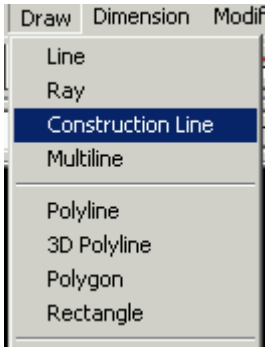
MAK112E Computer Aided Technical Drawing

AutoCAD - Basic Drawing Commands

Construction Line: This type of line gives you an idea about the part's border.



Change ORTHO mode to draw horizontal or vertical

and then select the End point of the object and select last point. To finish drawing construction line press return.

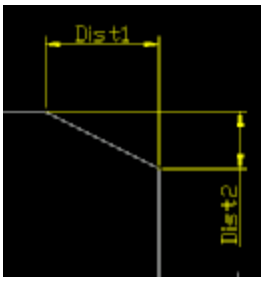
Chamfer 

Chamfer command to cut the corner with a given distance. You have to set the Dist1 and Dist2 using d

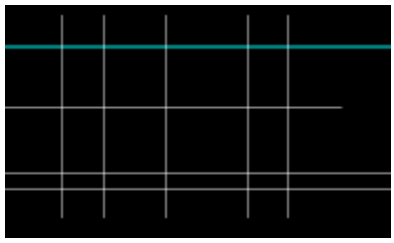
Dist1 = 10

Dist2 = 10

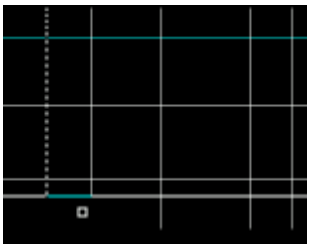
```
Command:
Command:
Command: _chamfer
(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000
Polyline/Distance/Angle/Trim/Method/<Select first line>: d
138.1174,5.5432 ,0.0000  SNAP GRID ORTHO OSNAP MODEL TILE
```



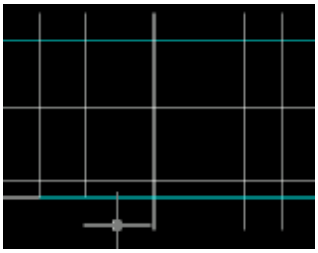

```
(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000
Polyline/Distance/Angle/Trim/Method/<Select first line>: d
Enter first chamfer distance <10.0000>: 0
Enter second chamfer distance <0.0000>:
Command:
272.4388,53.9735 ,0.0000  SNAP GRID ORTHO OSNAP MODEL TILE
```



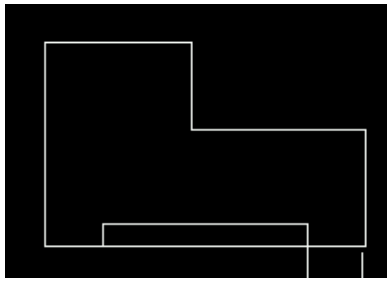
```
CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Polyline/Distance/Angle/Trim/Method/<Select first line>:
Select second line:
216.3742,98.3193 ,0.0000  SNAP GRID ORTHO OSNAP MODEL TIL
```



```
Command: _chamfer
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.000
Polyline/Distance/Angle/Trim/Method/<Select first line>:
Select second line:
Command:
234.4784,93.0678 ,0.0000  SNAP GRID ORTHO OSNAP MODEL T
```

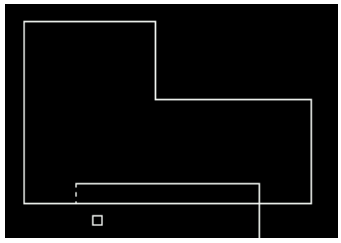


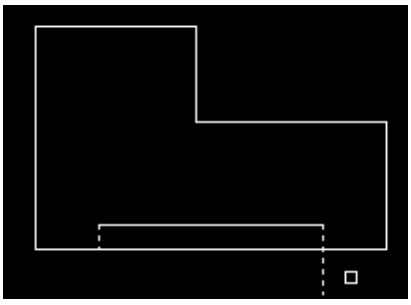
Repeat the chamfer command until getting this drawing

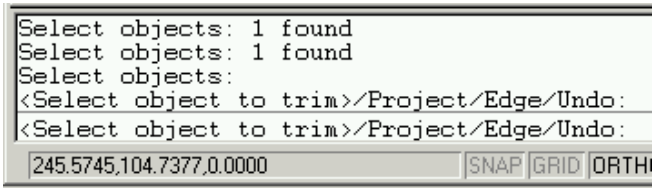


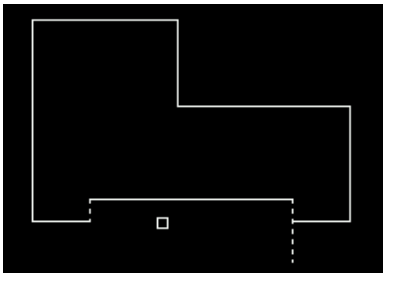
Trim 

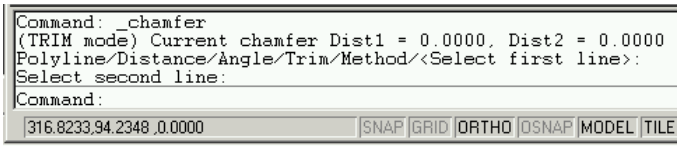
To use this command click trim icon

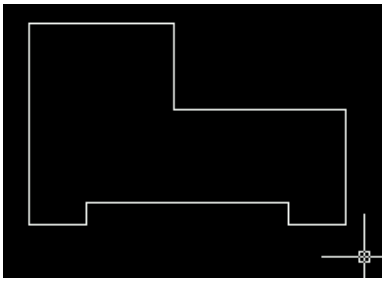




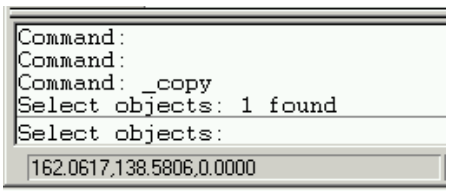


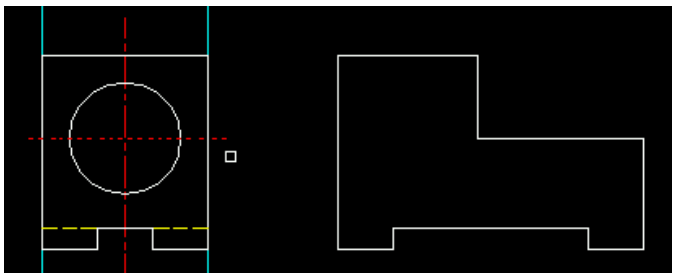




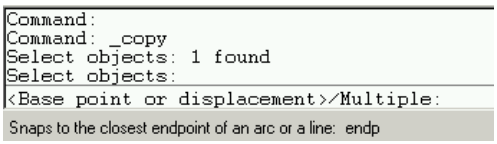


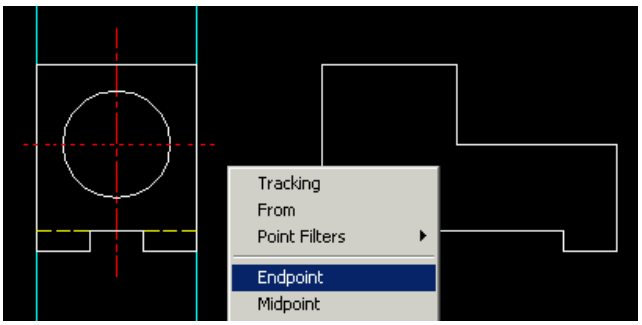
Copy Command 

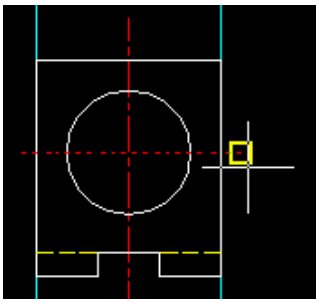


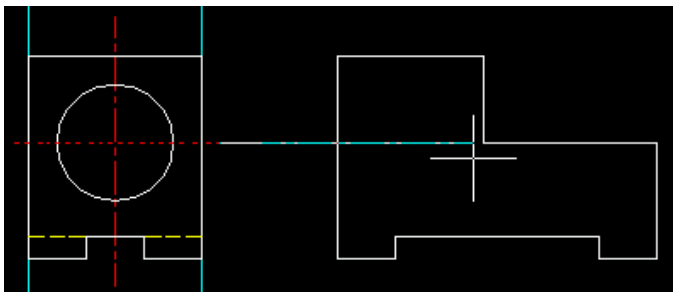


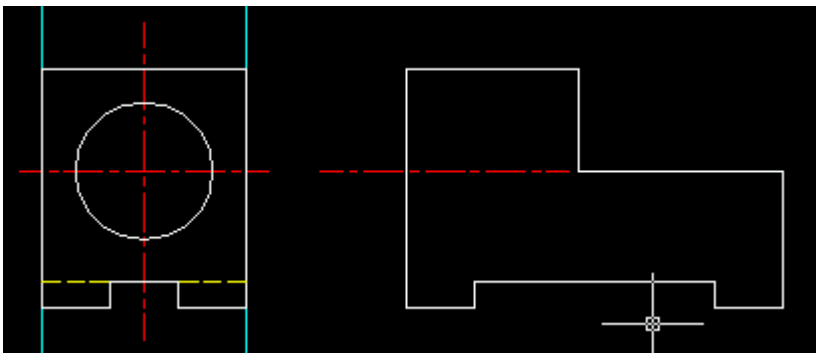
Press enter to finish selection and then select the base point of the object using Endpoint



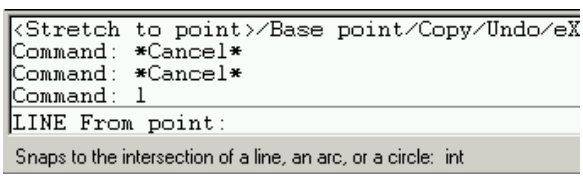


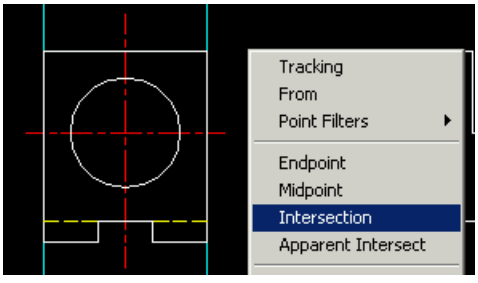




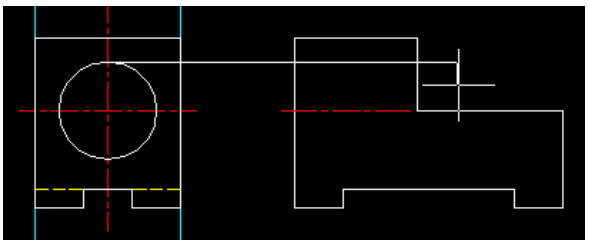


Line Command 

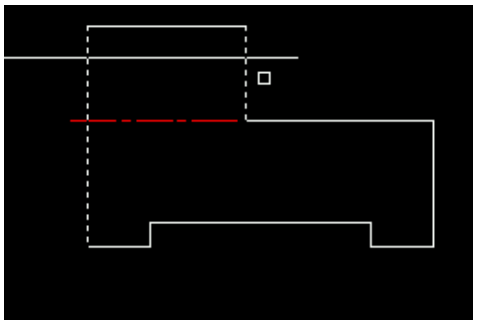




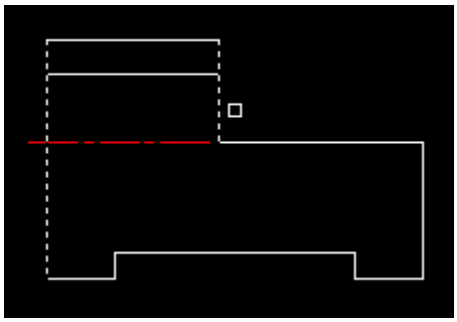
```
Command: *Cancel*
Command: l
LINE From point: _int of
To point:
To point:
267.1828,155.5020,0.0000
```



Trim Command 



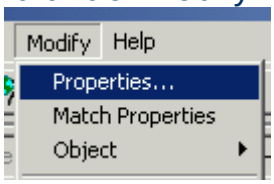
```
Select cutting edges: (Projmode = UCS, Edgemode = No extend)
Select objects: 1 found
Select objects: 1 found
Select objects:
<Select object to trim>/Project/Edge/Undo:
299.3031,96.5688,0.0000  SNAP GRID ORTHO OSNAP MODEL TILE
```



```
Select objects:
<Select object to trim>/Project/Edge/Undo:
<Select object to trim>/Project/Edge/Undo:
<Select object to trim>/Project/Edge/Undo:
<Select object to trim>/Project/Edge/Undo:
255.5026,154.3350,0.0000  SNAP GRID ORTI
```

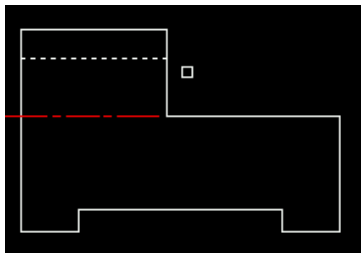
Changing Properties 

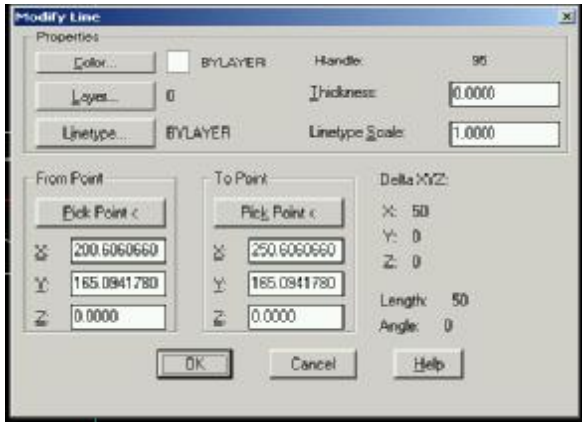
Or select Properties command under Modify Pull-Down menu

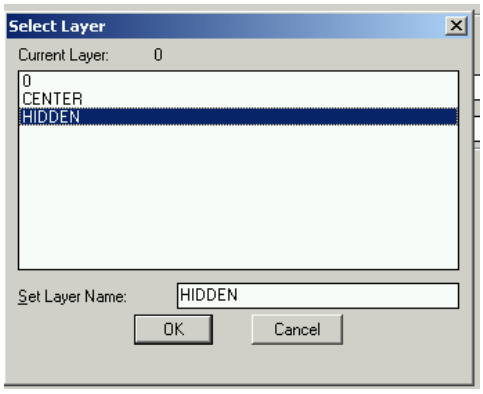


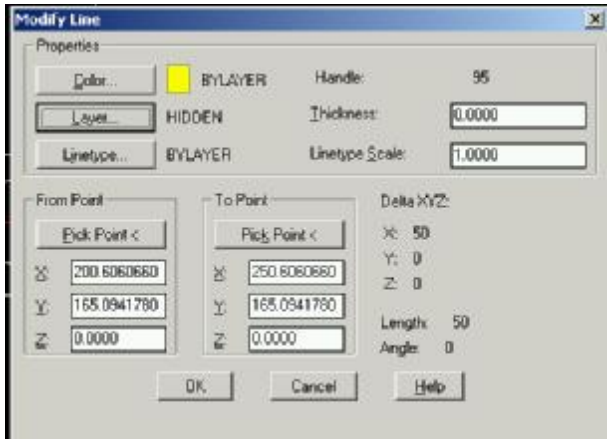

```
Command:
Command:
Command: ai_propchk
Select objects: Other corner: 1 found
Select objects:
257.8387,160.1699,0.0000  SNAP GRI
```

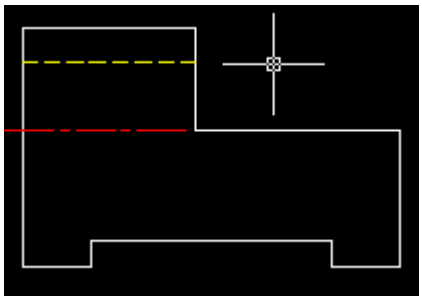
to finish selection press enter

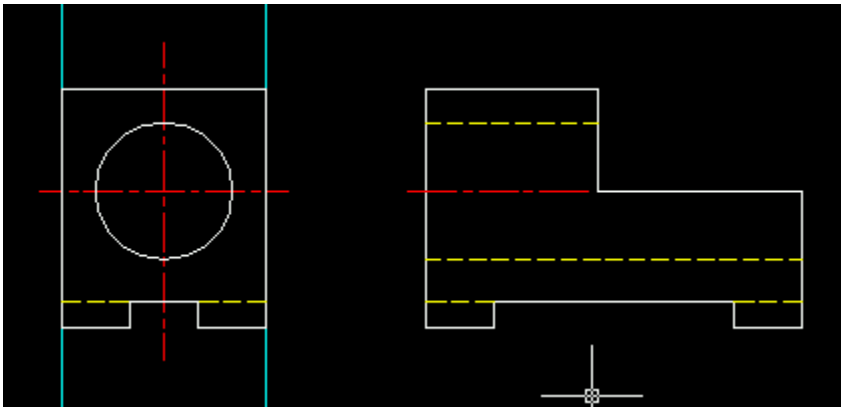










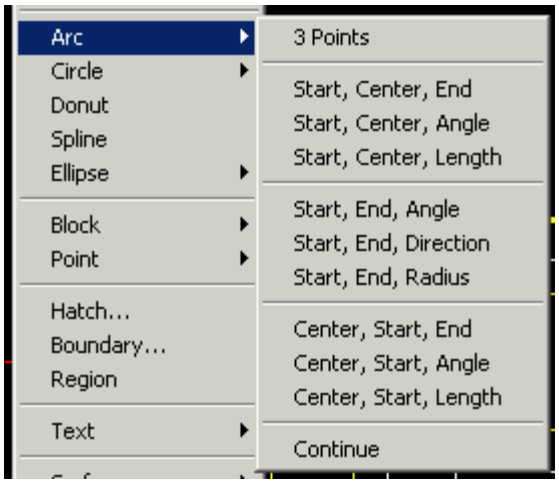


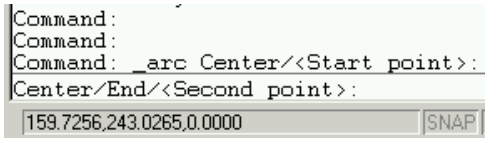
Use the following commands to generate the left side view.

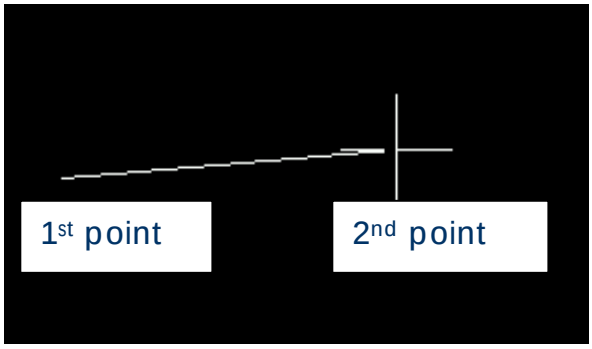
LINE OFFSET CHAMFER TRIM COPY PROPERTIES

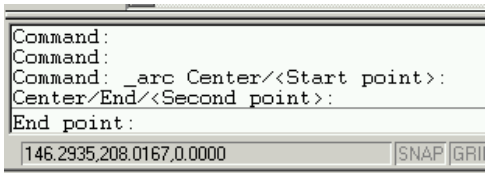
Arc command

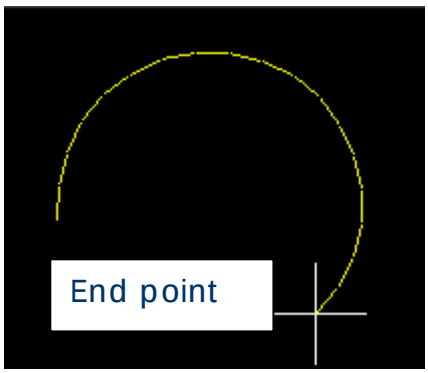
To draw arc (curve) you will use arc command or icon or use Draw pull-down menu and select arc. Don't forget that the positive direction is the counter-clock wise direction.





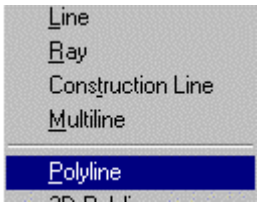






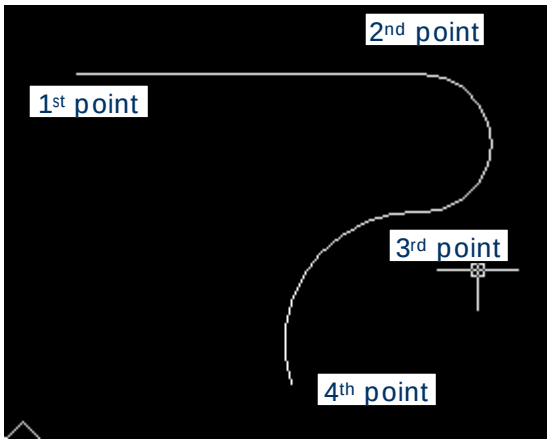
The Polyline Command

The Polyline PLINE command draws a series of connected lines and arcs that AutoCAD trasts as a single graphic object called a polyline. It is used on technical drawing top draw irregular curves and lines that have a width.



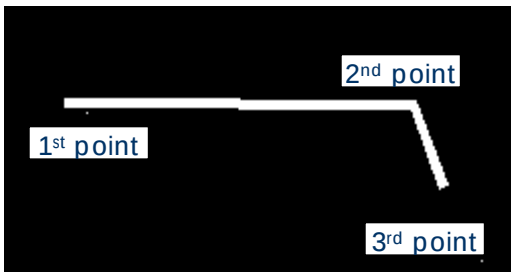
To draw line and then an arc type a for the arc and then select end point of an arc.

```
Command: _pline
From point:
Current line-width is 0.0000
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: a
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>:
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>:
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>:
Command:
```



It is possible to give width a drawing objetcs using polyline command. To do this on polyline command, type w for the width and then enter the width (i.e. 3).

```
PLINE
From point:
Current line-width is 0.0000
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: w
Starting width <0.0000>: 3
Ending width <3.0000>:
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Command: *Cancel*
Command:
```

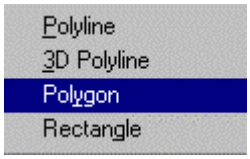


You may give different starting and ending width .

```
PLINE
From point:
Current line-width is 1.0000
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: w
Starting width <1.0000>: 5
Ending width <5.0000>: 1
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
Command:
```

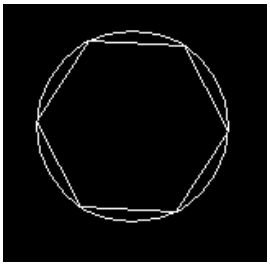
The Polygon Command

The Polygon  command draws regular polygon with 3 to 1024 slides. Polygons are either inscribed in or circumscribed about a circle.



Inscribed in circle option :
First of all, you have to enter number of sides (i.e. 6) after pointing the center of polygon, to draw polygon inscribed in circle type i and give the radius of inscribed circle.

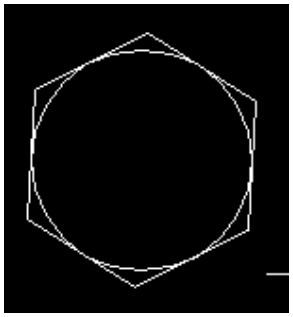
```
Command:
Command: _polygon Number of sides <6>: 6
Edge/<Center of polygon>: _cen of
Inscribed in circle/Circumscribed about circle (I/C) <C>: i
Radius of circle: _nea to
Command: *Cancel*
Command:
```



Circumscribed about a circle option :

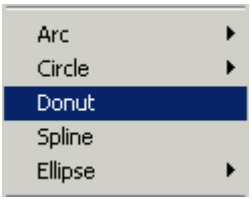
First of all, you have to enter number of sides (i.e. 6), after pointing the center of polygon, to draw polygon circumscribed about a circle type c and give the radius of circumscribed about a circle.

```
Command:
Command:
Command: _polygon Number of sides <6>: 6
Edge/<Center of polygon>: _cen of
Inscribed in circle/Circumscribed about circle (I/C) <I>: c
Radius of circle: _nea to
Command:
```



Donut Command

Donut command draws filled circles or cocentric filled circle




```
Command:
DONUT
Inside diameter <1.0000>: 5
Outside diameter <5.0000>: 10
Center of doughnut:
```