2 D FRAME ANALYSIS

Discover the Beauty of Structural Analysis

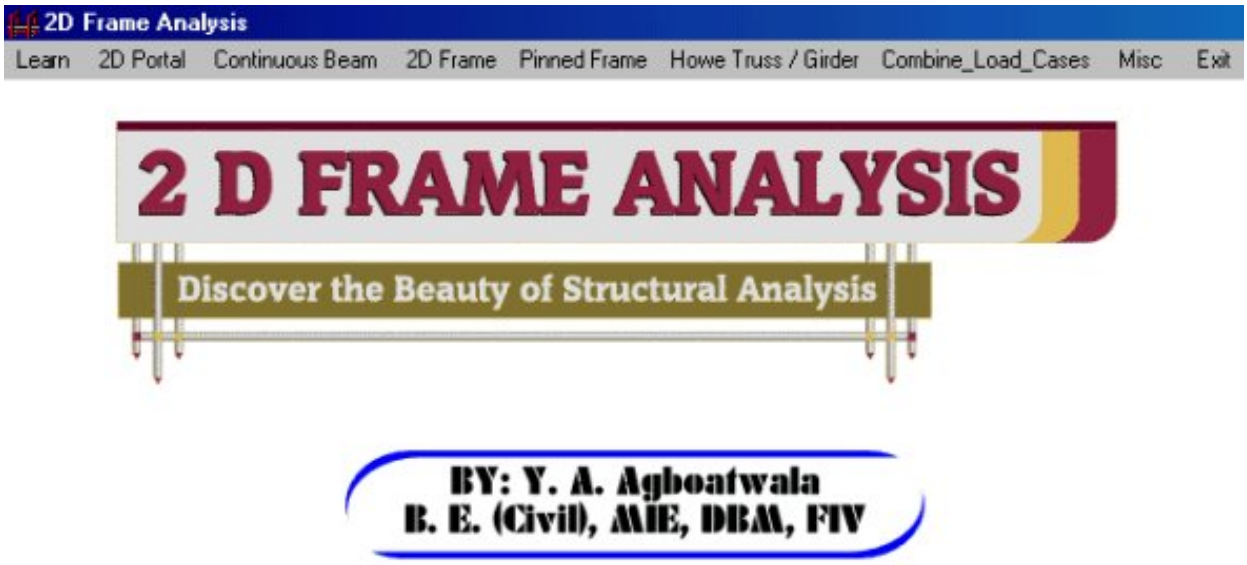
By :
Y.A. Agboatwala &
Fatima.Y. Agboatwala
1802, Jamuna Amrut,
219, Patel Estate, S.V.Road,
Jogeshwari(W), Mumbai - 400102
Phone: 09820792254 , (022) 26783525
Url: www.supercivilcd.com,
www.agboatwala.com
Email : yaa@supercivilcd.com,
yaa@agboatwala.com,
supercivilcd@gmail.com

LEARN 2D FRAME ANALYSIS

	GENERAL : FILE CREATION, COPY & DELETION
	PORTAL FRAME (Page No. 1)
	PORTAL FRAME (Page No. 2)
	CONTINUOUS BEAM
	2D FRAME - GENERAL
	PINNED FRAME - GENERAL
	HOWE TRUSS / OPEN WEB GIRDER
	COMBINE LOAD CASES
	TIPS

LEARN 2D FRAME ANALYSIS STEP BY STEP

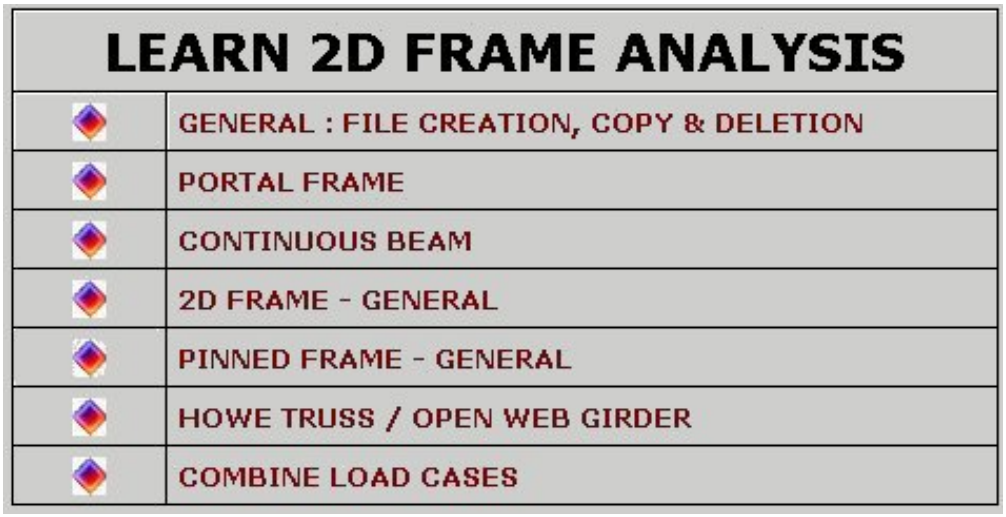
GENERAL INFO : FILE CREATION, COPY & DELETION



When Program starts, the graphics above is displayed. The Menu bar contains following options.

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

Click the LEARN option in the MENU bar. The following window will open.



The 1st option is : GENERAL : FILE CREATION, COPY & DELETION.
This option is same to all the 5 Programs i.e. " 2D Portal ", " Continuous Beam ", " 2D Frame ", " Pinned Frame " & " Howe Truss / Girder ".
Please note that following file extensions are used to differentiate between various analysis programs.
1. PORTAL FRAME ANALYSIS : ".DAT" as file extension.
2. CONTINUOUS BEAM ANALYSIS : ".00B" as file extension.
3. 2D FRAME - GENERAL ANALYSIS : ".2DF" as file extension.
4. PINNED FRAME - GENERAL ANALYSIS : ".2DP" as file extension.
5. HOWE TRUSS / OPEN WEB GIRDER ANALYSIS : ".HWE" as file extension.

Different Directories shall be created for each type of analysis. Do not store Portal Analysis files in say 2D Frame Analysis folder. Many files will over-write each other.

Difference between various Programs :

1. Portal Analysis is for structural cum seismic analysis of 2D multistory building frames. The Joint Co-Ordinates are automatically generated by just giving No. of Bays & Storeys.

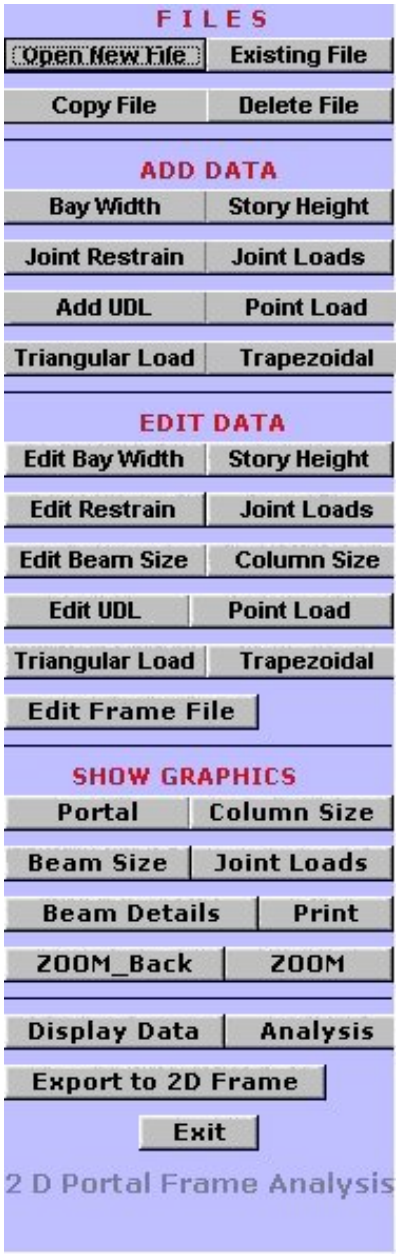
2. Continuous Beam Analysis is for structural analysis of Continuous Beams. The Joint Co-Ordinates are automatically generated by just giving No. of Bays.

3. 2D Frame Analysis is for structural analysis of General 2D Frames. The Joint Co-Ordinates are to be given manually in Table form.

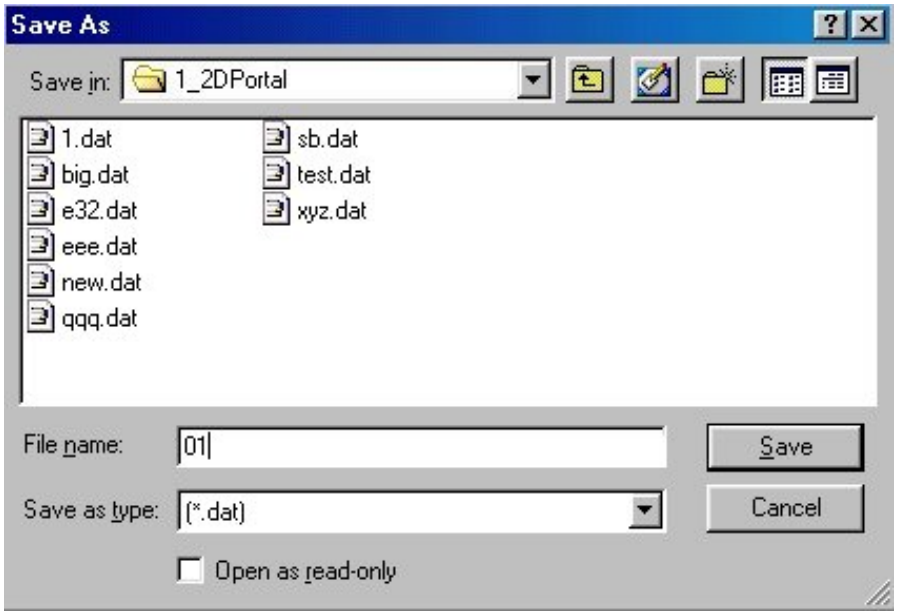
4. 2D Pinned Frame Analysis is for structural analysis of General 2D Pinned Frames / Trusses. The Joint Co-Ordinates are to be given manually in Table form.

5. Howe Truss / Open Web Girder Analysis is for structural analysis of 2D Pinned Howe Trusses & Open Web Girders. The Joint Co-Ordinates & Member Numbers are automatically generated by just giving No. of Panels & Height of Howe Truss / Girder.

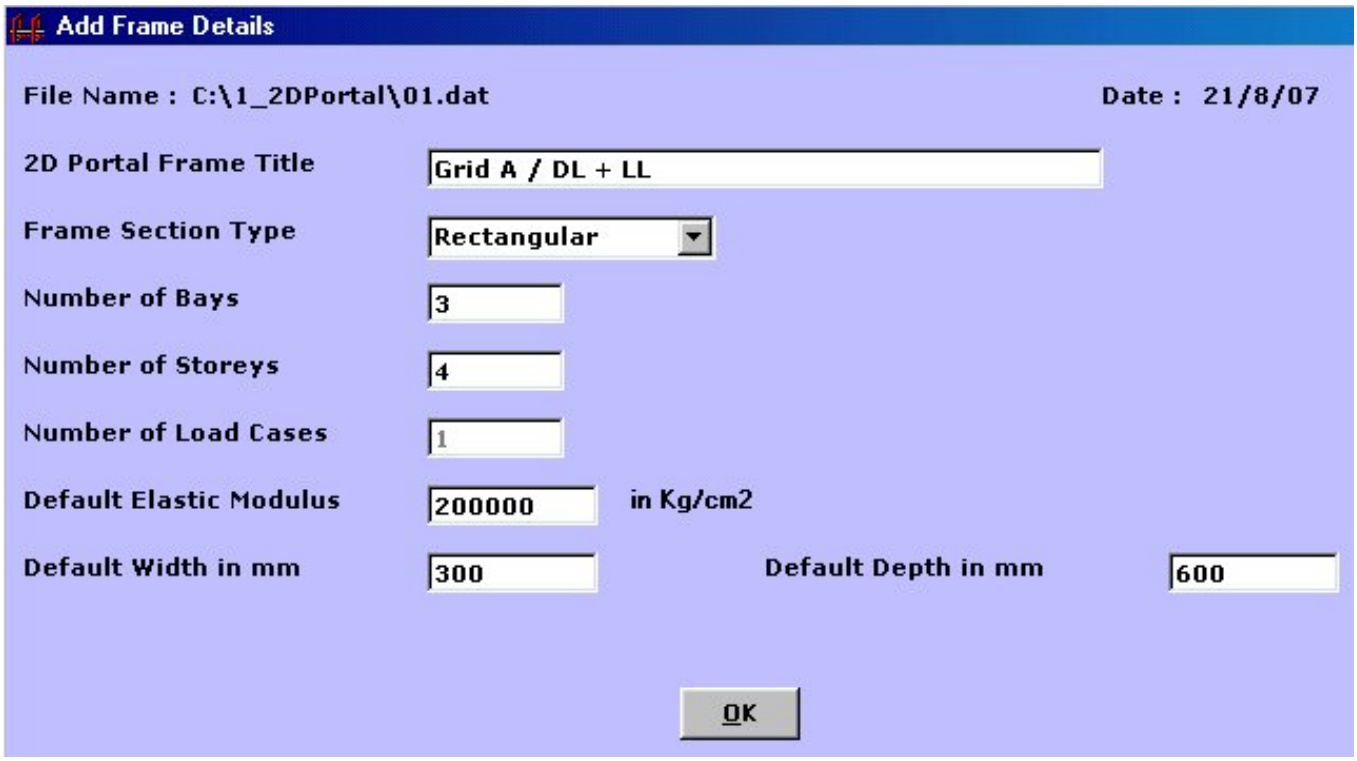
Now let us take the example from 2D PORTAL Analysis. Click 2D PORTAL menu. Following menu will appear.



Now click "Open New File" option. The following window will open.
You must create a separate Folder / Directory to store your files.
I have created a Directory called " 1_2DPortal " in C drive to store my 2D Portal Analysis files.
Now go to this folder & give a file name to your project. I have given " 01 " as the name of my new project file. Click the save button.



➤ **When you click the save button, following project window will open.**



➤ **The window requires various project details. Whatever values you will fill here will serve as default values for the project.**

I have filled up the above values as required by my new project " 01 ".
You can change these values now OR later by clicking "Edit Frame File" option in "EDIT DATA" Menu.
If a user wants to use Non-Rectangular Section, he can choose it by giving Area & Ixx (Moment of Inertia) in cm2 & cm4 respectively. For this "Frame Section Type" shall be selected as "Non-Rectangular", & user should click on "Default Width in mm" field. It will automatically change to Area and Ixx fields from existing Width & Depth Field.

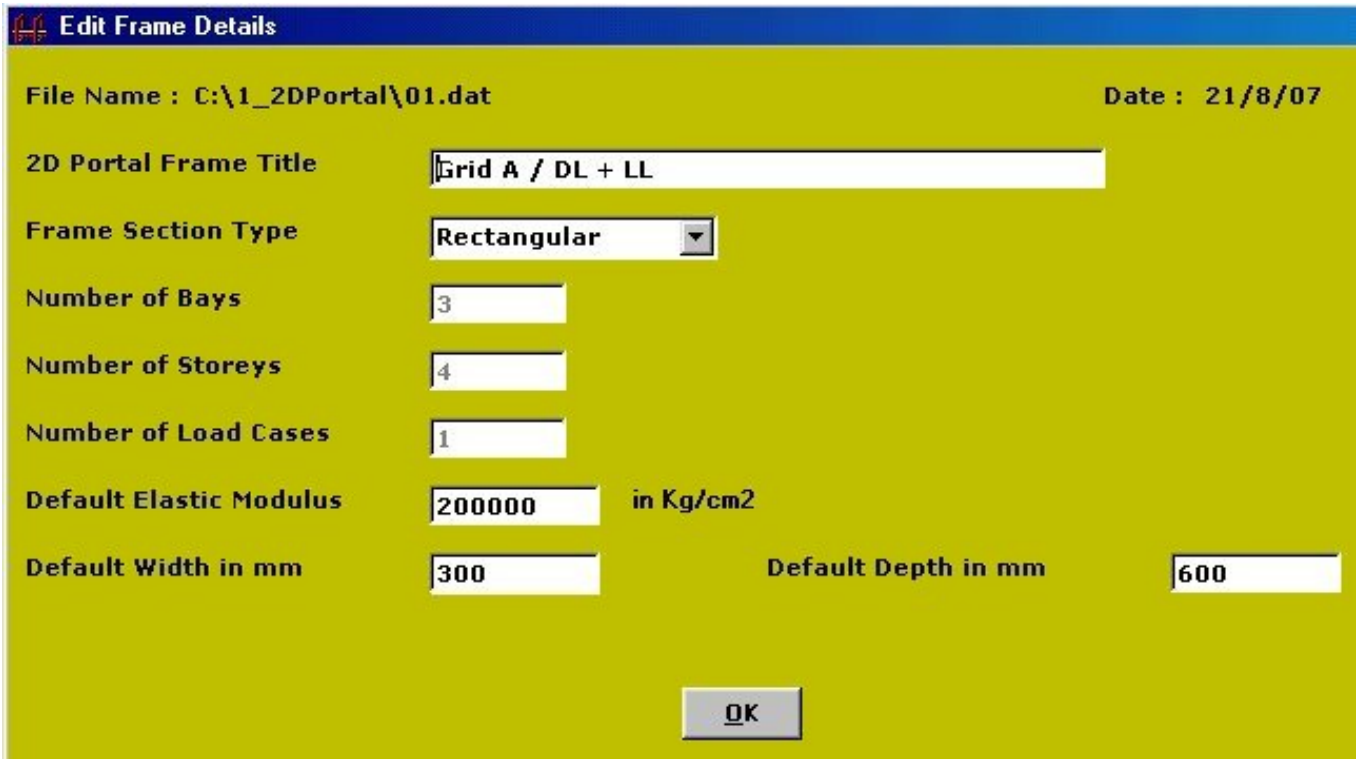
● **Note that Number of Load Cases is always shown as "1". A user should create another file by copying the existing main file & change its loading. This will act as another Load Case. Thus any number of Load Case files can be created. After all the load case files are analyzed, use " Combine_Load_Cases" option to sort the analysis results (Direct Force, Shear & BM) in Ascending OR Descending order.**

Now click the OK button, following window will appear.

Click OK button. Now project File creation is complete.
The above window gives the following five vital information.
1. Add Building Data (For Joint Co-ordinates) .
2. Add Member Data.
3. Add Joint Restrains (Roller / Hinged / Fixed).
4. Add Loads at Joints (Horizontal / Vertical / Moment).
5. Add Member Loads (UDL / Point / Triangular / Trapezoidal).

EDIT PROJECT DETAILS

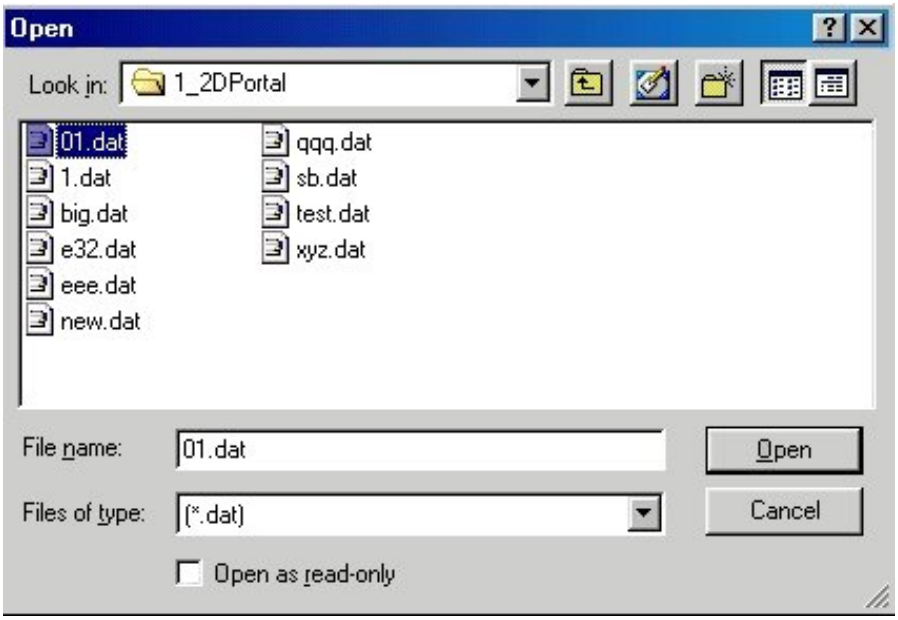
If you click the "Edit Frame File" option in 2D PORTAL menu, following window is displayed.



Use the above "Edit Frame File" option with Caution. Note that "Number of Bays" & "Storeys" are not editable. However Member properties are editable. The existing Member details will be over-written by default values.

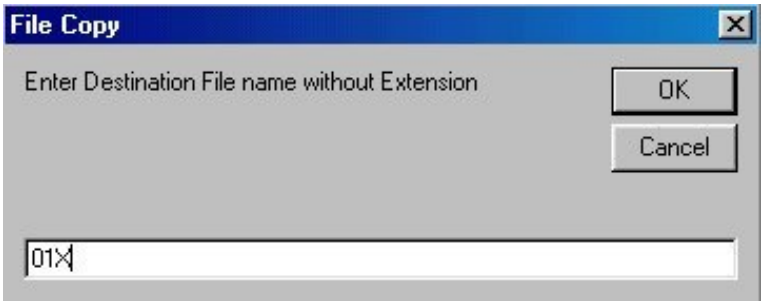
EXISTING FILE

Now click "Existing File" option from "2D PORTAL" menu. The following window will open.
You have to navigate the Windows File Open dialogue Box to the Directory / Folder, where you have stored your file.
I have created a Directory called " 1_2DPortal " in C drive to store my 2D Portal Analysis files.
I will go to this folder & select the " 01 " file & click the open button.
Now file "01" is open & I can use it to display the structure, provided Building (Joint) Data, Member Data & Restrain Data have been added.

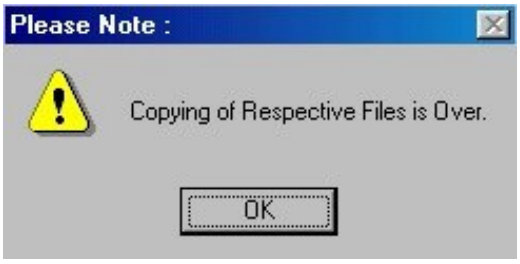


COPY FILE

Now click "Copy File" option from "2D PORTAL" menu. The window shown above will open.
You have to navigate the Windows File Open dialogue Box to the Directory / Folder, where you have stored your file.
I have created a Directory called " 1_2DPortal " in C drive to store my 2D Portal Analysis files.
I will go to this folder & select the " 01 " file & click the open button.
A new member will be displayed, asking for Destination File Name without Extension.
I have given the new file name as 01X. Now the contents of "01" file is copied to "01X" file.

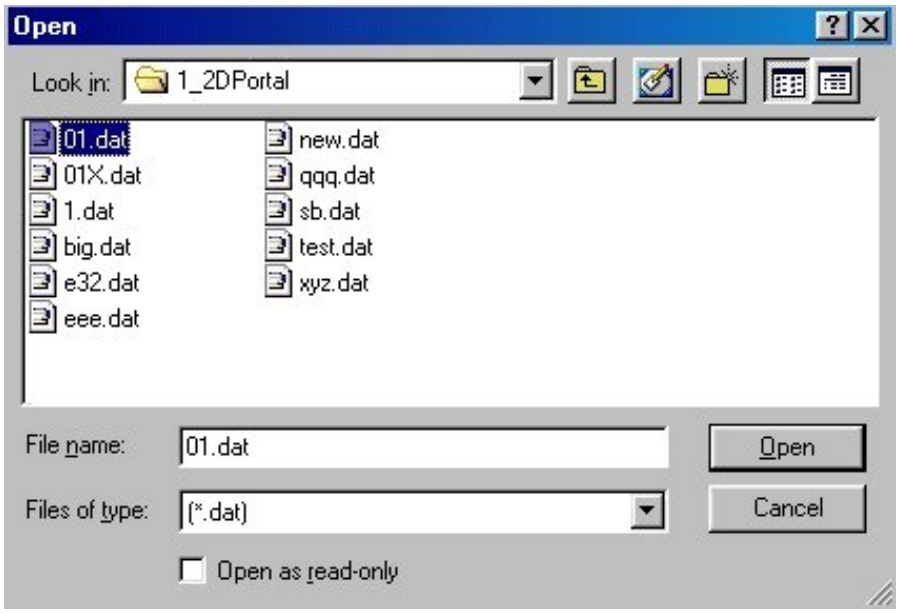


A new message is displayed as follows confirming copying of files.

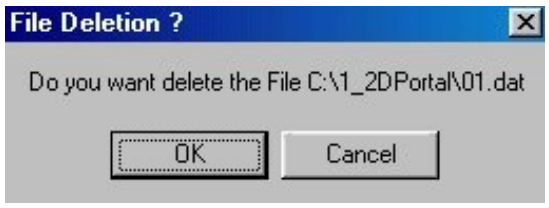


DELETE FILE

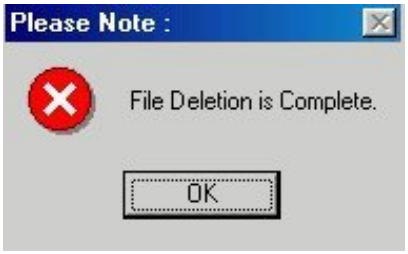
Now click "Delete File" option from "2D PORTAL" menu. The window shown below will open. You have to navigate the Windows File Open dialogue Box to the Directory / Folder, where you have stored your file.



I have decided to delete "01" file. Click open. Following message is displayed.

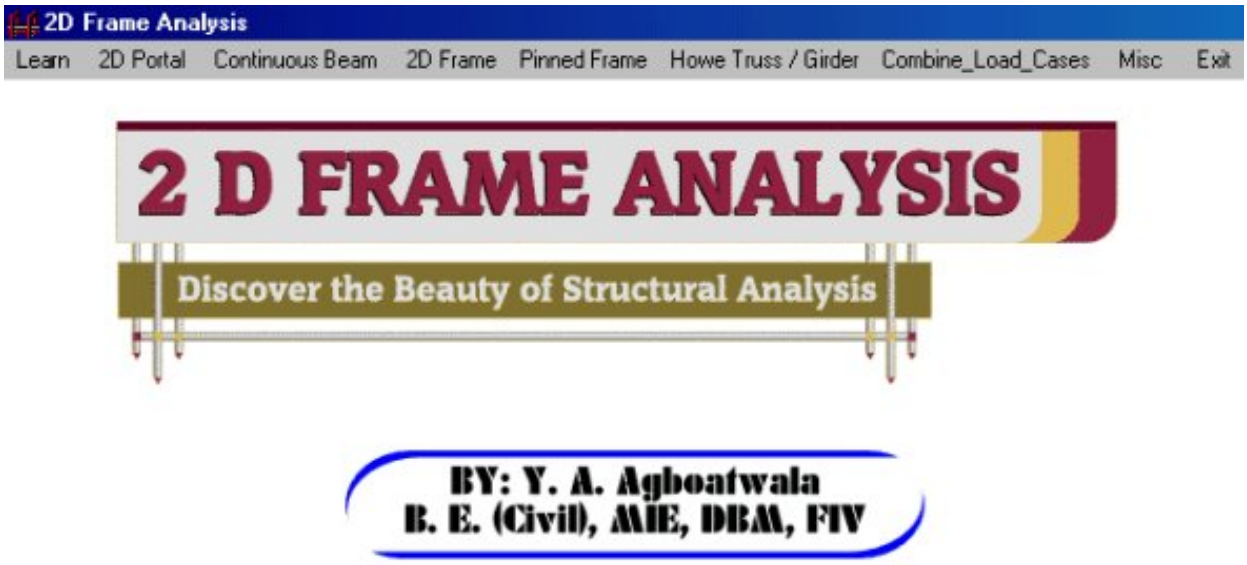


If "OK" button is pressed, file "01" will be deleted. If "Cancel" is pressed than file will remain intact. Press "OK" button. Following message is displayed.



This concludes the GENERAL INFO, FILE CREATION, COPY & DELETION option.

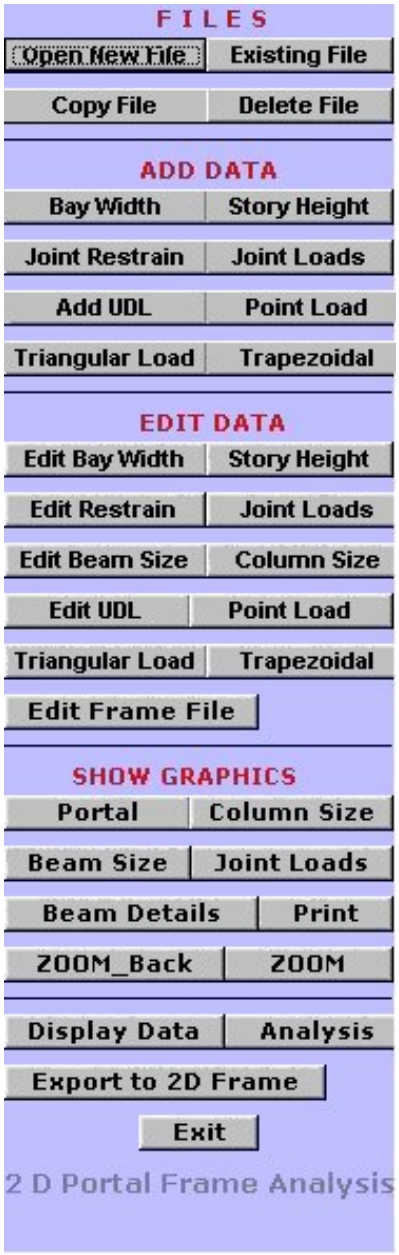
ANALYSIS OF MULTISTORY BUILDING 2D PORTAL FRAMES



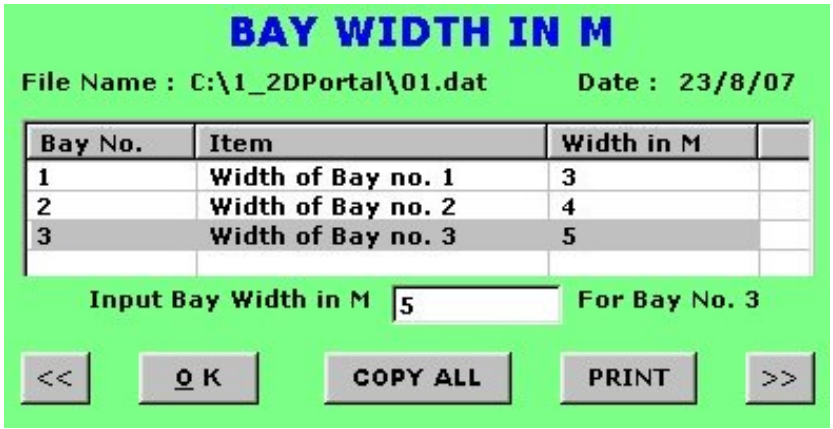
➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

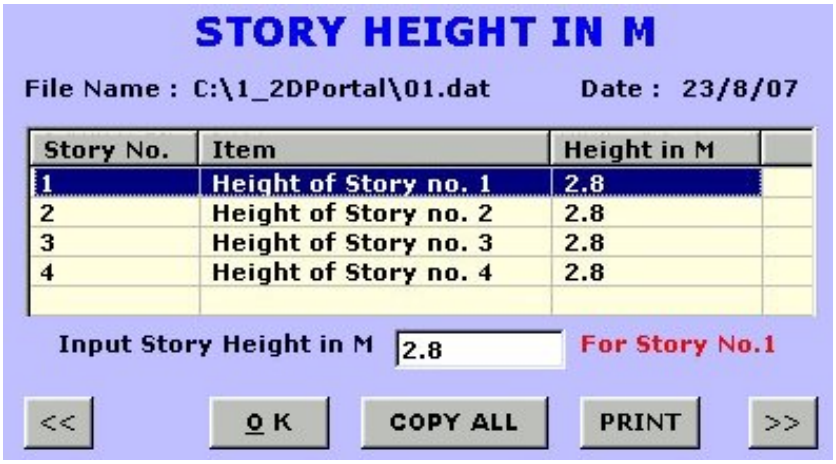
➤ **Click 2D PORTAL Option. Following menu will appear.**



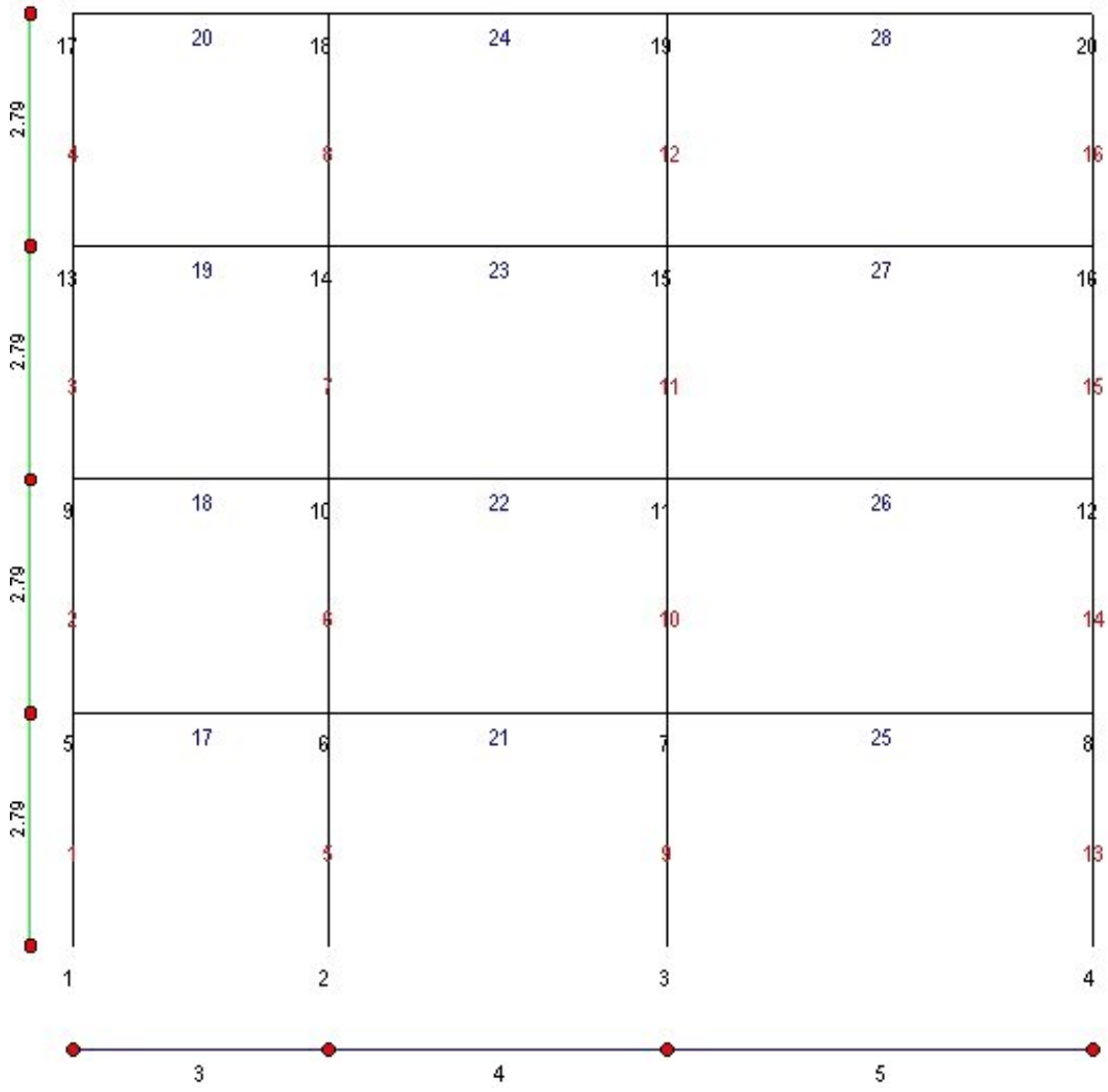
Go through GENERAL INFO : FILE CREATION, COPY & DELETION chapter before starting this option. File "01" is used through out for example.
The 1st item under "ADD DATA" menu is "Bay Width". When this option is clicked, following window will appear.



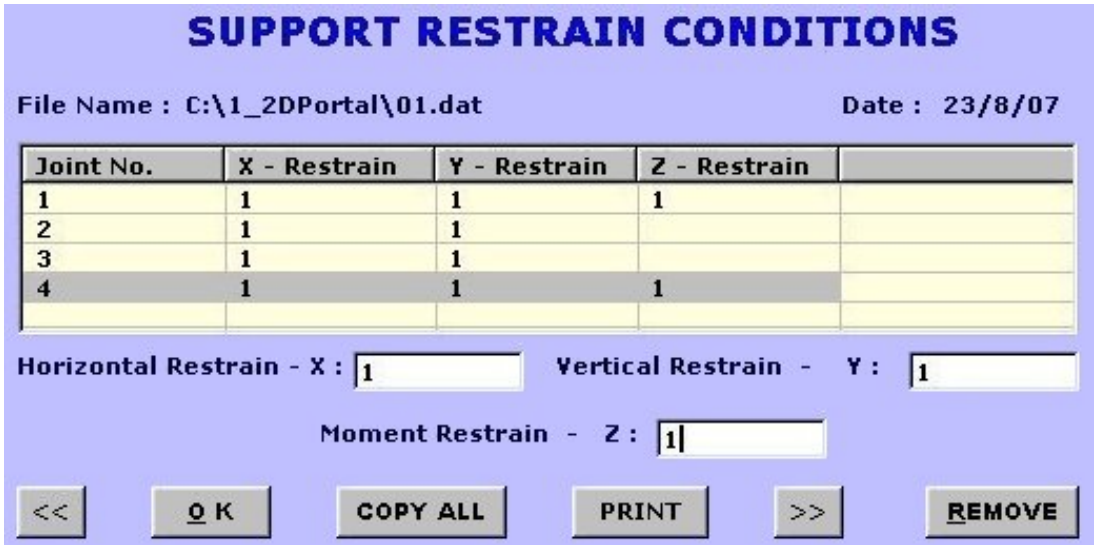
Enter the Bay Widths for individual Bays & click "OK" button. A user can also use "COPY ALL" button, if all bays are of same size. Use ">>" or "<<" button to move up or down a record, as an alternative to using mouse.
The next item under "ADD DATA" menu is "Story Height". When this option is clicked, following window will appear.



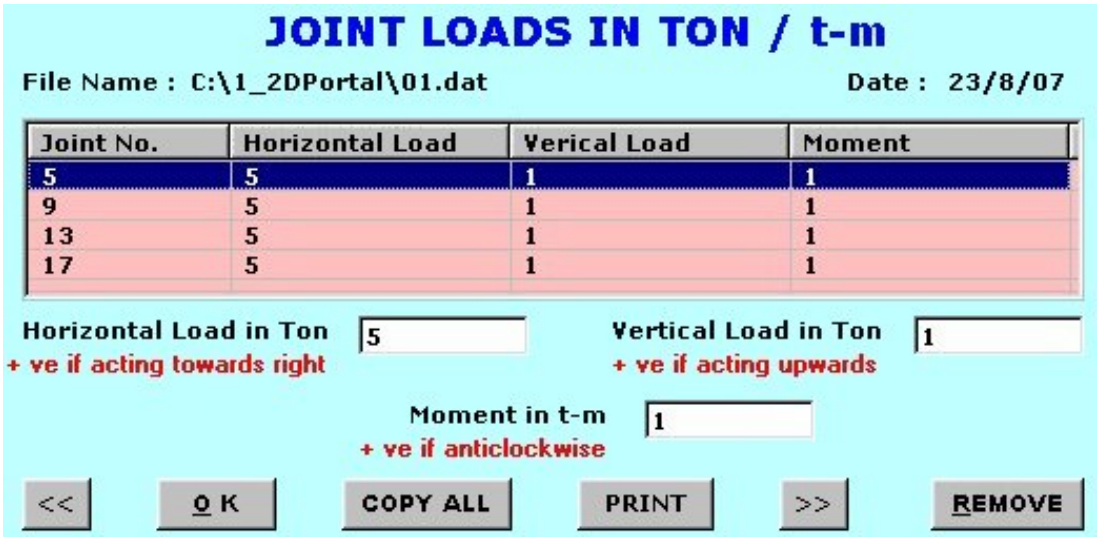
Enter the Story Heights for individual Bays & click "OK" button.
Now click "Portal" button under "SHOW GRAPHICS". Following Graphics will appear. It shows Joint #, Member #, Horizontal & Vertical Building dimensions in M.



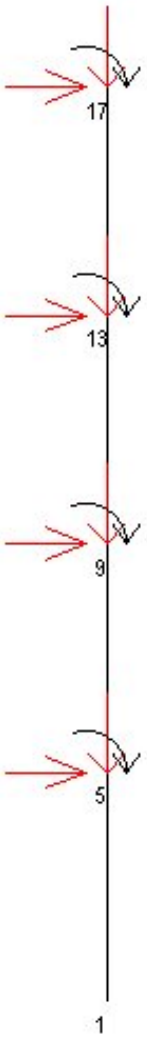
The next item under "ADD DATA" menu is add "Joint Restrain". When this option is clicked, following window will appear.



Enter the Support Restrain Condition (Fixed, Hinged or Roller - 1,1,1 / 1,1,0 / 0,1,0) for Support Joints & click "OK" button. Note that we have added "REMOVE" button, in case a user wants to delete any support. However user should be extra careful while deleting any support restrain.
Now click "Portal" button under "SHOW GRAPHICS". The Supports will be shown Graphically.
The next item under "ADD DATA" menu is add "Add Joint Loads". When this option is clicked, following window will appear.



Enter Joints Loads (Horizontal, Vertical Or Applied Moments). Normally Wind Or Seismic Horizontal Loads are applied. If a user does not want to indicate any joint load than it can be kept Blank or could be deleted using "REMOVE" button. However it is suggested to keep the joint loads blank, so that they can be used later on when required.
Now click "Joint Loads" button under "SHOW GRAPHICS". The Joint Loads will be shown Graphically.



The next item under "ADD DATA" menu is add "Add UDL". When this option is clicked, following Window will appear.

UDL IN TON / M

File Name : C:\1_2DPortal\01.dat Date : 24/8/07

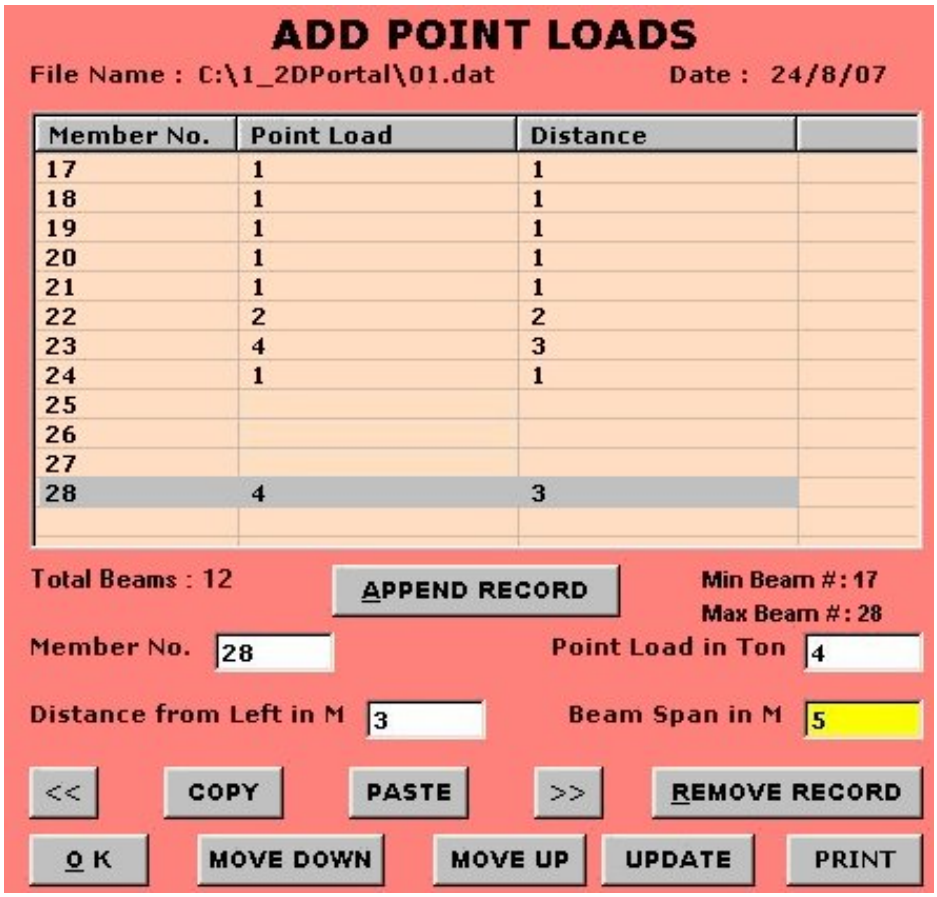
Member No.	Item	UDL in T/M
17	UDL on Beam Member No.17	1
18	UDL on Beam Member No.18	2
19	UDL on Beam Member No.19	3
20	UDL on Beam Member No.20	1
21	UDL on Beam Member No.21	2
22	UDL on Beam Member No.22	3
23	UDL on Beam Member No.23	1
24	UDL on Beam Member No.24	1
25	UDL on Beam Member No.25	1.5
26	UDL on Beam Member No.26	2
27	UDL on Beam Member No.27	2
28	UDL on Beam Member No.28	2

Input UDL in T/M [2] UDL on Beam Member No.28

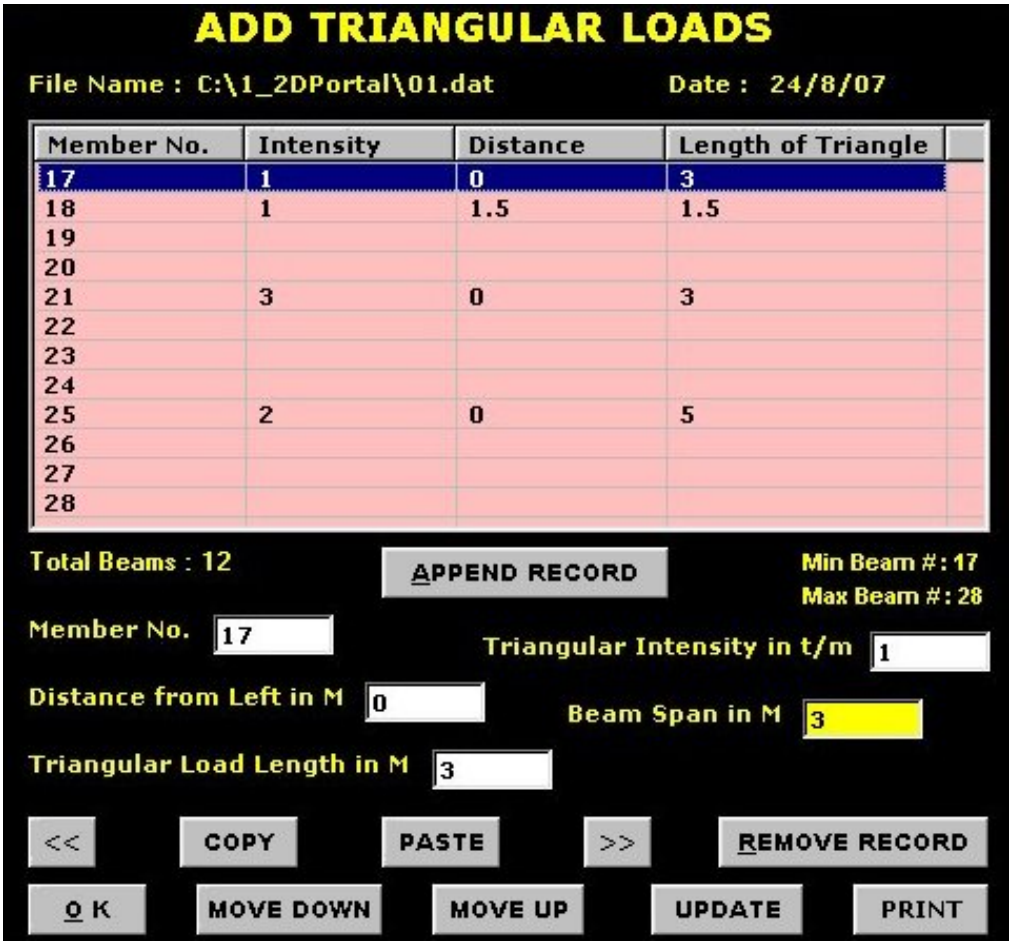
<< COPY PASTE >>

O K COPY ALL PRINT MOVE DOWN MOVE UP

User should enter all the desired UDL on Horizontal Members as displayed. No Horizontal member should be without UDL. UDL on vertical members are not indicated. Lateral Loads on Vertical members should be in the form of Horizontal Loads at Joints. *(All Joints should have two (2) members.)*. User can copy one record & paste it to other records one by one or use "copy all" button to copy UDL to all members. Use MOVE UP & MOVE DOWN buttons to rearrange the records if desired.
The next item under "ADD DATA" menu is add "Point Load". When this option is clicked, following Window will appear.



User can enter desired Point Loads on any Listed Horizontal Members. Any number of Point Loads can be added to one member by using APPEND RECORD button. While inputting records care should be taken that DISTANCE FROM LEFT should not exceed MEMBER SPAN. After adding multiple records use UPDATE button to re-arrange the records.
The next item under "ADD DATA" menu is add "Triangular Load". When this option is clicked, following Window will appear.

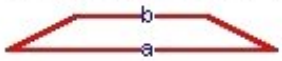


User can enter desired Triangular Loads on any Listed Horizontal Members. Any number of Triangular Loads can be added to one member by using APPEND RECORD button. While inputting records care should be taken that DISTANCE FROM LEFT + TRIANGULAR LOAD LENGTH should not exceed MEMBER SPAN. After adding multiple records use UPDATE button to re-arrange the records.

The next item under "ADD DATA" menu is add "Trapezoidal Load". When this option is clicked, following Window will appear.

ADD TRAPEZOIDAL LOADS

File Name : C:\1_2DPortal\01.dat



Date : 24/8/07

Member No.	Intensity	Distance	Base length	Top Length
17	1	0	3	2
18	1	0	2	1
19				
20				
21				
22				
23	2	2	2	.75
24				
25				
26				
27				
28				
17	3	1	2	1

Total Beams : 12

APPEND RECORD

Min Beam #: 17
Max Beam #: 28

Member No. 17

Trapezoidal Intensity in t/m 1

Distance from Left in M 0

Base Load Length (a) in M 3

Top Load Length (b) in M 2

Beam Span in M 3

<< COPY PASTE >> REMOVE RECORD

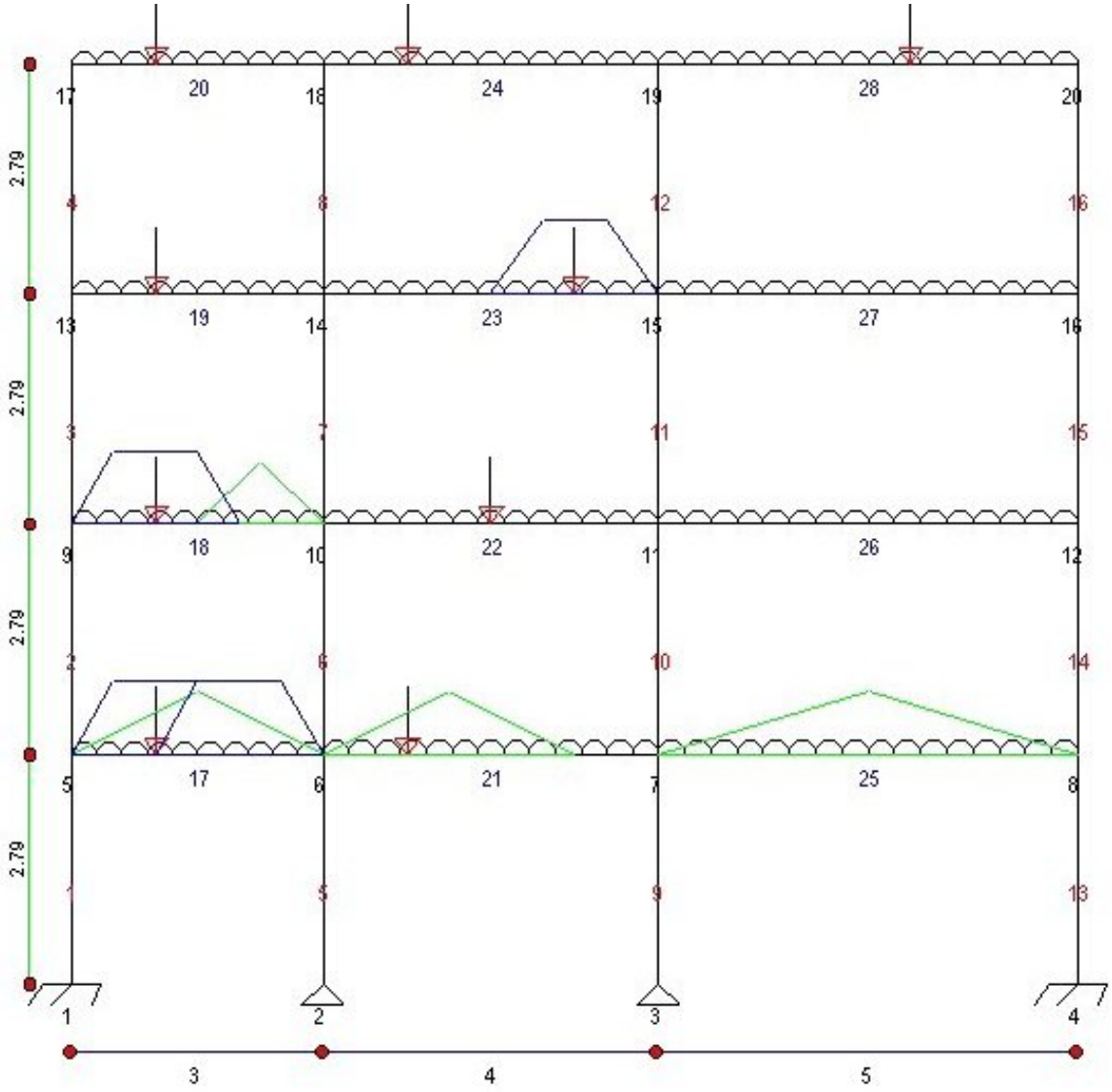
O K MOVE DOWN MOVE UP UPDATE PRINT

➤ **User can enter desired Trapezoidal Loads on any Listed Horizontal Members. Any number of Trapezoidal Loads can be added to one member by using APPEND RECORD button. While inputting records care should be taken that DISTANCE FROM LEFT + TRAPEZOIDAL BASE LOAD LENGTH should not exceed MEMBER SPAN. Also TOP LOAD LENGTH, should not exceed TRAPEZOIDAL BASE LOAD LENGTH.**

● **Note that PARTIAL UDL on BEAMS is a special case of TRAPEZOIDAL LOADS, when top & bottom load length are same. However a user should keep the top length less by at least 100 mm, else program will give error.**

After adding multiple records use UPDATE button to re-arrange the records.

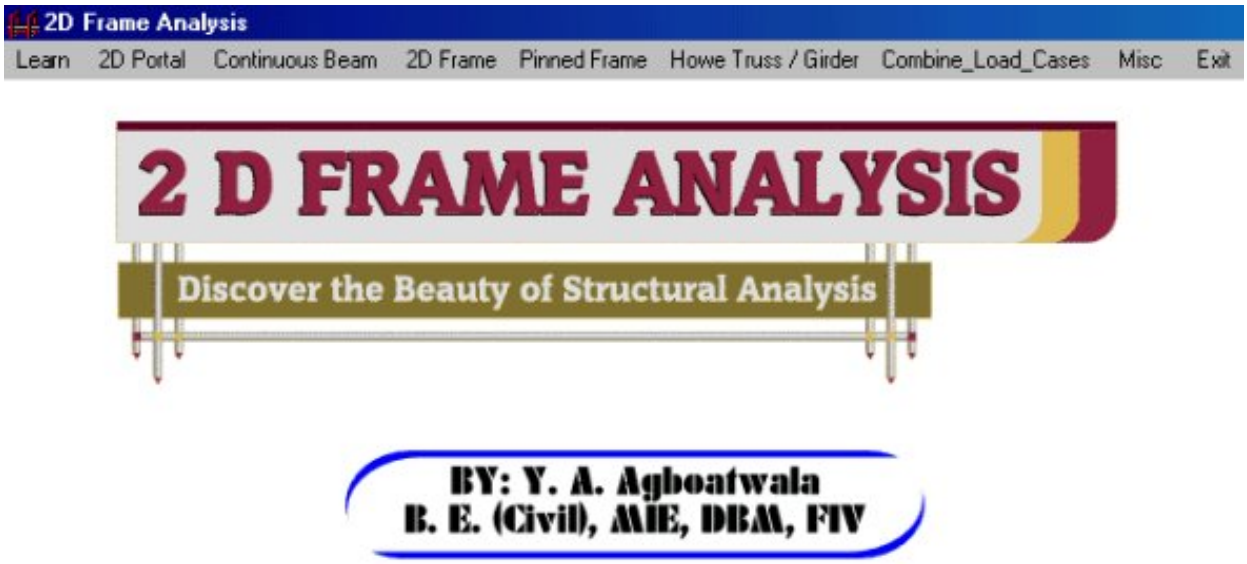
Now click "Portal" button under "SHOW GRAPHICS". Following Graphics will appear. It shows Joint #, Member #, Horizontal, Vertical Building dimensions, Loading & Supports.



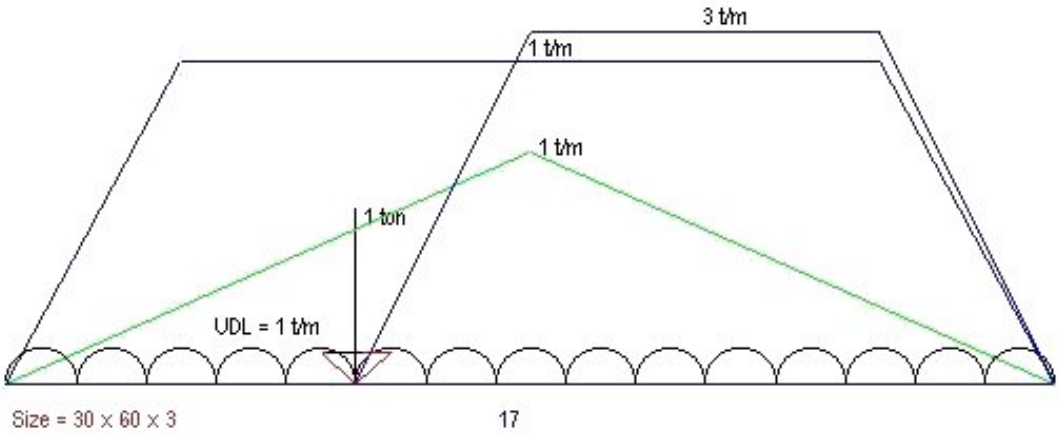
ZOOM : When ZOOM is clicked on SHOW GRAPHICS Menu & again clicked & Dragged on the display area a Rectangular wire line appears. When the mouse is released the selected area is ZOOMED. Use ZOOM_BACK button to restore original shape.
For better results use Square selection as far as possible.

This completes the ADD Menu.
The EDIT Menu is same as ADD Menu, except that it allows a user to Edit (Change) / Append (Add) the already existing records.
Click the next Page.

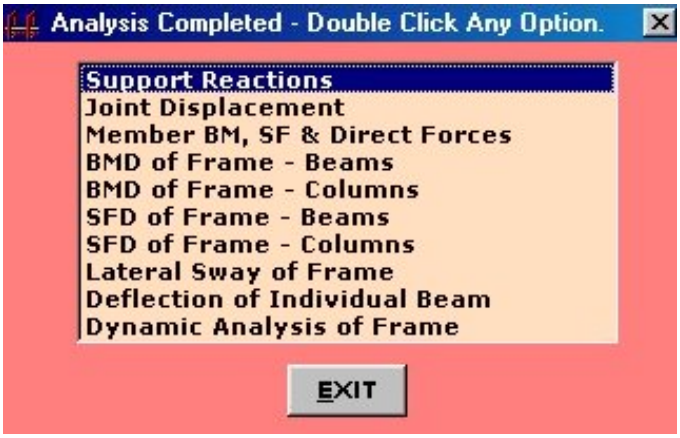
ANALYSIS OF MULTISTORY BUILDING 2D PORTAL FRAMES



➤ **Now click "Beam Detail" button under "SHOW GRAPHICS". Select Beam (Horizontal member). Following Graphics will appear. It shows Member #, Beam Size & Loading.**



➤ **All other Graphical options are self explanatory.**
Now click "ANALYSIS" option. Following Window opens up.



➤ **Click SUPPORT REACTION. Following window opens up.**

SUPPORT REACTIONS IN T / T-M

File Name : C:\1_2DPortal\01.dat Date : 25/7/07

Joint #	Horizontal	Vertical	Moment
1	-7.761	1.387	13.812
2	-2.037	42.479	0
3	-1.766	46.969	0
4	-8.438	30.653	14.231

<< | O K | PRINT | >>

+ ve if acting towards right + ve if acting upwards

+ ve if anticlockwise

➤ **Click JOINT DISPLACEMENT. Following window opens up.**

2D Portal Frame

JOINT DISPLACEMENT IN MM

File Name : C:\1_2DPortal\01.dat Date : 25/7/07

Joint #	Horizontal	Vertical	Rotational
1	0	0	0
2	0	0	-0.01102
3	0	0	-0.01063
4	0	0	0
5	2.38461	-0.0108	-0.00765
6	2.39451	-0.33039	-0.00363
7	2.37574	-0.36531	-0.00422
8	2.30706	-0.23841	-0.00628
9	4.36627	-0.03769	-0.00461
10	4.32787	-0.5184	-0.00424
11	4.30493	-0.63602	-0.0035
12	4.29673	-0.40606	-0.00372
13	5.70471	-0.06548	-0.00353
14	5.67459	-0.60939	-0.00252
15	5.65061	-0.80728	-0.00183
16	5.66195	-0.51844	-0.00253
17	6.47311	-0.0738	-0.00222
18	6.42974	-0.63626	-0.00178
19	6.38205	-0.88081	-0.00271
20	6.3304	-0.58082	0.00224

<< | O K | PRINT | >>

+ ve if acting towards right + ve if acting upwards

+ ve if anticlockwise

➤ **Click MEMBER BM, SF & DIRECT FORCES. Following window opens up.**

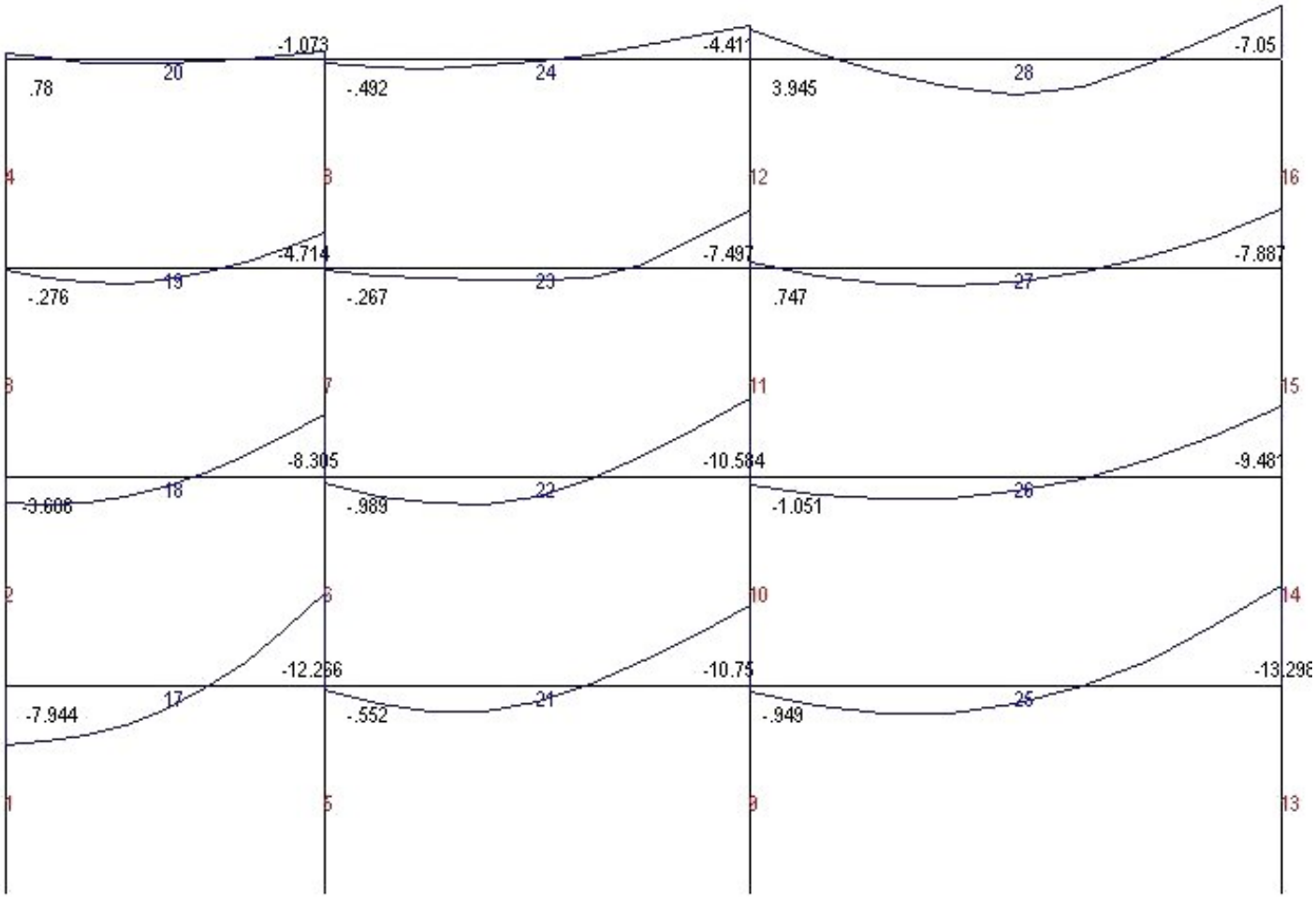
MEMBER FORCES IN T / T-M

File Name : C:\1_2DPortal\01.dat

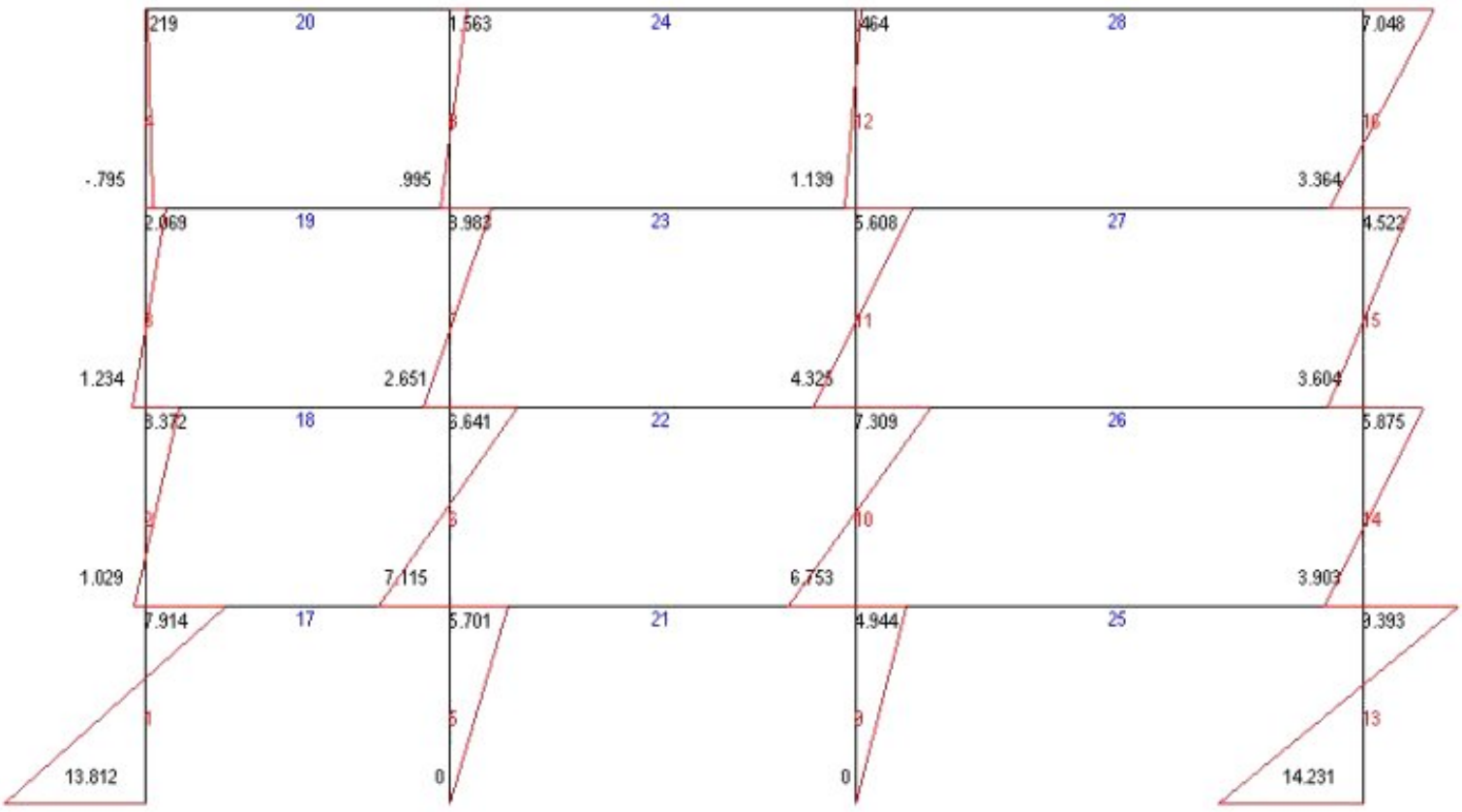
Date : 25/7/07

#	Near End	Axial	Shear	BM	Far End	Axial	Shear	BM	Span BM	Distance
1	1	1.387	7.76	13.812	5	-1.388	-7.761	7.914		
2	5	3.457	1.572	1.029	9	-3.458	-1.573	3.372		
3	9	3.573	1.18	1.234	13	-3.574	-1.181	2.069		
4	13	1.069	-0.206	-0.795	17	-1.07	.205	0.219		
5	2	42.479	2.036	0	6	-42.48	-2.037	5.701		
6	6	24.172	4.913	7.115	10	-24.173	-4.914	6.641		
7	10	11.699	2.369	2.651	14	-11.7	-2.37	3.983		
8	14	3.455	0.913	0.995	18	-3.456	-.914	1.563		
9	3	46.969	1.765	0	7	-46.97	-1.766	4.944		
10	7	34.805	5.022	6.753	11	-34.806	-5.023	7.309		
11	11	22.018	3.547	4.325	15	-22.019	-3.548	5.608		
12	15	9.454	0.572	1.139	19	-9.455	-.573	0.464		
13	4	30.653	8.437	14.231	8	-30.654	-8.438	9.393		
14	8	21.554	3.492	3.903	12	-21.555	-3.493	5.875		
15	12	14.448	2.902	3.604	16	-14.449	-2.903	4.522		
16	16	8.02	3.718	3.364	20	-8.021	-3.719	7.048		
17	5	-1.188	-1.07	-7.944	6	1.187	13.569	-12.265		
18	9	4.607	0.883	-3.608	10	-4.608	8.366	-8.305	3.78	.371
19	13	3.614	3.504	-0.276	14	-3.615	6.495	-4.713	2.279	1
20	17	5.205	2.069	0.78	18	-5.206	1.93	-1.072	0.791	1.069
21	6	1.688	4.737	-0.552	7	-1.689	8.762	-10.75	4.028	1.221
22	10	2.064	4.106	-0.989	11	-2.065	9.893	-10.584	3.799	1.368
23	14	2.158	1.747	-0.267	15	-2.159	8.992	-7.496	1.793	1.747
24	18	4.291	1.524	-0.492	19	-4.292	3.475	-4.41	1.516	1
25	7	4.945	3.4	-0.949	8	-4.946	9.099	-13.297	3.925	1.616
26	11	0.59	2.893	-1.051	12	-0.591	7.106	-9.481	3.144	1.446
27	15	-0.817	3.572	0.747	16	0.816	6.427	-7.887	2.442	1.786
28	19	3.718	5.979	3.945	20	-3.719	8.02	-7.049	4.993	2.989

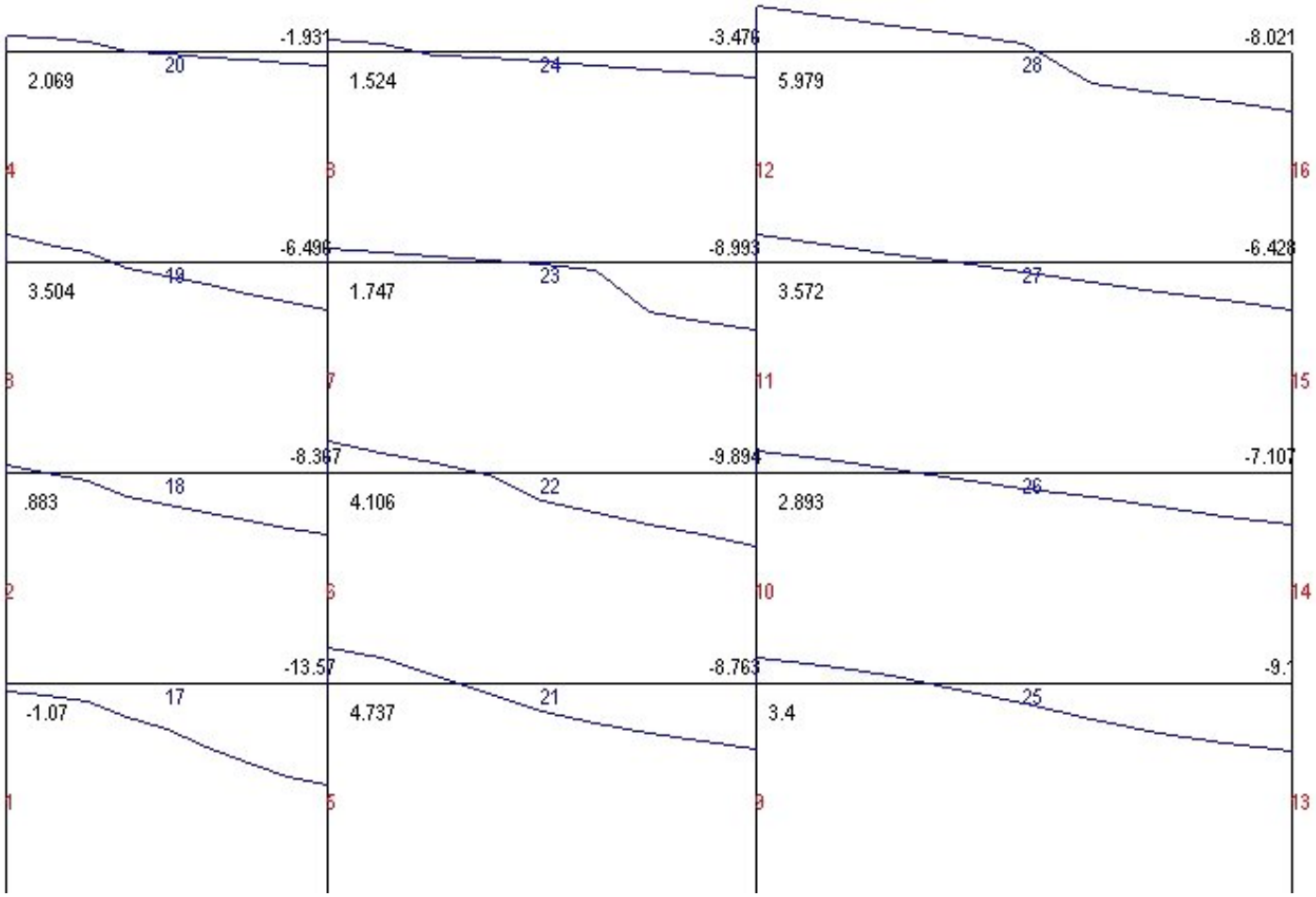
Click BMD OF FRAME - BEAMS. Enter Magnification Factor, say 10. Following window opens up.



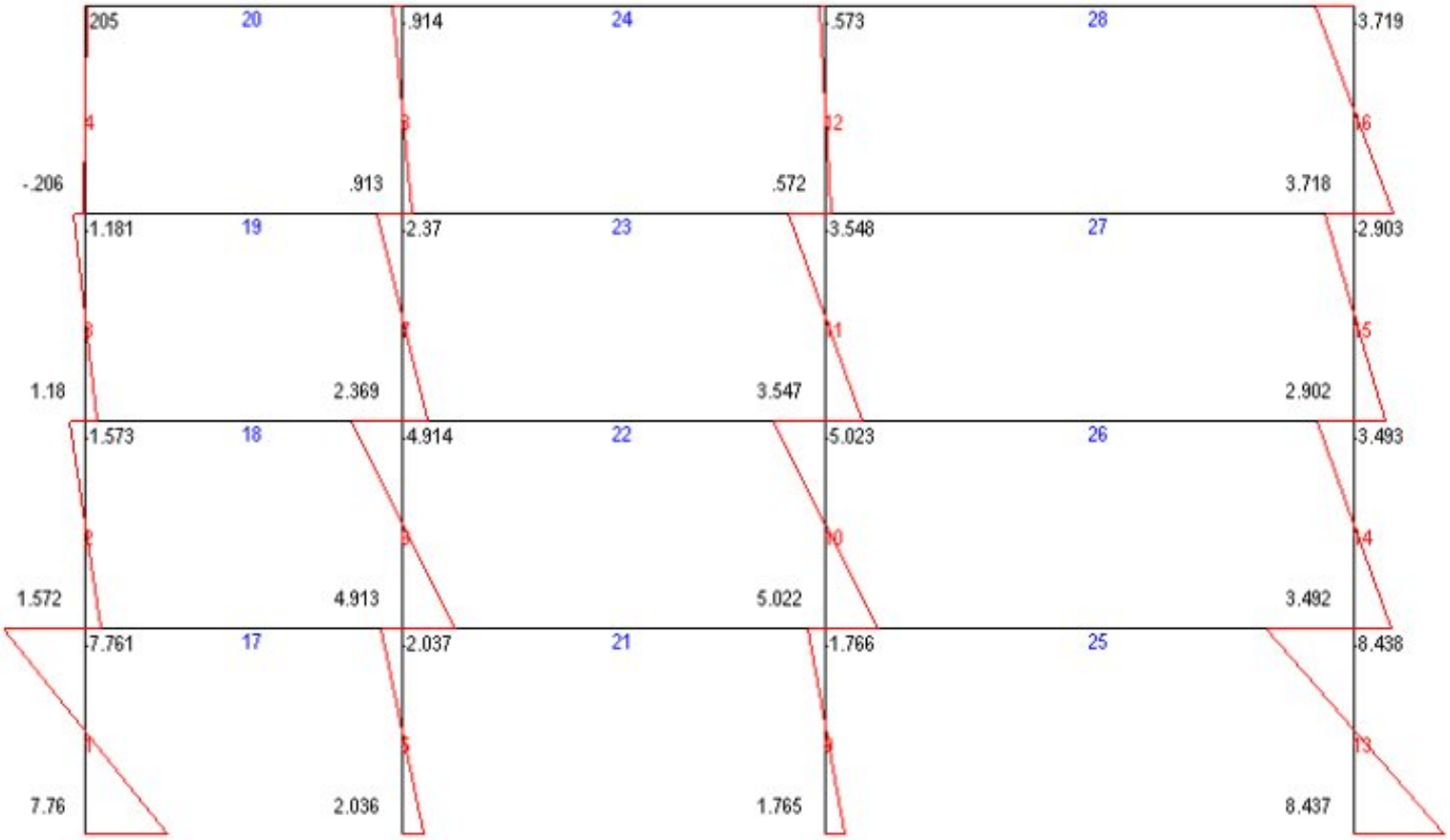
Click BMD OF FRAME - COLUMNS. Enter Magnification Factor, say 10. Following window opens up.



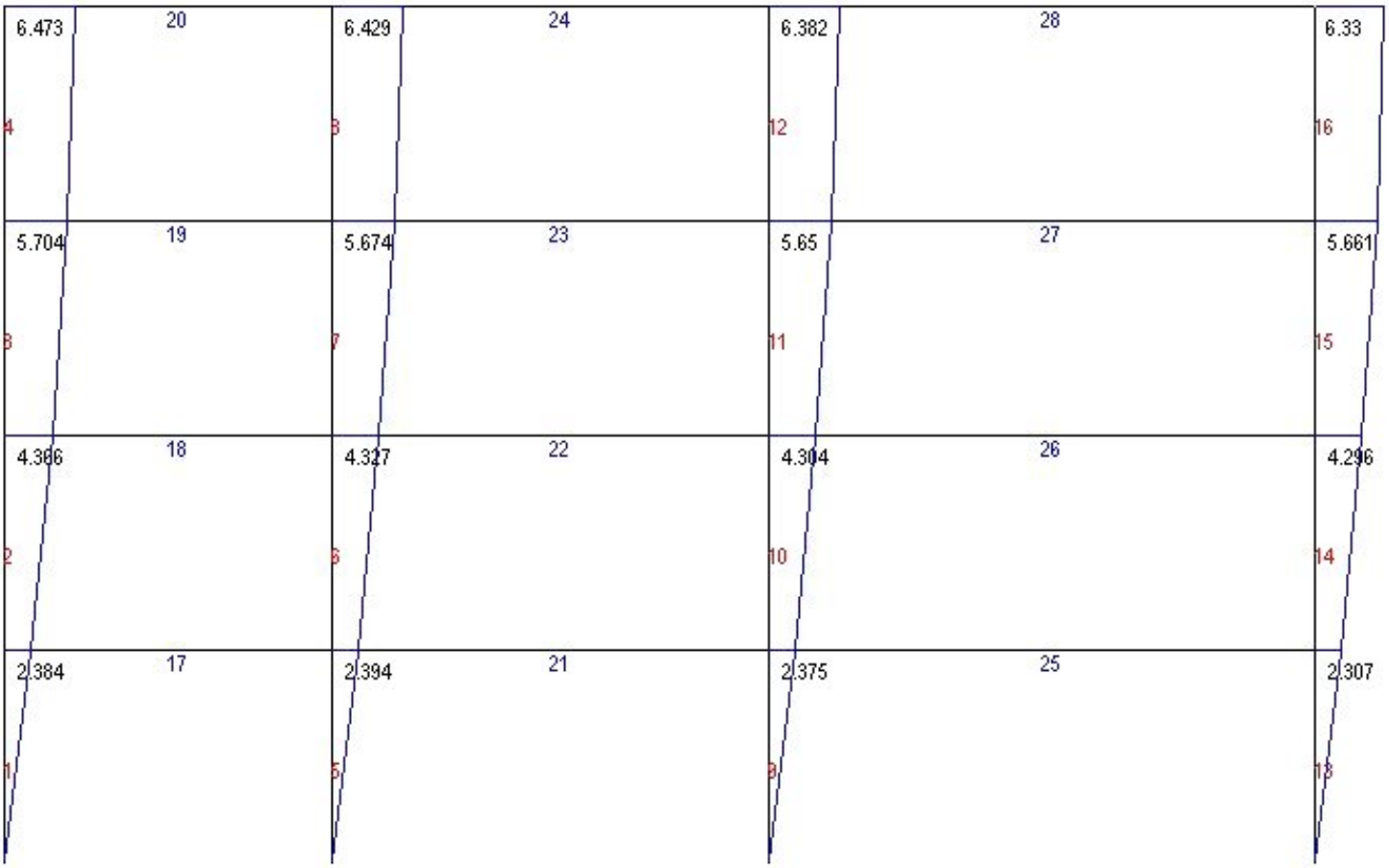
Click SFD OF FRAME - BEAMS. Enter Magnification Factor, say 10. Following window opens up.



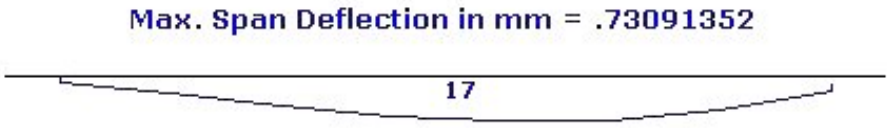
Click SFD OF FRAME - COLUMNS. Enter Magnification Factor, say 10. Following window opens up.



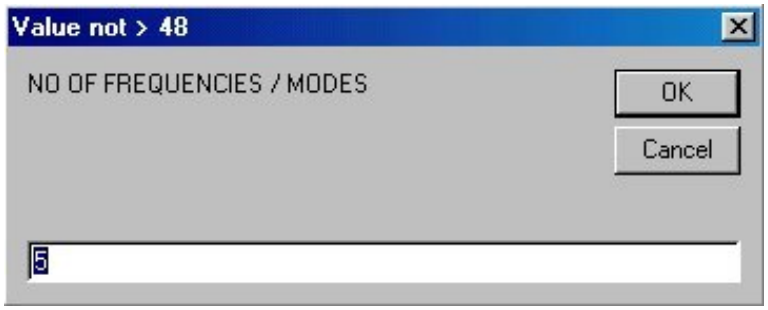
Click LATERAL SWAY OF FRAME. Enter Magnification Factor, say 10. Following window opens up.



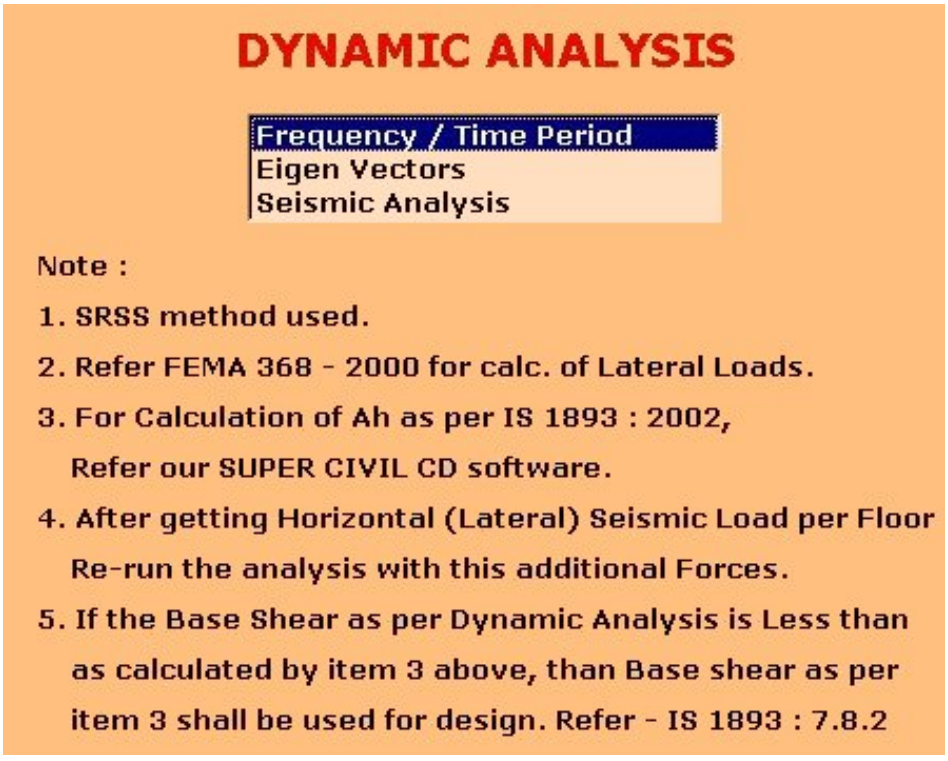
Click **BEAM DEFLECTION**. Enter Beam # as 17. Enter Magnification Factor, say 30. Following window opens up.



Click **DYNAMIC ANALYSIS OF FRAME**. Following window opens up.
Enter # of Frequencies / Mode of Vibration as 5. Click **OK**.
As per **IS:1893 : 2002 Clause 7.8.4.2** the no. of Modes should be such that sum total of Model Masses Under All Modes shall be atleast **90 %** of total Seismic Mass. Missing Mass correction beyond **33 %** shall be applied.
It is better to restrict Frequencies up to **33 Hz** (Cycle /Sec).
If very **High** Modes of Vibration is feed in to the Program, than Program may Generate **ERROR** Like division by ZERO. The Modes shall be kept to a bare **minimum**.



Following window opens up.
Read the NOTE carefully & click on **FREQUENCY / TIME PERIOD**.



Following window opens up.
Read the NOTE carefully & click on **FREQUENCY / TIME PERIOD**.
Eigen Values, Frequency & Time Period for desired Modes are displayed.

Time Period

DISPLAY OF MODE NO / EIGEN VALUE / FREQUANCY / TIME PERIOD IN SEC

File Name : C:\1_2DPortal\01.dat Date : 25/7/07

Mode	Eigen Value	Frequency	Time Period
1	0.00242	3.23212	0.30939
2	0.00029	9.33298	0.10714
3	0.00015	12.75137	0.07842
4	0.0001	15.34229	0.06517
5	0.00006	19.77882	0.05055

➤ **Now Click on Eigen Vectors. Following display contains Eigen Vectors at all Joints & for all the desired Modes of Vibration.**

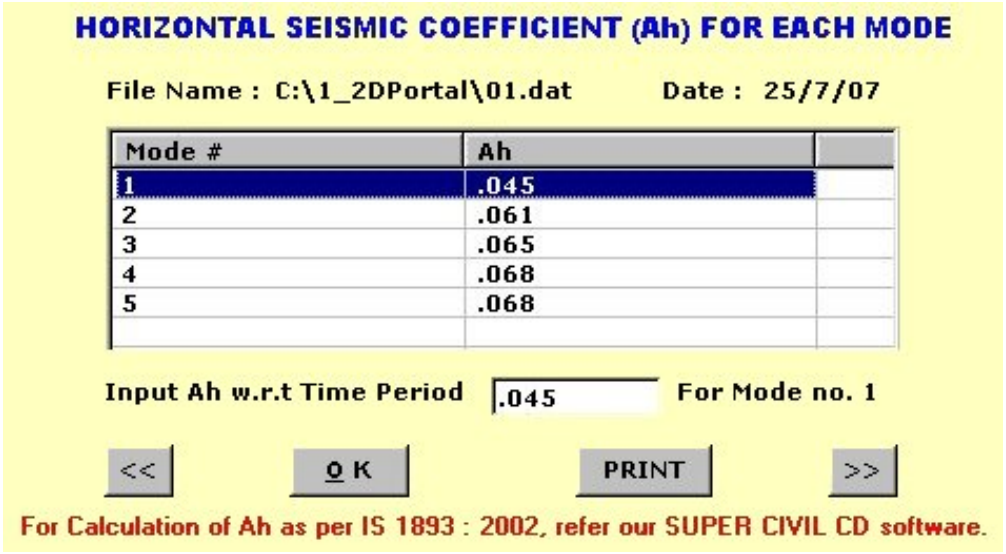
EIGEN VECTORS

File Name : C:\1_2DPortal\01.dat Date : 25/7/07

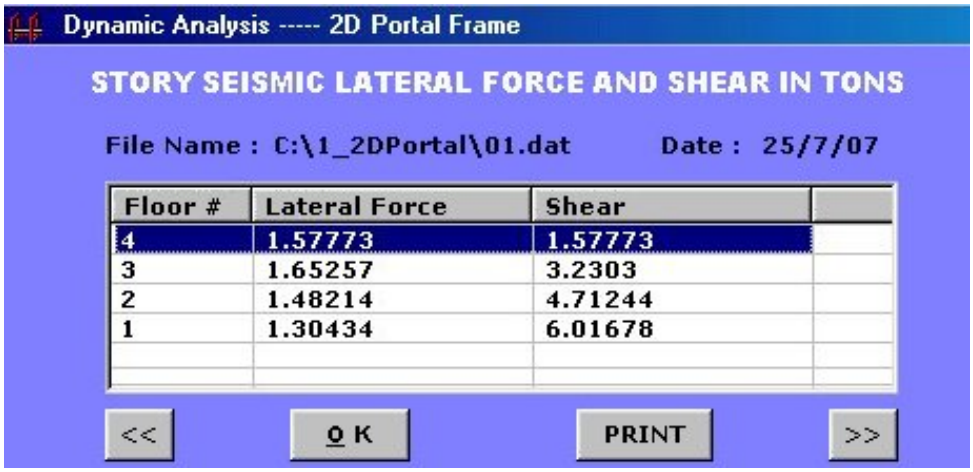
Mode	Joint #	Weight	Horizontal	Vertical	Rotational
1	5	-1.08	0.25205	0.02014	0.08808
1	6	18.3	0.25011	-0.00642	0.06293
1	7	12.16	0.24759	-0.00327	0.07041
1	8	9.09	0.24493	-0.0092	0.09694
1	9	0.88	0.59066	0.03258	0.08213
1	10	12.47	0.58545	-0.00963	0.06154
1	11	12.78	0.58156	-0.00561	0.06598
1	12	7.1	0.57981	-0.01507	0.08741
1	13	3.5	0.8531	0.03819	0.056
1	14	8.24	0.84638	-0.01044	0.04313
1	15	12.56	0.84128	-0.00689	0.0445
1	16	6.42	0.83871	-0.01793	0.05789
1	17	2.06	1	0.0397	0.03299
1	18	3.45	0.99319	-0.01034	0.02196
1	19	9.45	0.98776	-0.00733	0.01827
1	20	8.01	0.98578	-0.0188	0.0299
2	5	-1.08	-0.76256	0.07102	-0.15664
2	6	18.3	-0.70116	0.00022	-0.09102
2	7	12.16	-0.65003	-0.00633	-0.11268
2	8	9.09	-0.62111	-0.01319	-0.16268
2	9	0.88	-0.87293	0.15141	0.13469
2	10	12.47	-0.80926	-0.00955	0.08887
2	11	12.78	-0.75984	-0.01356	0.08106
2	12	7.1	-0.73612	-0.03198	0.10958
2	13	3.5	0.0326	0.2121	0.28566
2	14	8.24	0.03269	-0.01809	0.20134
2	15	12.56	0.0326	-0.0192	0.20023
2	16	6.42	0.03041	-0.04707	0.26042
2	17	2.06	1	0.237	0.22304
2	18	3.45	0.94663	-0.02059	0.13799

<< | O K | PRINT | >>

➤ **Click Seismic Analysis. Enter Values of Horizontal seismic Coefficient as per IS : 1893 : 2002. Click OK button.**

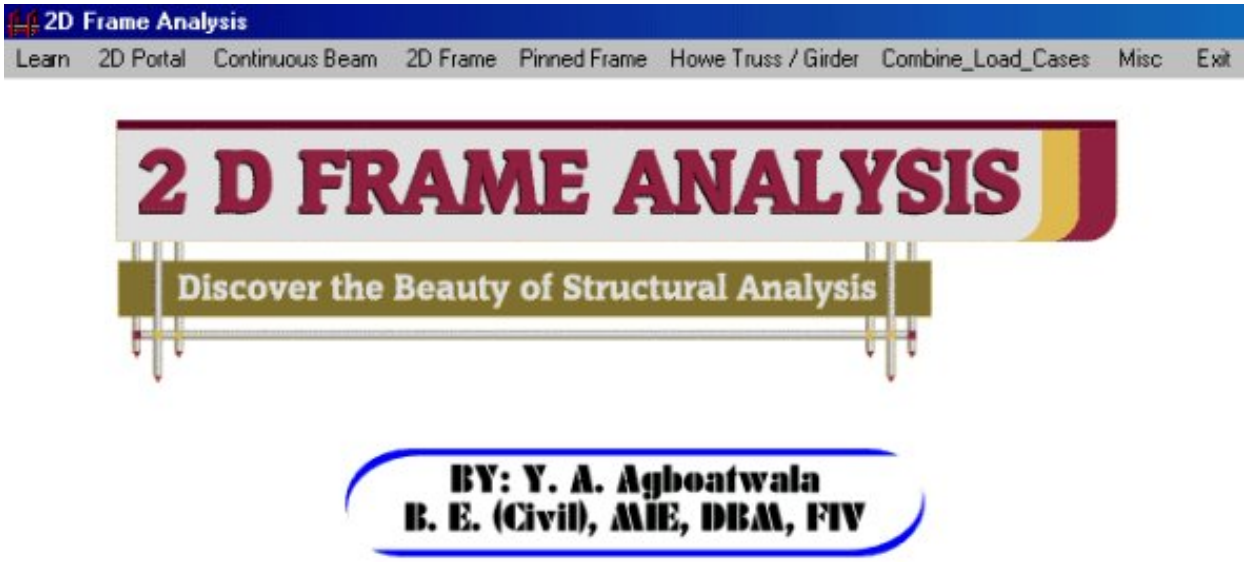


Following window opens up, displaying Lateral Seismic Force at Each Floor Level.
A user should re-run the Analysis after adding these horizontal forces at respective floor Joints.
Please note the Base Shear, if this value is Less Than as calculated by IS:1893-2002, than Increase the Base Shear & corresponding Floor Shears & Lateral Forces by multiplying with Following Factor.
factor = Base Shear as per IS:1893: 2002 ÷ Base Shear as per Dynamic Analysis



Learning of 2D Portal frame Analysis is Over.

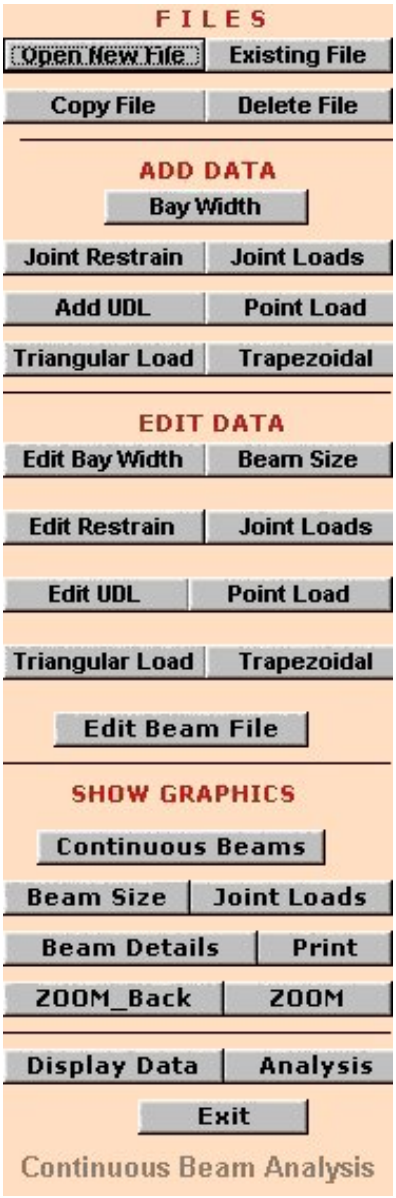
ANALYSIS OF CONTINUOUS BEAMS



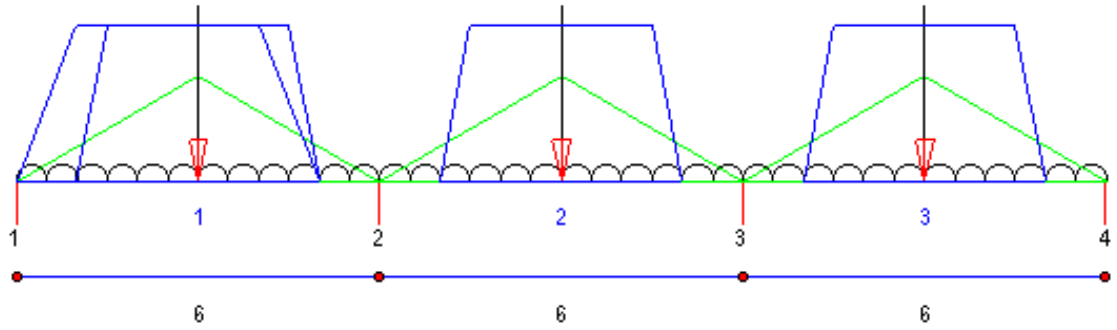
➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

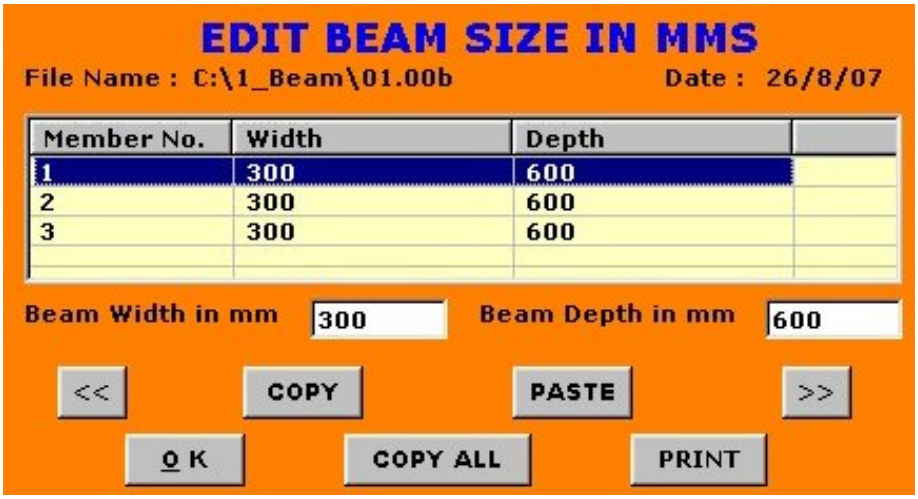
➤ **Click Continuous Beam Option. Following menu will appear.**



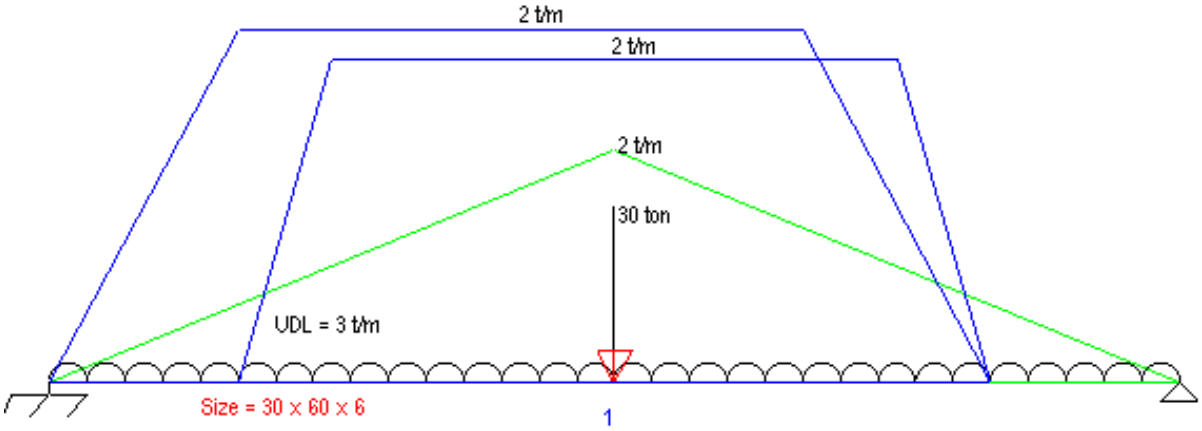
Go through GENERAL INFO : FILE CREATION, COPY & DELETION and "ANALYSIS OF 2D PORTAL FRAMES chapters before starting this option.
Continuous Beam Analysis is similar to 2D Portal Frame Analysis except, there is No STORY HEIGHT parameter. Dynamic Analysis is not envisaged.
We have created a typical 3 - span continuous beam file 01. Refer the Graphics below.



Since we have not covered EDIT option in 2D PORTAL analysis, we will describe it here. Click on BEAM SIZE option. Following window is displayed.

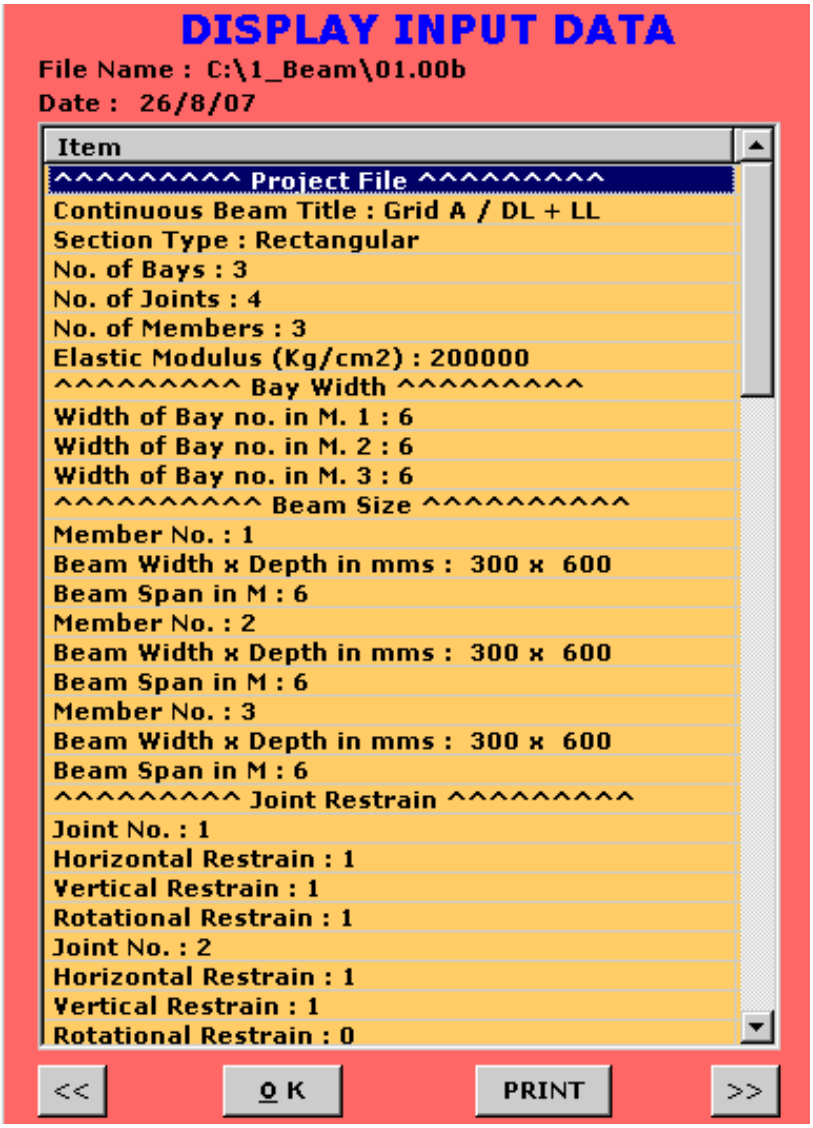


In this option a user can edit / change beam size. If only one record is filled & COPY ALL button is clicked, all records will be filled with the selected record. You can copy one record at a time by clicking COPY than >> (or <<) & PASTE button.
Click OK to exit this option.
Now click BEAM DETAILS option. The Program will ask for Beam Number. Enter Beam no. as 1. Following window is displayed.



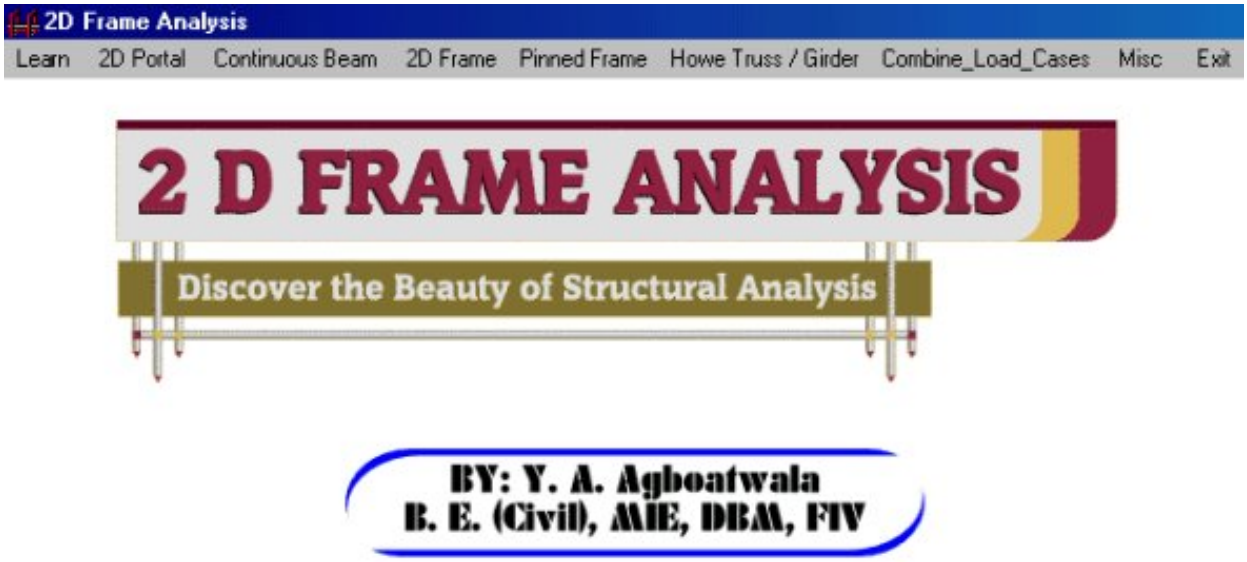
The above graphics displays Beam Loading, Size & Support type.

Click DISPLAY DATA option. This option display the complete INPUT DATA as entered by the user. This is a very useful display, incase all other Graphical Displays are congested & not clear.



Learning of Continuous Beam Analysis is Over.

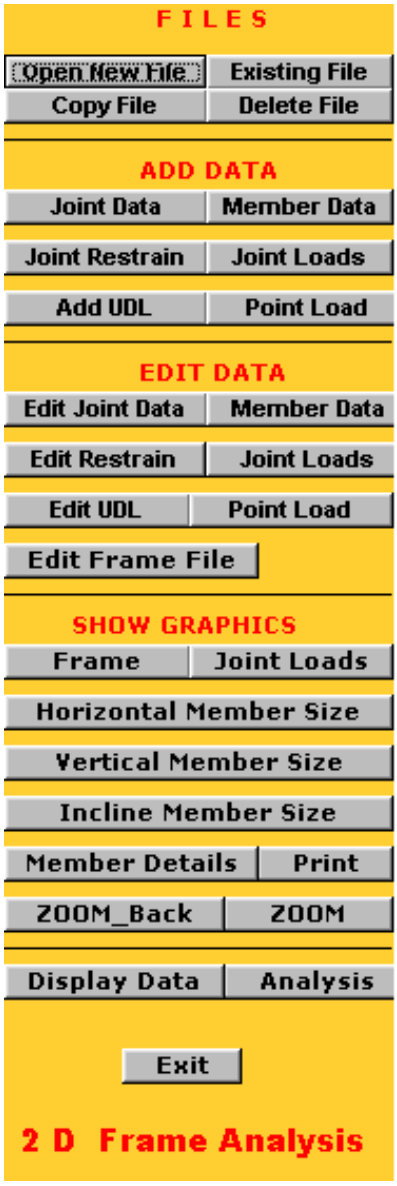
ANALYSIS OF 2D FRAMES - GENERAL



➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

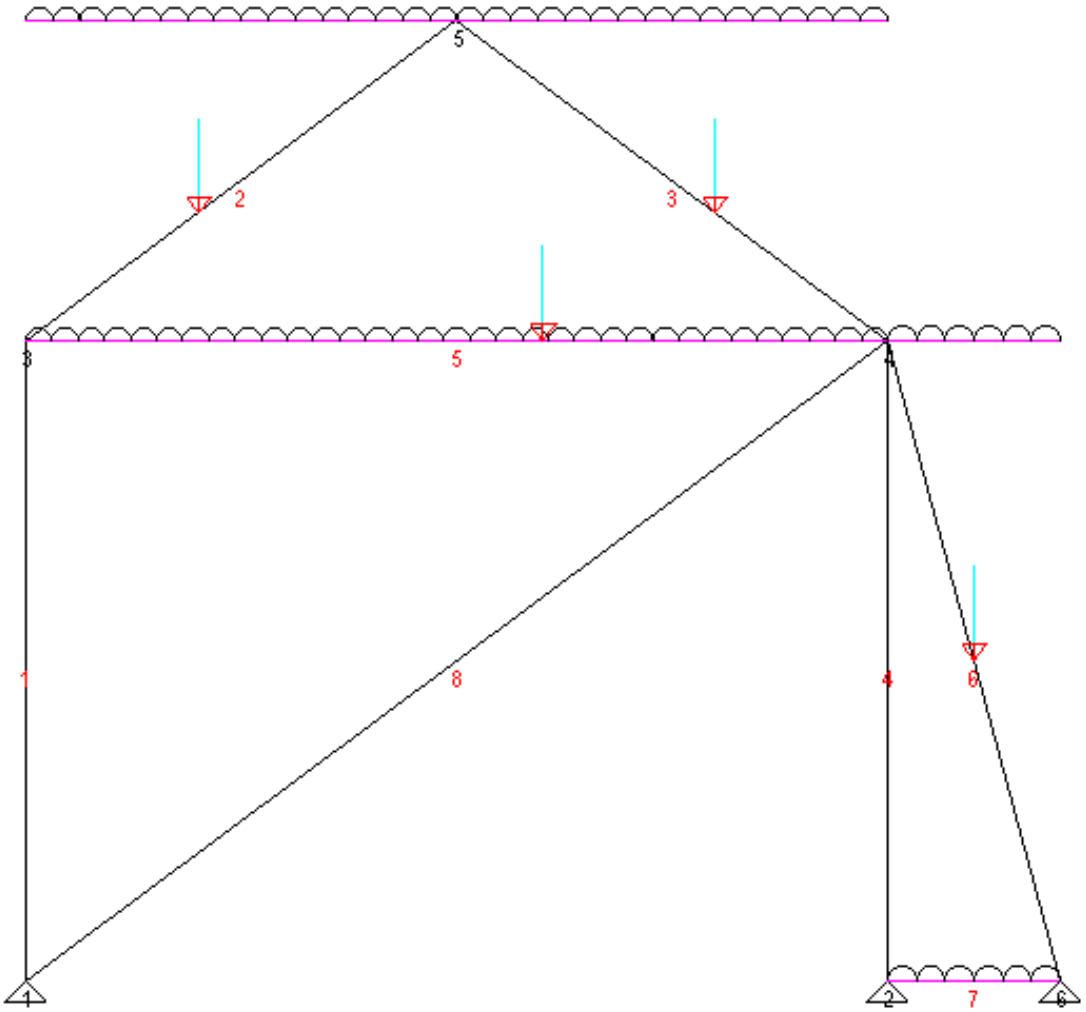
I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

➤ **Click 2D Frame Option. Following menu will appear.**



Go through GENERAL INFO : FILE CREATION, COPY & DELETION, ANALYSIS OF 2D PORTAL FRAMES and CONTINUOUS BEAM ANALYSIS chapters before starting this option.
2D FRAME Analysis is similar to 2D Portal Frame Analysis except, Co_Ordinates & Member Numbers are not generated automatically. A user has to feed Joint Co_Ordinates & Member Numbers.
2D Frame analysis is most General of all the Programs, any Shape can be analyzed. Hence we have given Export Links from other 3 Programs (Portal, Pinned & Howe Truss) to this Program. Following are the Limitations.
Only UDL and Point Loads are allowed.
Dynamic Analysis is not envisaged.
BMD is drawn taking in to account only joint Moments, Span moments are not considered.
Deflection of individual Beam is not Calculated.

We have created a typical 2D Frame File 01. Refer the Graphics below.



The Add Joint C0-Ordinates Display is given as under.
You can use APPEND RECORD button to add more Joints. Similarly Joint Numbers can be removed by just clicking the record (selecting) & pressing REMOVE RECORD button. Click UPDATE button to Re-Write Joint Numbers serially.

ADD JOINT CO-ORDINATES

File Name : C:\1_2DFrame\01.2df Date : 26/8/07

Joint #	X - Co_ordinate	Y - Co_ordinate
1	0	0
2	10	0
3	0	8
4	10	8
5	5	12
6	12	0

Joint No. 2 << O K PRINT >> UPDATE

X Co-Ordinate in M. 10 Y Co-Ordinate in M. 0

Joint No. willl be written serially APPEND RECORD REMOVE RECORD

The Member details are given as follows.

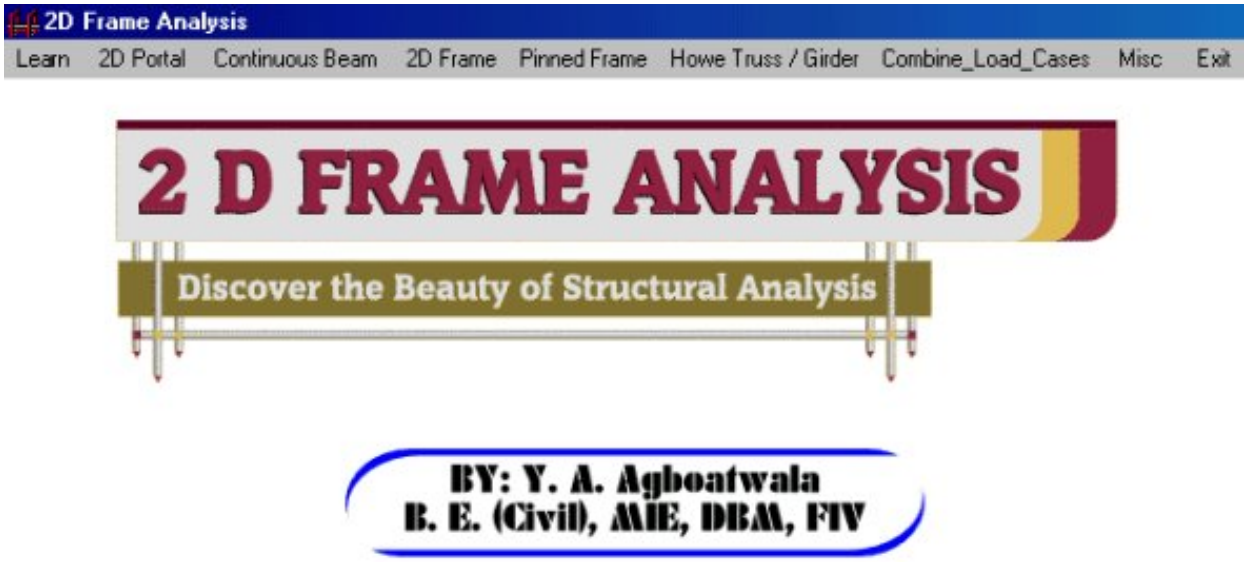
➤ **You can use APPEND RECORD button to add more Members. Similarly Members can be removed by just clicking the record (selecting) & pressing REMOVE RECORD button. Click UPDATE button to Re-Write Members serially.**

Incase you have edited Joints OR Member Numbers :
If you have Added or Removed any Joint, Edit the Relevant Member using Member Option, if required.
If you have Added or Removed any Member, Edit the corresponding Joint using Joint Option, if required.

➤ **All other Options are Same as 2D Portal Frame Analysis.**
Learning of 2D Frame Analysis is Over.

LEARN 2D FRAME ANALYSIS STEP BY STEP

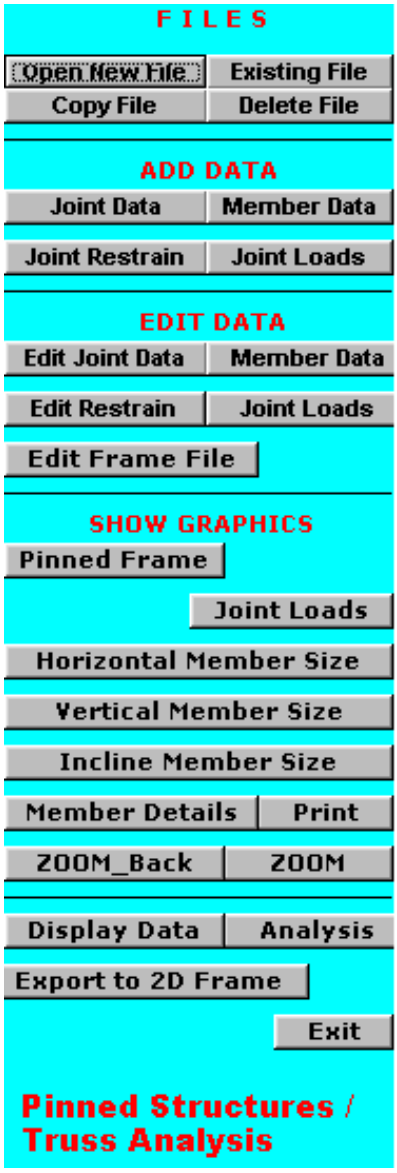
ANALYSIS OF 2D PINNED FRAMES / TRUSSES - GENERAL



➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

➤ **Click PINNED FRAME Option. Following menu will appear.**

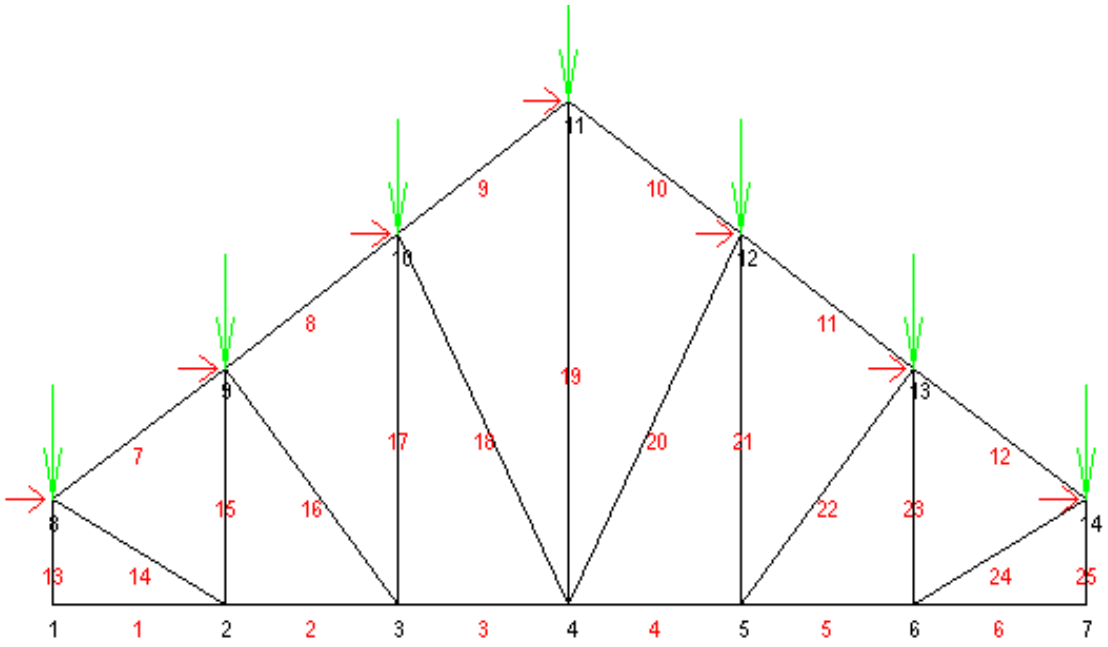


Go through GENERAL INFO : FILE CREATION, COPY & DELETION, ANALYSIS OF 2D PORTAL FRAMES, CONTINUOUS BEAM ANALYSIS and 2D FRAME ANALYSIS chapters before starting this option.

2D PINNED / TRUSS Analysis is similar to 2D Frame Analysis except the following.

- **All Joints are considered as Hinged & not Rigid, as the case with 2D FRAME analysis.**
- **In Member details only "AREA" is to be given.**
- **Joints Loads can be Horizontal OR Vertical. No Applied Moments allowed.**
- **Support Restrains could be Hinged or Roller.**
- **Member Loads are not allowed.**
- **Dynamic Analysis is not envisaged.**

We have created a typical 2D Frame File 01. Refer the Graphics below.



The Add Joint Loads Display is given as under.
You can use APPEND RECORD button to add more Joints. Similarly Joint Numbers & its Loads can be removed by just clicking the record (selecting) & pressing REMOVE RECORD button.
Note that Vertical Loads are given with (-) Negative sign, as they are acting Downwards.

ADD JOINT LOADS IN TON

File Name : C:\1_Pinned\01.2dp Date : 26/8/07

Joint No.	Horizontal	Vertical
8	0.01	-.35
9	0.01	-.35
10	0.01	-.35
11	0.01	-.35
12	0.01	-.35
13	0.01	-.35
14	0.01	-.35

APPEND RECORD Joint No 8

Horizontal Load in Ton 0.01 + ve if acting towards right

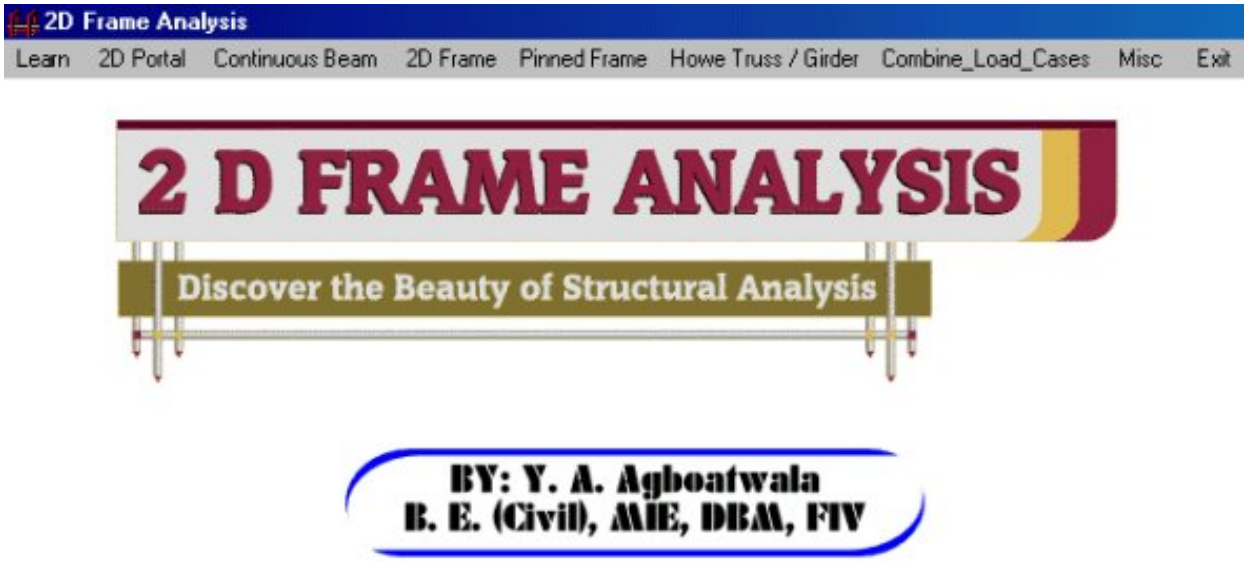
Vertical Load in Ton -.35 + ve if acting upwards

<< | O K | COPY ALL | PRINT | >> | REMOVE RECORD

Incase you have edited Joints OR Member Numbers :
If you have Added or Removed any Joint, Edit the Relevant Member using Member Option, if required.
If you have Added or Removed any Member, Edit the corresponding Joint using Joint Option, if required.

All other Options are Same as 2D General Frame Analysis.
Learning of 2D Pinned Frame Analysis is Over.

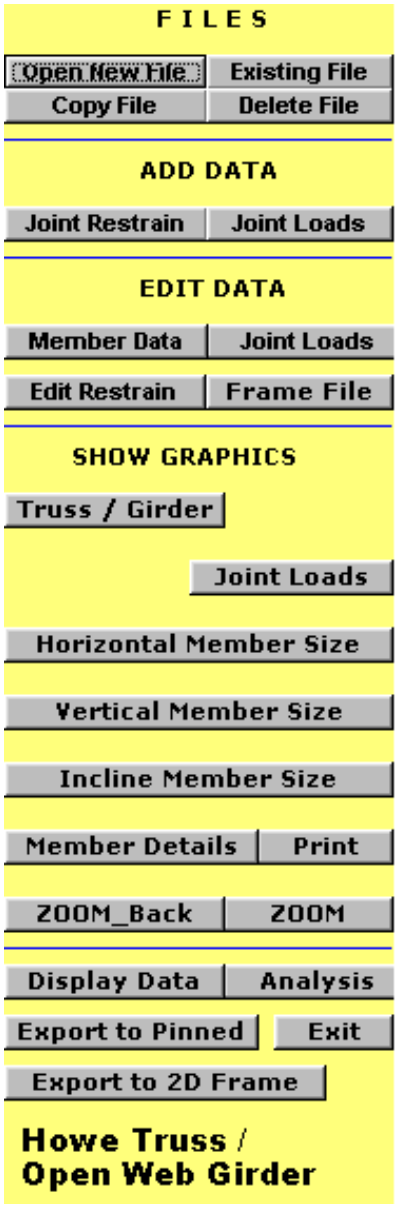
ANALYSIS OF 2D HOWE TRUSS / OPEN WEB GIRDER



➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

➤ **Click HOWE TRUSS / GIRDER Option. Following menu will appear.**



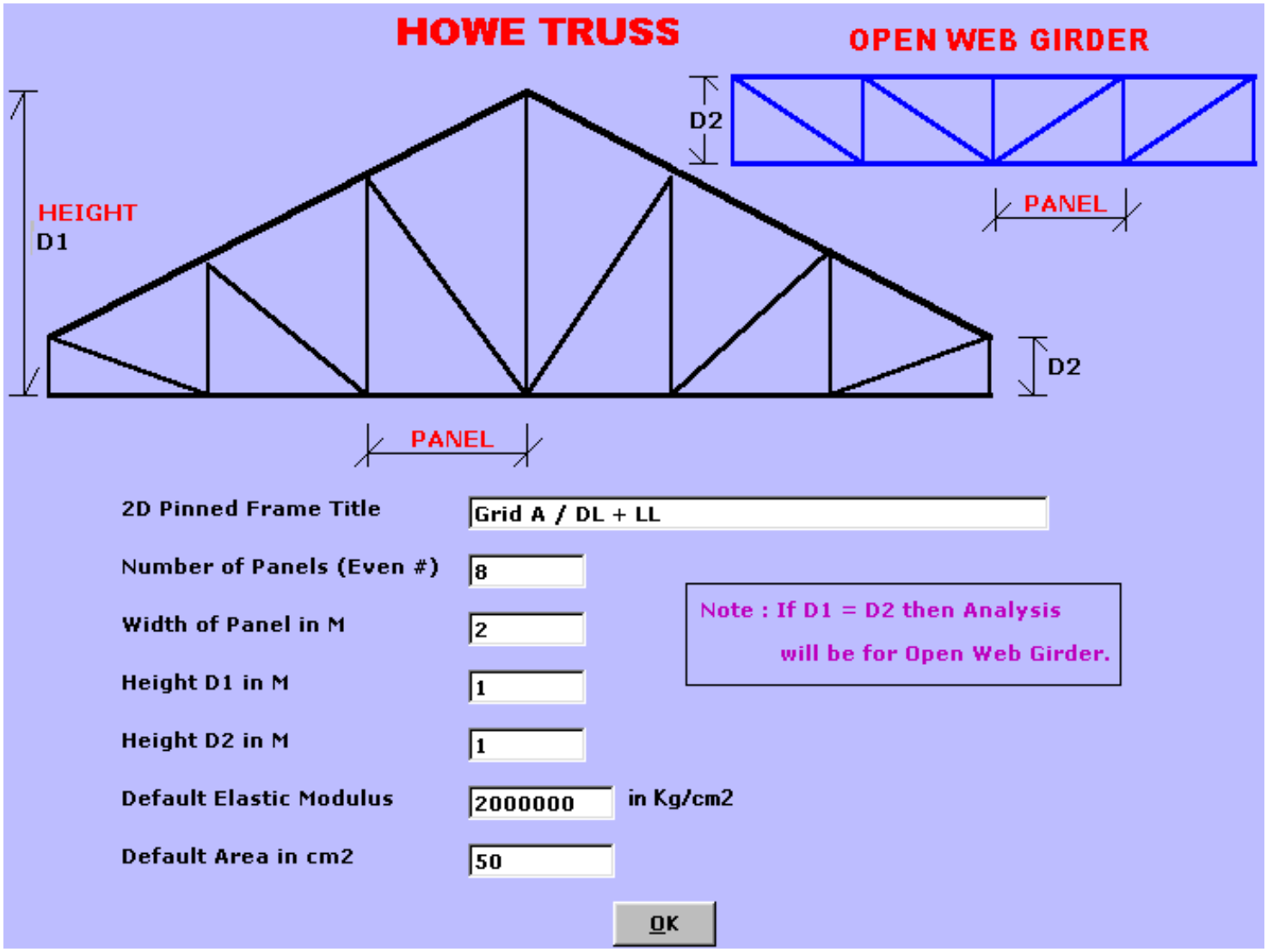
Go through GENERAL INFO : FILE CREATION, COPY & DELETION, ANALYSIS OF 2D PORTAL FRAMES, CONTINUOUS BEAM ANALYSIS, 2D FRAME ANALYSIS and PINNED FRAME chapters before starting this option.

2D HOWE TRUSS / OPEN WEB GIRDER is a special case of General - 2D Pinned Frame Analysis. The Joint Co-Ordinates & Member Numbers are automatically generated by just giving No. of Panels & Height of Howe Truss / Girder. No need to feed manually Joint Co-Ordinates & Member Numbers.

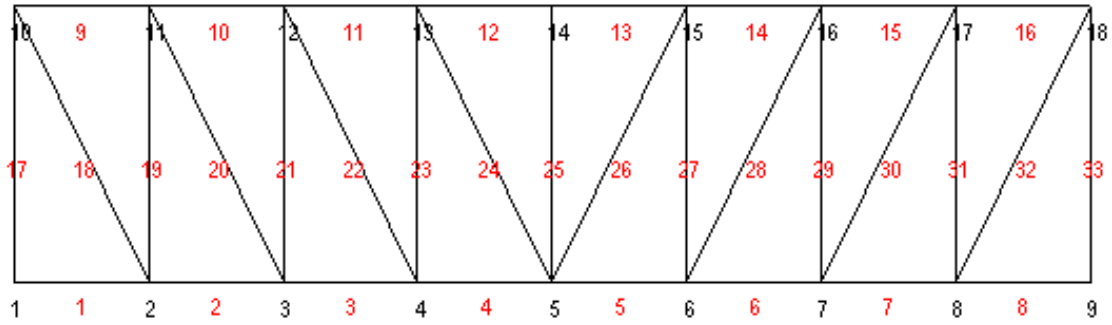
Click OPEN NEW FILES under FILES menu, a new window will open as follows.

Note that Open Web Girder is a special case of Howe Truss when D1 = D2.

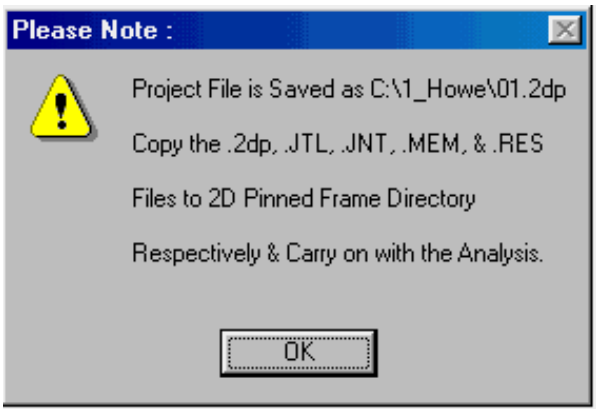
Enter the data as given below. The File name is 01.



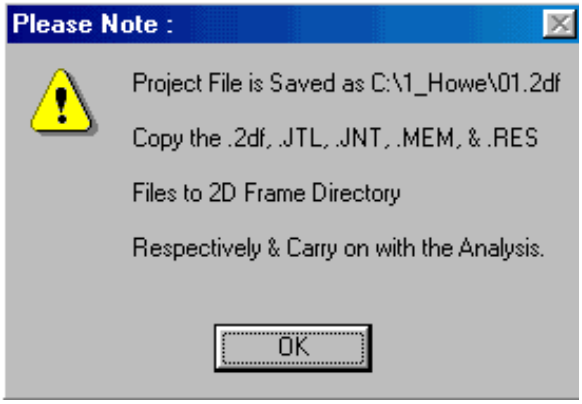
Click TRUSS / GIRDER button under SHOW GRAPHICS Menu. Following Graphics is displayed. Since we have given D1 = D2 = 1.0, Open web girder is displayed. We have already covered HOWE TRUSS under PINNED FRAME chapter.



Now Click EXPORT TO PINNED FRAME Program under SHOW GRAPHICS menu. Following message is displayed. The files are created in the same (Howe Directory) Folder. A user has to manually copy the following files to PINNED Frame directory. Once these files are re-created by Pinned Frame program you can add more Joints / Members / Loads etc. & Analyze the structure.

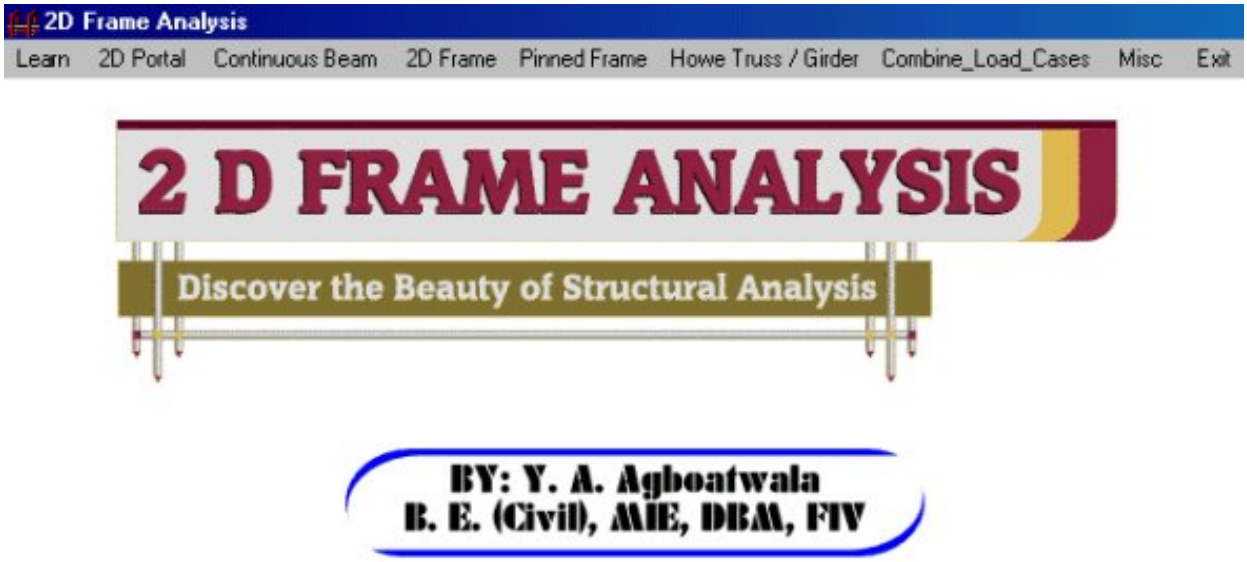


Now Click EXPORT TO 2D FRAME Program under SHOW GRAPHICS menu. Following message is displayed. The files are created in the same (Howe Directory) Folder. A user has to manually copy the following files to 2D Frame directory. Once these files are re-created by 2D Frame General program you can add more Joints / Members / Loads etc. & Analyze the structure. Note that in 2D Frame all joints are RIGID, moreover a user can ADD Member Loads Like UDL and Point Loads.



All other Options are Same as 2D Pinned Frame Analysis.
Learning of Howe Truss / Open Web Girder Analysis is Over.

COMBINE LOAD CASES



➤ **When Program starts, the graphics above is displayed. The Menu bar contains following options.**

I. Learn
II. 2D Portal
III. Continuous Beam
IV. 2D Frame
V. Pinned Frame
VI. Howe Truss / Girder
VII. Combine Load Cases
II X. Misc.
IX. Exit

➤ **Click COMBINE_LOAD_CASES Option, following is displayed.**
Note that "Number of Load Cases" is always shown as "1" in the Project File Creation. A user should create another file by copying the existing main file & change its loading. This will act as another Load Case. Thus any number of Load Case files can be created. After all the load case files are analyzed, use Combine_Load_Cases option to Add (Combine) & Sort the analysis Results (Direct Force, Shear & BM) in Ascending OR Descending order.
We have created 2 Load Case Files.
1. File 01 -----> Load Case No. 1
2. File 01X ---> Load Case No. 2
Both Files are from the Same Structure, but having different Loadings & Analysis Results. Before Combining Analysis must be performed. File Extensions .DAT are Same for Both. Different File Extensions Cannot be Added. You cannot add PORTAL TO 2D FRAME OR PINNED FRAME. Apples to Apples, Oranges to Oranges.

Click OK Button. Combined Analysis Results are displayed as under.
To Sort any COLUMN, Just Click Its Heading. Sorting will be done in Ascending or Descending order, as well as for Negative Results.

MEMBER FORCES IN T / T-M

Note : To Sort Column, Just Click its Header.

No.	Near End	Axial	Shear	BM	Far End	Axial	Shear	BM	Span BM	Dist
1	1	1.387	7.76	13.812	5	-1.388	-7.761	7.914		
2	5	3.457	1.572	1.029	9	-3.458	-1.573	3.372		
3	9	3.573	1.18	1.234	13	-3.574	-1.181	2.069		
4	13	1.069	-0.206	-0.795	17	-1.07	.205	0.219		
5	2	42.479	2.036	0	6	-42.48	-2.037	5.701		
6	6	24.172	4.913	7.115	10	-24.173	-4.914	6.641		
7	10	11.699	2.369	2.651	14	-11.7	-2.37	3.983		
8	14	3.455	0.913	0.995	18	-3.456	-.914	1.563		
9	3	46.969	1.765	0	7	-46.97	-1.766	4.944		
10	7	34.805	5.022	6.753	11	-34.806	-5.023	7.309		
11	11	22.018	3.547	4.325	15	-22.019	-3.548	5.608		
12	15	9.454	0.572	1.139	19	-9.455	-.573	0.464		
13	4	30.653	8.437	14.231	8	-30.654	-8.438	9.393		
14	8	21.554	3.492	3.903	12	-21.555	-3.493	5.875		
15	12	14.448	2.902	3.604	16	-14.449	-2.903	4.522		
16	16	8.02	3.718	3.364	20	-8.021	-3.719	7.048		
17	5	-1.188	-1.07	-7.944	6	1.187	13.569	-12.265		
18	9	4.607	0.883	-3.608	10	-4.608	8.366	-8.305	3.78	.371
19	13	3.614	3.504	-0.276	14	-3.615	6.495	-4.713	2.279	1
20	17	5.205	2.069	0.78	18	-5.206	1.93	-1.072	0.791	1.06
21	6	1.688	4.737	-0.552	7	-1.689	8.762	-10.75	4.028	1.22
22	10	2.064	4.106	-0.989	11	-2.065	9.893	-10.584	3.799	1.36
23	14	2.158	1.747	-0.267	15	-2.159	8.992	-7.496	1.793	1.74
24	18	4.291	1.524	-0.492	19	-4.292	3.475	-4.41	1.516	1
25	7	4.945	3.4	-0.949	8	-4.946	9.099	-13.297	3.925	1.61
26	11	0.59	2.893	-1.051	12	-0.591	7.106	-9.481	3.144	1.44
27	15	-0.817	3.572	0.747	16	0.816	6.427	-7.887	2.442	1.78
28	19	3.718	5.979	3.945	20	-3.719	8.02	-7.049	4.993	2.98
1	1	18.645	-0.831	-0.841	5	-18.646	.83	-1.487		
2	5	13.255	-1.029	-1.522	9	-13.256	1.028	-1.359		
3	9	8.244	-1.053	-1.414	13	-8.245	1.052	-1.533		
4	13	2.573	-1.16	-1.635	17	-2.574	1.159	-1.614		

This completes our Combing of Load Cases.
The MISC. option in the main Menu is Self Explanatory.

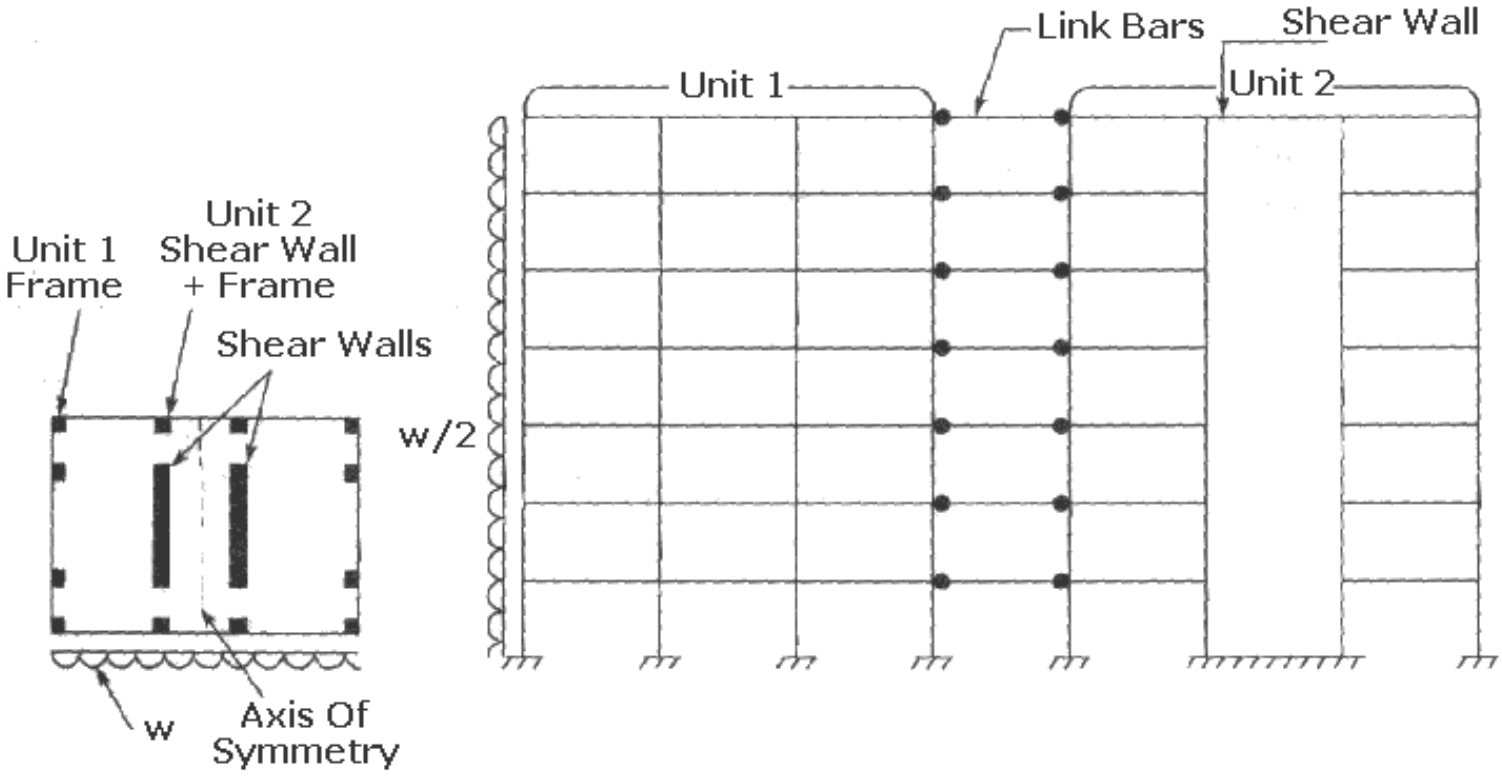
TIPS

- **In Case of 2D Frame Analysis all joints are Rigid. Suppose you want to make a particular Member Pinned (Hinged) at both the ends, than simply Change the Moment of Inertia (Ixx) to a very small value (say 0.001, Not Zero). Now when analysis is performed, Both Joint Moments will be Zero or Negligible. This is very Helpful in a 2D Frame, when you want to incorporate CROSS BRACINGS in a particular Bay.**

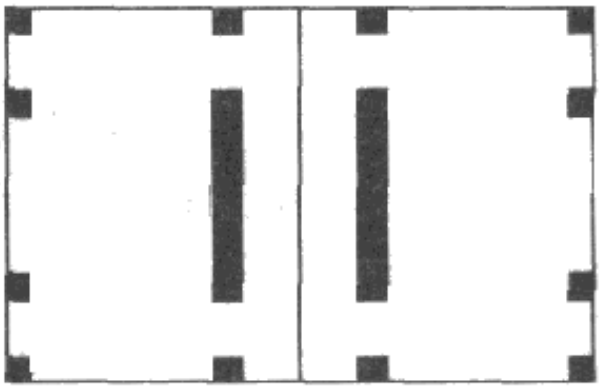
- **Suppose we want to Release a Member for its AXIAL STRAIN. Just Change AREA to a very small value (say 0.001, Not Zero). Now when analysis is performed, Both Joint AXIAL LOADS will be Zero or Negligible. This is very Helpful in a Frame, when you want to incorporate the effect of Thermal Expansion or Contraction of Members.**

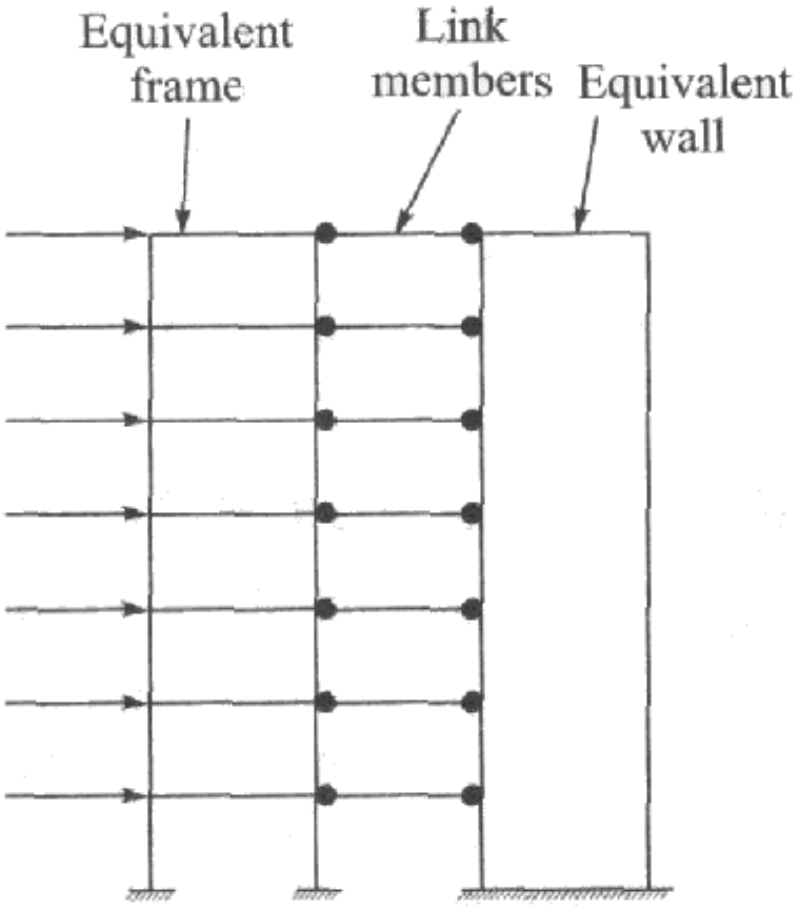
CONVERT 3D FRAME TO 2D FRAME ANALYSIS

When the plan consists of combination of parallel frames and frame-shear walls, then single idealized plane frame model of all such frames can be represented as shown in Graphics Below. Different units, such as frame 1, frame 2, etc., represent the lateral resisting frame along each line. These units are then connected at story levels by rigid links, (Give Very High Area & Negligible Ixx) which simulate the in-plane rigidity of floors. The finite width of core and wall are taken into account as represented by beam with rigid ends. This type of plane frame model can be analyzed by our 2D Portal Frame Program.

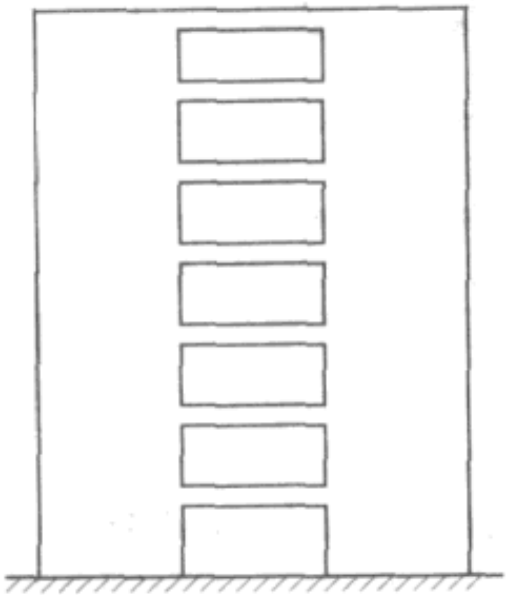


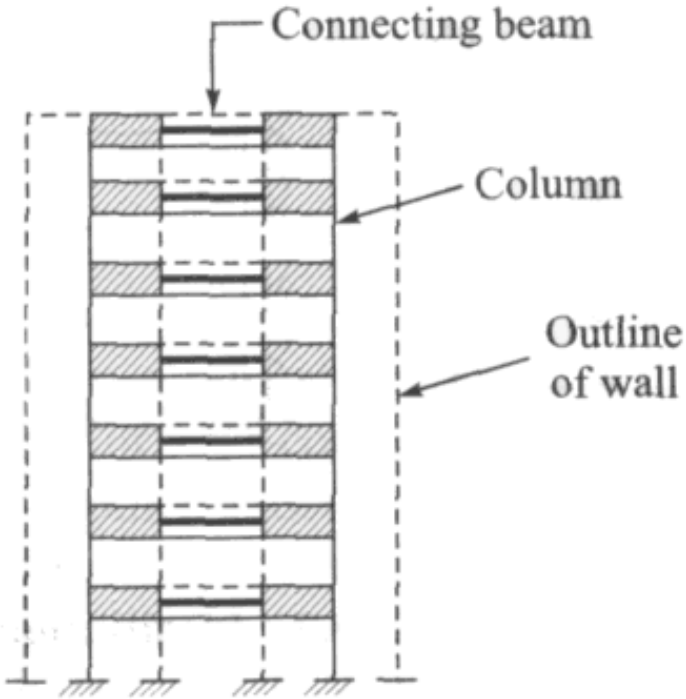
- **In order to analyze the building with parallel frames in plan as shown in Graphics below, an equivalent shear wall-frame model is established. The equivalent frame is obtained by lumping together all the frames into one bay equivalent frame, and combining all shear walls into an equivalent shear wall. This equivalent frame-shear wall system is analyzed for total lateral loads on the building in the particular direction. Subsequently the forces computed in the equivalent frame are distributed to the component elements from which the equivalent frame was composed in proportion to the lateral stiffness.**





Plane Frame Model of Coupled Shear Walls:
The multi-storeyed shear walls with openings are called coupled shear walls, these can be idealized by a frame with finite joints. The coupled wall is thus represented as a frame except that the finite width of the columns in comparison with the beam is recognized. A typical representation of coupled wall by a frame model is shown in Graphics below.





Modelling of coupled shear wall by a plane frame

Refer Our SUPER CIVIL CD Software for the Following :

DUCTILE DETAILING AS PER IS: 13920 of

- **Beams**
- **Column**
- **Coupling Beam**
- **Shear Wall With Flange**
- **Shear Wall Without Flange**
- **Cantilever Shear Wall**
- **Time Period & Seismic Base Shear As Per IS:1893-02**
- **Time Period & Seismic Base Shear As Per UBC**

This completes our Tips.

OTHER SOFTWARES:

SUPER CIVIL CD - Single Point Solution To Your Civil Engineering Needs

SUPER RATE ANALYSIS - Rate Analysis Of 1299 Nos. Of Civil Engineering Items

R C F - A Software for Analysis, Design, Estimation & Costing of RCC Floors

S S F - Analysis, Design, Estimation & Costing of Steel Buildings, revised as per IS 800 : 2007

Q T Y - Quantity Estimation & Cost, Project Control

SUPER REAL VALUATION - A Software For Immovable Properties

ROADS - Pavement Design & Rate Analysis Of Road Items

ROAD ESTIMATE - Quantity Estimation & Cost, Project Control For Road

ELECTRIC COST - Costing, Project Control & MDS For Electrical Projects

HVAC COST - Costing, Project Control & Design For HVAC Engineers

BILLING JI - A Database Management Software For General Billing

RA BILL - A Database Management Software For Item Rate Contract Billing

BUILDERS BILL - A Database Management Software for Billing of Lump sum Contracts

BID ANALYSIS - A Software For Technical & Commercial Tender Analysis

RAFT FOUNDATION - Analysis, Design, Estimation, Costing & Drawing of RCC Raft Foundation

STEEL 2007 - Limit State design of Steel as per IS 800 : 2007

SITE CONTROL - A Management Software for Resource Control At Site.

DESIGN & DRAWING CONTROL - A DBM Software for Control of Design & Drawing Manhours.

COMPOSITE - A Software for Analysis, Design, Costing & Drawing of Composite Floor Buildings

INSTA COST - A Software for Estimating Project Cost & Tender SOQ Instantly

FLAT SLAB - A Software for Analysis, Design, Estimation, Costing & Drawings of Flat Slabs

FLAT RAFT - A Software for Analysis, Design, Estimation, Costing & Drawings of Rigid RCC Flat Rafts

OPTIMIZE BAR - A Software for Optimization of Reinforcements from Existing Bar Bending Schedule

OPTIMIZE STEEL - A Software for Optimization of Steel Sections from Existing Fabrication Drawing

AutoQty - A Software for Automatic Quantity & Cost Estimation from AutoCAD Drawings