OPERATION MANUAL FOR
CADMAX– SOFTWARE
FOR STRUCTURAL ANALYSIS, DESIGN AND REINFROCEMENT DETAILING OF R.C.C.FRAMED STRUCTURES.

DEVELOPED BY
M.K.SUGUMARAN,B.E.(Hons),M.E, P
NIROOBINI TECHNOLOGIES
https://nirts.in
e-mail id: contact@nirts.com

INTRODUCTION

I would like to introduce myself as the developer of this software and as well as a practicing structural engineer for past 17 years. I developed this software in AutoCAD environment since, the most of the civil engineers and Architects are already exposed to it and is being adopted as the world leading design and drawing software. I started developing the this software from the year 1983, when I completed my graduation in civil engineering and my exposure to the AutoCAD was from 1987 when I did my post graduation at Coimbatore Institute of Technology, Coimbatore. Then I developed small software components in AutoCAD such as design of slabs, beams, columns and footings etc. with regular update as per my requirements. I never expected that adding all components with an effective interaction with AutoCAD will become this wonderful **"CADMAX SOFTWARE"** a boon for the practicing structural and civil engineers.

Like other available structural analysis and design software, it contains no loose talks to show it high or less. It is not developed for International standards and is only for Indian standards as per IS-456-2000 and design aids. The wind load and earthquake load analysis are as per corresponding IS-code provisions. But the method and the speed of approach to get structural design and detailed drawings will be seen unimaginable while you are using this software when compared with other structural design software. It is not a large software to occupy a large portion of your hard disk and it will occupy nearly 10MB of space of your hard disk. But it will do the great job for practicing structural engineers which has been failed by other structural software. The terminology used in this software is so simple that any civil engineer/ Architect can easily understand and operate.

This software is made into run with a pop-up menu in AutoCAD environment with the following main components:

1.Introduction of members such as columns, beams, slabs, walls and facia as drawing components in AutoCAD.

2.Calculation of beamloads from drawing entities automatically.

3.Introduction of wind load and earthquake components as drawing entities.

4.Analysis of the structure for dead loads, live loads, wind loads and earthquake loads as per requirements.

5.Design of members such as slabs, beams, columns and footings for critical forces of load combinations.

6.Reinforcement detailing of slabs, staircase, beams, columns and footings according to required type of gridlines with automatic grouping of members.

7.Preparation of detailed Design procedure for slabs, beams, columns and footings.

8.Requirement of steel quantities for the structure stage by stage levels as required.

9.The calculation of all civil work quantites is under development and will be appended to this software very soon which will be very useful for preparation of estimates.

I hope you will appreciate it, while using this software.

Please do not hesitate to write your comments to me regarding upgradation as per your requirement.

With best wishes,

M.K.SUGUMARAN
NIROOBINI TECHNLOOGIES

CHAPTER.1.INSTALLING THE SOFTWARE.

Installation of **AutoCAD 2000** or higher version software had to be preloaded in your computer for loading this **CADMAX SOFTWARE** and follow the steps listed below:

STEP.1.

Unzip the CADMAX SOFTWARE downloaded.

.**STEP.2.**

Using windows explorer copy all the unzipped files in a folder **CADMAX.**

STEP.3.

Now Select all the files in the folder using windows explorer and check for properties of files and if they are read-only files uncheck read only option.

STEP.4.

Then close the windows explorer.

STEP.5.

Right-click on the existing **AutoCAD** icon and then click on the "**create shortcut**" option to create a new **AutoCAD** icon.

STEP.6.

Right-click on the new **AutoCAD** icon and then click on the "**Rename**" option to rename the icon as **CADMAX** icon.

STEP.7.

Right-click on the new **CADMAX** icon and then click on the "**Properties**" option to set the new properties. In the **properties** dialog box change the **Start in** folder as "**C:\CADMAX**" that is where the software is copied and then **apply** and **close** the properties dialog box.

STEP.8.

Double click on the new **CADMAX** icon to start **AutoCAD** .

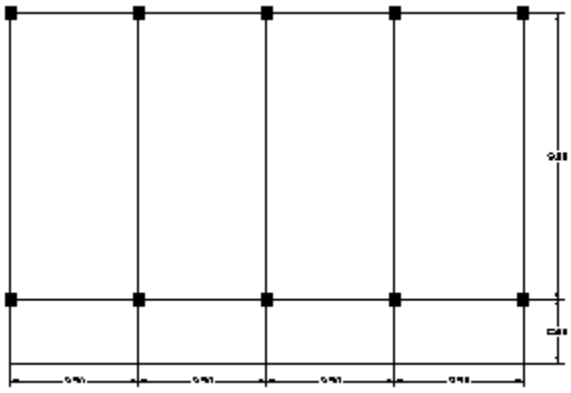
STEP.9.

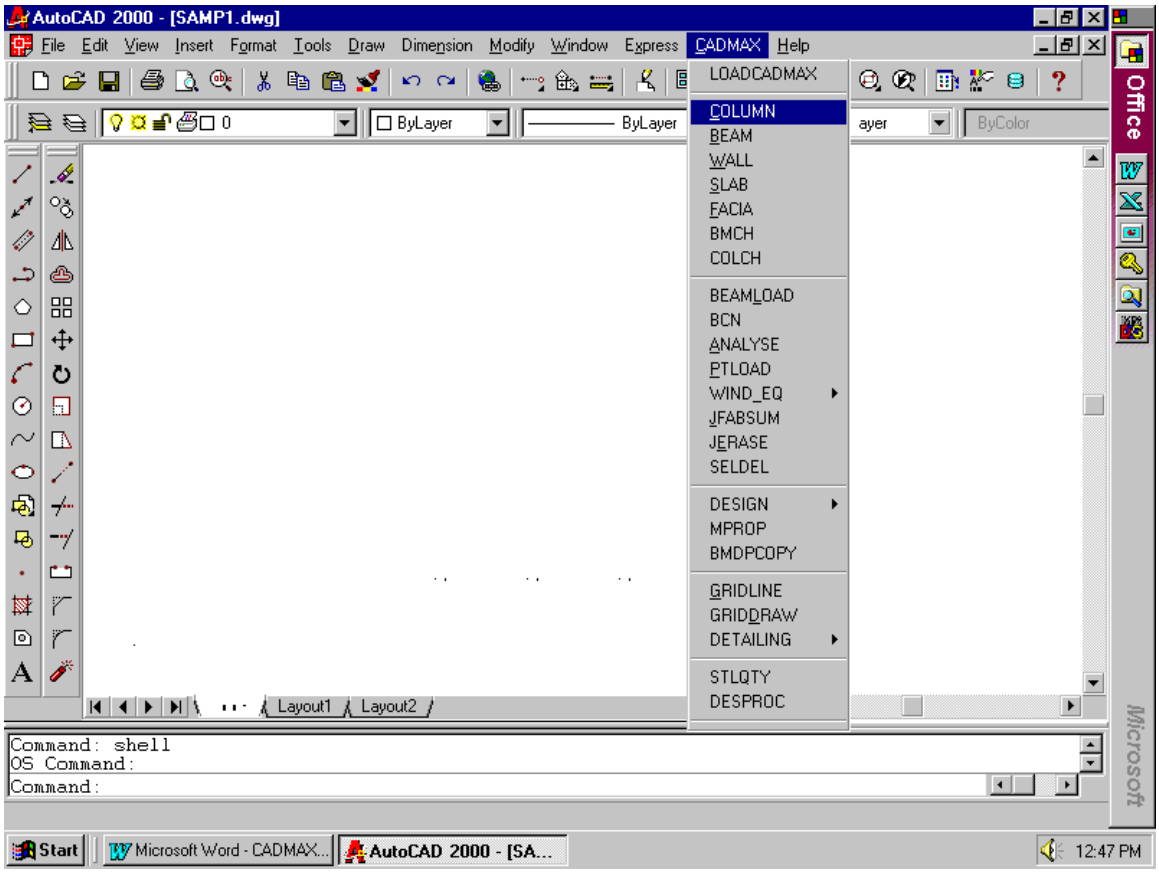
Click the **Tools** menu and then click **Customise menus** to load the **CADMAX.MNC** file in the **C:\CADMAX** folder and insert the **CADMAX** pop-up menu before **Help** menu. For AutoCAD 2007 or higher version **CADMAX.CUIX** file.

Now the **CADMAX** software is loaded and ready to run through pop-up menu or equivalent commands at the command prompt.

CHAPTER.2.INTRODUCING MEMBERS.

Let us consider a simple structure with two rows of columns with a span of 9m along Y-direction and 3.5m c/c along X-direction with four bays and cantilever projection of 2m along Y-direction in one side as shown below in the layer 0.(default).



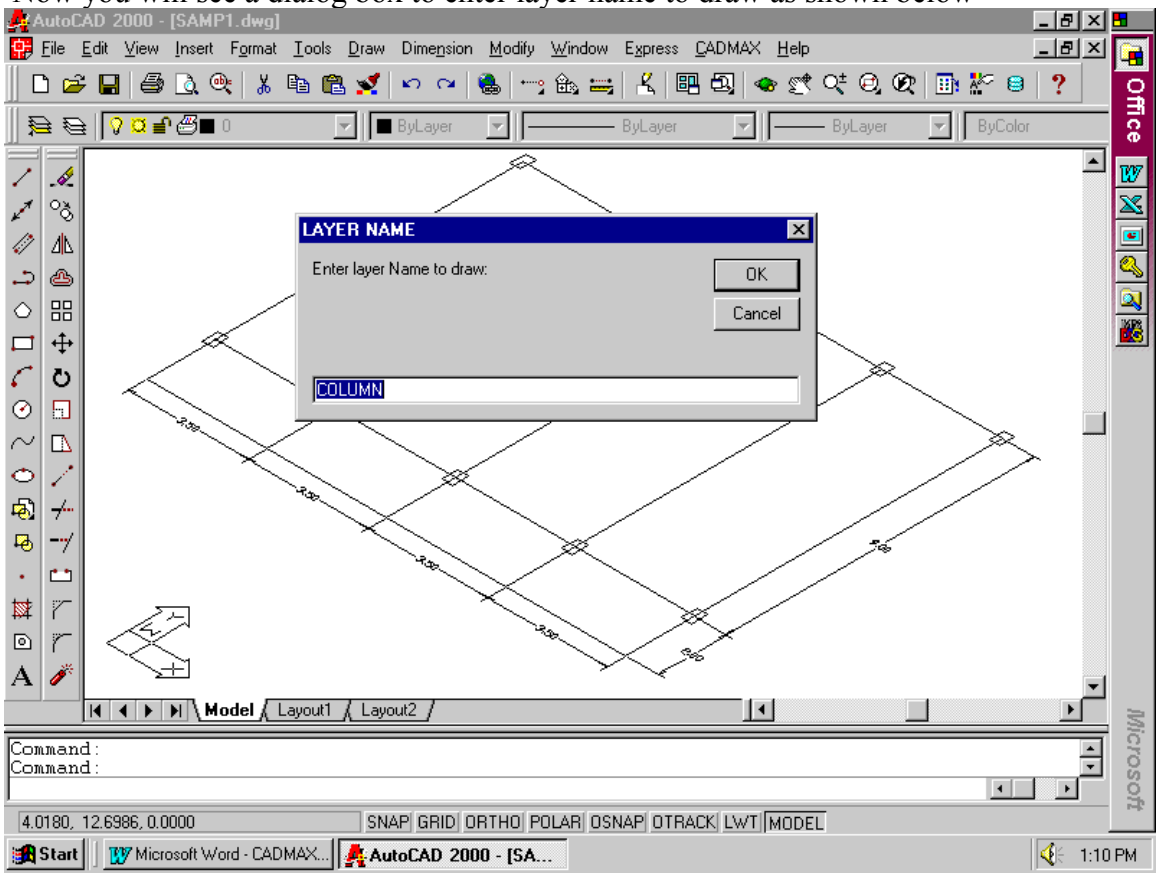

AutoCAD 2000 - [SAMP1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Window Express CADMAX Help
LOADCADMAX
COLUMN
BEAM
WALL
SLAB
FACIA
BMCH
COLCH
BEAMLOAD
BCN
ANALYSE
PTLOAD
WIND_EQ
JFABSUM
JERASE
SELDEL
DESIGN
MPROP
BMDPCOPY
GRIDLINE
GRIDDRAW
DETAILING
STLQTY
DESPROC
ByLayer
ByColor
Layout1
Layout2
Command: shell
OS Command:
Command:
Start
Microsoft Word - CADMAX...
AutoCAD 2000 - [SA...
12:47 PM
Office
Microsoft

1.Pick the "**LOADCADMAX**" command in the **CADMAX** Pop-up menu.
Or
Give **(LOAD"CADMAX1")** in **Command** prompt of **AutoCAD.**
(Now you will see the message "**CADMAX 2000 SOFTWARE LOADED!!nil**" at the **Command** prompt of **AutoCAD.**)

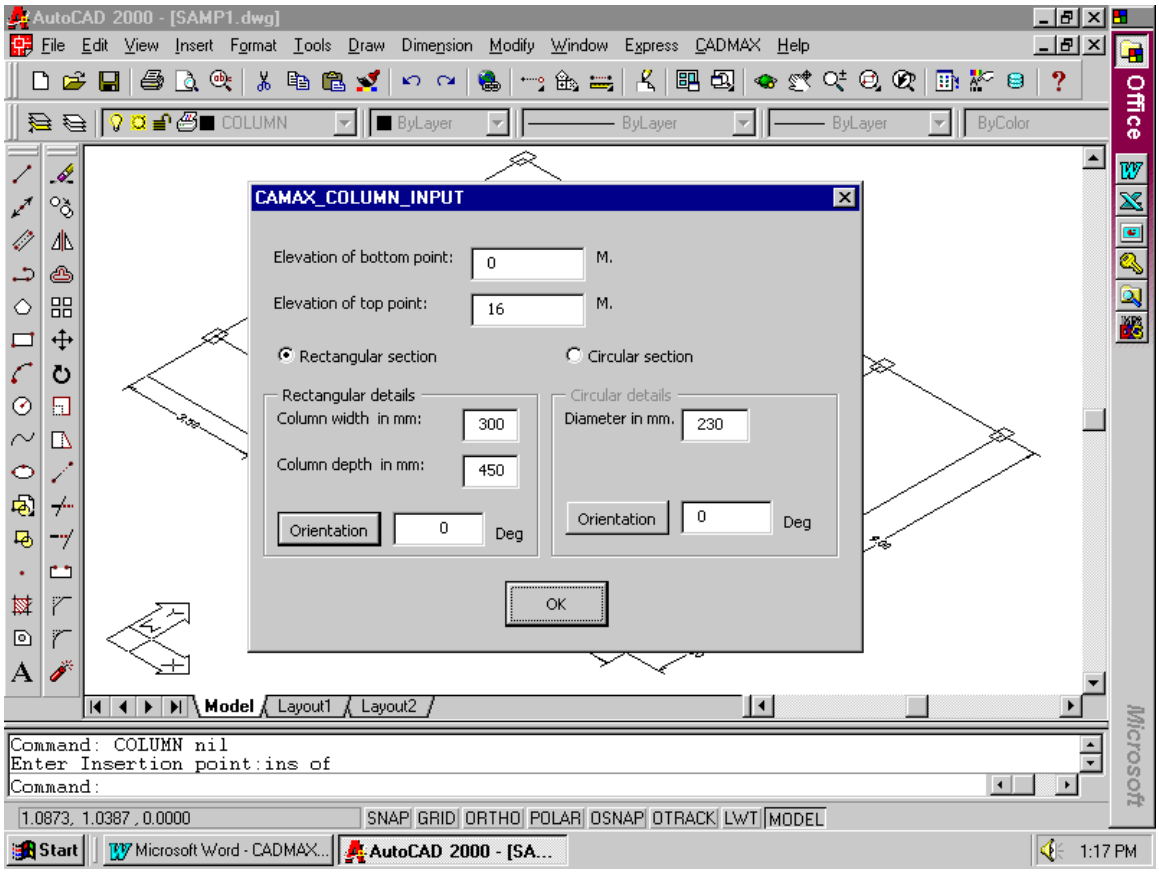
:
2. Give **VPOINT** command to get a convenient **3dview** to insert columns.
Command:VPOINT
<0.0000,0.000,1.000>:**1,-1,1** <enter>

Pick the **"COLUMN"** command in the **CADMAX** Pop-up menu.
Or
Give **COLUMN** in Command prompt of **AutoCAD.**
This command is used to introduce columns in the required locations.

Now you will see a dialog box to enter layer name to draw as shown below

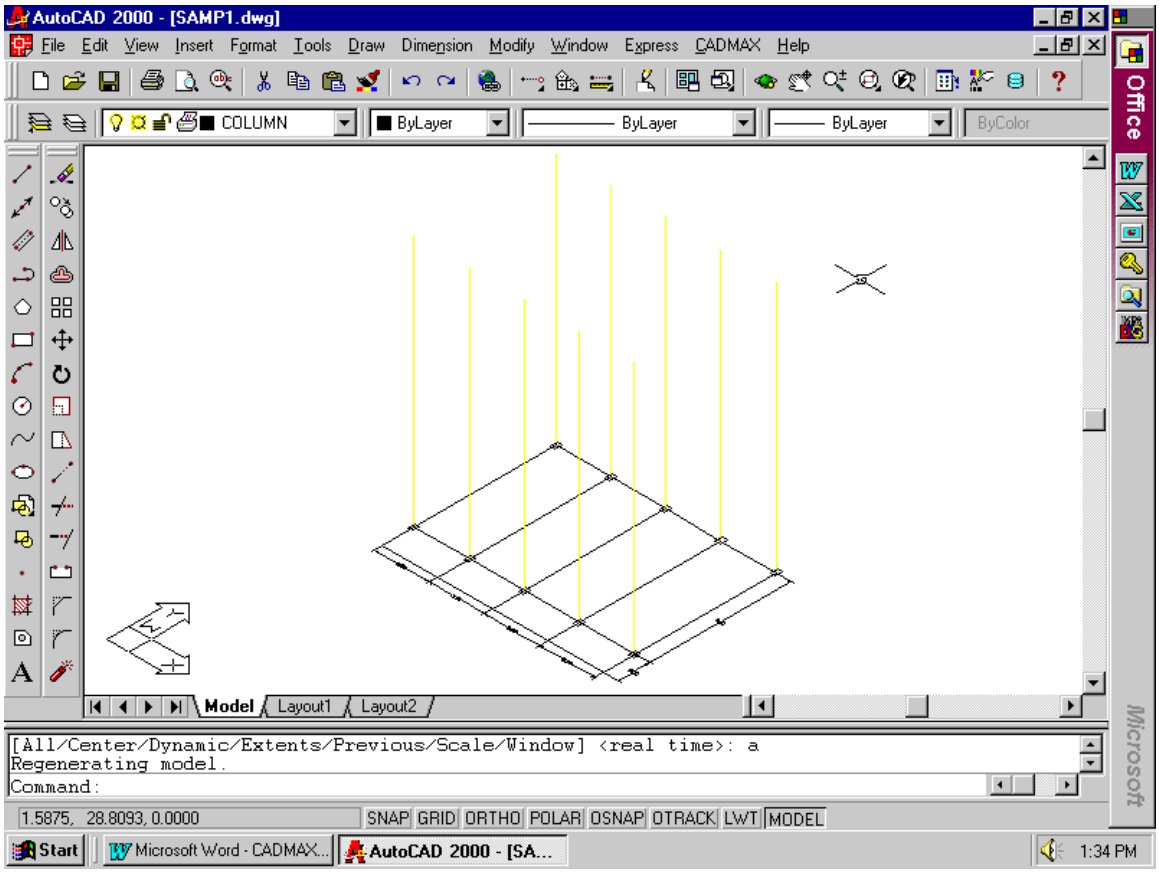


Fill in the input box for the layer name to draw as **COLUMN** to insert columns.
Command:COLUMN
Enter insertion point: ins of <pick the point>
Now you will see the following dialog box to fill the details of columns:



After filling in the details in the above dialog box click OK button.
Now you will see a column of required height at the insertion point in yellow color.
In this example the footing bottom elevation is 0 which is the elevation of bottom point given in dialog box and elevation of top point is given as 16 by assuming the structure as Ground + 3 floors with depth of footing below ground level as 1.5m and basement height above groung level as 0.5m and height of each floor as 3.5m to get a total of 16m(1.5+0.5+4*3.5=16m).

Now you can copy or array the drawn column to the appropriate locations.
Command:**ARRAY** <enter>
Select objects:**L**
Select objects: <enter>
Type of array<Rectangular/Polar>:<R>:**R** <enter>
Number of rows:**2** <enter>
Number of columns: **5** <enter>
Distance between rows: **9** <enter>
Distance between columns: **3.5** <enter>
Now you will see all the columns drawn in the structure as shown below:



After inserting columns now you can insert grade beams at required locations.

3.Pick the **"BEAM"** command in the **CADMAX** Pop-up menu.

Or

Give **BEAM** in Command prompt of **AutoCAD.**

This command is used to introduce beams in the required locations.

Now you will see a dialog box to enter layer name to draw as shown below

Enter layer name to draw as GBEAM in the input box and then click OK.

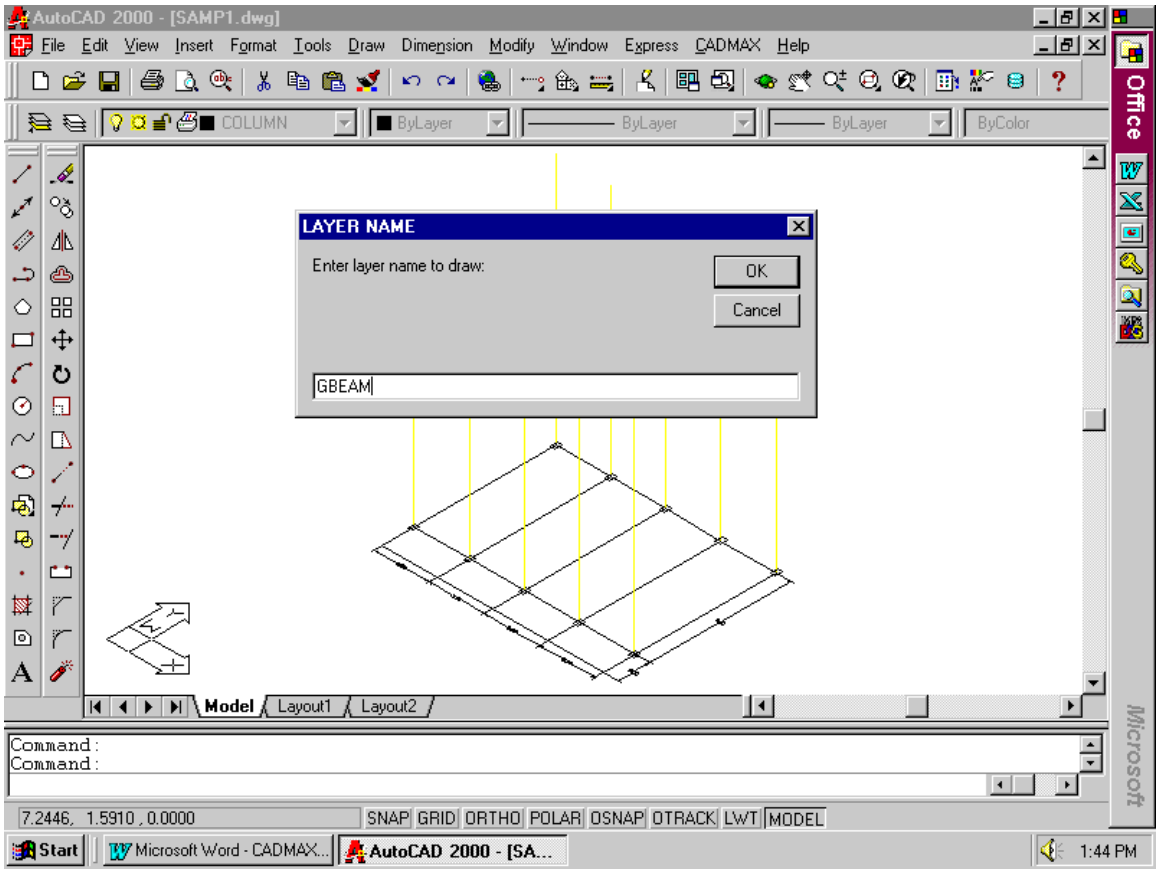
Command:BEAM

Enter start point:<pick the start point of beam>

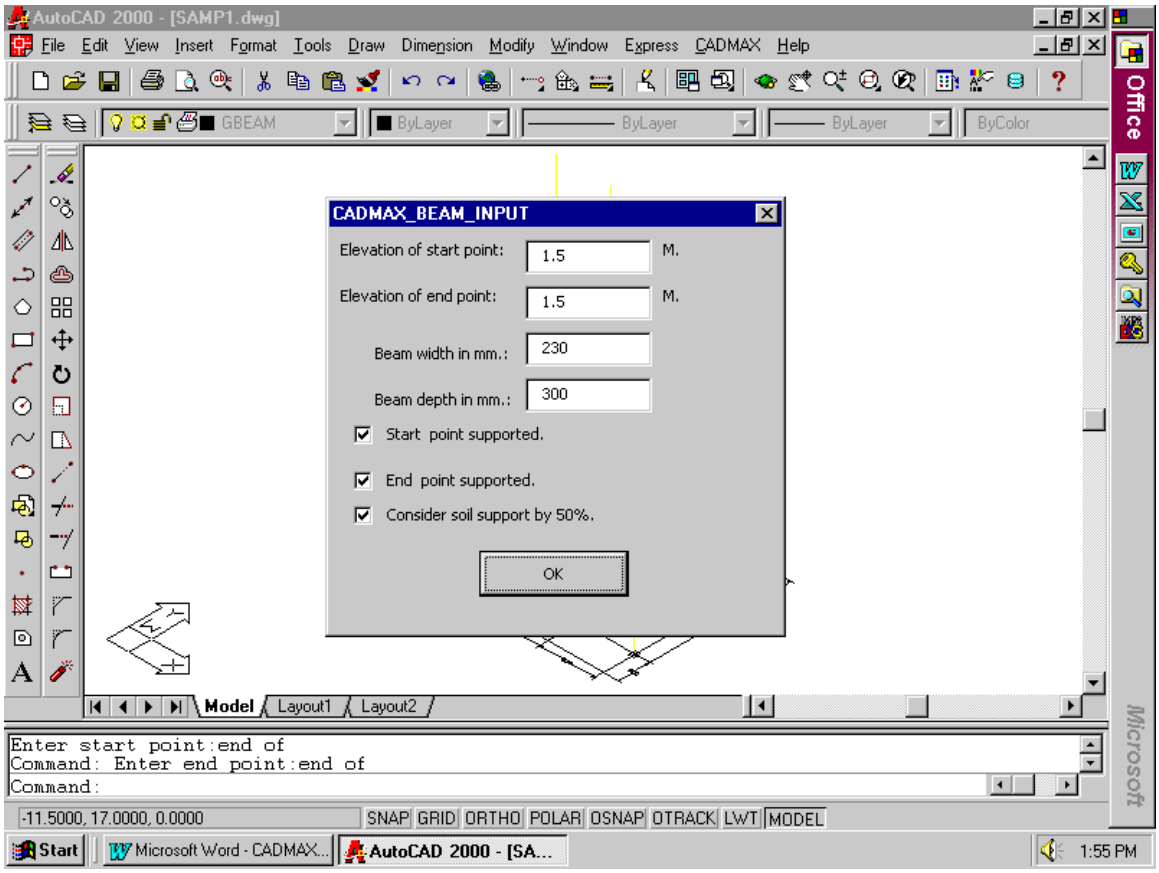
Enter end point: <pick the end point of beam>

(While picking the points do not worry about the elevation which will be given in the dialog box later)

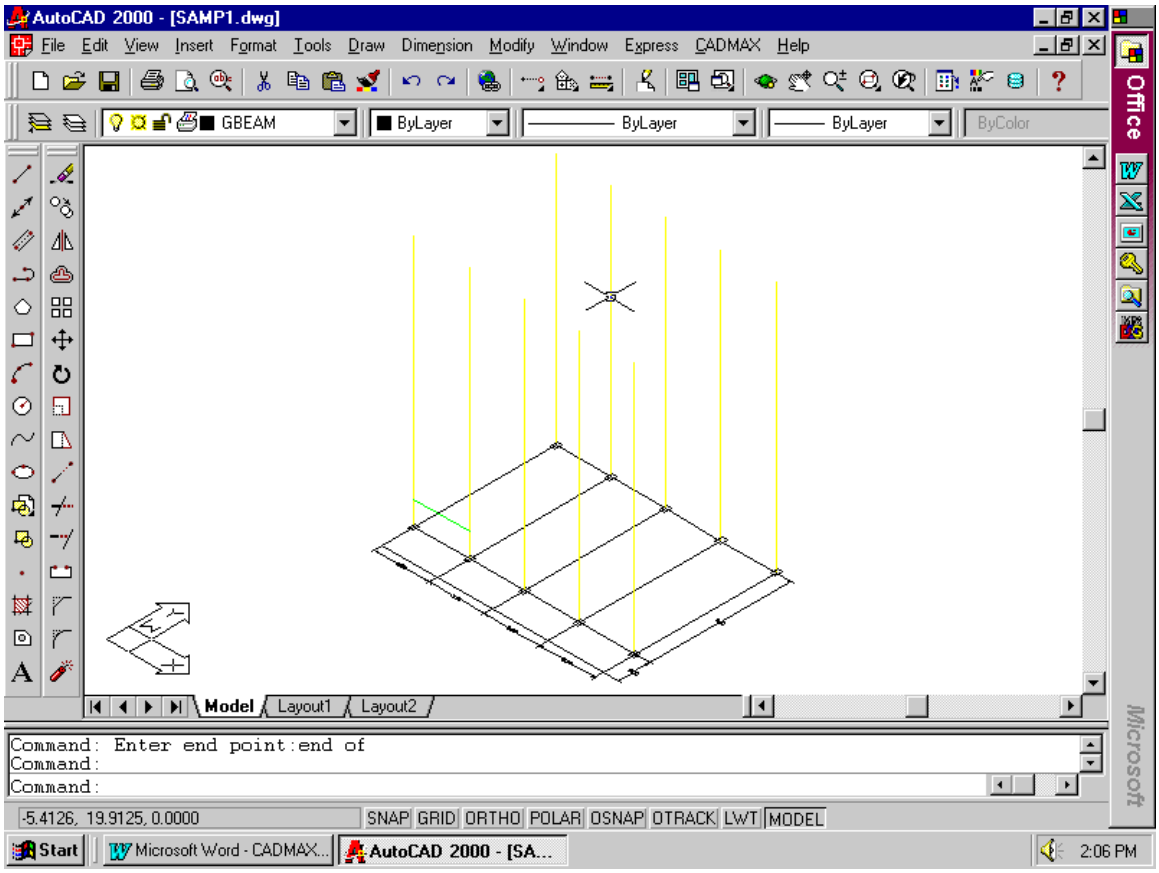
Now you will see the next dialog box to fill the required details as shown below:



Beam details dialog box shown below:
Fill the required details of beam **Elevation of start point**, **Elevation of end point**, **width of beam**, **depth of beam** and **click** the check boxes of **Start point supported**, **End point supported** and **Consider soil support by 50%** as per the requirement. Then click OK . Now you will see a beam in green color is drawn in the drawing as shown in the figure.



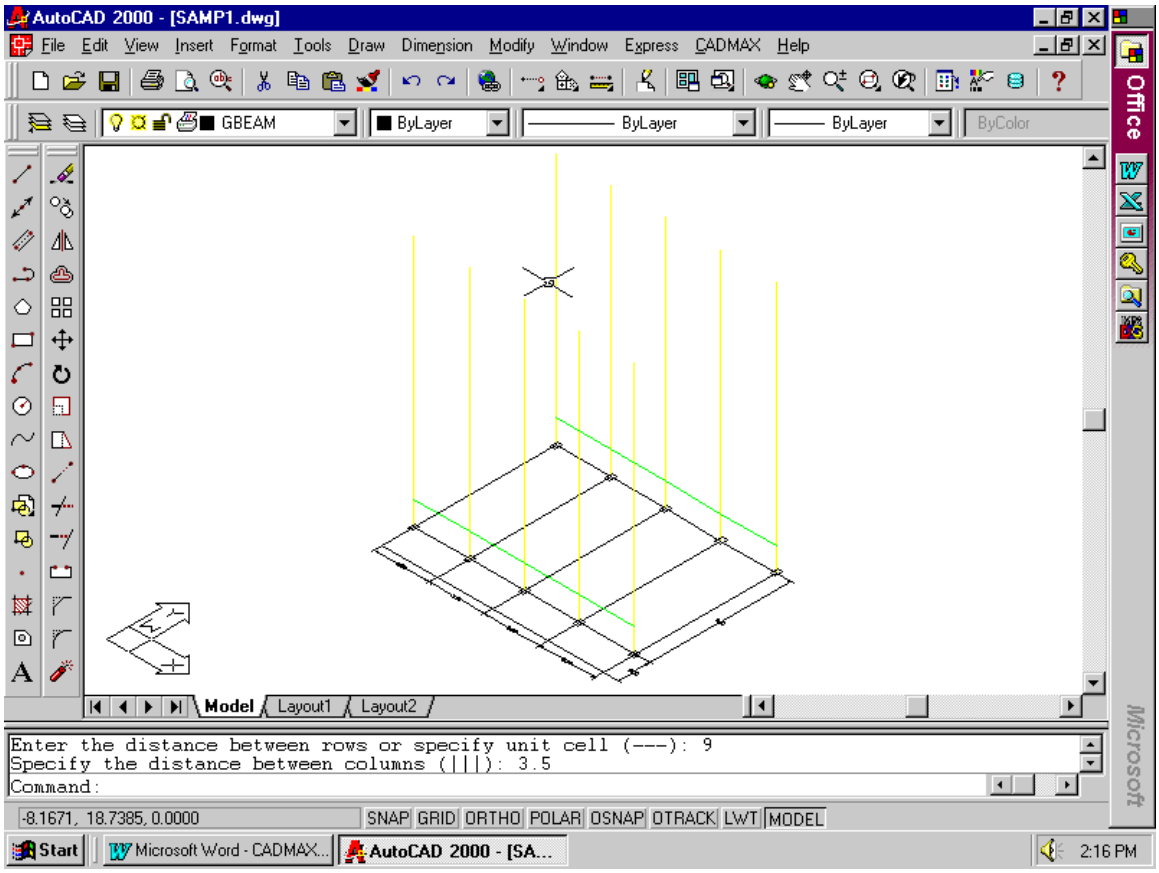
Drawing showing the first drawn grade beam at 1.5m elevation is given below:



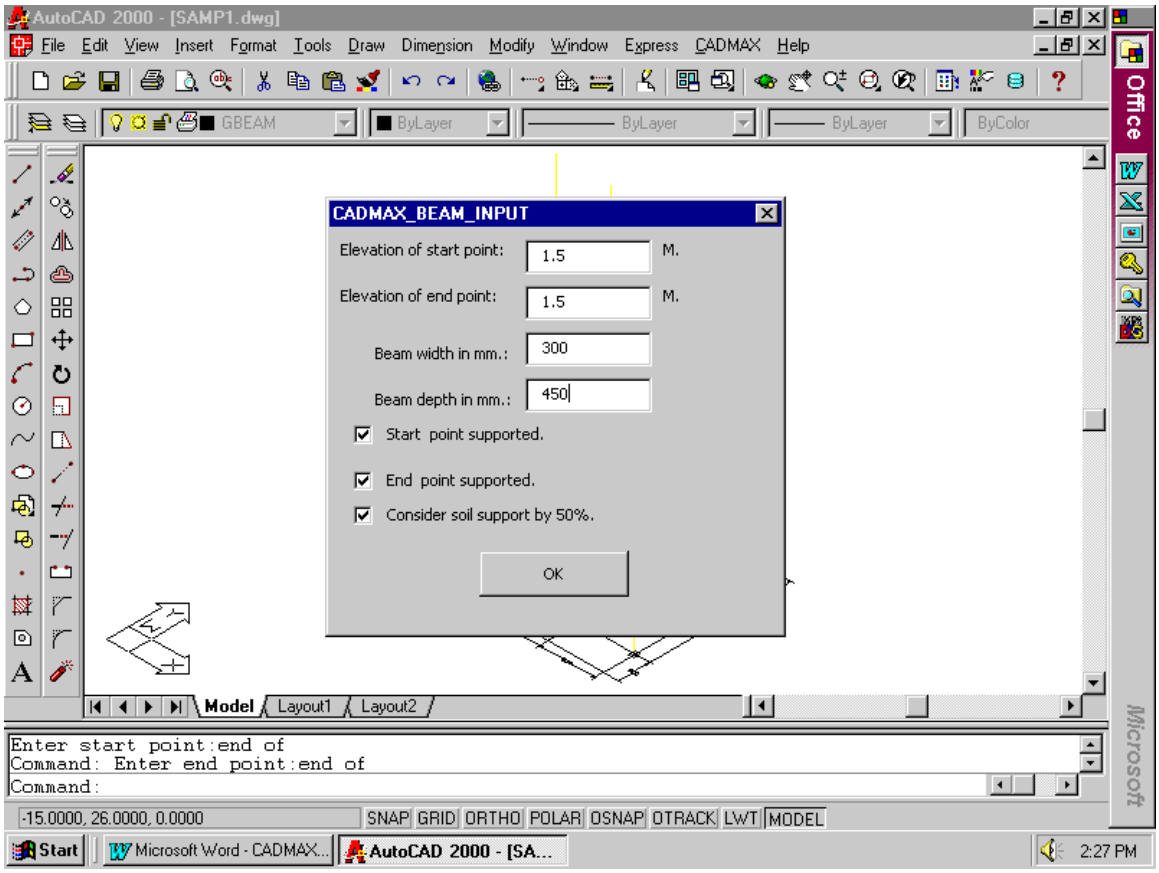
After drawing the first grade beam to adopt similar beams at other places you can copy or array the last drawn beam as given below:
Command:**ARRAY** <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array<Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **2** <enter>
Number of columns: **4** <enter>
Distance between rows: **9** <enter>
Distance between columns: **3.5** <enter>

Now you will see all the similar grade beams are drawn in the drawing will be as shown below:

Similarly you can draw the grade beam at 1.5m elevation in Y-direction by repeating the BEAM command as shown below in the same layer as GBEAM.



Command: BEAM
(fill in the layer name dialog box as “GBEAM”)
Enter start point:<pick the start point>
Enter end point: <pick the end point>
Now fill in the beam details dialog box as shown below and then click OK.
Now you will see a beam along Y-direction as shown in the next drawing.

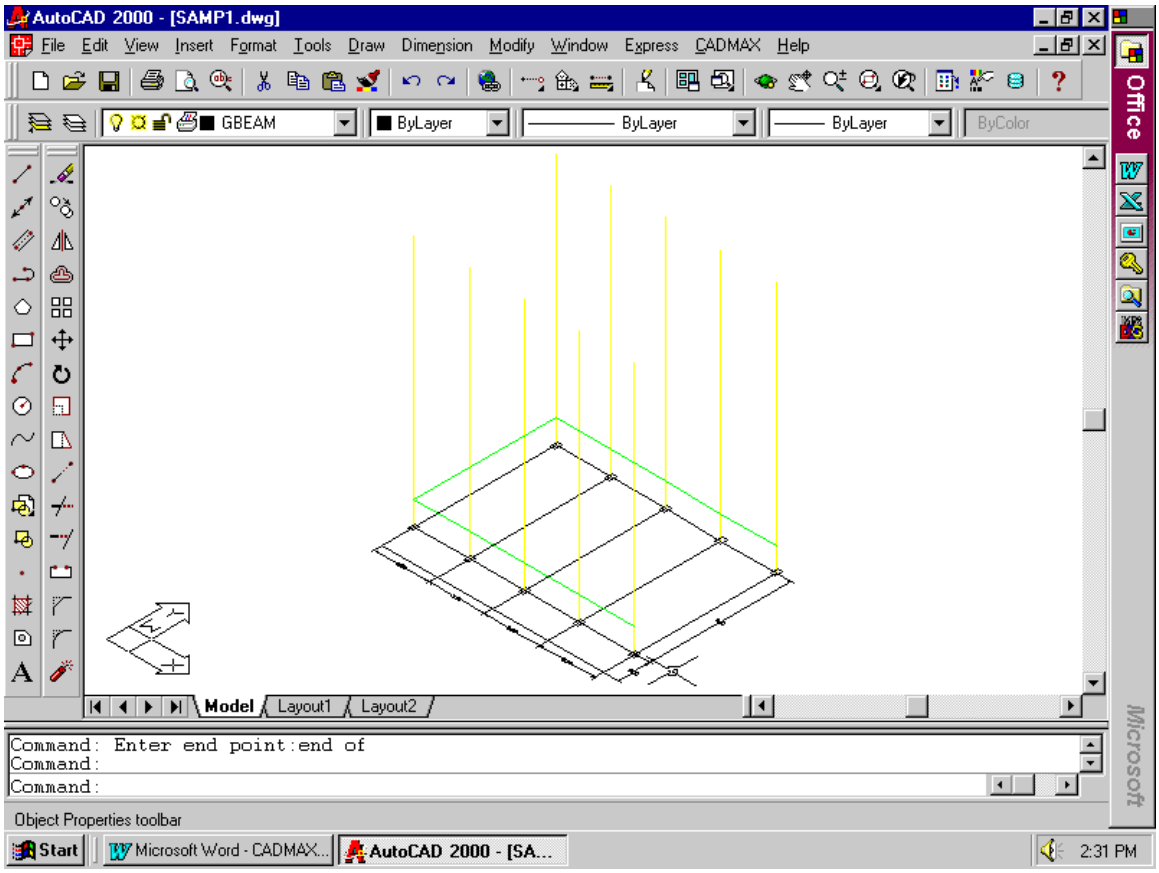


Now you will see the Y-direction grade beam as shown below:

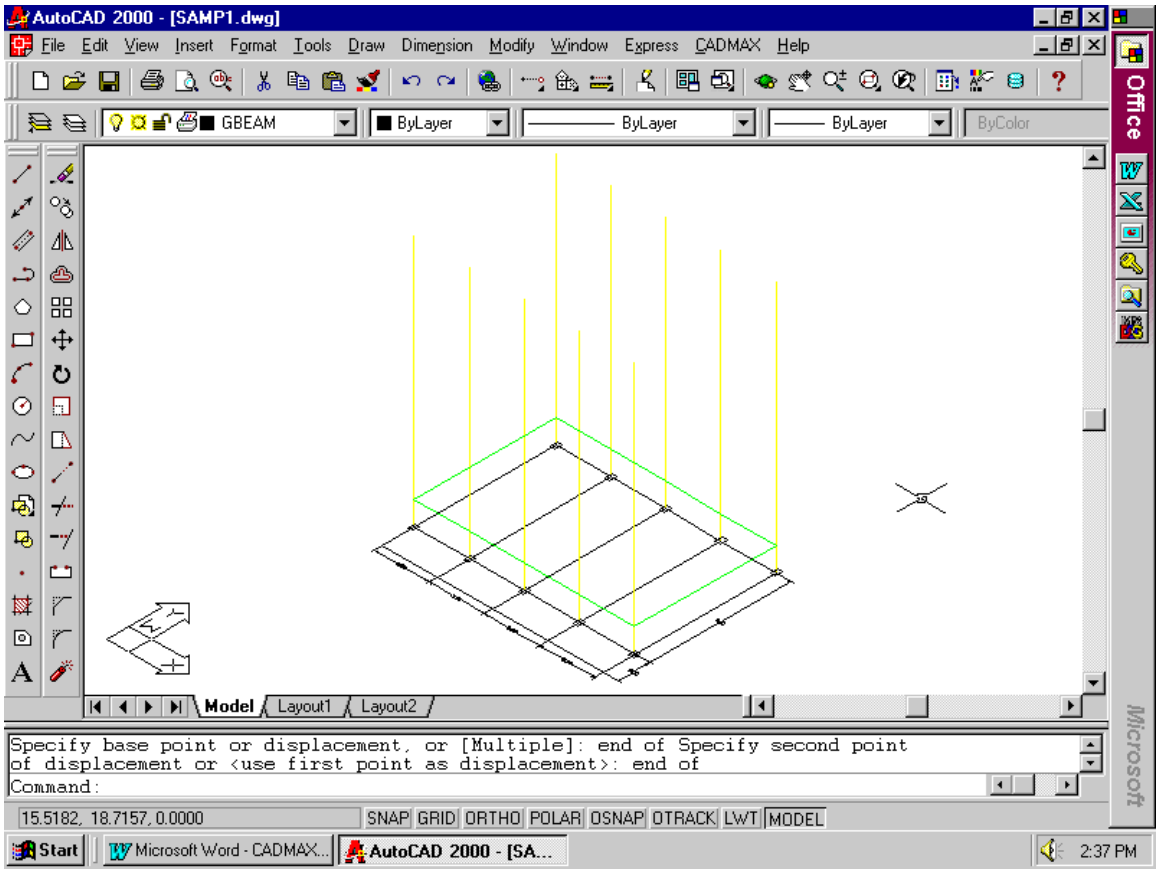
The last drawn grade beam can be copied to the other end using copy command:

Command: COPY <enter>
Select objects: L
Select objects: <enter>
Enter base point or [Multiple]:<pick the base point>
Enter displacement or next point to be copied: <pick the target point>

Now you will see the grade beam is copied to the other end location as shown in the next drawing.



Drawing showing the grade beam copied to the new location at the other end is shown below:



Now to introduce walls supported on the grade beam use the wall command as follows:

4.Pick the **"WALL"** command in the **CADMAX** Pop-up menu.

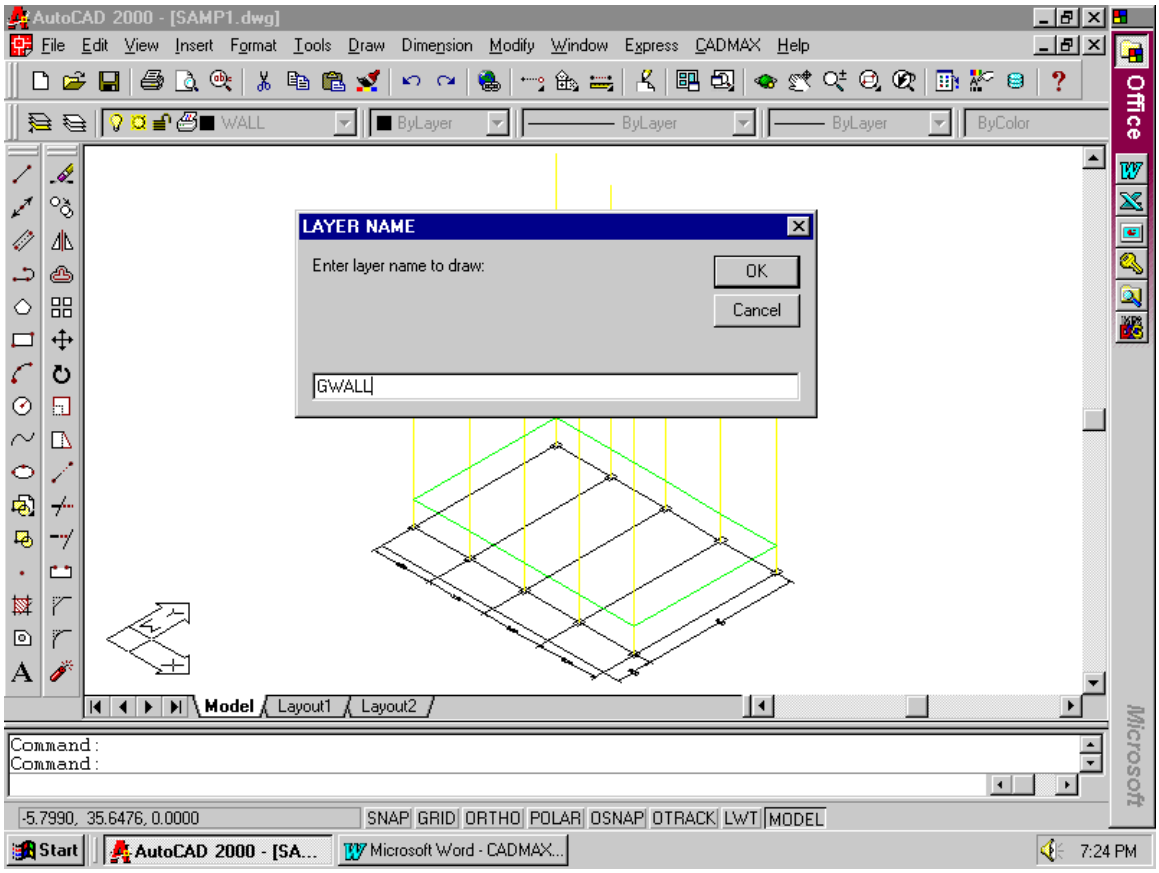
Or

Give **WALL** in Command prompt of **AutoCAD.**

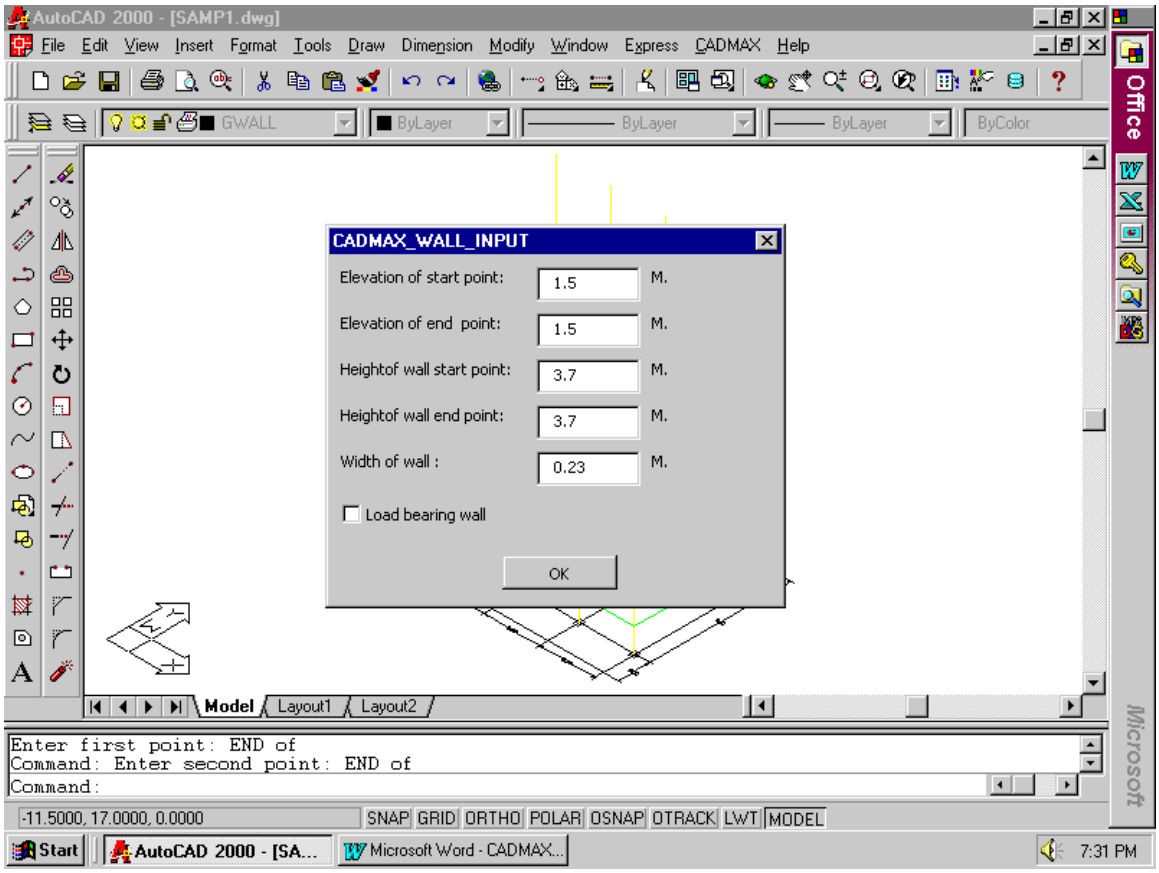
This command is used to introduce walls in the required locations.

Now you will see a dialog box to enter layer name to draw as shown below:

Enter layer name to draw as GWALL in the input box and then click OK.

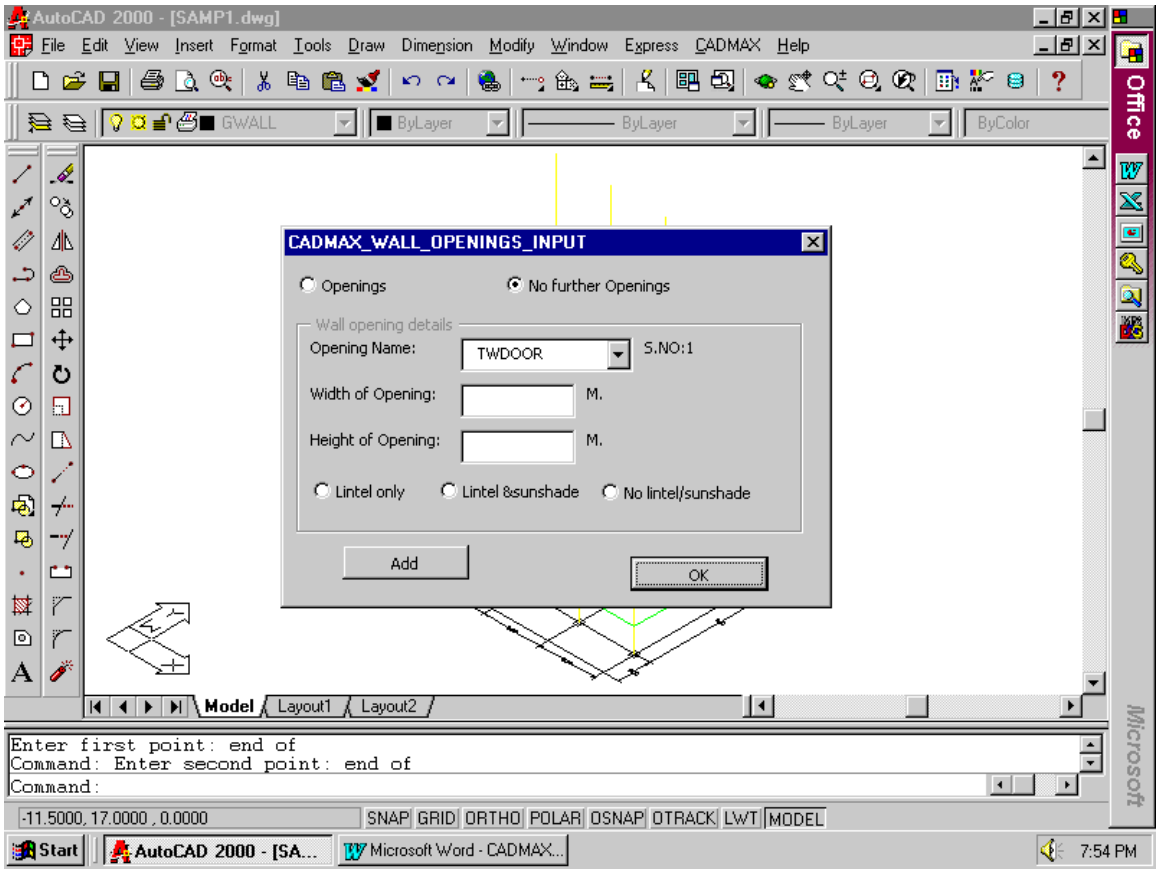


Command:**WALL**
Enter first point: <pick the start point of wall>
Enter second point: <pick the end point of wall>
(Do not worry about the elevation of points which will be filled in the next dialog box)
Now you will see the following dialog box to fill in the details of walls.
In the dialog box the **Elevation of start point:, Elevation of end point:** are filled in as **1.5m** since the walls are supported on the grade beam at the elevation of 1.5m. The **Height of wall at start point:, Height of wall at start point:** is filled in as **3.7m** since the height of basement is 0.5m and height of floor is 1.5m and after deducting 0.3m depth of beam at floor level (0.5+1.5-0.3=3.7m). The **Width of wall:** is filled in as **0.23m**. **Do not click** the **check box** for **Load bearing wall** since it is supported by a framed structure. Then **click OK**.

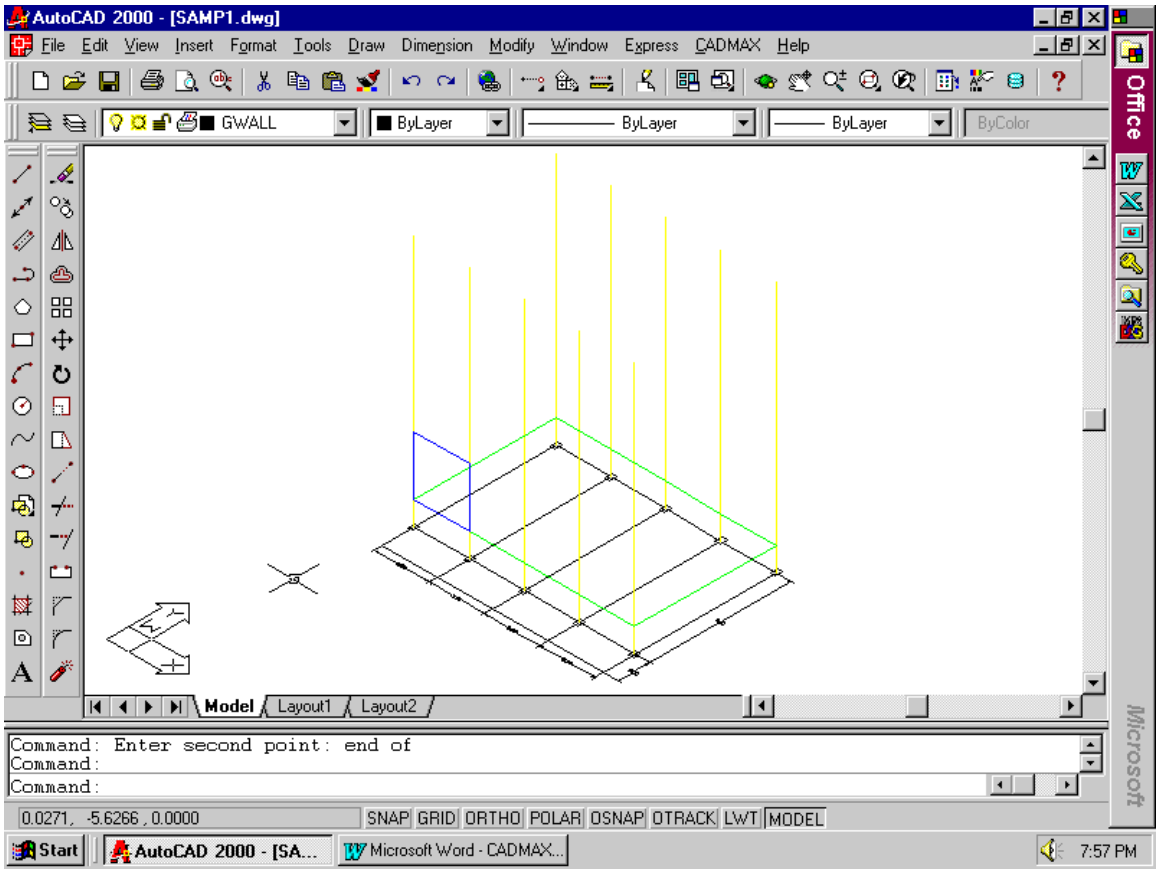


Then you will be shown a dialog box to fill in the opening details in the wall as shown below:

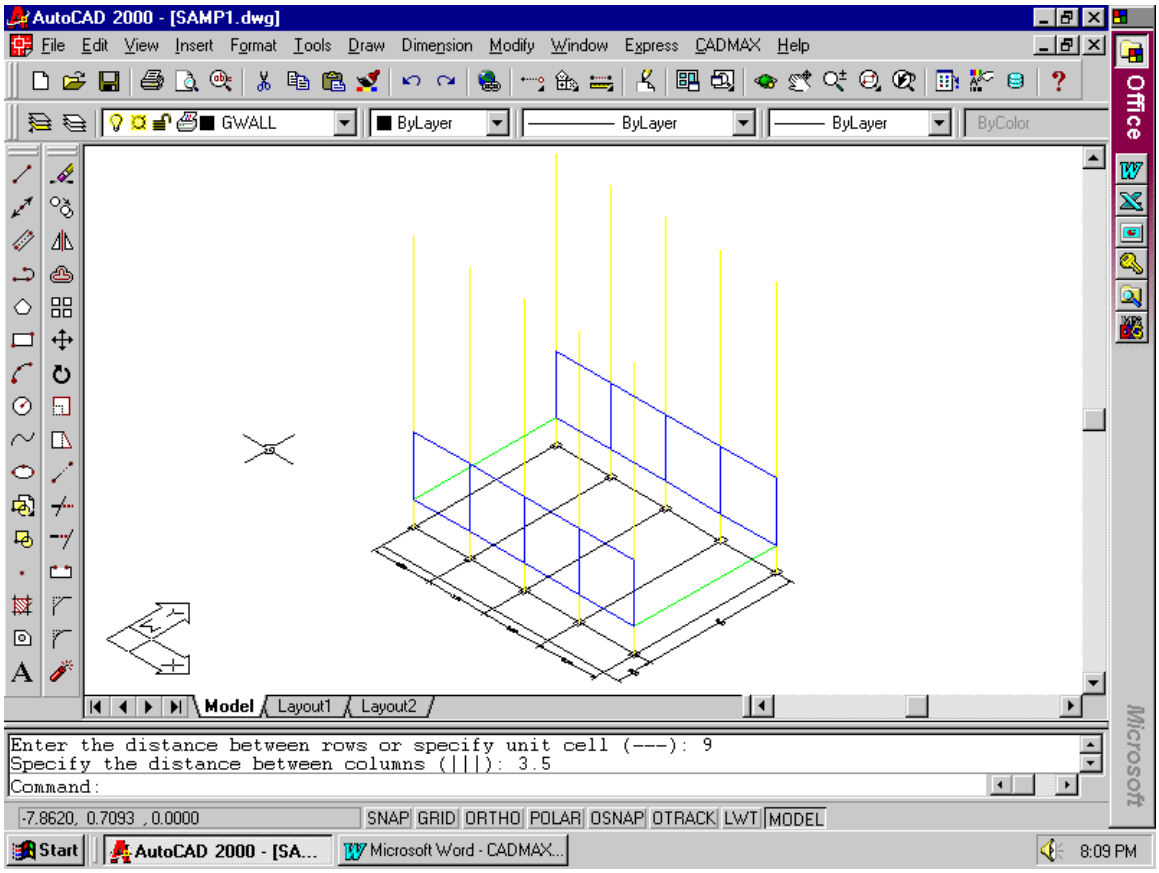
In the dialog box **click No further openings option** since the openings are generally omitted for the structural designs purpose. The provision for filling up of opening details is provided for the estimate purpose which is not dealt with presently. Then **click OK**.



Now you will see a wall in blue color introduced in appropriate location on the grade beam as shown in the drawing below:
The walls may be copied to similar locations using array or copy commands.
Command: ARRAY <enter>
Select objects: L <enter>
Select objects: <enter>
Type of array<Rectangular/Polar>:<R>: R <enter>
Number of rows: 2 <enter>
Number of columns: 4 <enter>
Distance between rows: 9 <enter>
Distance between columns: 3.5 <enter>
Now you will see that all walls along X-direction are drawn as shown in the drawing shown.

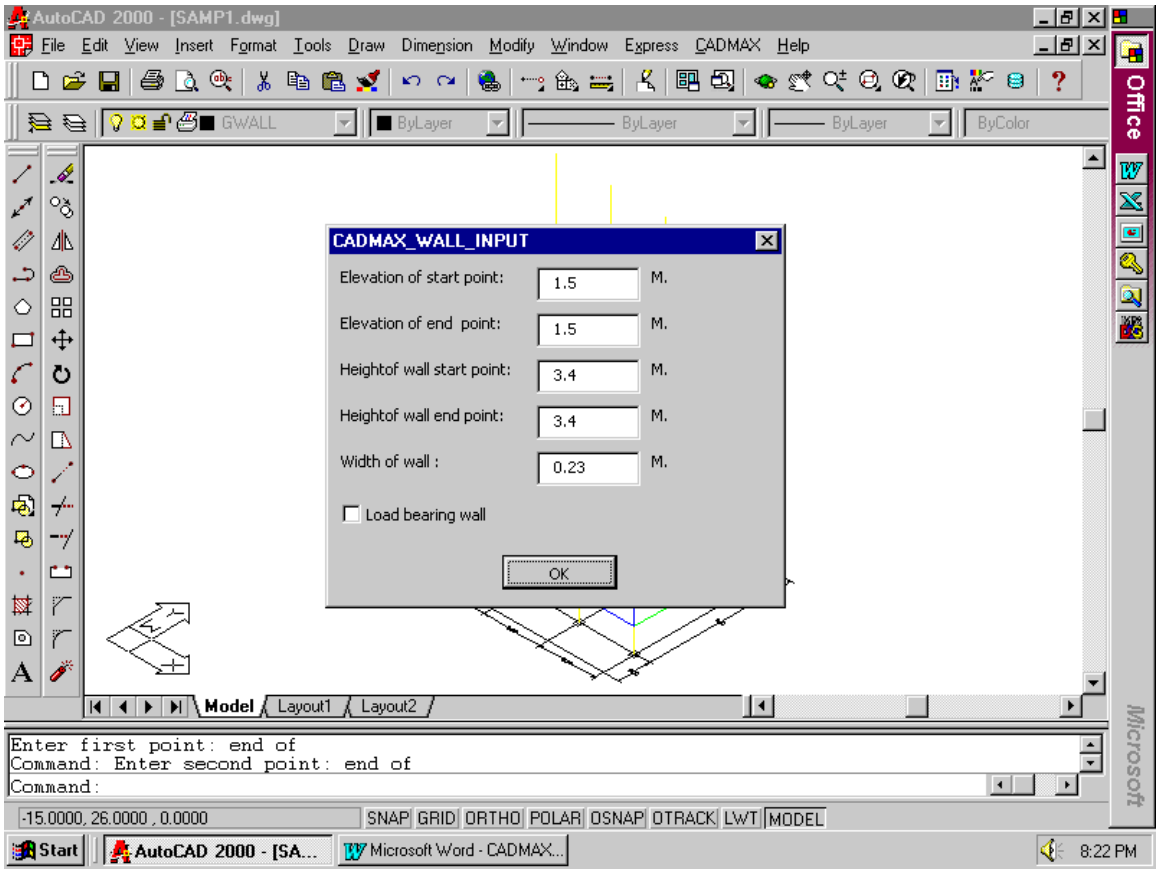


All the X-direction walls are drawn using above array command is shown below:

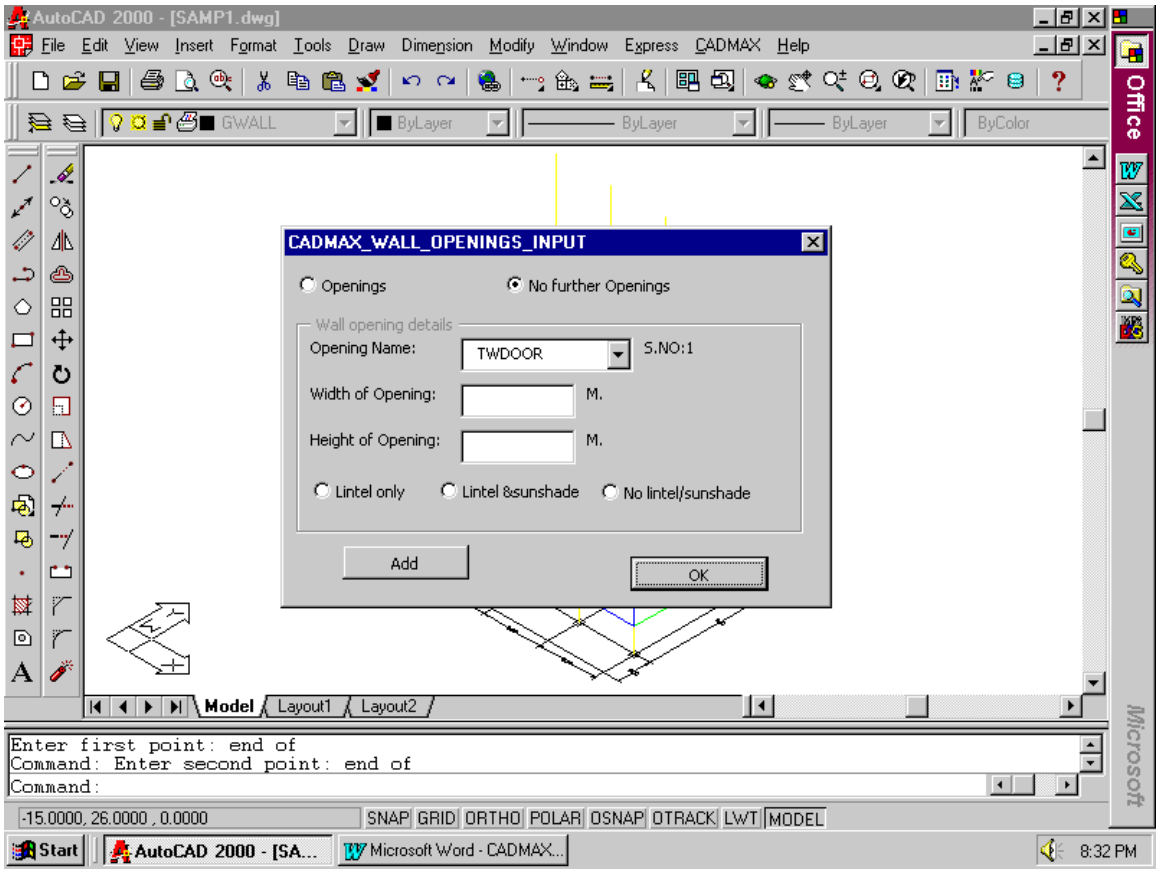


Similarly the walls along Y-direction is drawn and copied to the other end also as follows:
Give the **WALL** Command from **CADMAX** Pop-up menu or from Command prompt:
Enter the layer name to draw as **GWALL** as given previously.

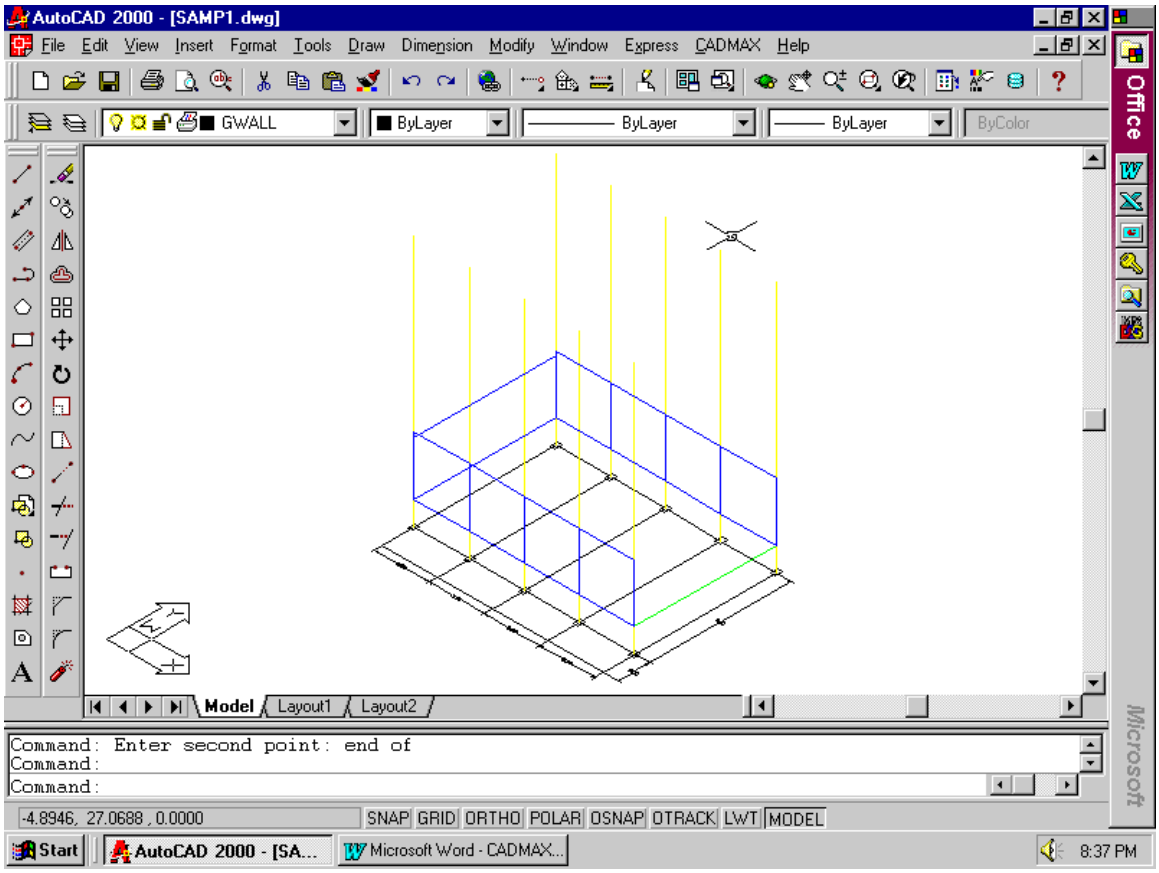
Command:WALL
Enter first point:<pick the start point>
Enter second point:<pick the end point>
Now you will see the following dialog box to fill in the details of wall supported on grade beam and also to fill in the opening details which will be opted for no further openings. The **elevation of start point and the elevation of end point is filled in as 1.5m** as given previously. But the **height of wall at start point and height of wall at end point is given as 3.4m** by adding the height of basement 0.5m and height of floor 3.5m and deducting the depth of beam at floor level as 0.6m(0.5+3.5-0.6=3.4). The width of wall is filled in as 0.23 and **check box for load bearing wall is not clicked**. Then **click OK.**



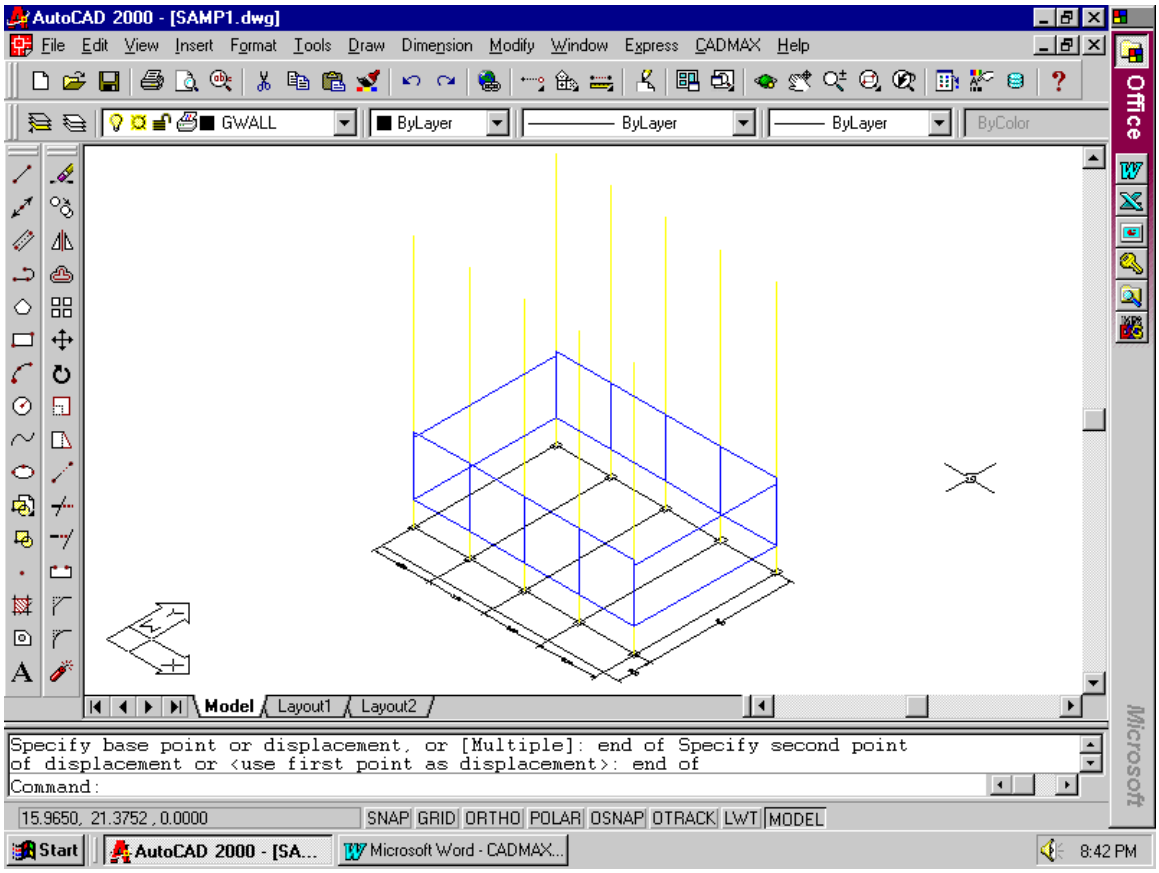
In the **wall openings dialog box** enter **No further openings option** and then **click OK**.



Now you will see the wall along Y-direction is drawn in appropriate location as shown below:

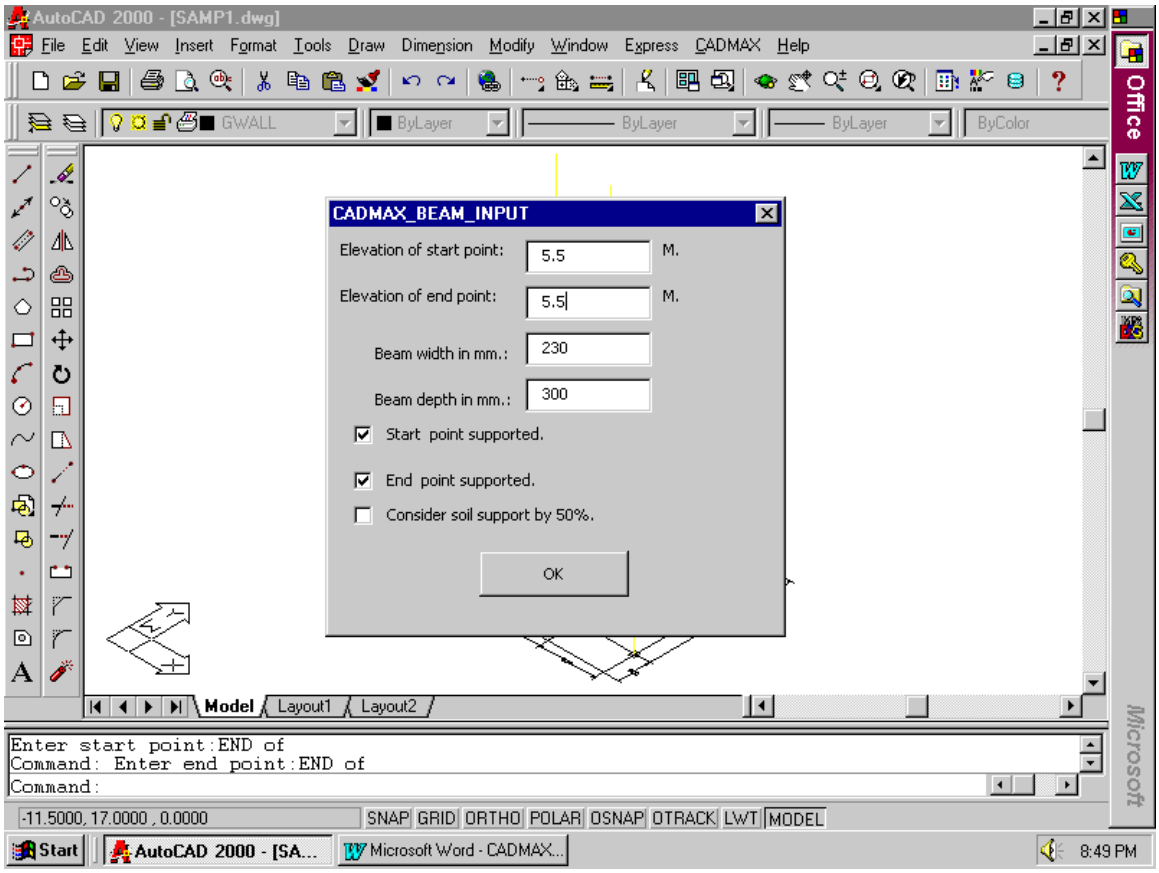


The last drawn wall may be copied to the other end using copy command as shown below:
Command: COPY <enter>
Select objects: L <enter>
Select objects: <enter>
Enter base point of displacement:<pick the base point>
Enter Second point or displacement:<pick the end point to copy to the required location>
Now the Drawing will be seen as given below:

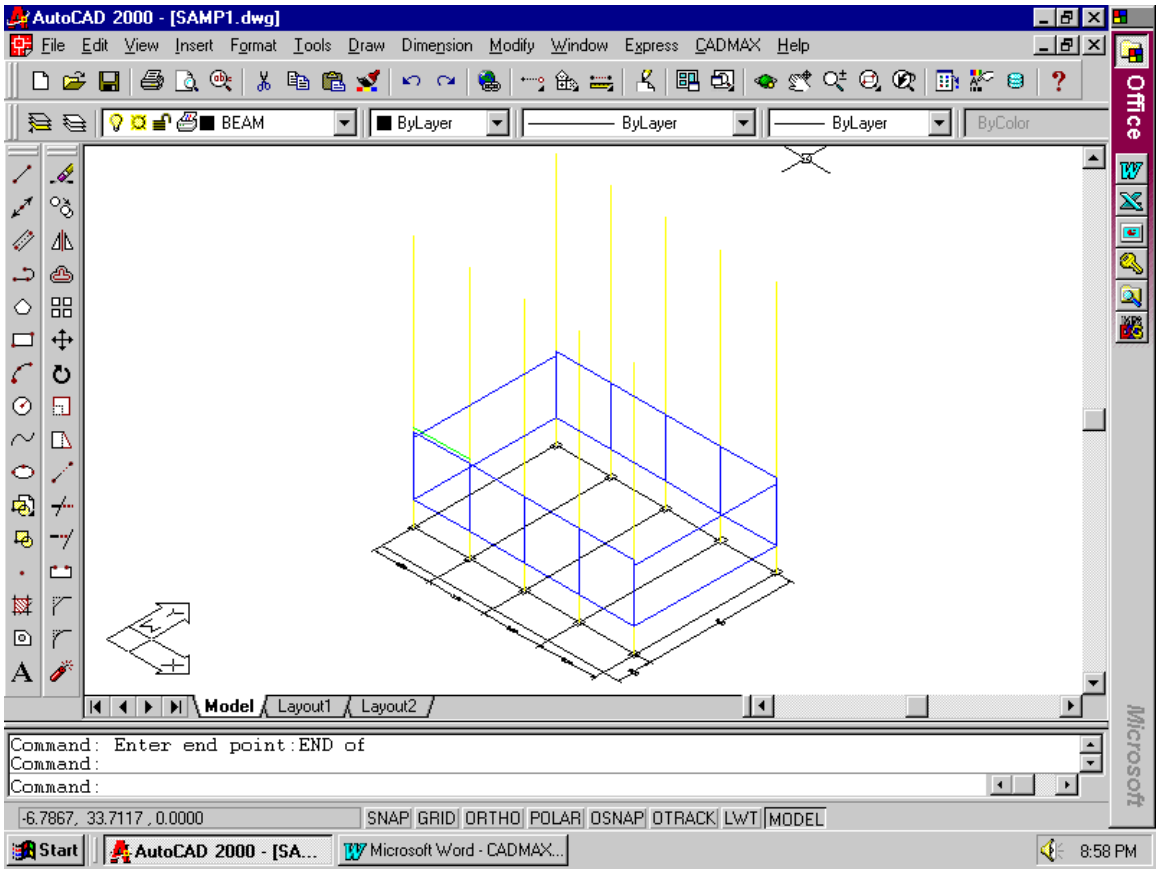


Now the grade beams and walls supported on the grade beams are introduced in the structure. Now let us insert the floor beams,slabs and walls as follows.

Pick the **BEAM** command from **CADMAX** Pop-up menu to introduce floor beams and enter the layer name to draw as **BEAM**. The beam details dialog box is filled in as shown below: The elevation of start point and the elevation of end point is shown as 5.5 since the floor level is at a height of 5.5m from footing bottom level. The size of the beam is filled in as 230 and 300 mm. The **check box for Start point supported and End point supported is clicked** since both ends are supported on columns. The **check box for Consider soil support by 50% is not clicked** since it is a beam at floor level. Then **click OK**.



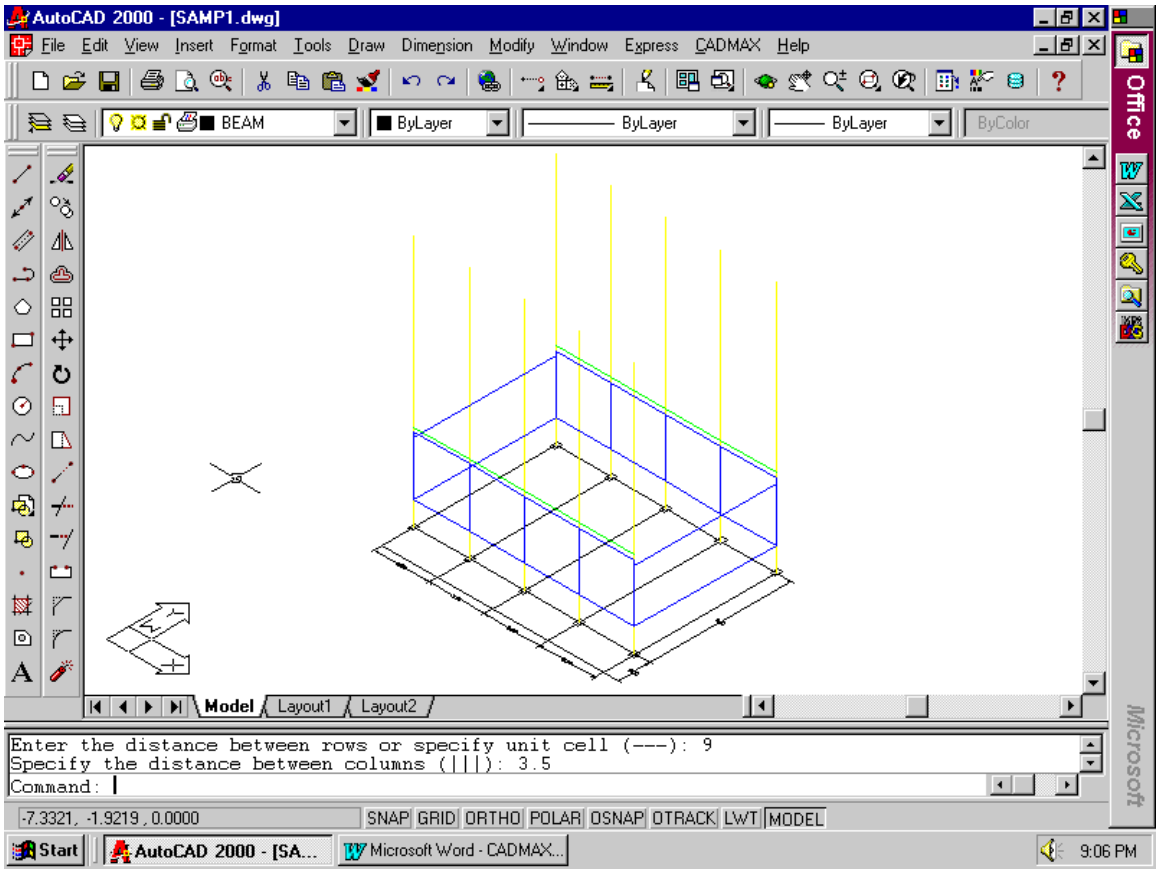
Now you will see a beam in green color is drawn in the appropriate location. To copy the beam to the similar locations use copy or array command.



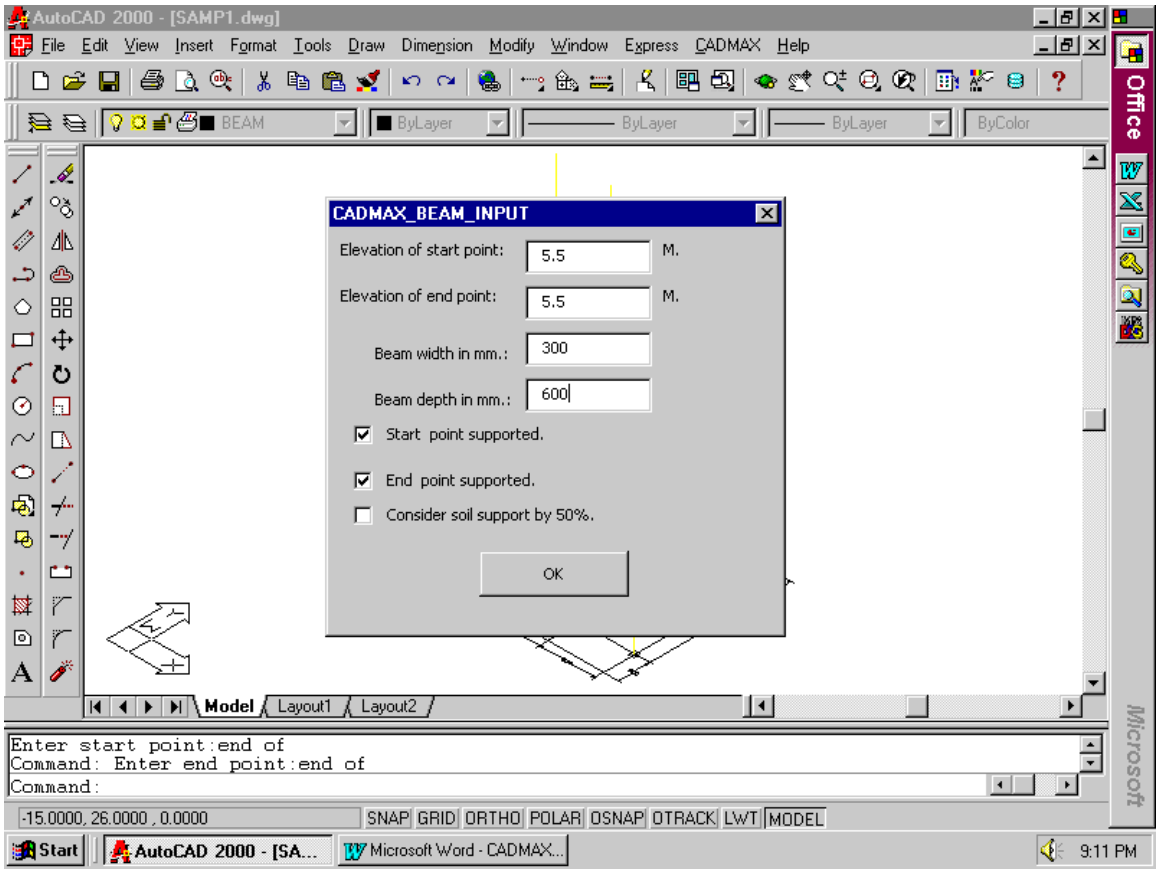
Give array command as follows to copy the similar beams.

Command: ARRAY <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array <Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **2** <enter>
Number of columns: **4** <enter>
Distance between rows: **9** <enter>
Distance between columns: **3.5** <enter>

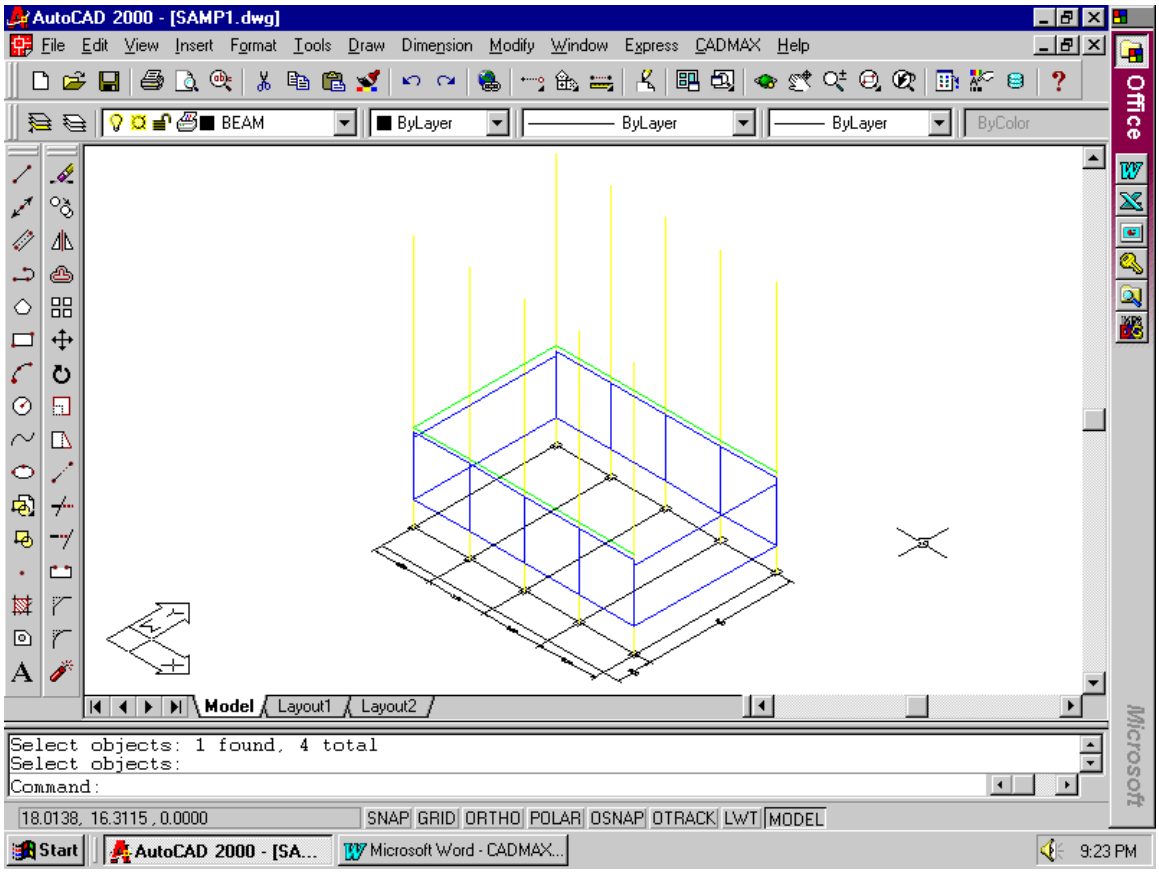
Now you will see all the X-direction floor beams are drawn as shown below:



Similarly you can draw the beam at floor level in Y-direction using the **BEAM** command from **CADMAX** Pop-up menu. Enter the layer name to draw as **BEAM** and fill in the dialog box for beam details as given below and then **click OK**.

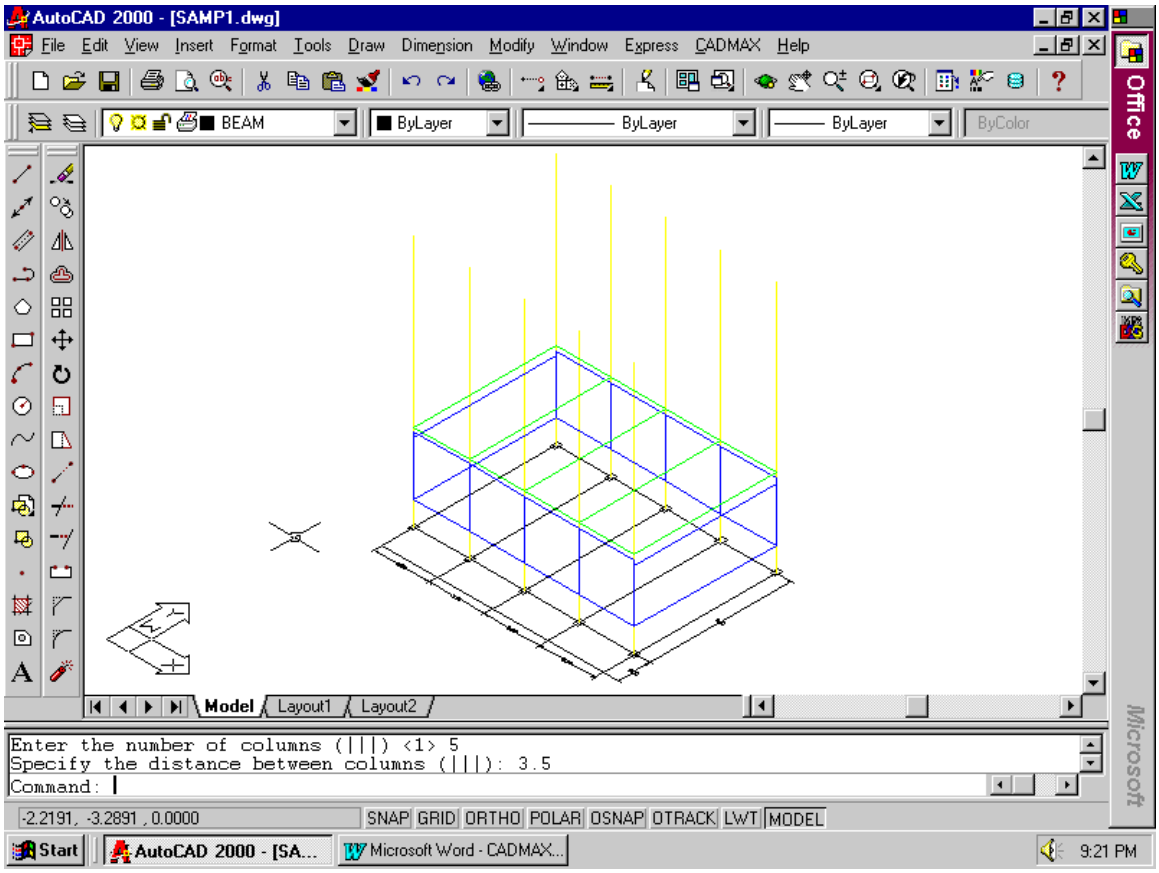


Now you will see the beam drawn in green color as shown below:

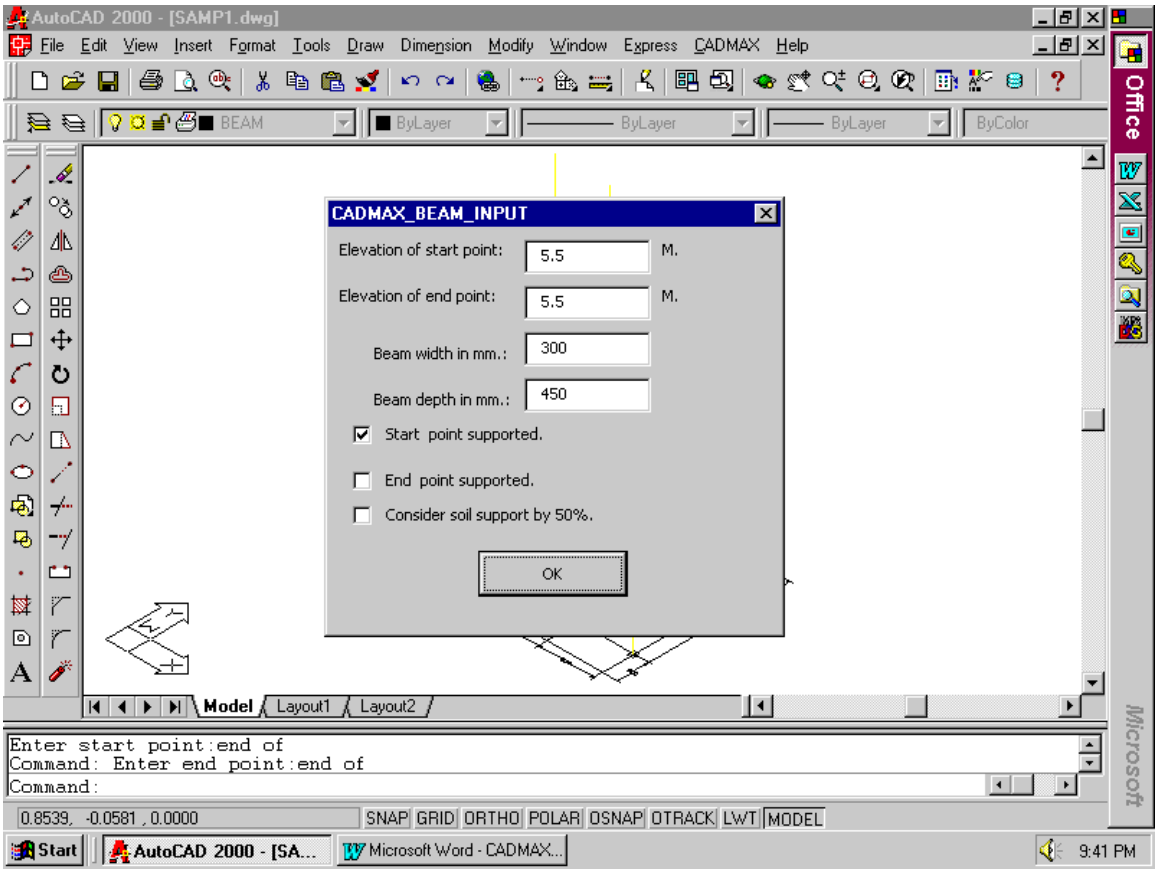


The similar beams can be copied at a distance of 3.5m c/c using array command as follows:
Command:**ARRAY** <enter>
Select objects**: L** <enter>
Select objects: <enter>
Type of array <Rectangular/Polar>:<R>: **R** <enter>
Number of rows: **1** <enter>
Number of columns: **5** <enter>
Distance between columns: **3.5** <enter>

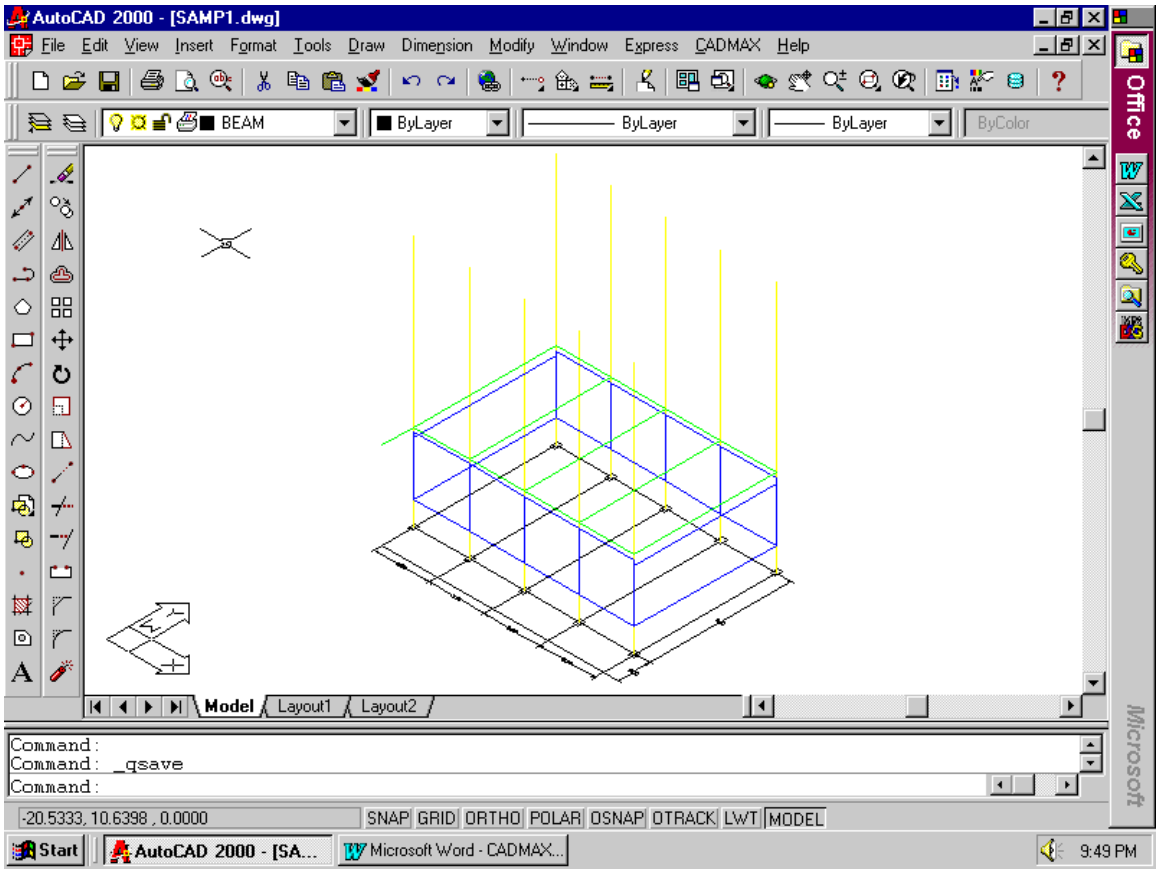
Now you will see the beams along Y-direction as shown below:



Now to insert cantilever beams at the front row use the **BEAM** command from **CADMAX** Pop-up menu. Enter the layer name to draw as **BEAM** and fill in the beam details dialog box as shown below: The **first point is selected at the column support point** and **the second point is selected at the cantilever projection end point**. The size of beam is filled in as **300 and 450** mm. The **elevation of start point and end point are filled in as 5.5m**. The **check box for start point supported is clicked** and the **check box for end point supported is not clicked since it is free end of the cantilever beam**. The **check box for consider soil support by 50% is not clicked** since it is at floor level.

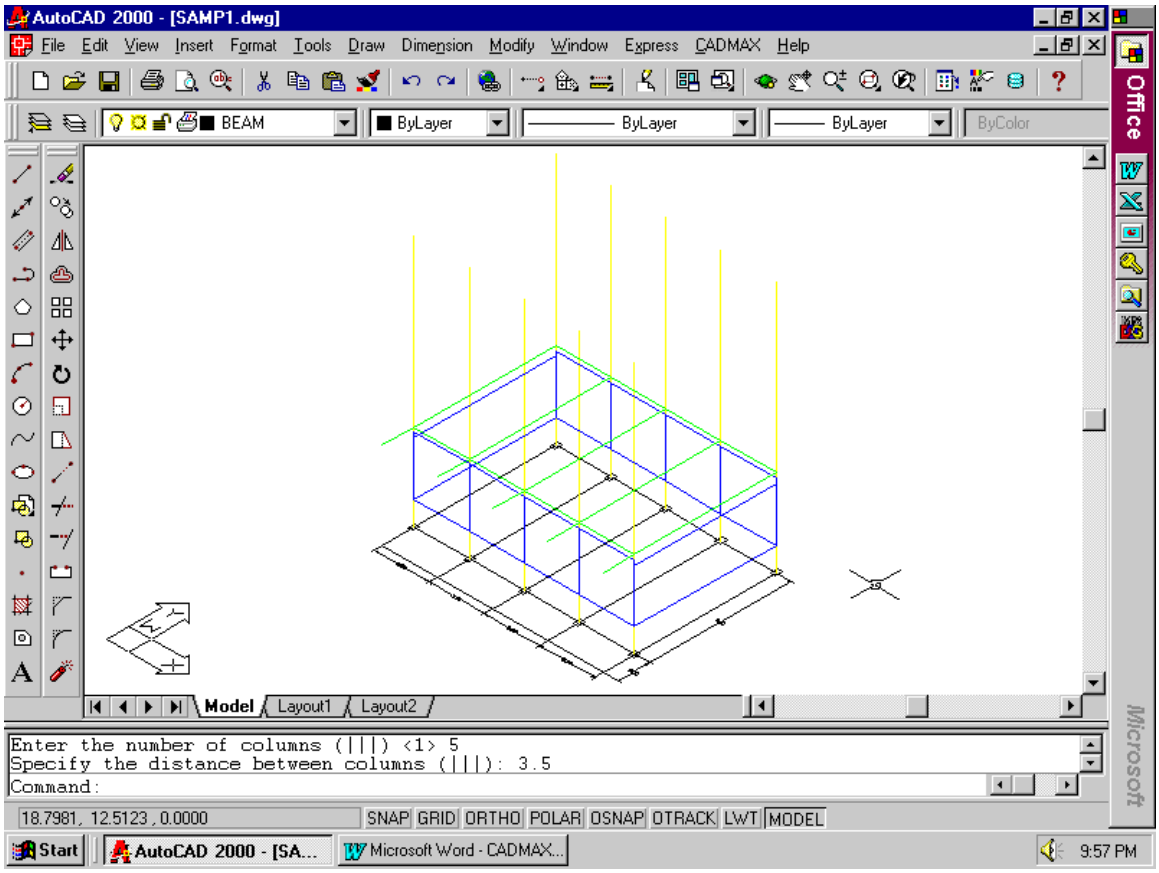


Now you will see the cantilever beam drawn as shown below:



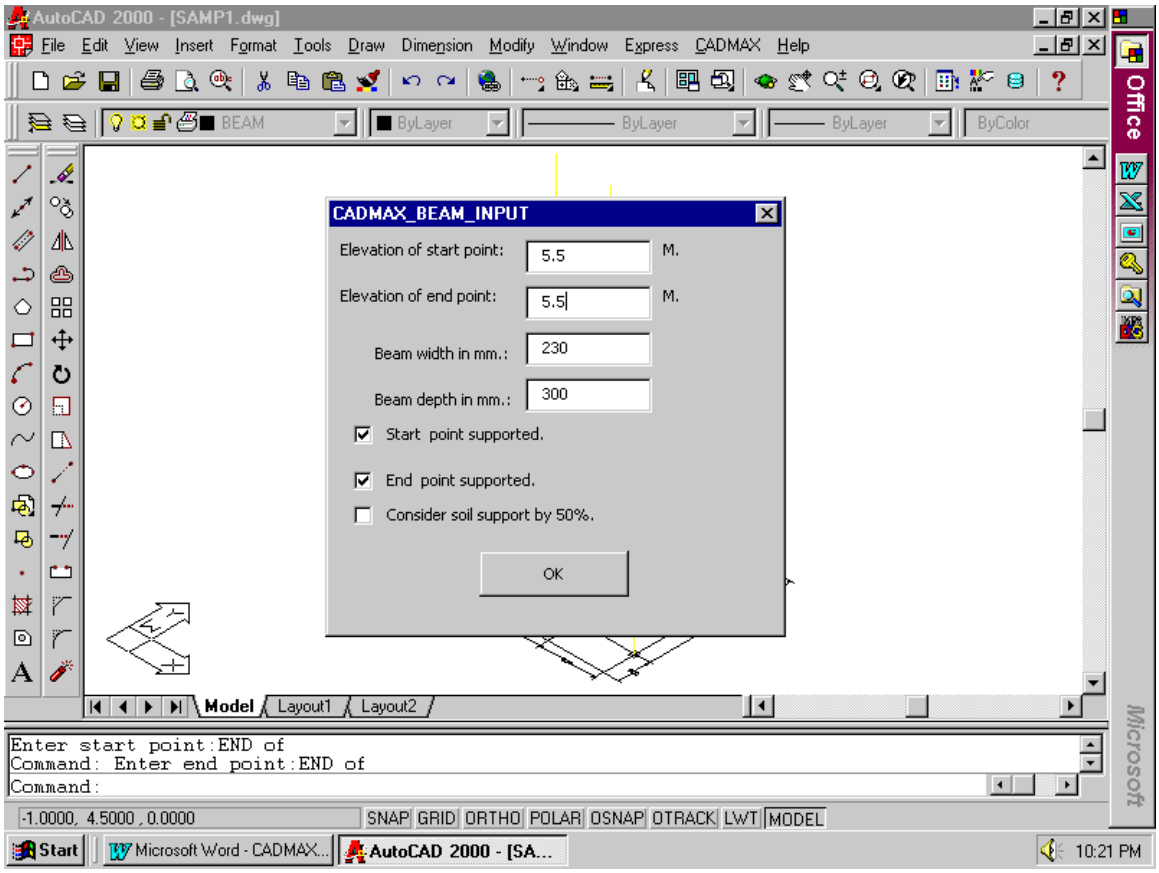
Now you can copy the last drawn cantilever beam to the appropriate locations using array command as shown below:

Command: **ARRAY** <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array <Rectangular/Polar>:<R>: **R** <enter>
Number of rows: **1** <enter>
Number of columns: **5** <enter>
Distance between columns: **3.5** <enter>
Now you will see the cantilever beams copied to the appropriate locations as shown below:

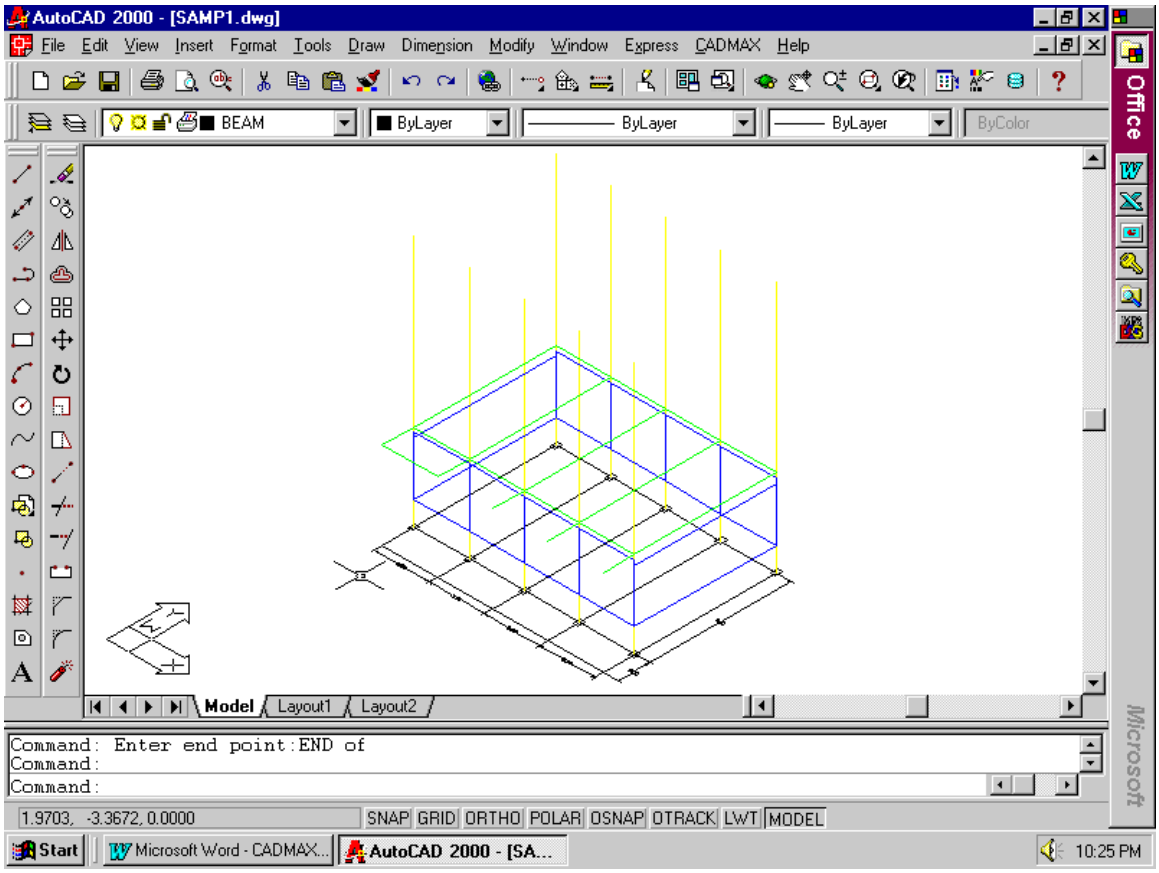


Now insert the edge beams at cantilever end points as shown below:
Use the **BEAM** command in the **CADMAX** Pop-up menu to draw the beam in the first bay. Enter the layer name to draw as **BEAM** and then proceed further as already explained.

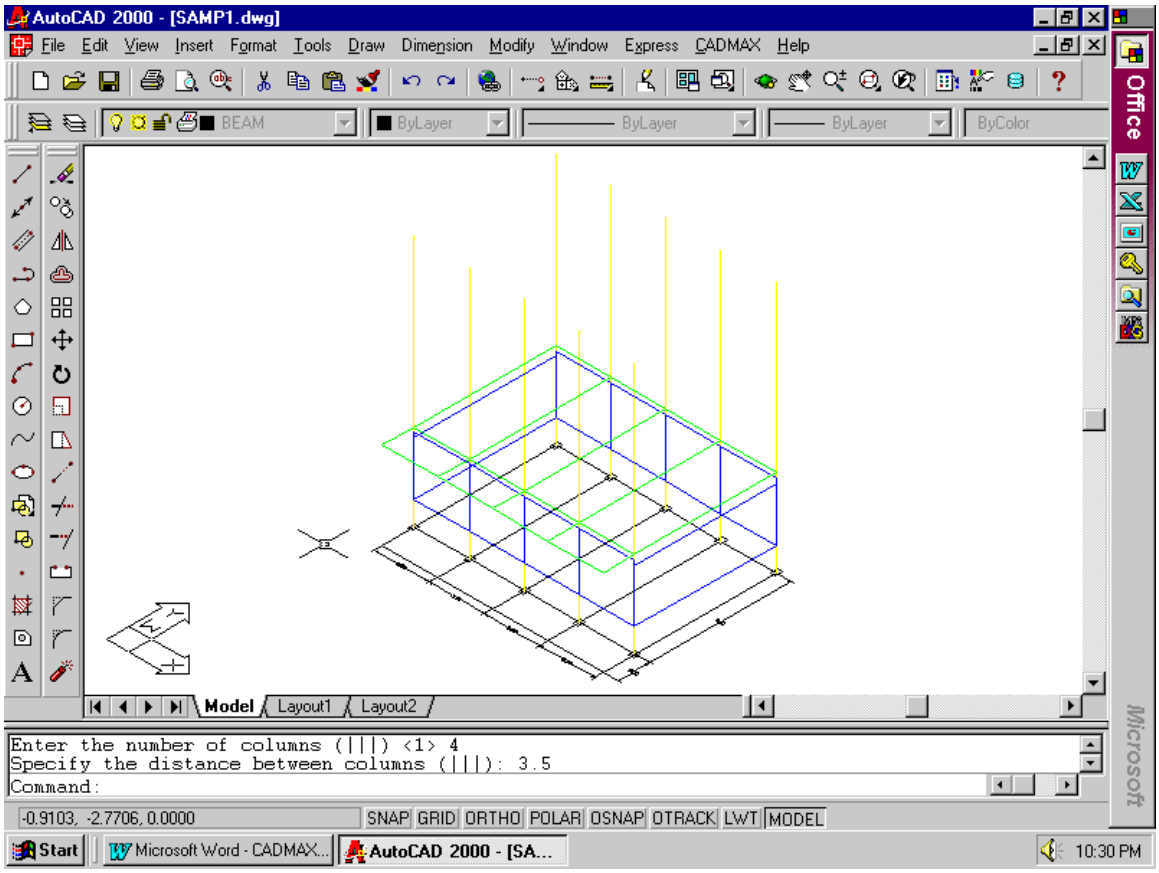
Command: **BEAM**
Enter start point:<pick the start point>
Enter end point:<pick the end point>
Then fill in the following dialog box as shown below and the **click OK.**



Now you will see the first edge beam is drawn as shown below:
Then copy the last drawn edge beam to the appropriate locations using array command.



Command**: ARRAY**
Select objects: **L** <enter>
Select objects: <enter>
Type of array <Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **1** <enter>
Number of columns: **4** <enter>
Distance between columns: **3.5** <enter>
Now all the edge beams are copied and the drawing will be seen as shown below:



Now all the beams at floor level are drawn and now the slabs between beams are introduced as follows:

4.Pick the **SLAB** command from **CADMAX** Pop-up menu

Or

Give **SLAB** command at the command prompt of **AutoCAD.**

Enter the layer name to draw as **SLAB** and continue further as follows:

Command: **SLAB**

Enter first point**:<pick the first point**>

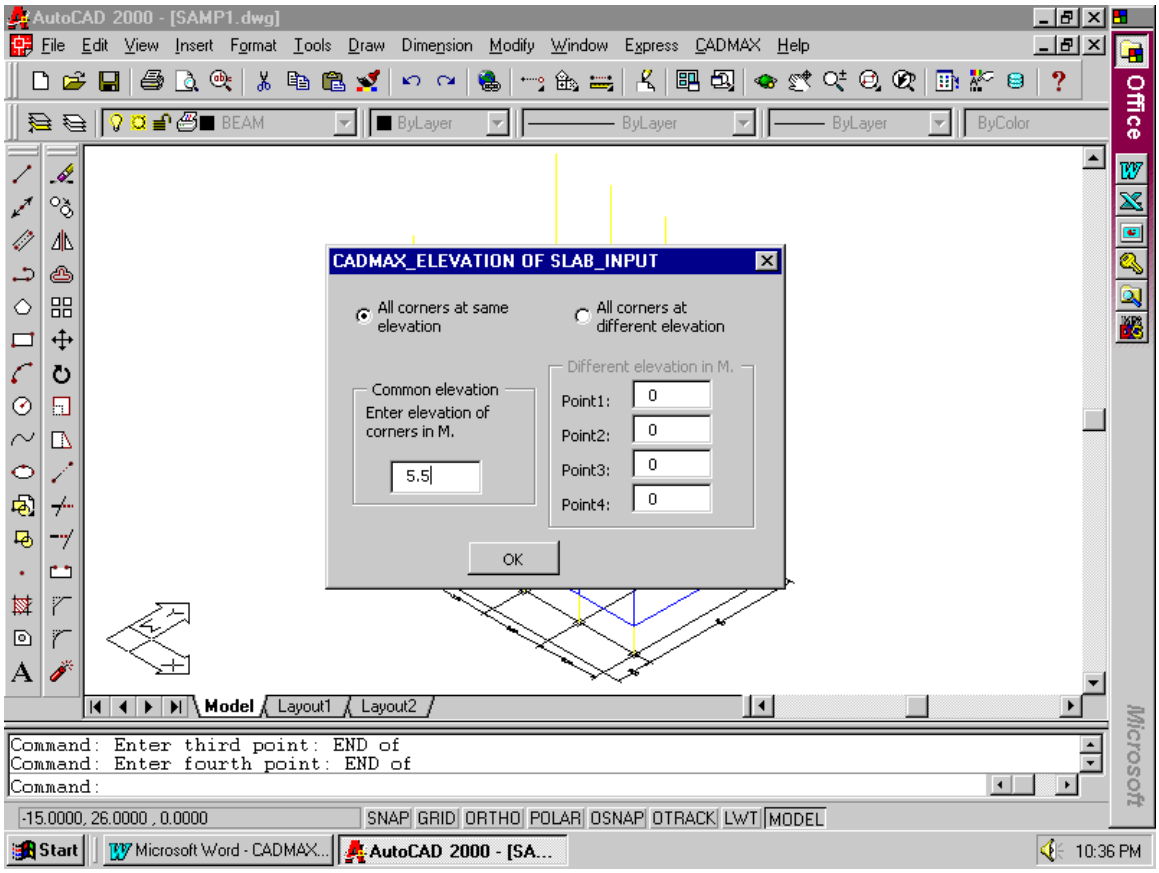
Enter second point: **<pick the second point>**

Enter third point: **<pick the third point>**

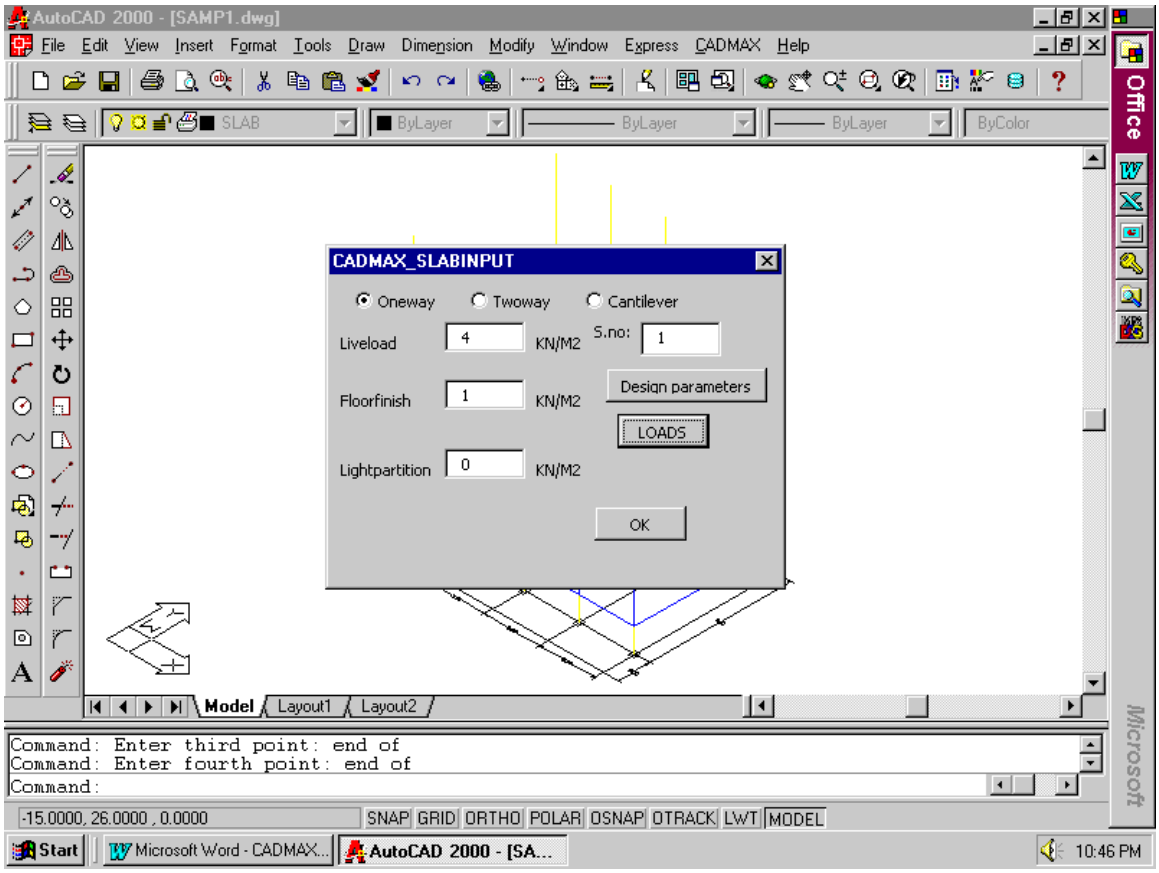
Enter fourth point: **<pick the fourth point>**

(All the above four points should be selected in anti-clock wise order)

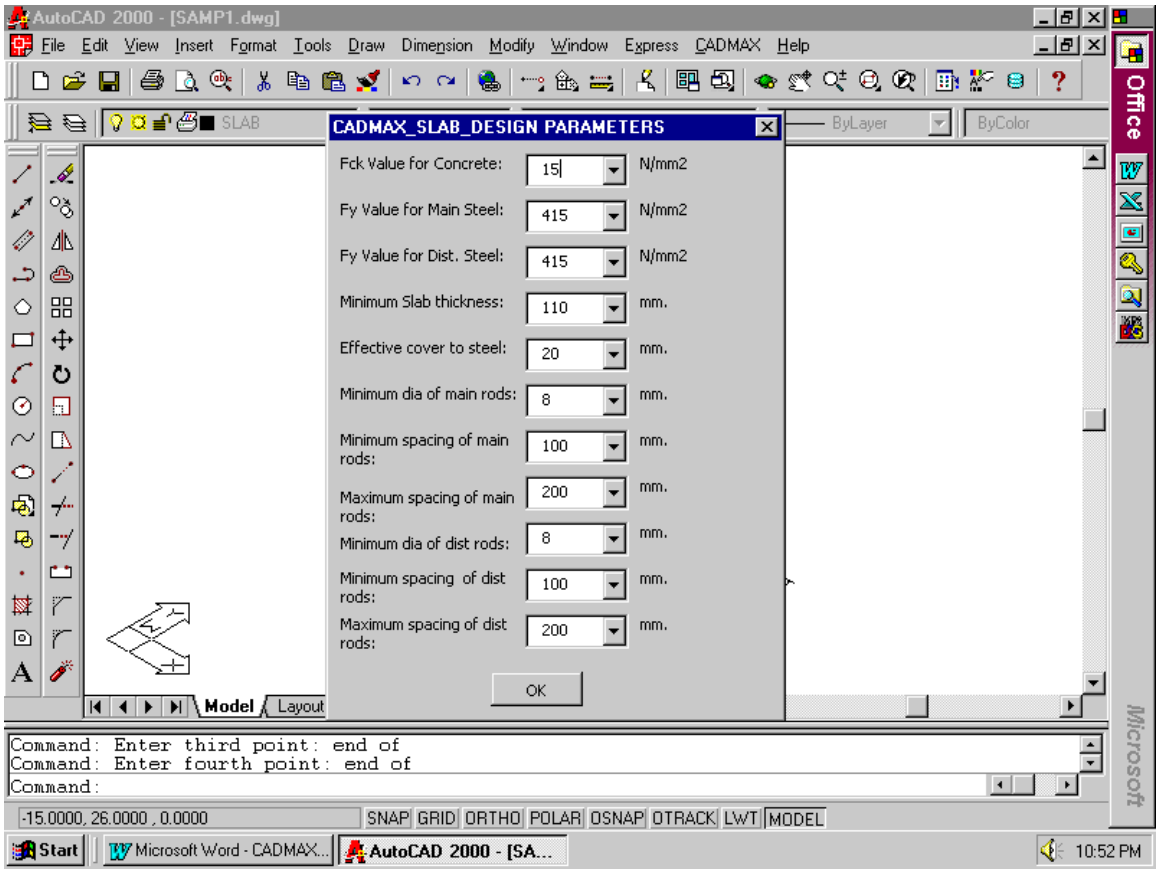
Then fill in the following dialog box as shown below:



After filling in the above dialog box **click OK**.
Then you will see another dialog box as shown below:



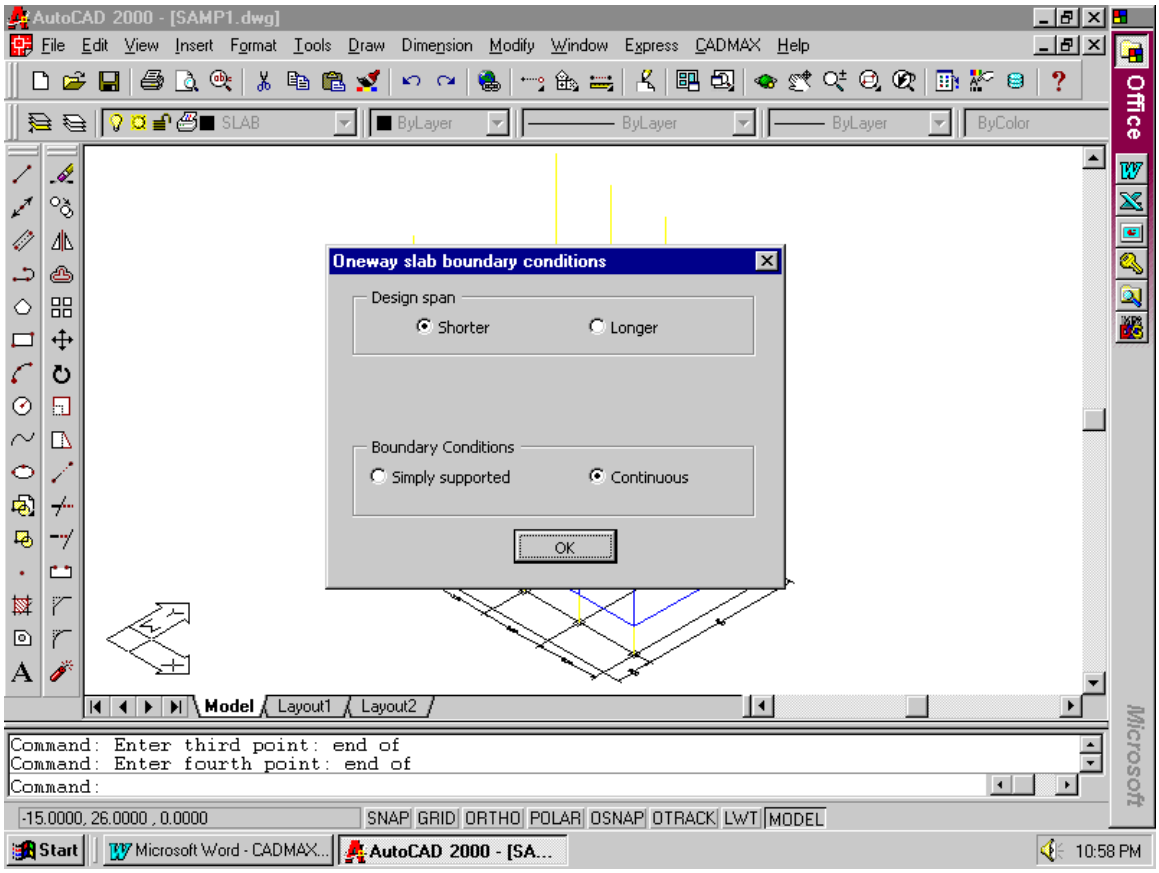
Fill in the above dialog box as shown above.
Click one way slab option since the size of slab is one way slab.
Click loads button to load default loads and then correct the loads as live load as **4**
Floor finish as **1** and light partition as **0** kN/m2 Then click design parameters button to give the design parameters dialog box as shown below:



Fill in the above design parameters according to the requirements as shown above and **then click OK**.

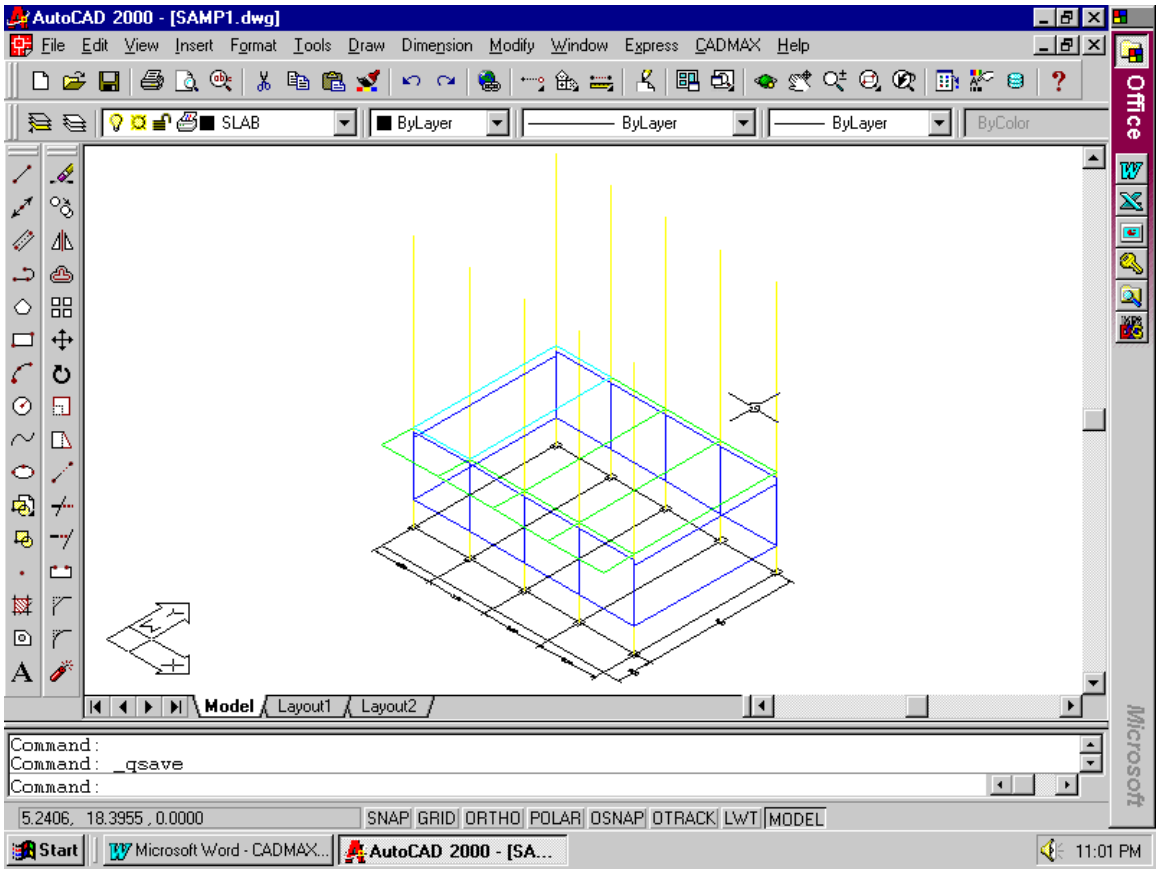
Now you will get the previous dialog box with updated design parameters and then click **OK**.

Now you will get another dialog box to select the boundary conditions and the design span for one way slab is shorter or longer span as shown below:

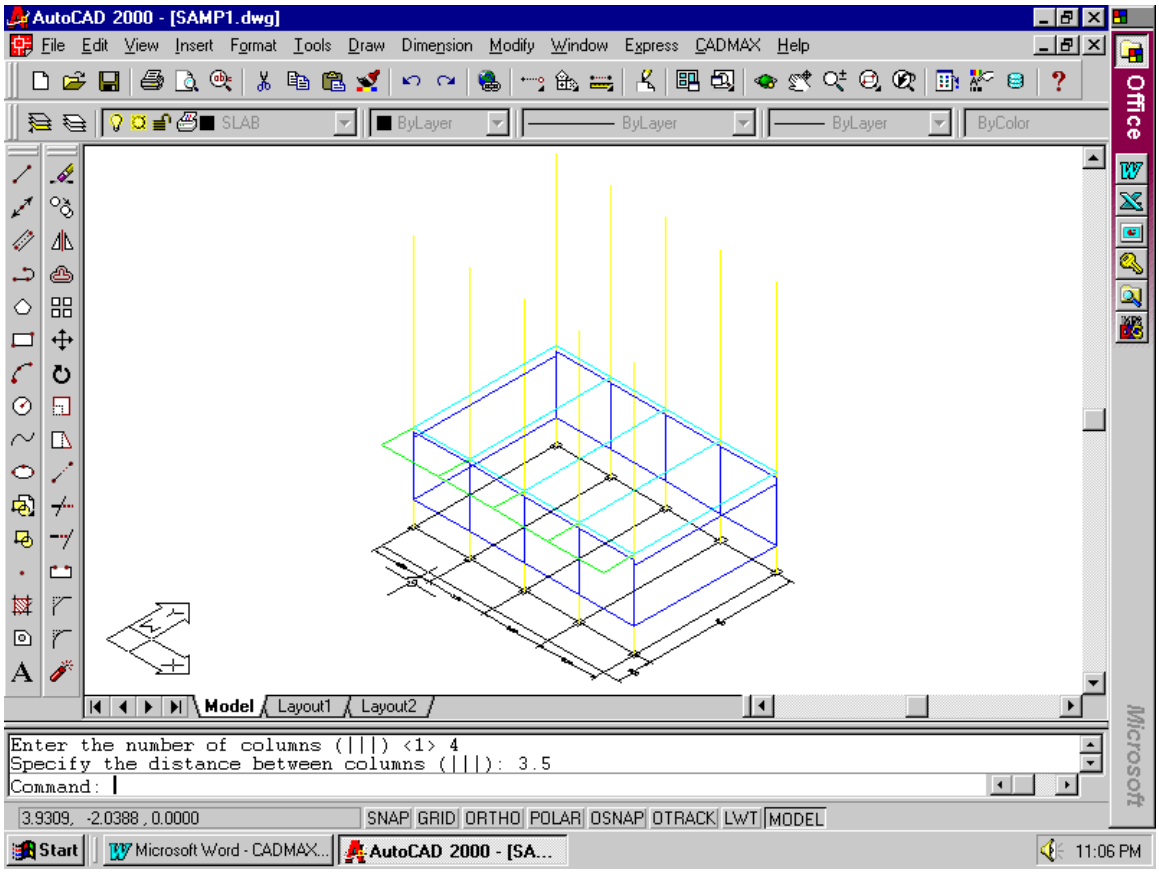


After clicking the required options in the above dialog box as shown above then click **OK**.

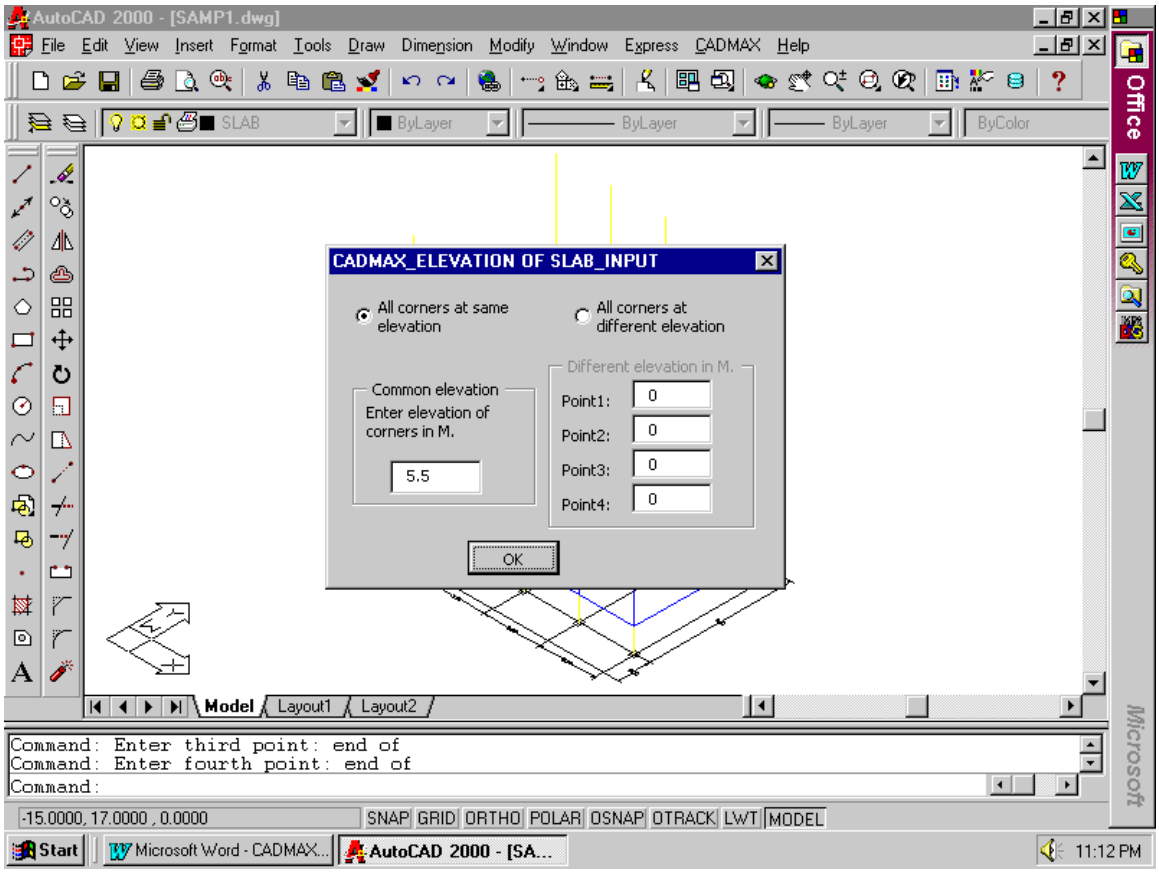
Now you will see a one way slab drawn in the required location in cyan color as shown below:



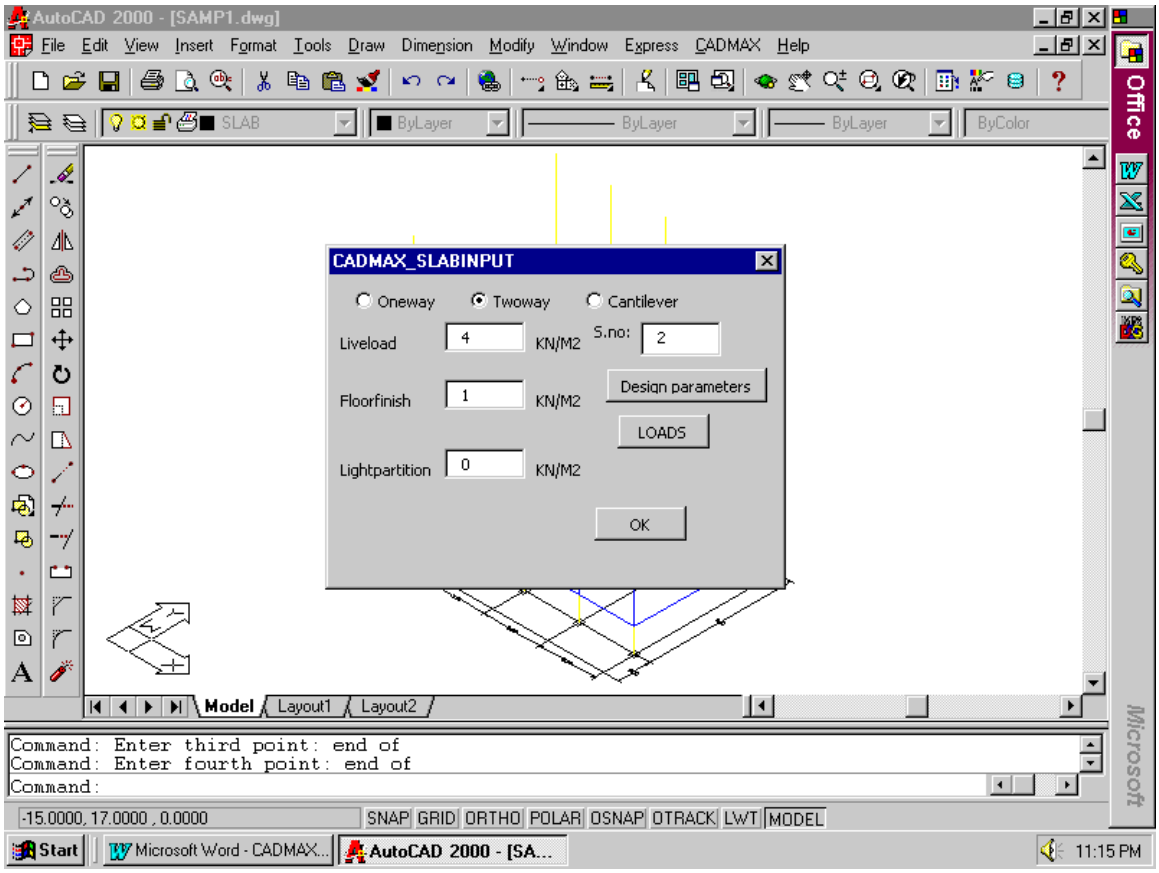
The last drawn slab can be copied to the similar locations using copy or array command as shown below:
Command: **ARRAY** <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array<Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **1** <enter>
Number of columns: **4** <enter>
Distance between columns: **3.5** <enter>
Now you will see the similar slabs are drawn in the required locations as shown below:



Now you will draw the slabs in the projections as follows:
Use the **SLAB** command in the CADMAX Pop-up menu and then enter the layer name to draw as **SLAB**.
Command: **SLAB**
Enter first point:<pick the first point>
Enter second point: <pick the second point>
Enter third point:<pick the third point>
Enter fourth point:<pick the fourth point>
Then fill in the following dialog box as shown below:



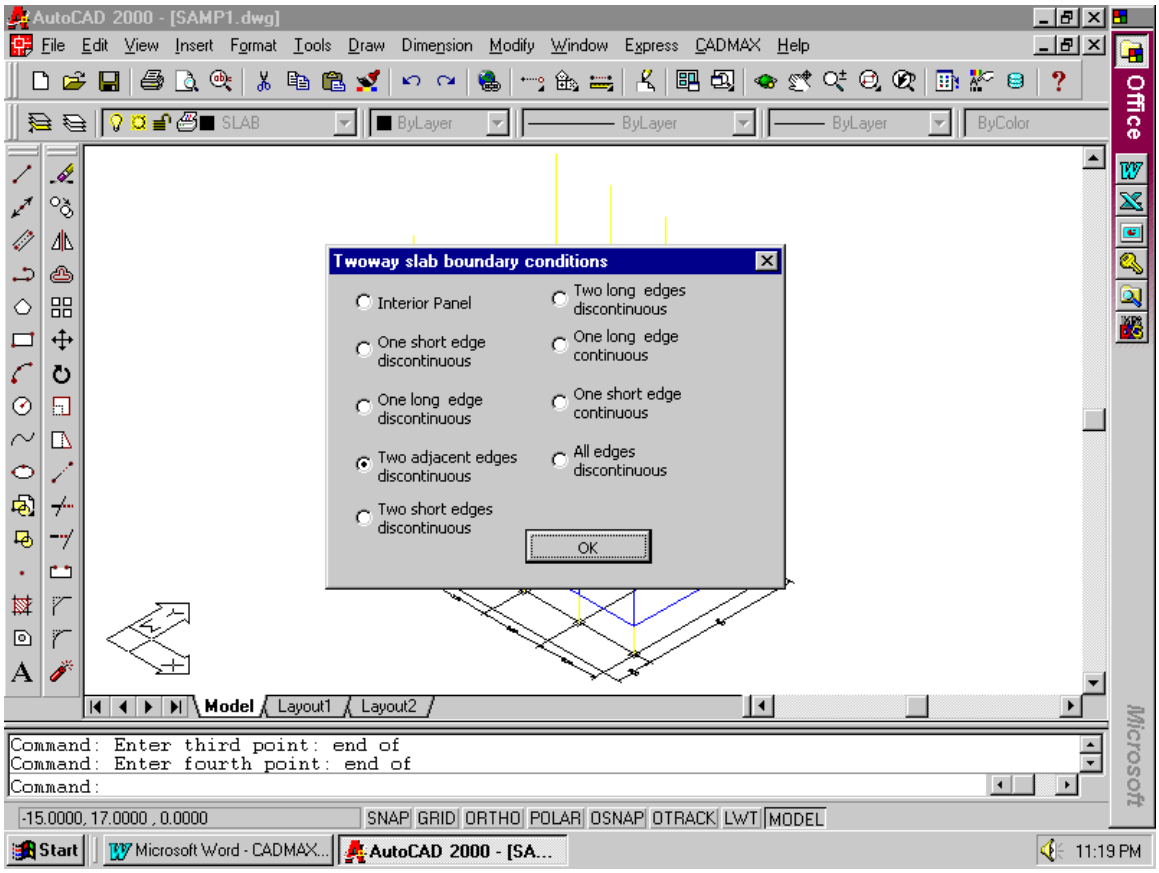
After filling in the above dialog box as above then **click OK**.
Now you will get the next dialog box as shown below:

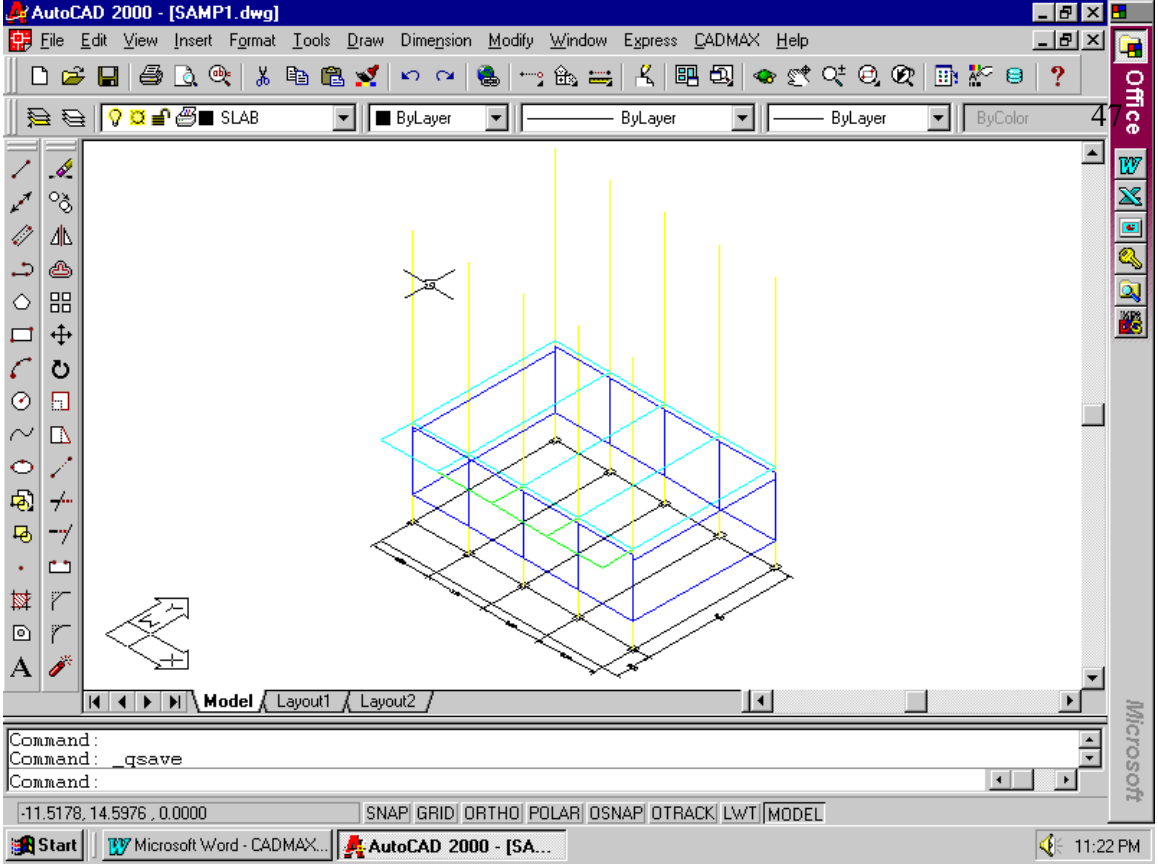


In the above dialog box fill the type of slab option as two way slab and slab no: is changed to 2 automatically and all other conditions are same and then click **OK**.
Now you will see the following dialog box to enter the boundary conditions for two way slab:

In the above dialog box pick the **two adjacent edges discontinuous** option and then click **OK.**

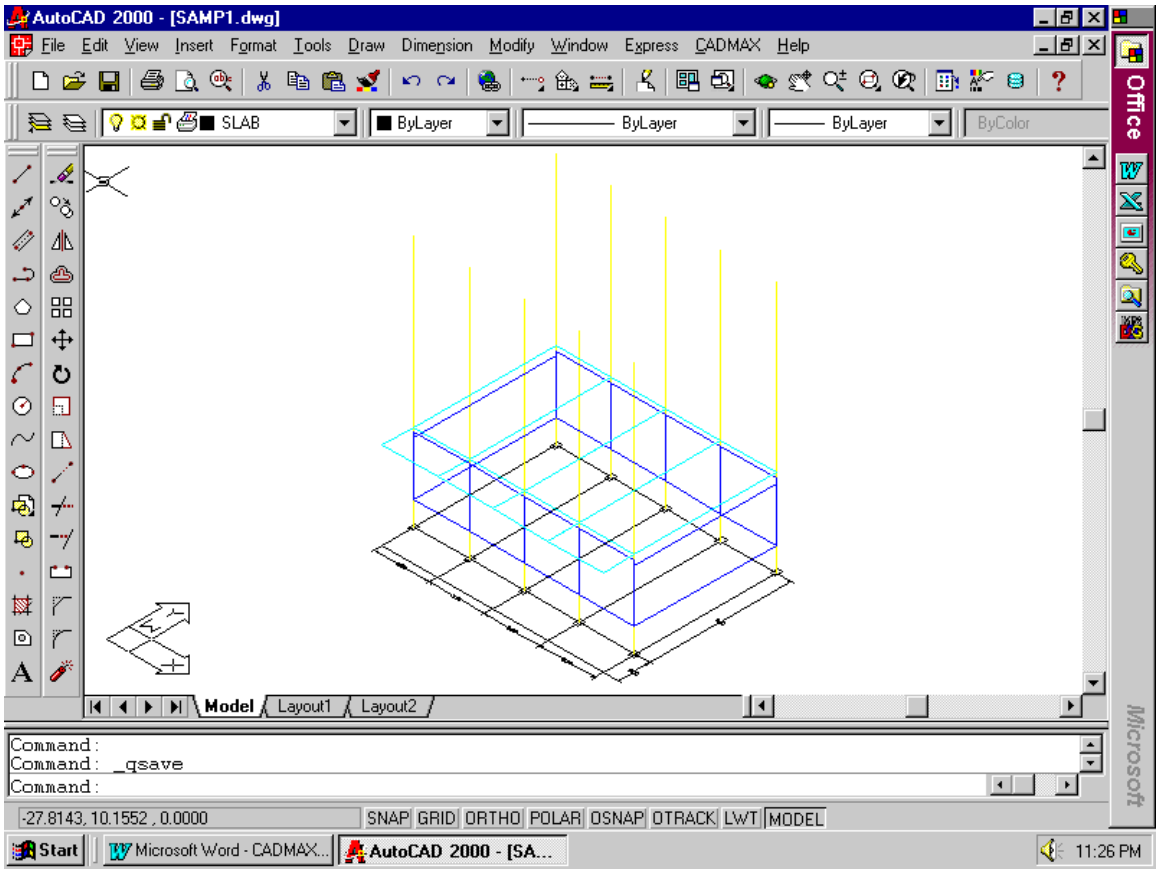
Now you will see the edge slab at projection is drawn as shown below


AutoCAD 2000 - [SAMP1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Window Express CADMAX Help
SLAB
ByLayer
ByLayer
ByLayer
ByColor
Twoway slab boundary conditions
Interior Panel
One short edge discontinuous
One long edge discontinuous
Two adjacent edges discontinuous
Two short edges discontinuous
Two long edges discontinuous
One long edge continuous
One short edge continuous
All edges discontinuous
OK
Model
Layout1
Layout2
Command: Enter third point: end of
Command: Enter fourth point: end of
Command:
-15.0000, 17.0000 , 0.0000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL
Start
Microsoft Word - CADMAX...
AutoCAD 2000 - [SA...
11:19 PM
Office
Microsoft



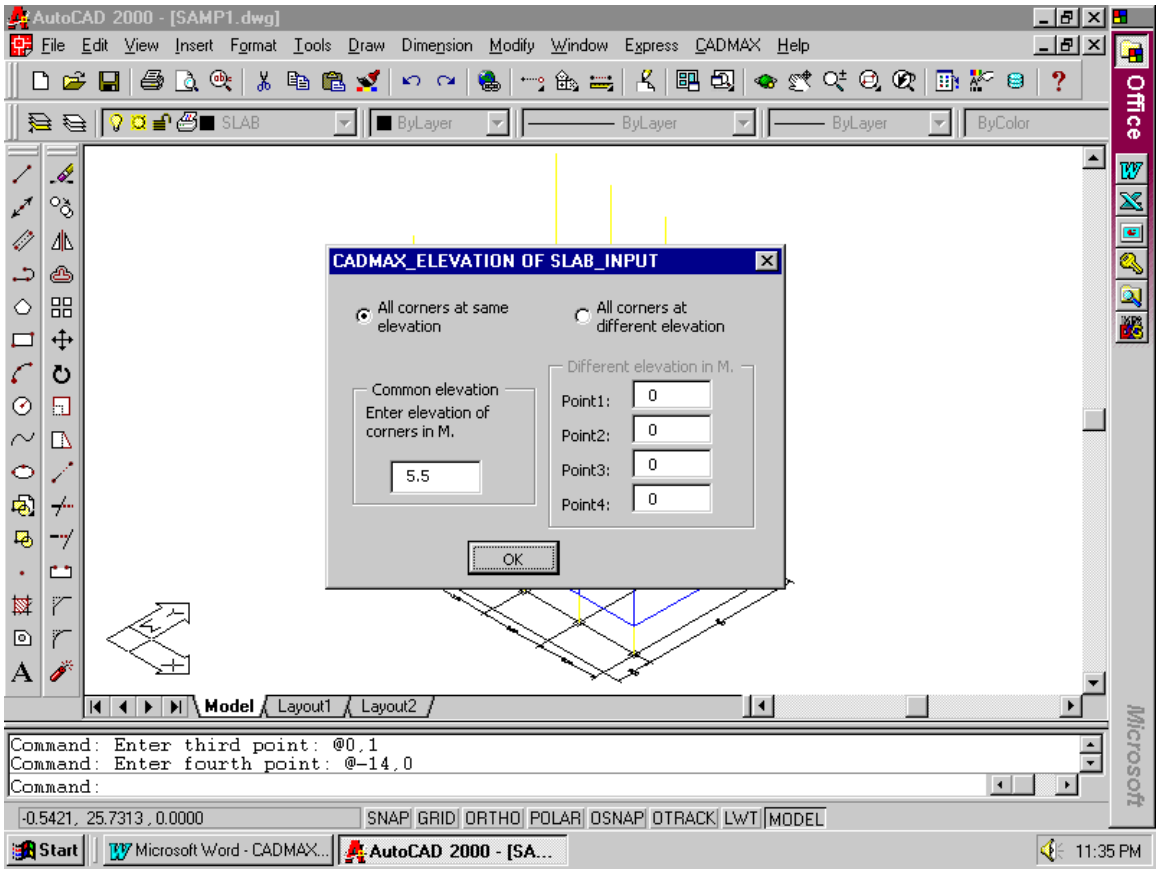
The slabs similar to the last drawn edge slab may be copied to the similar locations using array command as shown below:
:

Command: **ARRAY** <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array<Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **1** <enter>
Number of columns: **4** <enter>
Distance between columns: **3.5** <enter>
Now you will see the similar slabs are drawn in the required locations as shown below:

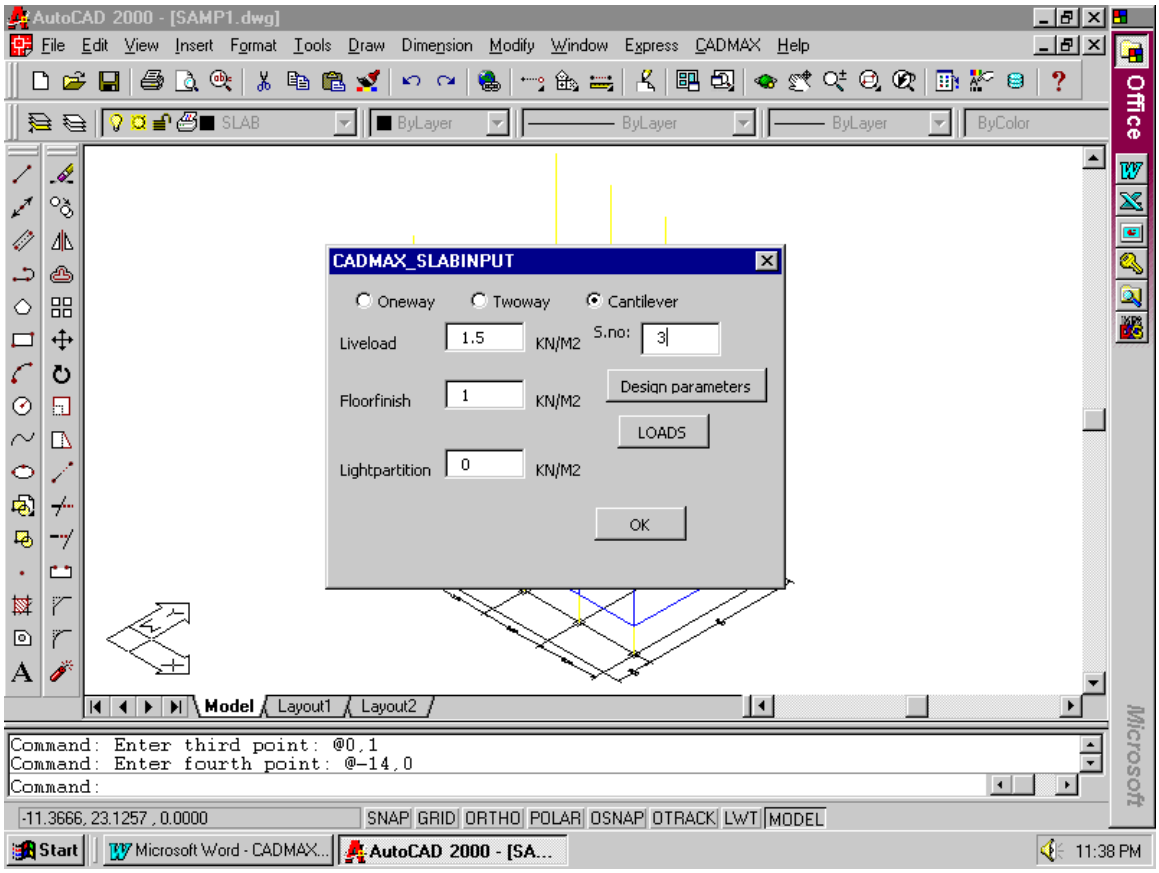


Now you can insert a cantilever slab in the rear side for a span of 1m as follows:

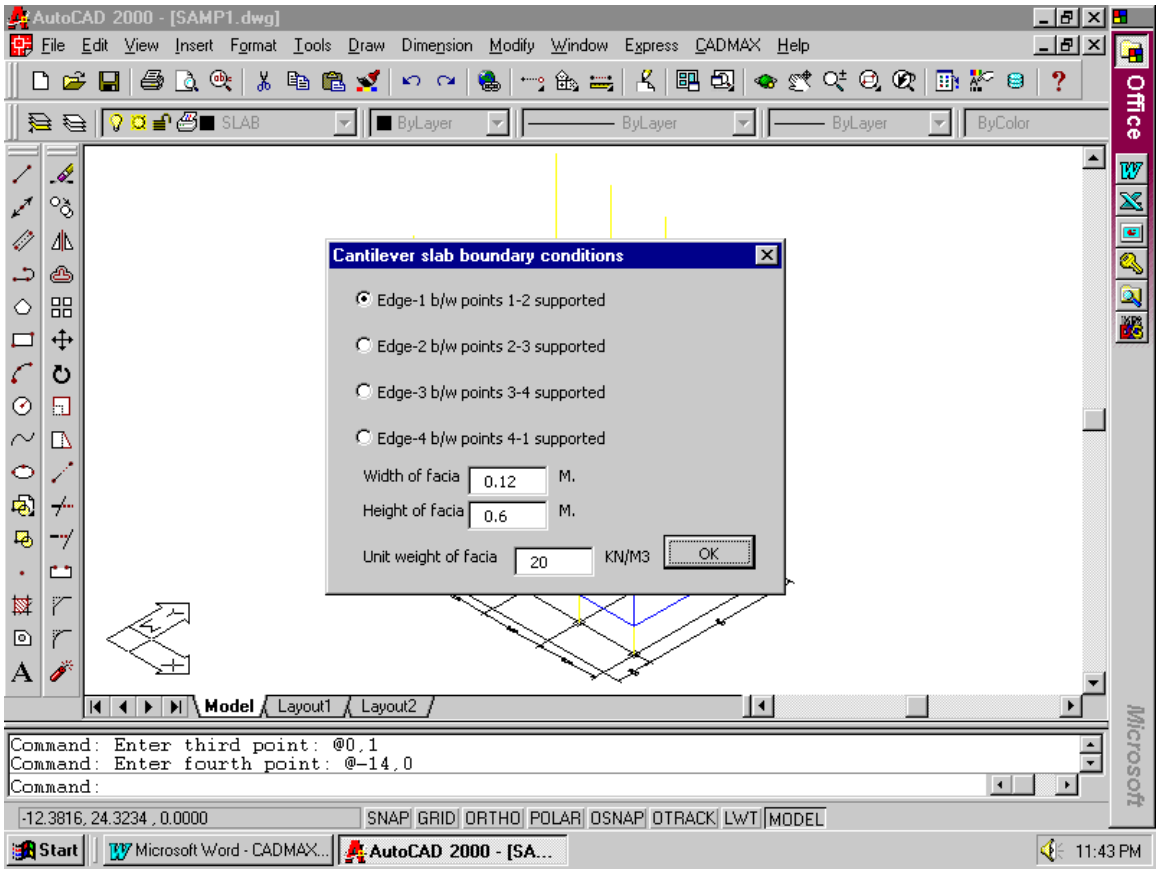
Use the **SLAB** command from **CADMAX** Pop-up menu and enter the layer name to draw as **SLAB**.
Command:**SLAB**
Enter first point: <pick the first point>
Enter second point: <pick the second point>**(@14,0)**
Enter third point: <pick the third point>**(@0,1)**
Enter fourth point: <pick the fourth point>**(@-14,0)**
Now enter the following dialog box as below:



After filling the above dialog box as above the dialog box given below is displayed.



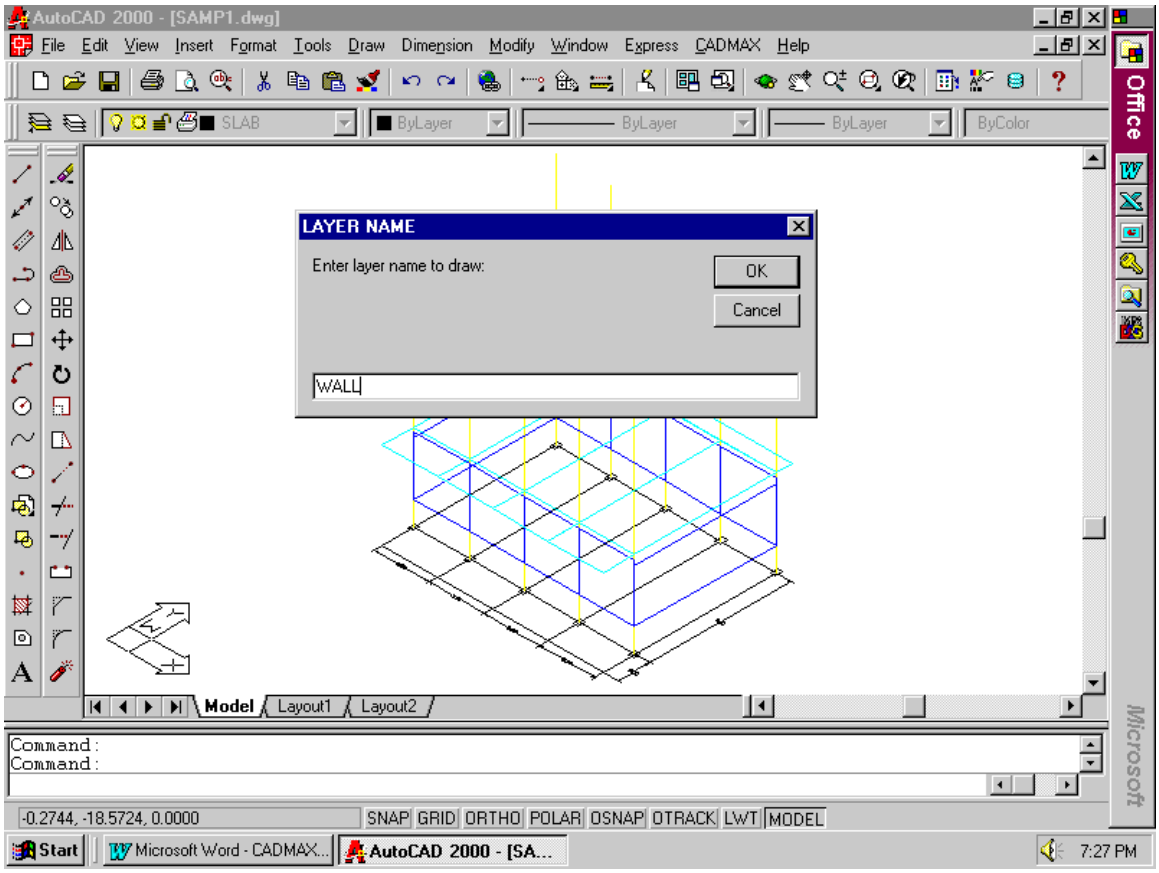
After filling the above dialog box as above for cantilever slab the next dialog box for boundary condition to select the supported edge of cantilever is displayed. In this case the first edge connecting point 1 and 2 will be the supported edge and the width of facia and height of facia and unit weight of facia are to be filled in the next dislog box as shown below:



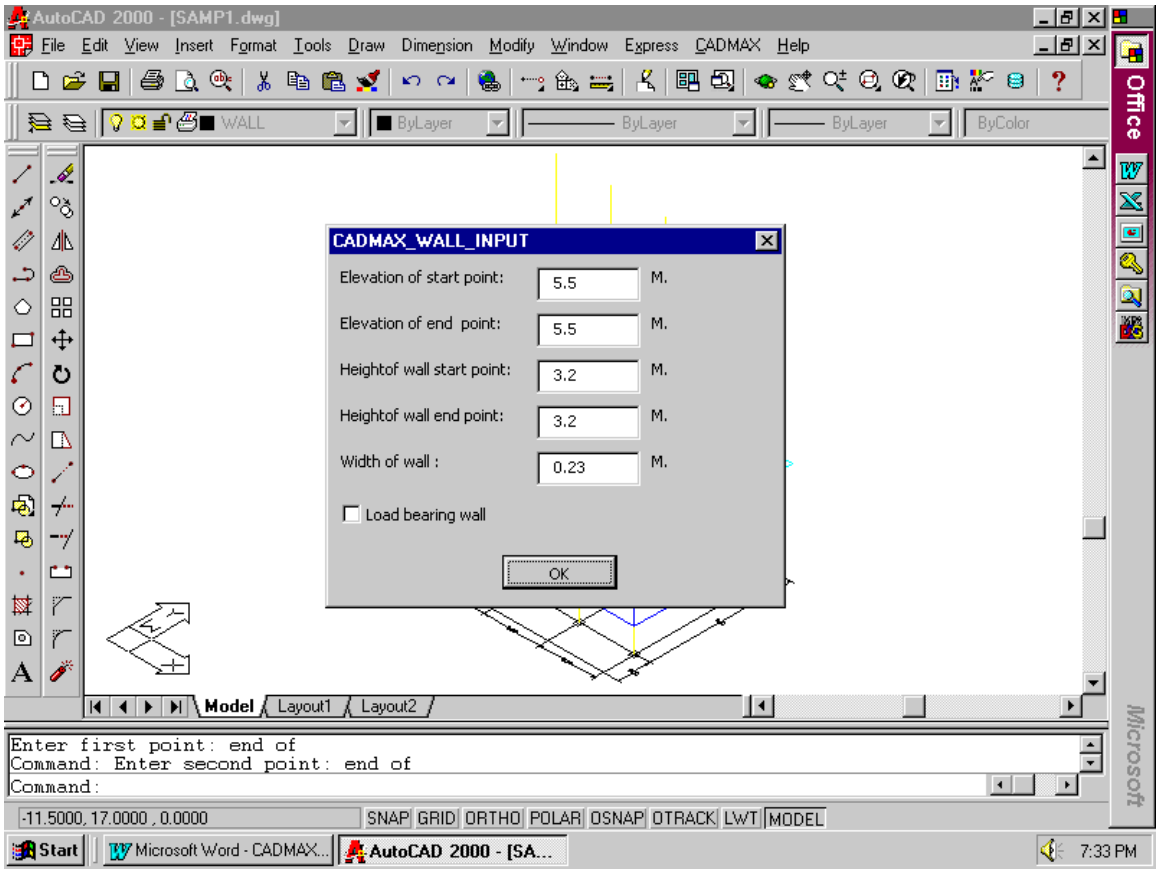
After filling in the above dialog box and then **click OK** . Now you will see the cantilever slab drawn in cyan color in the rear side of the structure.

Now you have to feed in the floor walls and facia details in the front projections as follows:

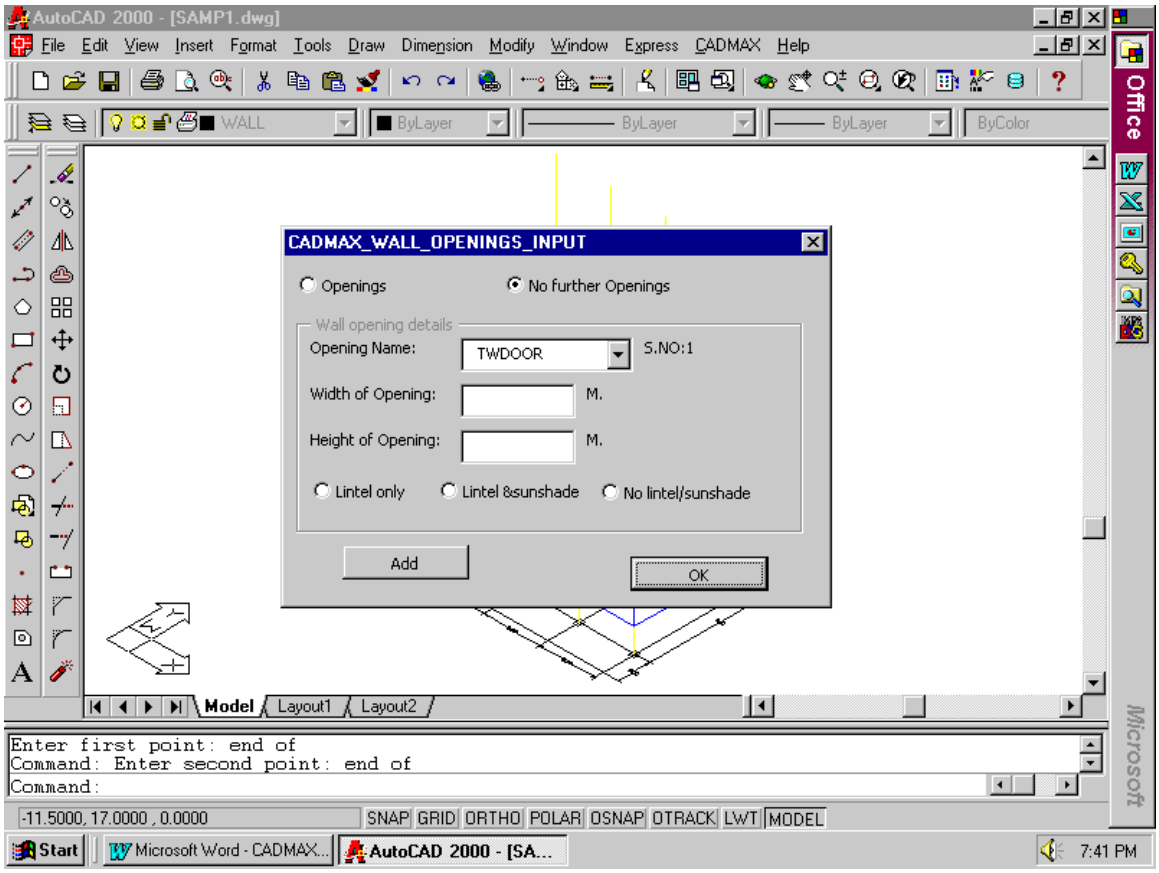
Pick the **WALL** command from **CADMAX** Pop-up menu and enter layer name to enter as **WALL** as follows and then click **OK**.



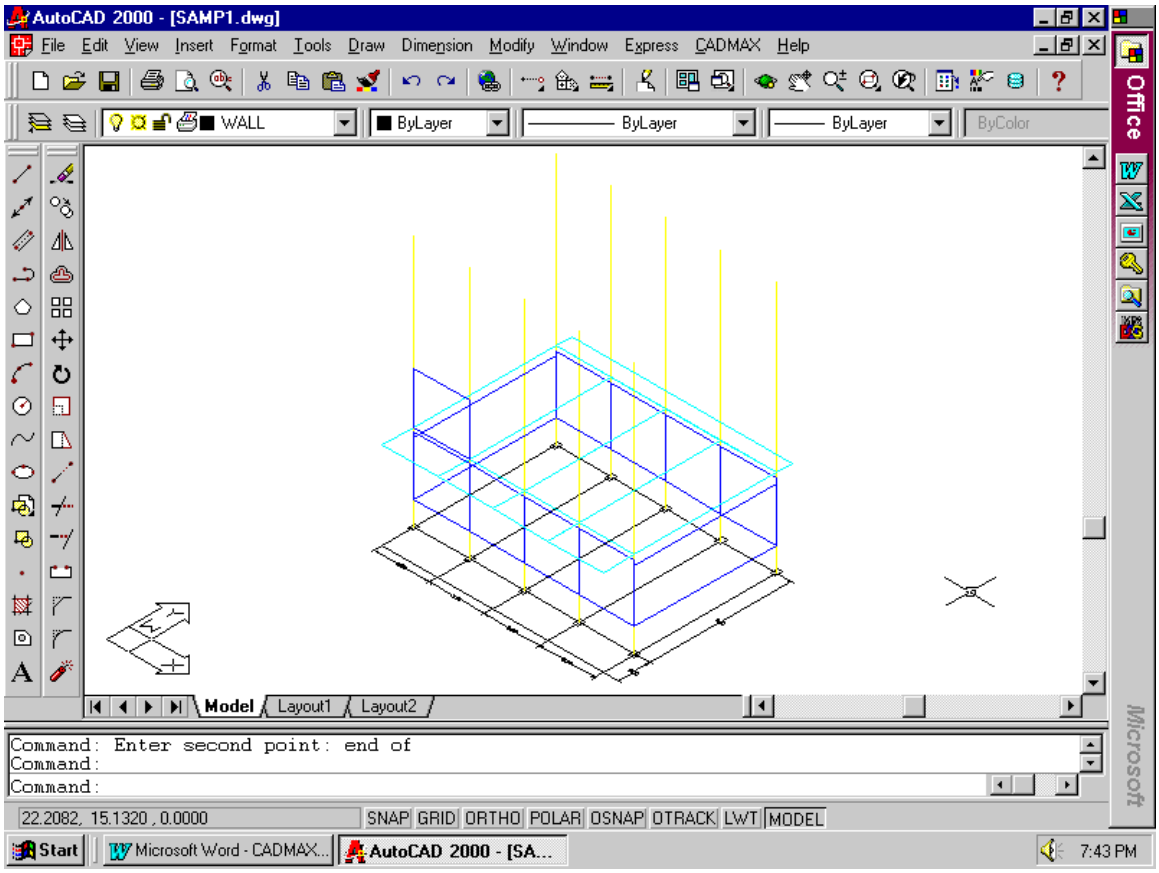
Then pick the first point and the second point as follows.
Command: **WALL**
Enter first point:<pick the start point>
Enter second point: <pick the end point>
Then fill in the wall details dialog box as follows:
In the wall details dialog box **enter elevation of start point and elevation of end point** as **5.5m** and **Height of wall at start point and Height of wall at end point as 3.2m**. The height of wall is arrived as the height of floor and then deduct depth of floor beam(3.5-0.3=3.2).
Then the width of wall is filled in as 0.23m.Then click **OK**.



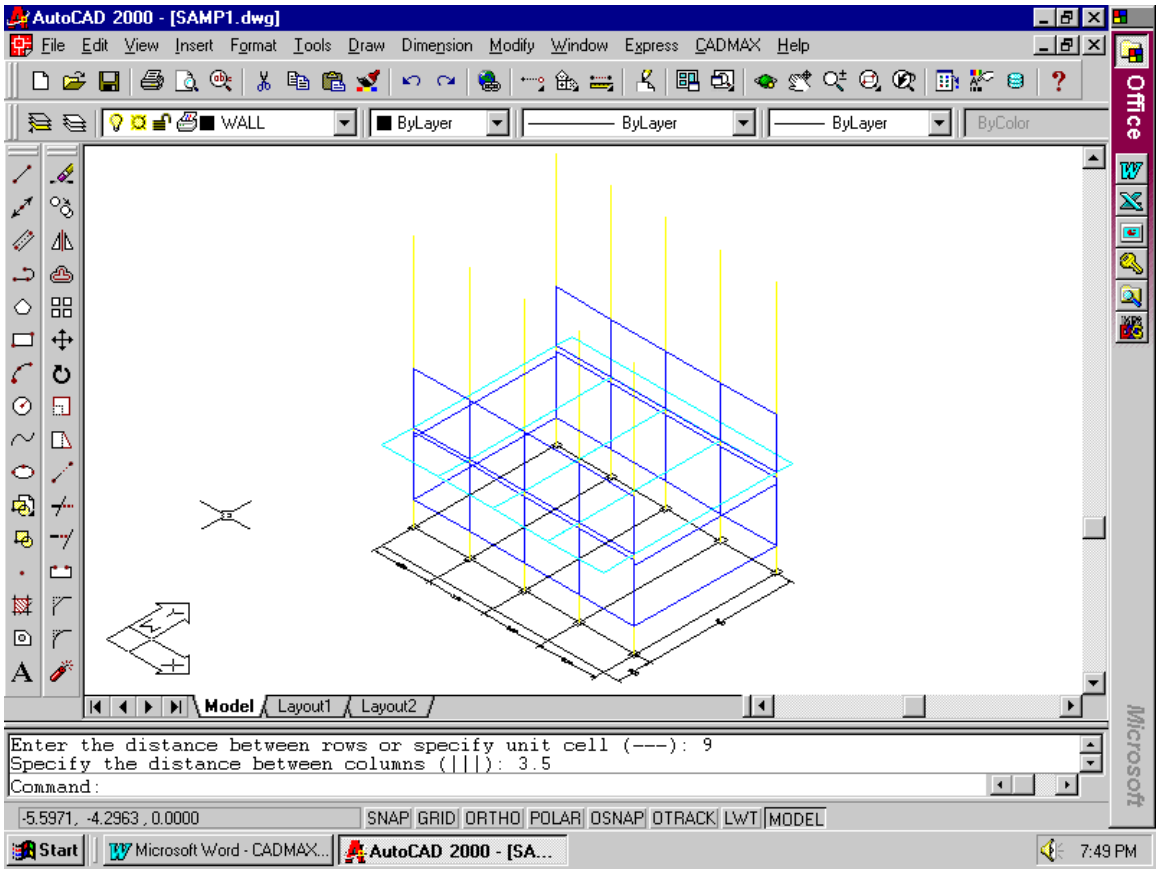
Now you will see the dialog box to fill in the wall opening details and for structural design purpose **select no further openings** option as follows and then click **OK**.



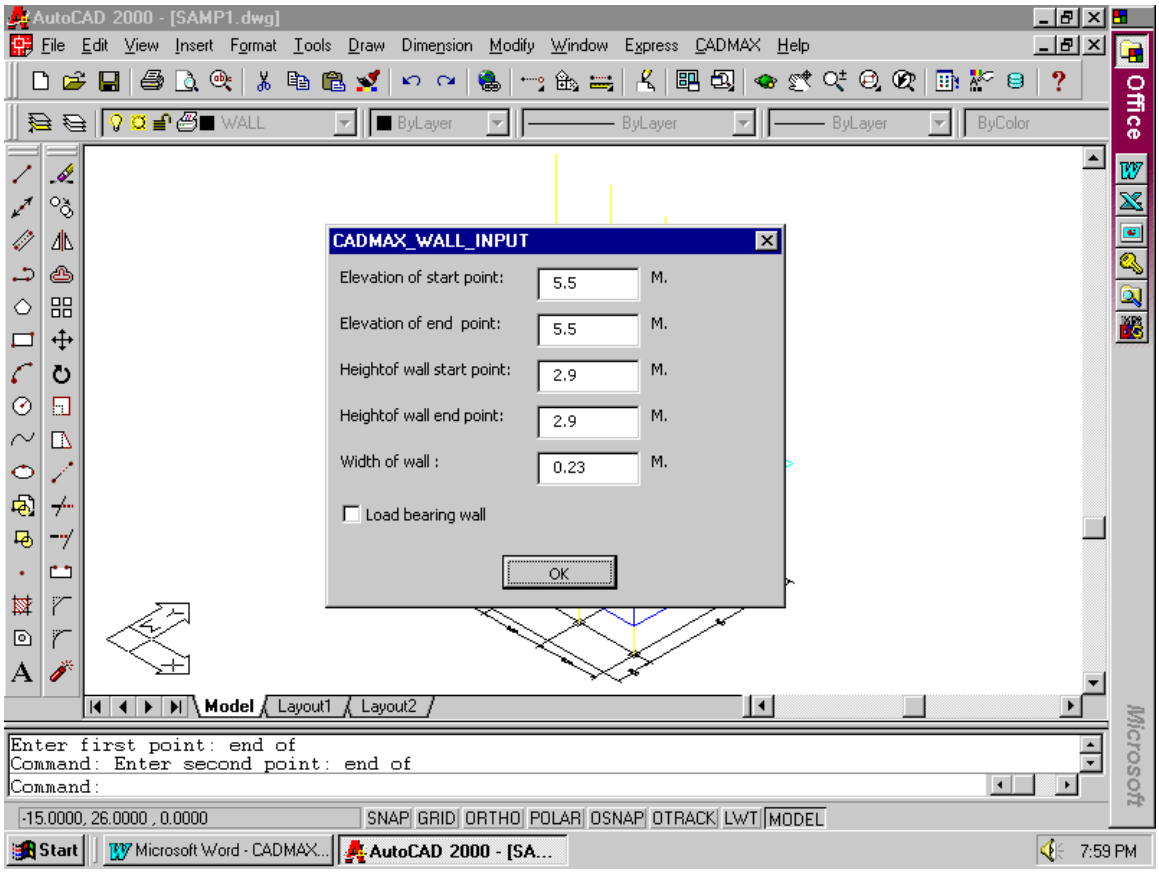
Now you will see a wall in blue color in appropriate location as given below:



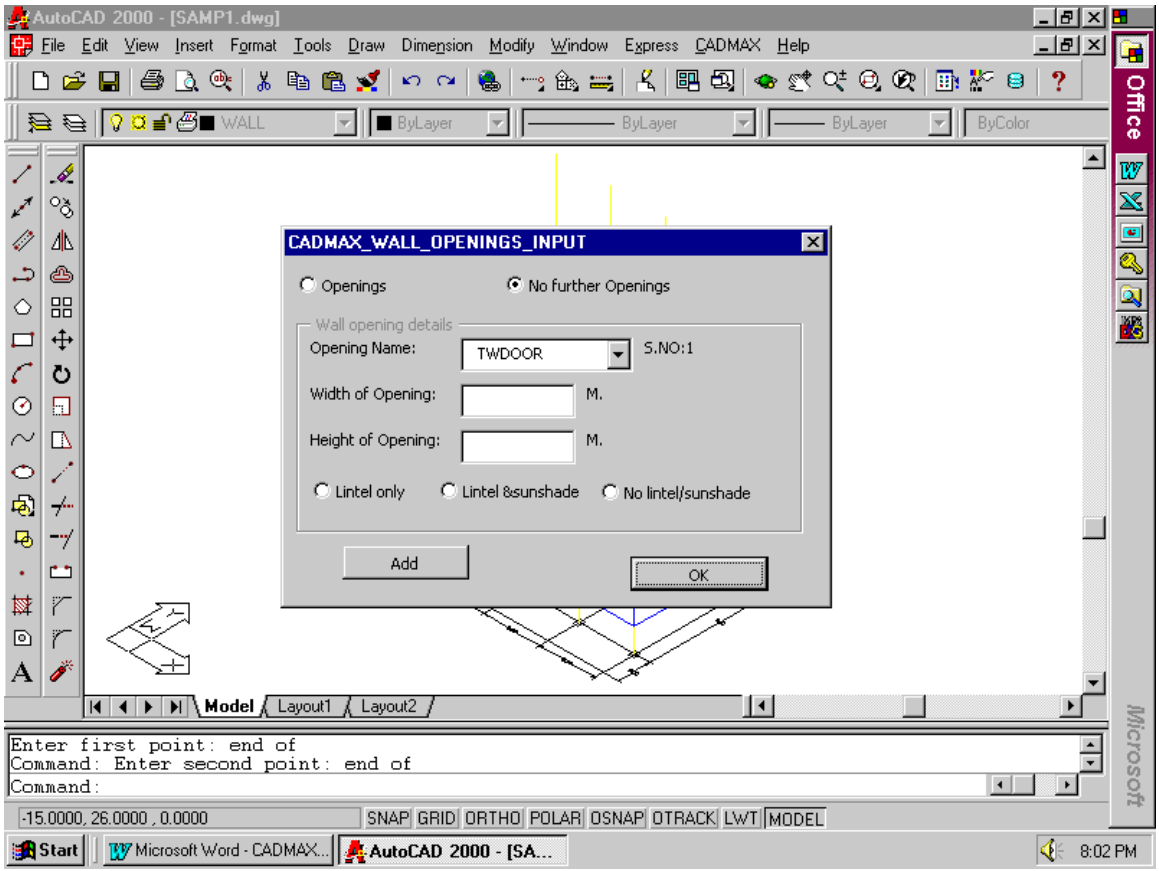
The walls similar to the last drawn can copied to the appropriate places using the array command as follows:
Command: **ARRAY** <enter>
Select objects: **L** <enter>
Select objects: <enter>
Type of array <Rectangular/Polar>:<R>:**R** <enter>
Number of rows: **2** <enter>
Number of columns: **4** <enter>
Distance between rows: **9** <enter>
Distance between columns: **3.5** <enter>
Now you will see that all the floor walls along X-direction is copied to the appropriate places and will be as shown below:



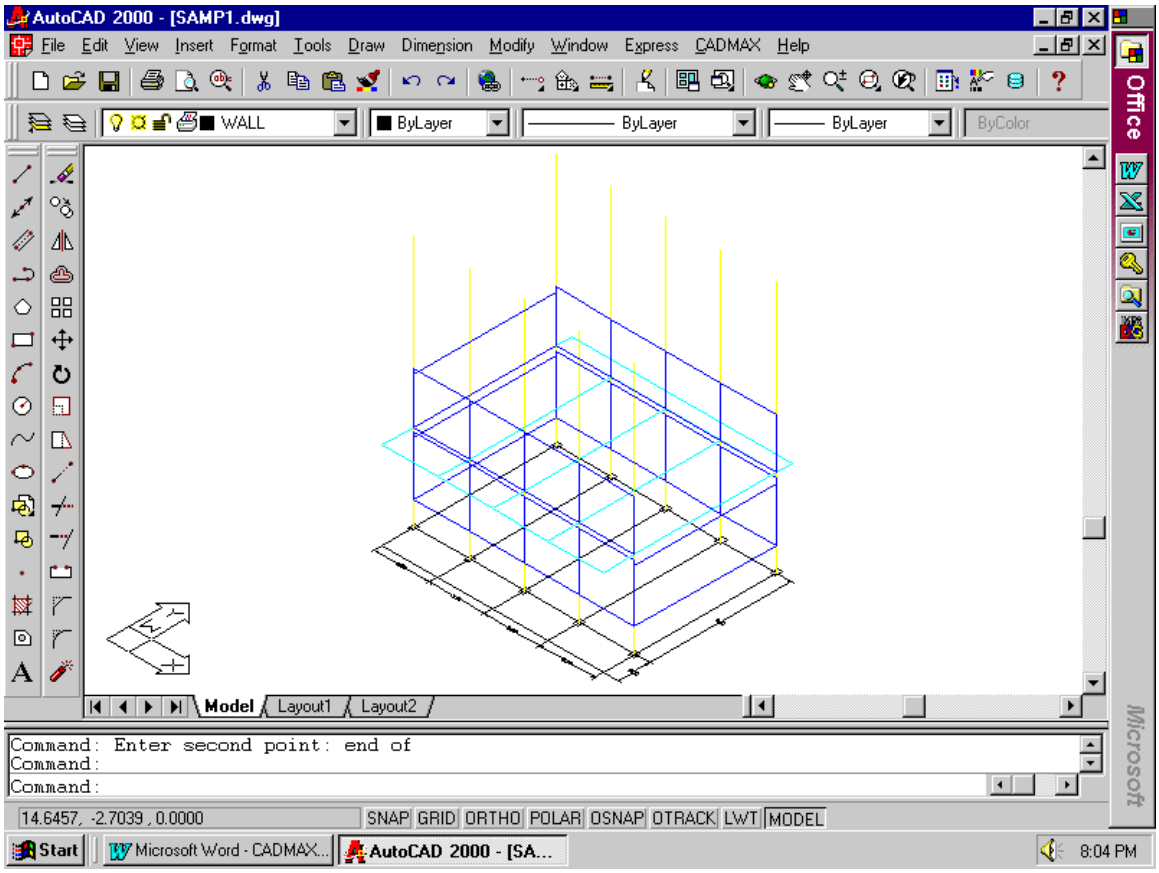
Similarly the walls along Y-direction is drawn as follows:
Pick the **WALL** command from the **CADMAX** Pop-up menu and then enter layer name to draw as **WALL** and then **OK**.
Now you will be prompted to pick the first point and second point as shown below:
Command: **WALL**
Enter first point: <enter first point>
Enter second point: <enter second point>
Now fill in the wall details dialog box as shown below: **The Elevation of start point and elevation of end point as 5.5m and the height of wall at start point and the height of wall at end point as 2.9m(3.5-0.6=2.9m).** The width of wall is filled in as 0.23m. Then click **OK**.



Then click the wall opening details as **No further openings option** as follows and then click **OK**.

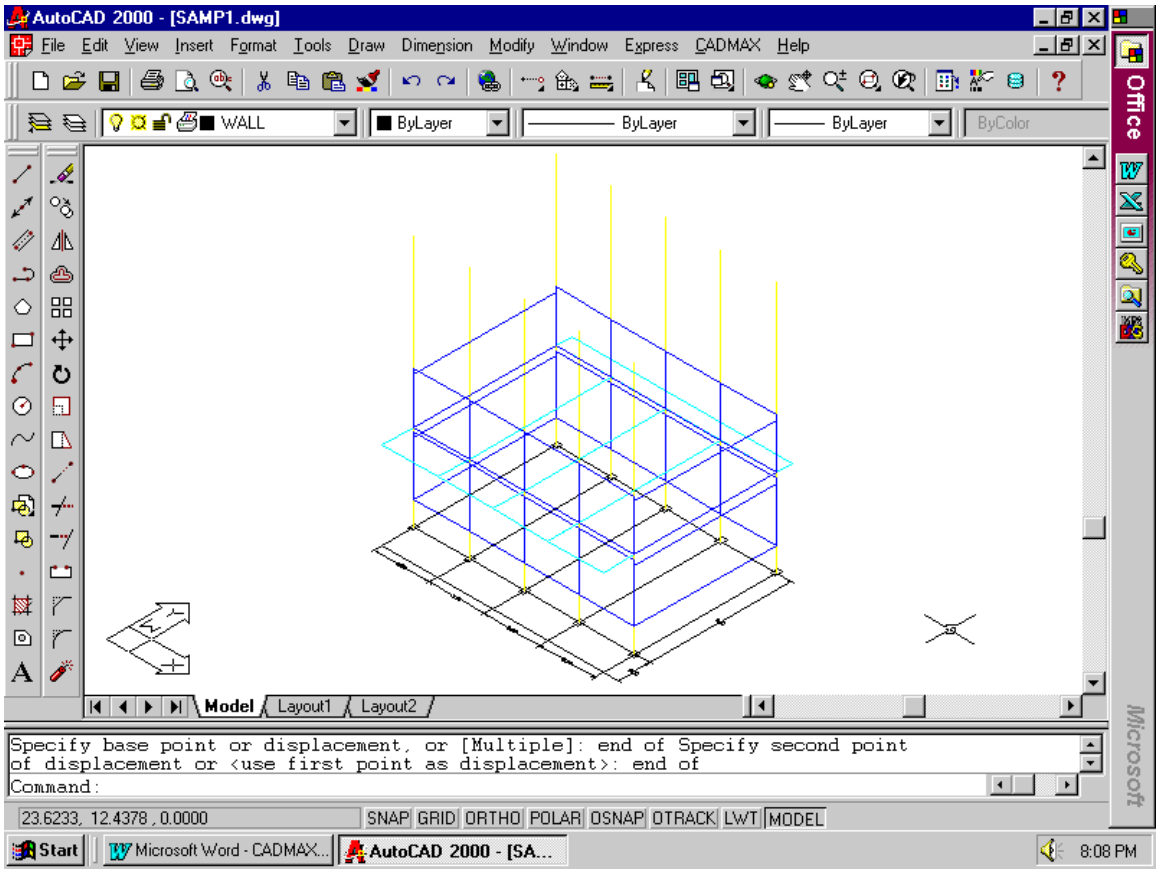


Now you will see the wall in blue color is inserted in the appropriate place as shown below:



The last drawn wall can be copied to the other end using the copy command as follows:
Command: COPY <enter>
Select objects: L <enter>
Select objects: <enter>
Enter base point: <pick the base point>
Enter second point or displacement:<pick the second point to copy to the other end>

Now you will see the wall is copied to the other end and will be seen as follows:



Now you can enter parapet walls in the front projections using the wall command as follows:

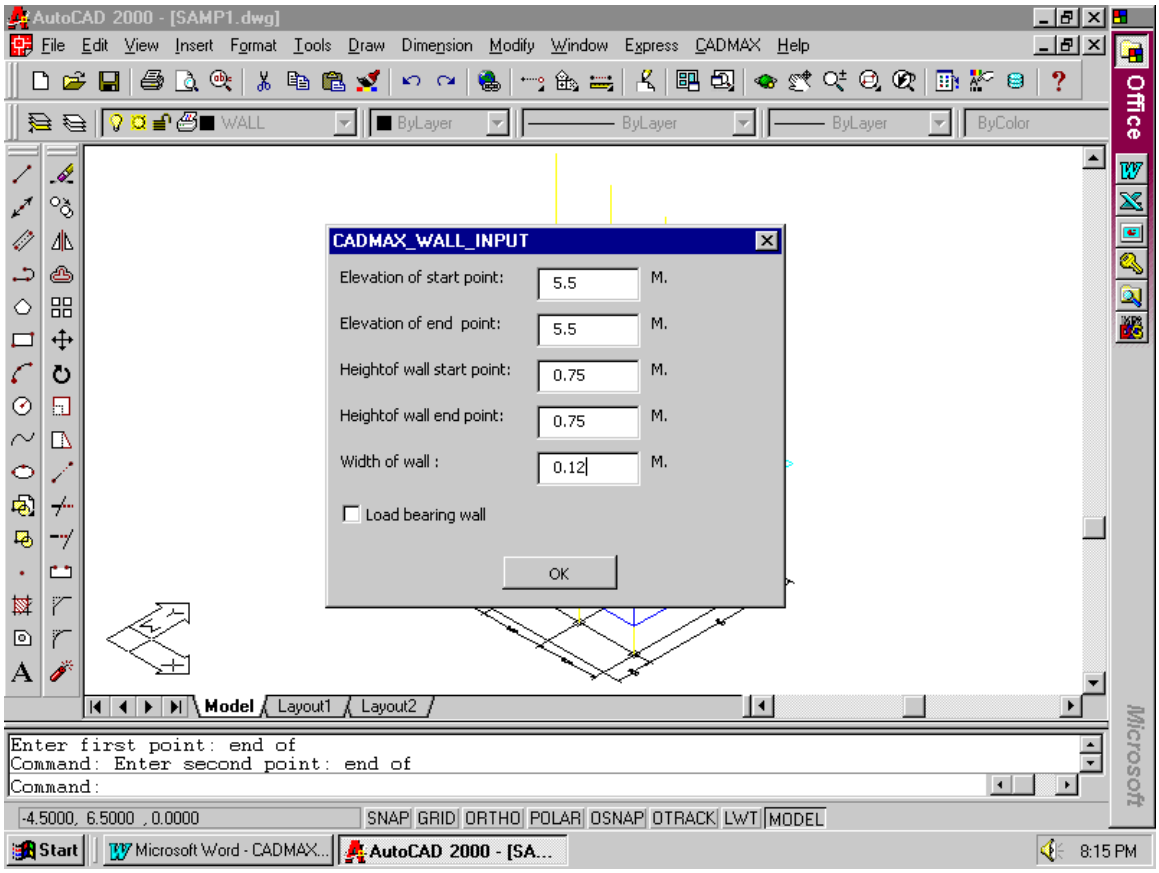
Pick the **WALL** from **CADMAX** Pop-up menu and enter the layer name to draw as **WALL** and then pick the start point and end point as follows:

Command: **WALL**

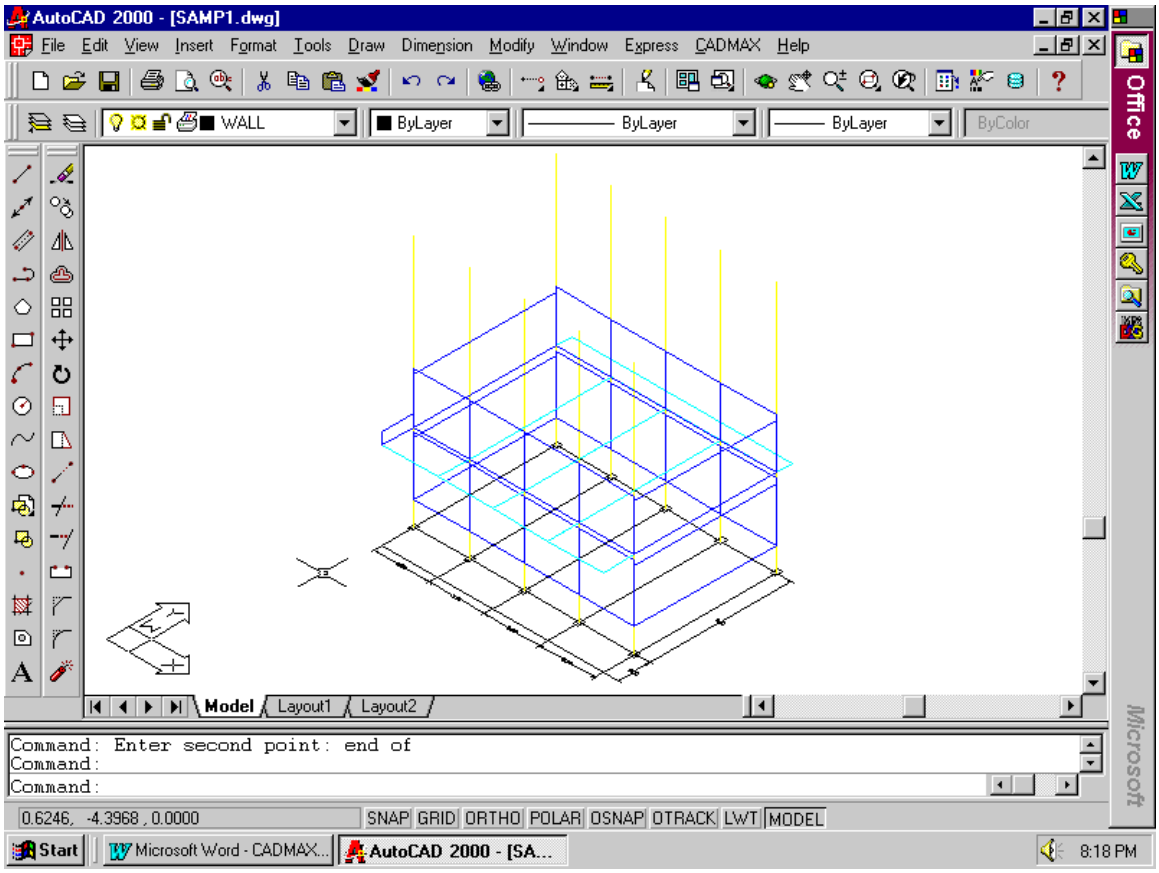
Enter first point: <pick the start point>

Enter second point: <pick the end point>

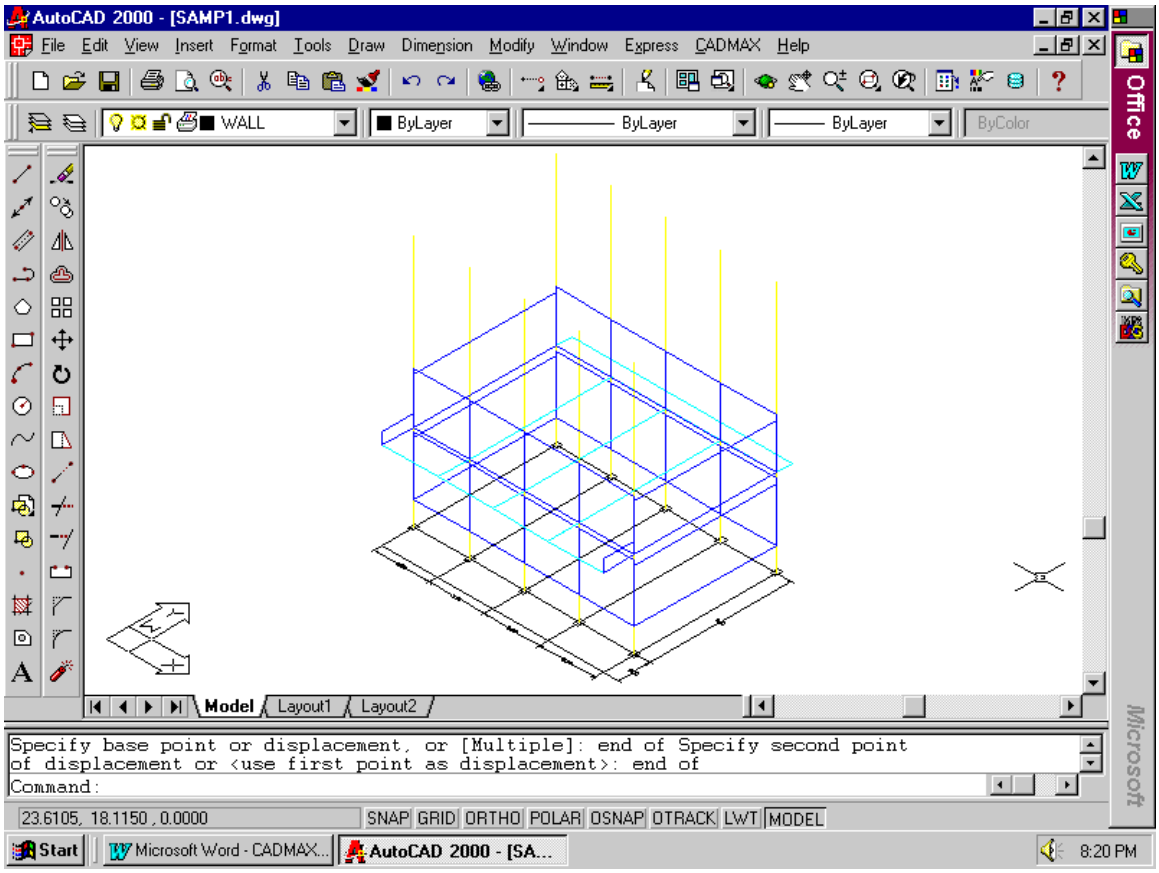
Now fill in the following wall details dialog box as follows:



For the next dialog box for wall opening details select the No further openings option in the wall opening details and then click OK as done previously.
Now the wall is inserted at the appropriate place and will seen as follows:

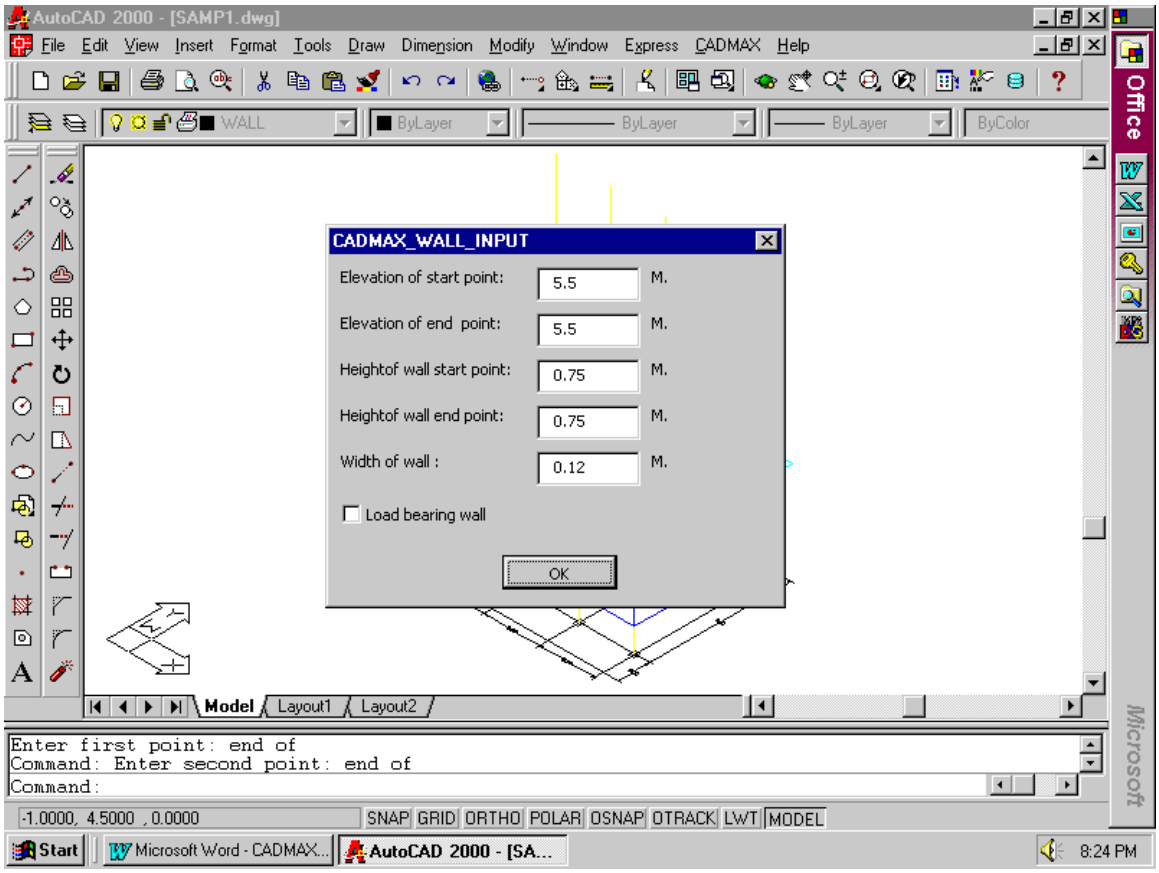


Copy the last drawn wall to the other end using copy command as done previously. Now the drawing will be seen as follows:

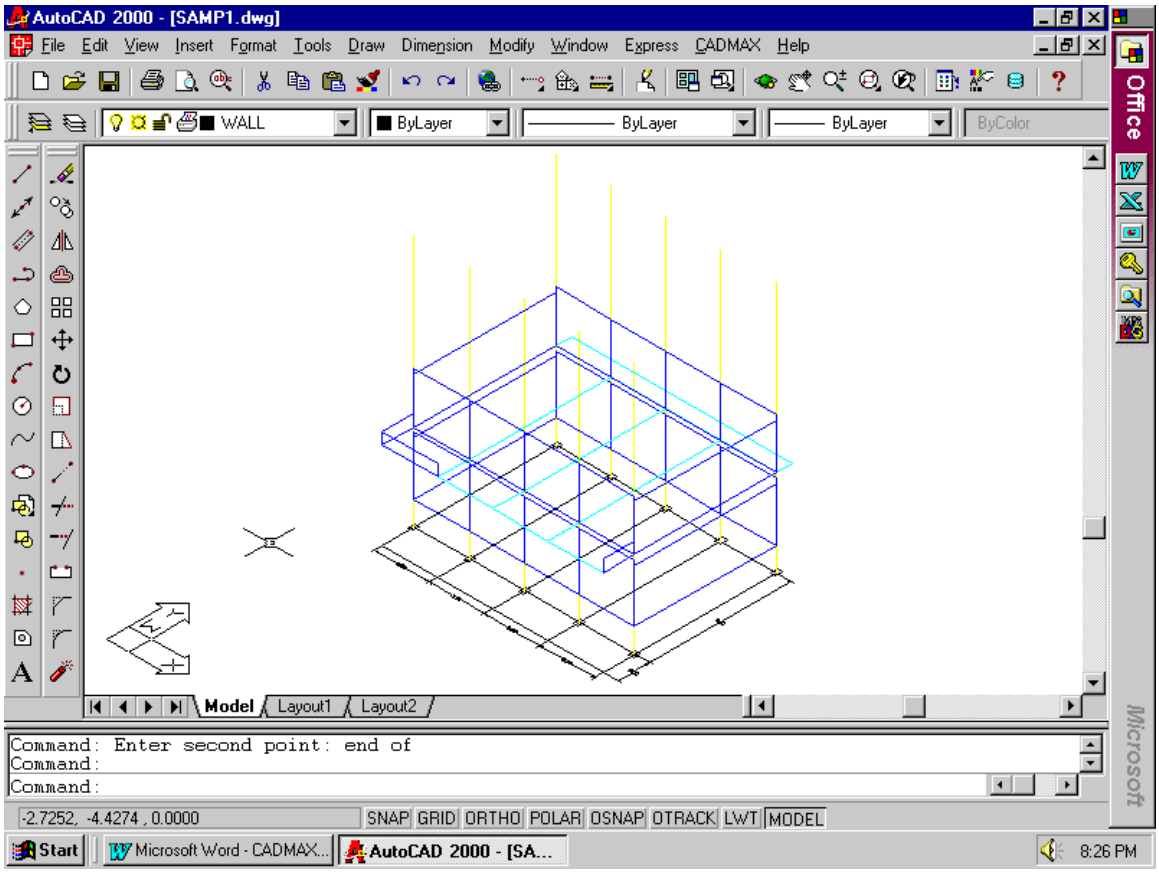


Now enter edge walls using the wall command in the same layer WALL as follows:

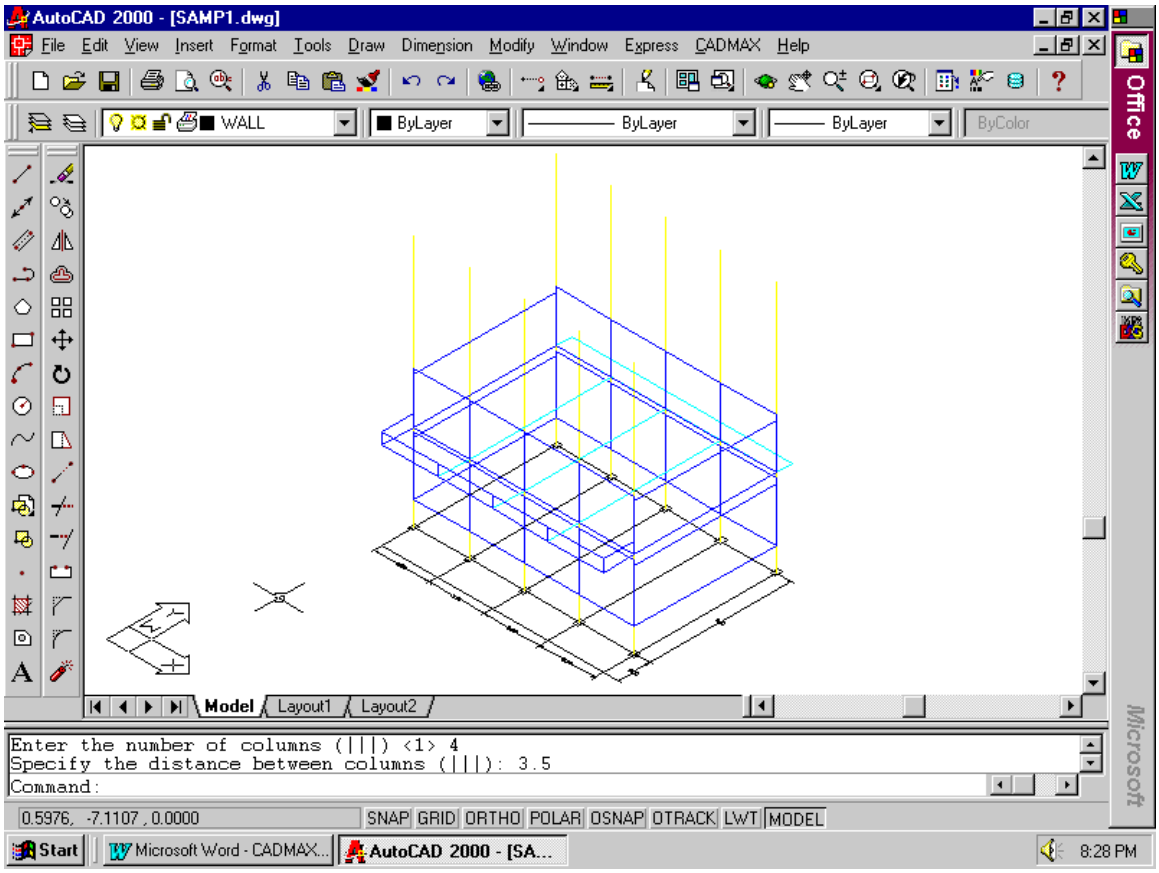
Command: **WALL**
Enter first point: <pick the start point>
Enter second point: <pick the second point>
Then fill in the following dialog box as follows:



Enter No futher openings option in the wall opening dialog box and then click OK as done previously. Now you will see the wall drawn in the appropriate place as shown below:

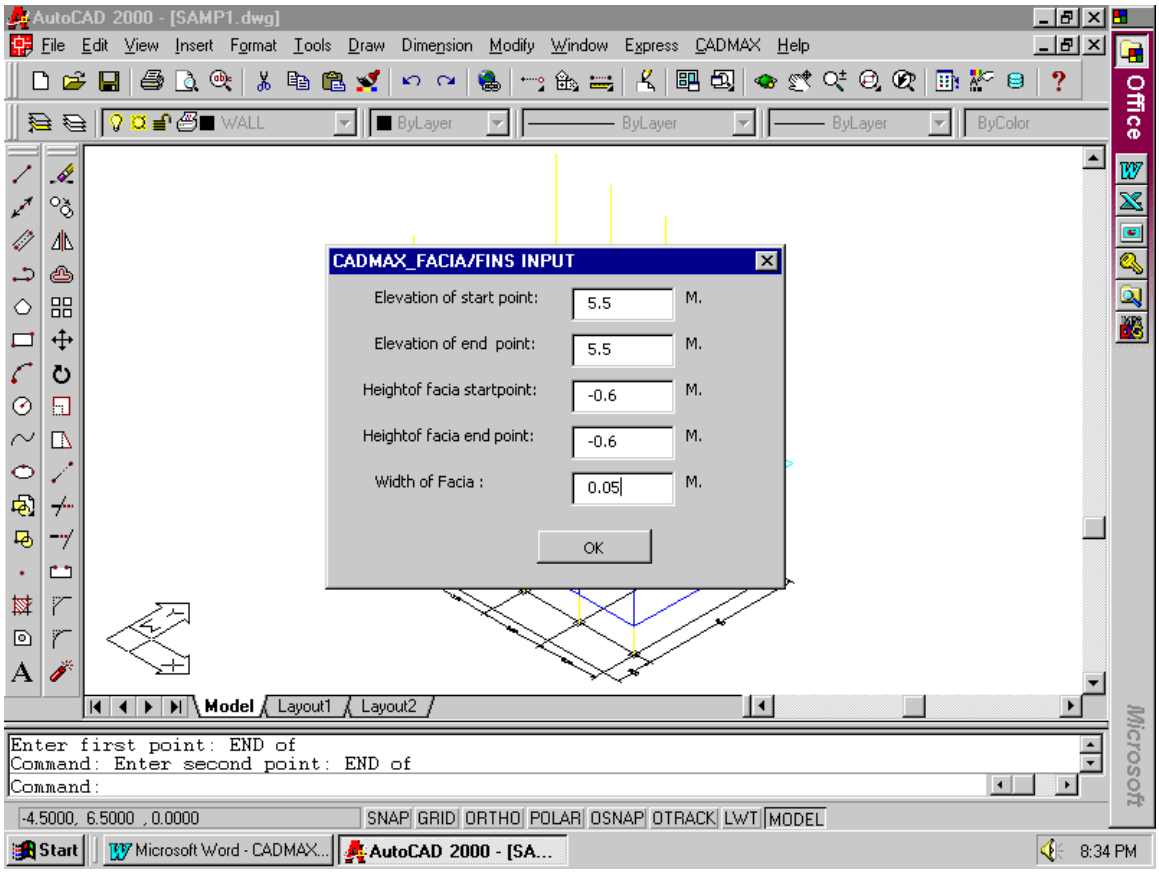


To copy the last drawn wall to the similar locations use the array command as done previously and now you see the drawing as follows:

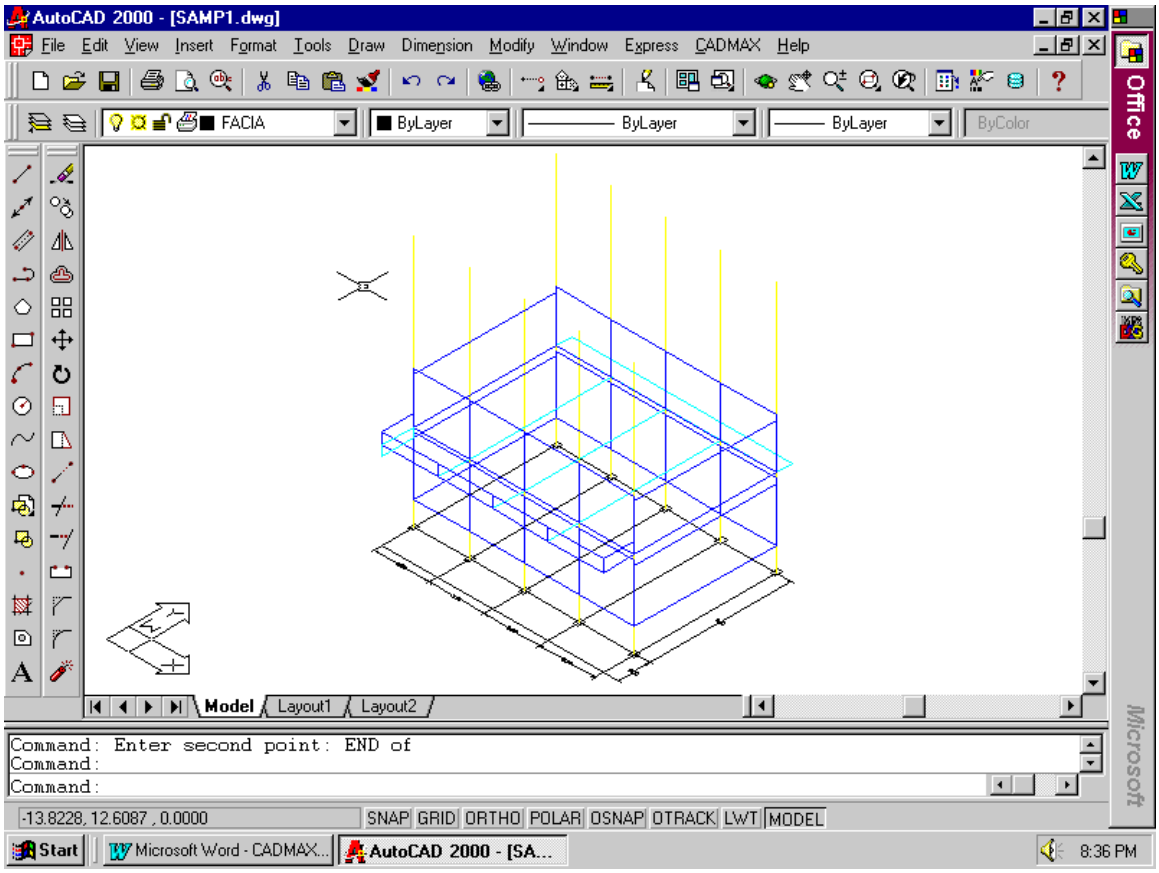


Now enter facia details on the edge beams as follows:

5.Pick the **FACIA** command from the **CADMAX** Pop-up menu and then enter the layer name to draw as **FACIA**.
Command**: FACIA**
Enter first point: <pick the first point>
Enter second point: <pick the second point>
Now enter the facia details dialog box as follows:

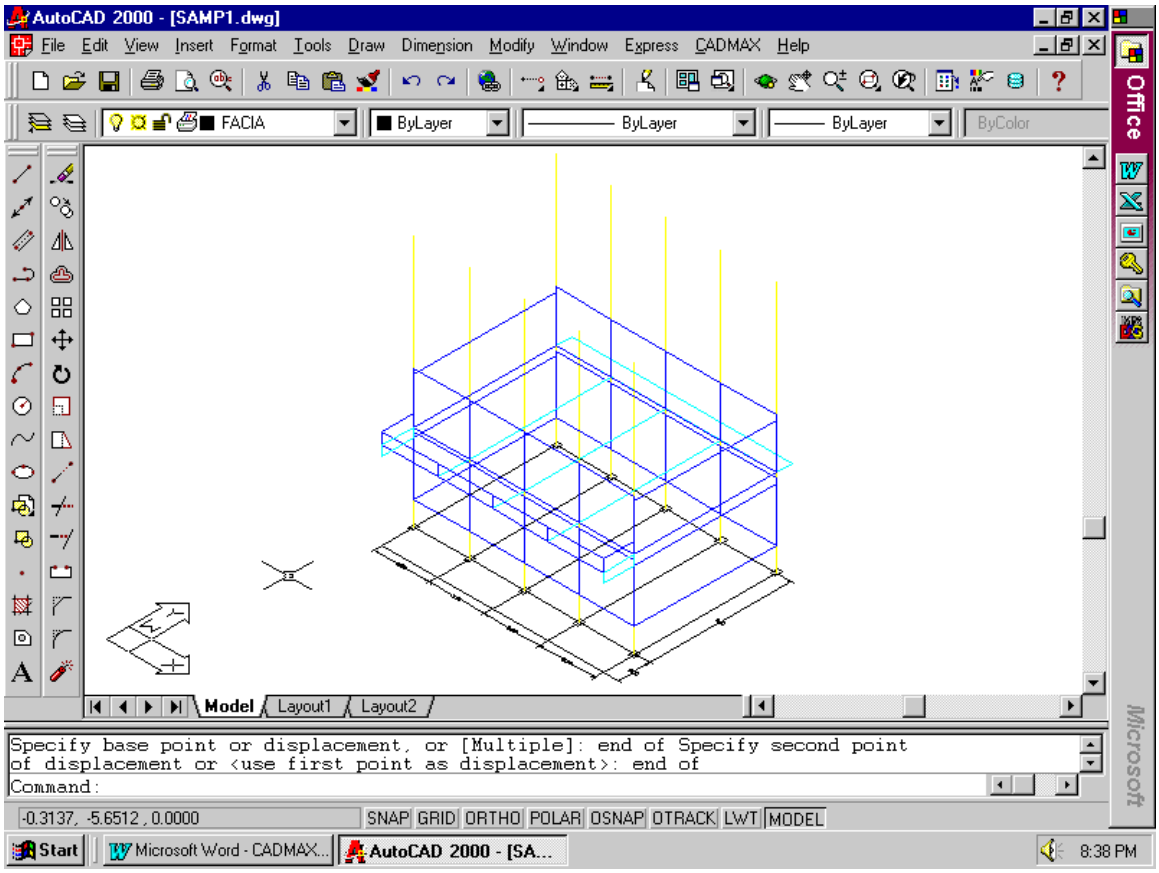


Now the facia in the appropriate location is drawn as follows in cyan color:

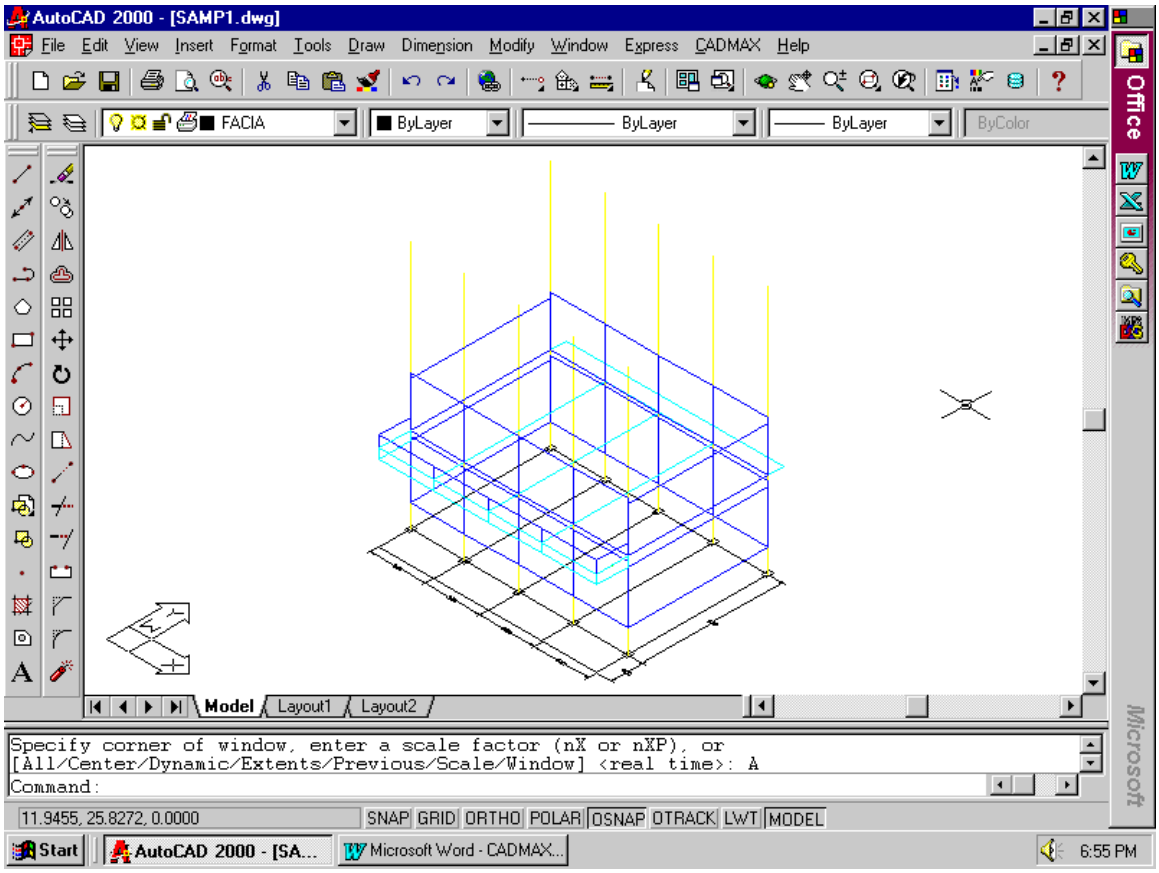


Now copy the last drawn facia to the other end using copy command as done previously.

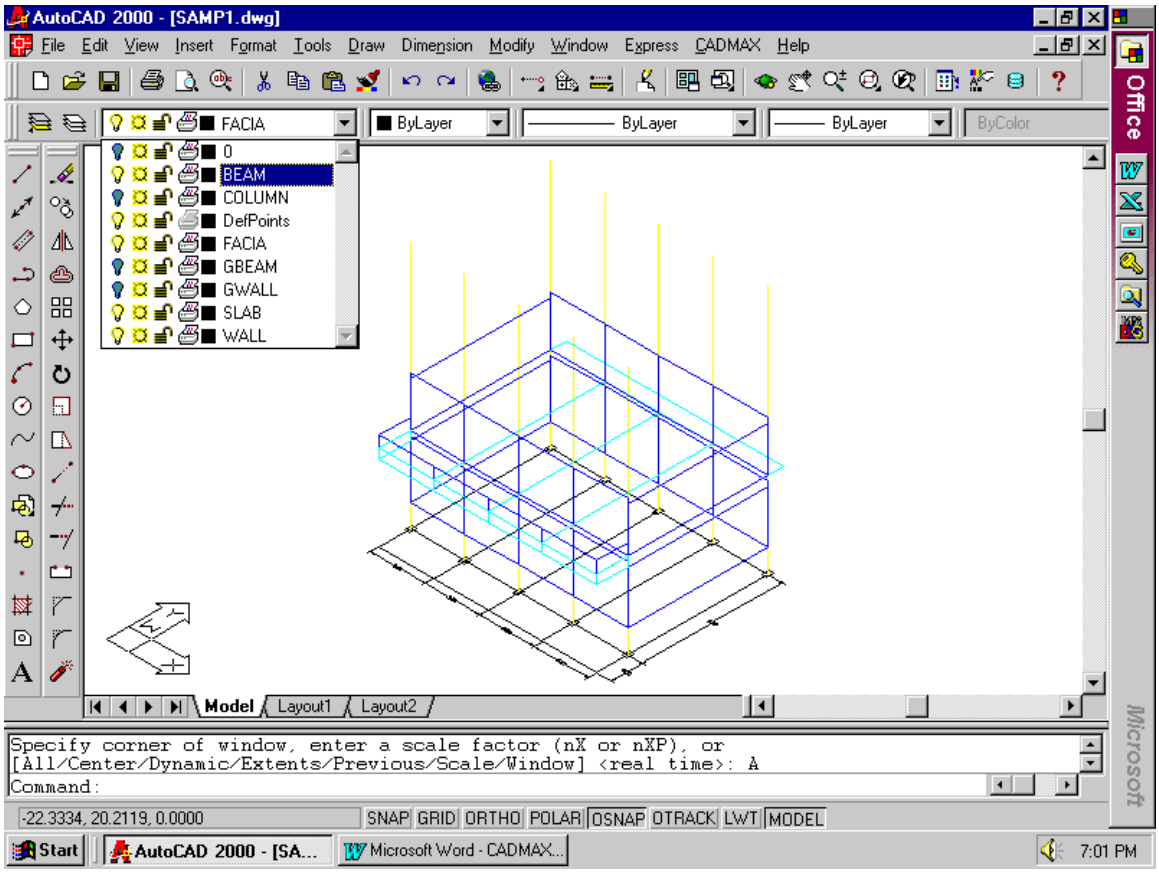
Now the drawing will be seen as follows:



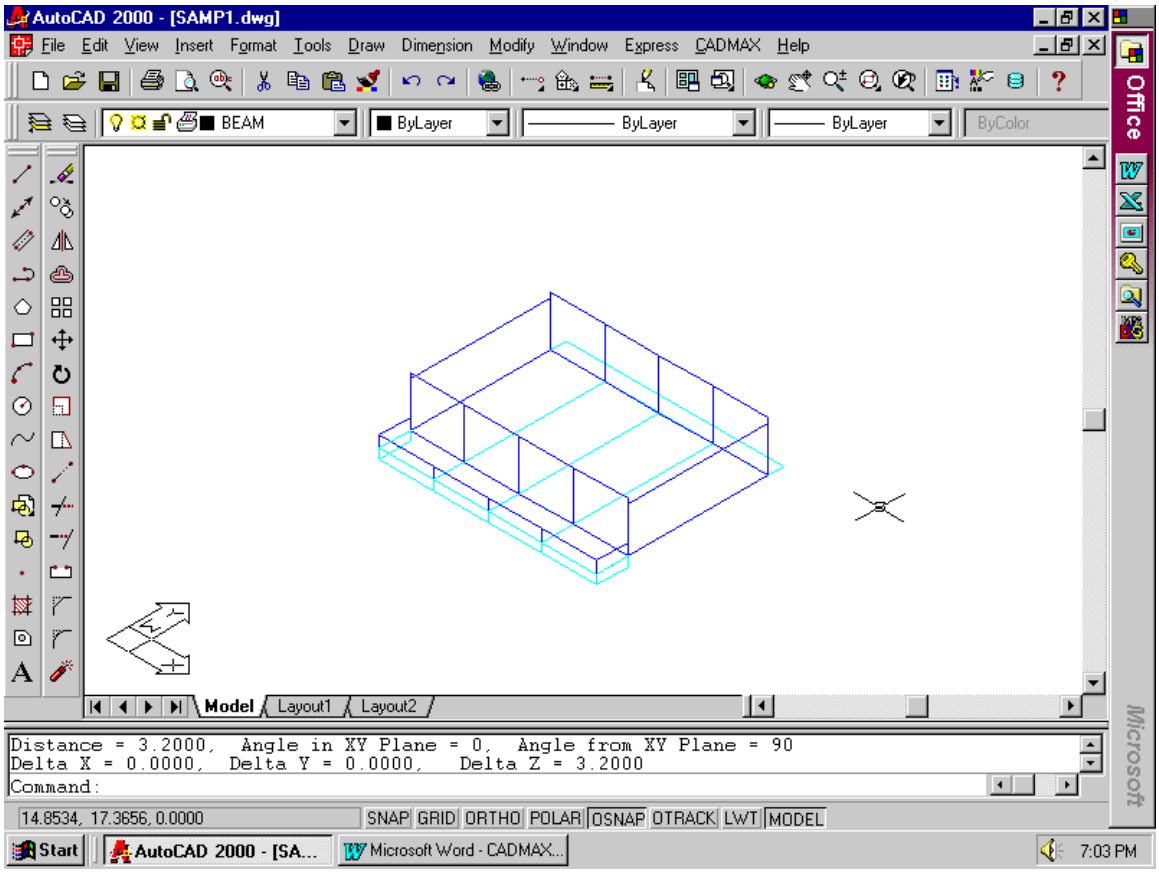
Similarly you can draw facia in the required places in the edges as done previously and then the drawing will be as shown below:



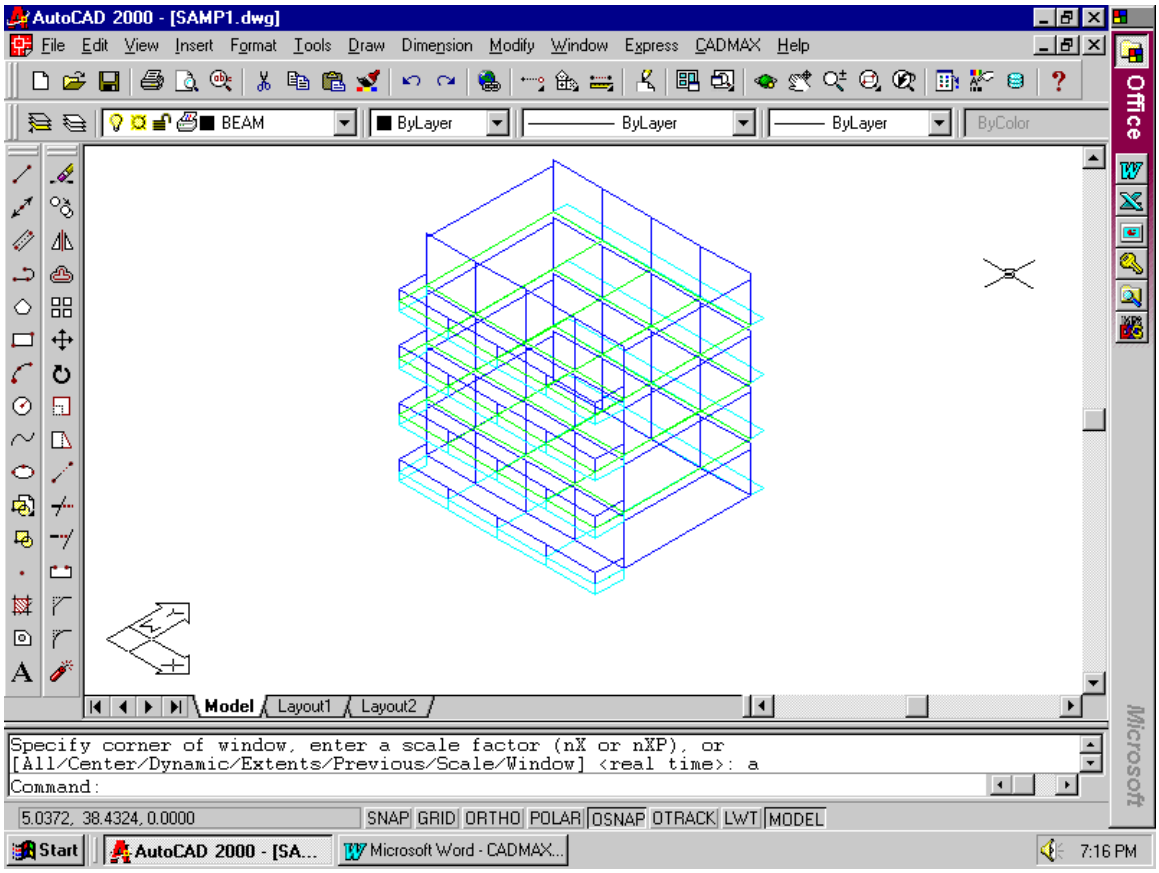
Now you can copy slabs,beams,walls,facia etc. to further floors by making off the layers "0,GBEAM,GWALL,COLUMN". The members to be copied to the other floors will be as shown below by making off the above mentioned layers.



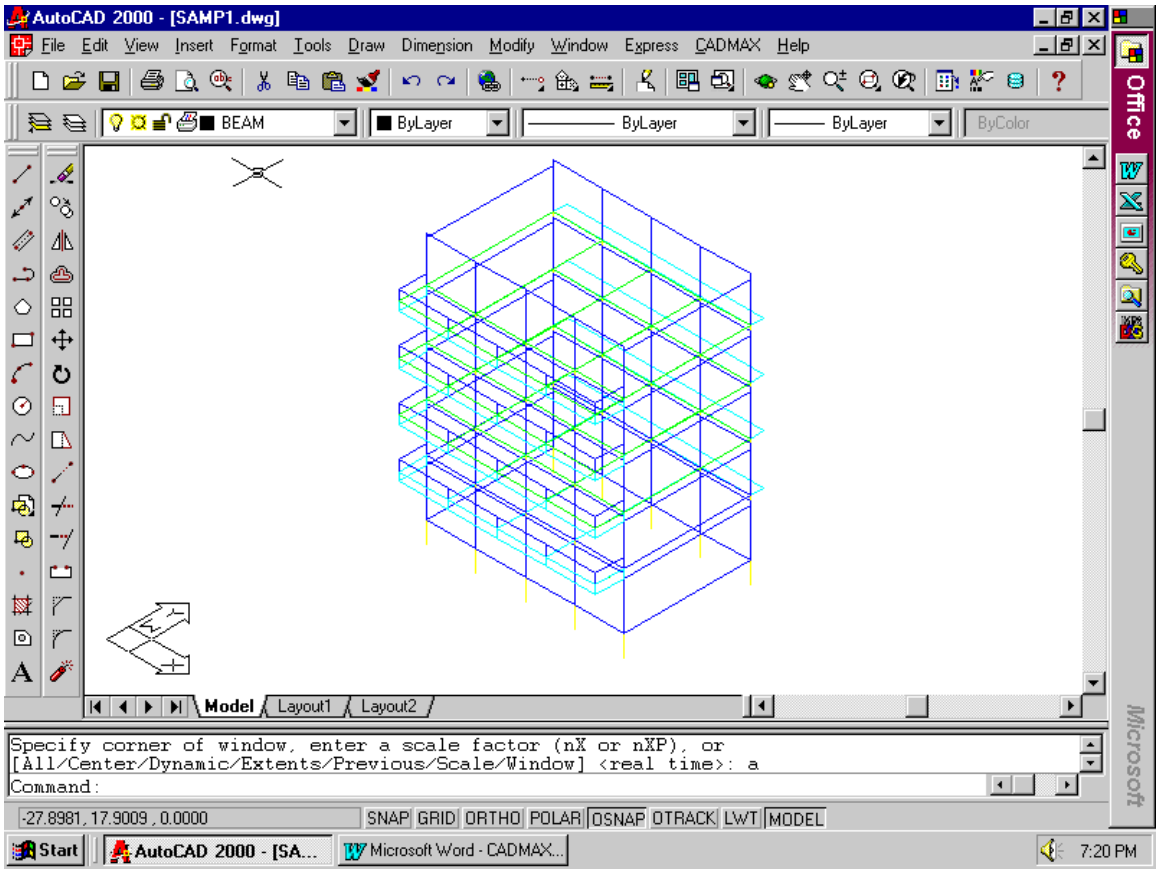
Now you copy the displayed objects at every floor heights as follows:



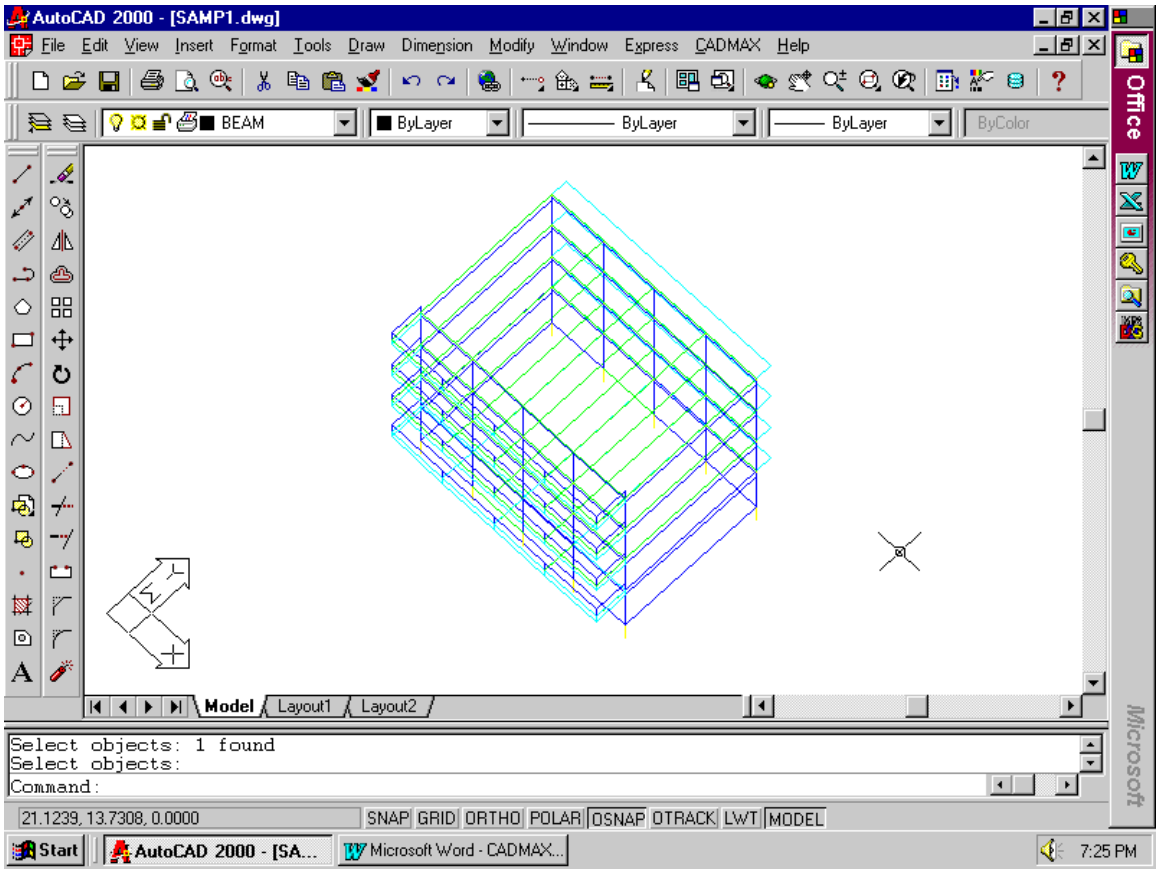
Command: **COPY** <enter>
Select objects: **W** <enter>
Enter first corner: <pick the first point>
Enter other corner: <pick the other corner so as to select all the displayed objects>
Enter base point or [Multiple]: **M** <enter>
Enter base point: **0,0,0** <enter>
Enter second point: **0,0,3.5** <enter>
Enter second point: **0,0,7.0** <enter>
Enter second point: **0,0,10.5** <enter>
Enter second point: <enter>
Now you will see the structure as shown below:



Now you can turn ON the layers “COLUMN,GBEAM,GWALL” for proceeding further. Now the drawing will be as shown below. Please note that the layer “0” is turned “OFF” for further analysis purpose.

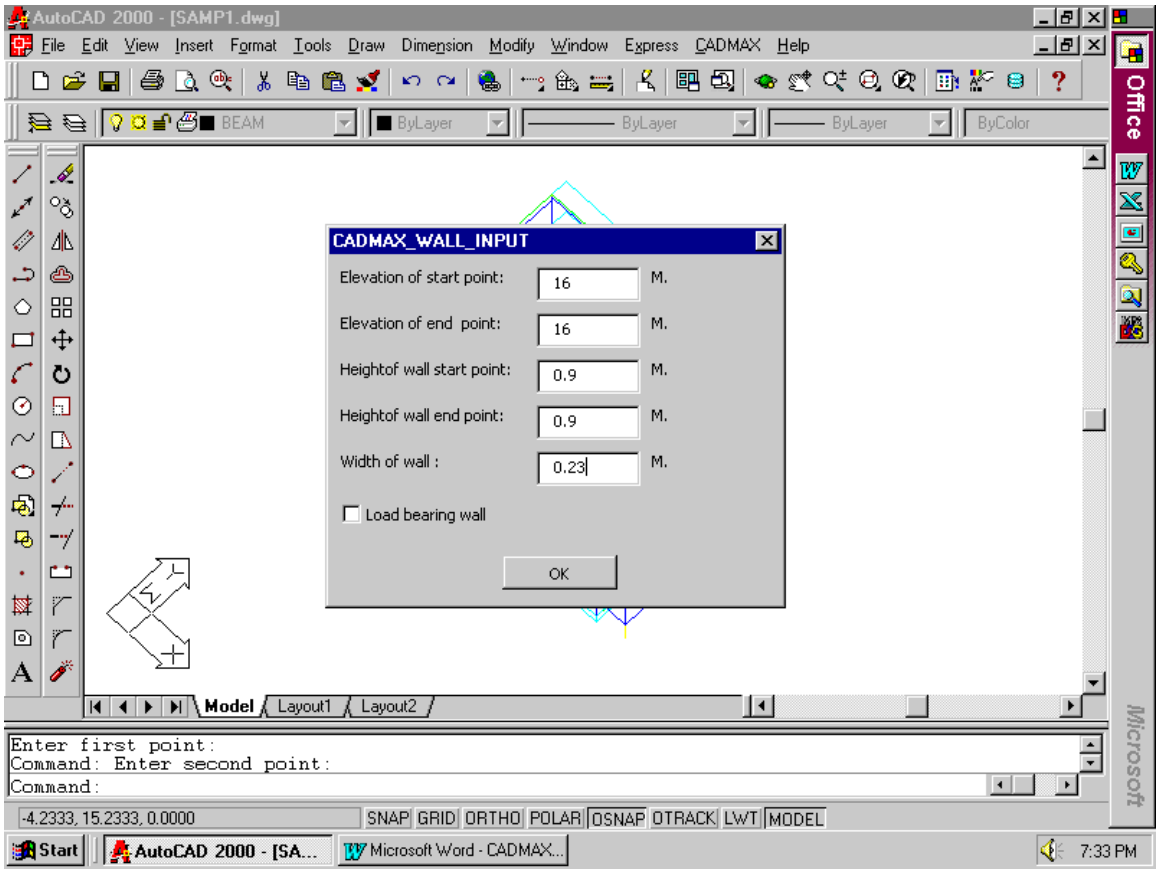


In the drawing shown now you will see high walls in the top floor instead of parapet walls. To make that changes erase high walls in the top floor and the drawing will be as shown below. To get a clear view of the structure you can give **"VPOINT"** command and change the vpoint as **"1,-1,3"** to get a following view.

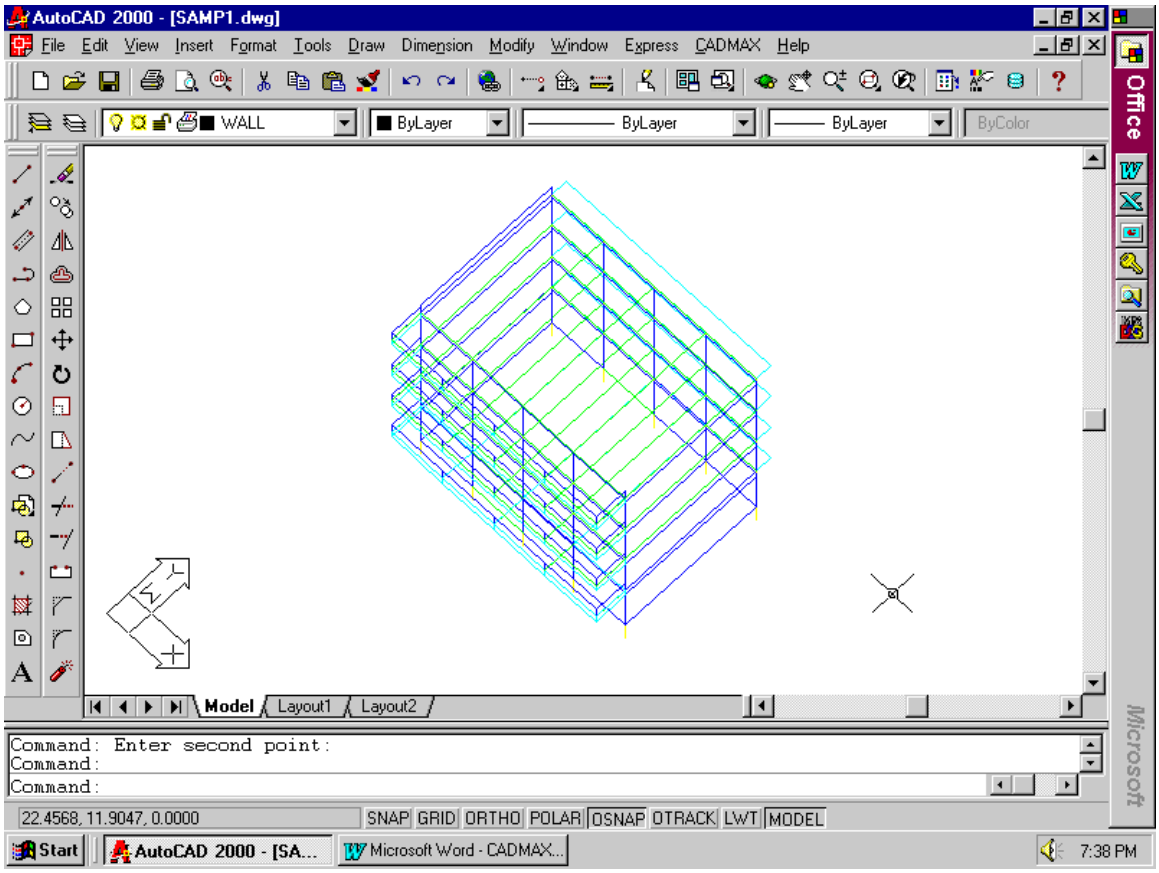


Now insert parapet walls at required places using **WALL** command from the **CADMAX** popup menu.

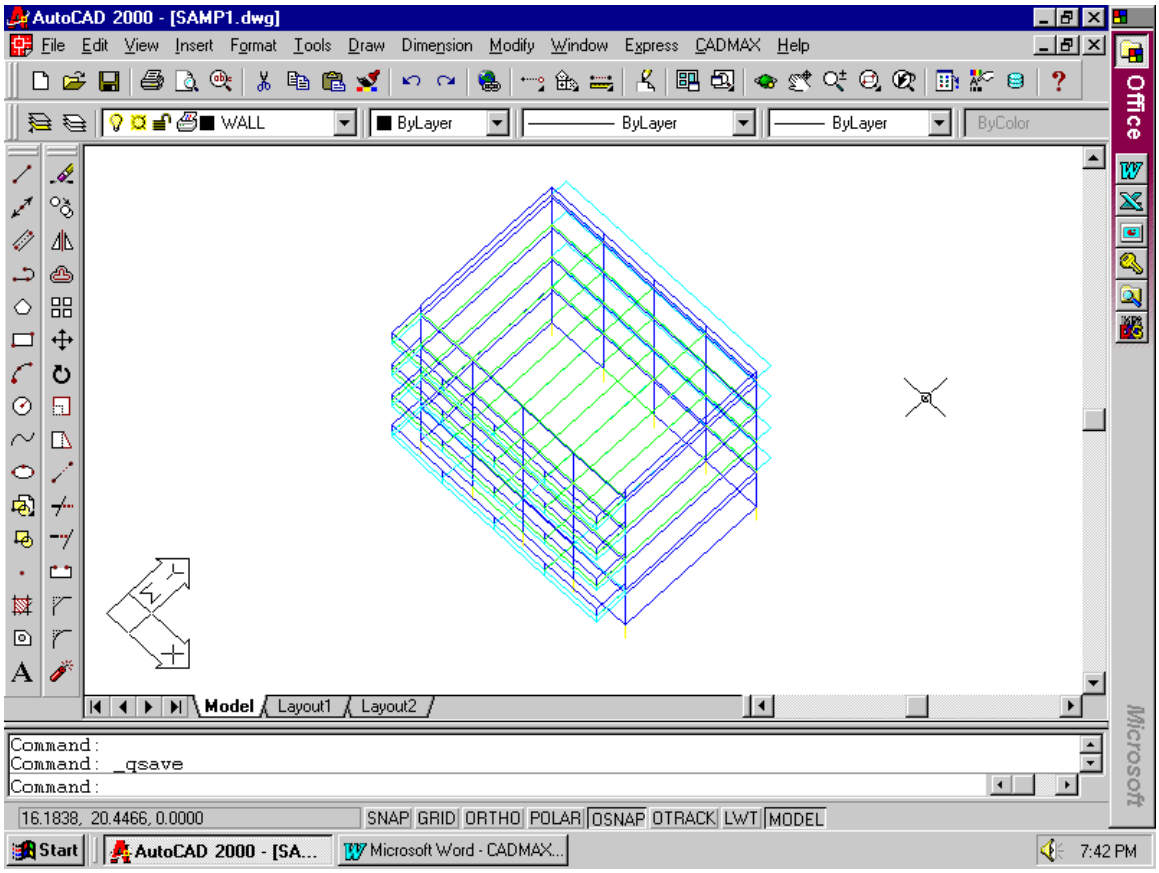
Command: **WALL**
(Enter layer name to draw as WALL and then OK.)
Enter first point: <pick the first point>
Enter second point: <pick the second point>
Now enter the wall details dialog box as follows and then click OK.



Now pick **"No further openings"** option in wall opening details dialog box. Now you will see the parapet wall as shown below.



Now you copy the last drawn wall to the right end using copy command. And you can draw similar parapet walls in rear side also using the same wall command and then copy or array command. Now you will see the final structure as shown below.



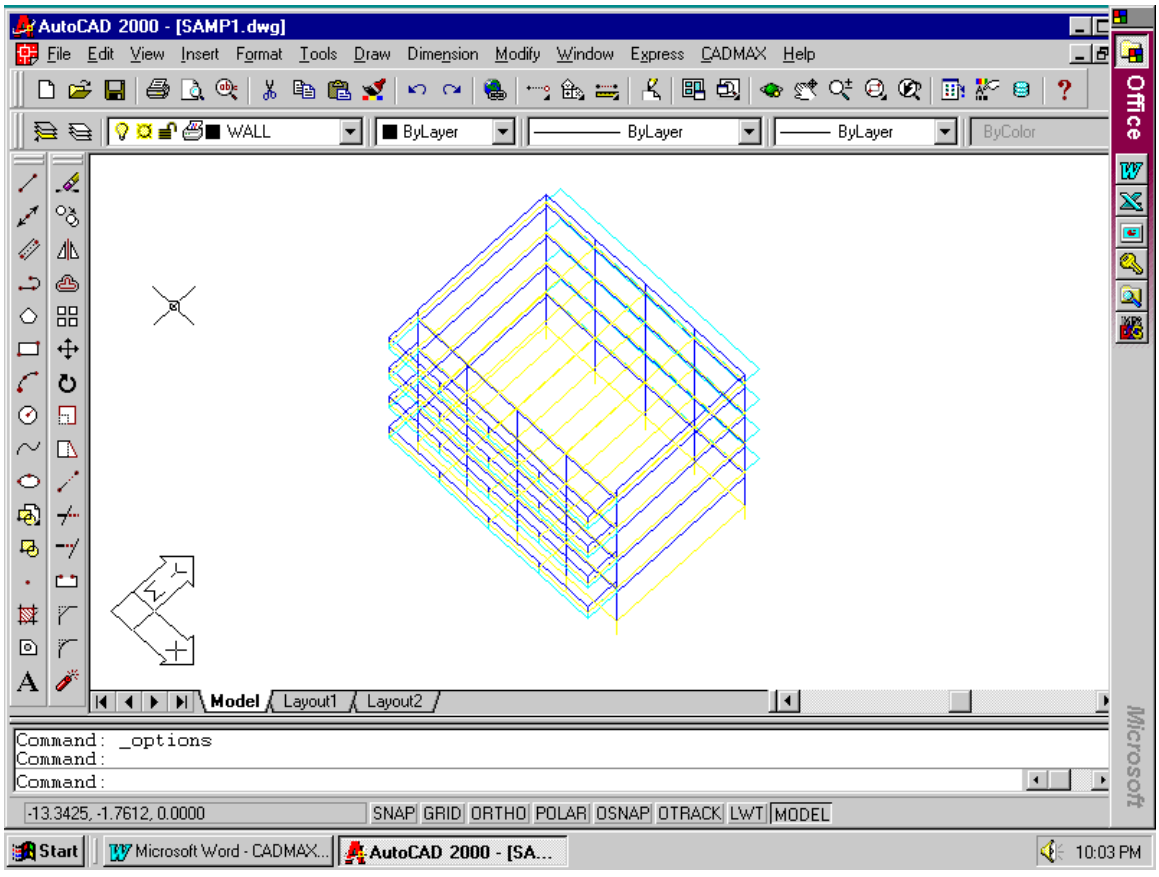
Now you have completed inserting all the members of the structure and the structure is ready for analysis. Now save the drawing and proceed to the next chapter.

CHAPTER.3.ANALYSIS OF THE STRUCTURE.

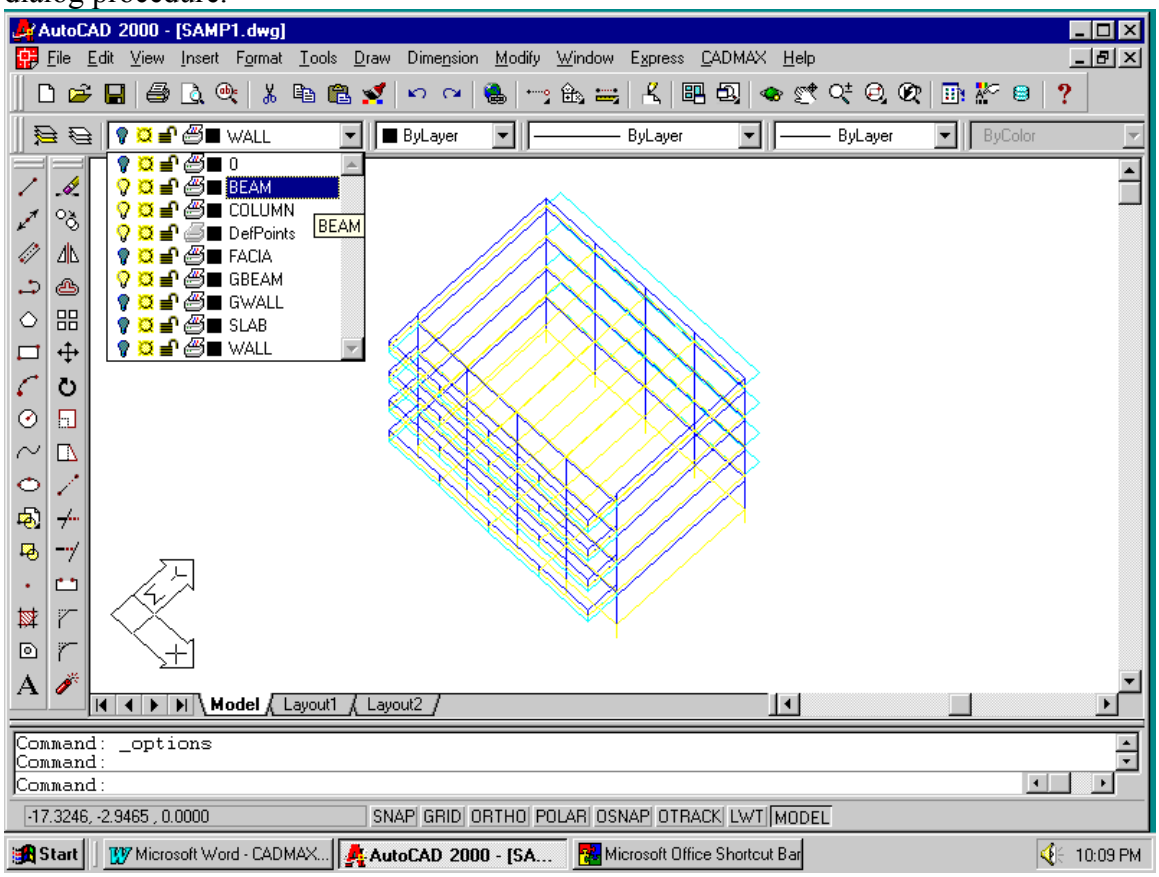
In general the structural analysis of the building is done by stiffness method by the CADMAX Software. You can use it for plane frame analysis or three dimensional analysis as you like it. The loads tranfered to the beams from slabs and walls are done automatically using beamload command. Afterwards analysis is made very simple. The following procedure will explain the steps involved in the analysis of the structure developed in the present worked out example.

6.Pick the **BEAMLOAD** command from the **CADMAX** popup menu.
Or Give the **BEAMLOAD** command at the command prompt.

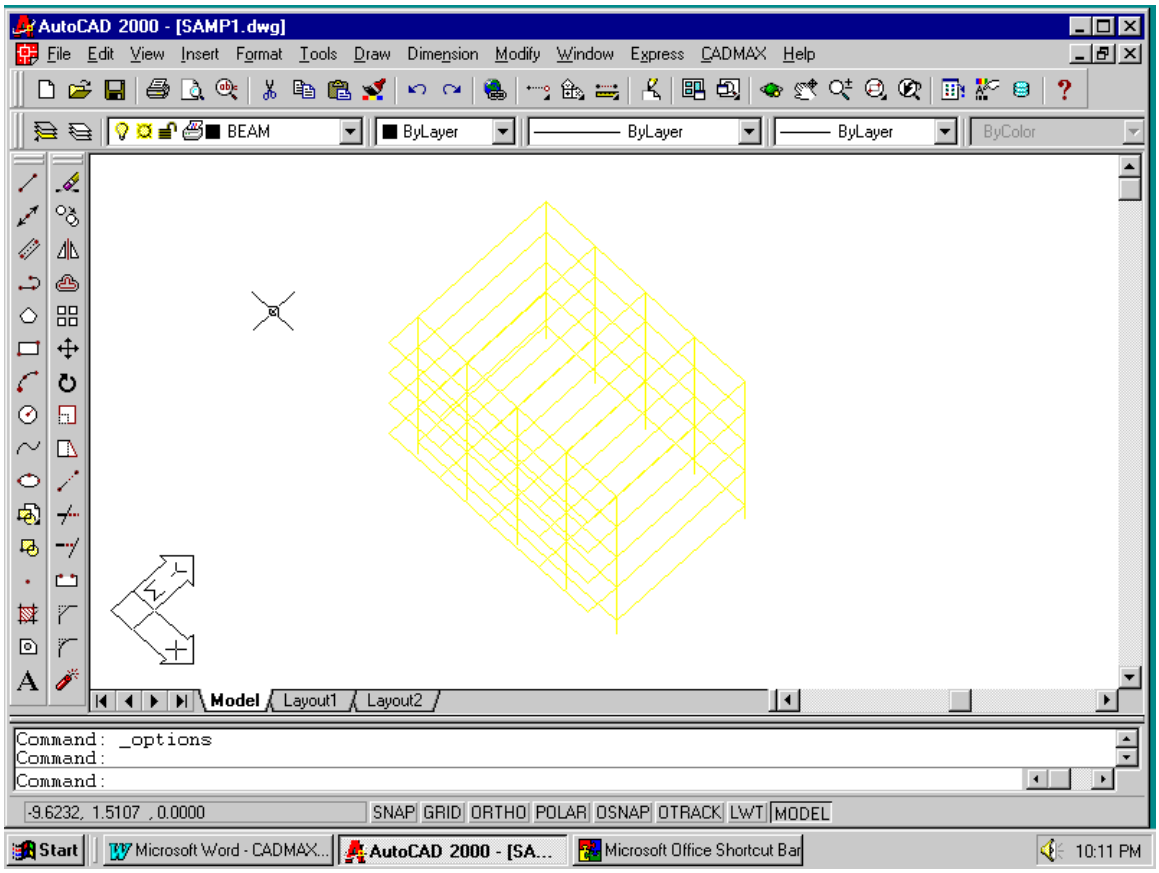
Command: **BEAMLOAD**
Select Objects: <select all the objects on the display>
(Now it will take few minutes to take the loads from the walls and slabs and then all selected grips will be released at the same time the color of the beams will be changed into yellow color. Now all the loads from slabs and walls are transferred to all the beams in the structure.)
Now the structure will be as shown below:



Now the structure is ready for analysis. Before analysis of the structure as a whole, the edge beams in the front side of the structure is to be analysed to get the point loads trasmitted to the cantilever beams. Since the loads from walls and slabs were transferred to the beams the layers containing walls,slabs,facia etc. may be off. Using the following dialog procedure.

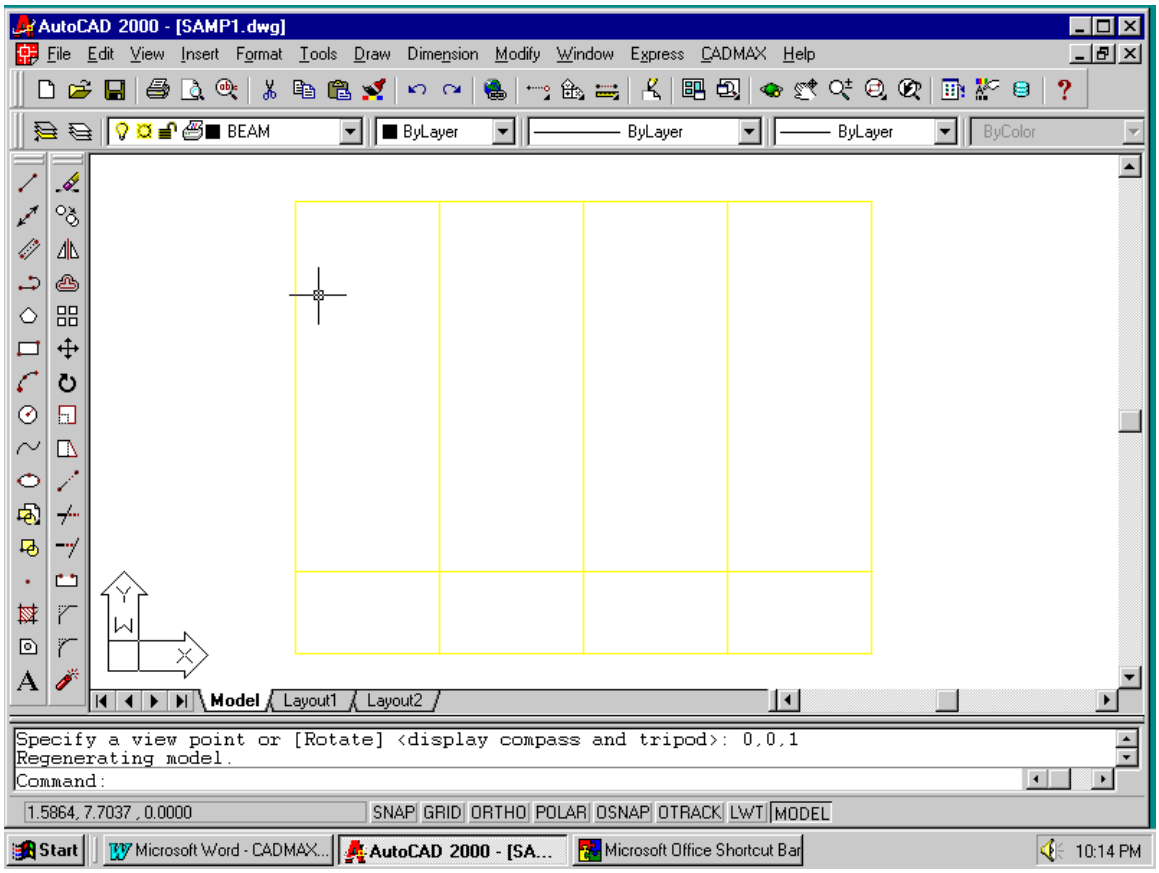


After making off the above layers the stucture will look like as shown below:



To analyse the beams along edge beams only, use the **vpoint** command to get the plan view of the structure.

Command: **VPOINT** <enter>
Enter vpoint:<1.000,-1.000,3.000>: **0,0,1** <enter>
Now you will get the planview of the structure as shown below:



Now to analyse the edge frame only as continuous beams transferring loads to the cantilever beams.

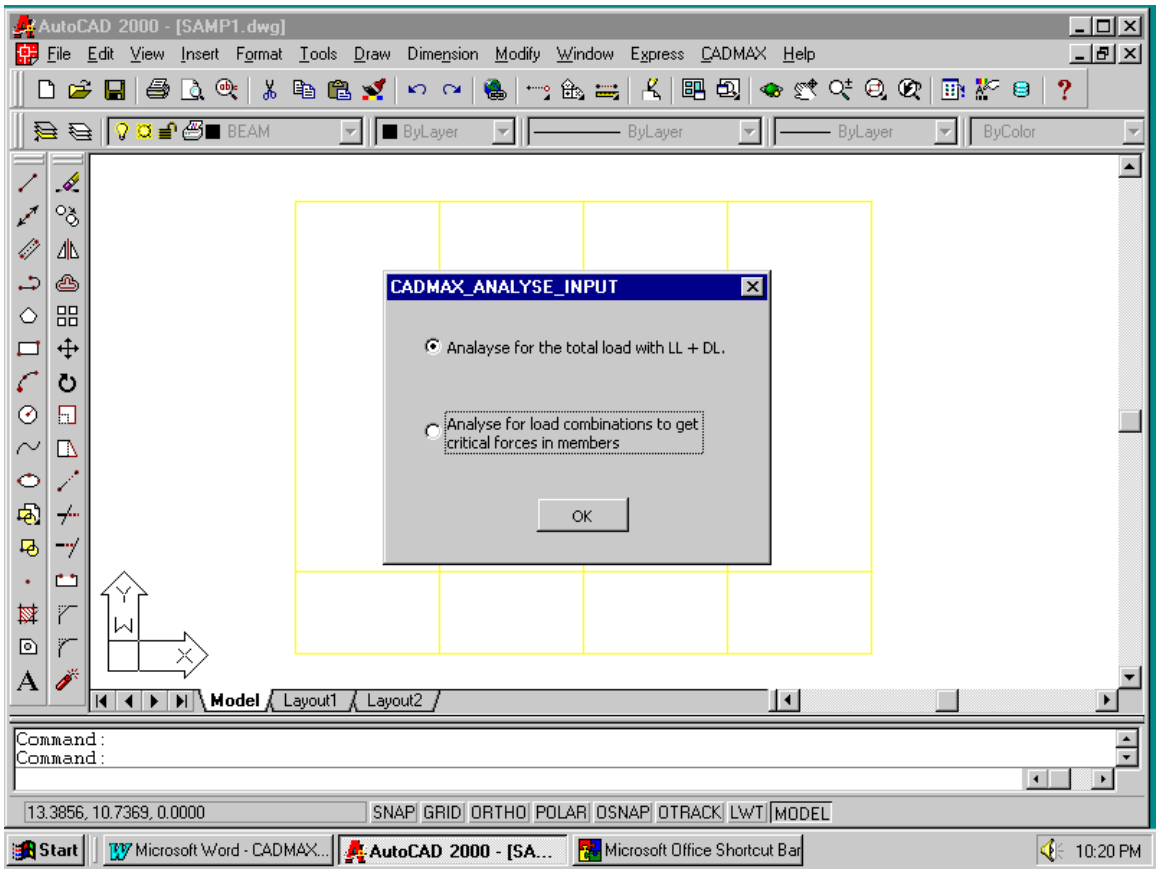
7.Pick the **ANALYSE** command from the **CADMAX** Popup menu.
Or Give the **ANALYSE** command at the command prompt.

Command: **ANALYSE** <enter>

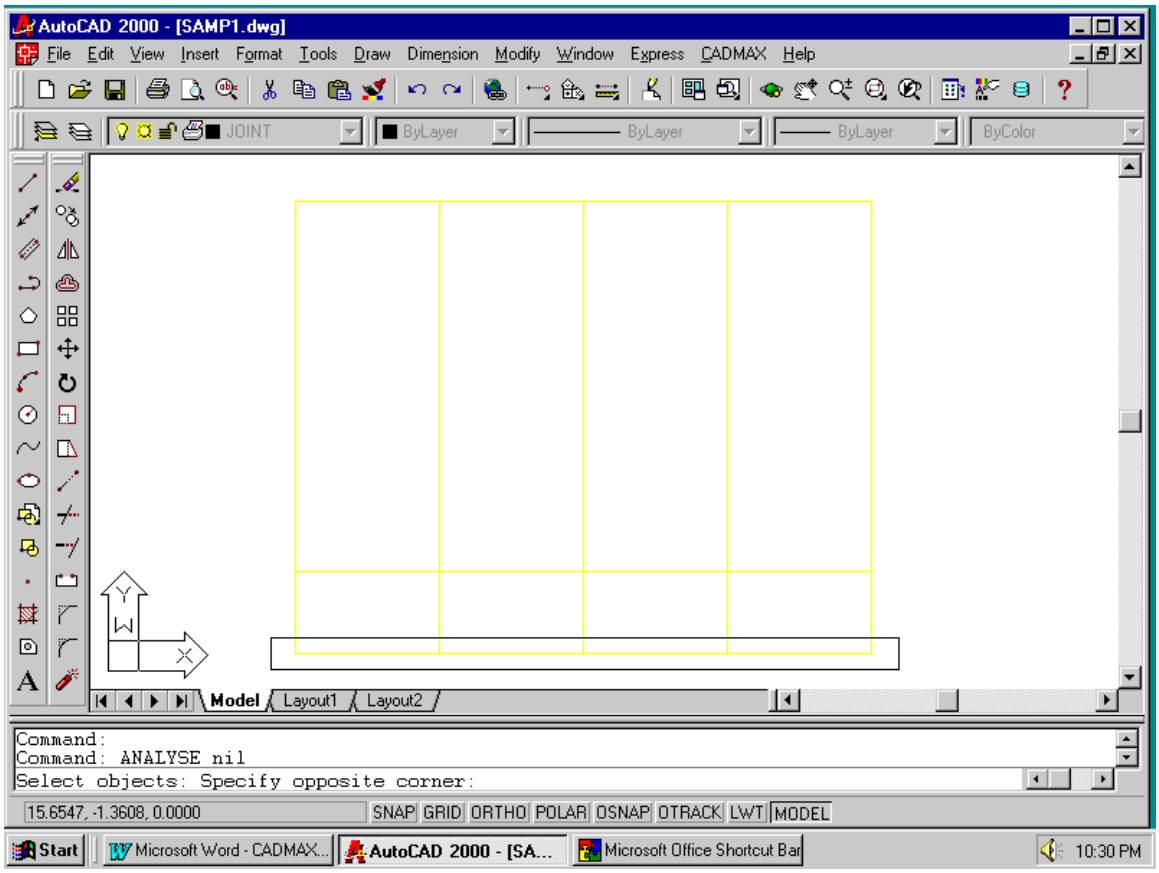
Now you will see the following dialog box to select the type of analysis with two options as follows:
1.Analyse the frame or structure by assuming total live load and dead load on the beams.

2.Analyse the frame or structure by considering load combinations of live load and dead load to get critical support moments and mid span moments.

In this example select the default option for analysis using total live and dead loads.
And then click **OK**.

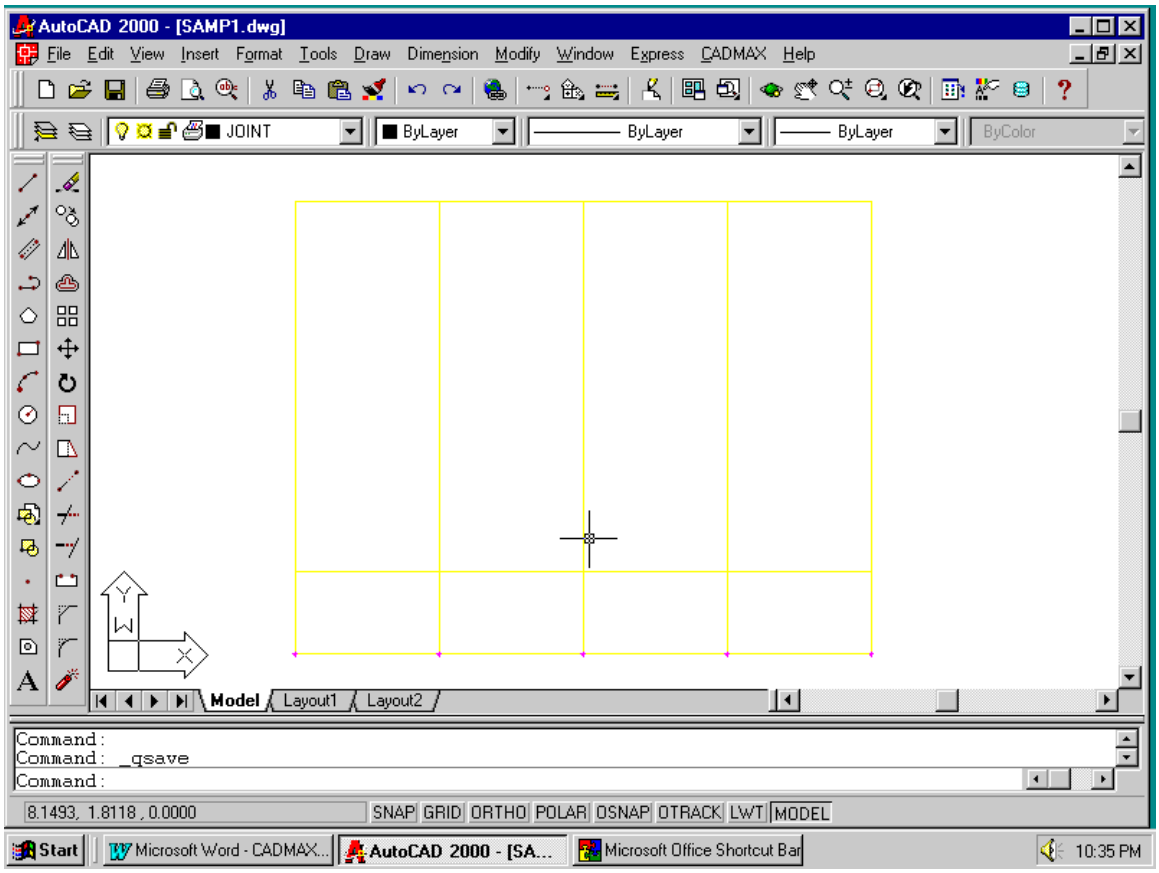


Then the dialog box will close and you will be asked to select object to be analysed. Now you select all the beams along the edge beams only as follows.



Now the edge frame is analysed within few seconds and you will see magenta color joints at the ends of beams, which shows the concentrated loads acting at the joints.

Now the drawing will be as shown below:



Now to transmit the point loads from edge beams to the cantilever beams use the **PTLOAD** command as listed below:

8.Pick the **PTLOAD** command from the **CADMAX** popup menu.
Or Give the **PTLOAD** command at the command prompt.

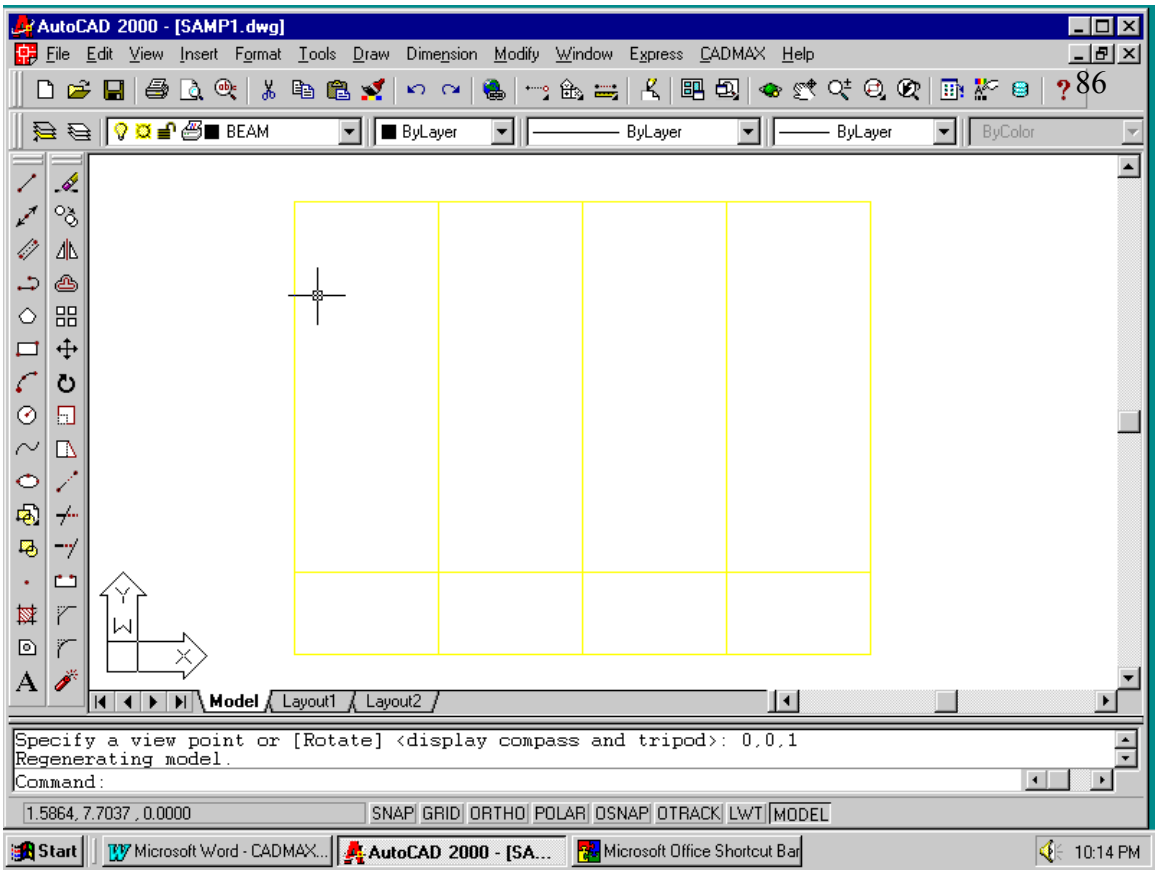
Command: **PTLOAD** <enter>
Select objects: Specify opposite corner: 36 found
(Now select all the point loads and beams along the edge beams)
Select objects:

Command: Select the beams support point loads:
Select objects: Specify opposite corner: 20 found
(Now Select the beams which supports the edge beams)
Select objects:

Now you will see that all the magenta color joints will disappear from the screen which shows that all the point loads have been transferred to the cantilever beams.

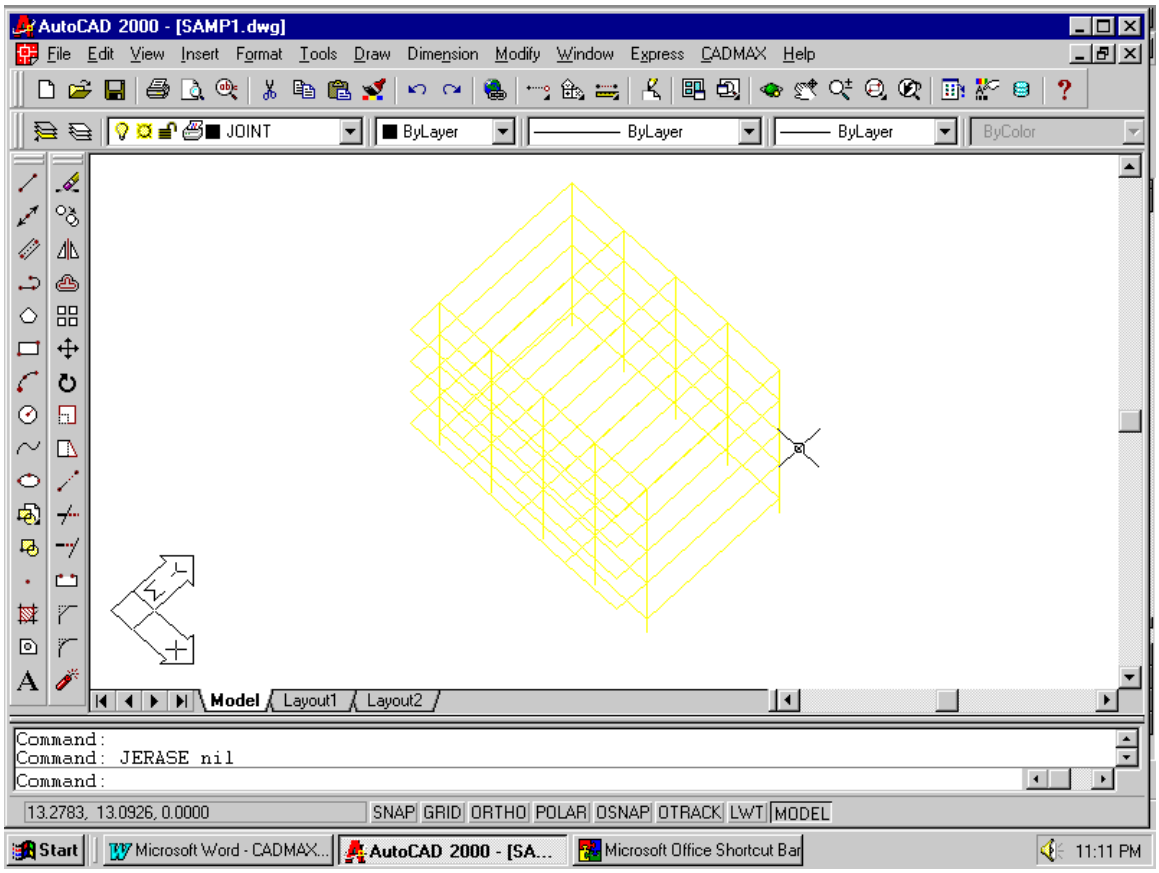
Now you can analyse the structure fully as three dimensional structure as space frame.

Now change the vpoint to 1,-1,3 to have a three dimensional view of the structure.



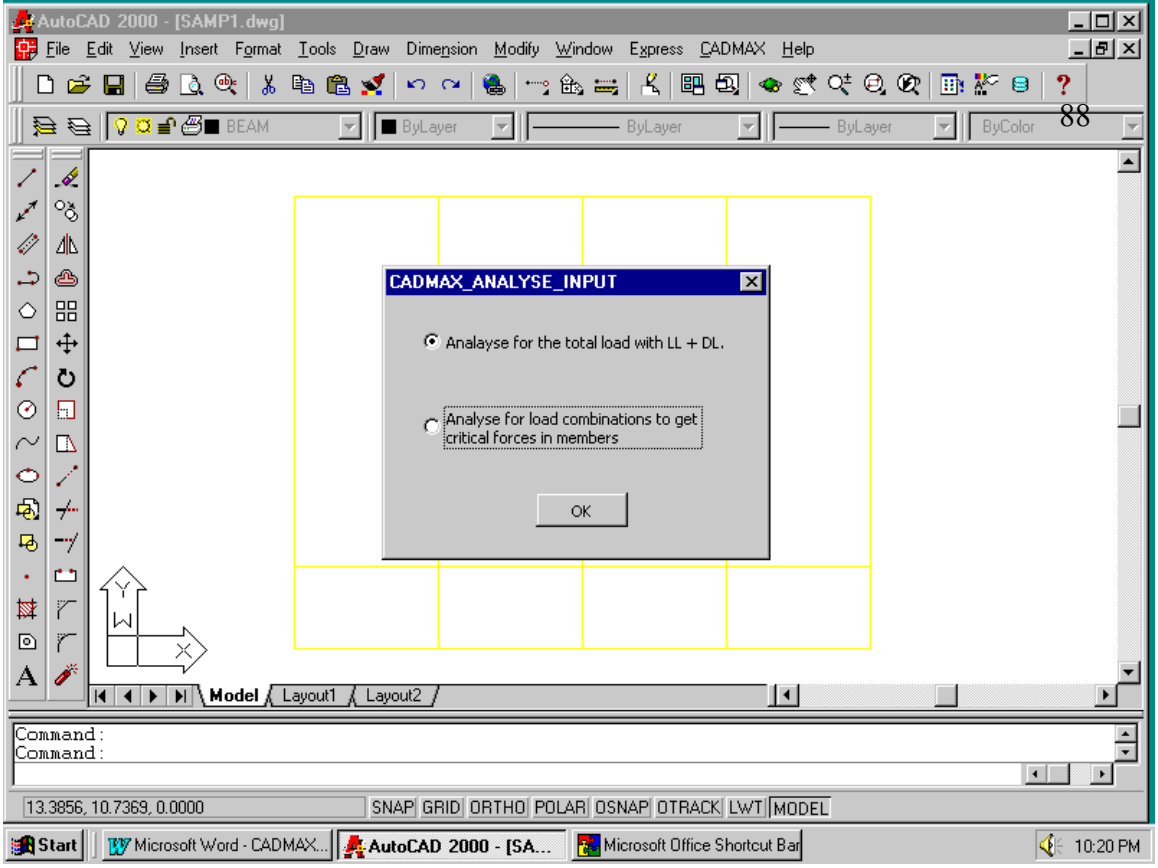
Command: **VPOINT** <enter>
Vpoint :<0.000,0.000,1.000>: **1,-1,3** <enter>

Now you will see the structure as given below:



Now to analyse the structure fully use the same **ANALYSE** command. Pick the **ANALYSE** command from the **CADMAX** Popup menu Or give the **ANALYSE** command at the command prompt.

Command: **ANALYSE**
Now you will get the same dialog box as before as shown below:



Select the default option of full live and dead load and then click **OK**.

Now you will be asked to select the objects for analysis.

Command: **ANALYSE**

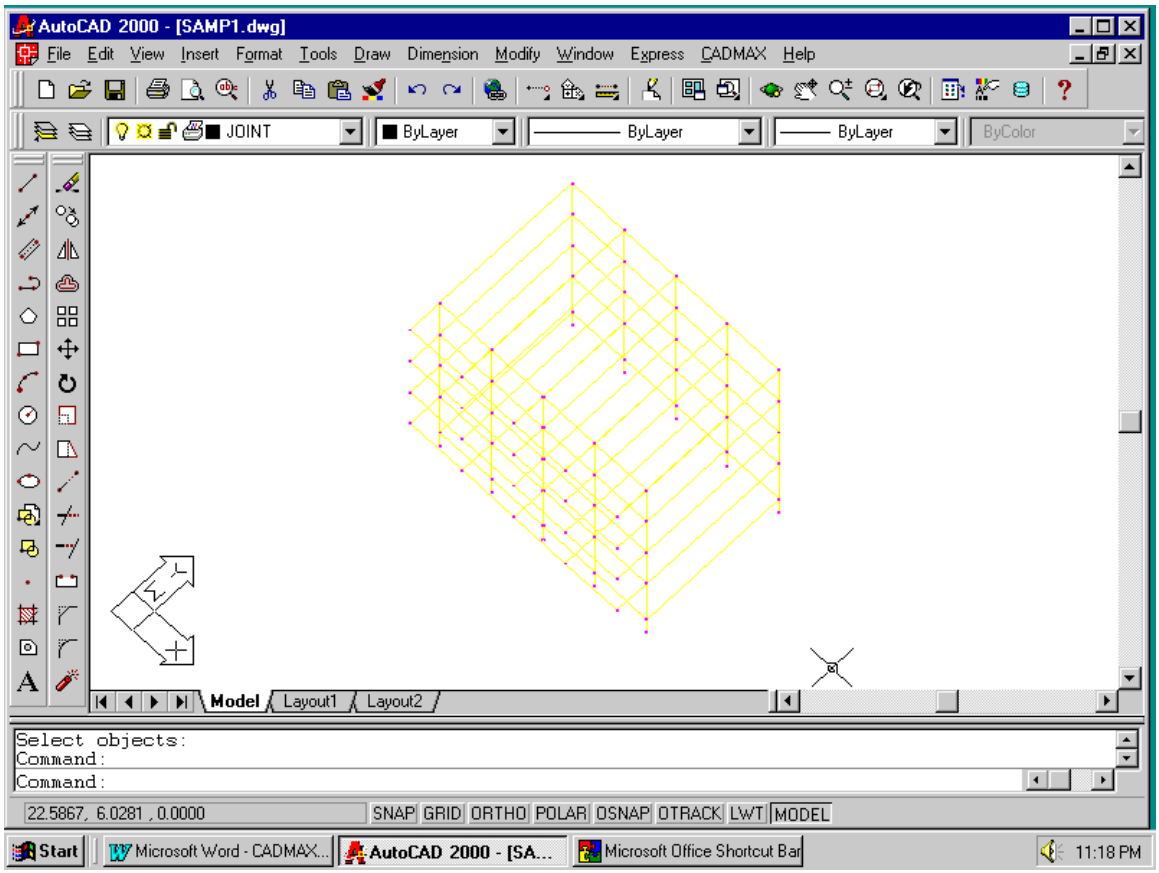
Select objects: Specify opposite corner: 108 found

Select objects:

Select objects: Specify opposite corner: 108 found
(Select all the objects in the display for 3D-Analysis.)
Select objects:

Now it will take few minutes to analyse the structure and you will see magenta color joints at all the junctions of beams and columns and the ends of beams which shows that analysis is over.

Now you will see the drawing as shown below:



Now you have analysed the structure for dead loads and live loads. If you click the option for critical forces in members for load combinations , it will take some more time to get the critical forces in all the members by load combinations of dead and live loads. You can also select the members frame by frame also as already done for edge frame analysis. When all the frames along X, Y or in all directions are over you have to use the following command to add the joint forces in all the directions.

10. Command: **JFABSUM** <enter>
You can pick the command from **CADMAX** Pop-up menu also.

Select objects: <Select all the objects in the display>
Select objects: <enter>

Now all the forces in various directions at the joints are combined by absolute sum of the forces corresponding to their respective directions and a single joint is created at each junction.

11. Command: **JERASE** <enter>

This command is used to erase all the joints if you have done something wrong during analysis and to redo the analysis again. Otherwise this command is not necessary.

12. Command: **SELDEL** <enter>

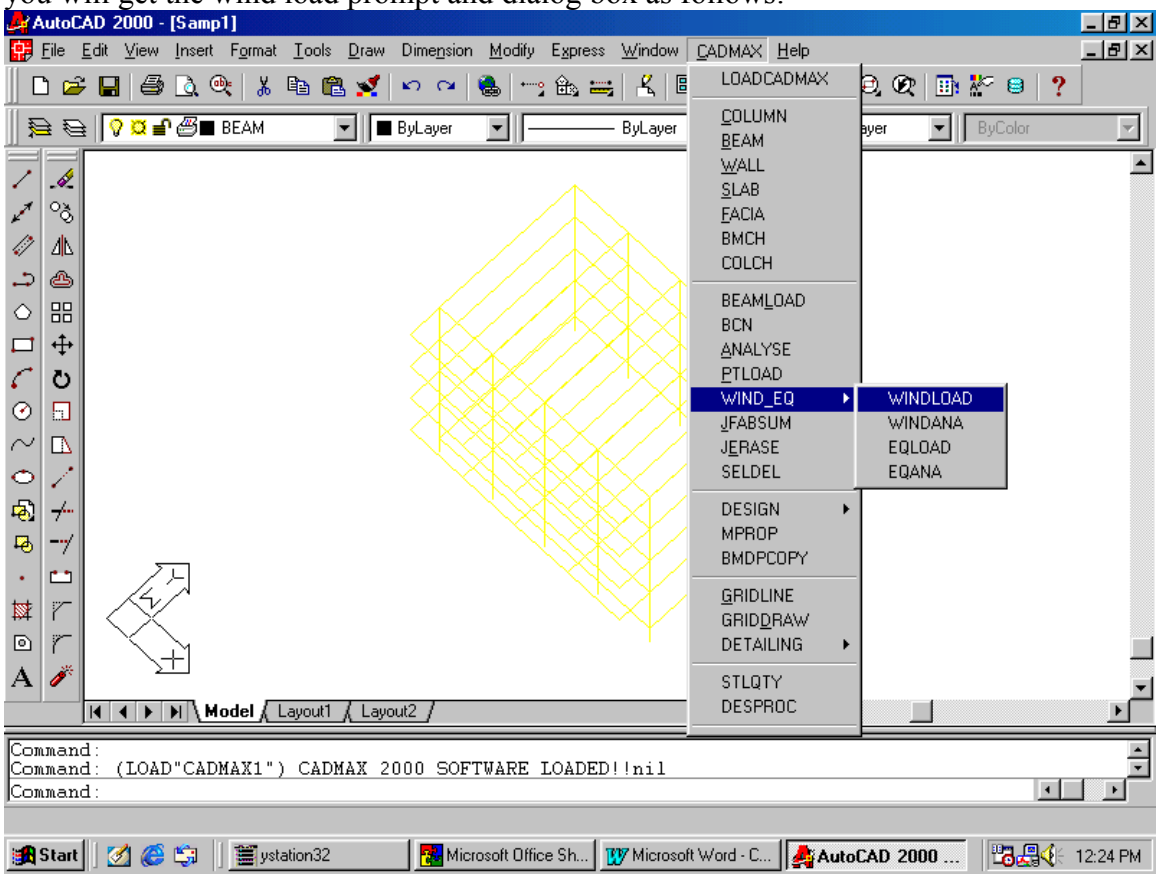
This command is used whenever you get error "The selection set already exists" . After using this command only you can use other commands whenever you met with above error.

13. To do wind load analysis and earth quake analysis use submenu of **WIND_EQ** .

To do wind and earth quake analysis there will be four commands in the submenu

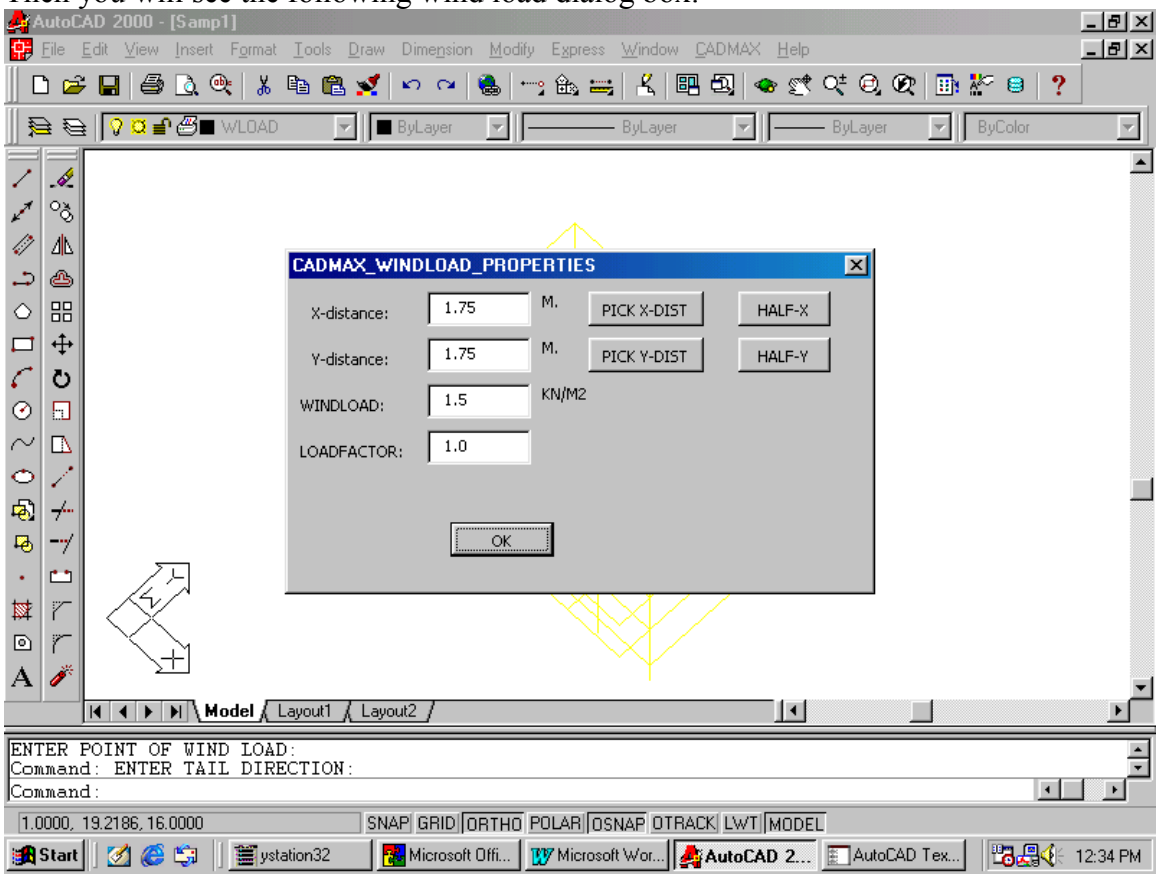
14. WINDLOAD <enter>

You can pick this command from the Pop-up menu also. This command is used to insert wind loads at the required junctions. For easy selection of junction points you make the layer JOINT off while inserting the wind loads. To insert wind loads acting at joints depending upon the intensity of wind pressure, Give the WINDLOAD command. Now you will get the wind load prompt and dialog box as follows:



Command: **WINDLOAD** <enter>

ENTER POINT OF WIND LOAD:<pick the point where the wind load acts>
Command: **ENTER TAIL DIRECTION**:<enter tail direction of win load acting from>
Then you will see the following wind load dialog box:



If you know the horizontal distance for wind load area you can directly fill the value in the above dialog box for X-distance as 1.75 or you can pick the button **PICK X-DIST** to specify the horizontal distance by picking the point in the drawing itself. Now you will get the following command prompt:
Command: **ENTER HORIZONTAL DISTANCE FOR WINDLOAD AREA:** Specify second point:<pick the point to show the horizontal distance of wind area>

Now you will see the dialog box again in which the horizontal distance is shown in the dialog box. If you want to take the half the distance in the dialog box you can pick the button **HALF-X** which will change the value to the half the distance.

If you know the vertical distance for wind load area you can directly fill the value in the above dialog box for Y-distance as 1.75 or you can pick the button **PICK Y-DIST** to specify the vertical distance by picking the point in the drawing itself. Now you will get the following command prompt:

Command: **ENTER VERTICAL DISTANCE FOR WINDLOAD AREA**: Specify second point:<pick the point to show the vertical distance of wind area>

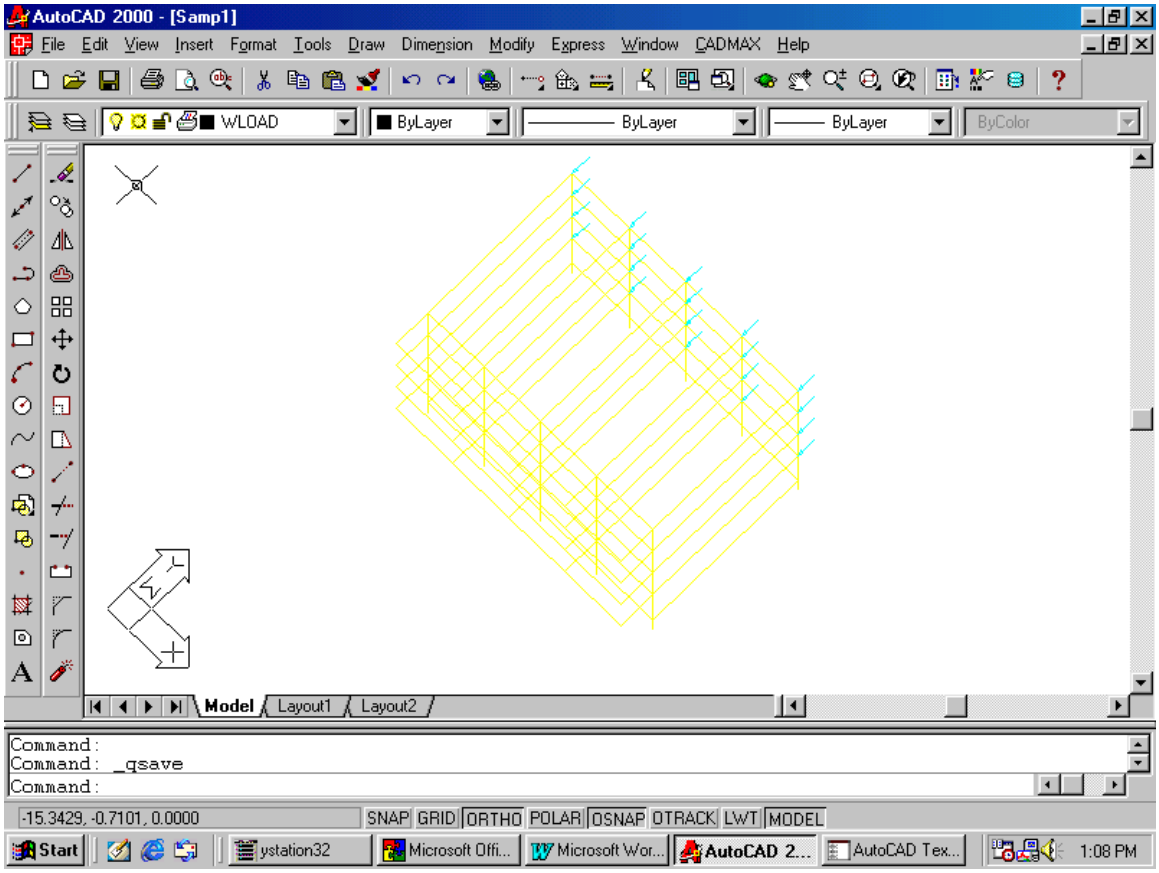
Now you will see the dialog box again in which the vertical distance is shown in the dialog box. If you want to take the half the distance in the dialog box you can pick the button **HALF-Y** which will change the value to the half the distance.

If you want to change the value of intensity of the wind load you can change the value as per your requirement. Similarly you change the load factor also depending upon the openings in the area of wind load other wise you can use same load factor value of 1.0. In this case wind load intensity is taken as 1.5 KN/m2.

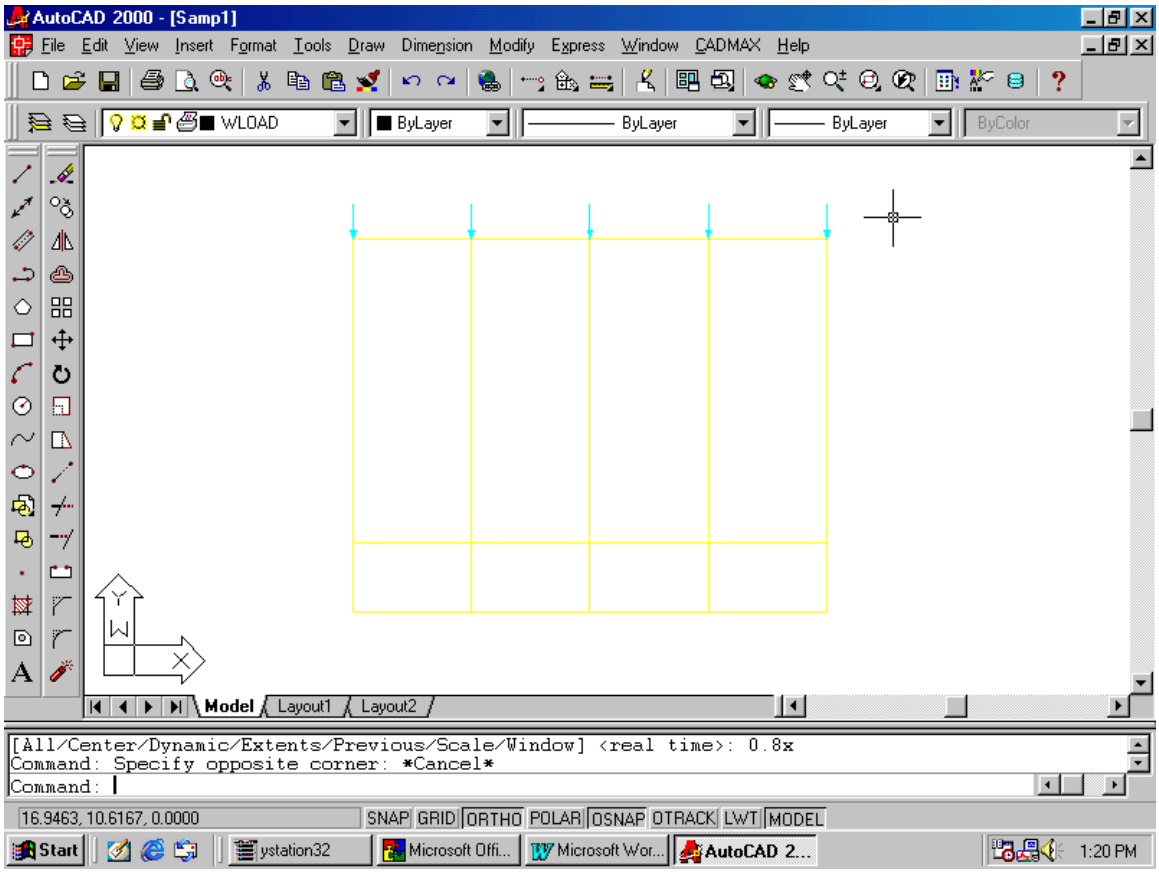
After entering the above values in the dialog box correctly you click **OK** button which will close the dialog box and a wind load with an arrow head in the required direction is seen the drawing which will have the all the values given as properties and calculated the point load acting at that joint.

Similarly you can give wind loads at all other junction points before wind analysis.

After entering all the wind loads your drawing will be shown as given below:

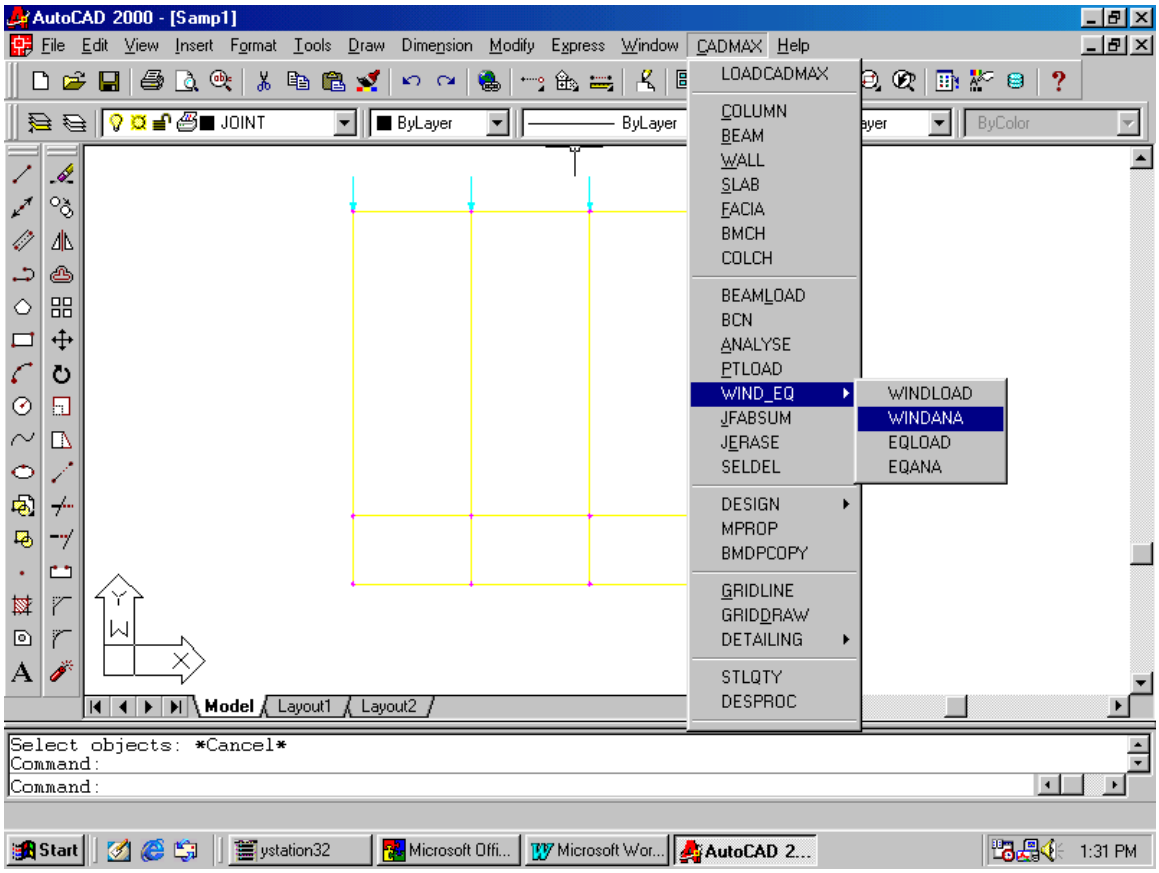


To analyse for wind load you should bring the drawing for plan view by giving vpoint as 0,0,1. Now you can do wind analysis for each frame by giving **WINDANA** command. The plan view will be as shown below:

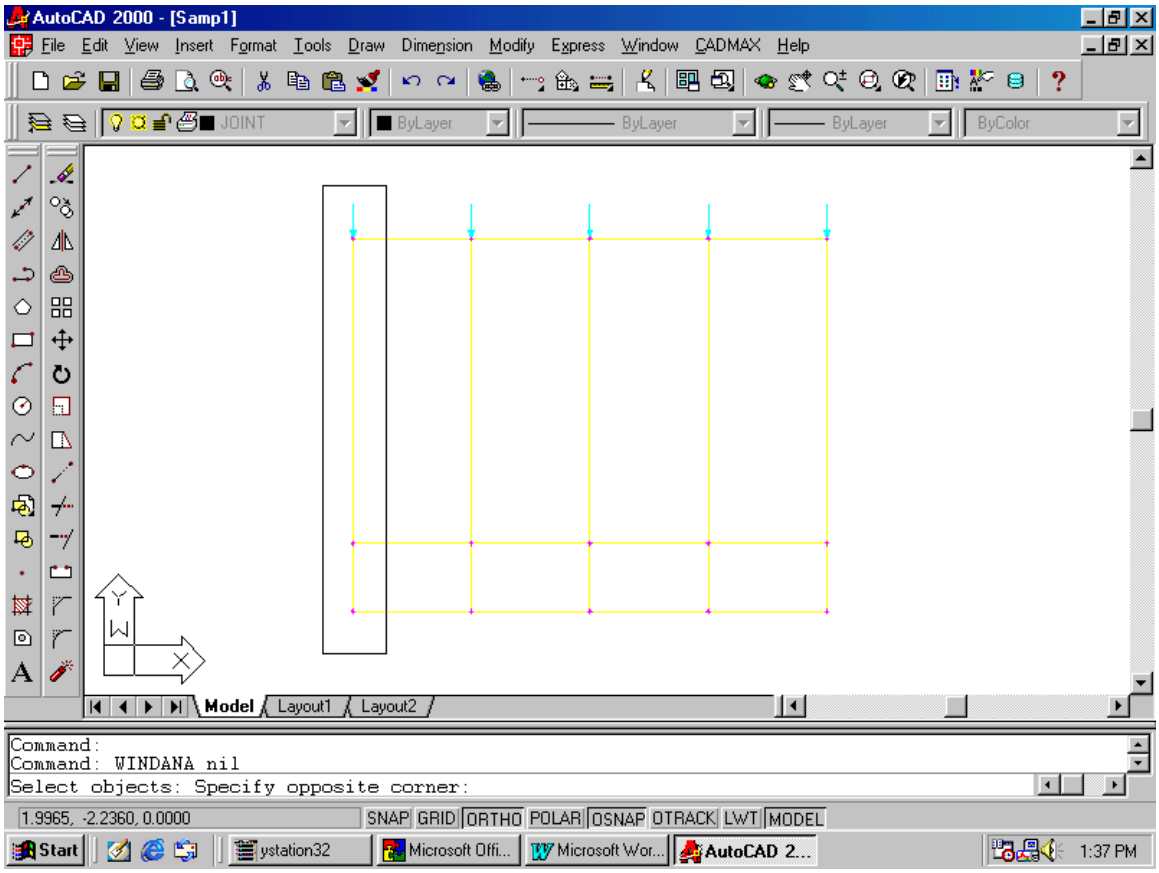


15. WINDANA

You can pick this command from the Pop-up menu also. This command is used to do wind analysis of the frames. You should analyse for wind load by frame analysis method only for each frame individually. Before Wind analysis the Layer **"JOINT"** which is made **"OFF"** before the insertion of wind loads should be made **"ON".**



Command:**WINDANA** <enter>
Select objects:<select all the objects in the frame as shown below>

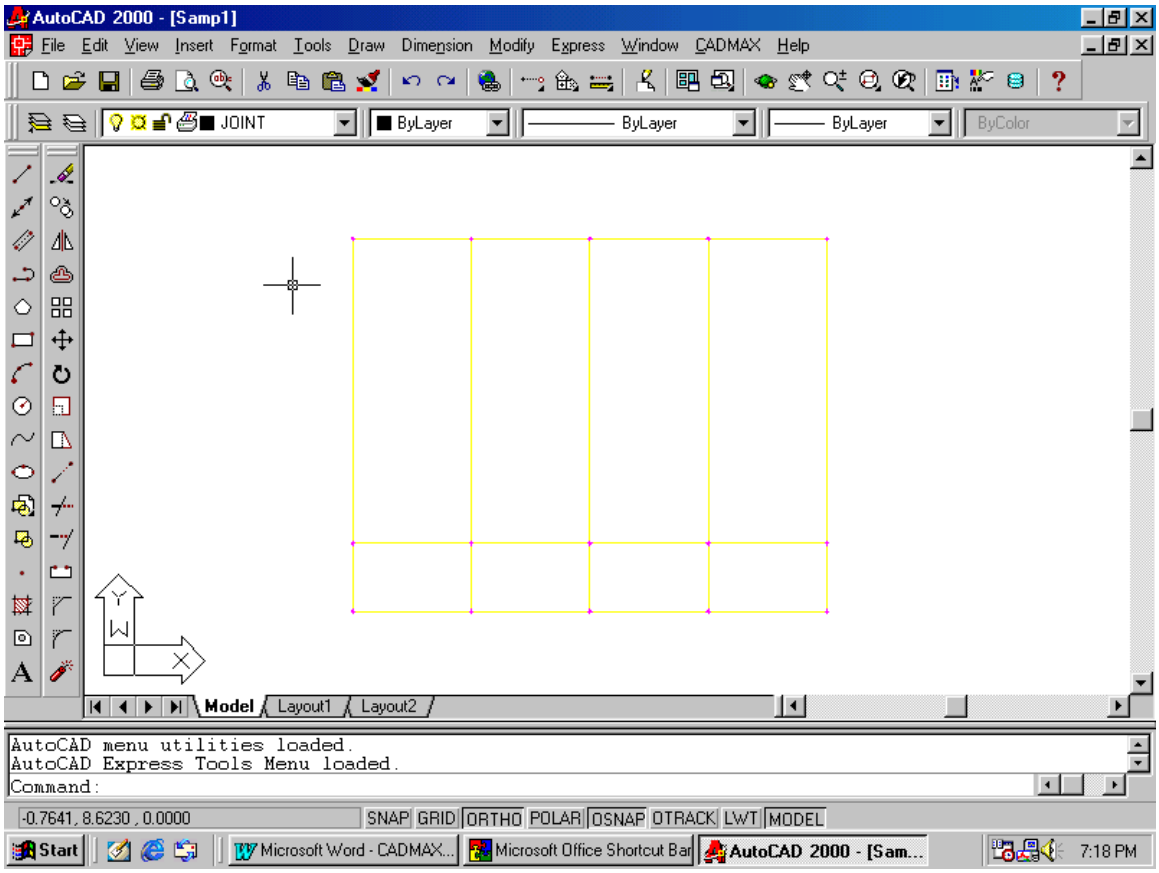


Now you will be prompted to enter Natural Ground level which is entered as 1.5m in this case and then click **OK**. Now the wind analysis for the selected frame is done. Similarly the wind analysis for the other frames are done. While designing the frame members the reversal of wind loads will be automatically considered for getting worst conditions.

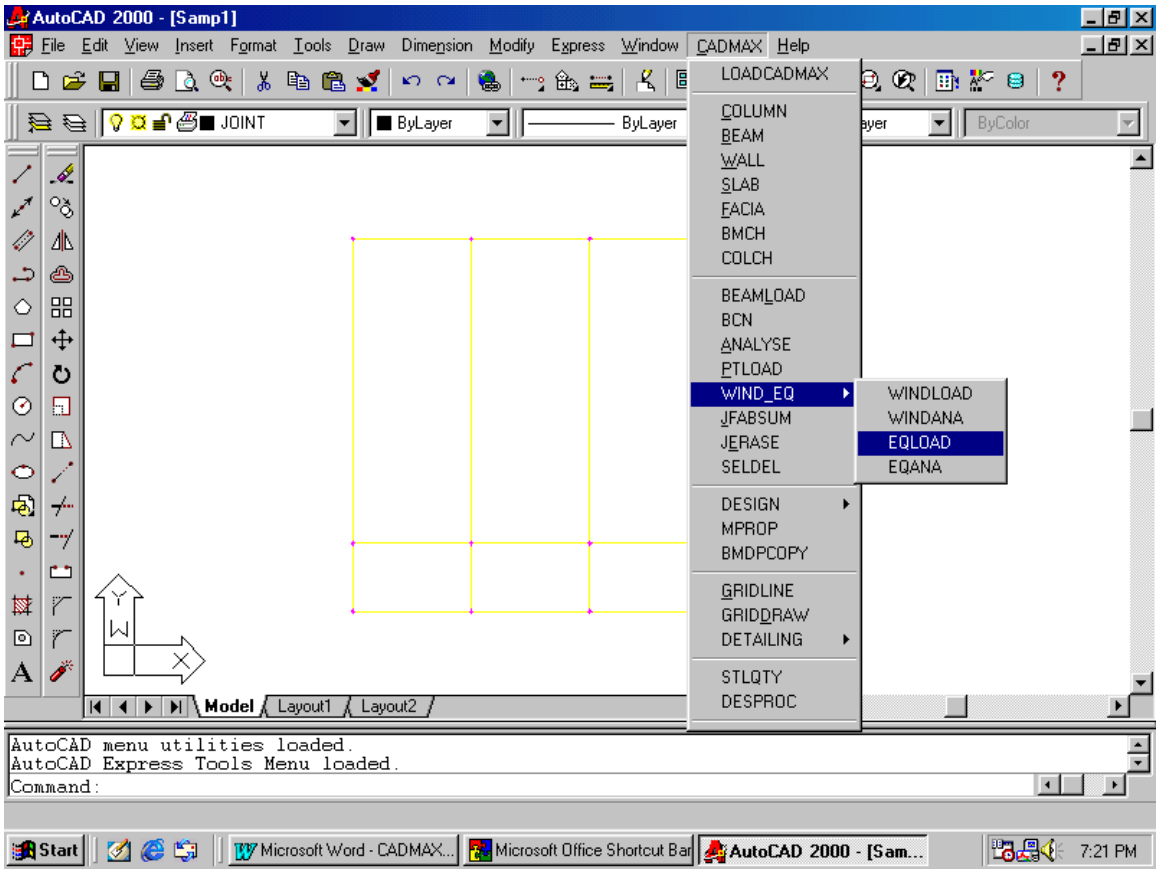
16. EQLOAD <enter>

You can pick this command from Pop-up menu also. This command is used to insert earth quake zone properties such as basic horizontal seizmic coefficient , live load factor, Importance factor of the structure and live load factor as per IS-code-1893.

Before inserting earth quake load properties, make the layer “WLOAD” “OFF” since the wind load analysis is already over.
Now the drawing will be as shown below:



Now insert the Earth Quake load from the pop-up menu as shown below:(or you can give directly give “EQLOAD” command at the command prompt.

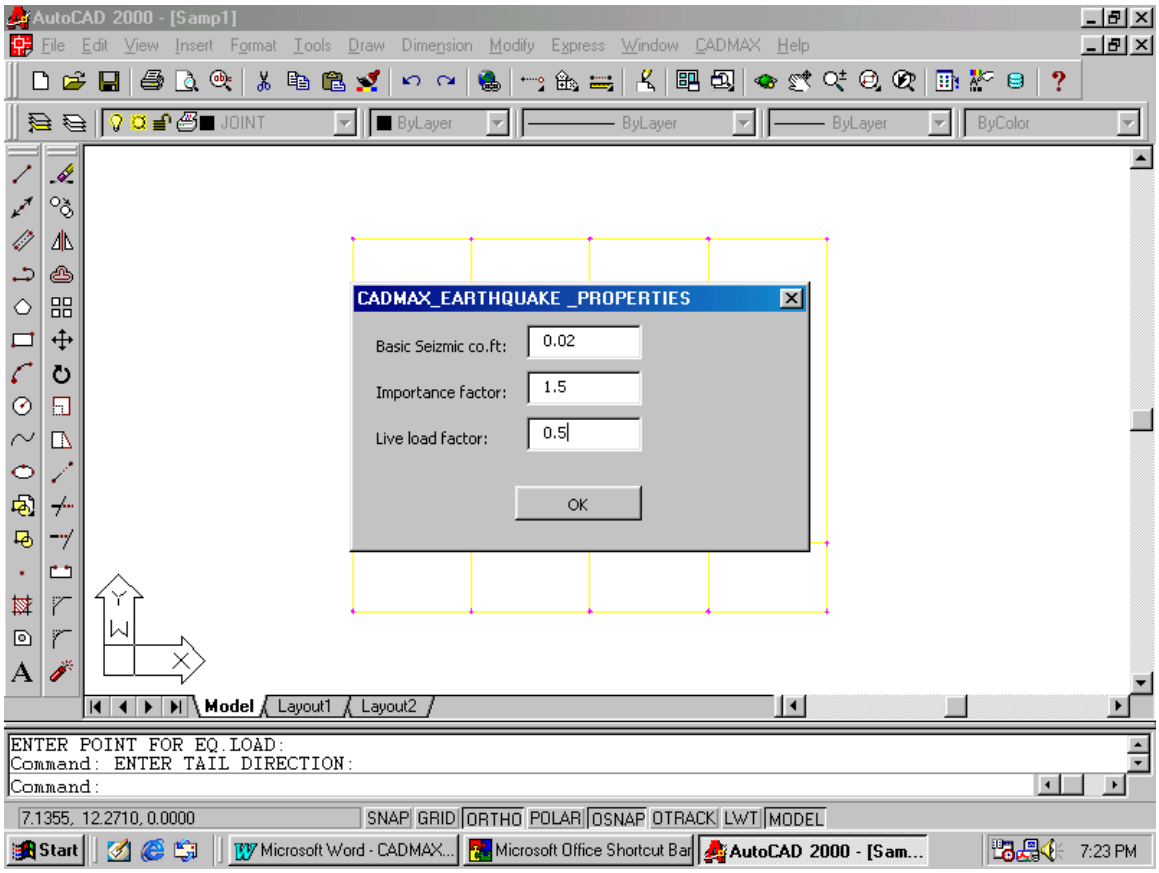


Command**: EQLOAD**

ENTER POINT FOR EQ.LOAD:<please pick a point outside the drawing>

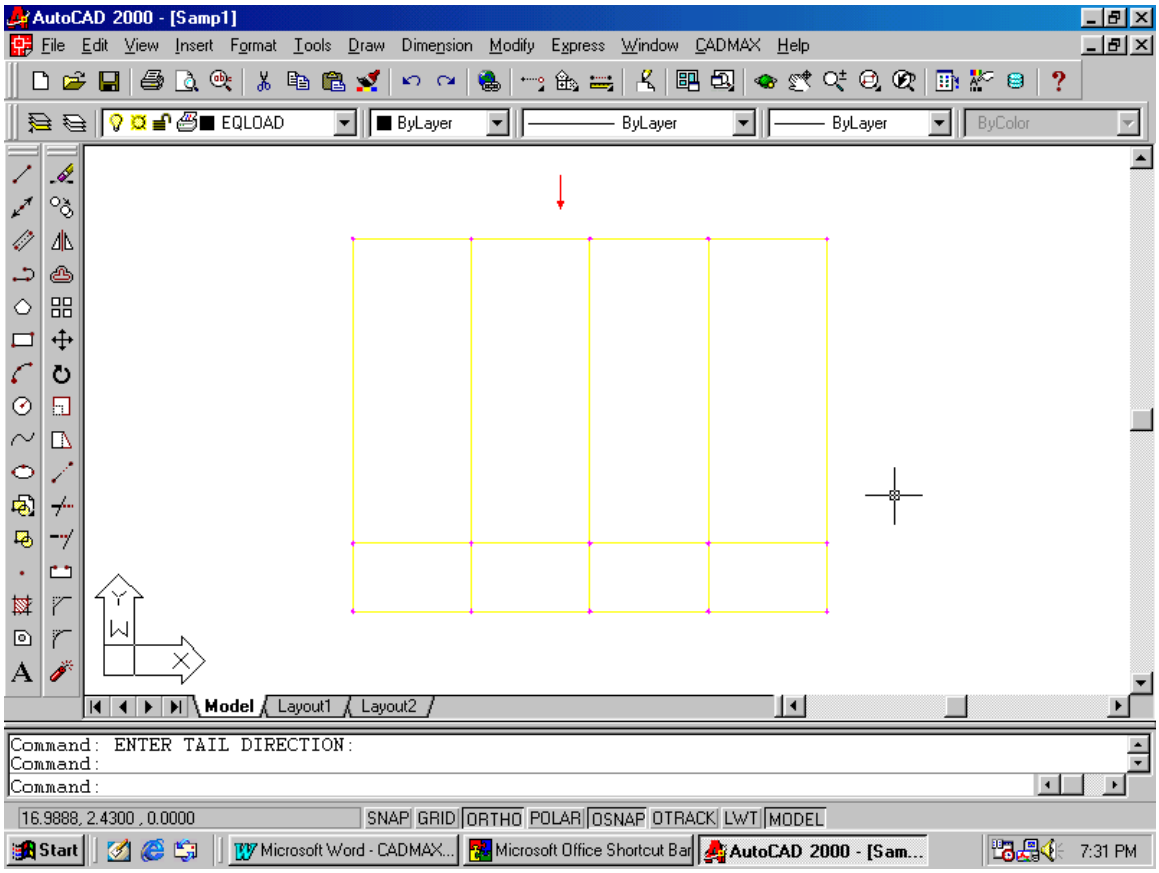
Command: **ENTER TAIL DIRECTION**: <pick other point to enter the critical direction>

Now you will get the following dialog box to enter the earth quake properties of the zone as per IS-1893 as follows:



Enter the Basic seizmic co-efficient, Importance factor, and live load factor as per IS-1893 code specifications in the above dialog box and then click **OK**.

Now you will see an arrow in red color in the drawing with given earth quake properties as shown below:



Now you can analyse the structure for earth quake load as follows:

17. EQANA <enter>

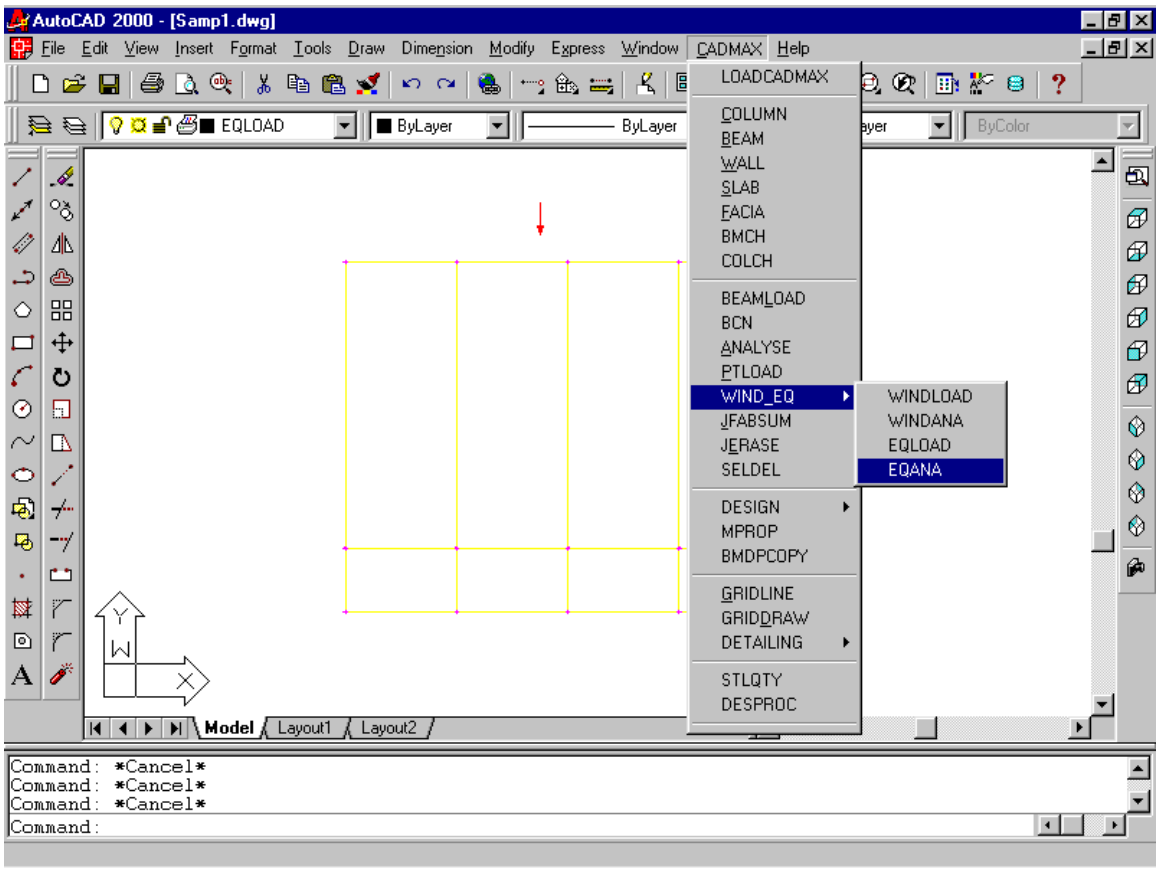
You can pick this command from Pop-up menu also. This command is used to do earth quake analysis of the frames. You should analyse for earth quake load for each frame separately but the analysis will take care of total loads at each joints from all directions for vertical and horizontal acceleration due earth quake forces.

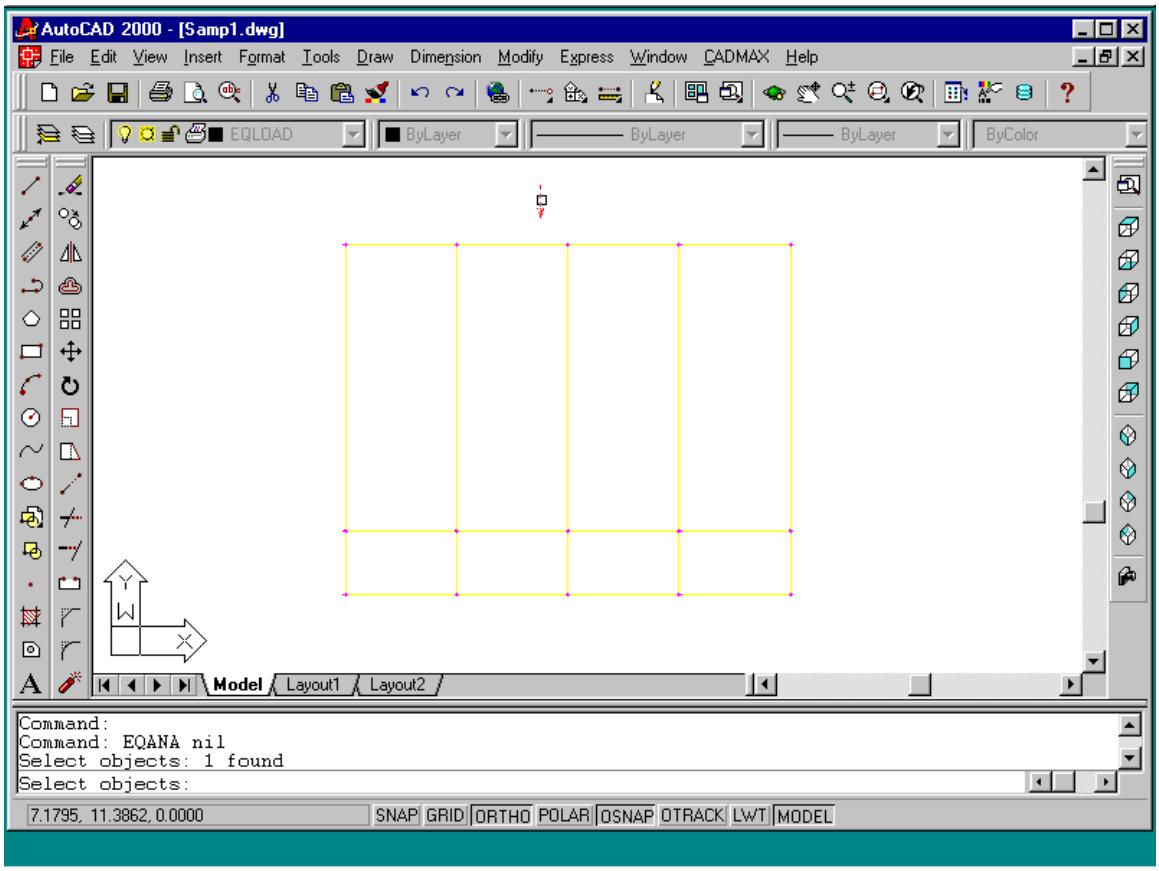
Command: **EQANA** <enter>
nil
Select objects: 1 found <First you have to pick the arrow formed during EQLOAD command as shown in figure then you have to select all members along one frame as shown in the next figure that is first frame along Y-direction>

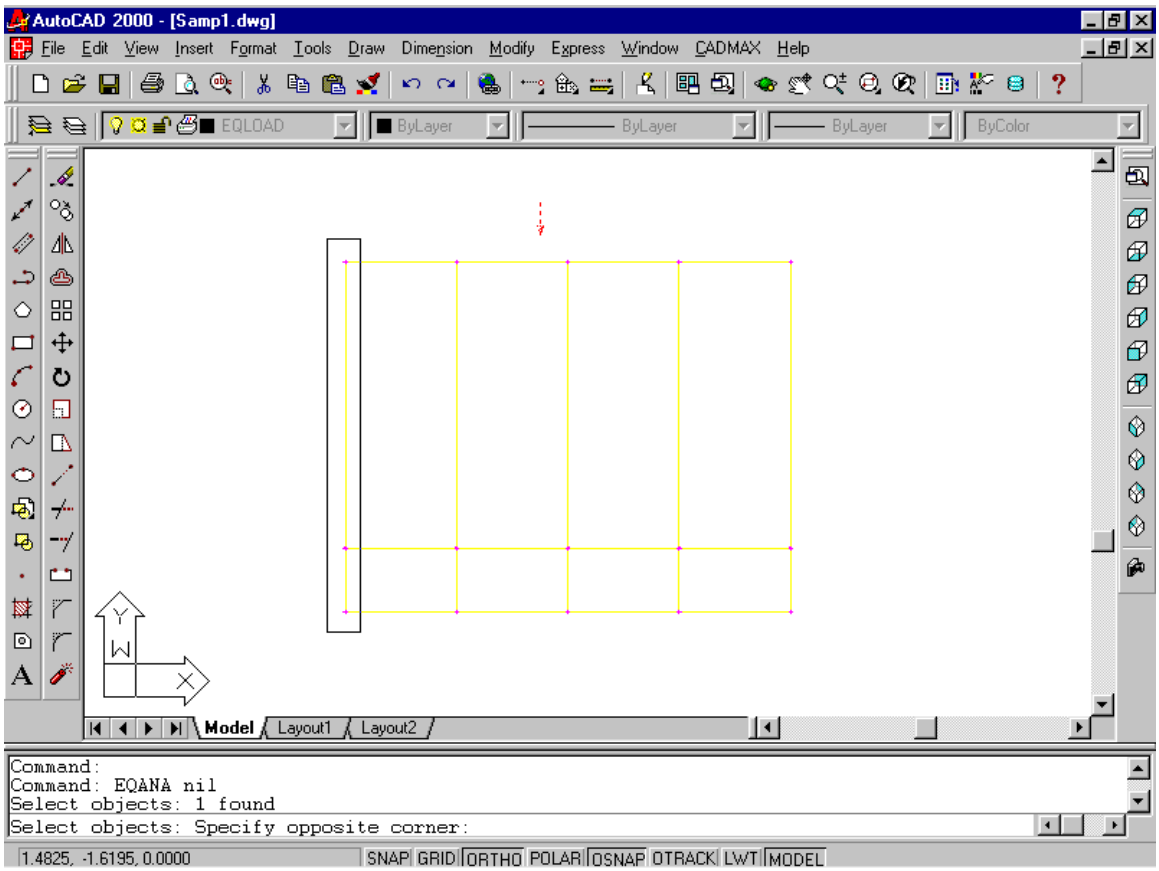
Select objects: Specify opposite corner: 27 found, 28 total

Select objects:

Command:


AutoCAD 2000 - [Samp1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Express Window CADMAX Help
EQLOAD
ByLayer
ByLayer
ByColor
LOADCADMAX
COLUMN
BEAM
WALL
SLAB
FACIA
BMCH
COLCH
BEAMLOAD
BCN
ANALYSE
PTLOAD
WIND_EQ
JFABSUM
JERASE
SELDEL
DESIGN
MPROP
BMDPCOPY
GRIDLINE
GRIDDRAW
DETAILING
STLQTY
DESPROC
WINDLOAD
WINDANA
EQLOAD
EQANA
Model
Layout1
Layout2
Command: *Cancel*
Command: *Cancel*
Command: *Cancel*
Command:


AutoCAD 2000 - [Samp1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Express Window CADMAX Help
EQLOAD
ByLayer
ByLayer
ByLayer
ByColor
Model
Layout1
Layout2
Command:
Command: EQANA nil
Select objects: 1 found
Select objects:
7.1795, 11.3862, 0.0000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL



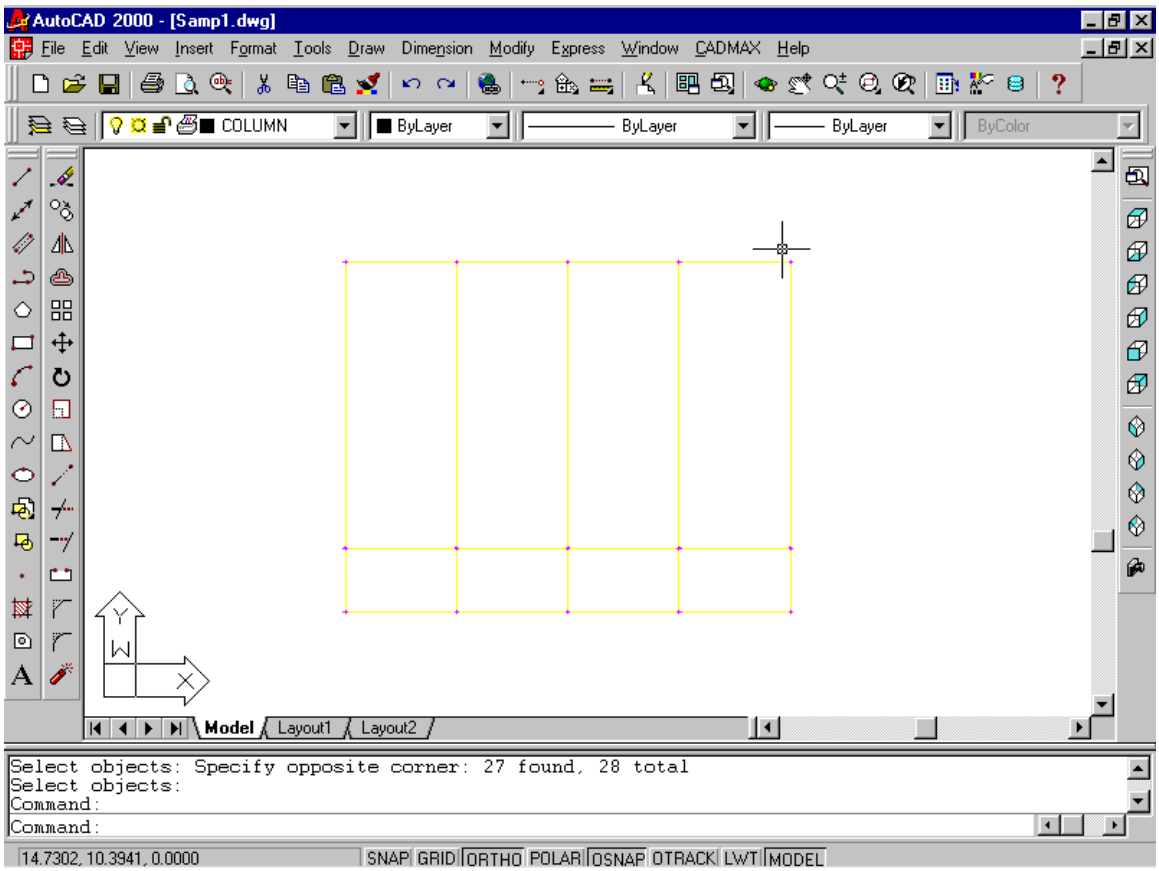
Now the Earth Quake analysis for the first frame along Y-direction that is for selected members only is done. The analysis will consider the reversal of loads also for getting the critical forces in the members.

Similarly the Earth Quake Analysis for the other frames can also be done as described above by selecting the EQLOAD arrow and the members of the other frames seperately.

With above commands you have completed all the analysis of the above structure such as dead load and live loads, wind load analysis, earth quake analysis. If you do not want Wind load and Earth Quake load Analysis, you can skip them. But it is important to note that Wind load and Earth Quake Analysis should be done only after completion of Dead Load and Live Load Analysis.

After Earth Quake Analysis, You Can make the layer “EQLOAD” to “OFF” and “SET” the current layer as “COLUMN” to go for design of members.

Now, the drawing will be as shown below.



Now, the Analysis part is over and we will go for Design of Members.

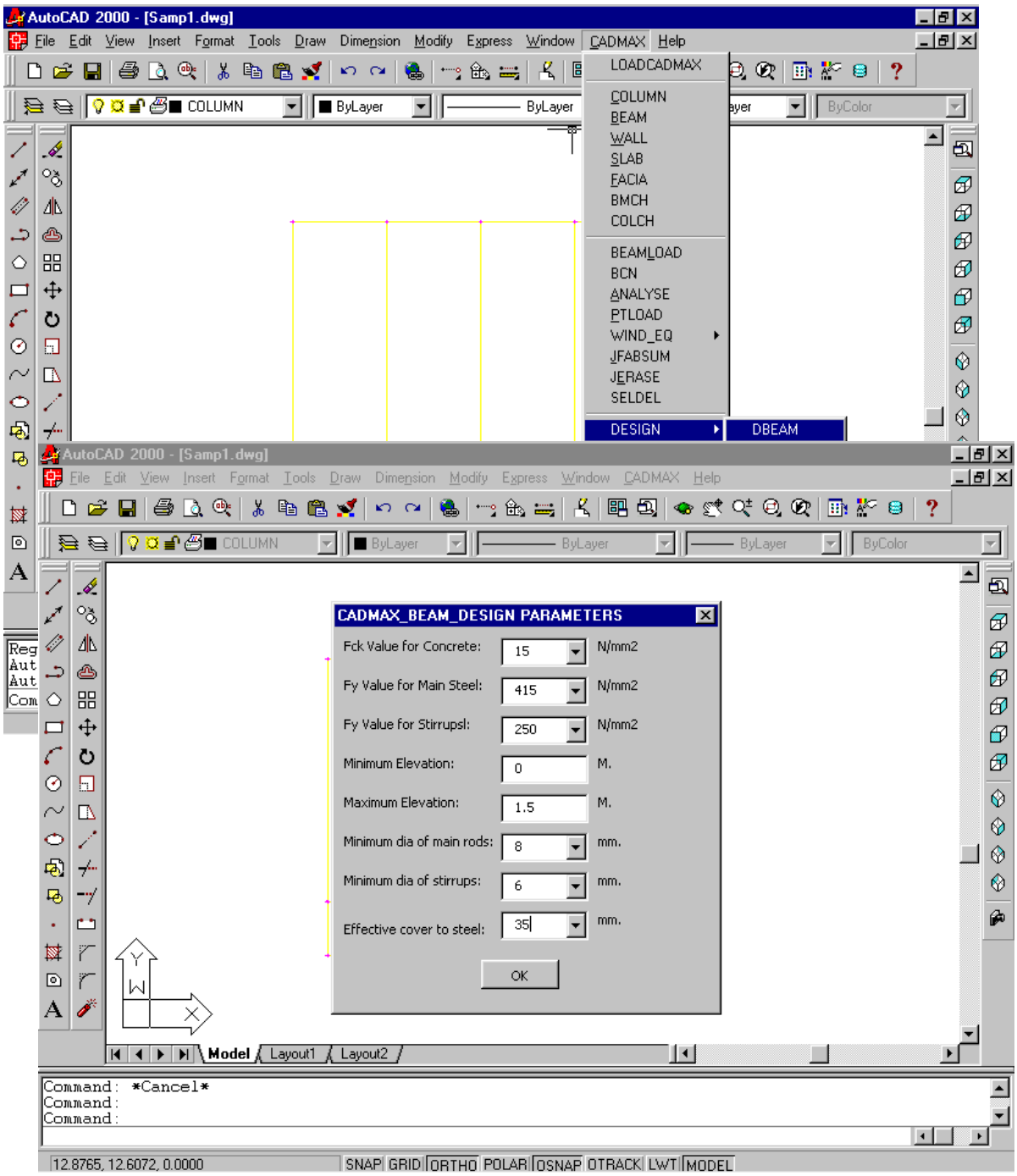
CHAPTER.4.DESIGN OF MEMBERS OF STRUCTURE.

In this Chapter we will see about the design of members. The slab designs are done while giving "**SLAB**" command itself. The other members such as Design of Beams, Columns and Footings are to be done in this Chapter.

18.DBEAM <enter>

You can pick this command from Pop-up menu also. This command is used to do Design of Beams in the Total Structure. The Design procedure followed is as per Limit State Method in IS-456-1983 and Design Aids for IS-456. The Design forces considered is as per the critical forces arrived at the combination of Dead load, Live Load, Wind Load and Earth Quake loads as per IS code provisions.

The pop-up menu to give "DBEAM" command is shown below:

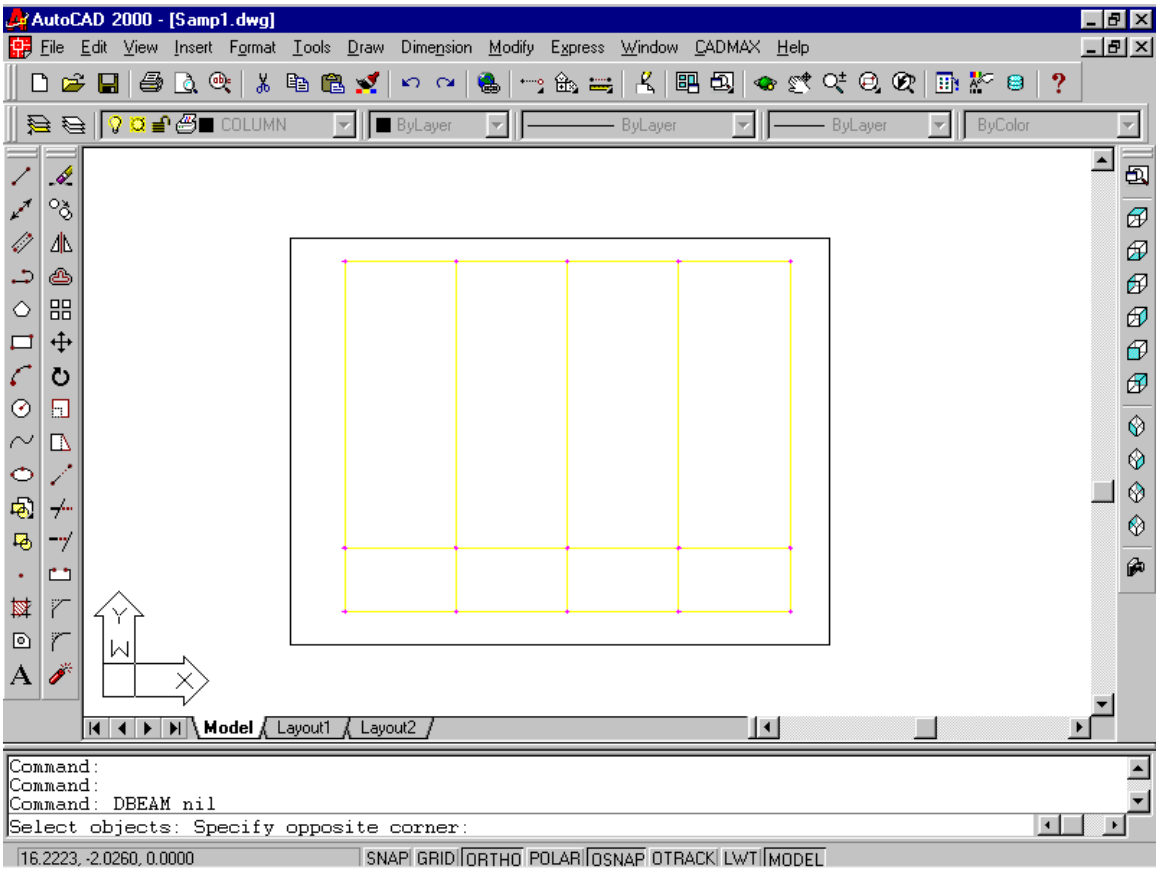


Command: **DBEAM**<enter>
Now you will see a dialog box to select the Design Parameters for Design of Beams as shown below:

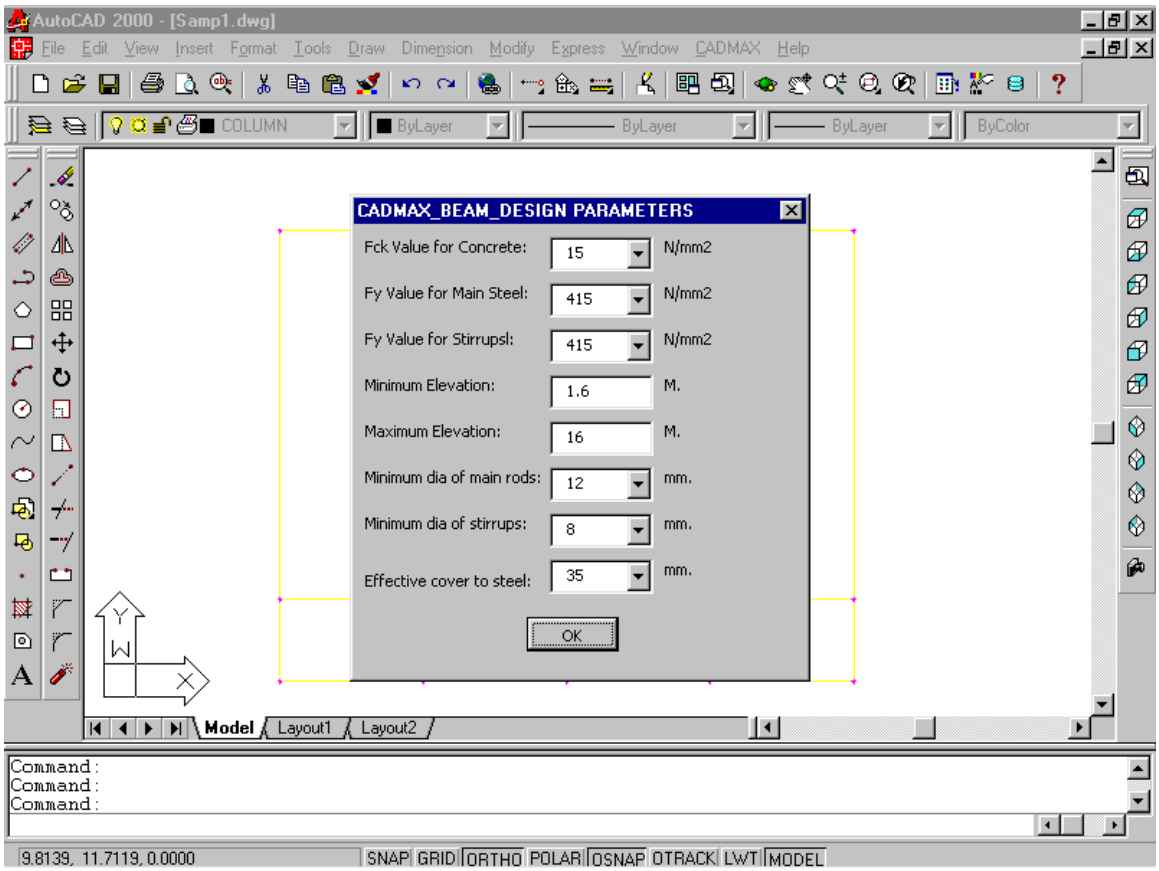
In the above dialog box you select the Fck Value of concrete, Fy Value of Main steel, Fy Value of stirrups used from combo boxes provided. The Minimum elevation value in M and Maximum elevation Value in M is given such that the members selected for design

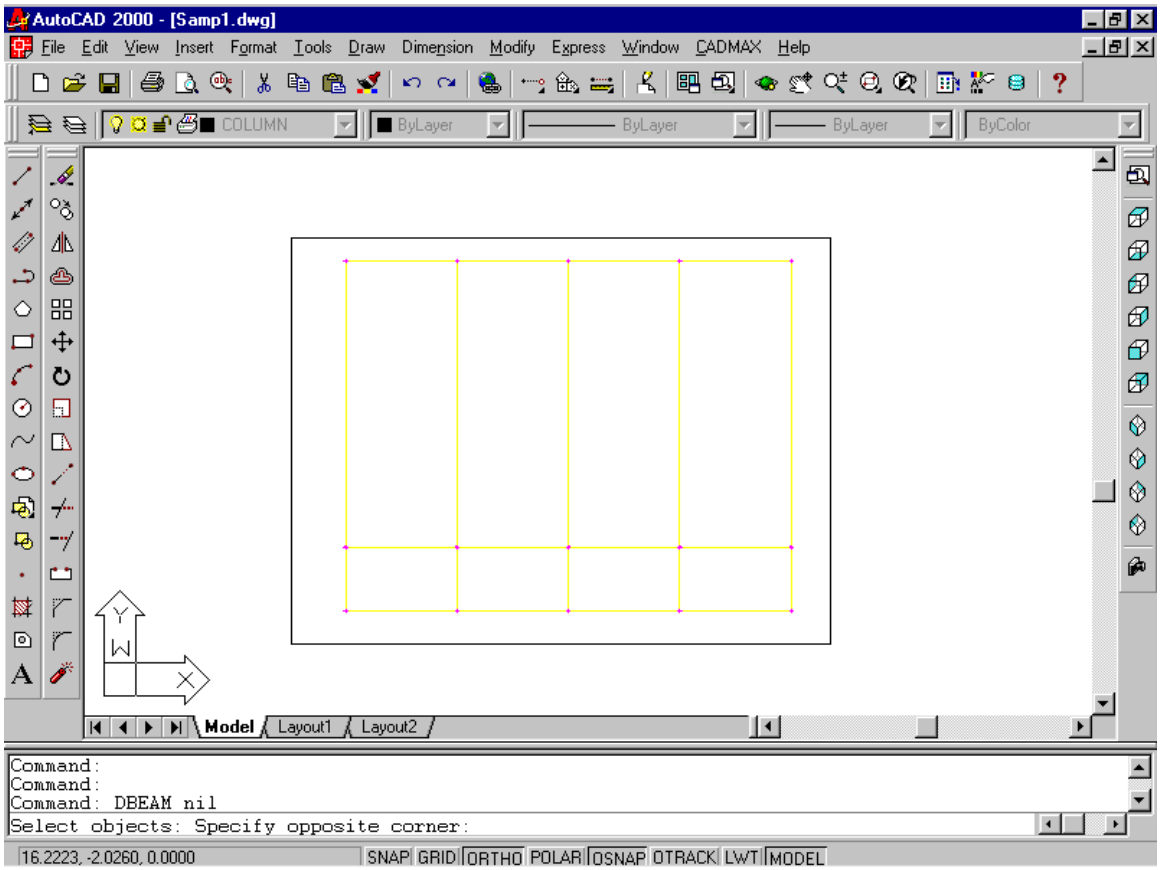
will be in between the given range of elevations. The Minimum dia of main rods will be selected from the combo box values. The Minimum dia of stirrups will be selected from the combo box values. The Effective cover to steel will be selected from the combo box values. Then click **OK** button. Now the given Values in the above dialog box will be considered for the design of beams. Then you will be prompted for select objects. In the above dialog box the Values are selected such that the Design of beams for Grade Beams between the elevations of 0.0m to 1.5m is done.

Select Objects:<pick all the members as shown below>
Select Objects:<enter>



After completion of Design of beams as per Design parameters given the selected objects will be released and ready for next command. The remaining beams between elevation of 1.6m to 16m and Minimum dia of main rods 12mm and Minimum dia of stirrups are 8mm and Fy Value of stirrups is given as 415 and all other Values are same as previous dialog box and then select objects as done previously. Now all the remaining beams between elevation of 1.6m to 16m is done. The dialog box and Selection of objects screens are shown in the following figures.


AutoCAD 2000 - [Samp1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Express Window CADMAX Help
COLUMN
ByLayer
ByLayer
ByLayer
ByColor
CADMAX_BEAM_DESIGN PARAMETERS
Fck Value for Concrete: 15 N/mm2
Fy Value for Main Steel: 415 N/mm2
Fy Value for Stirrupsl: 415 N/mm2
Minimum Elevation: 1.6 M.
Maximum Elevation: 16 M.
Minimum dia of main rods: 12 mm.
Minimum dia of stirrups: 8 mm.
Effective cover to steel: 35 mm.
OK
Model
Layout1
Layout2
Command:
Command:
Command:
9.8139, 11.7119, 0.0000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL

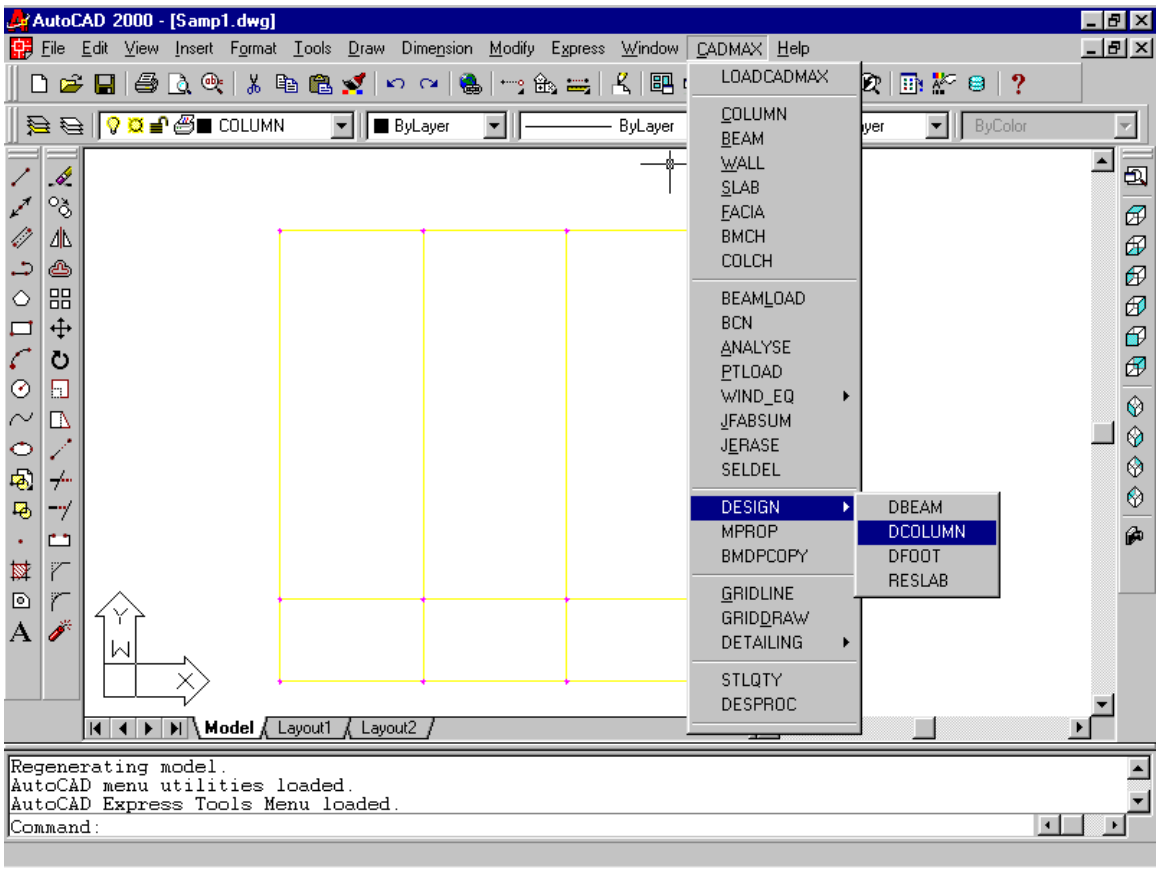


Now all the beams in the structure is designed.

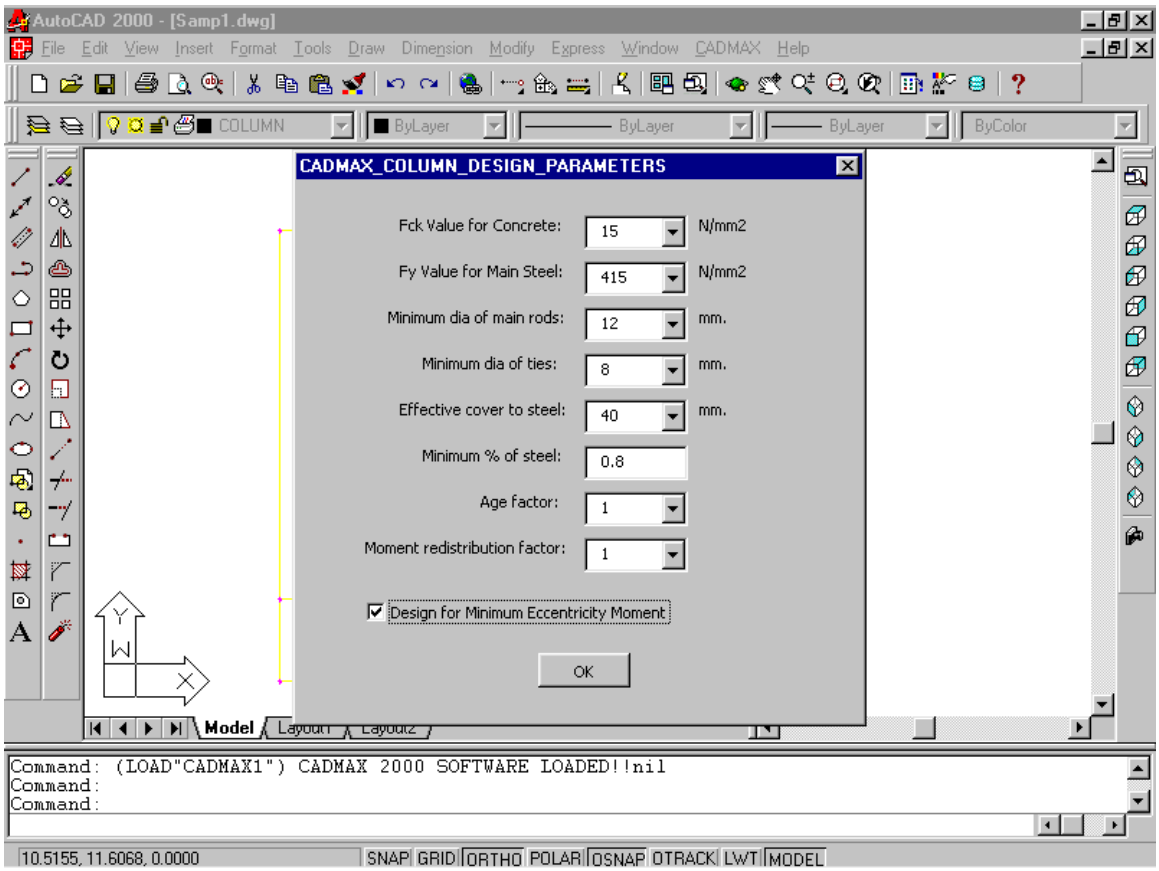
19.DCOLUMN <enter>

You can pick this command from Pop-up menu also. This command is used to do Design of Columns in the Total Structure. The Design procedure followed is as per Limit State Method in IS-456-1983 and Design Aids for IS-456. The Design forces considered is as per the critical forces arrived at the combination of Dead load, Live Load, Wind Load and Earth Quake loads as per IS code provisions.

The pop-up menu to give "DCOLUMN" command is shown below:

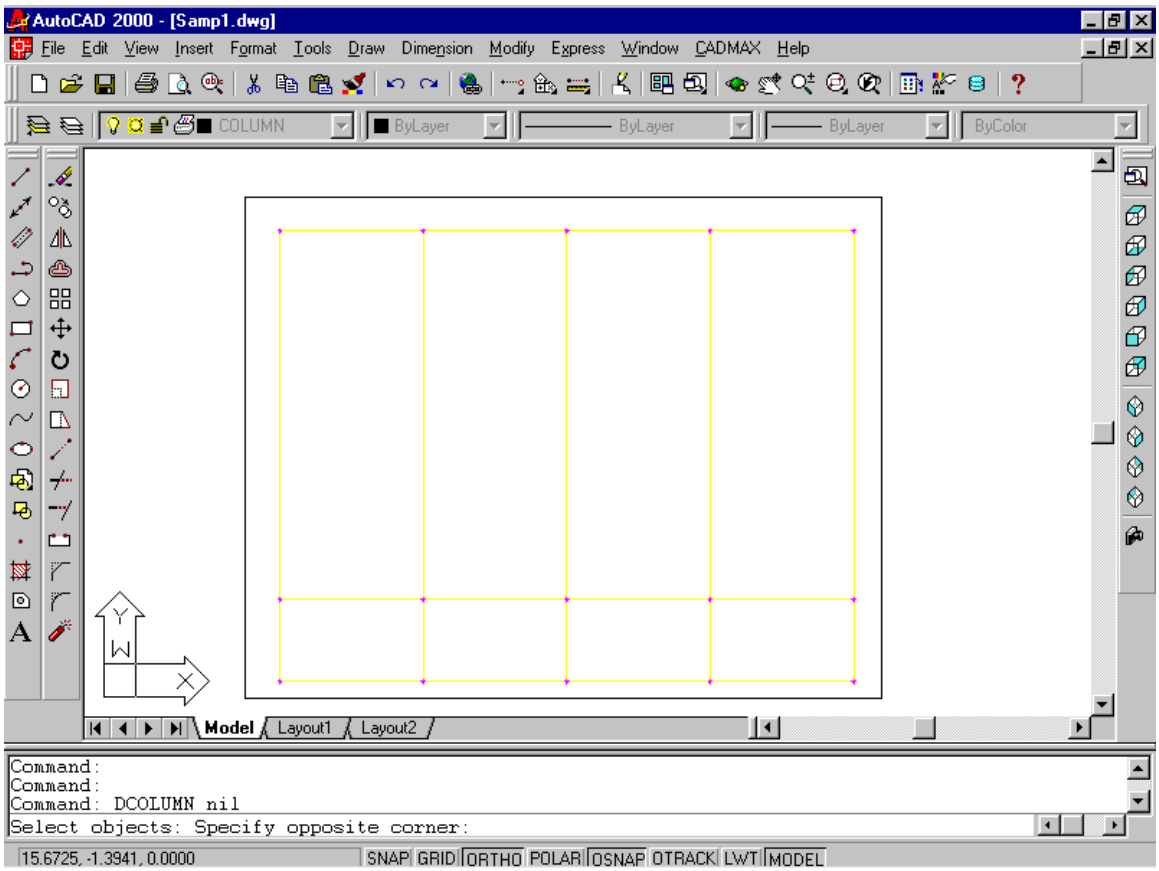


After giving “DCOLUMN” command you will be prompted with a Design Parameters Dialog box as shown below:



In the Column Design Parameters dialog box shown above, Fck Value of Concrete, Fy Value for Main Steel, Minimum dia of main rods, Minimum dia of Ties, Effective Cover to steel are selected from the combo box values. The Minimum % of steel is 0.8% default value is automatically given in the dialog box. If you want to change the Minimum % of steel you can change in the corresponding text box. The Value of Age factor is 1 as default value automatically given in the dialog box. If you want to change the Age factor to 0.8 as per British code you can select it from the combo box values. The Fck Value of Concrete is multiplied by the Age Factor during design procedure. The Moment redistribution factor default value is 1. If you want to change the value of Moment redistribution factor to a lesser value you can select it from the combo box values. The Design moment for column is multiplied by the Moment redistribution factor during design of columns. If you click the check box for "Design for Minimum Eccentricity Moment", the Minimum Eccentricity Moment is considered if the Design Moment is less than the Minimum Eccentricity Moment. The Values given in the present design procedure is given in the above dialog box. Then click "**OK**" button. Now you will be prompted for Select Objects .

Command: **DCOLUMN** <enter>
Select Objects: <Select all the objects as shown below>
Select Objects:<enter>
Command:

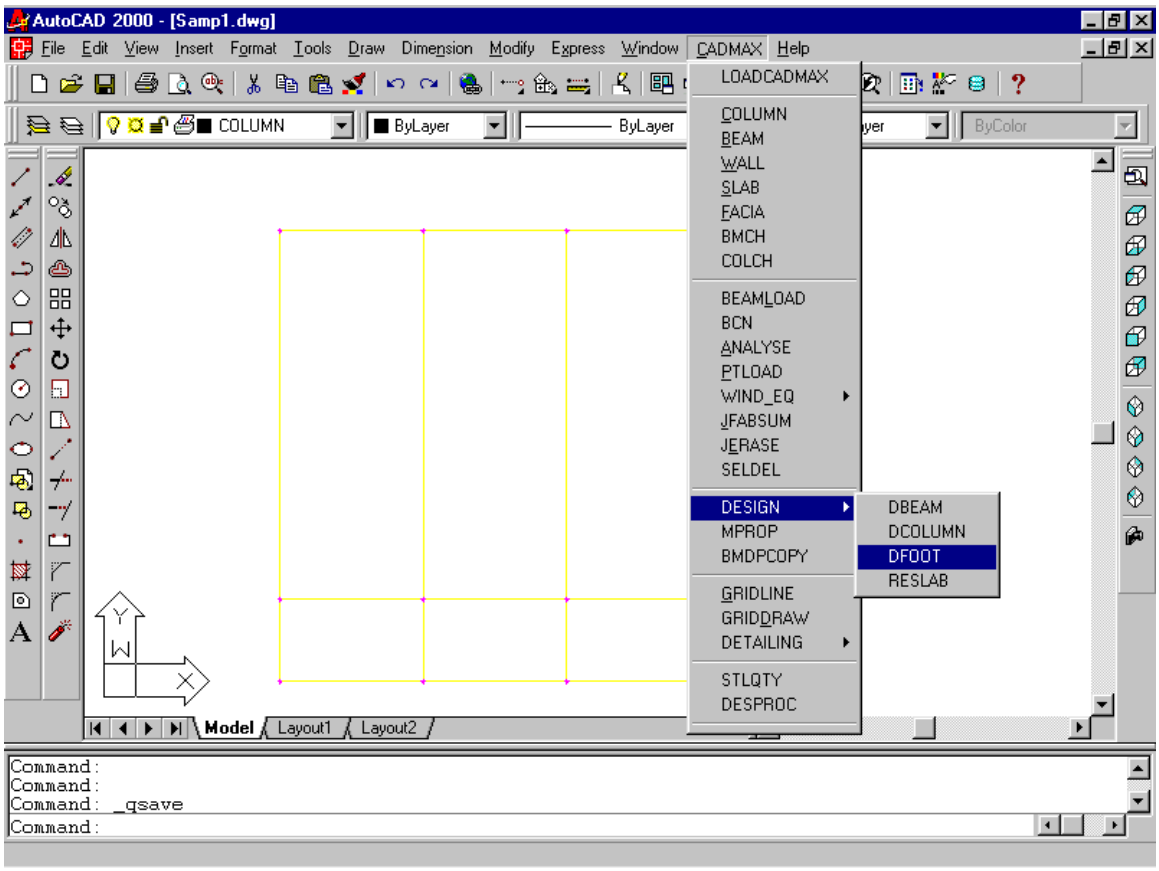


When selected objects are released, the design of all the columns are completed as per design parameters given.

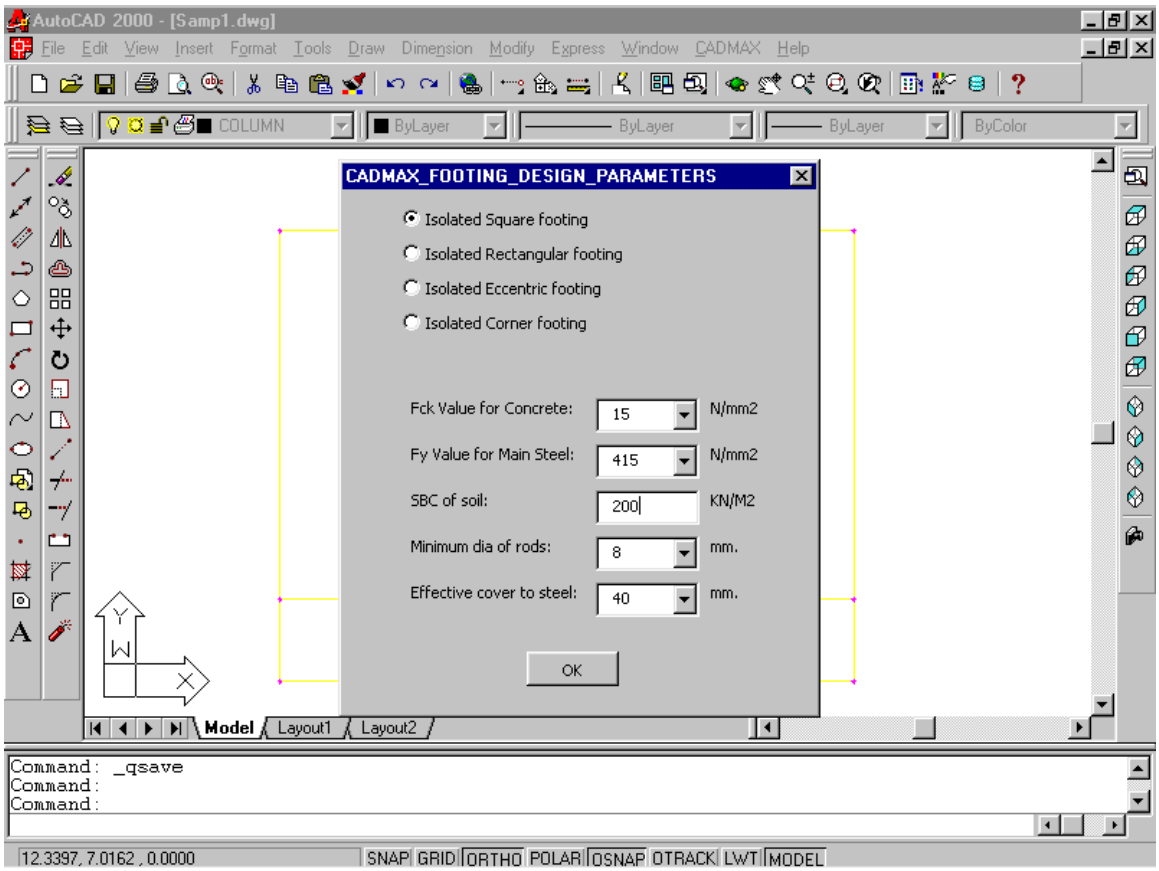
20.DFOOT <enter>

You can pick this command from Pop-up menu also. This command is used to do Design of Footings in the Total Structure. The Design procedure followed is as per Limit State Method in IS-456-1983 and Design Aids for IS-456. The Design forces considered is as per the critical forces arrived at the combination of Dead load, Live Load, Wind Load and Earth Quake loads as per IS code provisions.

The pop-up menu to give “DFOOT” command is shown below:

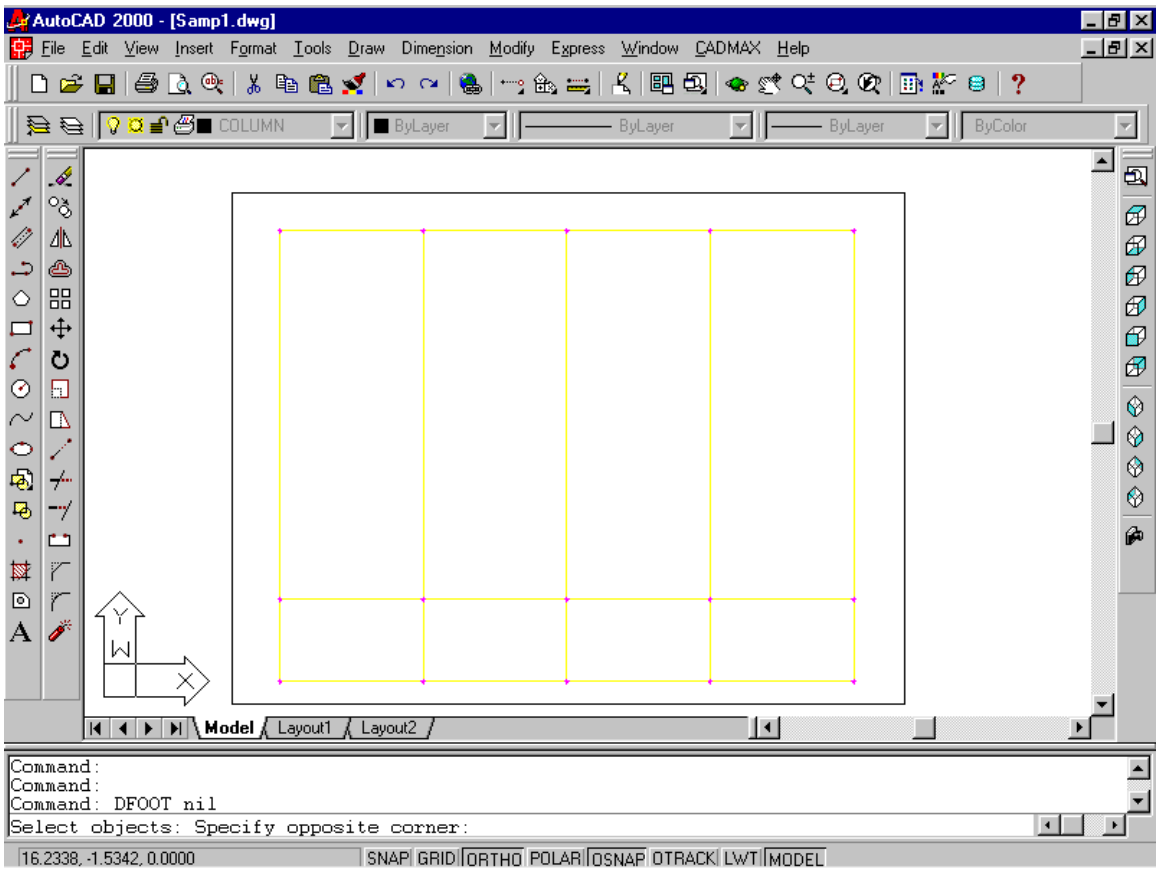


After selection of "DFOOT" Command, you will be prompted with Design Parameters Dialog box as shown below:

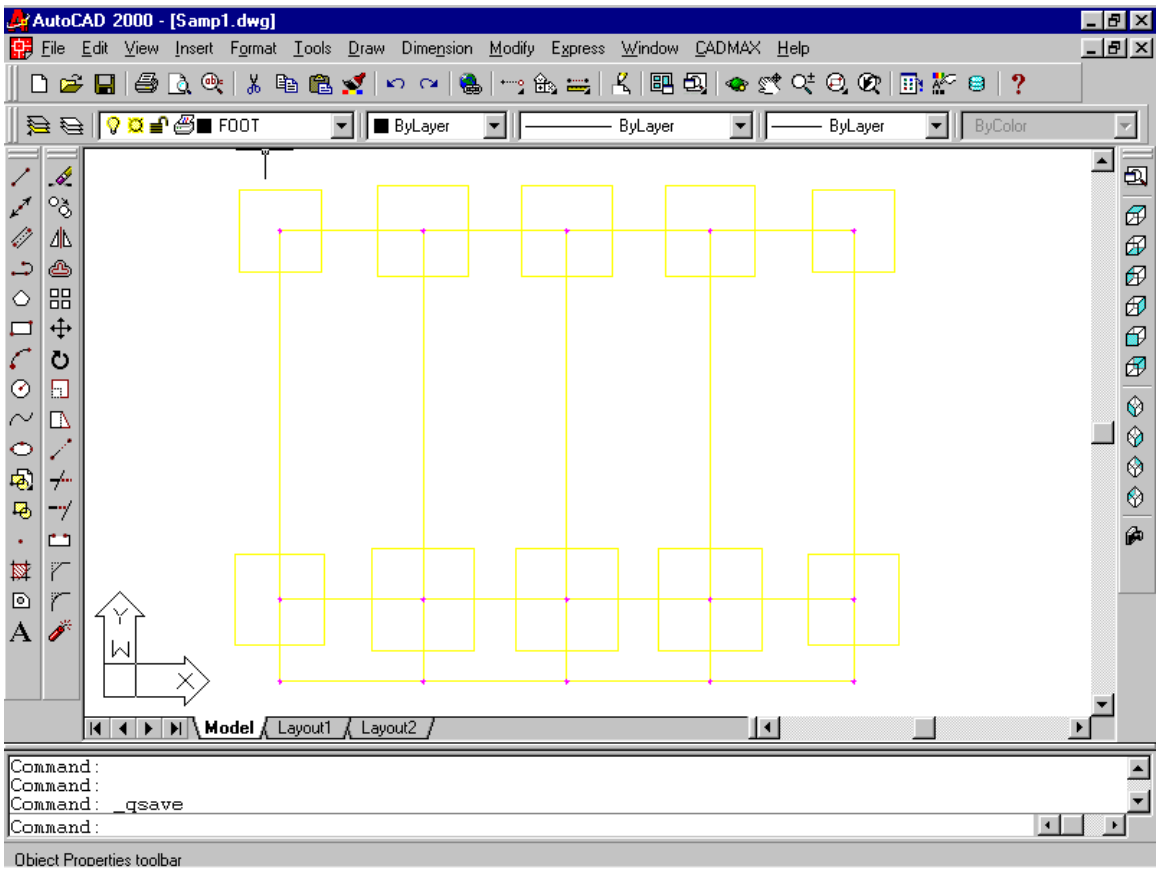


In the above Footing Design Parameters, the option box for Isolated square footings is selected and the other options will be discussed later. The Fck Value for Concrete, Fy Value for Main steel is selected from the corresponding Combo box values. The Safe Bearing Capacity of Soil is given in corresponding Text box. The Minimum dia of rods and effective cover to steel is selected from the corresponding Combo box values. Then click "**OK**" button. Now you will be prompted for Select objects.

Command: **DFOOT** <enter>
Select Objects: <Select all the objects as shown below>
Select Objects: <enter>
Command:



After the selected objects are released, all the footing designs are completed and you will see the all the footings sizes are marked in a separate layer “FOOT” as shown below:



Now, all the members of the structure such as Beams, Columns and Footings are Designed. Now you have to go for detailing of the Structure.

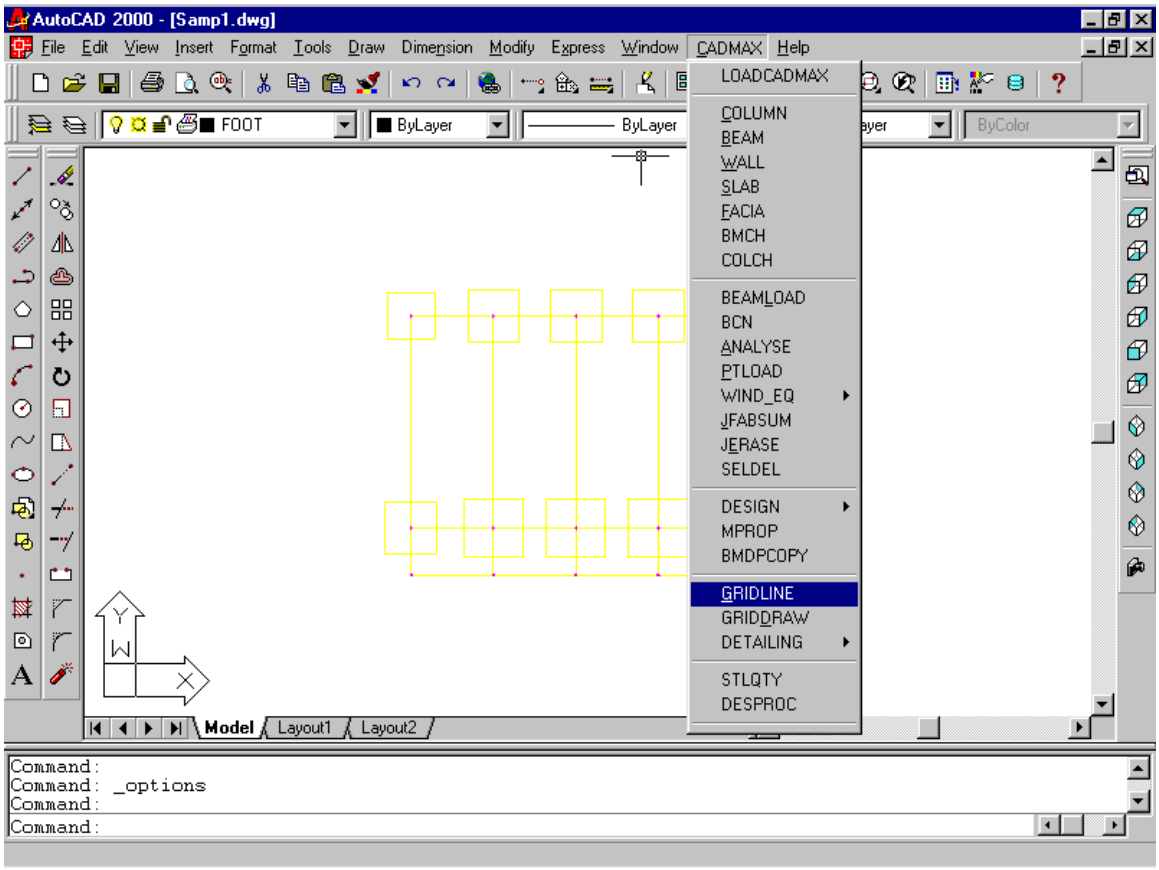
CHAPTER.5.DETAILING OF THE STRUCTURE.

Before detailing of the structure, it is important to draw the grid lines along X, Y directions and name them as A,B,C,D…. along one direction and 1,2,3,4….. along other direction depending upon our convenience. This notation of grid lines along X and Y directions will help us to get an easy way to identify the members such as beams, columns and footings. The detailing of the structures using the above grid lines will be very useful to identify the members easily from the detailed drawings and to adopt them in the field also. The gridlines may be adopted as X1,X2,X3…along one direction and Y1,Y2,Y3…along other direction also. If you want to draw the grid lines for a non regular structure, arch or circular structures named grid lines method may be followed in which each and every junction in plan is given a named joint such as A1,A2,B1,B2….etc as per the convenience and understanding. First we will see how to draw grid lines for the current problem as given below.

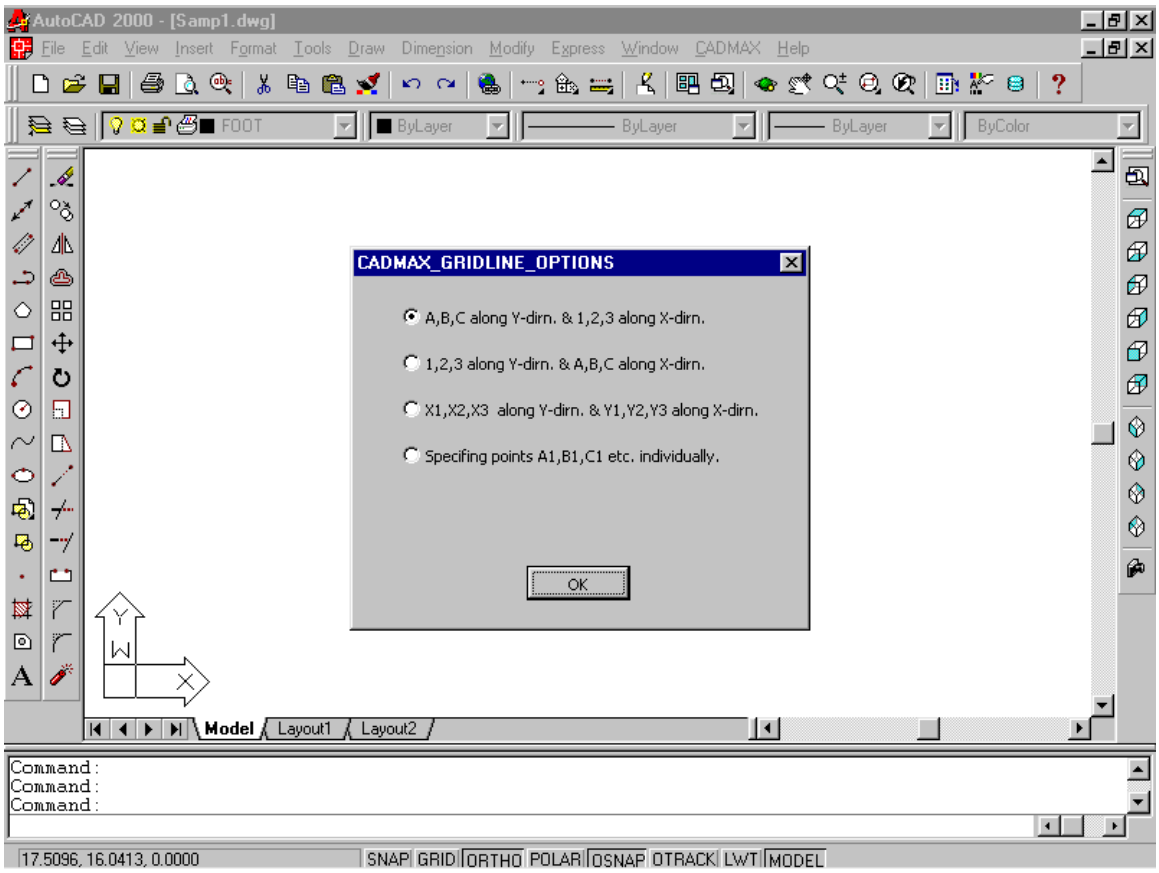
21.GRIDLINE <enter>

You can pick this command from Pop-up menu also. This command is used to do automatically the arrangement of grid lines as per options chosen by us. The drawing can be zoomed to smaller size to enable easy selection of members and detailing of locations of grid lines in the drawing.

The pop-up menu to give "GRIDLINE" command is shown below:



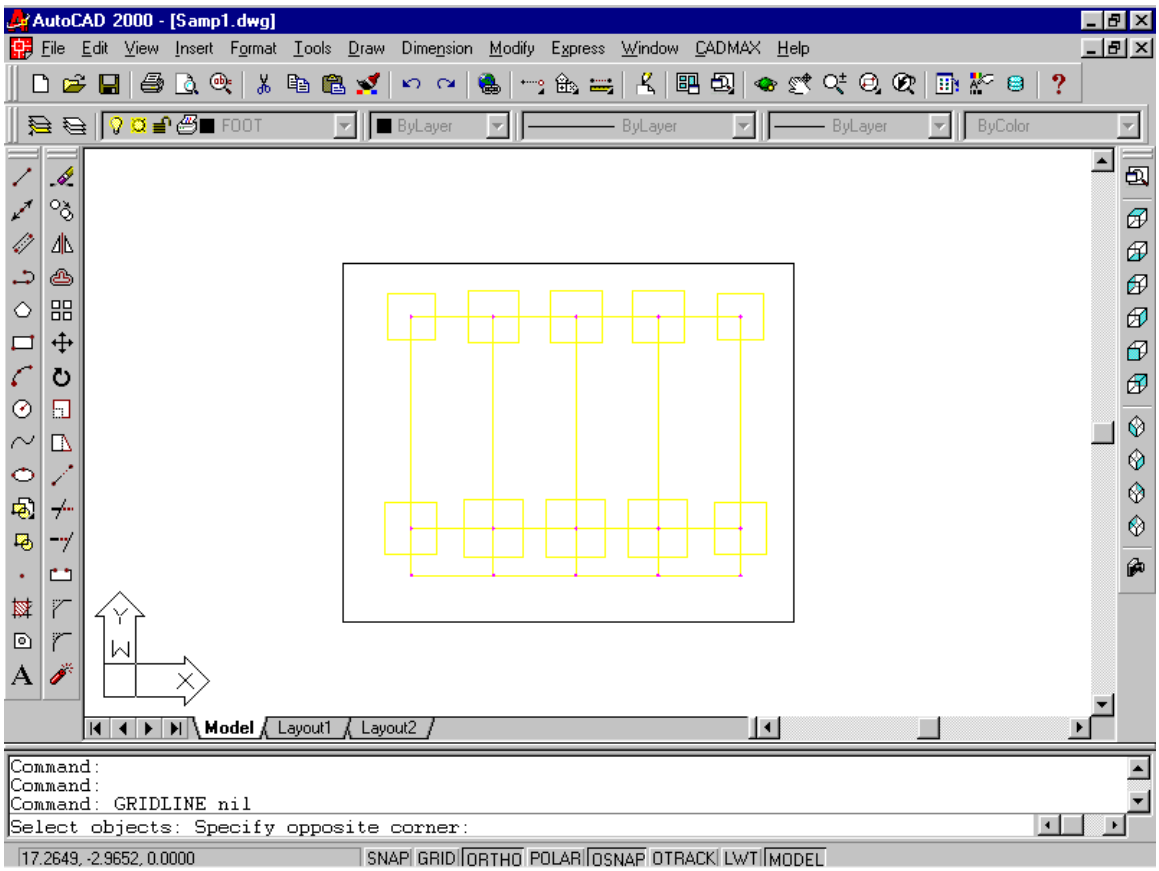
After gridline command is given, you will be prompted for options dialog box as shown below:



In this case the first option A,B,C numbering along X-direction and 1,2,3 numbering along Y-direction is clicked and then press "**OK**" button.

Now, you will be prompted for selection of objects. Now select all the objects in the screen as shown below.

Command: **GRIDLINE** <enter>
Select Objects: <Select all the objects in the drawing as shown below>
Select Objects: <enter>

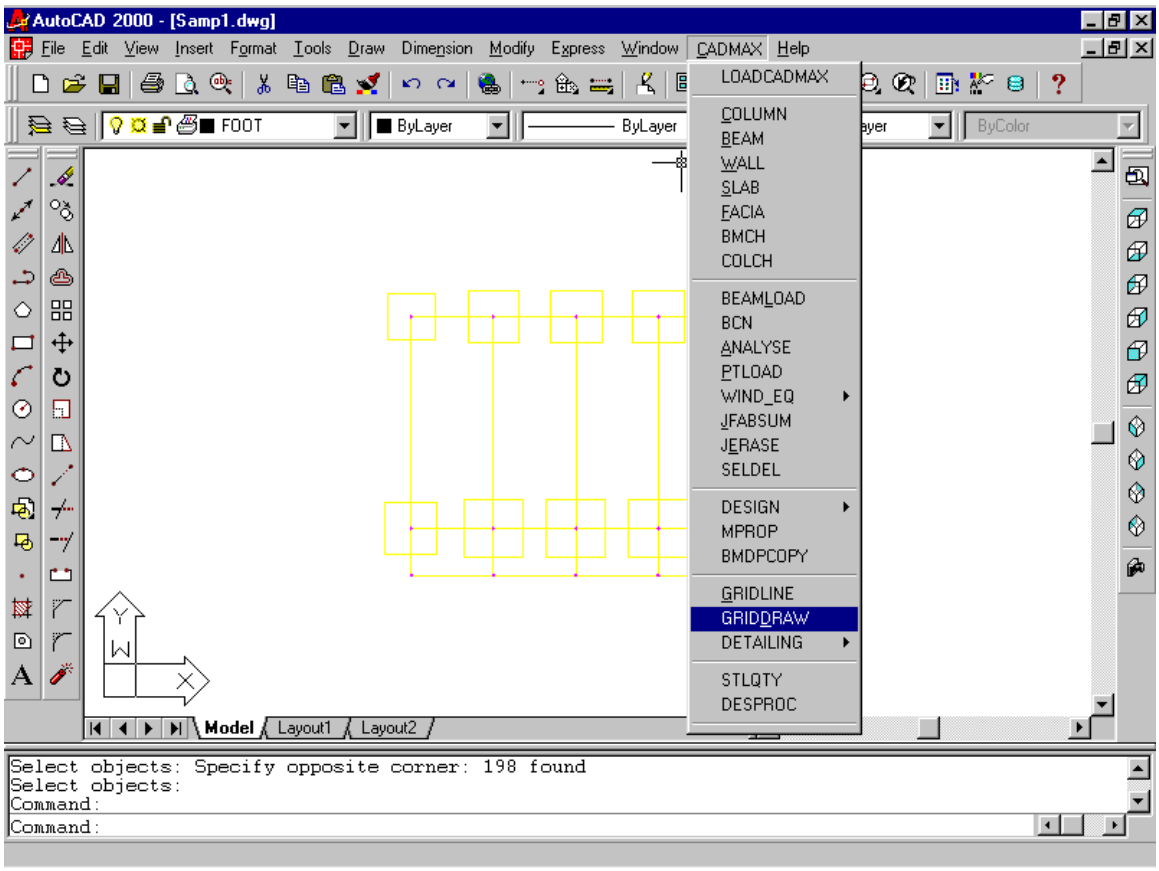


After the selection set is released, the arrangement of grid lines is finalized and saved in the drawing according to the options chosen.
Now you have to draw the grid lines that have been finalized in the "GRIDLINE" command in the previous step. For this "GRIDDRAW" command is used.

22.GRIDDRAW <enter>

You can pick this command from Pop-up menu also. This command is used to do draw automatically the arrangement of grid lines as per options chosen by us. It is important to note that "OSNAP" button is made off before giving this command.

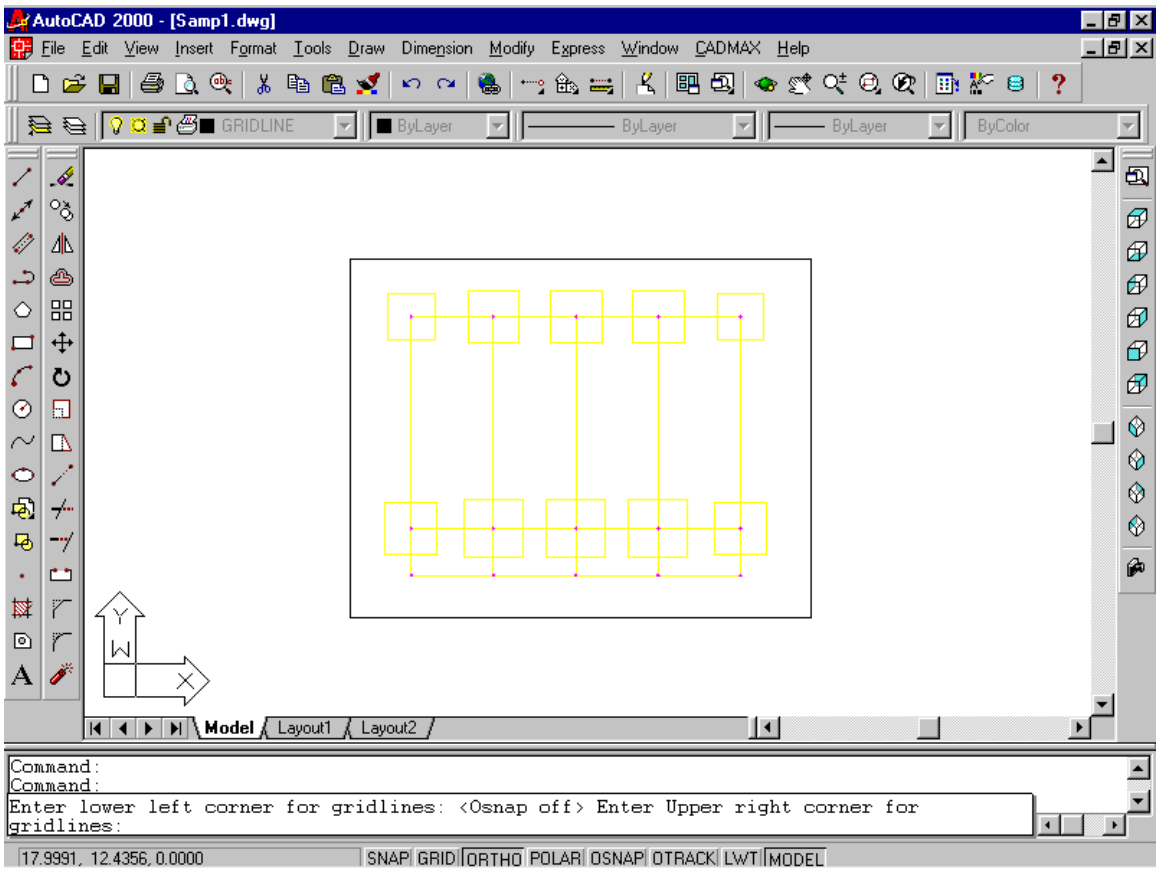
The pop-up menu to give "GRIDDRAW" command is shown below:



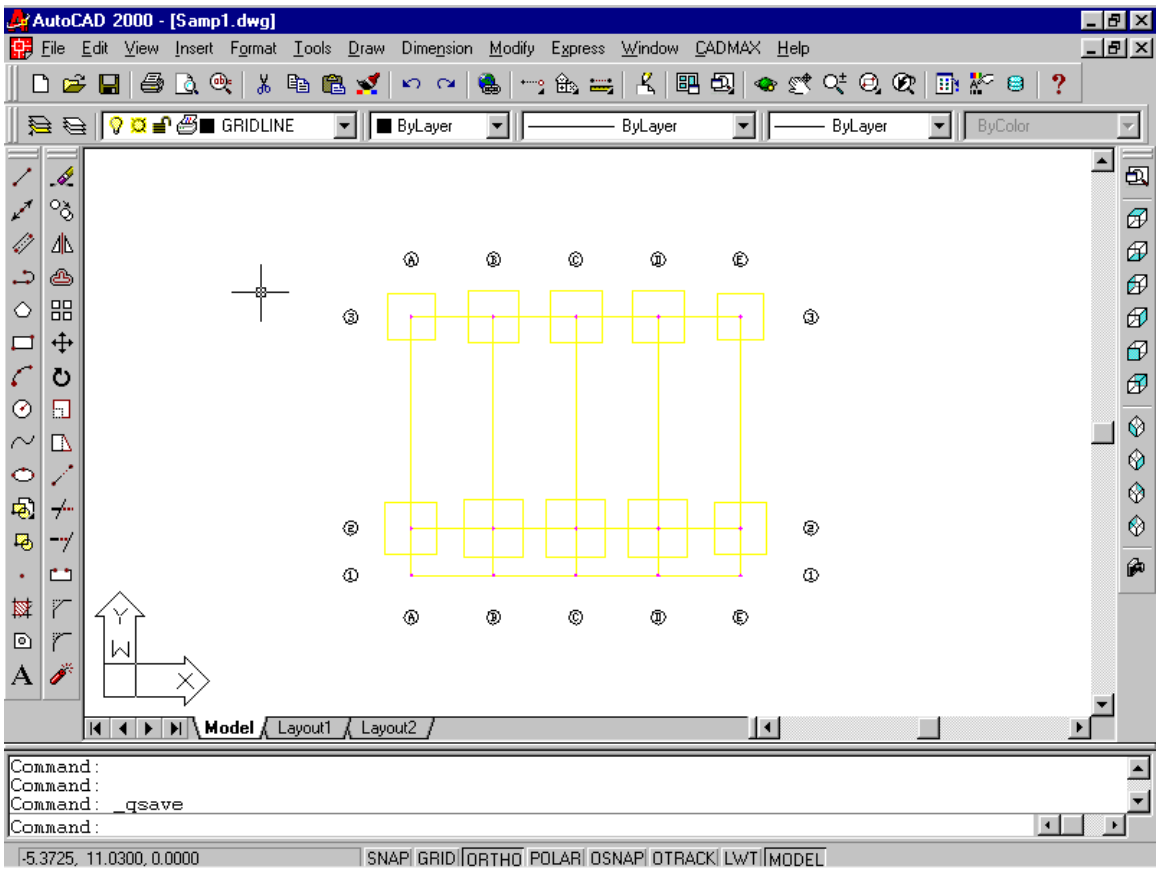
Now, You will be prompted that "You want to use named gridlines:<Y/N>:" and answer the above question with "N" since in this case we have not opted for named grid lines in the previous GRIDLINE command. Then you will be prompted for selecting the lower left corner and upper right corner in the drawing. The corners should be selected in such a way that the grid notations are written well located outside the drawing as shown below.

Command: **GRIDDRAW** <enter>
You want to use named gridlines:<Y/N>:n
Enter lower left corner for gridlines: <pick the lower left corner as shown below>
Enter Upper right corner for gridlines: <pick the upper right corner as shown below>

Command:



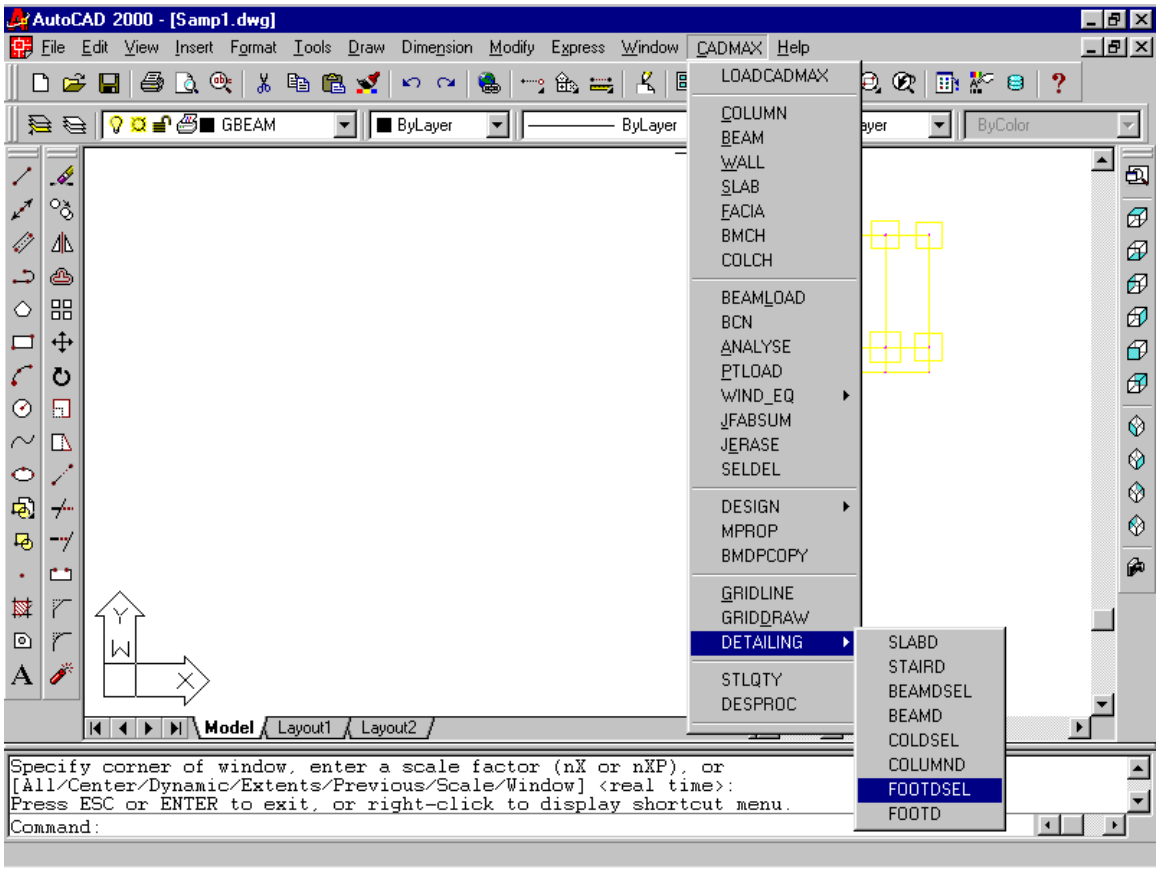
After the selection set is released, the Gridlines according to our option will be drawn in the drawing as shown below. The grid line notations written within the circles will be drawn in a separate layer “GRIDLINE” automatically so that you can “ON” or “OFF” the “GRIDLINE” layer as per your requirement.



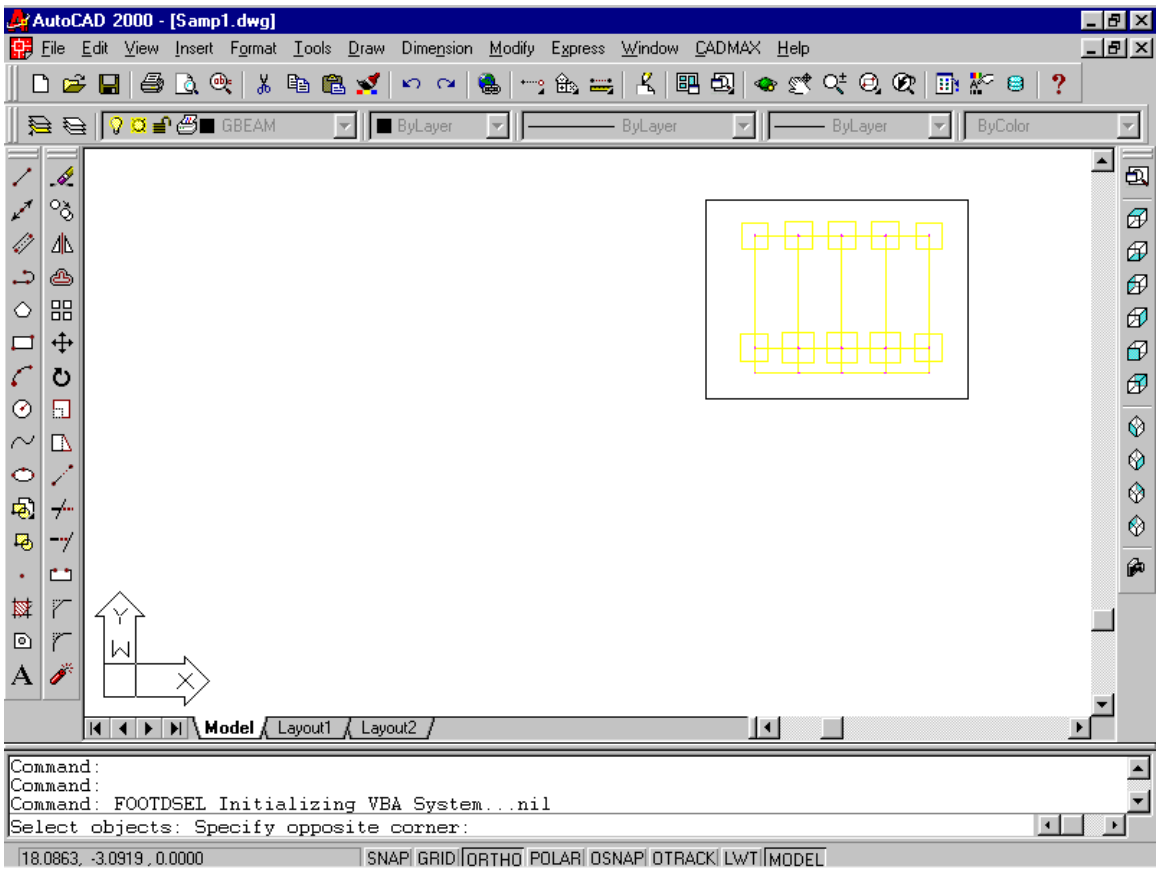
23.FOOTDSEL <enter>

You can pick this command from Pop-up menu also. This command is used to do selection of members for footing detailing. It is important to note that "OSNAP" button is made off before giving this command. To have more space for detailing the drawing may be zoomed into smaller size and pan to right top corner. The layer "GRIDLINE" is to be made "OFF" to avoid selecting them.(The selection of grid notations in "GRIDLINE" layer will lead to errors while detailing. Hence it is better to "OFF" that layer). Set the current layer as "GBEAM" or "BEAM" or "COLUMN".

The pop-up menu to give "FOOTDSEL" command is shown below:



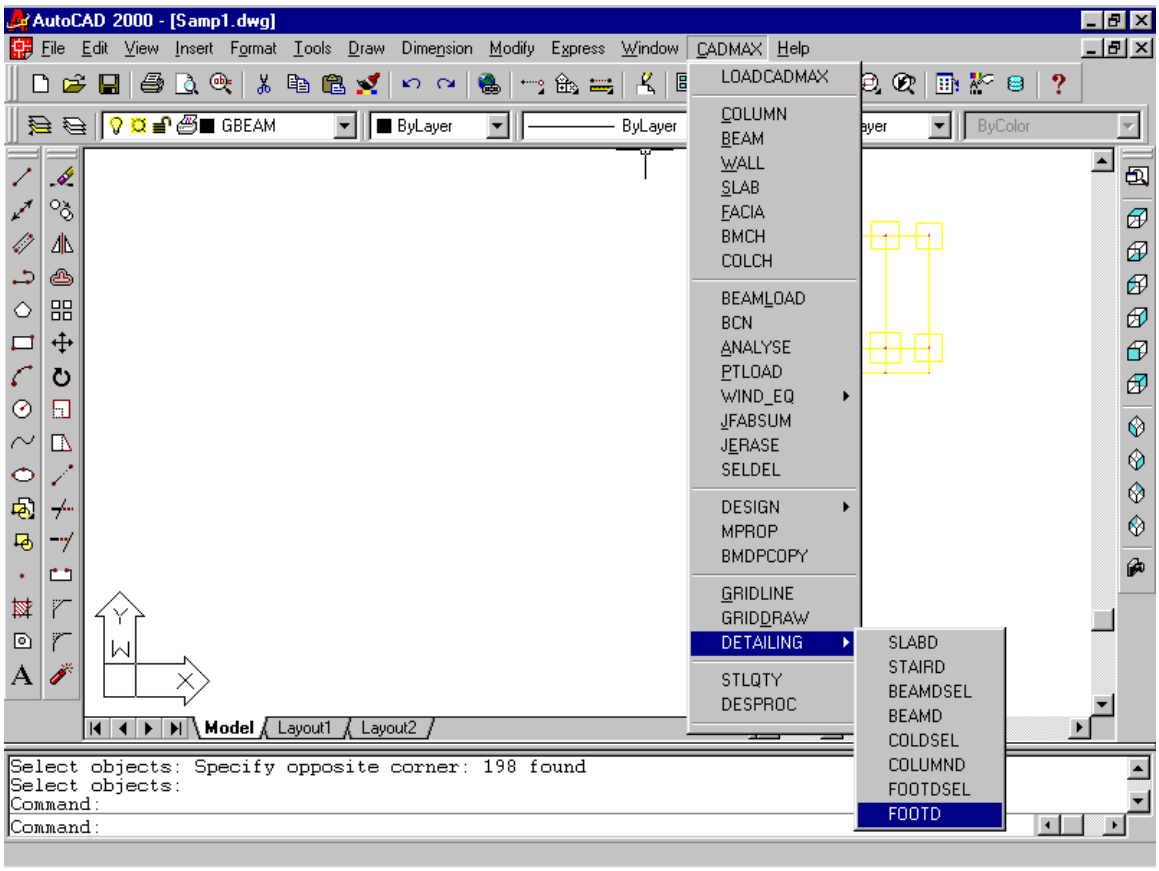
Command: **FOOTDSEL** <enter>
Select Objects: <select all the objects in the drawing as shown below>
Select Objects: <enter>



When the selection set is released, the command is over and now it is ready to draw the detailing of the footings.

24.FOOTD <enter>

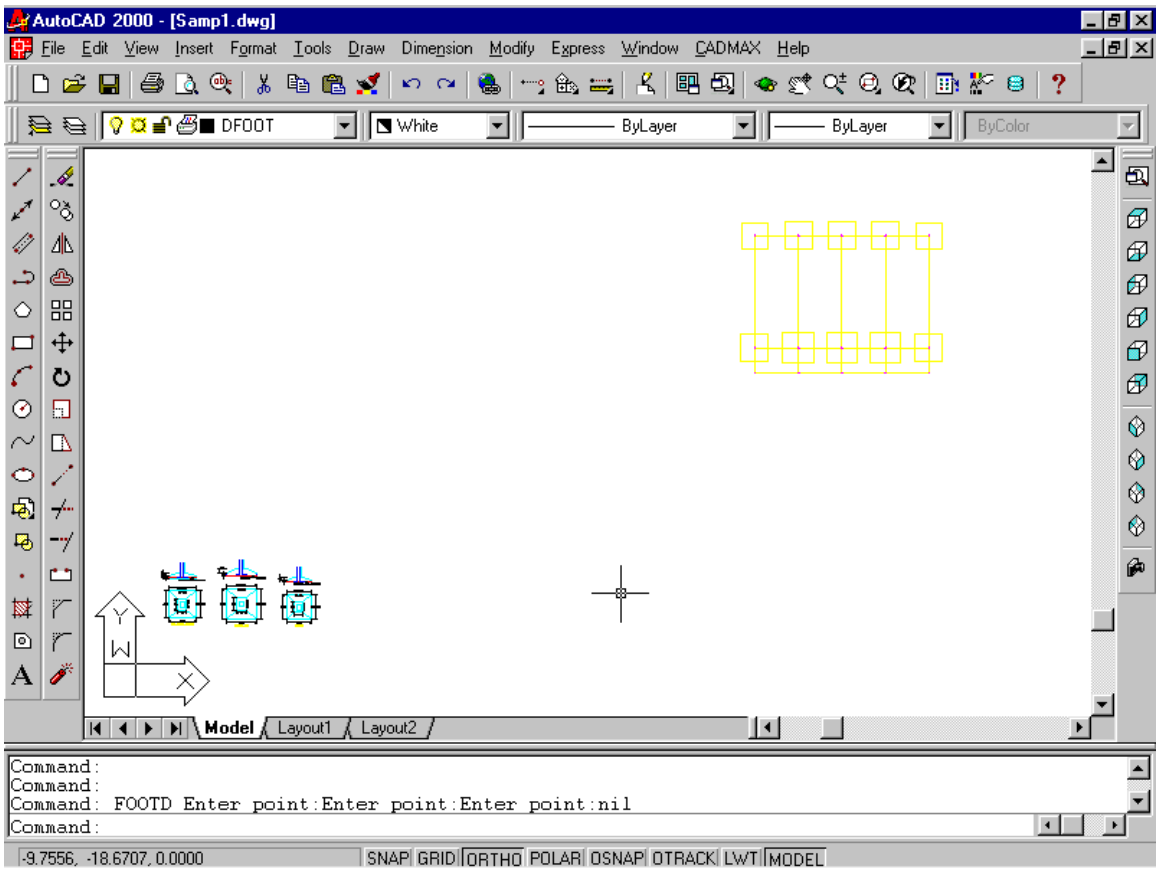
You can pick this command from Pop-up menu also. This command is used to do draw footing detailing along with grouping of similar footings. It is important to note that "OSNAP" button is made off before giving this command. To have more space for detailing the drawing may be zoomed into smaller size and pan to right top corner.
The pop-up menu to give "FOOTD" command is shown below:



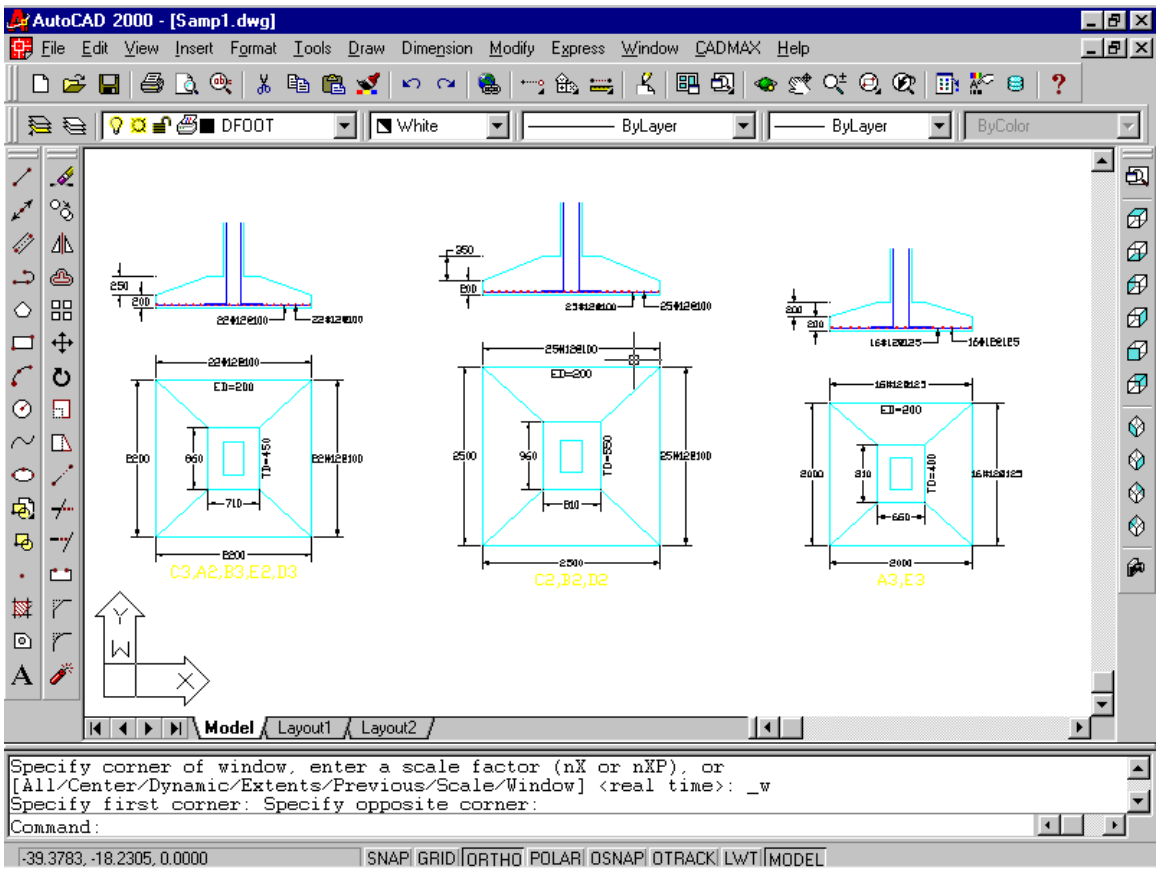
Command: **FOOTD** <enter>
Enter point: <pick a point in the lower left area free space in the drawing>
(Now you will see a footing details will be drawn just above the picked point and grouping of footings will be given there)
Enter point: <pick next point in the lower left free space next to the footing details drawn>
(Now you will see next footing details will be drawn just above the picked point and grouping of footings will be given there)
Enter point: <pick next point in the lower left free space next to the footing details drawn>
(Now you will see next footing details will be drawn just above the picked point and grouping of footings will be given there)

When all the footings are drawn, you will not be prompted for Enter point and the command prompt will come automatically. You can see that all the footing details are drawn in the separate layer "DFOOT" so that it will be convenient for us to "ON" or "OFF" the layer whenever required.

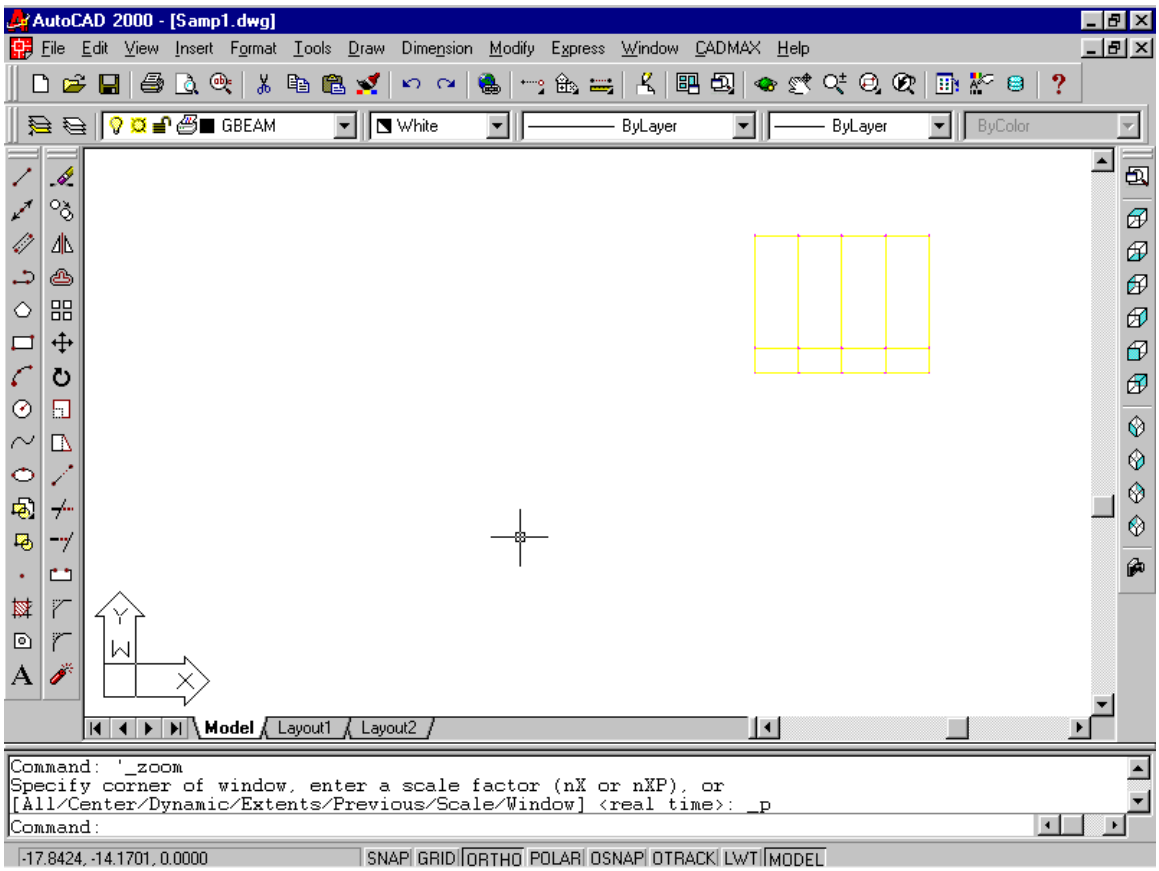
Now you will see the footing details as shown below:



To see the detailing of footings in clear, you can zoom the footing details as shown below:



By giving Zoom previous command you will get the previous view as shown previously. Now you can make "OFF" the layers "DFOOT" and "FOOT" and set the current layer as "COLUMN" or "BEAM" or "GBEAM" since the detailing of footings is completed. Now you will see the screen as shown below:

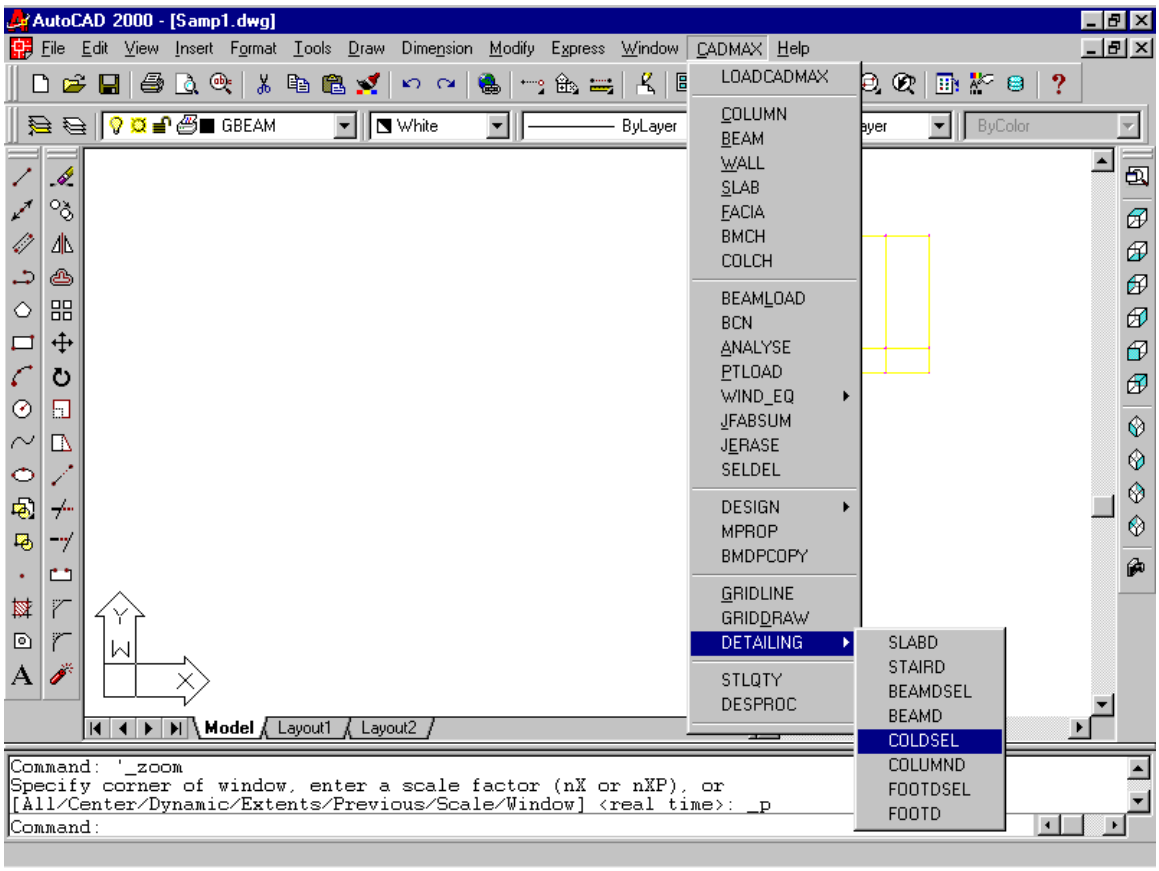


Now you can do column detailing as described below:

25.COLDSEL <enter>

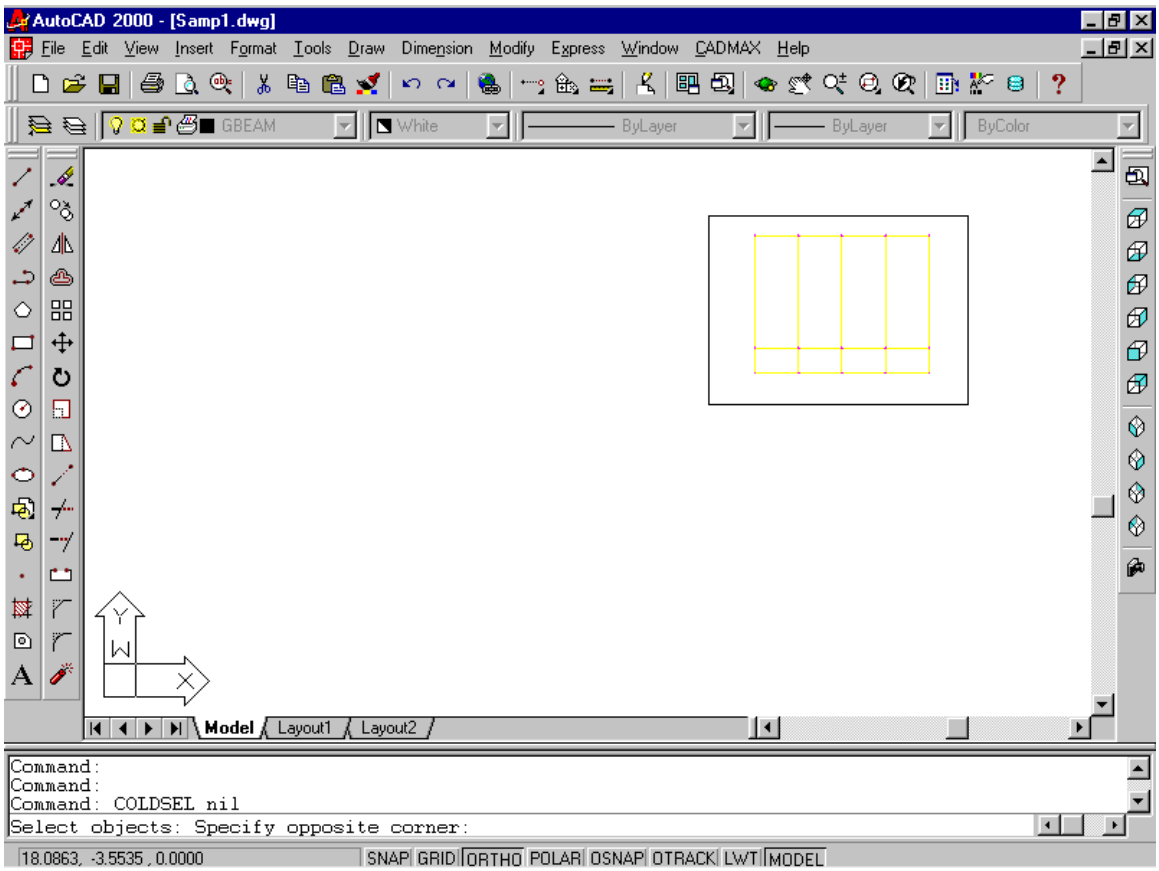
You can pick this command from Pop-up menu also. This command is used to do selection of members for column detailing and the elevation of columns between floor levels so that the curtailment of rods may vary between floor levels only. .

The pop-up menu to give “COLDSEL” command is shown below:

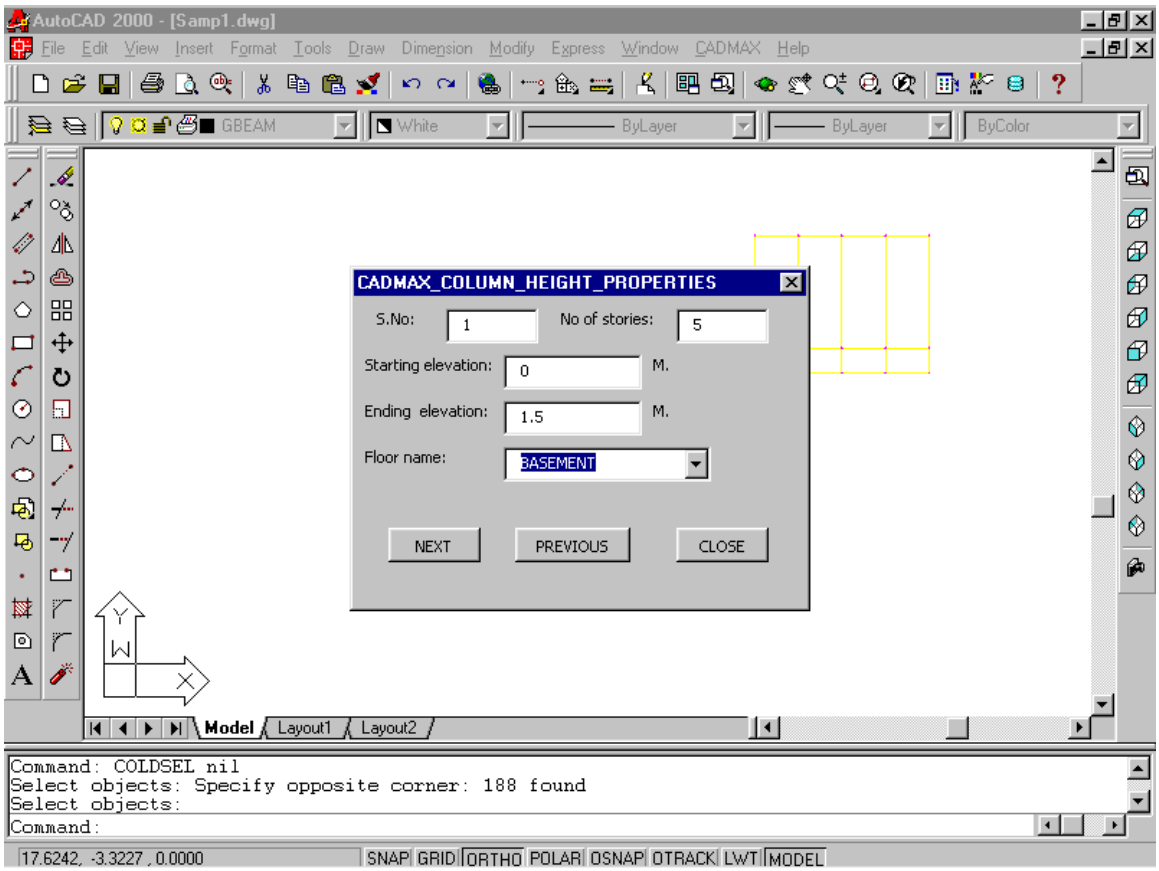


After giving the "COLDSEL" command you be prompted for select objects and after selection of all the members you will be prompted with a dialog box to give the elevation levels of each floor level .

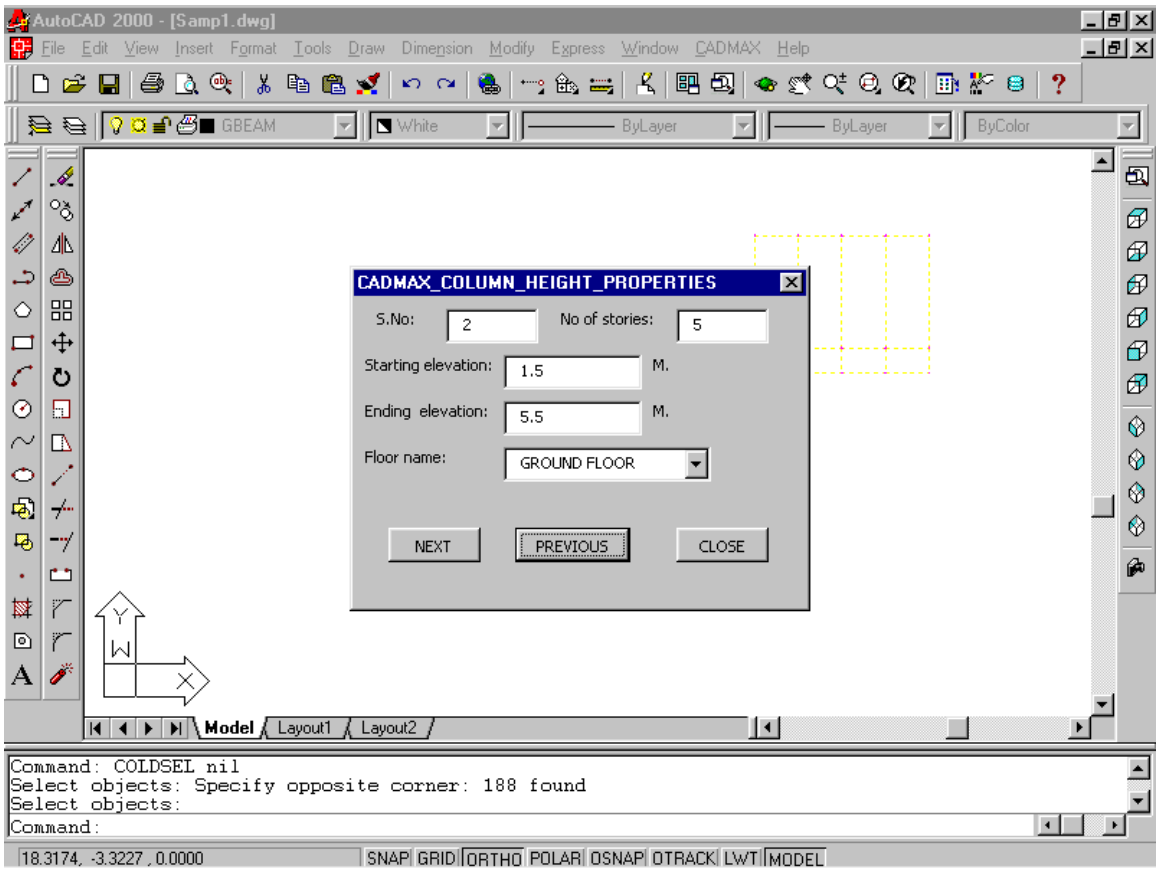
Command: **COLDSEL** <enter>
Select Objects: <select all the objects as shown below>
Select Objects: <enter>



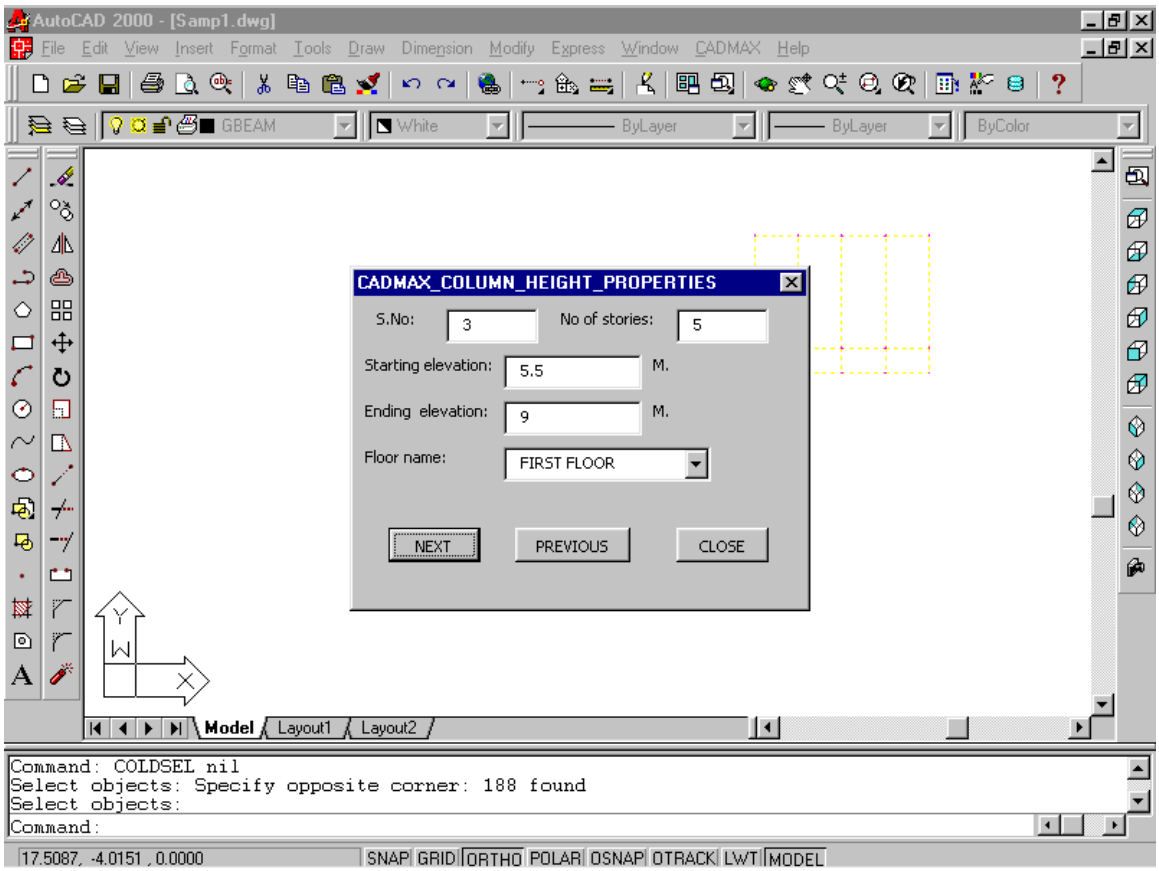
Now you will be prompted with the following dialog box , to give the details of elevation of floor levels:



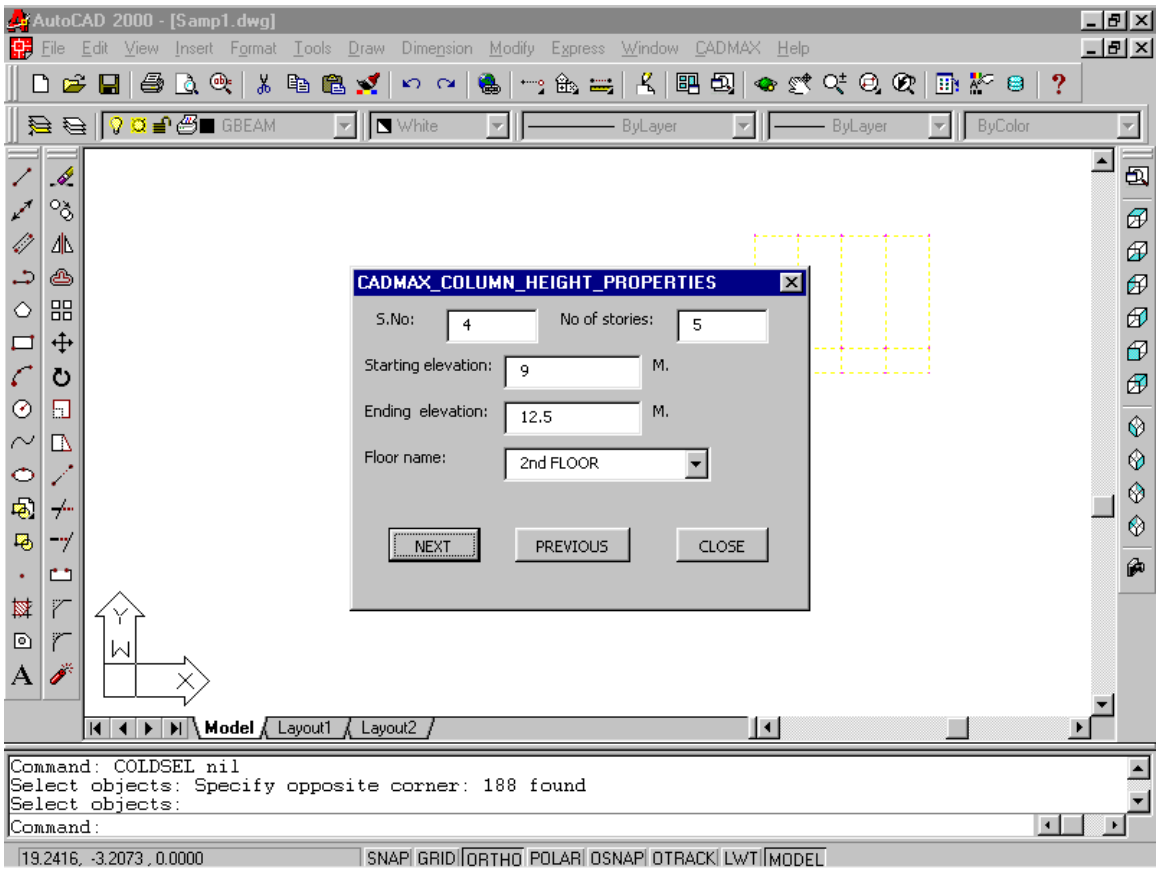
In the above dialog box , Fill the No of stories as 5 in the Textbox , and Starting elevation as 0 in the Textbox, and Ending elevation as 1.5 in the Textbox, and Floor name as "BASEMENT" from the Combo box and then press the "NEXT" button. Now you will see the same dialog box with S.No. Textbox changed automatically to 2 and remaining Textboxes such as Starting elevation, Ending elevation, Floor names are changed as follows and then "NEXT" button is pressed.



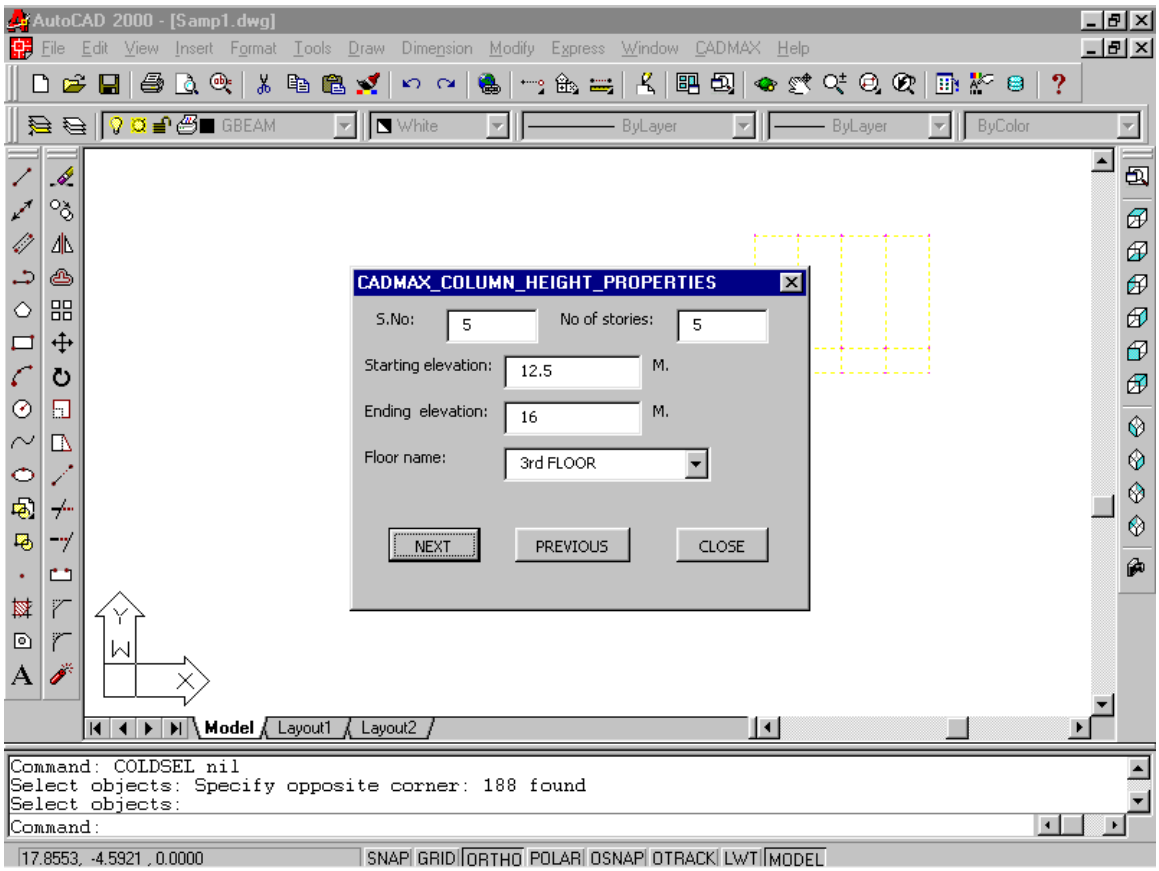
After pressing “NEXT” button, You will be prompted with the same dialog box with S.No. Textbox changed automatically to 3 and then Starting elevation, Ending elevation and Floor name are given as follows and then “NEXT” button is pressed.



After pressing “NEXT” button, You will be prompted with the same dialog box with S.No. Textbox changed automatically to 4 and then Starting elevation, Ending elevation and Floor name are given as follows and then “NEXT” button is pressed.



After pressing “NEXT” button, You will be prompted with the same dialog box with S.No. Textbox changed automatically to 5 and then Starting elevation, Ending elevation and Floor name are given as follows and then “NEXT” button is pressed.

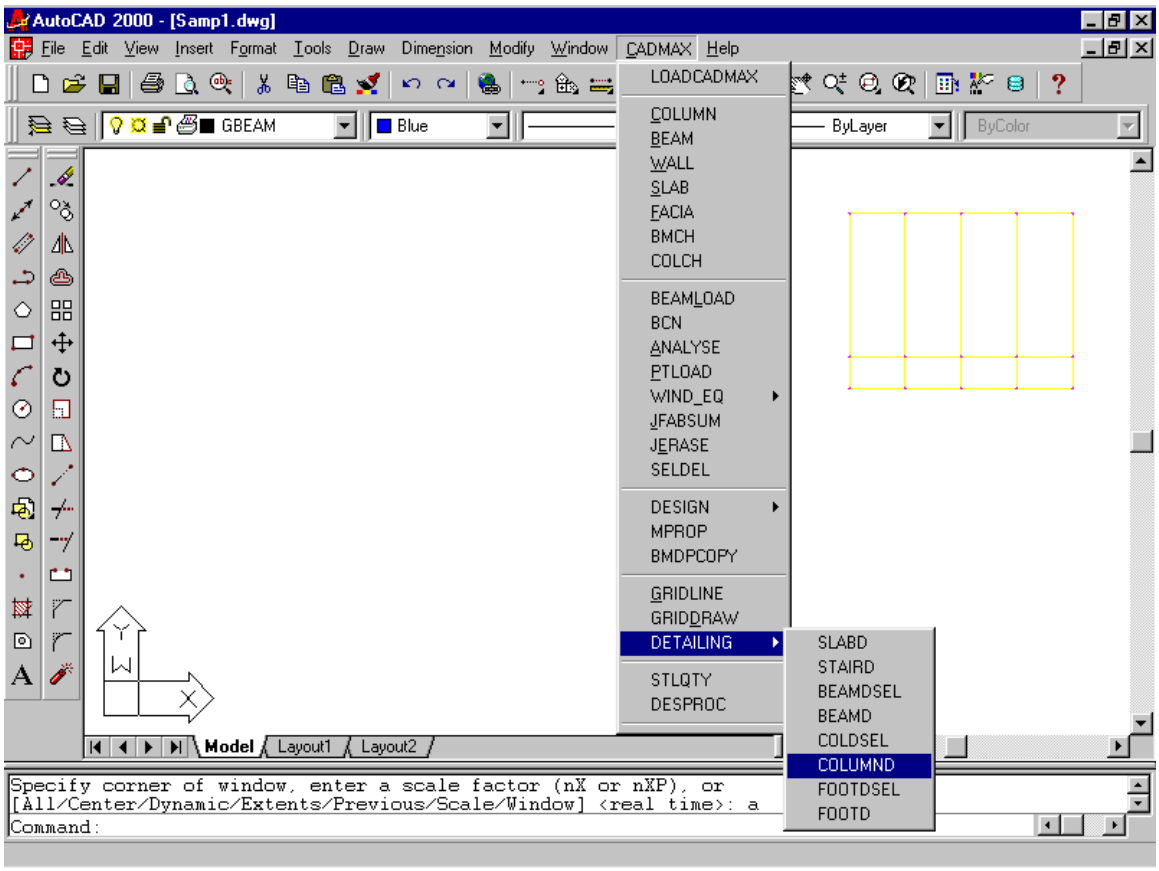


After pressing "NEXT" button you will not find change in S.No. Textbox, since the No of stories and S.No. are equal to 5. Now press "CLOSE" button. Now the dialog box will disappear. With this step the "COLDSEL" command for selection of columns for detailing is over. Now, the Column detailing is done by "COLUMND" command as follows:

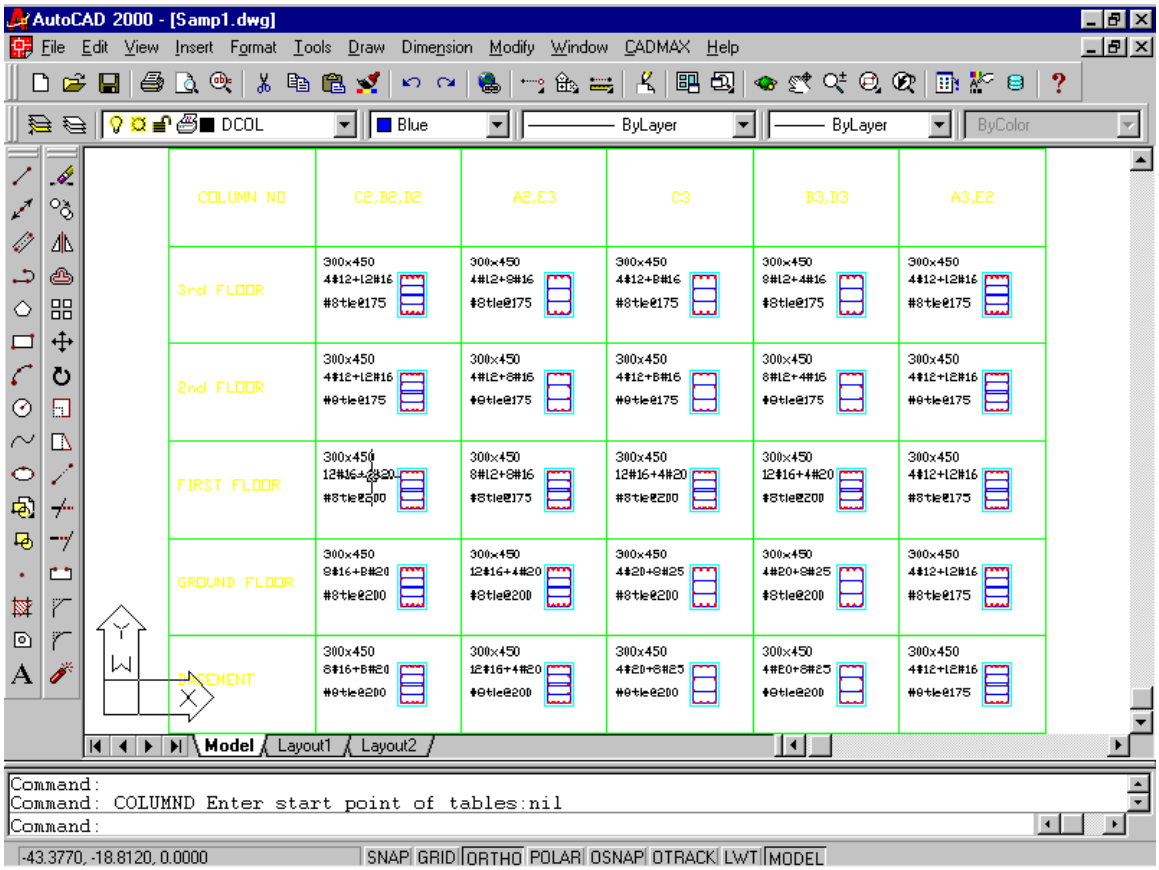
.**26.COLUMND** <enter>

You can pick this command from Pop-up menu also. This command is used to do draw column detailing of the members of the various floors between floor levels given in the previous command.

The pop-up menu to give "COLUMND" command is shown below:


AutoCAD 2000 - [Samp1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Window CADMAX Help
GBEAM
Blue
ByLayer
ByColor
LOADCADMAX
COLUMN
BEAM
WALL
SLAB
FACIA
BMCH
COLCH
BEAMLOAD
BCN
ANALYSE
PTLOAD
WIND_EQ
JFABSUM
JERASE
SELDEL
DESIGN
MPROP
BMDPCOPY
GRIDLINE
GRIDDRAW
DETAILING
STLQTY
DESPROC
SLABD
STAIRD
BEAMDSEL
BEAMD
COLDSEL
COLUMND
FOOTDSEL
FOOTD
Model
Layout1
Layout2
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: a
Command:

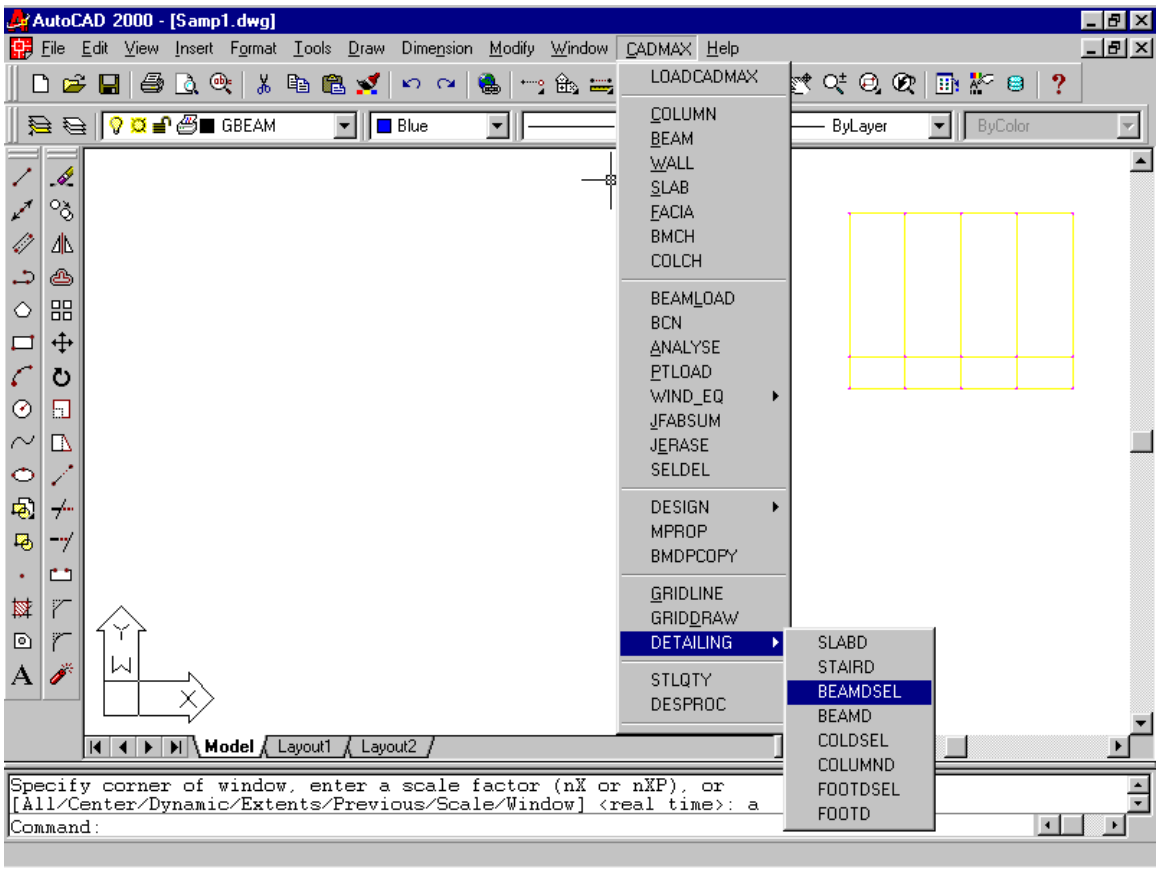
Command: **COLUMND** <enter>
Enter Start point of Tables: <pick a point to draw Detailing of columns in tabular form>
Now you will see the detailed drawing of tables in a separate layer "DCOL" as shown Below:



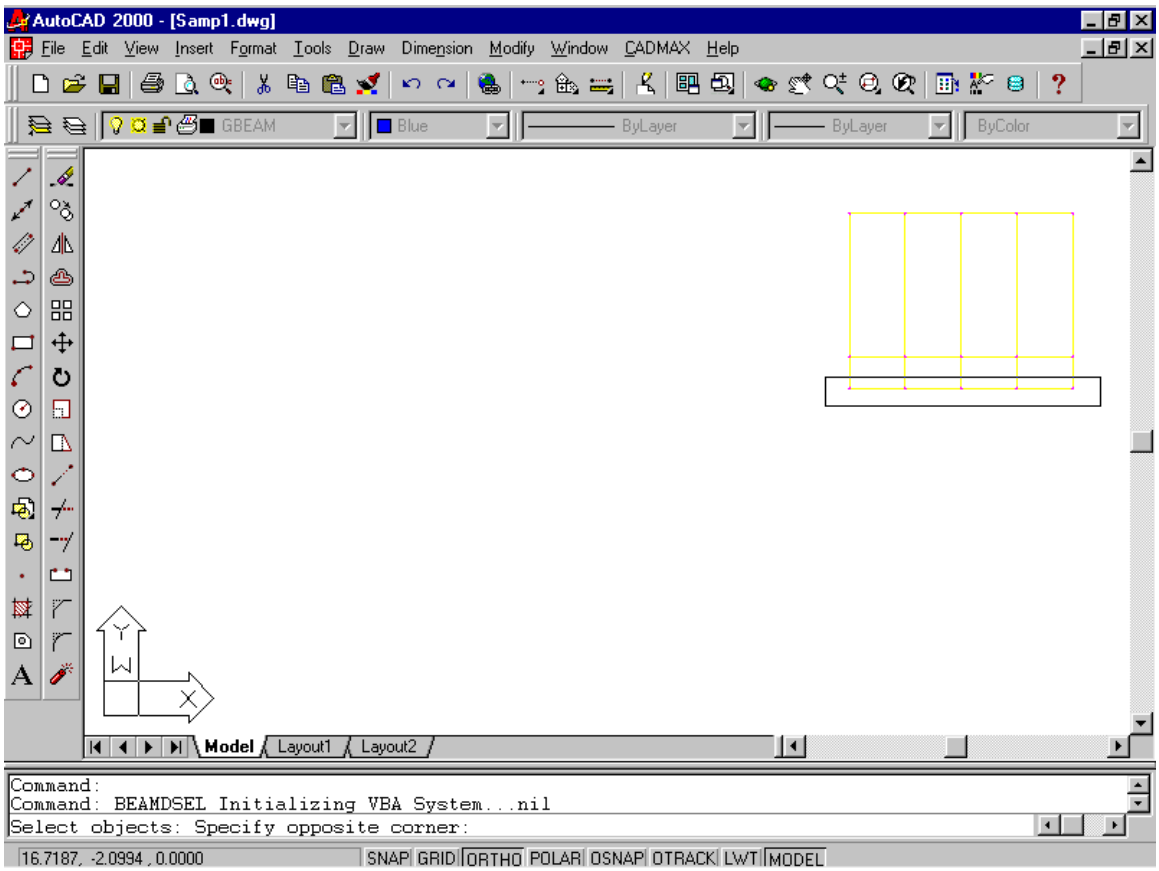
Now the detailing of columns for the structure is done. Now we can go for Detailing of beams with the following commands:

27. BEAMDSEL <enter>

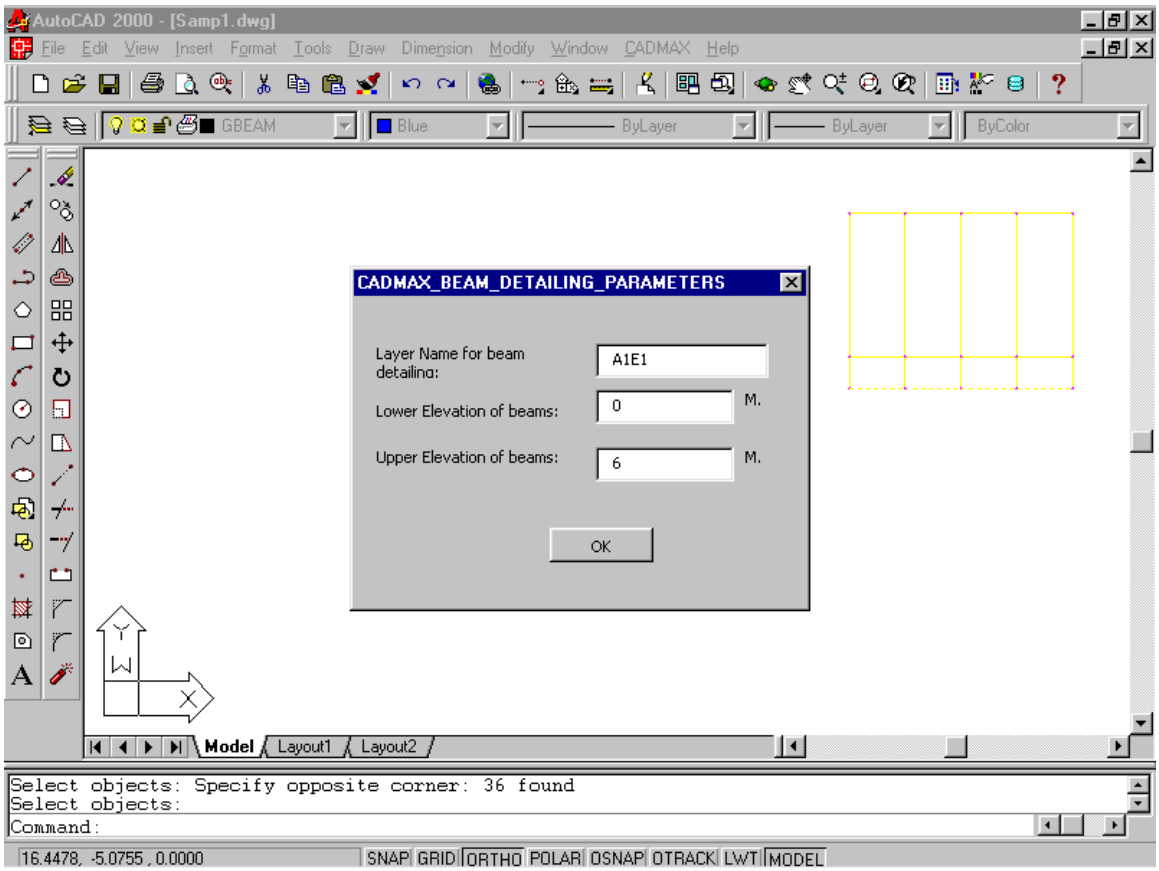
This command can be given directly at the command prompt or it can be can be selected from the CADMAX pop up menu as follows:



Command: **BEAMDSEL** <enter>
Select objects: <select the objects along single direction only>
For example now you are selecting edge beams on cantilever only as shown below:



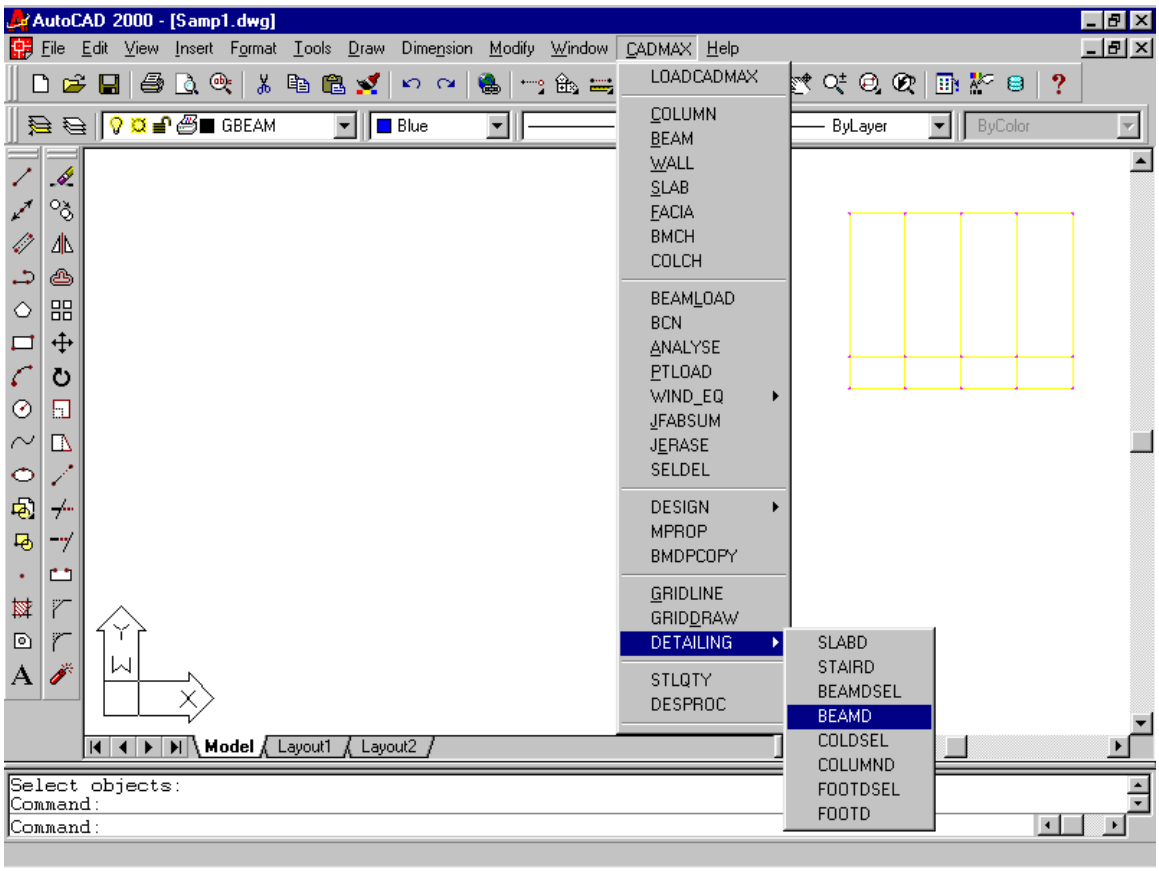
Now you selected the beams along single line along frame name A1E1 only.
After this, you will be prompted with dialog box to indicate the Layer name to draw the detailing of selected beams and the starting elevation of beams and ending elevation of beams to detail only the beams within the selected elevations as below:
Fill up the details as shown below to draw all the beams in between 0 to 6m only.



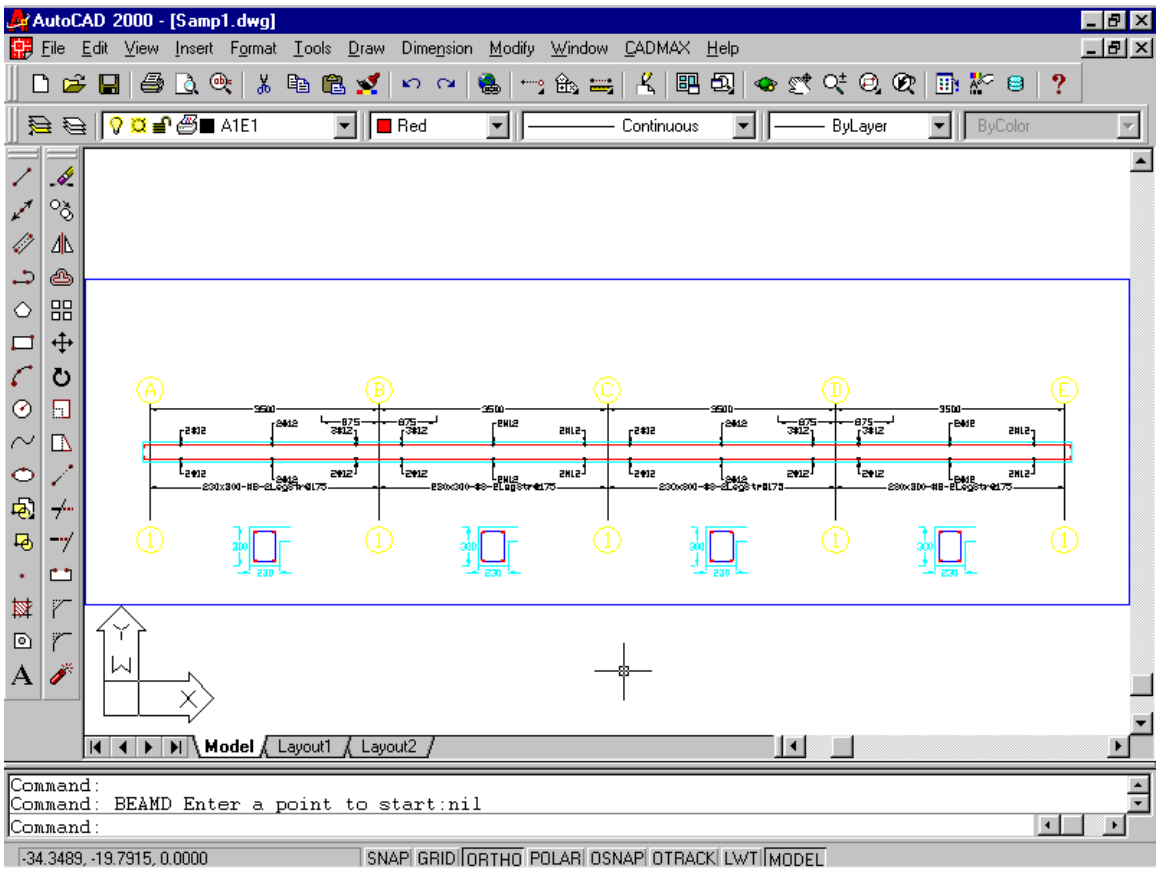
Then click OK. Now the from the selected objects only the beams within the elevation of 0 to 6m only selected for detailing and now the command prompt appears to draw the detailing of Beams selected using BEAMD command.

28.BEAMD <enter>

This command can be given directly at the command prompt or selected from CADMAX pop up menu as follows:

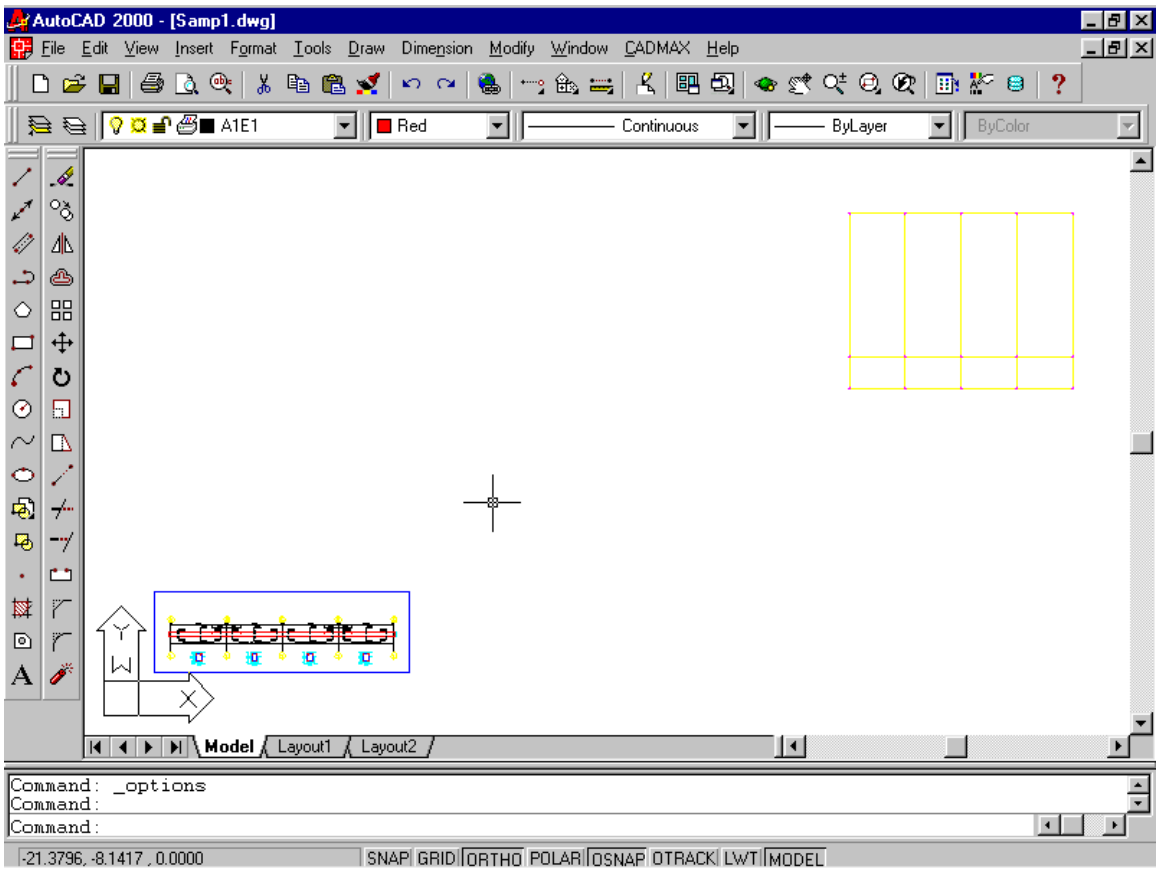


Command: **BEAMD** <enter>
Enter a point to start: <pick a point to start>
Now you will be prompted to pick a point to start and then the detailing of selected beams will be drawn automatically as shown below in the separate layer "A1E1" as indicated in the previous "BEAMDSEL" command.

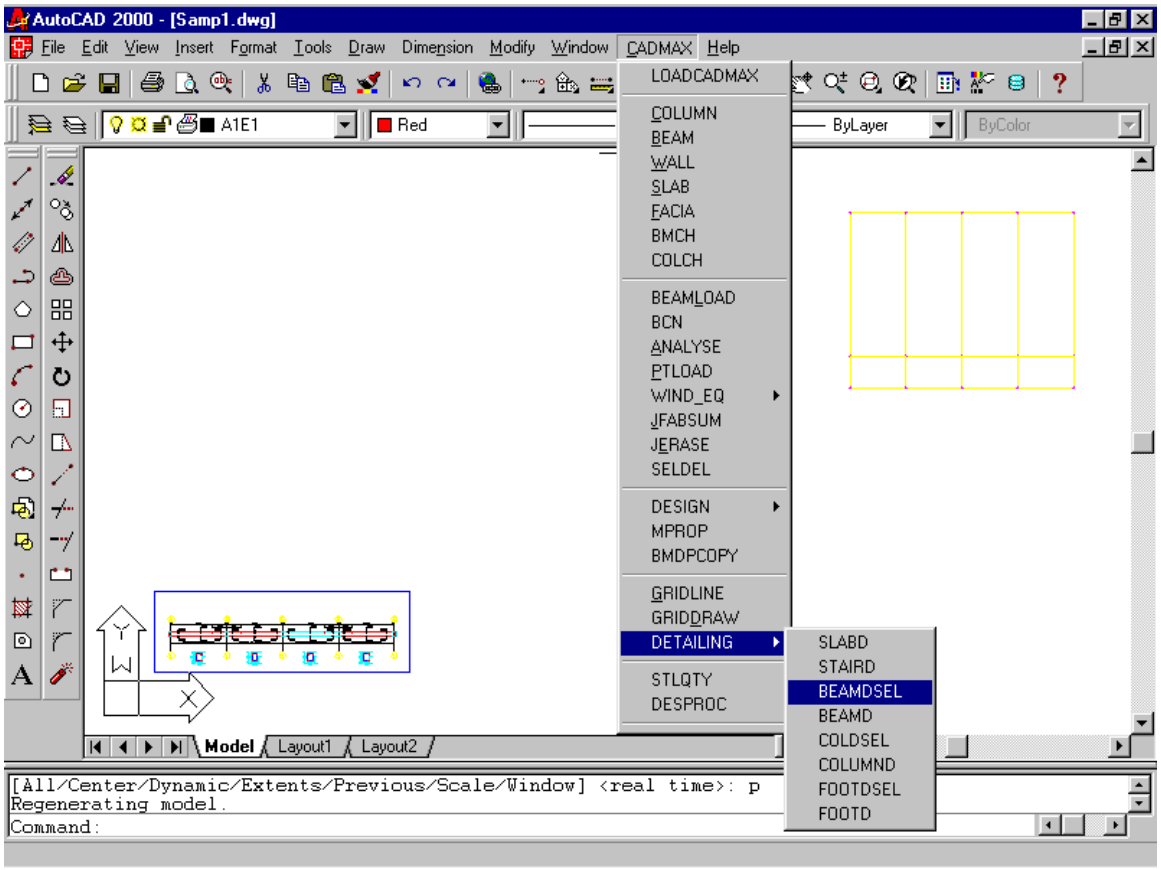


Similarly for other beams also detailed using “BEAMDSEL” and “BEAMD” command as detailed below:
Command:**ZOOM** <enter>
Select Options <All> **A** <enter>

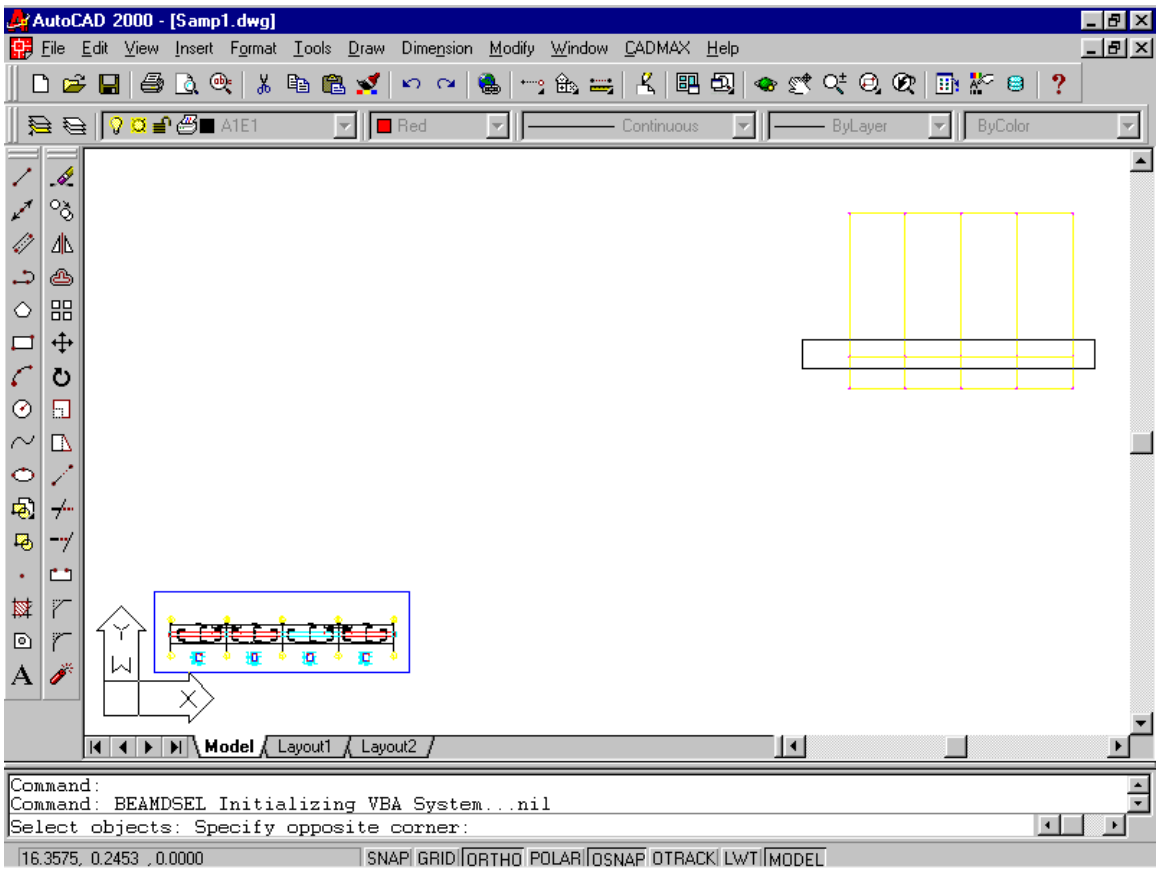
Now you will see the whole picture as shown below



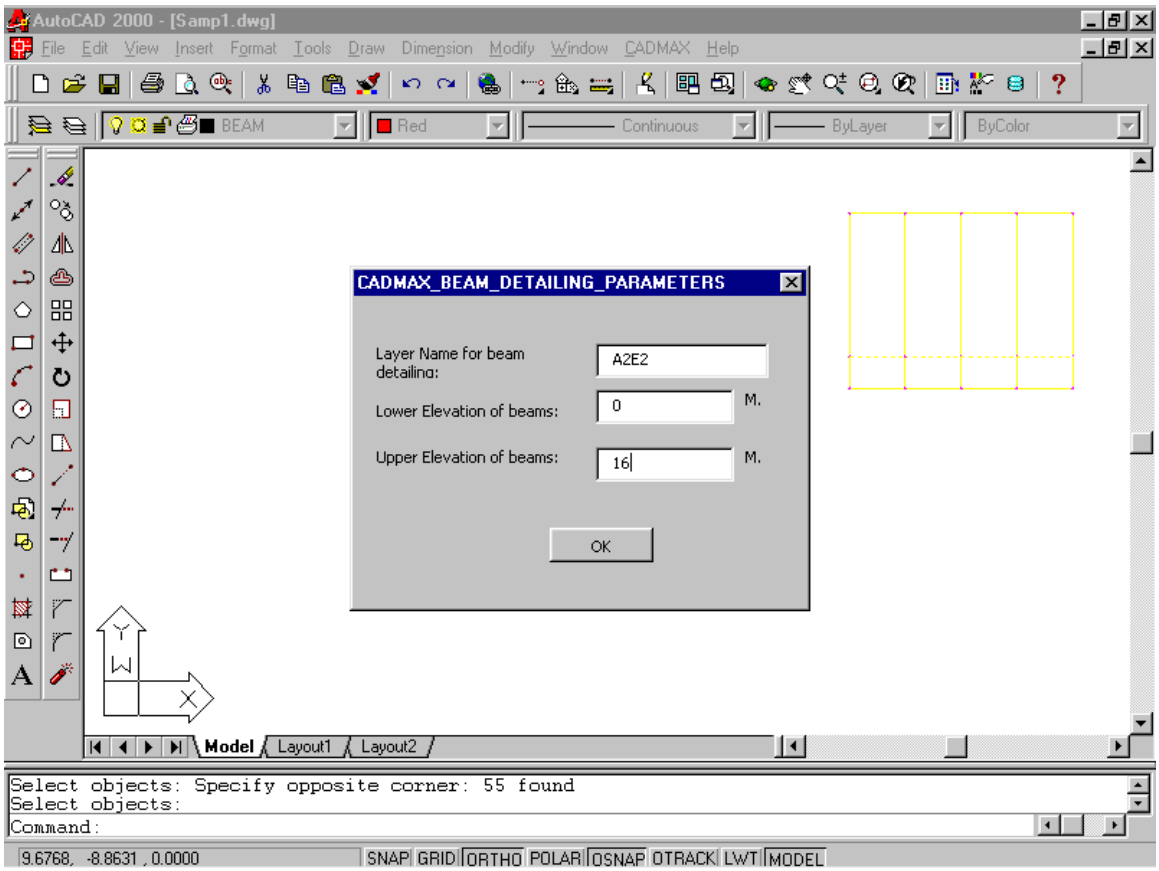
Now once again give “BEAMDSEL” command as shown below:



Now make the layer A1E1 off using layer command or layer popup menu.
Command:**BEAMDSEL** <enter>
Select objects: <select objects containing beams along single line as shown below>

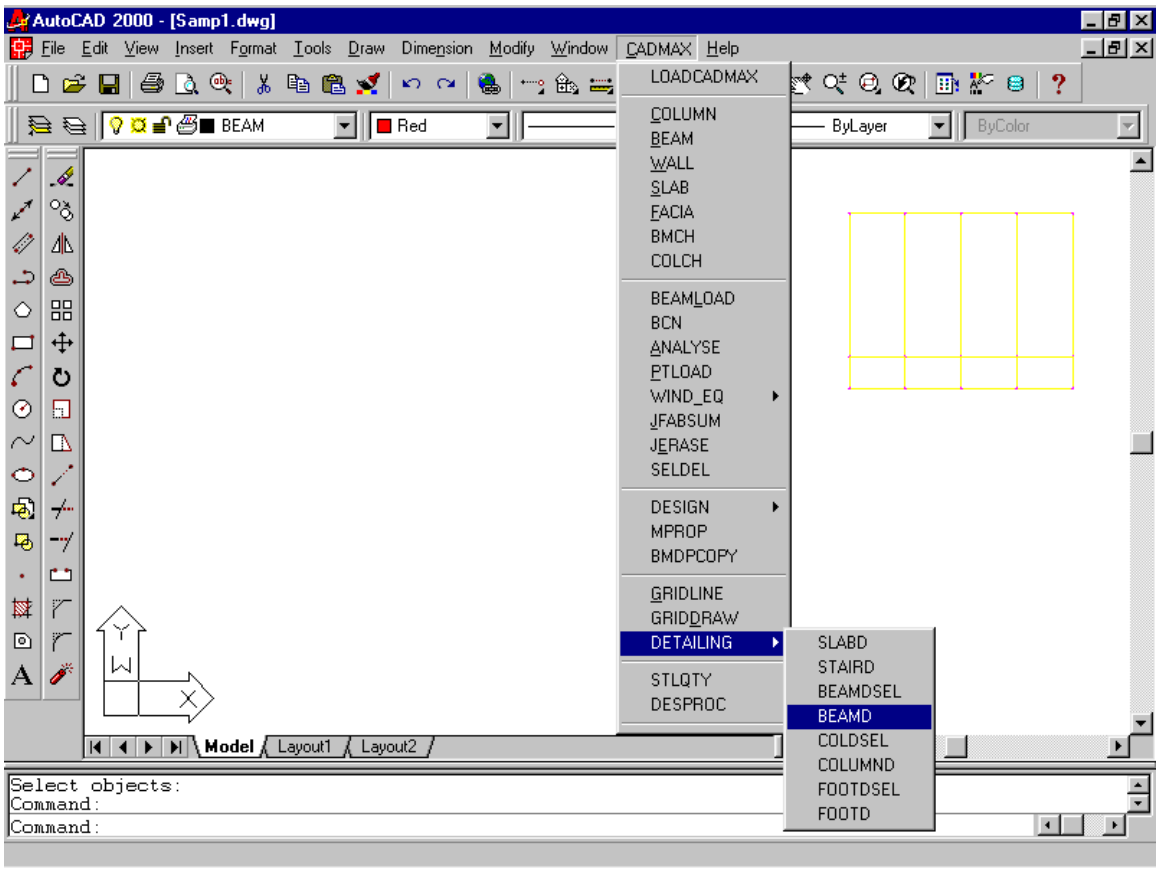


For the next dialog box fill up appropriate details as shown below:

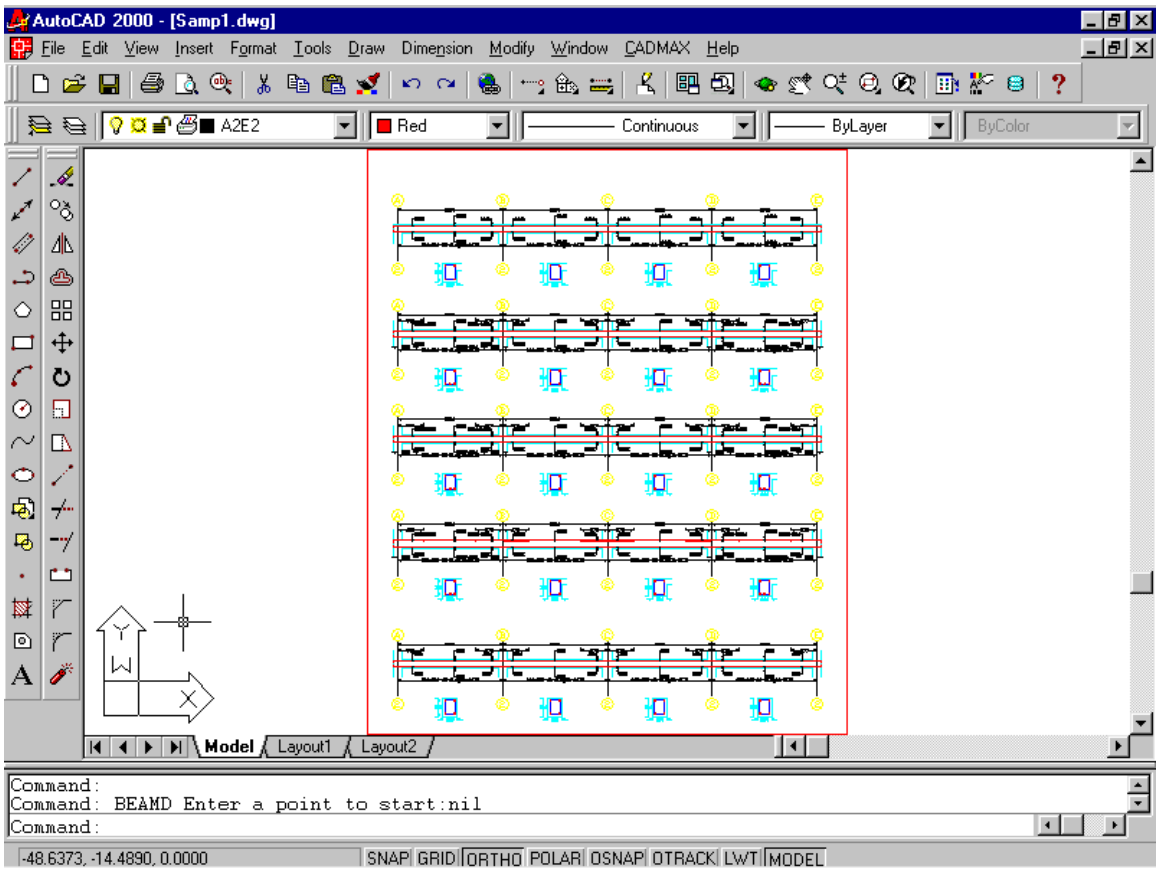


Now use the BEAMD command to draw detailing of the selected objects in layer A2E2 as shown below:

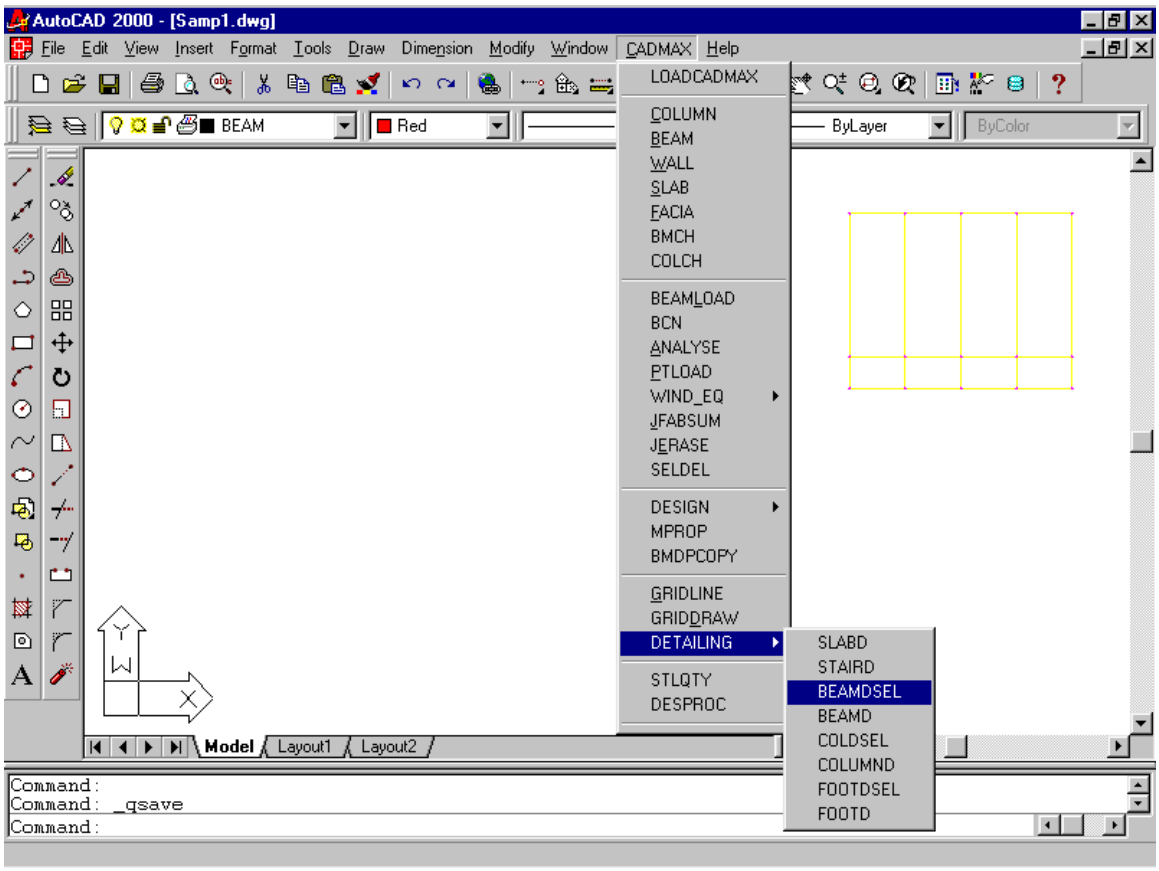
Command: **BEAMD** <enter>



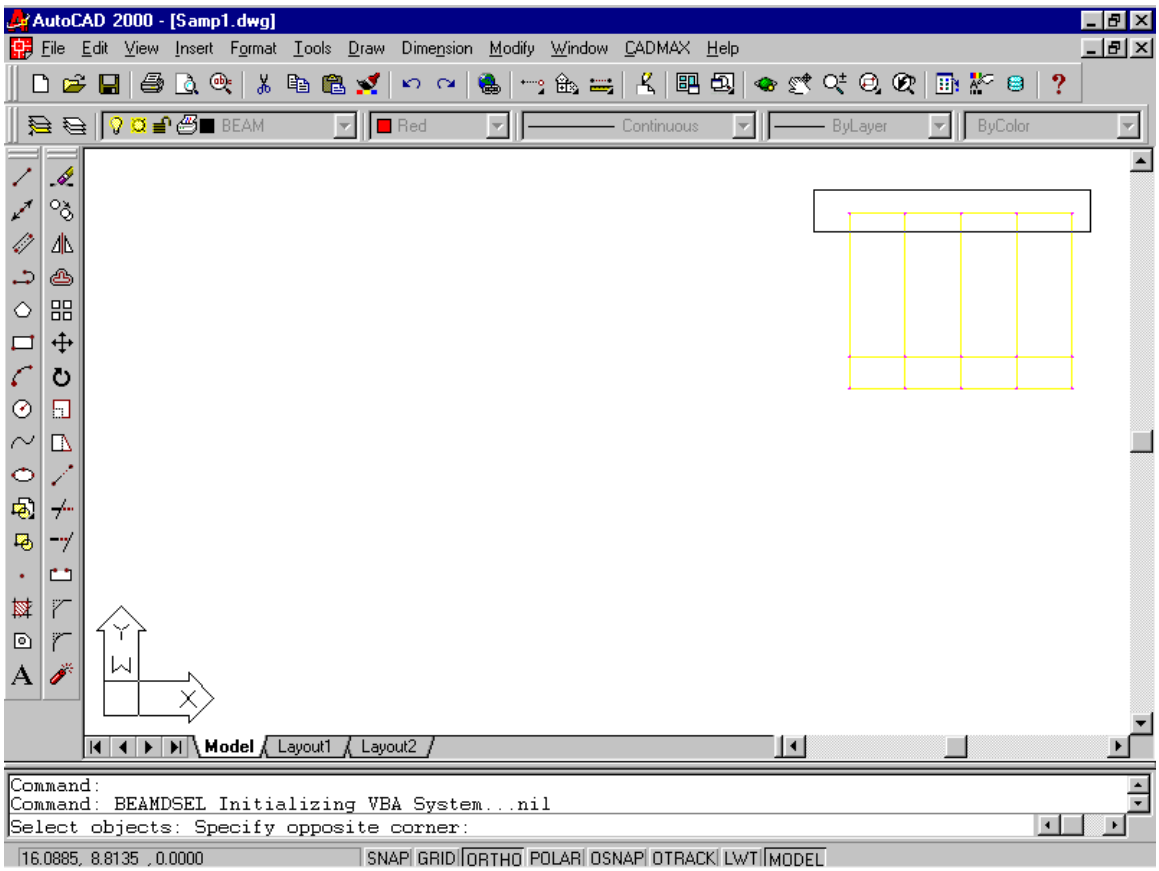
Now you will be asked to pick a point on the screen as shown below:



In the above drawing you are seeing the beam details of all the beams in the frame A2E2 and now use the Zoom All Command to detail further beams and make the layer A2E2 off and set the current layer as BEAM.

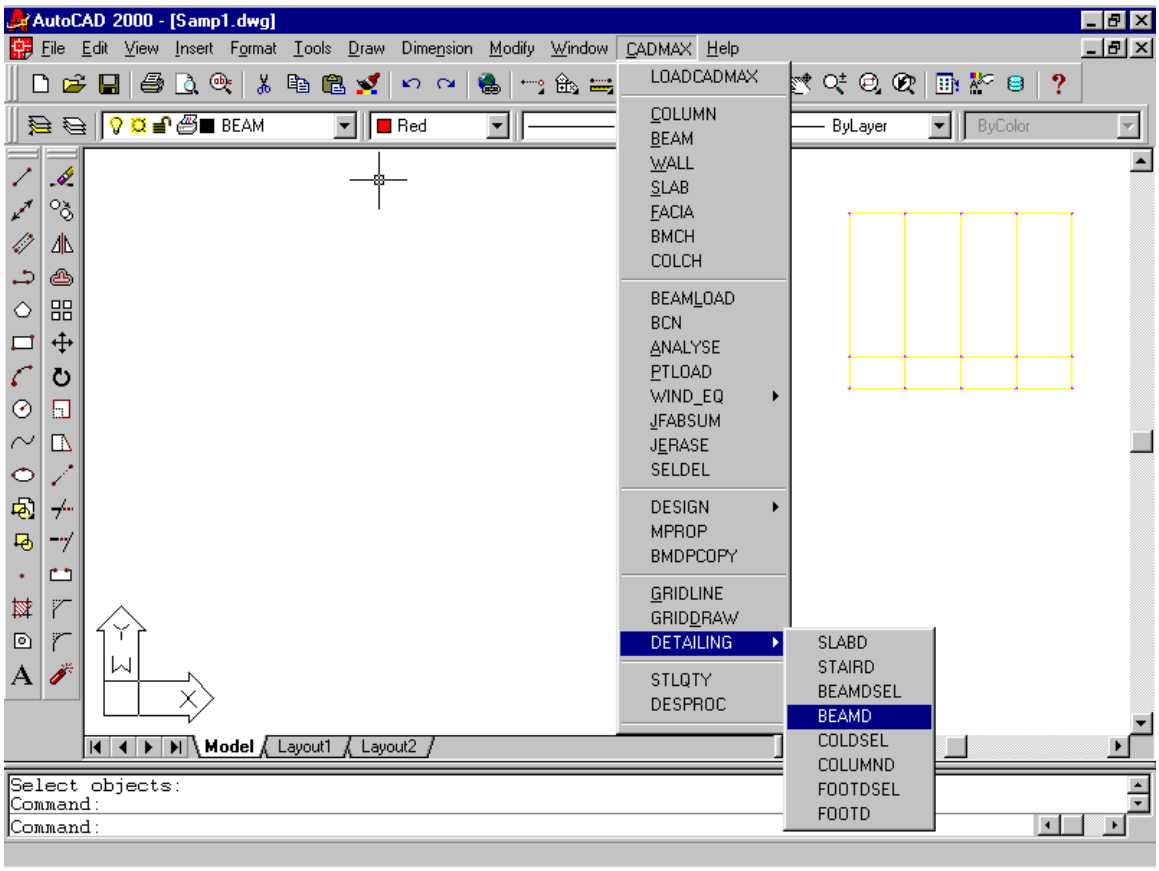


Command: **BEAMDSEL** <enter>
Select objects: <select all the beams along A3E3 as shown below:

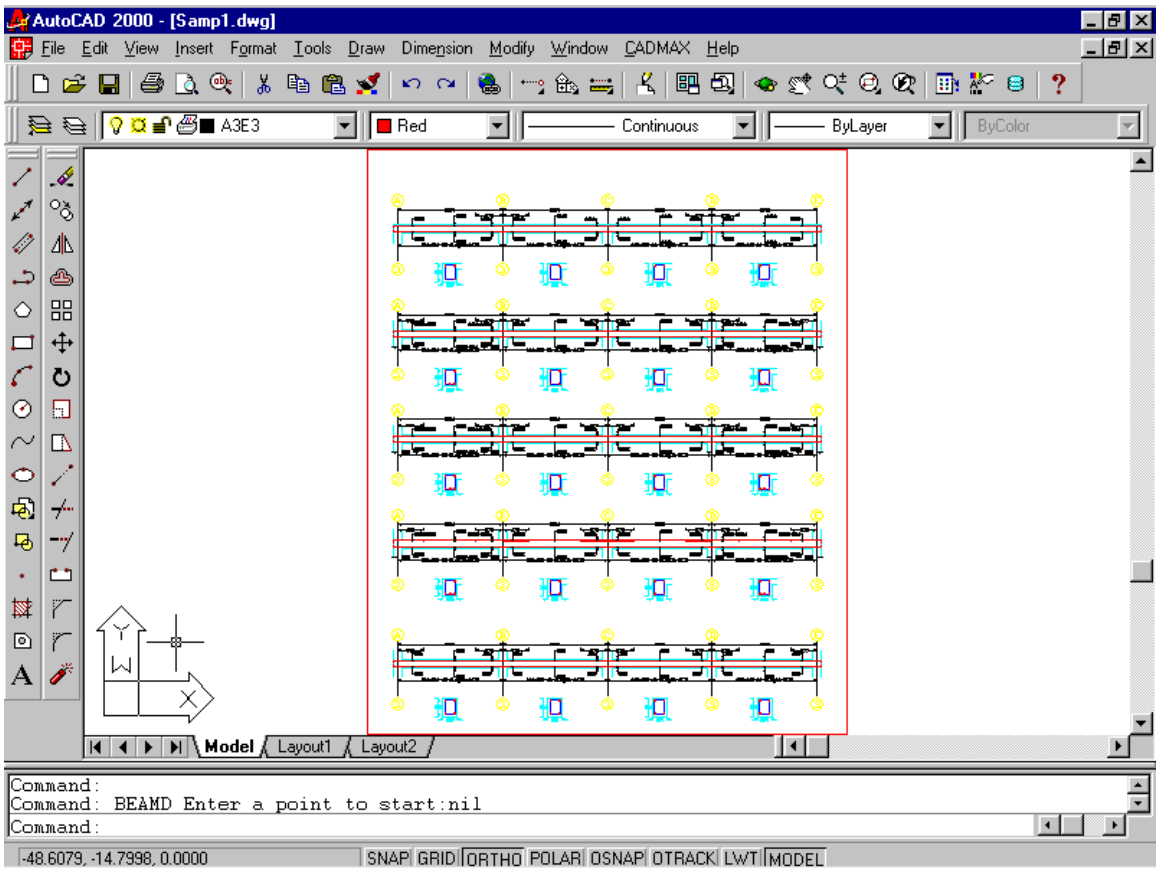


Now after selection you can use BEAMD command to detail the beams in the layer A3E3 as shown below:

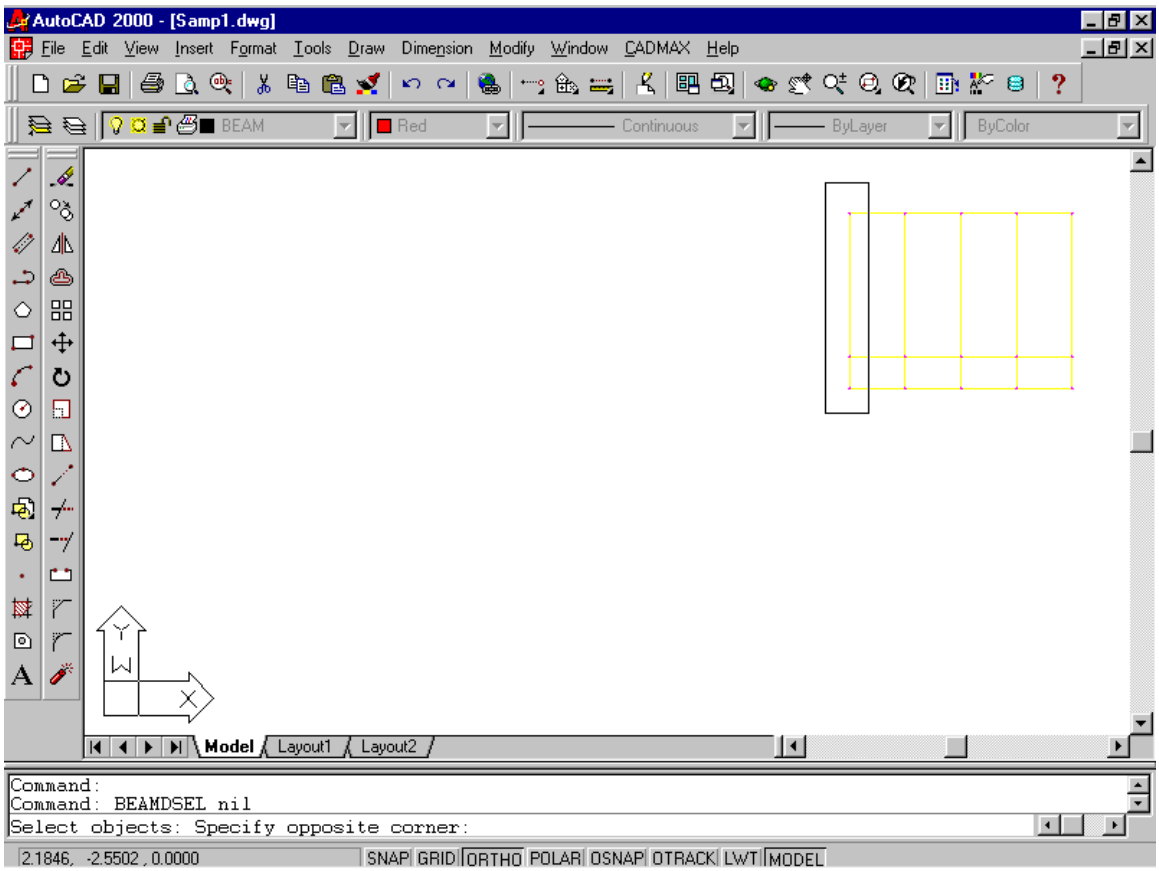
Command: **BEAMD** <enter>
Pick the start point to draw detailing: <pick a point to start detailing>



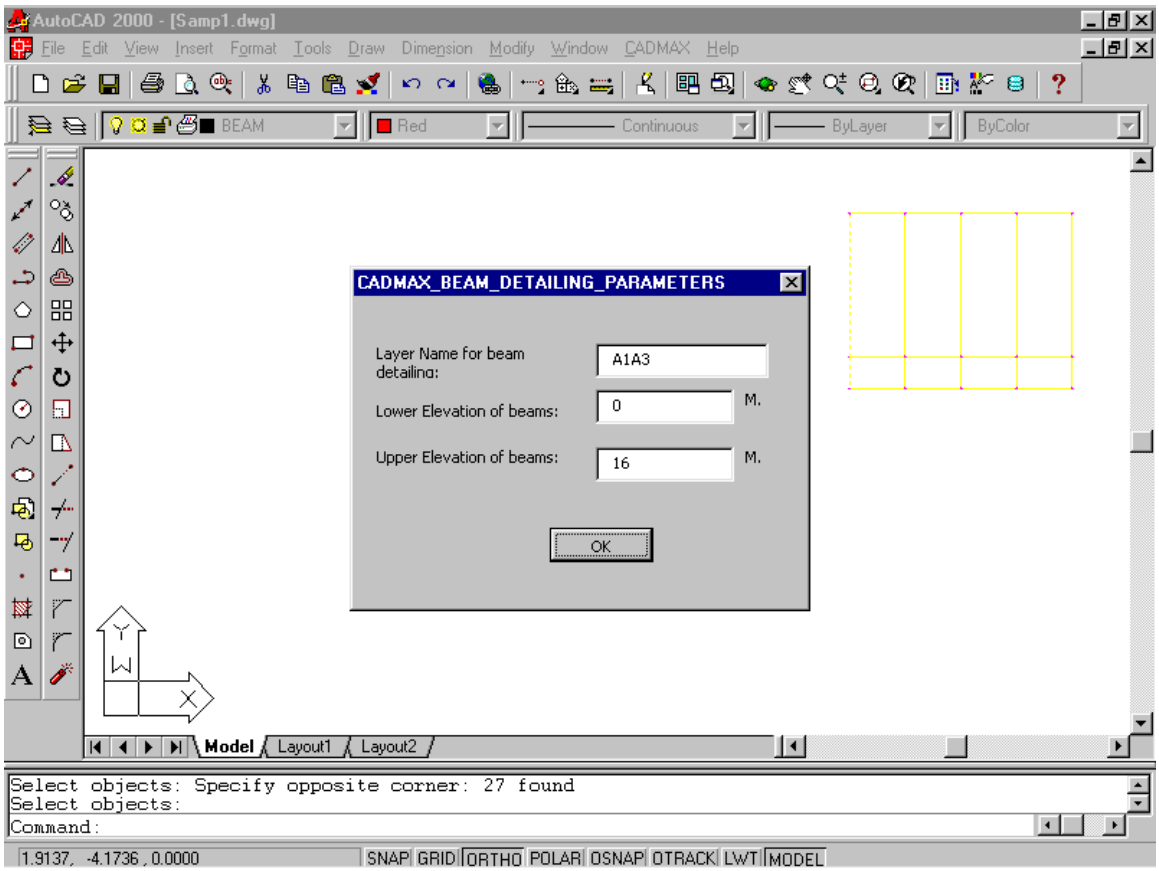
Now after picking the point you will the following beam details for Frame A3E3.



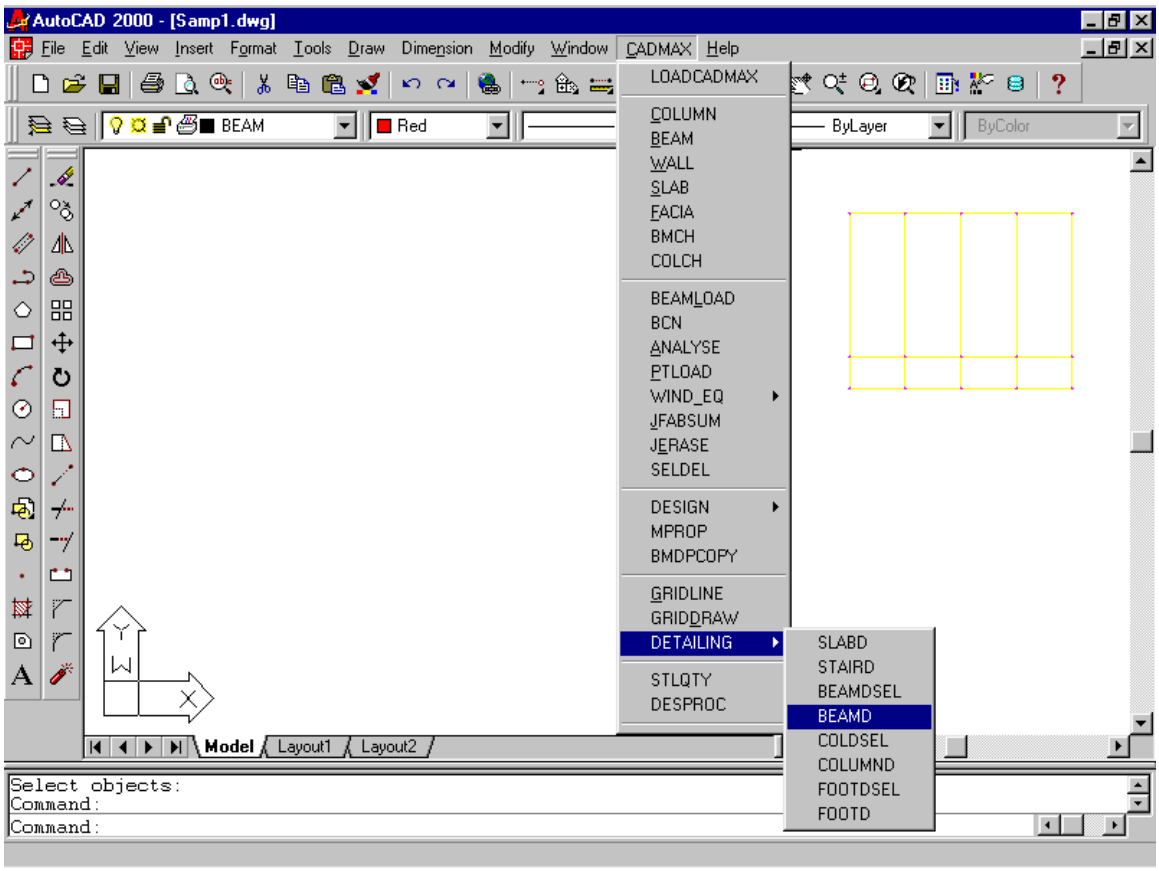
Now you can start detailing of beams along Y-direction also similarly as shown below:



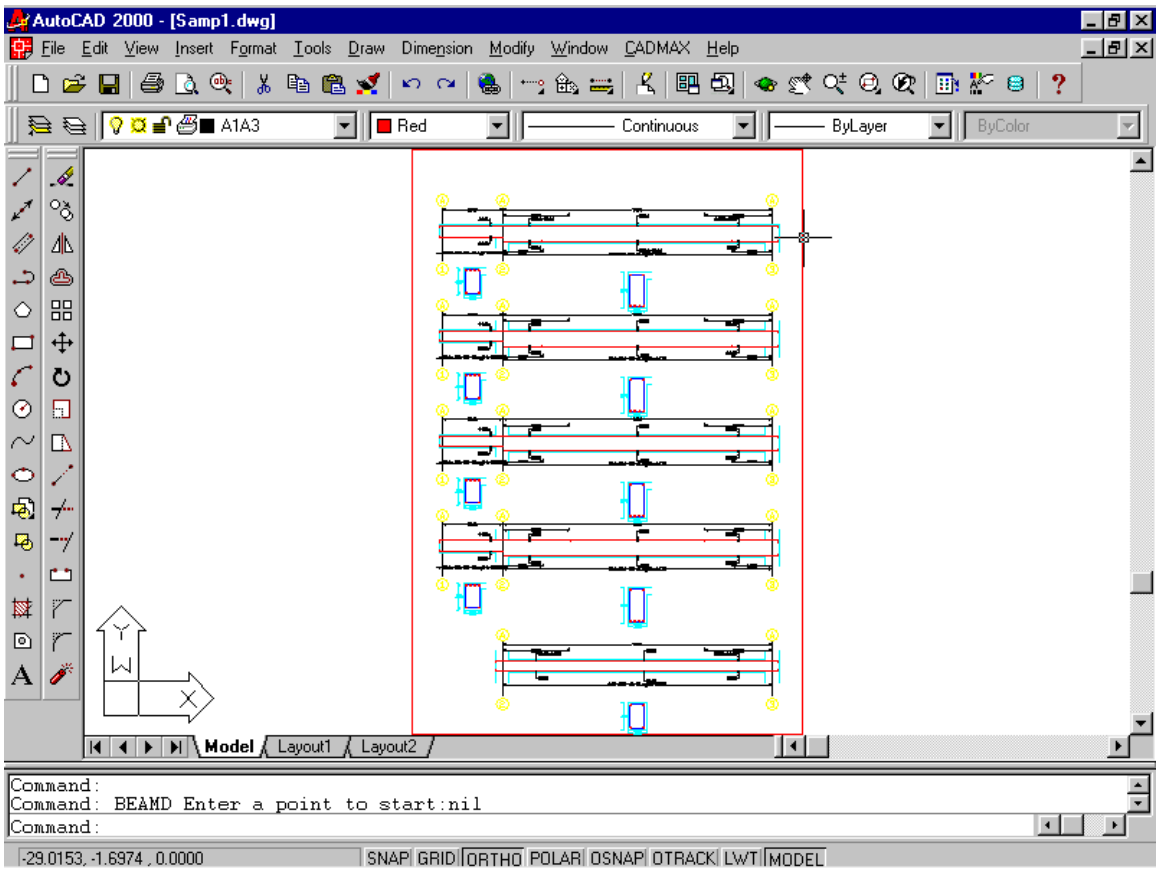
Now you can fill up the following dialog box as shown below appropriately.



Now you can use BEAMD command to draw the beam details of selected Frame A1A3 as shown below:

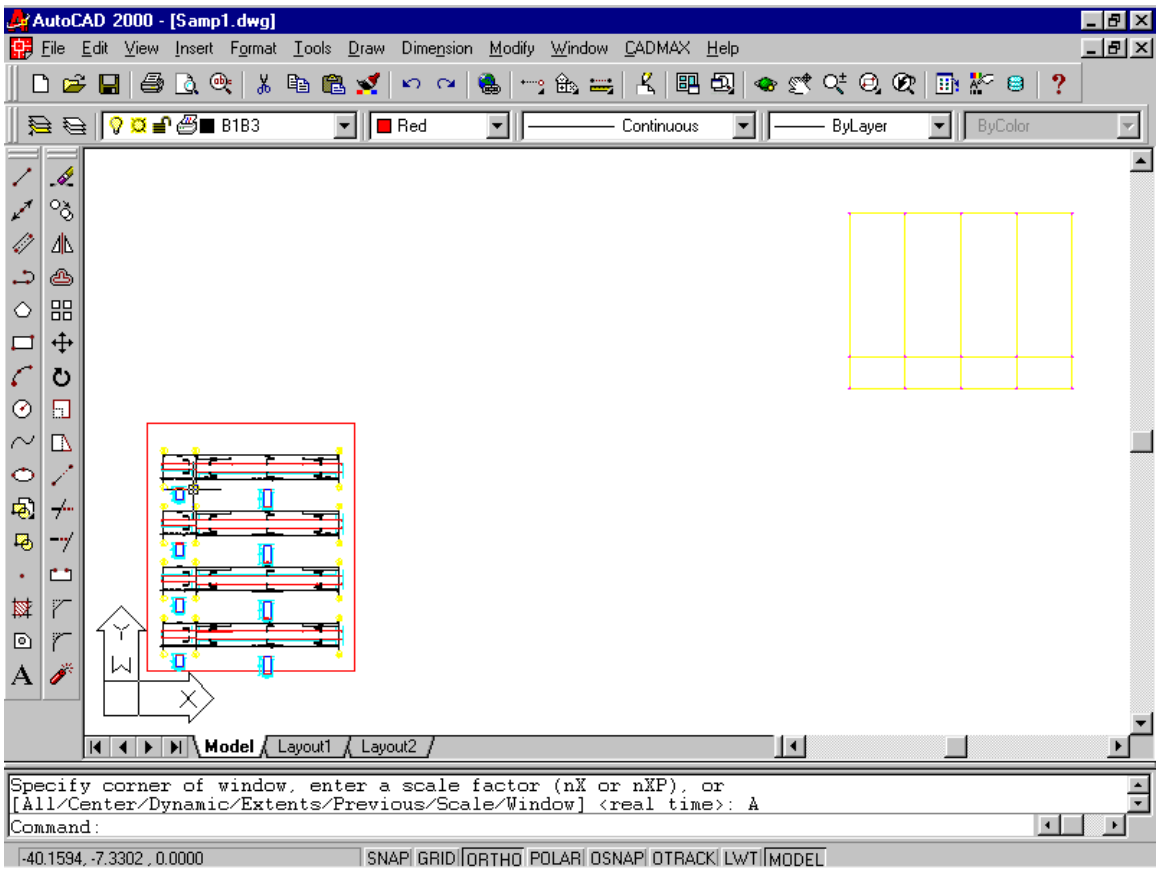


Now you can get the following drawing in the layer A1A3 as shown below:



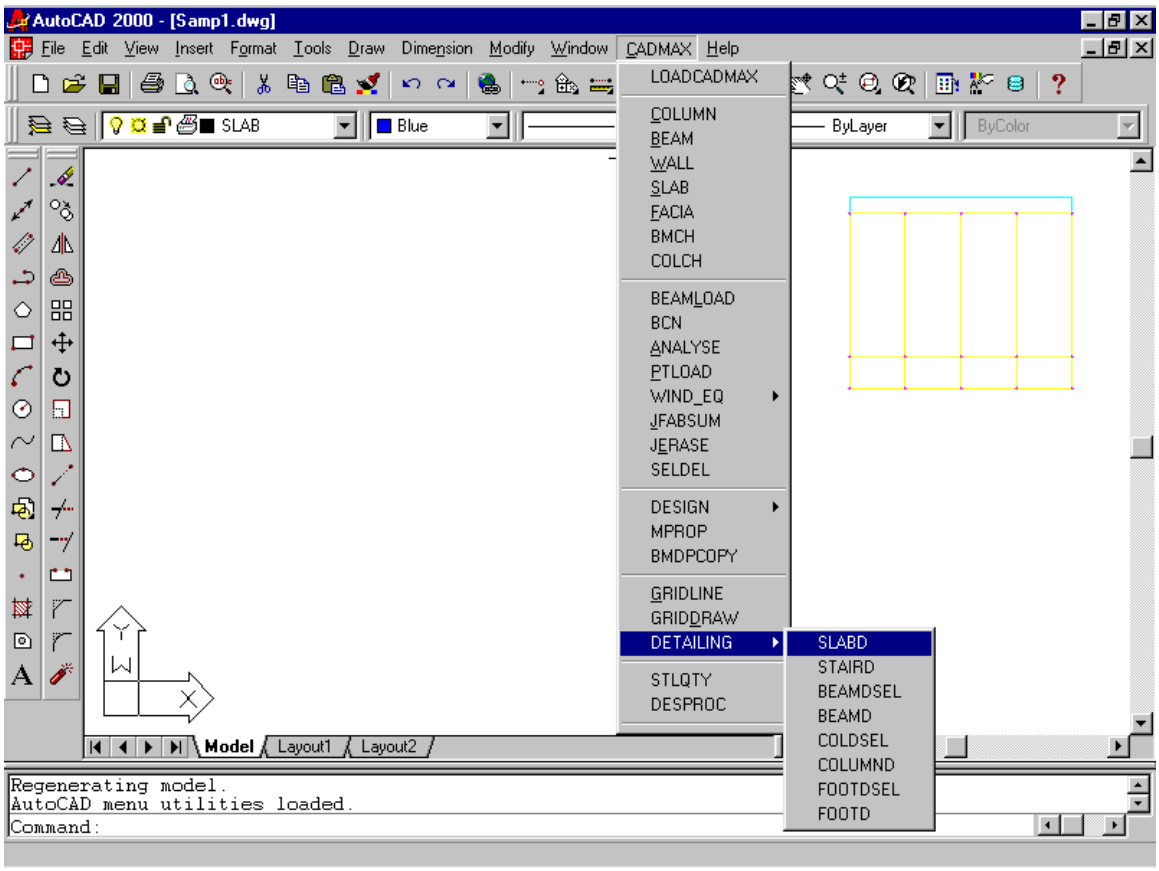
Now use Zoom All Command to go for other beam detailing and make the layer A1A3 off and set the current layer as BEAM.

Then use the BEAMDSEL command and select the beams along B1B3 and then fill up the dialog box as done in previous commands and you will get the following Detailing using the BEAMD command .

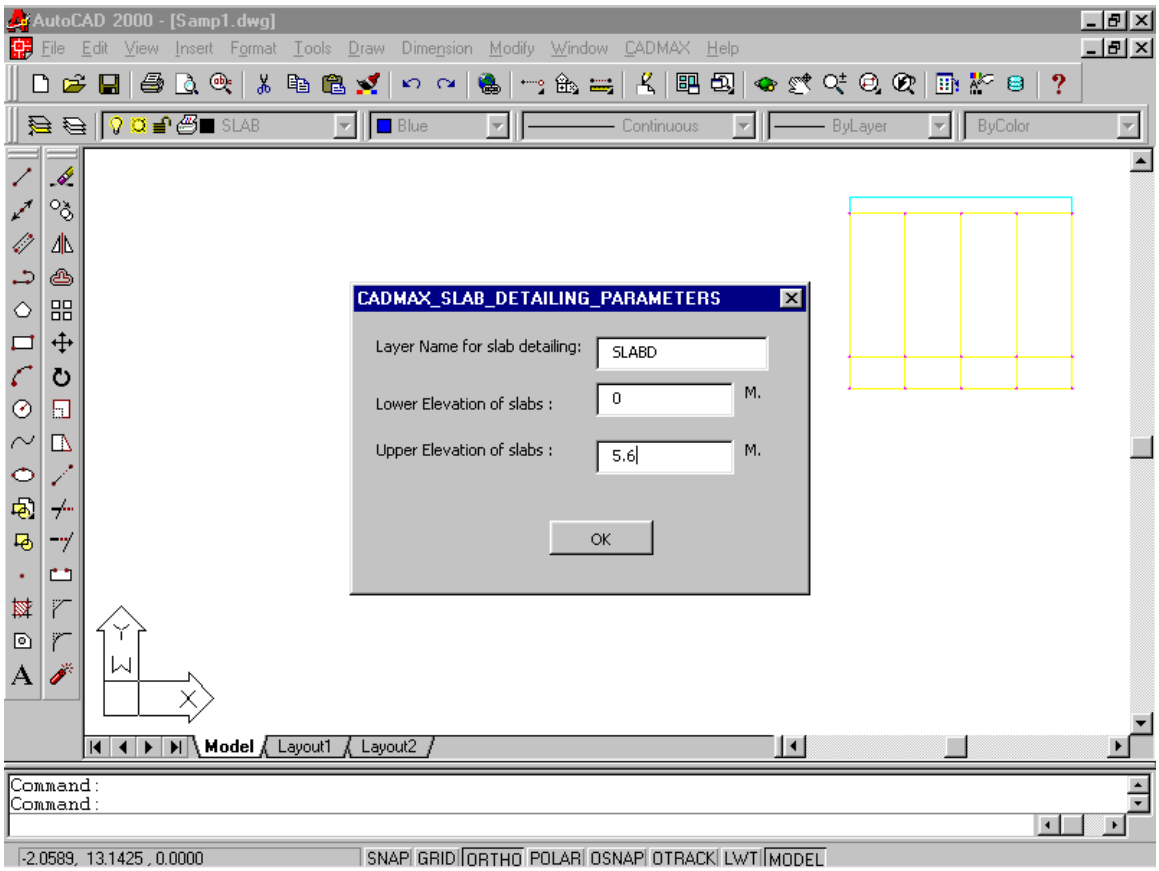


Similarly you can detail other beams also.

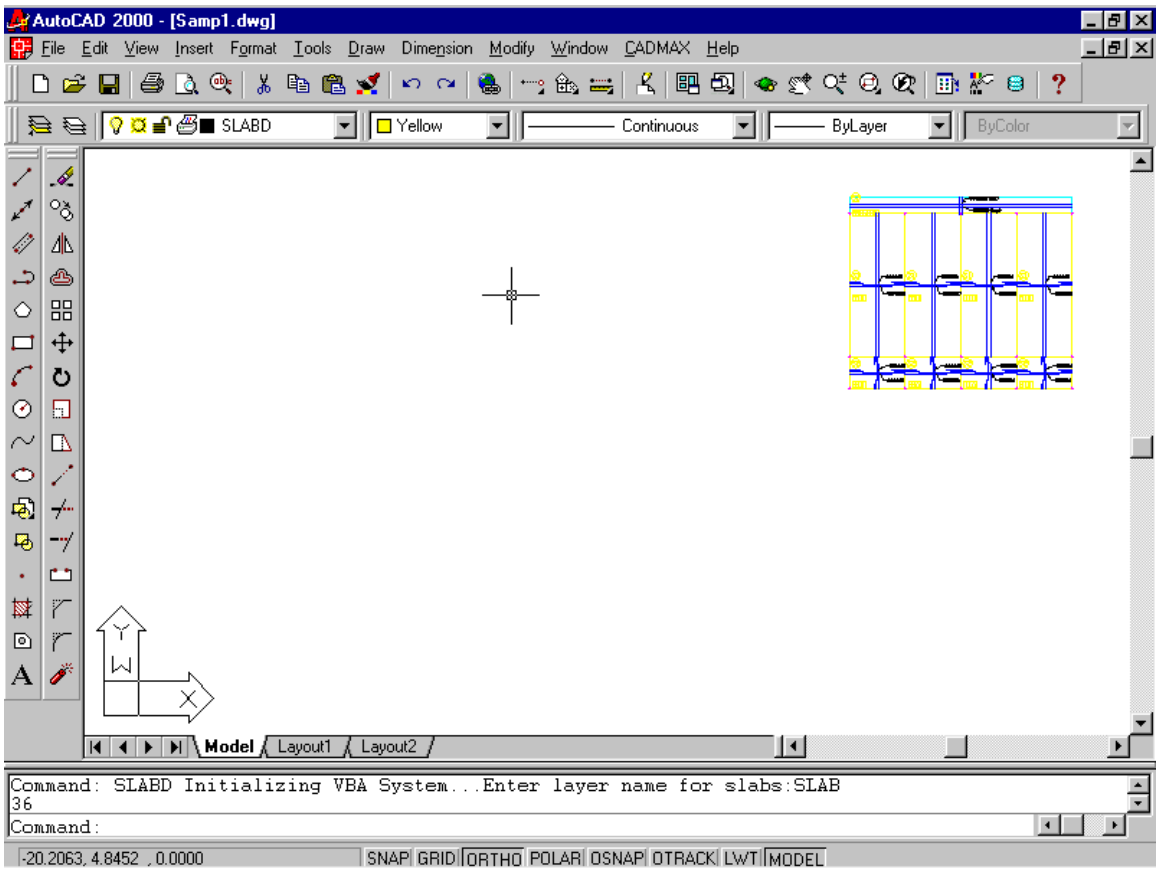
29.SLABD Command: This command is used for detailing of slabs as shown below:
(You should make the layer SLAB on before giving this command)
Command: **SLABD** <enter>
You can select the above command using pop up menu as shown below:



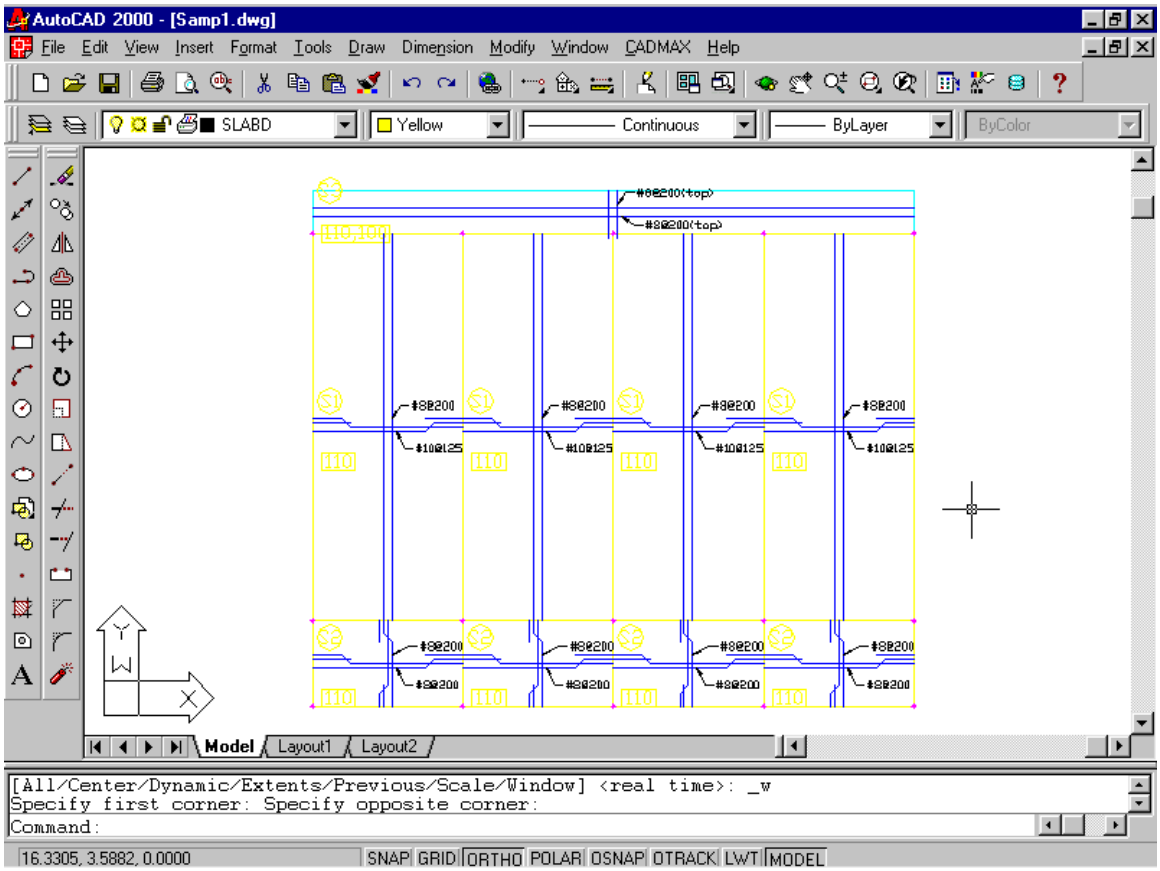
After issuing this command you will be asked to fill up the following dialog box shown below:



Now you asked for layer name in which the slabs are drawn using SLAB command already. In this example we have already drawn slabs in layer SLAB.
Enter layer of slabs: **SLAB** <enter>
Now you will see the slab details in the elevation between lower and upper elevation specified in the above dialog box as shown below:

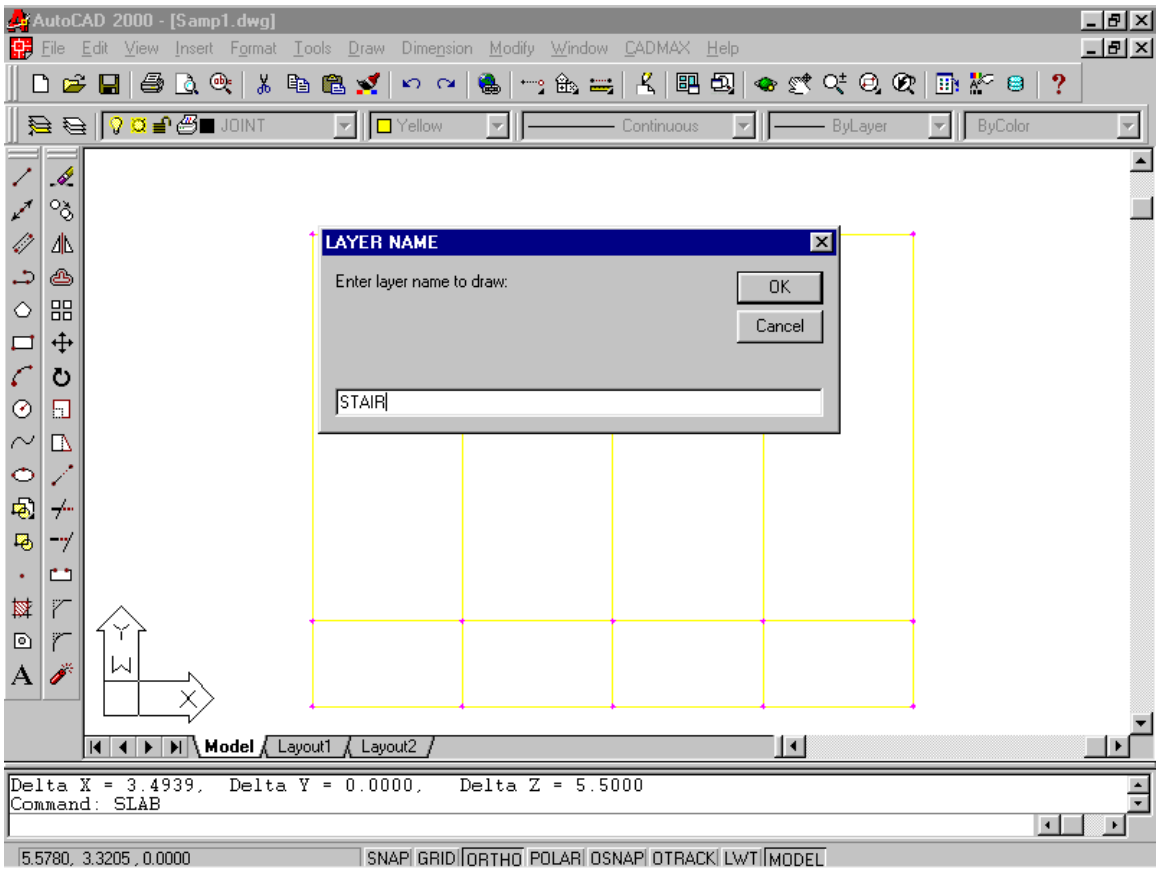


Now you can Zoom the drawing to see the slab details as shown below:

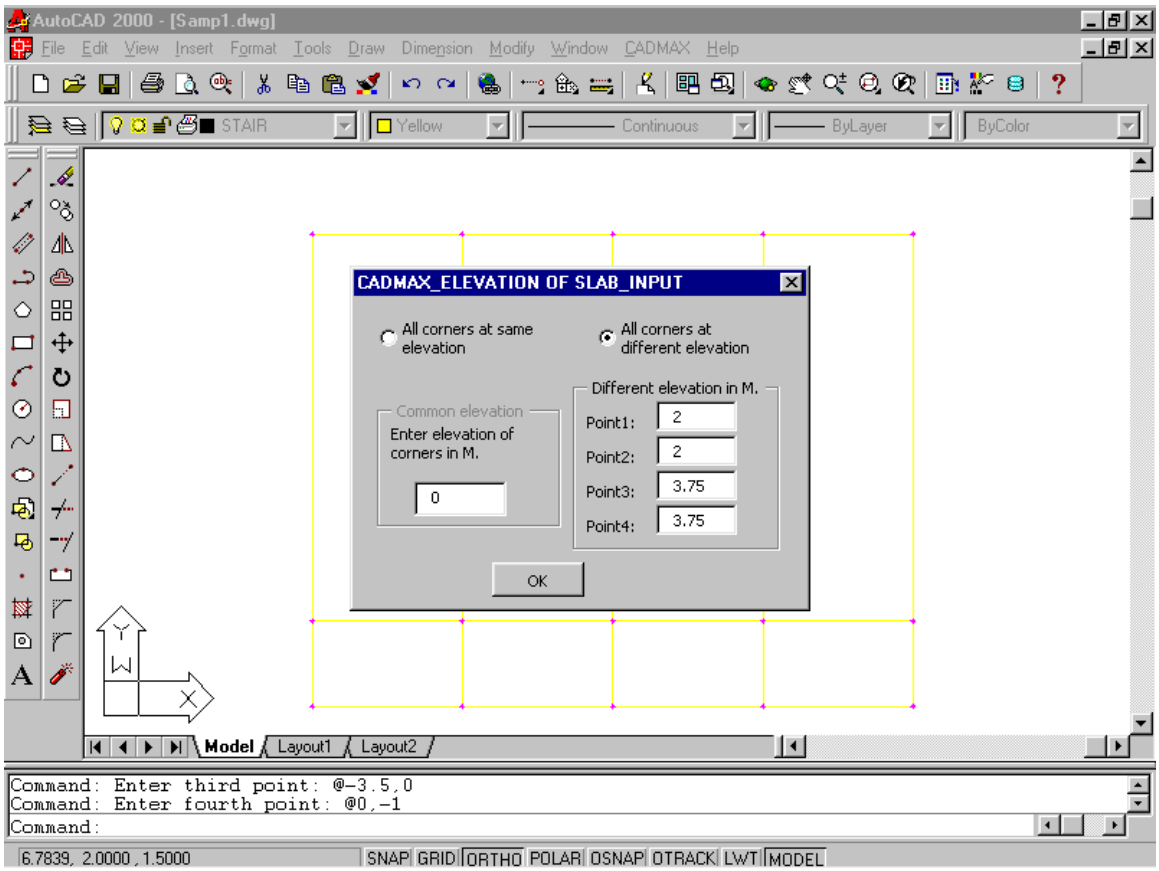


30.STAIRD Command: This command is used for staircase detailing.

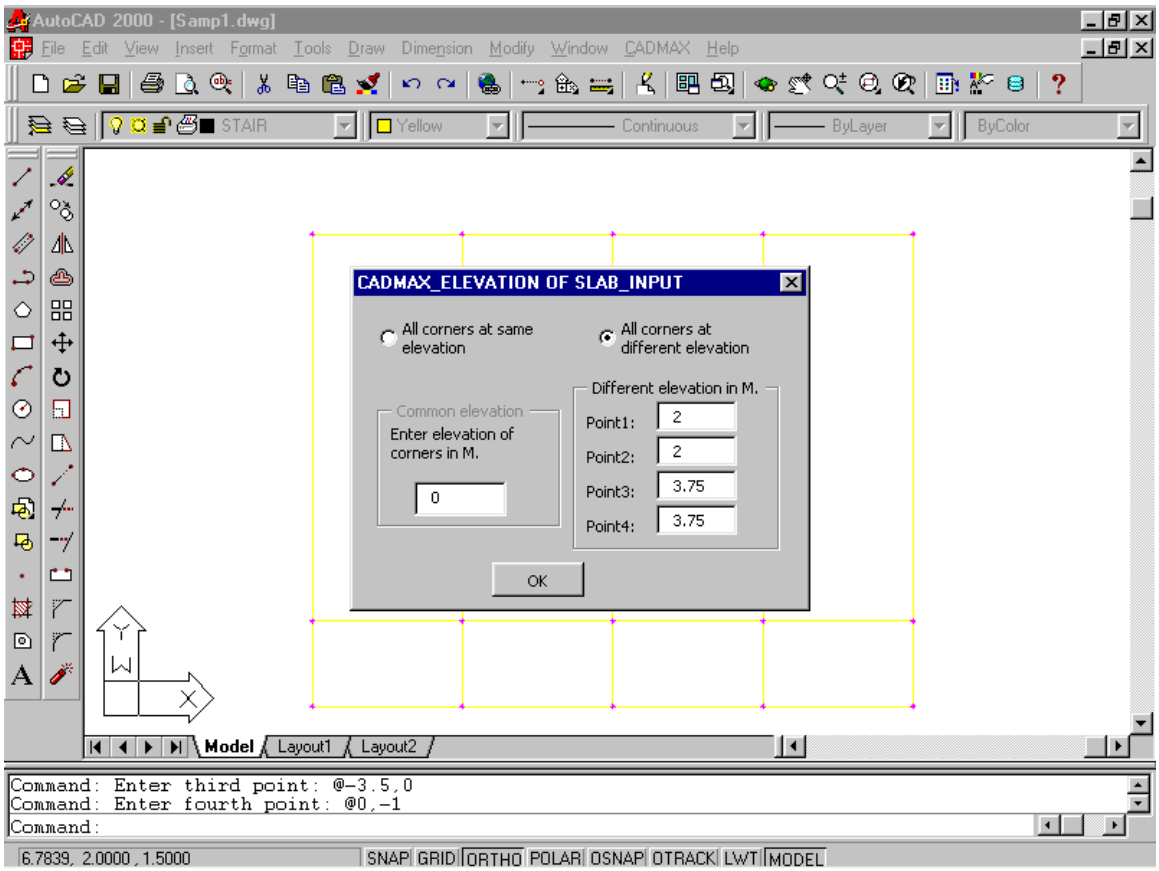
To run this command you should draw the staircase slab as sloped slab as given below.
In this drawing give slab command for stair as shown below: Before giving this command make the layers SLABD and SLAB off.
Command: **SLAB** <enter>

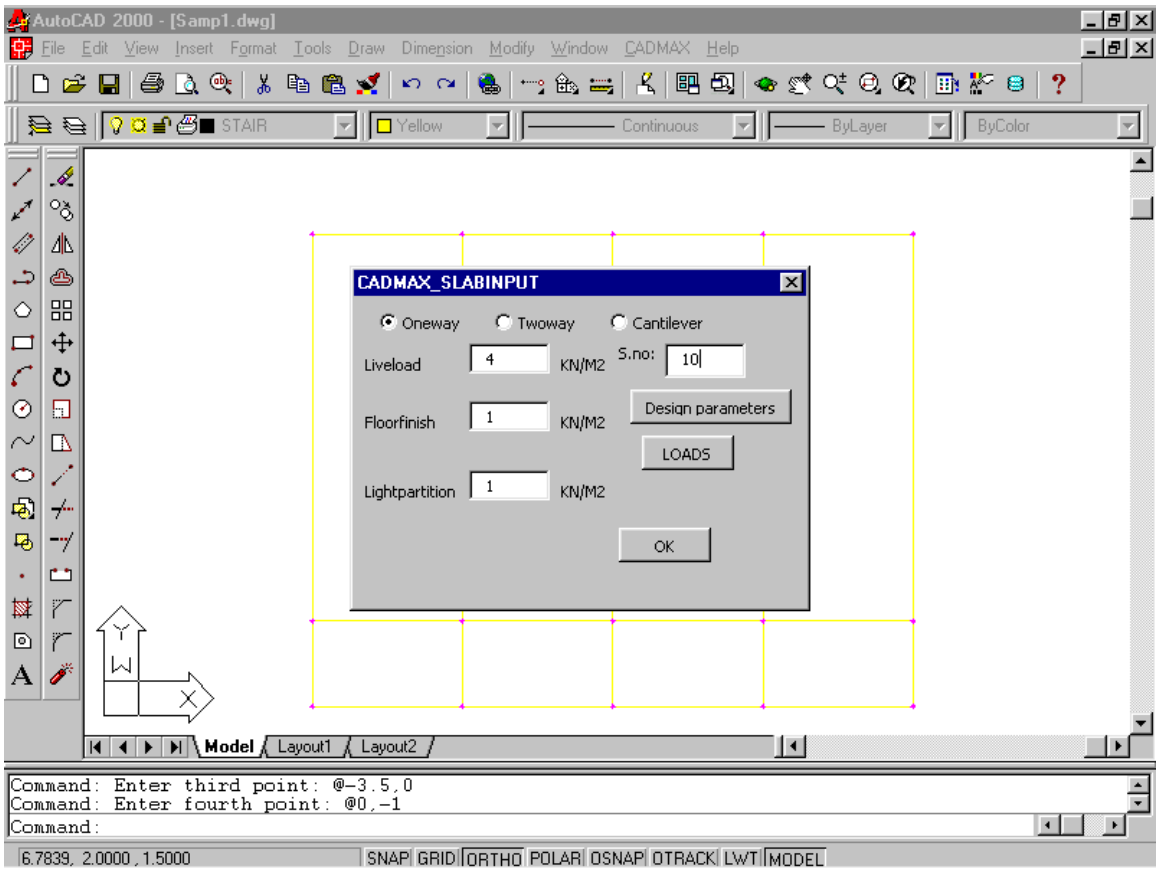


After filling the above dialog box you will be prompted for picking four corners of points and give them as previous slab commands. Then in the next dialog box select Corners at different elevations and fill them up as shown below:

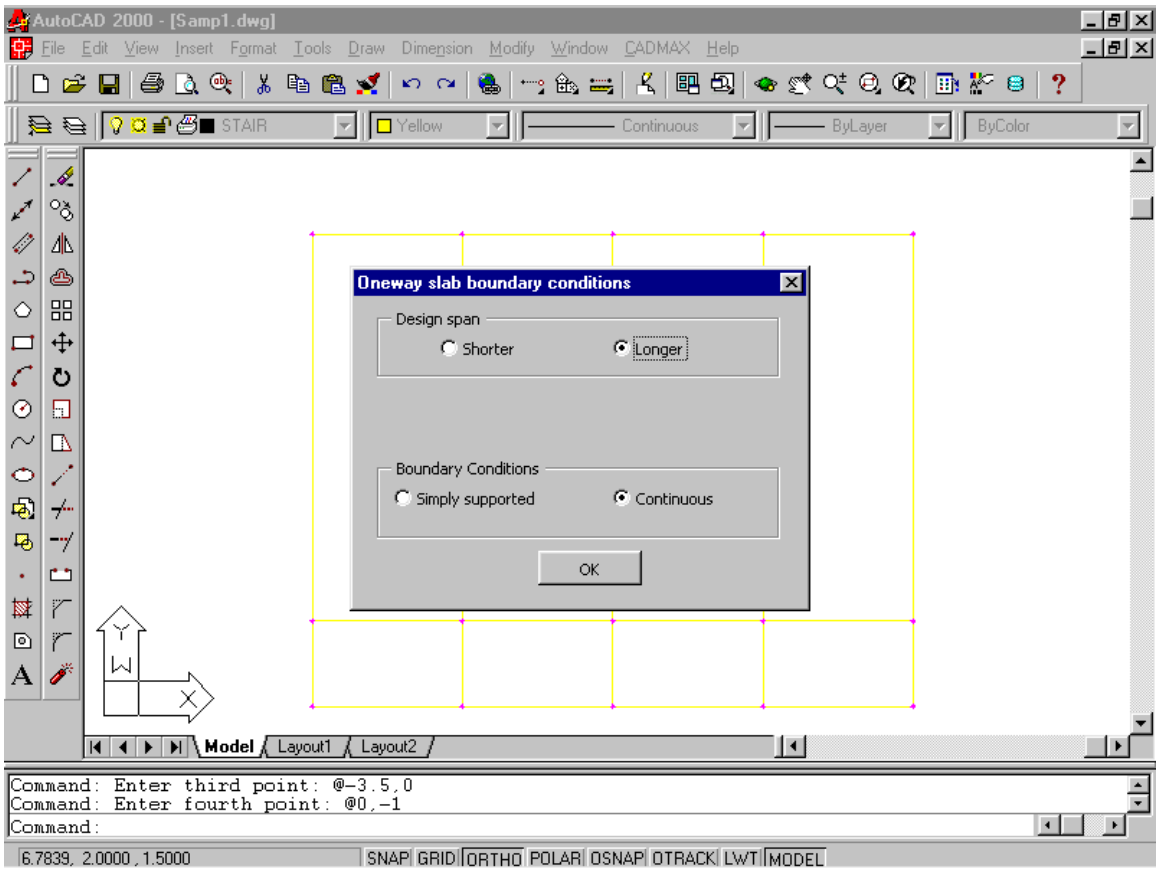


Now you will see a sloped slab is drawn in the figure as shown below:

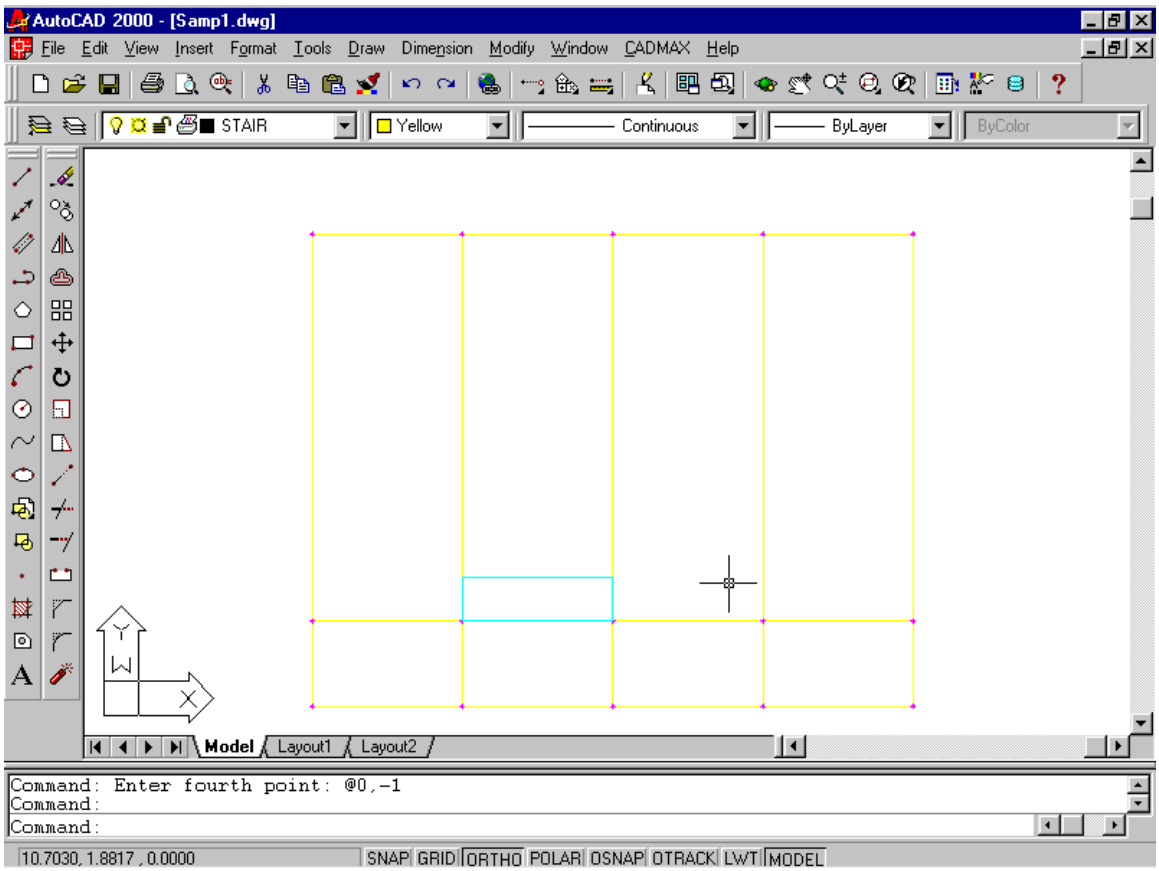

AutoCAD 2000 - [Samp1.dwg]
File Edit View Insert Format Tools Draw Dimension Modify Window CADMAX Help
STAIR
Yellow
Continuous
ByLayer
ByColor
CADMAX_ELEVATION OF SLAB_INPUT
All corners at same elevation
All corners at different elevation
Common elevation
Enter elevation of corners in M.
0
Different elevation in M.
Point1: 2
Point2: 2
Point3: 3.75
Point4: 3.75
OK
Model
Layout1
Layout2
Command: Enter third point: @-3.5,0
Command: Enter fourth point: @0,-1
Command:
6.7839, 2.0000, 1.5000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL



Now you will see the next dialog box shown below:

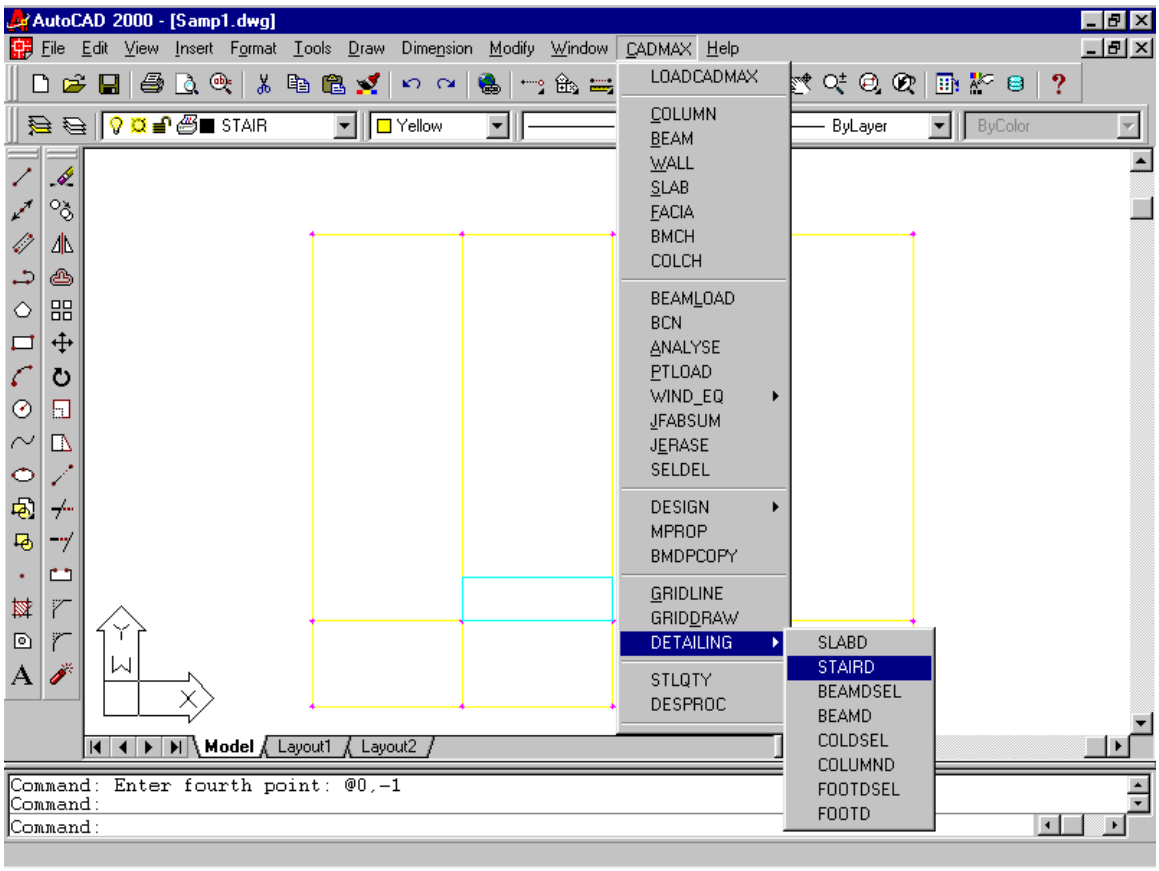


Select the longer span as design span for staircase flight slab and then press OK to complete command. Now you will see a flight slab is drawn in the drawing in the layer STAIR.

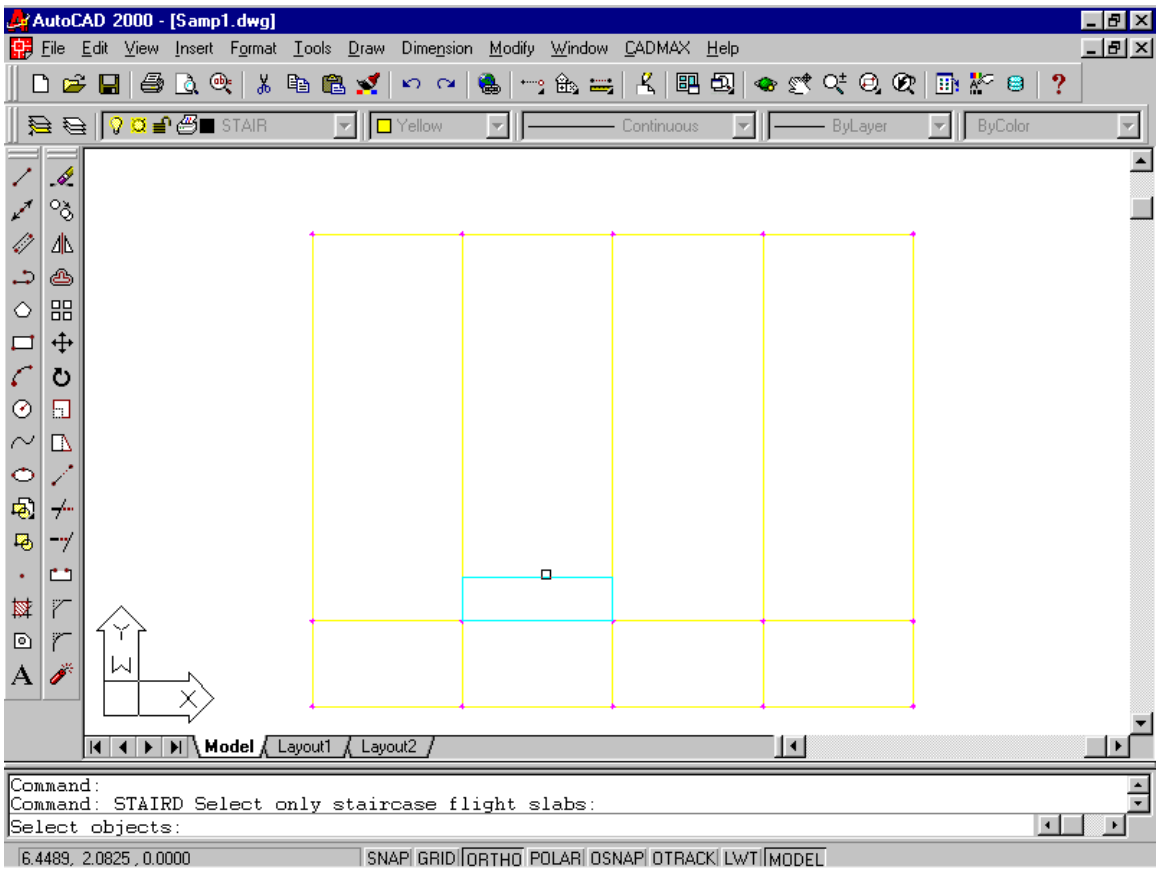


Now a Staircase flight slab is drawn as sloped slab. Now using the STAIRD command the flight slab is going to be detailed as shown below:

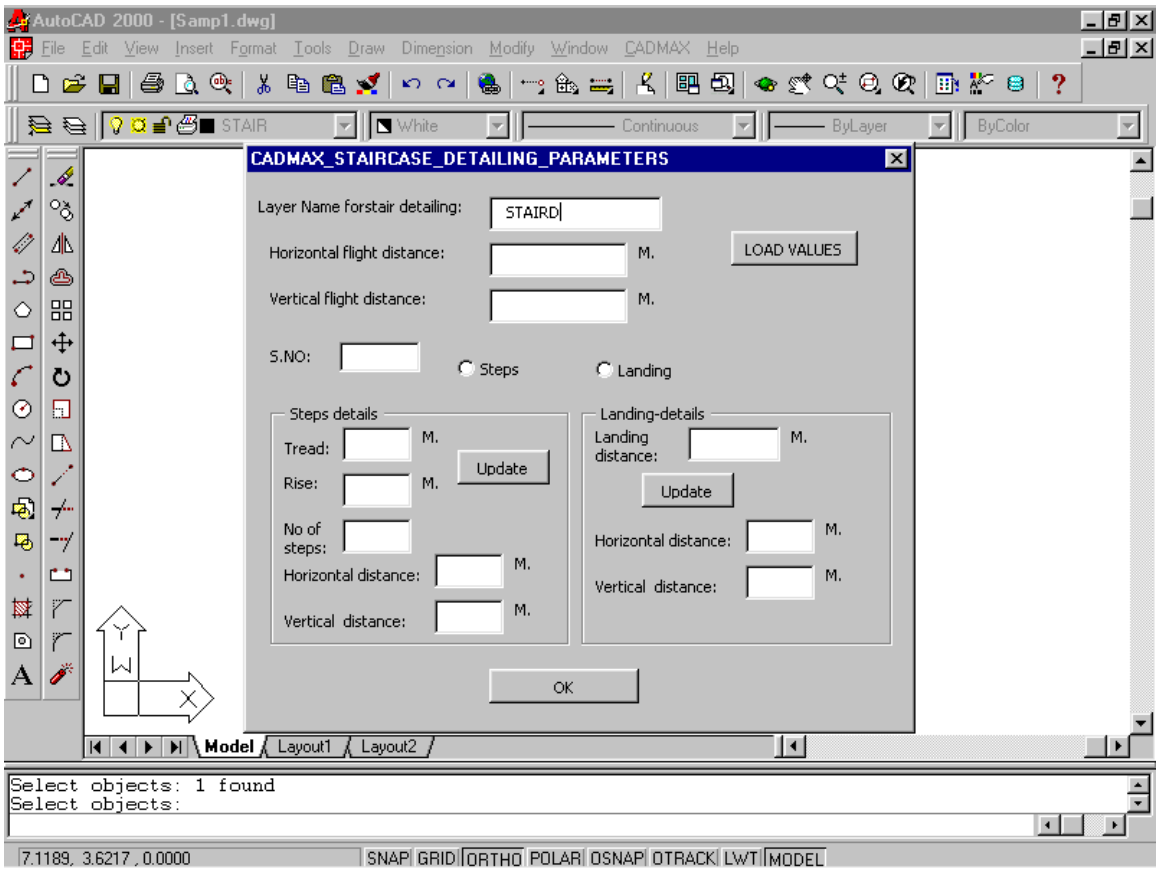
Command: **STAIRD** <enter>
The pop up menu is shown below:



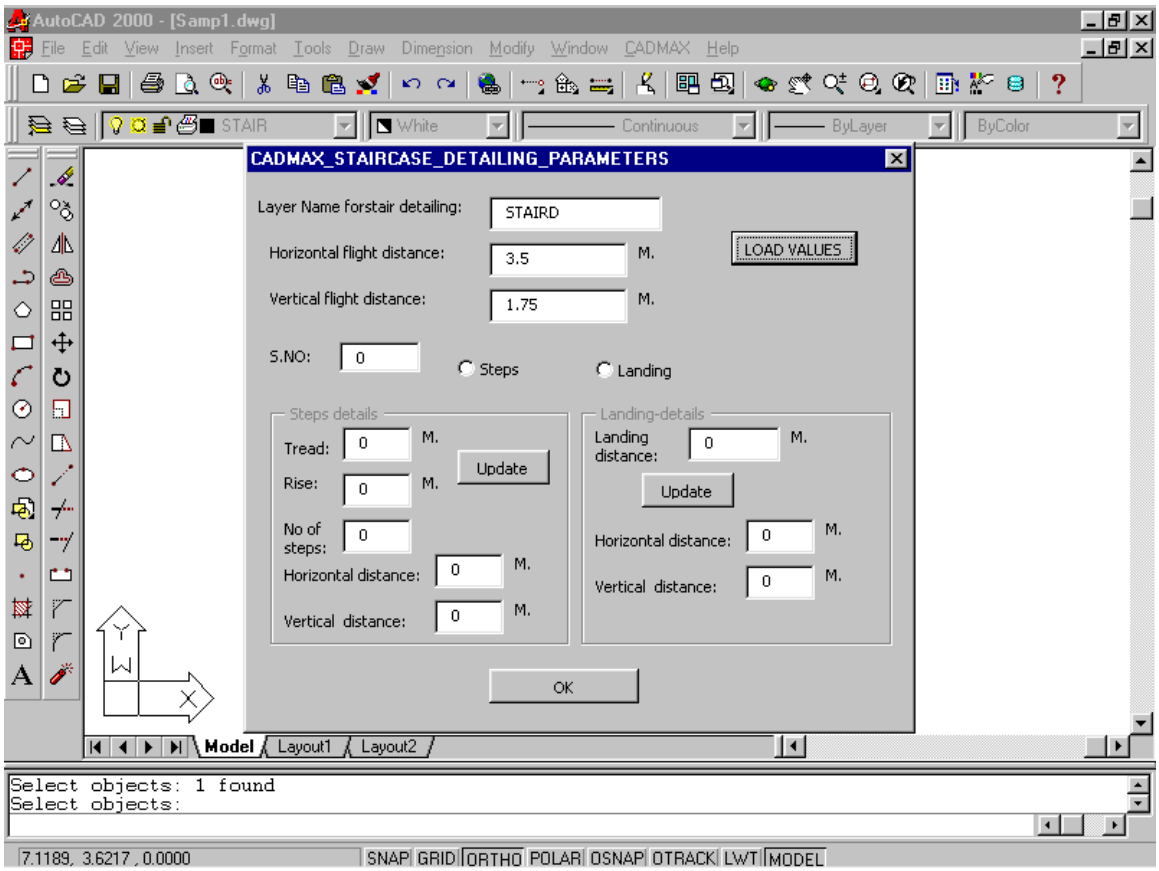
Now you will be asked to select the staircase flight slab to detail. Now select the flight slab last drawn as shown below:



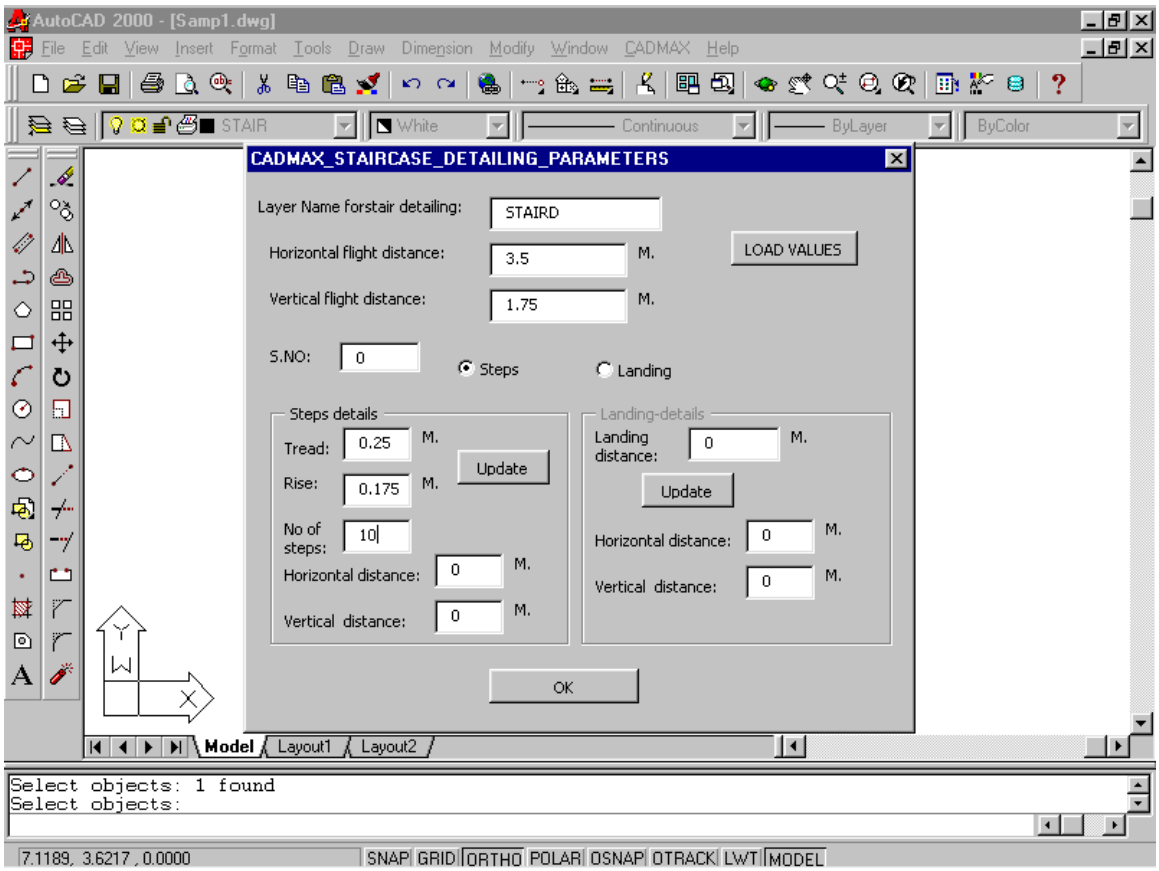
After selecting the slab now you will be asked to fill up the following dialog box.



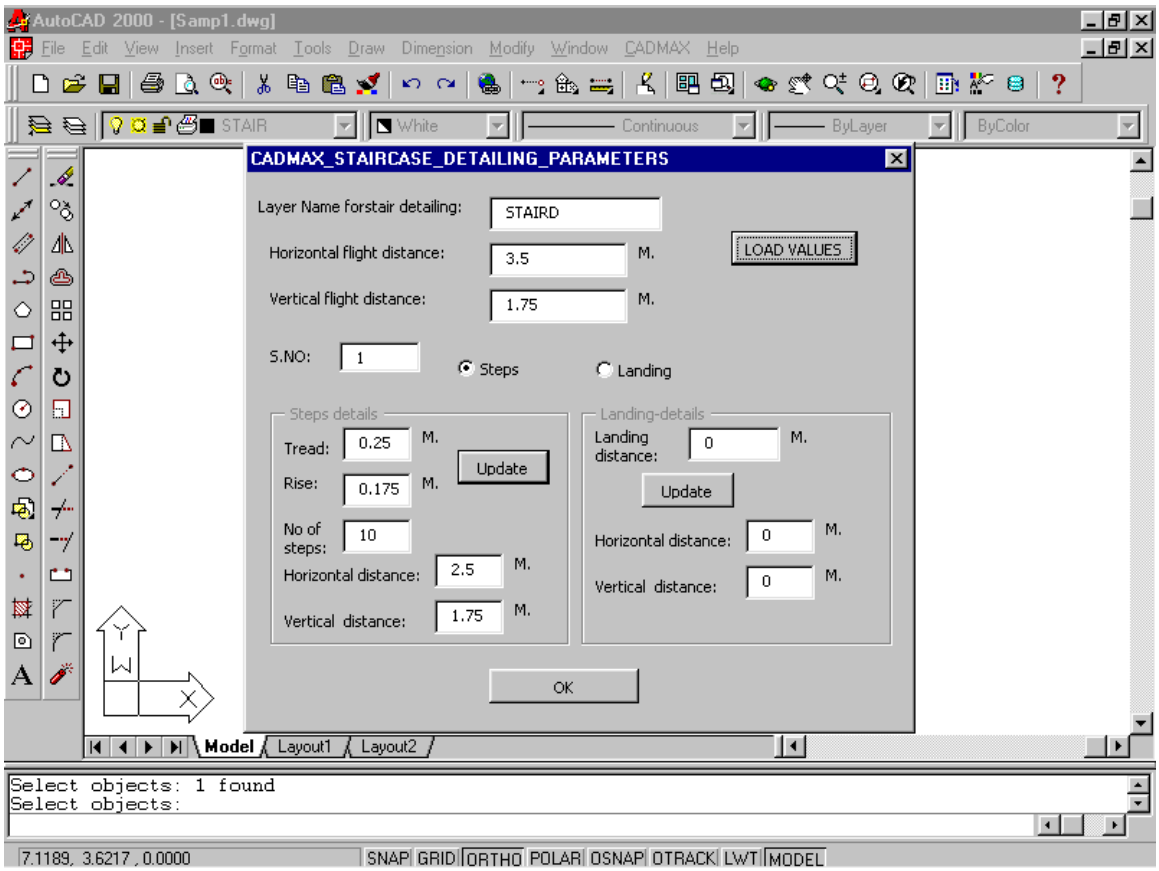
In the above dialog box you click the button LOAD VALUES and then you will see the change in dialog box with filled up values of Horizontal and Vertical distance of the flight slab is entered as shown below:



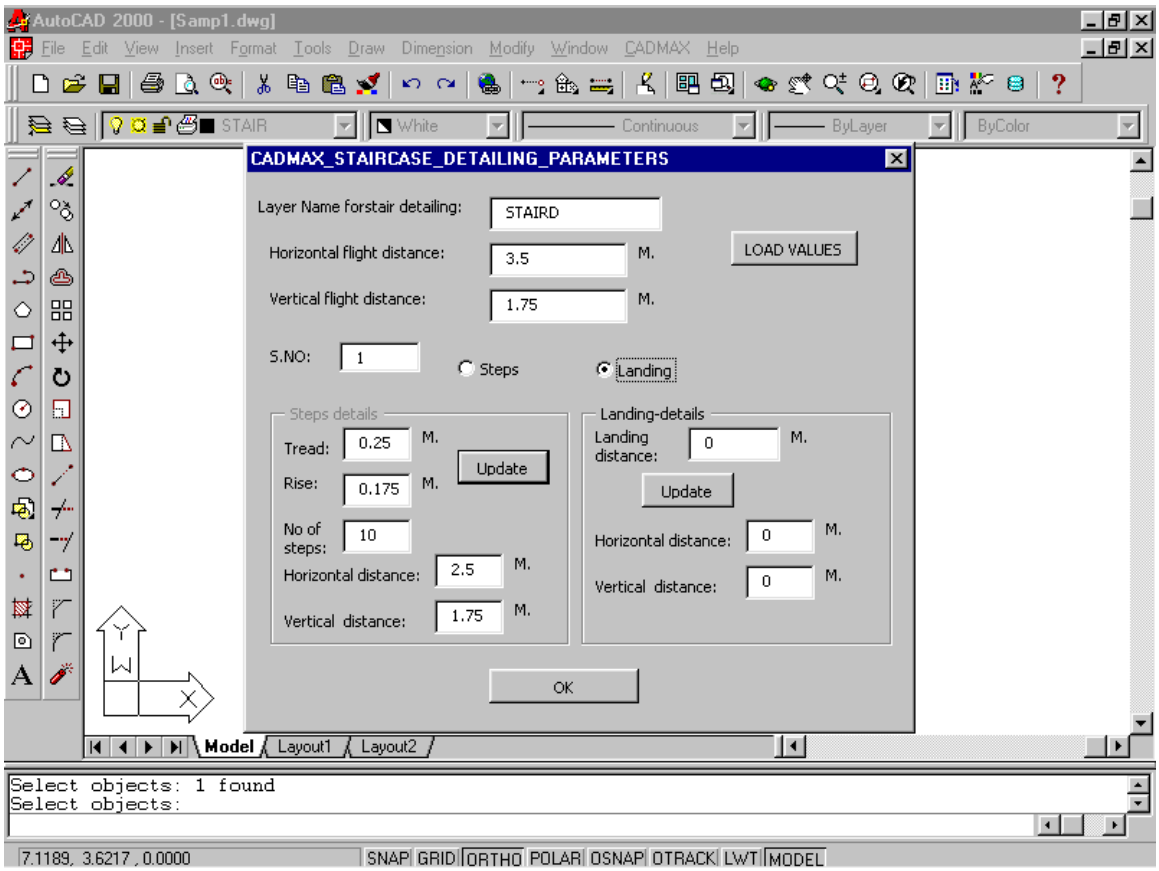
Now pick the option box Steps to draw steps first in the flight and then fill up tread and rise and No of steps as shown below:



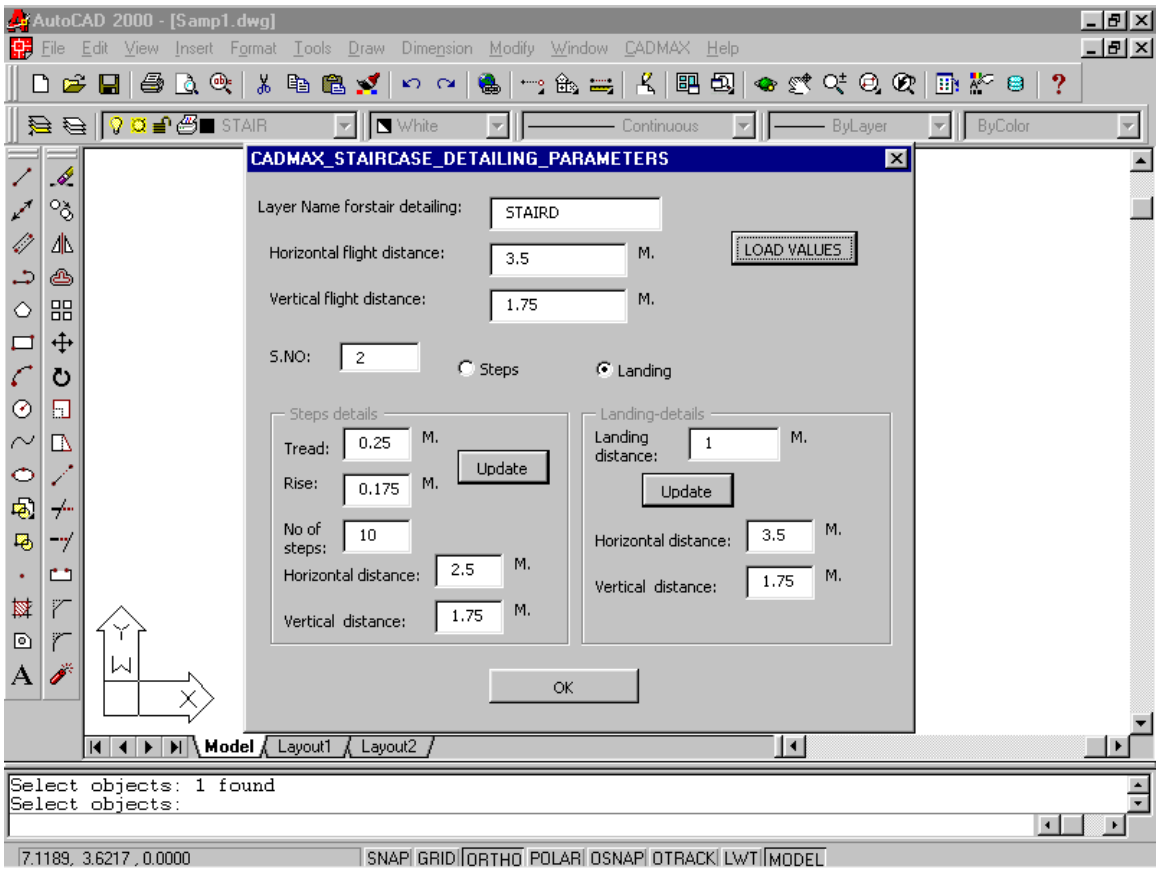
Next click Update button in steps details frame . Now you will see the Horizontal distance and vertical distance covered by the steps portion is filled up as shown below:



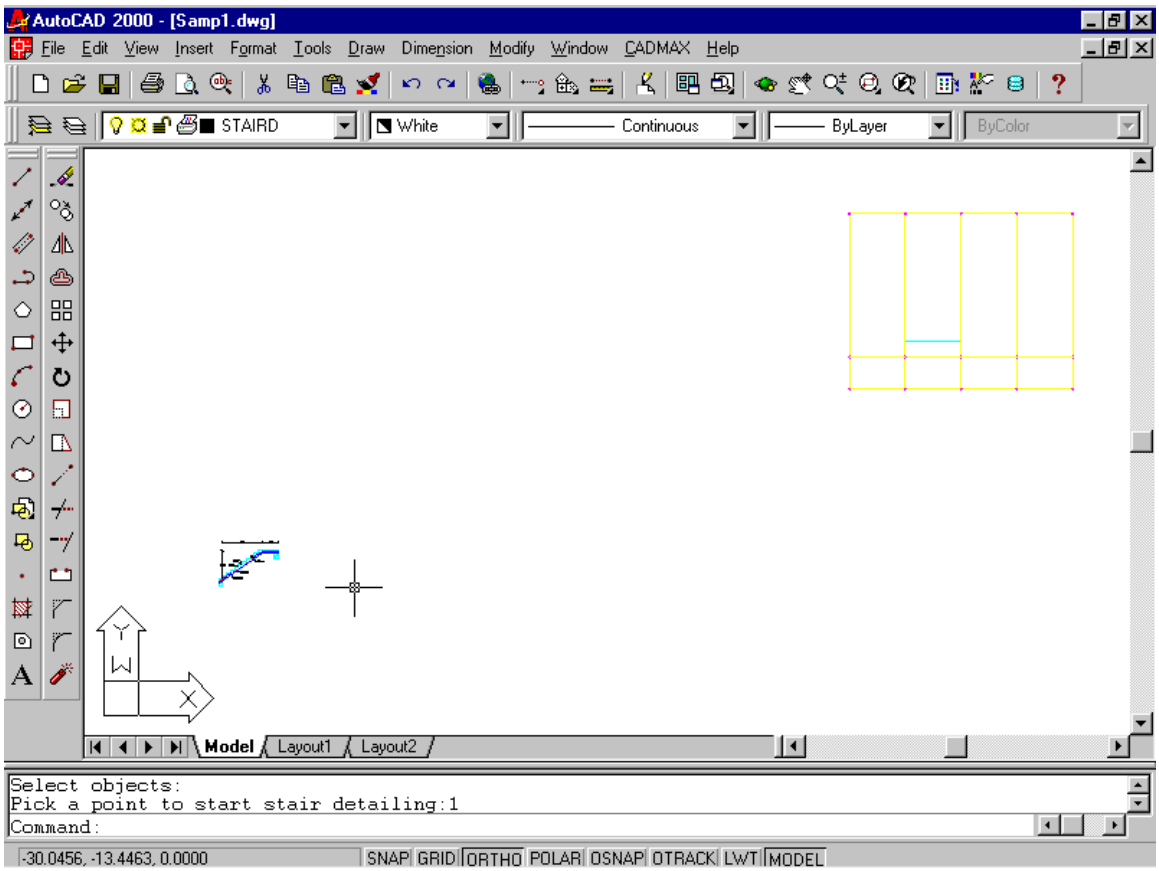
Now go to the Landing option box to fill up the landing details as shown below:



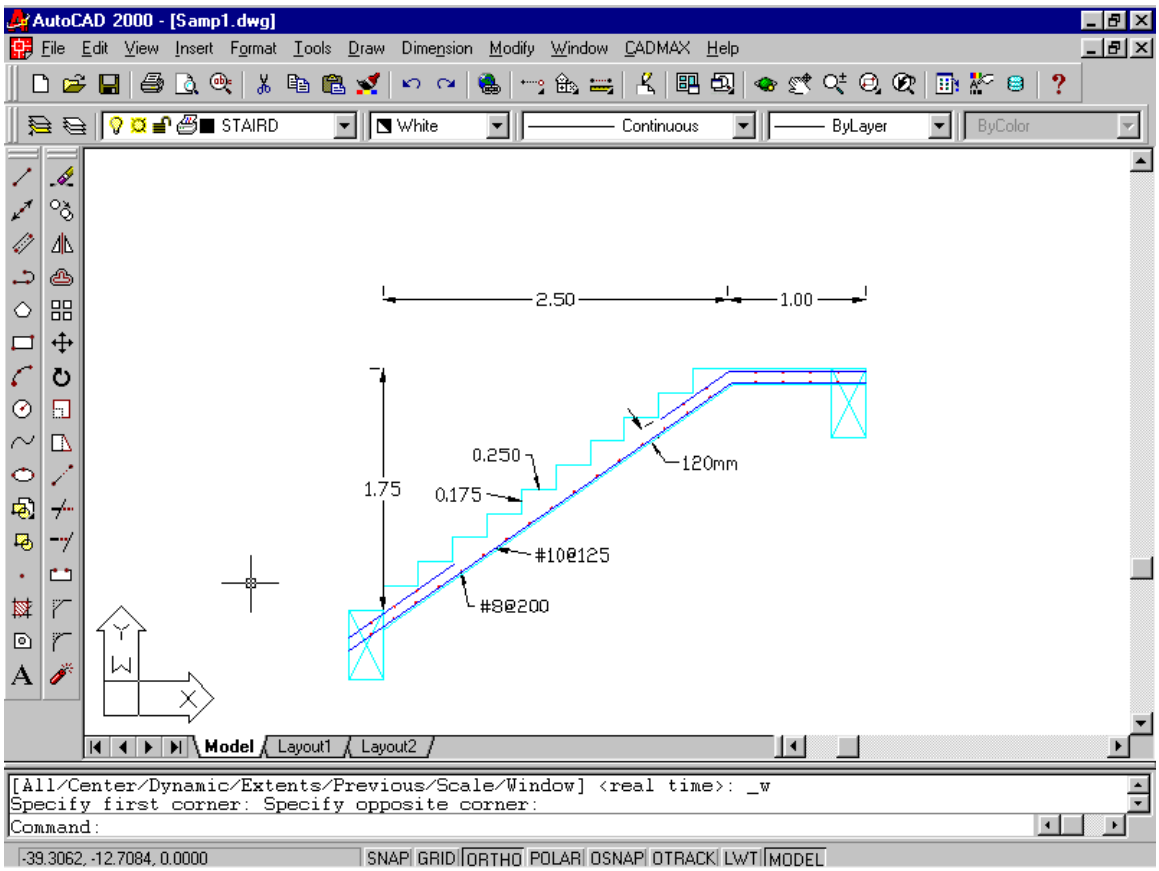
Now fill up the balance horizontal distance to be covered by landing as 1 and then click update button in landing portion as shown below:



Now you will see that the Horizontal distance and vertical distances tally with loaded values. Now Click OK button. Now you will be prompted to pick start point in the drawing and if you pick the point your stair case detailing will be drawn in the layer STAIRD as shown below:



Now zoom the drawing to see the details:



It is important to note that the OSNAP is off before giving the STAIRD command.

With this we conclude the detailing part of this software.