

**ENG-A2001 Computer-aided Tools in Engineering
AutoCAD Exercise Material**

AutoCAD Exercise Material

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1 Introduction

1.1 Prologue

This course follows such an arrangement, that the students can advance freely further ahead. There will be an assistant in the exercise sessions for those in need of help. If you are done with the exercise materials in the section 1 before the schedule, feel free to begin with the exercises.

Let us familiarize ourselves with the program before we begin the actual modelling. This material is for the AutoCAD 2020 program.

1.2 Properties of the User Interface and Fundamentals

Start AutoCAD through start menu in Windows. AutoCAD will start up with ribbon menus similar to those Windows uses. Choose acad.dwt as your template (Figure 1).

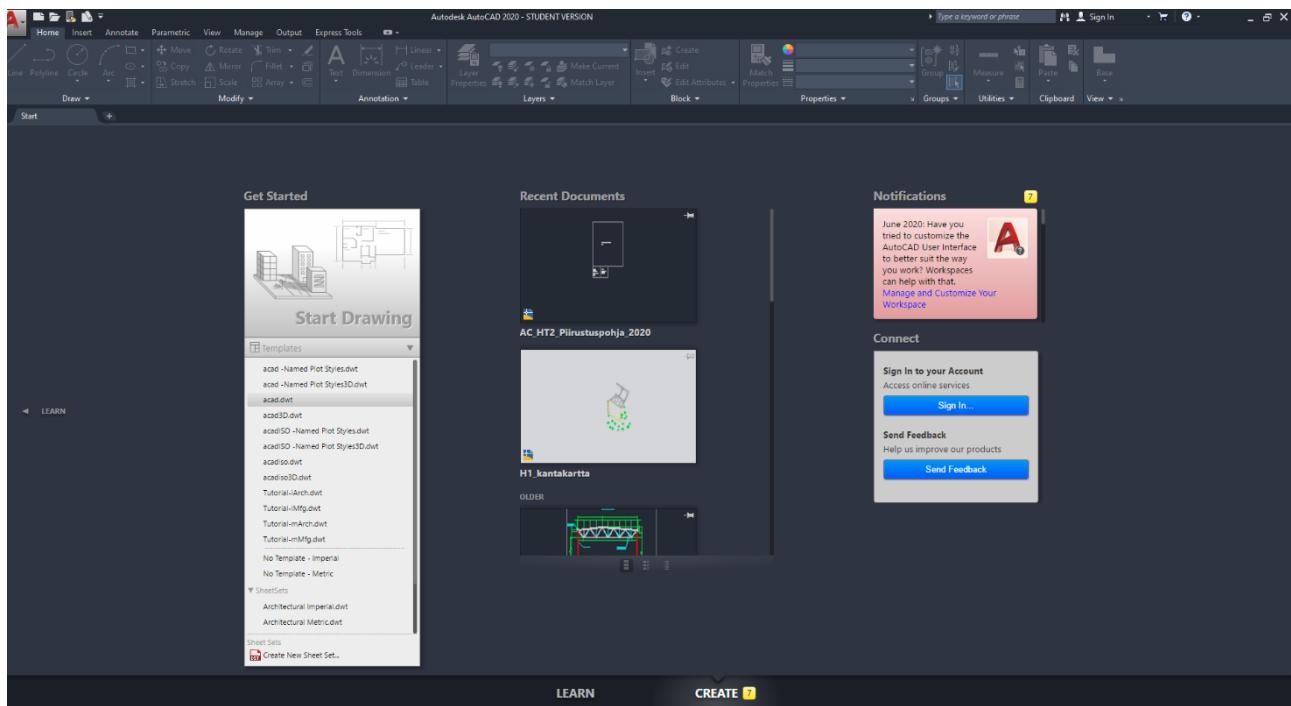


Figure 1. AutoCAD start up menu

Now the user interface should look similar to Figure 2. Some of the more important features are also shown in Figure 2. User interface can be modified by the user via menus, buttons, and commands. This can significantly increase the user efficiency of AutoCAD. There are two spaces in the drawing area: *Model space*, that is used for modelling, and *Paper Space*, that is used to create the printed views on a paper. Multiple viewports can be defined for a single model.

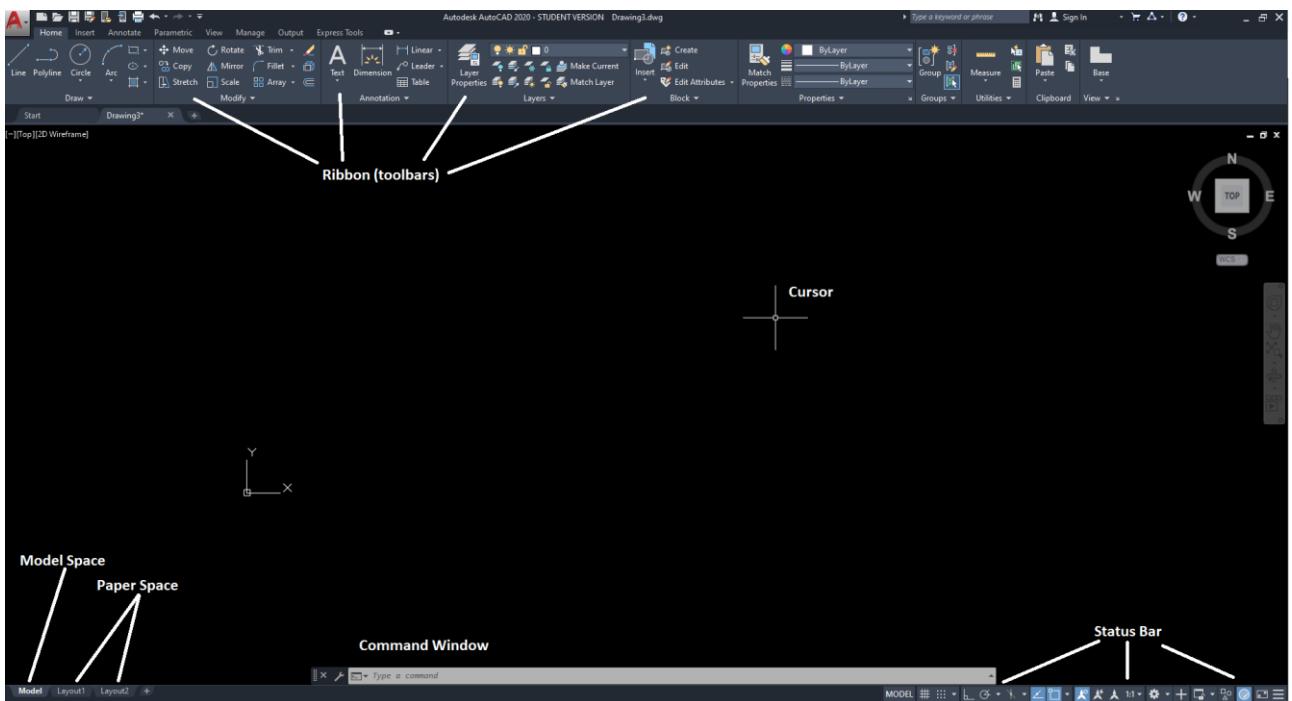


Figure 2. User interface

From toolbars you can find different tools and functions grouped by topic area. Toolbars can be modified by the user. This is done from *View – Toolbars* menu.

Command line is used to enter commands using the keyboard. Commands can be given either as whole or as abbreviations also known as *alias*. **The command line can also be used to see what AutoCAD is expecting to be the next action.** AutoCAD is expecting for user input if *Command:* prompt is visible. Arrow keys (\uparrow and \downarrow) can be used to browse through previous commands.

Status Bar can be used to turn on or off some utilities, such as Object snap and Ortho. With status bar, it is easy to switch between model spaces.

There are three ways to display the coordinates:

- *Static display* only shows the coordinate values when they are changing during the modelling
- *Dynamic display* updates coordinates real time when moving the cursor (keyboard hotkey is F12)
- *Distance and angle* shows length and angle real time. Only works with commands that require more than one coordinate point

1.3 Options

Command: **options, op** (AutoCAD start – Options (Figure 3))

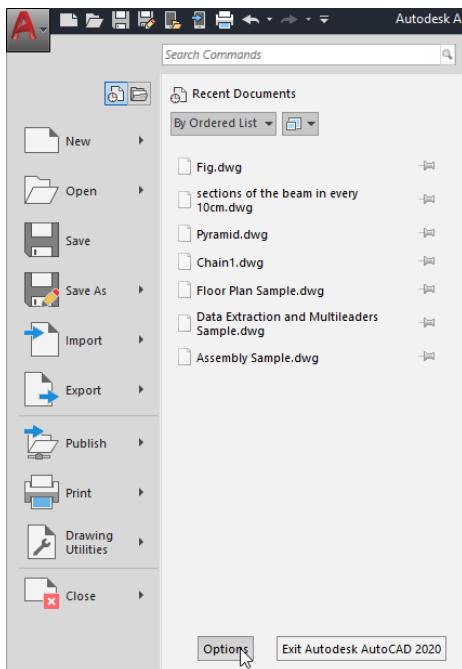


Figure 3. Options in start menu

Options command opens a dialog box (Figure 4). On that dialog boxes tabs, you can define file and search paths, display settings, open and save settings, and cursor settings. These settings can then be saved to a personal profile.

For example, let us change the color of model space to red by clicking *Colors* button in *Display* tab in *Options* menu. This will open Drawing Window Colors window, that has a dropdown menu called *Color* in the top right corner. Change the color from default black to red and apply and close to save the change. On the next chapter learn how to save and share these new personal modelling profiles. Please do note that the color of display does not have to be red later, this is all about personal preferences.

Changing the size of the crosshair is something else you might find useful. This is also done via the options menu's display tab. On the bottom right, there is a slider option for crosshair size. You can change this to 100 to make it so, that the crosshair touches borders of your drawing area.

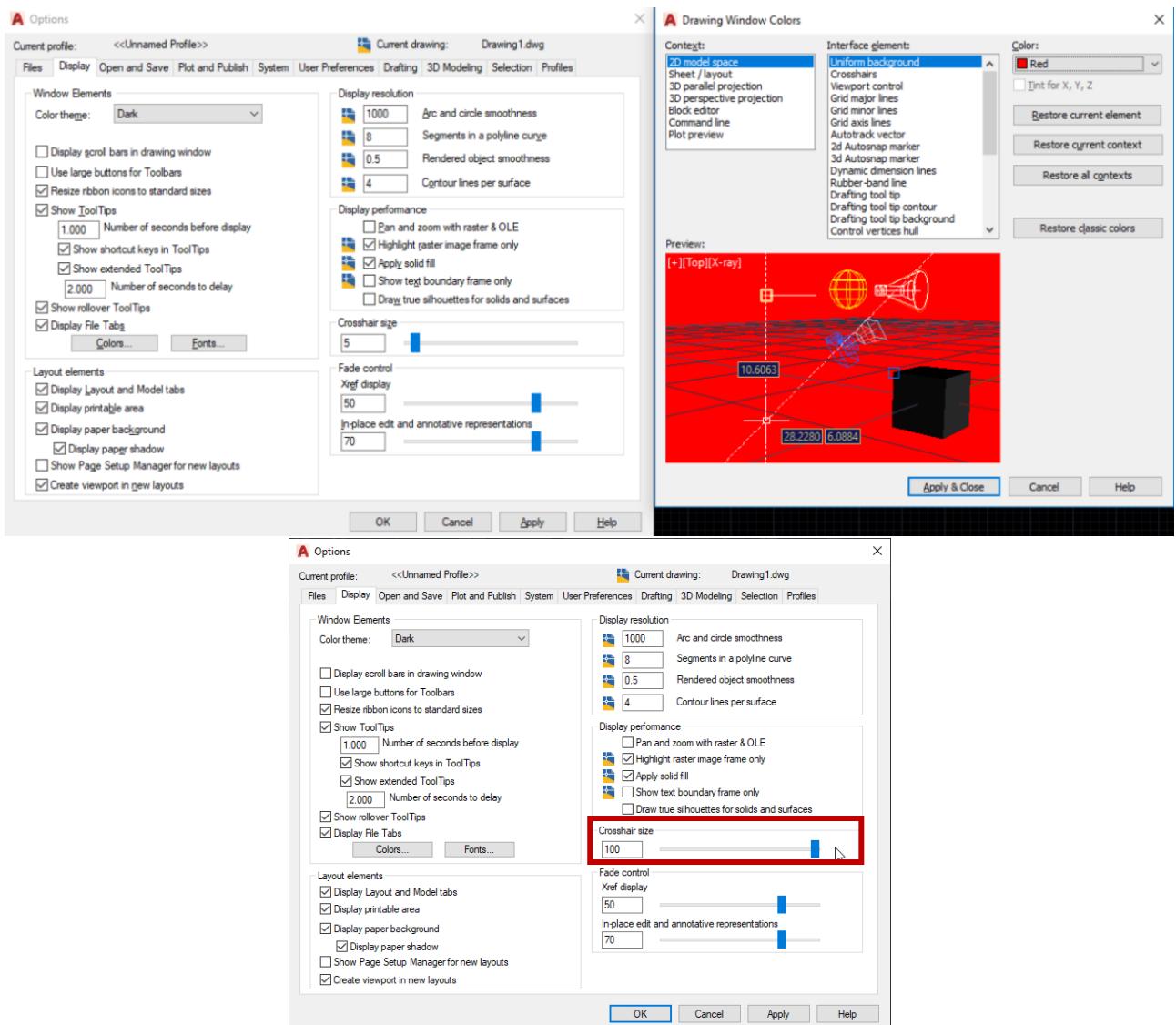


Figure 4. Option menu's Display tab and Drawing Window Colors view

Saving and importing a profile

Click Export button in the Profiles tab of Options window and select a file path to save the file. After this, download AC_Profile.arg file from the course's MyCourses page and save it on your computer so that you can find it. The file is then brought to AutoCAD by clicking the Import button in same Profiles tab. The file called AC_Profile should appear on the list and can now be set as a profile used by clicking the Set Current button. Click OK to save the changes.

Drawing units and precision

Command: **units, un** (AutoCAD start – Drawing utilities – Units (Figure 5))

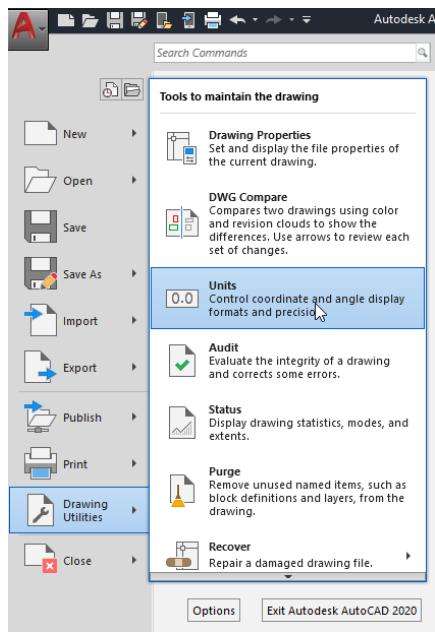


Figure 5. Units in start menu

Command **Units** opens Drawing Units menu, which is shown in Figure 6. Length menu is used to define the type of the measure of length and the number of decimals. Angle menu is used to define type and precision of angular units. Insertion Scale defines automatic scaling for content that is brought to the model between different units. Direction opens a Direction Control menu, that can be used to define the direction of a 0° angle. Clockwise (CW) option turns rotation direction of angular units from counter CW to CW. Therefore, the “positive” rotation direction is towards counterclockwise if not specified separately. Let us define the precision of units to 0 and angles to 0.

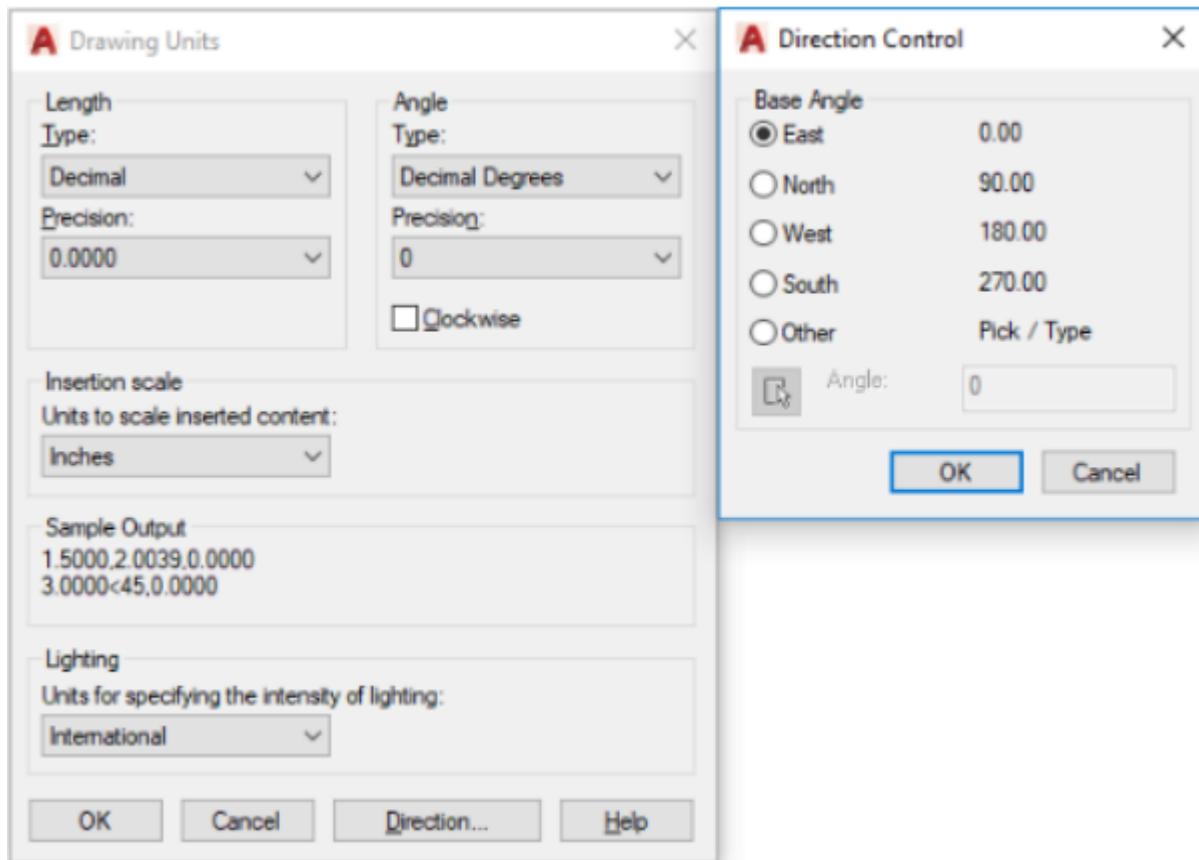


Figure 6. Units dialog box

1.4 Specifying the Drawing Area

Command: **limits**

Command **Limits** is used to define boundaries in the drawing area. AutoCAD will ask for two coordinate points, these being bottom left and top right corners. Grid is only drawn inside of the limits. Boundaries can also be changed during the modelling.

Command: **LIMITS**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000> : 0,0

Specify upper right corner <420.0000,270.0000>: 45000,55000

1.5 Executing and Undoing Commands

In AutoCAD commands can be issued with:

- Keyboard

- Dropdown Menus
- Tool Buttons

Command can be undone by command **Undo** or pressing **Ctrl+z**. Last undone action can be redone with command **Redo**. Pressing **Esc** interrupts the executing of ongoing commands. Commands can be accepted by pressing **Enter**, **Spacebar** or mouse's right click.

Command Abbreviations or Aliases

While using the command line, it is possible to use either commands name or abbreviated versions known as aliases. Aliases have been defined in acad.pgp file (found in the installation folder of AutoCAD), which is loaded on start up. This file can be changed during the session and can be reloaded with command **reinit**.

Hotkeys for Snap Settings

In AutoCAD, it is reasonable to try minimizing work done with the mouse, because use of keyboard is fast with your free hand. Especially, the use of snap settings can be improved with hotkeys that are shown in Table 1 below.

F1	Opens Help
F2	Expanded Command History
F3	Object Snap on/off
F4	3D Object Snap on/off (3D property)
F5	Isoplane on/off (3D property)
F6	Dynamic UCS* on/off
F7	Grid on/off
F8	Ortho on/off
F9	Snap on/off
F10	Polar Tracking on/off
F11	Object Snap Tracking on/off

Table 1. Function hotkeys

* User Coordinate System is used to align the XY plane of the UCS with a plane on a 3D solid.

1.6 Layers

Layers are important part of managing the structure of the model. Layers in a drawing are used to retrieve information and filtering, print management and graphic management. There are several company and industry specific recommendations and standards about the use of layers. Layers have properties like color, line type, line width and print style. They also have on/off switches that can be used to define visibility, freezing, locking, and plotting.

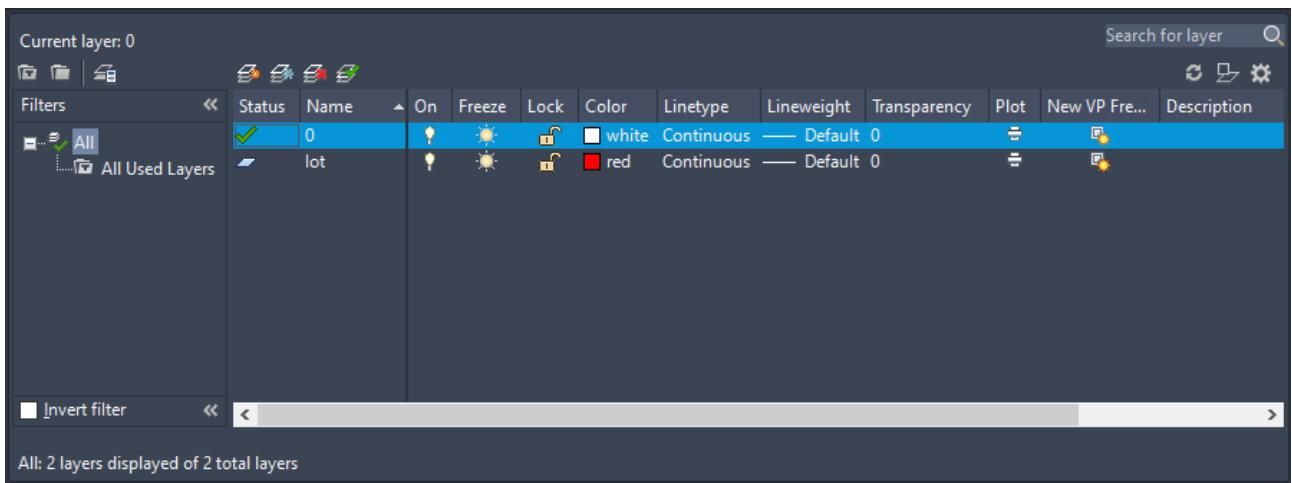


Figure 7. Layer menu

Using, editing, and deleting layers

Command: **layer, la** (*Ribbon menu – Home – Layer Properties*)

Command **layer** will open Layer Properties Manager menu to define and manage the layers.

New layer can be created either in Layer Properties Manager menu with **New** button or key combination **Alt+n**. Layer's name can be up to 255 characters long and can include spaces or Scandinavian characters, although they should be avoided.

Properties of a layer can be edited in Layer Properties Manager menu. The name of a layer can be changed by double clicking it. Other properties can be edited by clicking them once.

Deleting a layer is done in Layer Properties Manager menu by selecting layers to be deleted and clicking **Delete**. Layers "0" or "Defpoints" cannot be deleted. Nor can a layer be destroyed if it contains objects or is set current. References' layers must be deleted in actual reference drawing. Completed drawing can be purged from redundant or unused specifications with command **purge**. One of the options, that purge command has, is deleting unused layers.

Layer management

Layers can be sorted in ascending or descending order in the Layer Properties Manager dialog box based on the property of the column (Name, On, Freeze in all VP, lock and so on). Layers can be filtered by right clicking the Layer Properties Manager and selecting Layer Filters. *Invert Filters* inverts the selected filters.

1.7 Zooming and Panning

Command: **zoom, z** (*View – Zoom – ...*)

AutoCAD has several zoom commands, that can be used to observe the drawing from wanted distance. Drawing will be saved using the last view, since the zoom commands are saved with the drawing. Mouse wheel up and down is common and easy way to use zoom. However, often it is reasonable to a zoom command. Let us practice how that works next. Zoom options are shown in

Table 2. Should you want to zoom the drawing area to size of the limits before beginning to draw:

Command: z

Command: ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or
 [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real
 time>: a

Zoom options

Zoom Window



Command: **zoom** (option: Window)

Zooms to display an area specified by a rectangular window.
 With the cursor, you can define an area of the model to fill the entire window.

Zoom Previous



Command: **zoom** (option: Previous)

Zooms to display the previous view. You can restore up to 10 previous views.

Zoom Dynamic



Pans and zooms using a rectangular view box. The view box represents your view, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms to fill the viewport with the view inside the view box.

Zoom Scale



Zooms to change the magnification of a view using a scale factor.

- Enter a value followed by x to specify the scale relative to the current view.
- Enter a value followed by xp to specify the scale relative to paper space units.

Zoom in, Zoom out



Command: **zoom** (option as default: scale)

Zoom in and Zoom out are special cases of Zoom Scale. Zoom in uses scale factor of 2x and Zoom out uses the scale factor of 0.5x.

Zoom Center
 Command: zoom (option: Center)
Zooms to display a view defined by a center point and a magnification value or a height. A smaller value for the height increases the magnification. A larger value decreases the magnification. Not available in perspective projection.
Zoom All
 Command: zoom (option: All)
Adjusts the magnification of the drawing area to accommodate the extents of all visible objects in the drawing, or visual aids such as the grid limits (the LIMITS command), whichever is larger.
Zoom Extents
 Command: zoom (option: Extents)
The extents of each object in the model are calculated and used to determine how the model should fill the window.

Table 2. Zoom options

Command: **pan** (*View – Pan*)

Panning means movement of the view. Panning can be done by pressing down the mouse wheel. Pan command makes moving the view in the drawing area possible in real time. Two coordinate points can also be given for pan command to move view accurately. There is a useful function tied to double clicking mouse wheel, it is used as a hotkey to Zoom Extents. This means that by doing it you can zoom the drawing so, that it fills up your whole screen.

1.8 Coordinate Systems

AutoCAD uses both rectangular and polar coordinates. Coordinates can be input either absolute or relative.

Rectangular Coordinates

Rectangular coordinates are used when dimensions are given in the direction of the coordinate axes. Positive x-axis grows towards right and y-axis grows upwards.

Polar Coordinates

Polar coordinates are used when dimensions are given with radial distance and polar angle. Positive growing direction of a polar angle is counterclockwise.

Absolute Coordinates

Absolute coordinates refer always to the origin of a drawing (0,0,0). Absolute coordinates are given in terms of #X,Y,(Z). Decimals are separated with a dot [.]. For an example you could draw a lot using absolute corner points:

Command: LINE

Specify first point: #65234,43033

Specify next point or [Undo]: #92452,40987

Specify next point or [Undo]: #95213,20089

Specify next point or [Undo]: #64178,18945

Specify next point or [Undo]: c

Relative Coordinates

The given coordinate in relative coordinates refers to a point given before that. Relative coordinates can be used in both rectangular and polar coordinates. Relative coordinates in rectangular coordinates are given in following shape @dx,dy,(,dz), where dx, dy, and dz are transitions in corresponding axis. Polar coordinates are given in following form @radius<angle. @ character is rarely necessary, because AutoCAD assumes that relative coordinates are used when dynamic input is selected.

Command: LINE

Specify first point: 0,0

Specify next point or [Undo]: 600,0

Specify next point or [Undo]: 600<-120

Specify next point or [Undo]:

Giving coordinates

Coordinates can be input by:

- Pointing with a mouse
- Keyboard
- Using both mouse and keyboard
- Reading coordinates from an external point file and inputting objects with ready functions

Inputting points with mouse

Using mouse to input coordinate points is simply done by moving cursor on a point and clicking left mouse. Snap and Object snap settings can be used to make drawing by mouse accurate.

Inputting Points with Keyboard

When inputting coordinate points with keyboard, coordinates are given either in absolute or relative. Absolute form for coordinates is #X,Y,(Z). Point can also be expressed with transitions referring the last point using the format dx,dy,(dz) in rectangular coordinates and format length<angle in polar coordinates.

Inputting Points with Both Mouse and Keyboard

When **ORTHO** is activated, coordinates can be input quickly and accurately by showing the direction with mouse and length relative to the last point with keyboard. The next point can then be found from given length in a direction that the cursor showed. ORTHOMODE only allows points in directions of coordinates axes.

1.9 Editing the Workspace

The user interface can be edited to one's preference and then be saved to profile files the way you learned back in chapter 1.3. Every experienced AutoCAD user has their own preferences and many user interfaces are different from default view.

Command line might float in lower half of the drawing area in default view of the AutoCAD. You can attach it on the status bar by dragging it from the left margin. Command line can also be completely hidden with a command **commandlinehide**. To show it again use command **commandline**. You can also try key combination **Ctrl+9** to hide and show the command line.

It might help editing the objects later if you keep properties window open, use command **ch** to show. This can also be attached by dragging it to a preferred place from the window's top margin.

2 Exercise 1: Site Plan

2.1 Basic Settings

Before beginning with modelling, make sure that you have done these basic settings. They are portrayed more accurately in chapter 1. How to check that you have activated the *Dynamic Input*: click on the customization in bottom right corner and select *Dynamic Input*. After this make sure that the newly appeared Dynamic Input is selected (icon is blue).



Figure 8. Dynamic input selected

Units

Check that you are using millimeters as your distance unit. This is done with the command **UN (UNITS)**.

Command: UN

Change *Insertion Scale* to millimeters (if it is not yet) in window that just opened (Figure 6). Next open options with command **OP (options)** and make sure that you have Insertion Scale set as millimeters in *User Preferences* tab also.

Command: OP

Limits

Limits command allows you to specify boundaries of the drawing area. AutoCAD will ask for the coordinates of the lower left and top right corners. Grid will only be drawn inside of these limits. You can change the limits during the modelling. Please do note, that AutoCAD may suggest the last used coordinates, they will be shown in angle brackets. Contents inside of these angle brackets do not have any effects, except that you can select those as your next command by pressing enter. This can make drawing faster.

Command: LIMITS

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000> : 0,0

Specify upper right corner <420.0000,270.0000>: 45000,55000

2.2 Lot

Drawing of the site plan will begin by drawing the lot. Create layer “Lot” like the chapter 1.6 taught. Set the created layer as current (green icon in *Status*). Now we are ready to begin with the actual drawing.

Polyline



Command: **pline, pl** (*Draw – Polyline*)

Creates a 2D polyline, a single object that is composed of line and arc segments. The following prompts are displayed:

- Arc – **a**: creating arc segments
- Close – **c**: Connects the first and last segments to create a closed polyline.
- Halfwidth – **h**: Specifies the width from the center of a wide segment to an edge.
- Length – **l**: Creates a segment of a specified length at the same angle as the previous segment.
- Undo – **u**: Removes the most recently added segment.
- Width – **w**: Specifies the width of the next segment.

Polyline object can be edited like any other objects or with command **pedit** that is purely meant for editing polylines. Pedit can be used to join multiple polyline objects as one or turn line and arc objects into polylines.

Command: PL

PLINE

Specify start point: 10000,50000

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 35000<-30
(polar)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
10000<-120 (polar)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0,-
20000 (rectangular)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
35000<150 (polar)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: <Ortho
on> 25000 (ortho on (F8) and point up from last segment with mouse)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c
(closes the first and last segments)

Command: <Ortho off> (F8)

The lot should now look similar to Figure 9.

Remember to save your drawing once in a while by pressing **ctrl+s or Start → Save!**

Ortho can be turned on and off pressing F8!

When drawing the lot, limits must be set large enough. (Chapter 2.1)

You can get everything, that you have drawn, visible with Zoom extents command. You can execute zoom extents by double clicking mouse wheel.

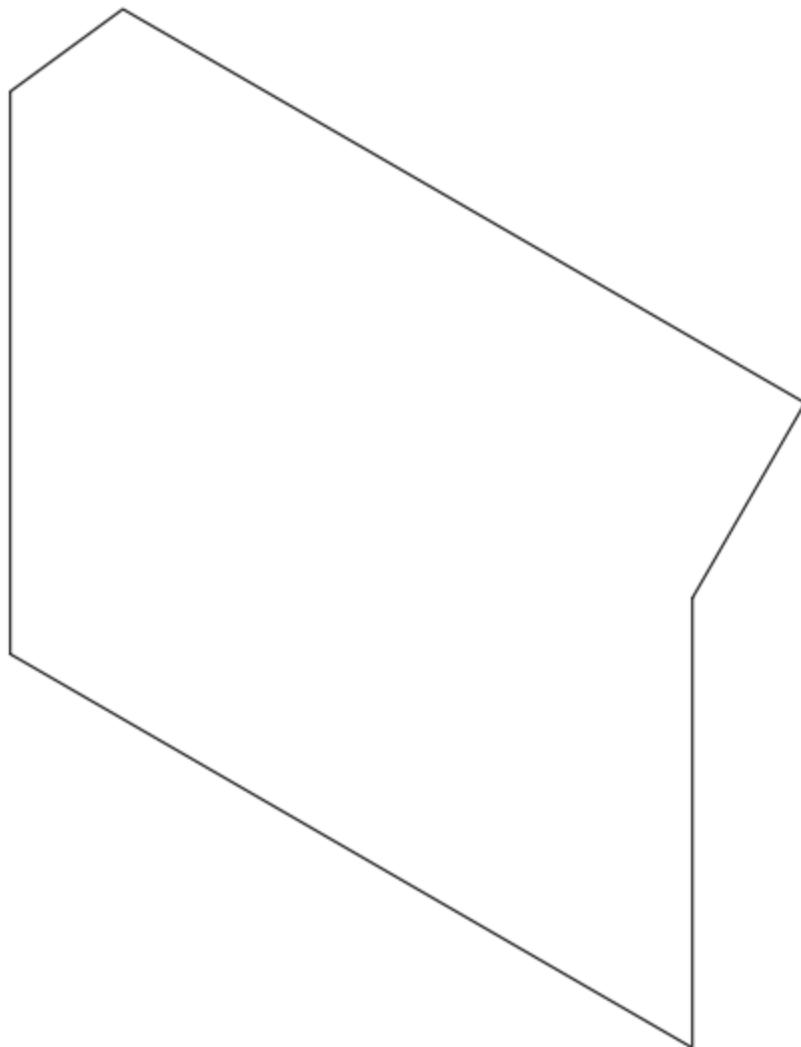


Figure 9. Shape of the lot

Snap settings

The use of snap settings is necessary if drawing of a new object is begun from an already existing object. It is reasonable that user can point accurately the wanted point with a mouse. Tools, that forces the selection to hit the end of a line for example, are needed. These tools can be administered with command **OS (OSNAP)**. Alternatively, you can access snap settings with **Shift + right click** and selecting *Osnap Settings*. Snap settings are then done in *Object Snap* tab in *Drafting Settings* dialog box. Continuous object snap can be turned on with function key F3 or OSNAP icon in status bar. There are multiple object snap options. You should familiarize yourself with them and remember as a basic rule, only keep the necessary ones activated at any time. The table below will introduce the snap settings.

	Endpoint, end Snaps to the closest endpoint or corner of a geometric object.
	Midpoint, mid Snaps to the midpoint of a geometric object.
	Intersection, int Snaps to the intersection of geometric objects. Intersections and extended intersections do not work with edges or corners of 3D solids.
	Apparent Intersection Snaps to the visual intersection of two objects that do not intersect in 3D space but may appear to intersect in the current view.
	Center, cen Snaps to the center of an arc, circle, ellipse, or elliptical arc. Selection point is objects circumference.
	Quadrant, qua Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc. In World Coordinate System (WCS), quadrants are in 0, 90, 180, 270 degrees from middle point.
	Perpendicular, per Snaps to a point perpendicular to the selected geometric object.
	Parallel, par Constrains a new line segment, polyline segment, ray or xline to be parallel to an existing linear object that you identify by hovering your cursor.
	Tangent, tan Snaps to the tangent of an arc, circle, ellipse, elliptical arc, polyline arc, or spline.
	Node, nod Snaps to a point object, dimension definition point, or dimension text origin.

	Insertion, ins Snaps to the insertion point of objects such as an attribute, a block, or text.
	Nearest, nea Snaps to the nearest point on an object such as an arc, circle, ellipse, elliptical arc, line, point, polyline, ray, spline, or xline
	None, non None revokes all selected snaps, meaning that none are selected. Points can be selected freely.
	Extension, ext Causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension.
	Temporary Tracking Point, tt Is used to reference existing geometry by a temporary origin, with the help of a reference point. Can only be used for horizontal and vertical transitions.
	Snap From, fro References to a temporary origin are done with relative coordinates.

Table 3. Snap settings

Measuring Tools

In AutoCAD, it is possible to measure drawn objects with tool **DI (DIST)**. While advancing with the drawing, it is wise to make sure occasionally that the dimensions are correct. Let us measure the top border of the lot.

Command: DI

DIST

Specify first point: (point with mouse)

Specify second point or [Multiple points]: (point with mouse)

Distance = 35000.0000, Angle in XY Plane = 330, Angle from XY Plane = 0

Delta X = 30310.8891, Delta Y = -17500.0000, Delta Z = 0.0000

This means that the distance is 35000 mm and it is at an angle 330° to the horizontal. AutoCAD also shows the measured dimensions in the direction of the coordinate axes: in the direction of

the X axis the distance is 30310.8891 mm and in the direction of Y axis the distance is -17500 mm. Figure 10 shows the first point in green and the second one in red.

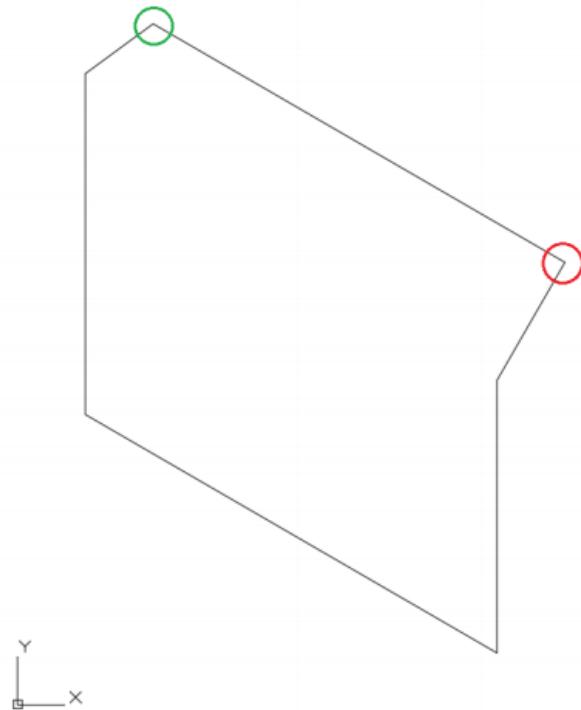


Figure 10. Using the measuring tool

2.3 Roads

Lets then define a new layer “roads”. Change color to gray and keep the other settings as default. Set the “roads” layer as current with the Set Current icon. Before drawing the roads, enable the *Object Snap* setting. Object snap can be enabled for a single grab or continuosly.

To start drawing the roads, use command **RAY** to draw a guideline (Figure 11).

Command: RAY (Draw - ray)

RAY Specify start point: (from the drawing with help of endpoint snap)

Specify through point: (from the drawing with help of endpoint snap)

Specify through point: <enter>

Infinitely long guidelines can be drawn with the ray command. The lines begin from the specified point and continues towards the direction given infinitely. Ray objects do not affect the zoom in any way, in example the Zoom Extents command is determined by other geometries. Ray objects will convert to line objects if they cut with a tool, such as **Trim**.

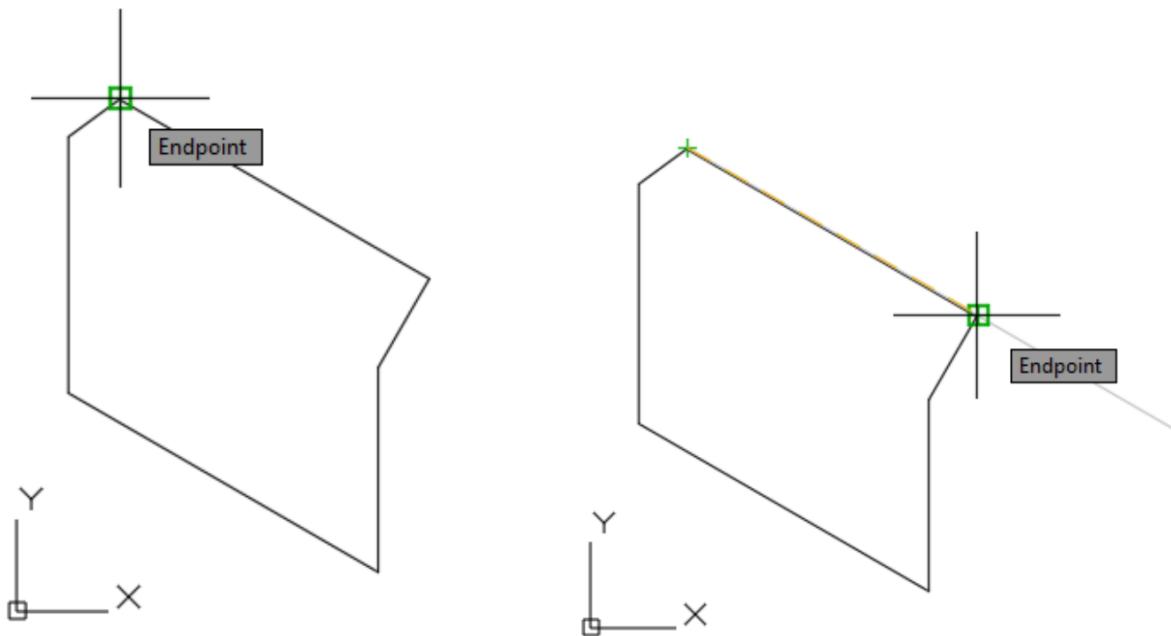


Figure 11. Drawing a rayline using endpoint snap

Road to the garage and position of the house is done with command **O (OFFSET)** from the guideline.

Offset



Command: **offset, o (Modify – Offset)**

You can offset an object at a specified distance or through a point. After you offset objects, you can trim and extend them as an efficient method to create drawings containing many parallel lines and curves.

The offset distance can be given with mouse or numerical, on top of this you must specify the side that copy will be made.

Command: O

OFFSET

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

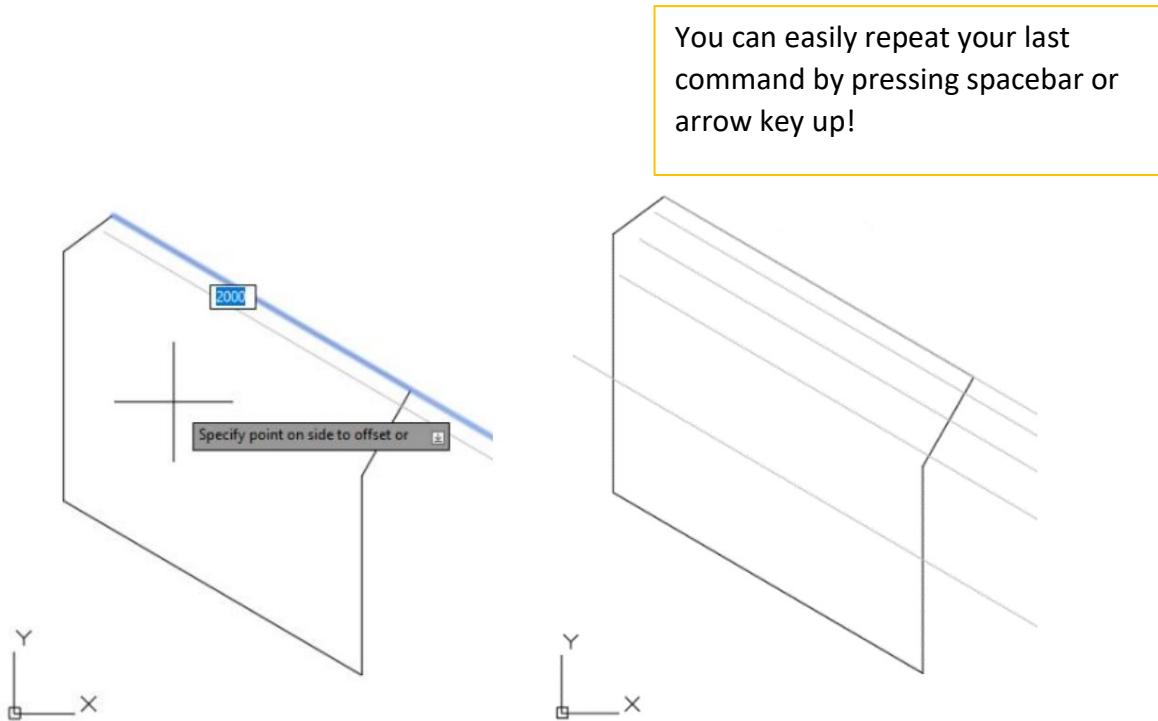
Specify offset distance or [Through/Erase/Layer] <Through>: 2000

Select object to offset or [Exit/Undo] <Exit>: (Ray line that was just drawn)

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: (Figure 12)

Select object to offset or [Exit/Undo] <Exit>:

Offset will be done three more times with distances 3000, 4000, 9000 **using last drawn line as a reference**. Figure 12 will clarify.



You can easily repeat your last command by pressing spacebar or arrow key up!

Figure 12. Drawing roads with the Offset

The roads that are the other way around will also be drawn with Ray and Offset commands. Offset distances are: 1500, 1000, 1000, 15000, 1000, 1000, 1500, 10000 and 1500 (Figure 13).

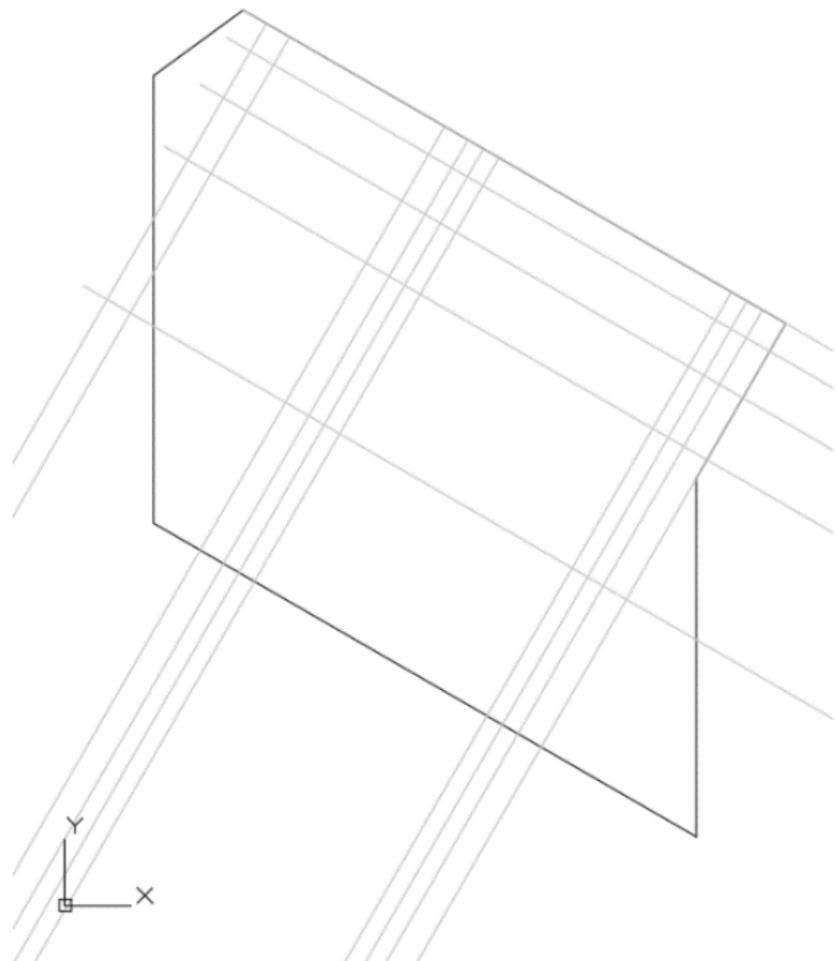


Figure 13. Drawing roads in the other direction

Check the distances of the roads, that you just drew, with measurement tool **DI (DIST)**.

Remember to save your drawing every 10 – 15 minutes (ctrl+s)!

Now that the lines are offset, use **E (ERASE)** command to erase lines that are overlapping the edges of the lot (Figure 14). There are two easy ways to do this, either the mentioned command **Erase** or clicking on top a line you want to delete, and press **Delete** on your keyboard.

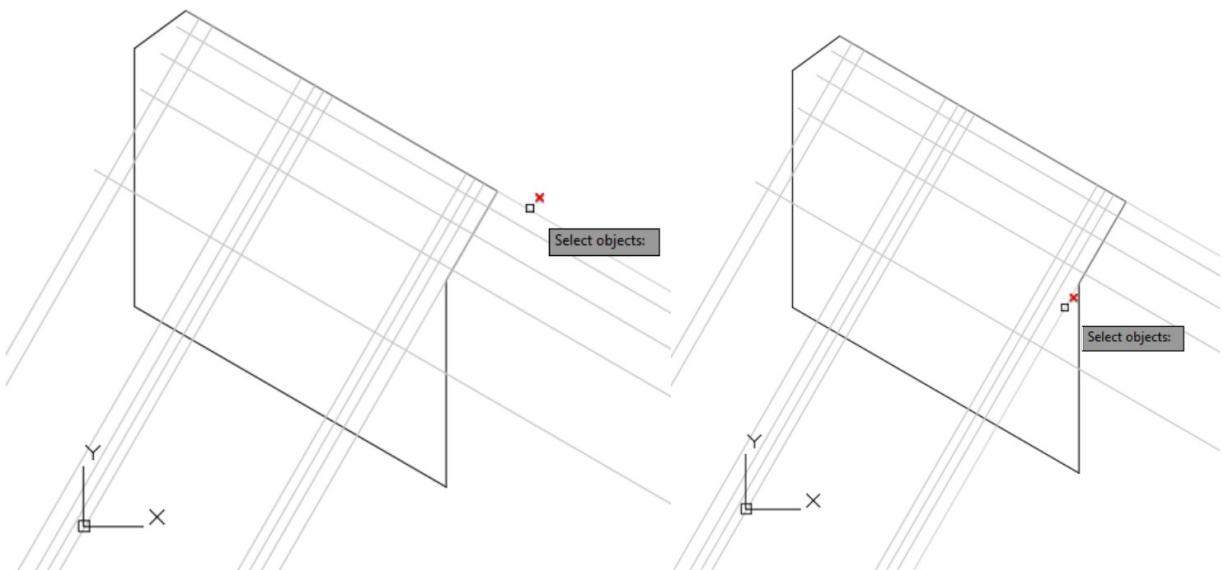


Figure 14. Deleting the overlapping lines

Trim



Command: **trim**, **tr** (*Modify – Trim*)

Trims objects to meet the edges of other objects. To trim objects, select the boundaries and press Enter. Then select the objects that you want to trim. To use all objects as boundaries, press Enter at the first Select Objects prompt.

Command: TR

TRIM

Current settings: Projection=UCS, Edge=None

Select cutting edges ... (top left in Figure 15)

Select objects or Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: (select line to be trimmed by clicking on the part to be cut off, top right in Figure 15)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: (repeat)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: (etc.)

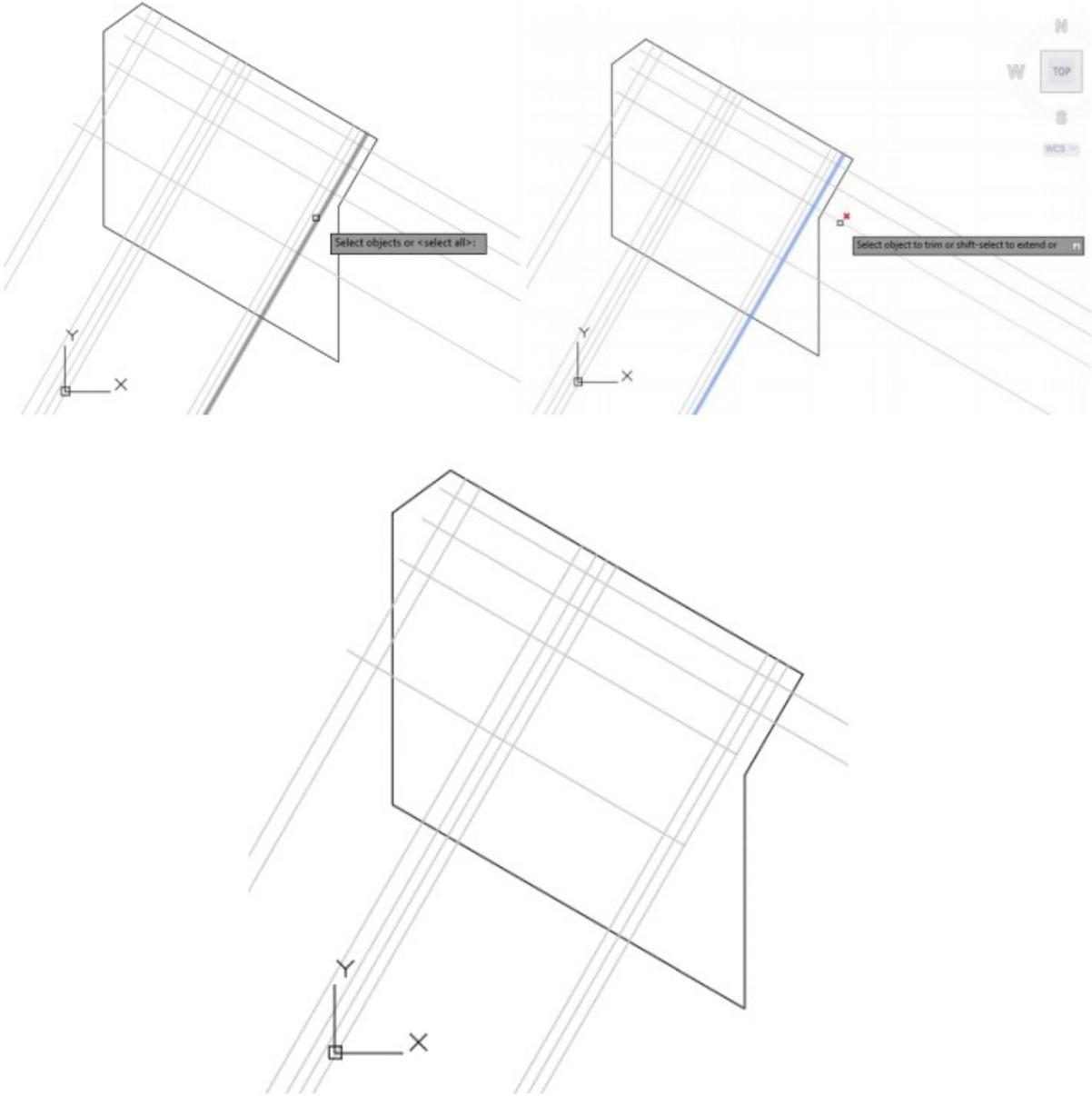


Figure 15. Tidying up the roads with the Trim

2.4 Buildings

Let us begin drawing the houses. First create layer “buildings”, change the color to **blue**, and leave everything else default.

Draw the outlines of the larger house first (blue rectangular in Figure 16). Draw the house using Polyline command and using the intersection snap setting. You can activate it from the Drafting Settings (command **OS, OSNAP**). Now trim some extra lines the way that Figure 16 shows.

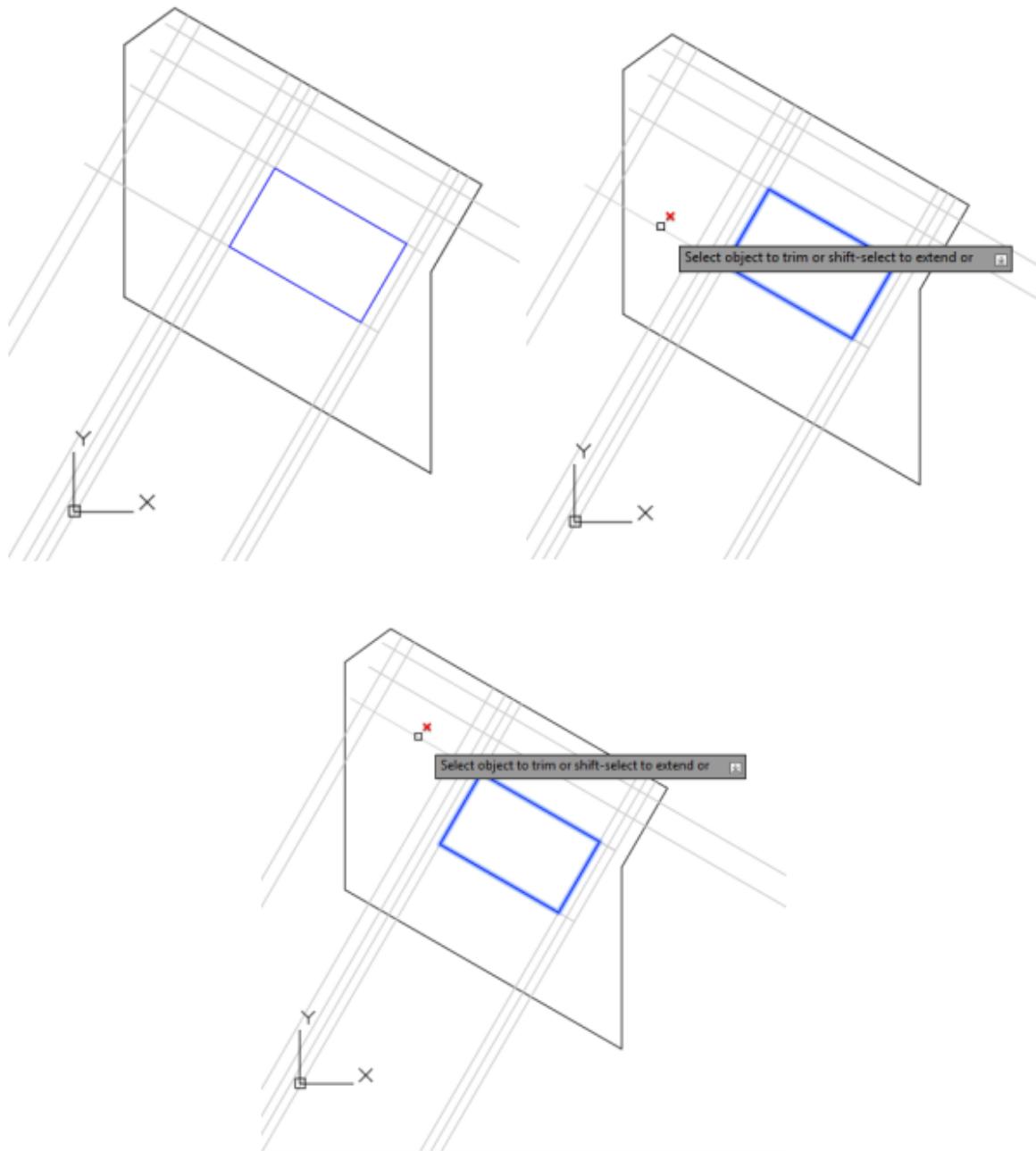


Figure 16. Outlines of the houses and their uses in trimming

We need one guideline more with offset distance of 5000 units for the smaller building (Figure 17, left)

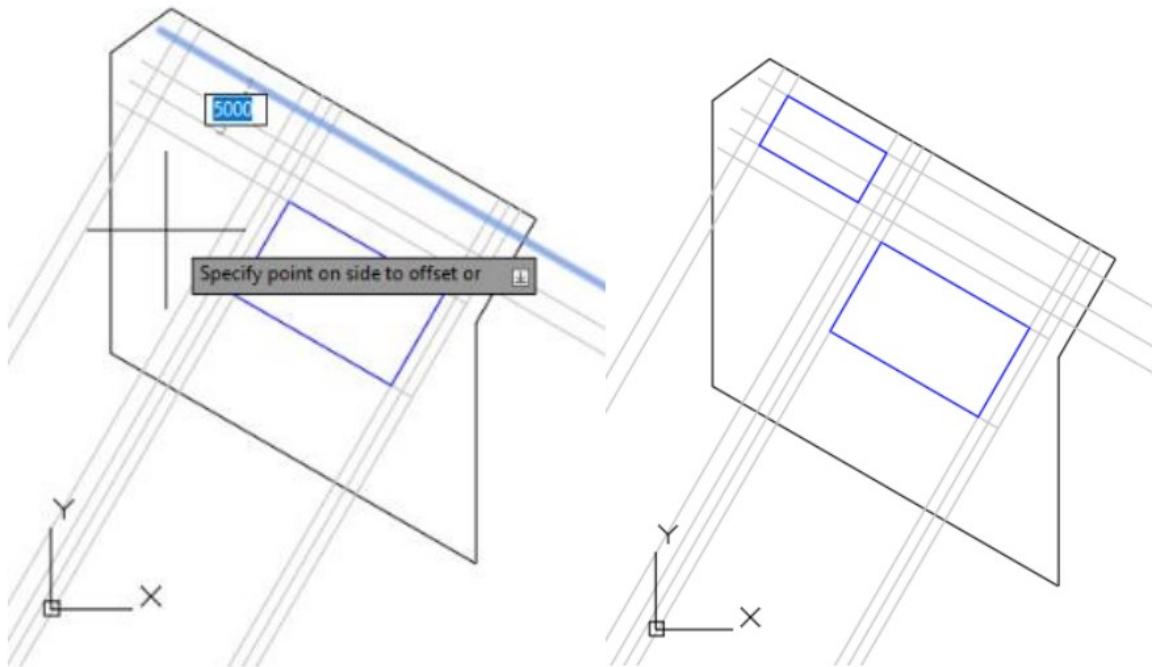


Figure 17. Drawing guideline and smaller building

The smaller building will be drawn the same way as the larger house, with polyline and intersection on (Figure 17, right)

Now you can erase five spare lines with the Erase, and after that your drawing should look like the model shown in Figure 18. Drawing after removing the spare lines. Figure 18.

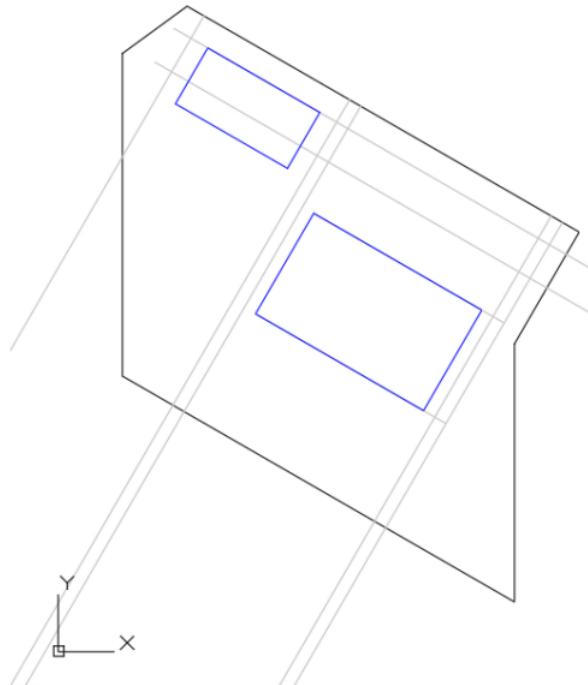


Figure 18. Drawing after removing the spare lines.

2.5 Courtyards and Lawns

The corridor at the right end of the larger building (width 1500, initial situation on the left) is modified similar to Figure 19 on the right with offset and trim commands. **The dimensions on the figure are to clarify and are not supposed to be drawn.**

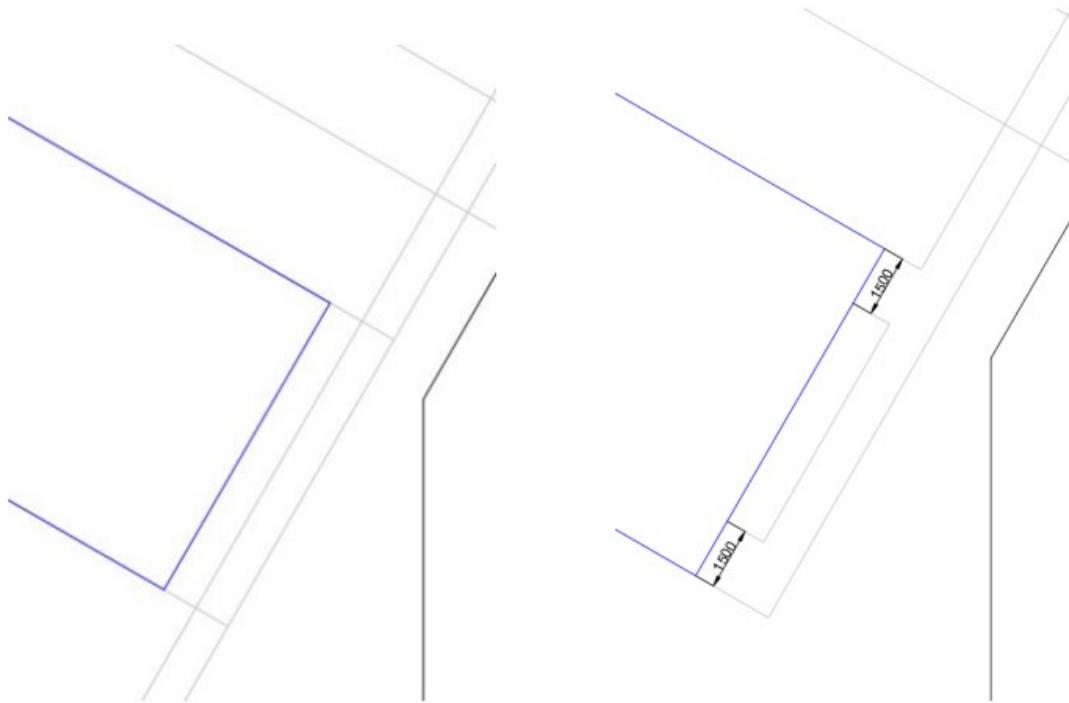


Figure 19. Drawing the courtyard, initial situation and final result

Next, we move on to editing the other side of the corridor. At the both ends of the corridor, there are chamfers with a size of 500x500. Use **BR (BREAK)** command to cut the line into two pieces.

Break



Command: **break, br (Modify – Break)**

You can create a gap between two specified points on an object, breaking it into two objects. If the points are off of an object, they are automatically projected on to the object. BREAK is often used to create space for a block or text. Use f (first) option to override the original first point where you selected the object with a new point that you specify.

Command: BR

BREAK

Select object: (top left)

Specify second break point or [First point]: (int snap, top right)

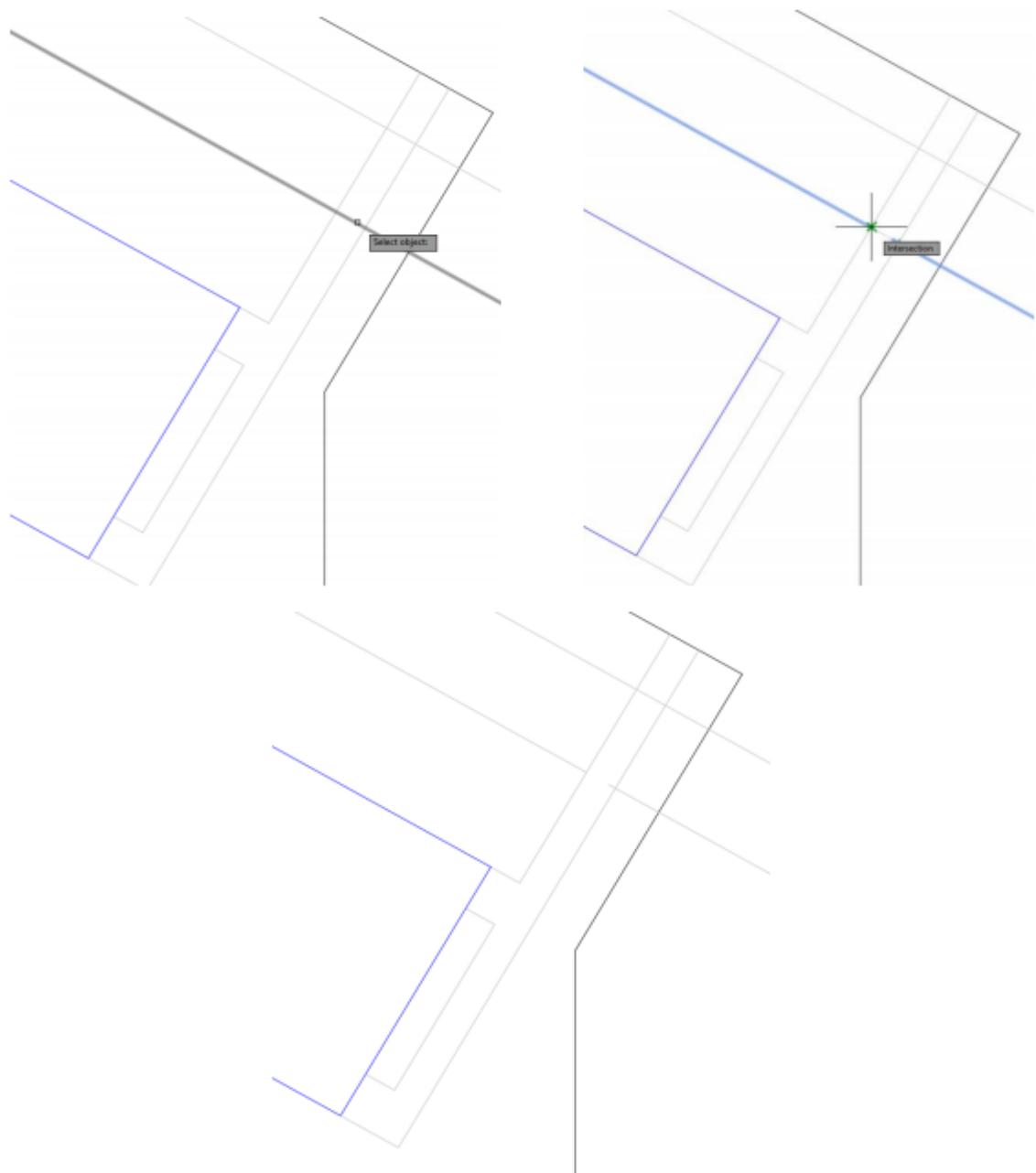


Figure 20. Using the Break command

Chamfers are drawn with the command **CHA (CHAMFER)**. If the trim option is used on chamfer, it will trim the extra objects.

Chamfer command is used to create the chamfers (Figure 21).

Chamfer

Command: **chamfer, cha** (*Modify – Chamfer*)

Chamfer can be used to chamfer non-parallel lines, polylines, and guidelines corners. The distances and angles that you specify are applied in the order that you select the objects. The distances and angles that you specify are applied in the order that you select the objects.

Command: CHA

CHAMFER (TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000

Select first line or

[Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: d (specify chamfer's distances)

Specify first chamfer distance : 500

Specify second chamfer distance : (accept the suggested 500 by pressing enter)

Select first line or

[Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: (top right Figure 21)

Select second line or shift-select to apply corner or

[Distance/Angle/Method]:

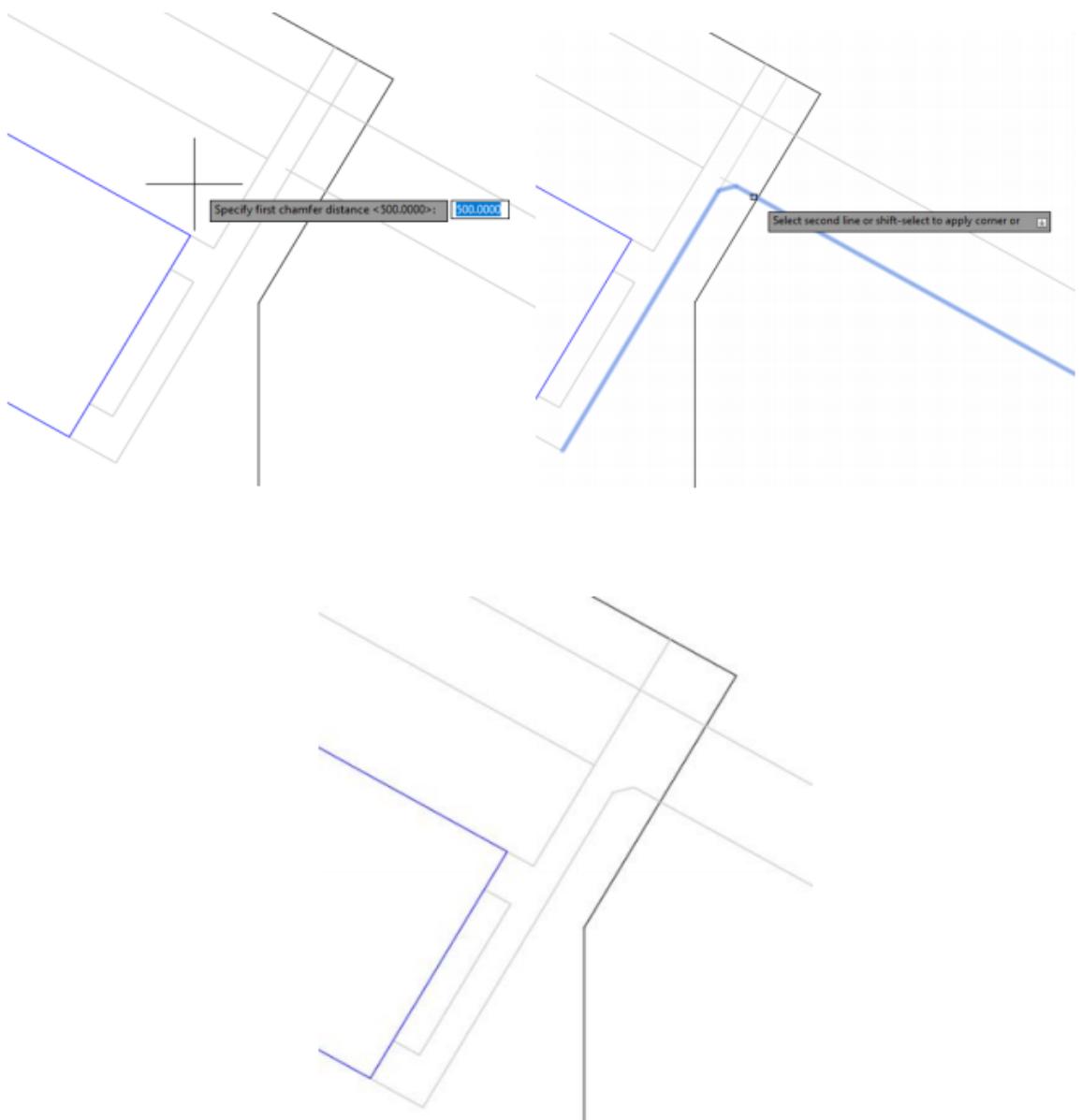


Figure 21. Chamfering

Chamfers with same dimensions are also drawn in the corridors shown in Figure 22. Remember that you can repeat the last command with **Enter** or **Spacebar**.

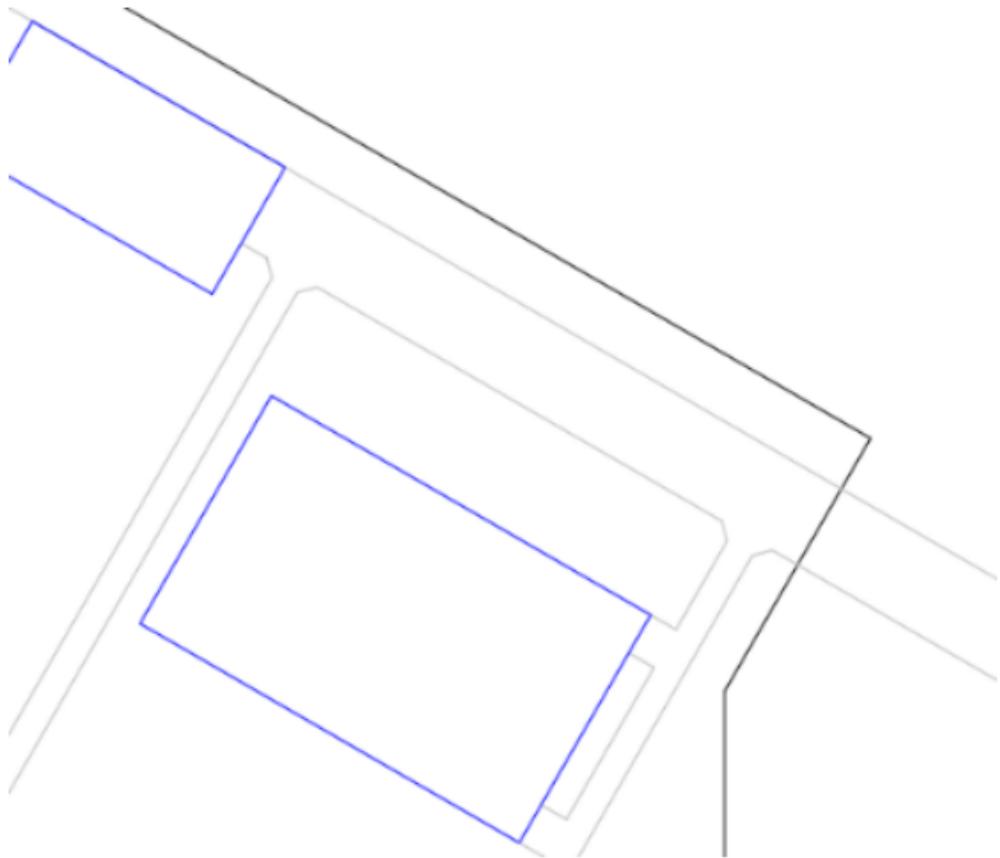


Figure 22. Finished chamfers

Curved corridor in the backyard is drawn with the Polyline command (Figure 23).

Command: PL

PLINE

Specify start point: (end snap from the corner of larger building)

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: a

Specify endpoint of arc (hold Ctrl to switch direction) or
[Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
a

Specify included angle: -90

Specify endpoint of arc (hold Ctrl to switch direction) or [CEnter/Radius]:
(use mouse to specify with int snap, Figure 23)

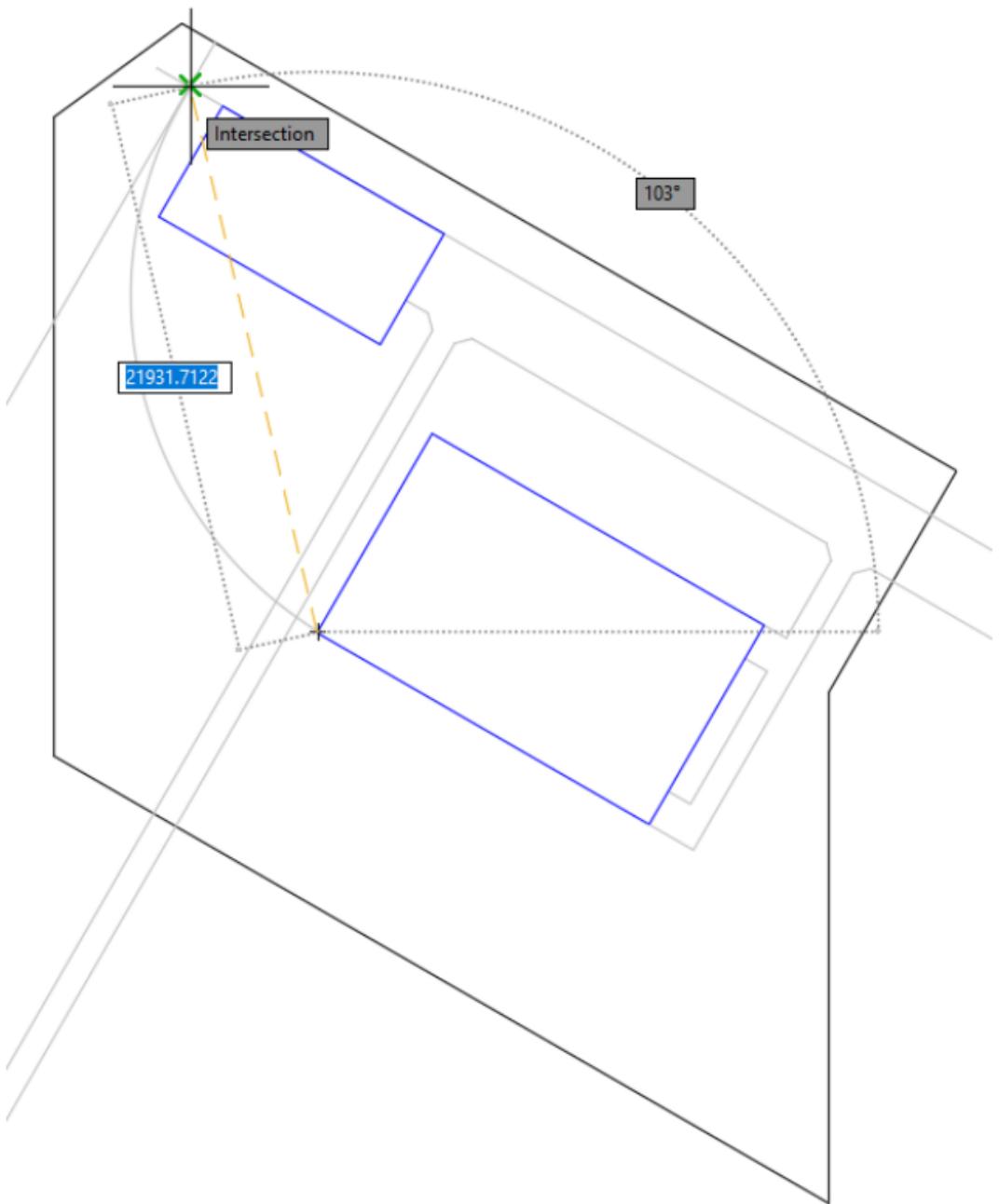


Figure 23. Drawing arc with polyline

Arc is then copied with offset with distance of 1500 units towards the buildings. After copying, the arc and lines are trimmed as shown in Figure 24.

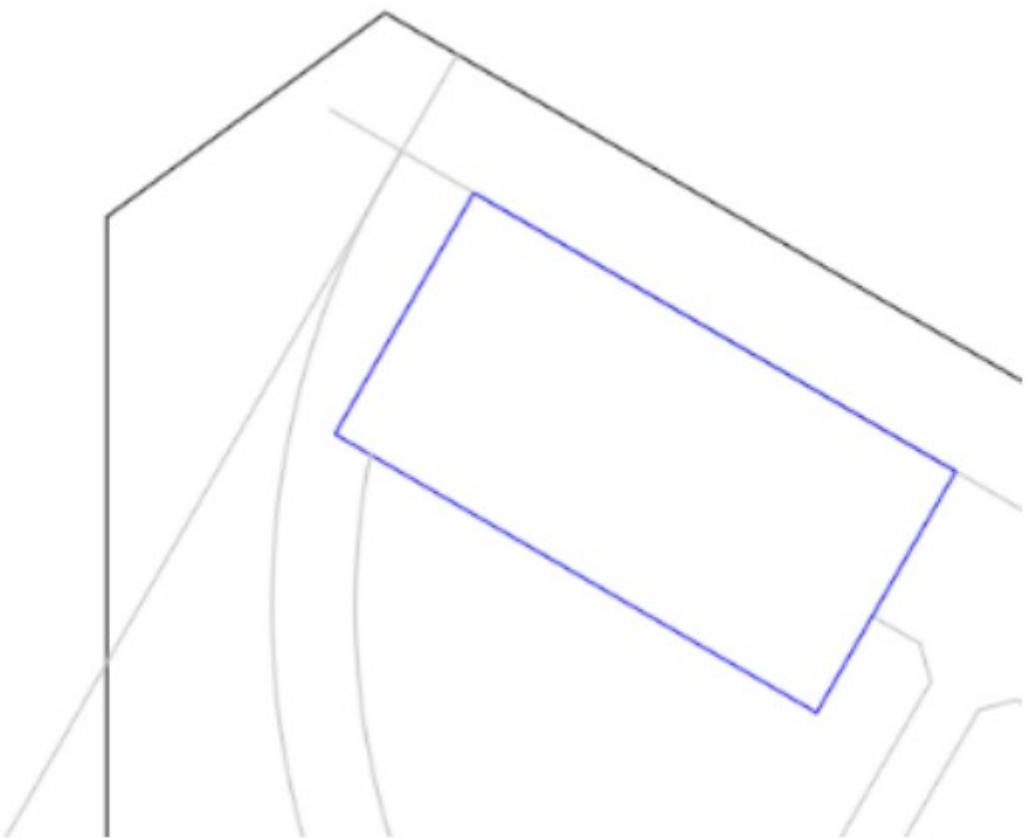


Figure 24. Trimming the arc

Finally, the courtyard between the buildings is shaped as shown in Figure 26. Shaping in started with **L (LINE)** command. Before the next phase, make sure that you have the following snap settings on: *Perpendicular*, *Extension*, and *Intersection*.

Line



Command: **line, l (Draw – Line)**

Line is a vector which's start and end points are known. Create a series of contiguous line segments. Each segment is a line object that can be edited separately. Line polygon can be closed with command **c (close)**.

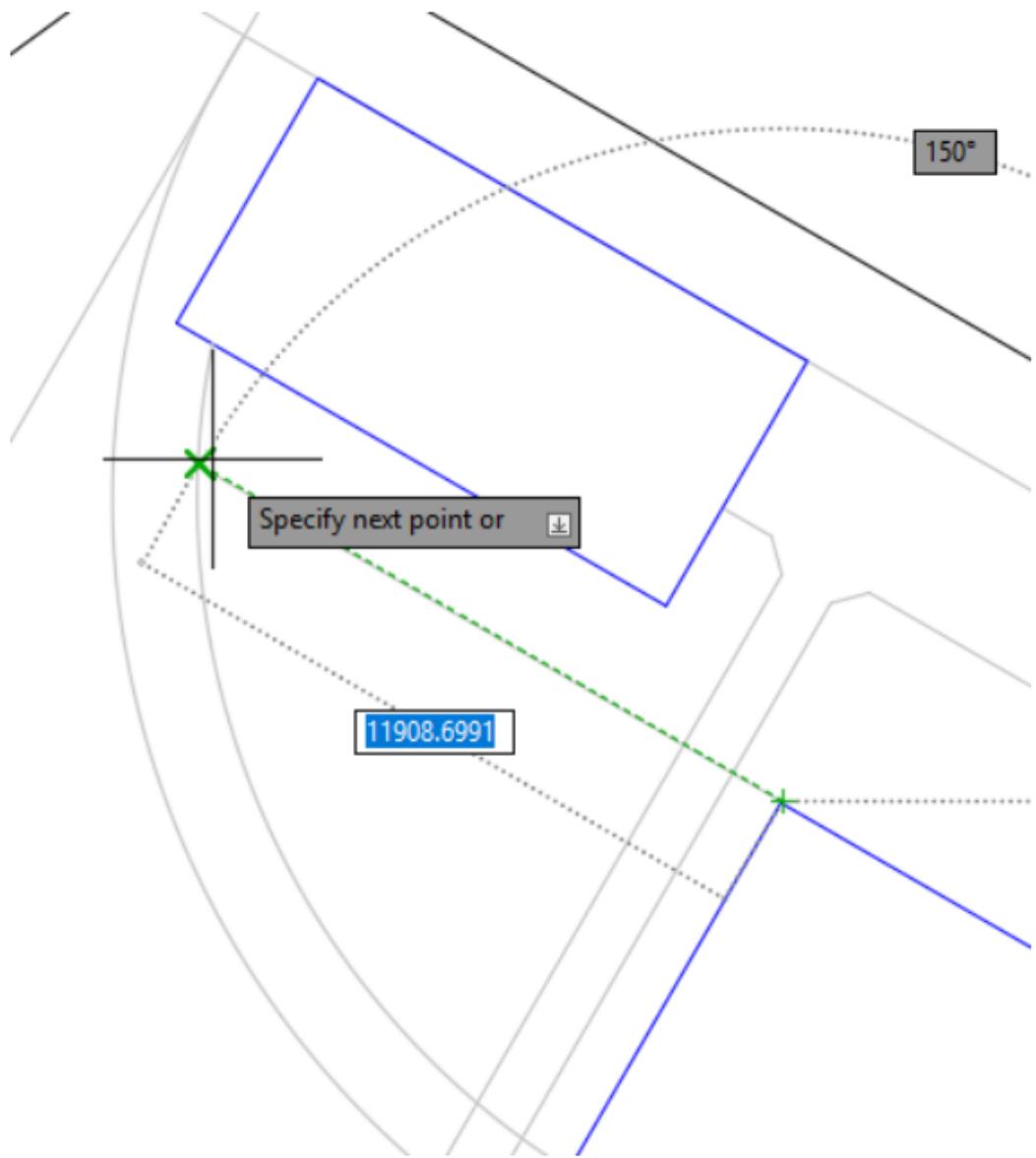


Figure 25. Drawing the edge of the road area with line command

Command: L

LINE

LINE Specify first point: (end snap to larger building's top left corner)

Specify next point or [Undo]: (mouse is brought to inner arc)

Specify next point or [Undo]:

Next the additional lines are trimmed as shown in Figure 26. The guideline next to the outer arc can be erased. Check that the road lines are at the correct layer. Layer of an object can be changed by selecting desired object(s) and changing their layer in the Properties menu.

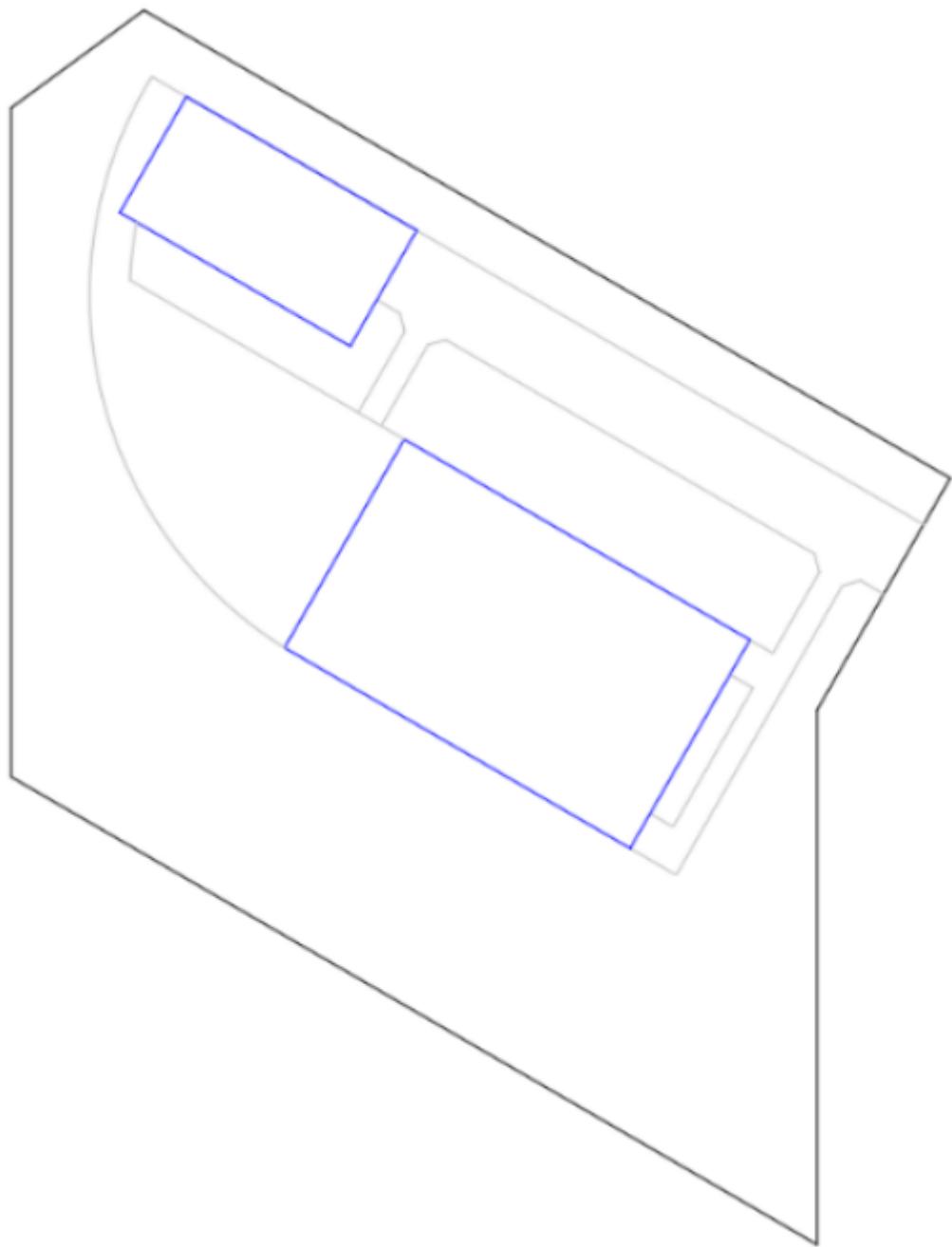


Figure 26. View of the drawing drawn so far

Lastly, corridors' corners are rounded with **FILLET** command. Fillet works similar to Chamfer. Notice that breaking the line before using the fillet makes things easier.

Fillet



Command: **fillet, f (Modify – Fillet)**

Rounds or fillets the edges of two 2D objects or the adjacent faces of a 3D solid. Size of the round is specified with radius. You can round lines, polylines, guidelines, arcs, and spline curves.

Command: FILLET

Current settings: Mode = TRIM, Radius = 100.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: r

Specify fillet radius <100.0000>: 500

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

Select second object or shift-select to apply corner or [Radius]:

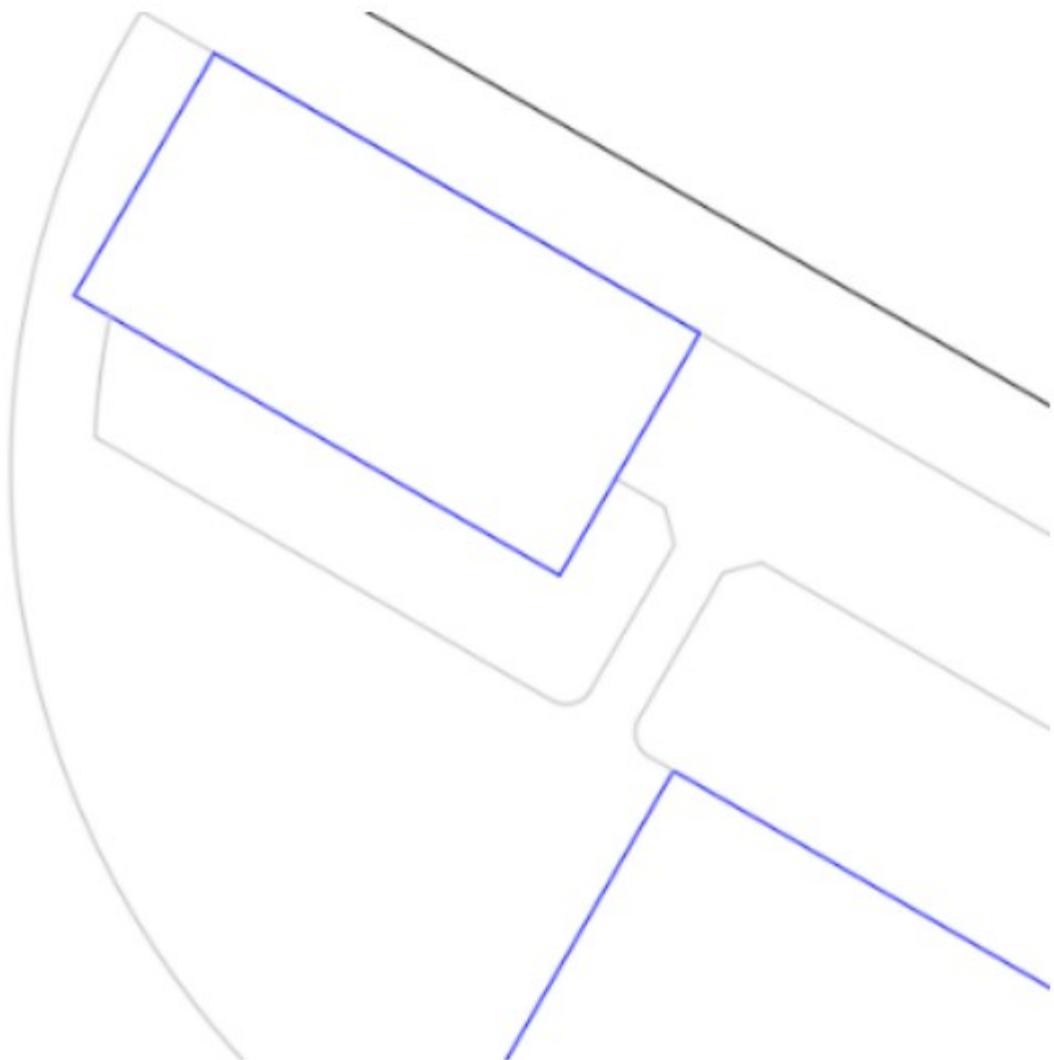


Figure 27. Finished fillets

The result is the plot of the site plan according to Figure 28.

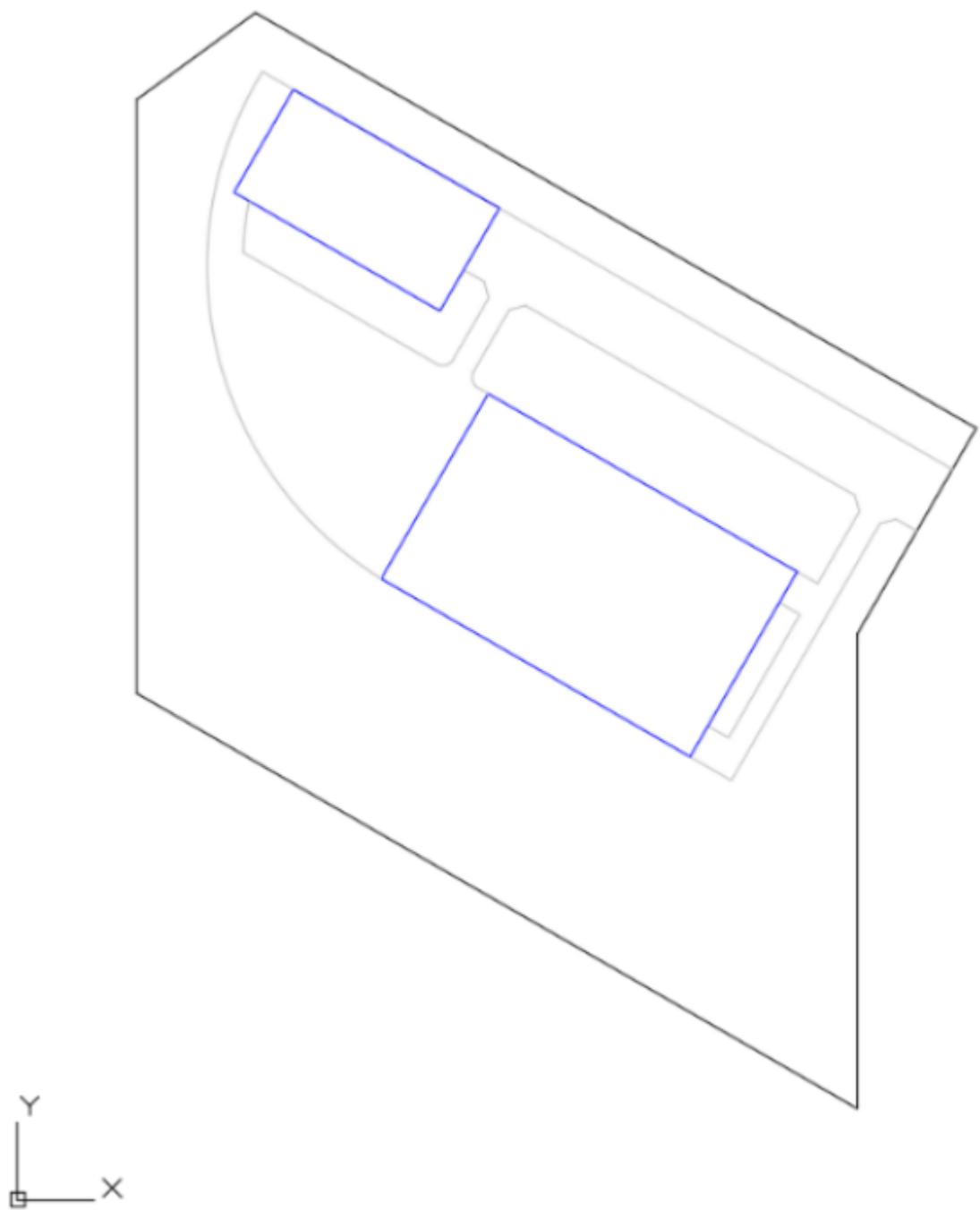


Figure 28. The lot of the site plan

Again, remember to save your work occasionally (ctrl + s)!

2.6 Some New Basic Settings

Grid and snaps

Command: **dsettings**

(Tools – Drafting Settings ...)

Command: **ddrmodes**

Command: **snap, grid**

Grid consists of a set of small dots. Density of the grid in the direction of the X and Y axes can be adjusted separately. Grid can be rotated just like a coordinate system. Grid can be turned on by hotkey **F7** or **GRIDMODE** icon in Status Bar. Snap is an invisible grid, that the cursor snaps on. Snap can also be adjusted like a Grid. Snap is turned on with hotkey **F9** or **SNAPMODE** icon in Status Bar.

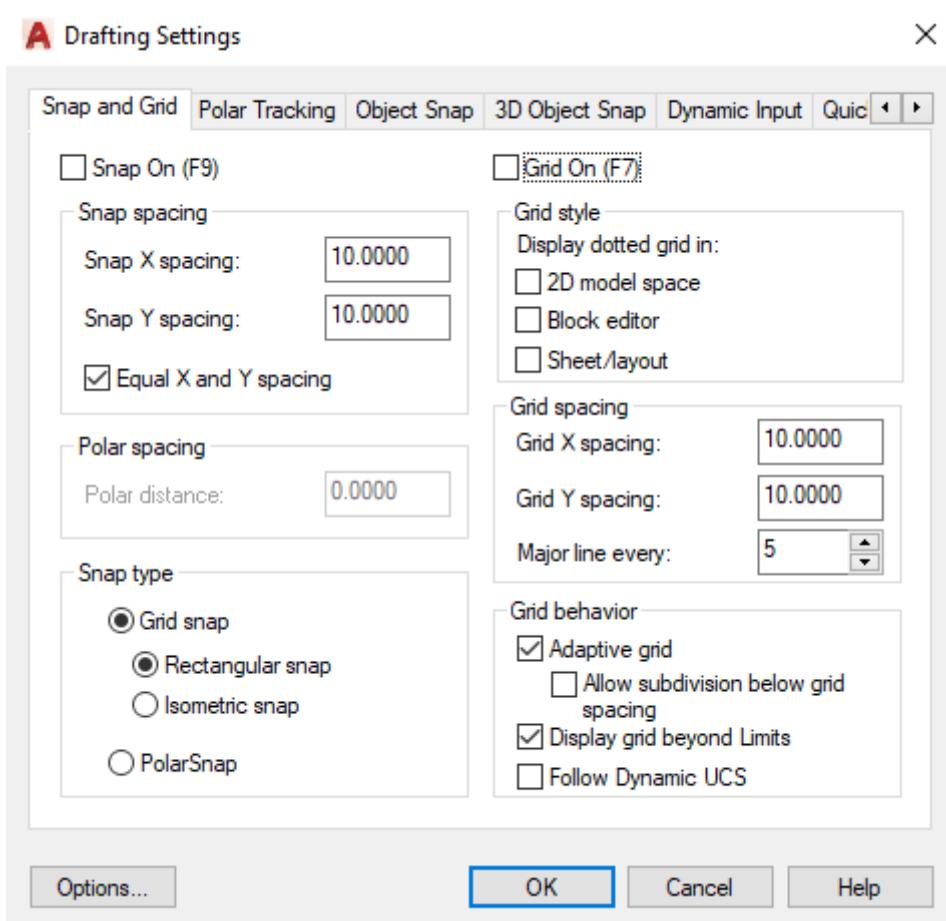


Figure 29. Drafting settings dialog box

Polar Tracking

Command: **dsettings**

(Tools – Drafting Settings ... – Polar Tracking)

Polar Tracking can be used to draw objects in a constant angle or on top of guidelines determined by already existing objects. Polar Tracking is turned on with hotkey **F9** or **Polar Tracking** icon in Status Bar. Polar Tracking settings are done in Polar tracking tab in Drafting Settings menu. *Increment angle* variable defines a constant angle that repeats. *Object Snap Tracking Settings* menu defines if track is for rectangular or both rectangular and polar angles. *Polar Angle Measurement* menu defines if polar coordinates uses absolute or relative coordinates. Grid and Snap Grid settings are modified in the Snap and Grid tab. Usually the grid and snap are set to be the same or multiples of one and another, allowing all coordinate points to be entered with mouse and working will be faster. At the same time, exact drawing is made possible which is the purpose of computer aided design.

2.7 Roofs

The buildings are missing roofs, there is no bushes or trees on the lot, and the dimensions are missing. Roofs are drawn on a “roofs” layer. Select **cyan** as the color of the layer and **DASHED** as the line type. This line type may not be included in the linetype list, so you need to download it into the list (*Layer – Linetype – Load*).

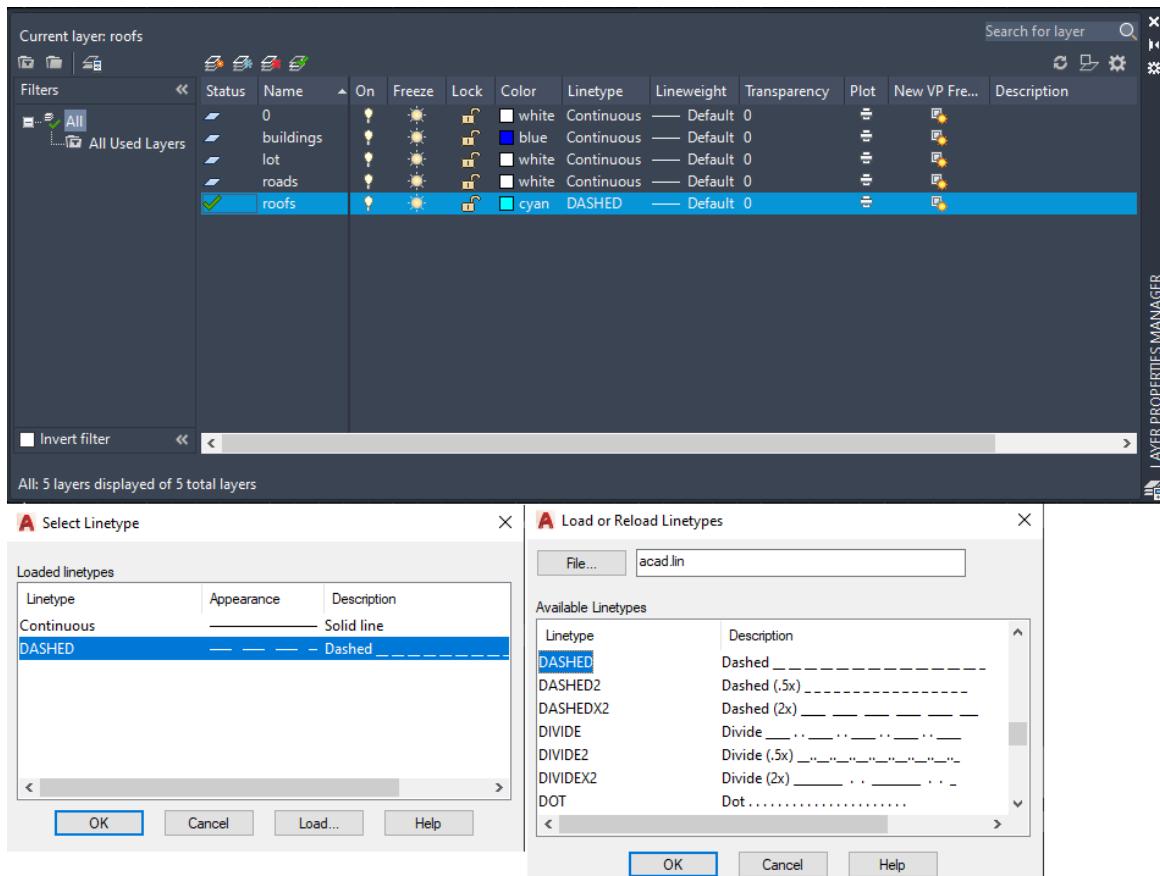


Figure 30. Defining a new linetype

The roof of the larger building is drawn first. The roof is replicated 500 units away from the walls with the Offset command. Duplicated object inherits the original properties, so the roof is drawn on the **buildings** layer. Layer can be changed by selecting the desired object and by selecting new layer from the Properties menu. The roof of a smaller building is drawn in the same way.

Although the linetype of the roof layer is dashed, the lines at the layer may appear solid because line scaling may be set to a scale that does not fit the image. The scaling is changed in a following way:

1. Select all lines on the layer
2. Open Properties menu with **CH (PROPERTIES)** command
3. Change *Linetype Scale* to 1000 (can be found under the General section). If the line is still solid, try scaling factor of 100 or 10.

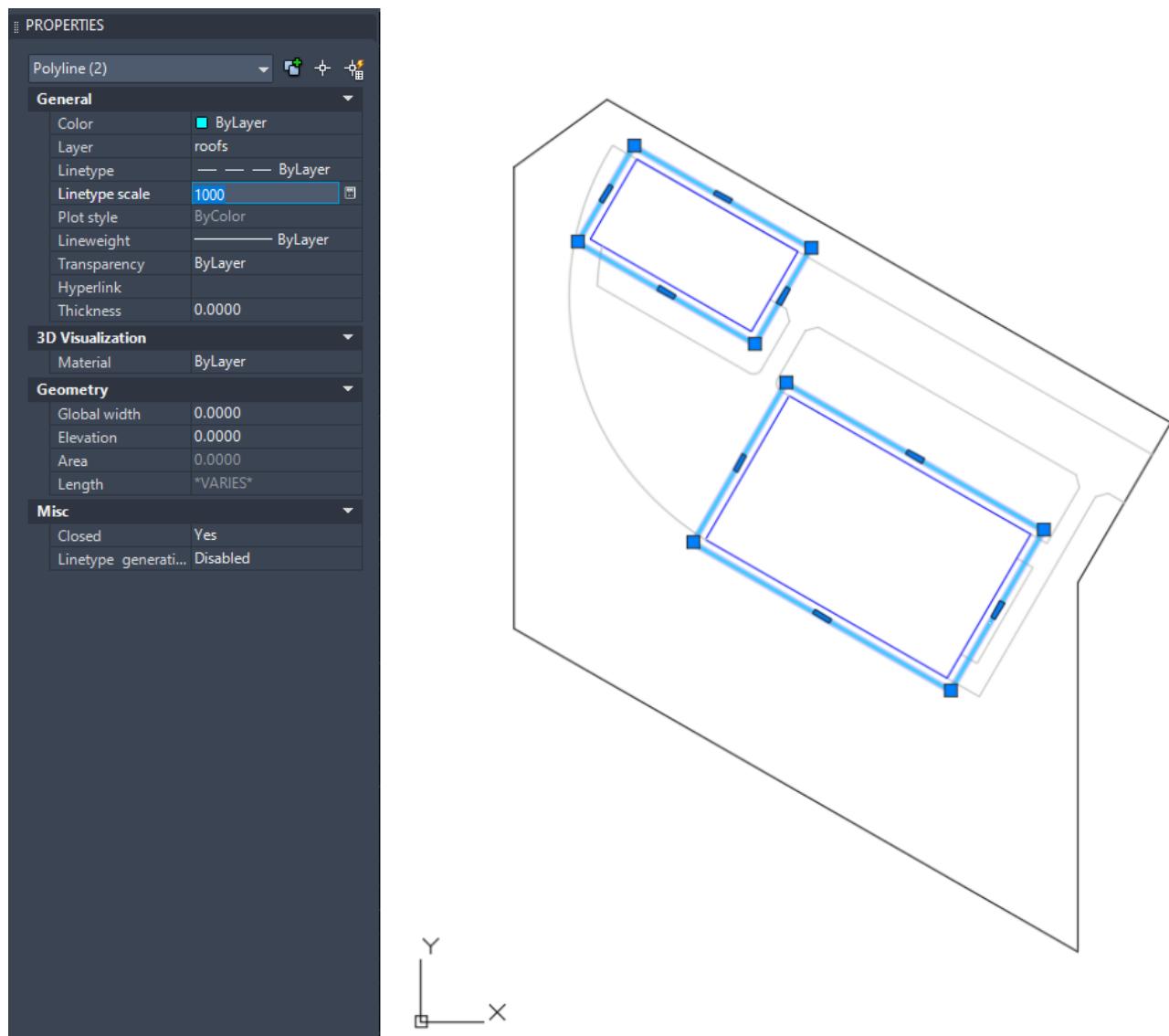


Figure 31. Changing the scaling factor

2.8 Trees and Bushes

Trees and Bushes are added to the drawing as blocks. Blocks are collections of objects that are combined into a single named object. The properties of a block depend on the layer at which it is created. If block is created on layer “0”, the properties comes from a layer that it is created on.

Begin drawing the Tree block by changing the layer to “0”. Tree block is formed from 5 arc objects. Draw the first arc with the **A (ARC)** command. Arc is drawn by giving it three points: starting, ending, and a point between them through which the arc must pass. After this, the arc is duplicated with the **AR (ARRAY)** command, so that the arcs form a tree according to the figures below.

Note! Draw the trees to a reasonable size in relation to the drawing. If the trees are sized wrong, they must be scaled later. This may cause some problems when importing the reference file.

Command: A

ARC

Specify start point of arc or [Center]:(with mouse)

Specify second point of arc or [Center/End]:(with mouse)

Specify end point of arc:(with mouse)

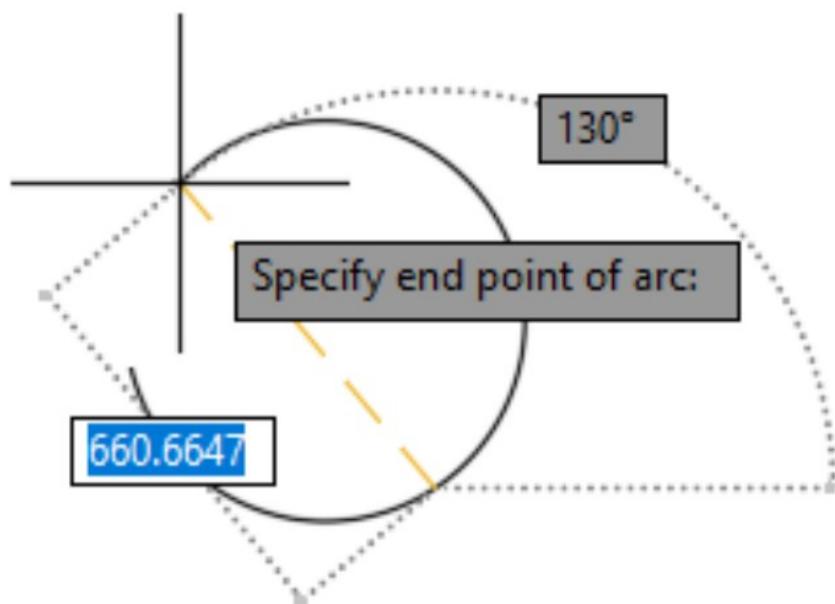


Figure 32. Drawing an arc

Command: AR

ARRAY

Select objects: 1 found

Select objects: (stop selecting by pressing enter)

Enter array type [Rectangular/PAtch/POlar] <Rectangular>: po (or by clicking Polar on the menu opening next to the cursor)

Type = Polar Associative = Yes

Specify center point of array or [Base point/Axis of rotation]:

(click approximately in the center of the missing arc section)

Select grip to edit array or [ASSociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit]<eXit>: I
(change the number of objects)

Enter number of items in array or [Expression] <6>: 5

Select grip to edit array or [ASSociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit]<eXit>: (enter)

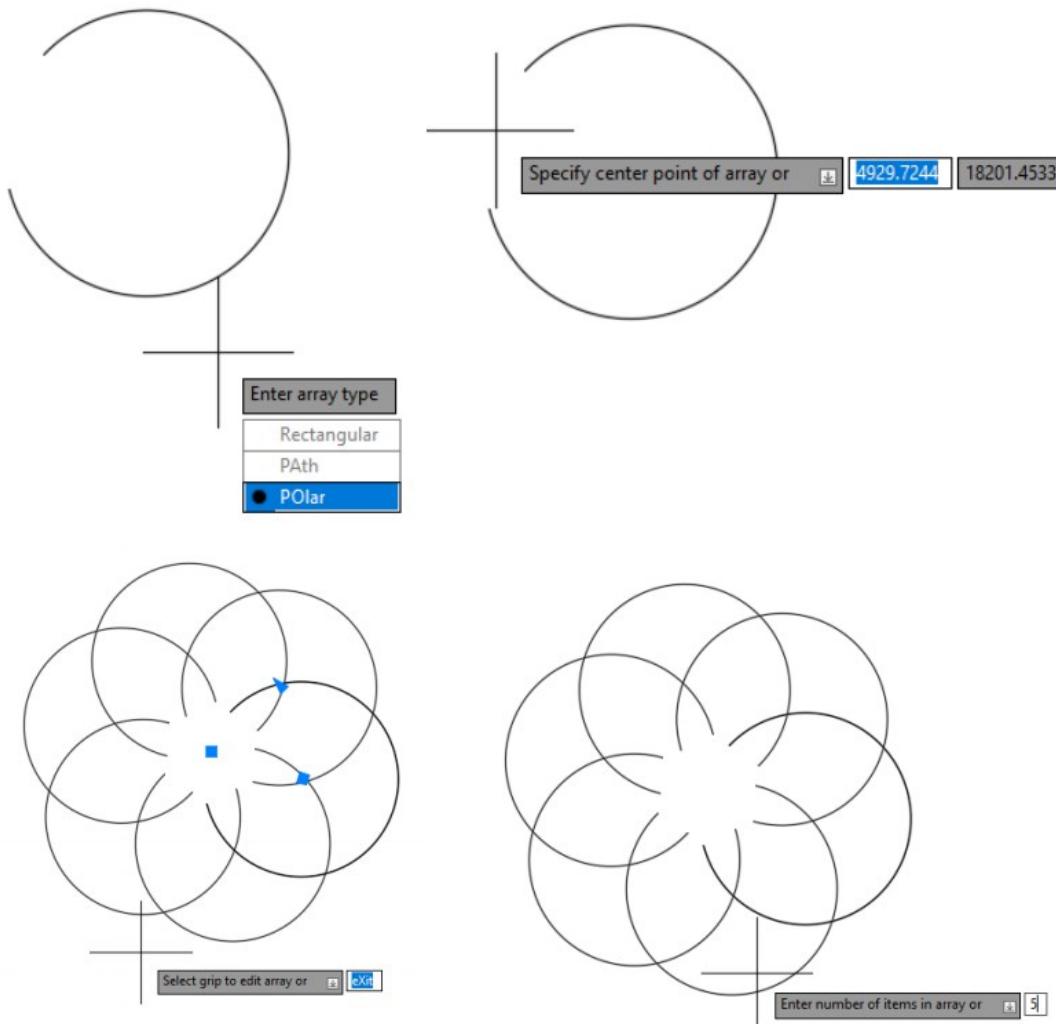


Figure 33. Using the Array command

Next the arc cluster will be made into a block with the **B (BLOCK)** command. Name the block a “Tree”. Base point means a point that is used to add the block into a drawing. Select the middle point of arc cluster as your Base Point by clicking *Pick point*. Select objects is used to specify the objects to be added on a block and select *Delete* option, which deletes the original objects from the drawing.

BLOCK

Select objects: 1 found

Select objects: (select 5 arcs of the tree)

Specify insertion base point: (click in the middle of the tree)

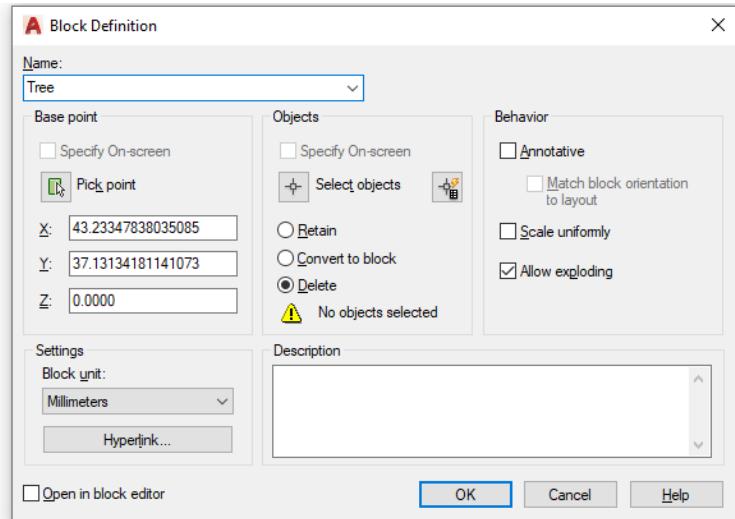
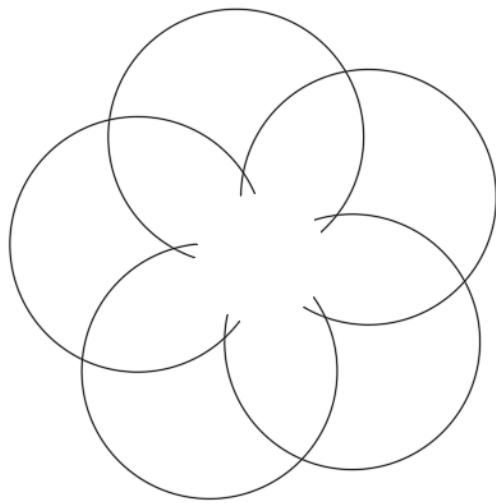


Figure 34. Block menu

Before adding blocks, create a new layer called “**trees and bushes**” (color green, linetype continuous). Blocks are inserted with the **I (INSERT)** command. Command will open a BLOCKS toolbar. The block that we just created should now be on the *Recent Blocks* list on the *Recent* tab. Keep the default options on the *Insertion Options* menu: *Insertion Point* selected, *Scale* variable as 1 in every direction, and *Rotation* variable as 0. Click on the tree block and insert it on a desired point.

Command: I

INSERT

Specify insertion point or [Basepoint/Scale/X/Y/Z/Rotate]:

(insert the tree on a desired point on the lot)

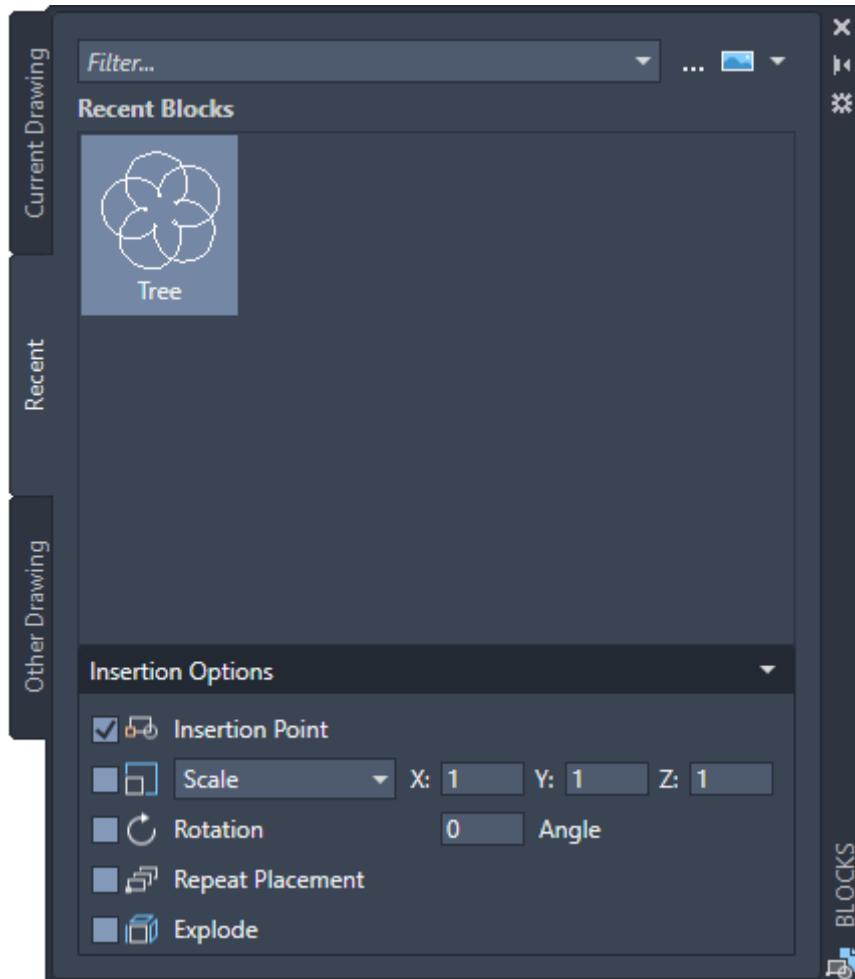


Figure 35. Adding a block with the Insert command

The remaining trees are then copied with the **CO (COPY)** command using ortho (**F8**) in line with the first. As a *Base Point*, select the middle point of a tree block. Next, enter a location for the next object. Keep on showing locations until you run out of space. The figure below will clarify the placement of the trees.

Command: CO

COPY

Select objects: 1 found

Select objects:

Current settings: Copy mode = Multiple

Specify base point or [Displacement/mOde] <Displacement>:

(with mouse)

Specify second point or [Array] <use first point as displacement>:(with mouse)

Specify second point or [Array/Exit/Undo] <Exit>:(etc)

Specify second point or [Array/Exit/Undo] <Exit>: *Cancel*

(press esc or enter)

First bush will be added with the Insert command. Bush is just a tree scaled down to half (with a scale factor of a 0.5), which can be done at the insertion stage. Select *Uniform Scale* variable as your Scaling option. Insert the block in a desired place and enter 0.5 as a scale factor in a window that opens. Copy the desired numbers of bushes with the Copy command.

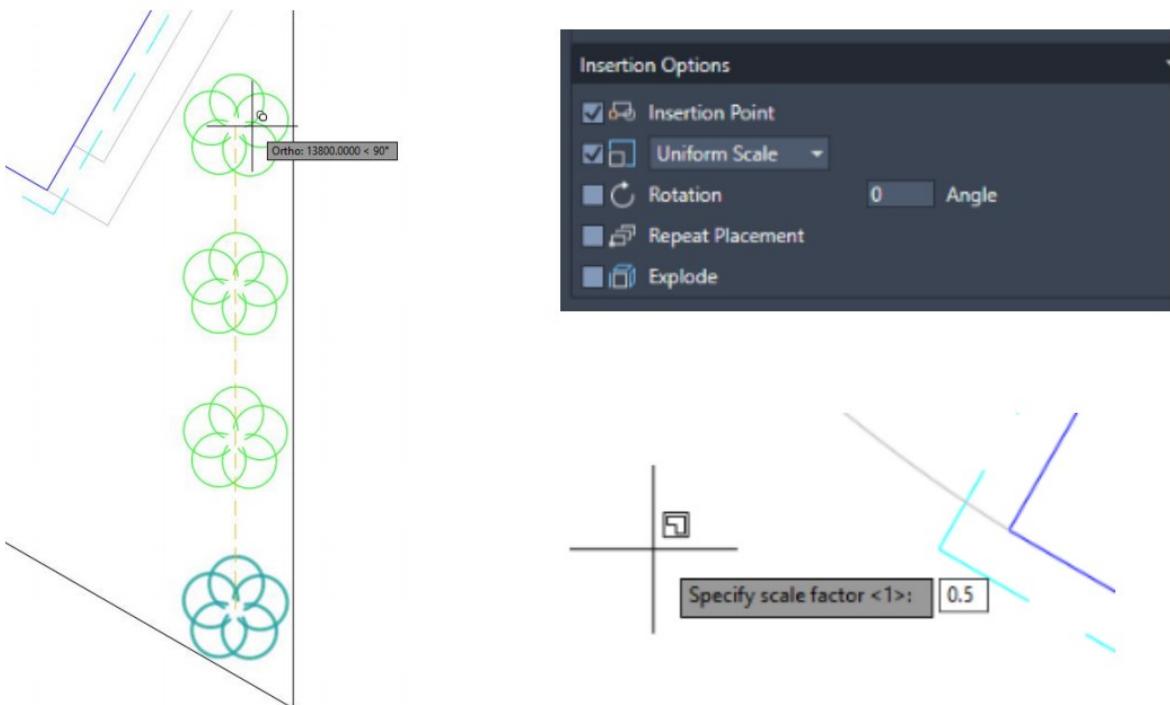


Figure 36. Copying the trees and scaling the bushes

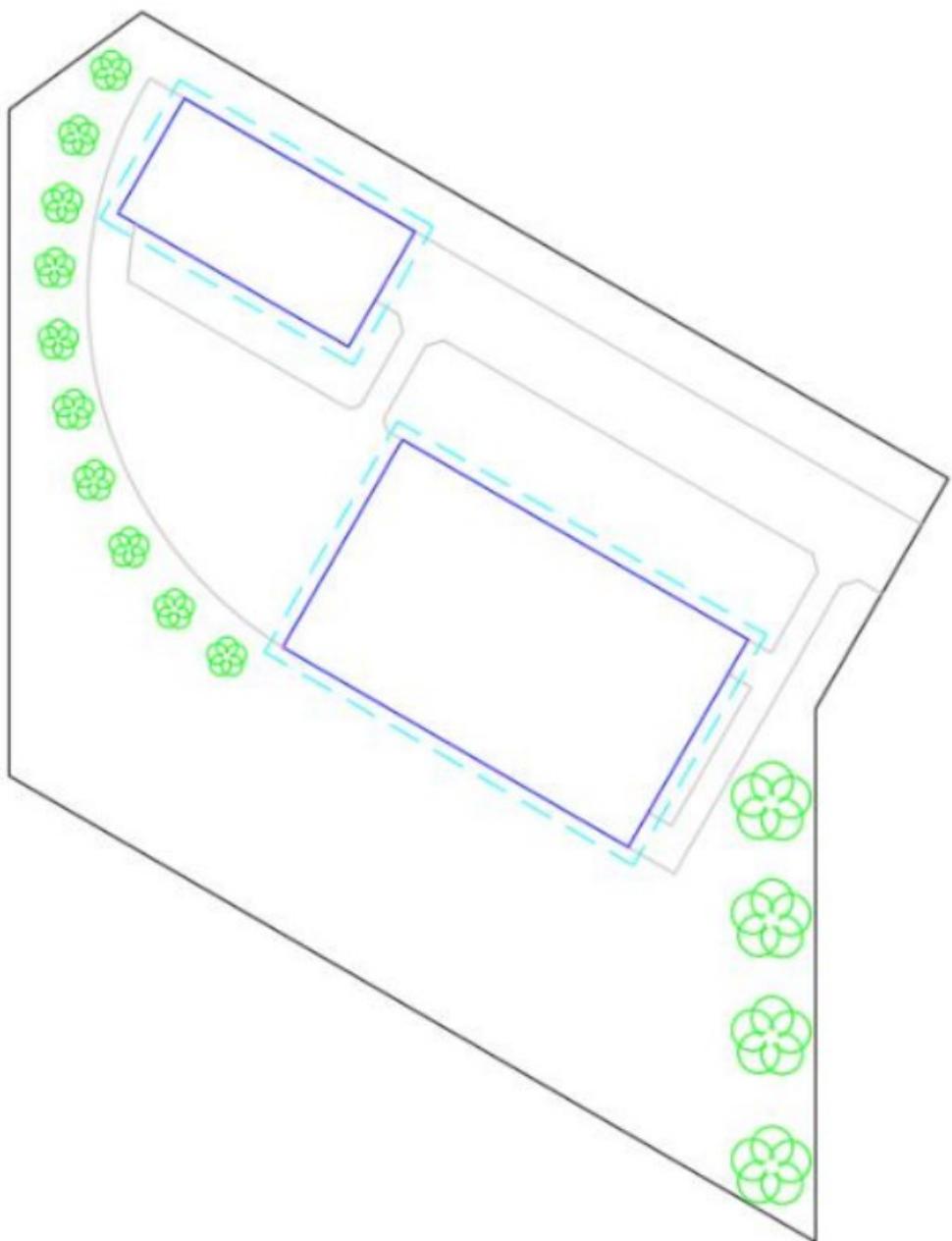


Figure 37. Finished lot with plantings

2.9 Import Reference File

Download EX1_Detail_Plan.dwg to your computer. Attaching it into you drawing is done with the **XR (XREF)** command and by clicking on *Attach DWG* on the External References menu, that just opened. Then select the desired file and press Open.

Select *Attachment* in the *Attach External Reference* window. If attachment is selected, the reference file EX1_Detail_Plan.dwg will be shown in a reference file made from the actual EX1.dwg. Select Reference Type: Overlay to only show the detail plan only in a file that it is imported to.

Next, input the location where to attach the reference file.

Specify insertion point or [Basepoint/Scale/X/Y/Z/Rotate]: 0,0

Change the imported reference file into a block, that is a part of the drawing, by right clicking on EX1_Detail_Plan file in the External References window and select Bind → Bind type: Insert.

NOTE! Reference file includes Tree blocks. If you did not draw reasonable sized trees, you will have to scale the trees in the reference file to a more reasonable size manually. You should do this by selecting all the trees, that needs scaling, and changing the Scale Factor in the Properties menu.

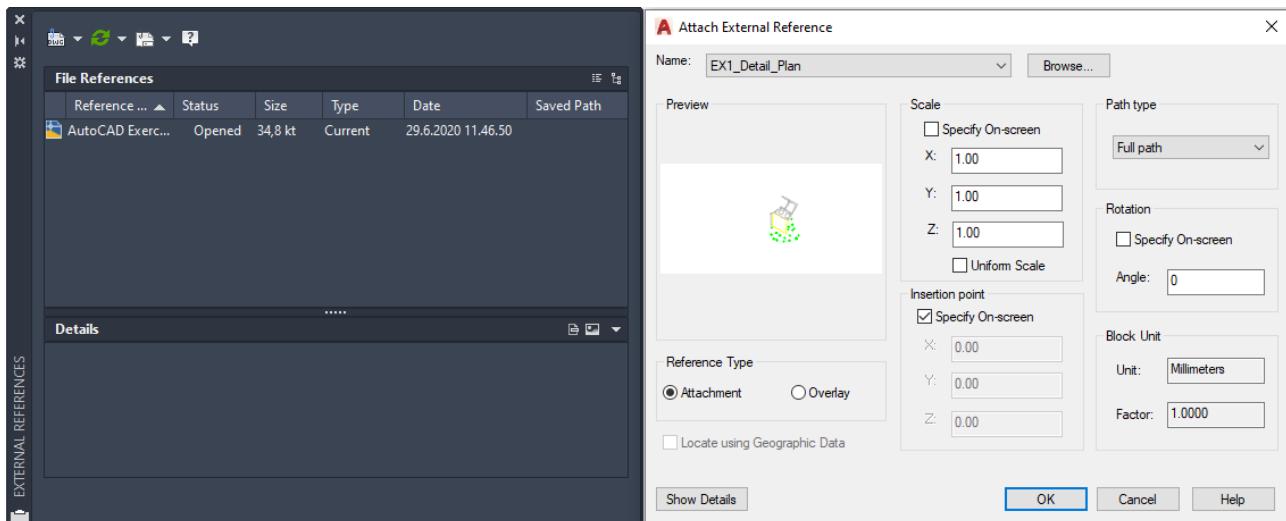


Figure 38. Importing a reference file

The yellow line the reference file should match exactly the lot borders, that you drew. The layers in a reference file will also be imported into your drawing. Explode the created block into component objects with the **X (EXPLODE)** command.

Explode



Command: **explode**, **x** (*Modify – Explode*)

Breaks a compound object into its component objects. Objects will revert to a lower definition level, in example polylines will revert into lines and arcs. Explode is often used to breaking down blocks, so they can be modified.

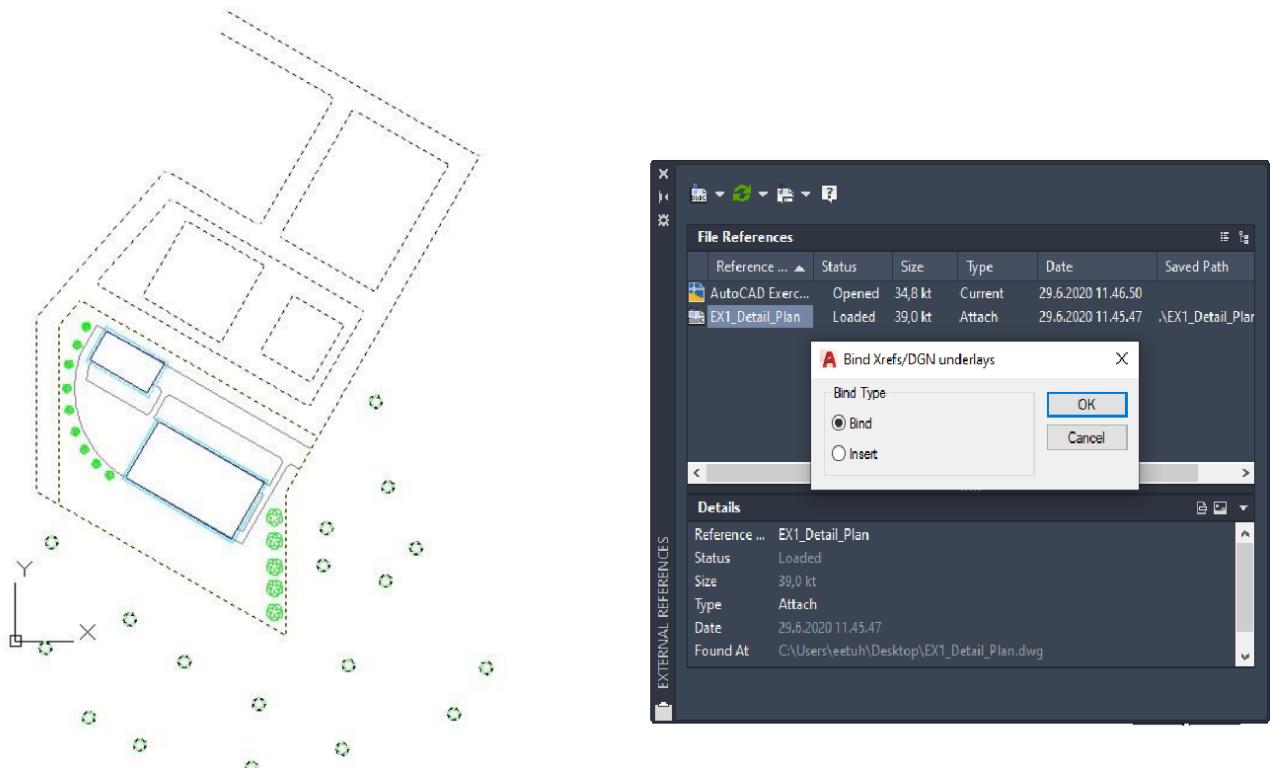


Figure 39. Attaching the reference file with the bind command

Command: X

EXPLODE

Select objects: 1 found (with mouse)

Select objects: (enter)

Check that the drawing does not have any redundant overlapping objects. Use the **OVERKILL** command for this. Hint, this can be useful in the exercise 2 also. Select all objects to be processed, in this situation you can select the whole drawing, and press enter. At the top of the Delete Duplicate Objects menu (Figure 40), there is a Tolerance value that determines how close objects have to be to one and another to be interpreted as similar (there is no need to change the value). Ignore Object Property can be used to filter out object properties. For example, by selecting Color and Layer fields and executing overkill for the whole drawing, the borders that you drew and the borders in the reference file will be interpreted as similar and one will be deleted. There is no need to use these options now, but if you suspect that there are overlapping objects in the drawing, it is recommended to try these out. In options menu, you should uncheck the *Optimize segments within polylines*, since often the use of partially overlapping polylines is justified. Then press OK and AutoCAD should erase overlapping objects.

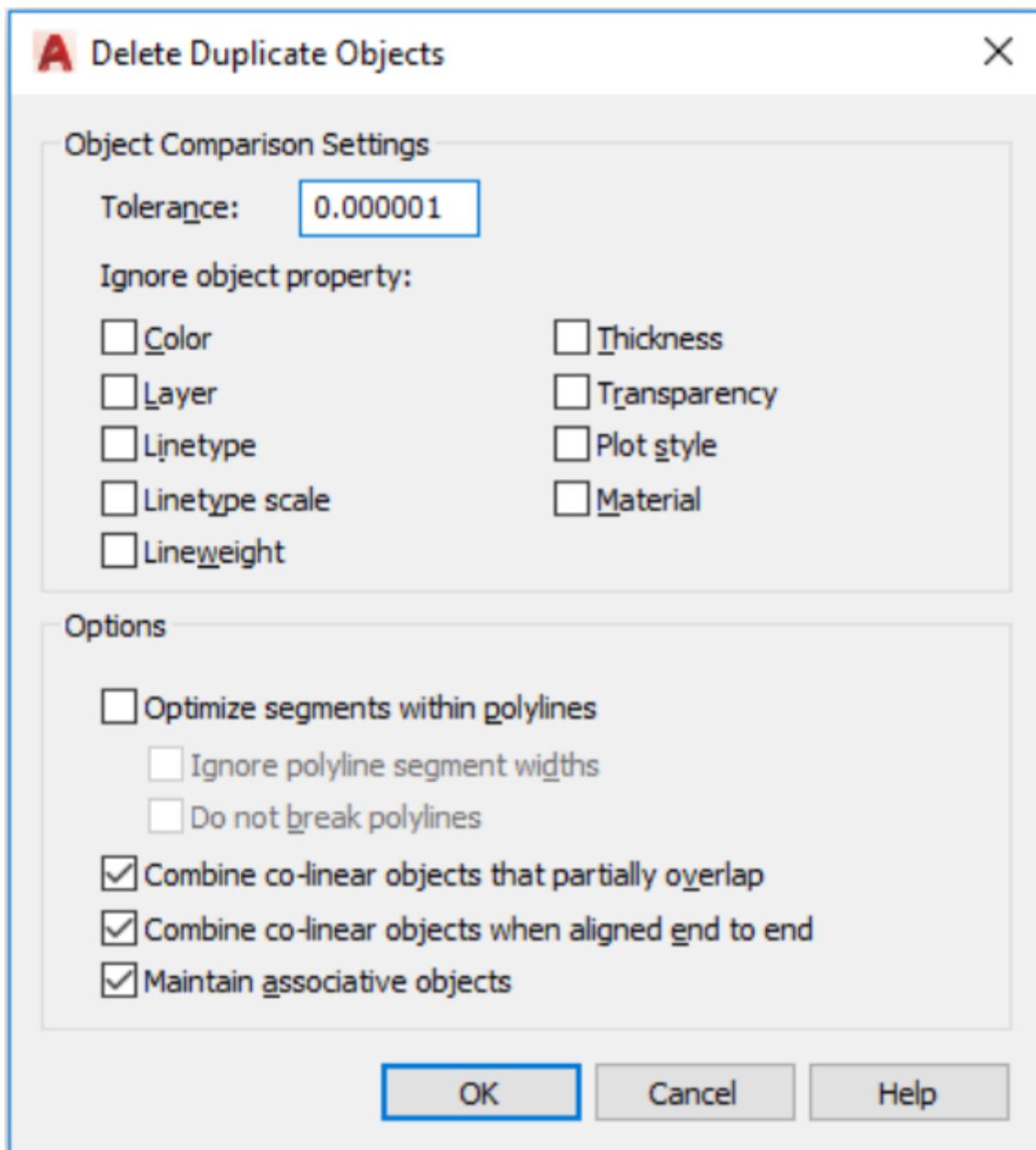


Figure 40. Delete Duplicate Objects dialog box

Remember to save your drawing at least every (ctrl + s)!

Units of the AutoCAD and the Purge Command

If you did not select your units correctly in the beginning, the reference file will now be way too small (the image you drew is about 25.4 times larger, because one inch is about 25.4 mm). The reference drawing is drawn in millimeters, and AutoCAD will not change the units between reference and the actual drawing.

If the imported reference does not match the dimensions of the drawing, it can be fixed by the following procedure:

1. Delete the imported reference (Delete or Erase)

Image will now be deleted from the view. In addition, it should be deleted from AutoCAD memory. This is done with the **PU (PURGE)** command. **NOTE! Purge can NOT be undone, so be careful when using it.**

2. Give the PURGE command. Find *Blocks* in the newly opened window and select EX1_Detail_Plan. After this click on the *Purge Checked Items*. Now the reference file is deleted from the memory.
- NOTE! DO NOT PRESS THE PURGE ALL! If you are unsure, seek advice from the assistant before continuing.**
3. When the detail plan is deleted, go to Drawing Units options (chapter 1.3) and change the *Units to scale inserted content* to millimeters.
4. Import the reference file again as described above.

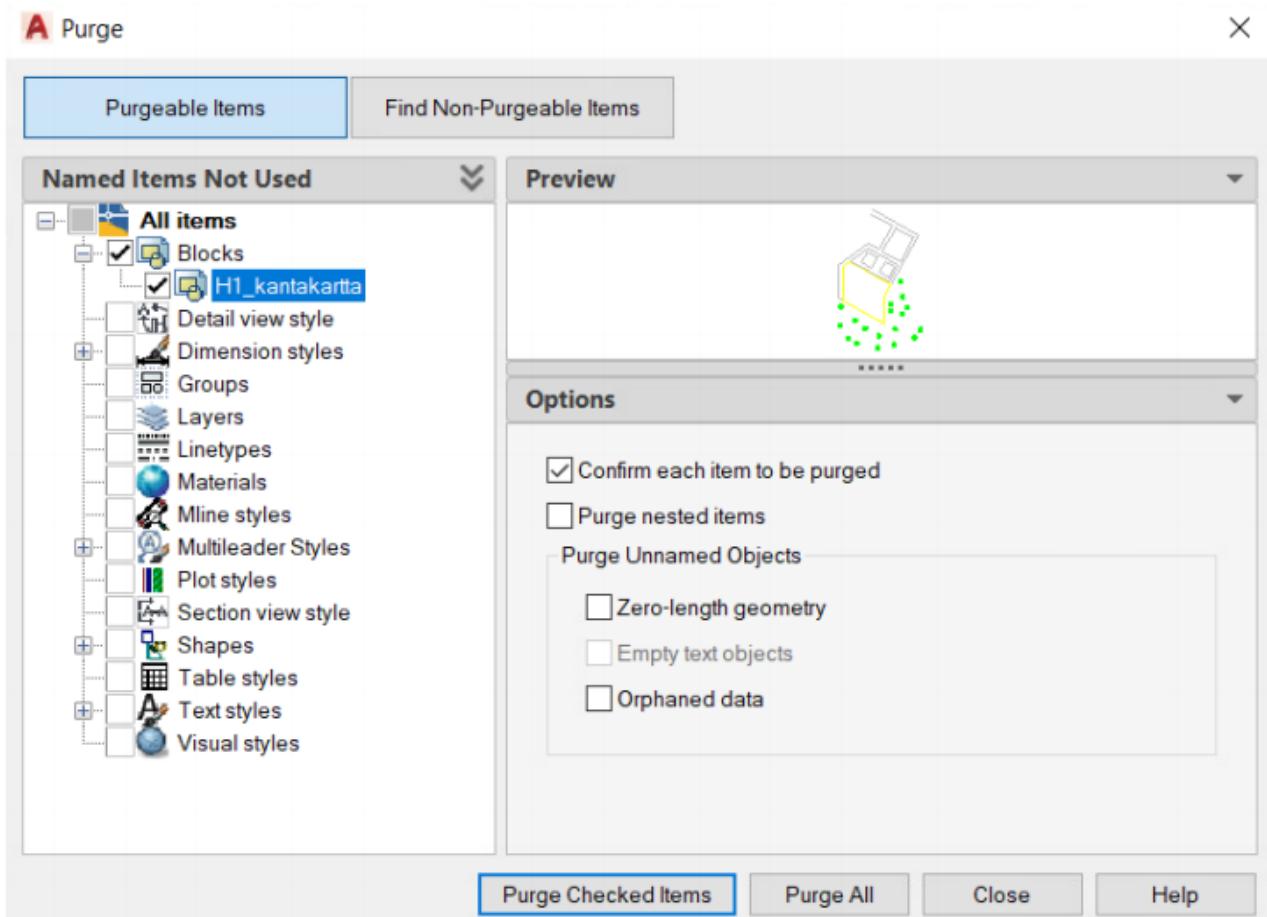


Figure 41. Purge dialog box

2.10 Dimensions

Before beginning the dimensioning, create new layer called “dimensions” (linetype **continuous**, color **magenta**). Dimensioning is started by creating a new dimensioning style with the **DIMSTY** (**DIMSTYLE**) command. Press **New** button to create a new style. Name the style as Dimensions-100, where the number 100 refers to a scale of 1:100. Standard dimension style is taken as the model and that dimension style is used for all dimensions. Continue creating the dimension style with the **Continue** button.

This color is magenta

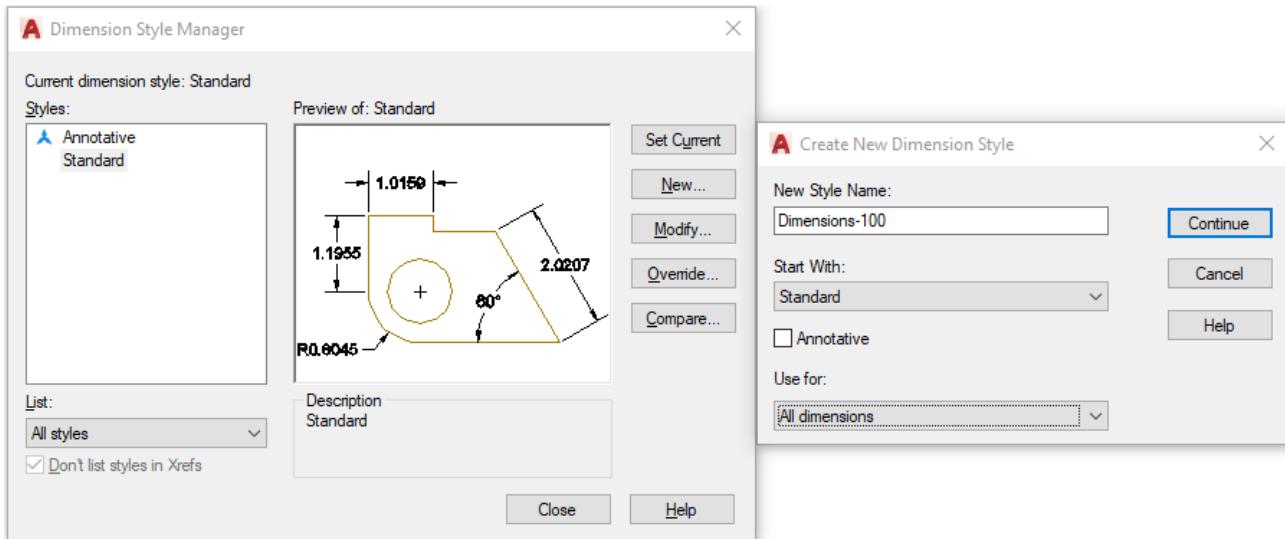


Figure 42. Creating a dimension style

Next, the Create New Dimension Style window opens, from which Lines tab is used to configure settings of the dimension lines, such as dimension line size and styles. *Offset from origin* specifies that how far from the selected point the guideline of the dimension line is started. *Extend beyond dim lines* defines how far the guides lines continue after the dimension line. With *Suppress Dim Line/ Ext Line* one of the main dimension lines or one of the guidelines can be taken off. Select *Closed filled arrows* as the arrowheads and 5 as the arrow size in the *Symbols and Arrows* tab. The preview image changes as you change the settings.

The *Text* tab is used to define settings of the dimensions text. You can change style, color, and height of the dimension text. The placement and the alignment can also be changed. *Offset from dim line* defines how far away the text will be from the guideline.

Fit tab specifies which ones of the dimensions will be shown if there is no room for both the arrow and the text. The most important thing on this tab is *Scale for Dimension Features*, which specifies the scale. In example, if you are drawing in a scale of 50, select 50 as *Scale for Dimension Feature*.

Primary units tab defines the units used in the dimensioning. Select precision, decimal pointer as well as possible rounding for the units. With *Prefix* and *Suffix*, you can add text at the beginning and end of the text, e.g. a plus-minus sign at the beginning. If *Measurement Scale* is set other than one, the numerical values of the dimensions are multiplied by the number indicated by the scaling factor. In the *Alternate Units*, you can define alternative units.

Tolerance tab defines the settings related to tolerances.

Define your dimension style Dimensions-100 as shown in Figure 43. After you are done with the settings, press OK to return to *Dimension Style Manager* window. Set Dimensions-100 as the current dimension style by pressing *Set Current* button.

NOTE! If the dimension style does not match the model, you can try and change the settings in the Dimstyle Manager. The most important thing is that the style roughly matches the model and that the dimensions are easily readable.

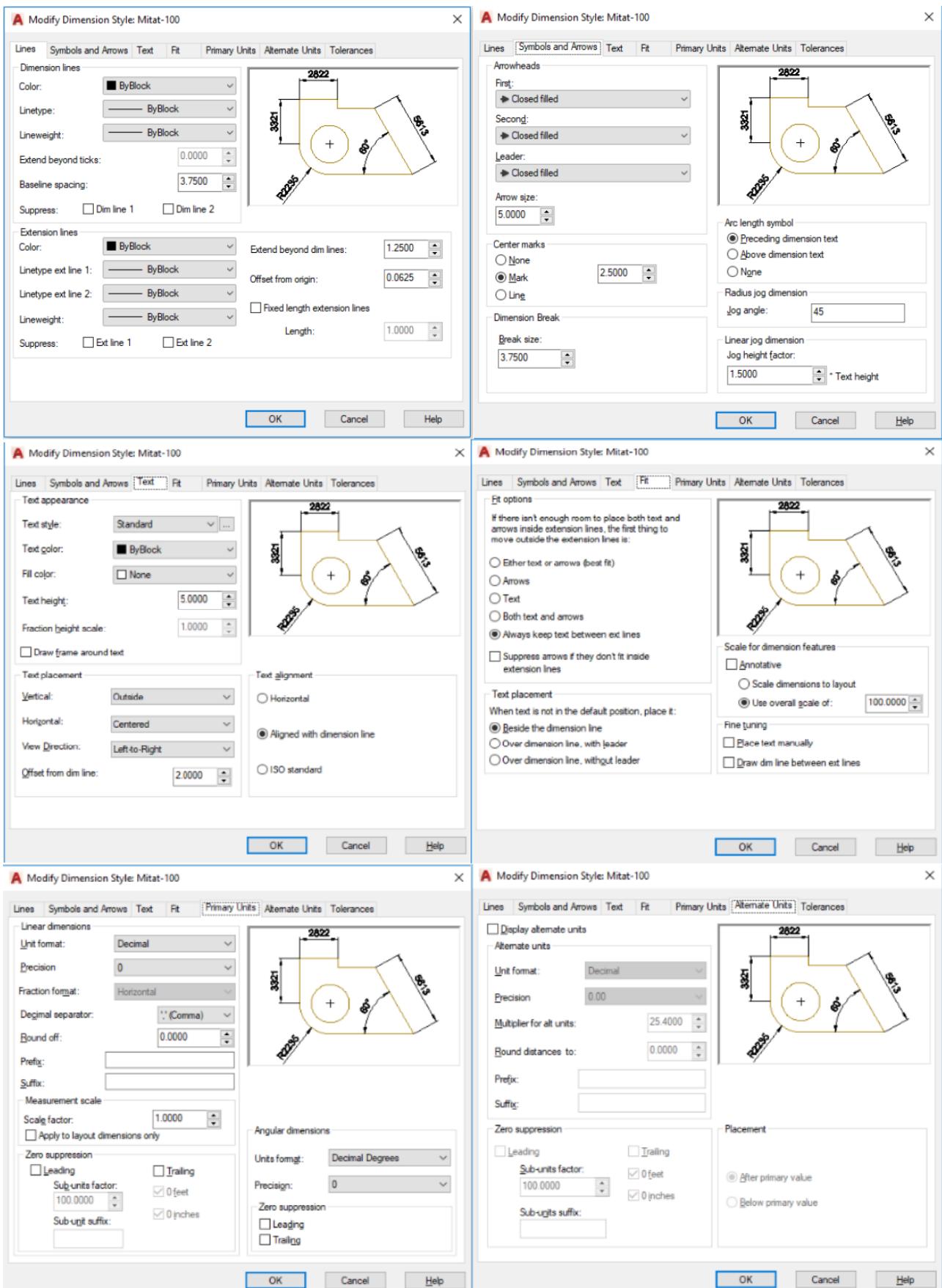


Figure 43. Dimension style settings

Now we can start with the dimensioning. For dimensioning, use Endpoint and Perpendicular snap options. Dimensions can be started with the **DIMALI (DIMALIGNED)** command, which is used to dimension objects that are not parallel with the coordinate axes. **DIMLIN (DIMLINEAR)** is used to dimension objects parallel to coordinate axes. First dimension is determined with the mouse as shown in Figure 44Figure 44.

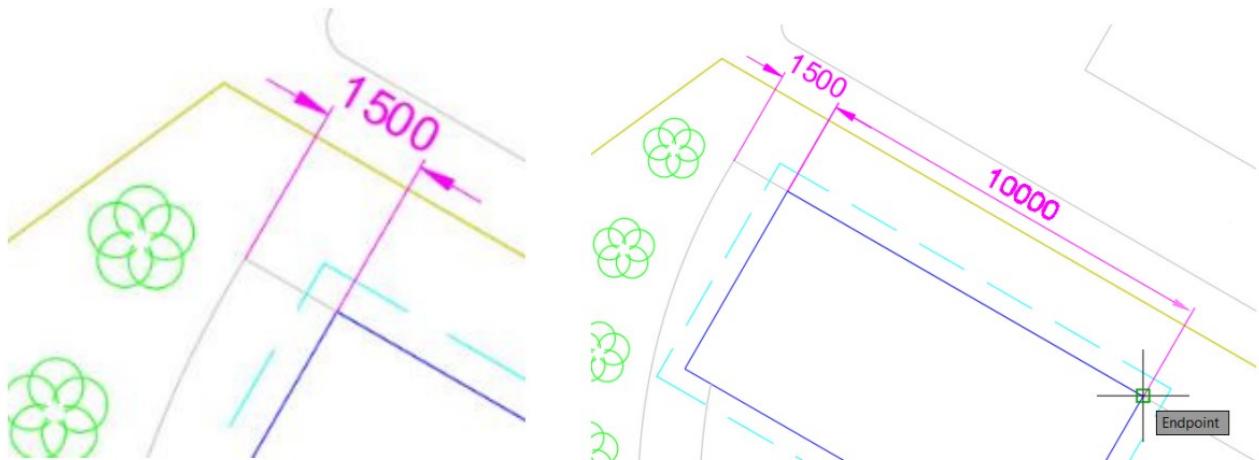


Figure 44. Dimensioning with Dimaligned and Dimcontinue commands

Command: **DIMALI**

DIMALIGNED

Specify first extension line origin or <select object>: (select the starting point of the dimension)

Specify second extension line origin: (select the endpoint of the dimension)

Specify dimension line location or

[Mtext/Text/Angle]: (click the dimension text where you want it)

Dimension text = 1500

Command: **DIMCONTINUE**

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 10000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1500

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 15000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 1500

Specify a second extension line origin or [Undo>Select] <Select>:

Select continued dimension: (enter)

Some of the dimension texts are overlapping in the newly created dimension style (Figure 45), so the dimension style has to be modified. Use the Dimstyle command, select the Dimensions-100 style and press modify. Try out some settings to make them readable.

Individual dimension texts can be moved by moving your cursor on top of the select point of the dimension text and by selecting *Move with Leader* (Figure 46).



Figure 45. Overlapping dimension texts

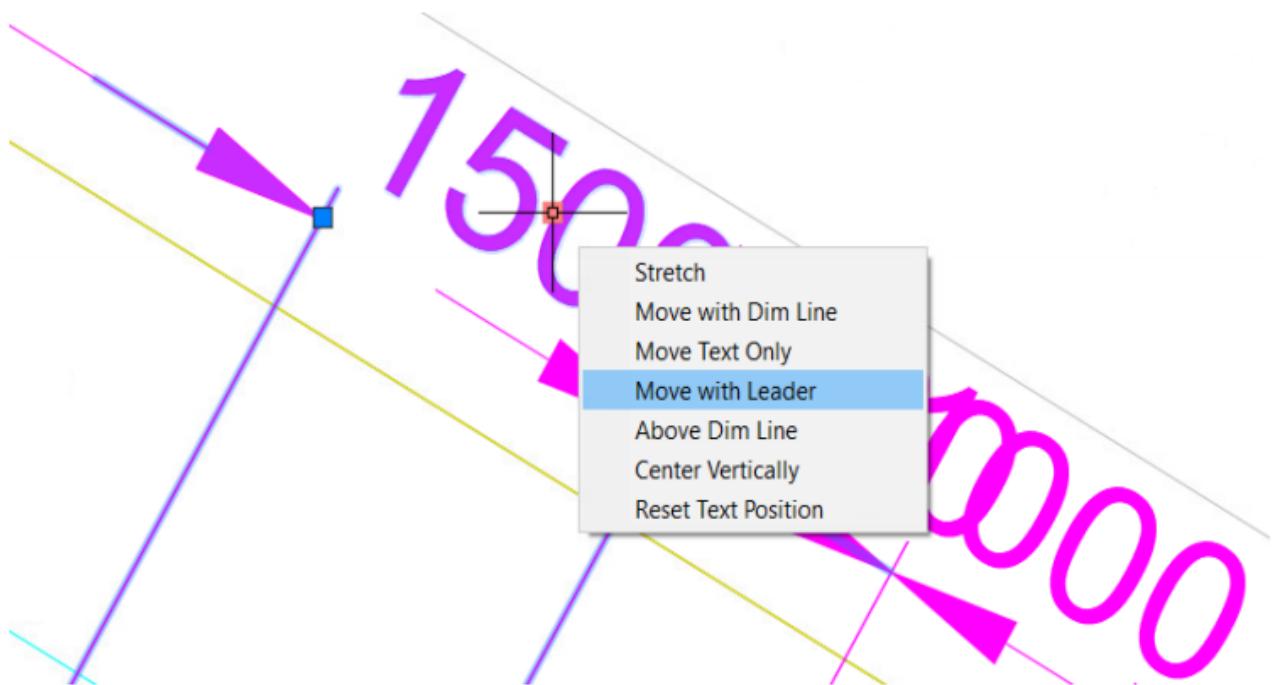


Figure 46. Moving an individual dimension text



Figure 47. All of the dimensions text are legible

Dimension the right side of the lot next. Use the Dimaligned command draw the first dimension. Continue the dimensioning with the **DIMBASE (DIMBASELINE)** command. Dimbaseline continues dimensioning from the point determined by the first dimension. Dimensions overlap a little, so you have to move some dimension lines. Dimension lines can be moved by selecting the desired dimension, so that the grips become visible. Drag the grip to a desired place with your cursor. Lastly, dimension the corridor in front of the building by using the Dimaligned command and setting the dimension to the same layer as the other dimensions.

Command: DIMALIGNED

Specify first extension line origin or <select object>:

Specify second extension line origin:

Non-associative dimension created.

Specify dimension line location or

[Mtext/Text/Angle]:

Dimension text = 2000

Command: DIMBASE

DIMBASELINE

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 5000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 9000

Specify a second extension line origin or [Undo>Select] <Select>:

Dimension text = 18000

Specify a second extension line origin or [Undo>Select] <Select>:

Select base dimension: *Cancel*

Command: DIMALI

DIMALIGNED

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or

[Mtext/Text/Angle]:

Dimension text = 1500

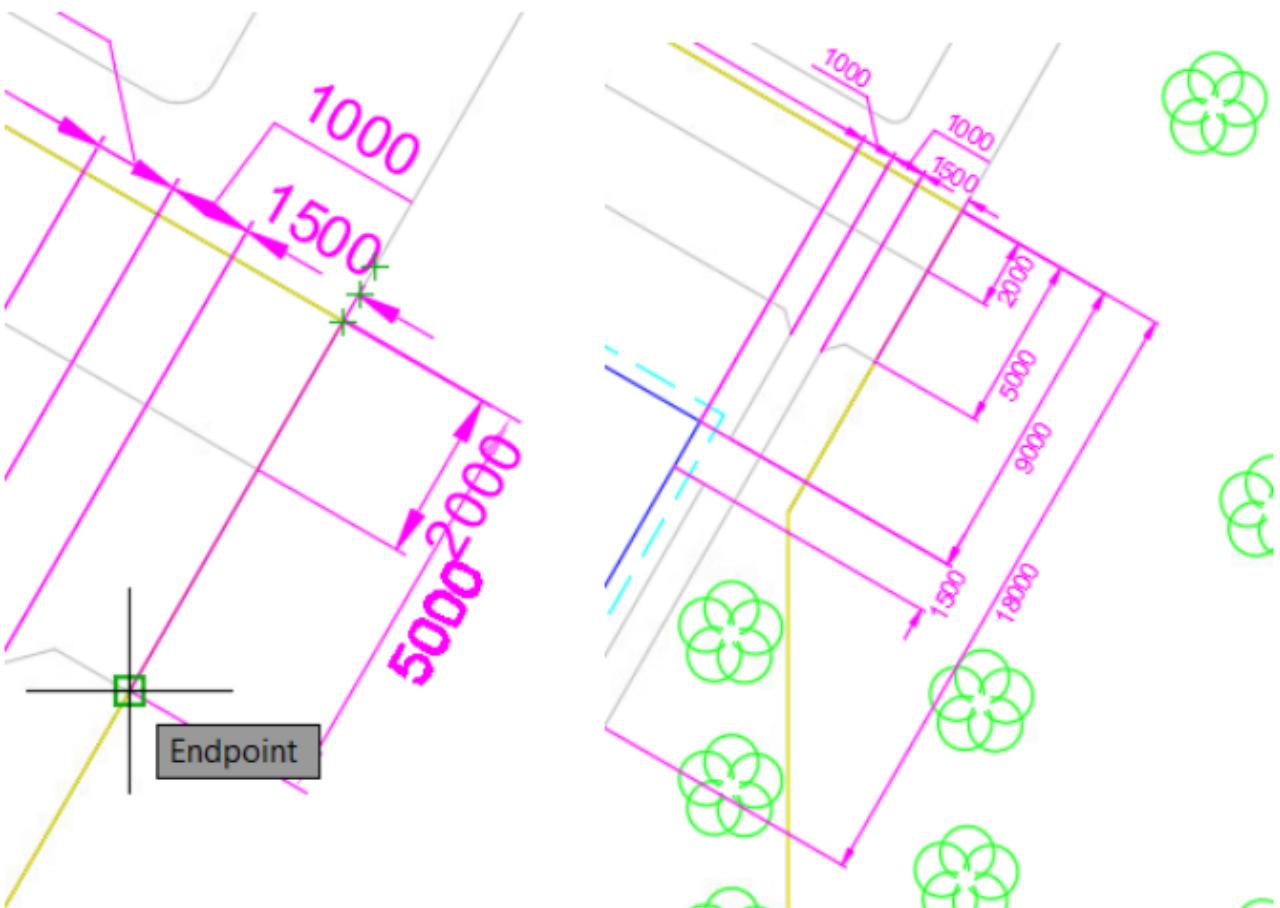


Figure 48. Dimensioning the right side

Then we need to dimension the angles specifying borders of the lot. Angles are dimensioned with the **DIMANG (DIMANGULAR)** command. Use the ortho (**F8**) while doing it. **Do the angular dimensioning to top left corner also (see Figure 52)!**

Command: DIMANG

DIMANGULAR

Select arc, circle, line, or <specify vertex>: (enter)

Specify angle vertex: (endpoint selection from the tip)

Specify first angle endpoint: (endpoint selection other end of the line)

Specify second angle endpoint: <Ortho on> (select a point with mouse and ortho)

Specify dimension arc line location or

[Mtext/Text/Angle/Quadrant]: (select inner angle with mouse, Figure 49)

Dimension text = 30

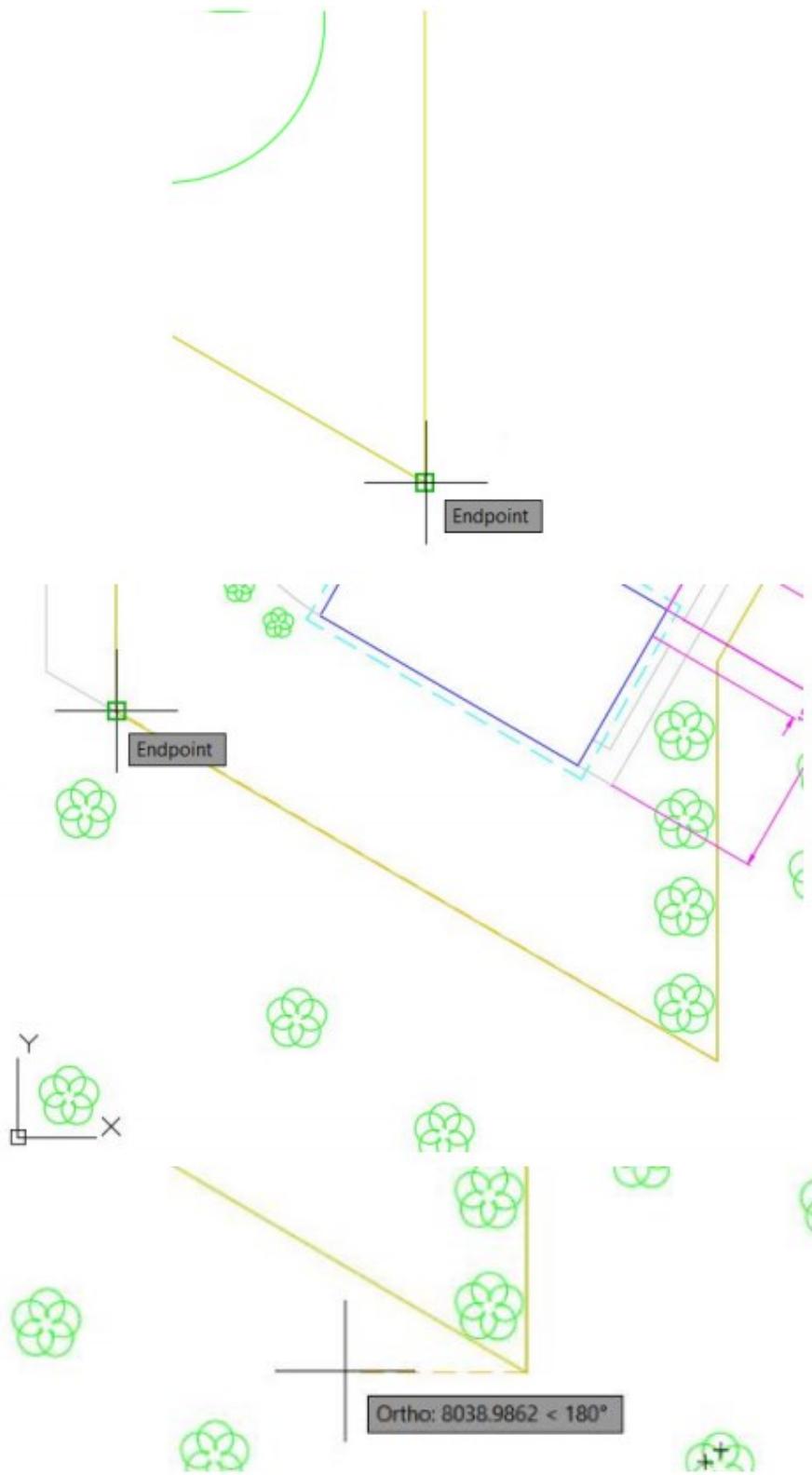


Figure 49. Angular dimensioning with the Dimangular command

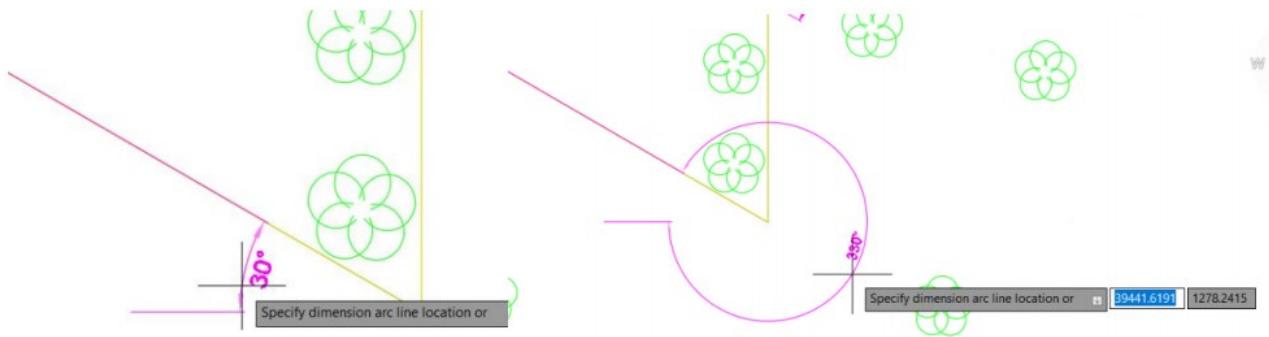


Figure 50. The layout of the angle affects the magnitude of the angle

Lastly, dimension radius of the fillet with the **DIMRAD (DIMRADIUS)** command.

Command: **DIMRAD**

DIMRADIUS

Select arc or circle: (click on the arc with mouse)

Dimension text = 500

Specify dimension line location or [Mtext/Text/Angle]: (click on the desired place with your cursor)

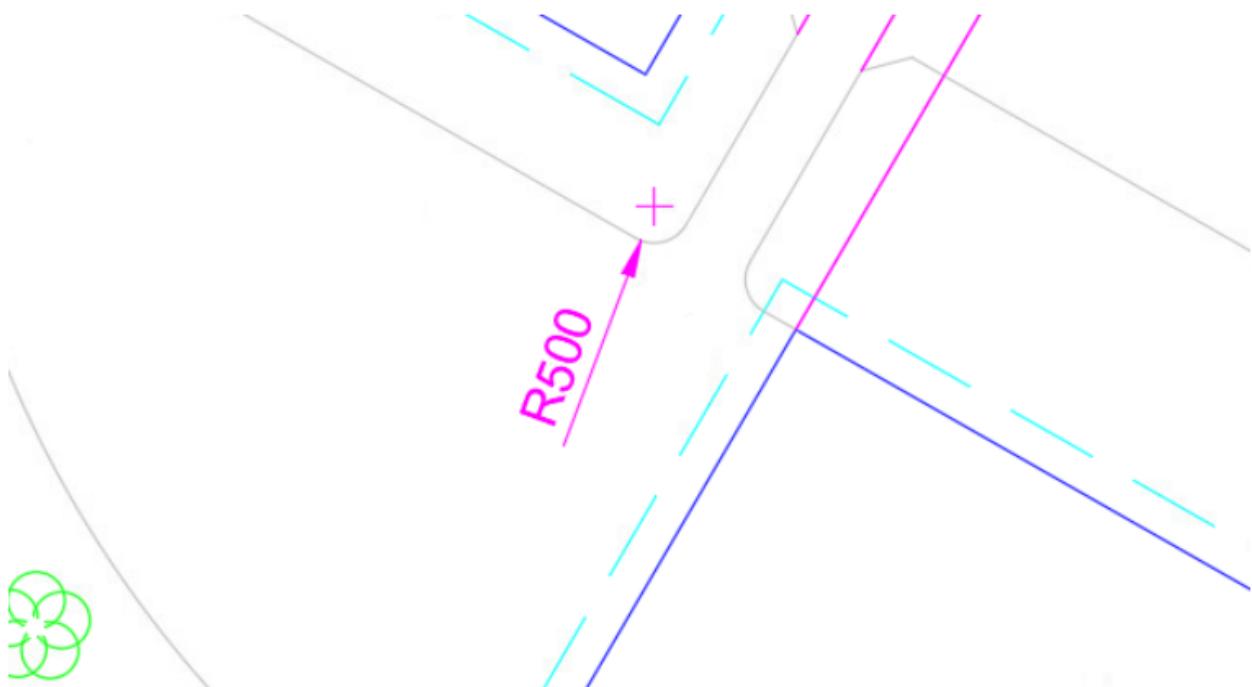


Figure 51. Dimensioning of the fillet radius

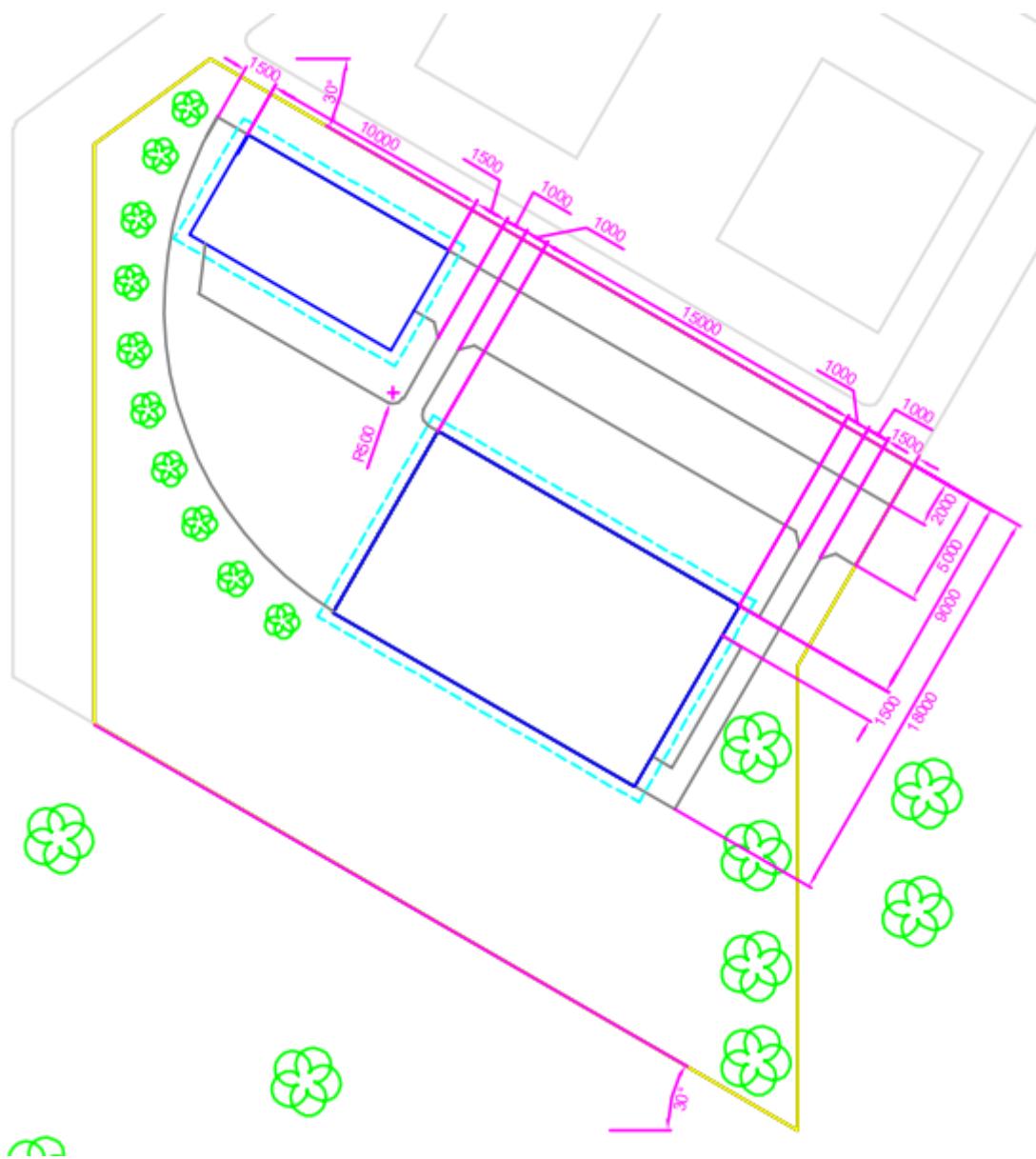


Figure 52. Finished dimensions

Save your drawing. According to instructions 2.11-2.13, print (student number)_1.pdf file on a A3 sheet at a scale of 1:500.

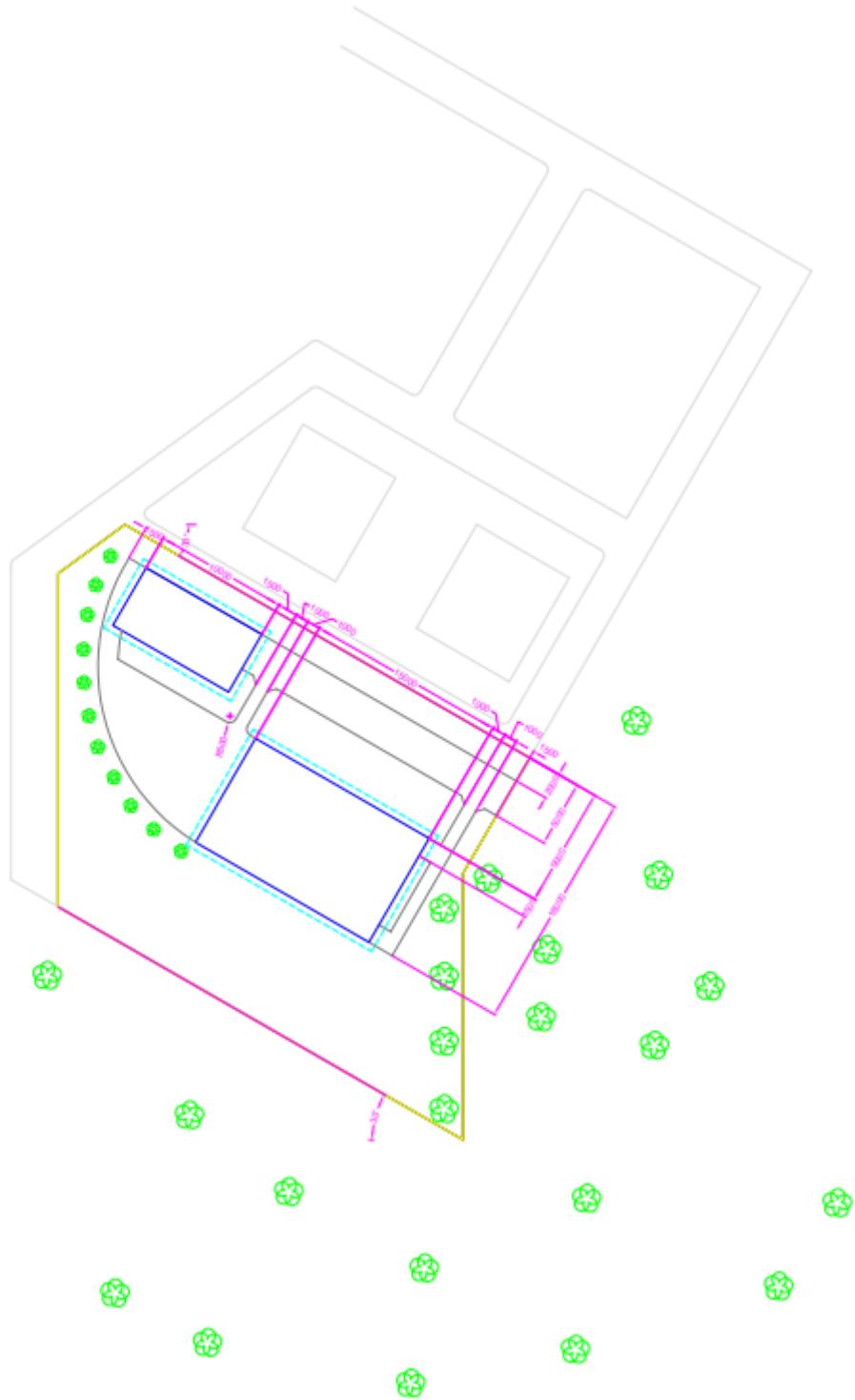


Figure 53. Finished site plan in the ModelSpace

2.11 Viewports

Drawing area has two spaces: Model Space that is used for the actual drawing or modelling and Paper Space where you create the printout sheet, layout of the sheet, and scaling to the right scale. There can be multiple layouts defined for a single drawing. Spaces can be changed from the

bottom left corner of the drawing area or from the status bar's MODEL/PAPER button. You can define a name for the layout by double clicking on it and renaming it.

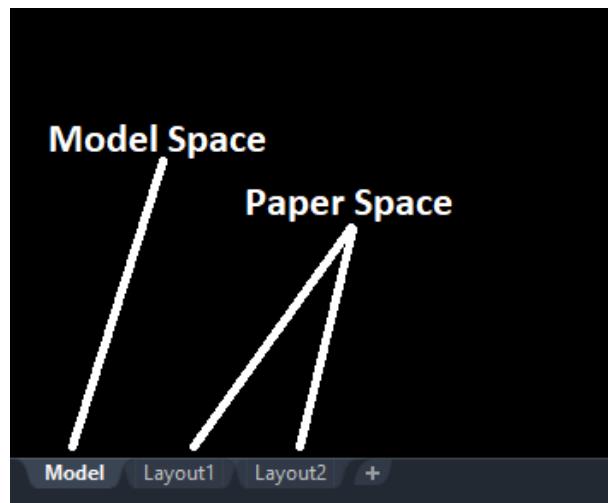


Figure 54. Spaces of the AutoCAD

Changing the settings of the Layout

To change the Paper Space settings, first activate the paper space by clicking on the *Layout1* button. After that right click on the same button and select *Page Setup Manager*. Select *Modify* in the opened window.

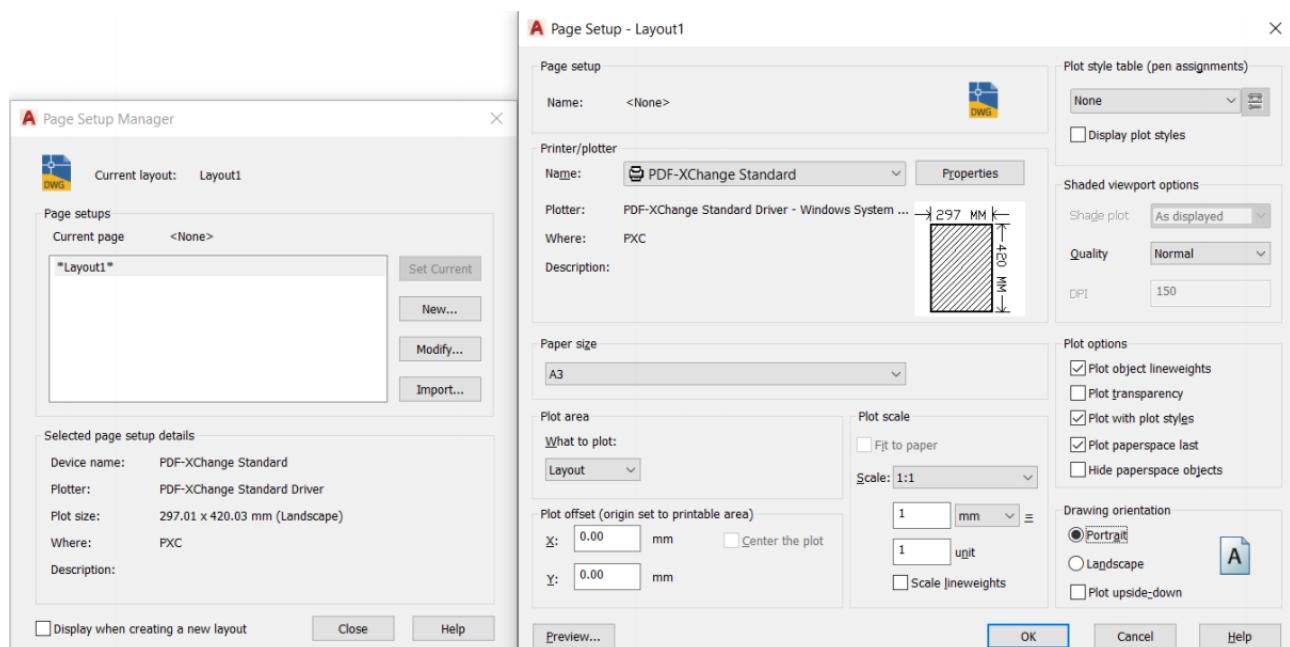


Figure 55. Page Setup Manager

Change *Plot Scale* to 1:1 in the *Page Setup* window (if it is not already). This drawing we will be using portrait A3 sheet. Select a fitting PDF printer from the *Print/Plotter* option like shown in Figure 55. On the top right corner, you can determine a plot style to be used. In exercise 1, we will not be using any plot styles, so you leave *None* as the selected option. Plot styles will be used in exercise 2.

Click OK when you are done with the *Page Setup Manager*, and AutoCAD will generate sheet of the selected size with one viewport in it. Delete the automatically generated viewport with *Erase* and draw your own view from the beginning onwards.

Create a “views” layer from which you can turn off printing by clicking on the printer image of the layer.

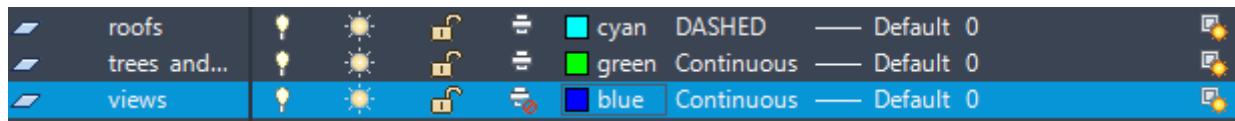


Figure 56. Settings of the views layer

New View can be made with the **MV (MVIEW)** command in Paper Space (you only need one in this exercise).

Command: mview

Specify corner of viewport or

[ON/OFF/Fit/Hideplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>:

(with mouse)

Specify opposite corner: (with mouse)

Regenerating model.

The size of the viewport can be changed by dragging from the corners.

Viewports can be of any shape. For example, the polygonal option can be used to draw polygonal viewport. To dive in the View, double click within the borders of the view. In this case, you will be working on the model space. The borders of the viewport will change to thicker when working on the model space. Drawing can be panned in the model space with the pan command or by holding down the mouse wheel. Be careful when using the mouse wheel, since scaling will change when you zoom in and out.

Next, the scale of the drawing will be changed to correct value. Site plan is shown on a scale of 1:500 in the printout. Change of scale is done in the model space (model space can be accessed by double clicking within the borders of the view) and selecting the desired scale from the status bar. There is no 1:500 as a default scale, so you have to create it by clicking on the *Custom...* and adding a new one. When the new scale is ready, it can be selected from the Scales list. See Figure 57.

When the scale is set right, center the drawing on the paper. If the drawing disappeared from your view after scaling, go inside the view by double clicking, zoom out, search the drawing, and zoom closer to the plot so that it is approximately in the center of the paper. Finally, correct the scale to 1:500 again and finetune the placement.

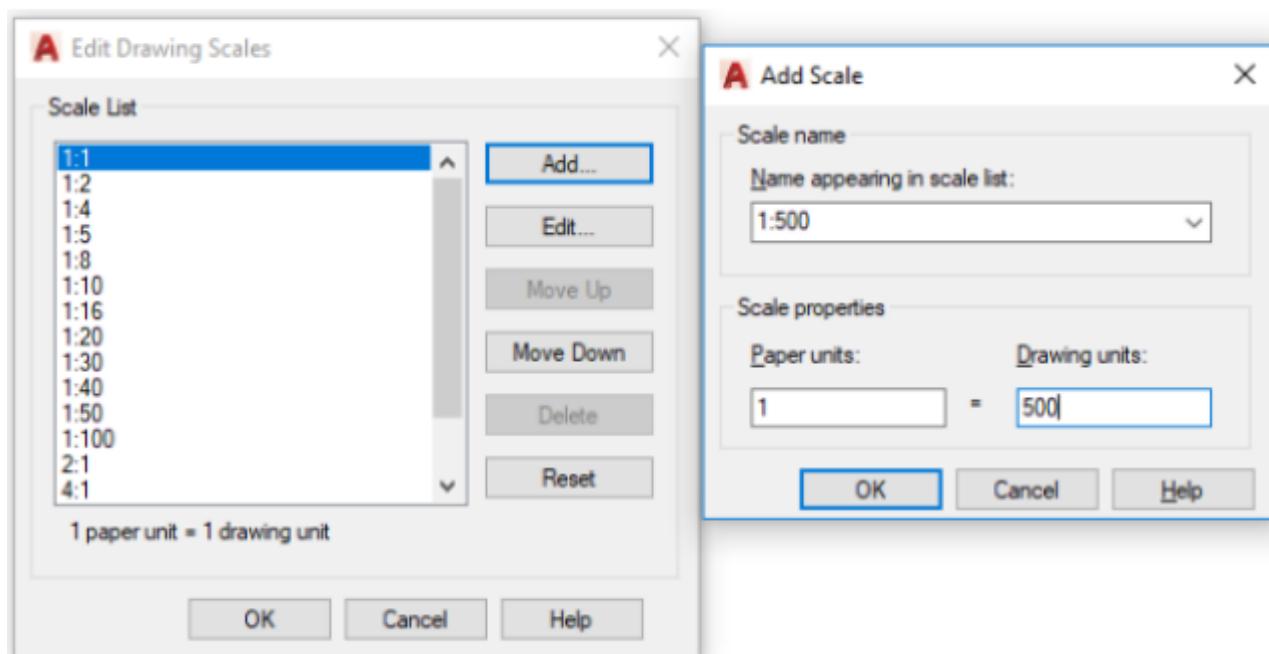
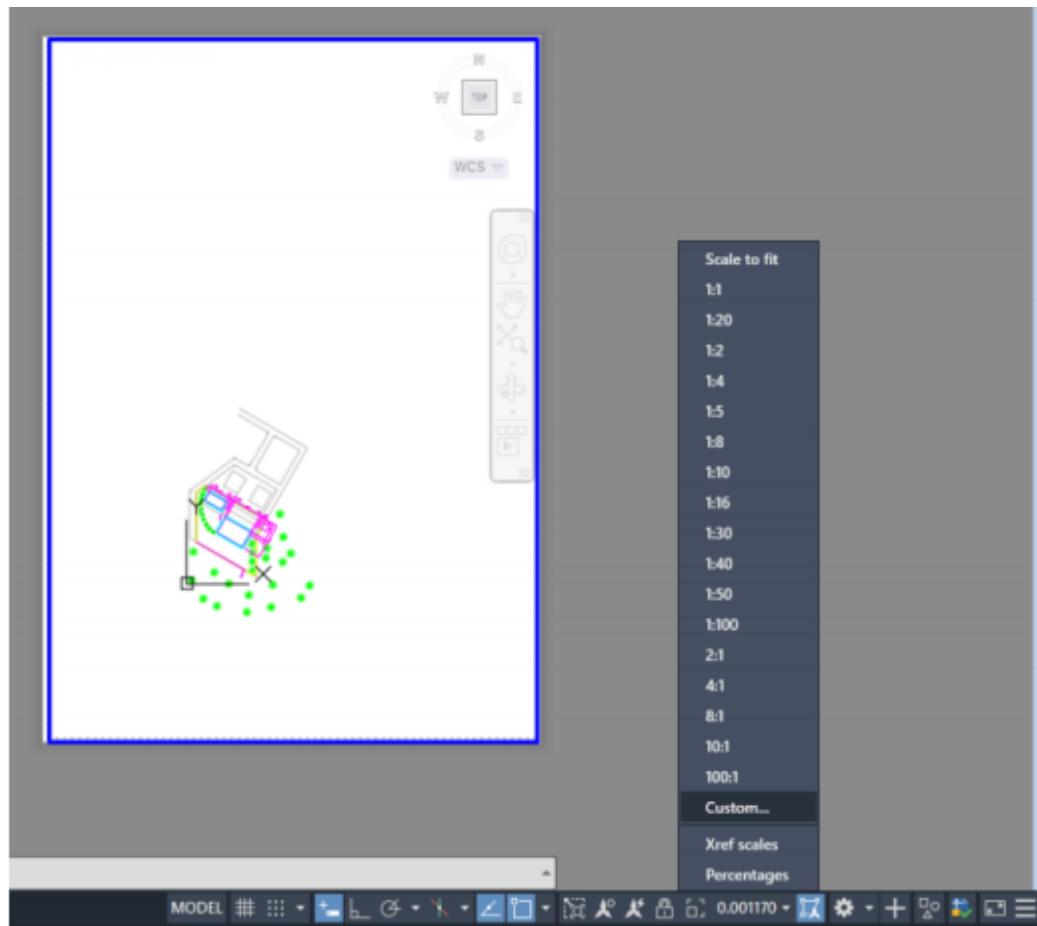


Figure 57. Setting the scale

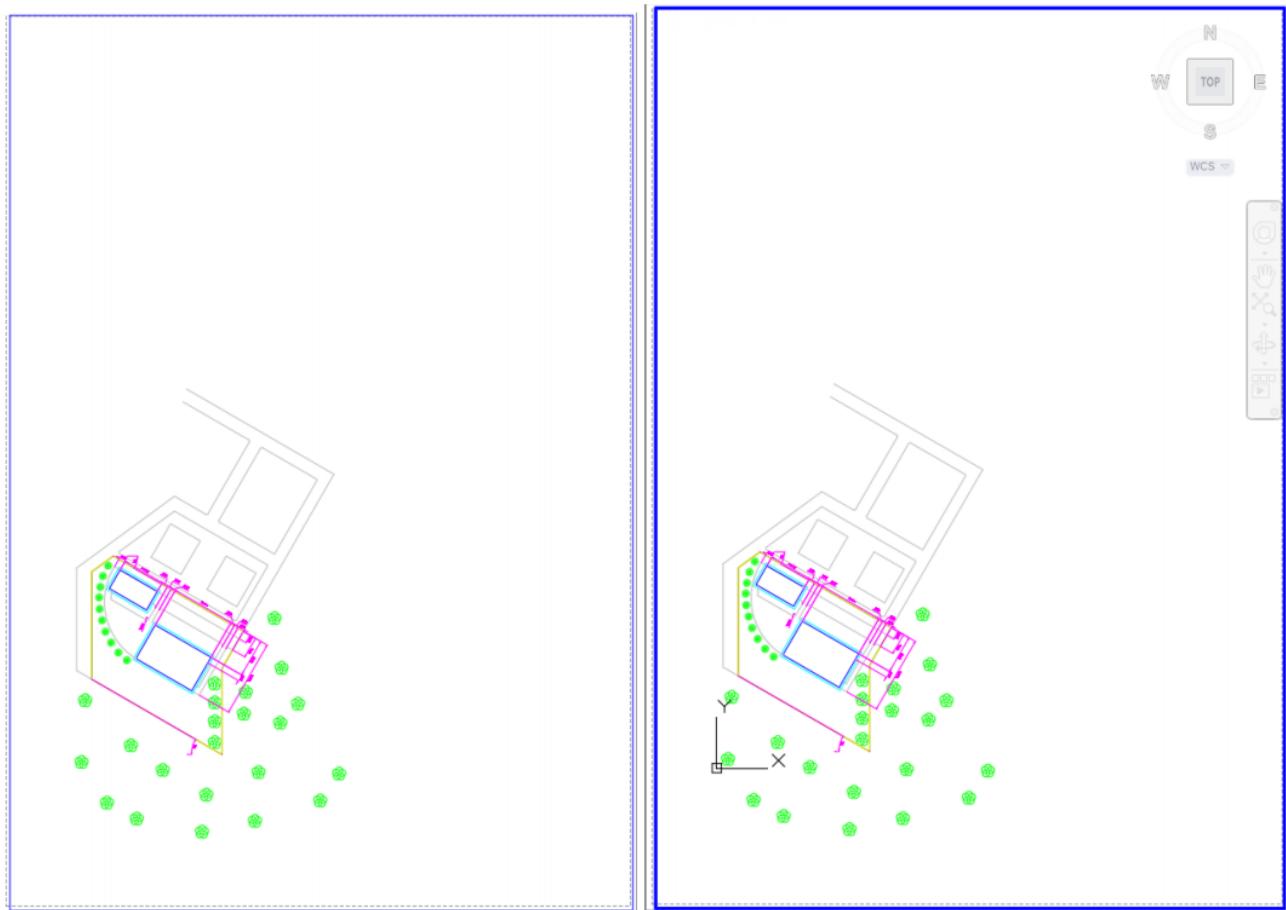


Figure 58. Paper Space on the left, Model Space on the right

2.12 Texts

Texts are used in title tables, part lists, note and help texts, and as parts of other image elements. When creating a drawing, pay attention to the scale, especially for texts. **Printing scale will affects the text height settings: text height in the drawing units = desired height in the print times scale.** The formatting and layout of the texts are defined in text styles that are inherited from the image template.

Creating a text style

Text style can be created and modified with the **ST (STYLE)** command.

Text

Text command is used to write a single line text. First, the place of the text is indicated, then height and angle are given, and finally the text itself. A little example follows after this. **Note that this part is not done in the exercise.**

Command: TEXT

Current text style: "Standard" Text height: 2.5000

Annotative: No Justify: Left

Specify start point of text or [Justify/Style]:

Specify height <2.5000>:

Specify rotation angle of text <0>:

(write text, line break with enter)

(end writing by pressing esc)

Texts can be edited with the **DDEDIT (TEXTEDIT)** command or with the Properties window.

Mtext

Command: **MText** (Draw – Text – Multilinetext)

MT (MTEXT), or so-called paragraph text, is meant for writing multi-line texts. Two points indicates the width of the paragraph, the height is automatically adjusted. If width is specified to 0 with the *Width* option, the row will continue until enter is pressed. After determining the paragraph's size, a text editor opens from which you can define e.g. font height, font type, italics, bold, underline, fractional format, color, and special characters with the *Symbols* option. In the editor bar, you can find *Find/Replace* function to find words and replace them in the paragraph text.

Texts will be written on the “texts” layer. Make sure that printing of the layer in question is on.

Write your own name and student number with the MTEXT command on paper space. Use 5 units for text height.

Command: MT

MTEXT

Current text style: "Standard" Text height: 2.5000

Annotative: No

Specify first corner: (left click on the starting point)

Specify opposite corner or [Height/Justify/Line

spacing/Rotation/Style/Width/Columns]: h (change the text height)

Specify height <2.5000>: 5

Specify opposite corner or [Height/Justify/Line

spacing/Rotation/Style/Width/Columns]: (click the second point of the text box and write the text. End writing by clicking outside of the text box)

In general, you should avoid placing texts on top of viewports!

2.13 Printing

Drawings can either be printed in model space or paper space. Printing is done in paper space, because images of different scales are often desired on the same paper. In paper space, scale is set to 1:1, because the views are already scaled to the right scale. Printing in model space is done via **File – Plot...** or with **PLOT** or **PRINT** commands. In the printing window, you must specify e.g. the scale and printable area. To see the preview of the drawing before printing, press *Preview* button. To exit the preview, press **Esc**. Result should look pretty similar to Figure 60.

AutoCAD may ask if you want to plot all of the print layouts while printing. We only want to print the created layout so select *Continue to plot a single sheet* option.

If all of the lines in your drawing are continuous in the paper space and the actual print, use the **LT (LINETYPE MANAGER)** command, click *Show Details* on the Linetype Manager window (Figure 59) and unselect the *Use Paper Space Units for Scaling* option. After this use the **REGENALL** command to update your drawing. Now the lines should appear similar to those in the model space and you can print a new PDF file.

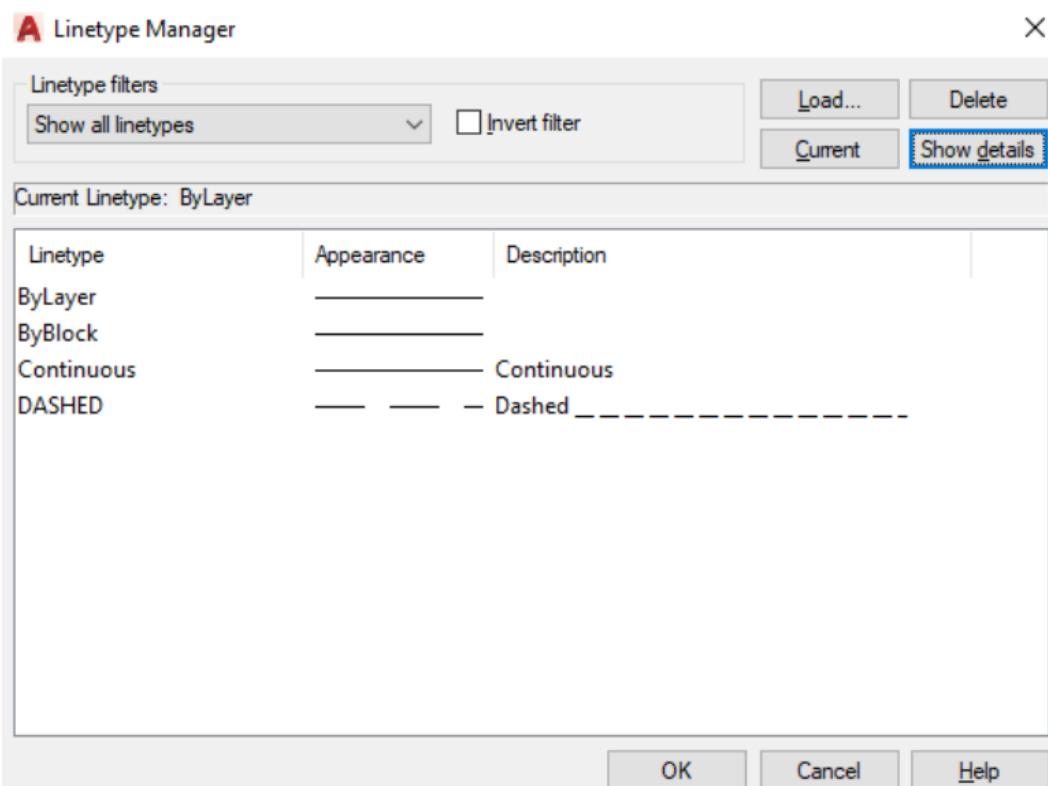


Figure 59. Linetype Manager dialog box

Teemu Teekkari
12345A



Figure 60. Finished site plan

Save your drawing as (your student number)_1.dwg and name the printout accordingly (your student number)_1.pdf