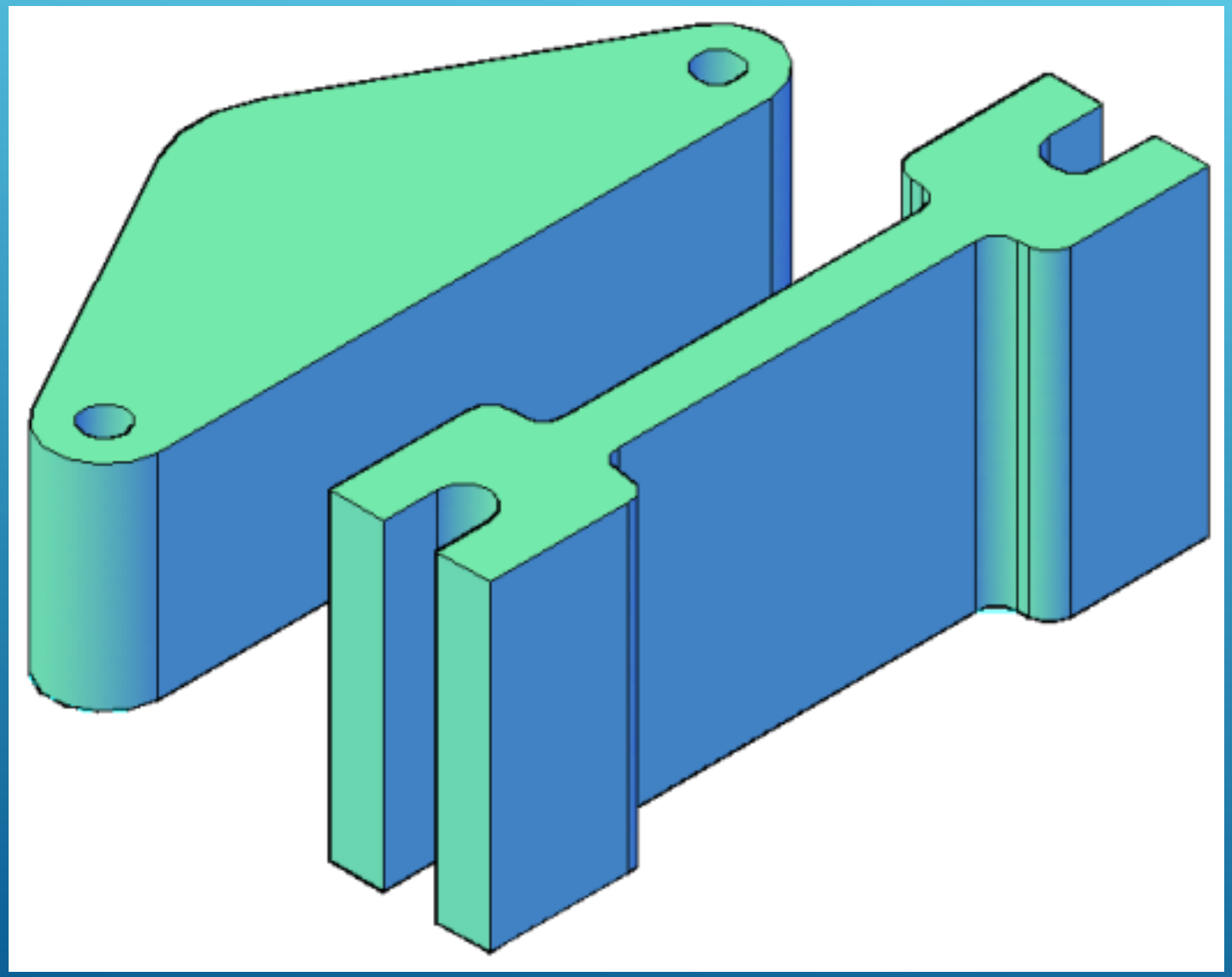

Extrude

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 aos furos (com Shift).

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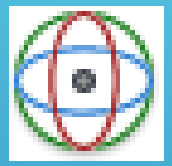
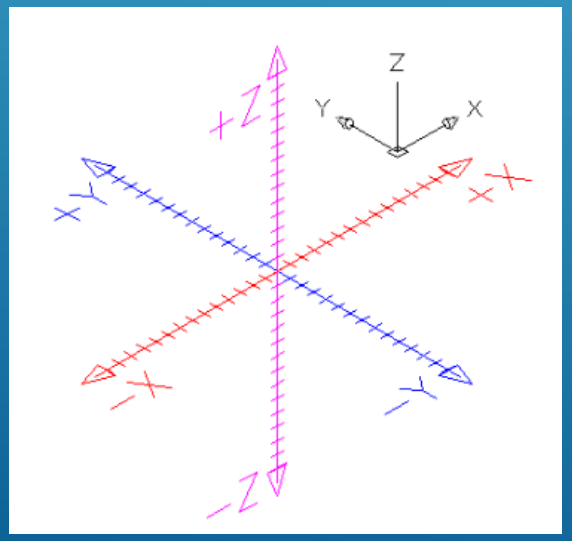
View > NE ISO

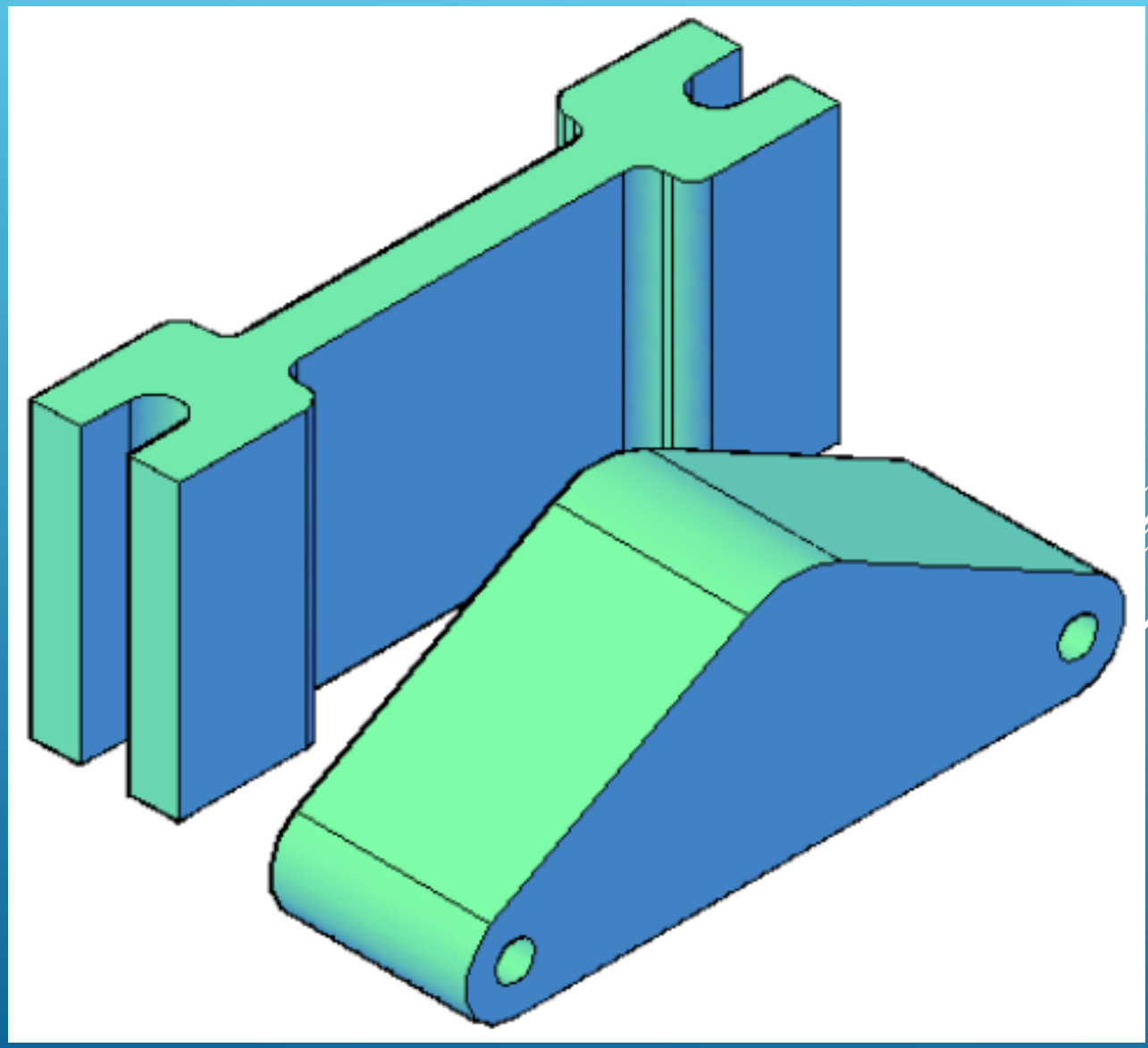
Modify > Rotate 3D



Seleccionar o desenho 1

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





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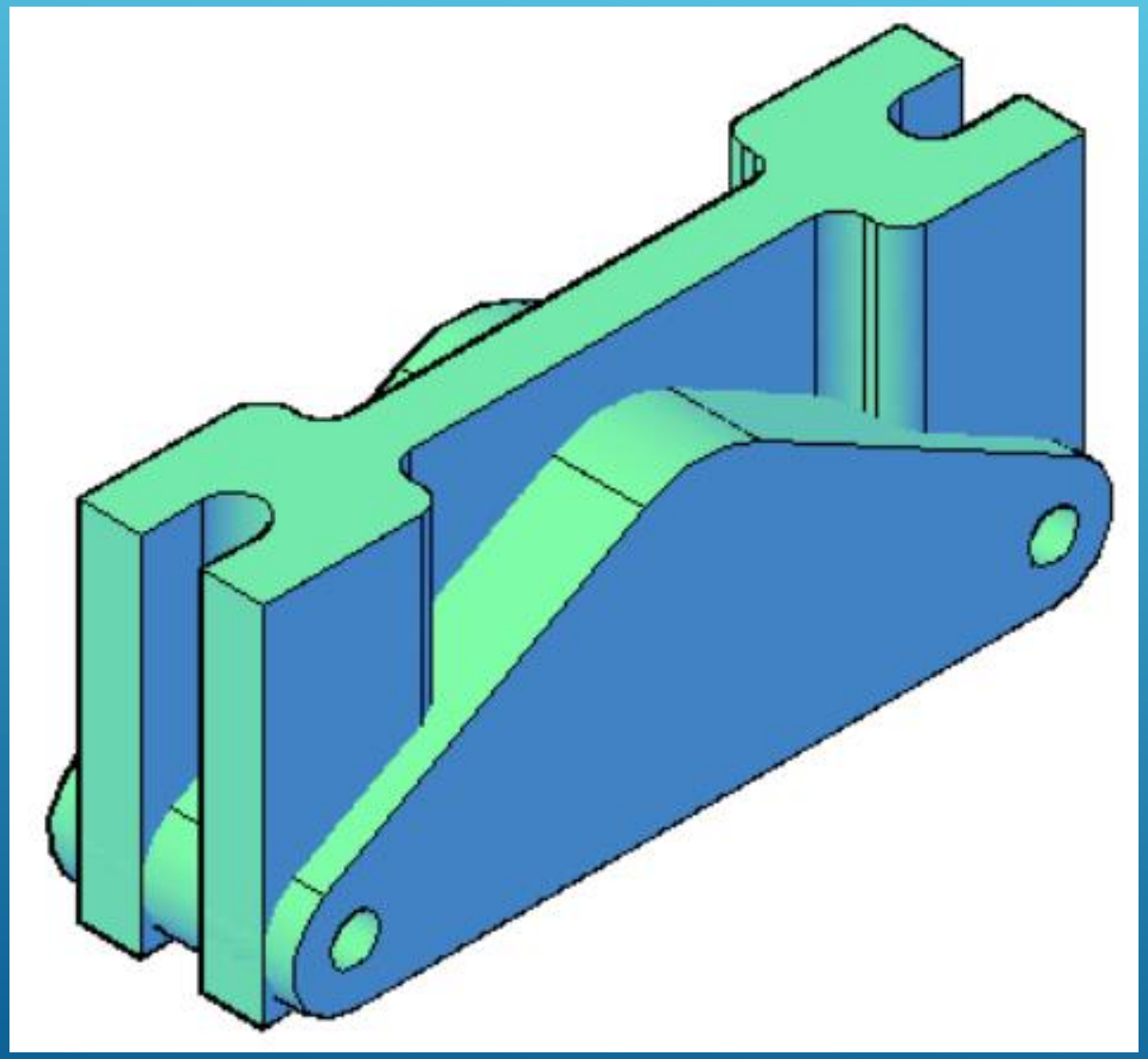

Posicionar o centro do desenho1 no centro do desenho 2

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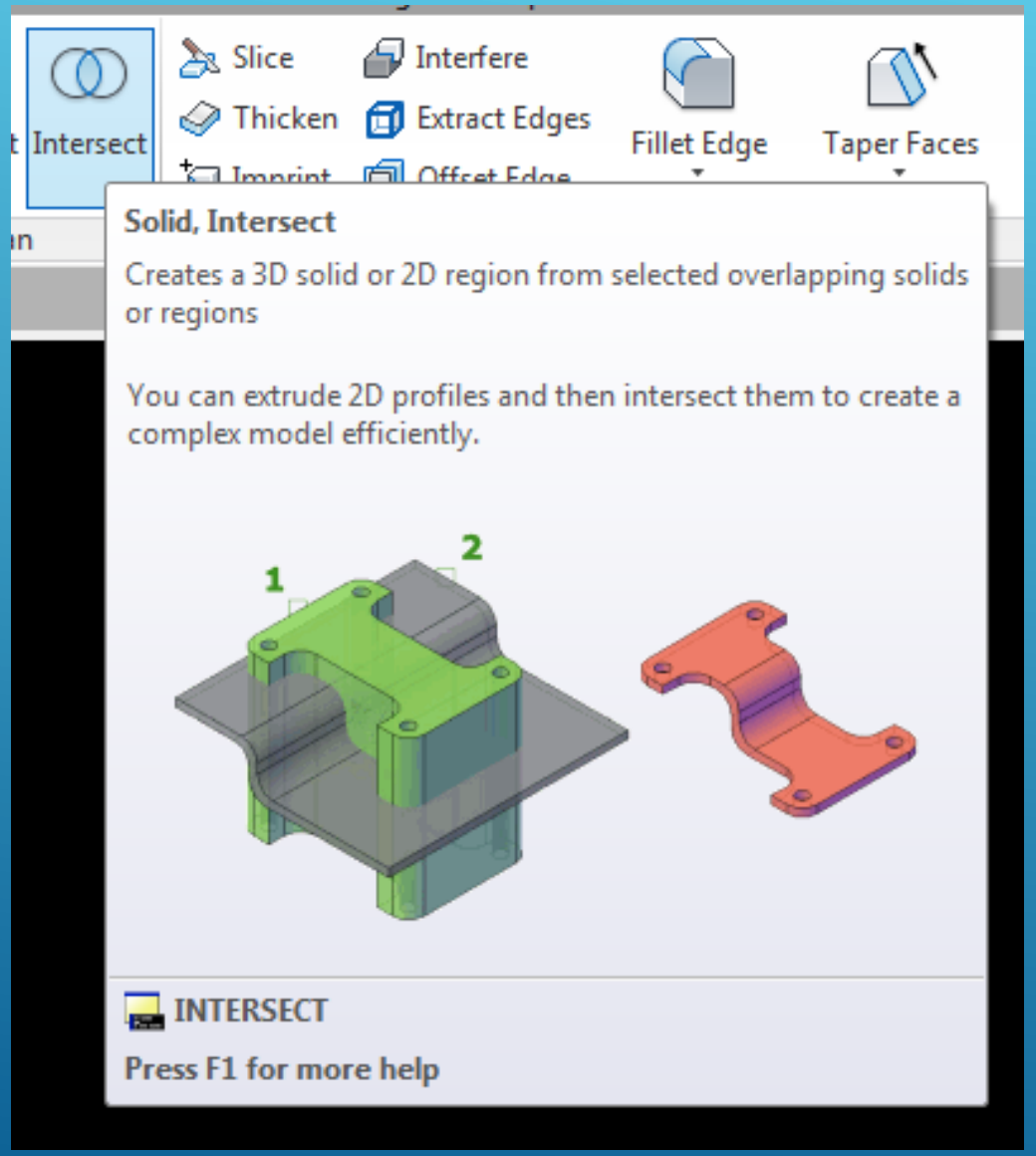

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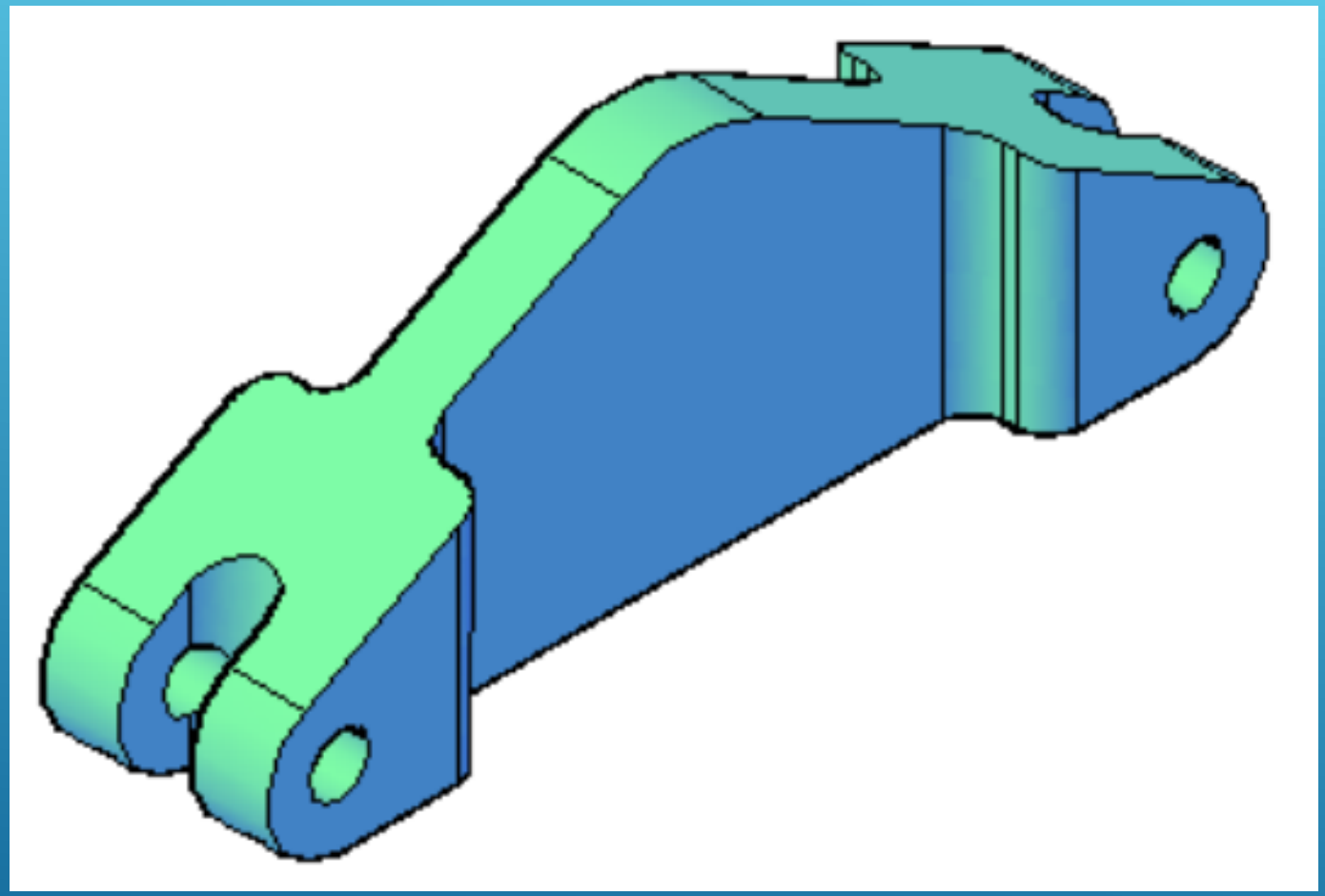

Efectuar a intercessão das duas figuras
(seleccionar as 2 figuras com shift + enter)

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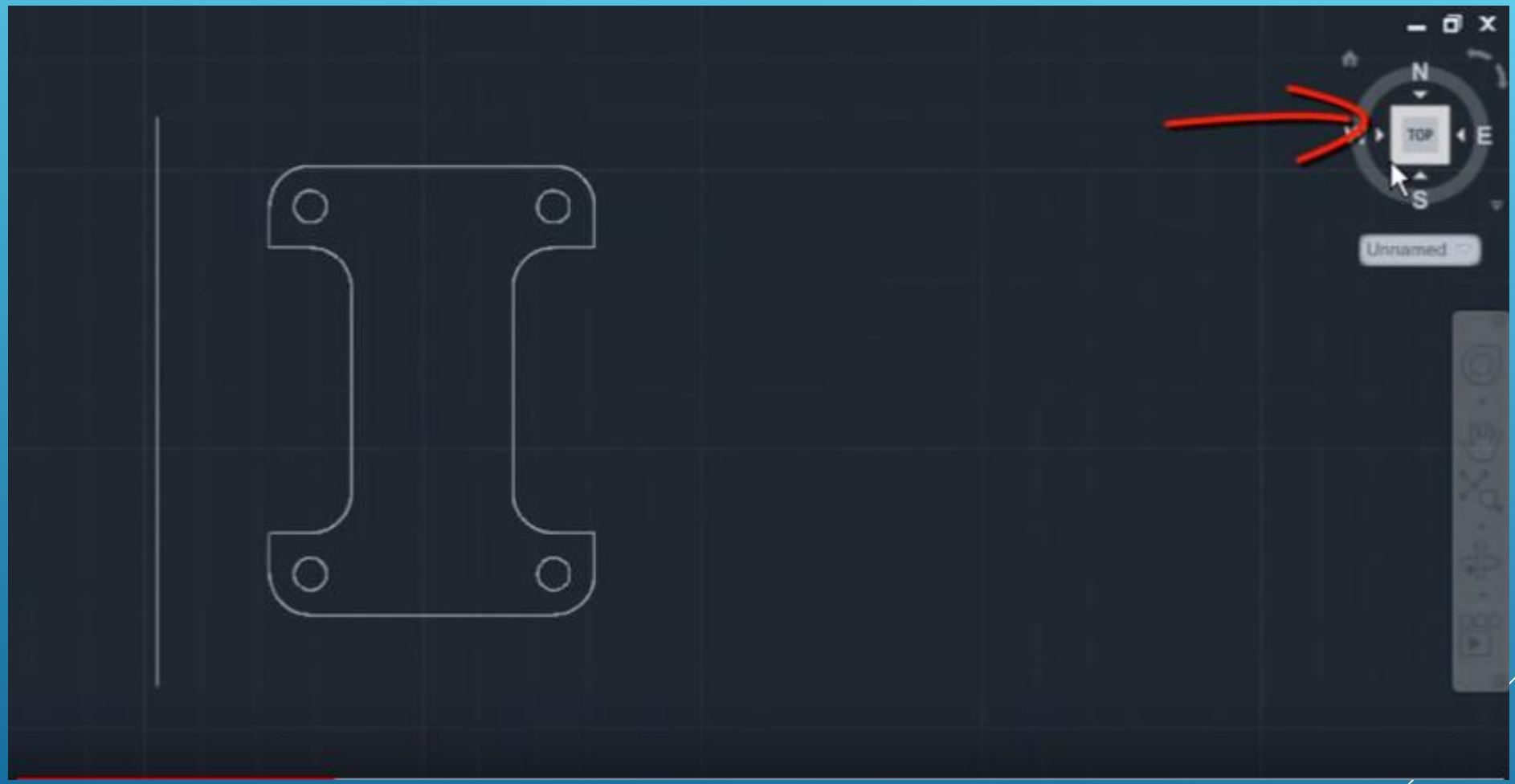

Ver intersect.dwg

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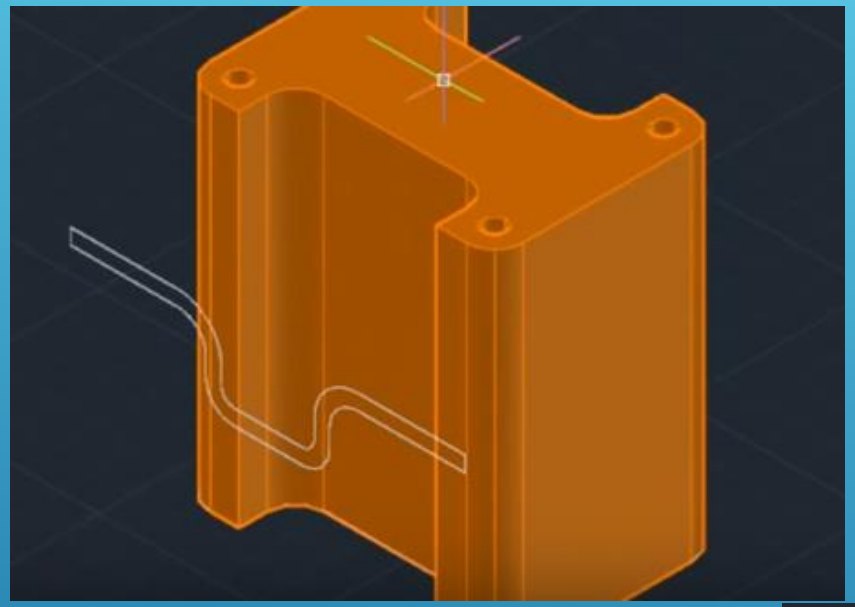

AULA 5 Desenho Técnico Assistido por Computador

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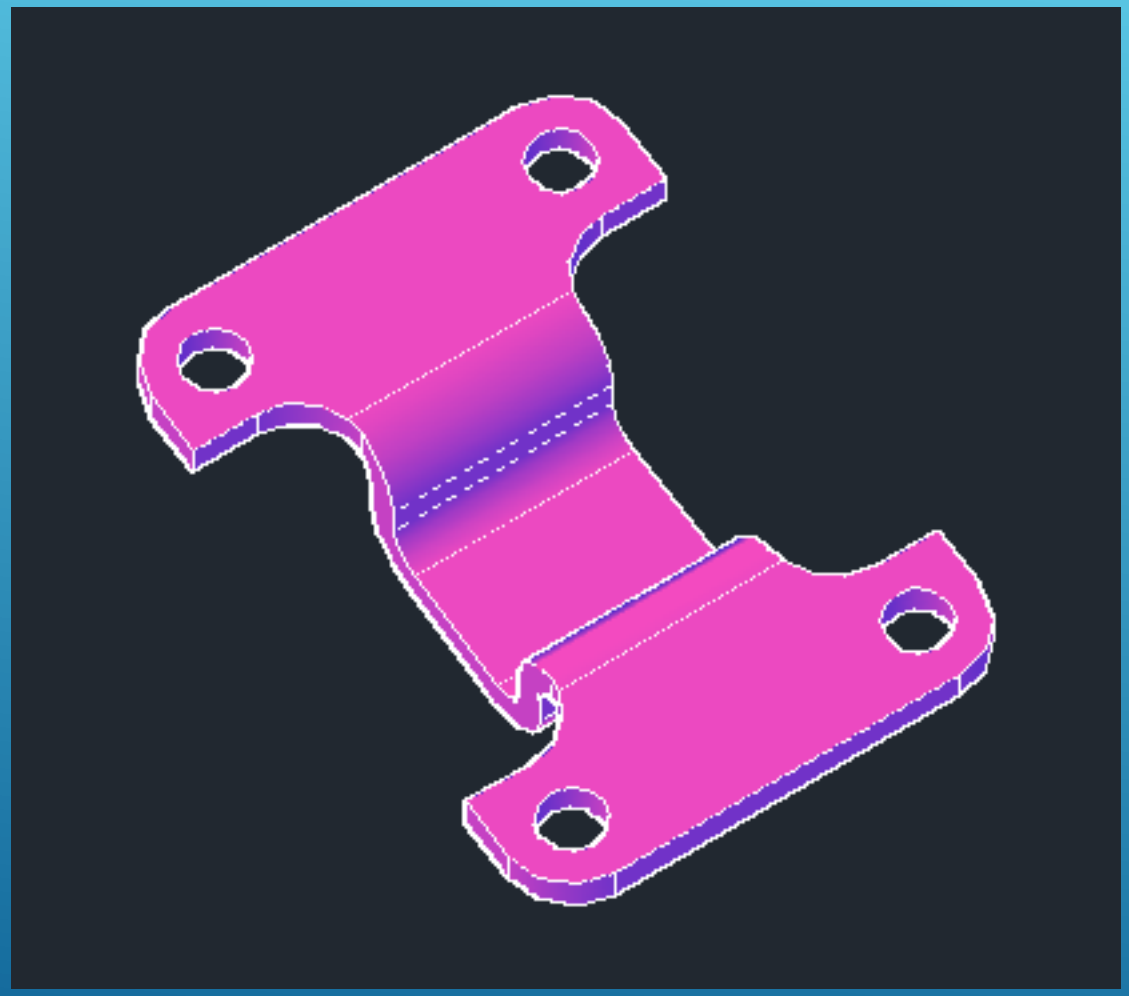

AULA 5 Desenho Técnico Assistido por Computador

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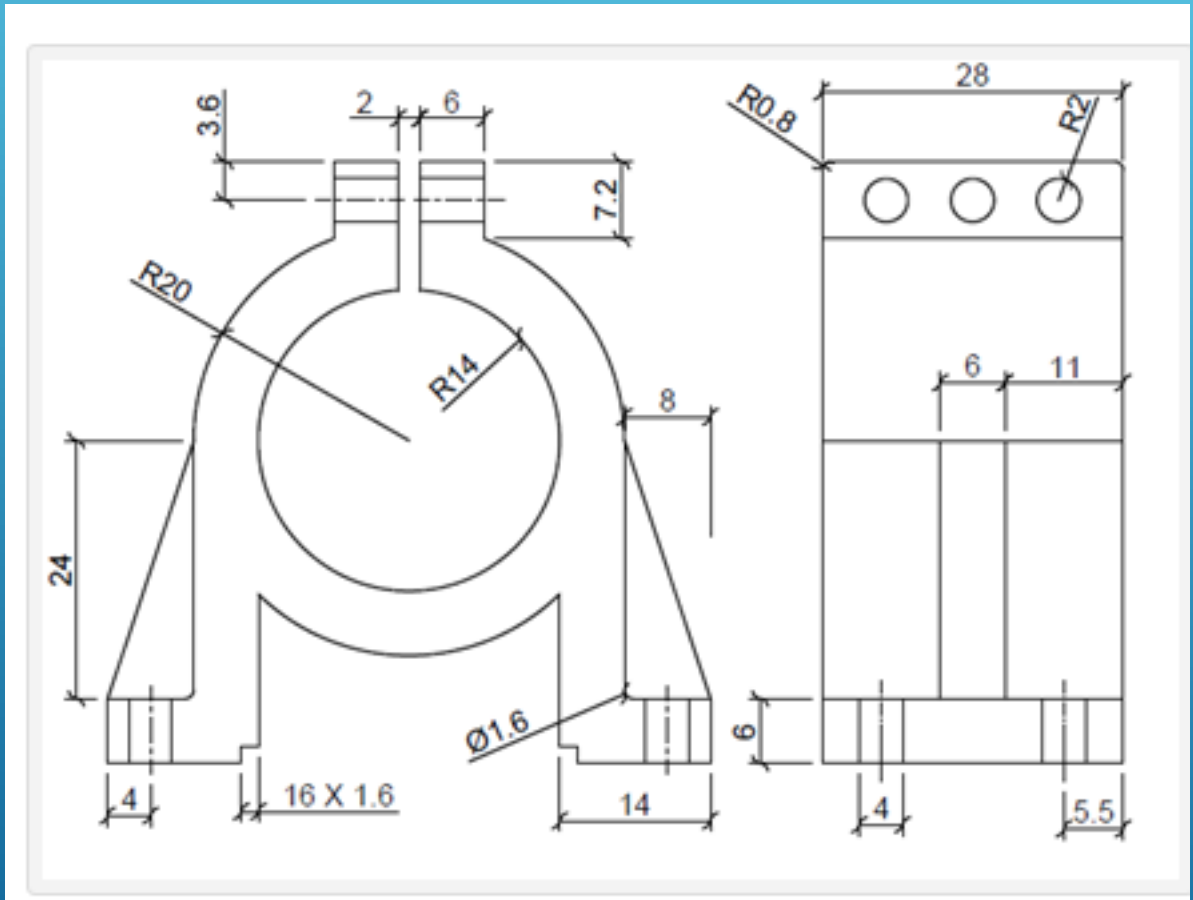

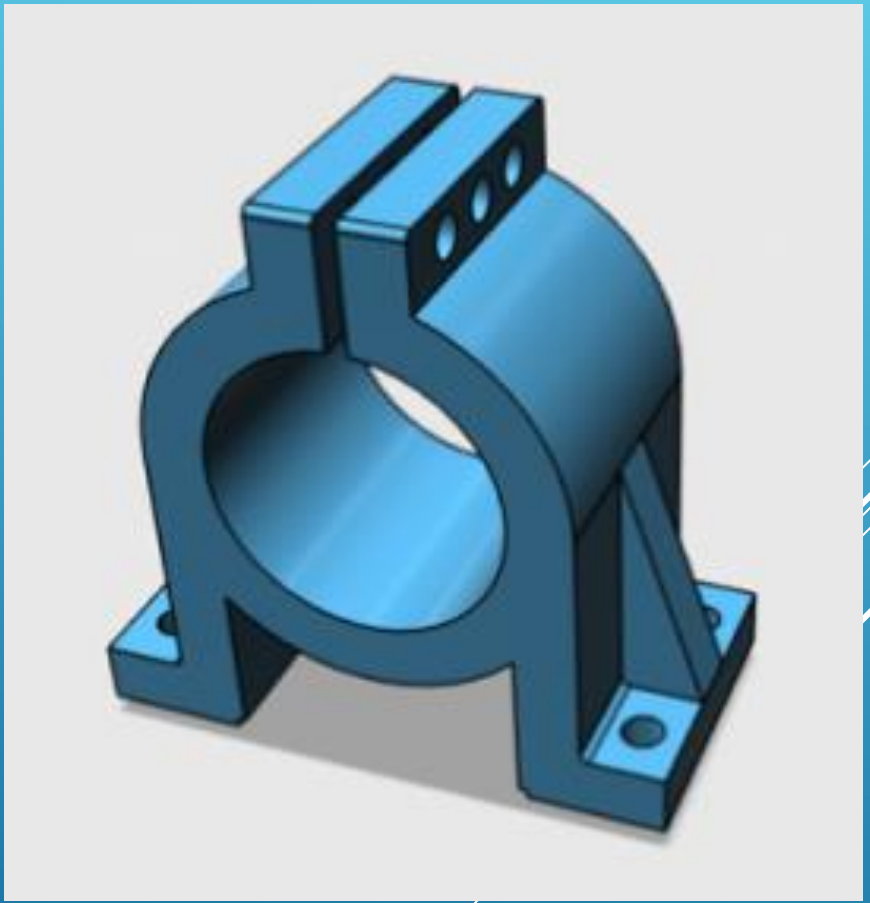

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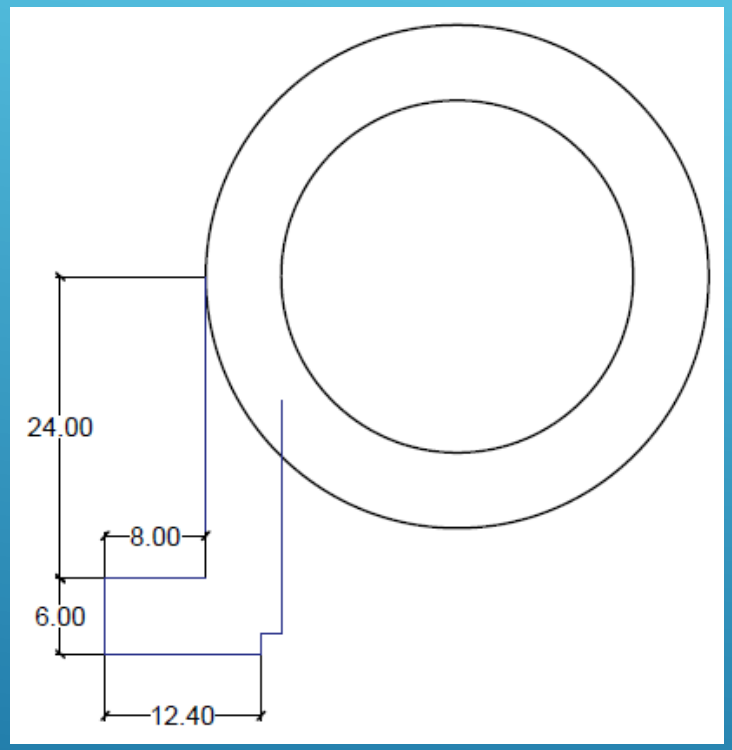

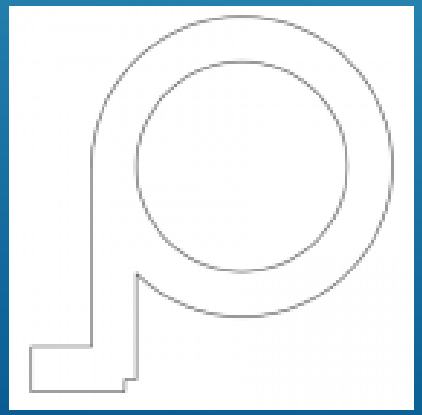

1.Desenhar 2 circunferências concêntricas de raios R20 e R14.
2.Desenhar uma linha vertical para baixo partindo do quadrante esquerdo da circunferência de raio maior com 24 unidades de comprimento.
3.Desenhar uma linha horizontal com 8 unidades de comprimento para a esquerda a partir da extremidade inferior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

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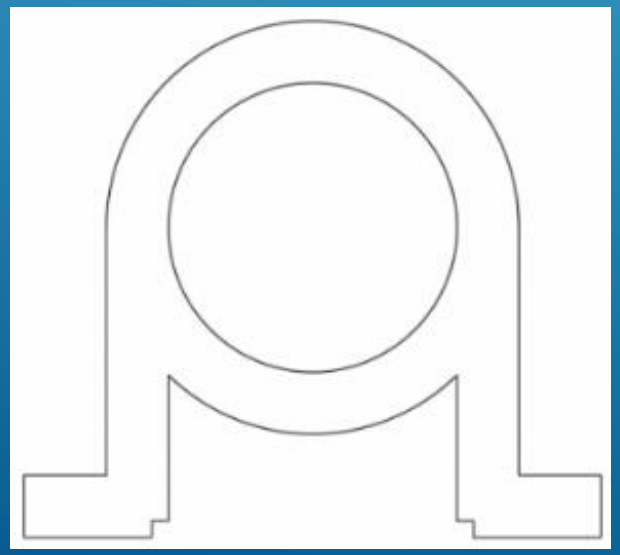
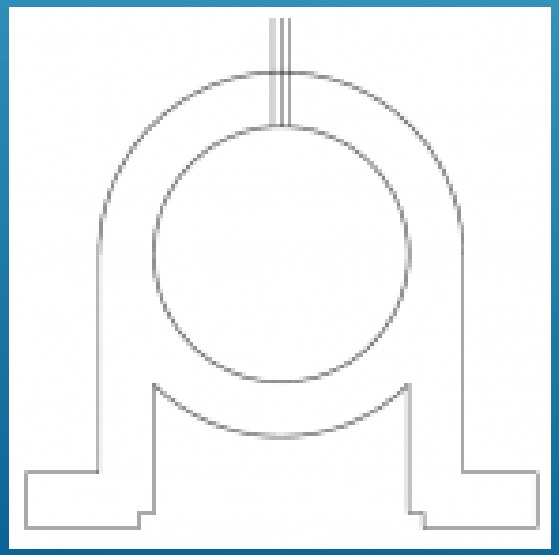
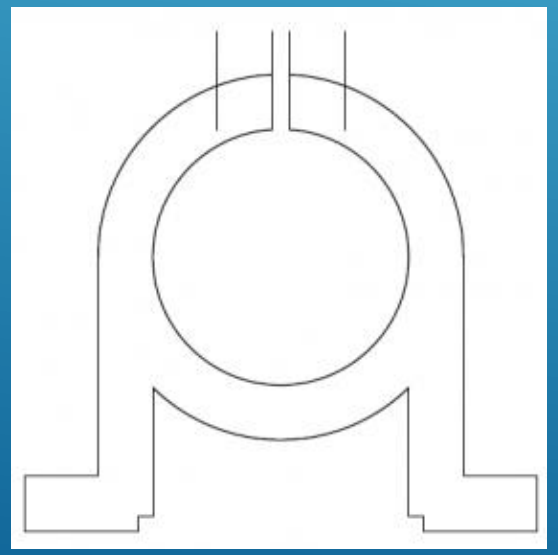
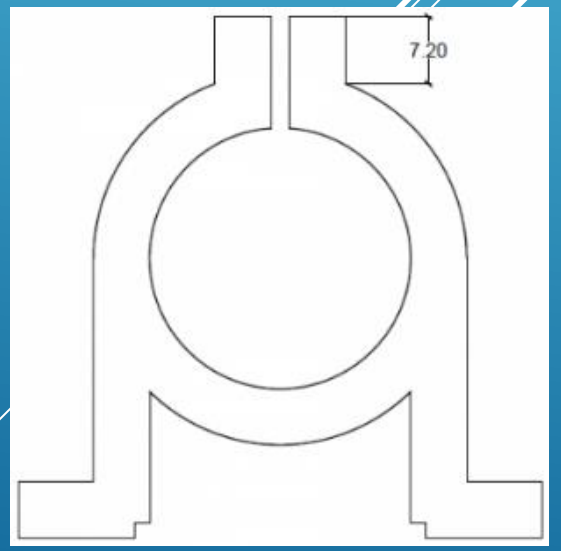
10.Utilize o comando Mirror para replicar a construção anterior para o lado direito do objecto.
11.Complete o desenho com o comando Trim.
12.Utilize o quadrante superior da circunferência interior para desenhar uma linha vertical que atravesse a circunferência exterior.
13.Utilize o comando Offset para 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 inferirores 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.









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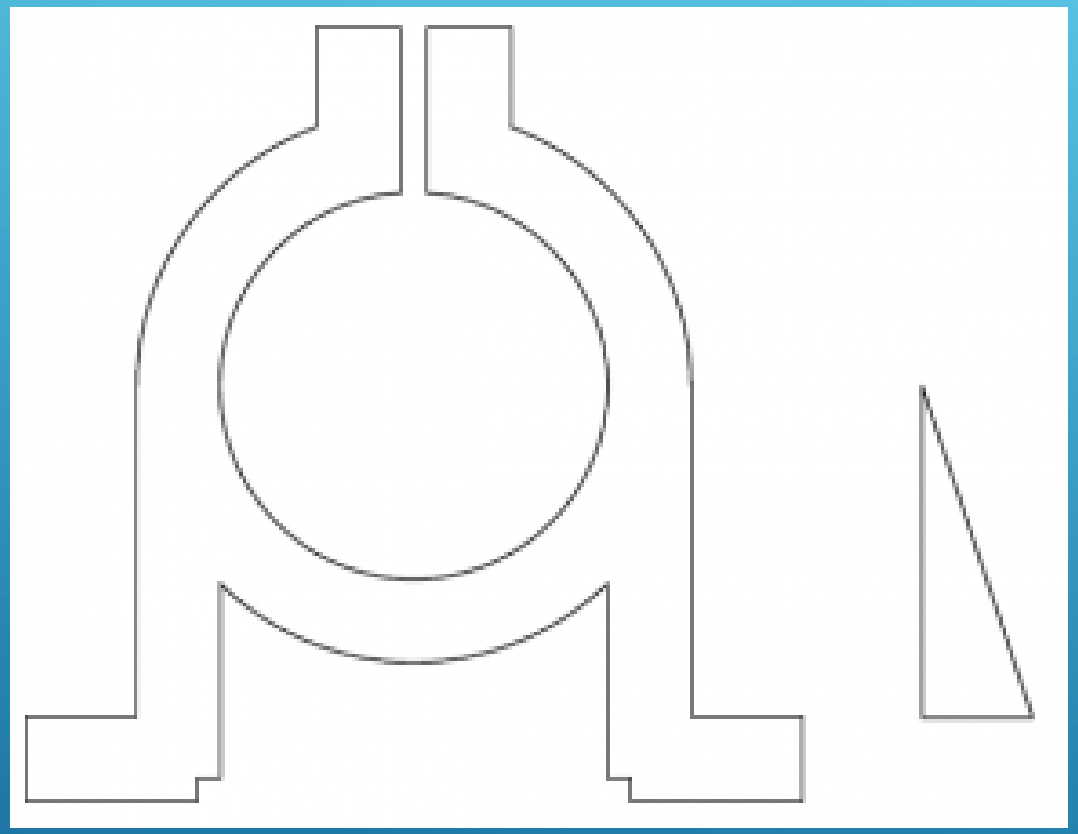
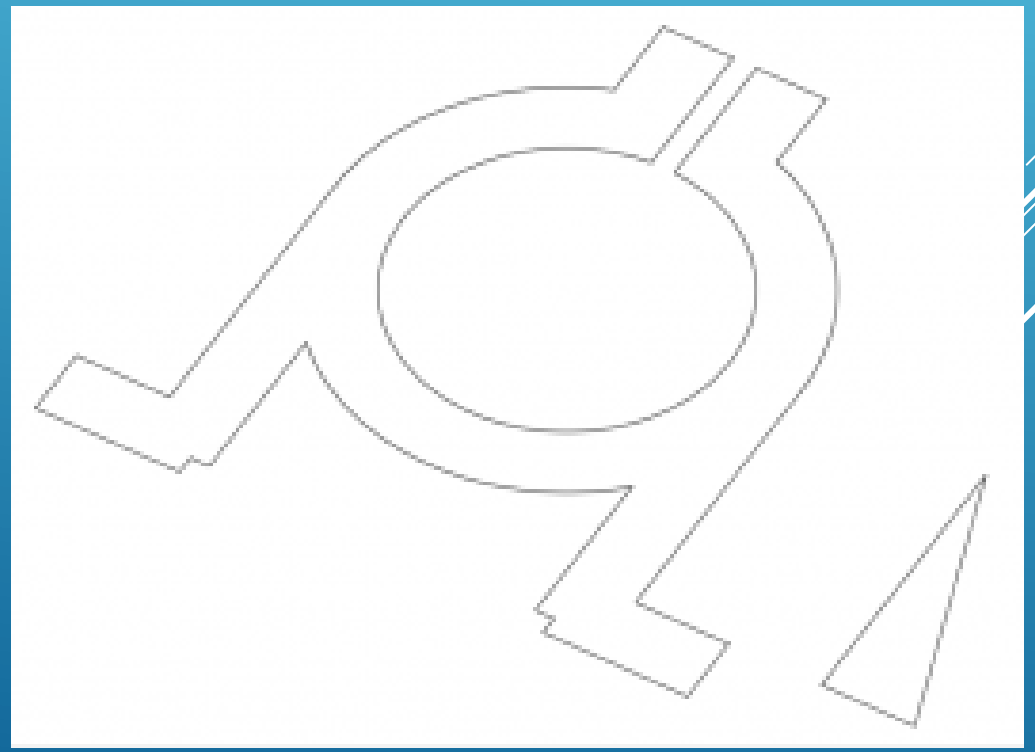


18.Desenhar o triângulo lateral, que tem o lado vertical com 24 unidades de comprimento e o lado horizontal com 8 unidades de comprimento.
19.Utilize o comando Region e forme 2 regiões.
20.Utilize o comando Free Orbit até obter uma vista semelhante à indicada na figura.



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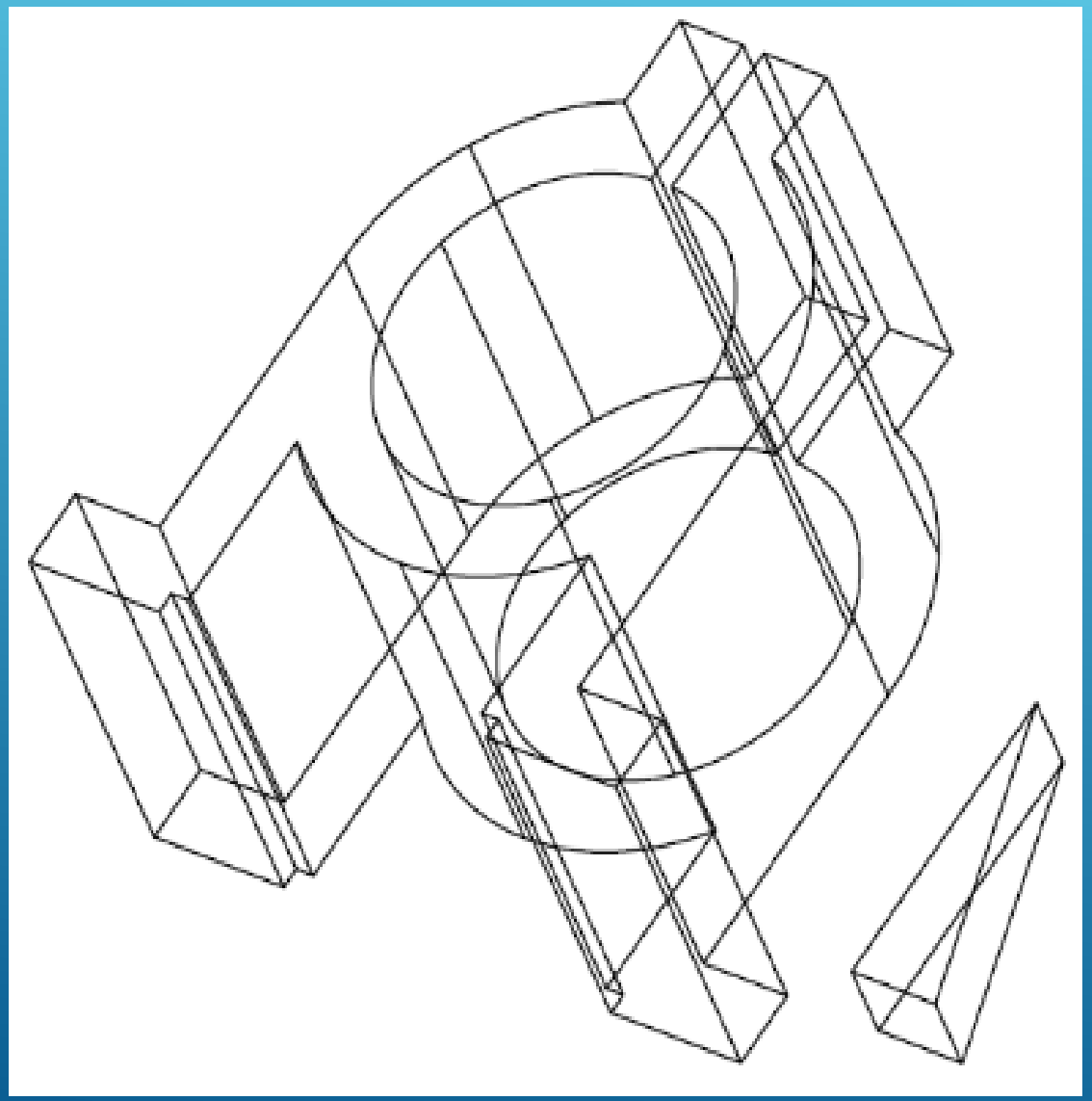
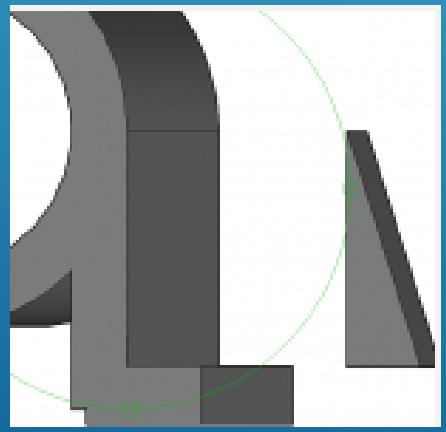


21.Utilize o comando Extrude com 28 unidades para o corpo principal e 6 unidades para o triângulo. Vizualize o modelo com o Visual Style Shades of Gray.
22.Na vista Top, utilize o comando Mirror para copiar o triângulo para o lado esquerdo do corpo principal e volte à vista anterior.
23.Utilize o comando 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 en Enter.



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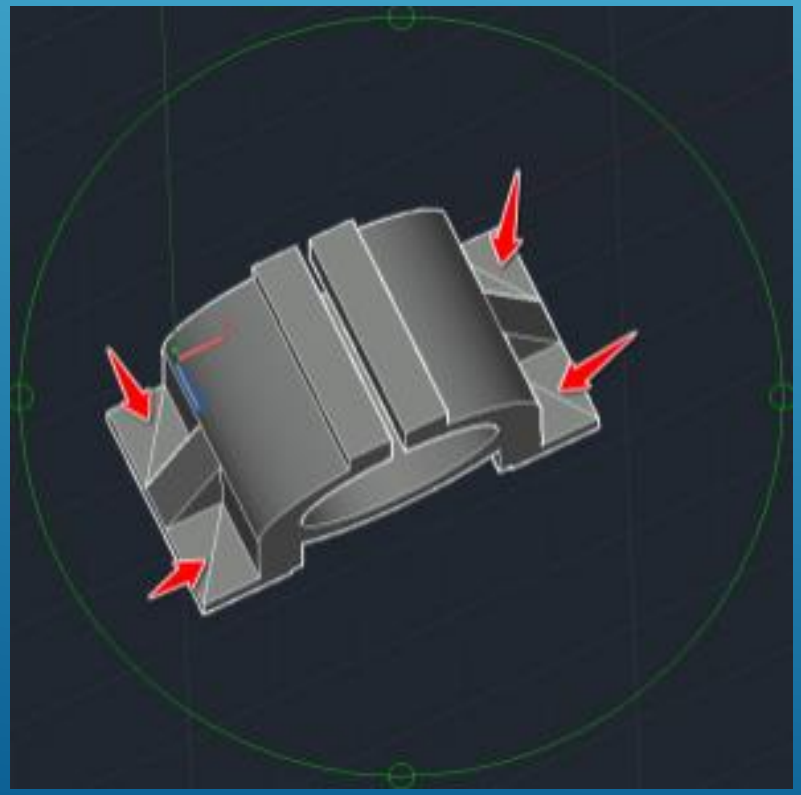
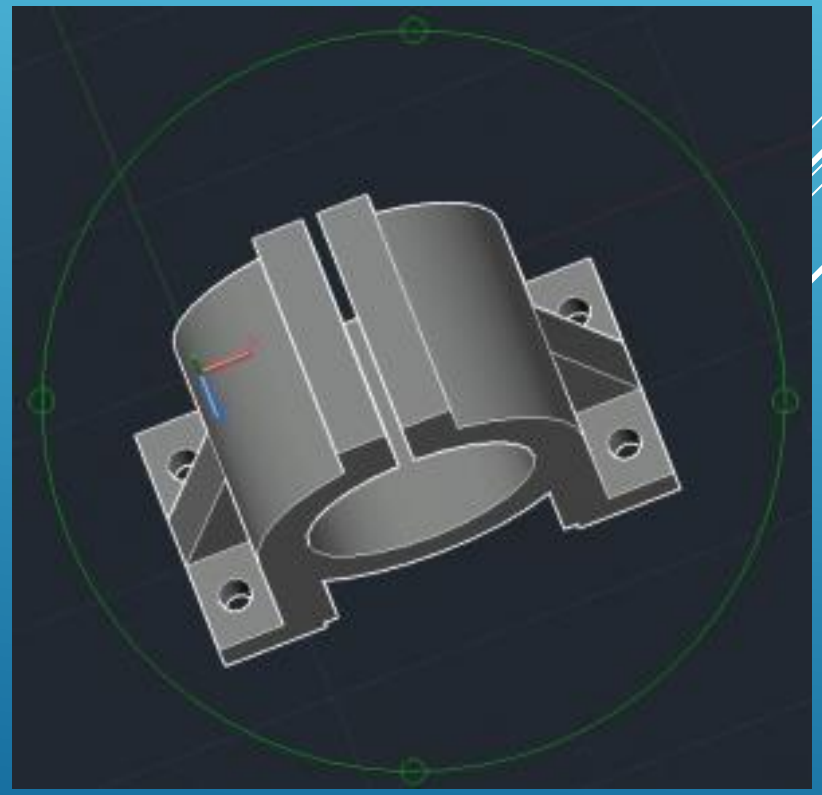
24.Visualize o modelo de forma a que consiga acrescentar as linhas diagonais indicadas na figura.
25.Crie circunferências com snap Midpoint em cada uma das linhas com raio 2 unidades
26. Efectue o Extrude dessas circunferências para baixo de tal forma que alcancem a outra superfície da peça.
27.Efectue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





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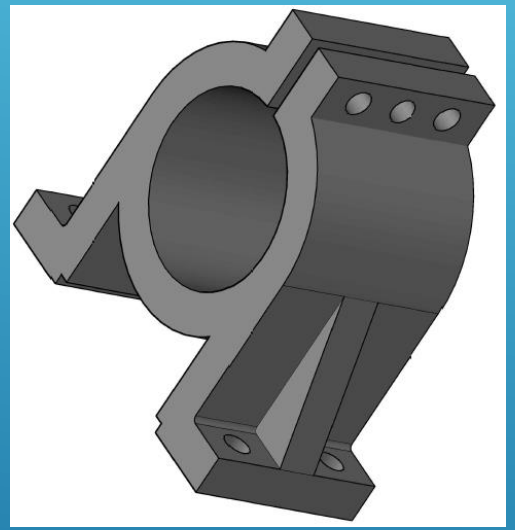
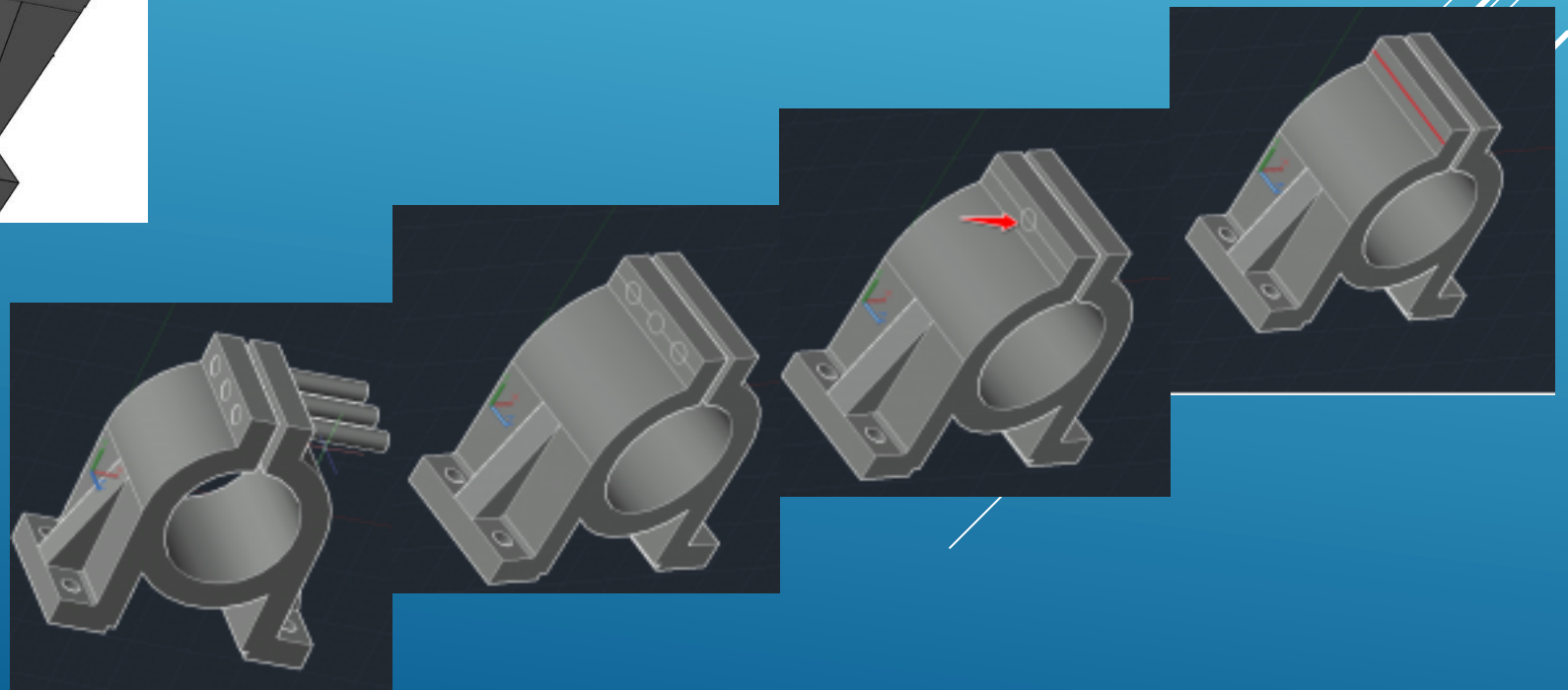
29.Utilize o comando Fillet para arredondar os vertices do modelo com R=0.8 unidades.



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 auxiliaries e efectue o Extrude das circunferências.
34.Subtraia os cilindros obtidos do modelo.



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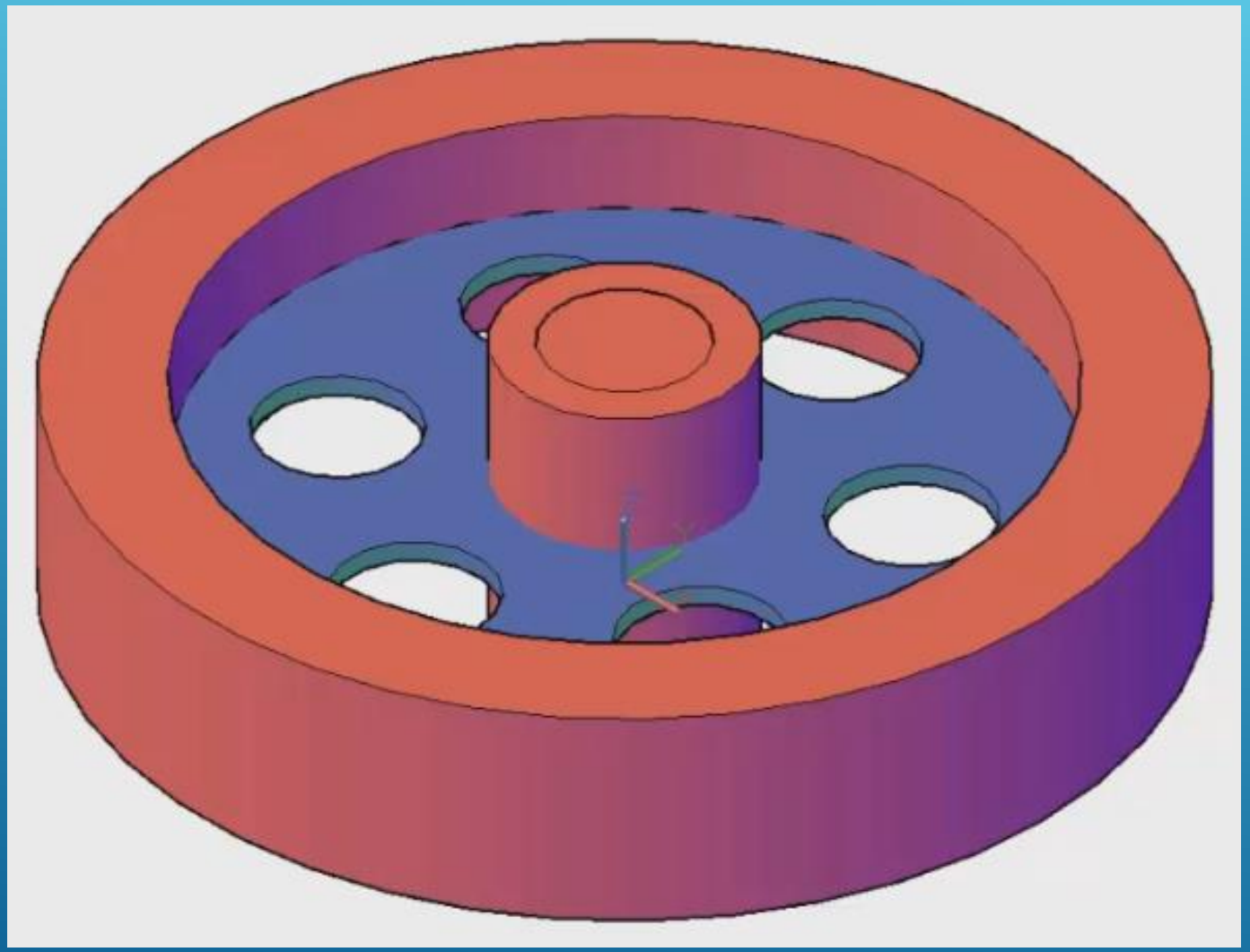

1. New Drawing
2. Criar 2 layers: Encarnado e Azul
3. Ambiente de trabalho: 3D Modeling
4. 3dWireframe
5. Solid Tab > Cylinder > Center point: **(0,0,0)** + enter
   Radius: **430** + enter
   Height: **180** + enter

1

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

2

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

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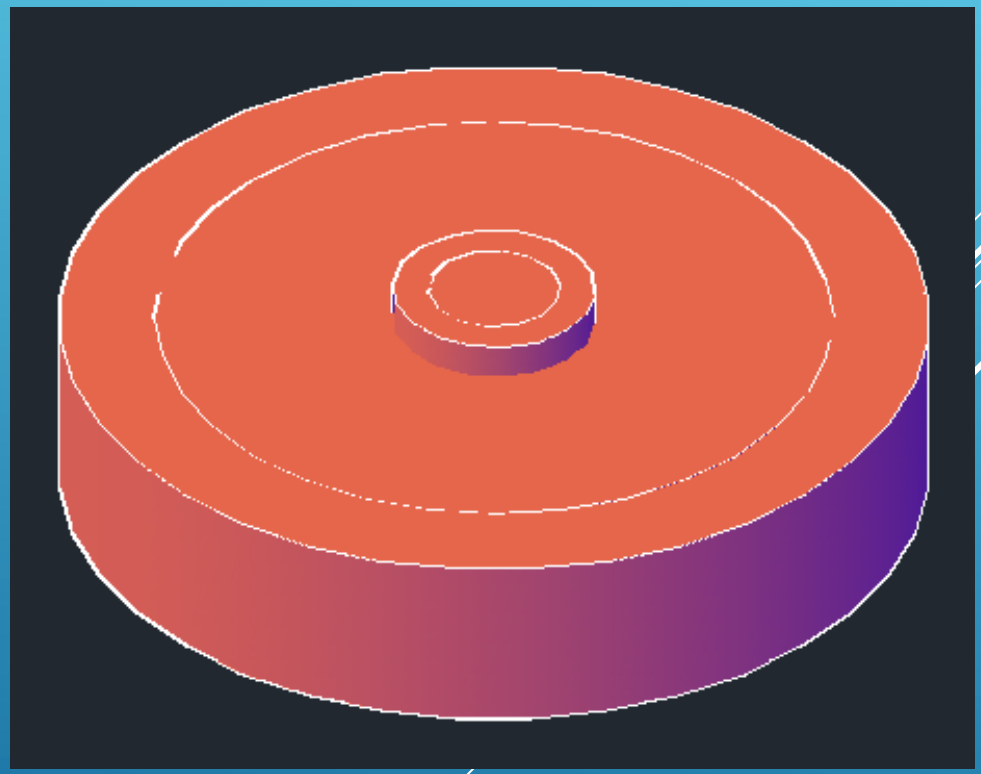
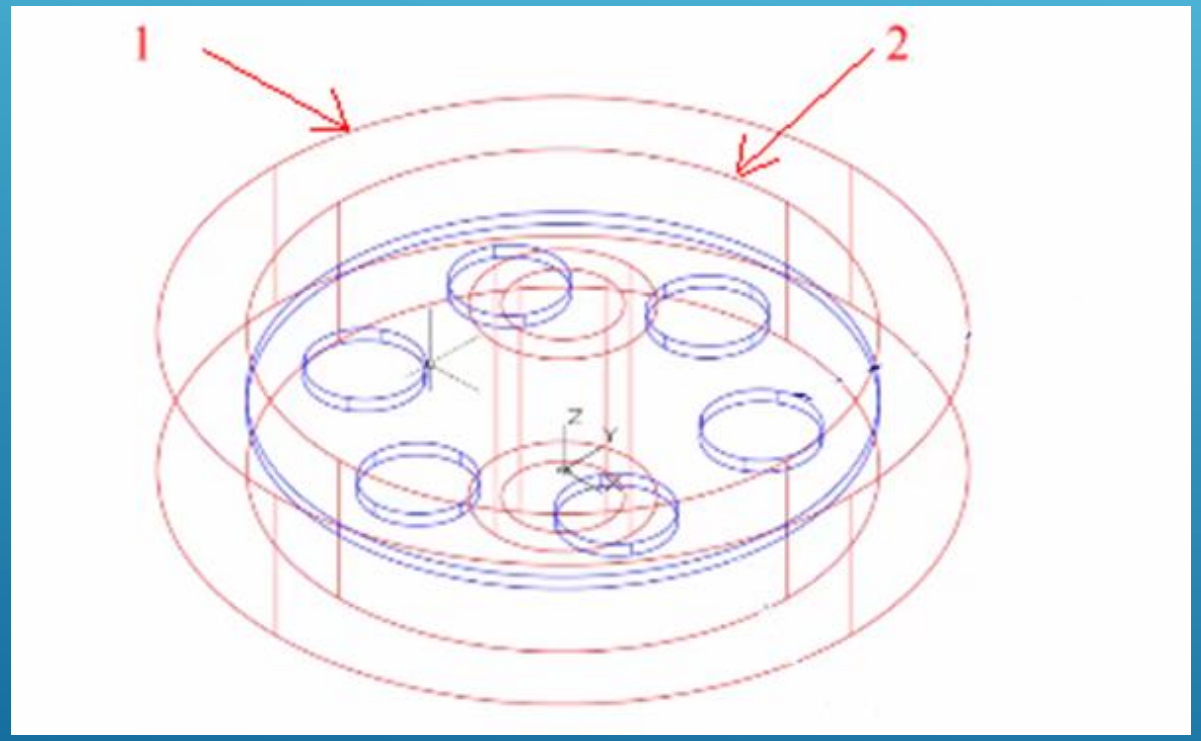
11. Home > Layers > Azul
12. *Solid Tab > Cylinder > Center point:* ***(0,0,82)*** *+ enter*
    Radius: ***335*** *+ enter*
    Height: ***16****+ enter*
13. *Solid Tab > Cylinder > Center point:* ***(0,-217.5,82)*** *+ enter*
    Radius: ***65*** *+ enter*
    Height: ***16*** *+ enter*
14. *Home > Modify > Polar Array*
    (0,0,82)
    6
    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*

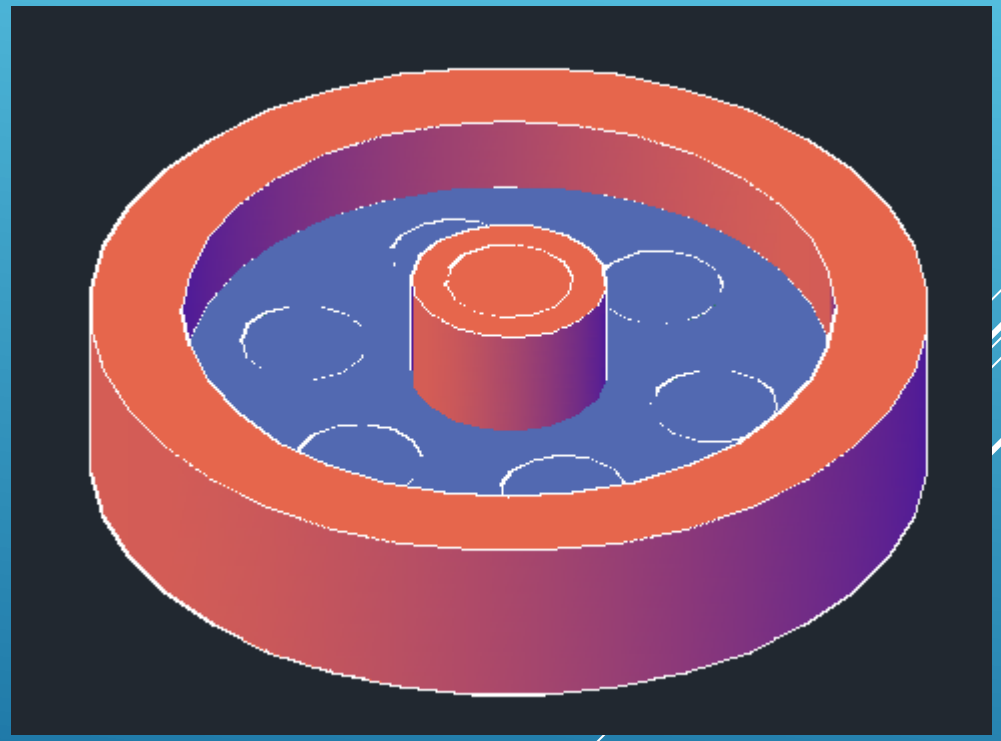
3

4



18. Solid Tab > Boolean > Subtract





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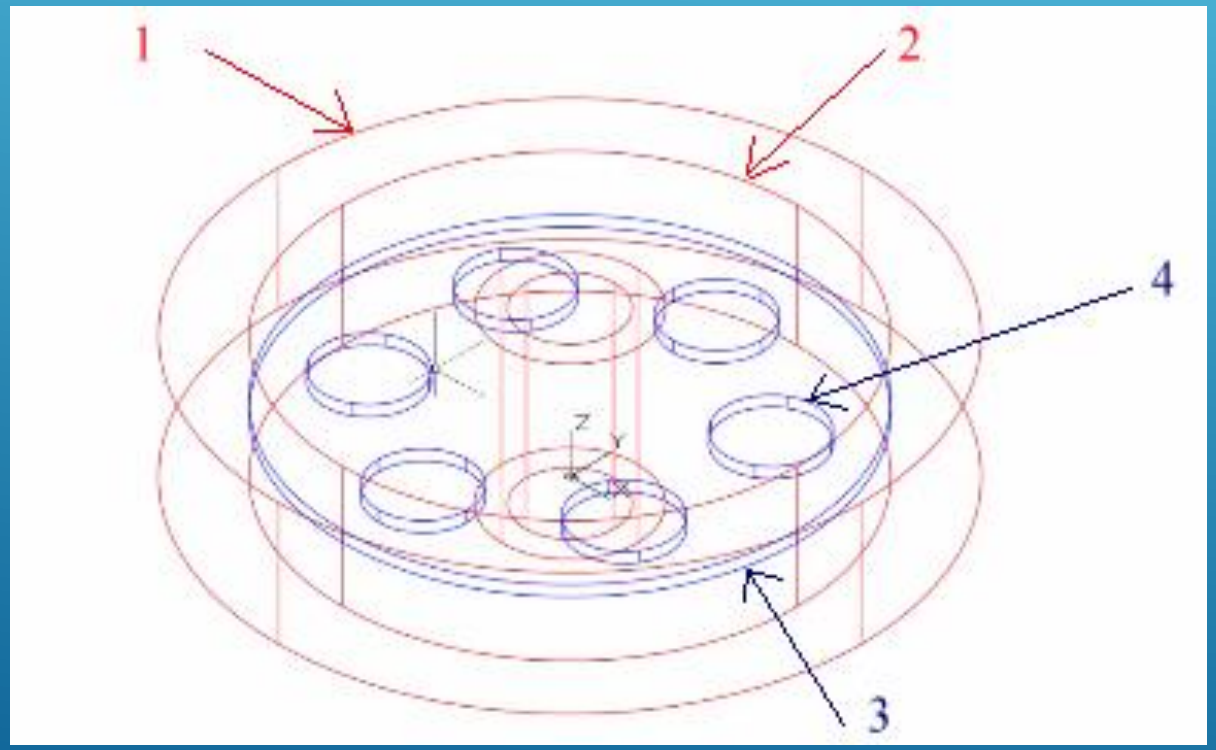
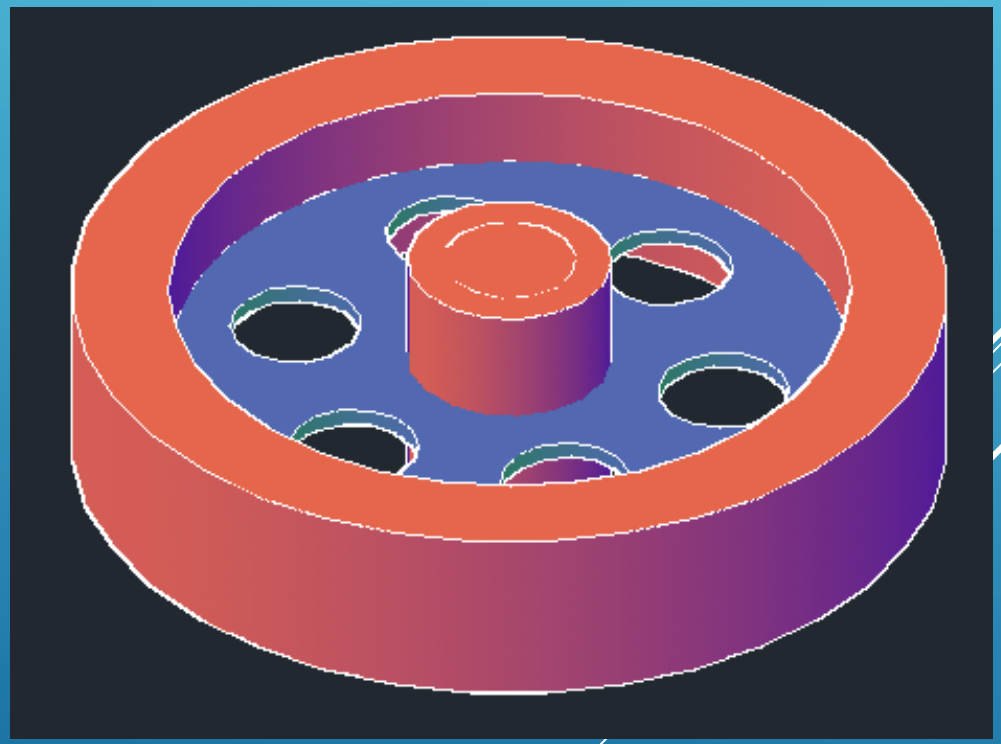
19. Solid Tab > Boolean > Subtract





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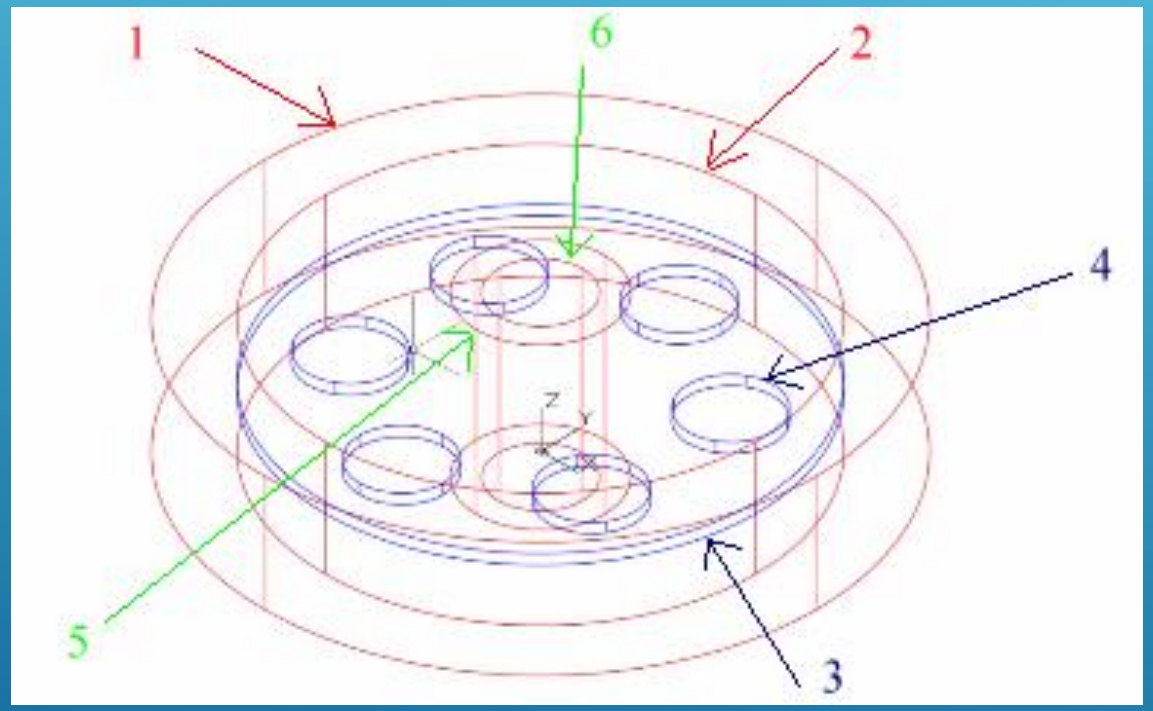
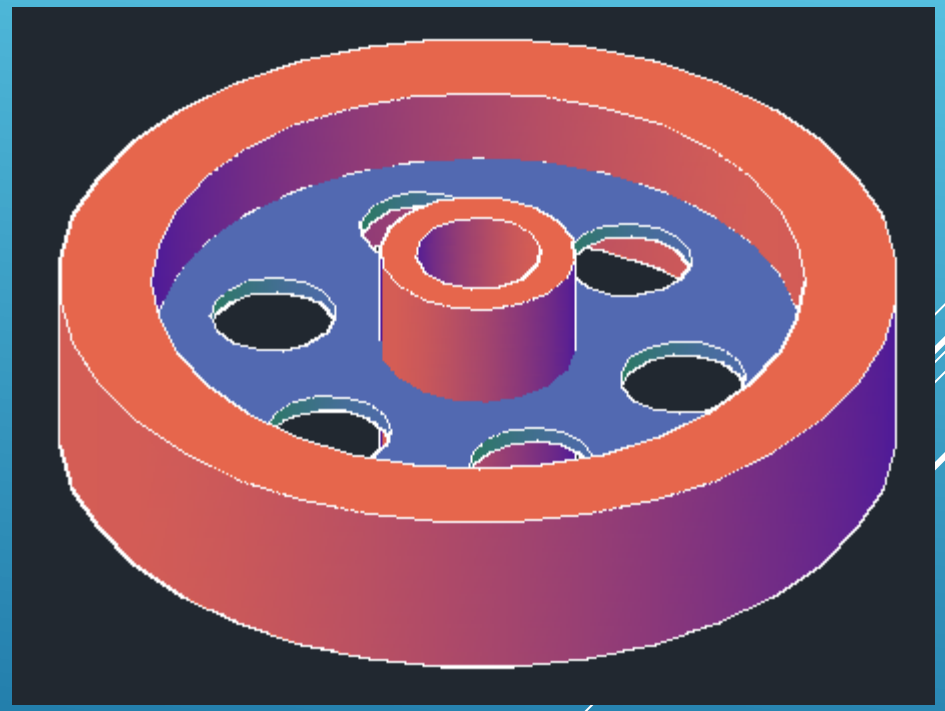
20. Solid Tab > Boolean > Subtract





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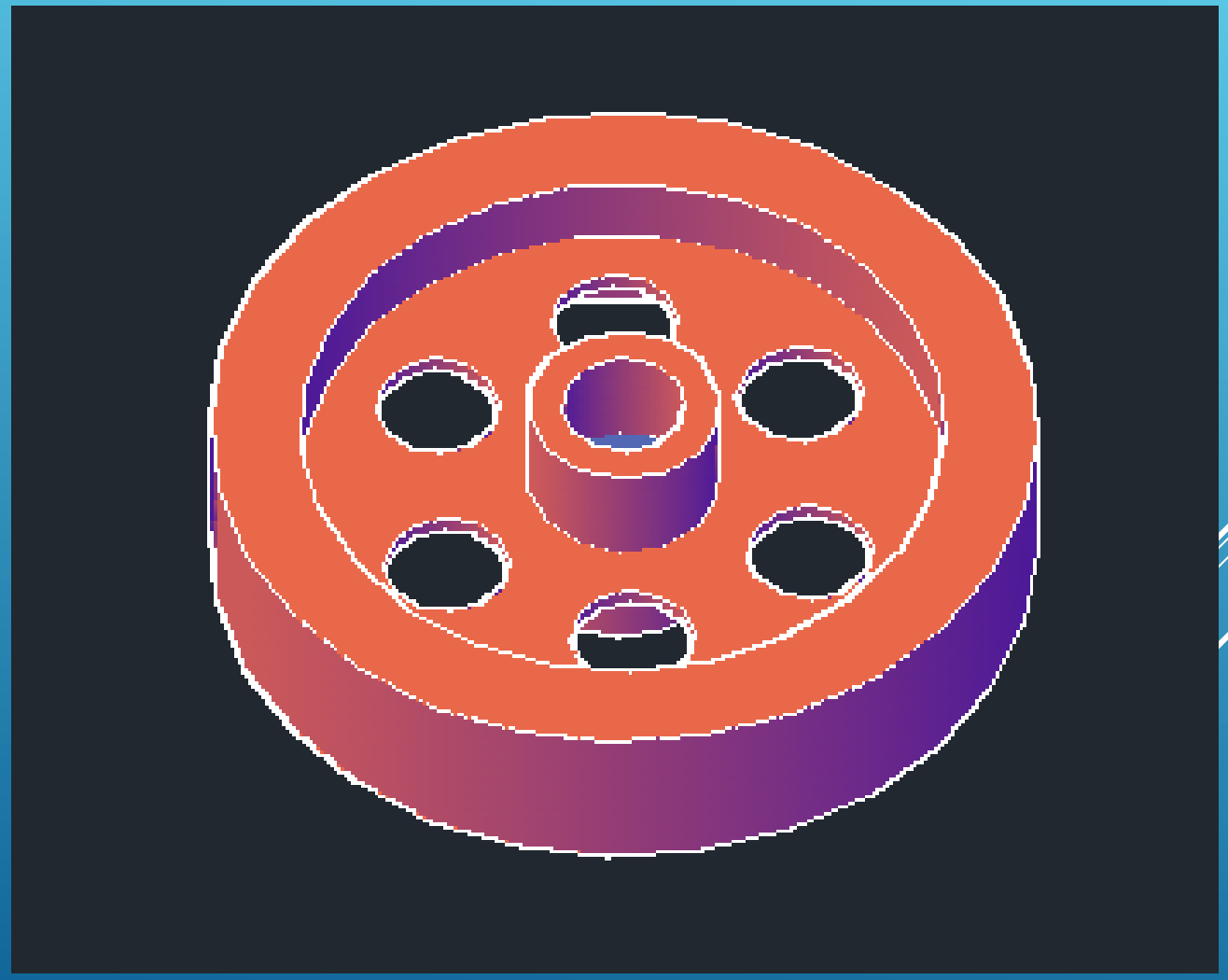
20. Solid Tab > Boolean > Union



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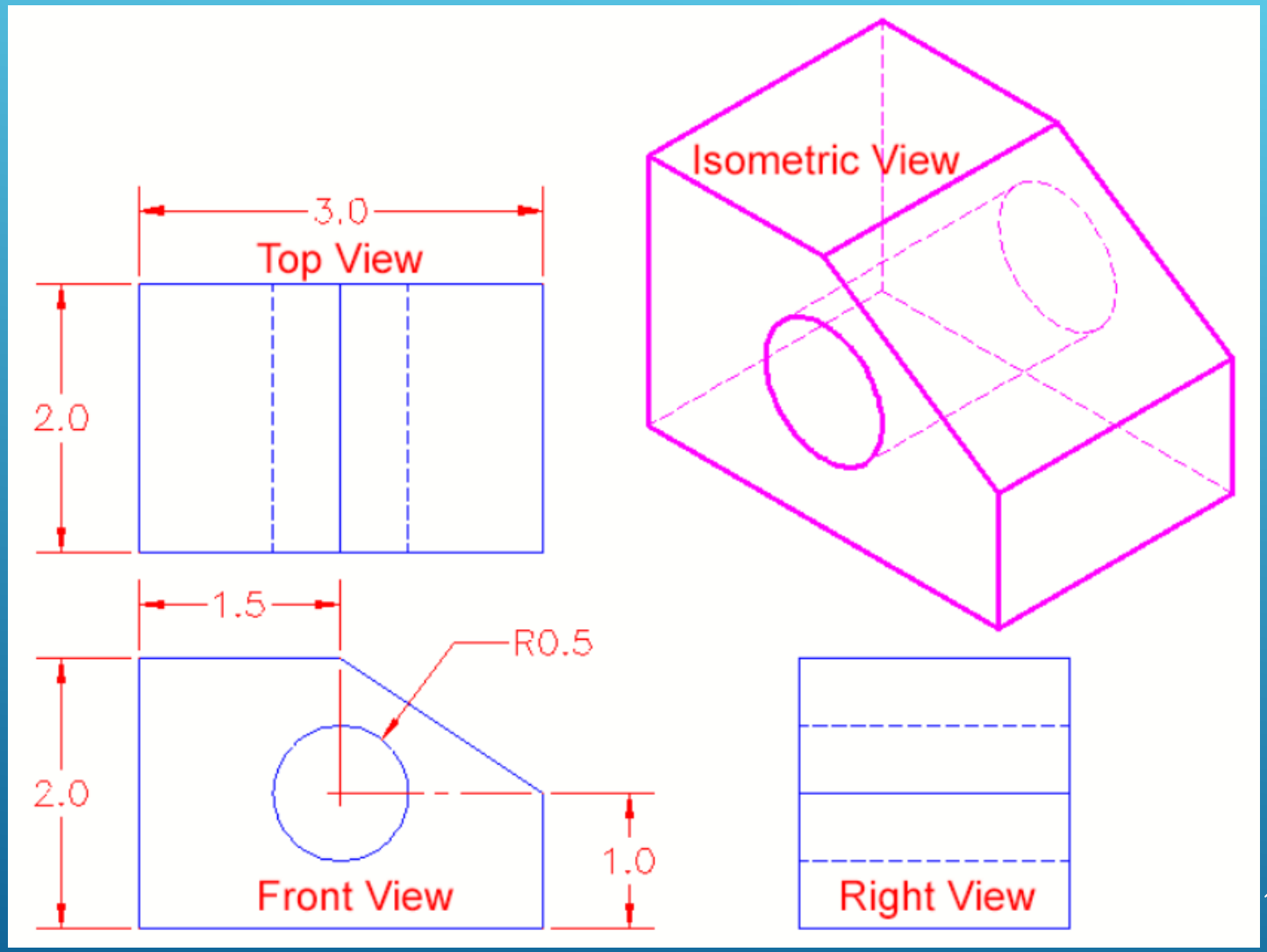

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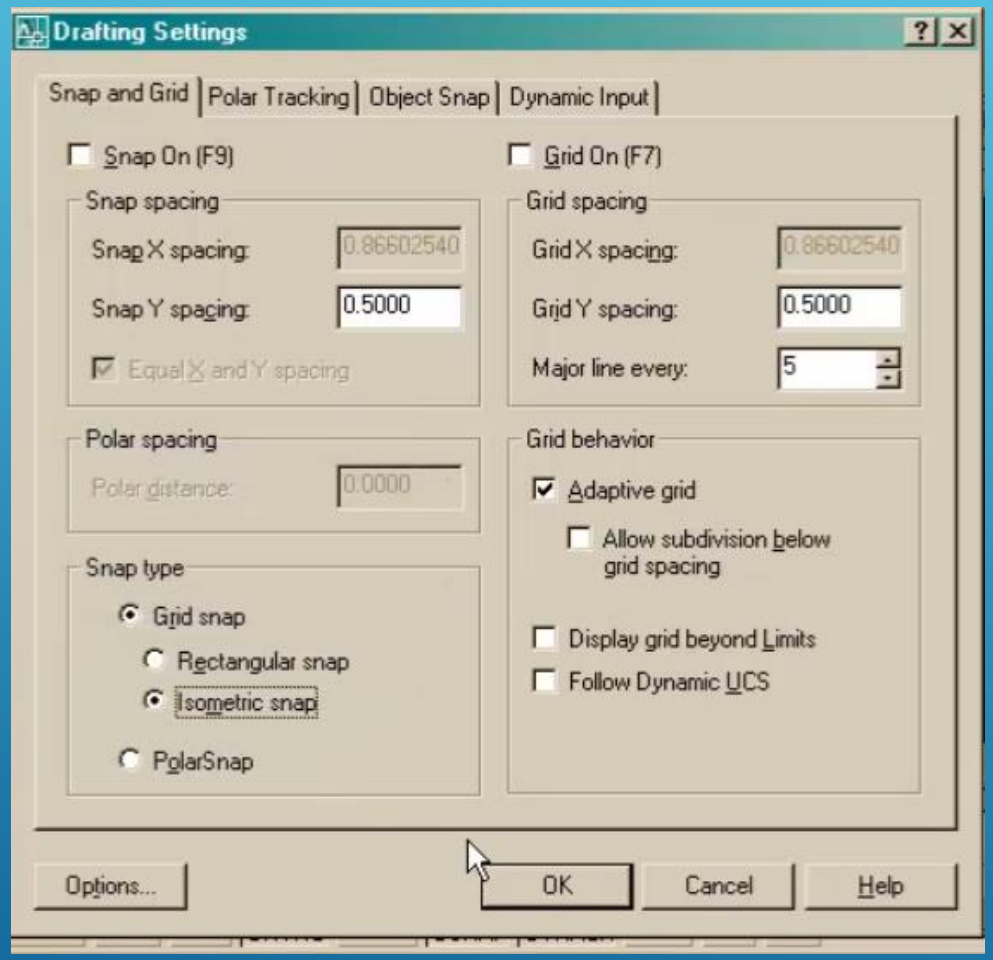

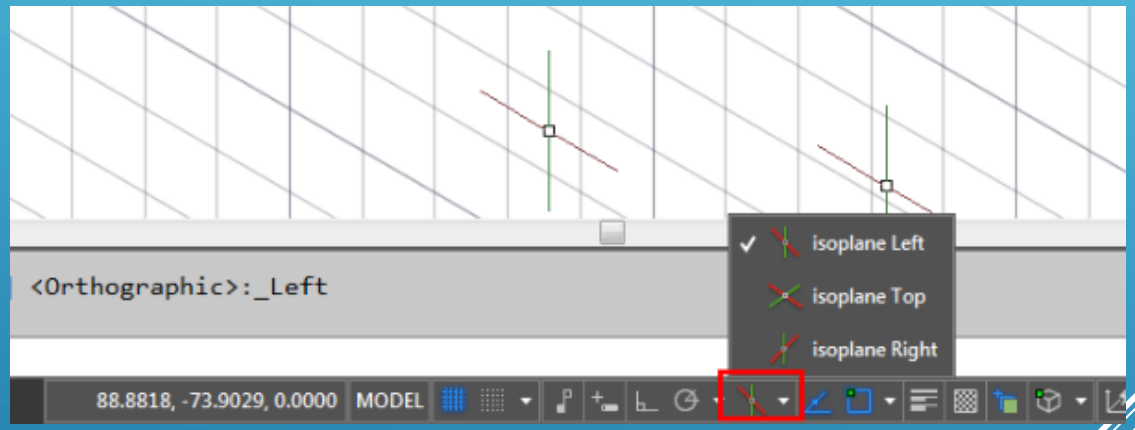

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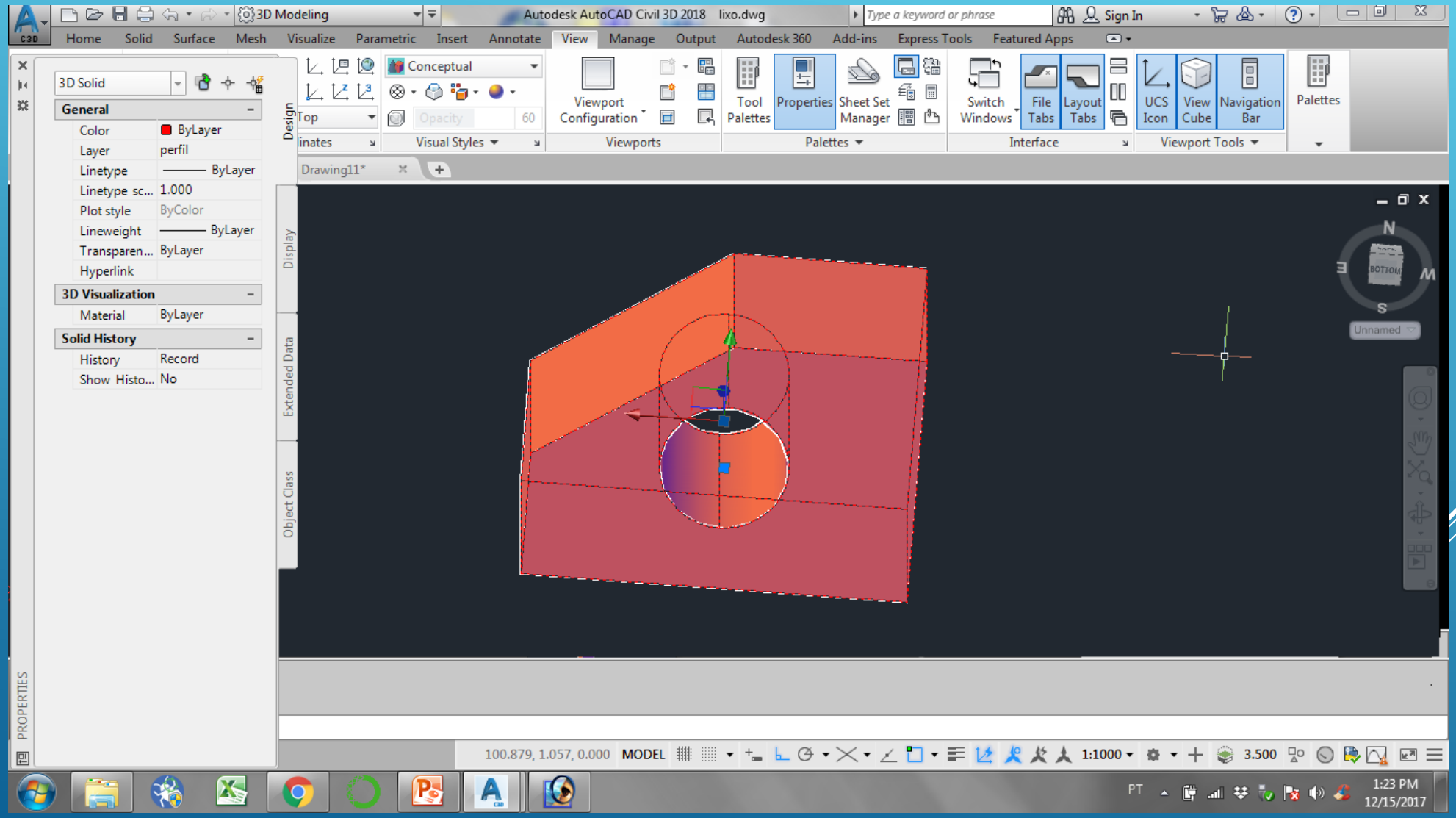

AULA 5 Desenho Técnico Assistido por Computador

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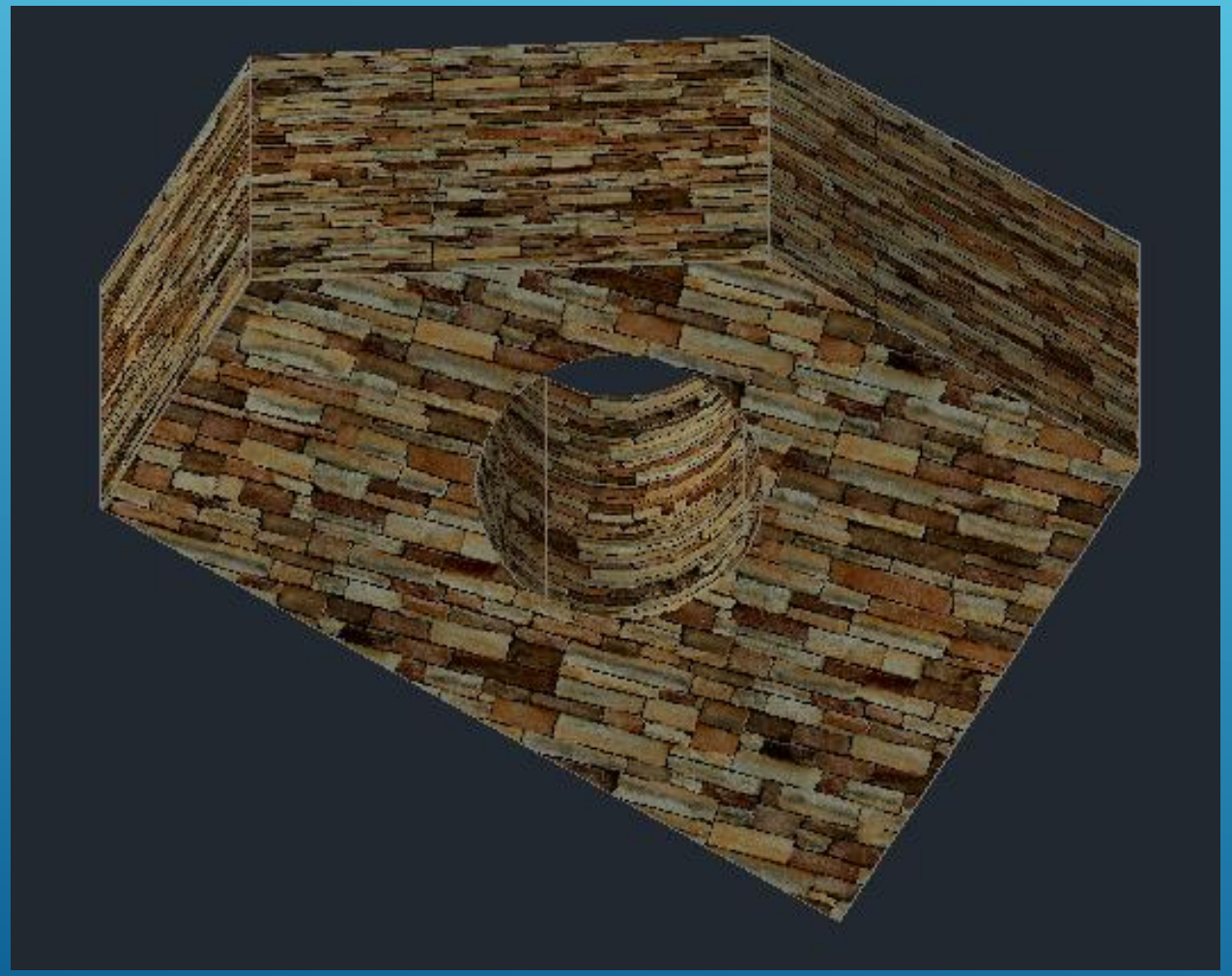

View > Realistic
RMAT (material Browser)
Seleccionar um material e arrasta-lo para o sólido

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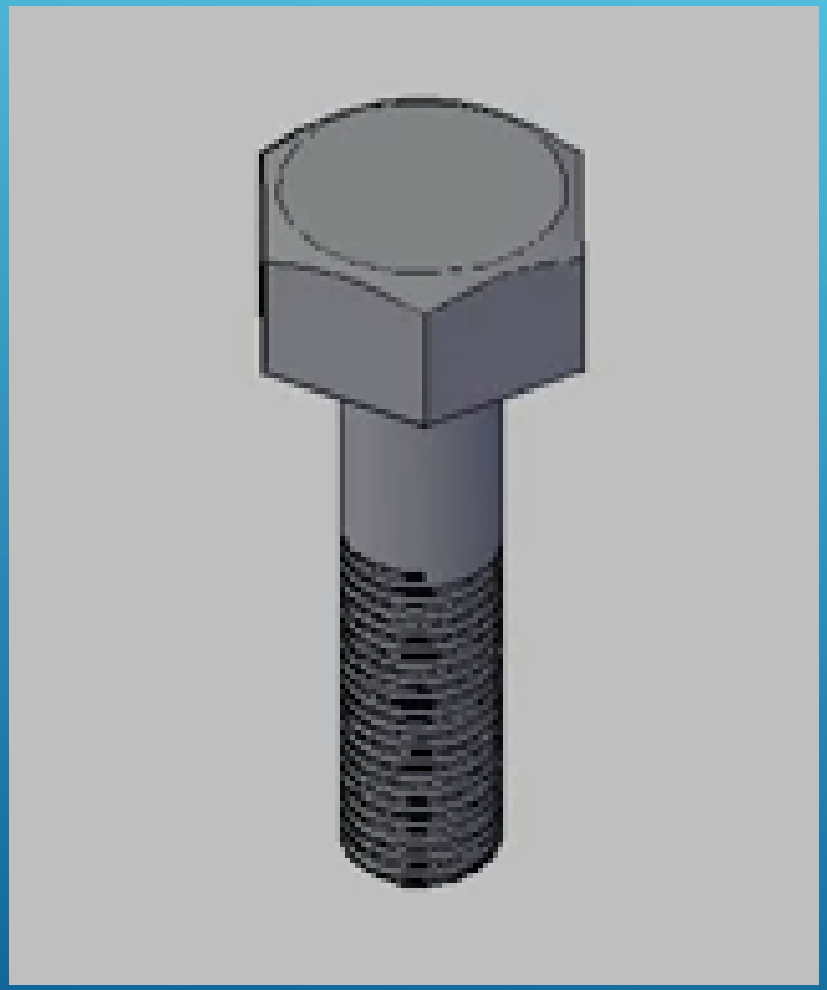

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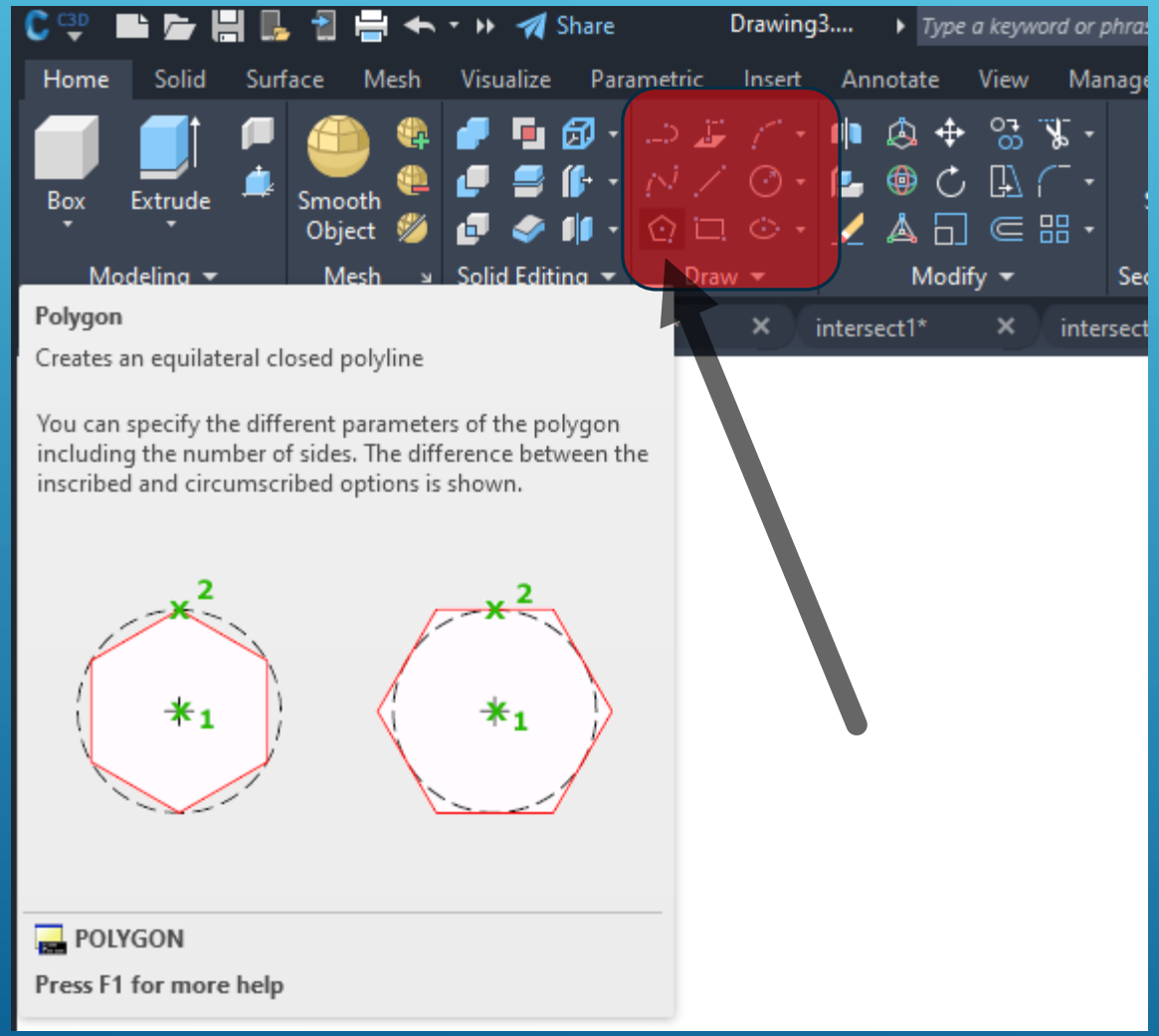


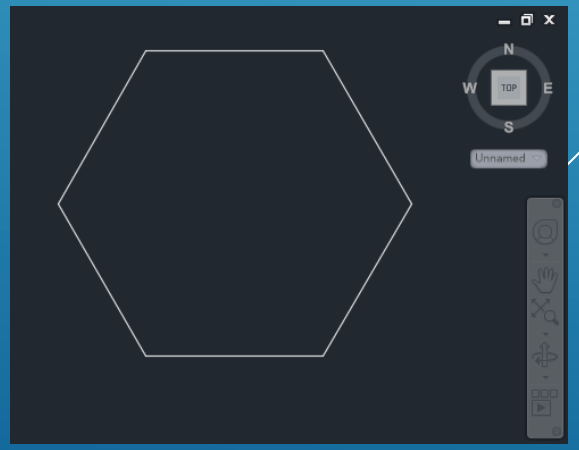

```
Command:
Command:
POLYGON _polygon Enter number of sides <6>:
```

```
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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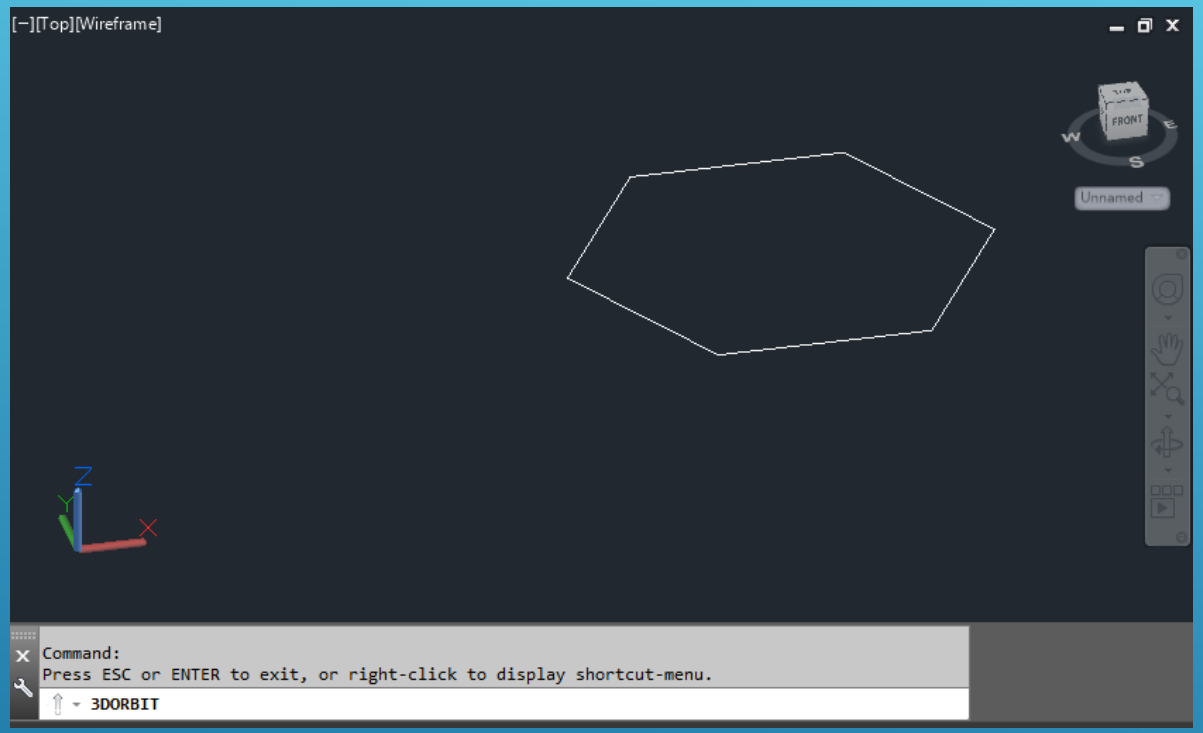

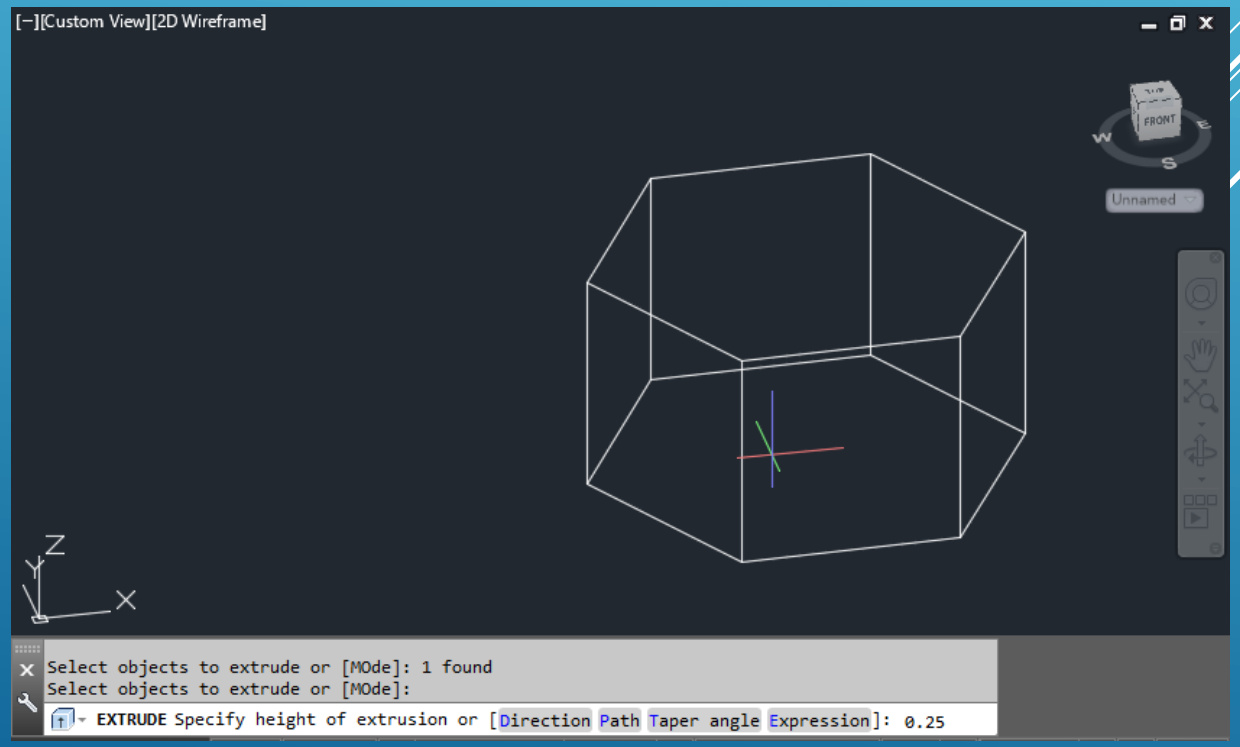

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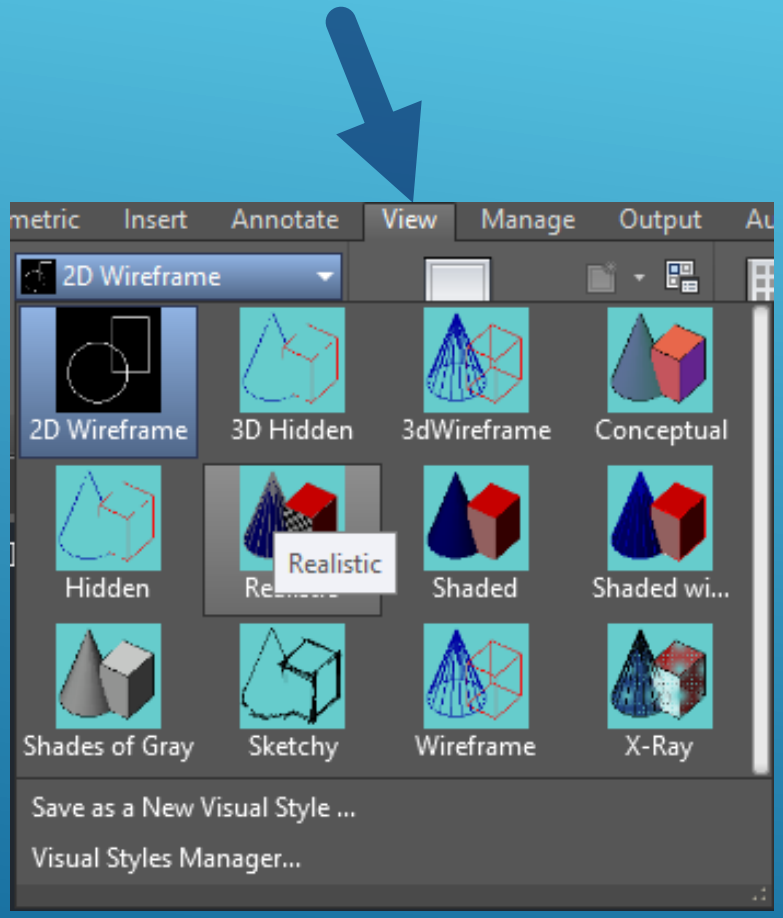

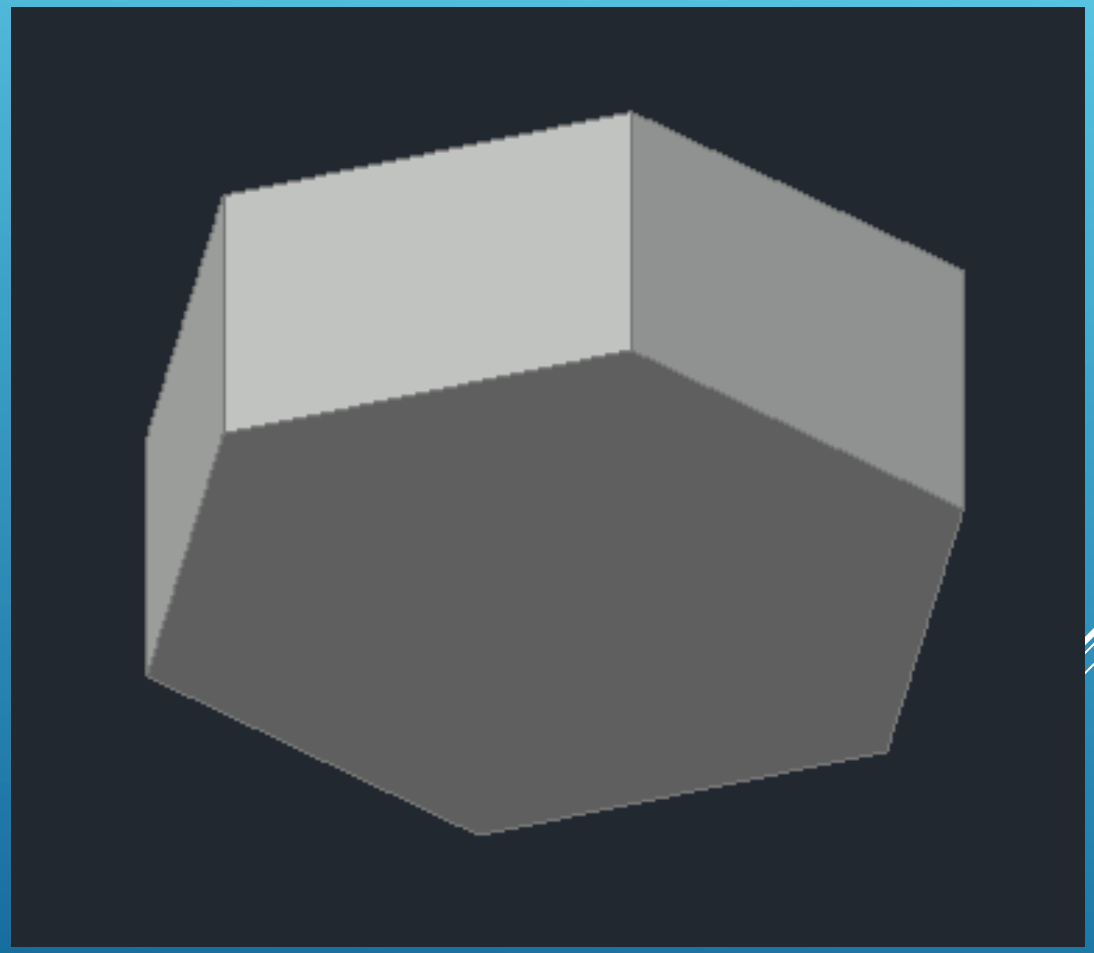

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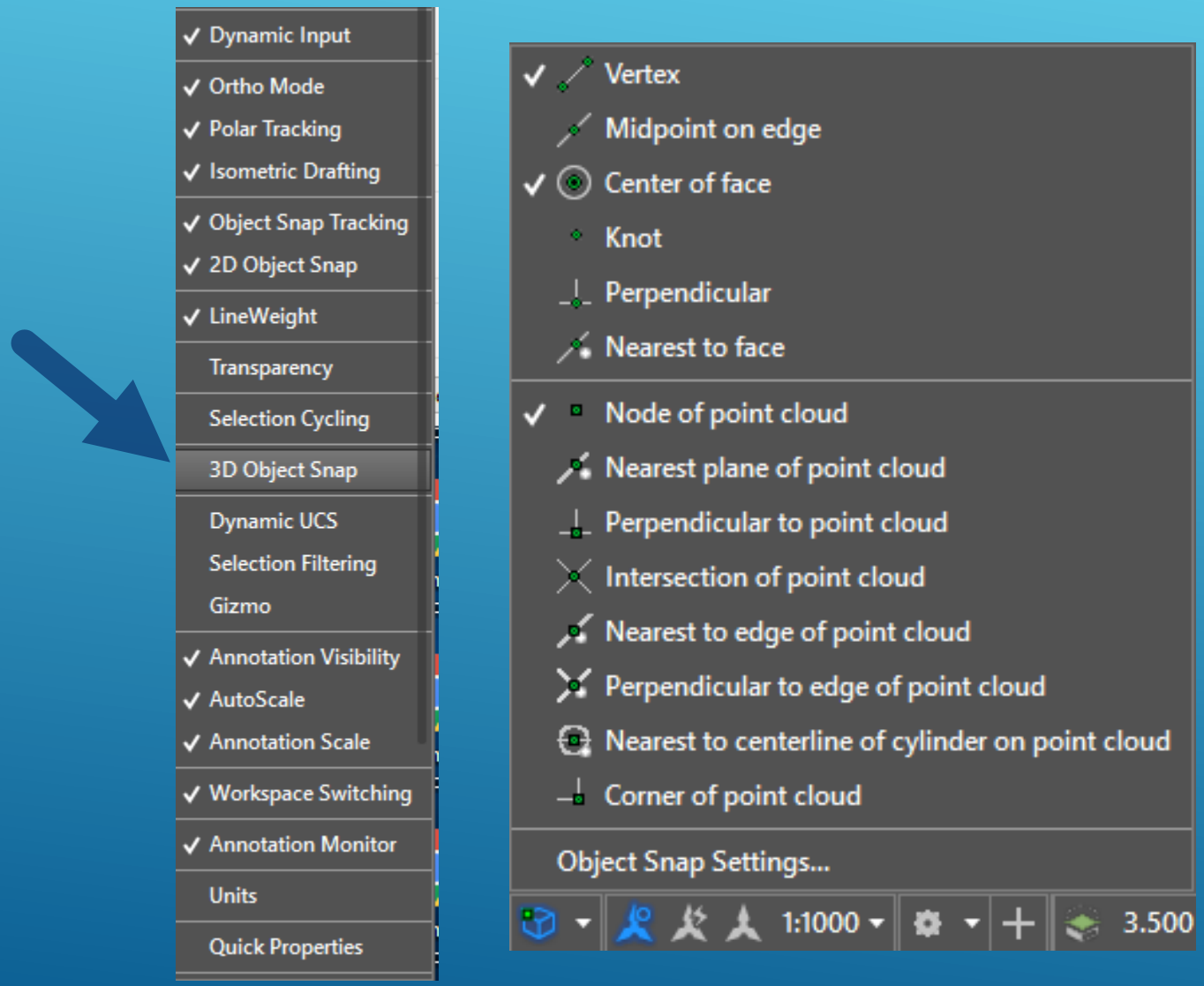

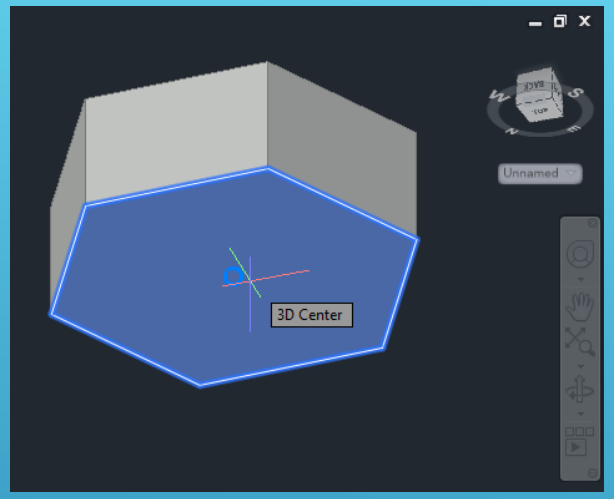

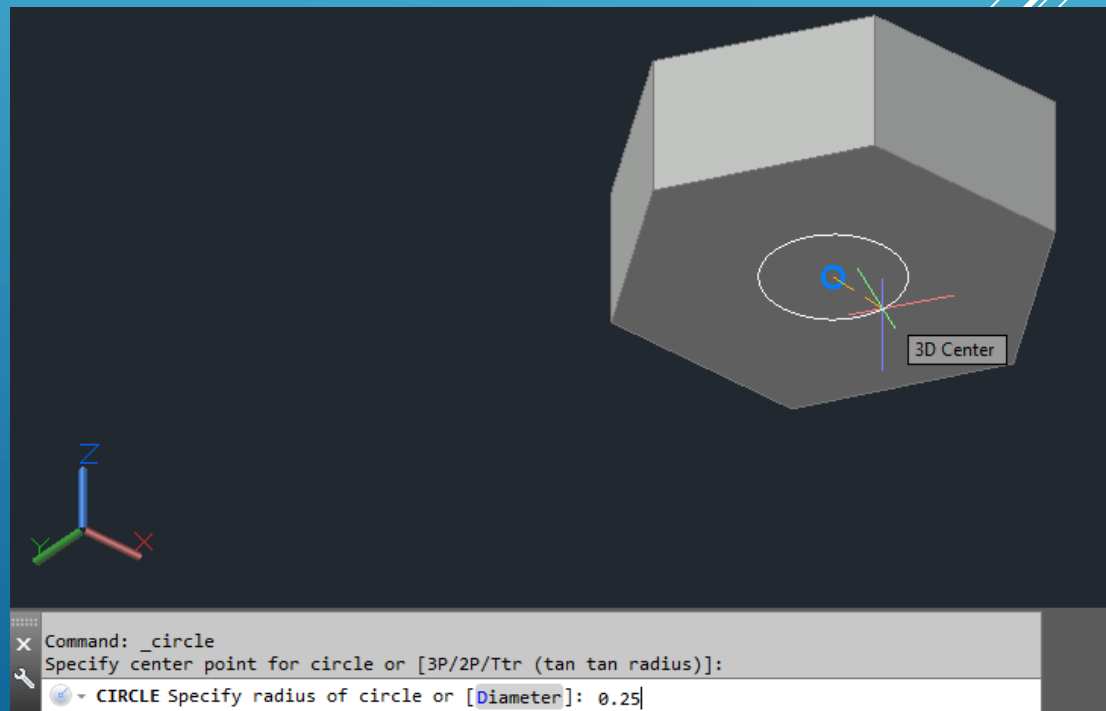

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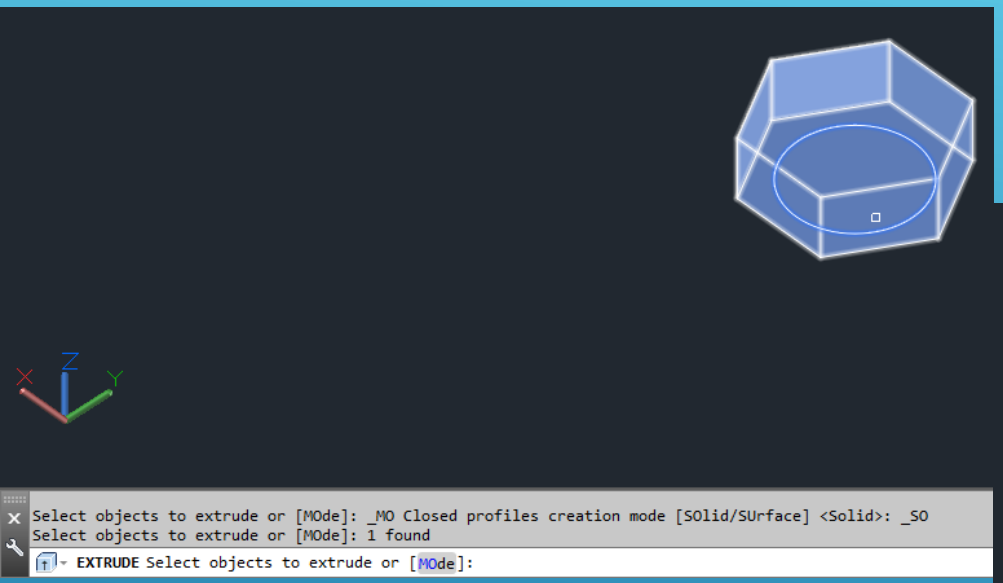

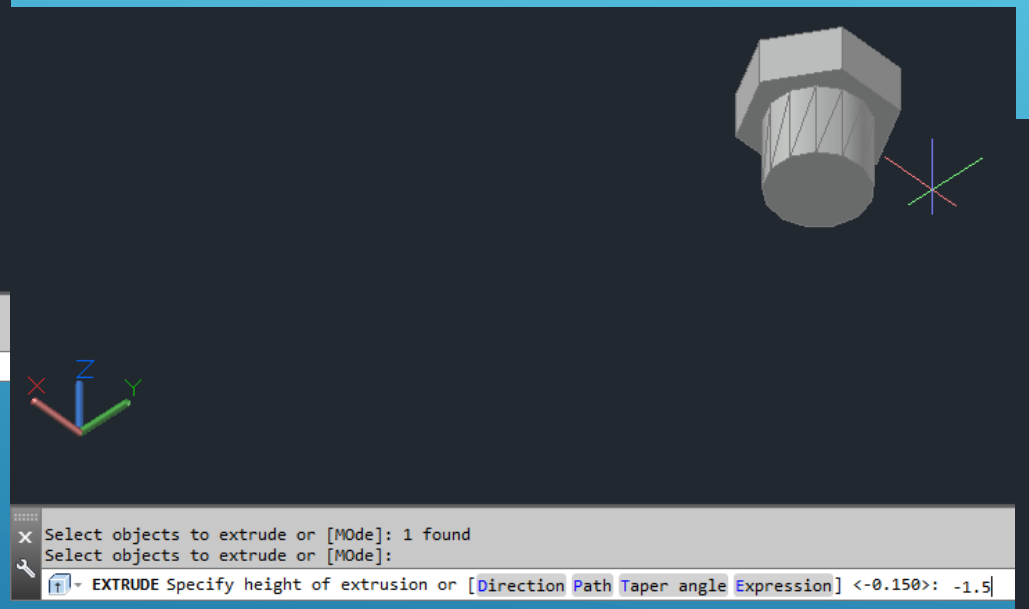

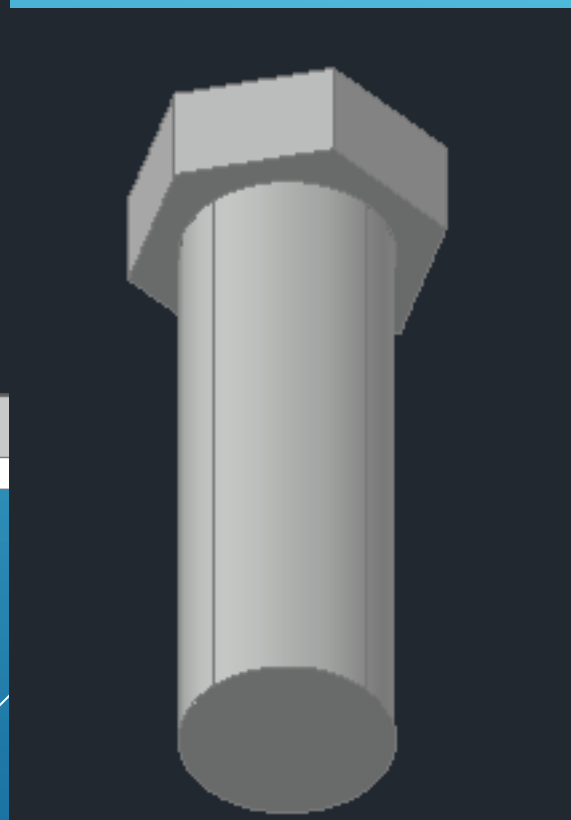

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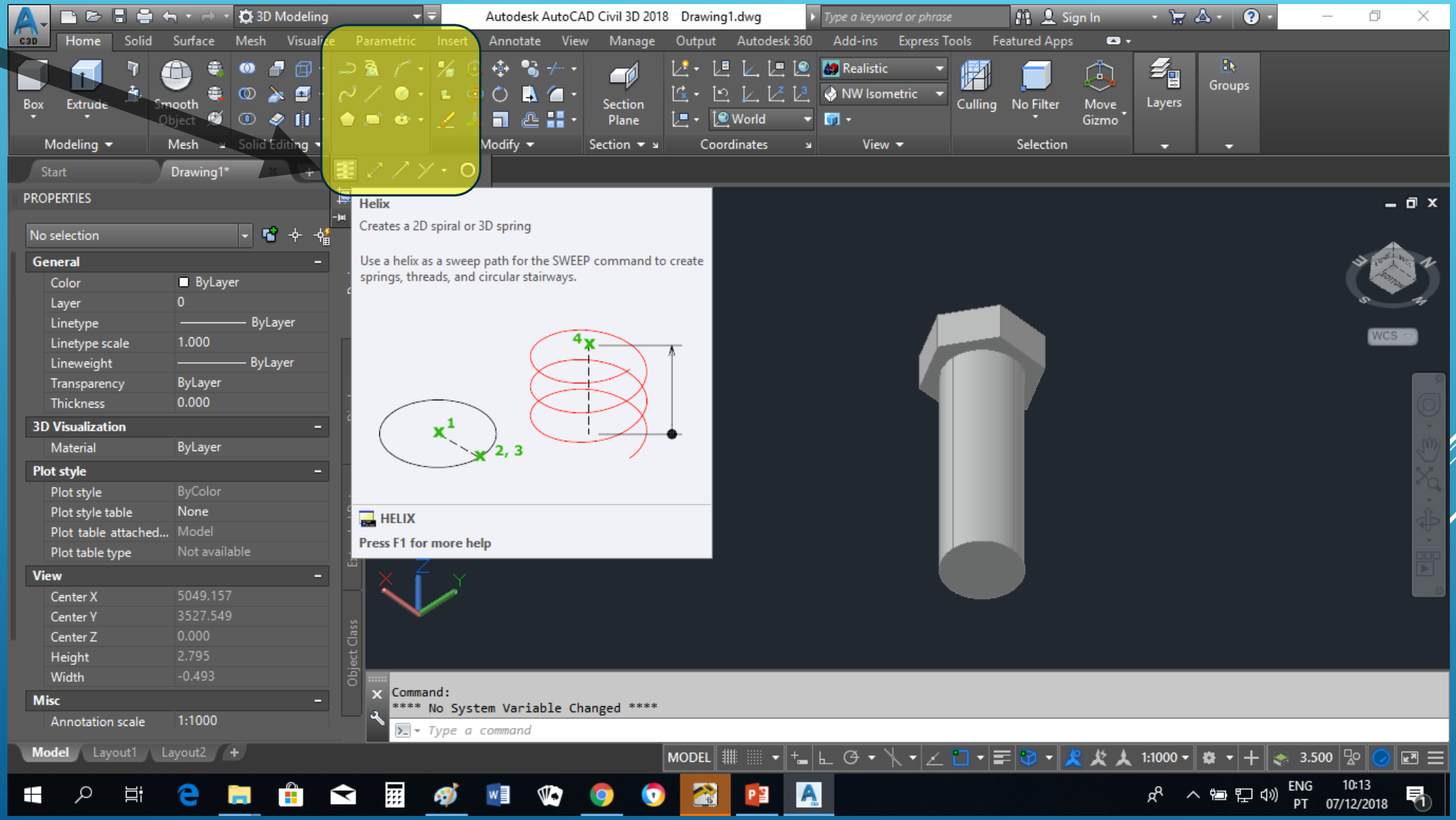

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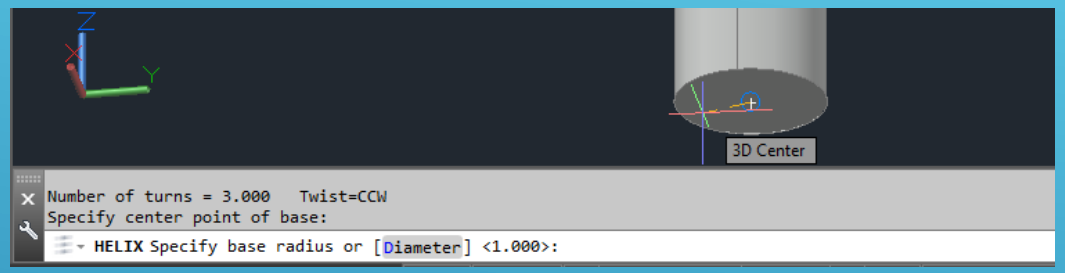

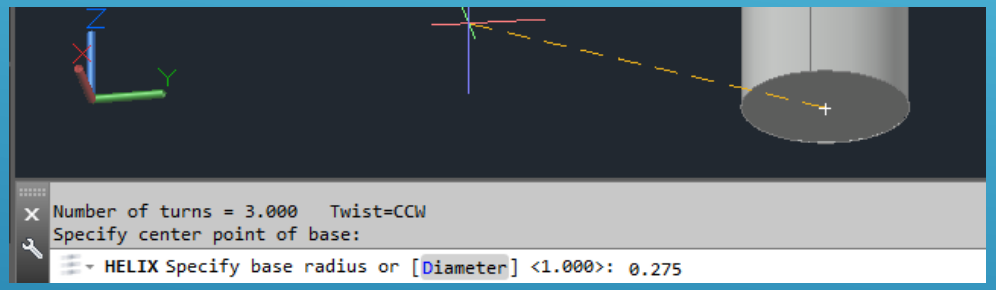

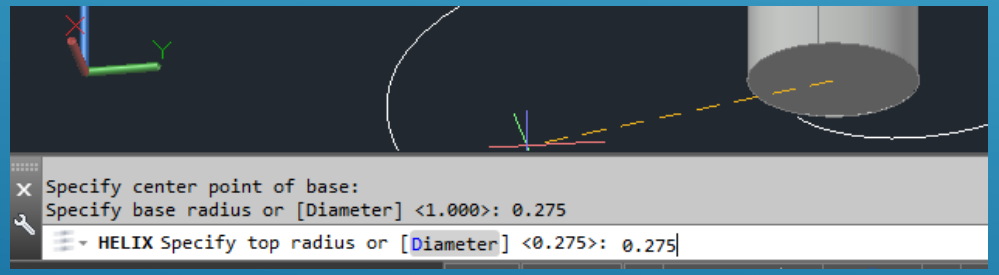

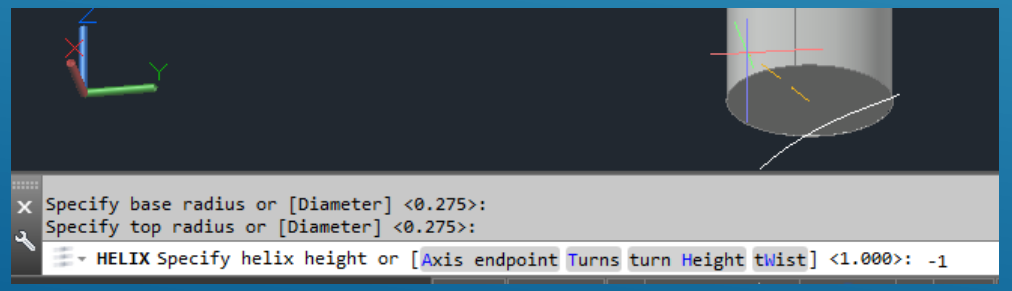

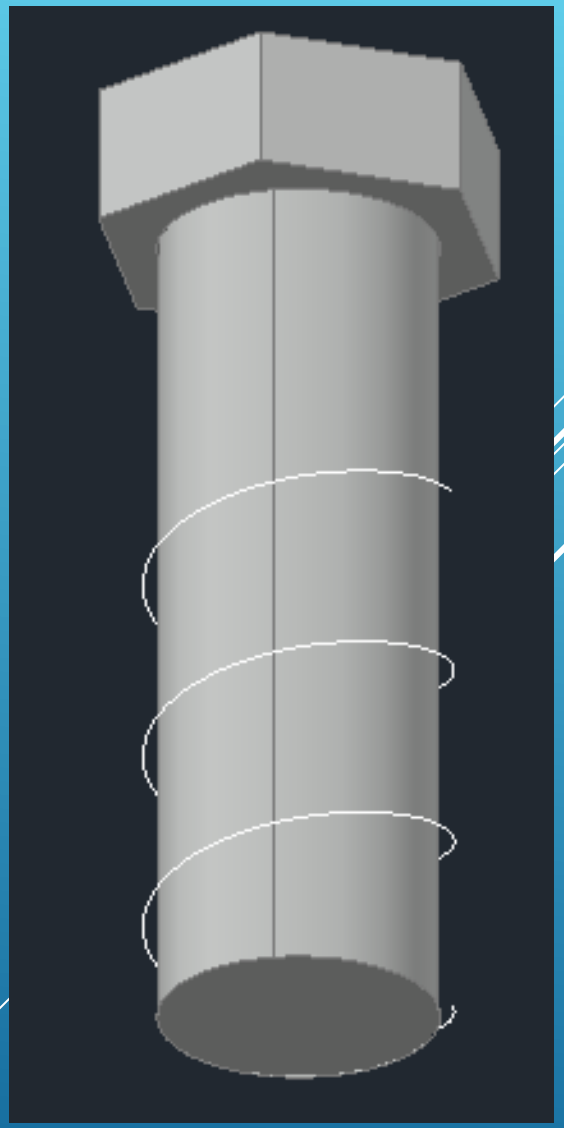

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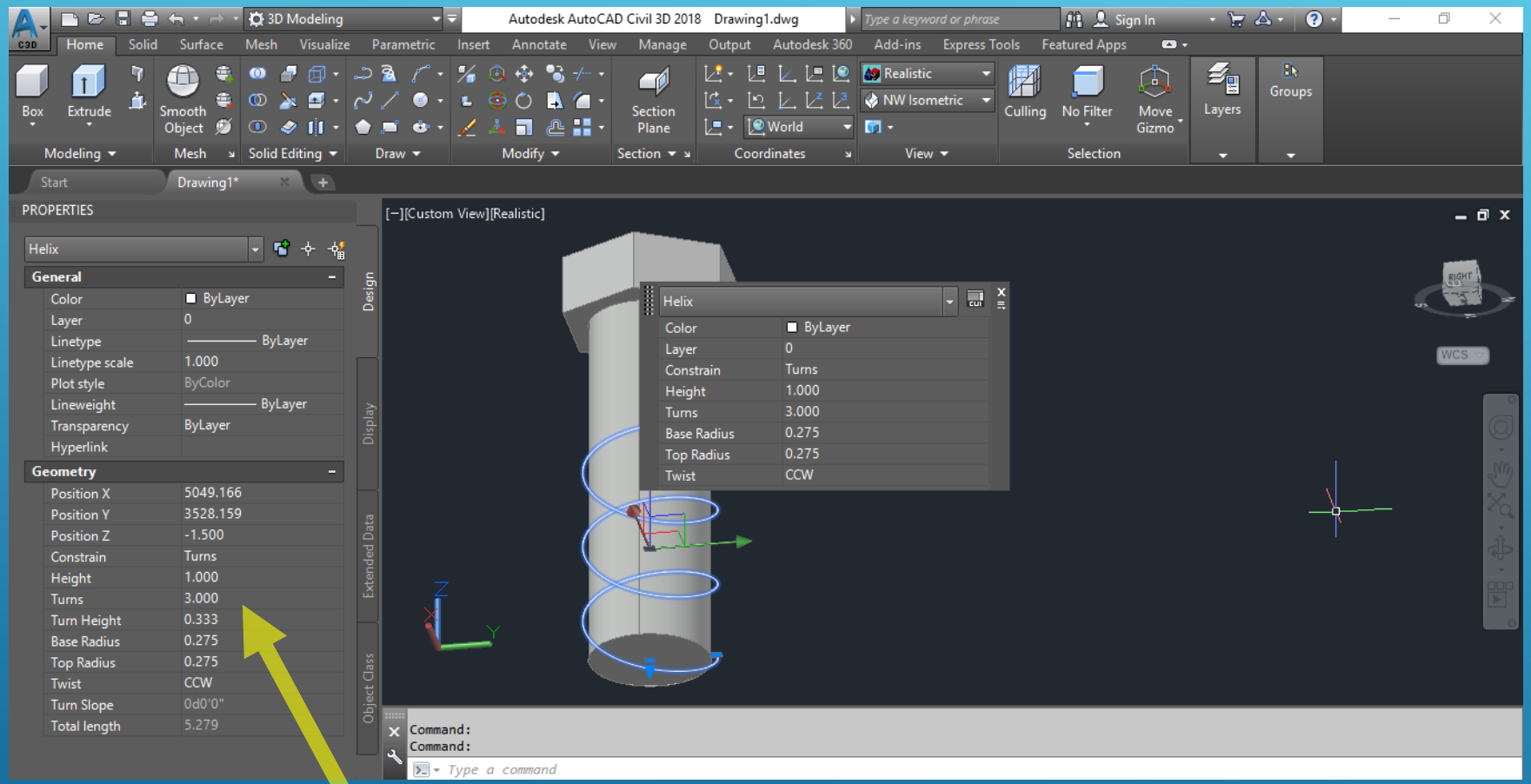
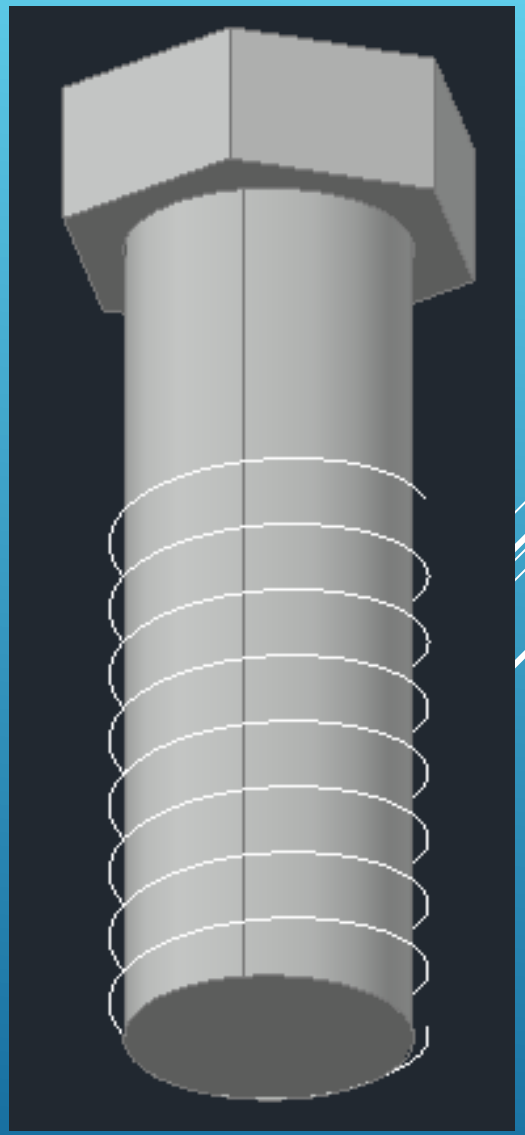


Turns: 8



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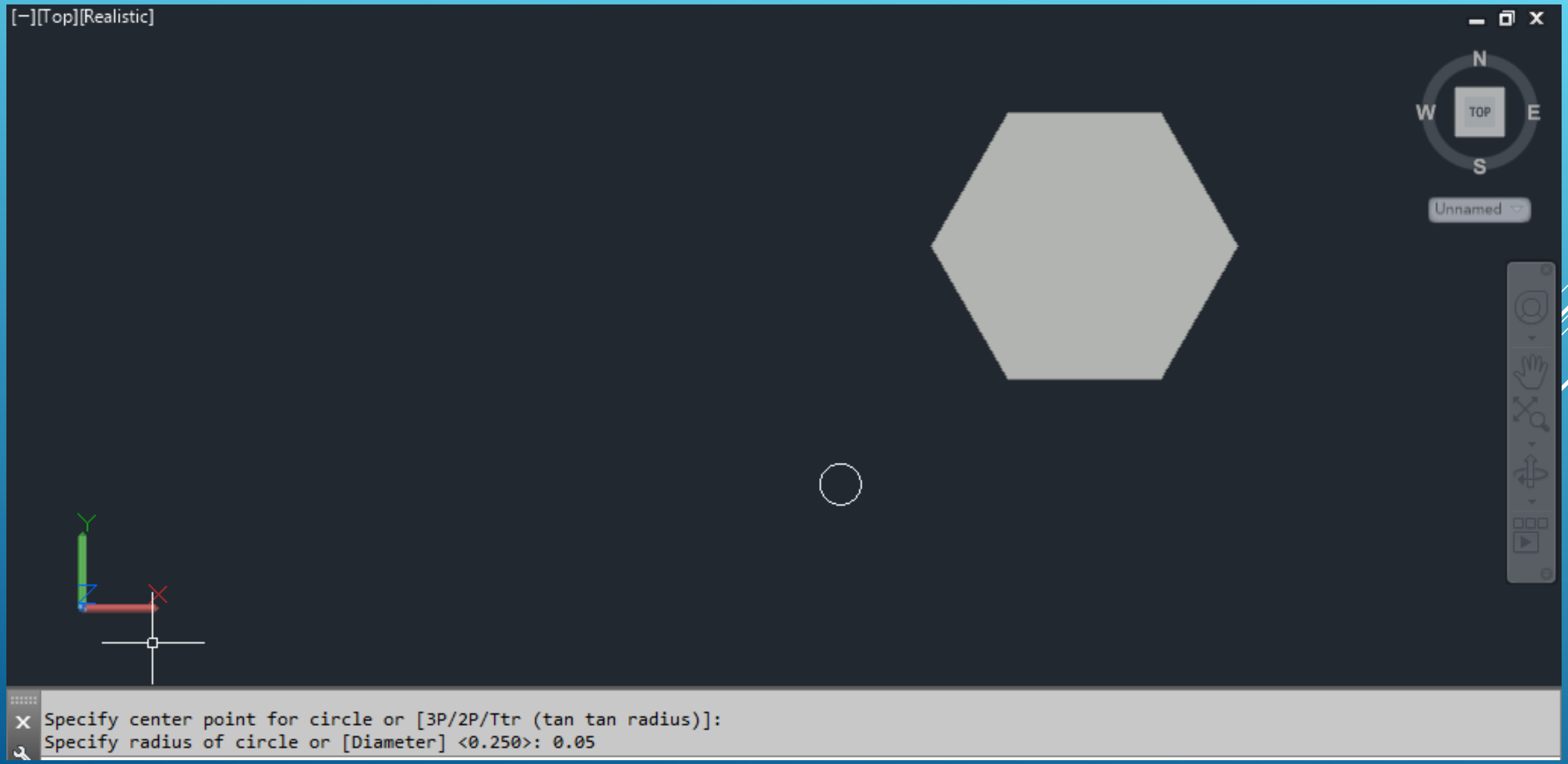

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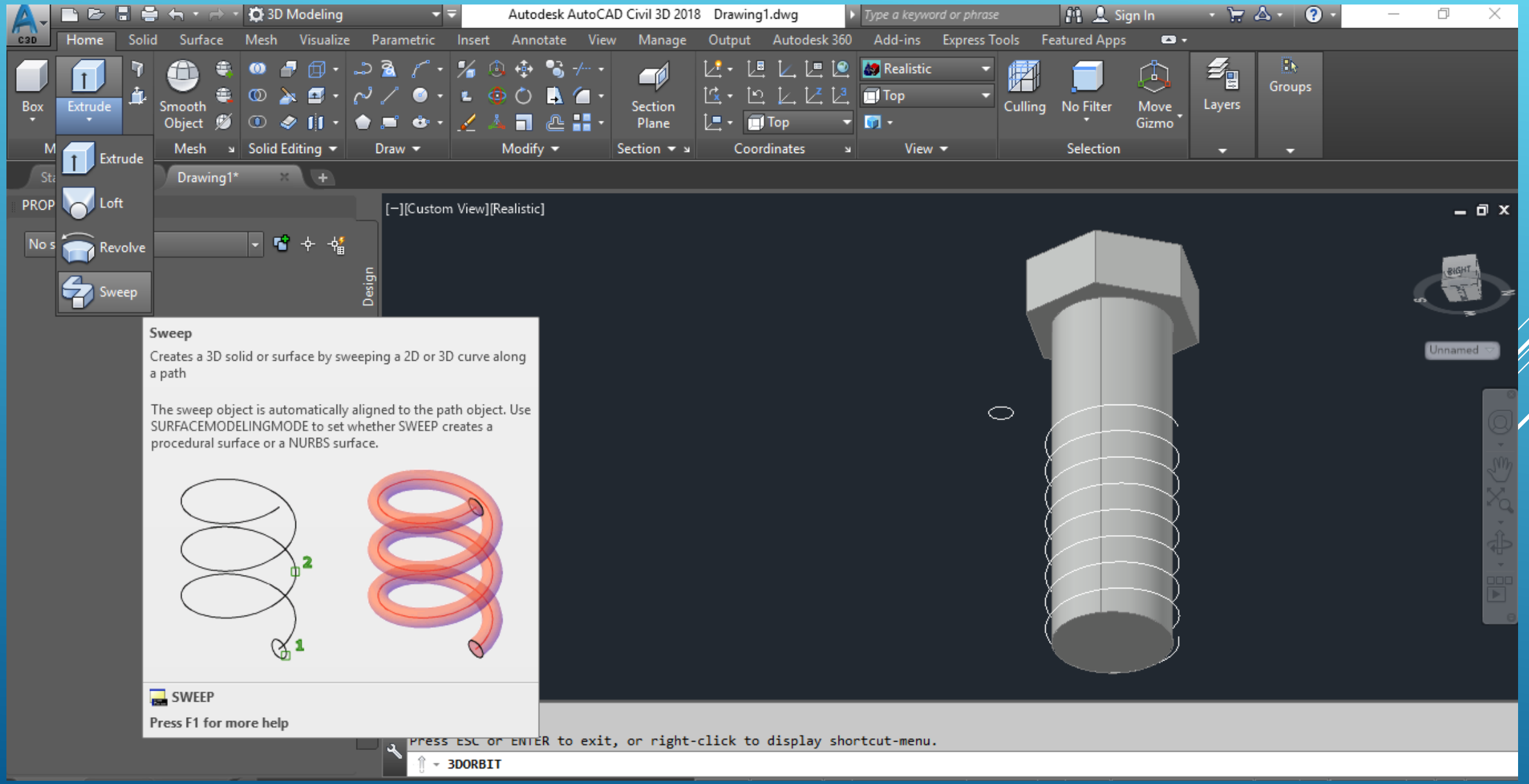

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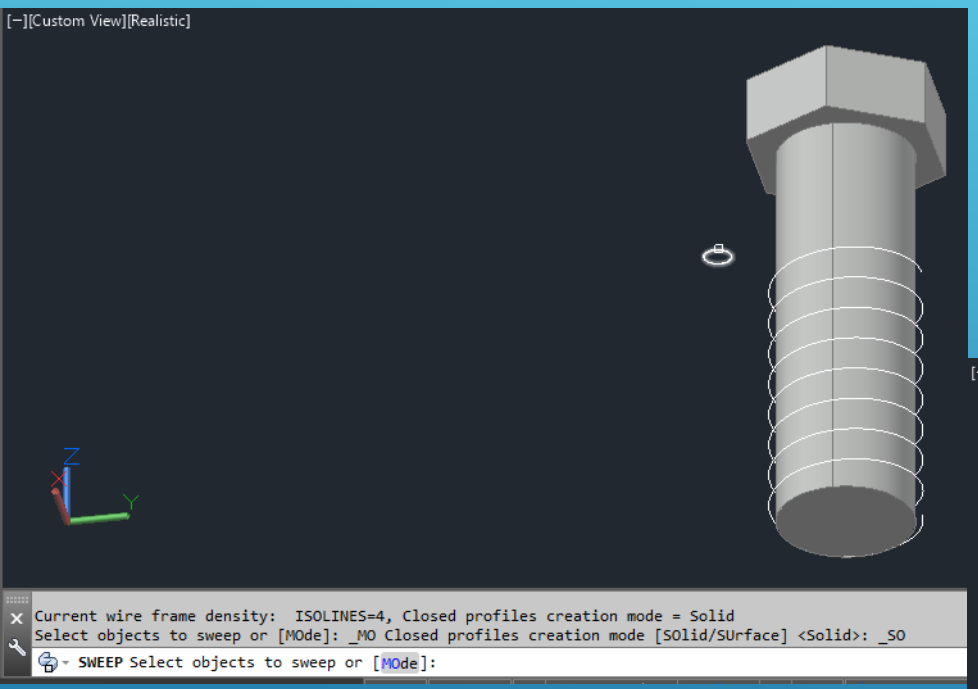

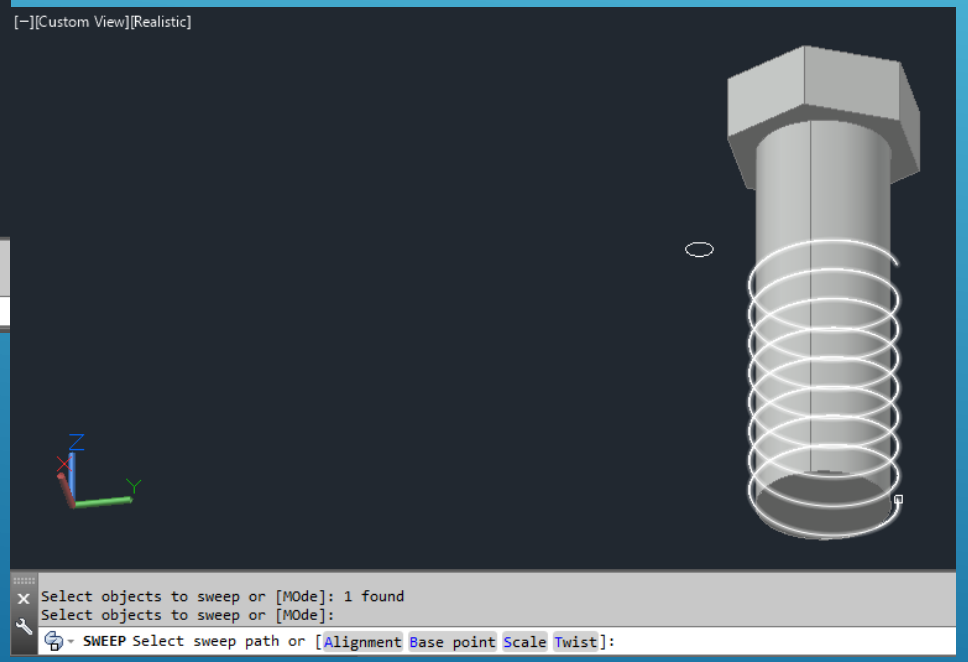

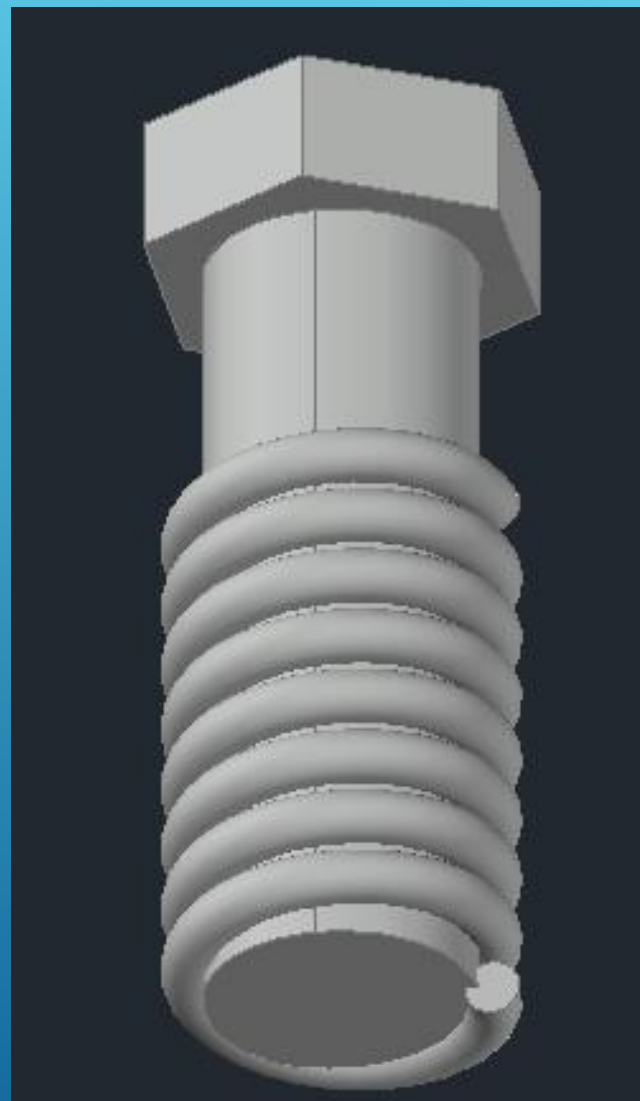

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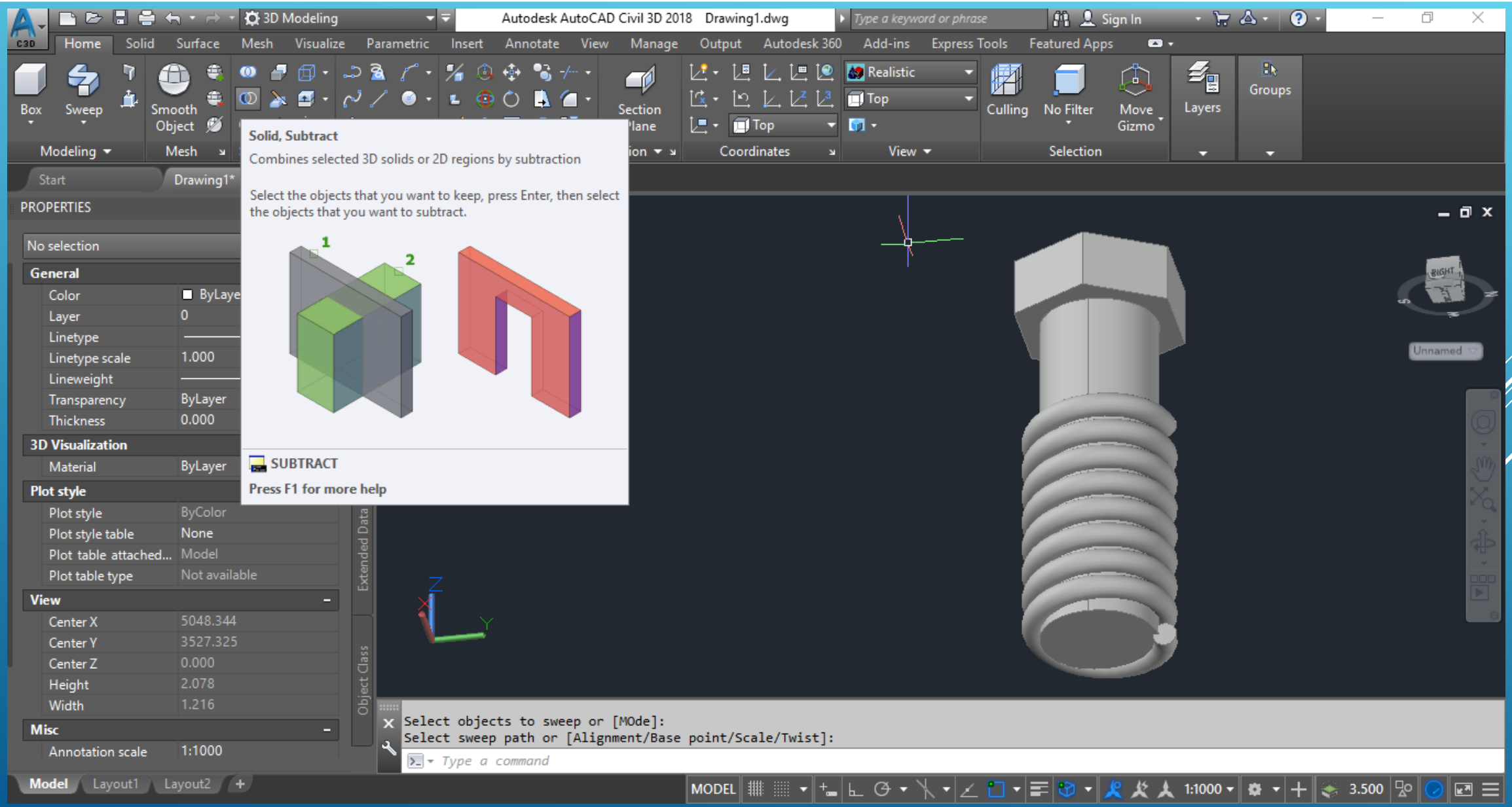

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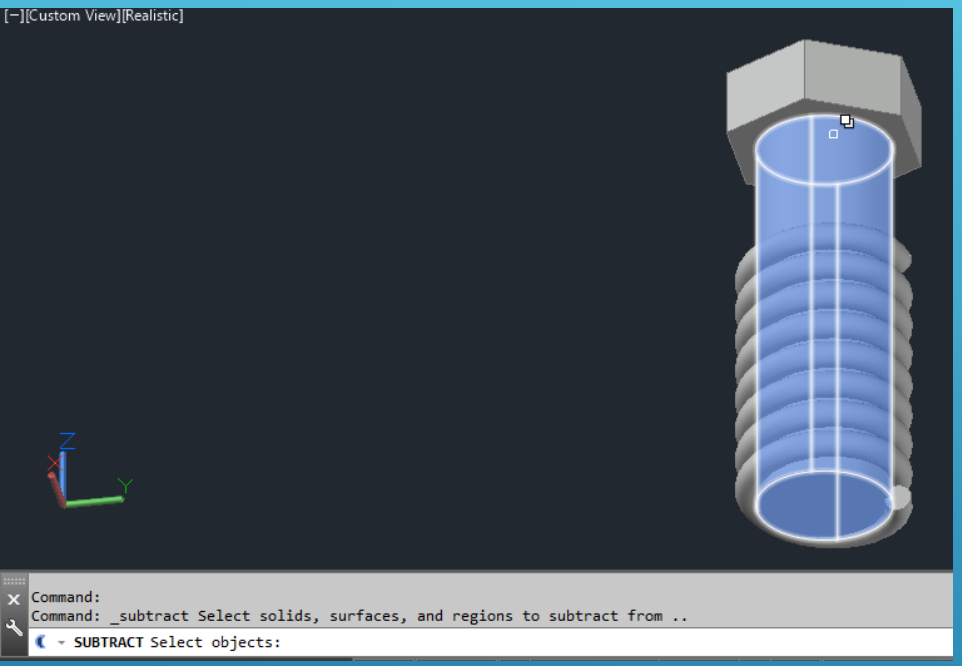
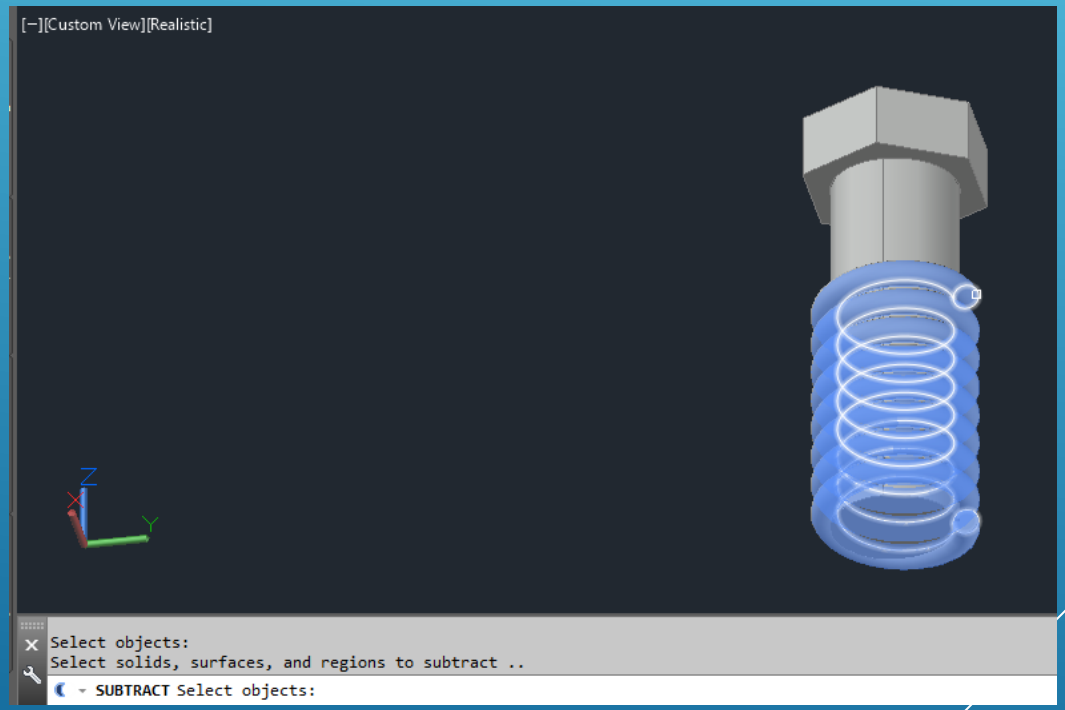
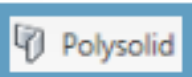




Primitive Solids

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

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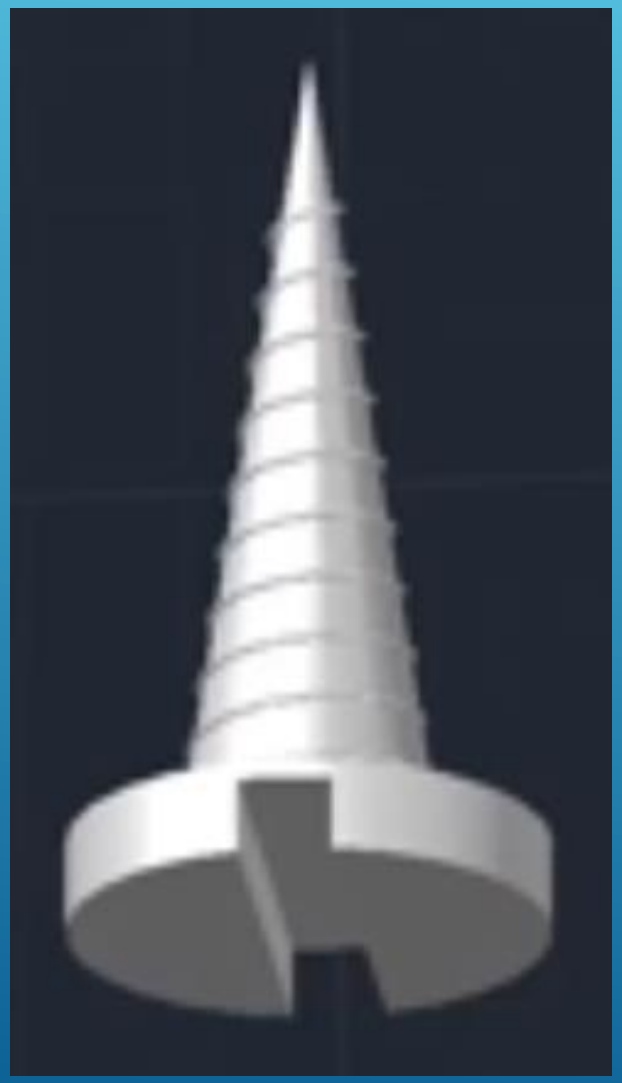
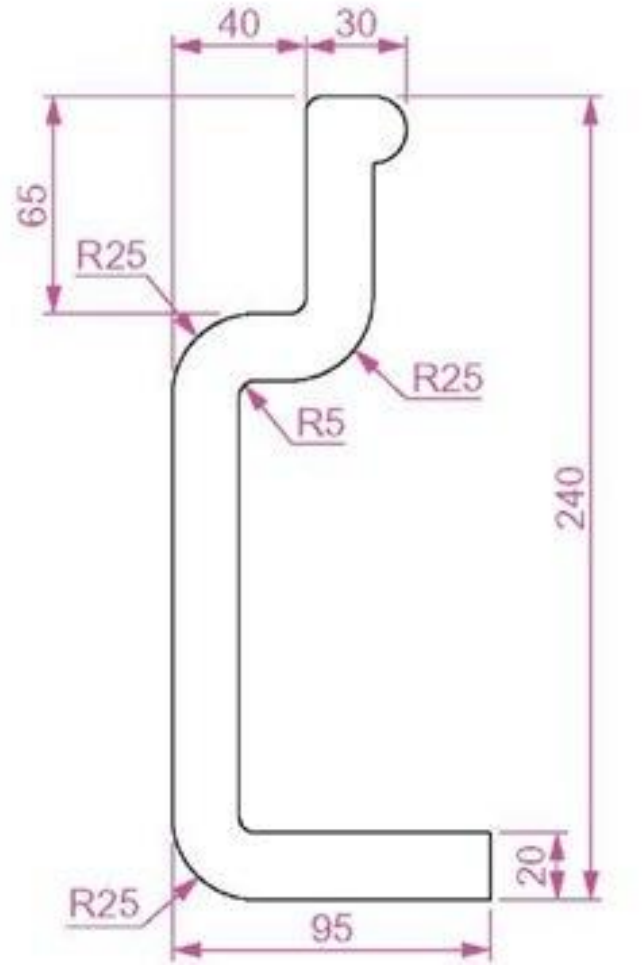


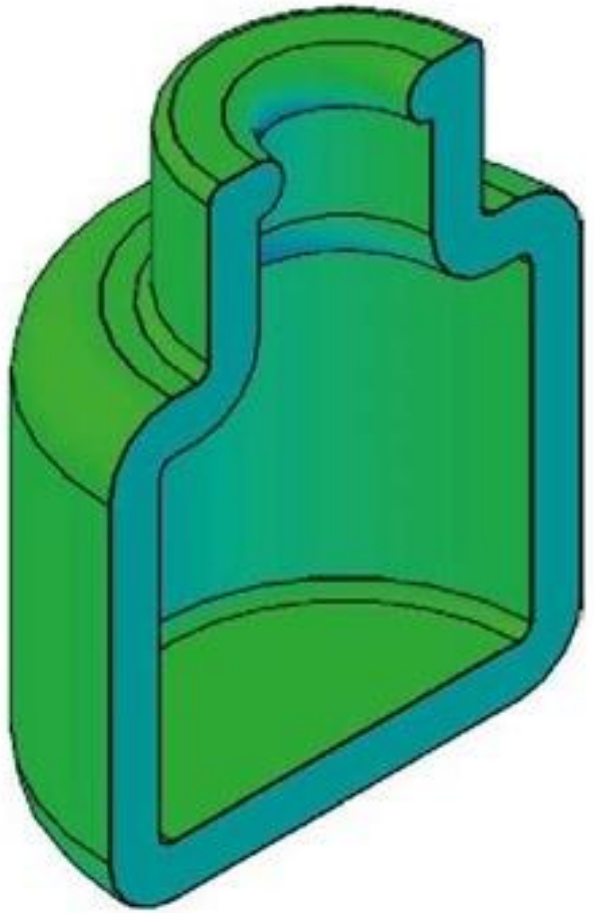
Desenhe como um sólido o parafuso com as características seguintes:

- 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 revolucão





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

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Desenhar o perfil do copo usando Lines e Arcs

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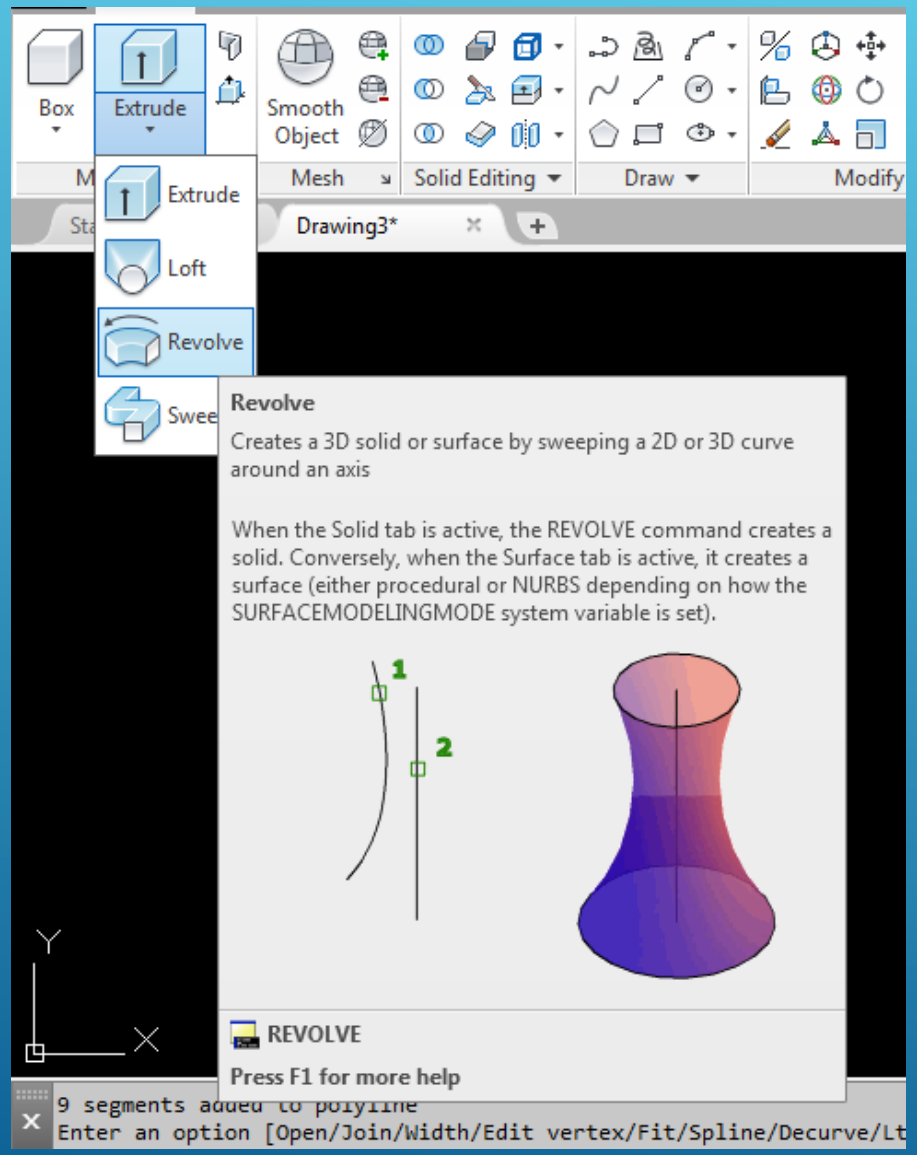

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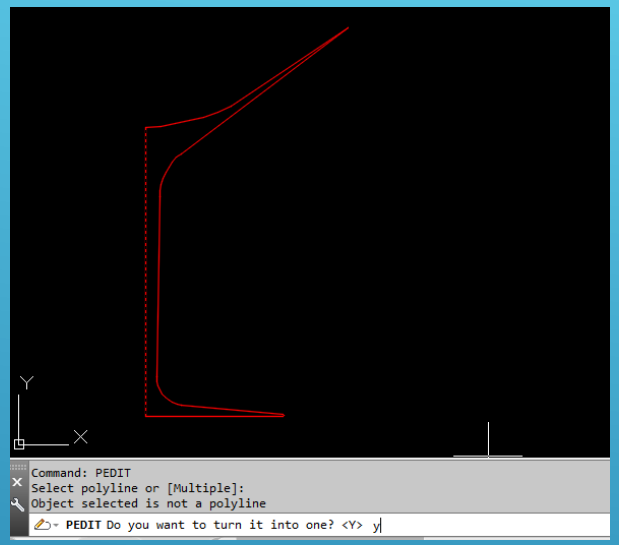

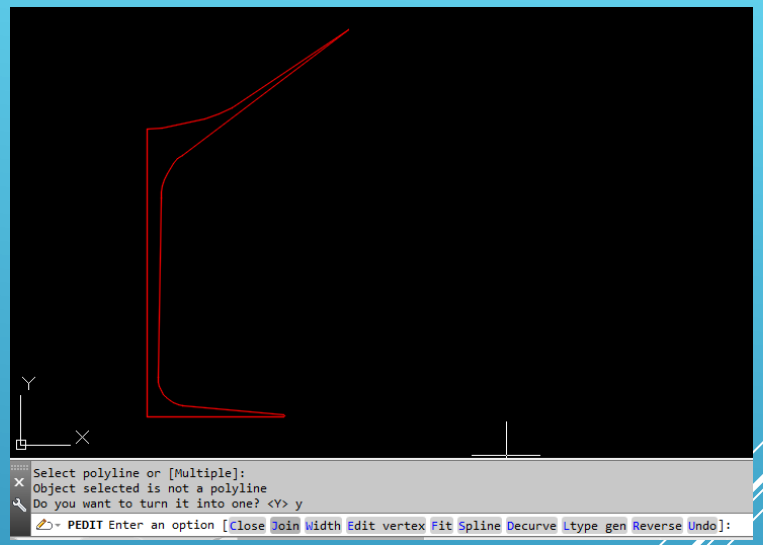

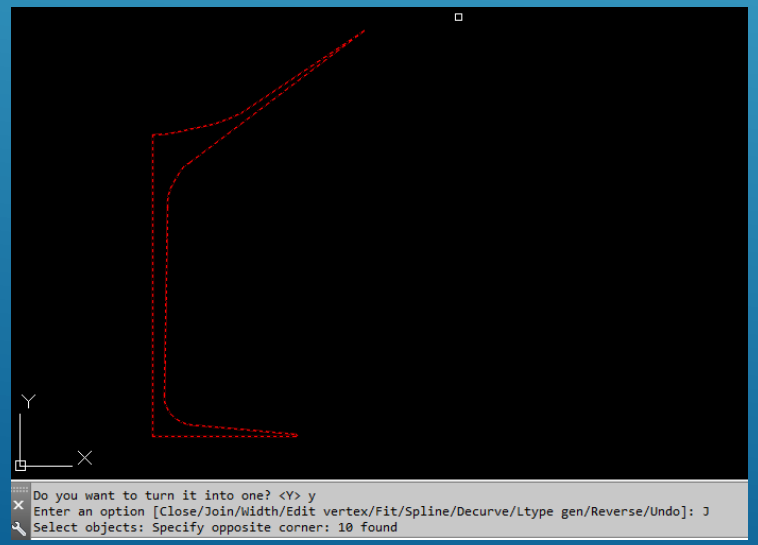

Usar o camando PEDIT para transformar os elementos LINES e ARCS numa POLYLINE

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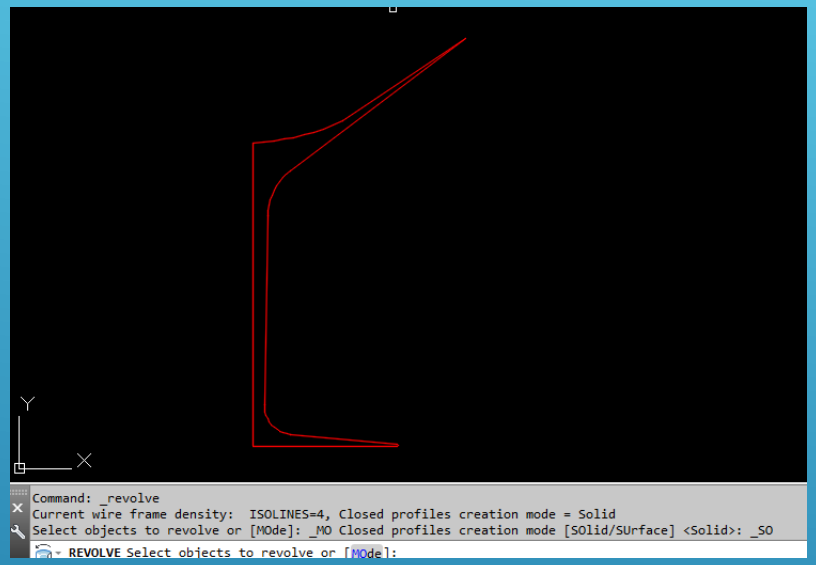

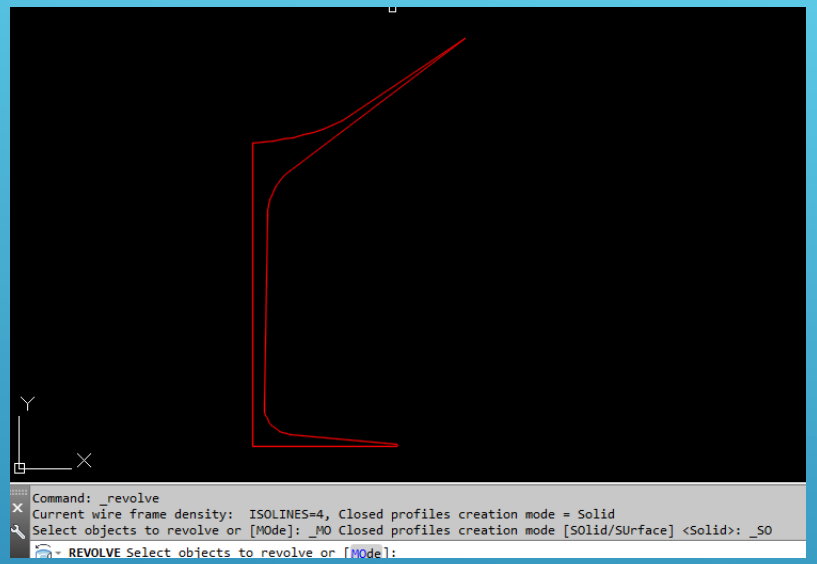

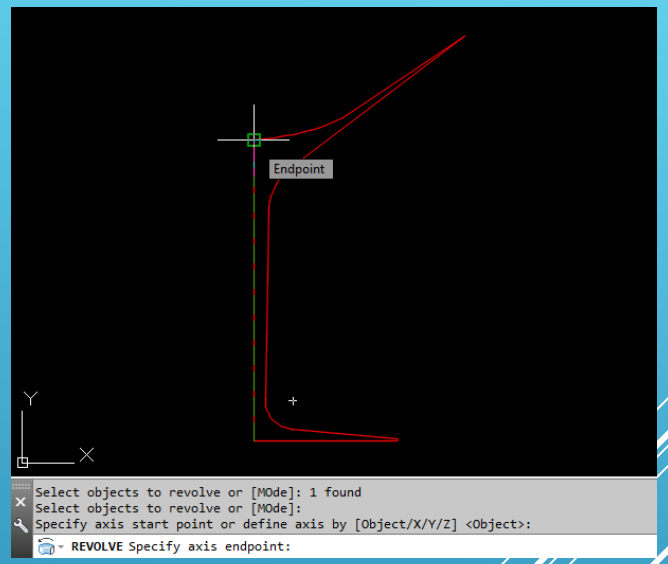

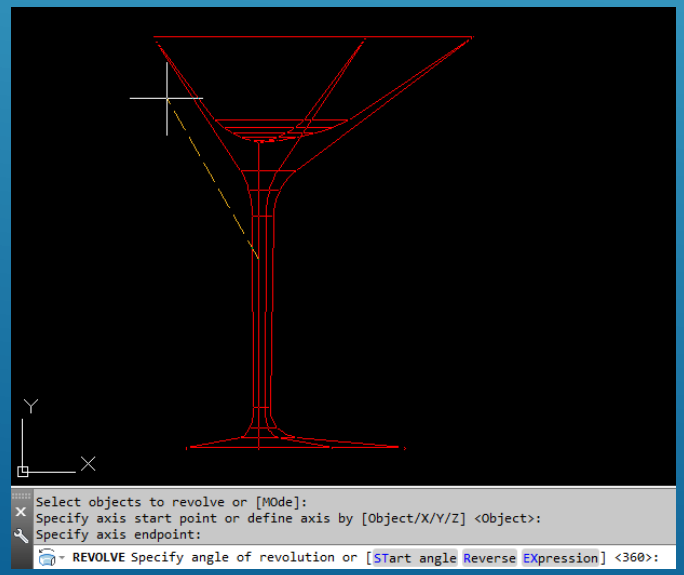

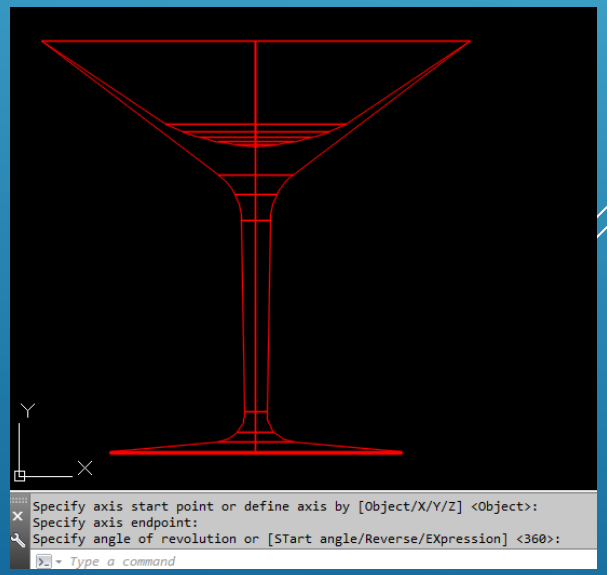

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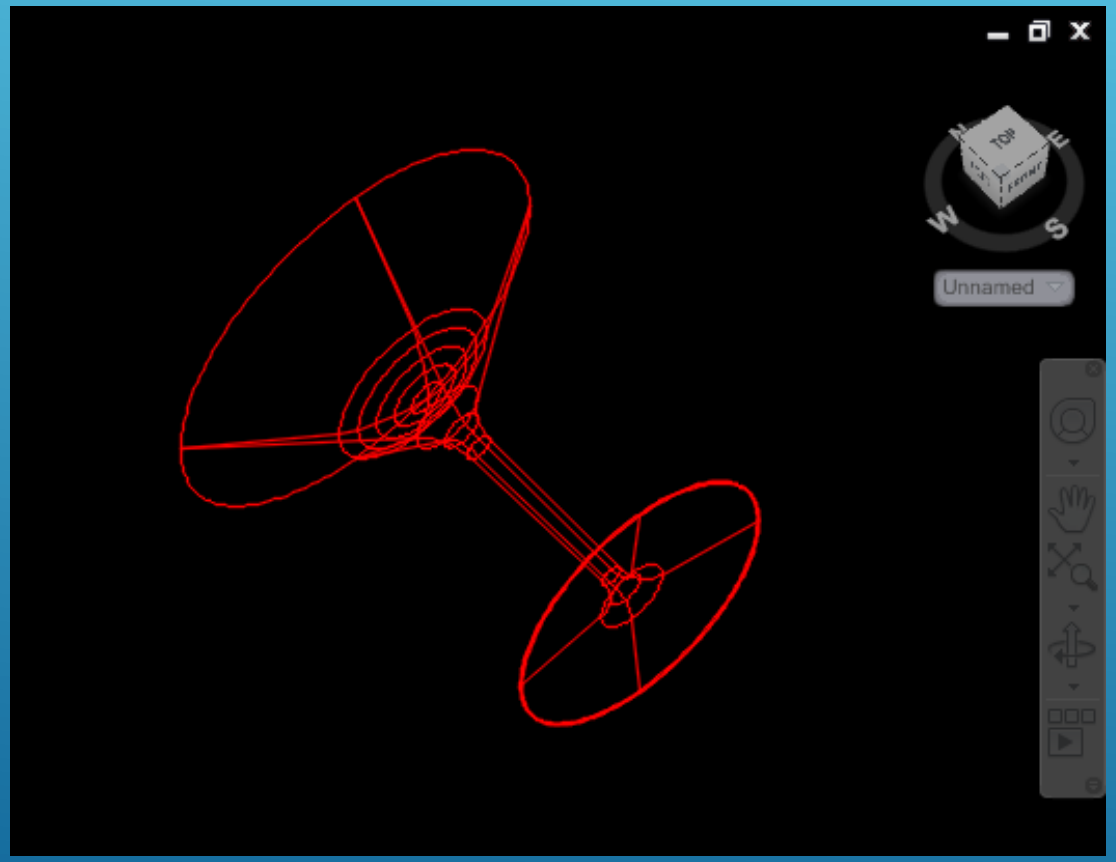

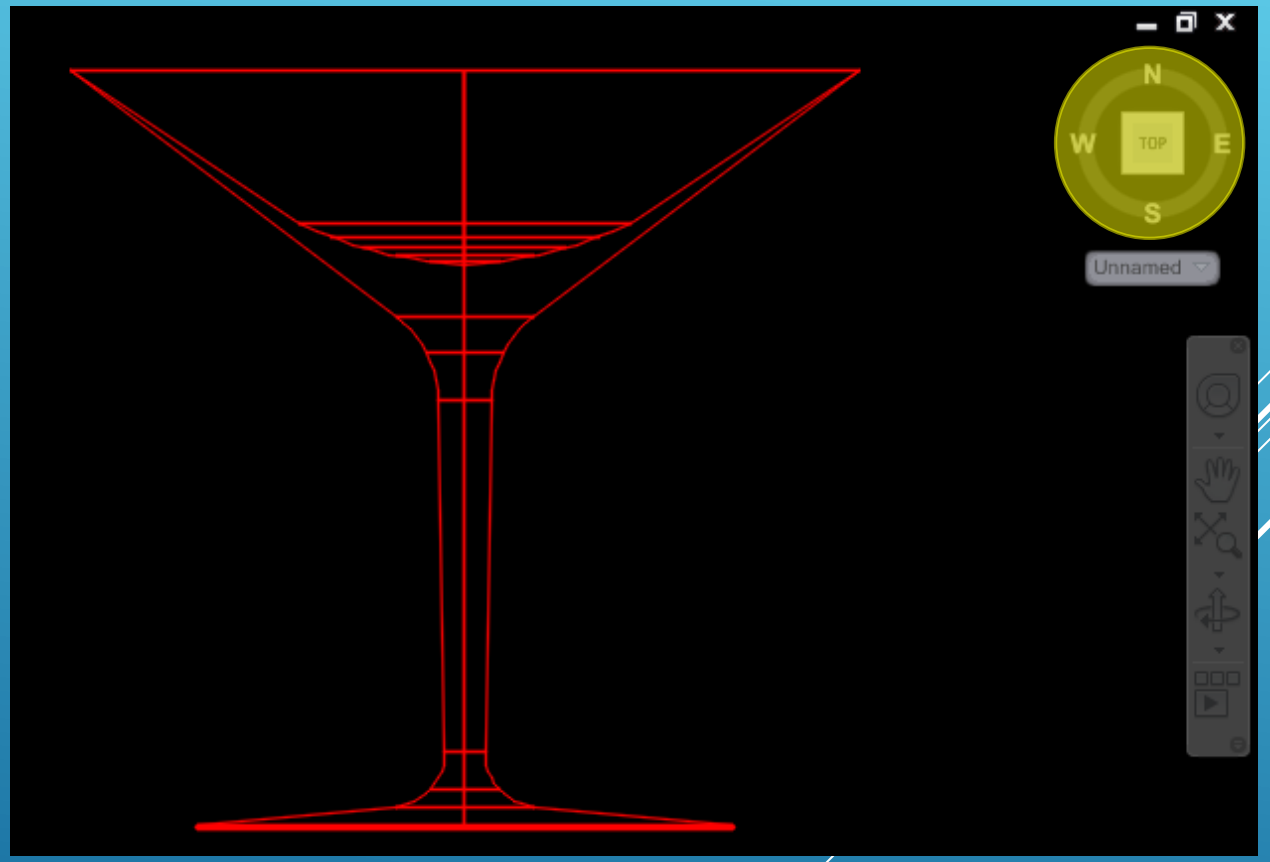

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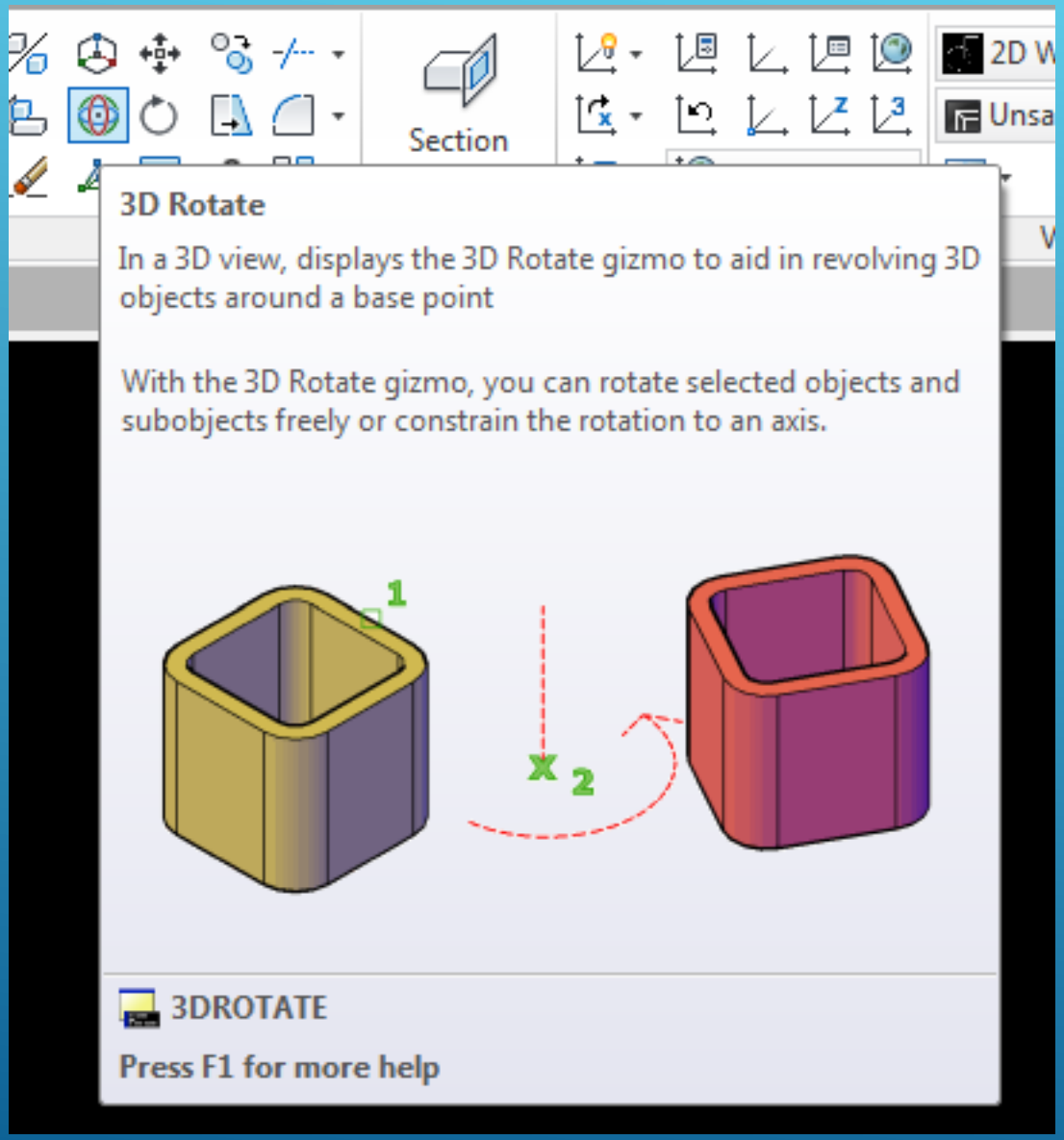

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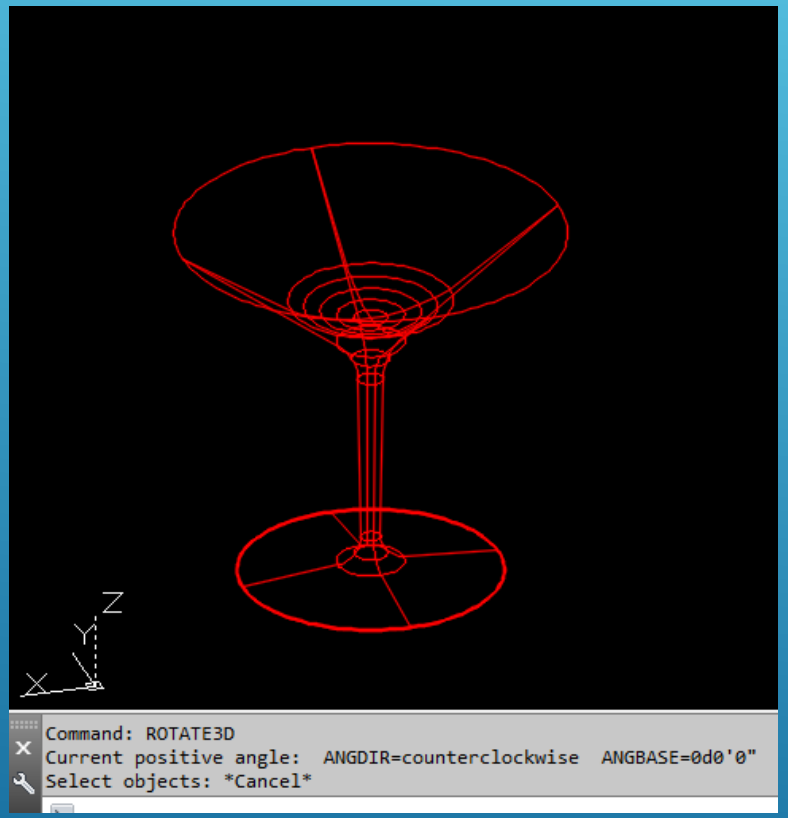

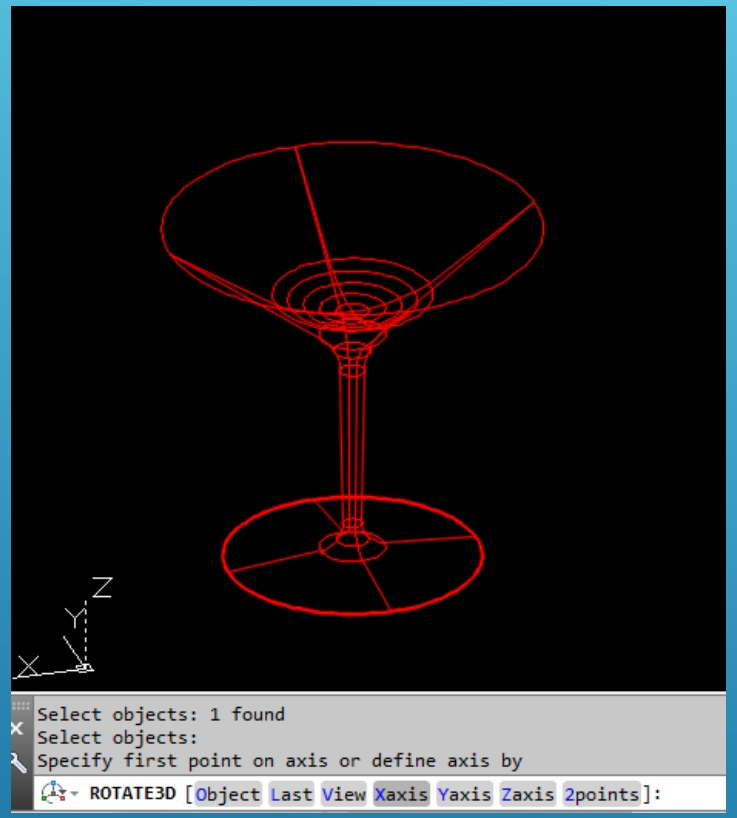

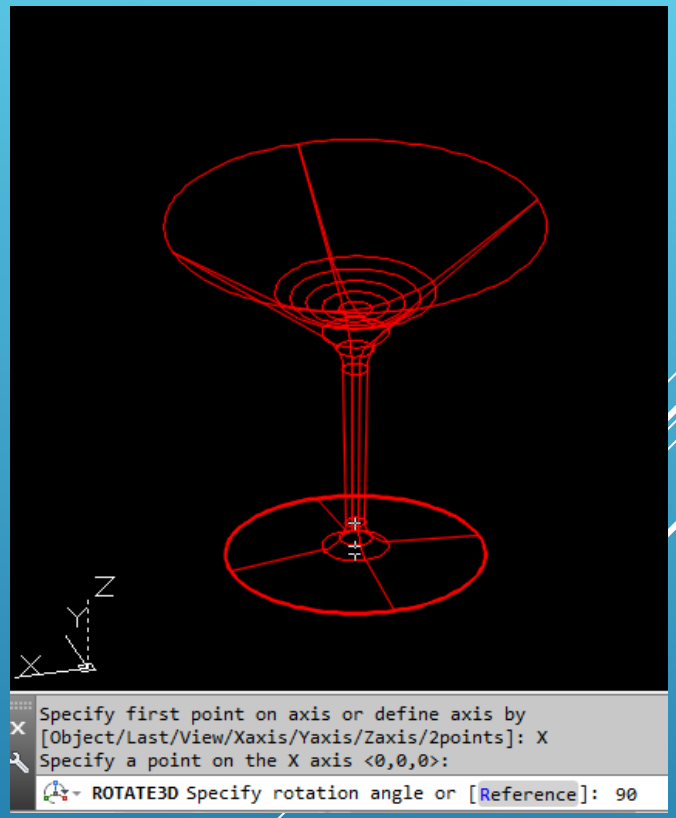

Rodar o copo em torno do eixo X para que na vista Top o copo apareça em pé

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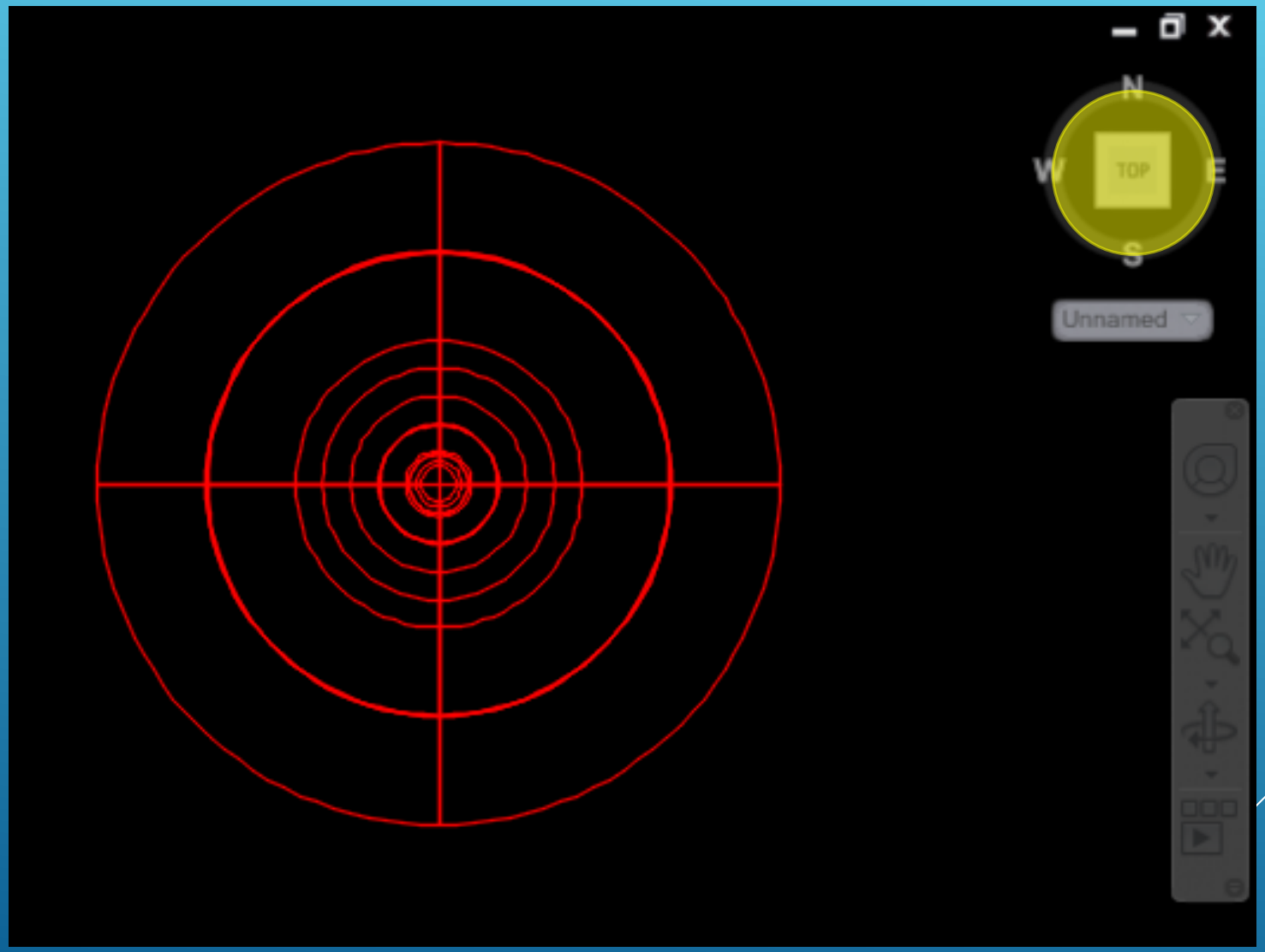

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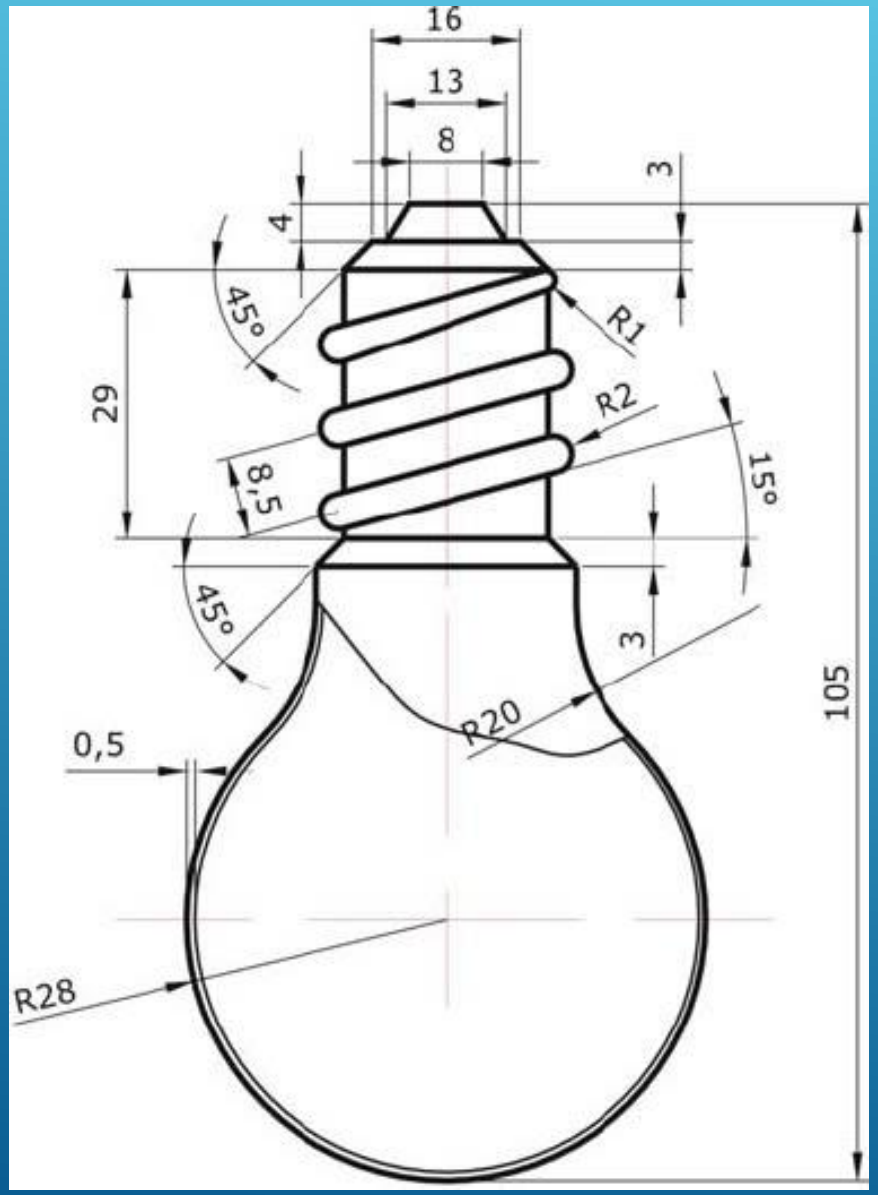

Represente a 3D a peça indicada na figura seguinte.

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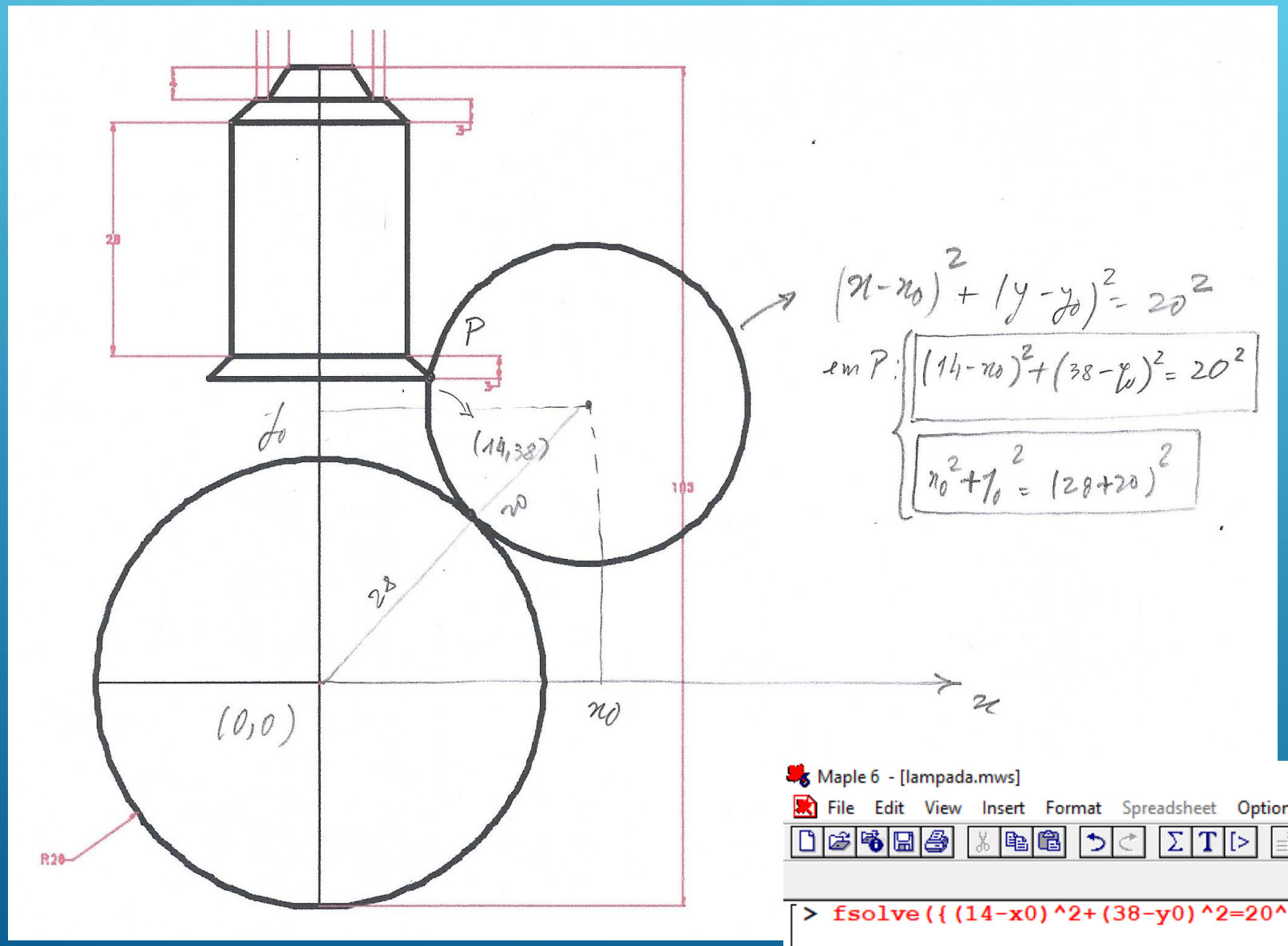
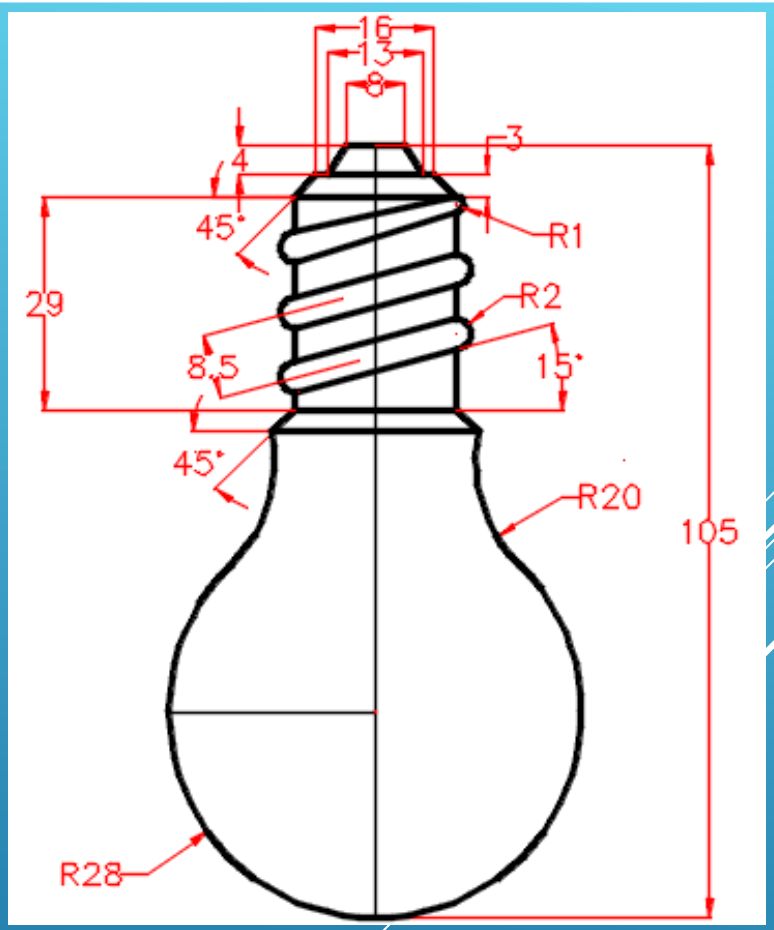


$$(x-x_0)^2+(y-y_0)^2=20^2$$

em P:
$$\begin{cases}(14-x_0)^2+(38-y_0)^2=20^2\\ x_0^2+y_0^2=(28+20)^2\end{cases}$$



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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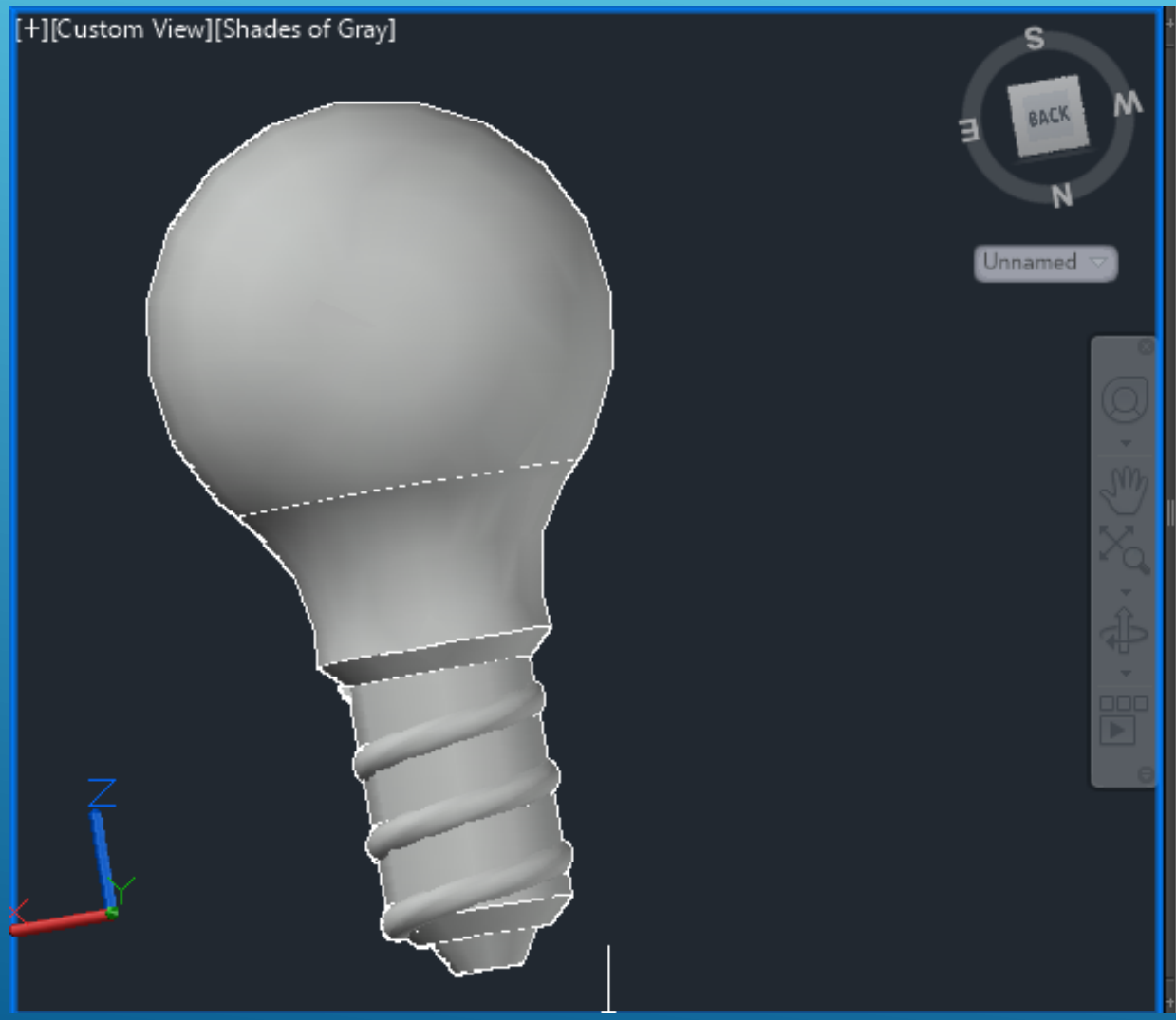

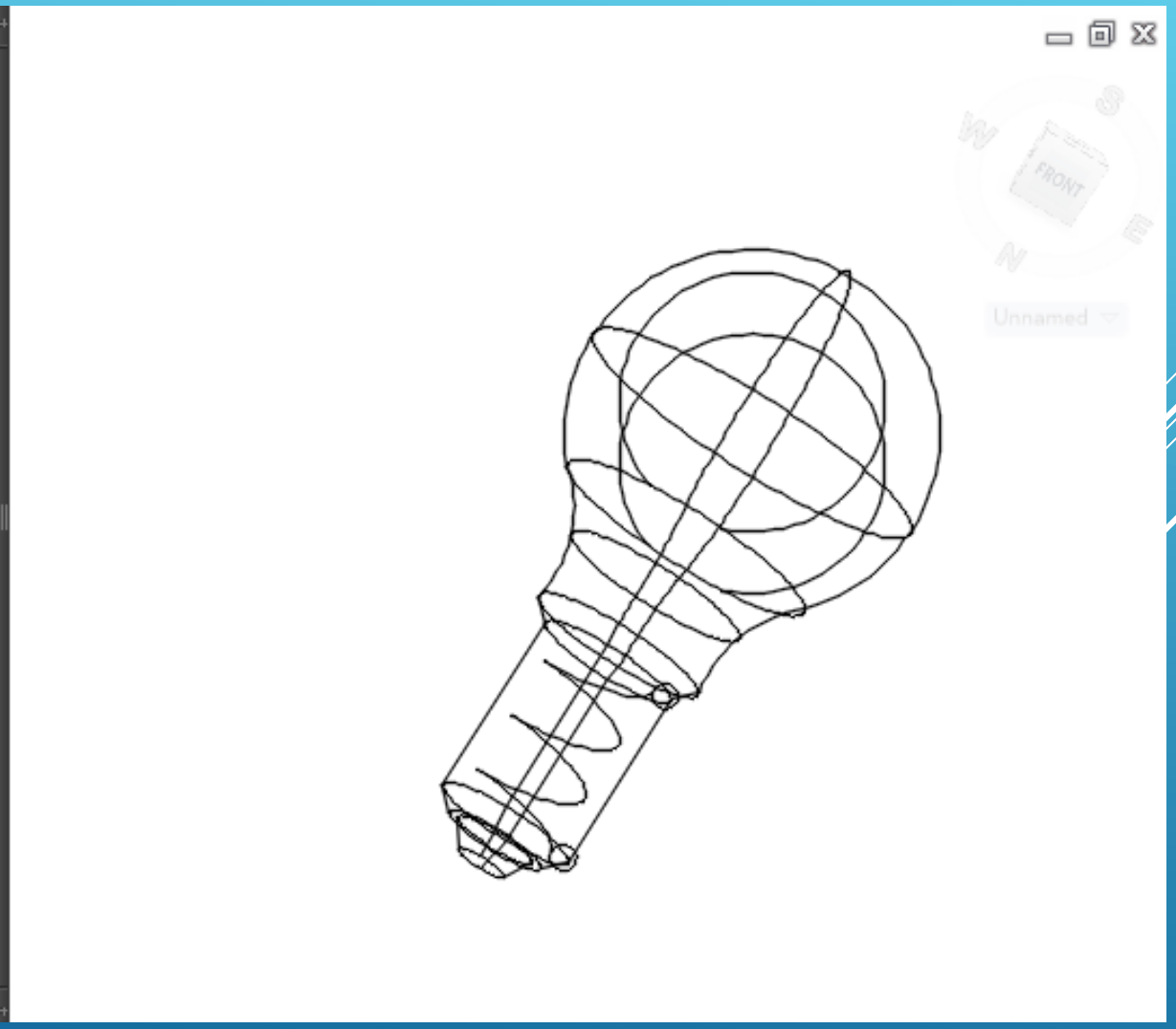

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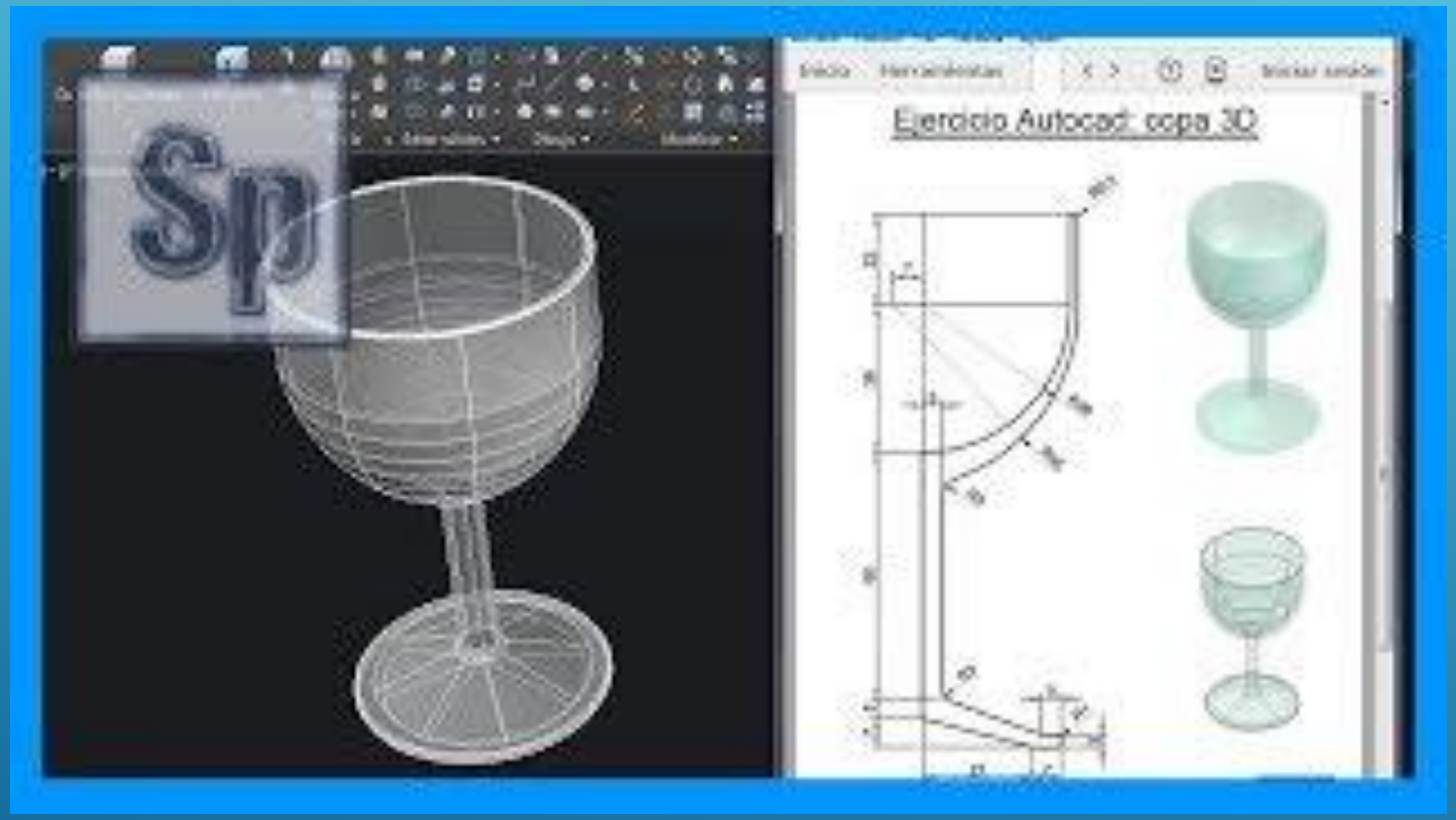

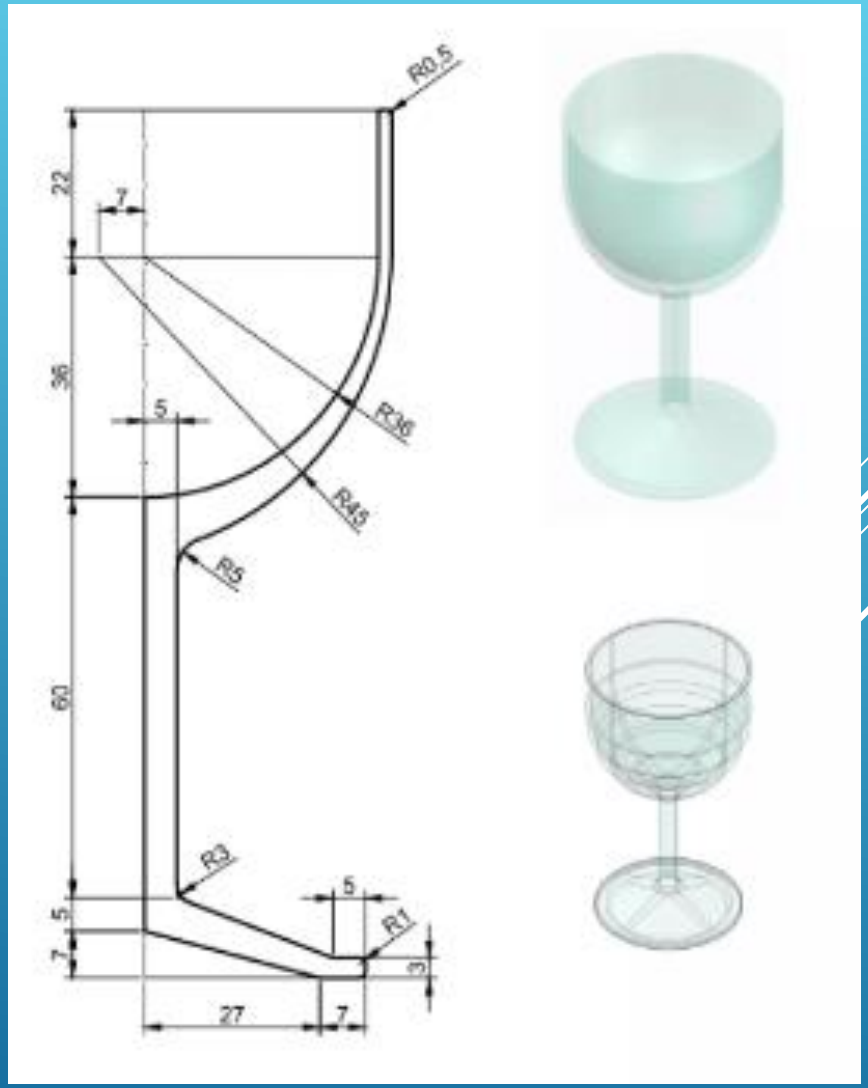

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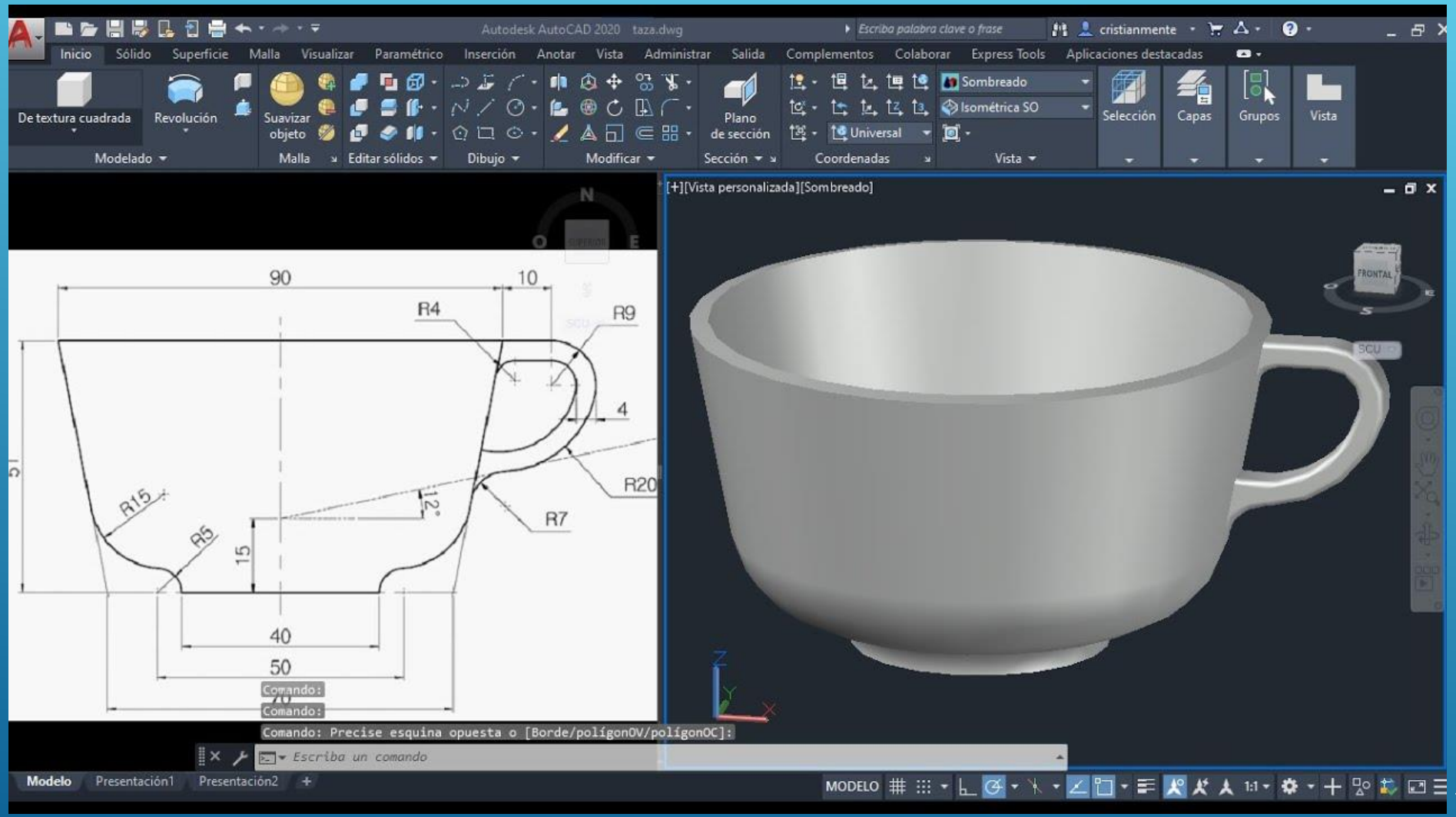

Desenhe a chávena como um sólido, supondo que a asa tem 4 unidades de largura. Após ter construído o sólido, use o comando Home>Solid Editing>Shell para criar um “buraco” no interior da chávena, identificando a superfície de topo da chávena e indicando duas unidades para a espessura da chávena. Apresente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

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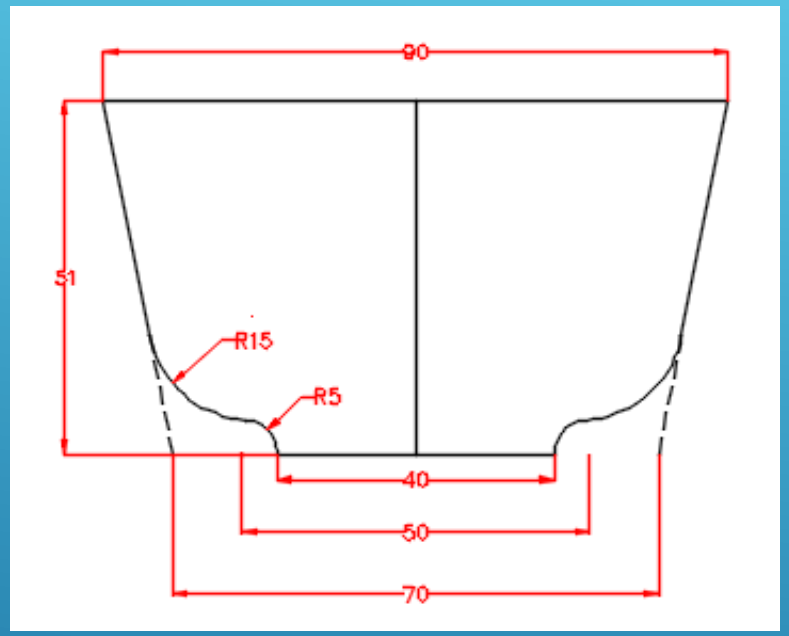

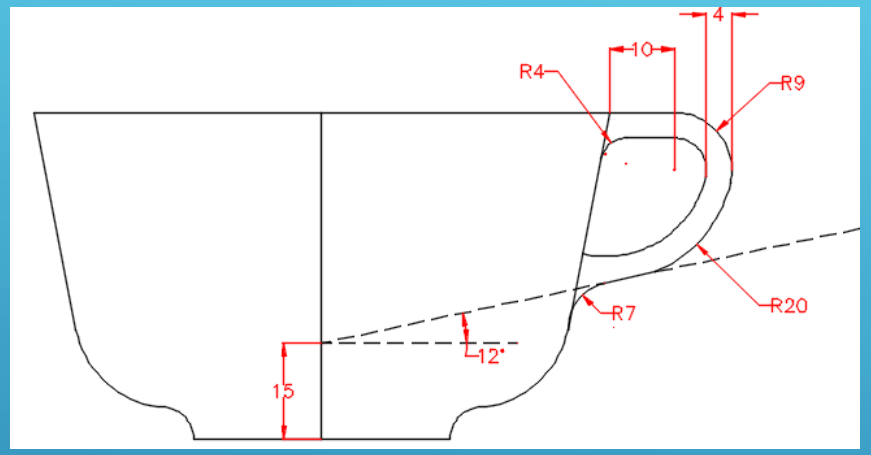

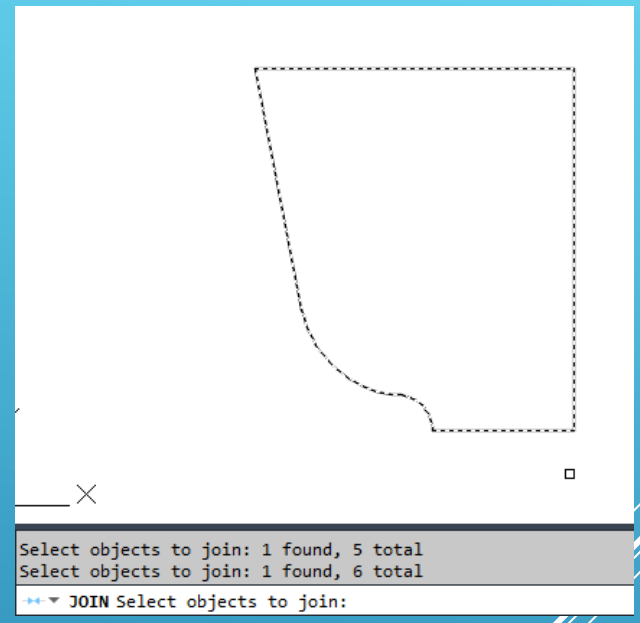

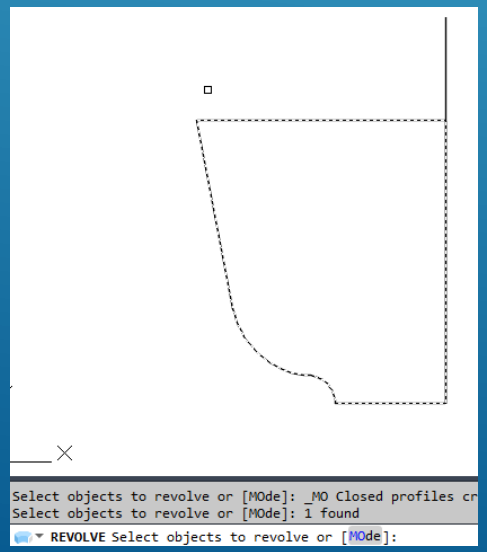

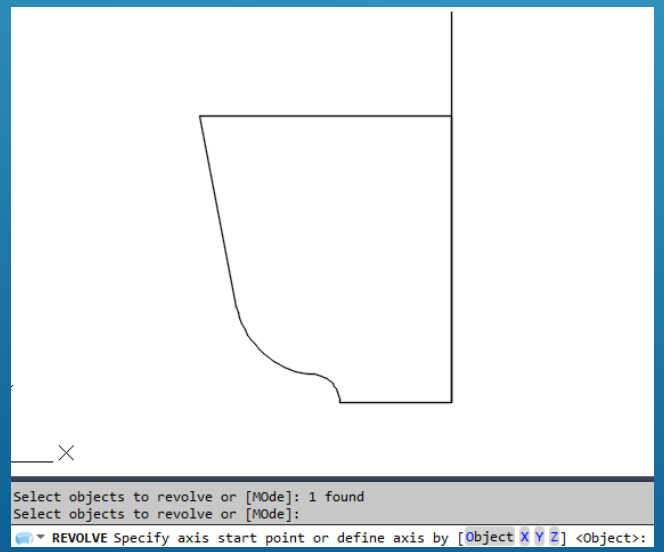

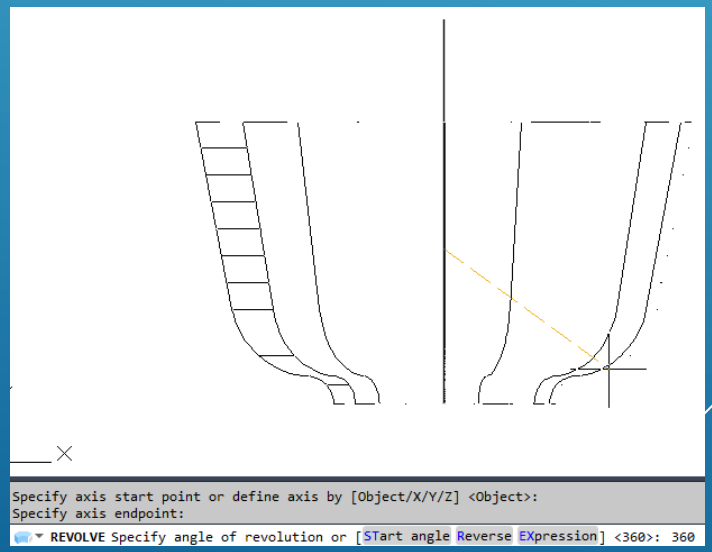

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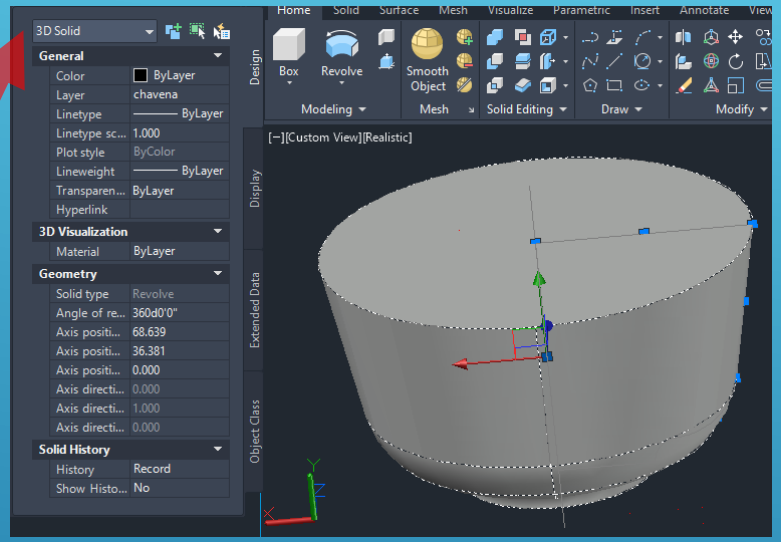

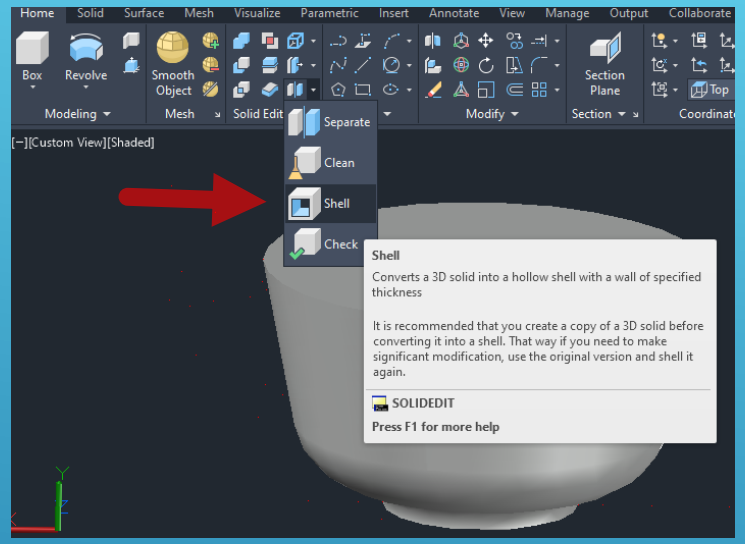

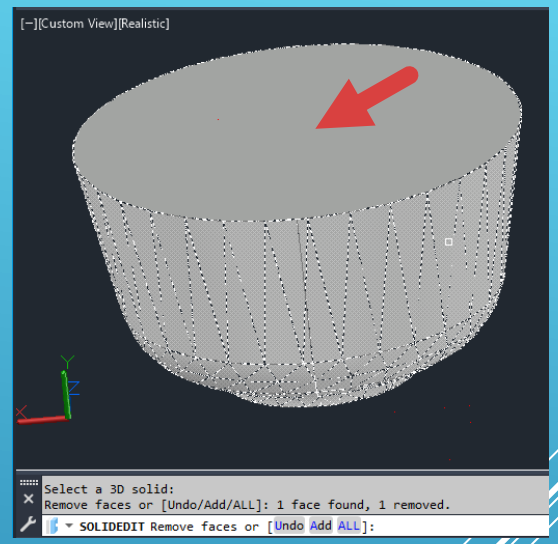

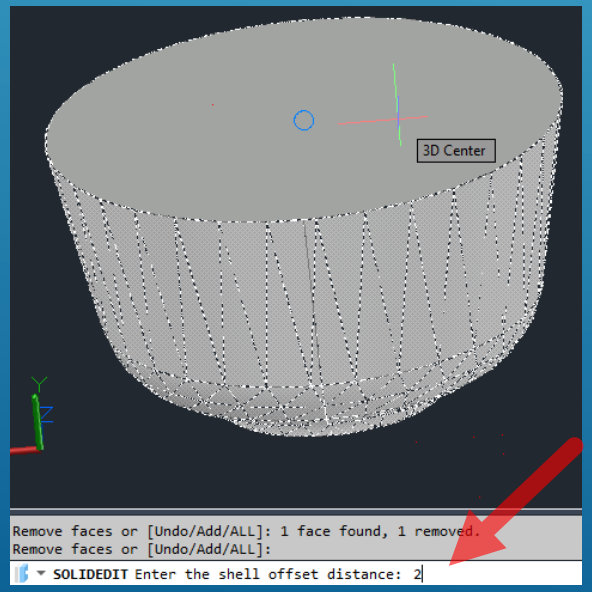

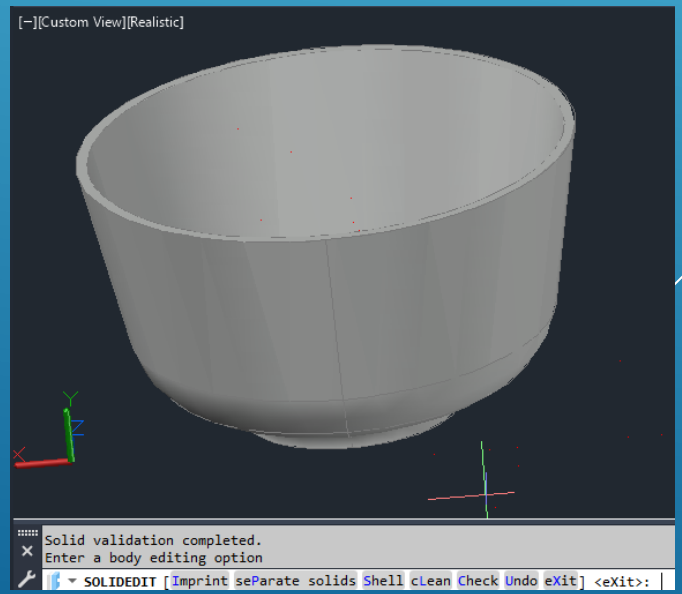

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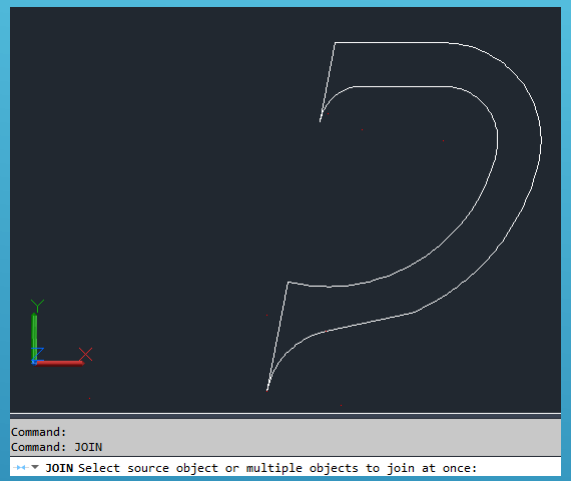

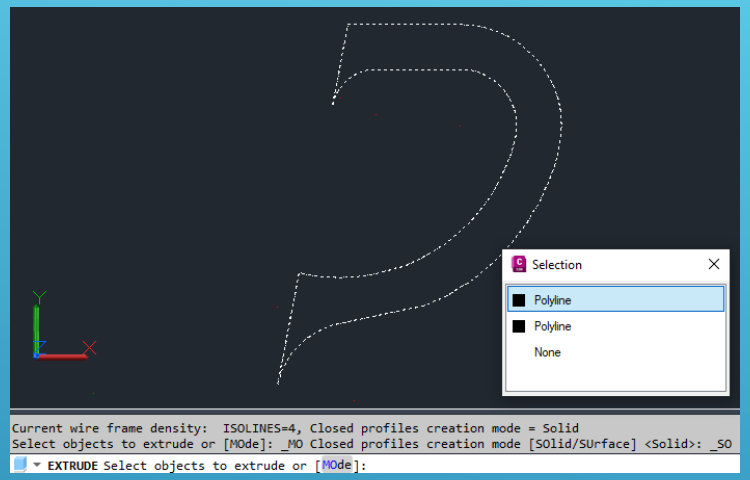

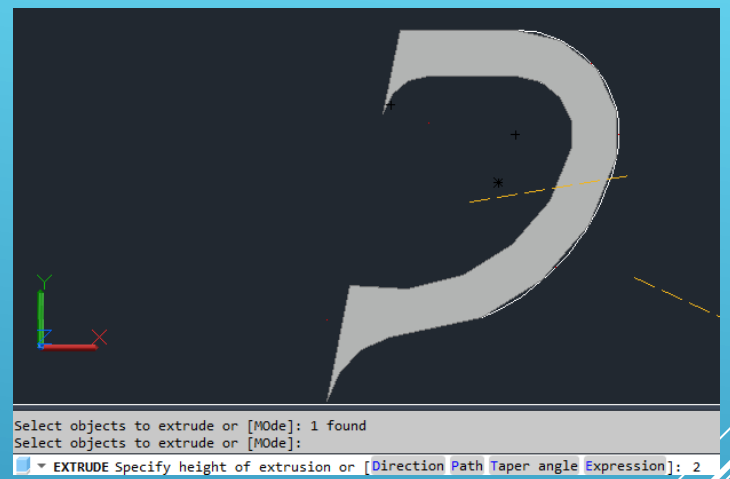

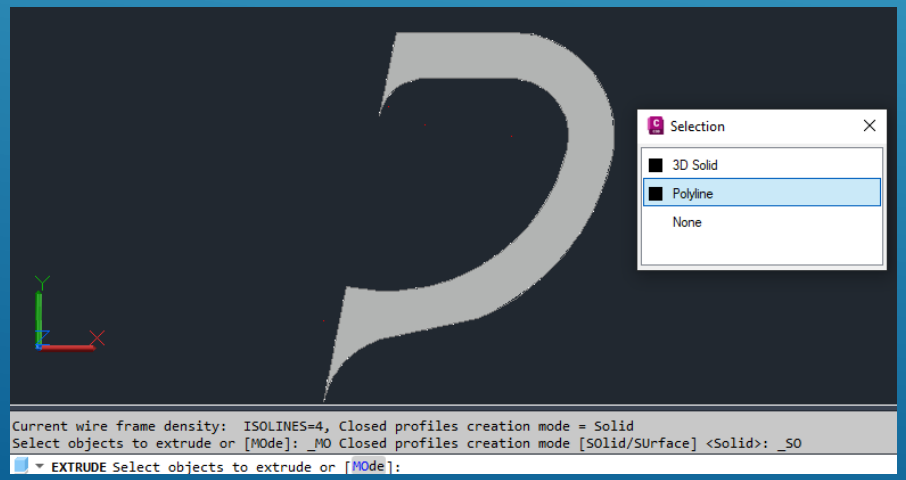

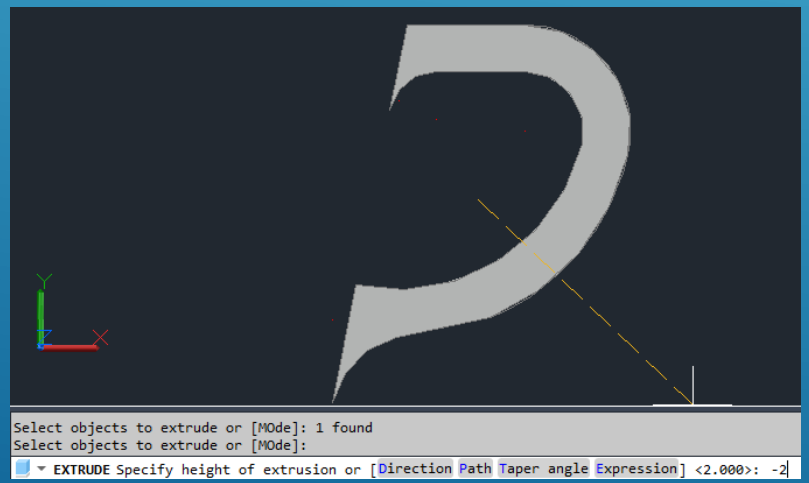

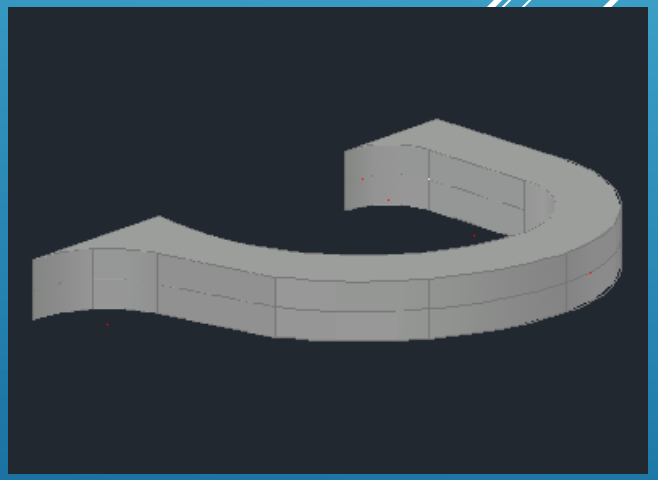

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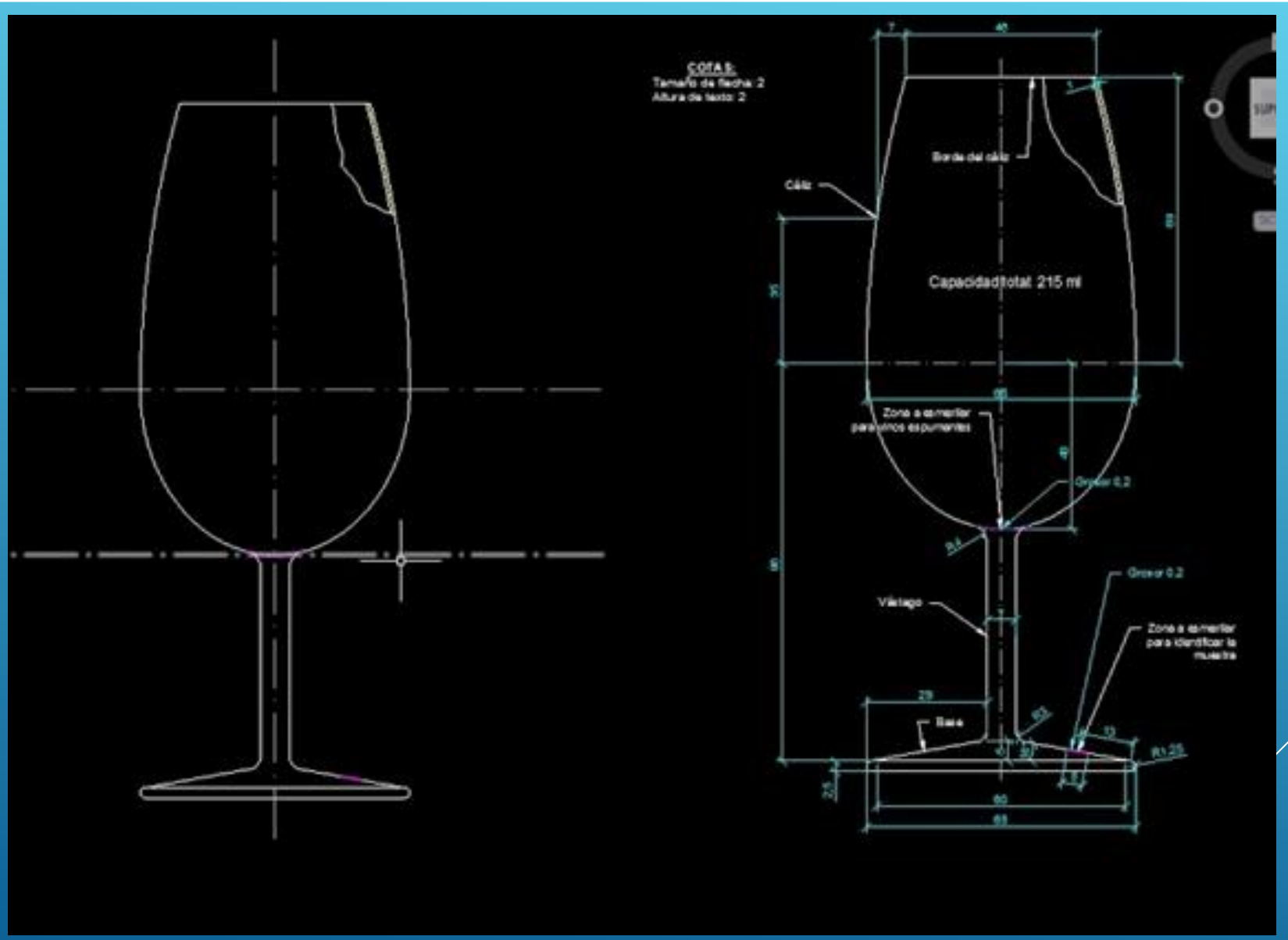

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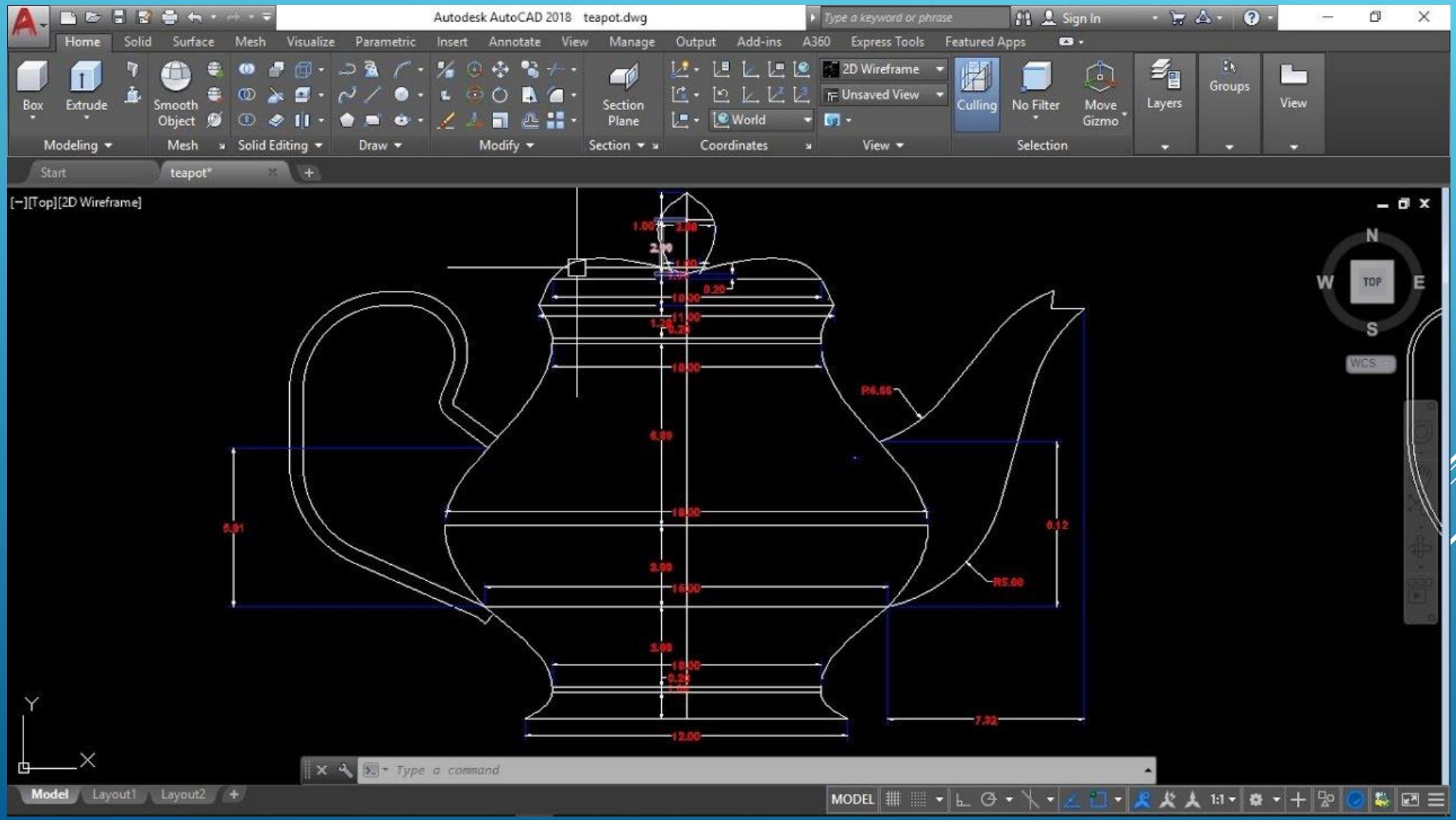

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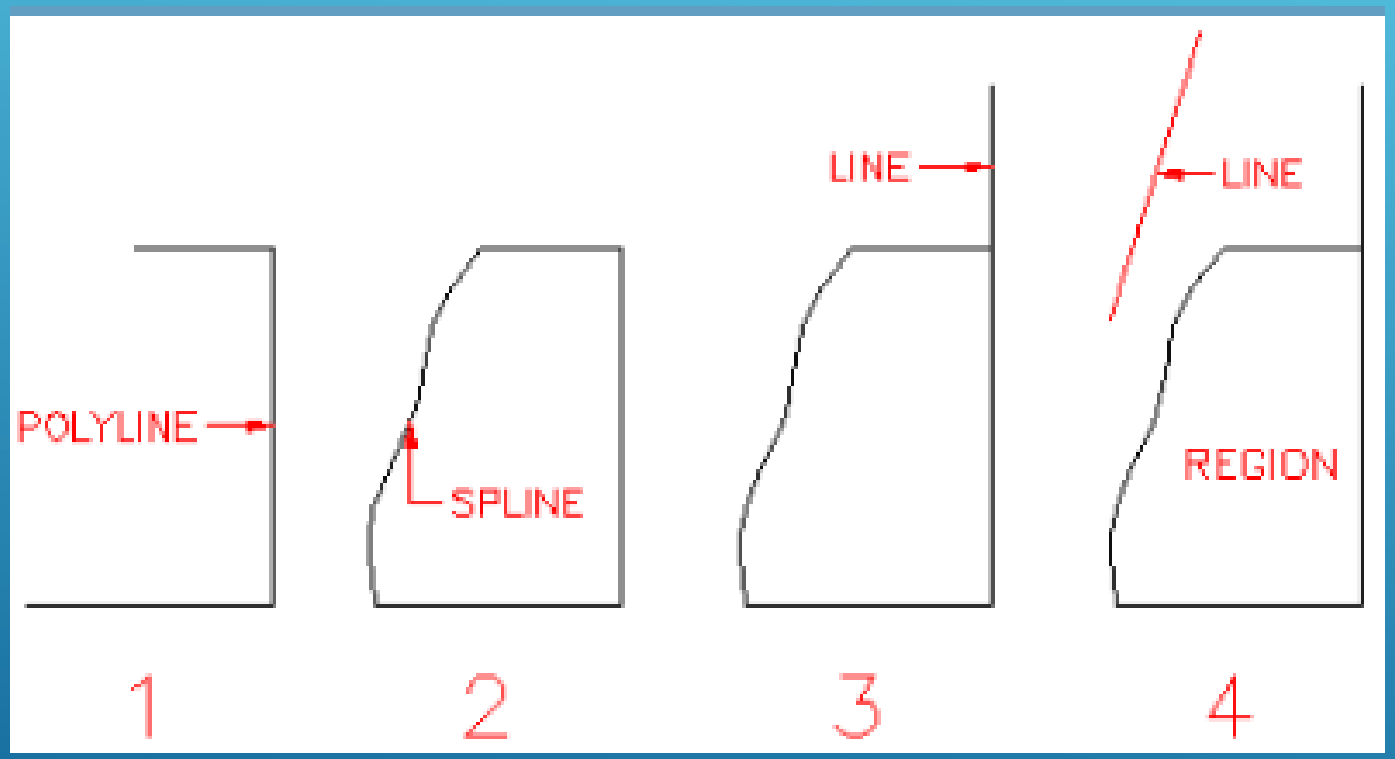

1. Criar 2 layers: base, abat-jour
2. Desenhar Polyline (layer base)
3. Desenhar Spline (layer base)
4. Desenhar Line vertical
5. Desenhar Line (layer abat-jout)
6. Definir Region (seleccionar Polyline e Spline, com Shift)
7. Revolve
8. Explode Region

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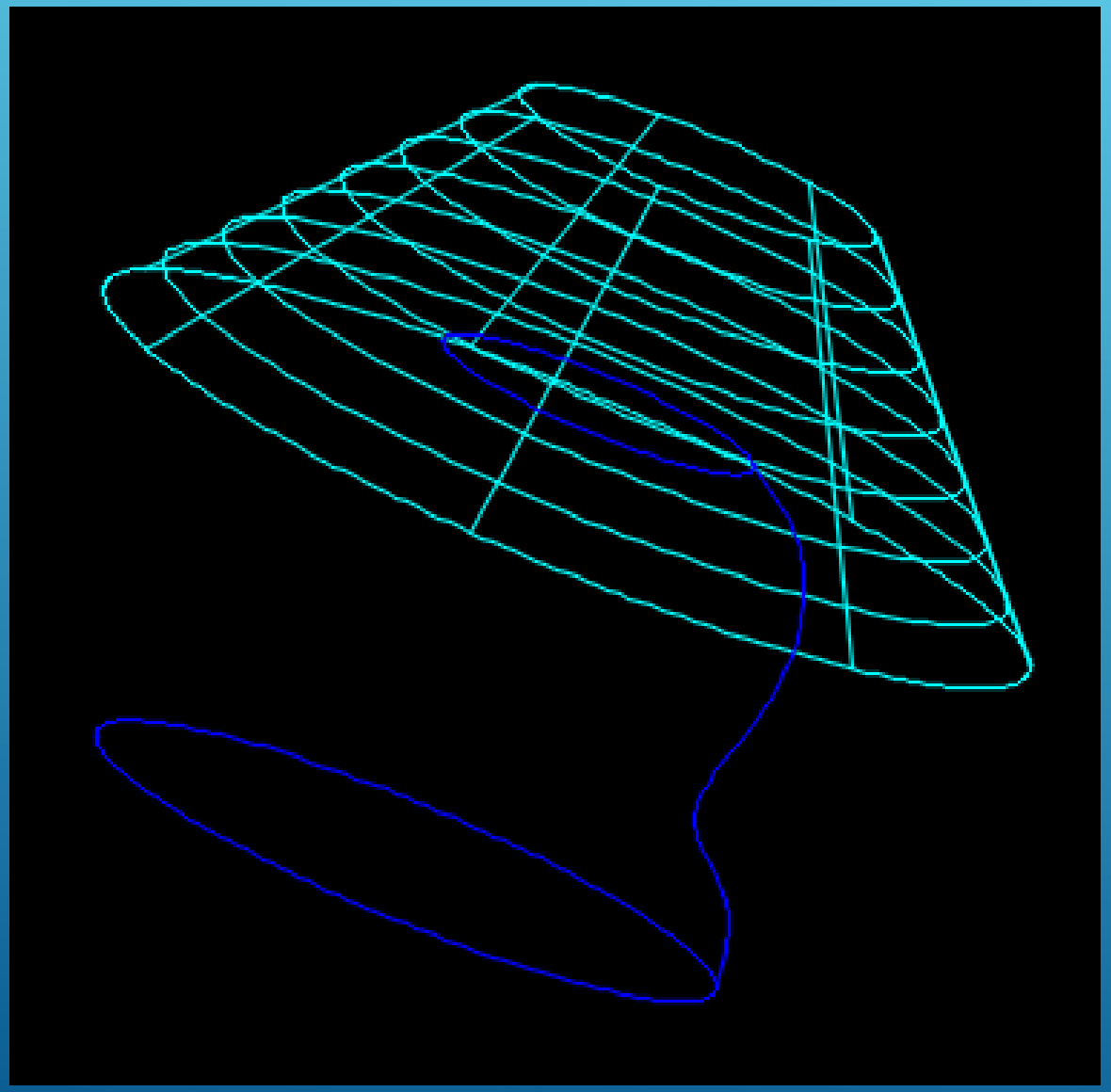
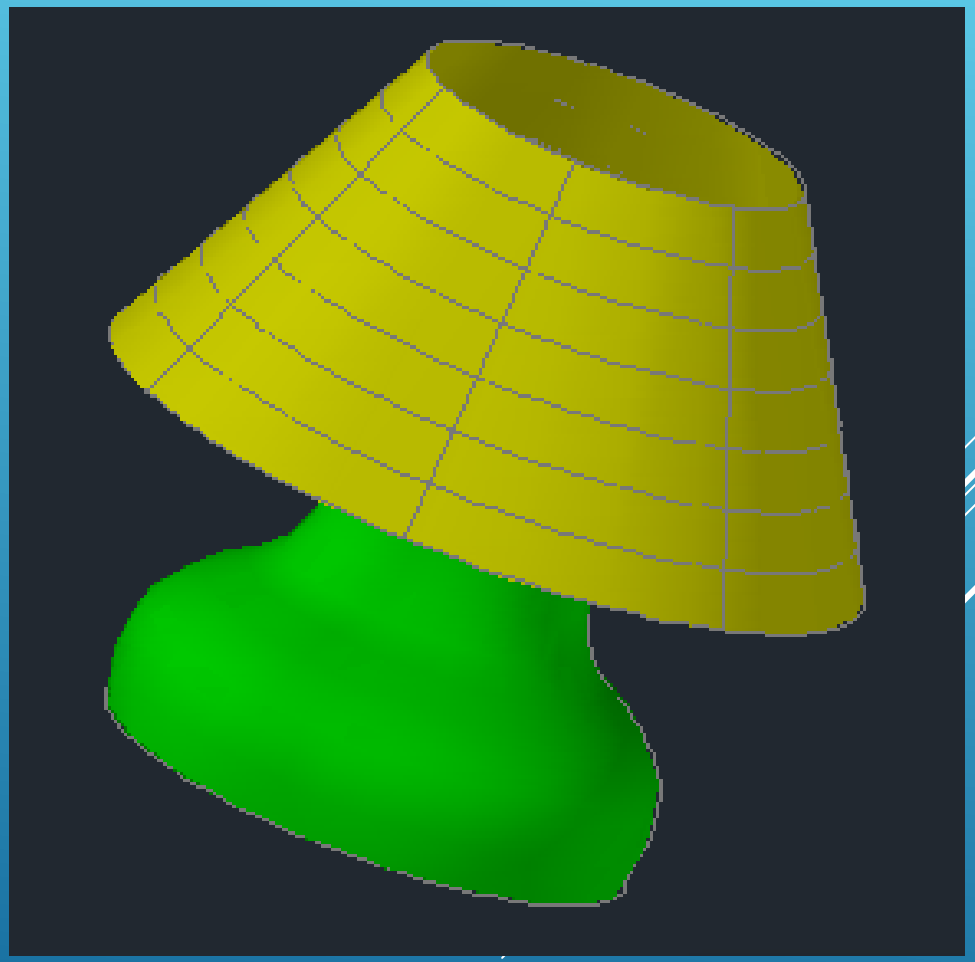
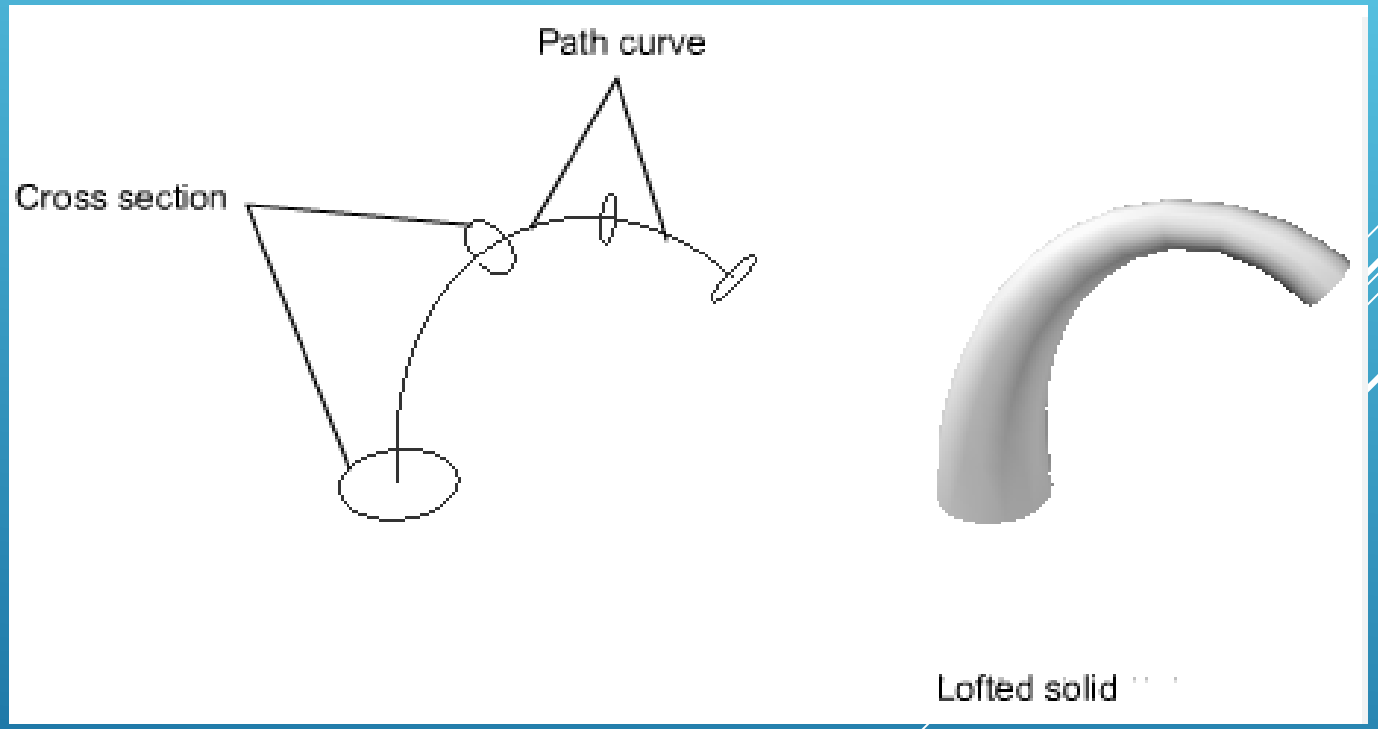




O comando **LOFT** é similar ao comando EXTRUDE mas é muito mais versátil: ao contrário do primeiro, onde a extrusão é efectuada sobre uma polyline fechada, neste caso a extrusão pode ser efectuada sobre diversas polylines fechadas e torna-las um objecto contínuo.



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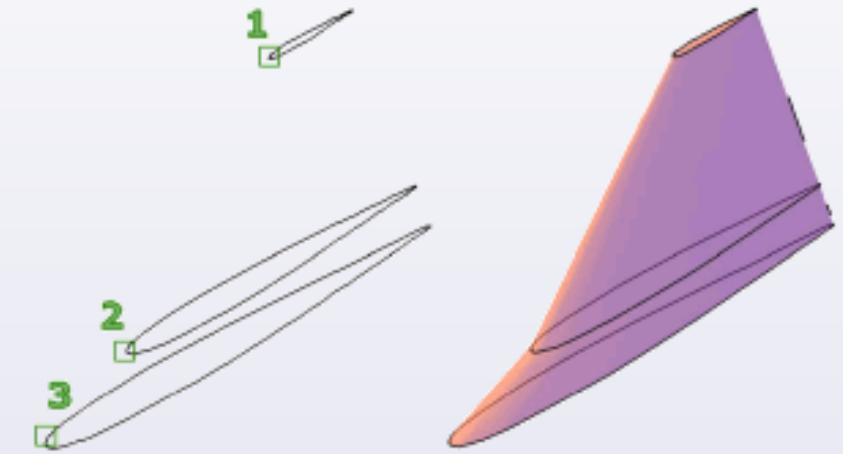
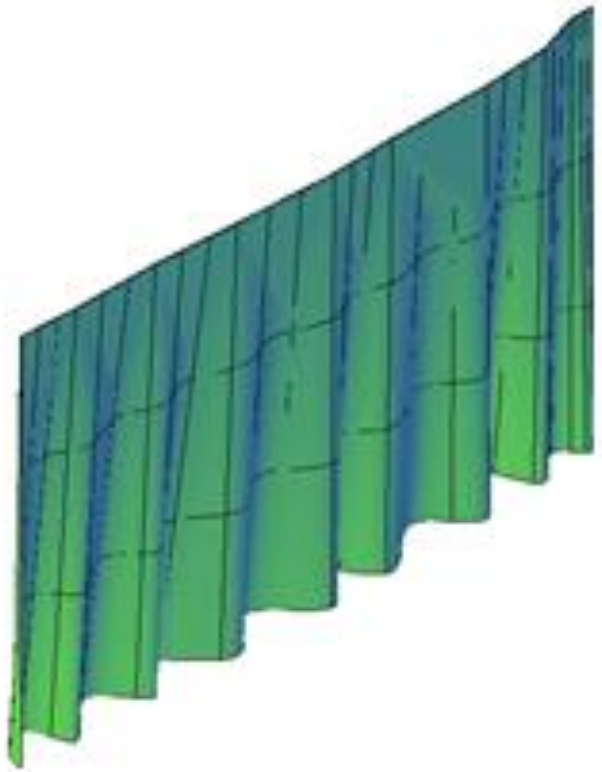
Loft

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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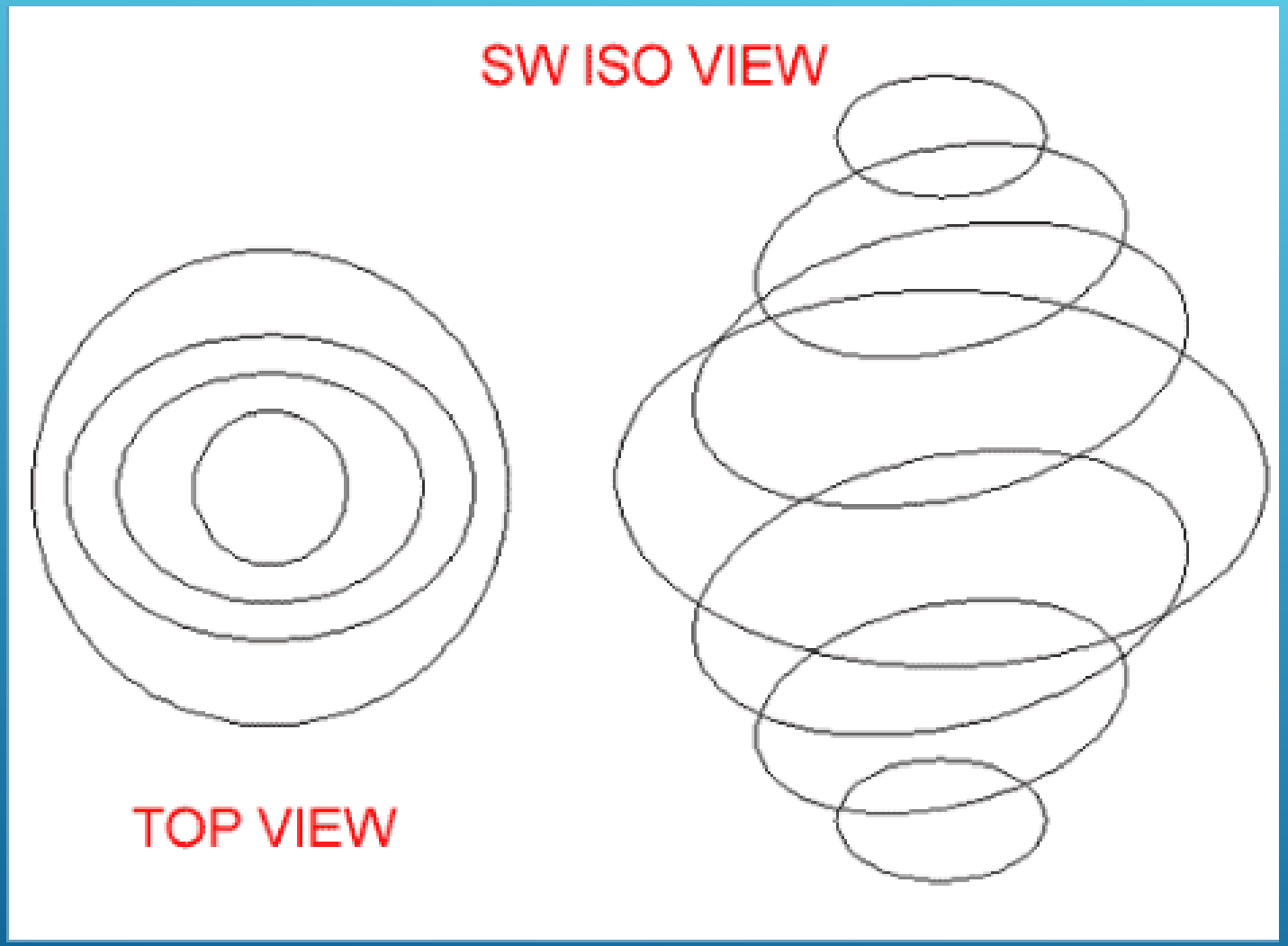

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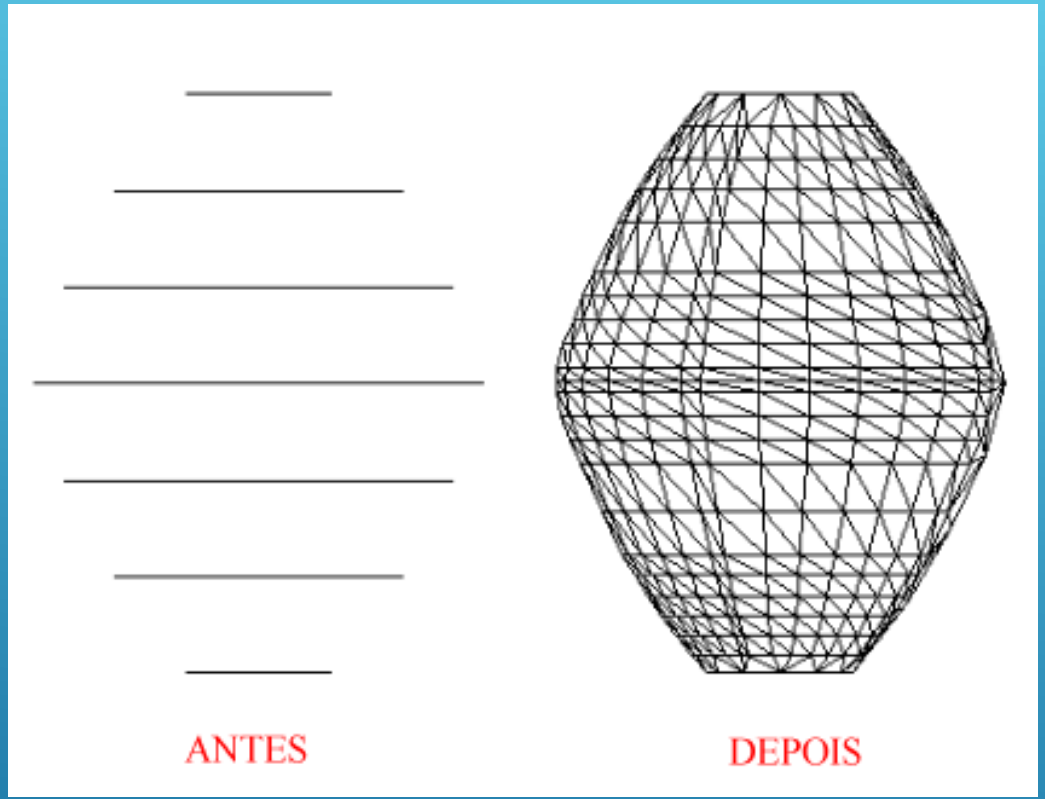

https://www.youtube.com/watch?v=-csG9hukV0w

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

https://www.youtube.com/watch?v=oq1332L_7U8

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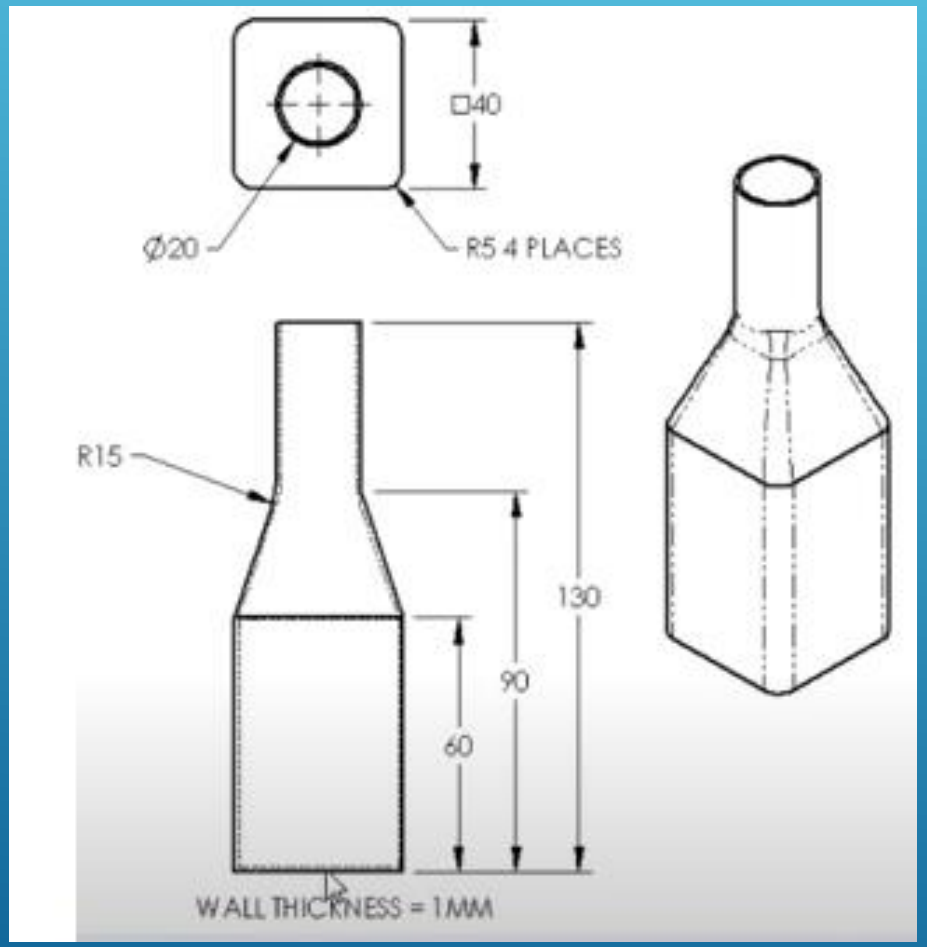

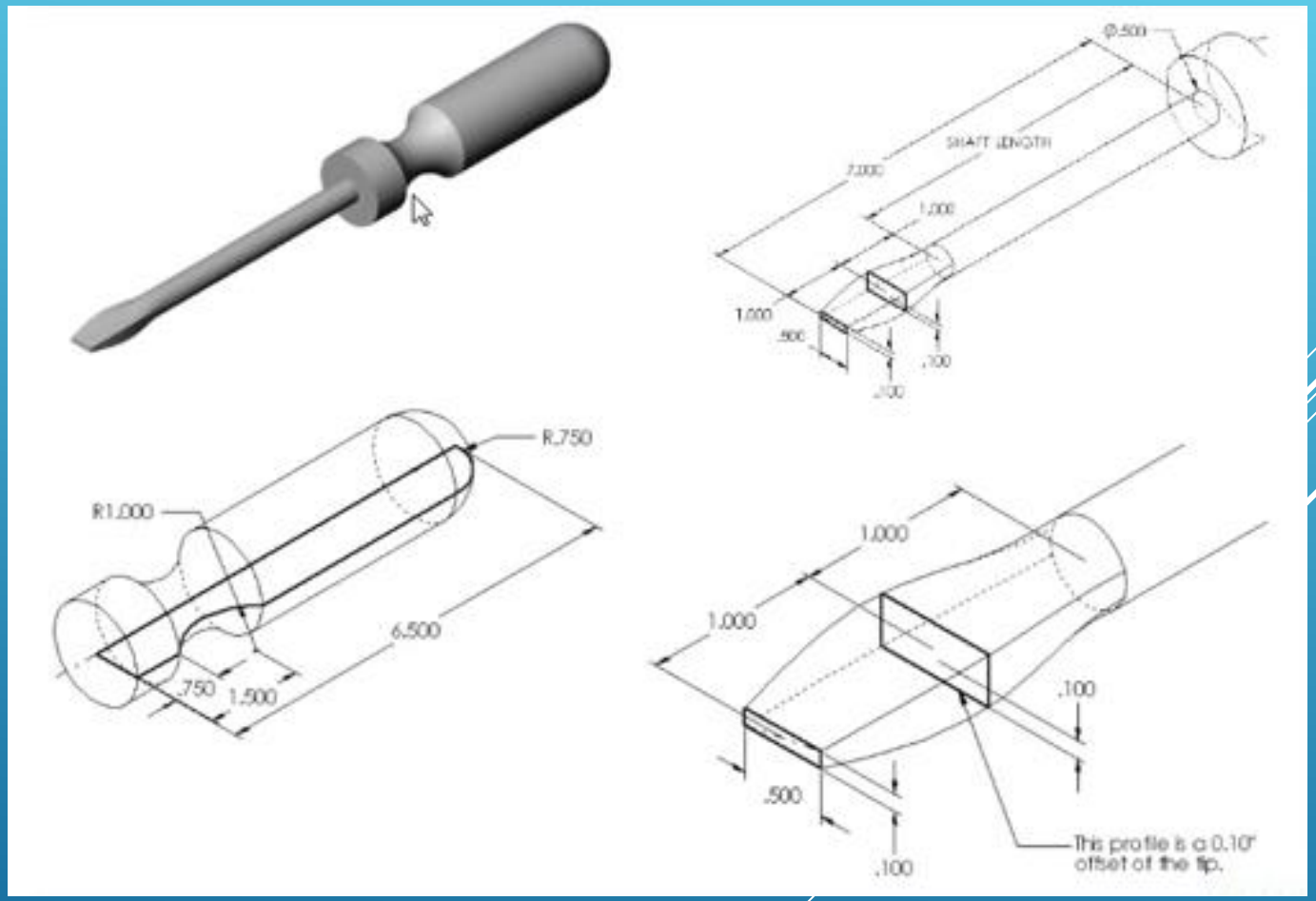

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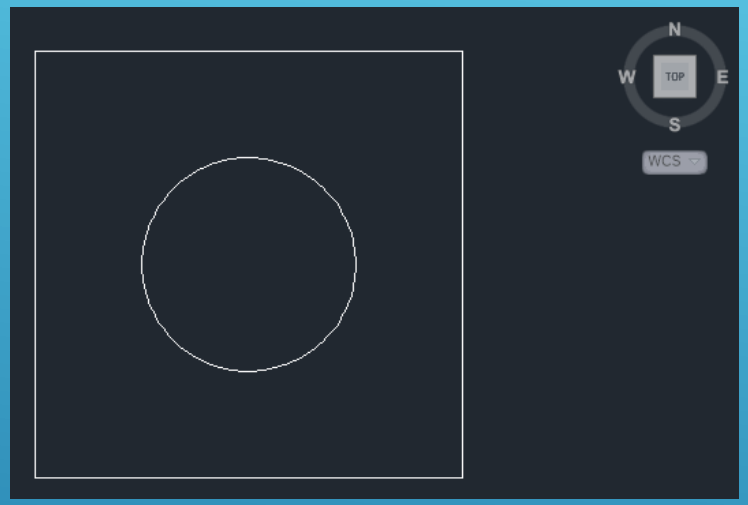
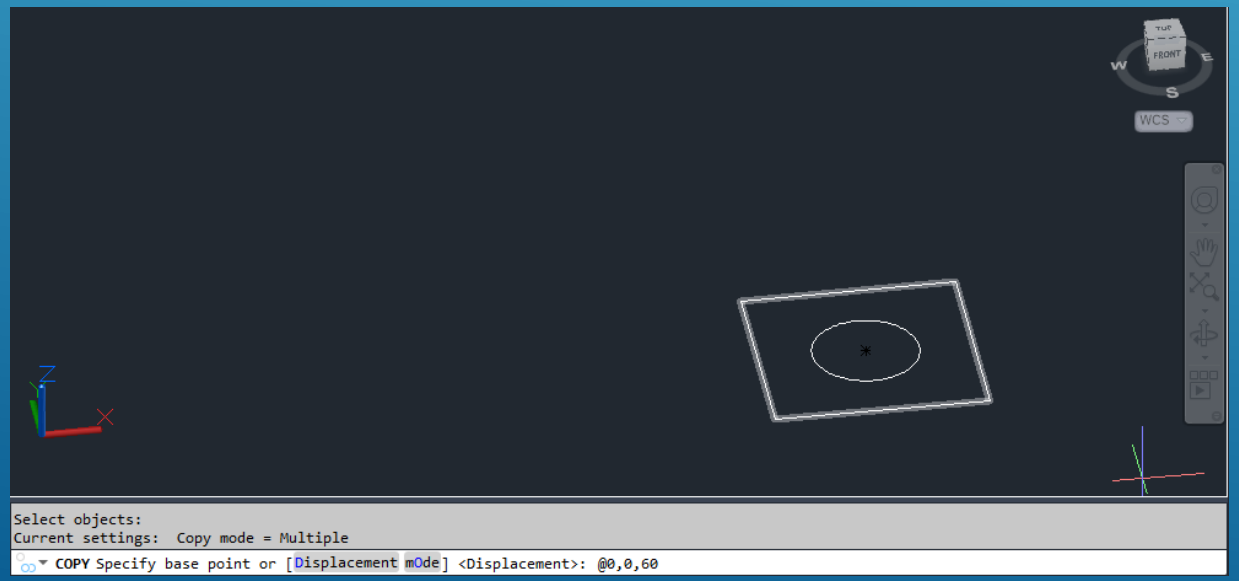
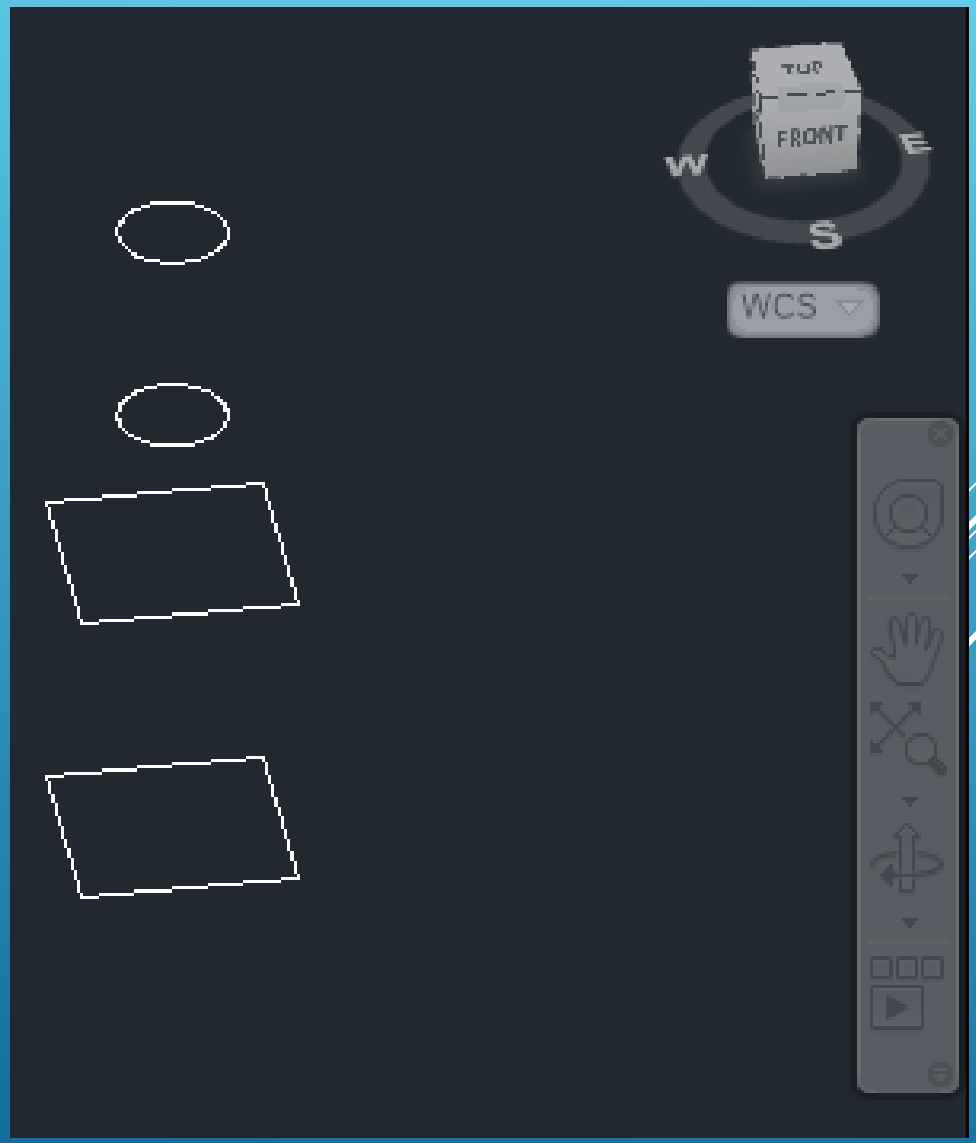


Rectangle
first point
dimensions
40
40

Circle
radius
10





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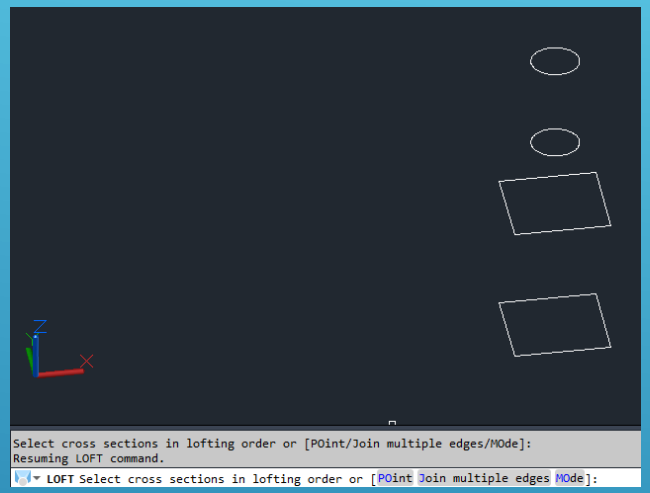

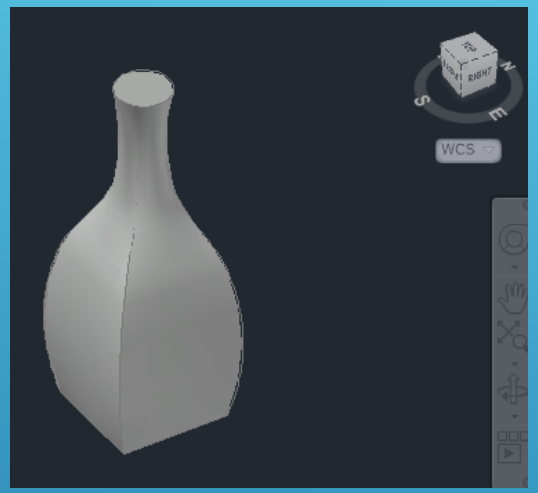

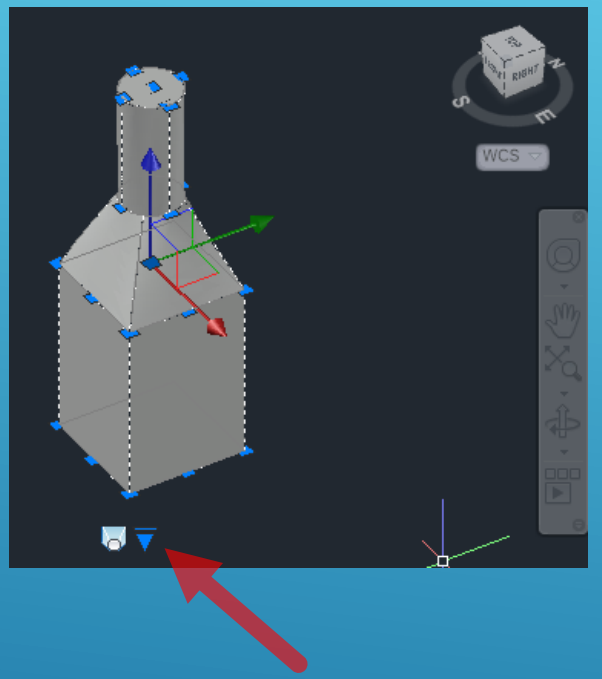

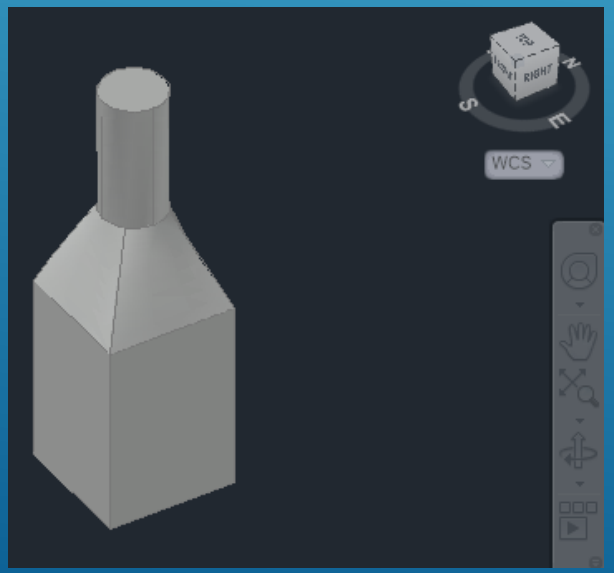

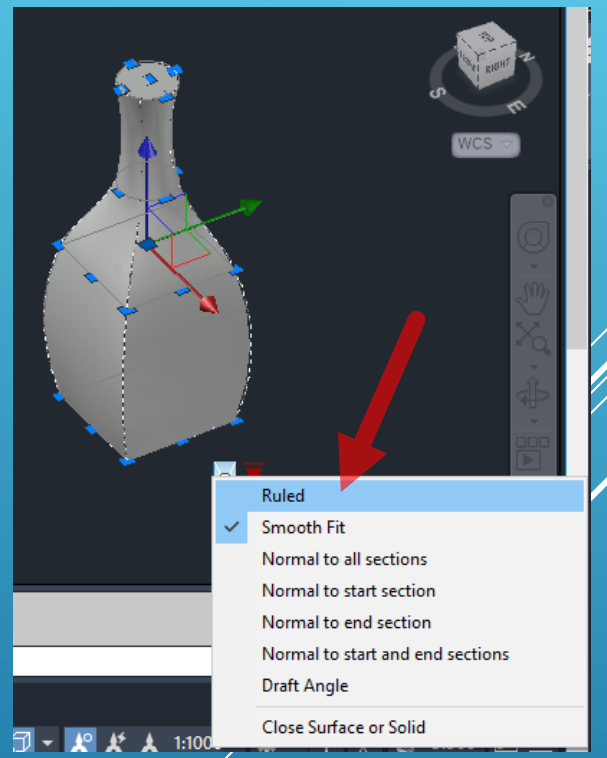

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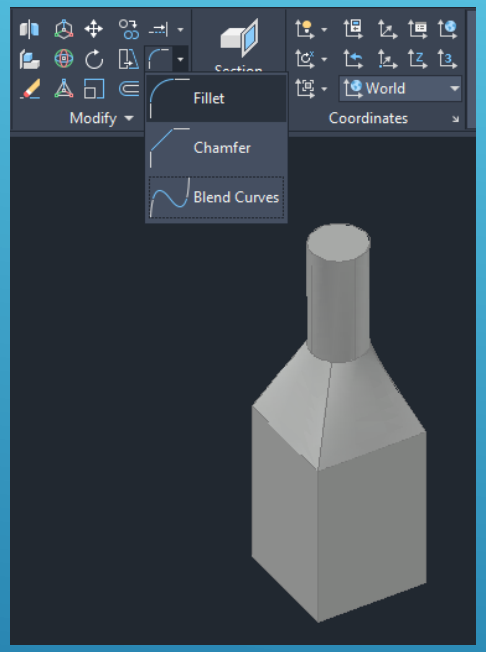

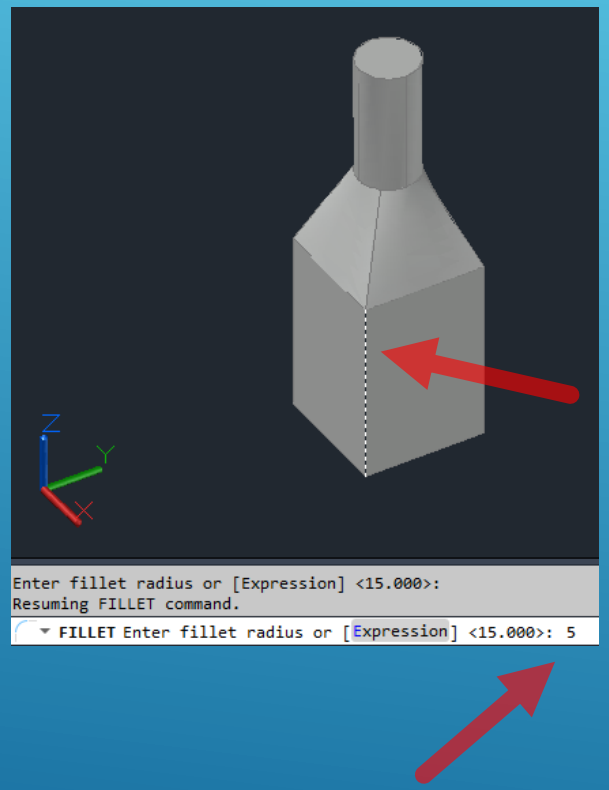

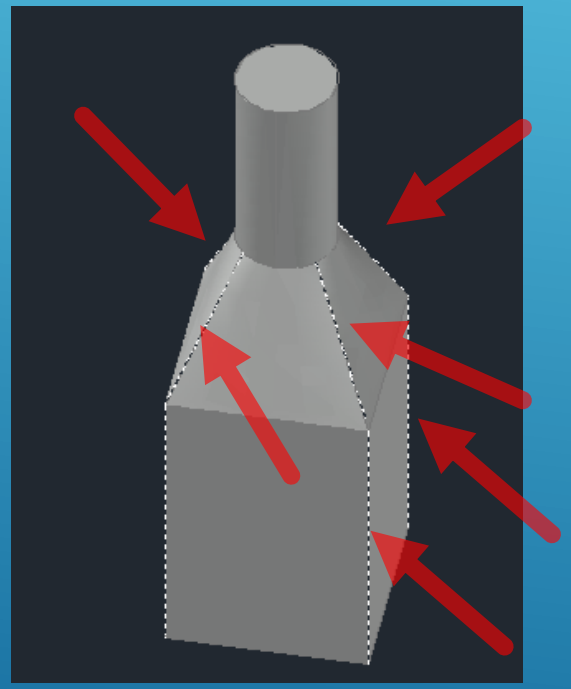

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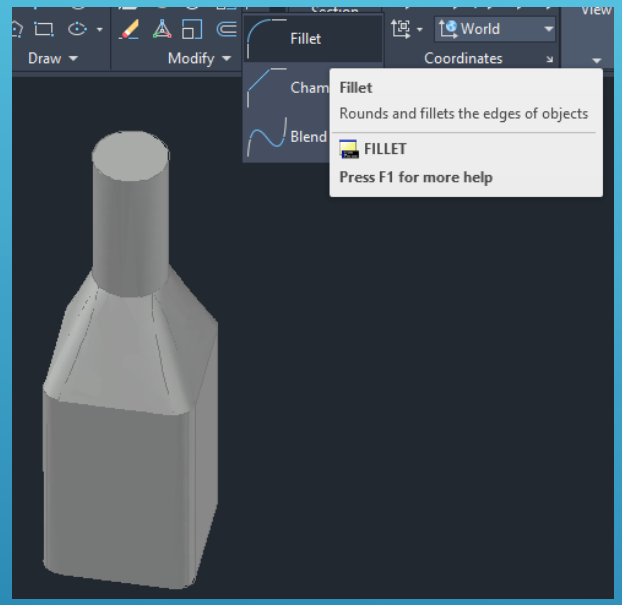

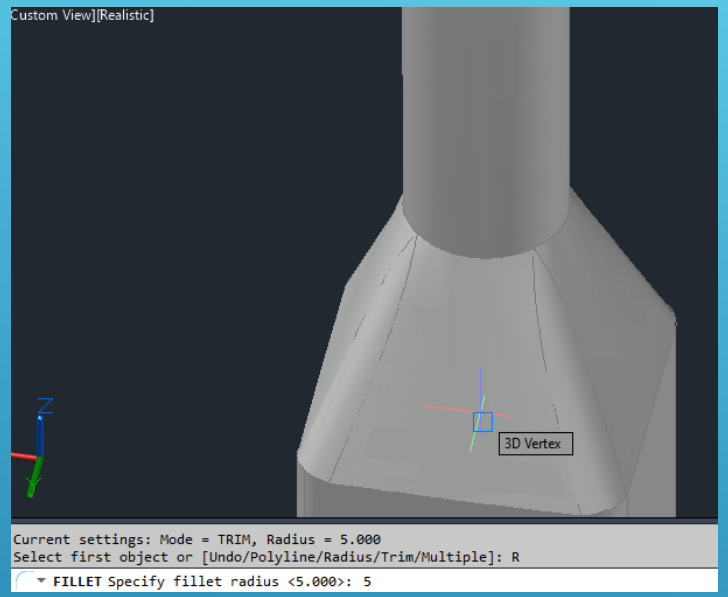

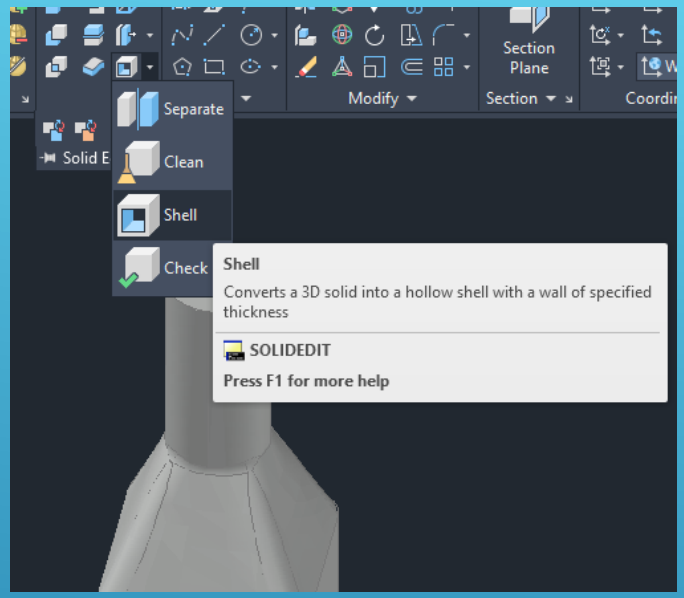

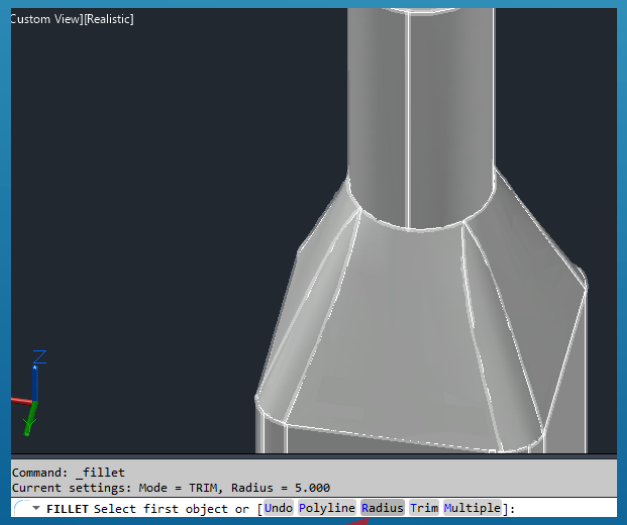

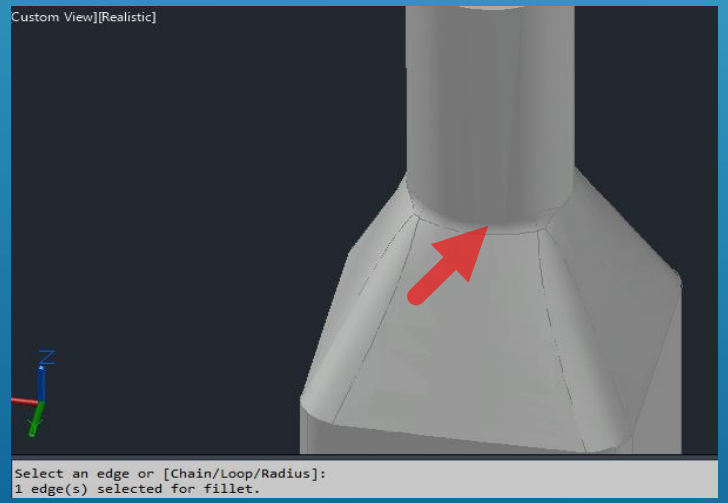

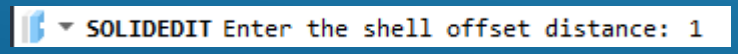

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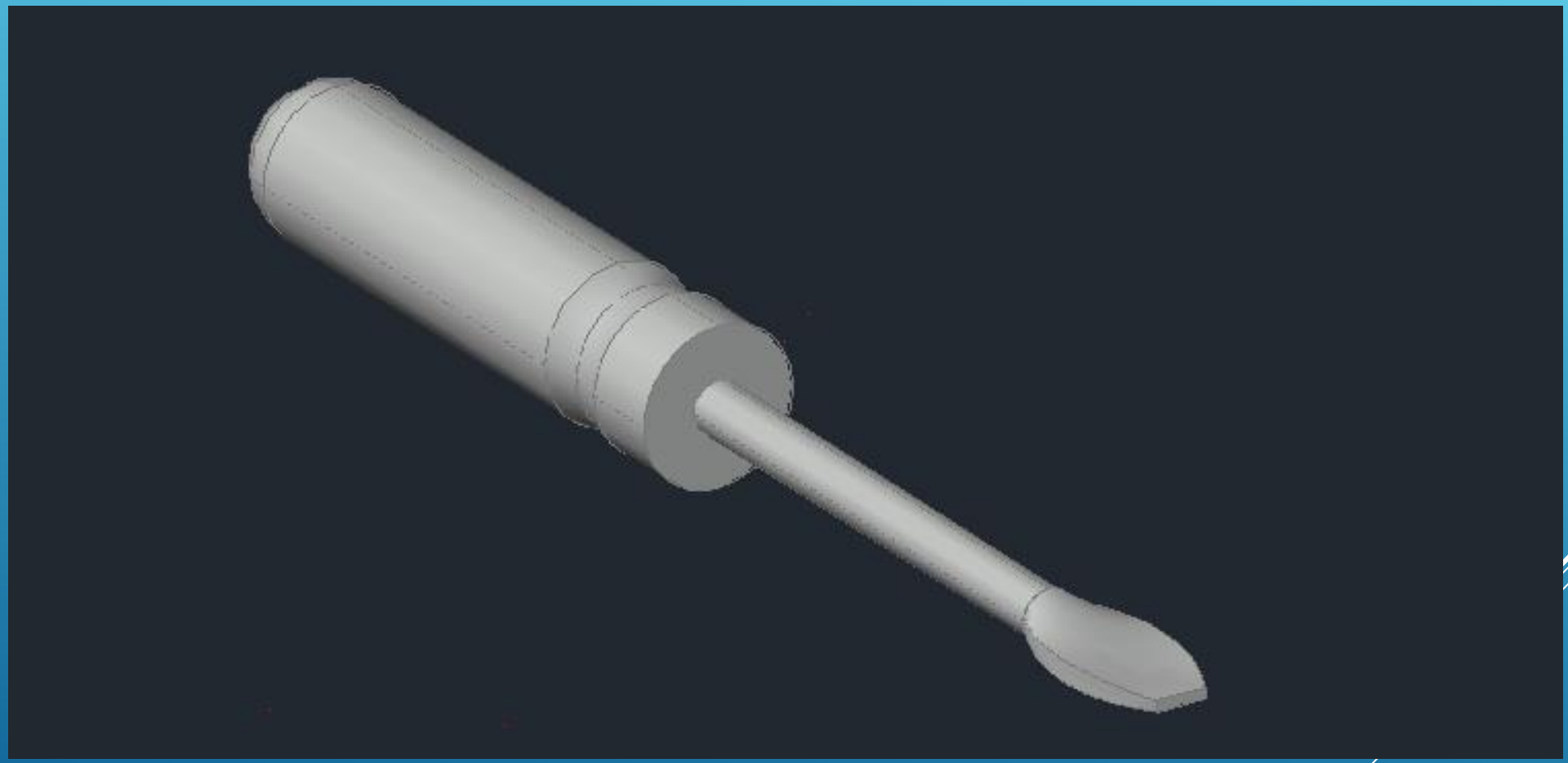

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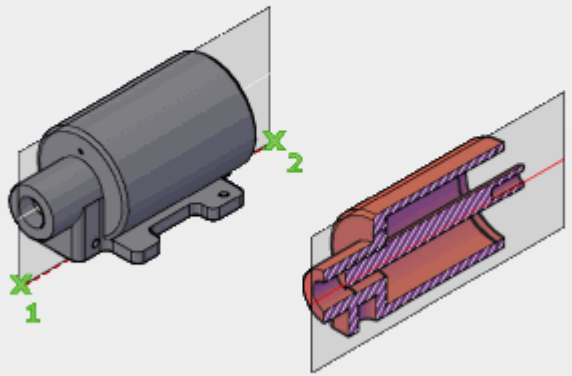
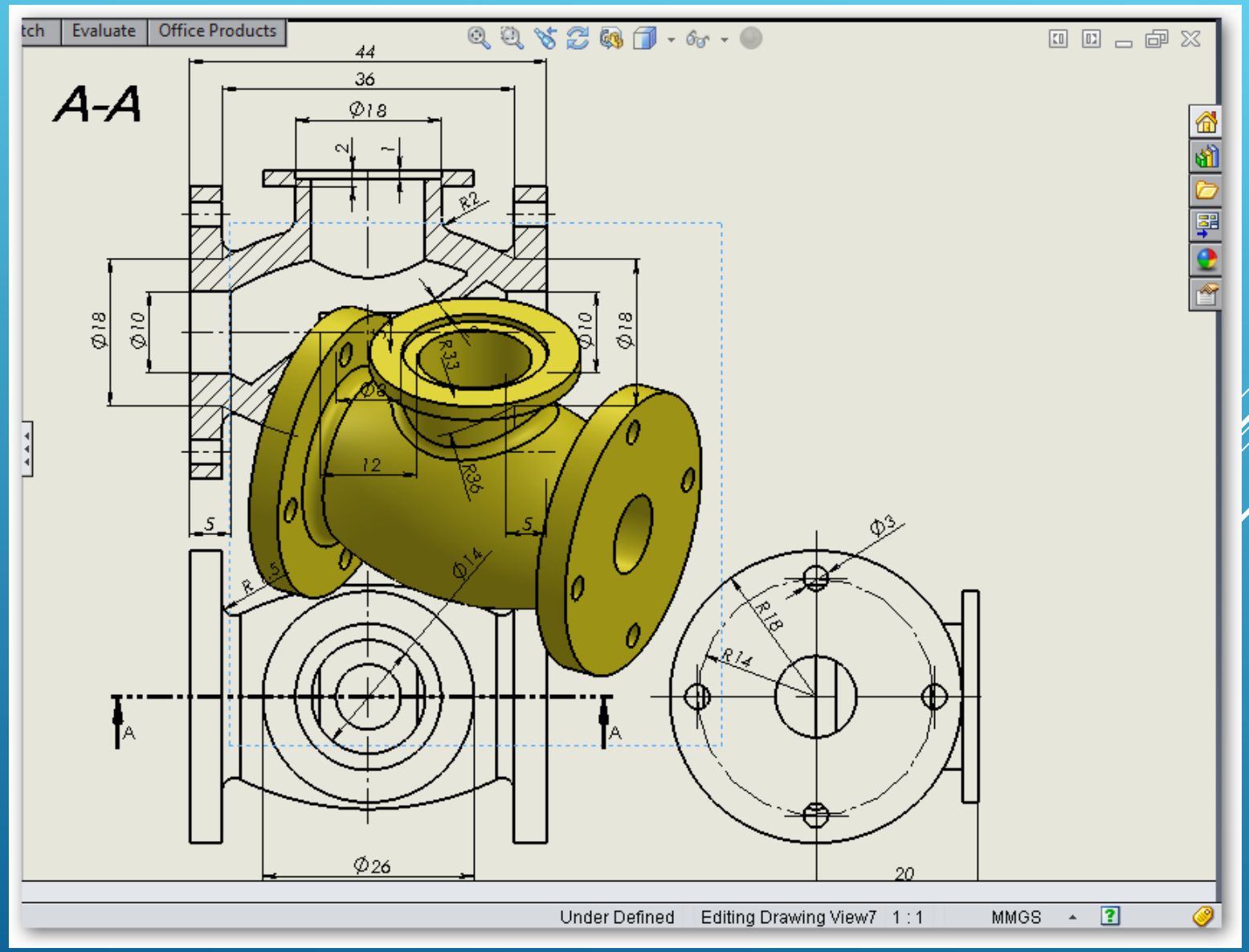
Creates a section object that acts as a cutting plane through 3D objects

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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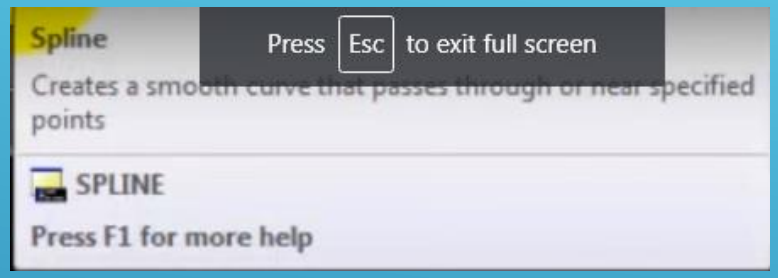

AULA 5 Desenho Técnico Assistido por Computador

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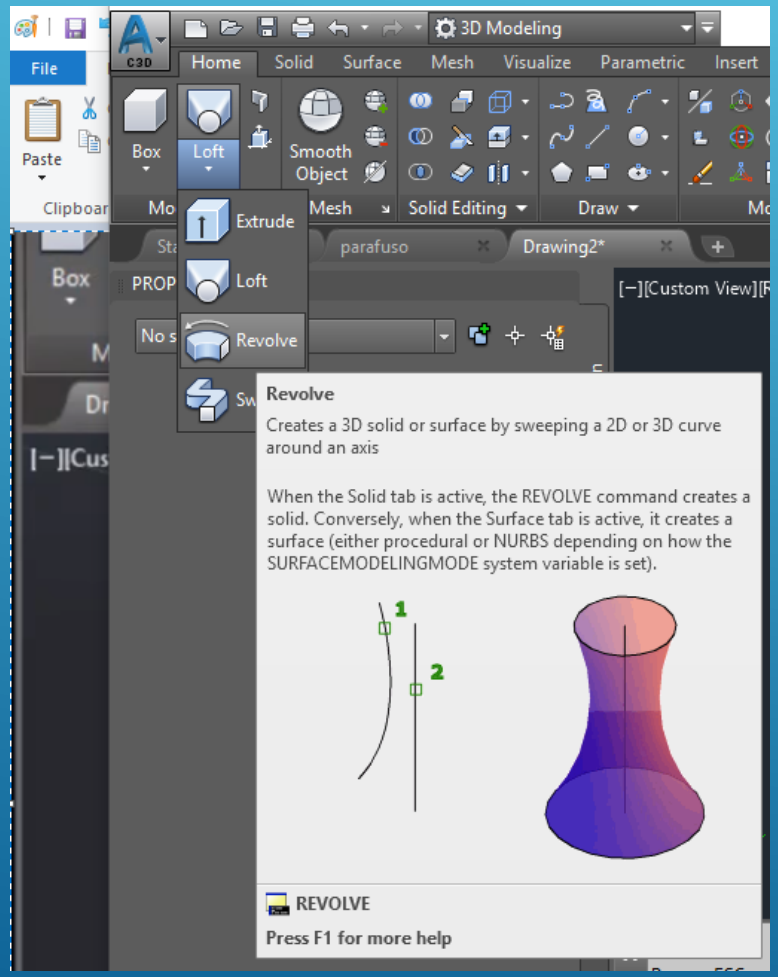

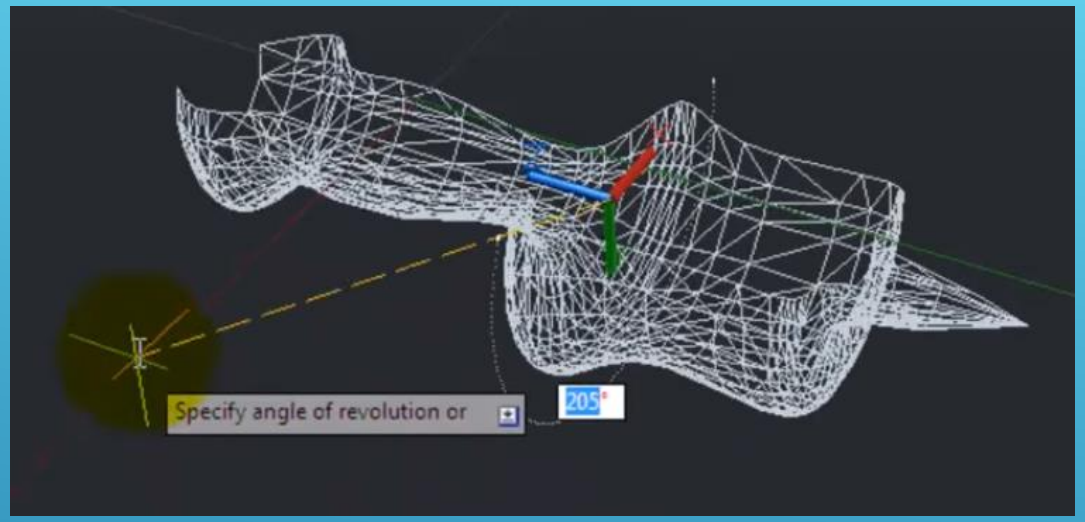

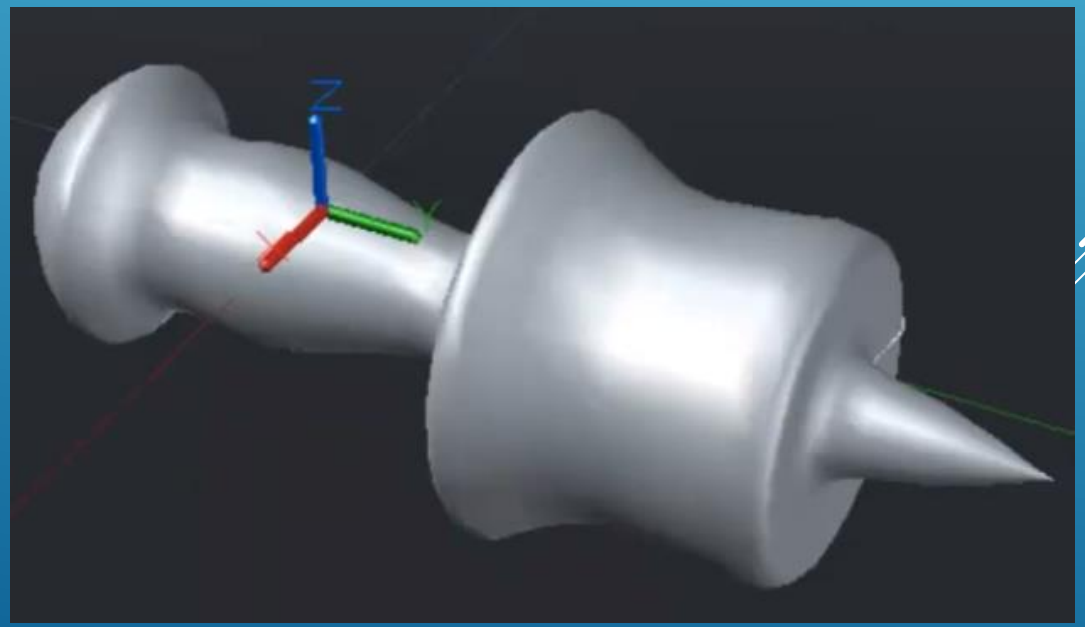

AULA 5 Desenho Técnico Assistido por Computador

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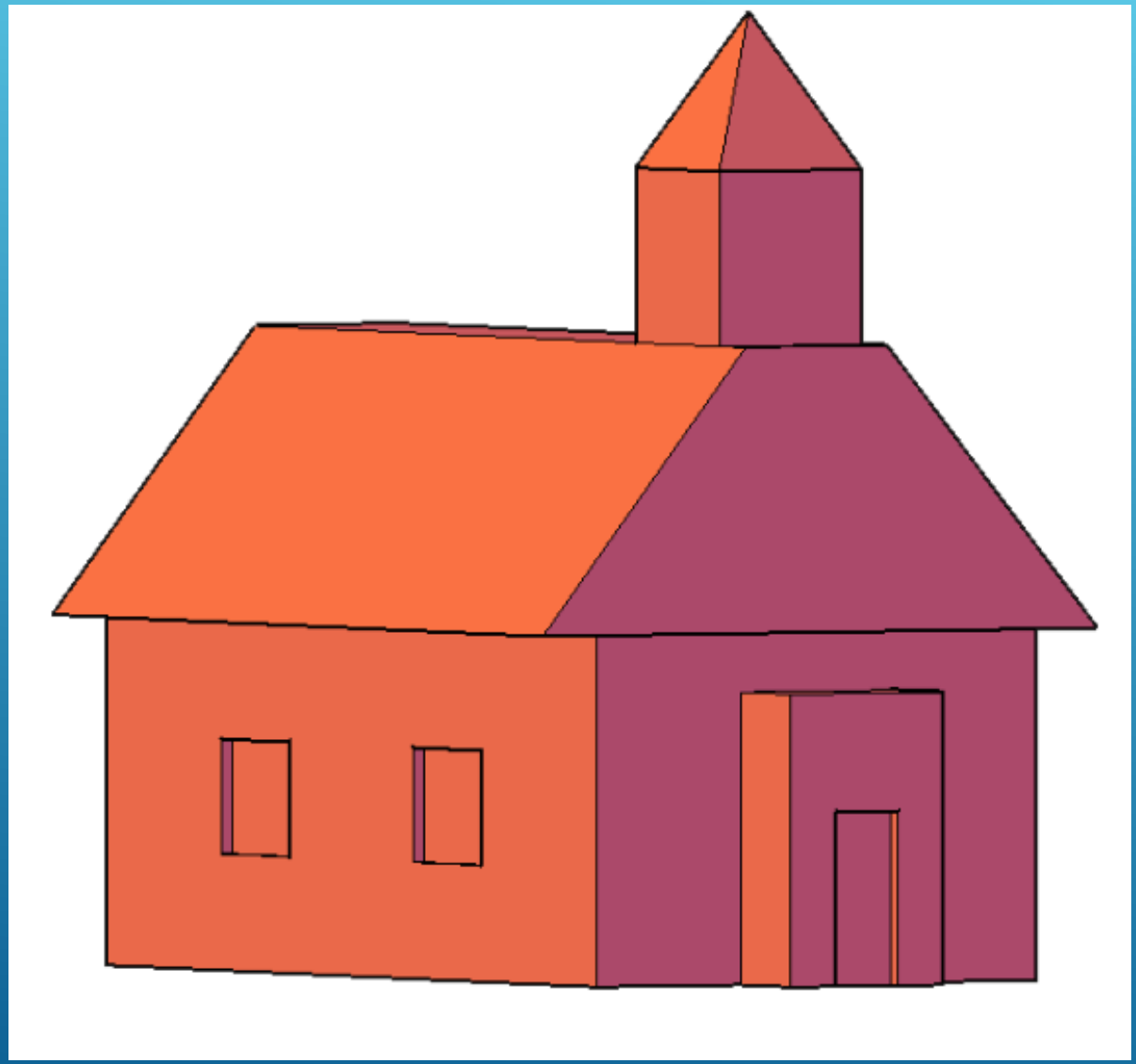

AULA 5 Desenho Técnico Assistido por Computador

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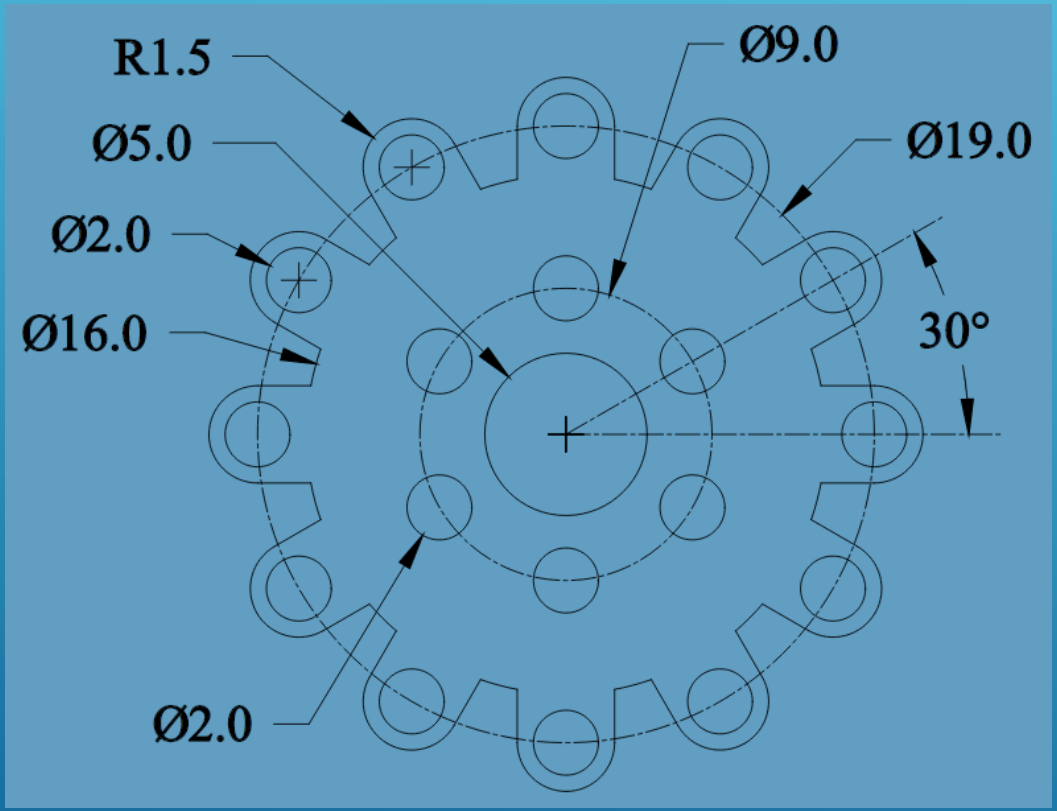

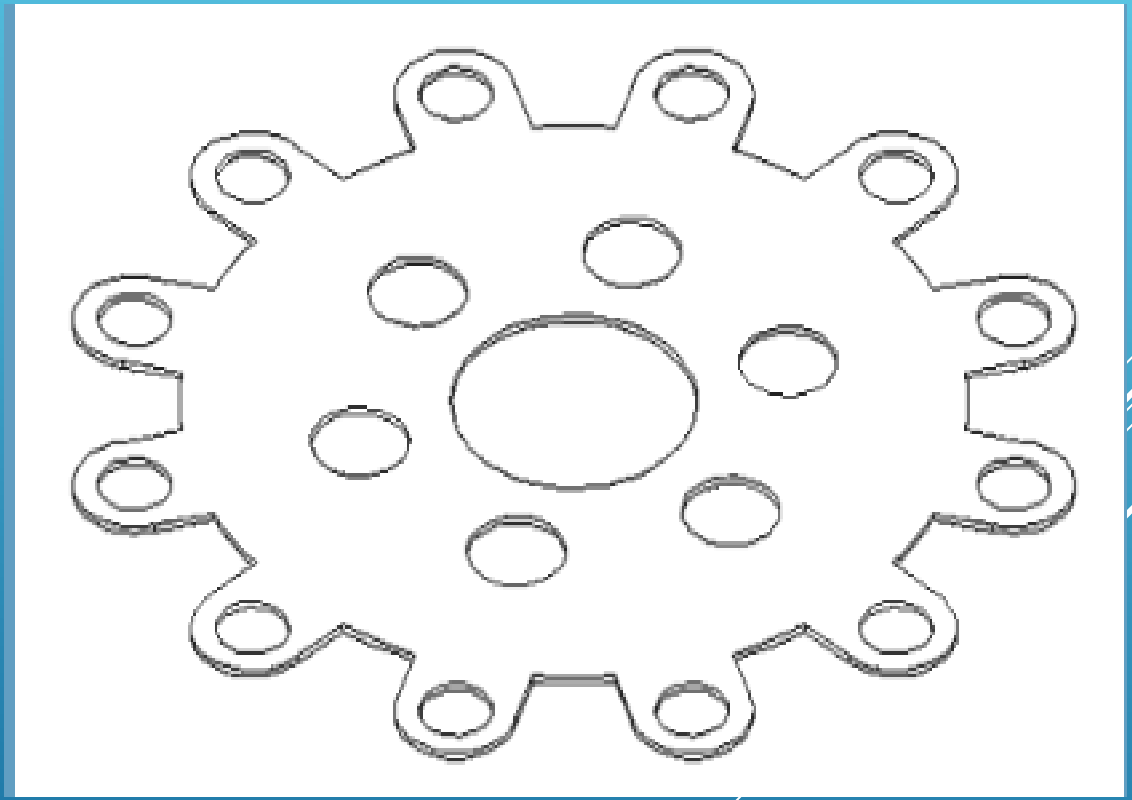

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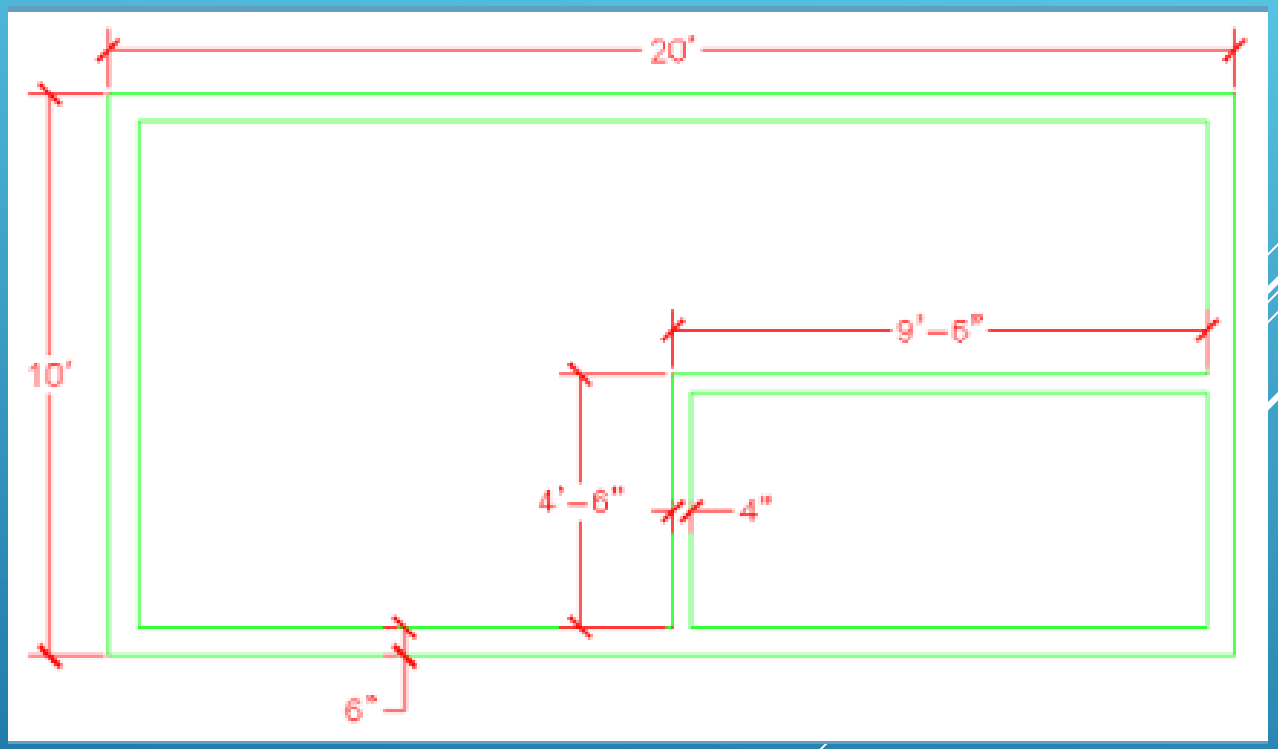

AULA 5 Desenho Técnico Assistido por Computador

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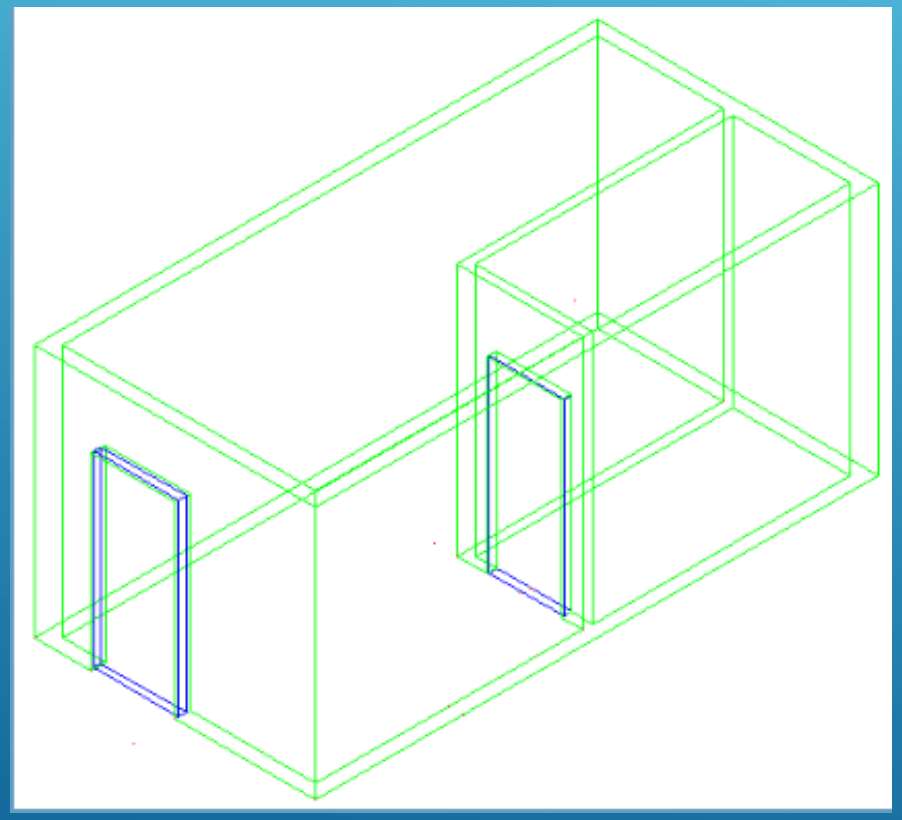

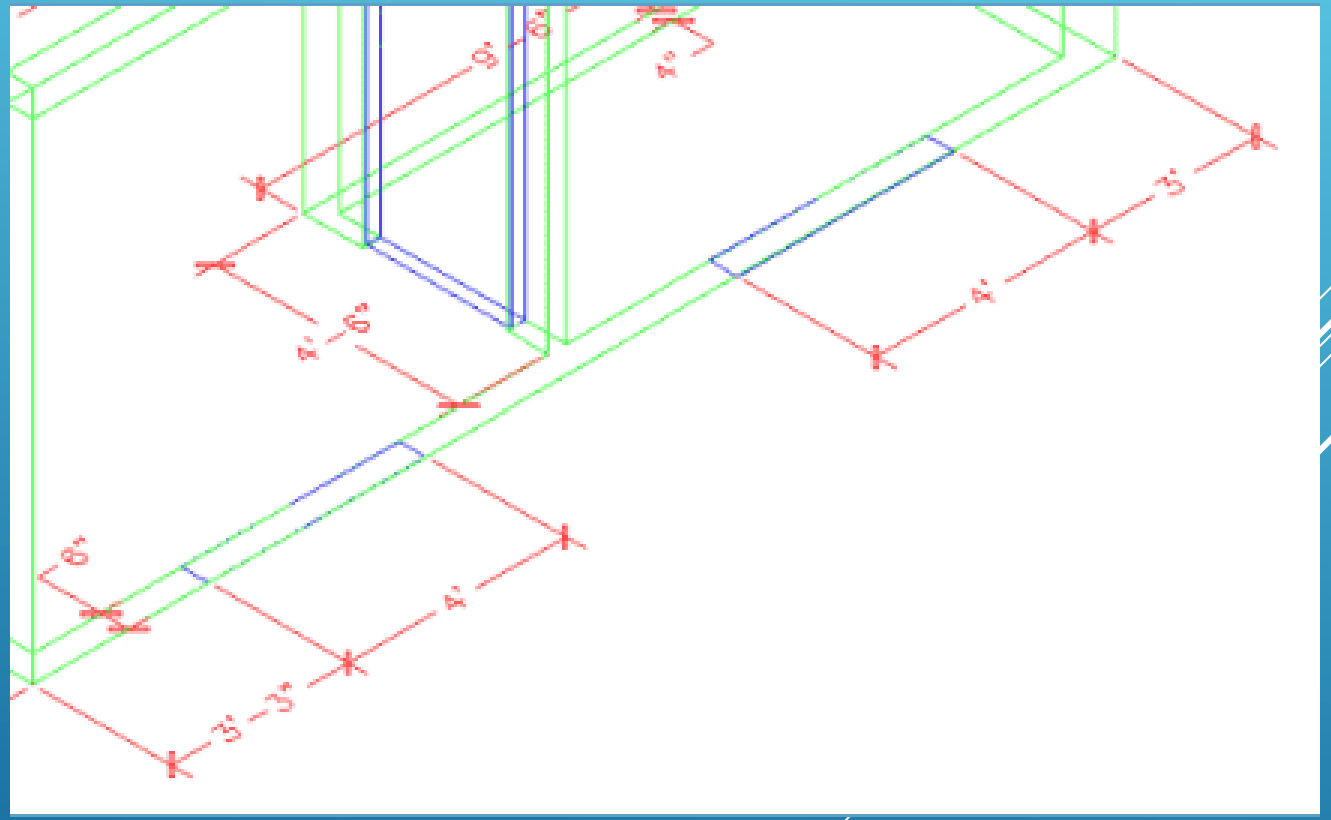

AULA 5 Desenho Técnico Assistido por Computador

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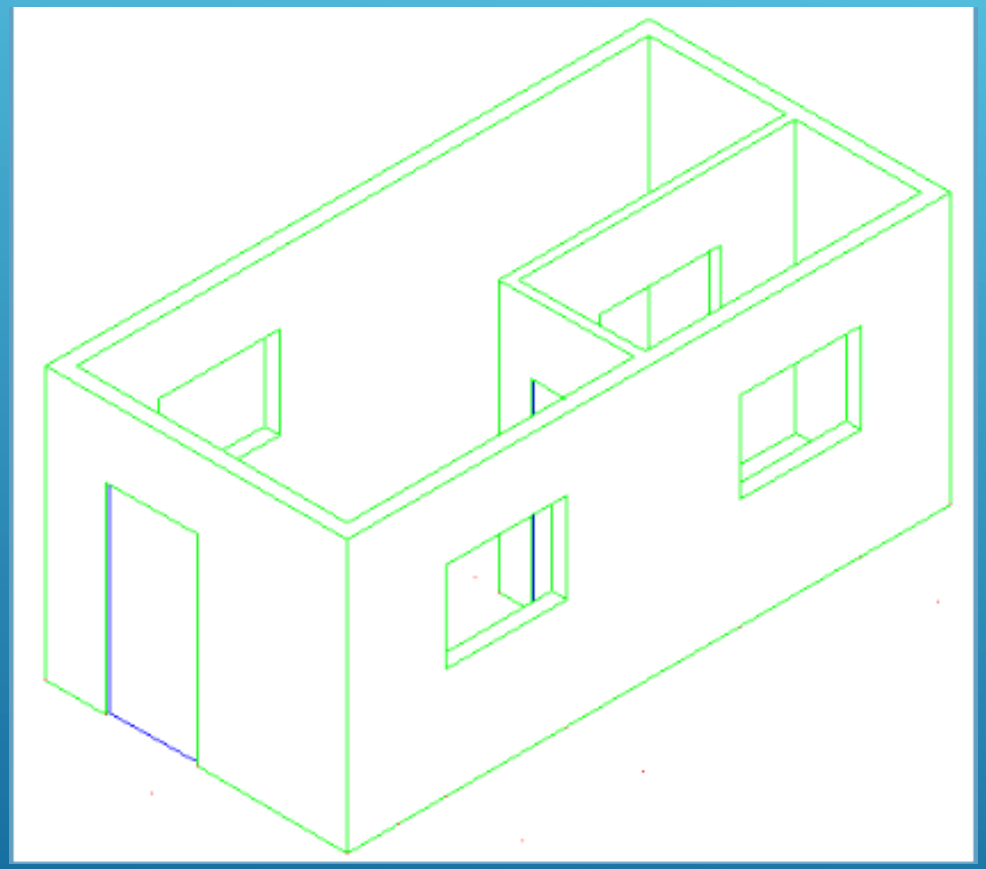

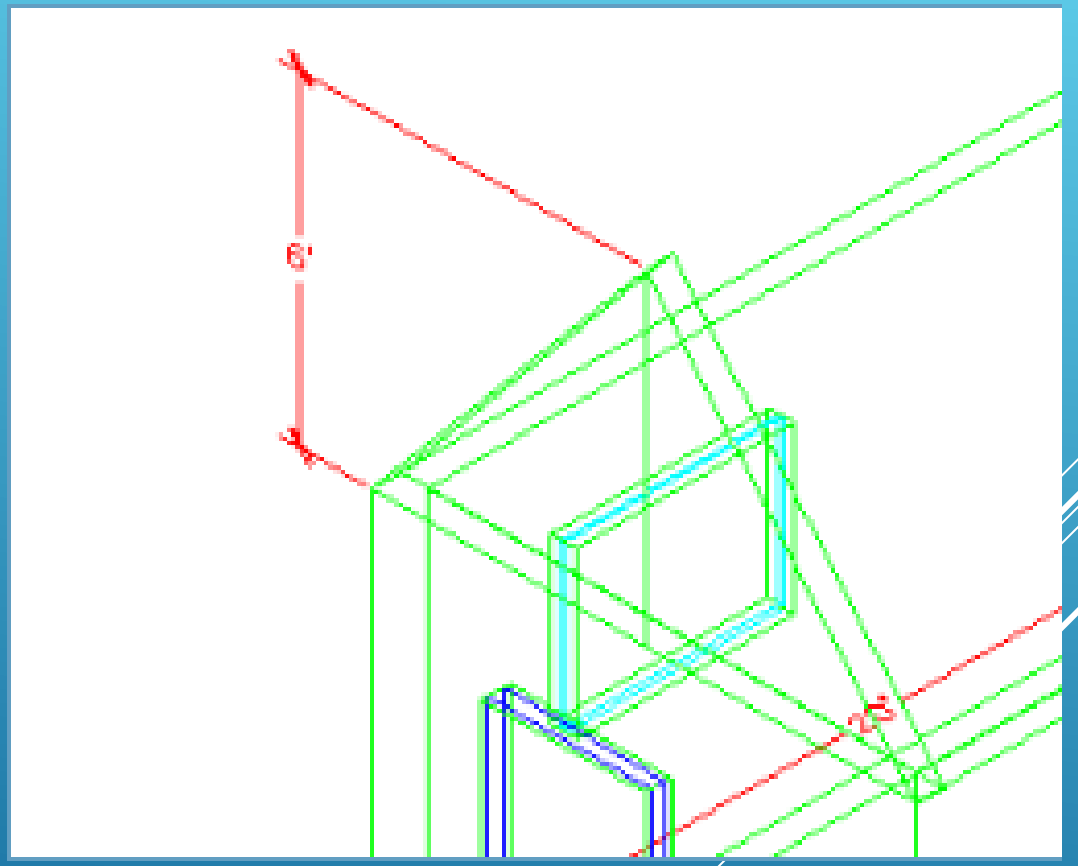

AULA 5 Desenho Técnico Assistido por Computador

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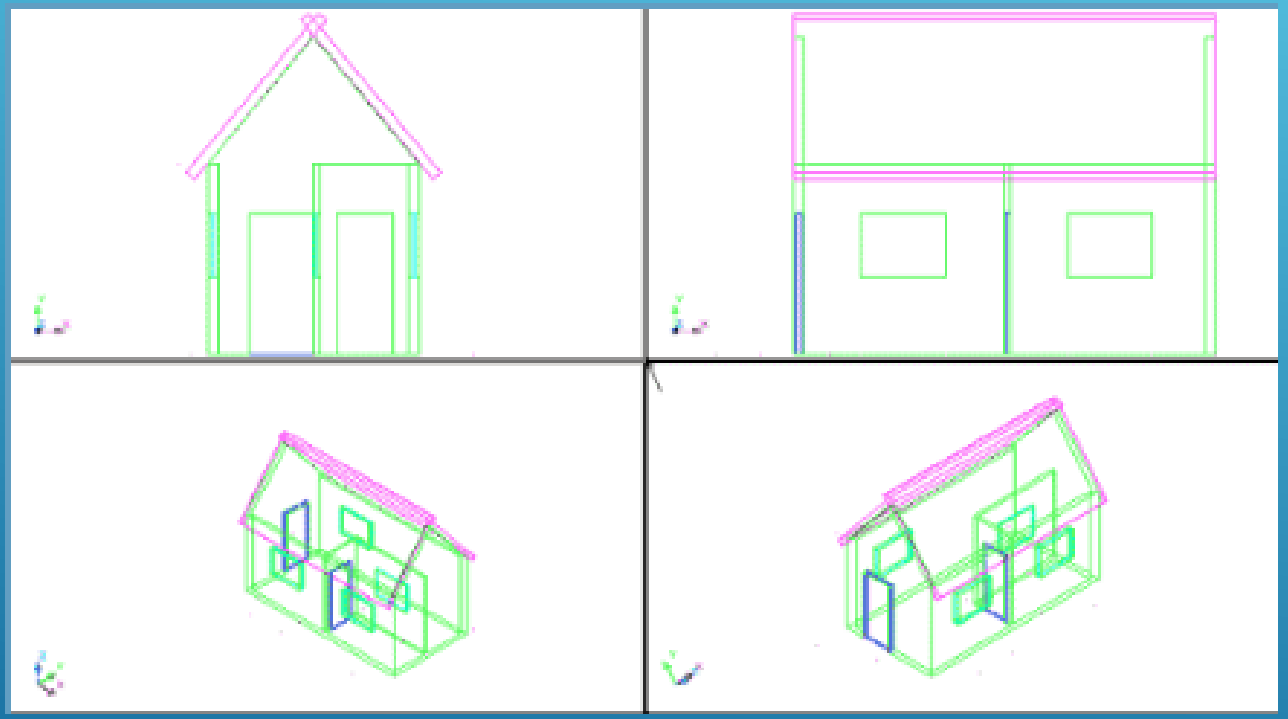

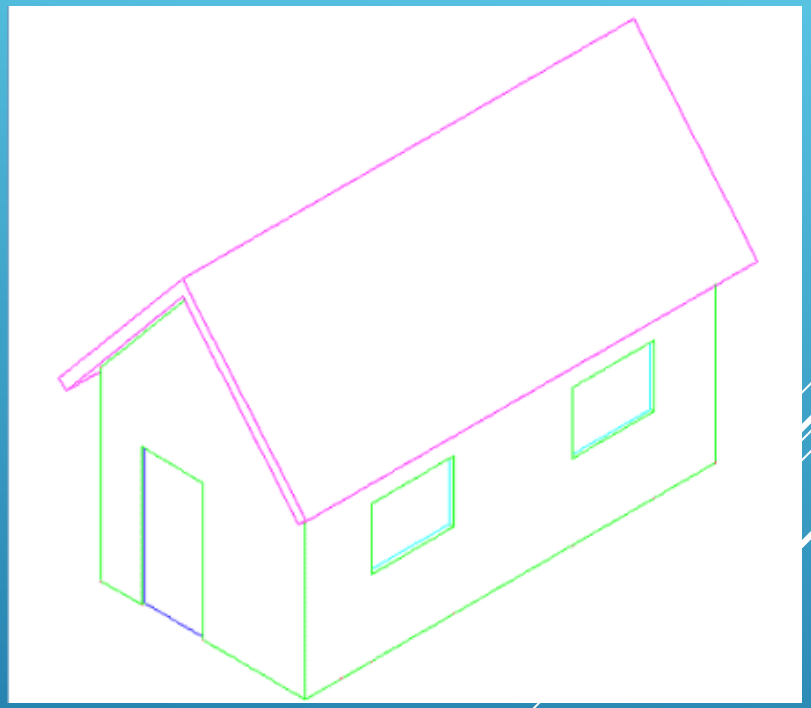

AULA 5 Desenho Técnico Assistido por Computador

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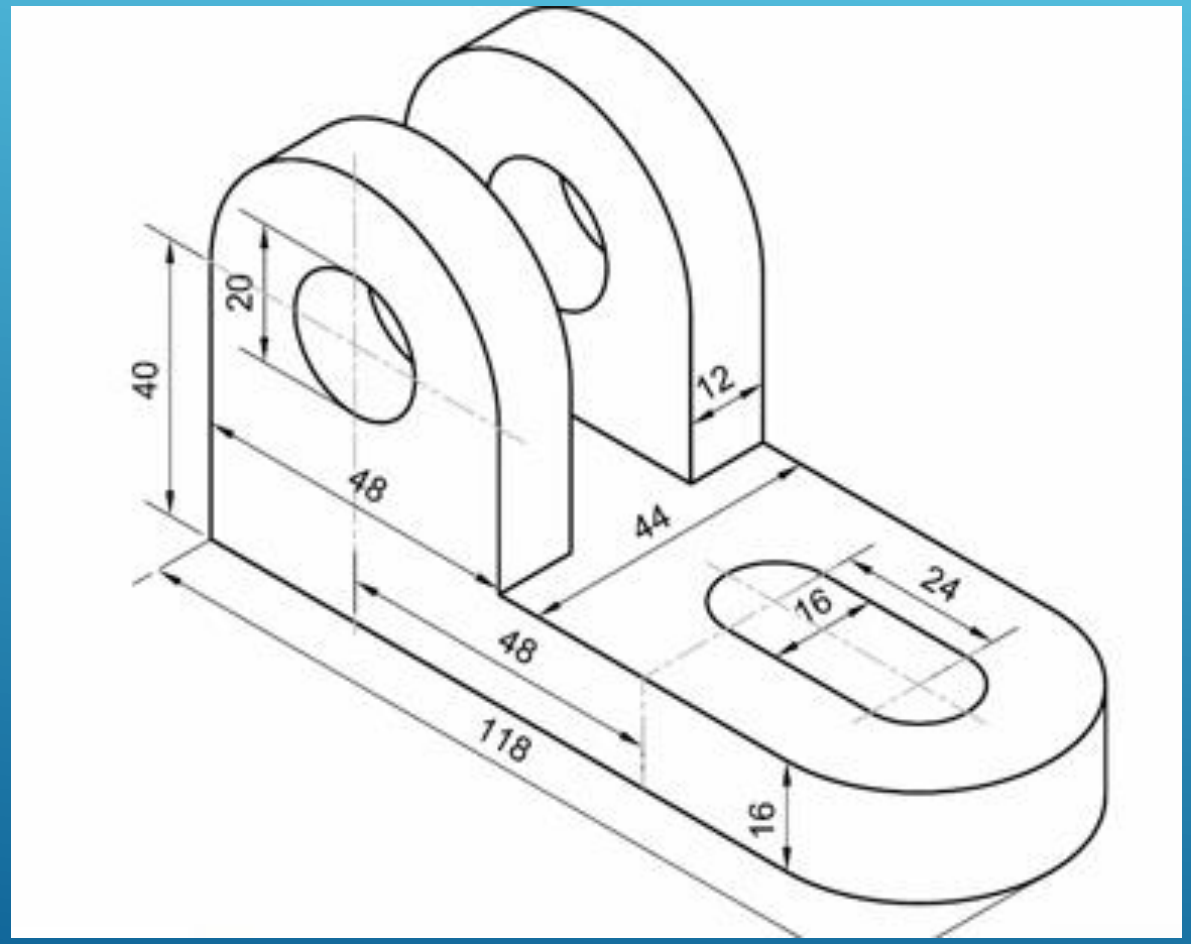

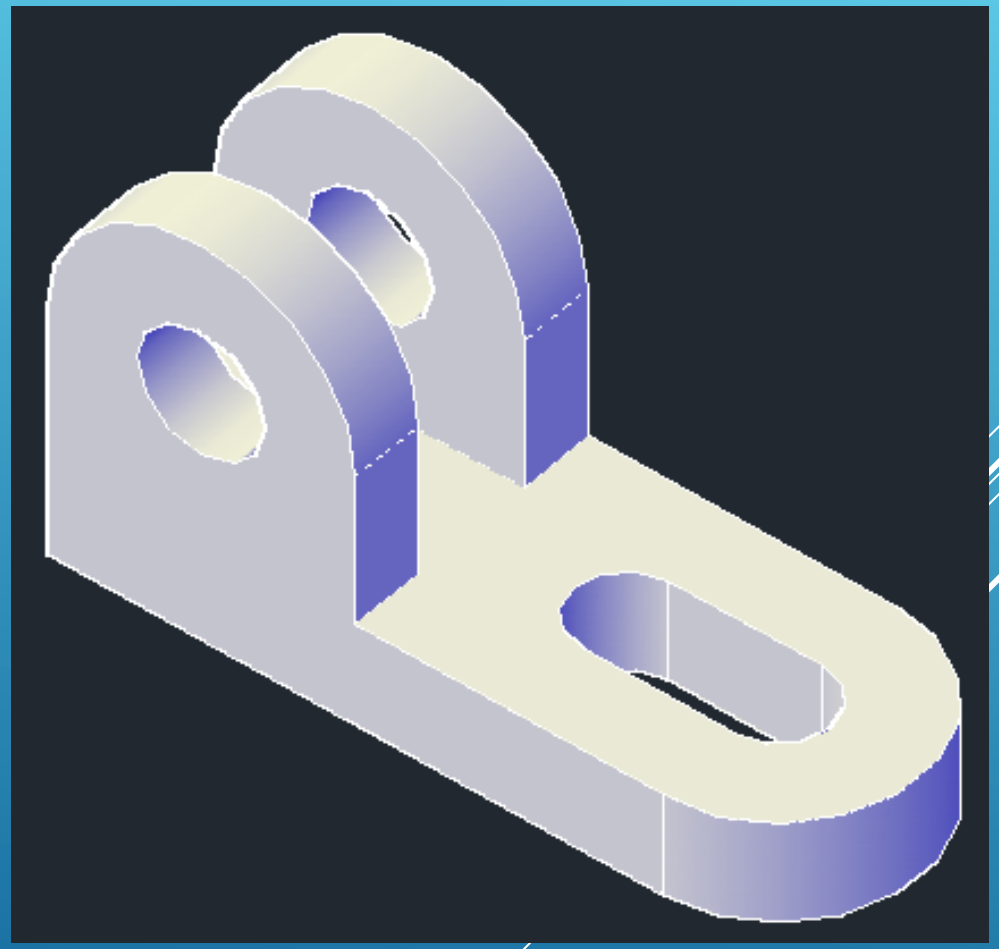

AULA 5 Desenho Técnico Assistido por Computador

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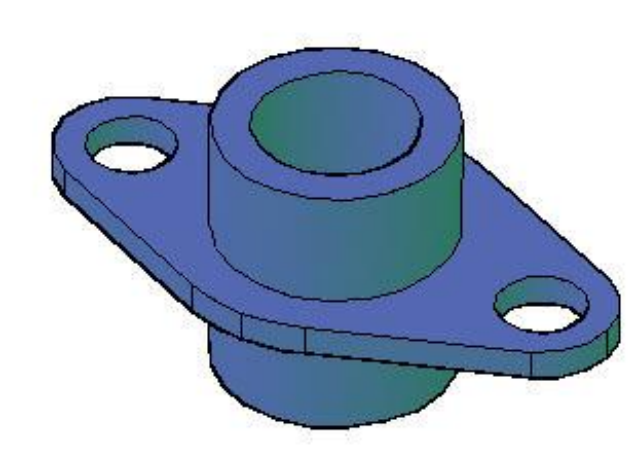

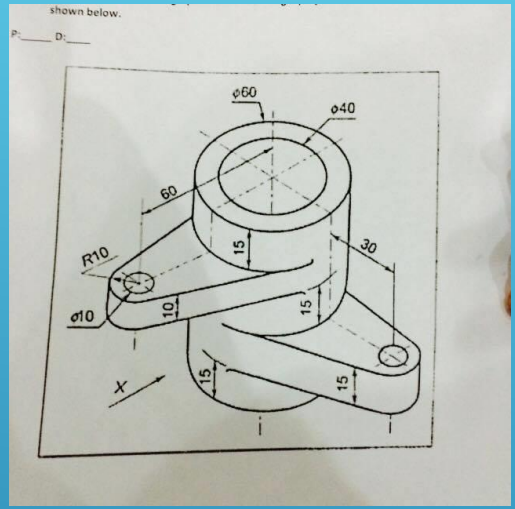

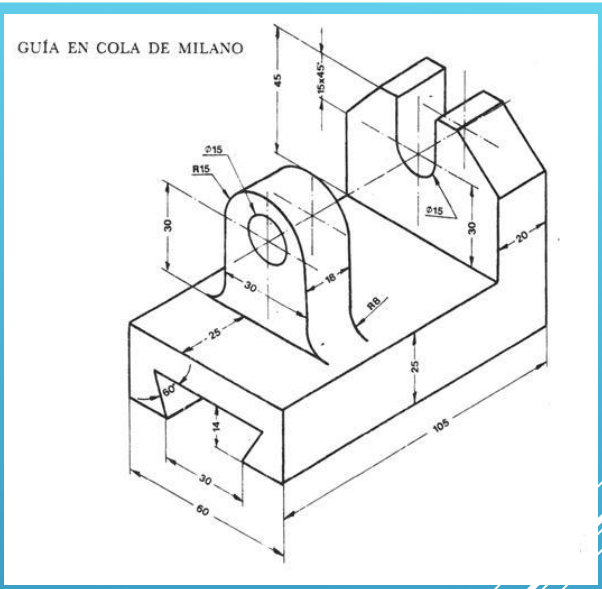

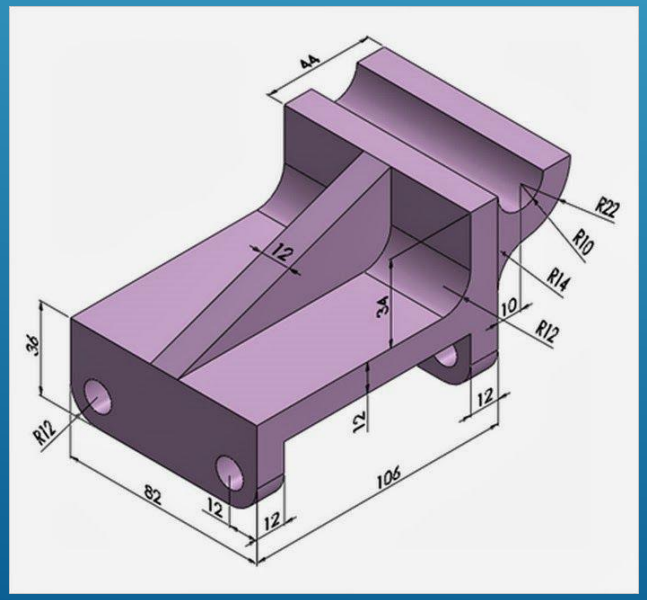

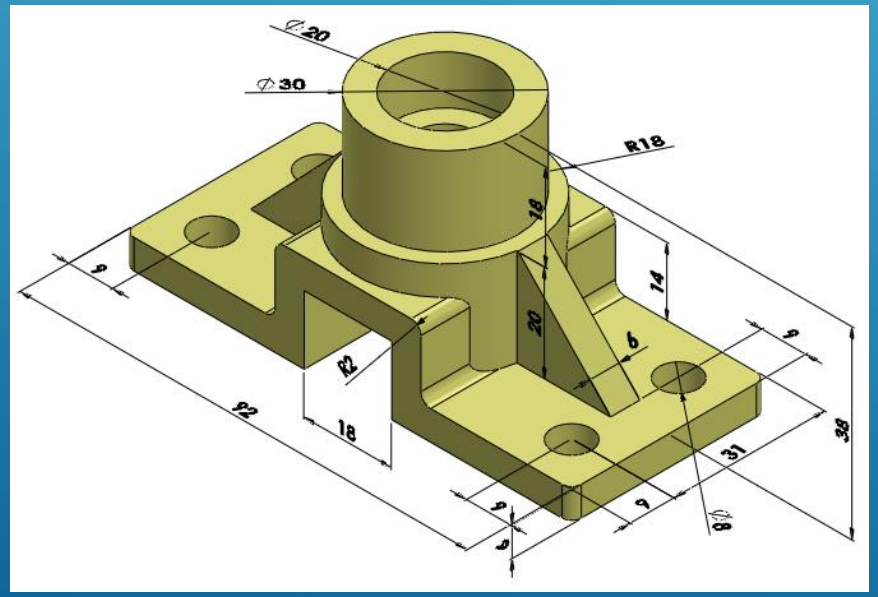

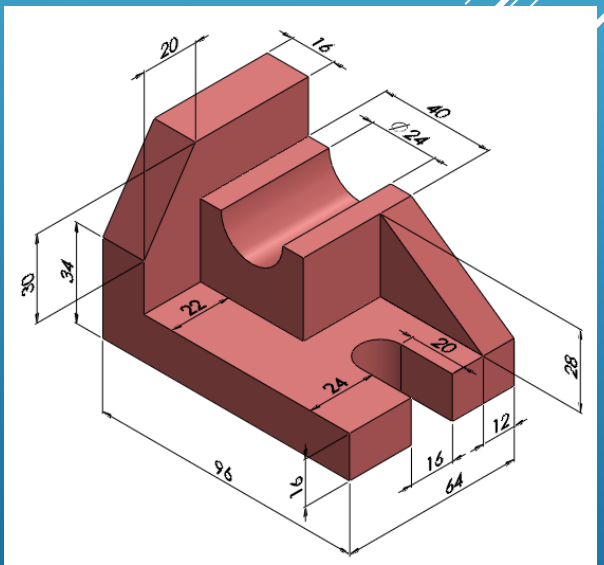

AULA 5 Desenho Técnico Assistido por Computador

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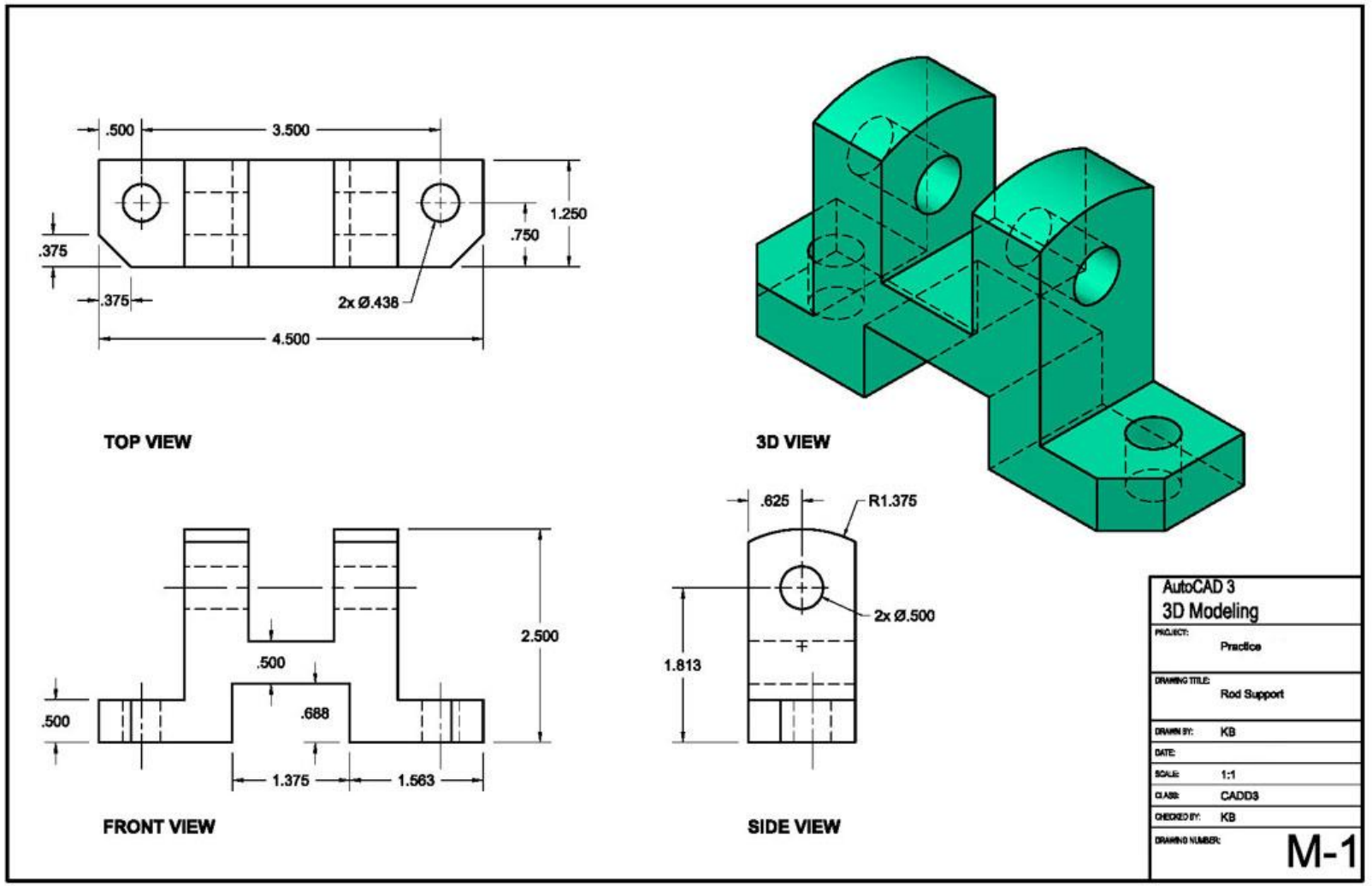

AULA 5 Desenho Técnico Assistido por Computador

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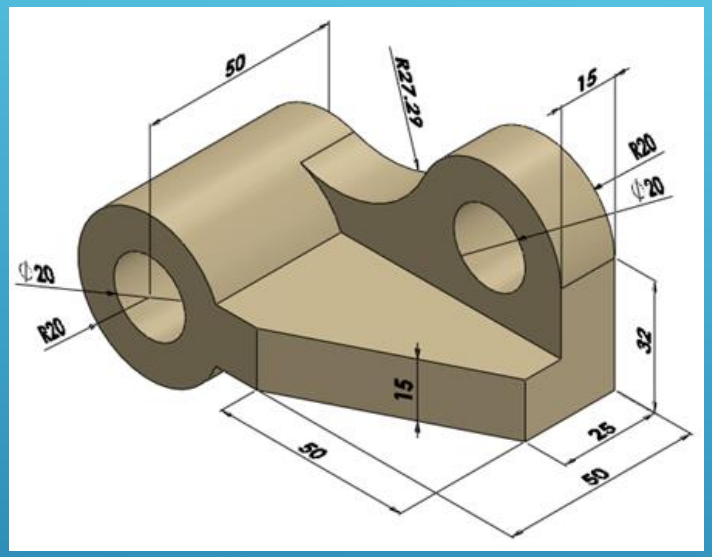

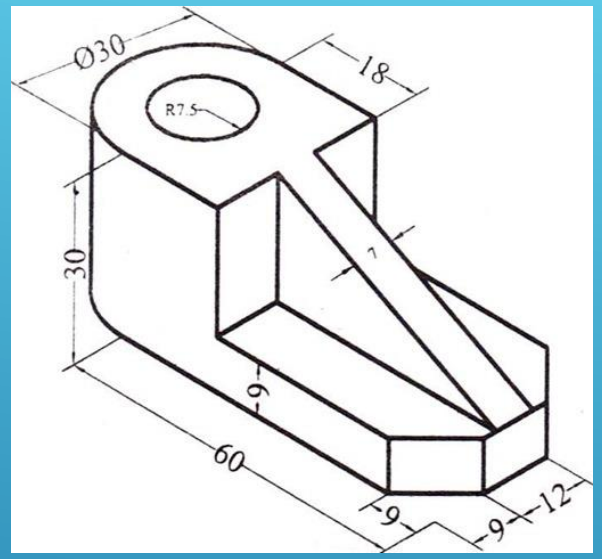

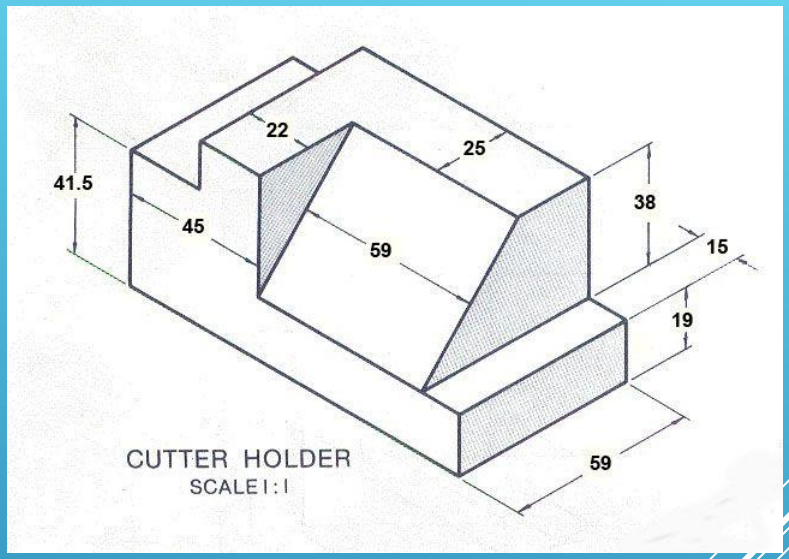

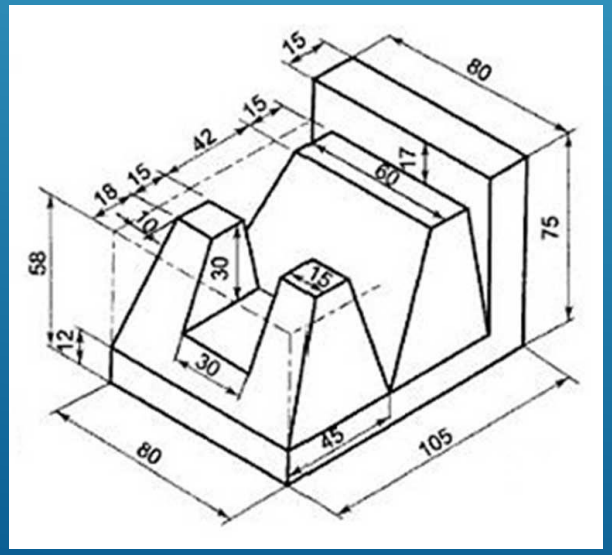

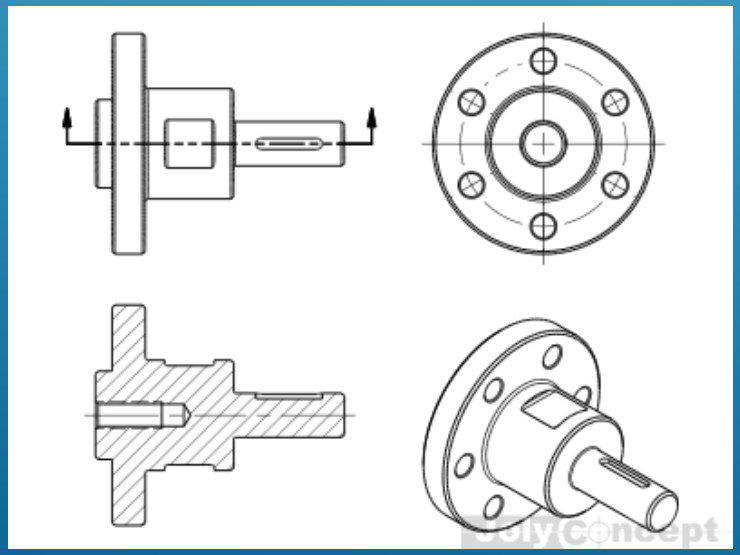

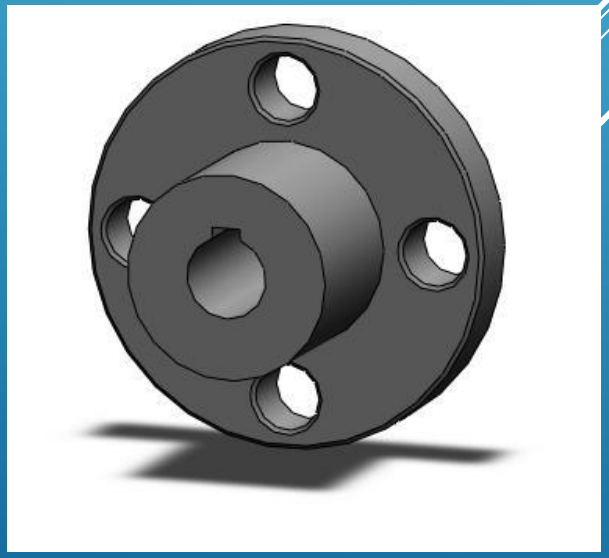

AULA 5 Desenho Técnico Assistido por Computador

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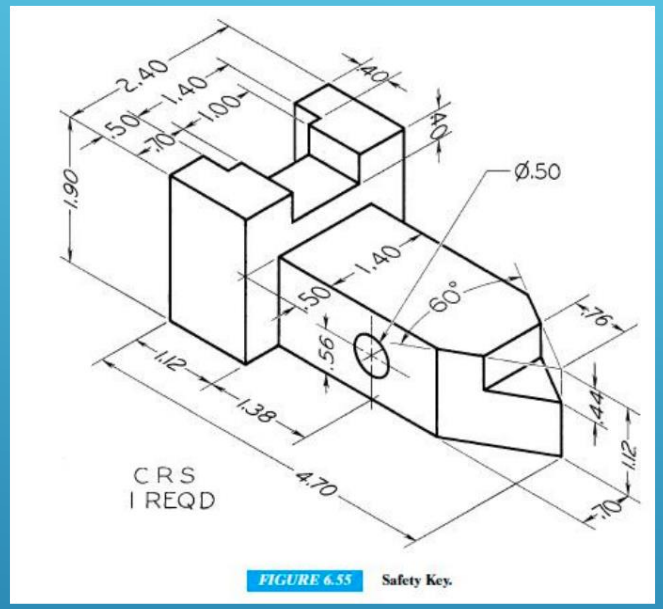
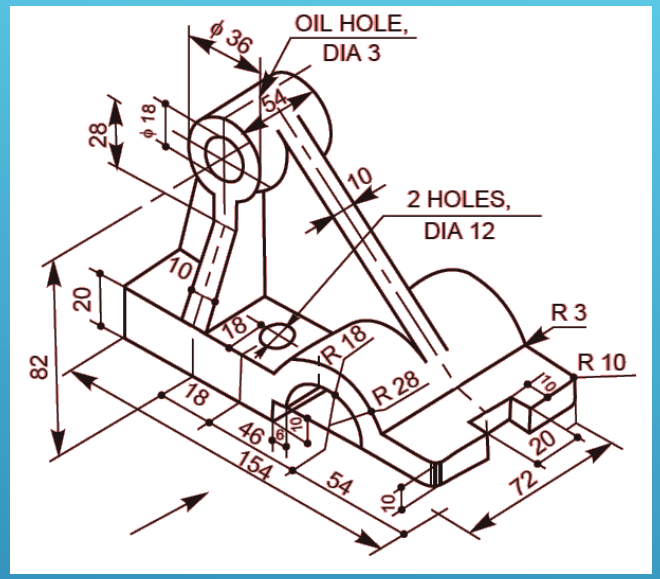
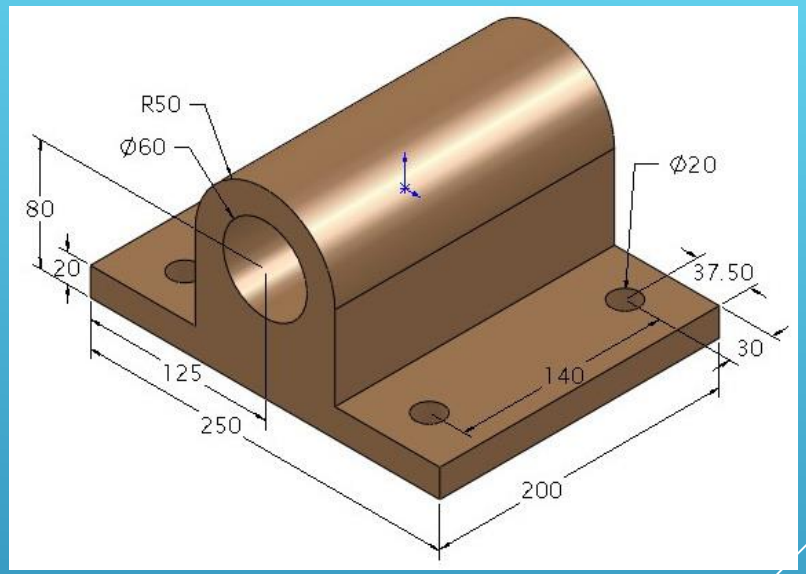
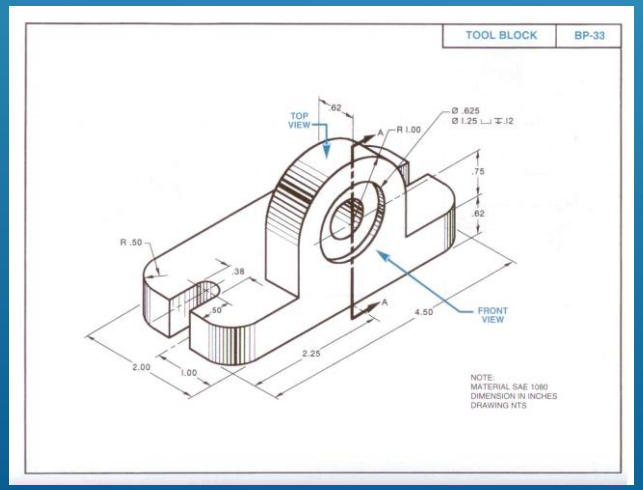
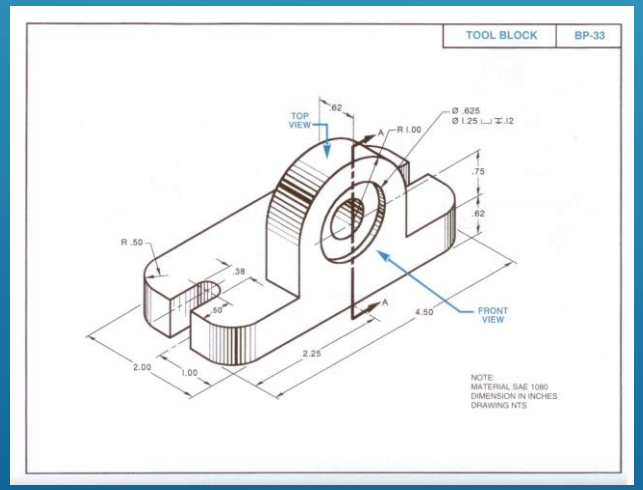


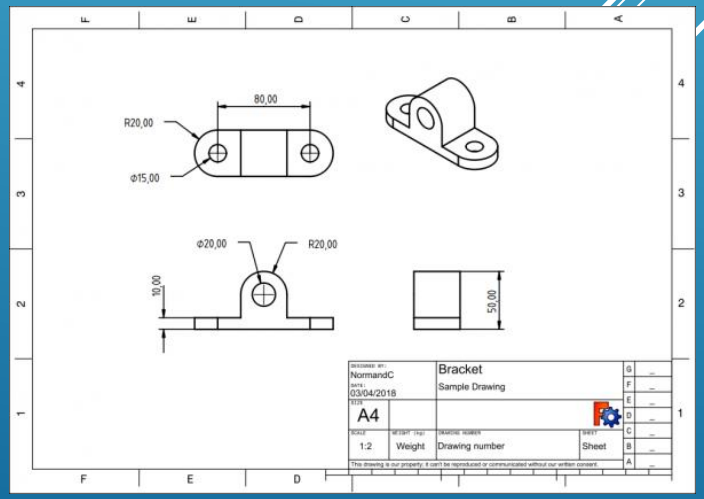
FIGURE 6.55 Safety Key.











AULA 5 Desenho Técnico Assistido por Computador

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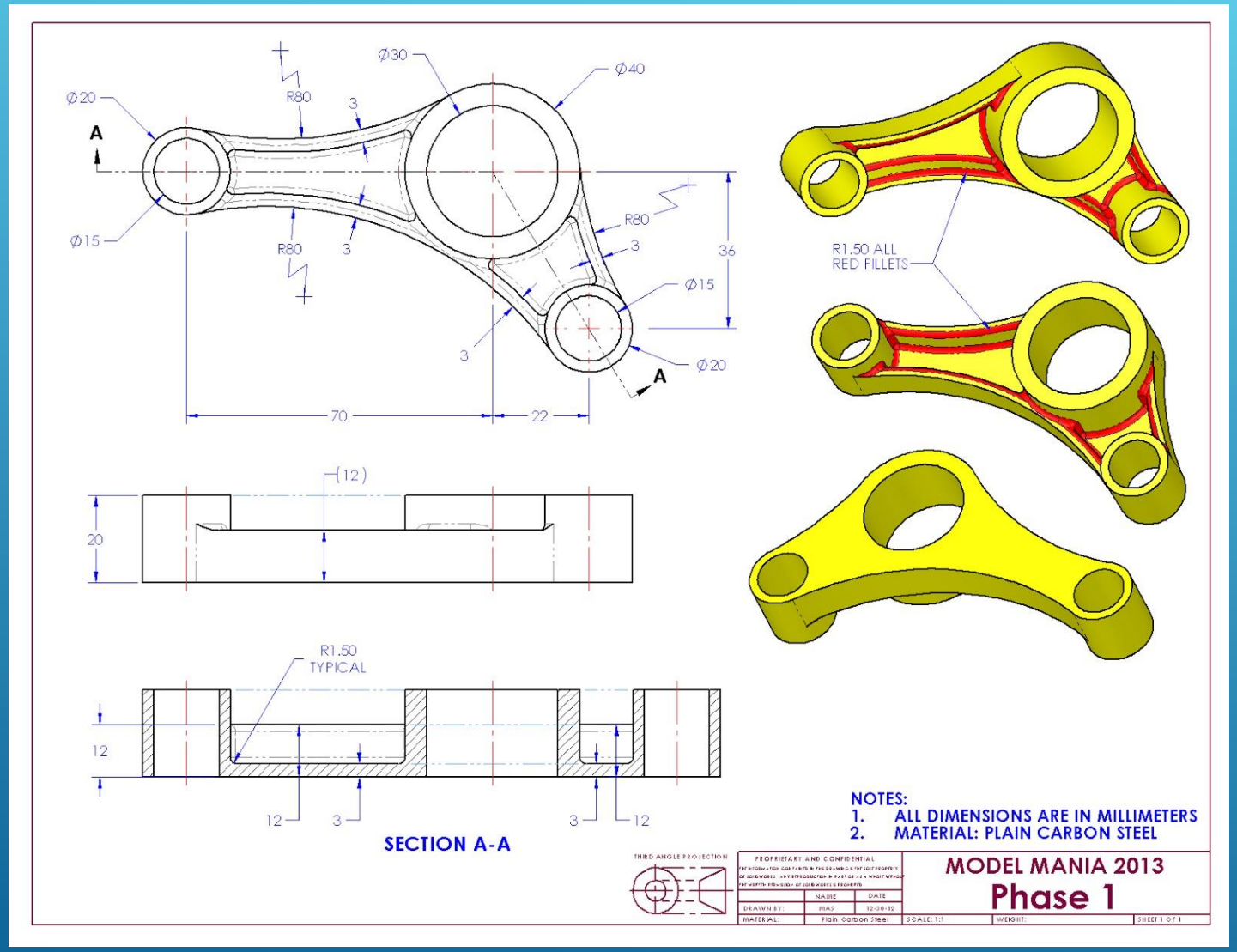

AULA 5 Desenho Técnico Assistido por Computador

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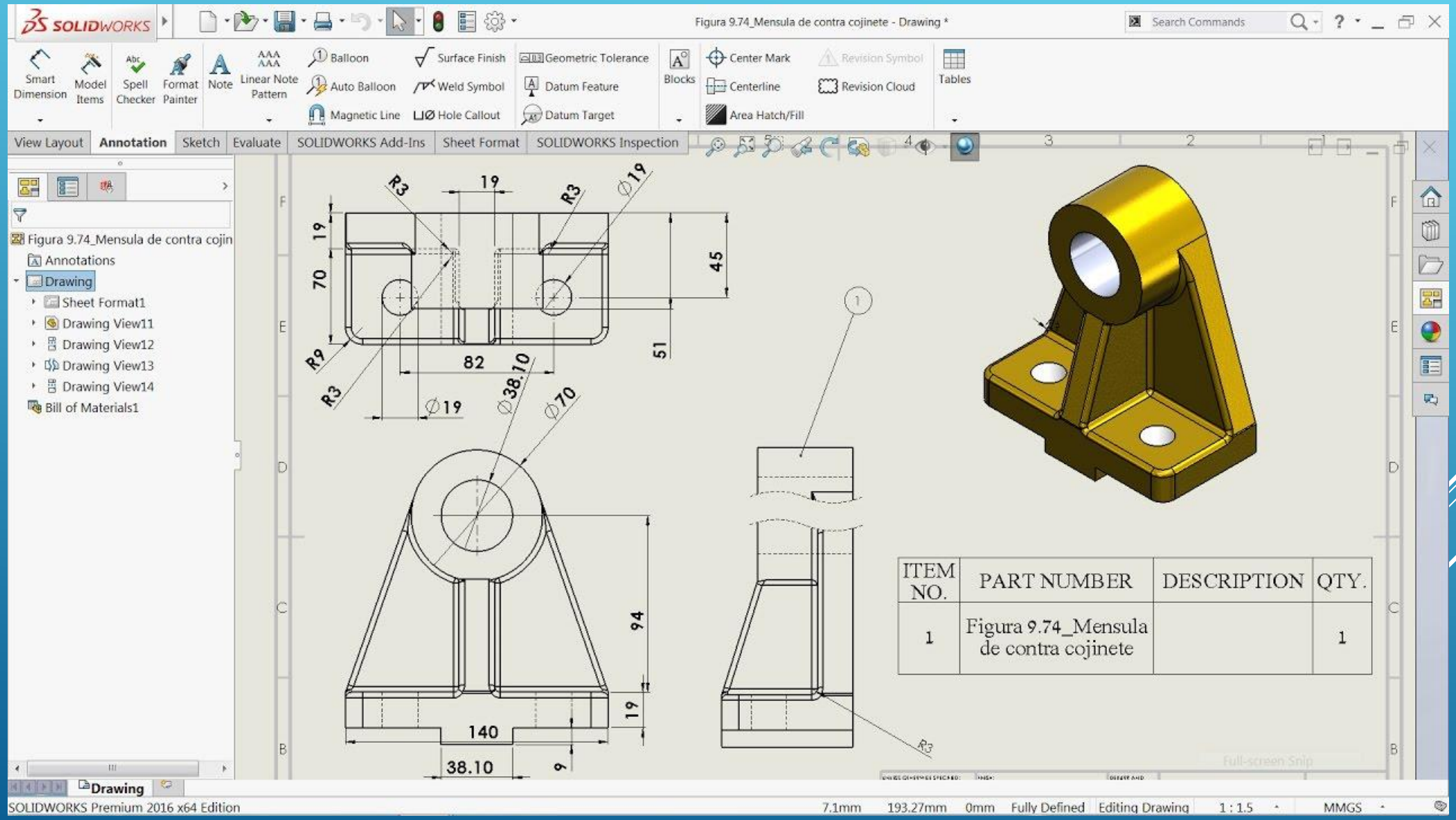

AULA 5 Desenho Técnico Assistido por Computador

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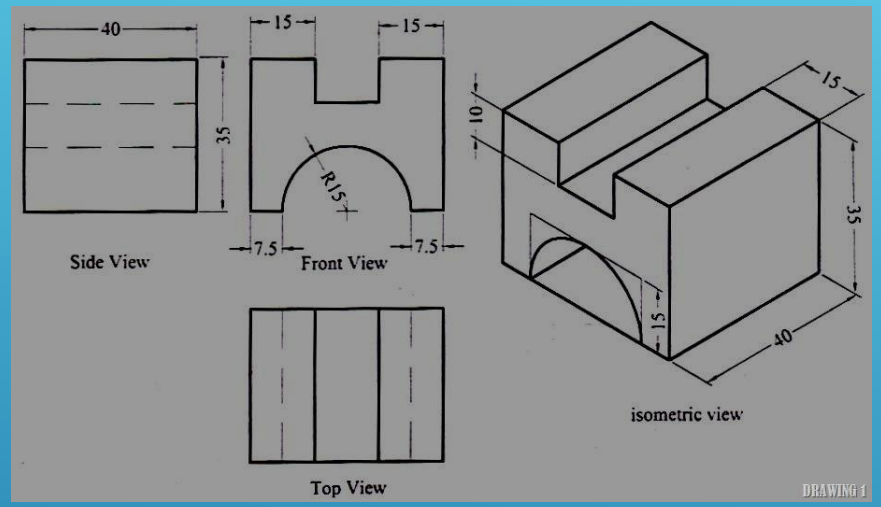
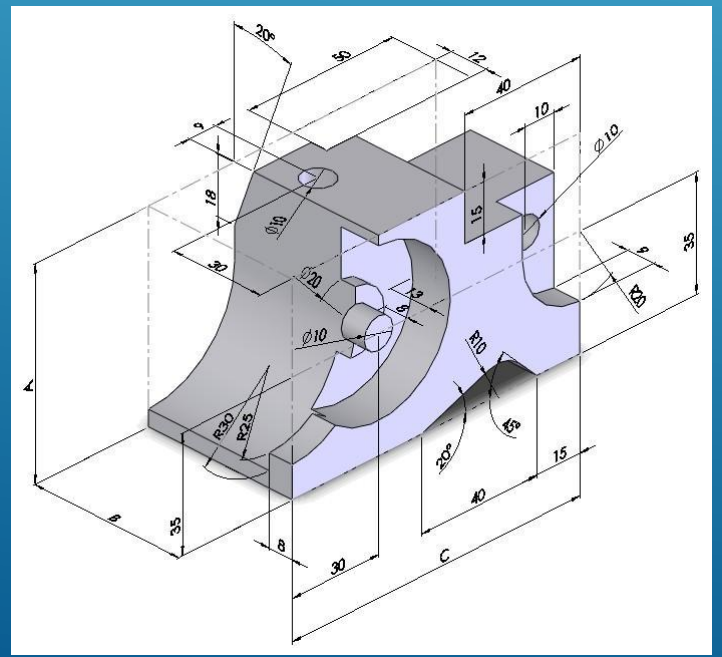




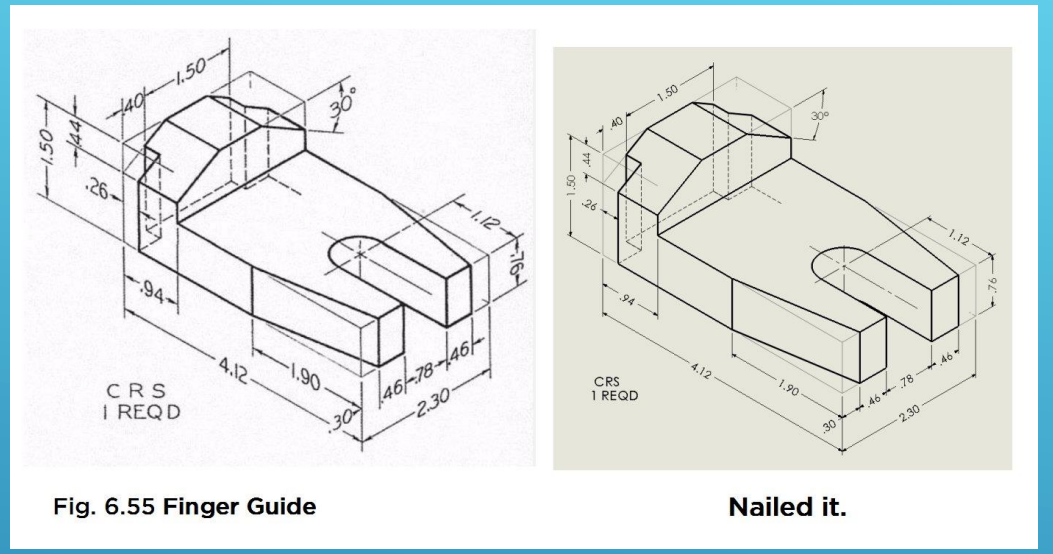
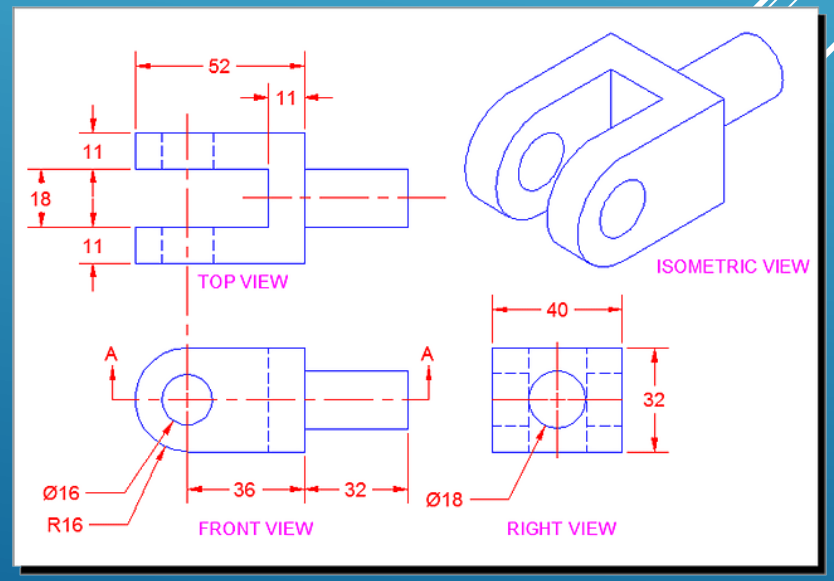


Fig. 6.55 Finger Guide

Nailed it.



AULA 5 Desenho Técnico Assistido por Computador

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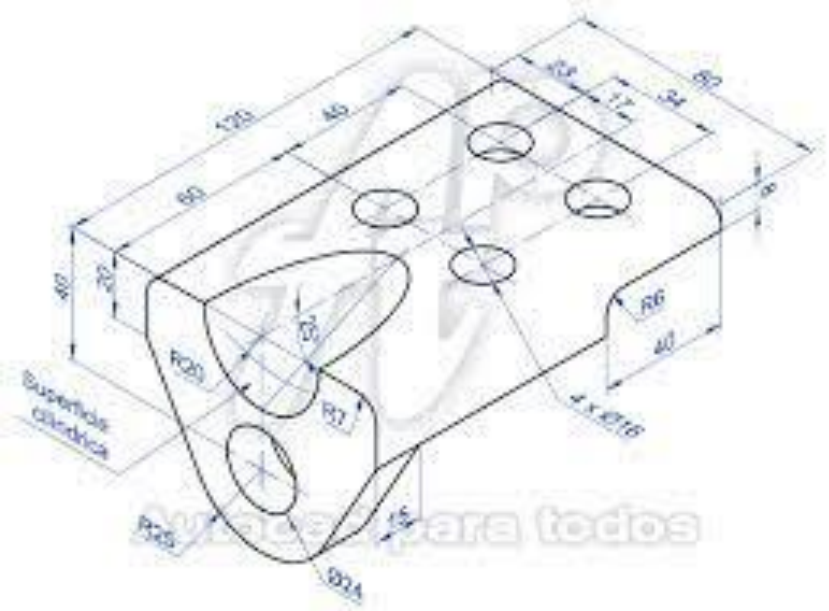
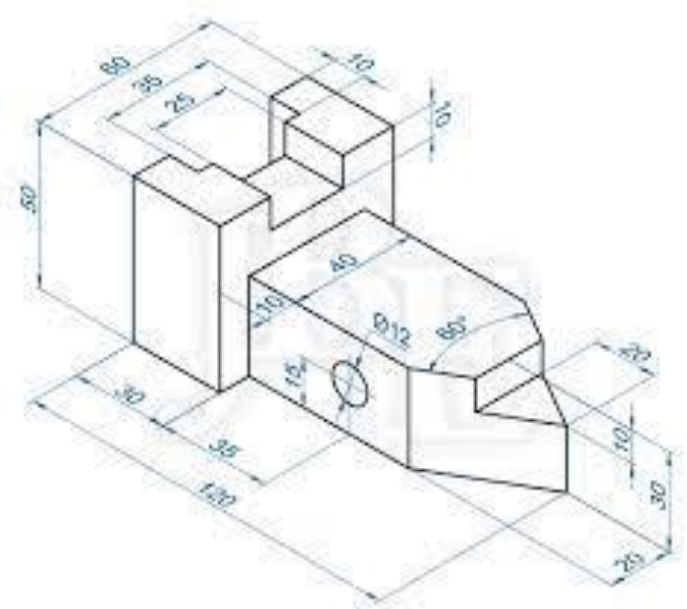
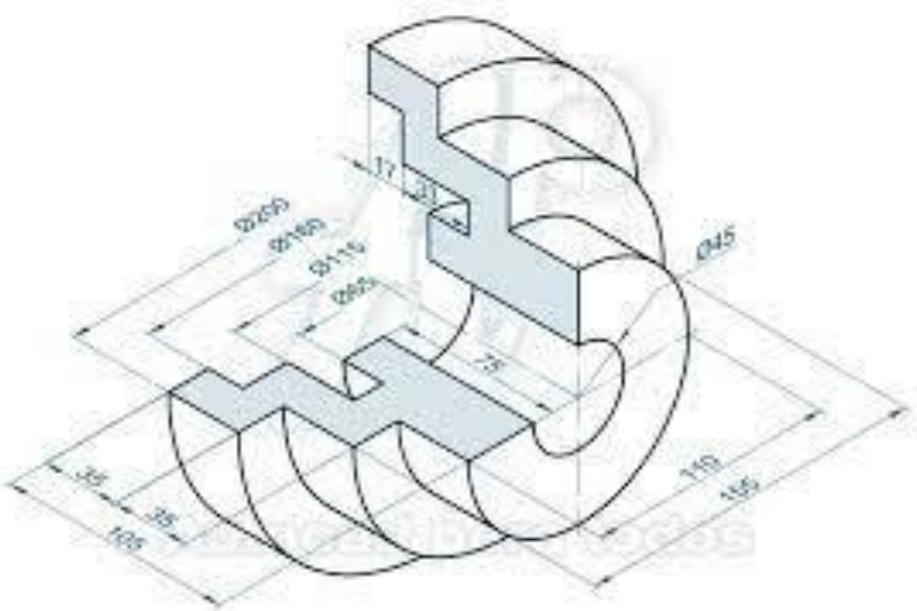
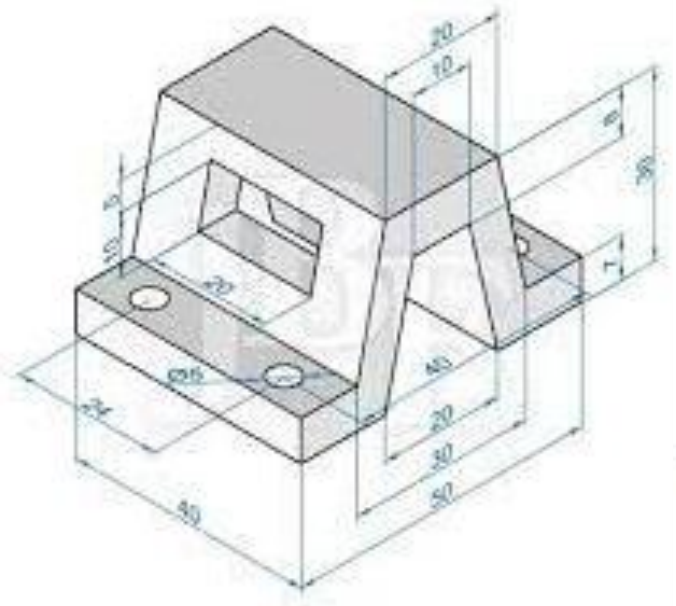
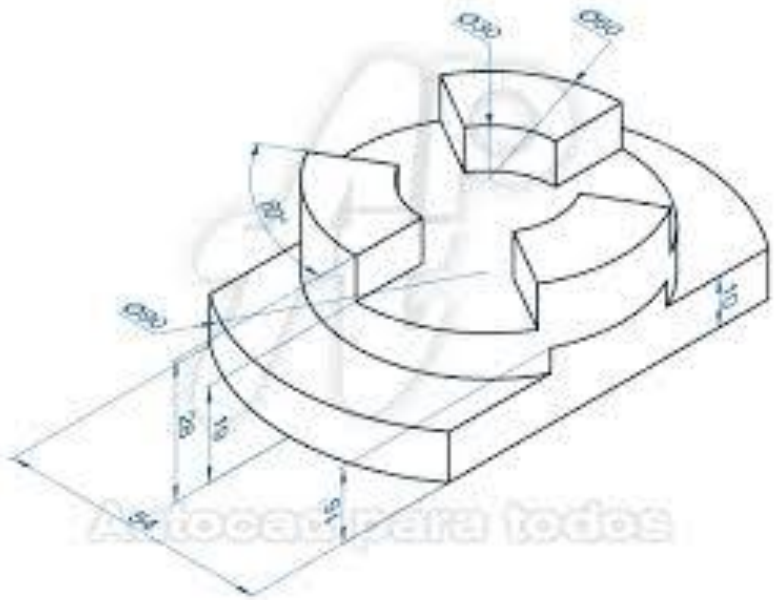
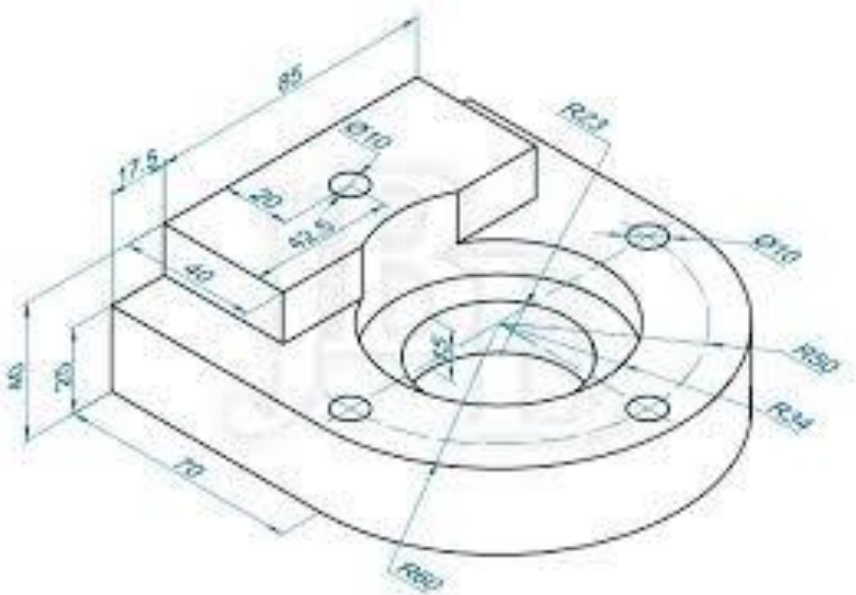

Superficie curvilínea
R6
4 x Ø16
R45
Ø24

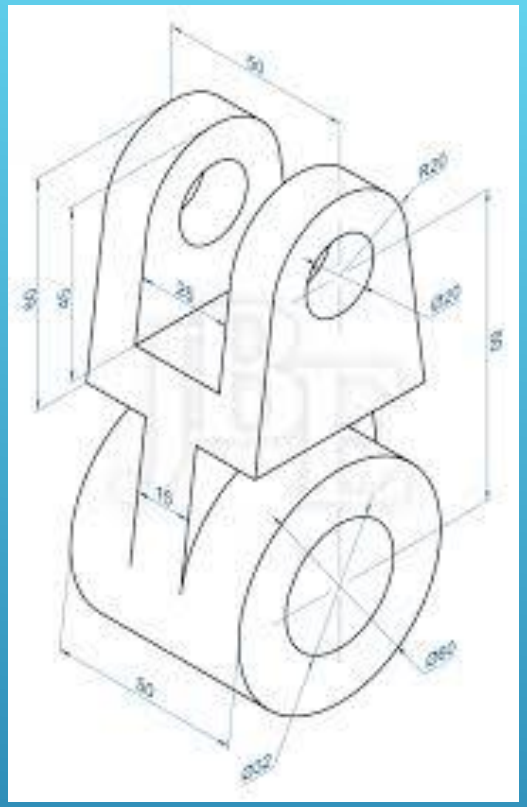

Ø12
85°

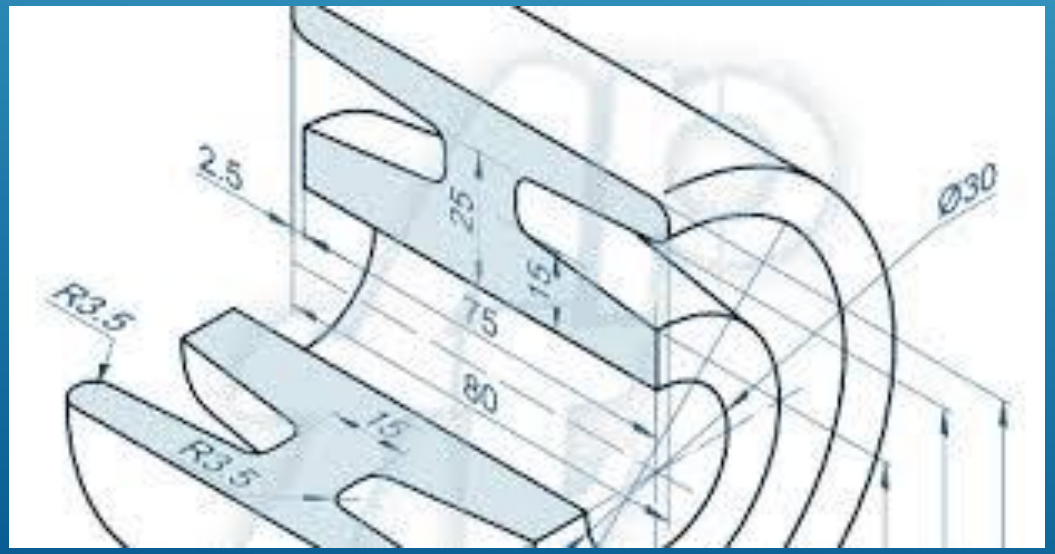

Ø45

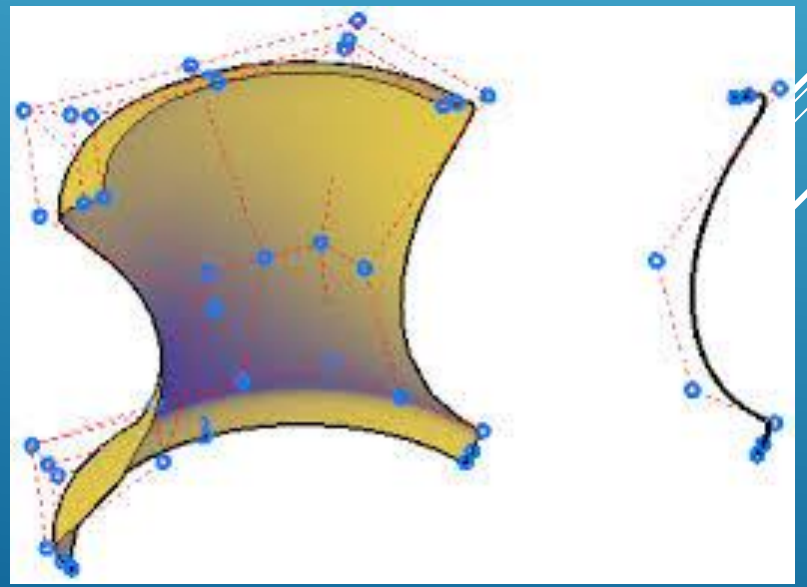


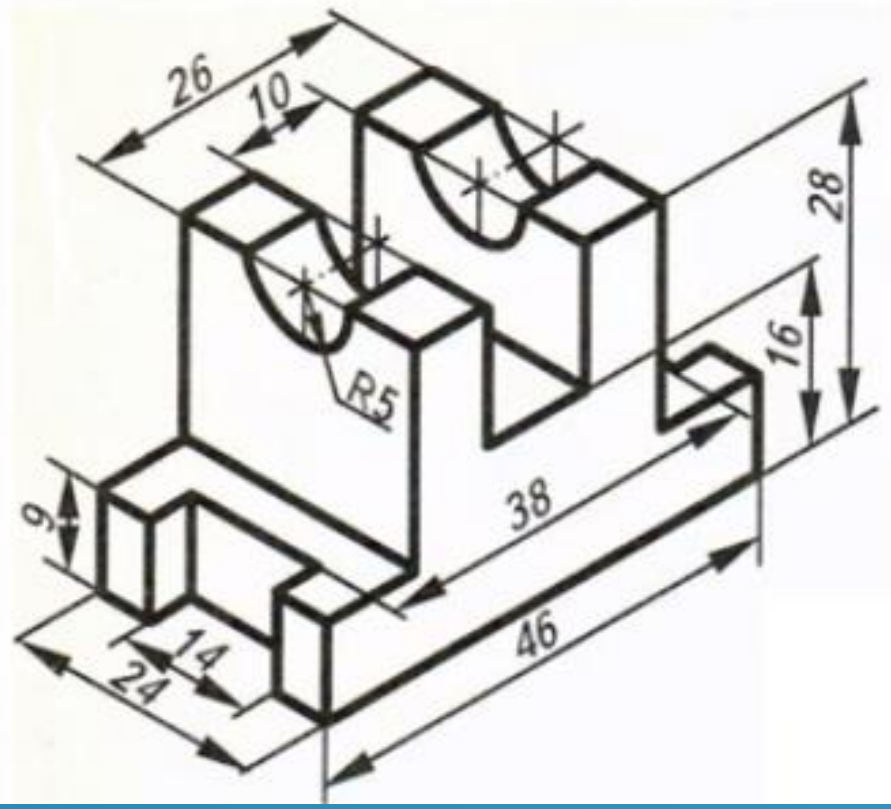



R23
Ø10
R30
R34
R80


50
R20
Ø20
25
50
65
55
15
Ø60
50
Ø32


2.5
25
15
75
80
15
R3.5
R3.5
Ø30





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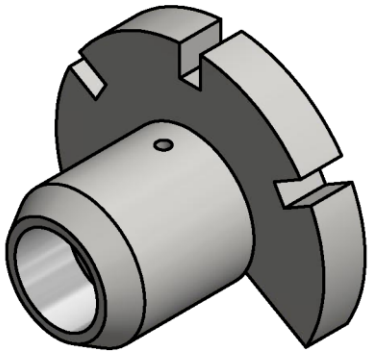
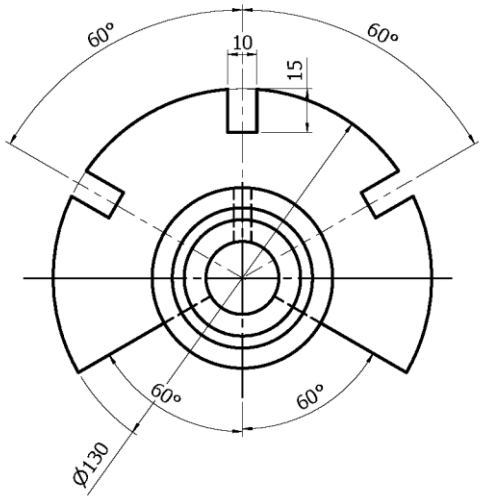
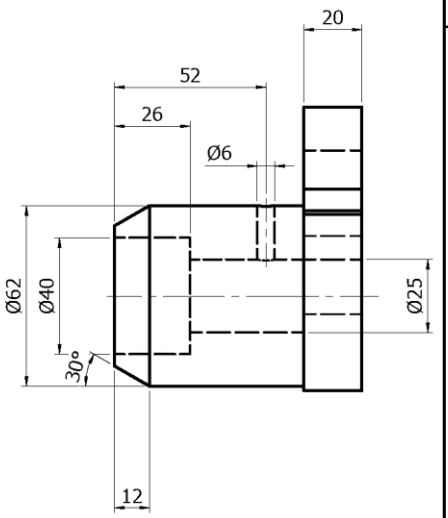
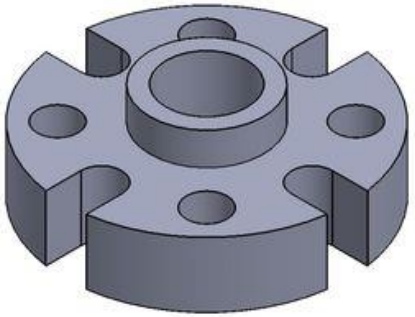
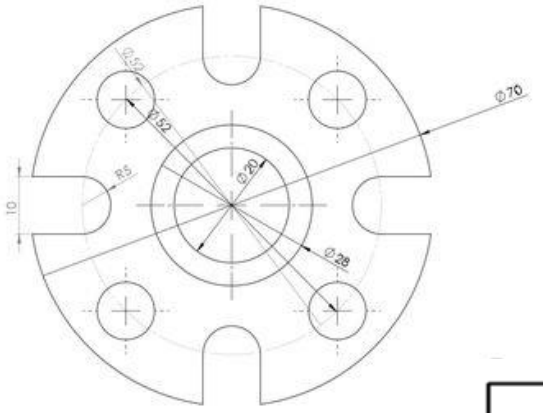


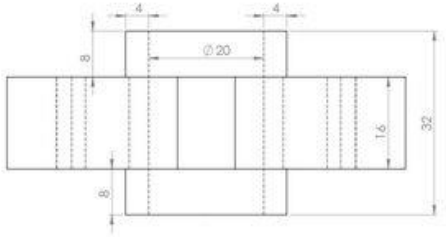




	Title	Date		Approve
	Modeling Practice Drawings 122	Design		
		Check		







3D EXERCISES

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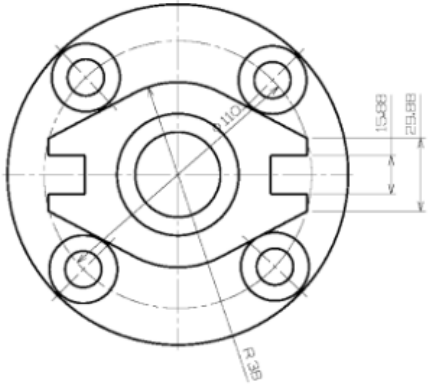
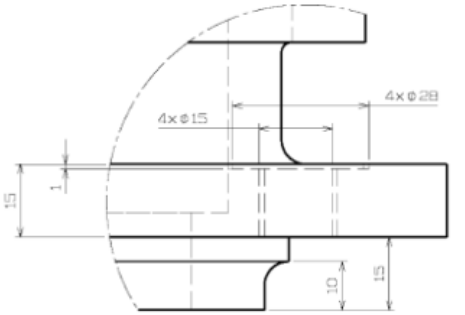
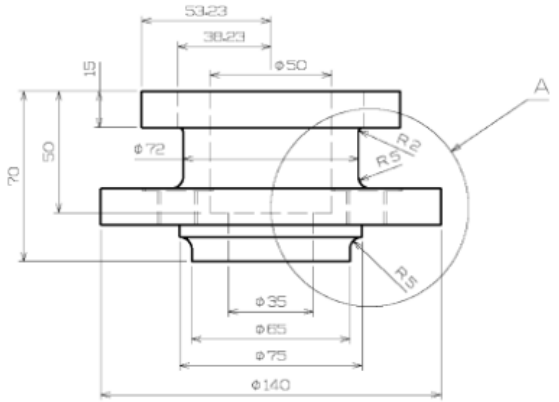
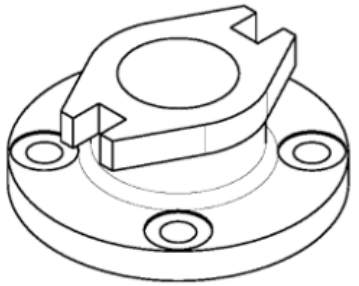
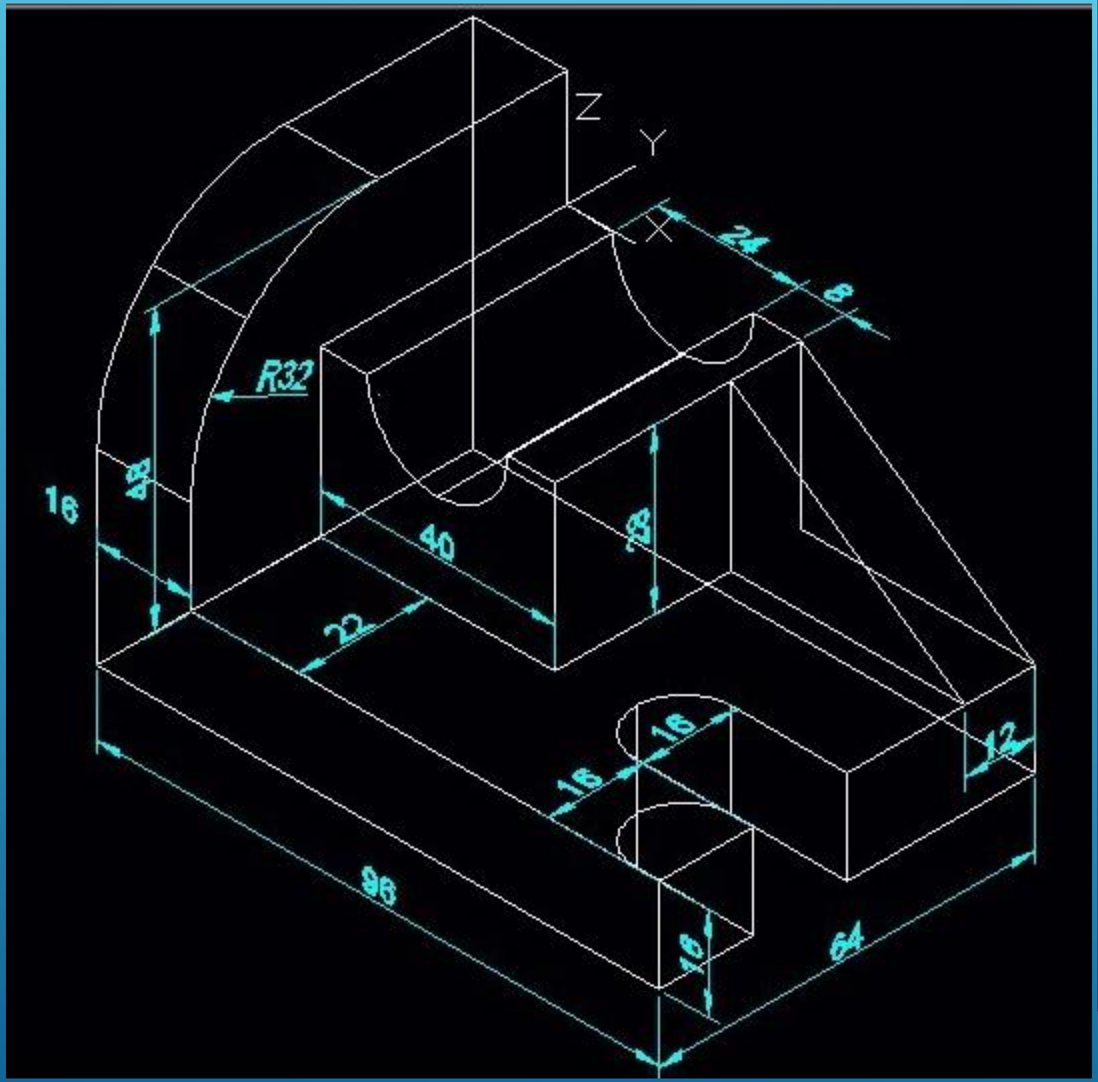


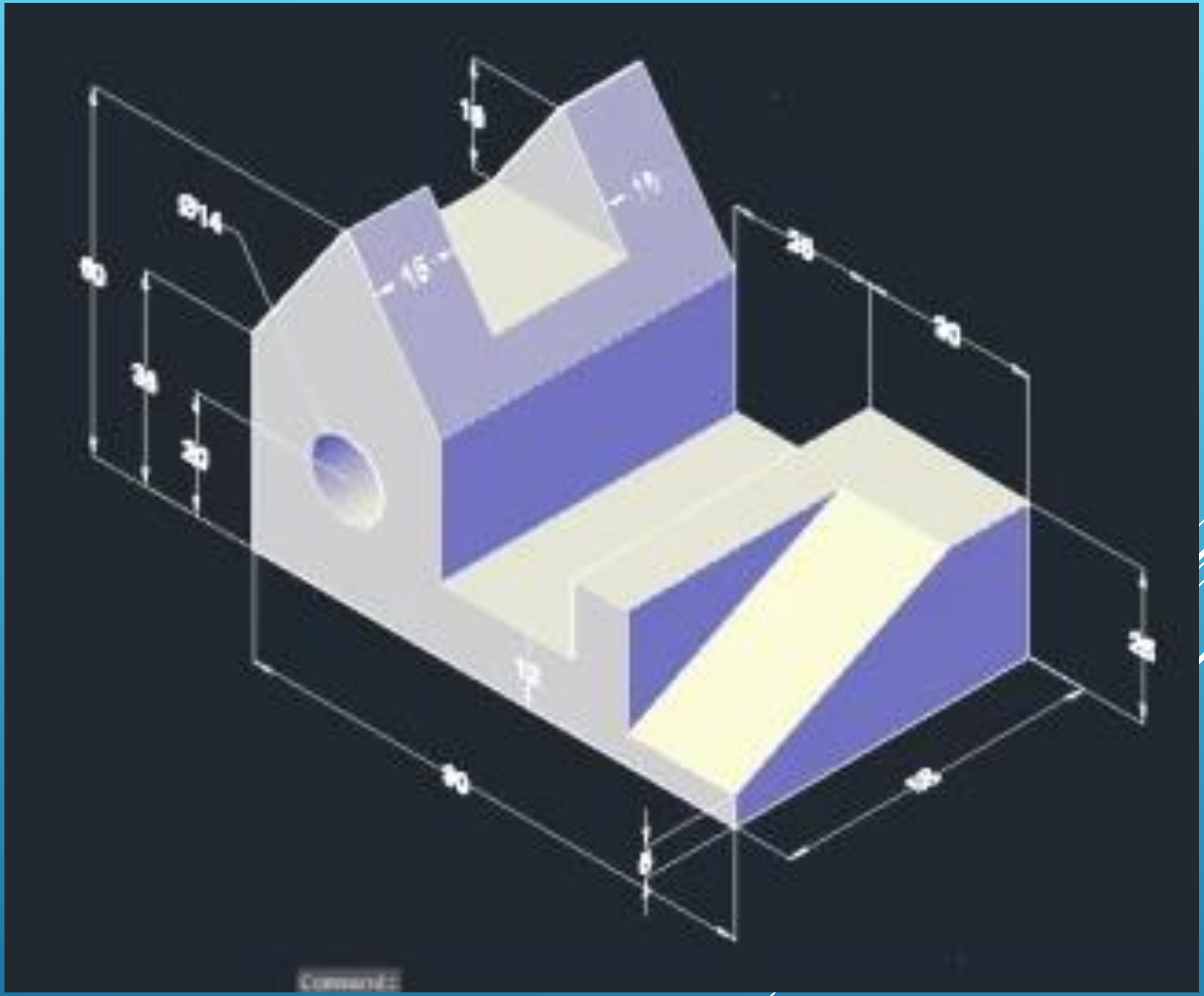


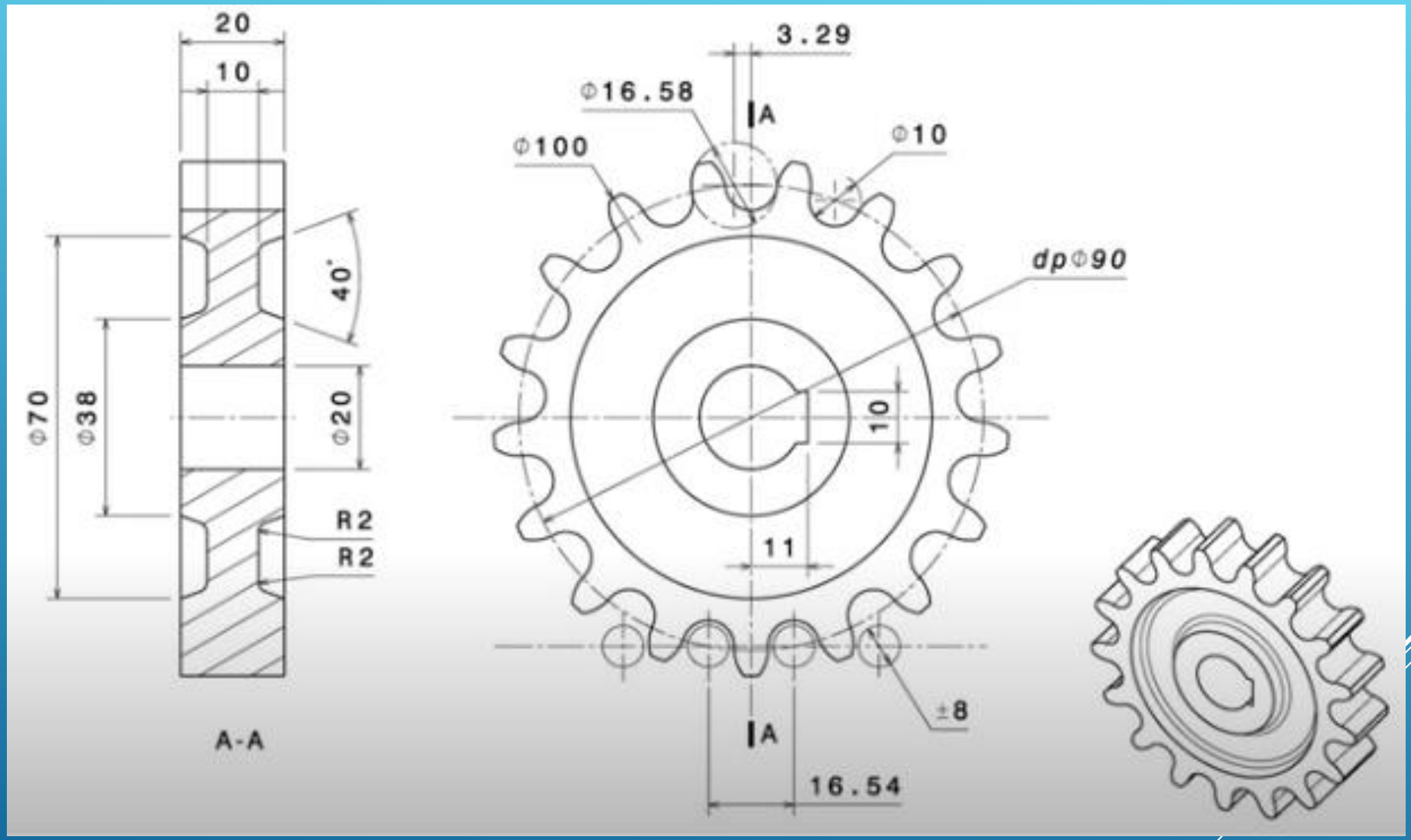
DETAIL A
SCALE 2:1

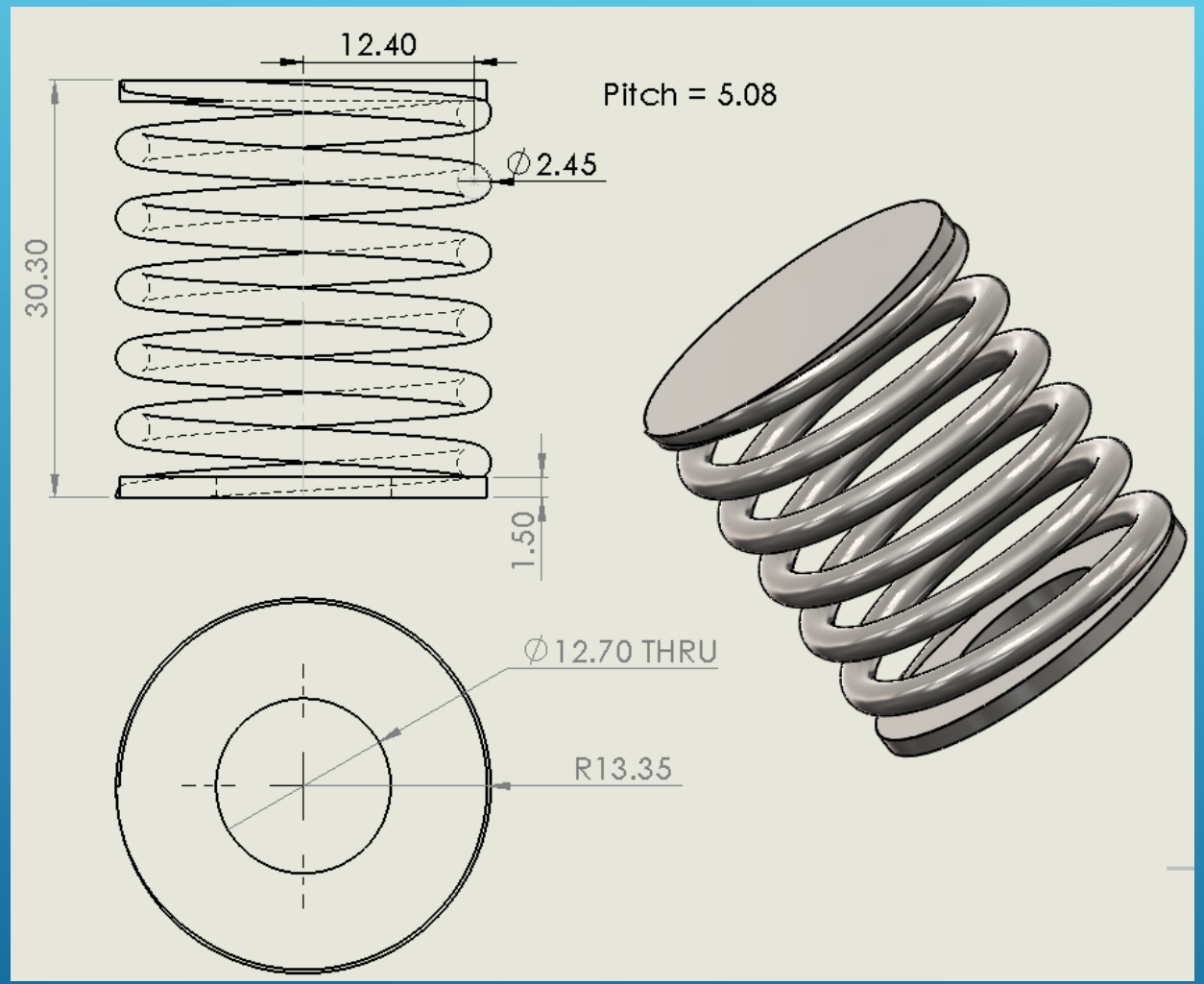


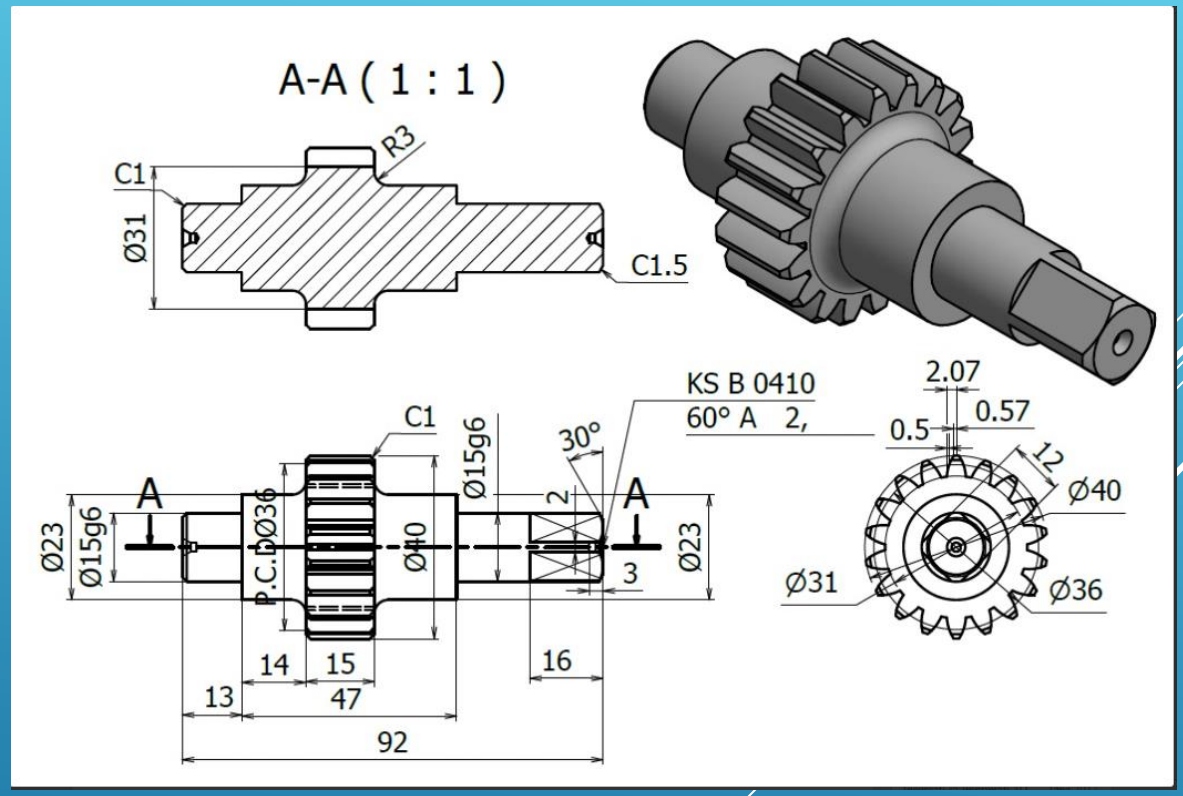


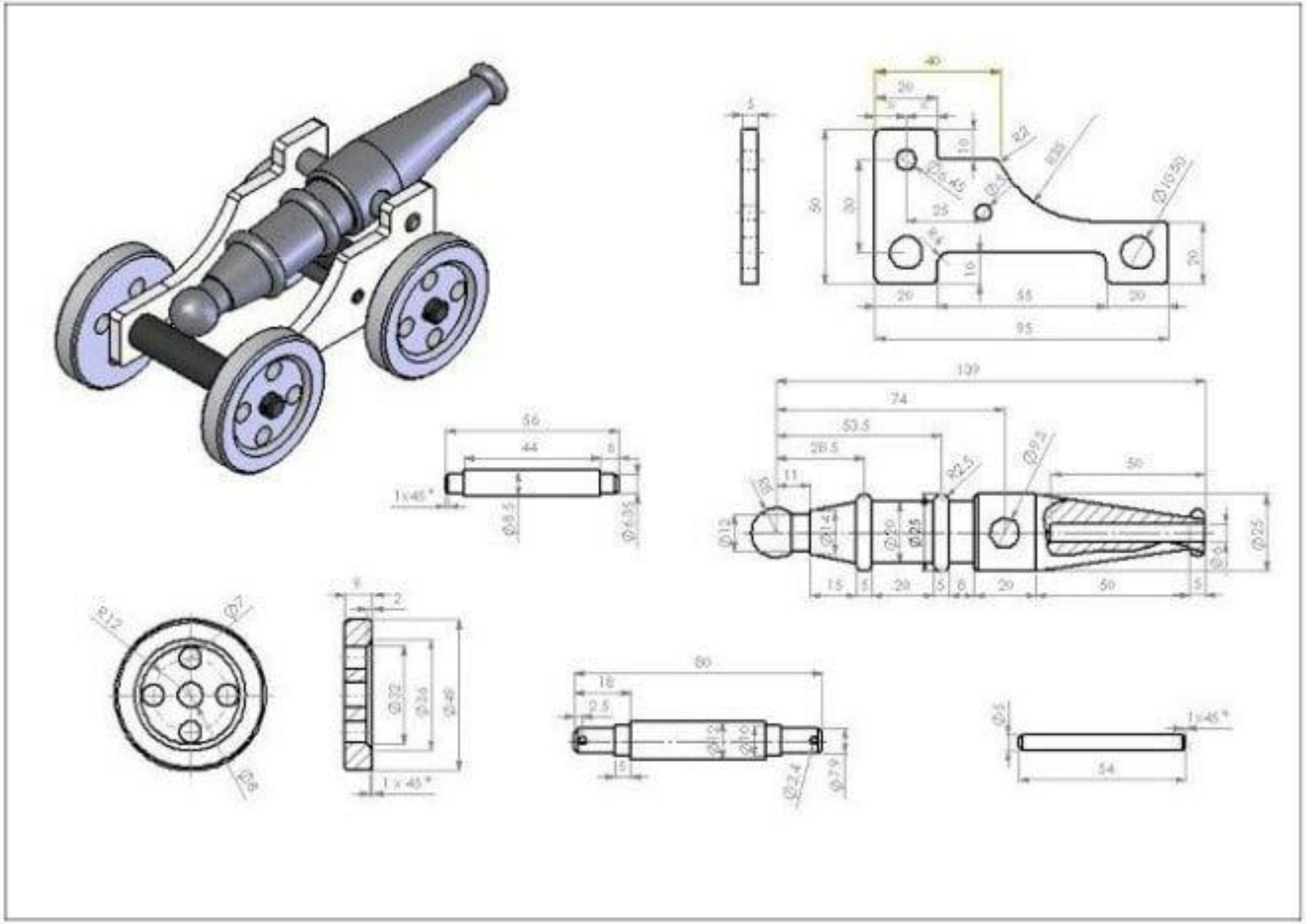

Z
Y
X
24
8
R32
16
48
40
22
16
16
12
96
64
16

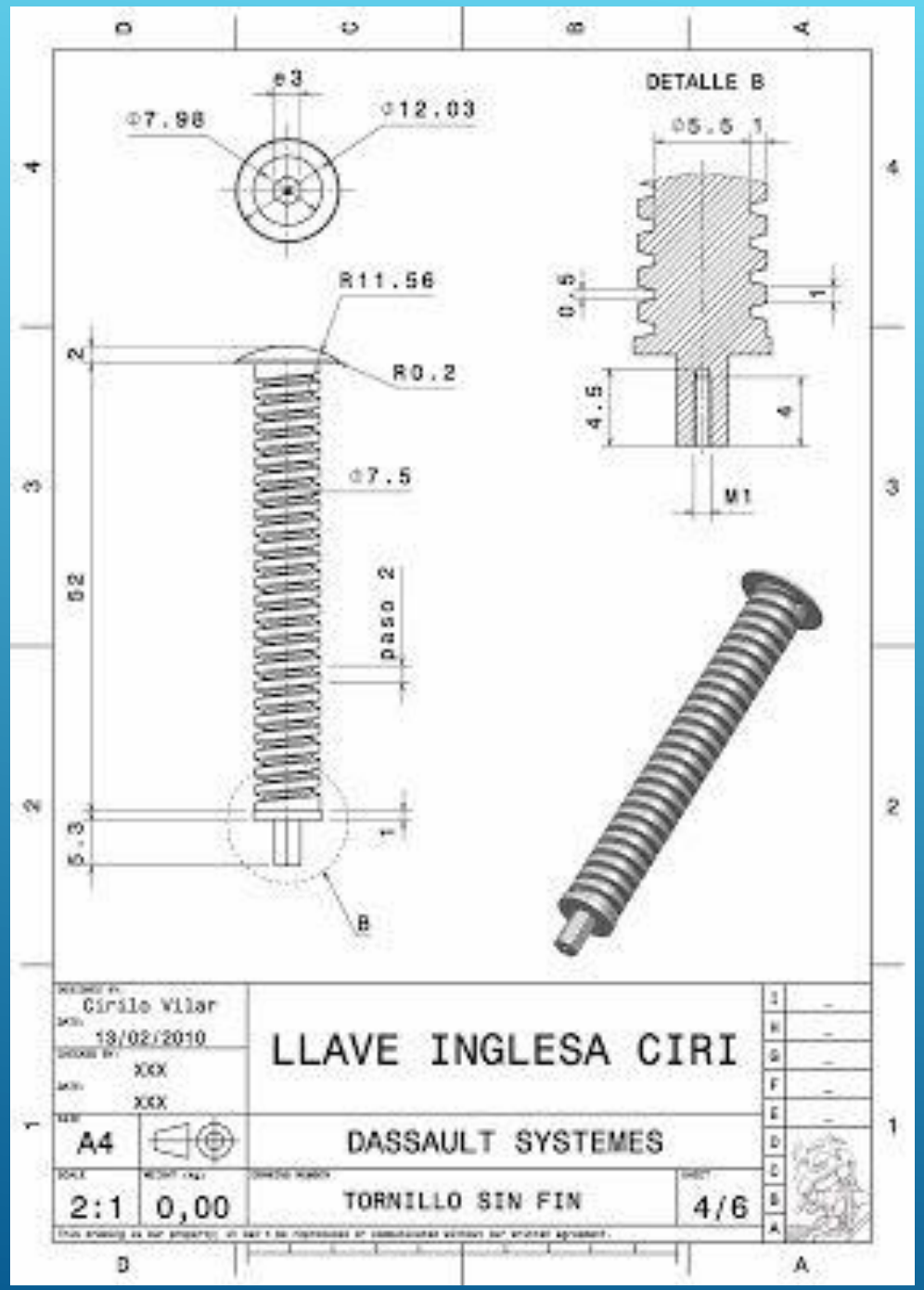

Ø14
60
34
20
28
30
90
Command:

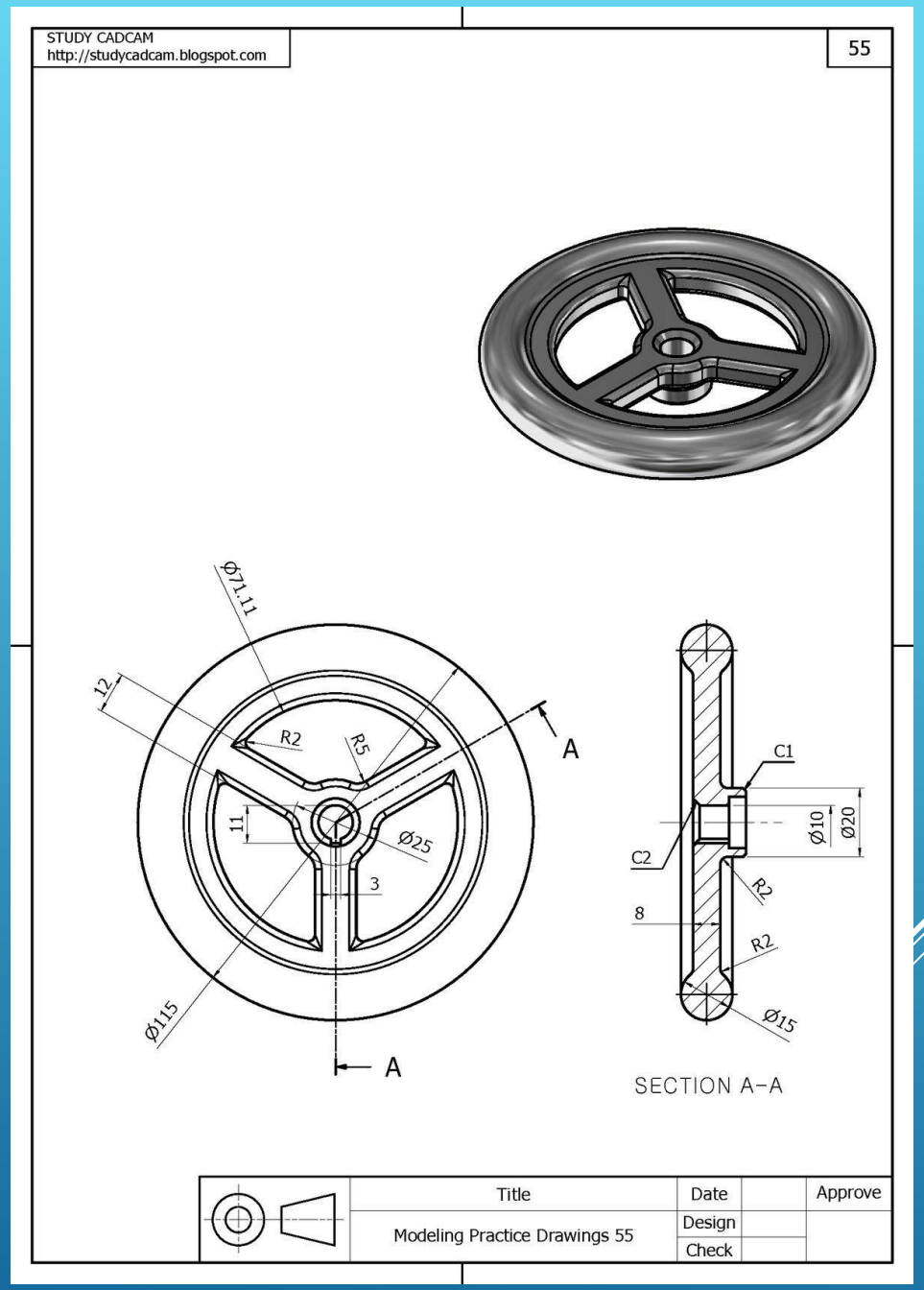

20
10
40°
Φ70
Φ38
Φ20
R2
R2
A-A
3.29
Φ16.58
A
Φ100
Φ10
dpΦ90
10
11
±8
A
16.54


12.40
Pitch = 5.08
∅2.45
30.30
1.50
∅12.70 THRU
R13.35


A-A (1 : 1)
R3
C1
Ø31
C1.5
KS B 0410
60° A 2,
2.07
0.57
0.5
12
Ø40
Ø31
Ø36
C1
Ø15g6
30°
A
A
Ø23
Ø15g6
P.C.D Ø36
Ø10
2
3
Ø23
14
15
16
13
47
92




DETALLE B
ø3
ø7.98
ø12.03
ø5.5
1
0.5
1
4.5
4
M1
R11.56
R0.2
2
ø7.5
52
paso 2
5.3
1
B
Cirilo Vilar
13/02/2010
XXX
XXX
LLAVE INGLESA CIRI
A4
DASSAULT SYSTEMES
2:1
0,00
TORNILLO SIN FIN
4/6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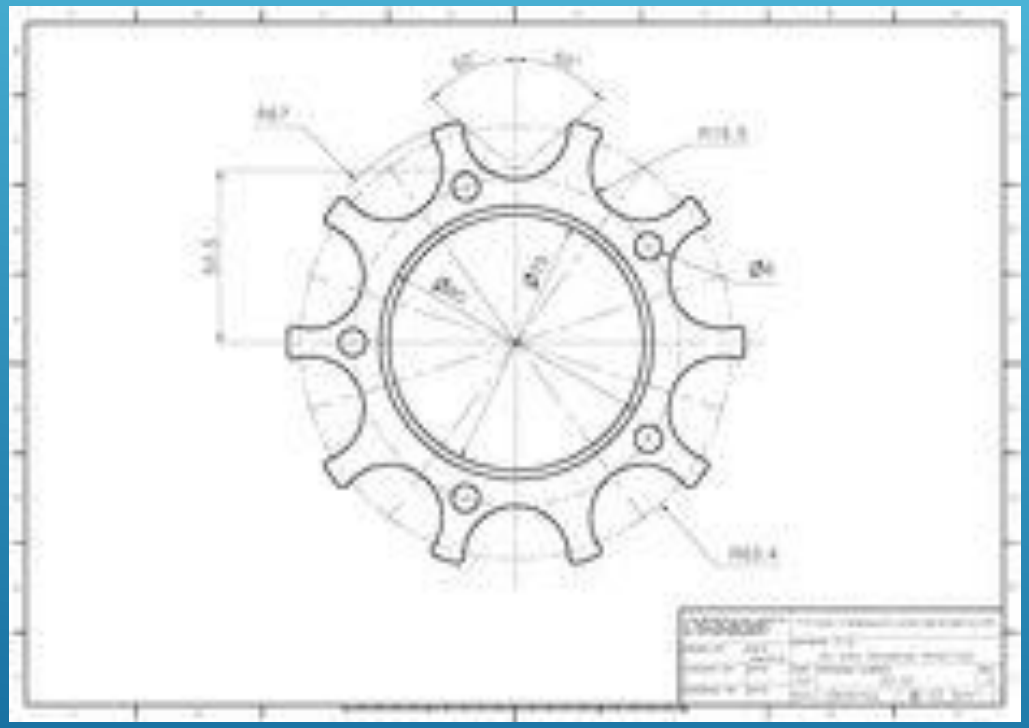
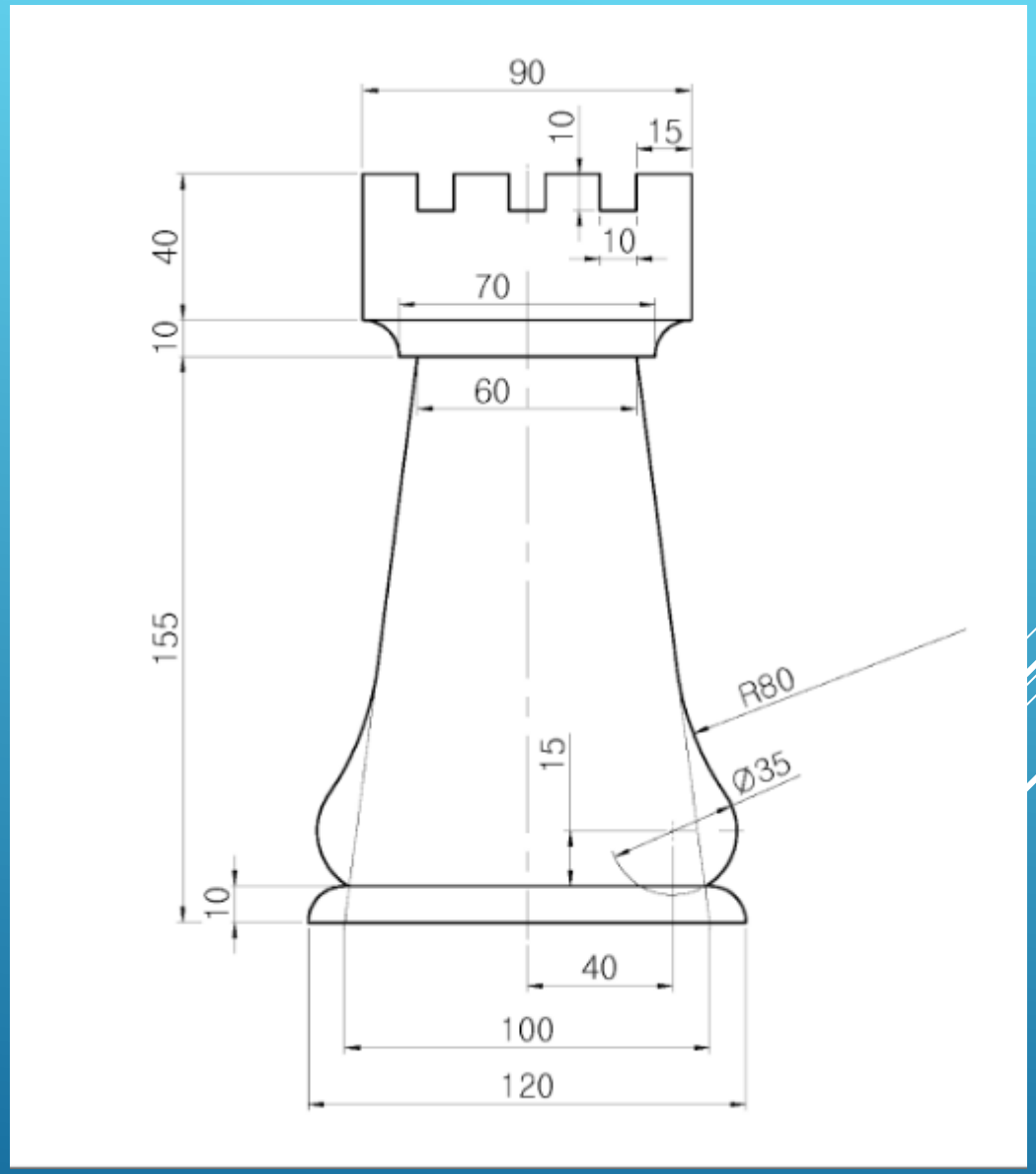
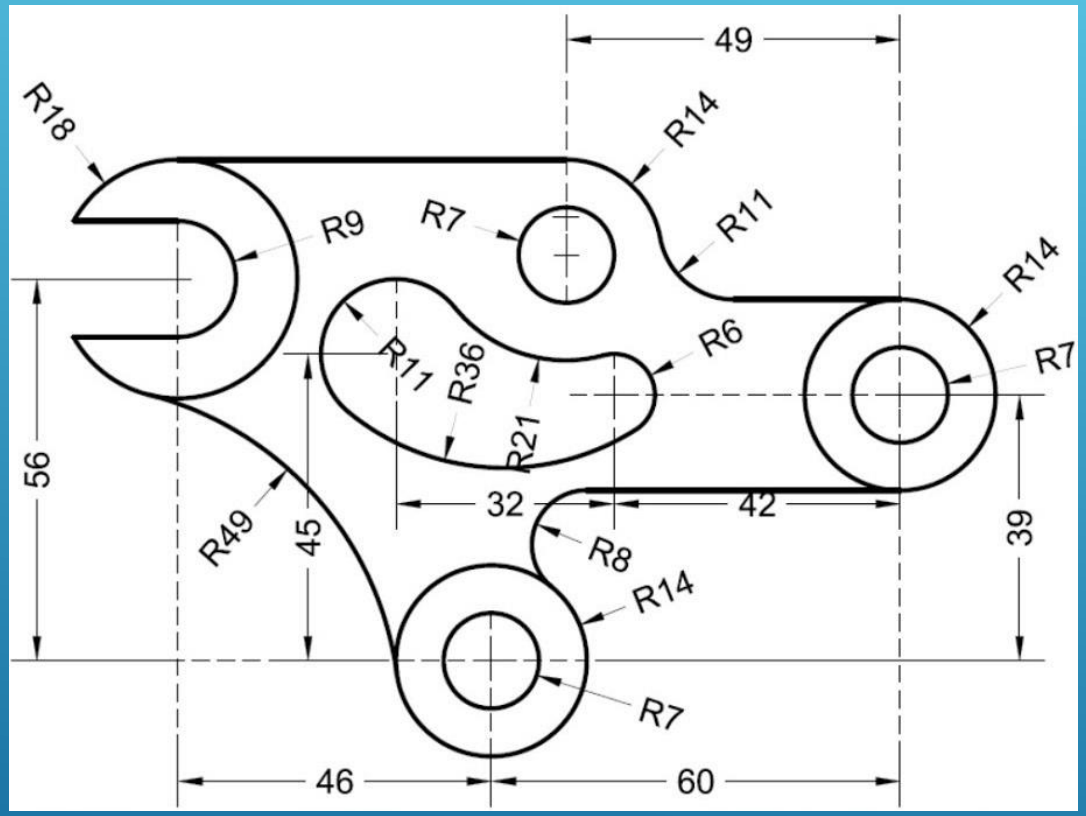
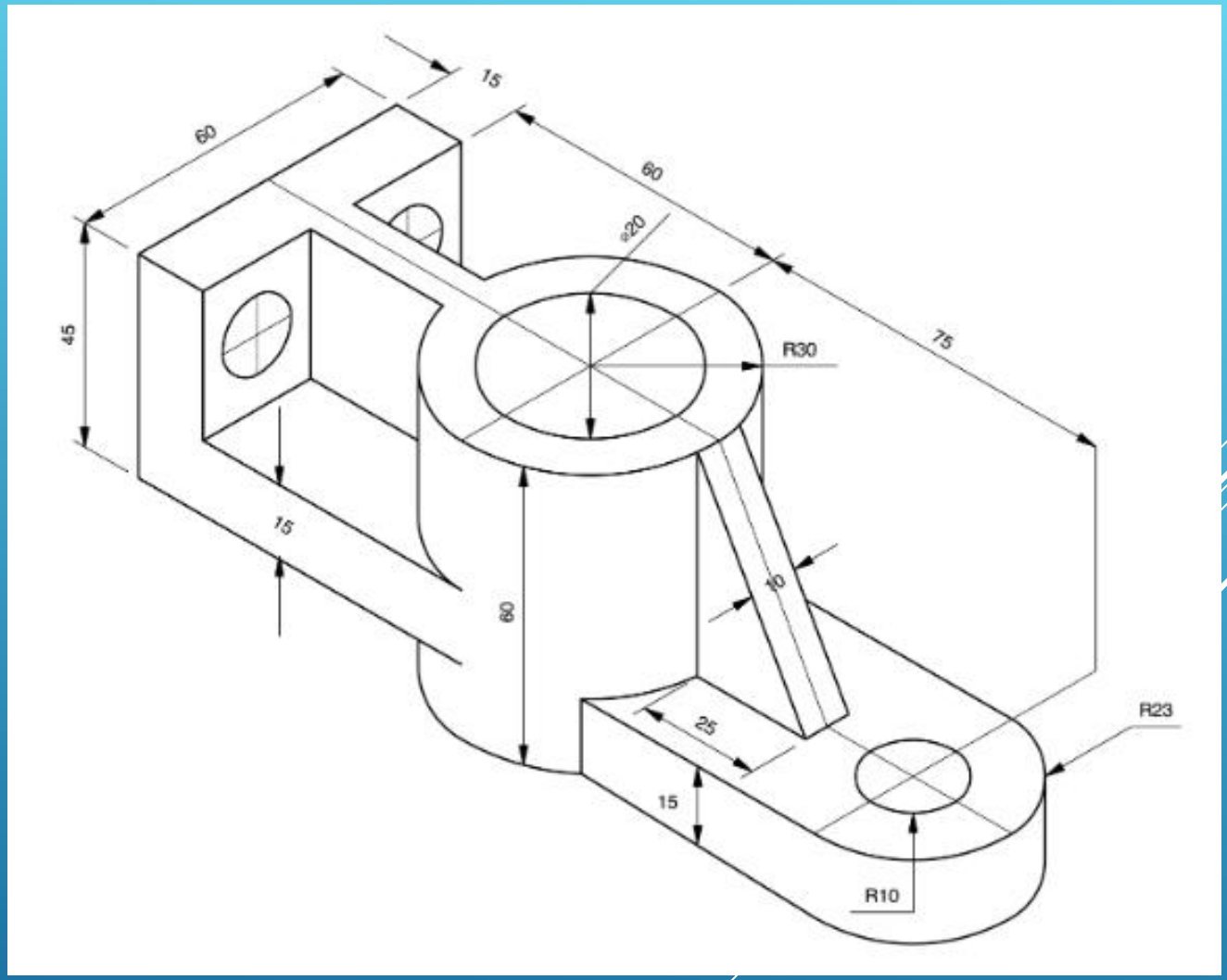
55
Ø71.11
12
R2
R5
A
11
Ø25
3
Ø115
A
C1
Ø10
Ø20
C2
R2
8
R2
Ø15
SECTION A-A
Title
Modeling Practice Drawings 55
Date
Design
Check
Approve




90
10
15
10
40
70
10
60
155
R80
15
Ø35
10
40
100
120


49
R18
R14
R9
R7
R11
R14
R6
R7
R11
R36
R21
56
R49
45
32
42
39
R8
R14
R7
46
60


15
60
60
ø20
45
R30
75
15
10
60
25
15
R23
R10