Solid Work

Guide Note

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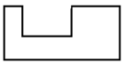
کتاب درسی:
مجموعه کتاب های مثلث نارنجی، خود آموز طراحی مکانیکی با
Solid Work
مهندس هادی جعفری
انتشارات نشر آفرنگ

جلسه اول

1. Being familiar with Part area
2. To remove Task pane (پنجره), go to: View/Task Pane
3. Sketch  through Front Plane
4. Feature and Extrude[1]
5. Go to Display Style
6. Go to View Orientation to have Isometric, and all Views
7. Then 2 Views and 3 Views
8. Adjust US method, if needed- Through:
   Tools/Option/System Option/Display/Third Angle Projection

7. Sketch through Top Plane and Extrude it, (same shape however different views will be appeared)
8. Go to View Orientation to have Isometric, and all Views
9. Clarify the difference of Top, Front and Right side

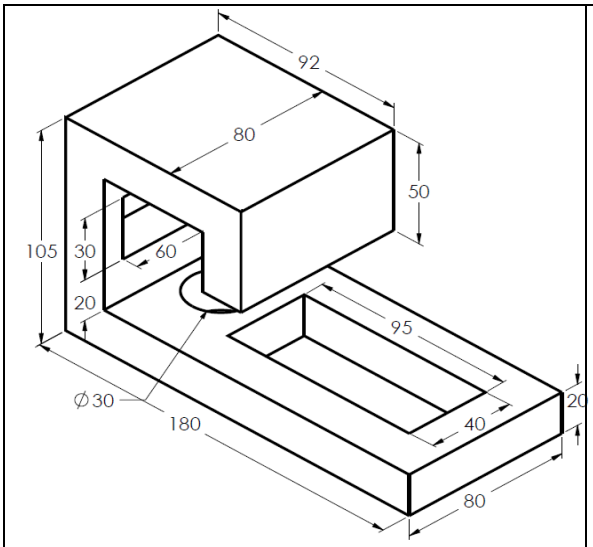


Fig. 7-32

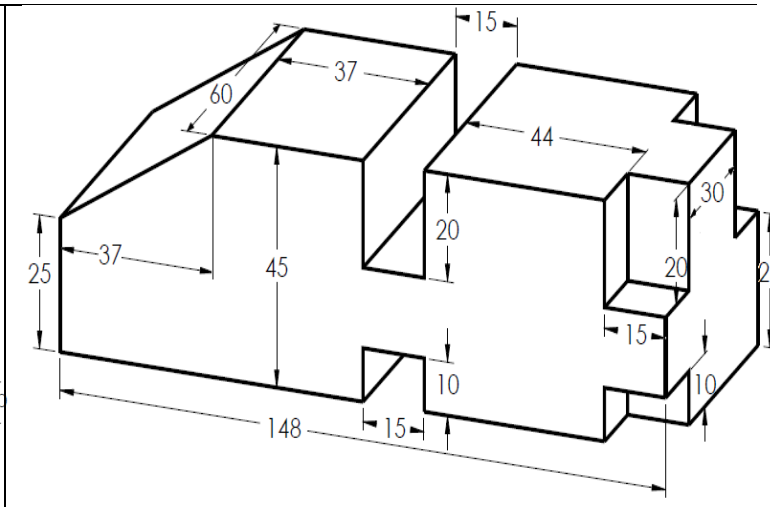
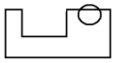
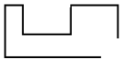
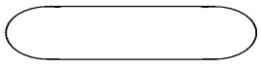
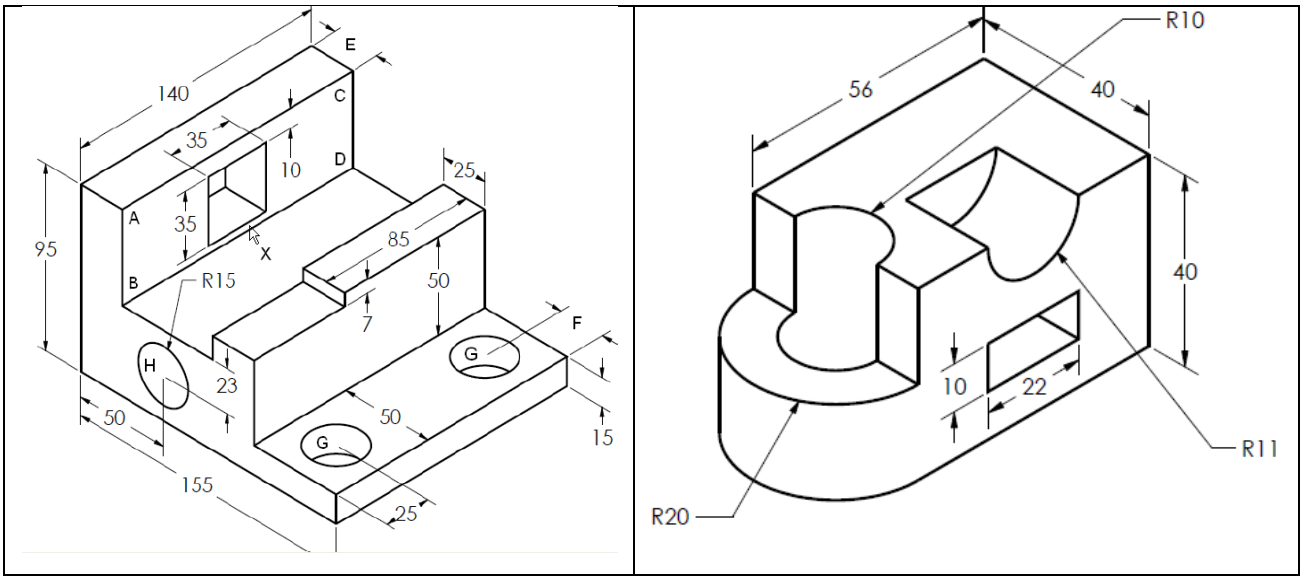


Fig.6-87

[1] . To slow down view transition, Go to Tools/Option/System Option/View/ Transition/View Transition/Slow

جلسه دوم

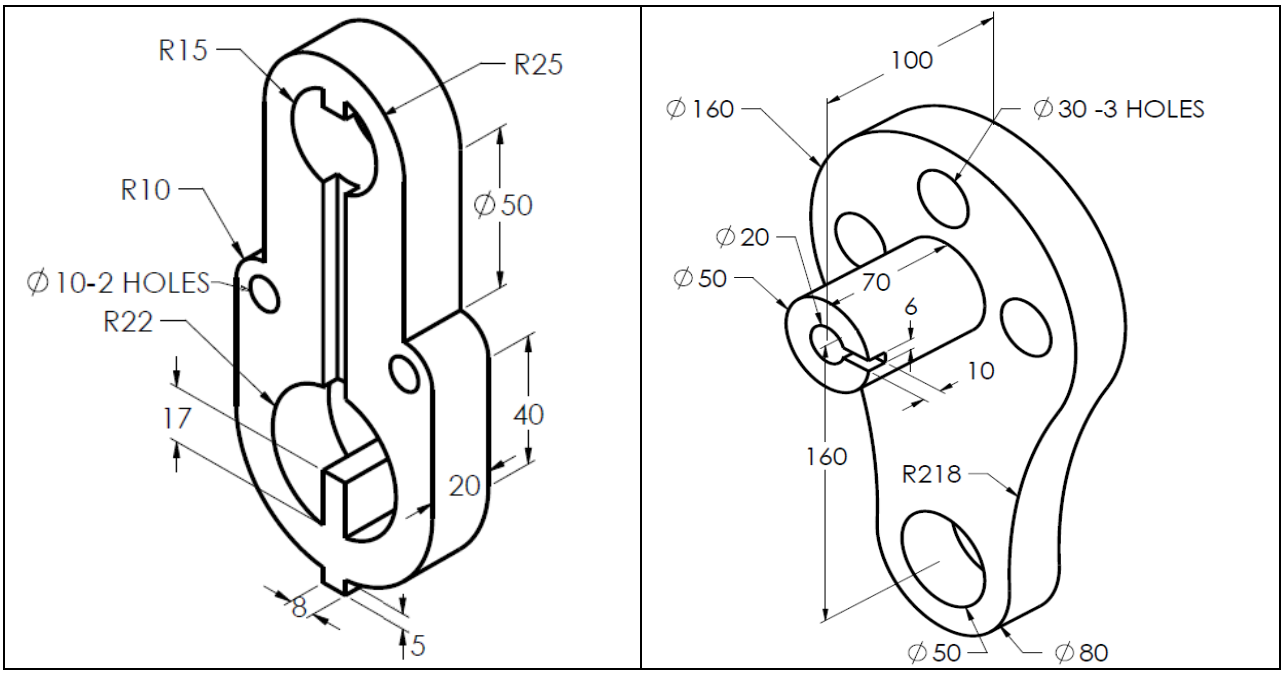
1. Zoom in and Zoom out[2]
2. Sketch all shapes, Line, Rectangle, arc, Polygon, slot
3. Sketch all shapes use Dimension- Convert IPS to MMGS
4. Extrude some of them
5. The problem in Extrude with wrong shapes like  or 
6. Trim, Split Entity
7. In sketch: Linear Pattern and Fillet and Chamfer, Circular Pattern
8. Draw  , go to Extrude and check symbol in Direction 1 and give 20 mm, check symbol in Thin Feature and give 5 mm, Note: in this case open shape like can be also extruded

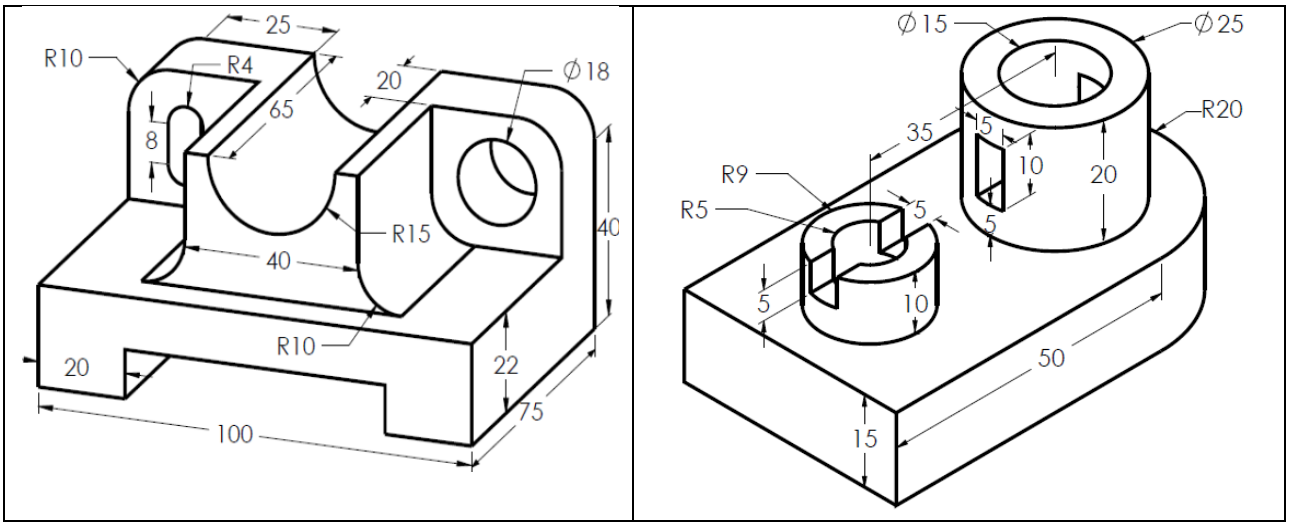


[2] . If the sketch is disappeared go to: View/Workspace/Default

جلسه سوم

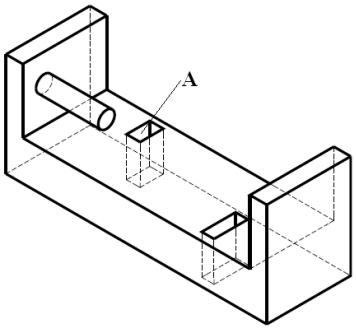
1. Normal to and Do some shapes on a rectangle cube
2. Relations, Lines, Points (merge), Fix, etc.
3. Relation, Tangent , Concentric, Coradial,
4. Edit area, enter and exit
5. Copy (Ctrl C -Ctrl V) and Move (just take it)





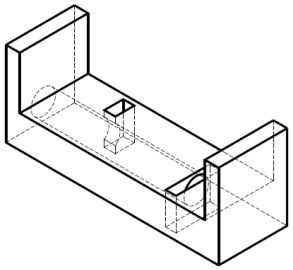
جلسه چهارم

1. Base Extrude. Blind, Through all,
   a) Up to surface(e.g. A), etc

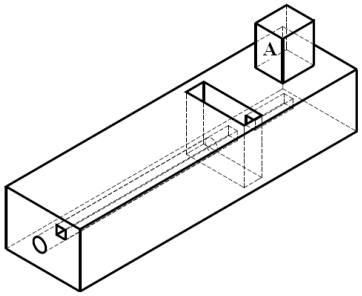


   b) Offset from Surface
   c) Mid Plane
   d) Up To body does not work

2. Cut Extrude, open shapes (line)
3. Cut Extrude, close shapes (rectangle or circle),
   a) blind, Through all,
   b) Mid plane
   c) Up to next (up to next feature, the next feature must be greater than cut profile),



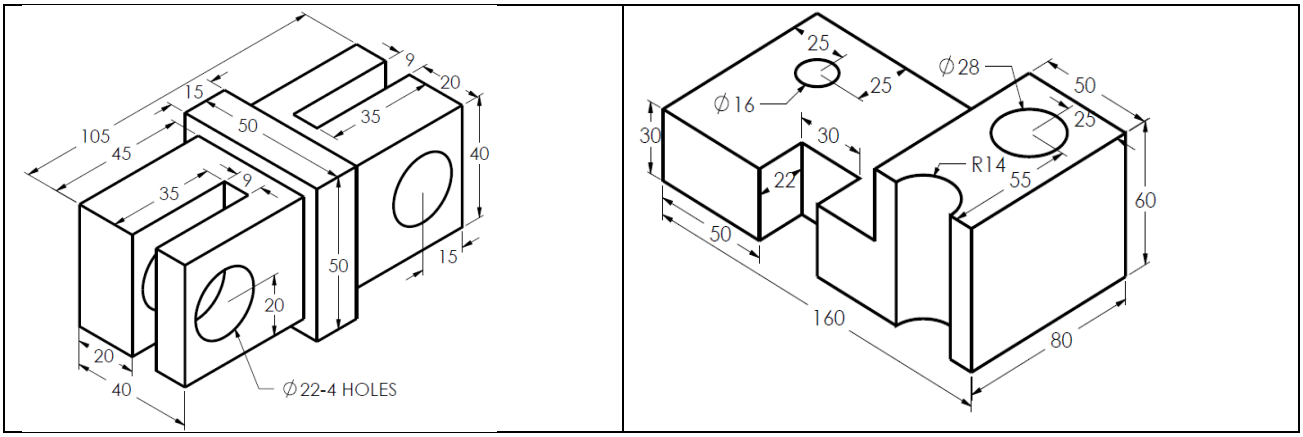
   d) Up to surface, (e.g. surface A)

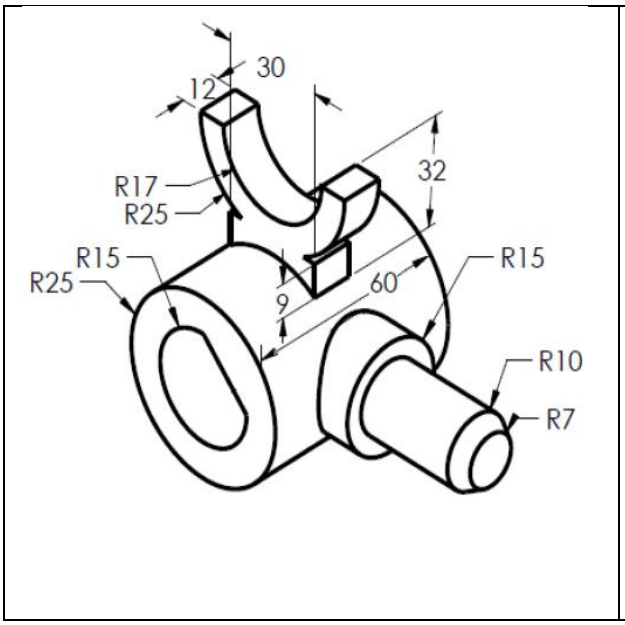


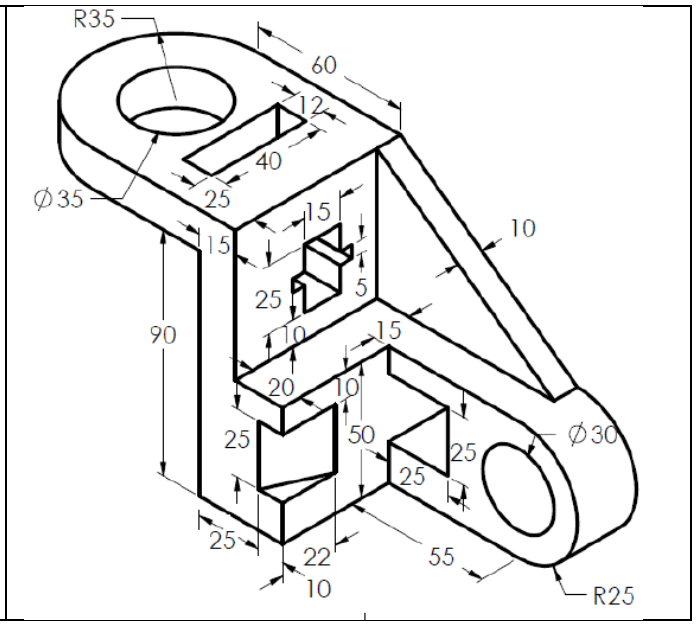
   e) Cut on a path,
   f) Off-set from surface
   g) the icon of **Up to body** is not available in some shapes, usually is used for 2 bodies with 2 planes

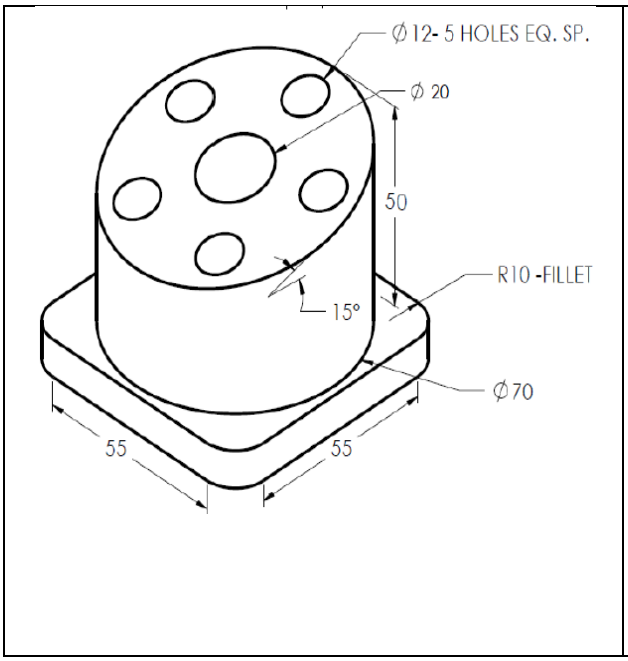
جلسه پنجم

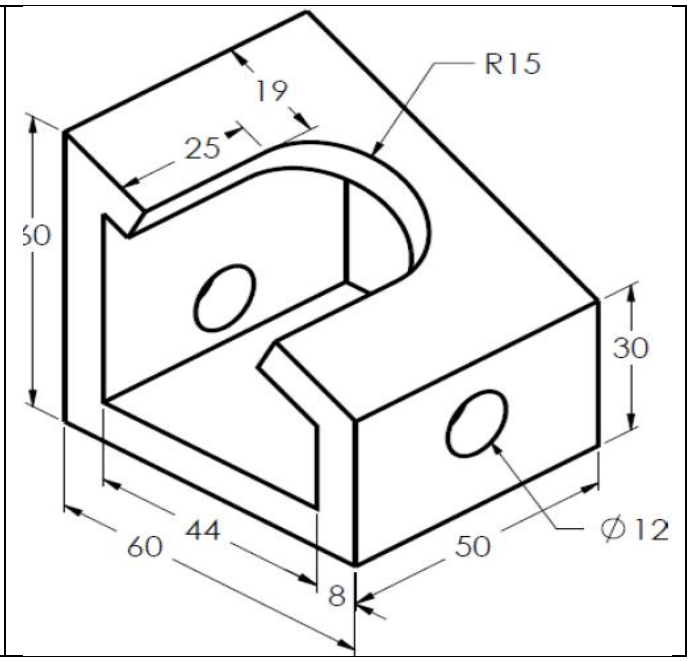
1. Sketch Mirror
2. Offset Entity
3. In Feature Edit, enter and exit edit
4. Revolved Boss
5. Revolved Cut
6. Sweep (Follow Path, Keep Normal Constant)
7. Sweep (Follow path and 1st Guide Curve, **Path= Straight Line**, Guide=3 point Arc)
   Note:
   a. draw a Circle in Top Plane,
   b. exit sketch
   c. Normal to in Front Plane and edit,
   d. draw path which is a straight line at the middle of the circle and then exit sketch,
   e. edit Front Plane again,
   f. draw Guide Curve which is 3-point -arc and then exit edit,
   g. go to feature and then Sweep
8. Sweep (Sweep Twist)
9. Sweep Cut
10. Plane, Front, Top, Side and 45.
11. Draw a sphere with a hollow spherical space inside
12. Draw a solid sphere and cut a corner by a spherical shape




30
12
32
R17
R25
R15
R25
R15
60
9
R10
R7

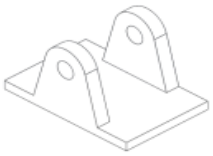

R35
60
12
40
25
Ø35
15
15
10
25
5
90
10
15
20
10
25
50
25
25
Ø30
25
22
55
10
R25


Ø12- 5 HOLES EQ. SP.
Ø 20
50
R10 -FILLET
15°
Ø70
55
55

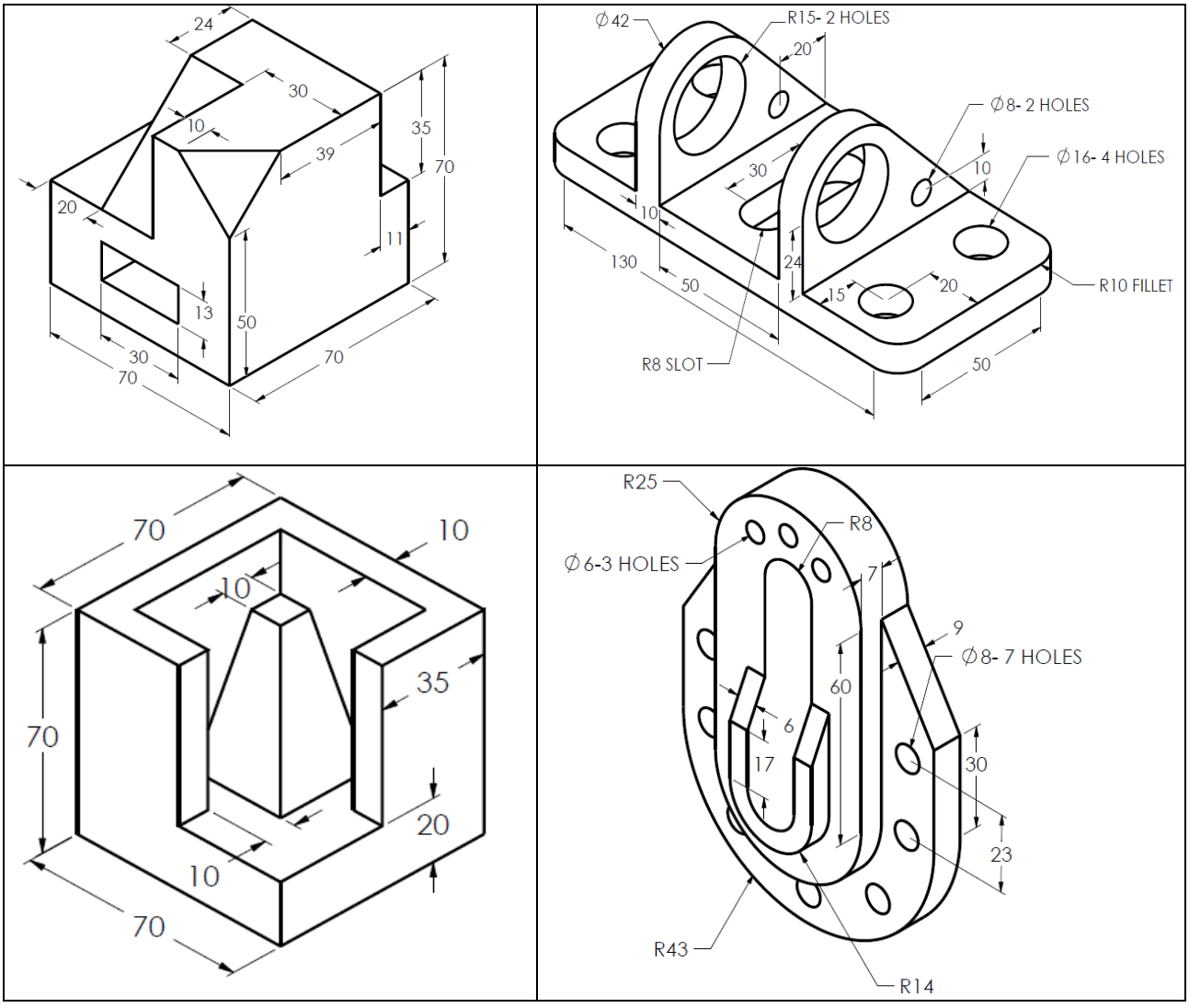

R15
19
25
50
30
Ø12
60
44
50
8

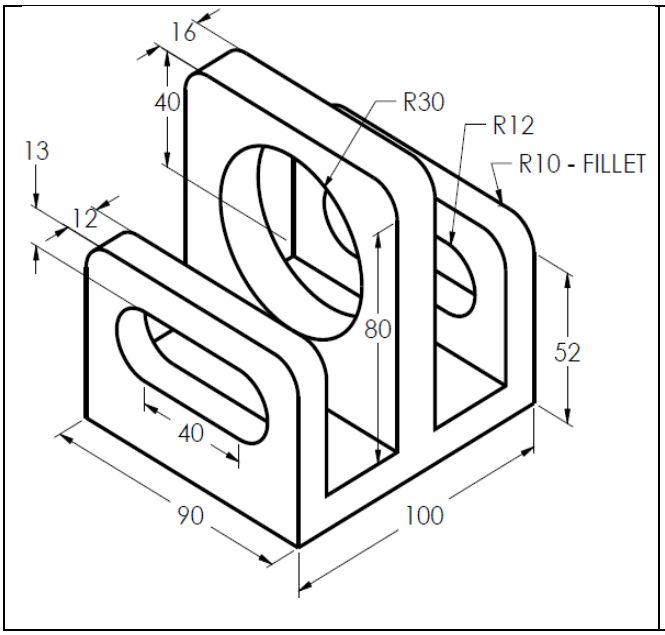
جلسه ششم

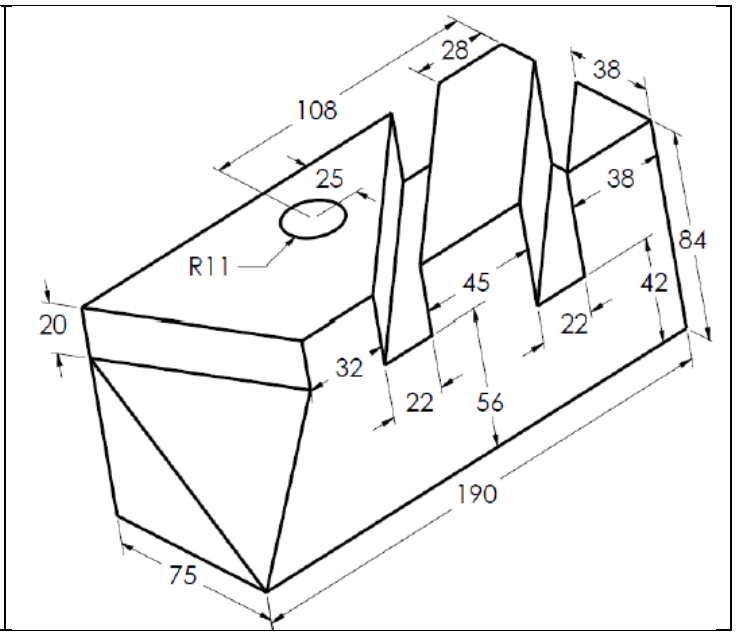
1. Loft and Loft cut (for Guide curve, Pierce Relation should be done for both circles)
2. Fillet and Chamfer(vertex) (Variable Radius Fillet, in **Variable Radius Parameters** a. click on **V1** and then give a radius, b. click on **V2** and then give another radius)
3. Linear Pattern in Feature
4. Circular Pattern in Feature
5. Shell (first extrude the shape of and then use shell)

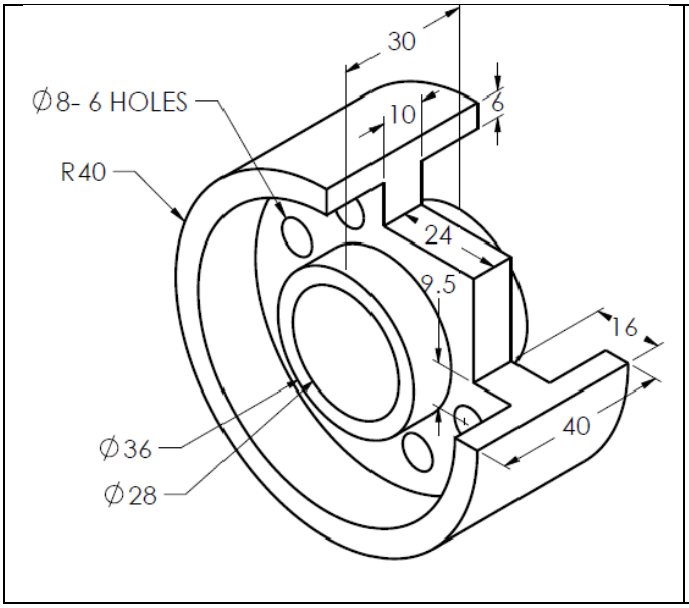


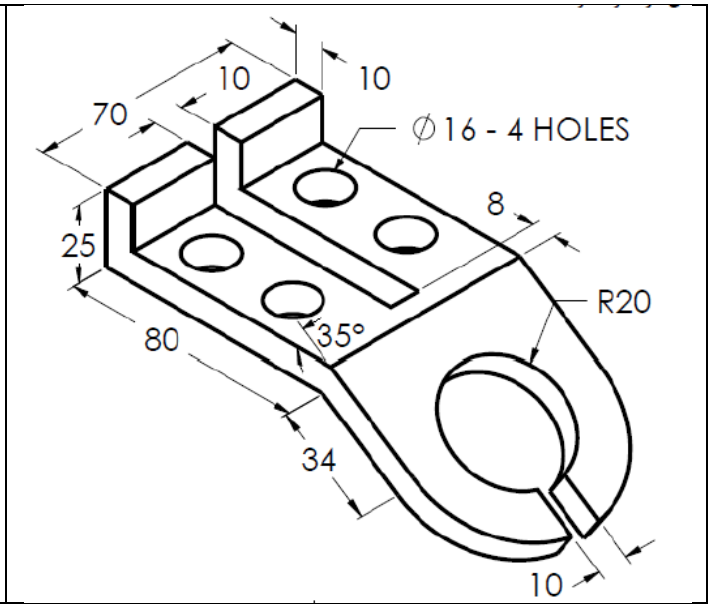
6. Mirror in Feature
7. Dome
8. Flex (from Insert/Feature/Flex), twist , Tapering and Bending, Stretching, بدون تغییر مقطع)




16
40
R30
R12
R10 - FILLET
13
12
80
52
40
90
100

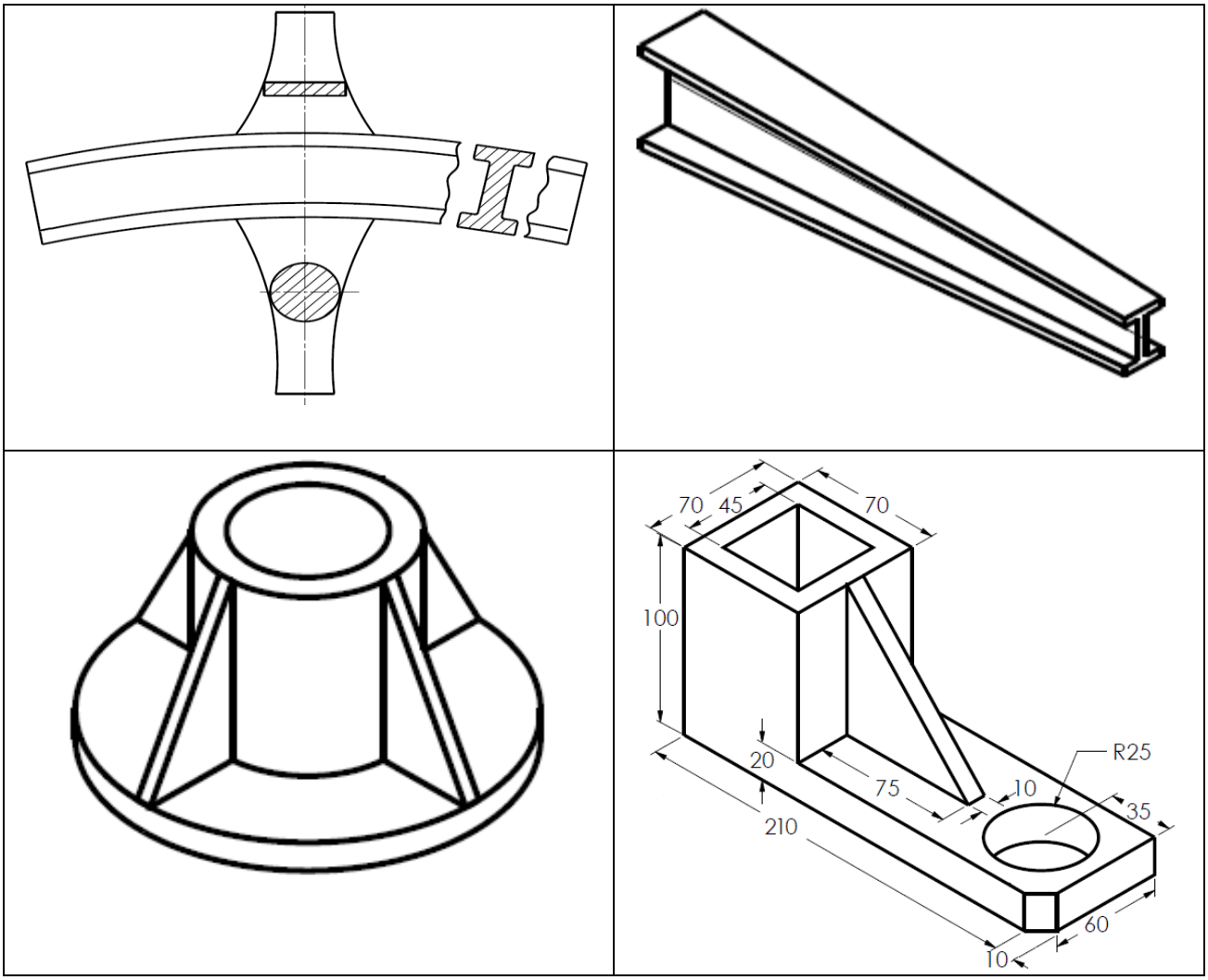

28
38
108
25
38
R11
84
45
42
20
22
32
22
56
190
75


30
Ø8- 6 HOLES
10
6
R40
24
9.5
16
Ø36
Ø28
40


10
10
70
Ø16 - 4 HOLES
8
25
R20
80
35°
34
10

جلسه هفتم

1. **Normal to** on a small surface of a cube and draw a large sketch on it
2. Make a Shaft and try to sketch on its curved side (which is not possible)
3. Reference geometry and sketch on a circular surface
4. First Reference and a point as the second Reference
5. First Reference and a Face as the second Reference(45° Plane)
6. Rib (simple and complex)
7. Flange
8. Spring

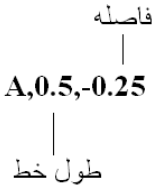


جلسه هشتم

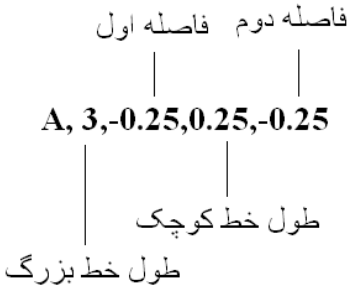
1. **Drawing Area** for 3-views and its adjustment;

To bring back Task Pane (پنجره), go to: View/Task Pane

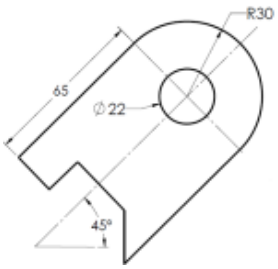
a) Scale
b) Line Font
c) Hidden Line: Tools/Option/Document Properties/Line Style/Dashed or Hidden Line/



d) Center Line: Tools/Option/Document Properties/Line Style/Center Line/



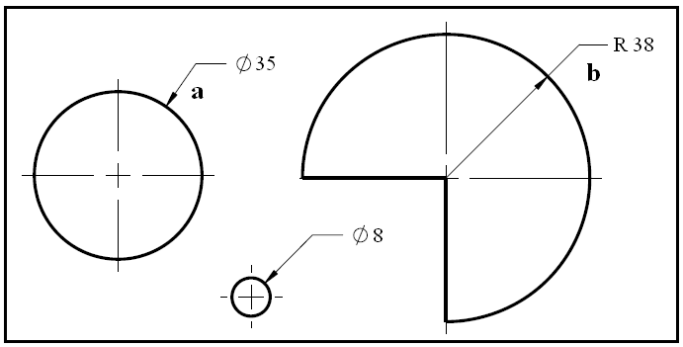
e) To bring Automatic Center Line Go to: Annotation/Center Mark
f) Click the Center mark in Corresponding View, Then remove the Green Tick in
Click on the Center mark in Corresponding View, Then remove the Green Tick in the box □Use Document Default and Then put a Tick in √□Center Line Font



g) Use angle in case of not needing horizontal Center Line e.g.

2. Sketch in Drawing Area, Change Layer

3. Table and Frame

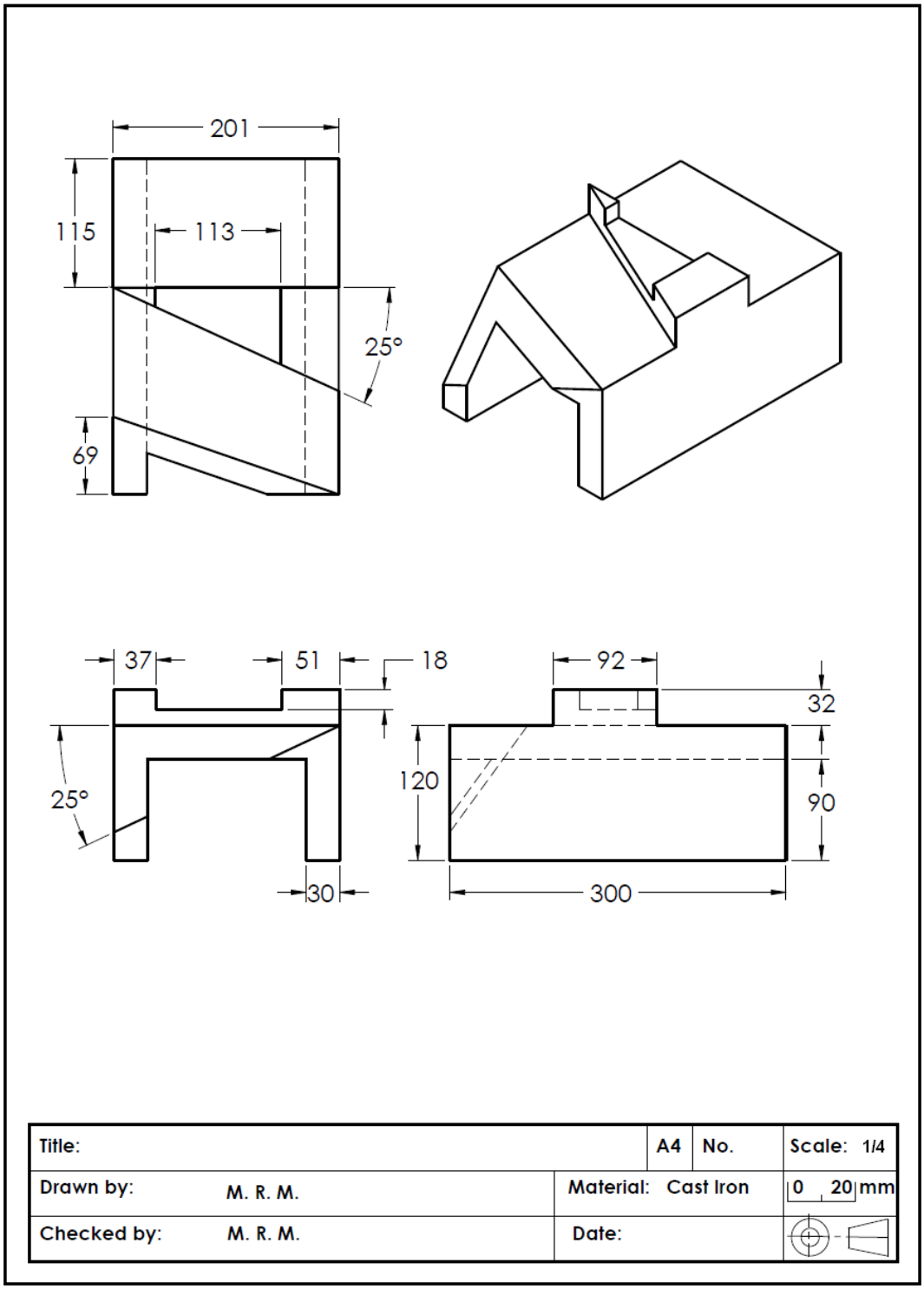


a. go to Dimension/Leaders/Outside/Diameter

(Note: if the arrow is inside of the circle, just take it and bring it out)

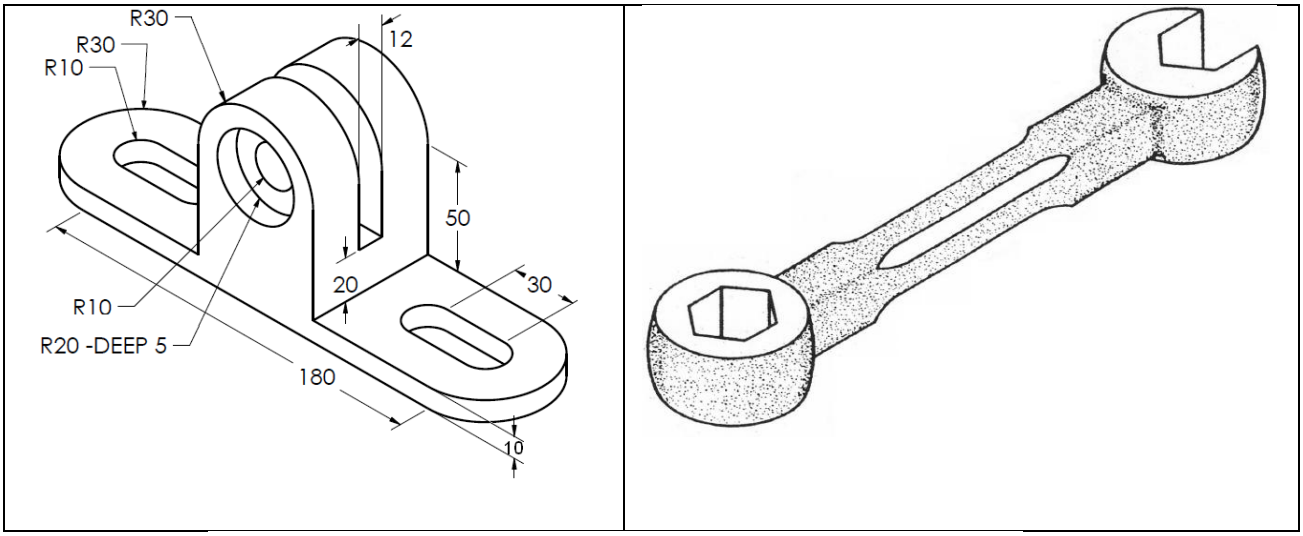
b. go to Dimension/Leaders/inside/Radius

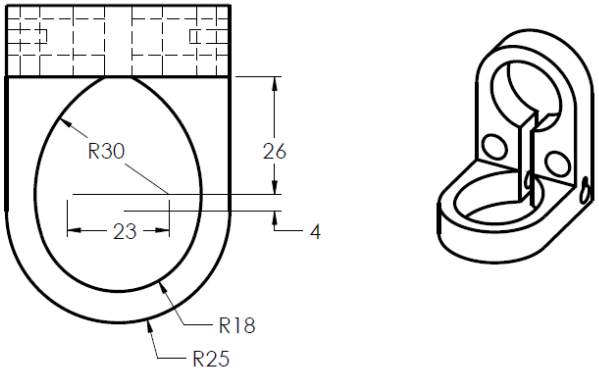
(Note: if the arrow does not go inside of the circle, just click on the arrow)

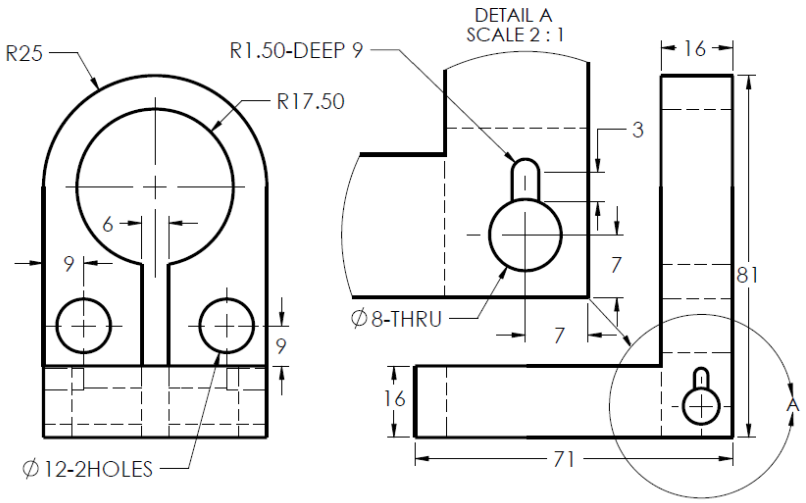


جلسه نهم

1. Sections, Hatching
2. Removed Section
3. Detail View

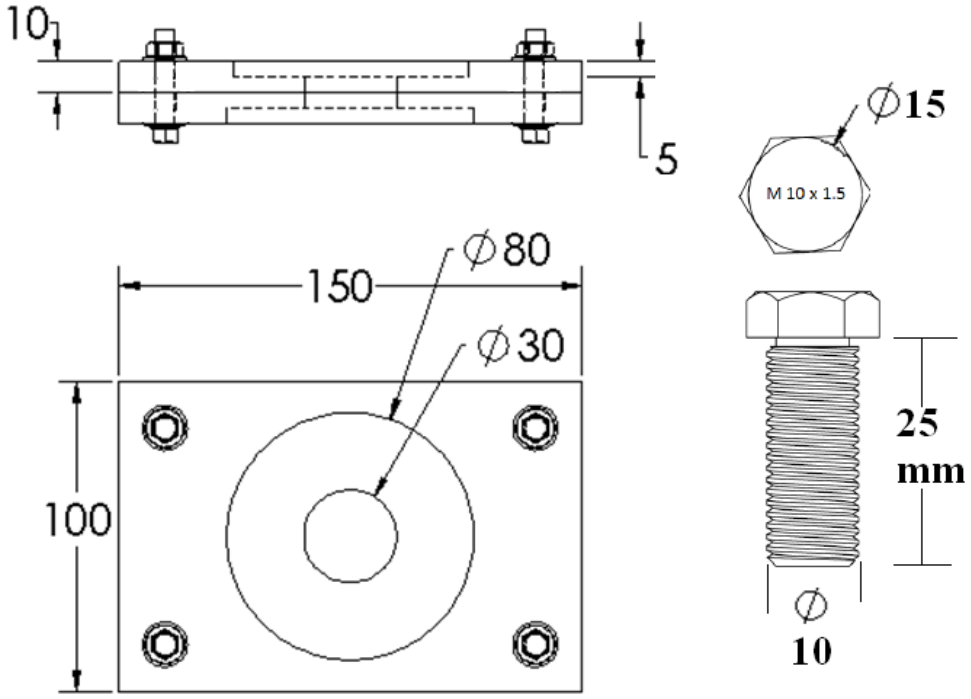


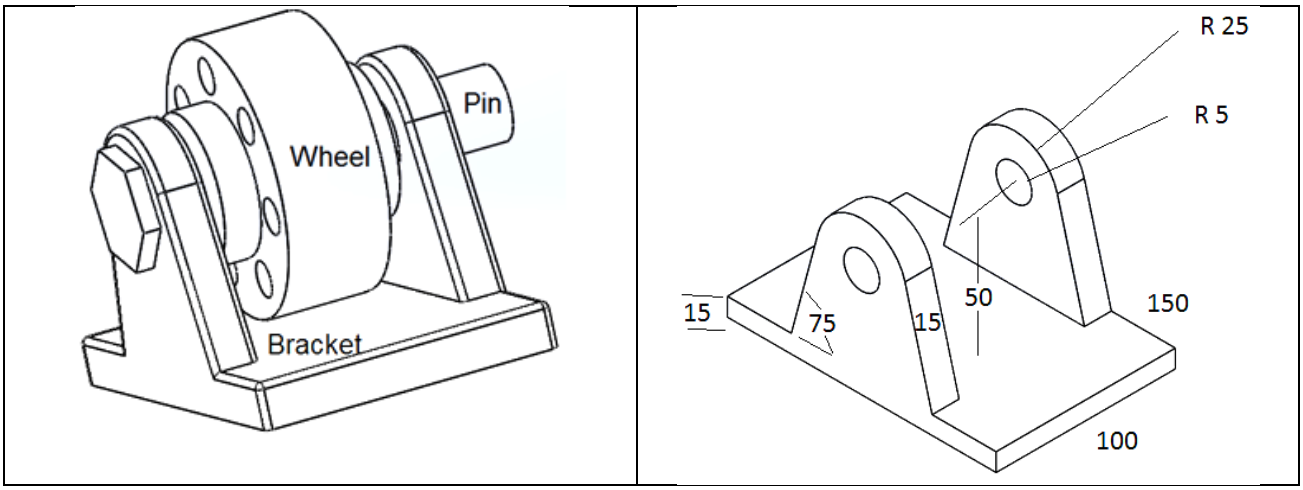




جلسه دهم

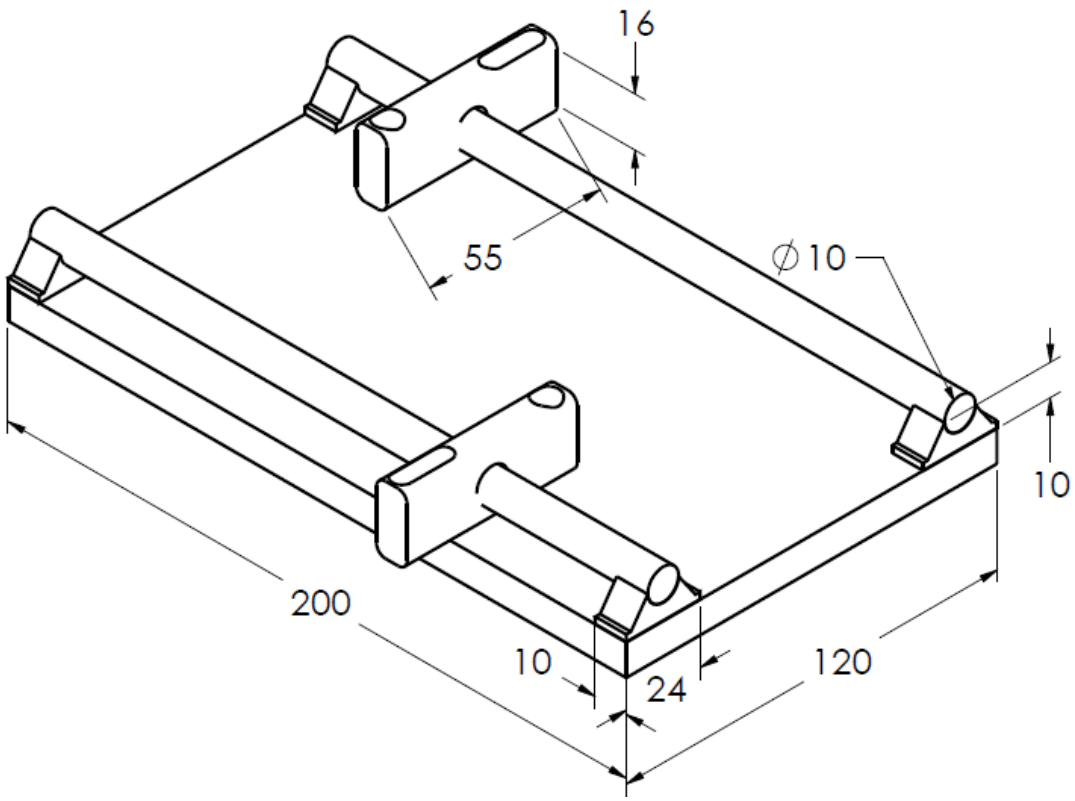
1. Basic Assembly
2. Exploded View





جلسه یازدهم

1. Derived Sketch
    a. Click on the sketch of the rectangle feature made in Top Plane
    b. Press Ctrl
    c. Click on the surface you want to be deprived that sketch
    d. Release Ctrl
    e. Go to: insert/derived sketch
    f. Go to: Feature/Extruded Cut
    g. Change the sketch in the Top Plane
2. Copy Feature
    a. Click on the Feature you want to copy
    b. Go to: Edit/Copy
    c. Right click on the surface you want to paste on it and do Normal to
    d. Go to: Edit/Paste
    e. Note that the extrude is as much as the length of the source feature
3. Advanced Assemblies, Limit Mate, Piston and Cylinder
4. Fix Mate, to fix any part in assemblies Area Right click on the part and press Fix
5. Advanced Assemblies, Linear Coupler

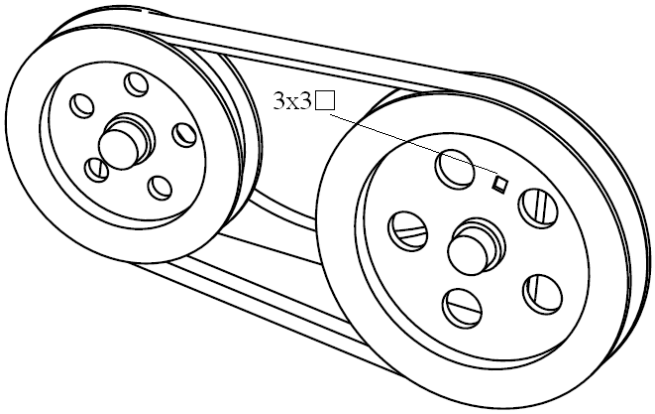


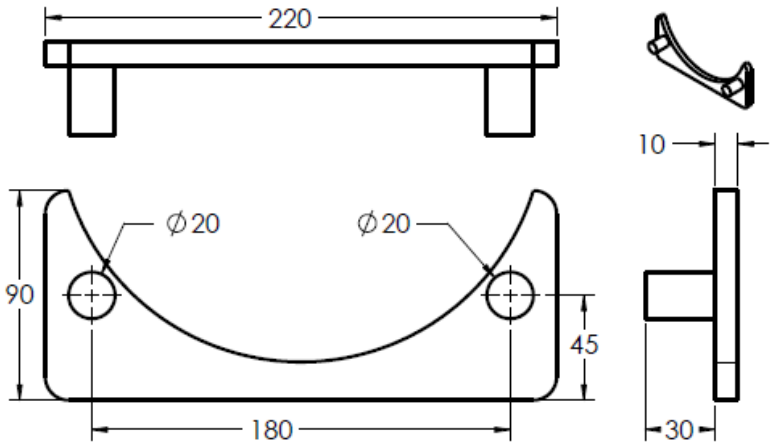
جلسه دوازدهم

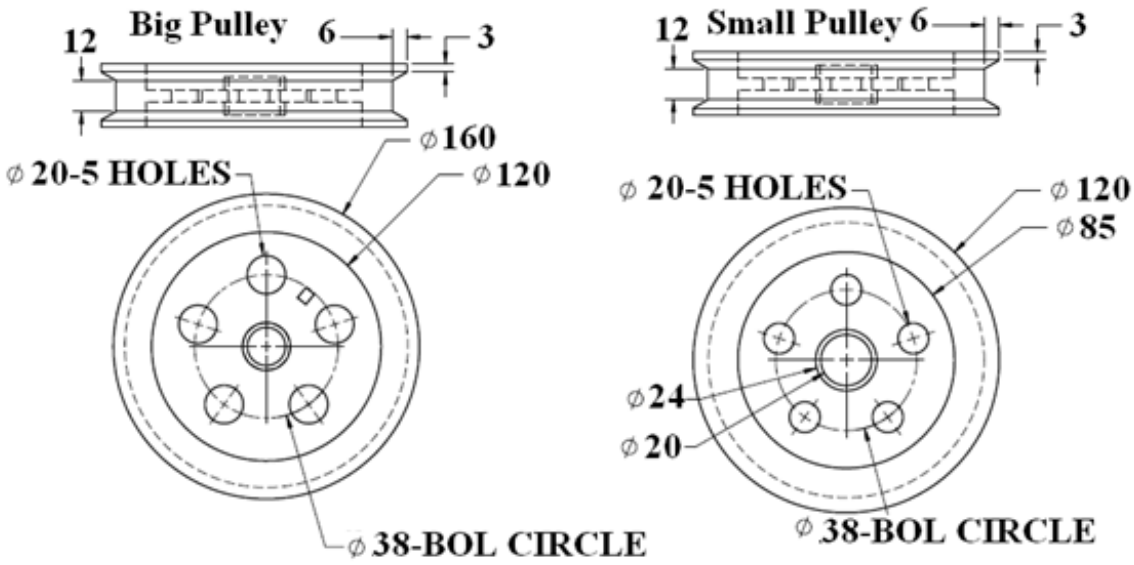
1. Advanced Assemblies, Width Mate

a. Make a shaft with 20 mm diameter and 100 mm length
b. Make a collar with 20 mm inside diameter and 35 mm outside diameter
c. Go to New Assemblies and put them together using Standard Mate
d. Go to Advanced Mate and select Width
e. In **width Selection** box select both side of the shaft
f. In **Tab Selection** box select both side of the collar

2. Advanced Assemblies, Belt



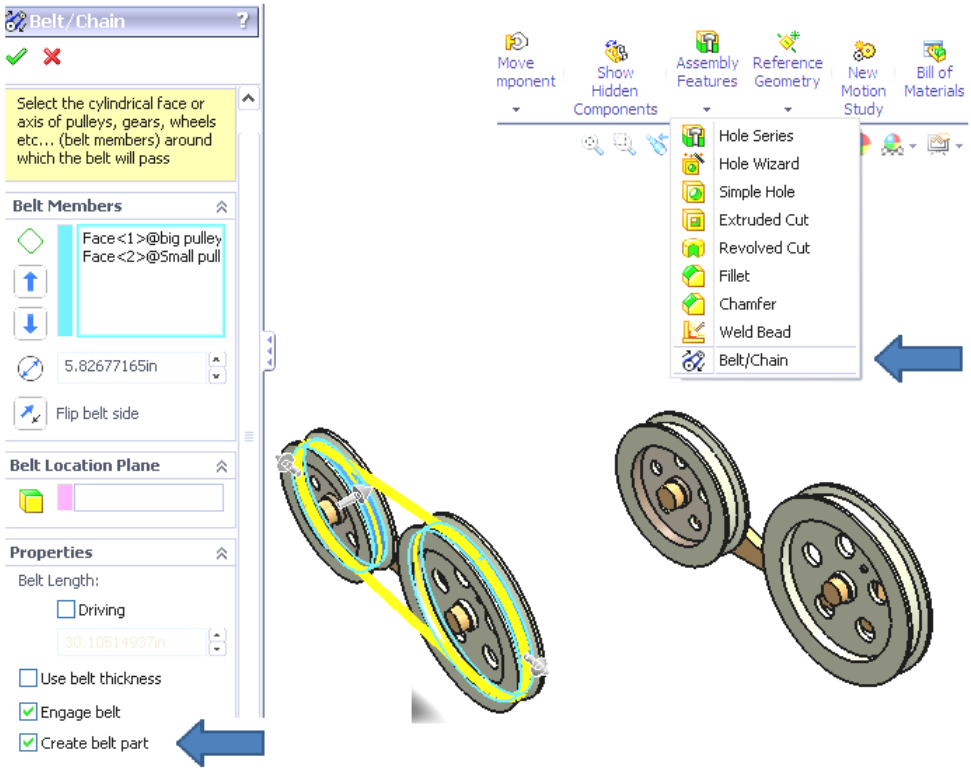




a. Open an New/Assembly

b. Assemble all components

c. From Assemblies Features, Select Belt & Chain



d. Select the face of pulley groove face

e. Put a check symbol in Create Belt Part

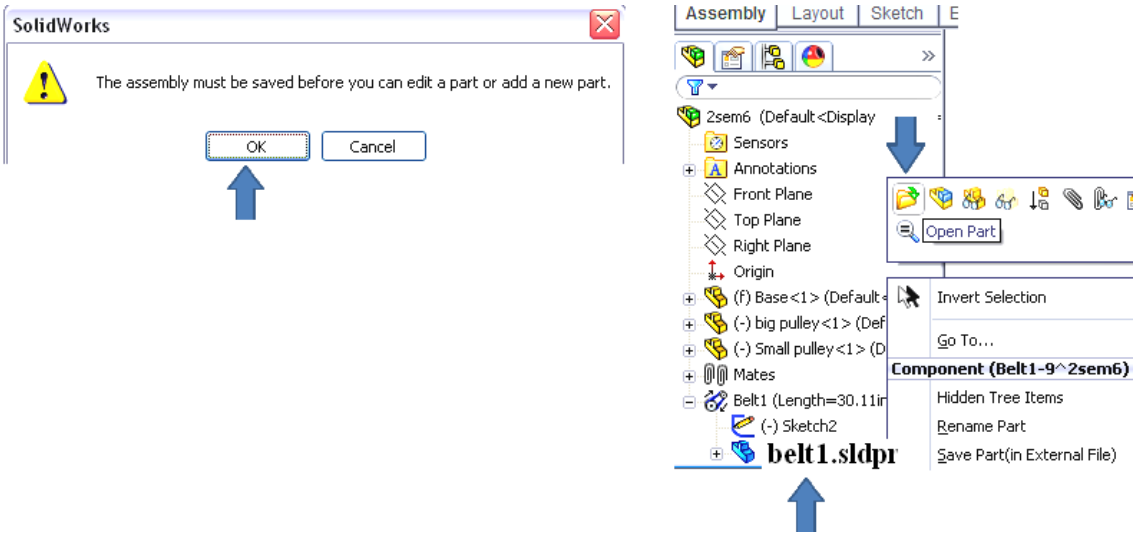
f. Press green Check symbol

g. Press **ok** and save the assembly

h. Open belt

i. Right Click on belt1

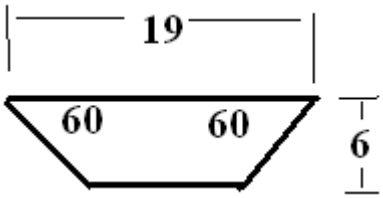
j. Open Part



k. The belt will be appeared

l. Reference Geometry / select a proper plane, then second Reference/ the point junction of the line and the **small curve** part of the belt

m. Using the same dimension of the pulley's groove make a profile for the belt



n. Using **Sweep** finish the belt and exit

o. Save changes to **belt1.sldpr**

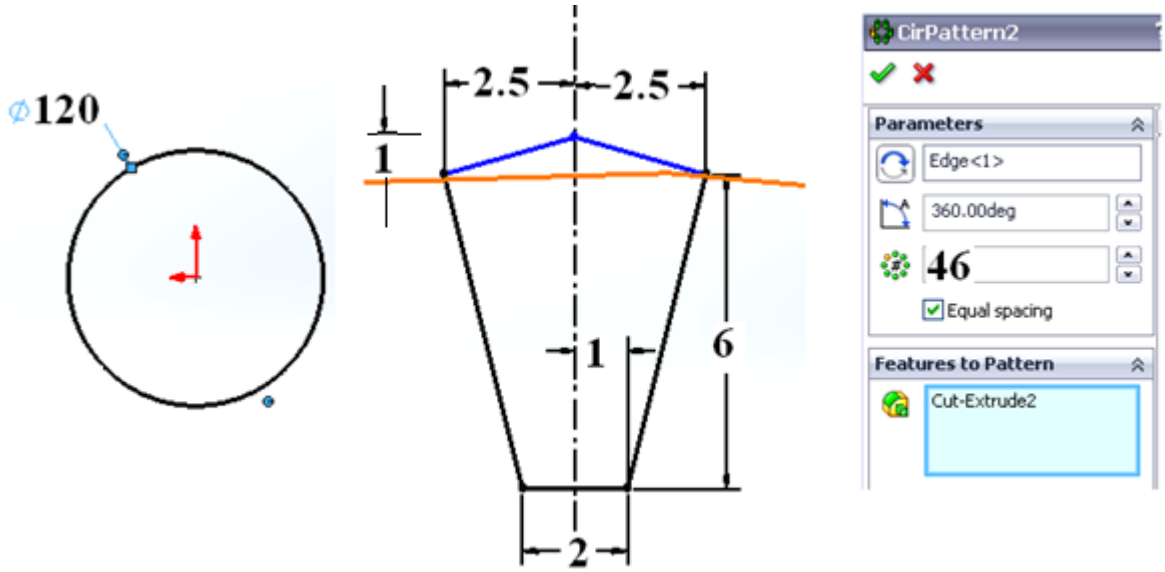
p. Press Yes

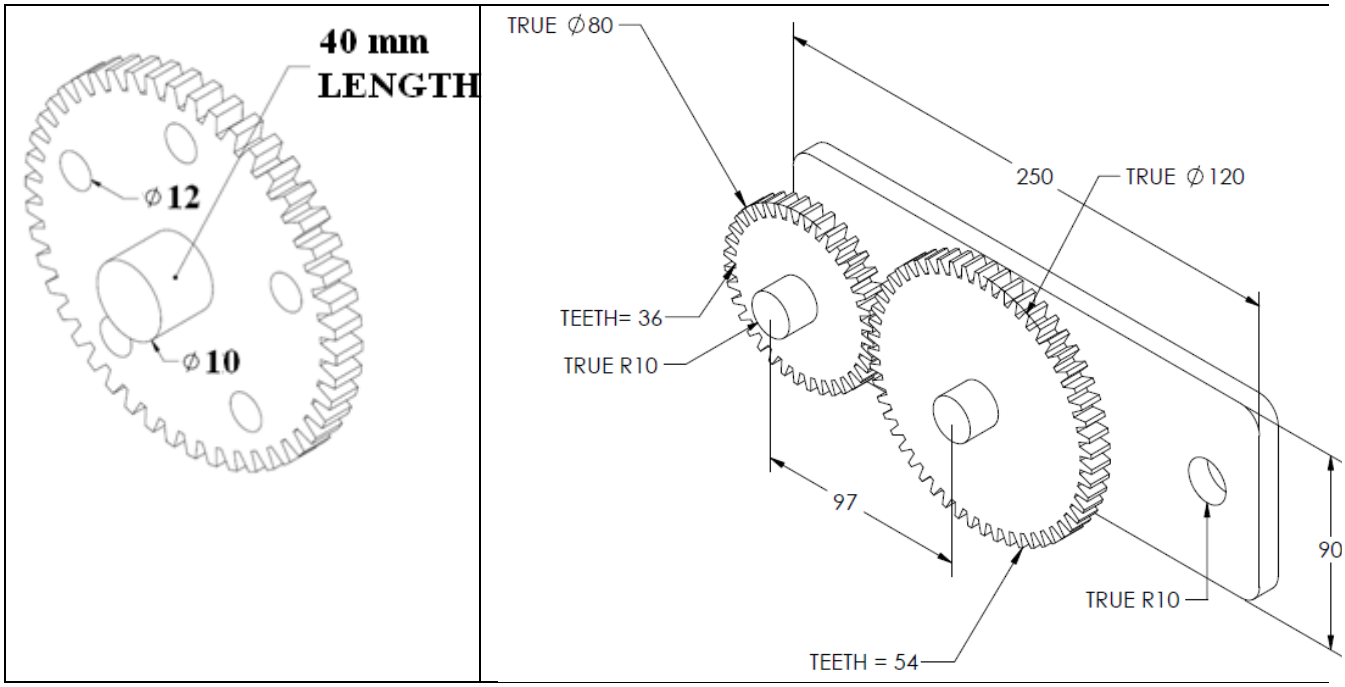
q. Hide the belt Line

r. Save it again

s. Note that the belt does not rotate

3. Advanced Assemblies, Gear

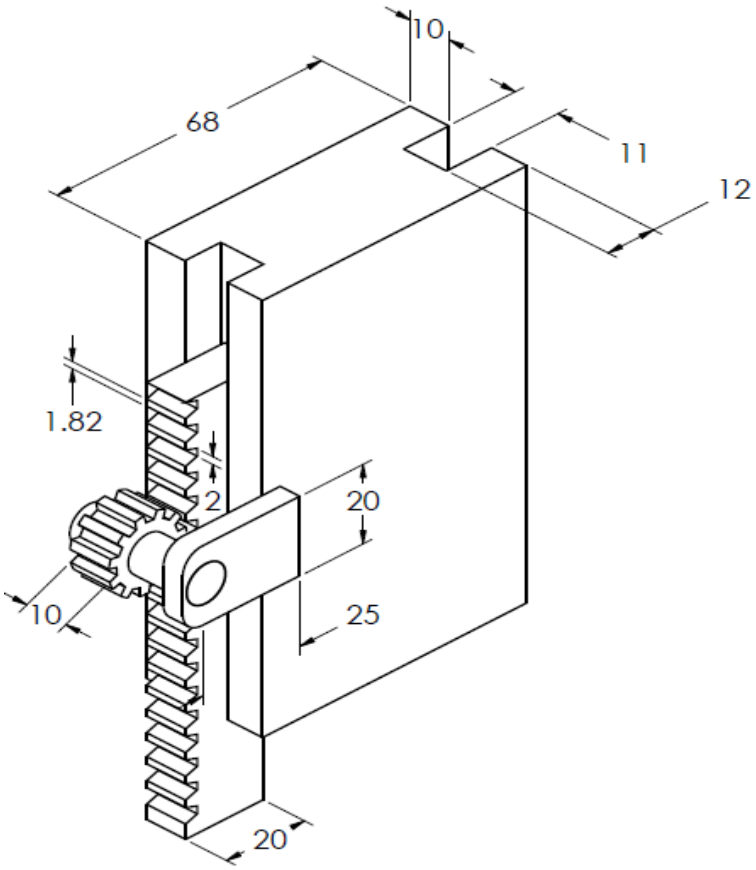

∅120
2.5
2.5
1
1
6
2
CirPattern2
Parameters
Edge<1>
360.00deg
46
Equal spacing
Features to Pattern
Cut-Extrude2

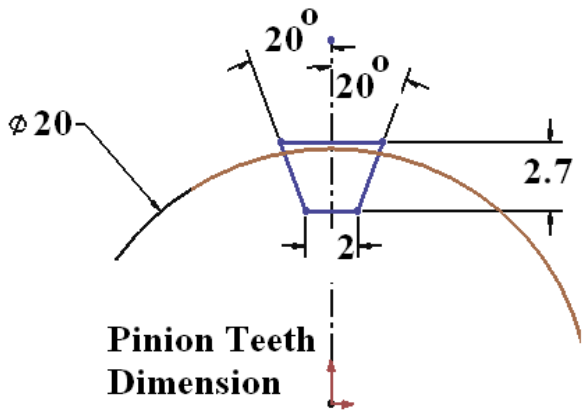

40 mm
LENGTH
∅12
∅10
TRUE ∅80
250
TRUE ∅120
TEETH= 36
TRUE R10
97
90
TRUE R10
TEETH = 54

جلسه سیزدهم

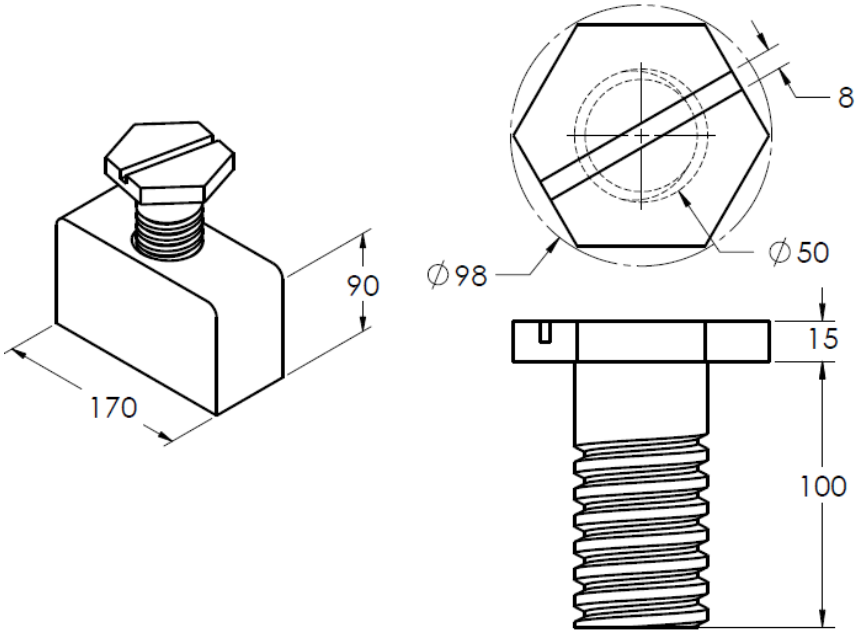
1. Advanced Assemblies, Rack & Pinion

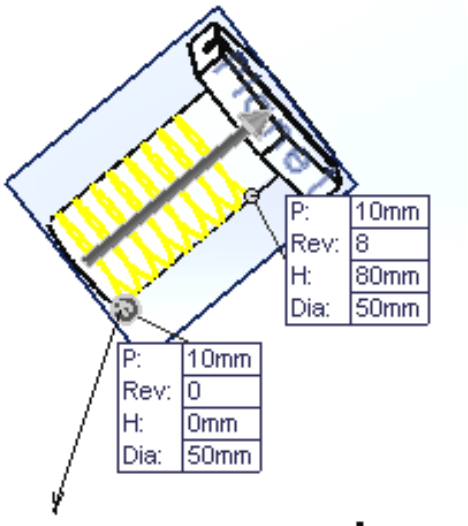
Note: the edge of the rack and the edge of the pinion shaft must be selected

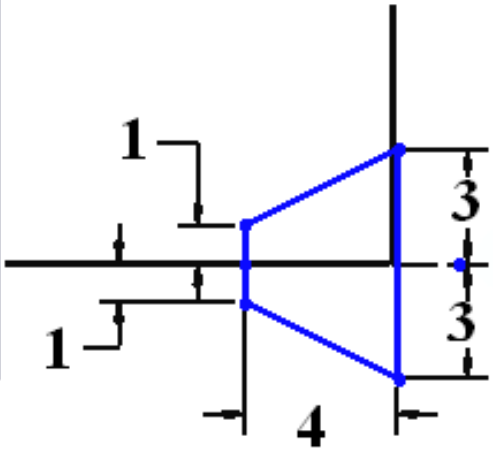




2. Advanced Assemblies, Screw



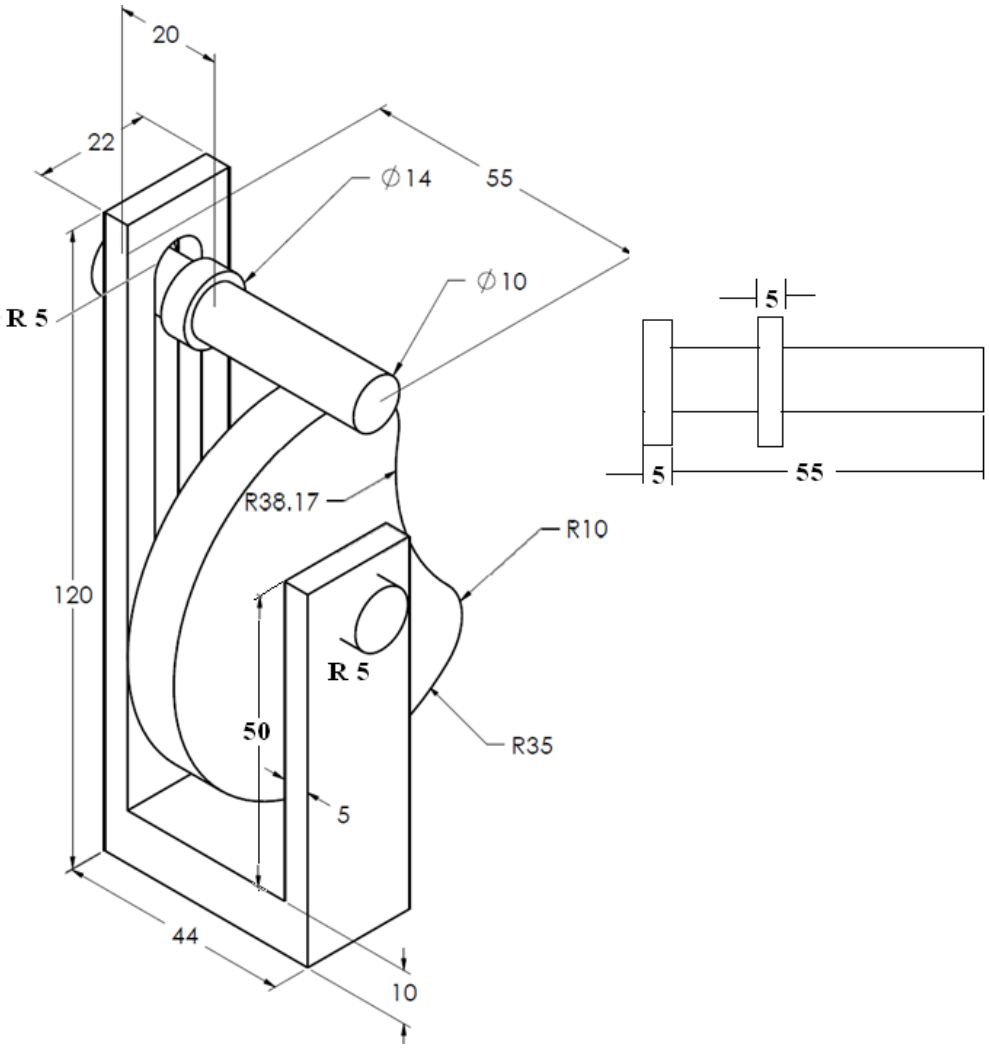


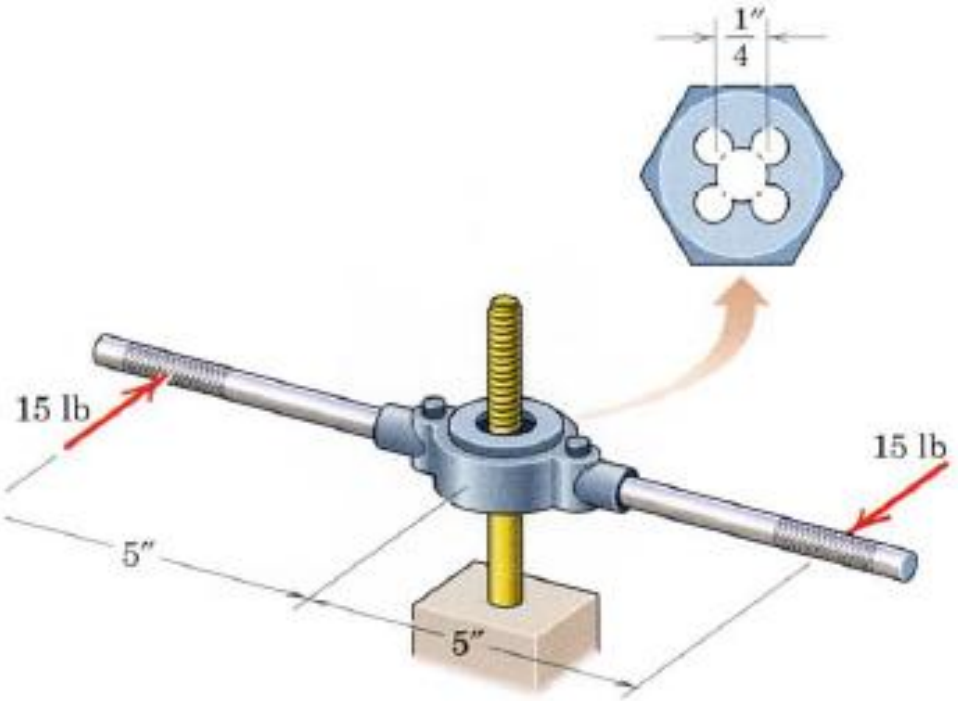


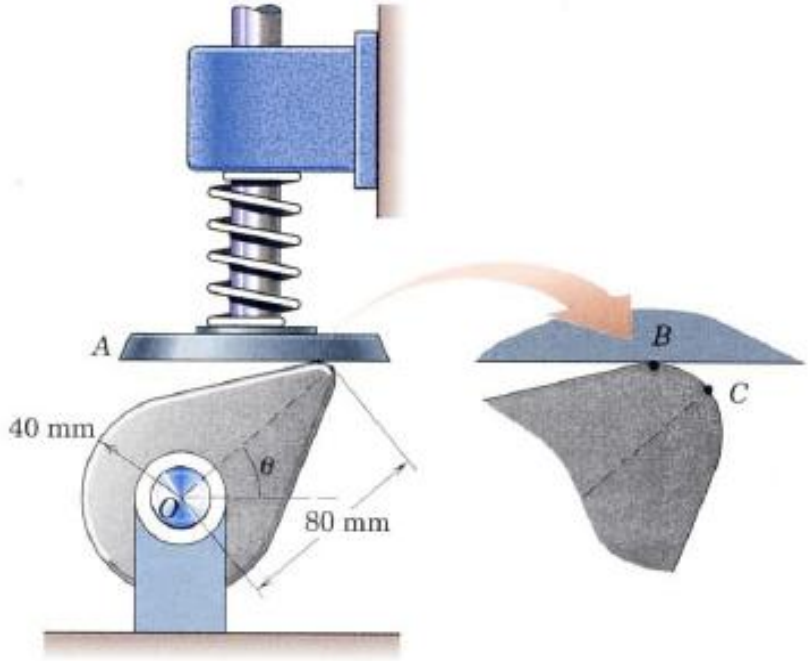
جلسه چهاردهم

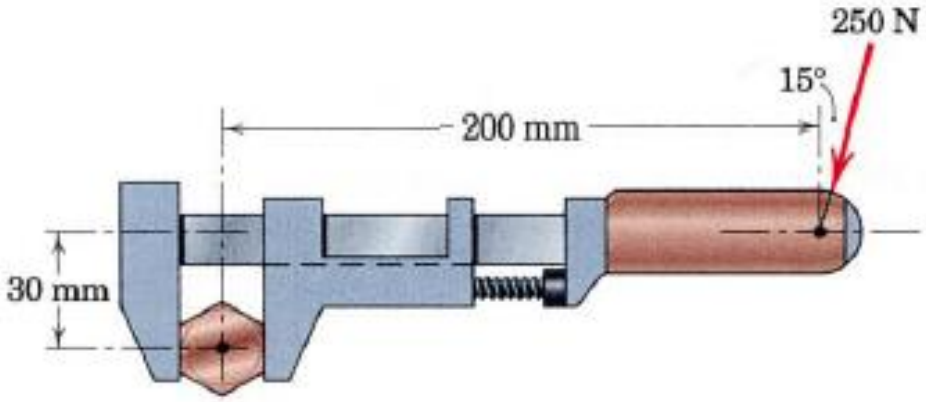
1. Advanced Assemblies, Cam and Cam Follower

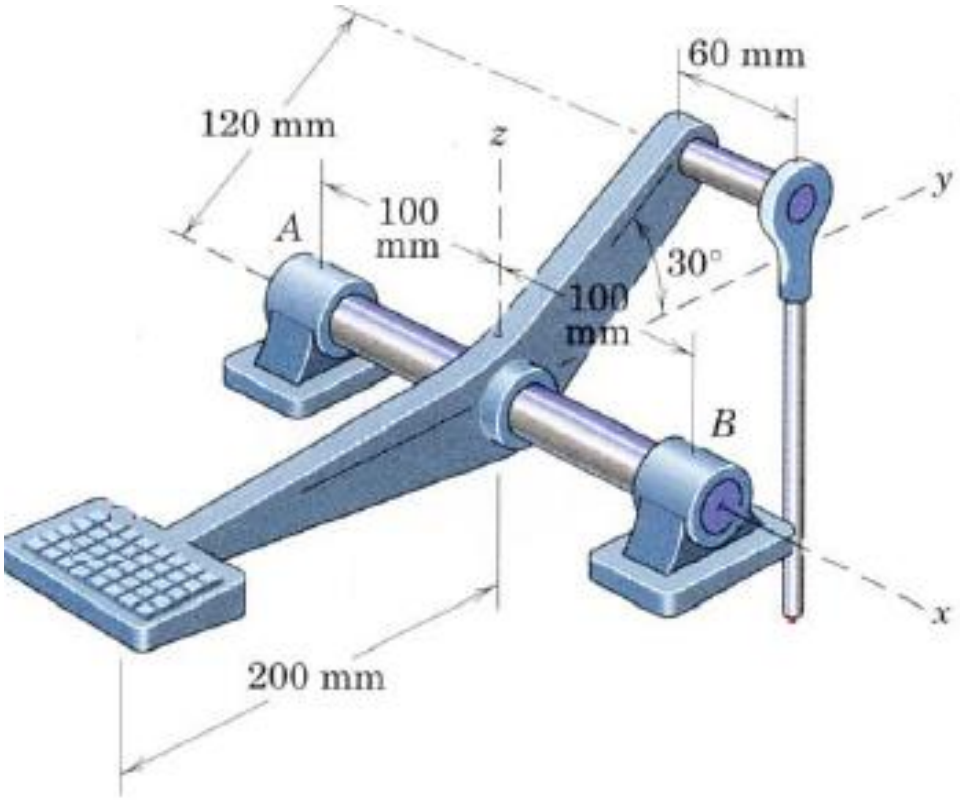
   Note: Mate/Mechanical Mate/Select the face of Cam and in box of Cam Follower select Cam Follower

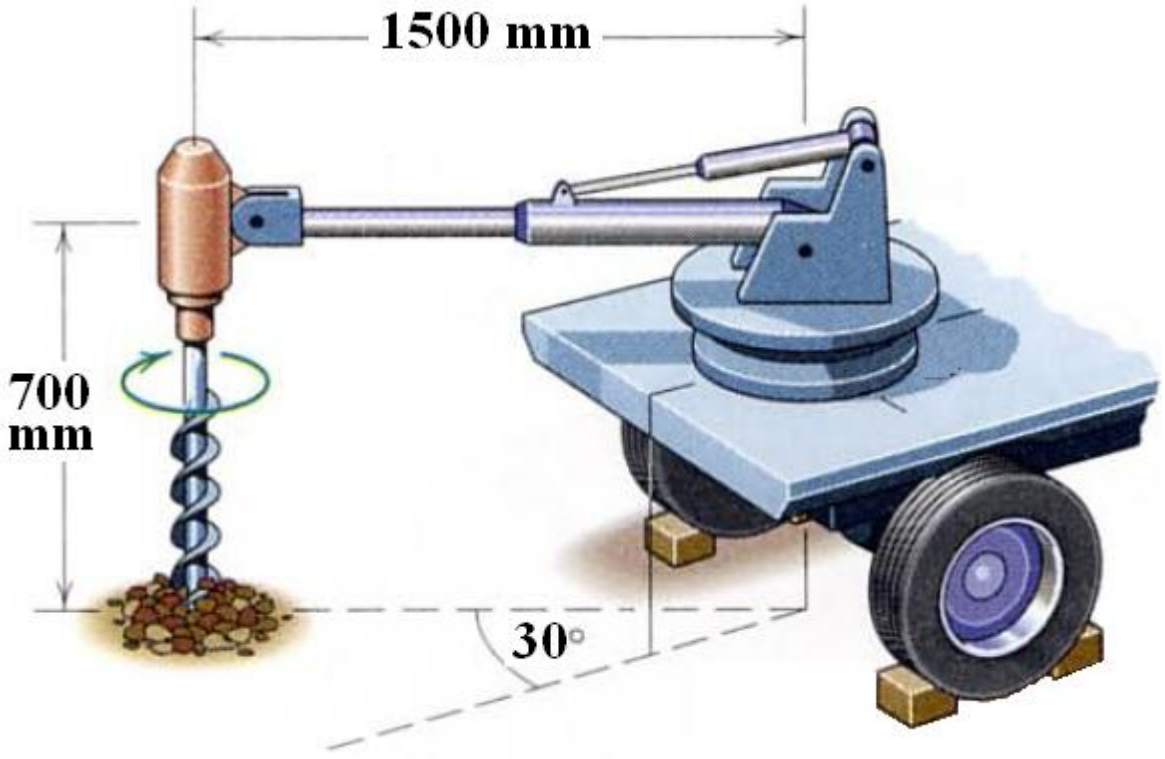



1″
4
15 lb
15 lb
5″
5″


A
B
C
40 mm
θ
O
80 mm


250 N
15°
200 mm
30 mm


60 mm
120 mm
z
y
A
100
mm
30°
100
mm
B
x
200 mm


1500 mm
700
mm
30°