LEARN COMPOSITE

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

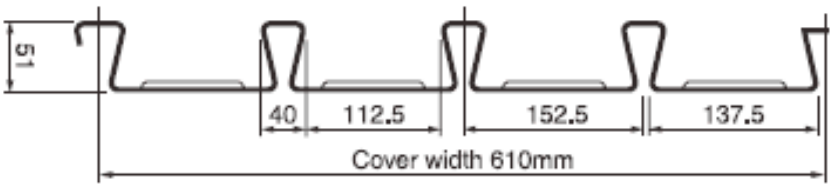
By :
Y.A. Agboatwala &
Fatima.Y. Agboatwala
1802, Jamuna Amrut,
219, Patel Estate, S.V.Road,
Jogeshwari(W), Mumbai - 400102
Phone: 09820792254 , (022) 26783525
Url: www.supercivilcd.com,
www.agboatwala.com
Email : yaa@supercivilcd.com,
yaa@agboatwala.com,
supercivilcd@gmail.com

LEARN COMPOSITE

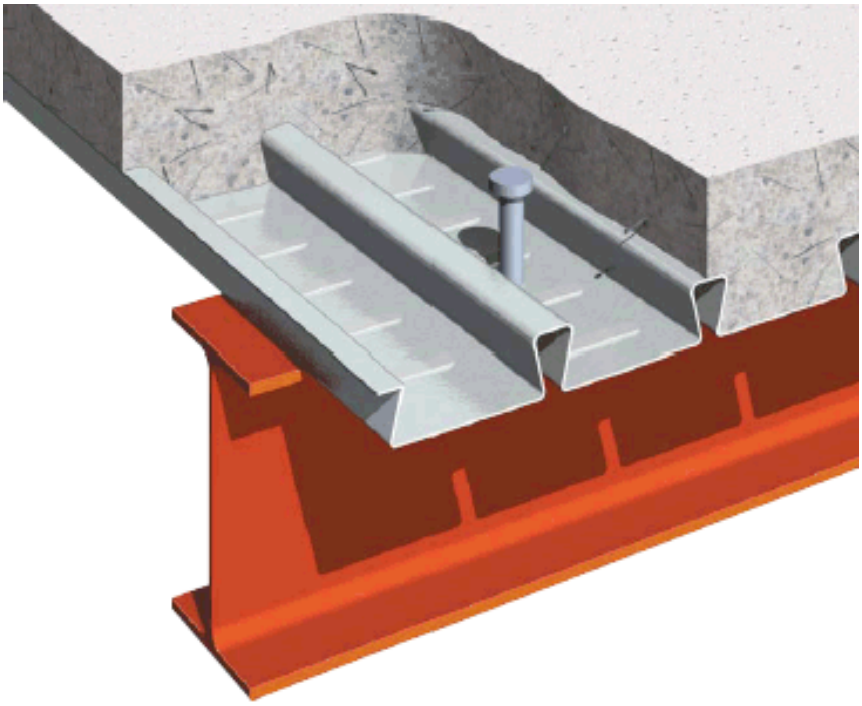
A Software for Analysis, Design, Estimation, Costing and AutoCAD Drawing of Multistory Composite Steel Buildings As per Euro 4 & IS 800 : 2007

DECK PROFILE DETAILS

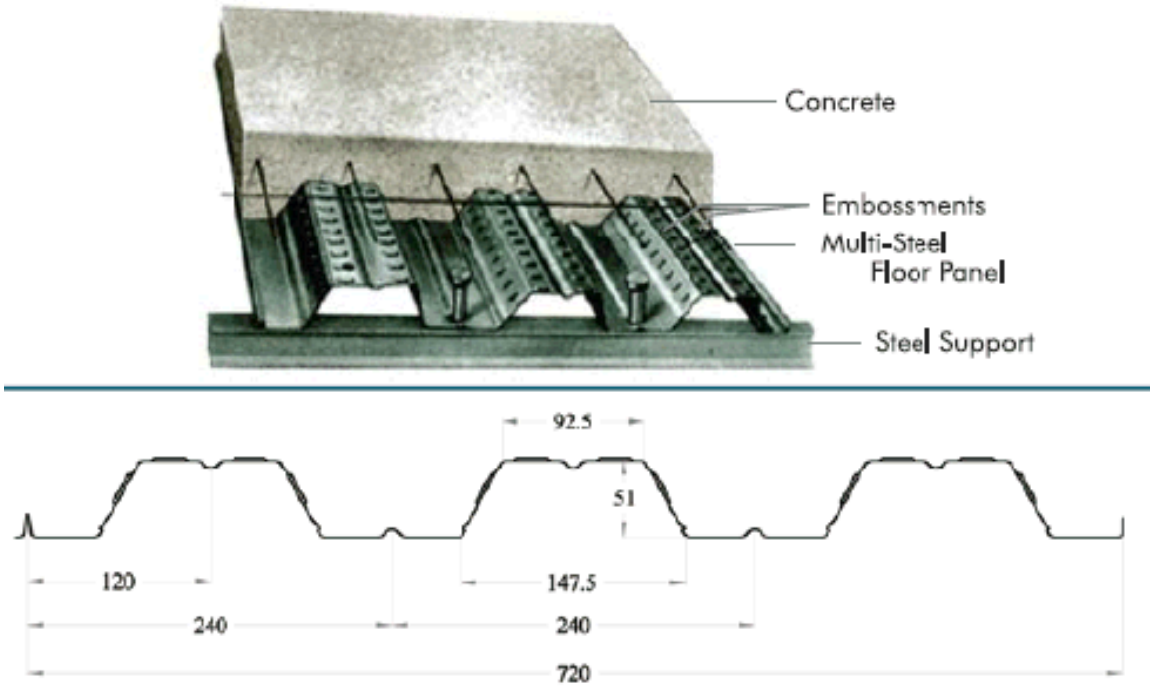
RE-ENTRANT DECK PROFILE



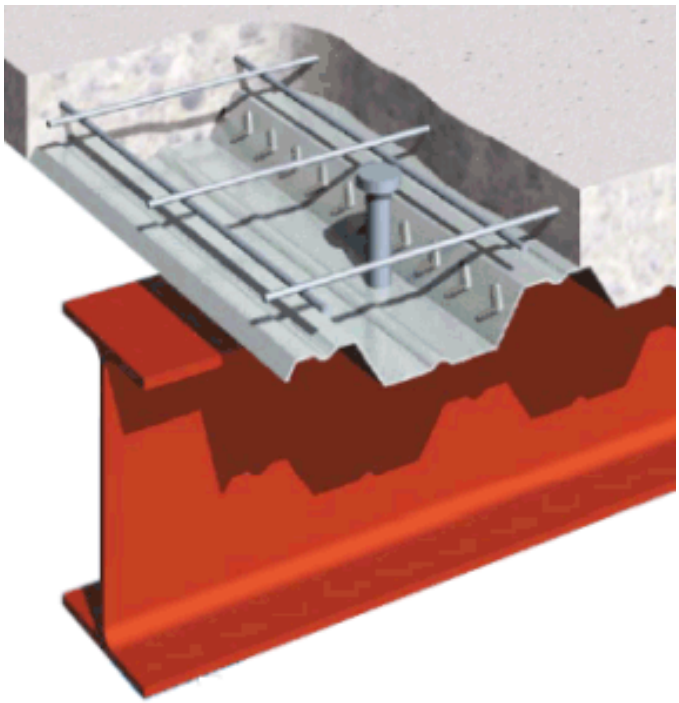
SINGLE STUD WITH RE-ENTRANT PROFILE



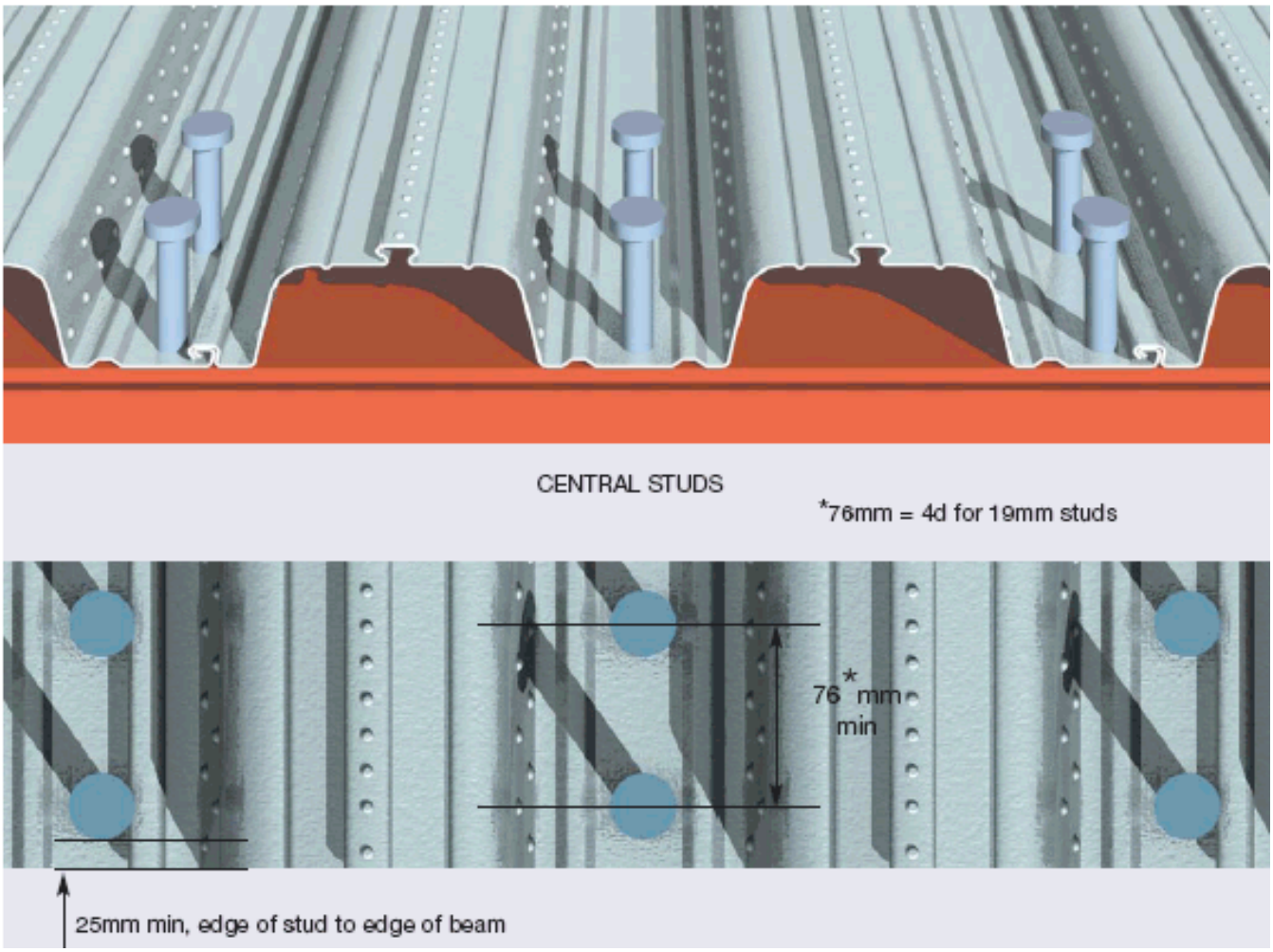
TRAPEZOIDAL DECK PROFILE



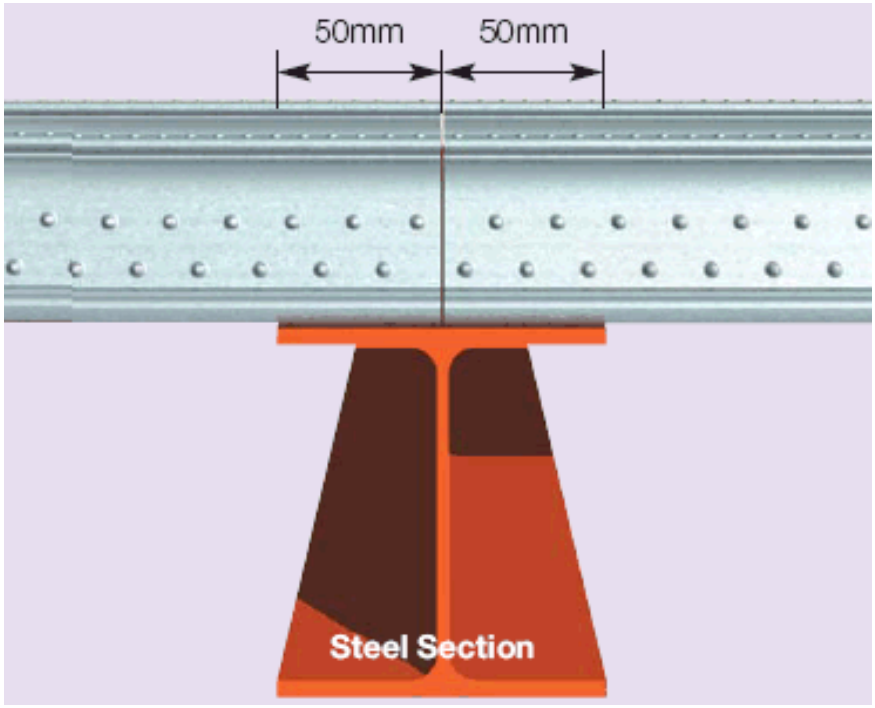
SINGLE STUD WITH TRAPEZOIDAL PROFILE



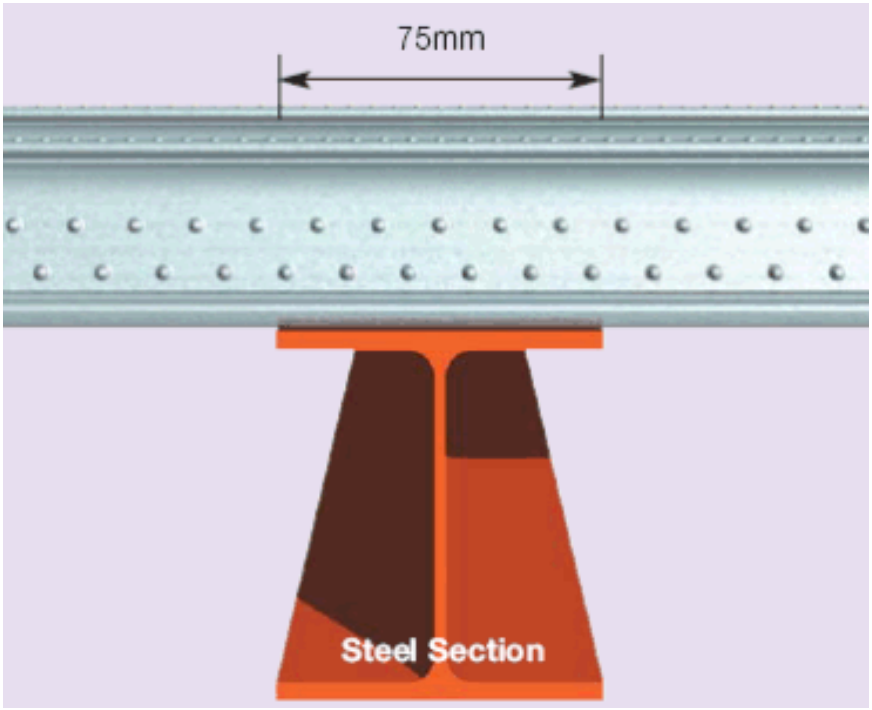
DOUBLE STUD WITH TRAPEZOIDAL PROFILE



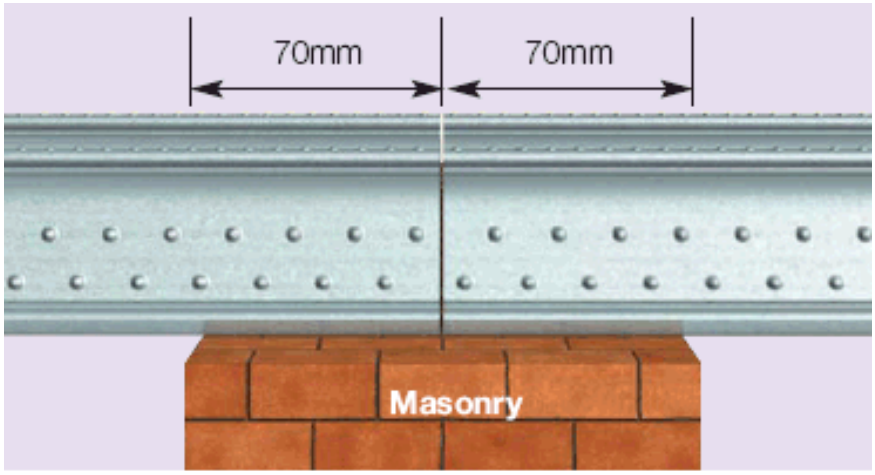
BEARING ON STEEL : DISCONTINUOUS PROFILE



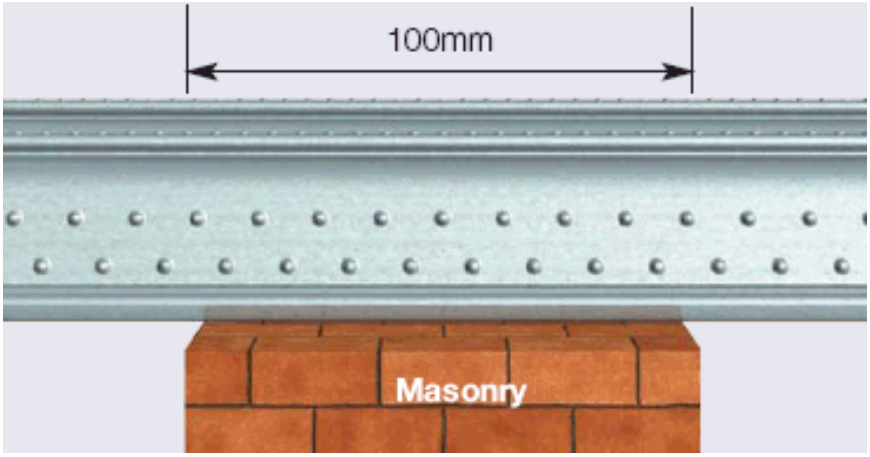
BEARING ON STEEL : CONTINUOUS PROFILE



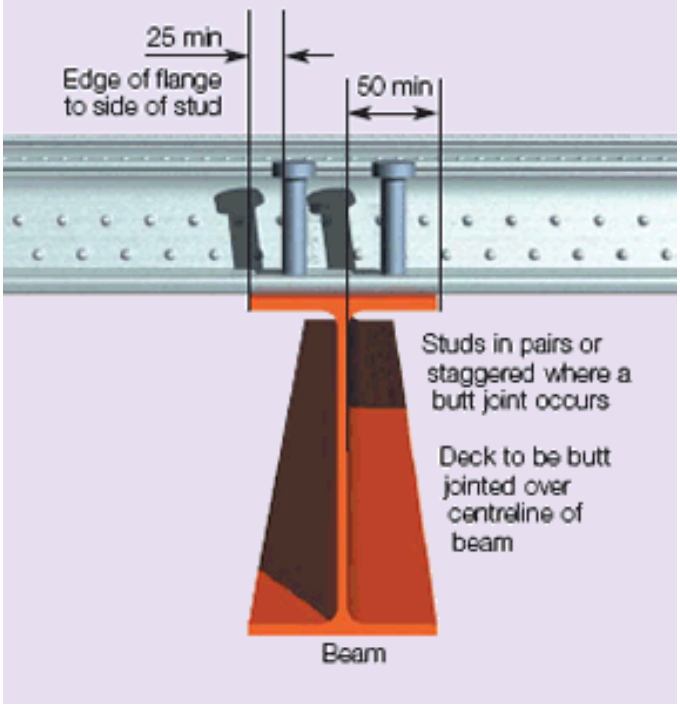
BEARING ON MASONRY : DISCONTINUOUS PROFILE



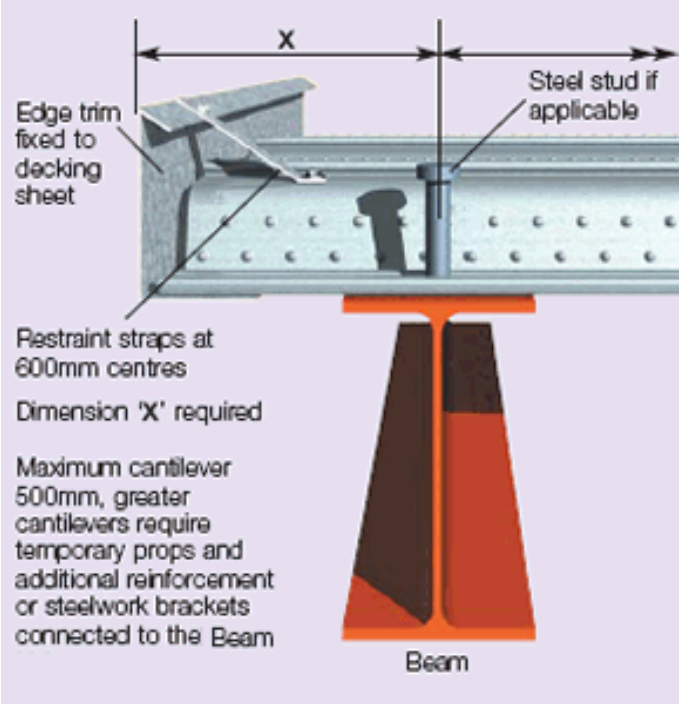
BEARING ON MASONRY : CONTINUOUS PROFILE



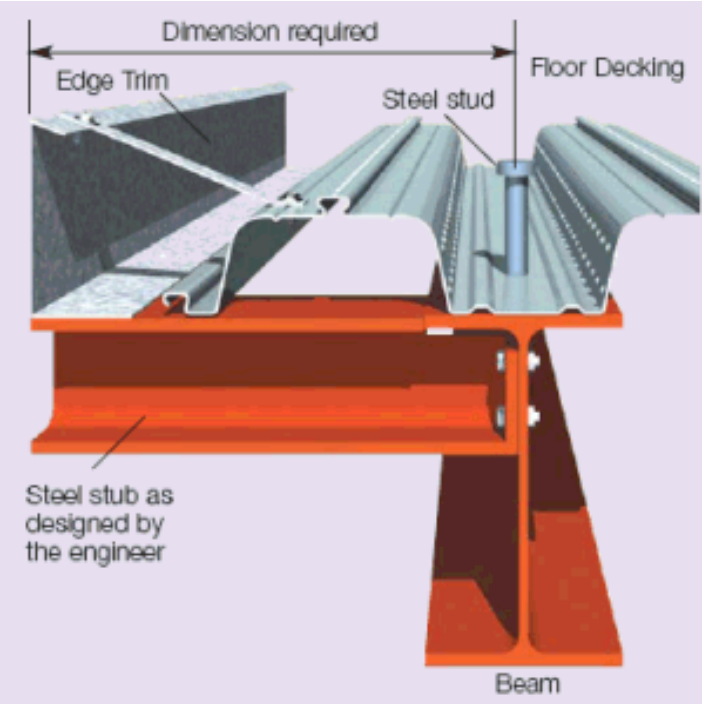
STUD CLEARANCE



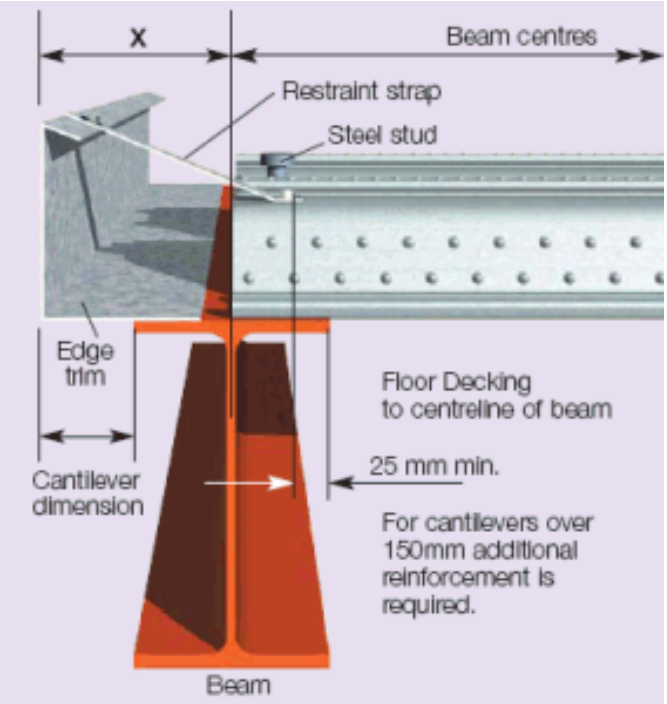
CANTILEVER DECK



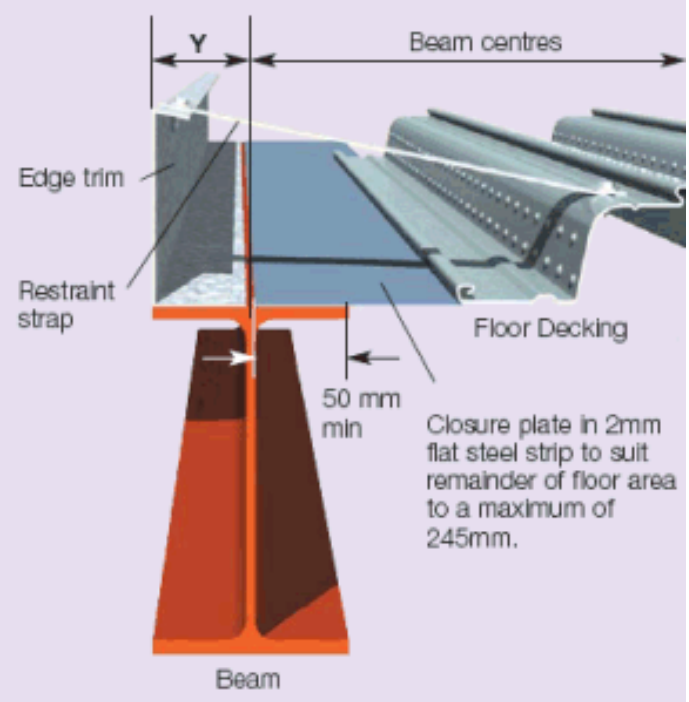
CANTILEVER : DECK PARALLEL TO BEAM



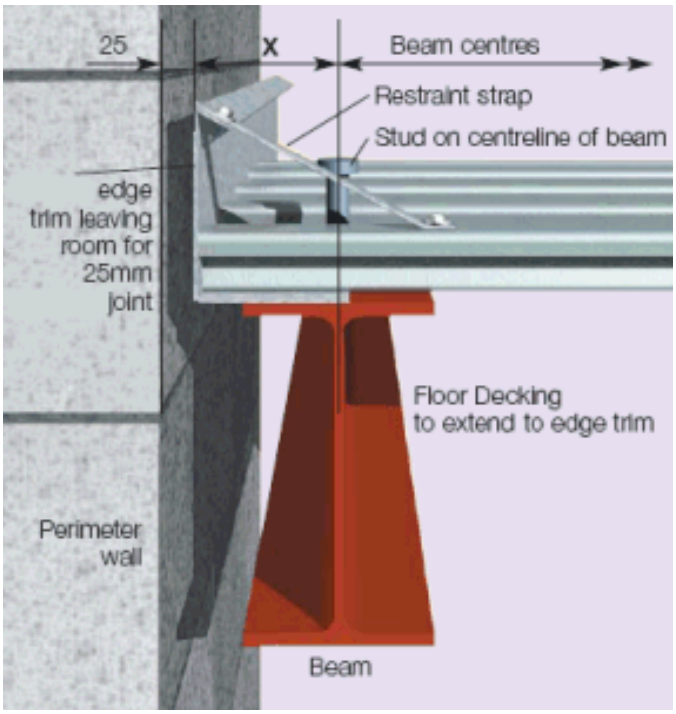
END DETAIL WITH CANTILEVER < = 150 MM



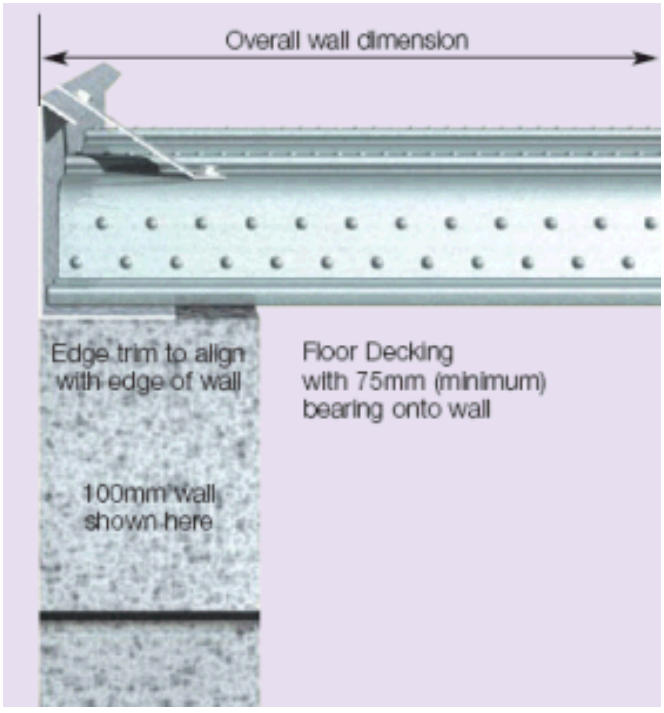
END DETAIL WITH CLOSER PLATE FOR DECK PARALLEL TO BEAM



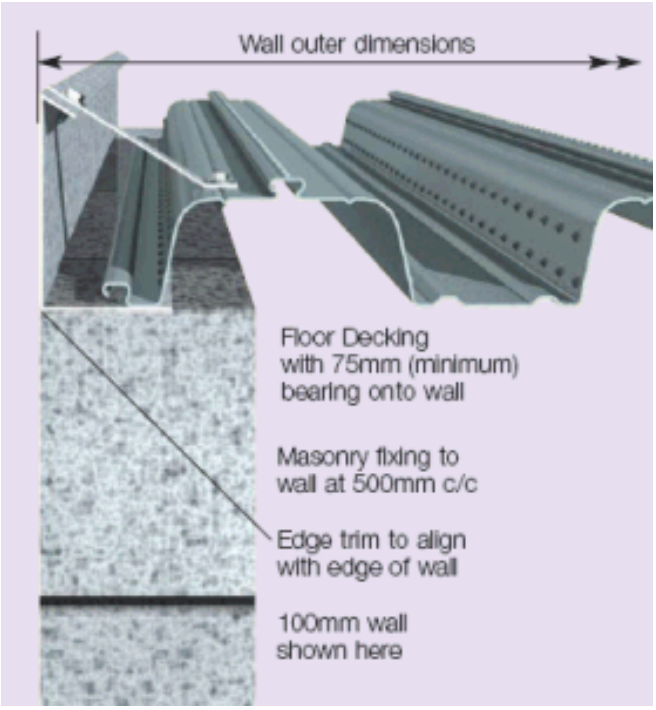
END DETAIL WITH 25 MM WALL CLARENCE



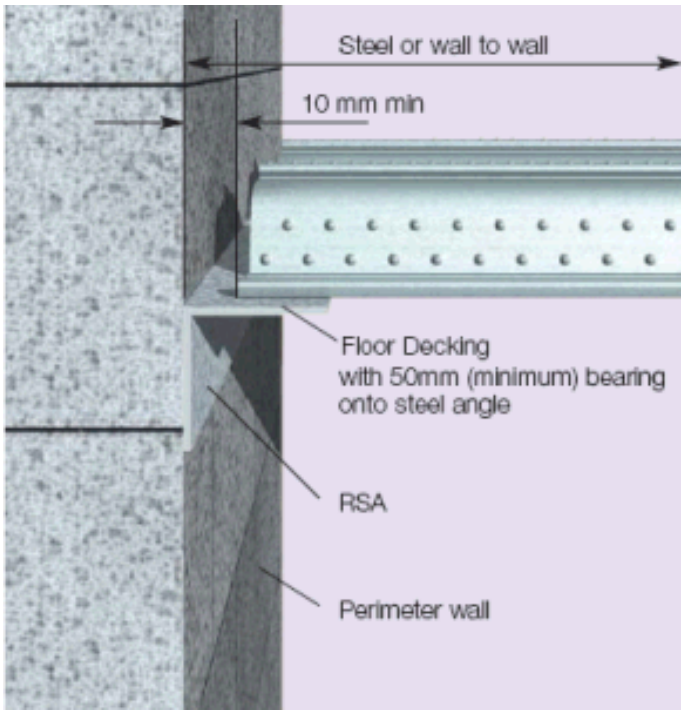
DECK PROFILE ON END WALL



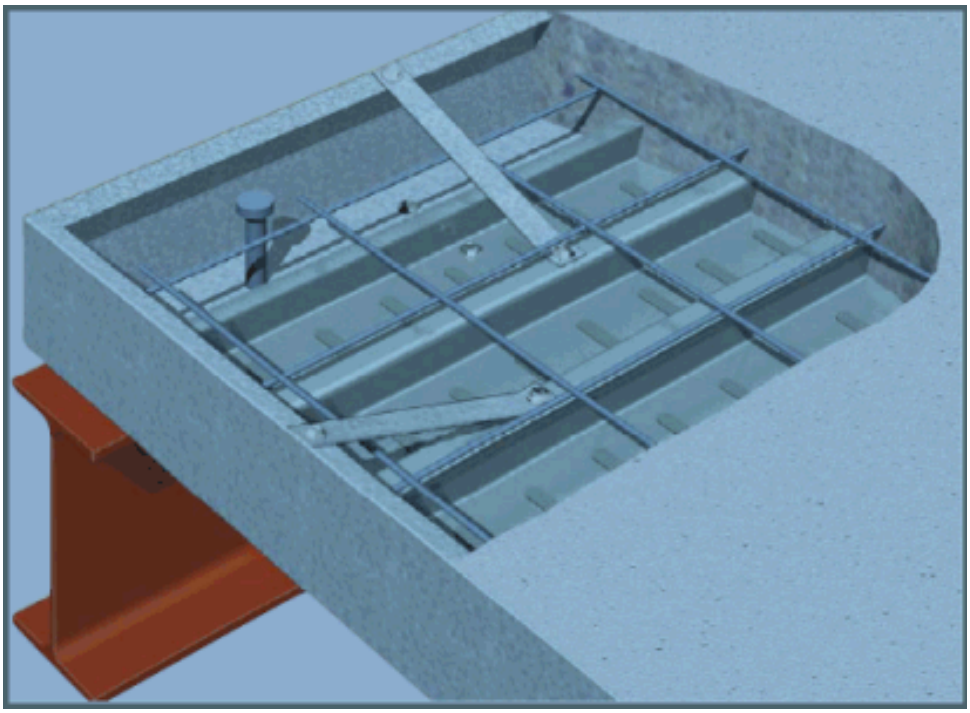
END DETAIL WITH DECK PARALLEL TO WALL



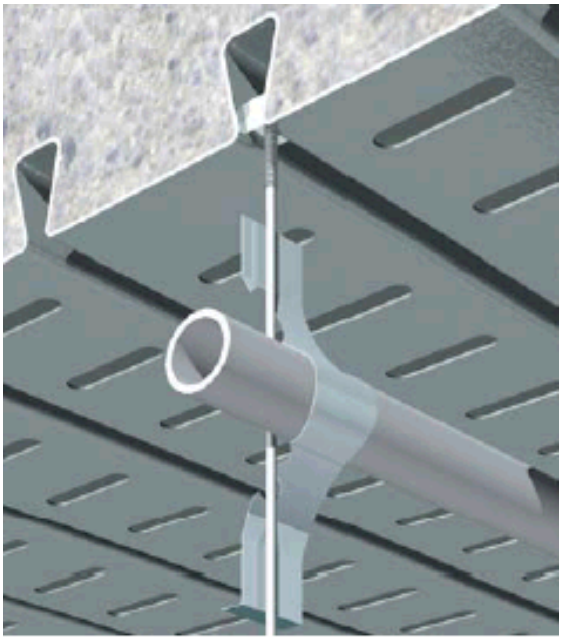
END DETAIL WITH DECK PERPENDICULAR TO WALL



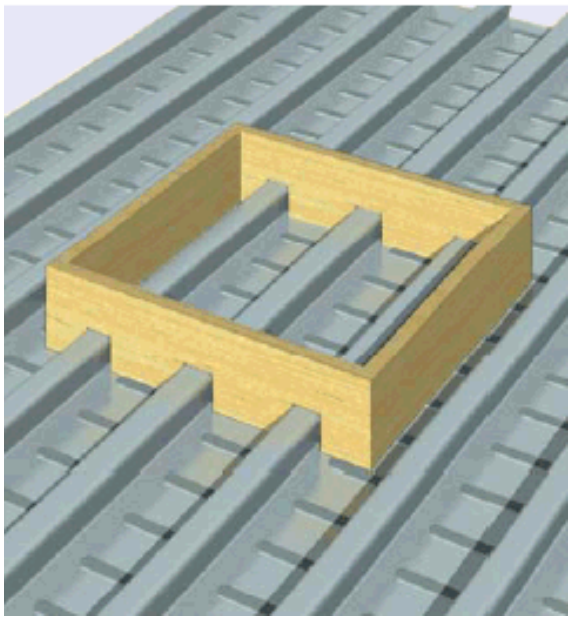
DETAILS OF EDGE TRIMS



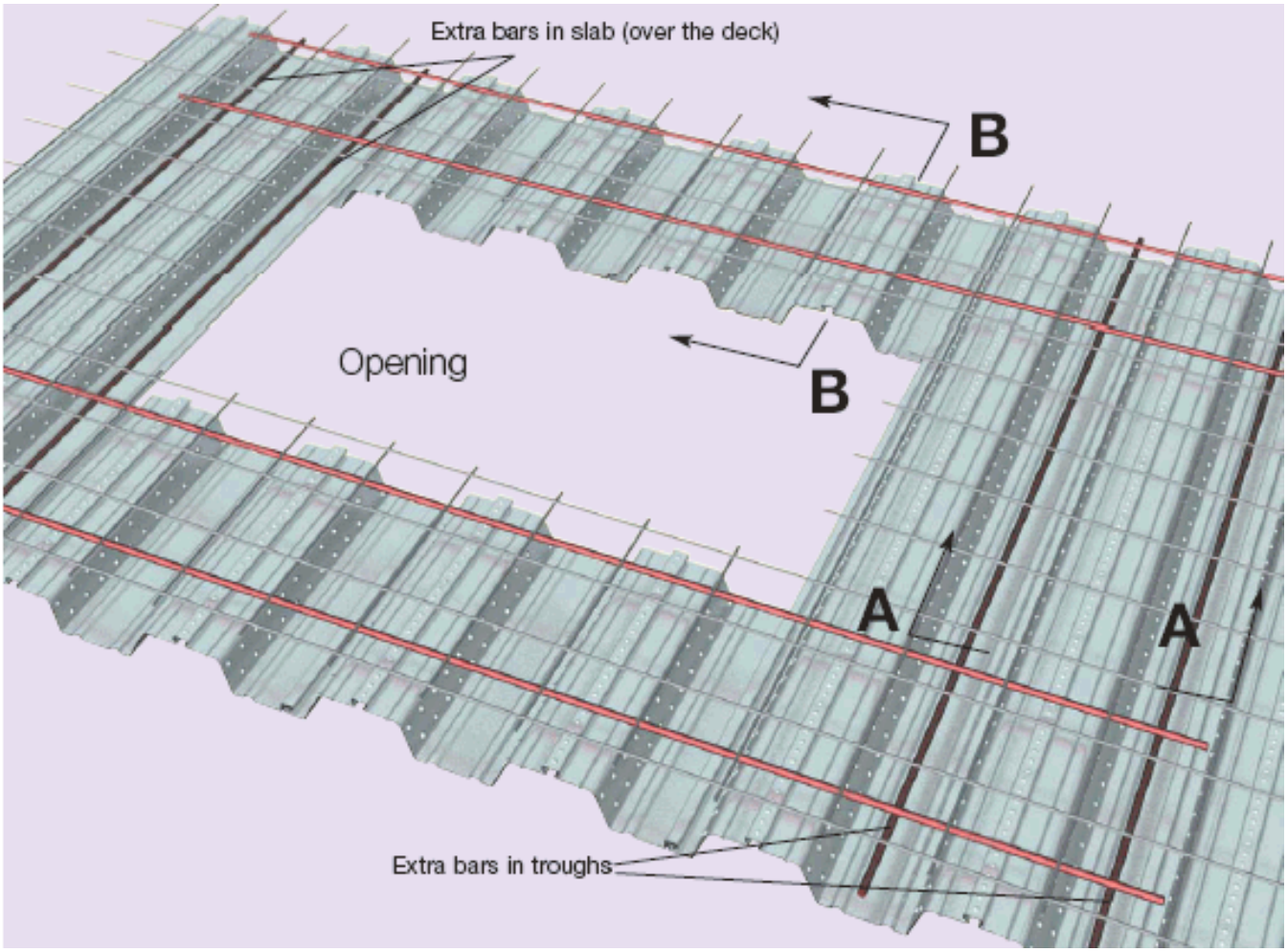
DETAILS OF HANGER



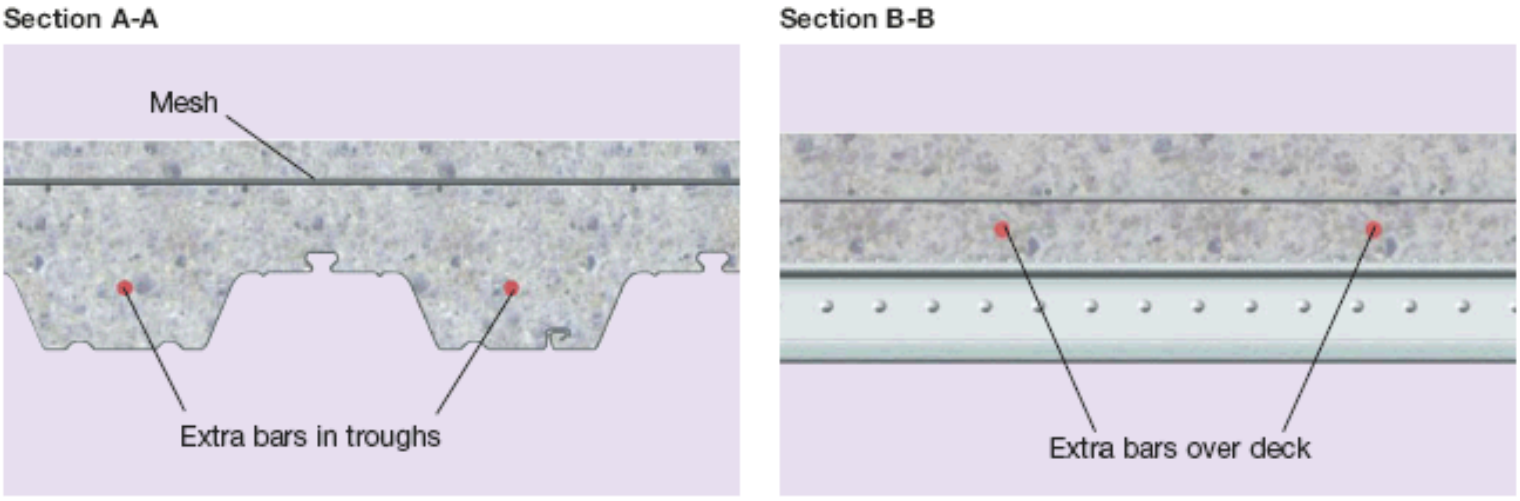
CUTOUT IN DECK USING TIMBER SHUTTERING



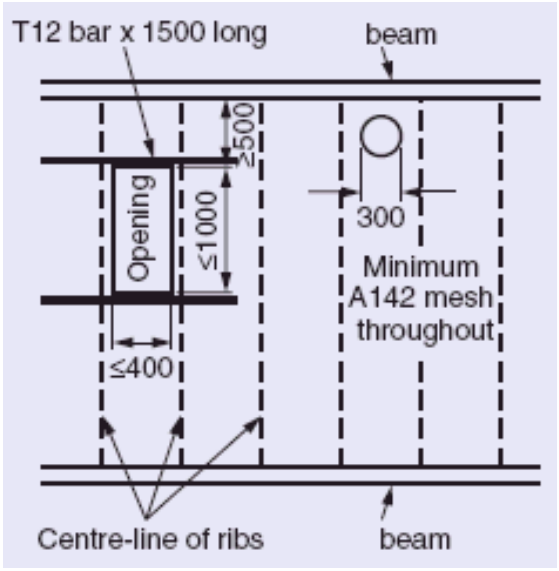
TYPICAL CUTOUT WITH REINFORCEMENT DETAILS



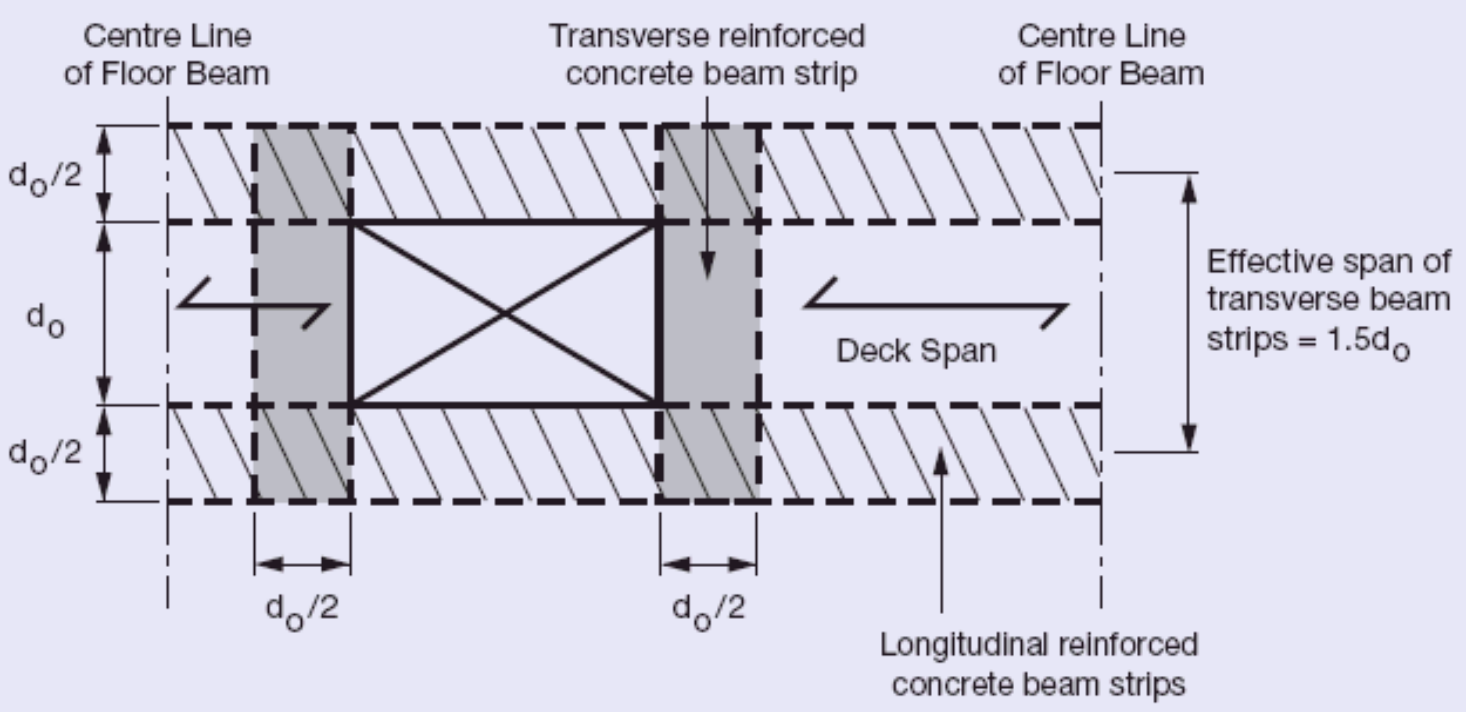
SECTION A - A & B - B



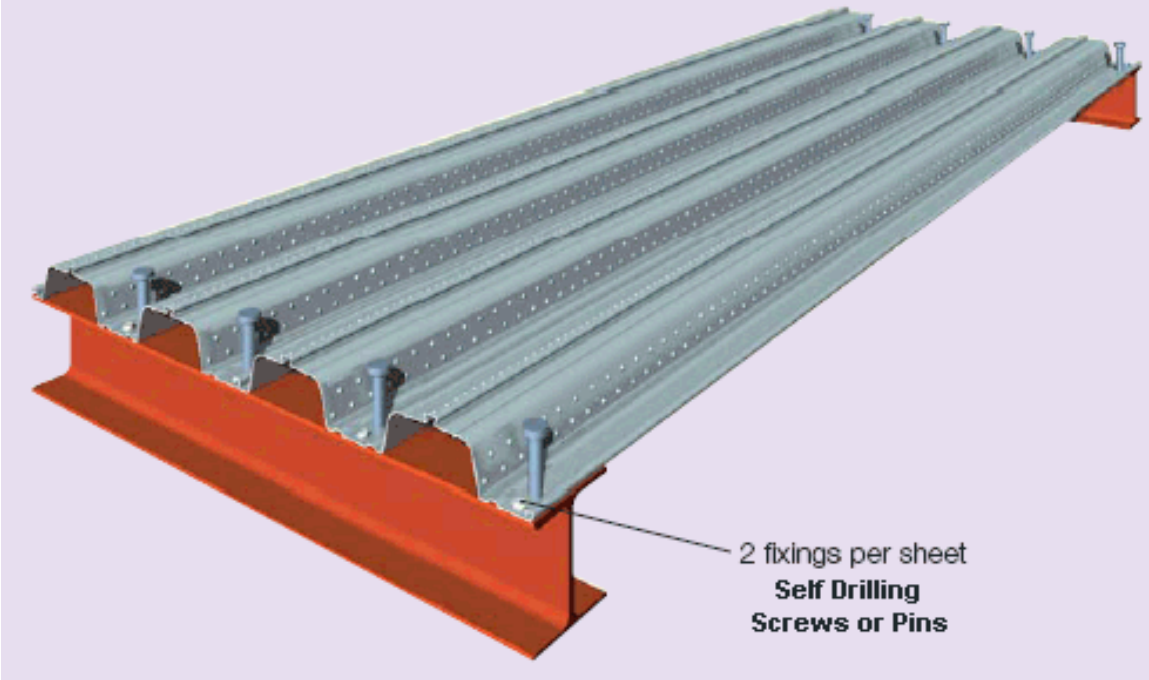
CUTOUT / OPENING DETAILS THROUGH DECK



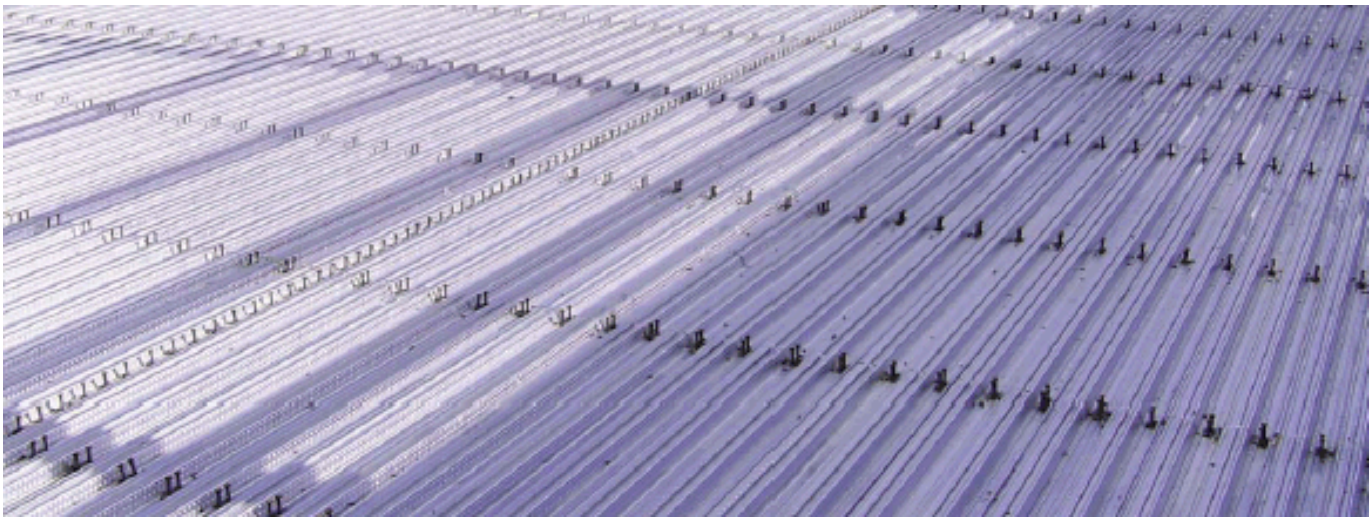
REINFORCEMENT STRIP AROUND OPENING



SELF DRILLING SCREWS / PINS + STUDS

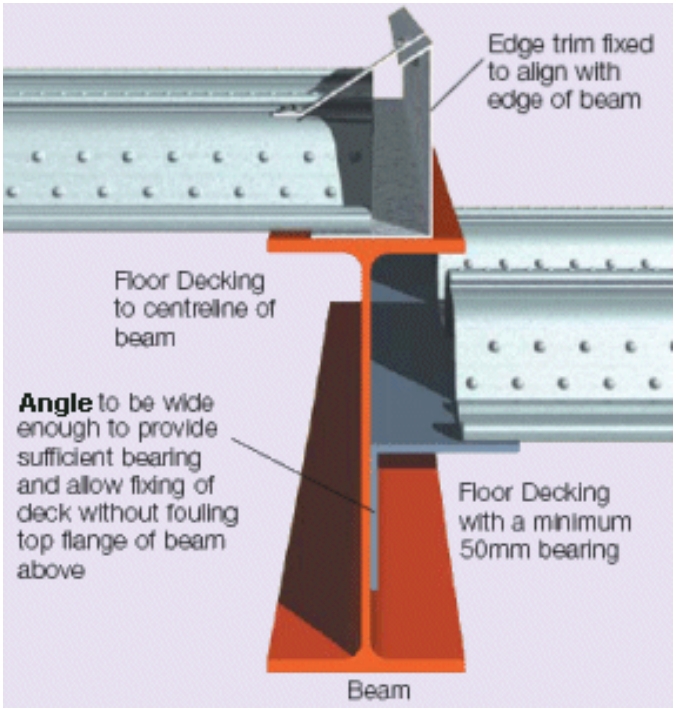


TOP VIEW OF DECK PROFILE + STUDS



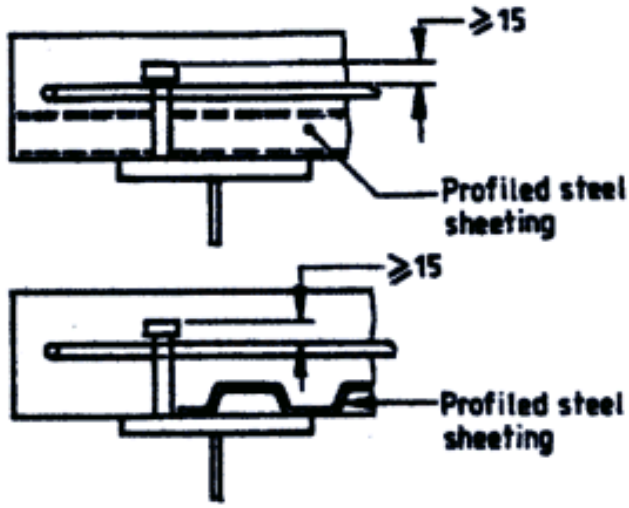
TYPICAL SOFFIT VIEW OF DECK

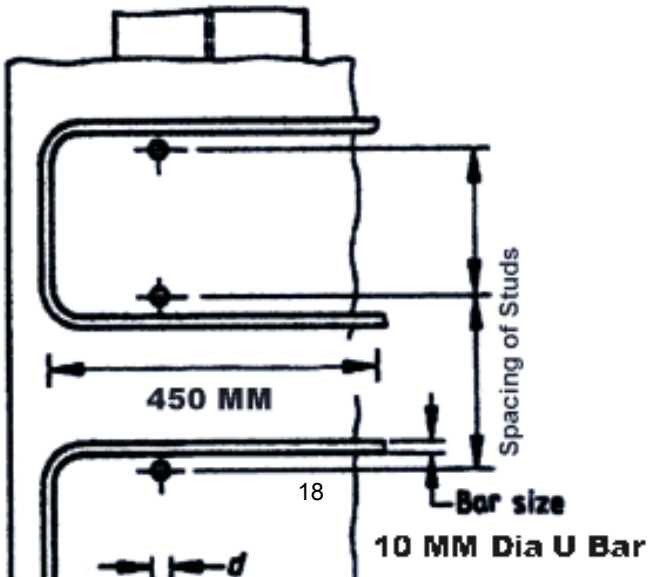
STEP IN DECK FLOOR

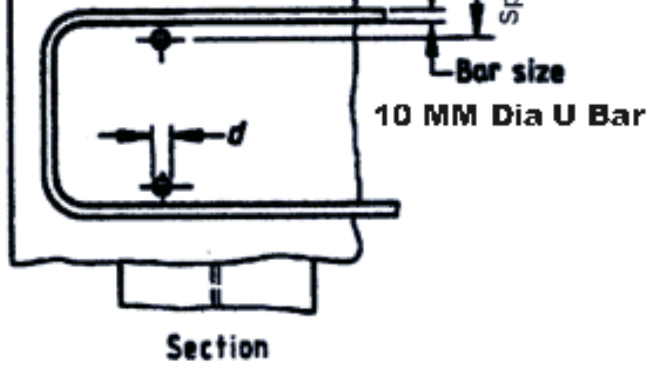


EDGE BEAM DETAILS

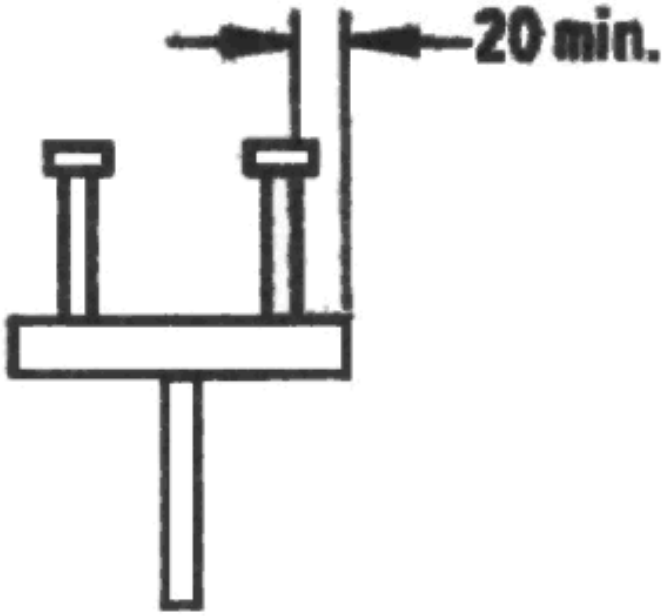
PROVIDE 10 MM Ø U BARS - 450 LONG EXTRA STEEL



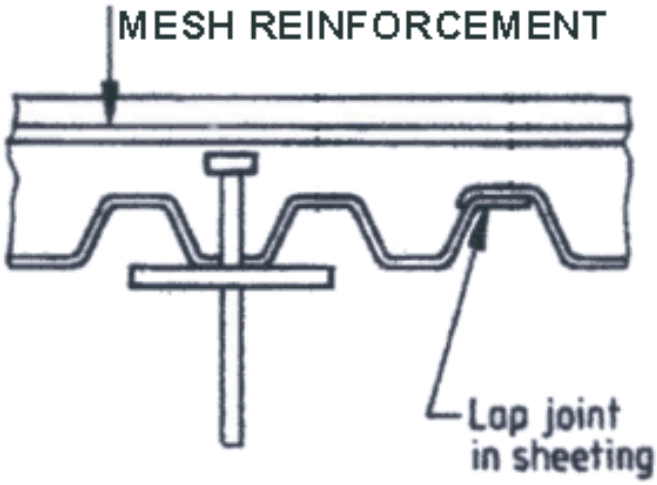




EDGE DISTANCE OF STUD SHEAR CONNECTOR



LAP JOINT IN DECK PROFILE : DECK PARALLEL TO BEAM



LEARN COMPOSITE

INTRO, LIMITATIONS & SPECS

- **Composite Floor Design is as per Euro 4 & IS 800 : 2007.**

  Composite floor using profiled sheet decking consists of steel beams, steel deck, shear connectors, steel mesh and cast in-situ concrete in such a manner that they would act monolithically. Composite floor comprises profiled steel decking as the permanent formwork to the underside of concrete floor spanning between support beams. The decking acts compositely with the concrete slab. It also supports the loads applied to it before the concrete has gained adequate strength (during construction stage). A steel mesh is placed in the concrete floor to avoid effect of cracks and shrinkage.

- Read all Beam to Beam, Beam to Column Shear Connection design with reference to standard details given in the software.

- **The software performs Analysis, Design, Estimation & Costing of Composite Steel Floors and Columns. Floor Plan shall be at a given Uniform Level (2D). Multiple Level Floors (3D) cannot be analyzed.**
  All Steel Columns shall be Box type i.e. 2MC Toe to Toe OR 2 MB Toe to Toe Welded without any Gap.
  The Software basically requires a User to enter floor data for Joints, Columns, Beams, Concrete Slab + Deck profile & Point load. All beams are Simple Supported. The rest of the things are taken care of by the software.

  The results are displayed in the form of BM & SF, Beam Design Details, Quantities, Cost & Column Loads.

  Graphics option are available for display and tabular Format is available for Editing and Deleting Data.

  A User should Delete / Edit Input-Data through the various Program Options only. If any editing is done outside the design environment than Data files may become corrupted. All Data should be Strictly "Entered" as explained in following steps.

  Extensive Printing options are available under each display. Printing is straight forward with default set of values (Arial Font, 8 mm Thick, Bold, Portrait).

  The best way to go about the software is to Mark on the Floor Plan, Joint, Beam, Column and Slab Numbers. A Joint represents a Column location or an intersection between 2 Beams. The Beams are represented by its location in the form of Right Hand Side (RHS) & Left Hand Side (LHS) Joint numbers. The Slabs are represented by LEFT BOTTOM & RIGHT TOP joint numbers. All Joints will have X & Y Co-Ordinates, Top Left corner is taken as origin (0, 0). Joint / Beam / Column / Slab numbers should start with " 1 " and should not be repeated.

The Program will generate automatic Joint, Beam, Column & Slab Numbers from the information given in Project File. Some of these Numbers / Members may not be required & shall be deleted in a systematic manner as explained in the following chapters. The Final Plan Graphics should look exactly as the Floor Plan.

- **Cantilever beams cannot be analyzed.**
  Only Vertical Loads & Axial loads on Beams are Envisaged.
  Lateral Loads & End BMs are on Beams are not Permitted.

- **Beams / Columns / Slabs shall be along two mutually perpendicular axis (Z and Y). Polygonal (Multi-sided) Floors cannot be analyzed.**

- **Only Dead + Live Load Case is analyzed by the software.**

It is assumed that structure is braced and lateral forces are resisted by only Braced Frame. A Frame which is braced as well as non sway will be preferred for composite construction. Non Braced elements are designed only for DL + LL. Refer the following Euro 4 Code (EC4) explanation.

- **Intersecting Joints between two Beams (Main & Secondary) is assumed as Hinged. Hence no Moment transfer is envisaged.**

- **Connection between End Column and Beam is considered as Hinged. Hence no Moment transfer is envisaged between Column and Beam.**

- **After data input, the user has to switch over to graphic option for visual checking of joints / columns / beams / slab nos. When the data is error free the user can run the Analysis, Design and Quantity options. The various results are also available through display or print options.**

- **Analysis, Design and Quantity options should be run in strict order, else program will give unexpected results.**

- **Program creates automatic Joint numbers as per nos. of Horizontal & Vertical Grids. Here Grids means Beams coming along Column center lines as well as all Internal Beams not aligned with columns. A user has to input Information regarding Horizontal & Vertical Grids while creating Project File.**

- **A user can delete the Joints not required by using Joint Option.**

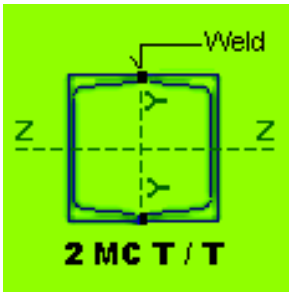
- **Joints will be automatically re-numbered when "UPDATE" button is clicked or at "EXIT".**

- **Remember to Delete / Edit Corresponding Beam / Column / Slab Member, whose Joint has been deleted.**

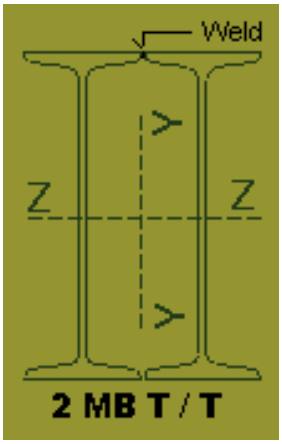
- **Always delete Beam / Column / Slab member from the "END" to facilitate further Editing. After Deleting press "UPDATE" button for re-numbering of members.**

- **After Deleting corresponding Beam / Column / Slab Member & Updating, edit the required Joint Numbers of affected Beam / Column / Slab Members.**

- **Go through the "READ ME" Button for better understanding of that particular Option.**
- **Only 2 Types of Beam are Possible.**
  (a) MC
  (b) MB
- **Only Two Types of Columns are Possible.**
  (a) MC welded Toe to Toe (T / T)
  (b) MB welded Toe to Toe (T / T)



2 MC T / T



2 MB T / T

- **Following Sections are Possible for Beams & Columns.**

  (a) ISMC Section: ISMC 100, 125, 150, 200, 250, 300, 350 & 400.

  (B) ISMB Section: ISMB 100, 150, 200, 250, 300, 350, 400, 450, 500, 550 & 600.

Braced and unbraced frames

When bracing is provided it is normally used to prevent, or at least to restrict, sway in multi-storey frames. Common bracing systems are trusses or shear walls (Figure 9).

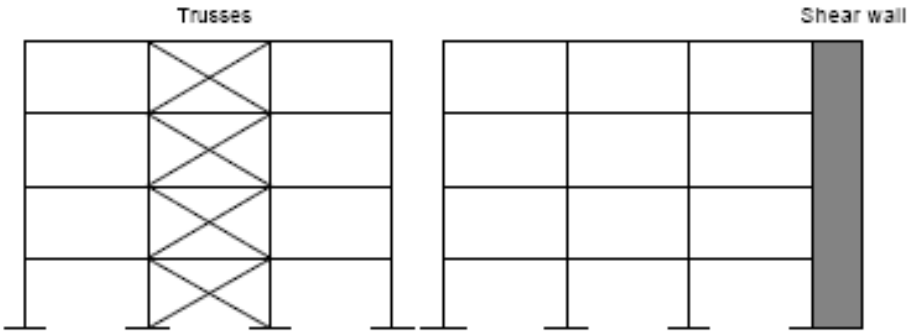


Figure 9 - Common bracing systems

For a frame to be classified as a **braced frame**, it must possess a bracing system which is adequately stiff.

When it is justified to classify the frame as **braced**, it is possible to analyse the frame and the bracing system separately as follows :
The frame without the bracing system can be treated as fully supported laterally and as having to resist the action of the vertical loads only.
The bracing system resists all the horizontal loads applied to the frames it braces, any vertical loads applied to the bracing system and the effects of the initial sway imperfections from the frames it braces and from the bracing system itself.

It should be noted that in a frame with a truss type or frame type bracing system, some members participate in the bracing system in addition to being part of the frame structure (without bracing).

For frames without a bracing system, and also for frames with a bracing system but which is not sufficiently stiff to allow classification of the frame as braced, the structure is classified as **unbraced**. In all case of unbraced frames, a single structural system, consisting of the frame and of the bracing when present, shall be analysed for both the vertical and horizontal loads acting together as well as for the effects of imperfections.

Sway and non-sway frames

The term **non-sway frame** is applicable when the frame response to in-plane horizontal forces is sufficiently stiff for it to be acceptable to neglect any additional forces or moments arising from horizontal displacements of its nodes. The global second-order effects (i.e. the P-Δ sway effects) may be neglected for a non-sway frame.

When the global second-order effects are not negligible, the frame is said to be a **sway frame**.

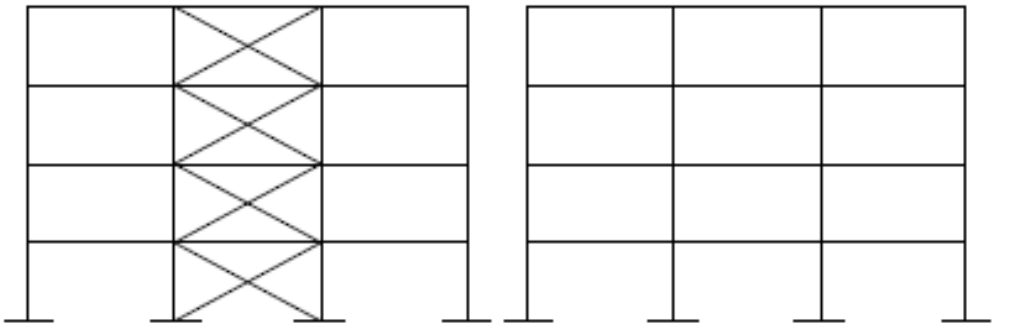


Figure 10 - Braced and unbraced frame

Normally a frame with bracing is likely to be classified as *non-sway*, while an unbraced frame is likely to be classified as *sway*. However, it is important to note that it is theoretically possible for an unbraced frame to be classified as non-sway (this is often the case of one storey portal frame buildings) while a frame with bracing may be classified as sway (possible for multi-storey buildings) (see Figure 10).

When a frame is classified as *non-sway*, a first-order analysis may always be used.

When a frame is classified as *sway*, a second-order analysis shall be used. A procedure involving iterations on a first-order elastic analysis is usually adequate for this purpose. Furthermore, on condition that the structure meets certain conditions, a first-order analysis (without any iteration process) may be used either by making a nominal correction to member end forces to allow for the global second-order effects or by analysing for vertical loads and for sway load effects (to be magnified for design) separately.

It should be noted that bracing systems which are themselves frames (or sub frames) must also be classified as sway or non-sway .

- **End Reactions are obtained by running 2-D or 3-D Portal Frame Analysis programs separately. User may use our 2-D Frame Analysis software for calculation of End Reactions of Braced Frames.**

IMPORTANT INFO / SPECS

1. Designer to Select Deck Profile as per Allowable total Loading Intensity corresponding to Maximum span of Deck, from the Load Table provided by Deck Manufacturer. The Deck deflection shall not exceed Span / 180 or 20 MM whichever is less. The Minimum Yield stress of DECK shall not be less than that of Structural Steel.

2. The Composite floor design is based on the assumption that all beams are simply supported. There is no Hogging bending moment due to DL + LL or Wind or Seismic.

3. The Complete System shall be Braced with Shear Wall or with Crossed Braced Frames, resisting all lateral Forces.

4. It is Preferred to Design all Braced Frames as Non-Sway Frames, refer IS 800 : 2007.

5. Select the Deck Span requiring no Props.

6. Restrict thickness of Galvanized Deck profile to 1.25 - 1.50 MM.

7. Diameter of automatically welded Studs are restricted to 19 MM.

8. Depth of Deck Profile > 85 & < 35 MM not Permitted.

9. Decking Sheets shall be attached to each other & to all permanent supports using screws or shot fired nails.

10. Thickness of Concrete above Rib < 55 MM not Permitted. Fire Rating of 2 Hours.

11. Deck Width (br) < 50 MM not Permitted.

12. Minimum Clear Cover to Stud from Concrete Top = 20 MM.

13. Deck Sheeting Longitudinal over lap = 150 ~ 300 MM & Min. Side Over lap = 30 MM as per Manufacturer's Specs.

14. Min. Bearing of Deck Profile on Steel = 50 MM.

15. Aggregate Size not > than 0.4 * Conc. thickness above Rib.

16. Attach minimum of one Stud to every decking trough.

17. However 2 Studs can be attached if required by design.

18. Stud Transverse Spacing not > 4 Ø.

19. Stud Longitudinal Spacing > 5 Ø.

20. If Frequency of Vibration of Beams < 4 Cycles/Sec,
Designer to change Section.

21. Height of Stud > 4 * Ø of Stud for Ductile classification.

22. Head Ø of Stud = 1.5 * Ø of Stud.

23. Minimum Yield Strength Fy for Deck Profile Shall be 250 N/MM2

24. Clear Cover to Stud from Top shall be 20 MM.

25. Clear Cover to Mesh Reinforcement from Top shall be 30 MM.

- **Export to Excel :**

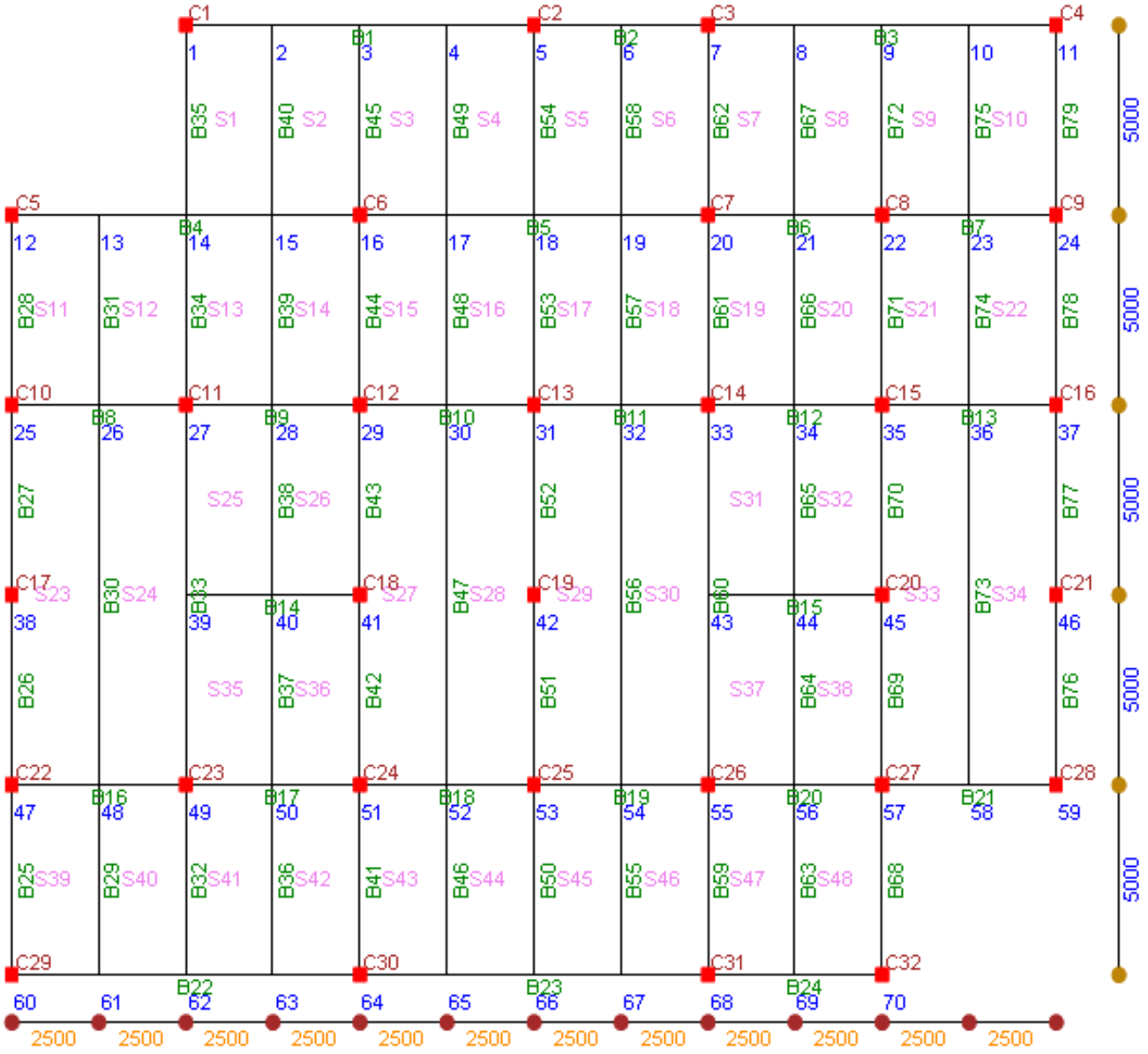
When the " Analysis Result -> Bending Moment & Reaction " option is Run, a Text file is automatically created. This File will open in Any Text Editor. You can also Open this Text File in EXCEL.
Start Excel -> File -> Open -> Delimited ->Next : Delimiters -> Comma ->Next -> Finish.
Now you will notice that Complete Data is displayed in Excel Spread Sheet.
If more than One File is Created, Corresponding to Each of Load Cases, than Open Excel Sheet for Each File (Load Case). In Excel Sheet Editing, Deleting, Sorting, Printing & Merging of Data/Files/Excel Sheets is Extremely Easy. This way any no. of Load Cases can be Manipulated.

Similar Text files are created in " Shear Corrected BM & SF " (Design BM & SF) & " Column Loads " option for Exporting Results to Excel Spread Sheet & its subsequent Manipulation.

- Minimum Computer RAM memory of 2 GB is recommended.
- Use Laser OR Ink Jet Printer.

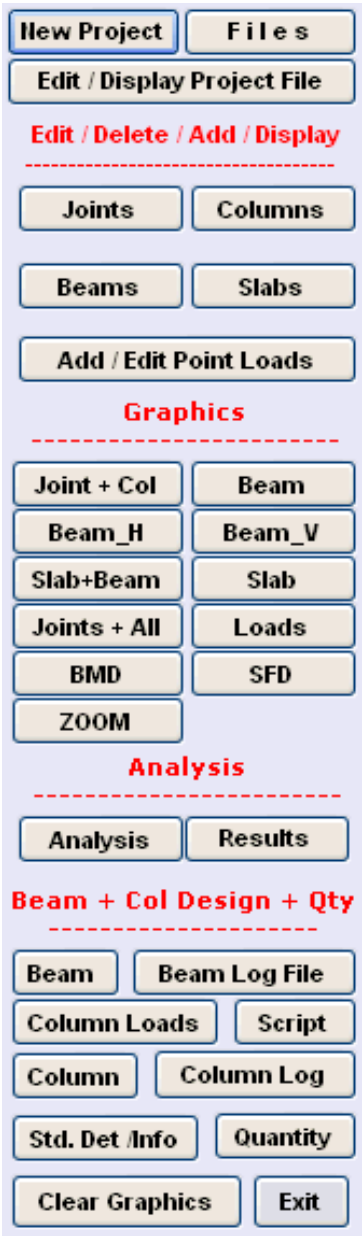
LEARN COMPOSITE STEP BY STEP

STEP NO. 1 : New Project (File) Creation



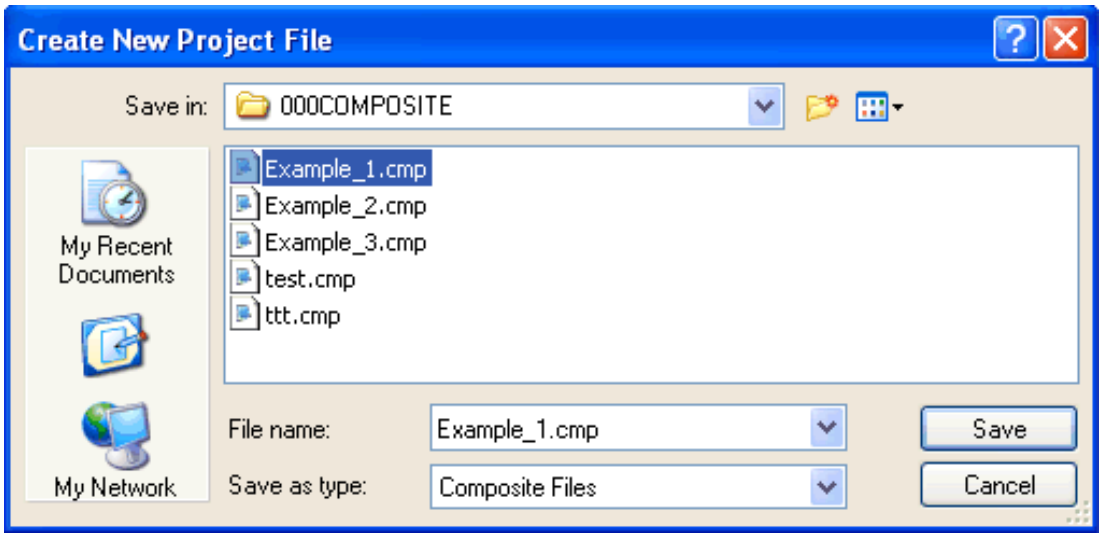
ACTUAL REQUIRED FLOOR PLAN

- **Refer the above Floor Drawing. Our Intention is to perform Analysis, Design, Estimation & Costing of the above Floor. The above floor has 70 # of Joints, 32 # of Columns, 79 # of Beams and 34 # of Slabs (Chequered Plate / Grating).**
  Please go through the following steps carefully, so that we can achieve our object efficiently.



- **When Program starts, the graphics above is displayed. Consider the " New Project Option ".**

Click the " New Project " option in the MENU bar. The following window will open.



You must create a separate Folder / Directory to store your files.
I have created a Directory called " 000COMPOSITE " in D: drive to store my Project files.
Now go to this folder & give a file name to your project. I have given " Example_1 " as the name of my new project file. Click the save button. Following project window will open.

Add Project Details :

File Name : D:\000COMPOSITE\Example_1 .cmp

Date : 04 June 2010

Organization	Super Civil CD	Net Height of Brick Wall in M	
Project	20 Story Bldg.	Thickness of Brick Wall in MM	
Project No.	8912	Total Flooring Dead Load in Kg/M2 (Concrete + Deck Profile + Partition + Celling + FF Loads)	430
Building ID	Admin	Default LL on Slab in Ton / M2	0.20
Floor No.	12	Column Type	2MC T/T
Floor Level	36.0	Column Section	MC-200
Floor Width (X Axis- Horiz. Dist.) in MM	30000	Default Storey Height in M	3
Floor Length (Y Axis- Vert. Dist.) in MM	25000	Structural Steel Rate in Rs / Ton	50000
No. of Vertical Grids (For Horiz. Dist.) Each for Every Beam and Column	7	Masonry Work in Rs / M2	850
No. of Horizontal Grids (For Vert. Dist.) Each for Every Beam and Column	6	Plastering in Rs / M2	400
Floor Spanning Along (Deck Profile)	X Axis	Painting in Rs / M2	100
Fy:	250	Total Door + Window Area in M2	
Beam Type	MB	Door / Window Rate in Rs / M2	2500
Beam Section	MB-200	Concrete Rate in Rs / M3	5000
Permissible Deflection Ratio - Beam	300	Reinforcement Rate in Rs / Ton	50000
Ultimate Tensile Stress of Steel in Mpa	410	Rate of Deck profile + Studs in Rs / M2	500
Ultimate Tensile Stress of Studs in Mpa	450	Rate of False Ceiling in Rs / M2	400
Concrete Grade	M20	Area of Deck Steel in MM2 / M	2118
Total Thickness of Concrete in MM	105	Height of Neural Axis of Deck in MM (From bottom of Deck Profile)	16.72
Thickness of Concrete above Rib in MM	55	Effective Width of Trough {br} in MM	93
Stud Diameter in MM	19		
Maximum Span of Deck in MM	2500		

EXIT | NEXT PAGE | READ ME | PRINT

- **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 " Example_1 ".
  Please note that you cannot Change / Edit all these values later. Hence be-careful in feeding these initial values. Few values will be allowed to change. The total floor width & length values will be used to tally the sum of individual Vertical and Horizontal Grids.
  Here the meaning of Grids means Center Lines Between Column-Beams and also Center Lines of Beams without Columns. Even any isolated Beam is treated as a Grid.
  The automatic creation of Joint Numbers & Co-Ordinate system depends up on total width, length & No. of vertical & horizontal Grids of floor.

Click on Read Me Button, following important points are displayed.

1. Designer to Select Deck Profile as per Allowable, Total Un-factored Loading Intensity corresponding to Maximum span of Deck, from the Load Table provided by Deck Manufacturer.

2. The Deck deflection shall not exceed Span / 180 or 20 MM whichever is less.

3. The Composite floor design is based on the assumption that all beams are simply supported. There is no Hogging bending moment due to DL + LL or Wind or EQ.

4. The Complete System shall be Braced with Shear Wall or with Crossed Braced Frames, resisting all lateral Forces.

5. It is Preferred to Design all Braced Frames as Non-Sway Frames, refer IS 800 : 2007.

6. Select the Deck Span requiring no Props.

7. Restrict thickness of Galvanized Deck profile to 1.25 MM.

8. Diameter of automatically welded Studs are restricted to 19 MM.

9. Thickness of Deck Profile > 85 & < 35 MM not Permitted.

10. Decking Sheets shall be attached to each other & to all permanent supports using screws or shot fired nails.

11. Thickness of Concrete above Rib < 55 MM not Permitted.

12. Deck Width (br) < 50 MM not Permitted.

13. Min. Clear Cover to Stud from Concrete Top = 20 MM.

14. Deck Sheeting over lap = 150 ~ 300 MM as per Manufacturers details.

15. Min. Bearing of Deck Profile on Steel = 50 MM.

16. Aggregate Size not > than 0.4 * Conc. thickness above Rib.

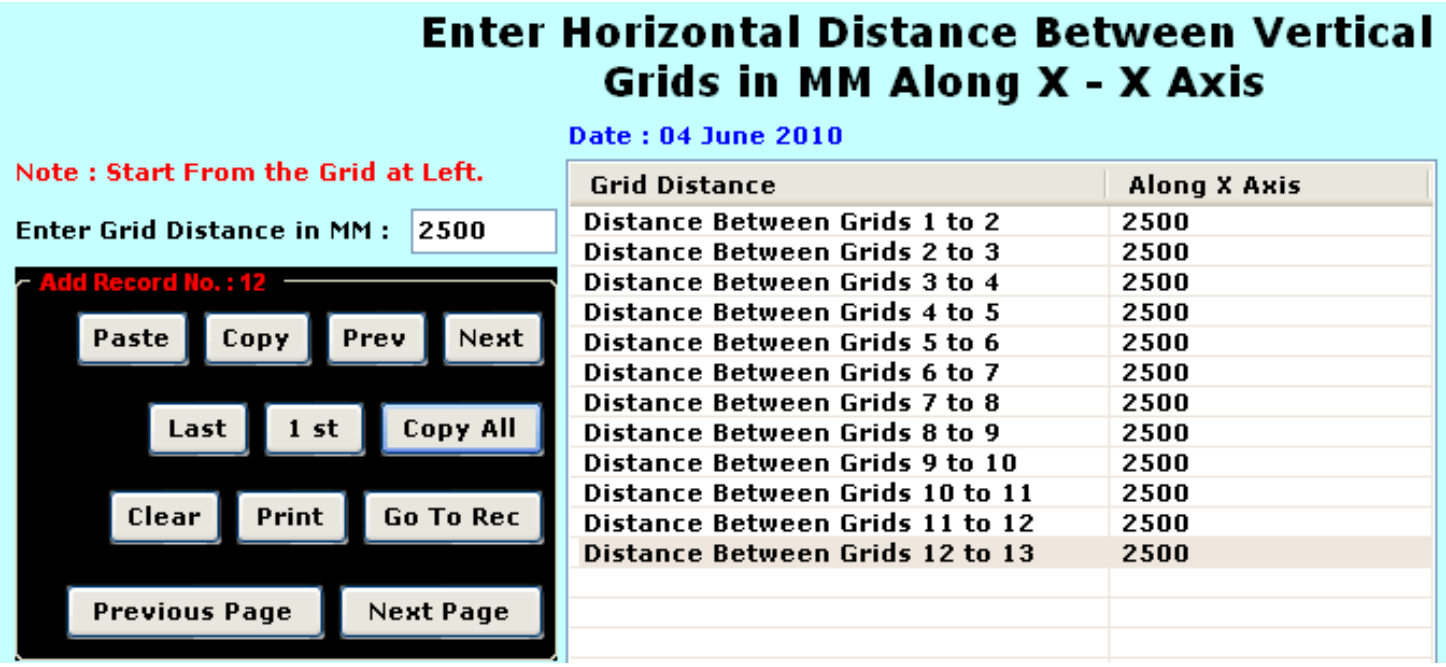
17. Attach minimum of one Stud to every decking trough.

18. However 2 Studs can be attached if required by design.

19. Stud Transverse Spacing not > 4 diameter.

20. Stud Longitudinal Spacing > 5 diameter.

Now click the " Next Page" button, following window will appear.



- **I have entered the Horizontal Grid distance as 25000 mm for each Bay. The total is 30000 mm, which tally's with the total floor width of 30000 mm which was entered in the earlier page. If there is a mis-match between the two then an error will be displayed. A user can click " Previous Page " button to display the previous page & verify the required total width. Note that distance between vertical Grids means horizontal distance. Start from leftmost grid by referring to the Floor Plan.**

If all grid distances are same then a user can enter the grid distance once & use " Copy All " button to copy the values to all ROWS.

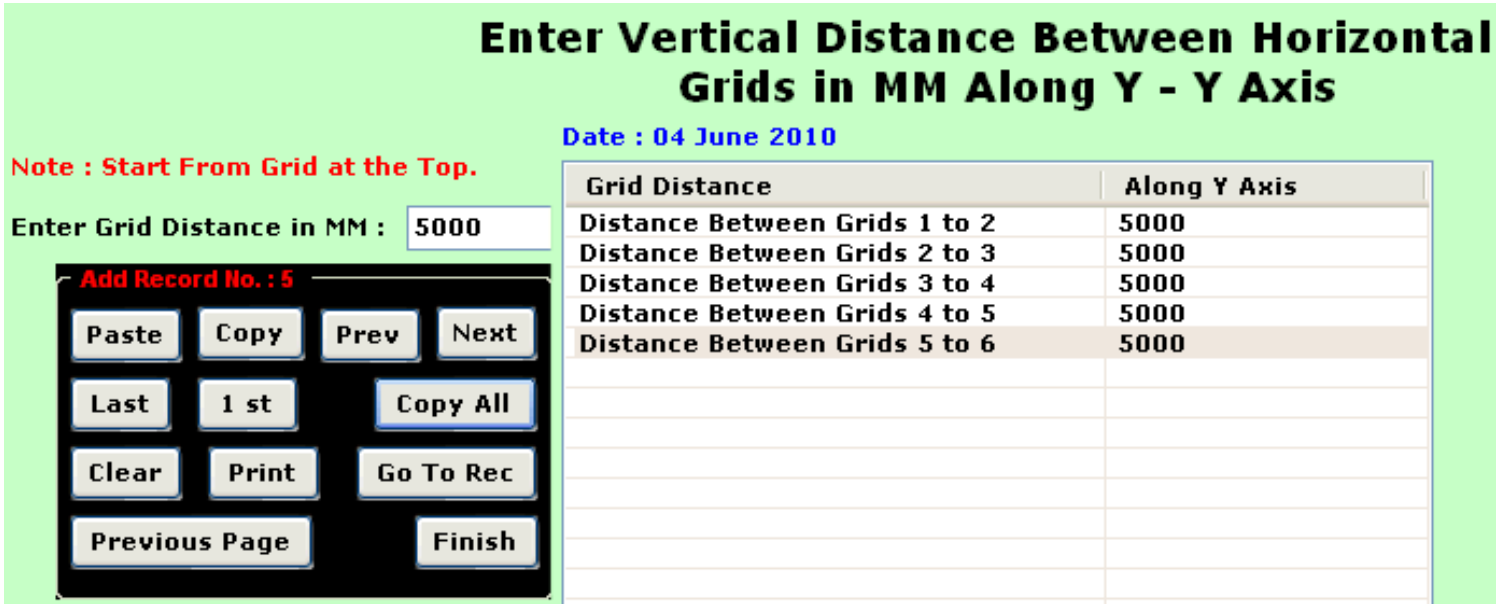
Use Copy & Paste Button to copy & paste values to different rows, in case the grid distances are not same.

The " Prev ", " Next ", " Last ", " 1 st ", & " Go to Rec " Buttons are for displaying / Focusing the cursor on Previous, Next, First or required Record Number.

The " Clear " Button clears all grid Distance values.

The " Print " Button is for printing of values from the Table. Use laser OR Inkjet Printer.

Now click the " Next Page " button, following window will appear.



- **I have entered the Horizontal Grid distance as 5000 mm for each Bay. The total is 25000 mm, which tally's with the total floor width of 25000 mm which was entered in the earlier page. If there is a mis-match between the two then an error will be displayed. A user can click " Previous Page " button to display the previous page & verify the required total width. Note that distance between Horizontal Grids means Vertical distance from Top Down. Start from Top Left grid by referring to the Floor Plan.**

If all grid distances are same then a user can enter the grid distance once & use " Copy All " button to copy the values to all ROWS.

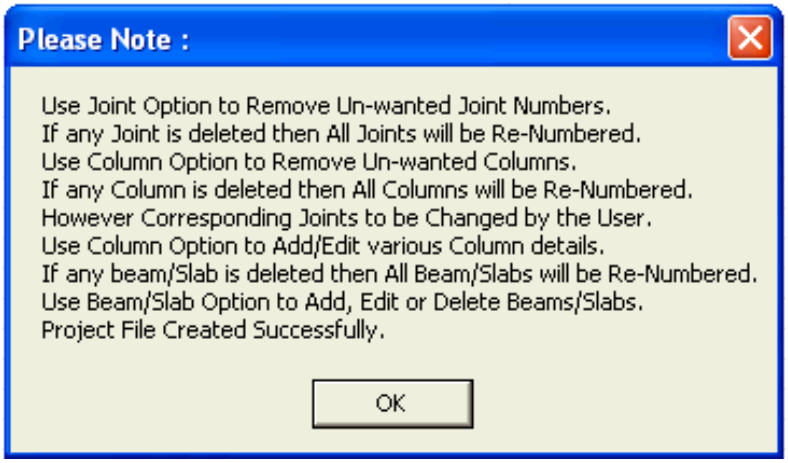
Use Copy & Paste Button to copy & paste values to different rows, in case the grid distances are not same.

The " Prev ", " Next ", " Last ", " 1 st ", & " Go to Rec " Buttons are for displaying / Focusing the cursor on Previous, Next, First or required Record Number.

The " Clear " Button clears all grid Distance values.

The " Print " Button is for printing of values from the Table. Use laser OR Inkjet Printer.

Now click the " Finish " button, following window will appear.



- **Note the above very important message.**
  If any joint no. is deleted then Joint numbers will be re-numbered. Delete the corresponding Un-Wanted Columns, Beams & Slabs, which belonged to above mentioned deleted Joints. Now the Columns, Beams & Slabs will be automatically re-numbered as Un-wanted members are deleted. Now user should manually change the Joint Numbers of Columns. Similarly RHS & LHS joint numbers of Beams should be changed manually as per the revised (Re-Numbered) joint numbers.

- **If a User would like to Part Edit / View the Project File, just click " Edit / Display Project File " Option. Following window will display the project file.**

Edit Project Details :

File Name : D:\000COMPOSITE\Example_1.cmp

Date : 04 June 2010

Organization	Super Civil CD	Total Flooring Dead Load in Kg / M2	430
Project	20 Story Bldg.	Default LL on Slab in Ton / M2	0.50
Project No.	8912	Column Type	2MC T/T
Building ID	Admin	Column Section	MC-200
Floor No.	12	Floor Level 36.0	Default Storey Height in M 3
Floor Width (X Axis- Horiz. Dist.) in MM	30000	Structural Steel Rate in Rs / Ton	50000
Floor Length (Y Axis- Vert. Dist.) in MM	25000	Masonry Work in Rs / M2	850
No. of Vertical Grids (For Horiz. Dist.)	13	Plastering in Rs / M2	400
No. of Horizontal Grids (For Vert. Dist.)	6	Painting in Rs / M2	100
Floor Spanning Along X Axis Fy: 250		Total Door + Window Area in M2	92.4
Beam Type MB Beam Section MB-200		Door / Window Rate in Rs / M2	2500
Permissible Deflection Ratio - Beam 300		Concrete Rate in Rs / M3	5000
Ultimate Tensile Stress of Steel in Mpa	410	Reinforcement Rate in Rs / Ton	50000
Ultimate Tensile Stress of Studs in Mpa	450	Rate of Deck profile + Studs in Rs / M2	500

Concrete Grade	M30	Rate of False Ceiling in Rs / M2	400
Total Thickness of Concrete in MM	130	Area of Deck Steel in MM2 / M	2118
Thickness of Concrete above Rib in MM	80	Height of Neural Axis of Deck in MM	16.72
Stud Diameter in MM	19	Effective Width of Trough {br} in MM	93
Maximum Span of Deck in MM	2500	EXIT READ ME PRINT	

Note from above display that AREAS which are Grayed cannot be edited, rest of them can be changed.

STEP NO. 1 IS OVER.

LEARN COMPOSITE STEP BY STEP

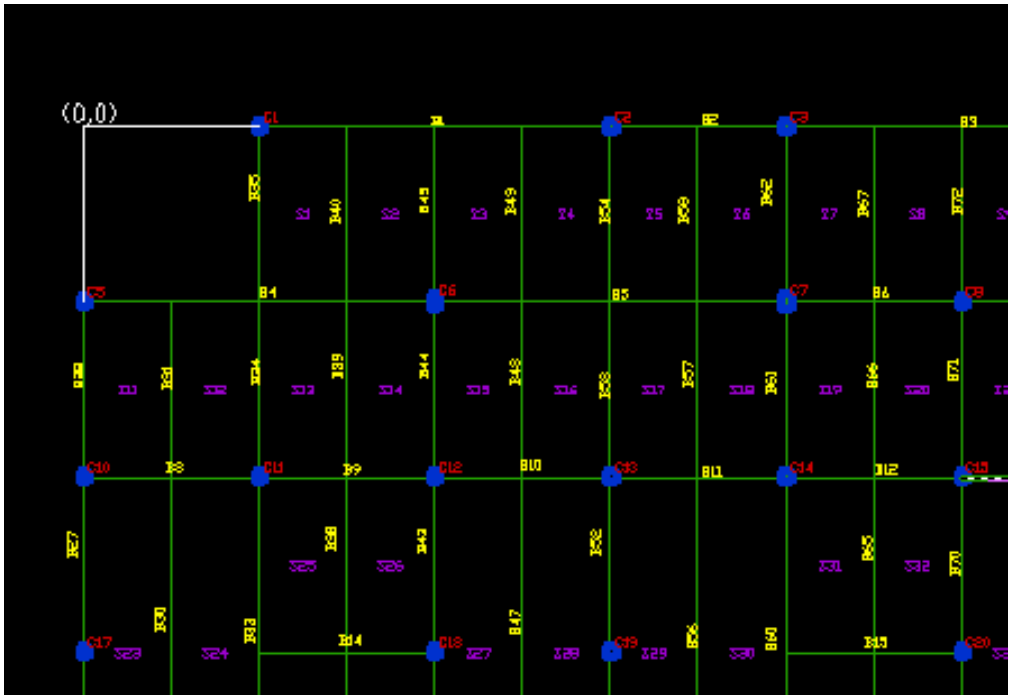
STEP NO. 2 (Alternate) : Scan Joint, Beam, Column & Slab Data from AutoCAD Drawing

In order to Read the AutoCAD drawing in COMPOSITE , the various drawing components should be drawn in their respective layers as shown below.

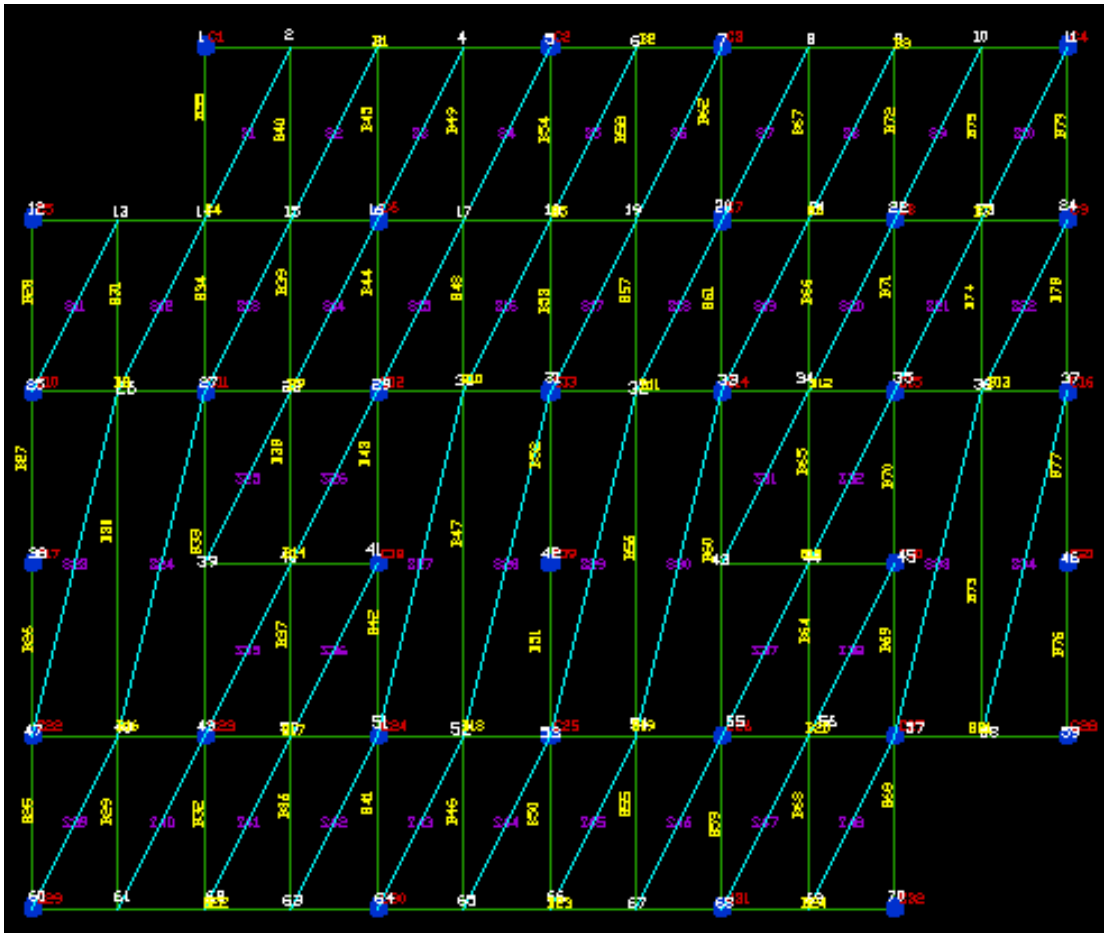
- **The Drawing Components to be drawn to exact scale and in Millimeter (MM).**

During the course of a project, a Floor can be extended by adding new Joints, Beams, Columns and Slabs.

Note that the plan should be drawn, such that the coordinate of Top Left corner should be located / shifted (in case of existing drawing) at 0,0 as shown below.



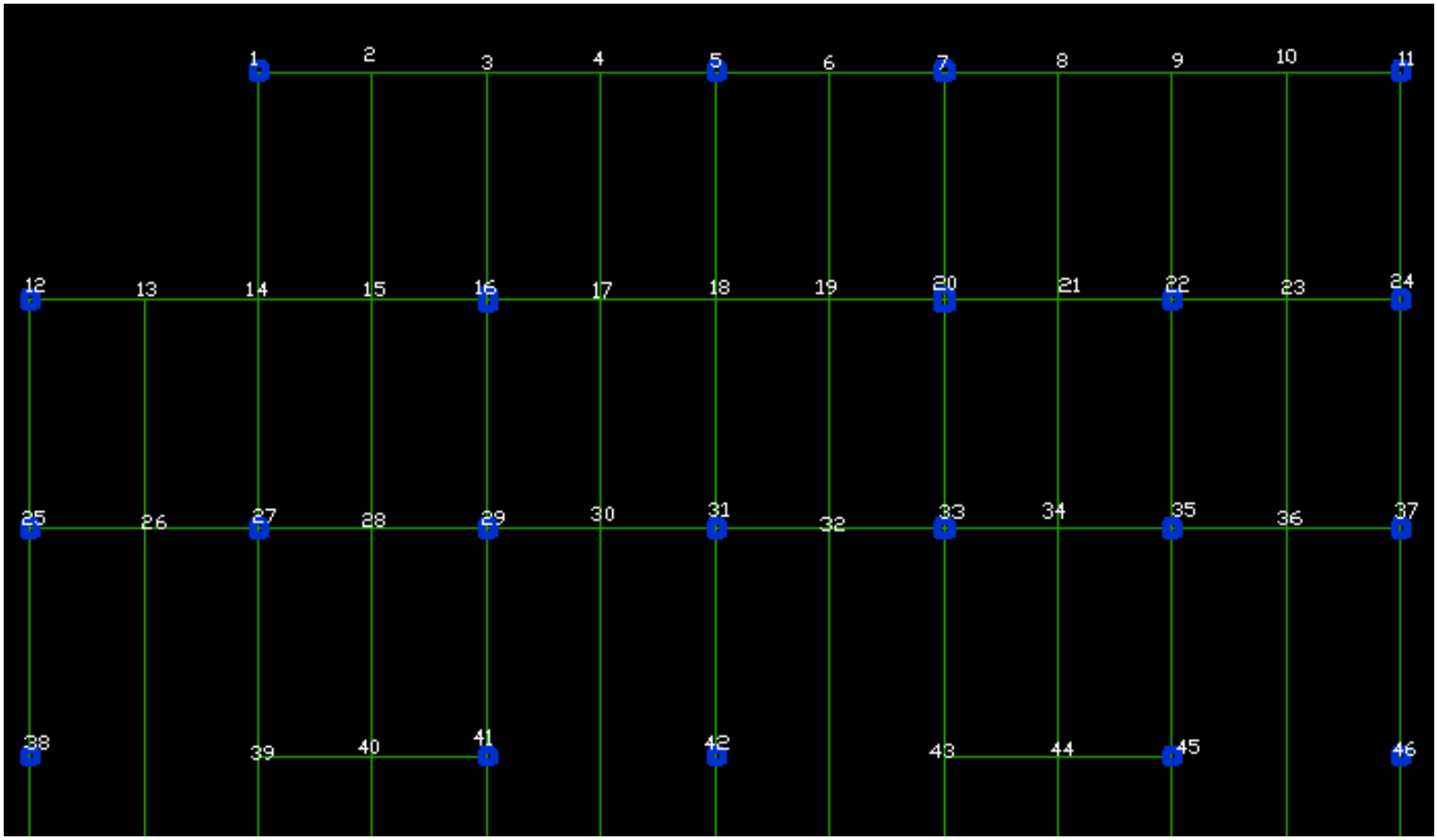
Shown below is a Typical Plan in AutoCAD :



The Layers are explained as follows:

JOINTS

A Joint represents a column location or an intersection between 2 beams.
All Joint Numbers should be in the Layer JOINTS
Draw text using 'Single Line Text' option in AutoCAD.
Joint Nos should not be repeated.
Joints should be Serially Numbered.
Joint Nos should not have any Prefix.
If a Joint No is deleted, then the consecutive joint nos should be serially Re-Numbered.
However a Joint can be added at any time by giving the Joint number as last Joint No. + 1

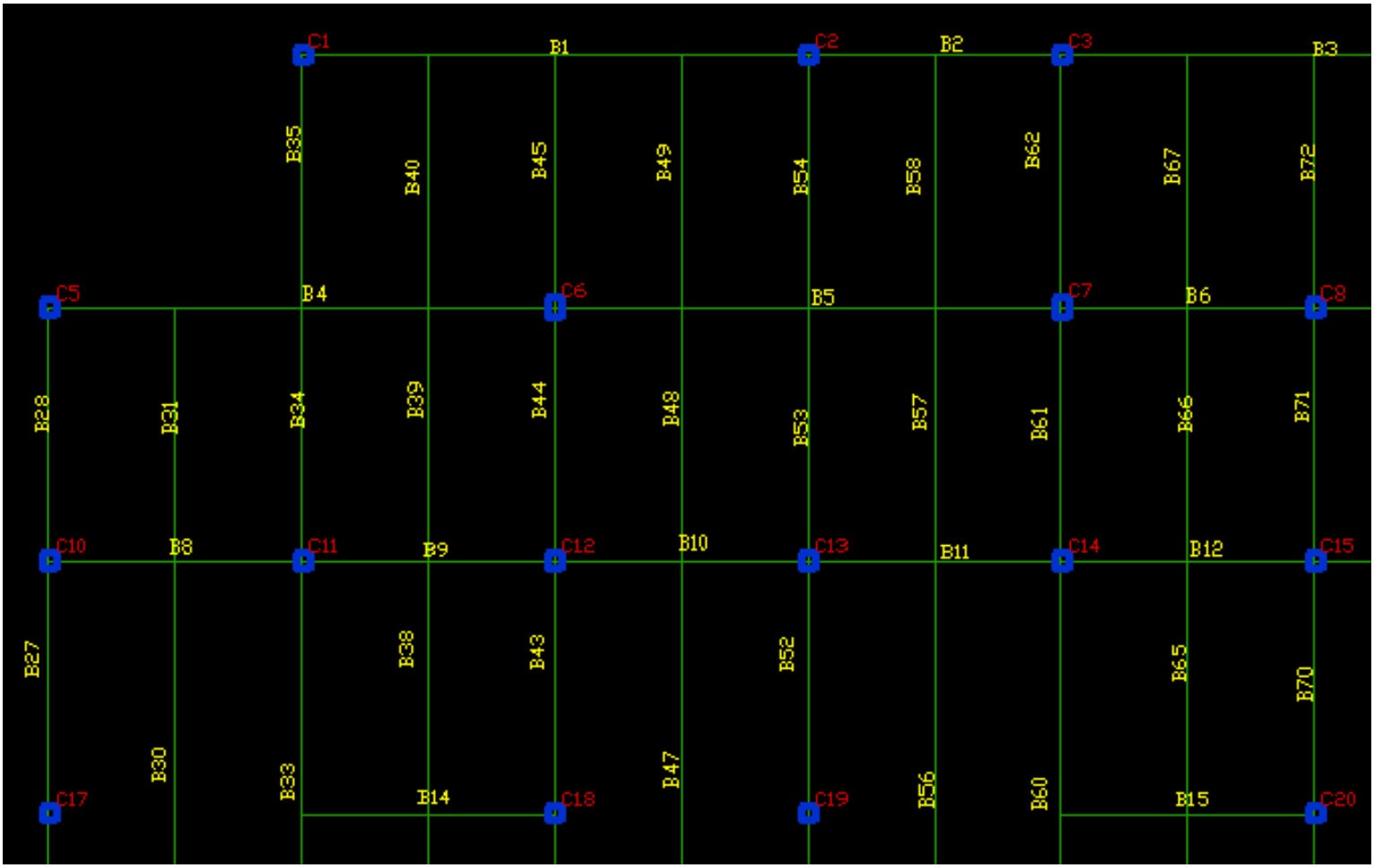


BEAM

All Beam Lines should be drawn under Layer CEN.
Only the Beam Centre line is to be drawn.
Beams to be drawn at 0 or 90 degrees only.
Inclined Beams are not permitted.
Keep "ORTHO" Option ON while drafting.
Every Beams should be a complete line touching Beam /Column Centre.
Every line in layer 'CEN' will be considered as a beam.
Beam Width will not be scanned from AutoCAD Drawing.
User to indicate Beam Width using Beam Option.
If a Beam is deleted, then the consecutive Beam Nos should be serially Re-Numbered.
However a Beam can be added at any time by giving the Beam number as last Beam No. + 1

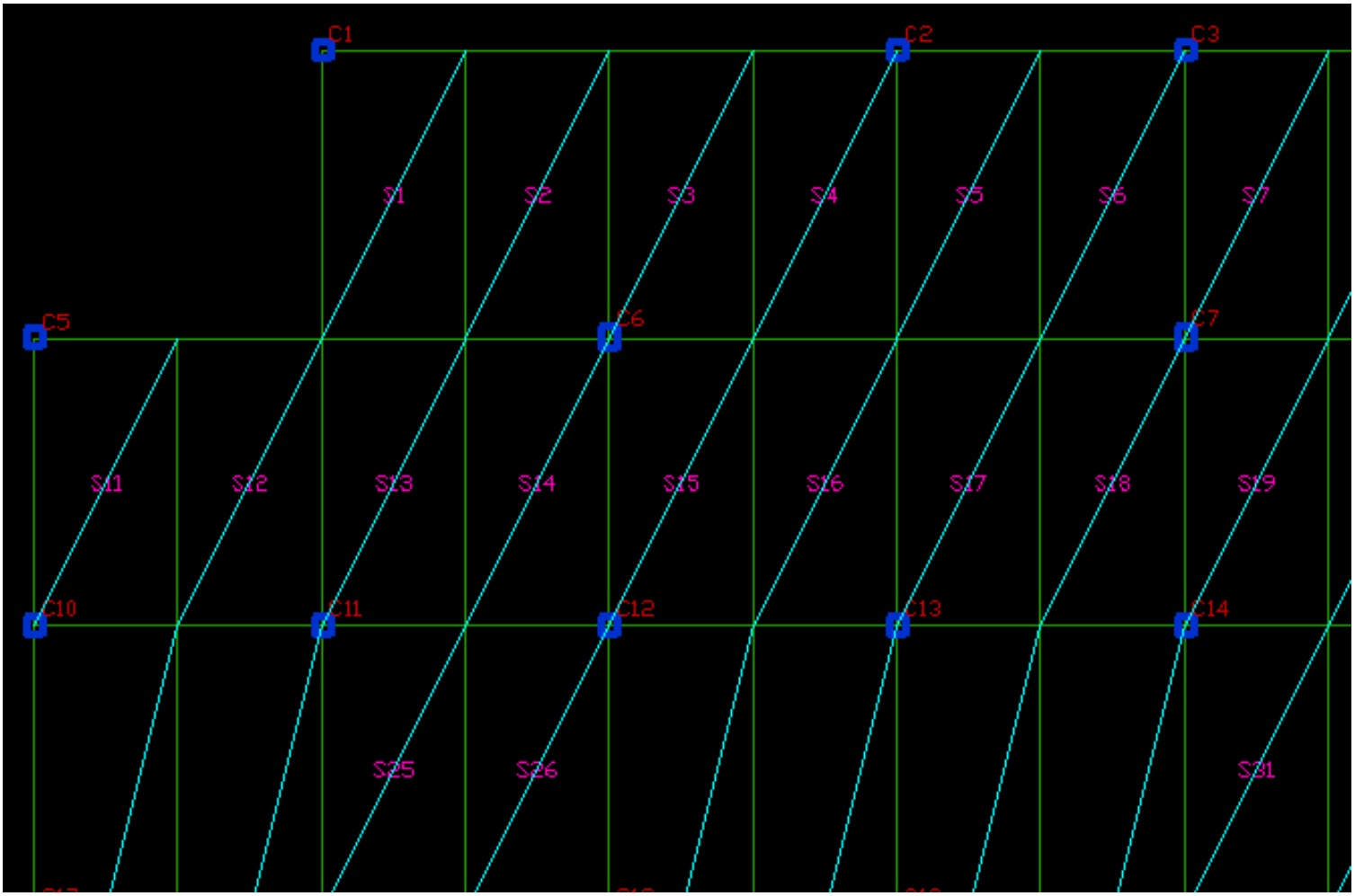
BEAM NUMBERS

All Beam Numbers should be in the Layer BEAMTEXT.
Draw text using 'Single Line Text' option in AutoCAD.
The angle of Inclination of Beam No's should be the same as the Beam.
(ie. If the Beam is inclined at an angle of 90 degrees, the text of the beam should also be inclines at 90 degrees.
Beam nos should be as close as possible to the centre of the Beam Line.
Beam Nos should not be repeated.
Beams should be Serially Numbered.
Beam Nos should be prefixed with a "B" (ie. B1, B2)



SLAB

In Order to mark the Extent of Slab, a Diagonal Line should be drawn from left bottom corner to right top corner of Slab as shown below.
The Diagonal Lines are to be drawn in the layer SLAB
Diagonal Lines should be drawn intersecting Beams or Columns.
If a Slab is deleted, then the consecutive Slab Nos should be serially Re-Numbered.
However a Slab can be added at any time by giving the Slab number as last Slab No. + 1



SLAB NUMBERS

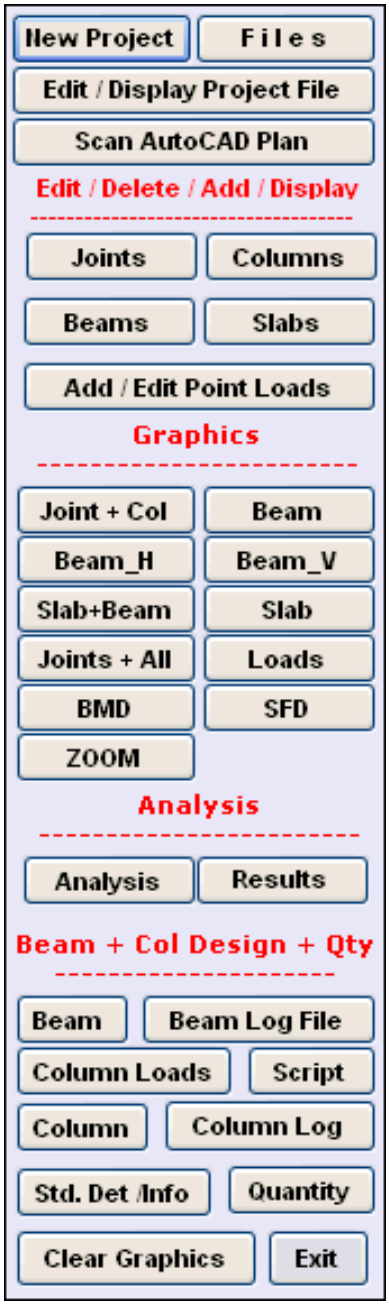
All Slab Numbers should be in the Layer SLABTEXT.
The Slab Text (No.) to be drawn near to the centre of the Slab .
Draw text using 'Single Line Text' option in AutoCAD.
Slab Nos. should not be repeated.
Slabs should be Serially Numbered.
Slab Nos should be prefixed with a "S" (ie. S1, S2)
Slab Text shall not be inclined.
It should be drawn at zero degrees.

COLUMN NUMBERS

All Column Numbers should be in the Layer COLUMNTEXT.
Column Nos should be as marked near its Joint.
Draw text using 'Single Line Text' option in AutoCAD.
Column Nos should not be repeated.
Columns should be Serially Numbered.
Column Nos should be prefixed with a "C" (ie. C1, C2)
Column Size will not be scanned from AutoCAD Drawing.
User to indicate Column Size in Column Option.
If a Column is deleted, then the consecutive Column Nos should be serially Re-Numbered.
However a Column can be added at any time by giving the Column number as last Column No. + 1

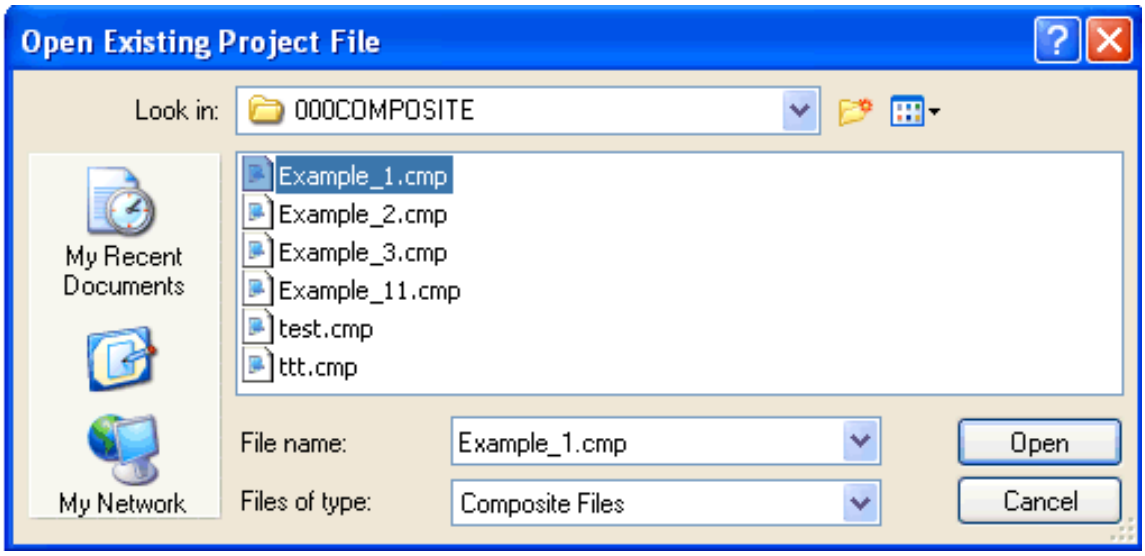
- **Once the drawing is completed, save the drawing in AutoCAD's DXF Format.**

- **Now Start COMPOSITE.**

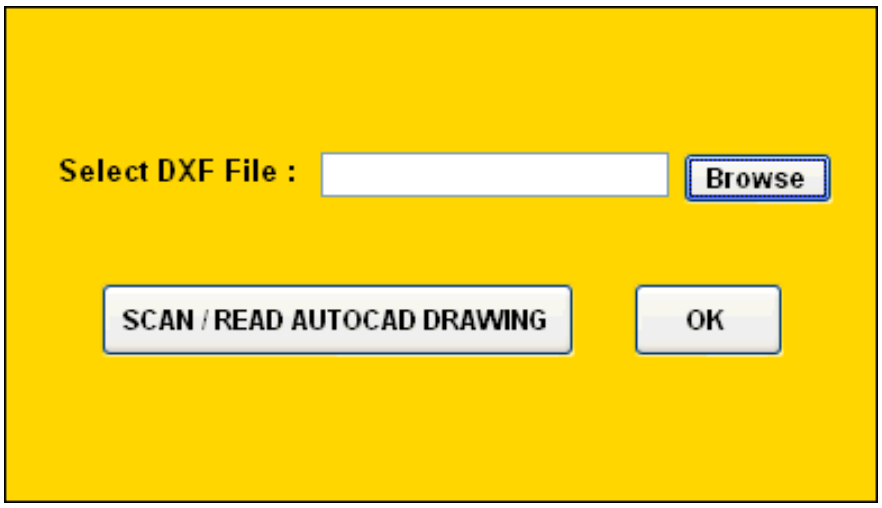


- **When Program starts, the graphics above is displayed.**

  Click the " Scan AutoCAD Plan" option in the MENU bar.
  The following window will open.



- **Now select Example _1.cmp File.**
  Following Graphics will be displayed.



- **Click on browse to select the AutoCAD Drawing.**
  Next click on "Scan/ Read AutoCAD Drawing" button.

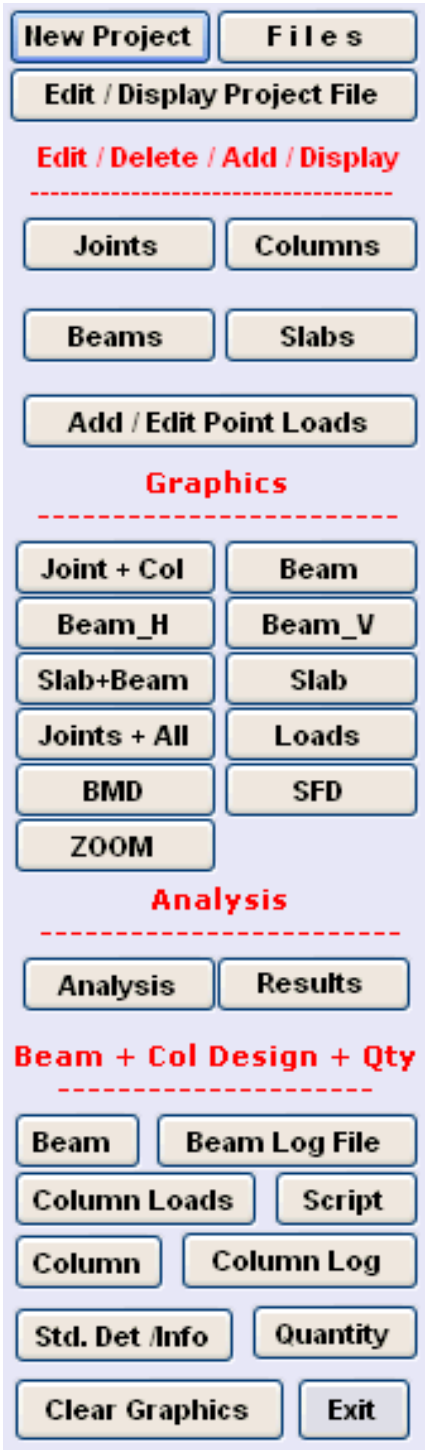
  The Imported data shall be verified using Edit/ Delete/ Add/ Display Joint, Beam, Column and Slab as well as Graphics Option of Joint, Beam, Column and Slab.
  The Left hand side Joint No, Right hand side Joint No and Span of Beams should be throughly checked using Add/ Edit Beam Option.
  The Graphic Display and AutoCAD Drawing should appear same.
  Do not perform analysis, if there is any discrepancy in drawings shown in various Graphic Options and AutoCAD.

STEP NO. 2 IS OVER.

go back page top print

LEARN COMPOSITE STEP BY STEP

STEP NO. 2 : Automatic Joint Number Creation

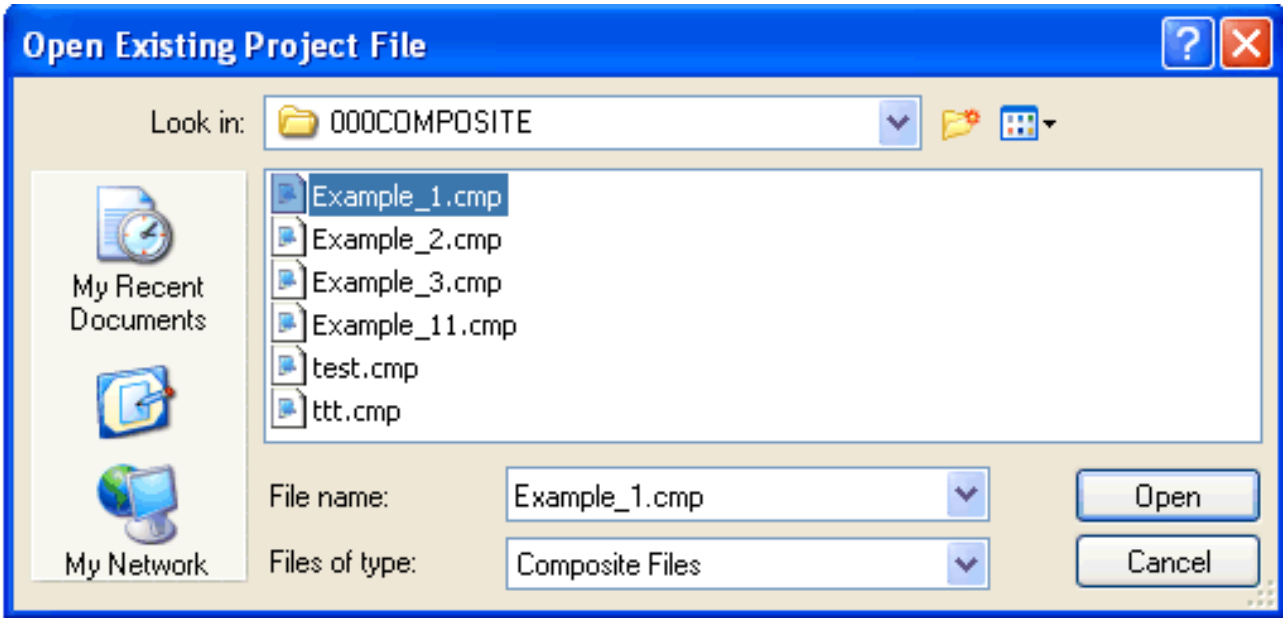


When Program starts, the Menu above is displayed. Under the Graphics Heading following options are displayed.

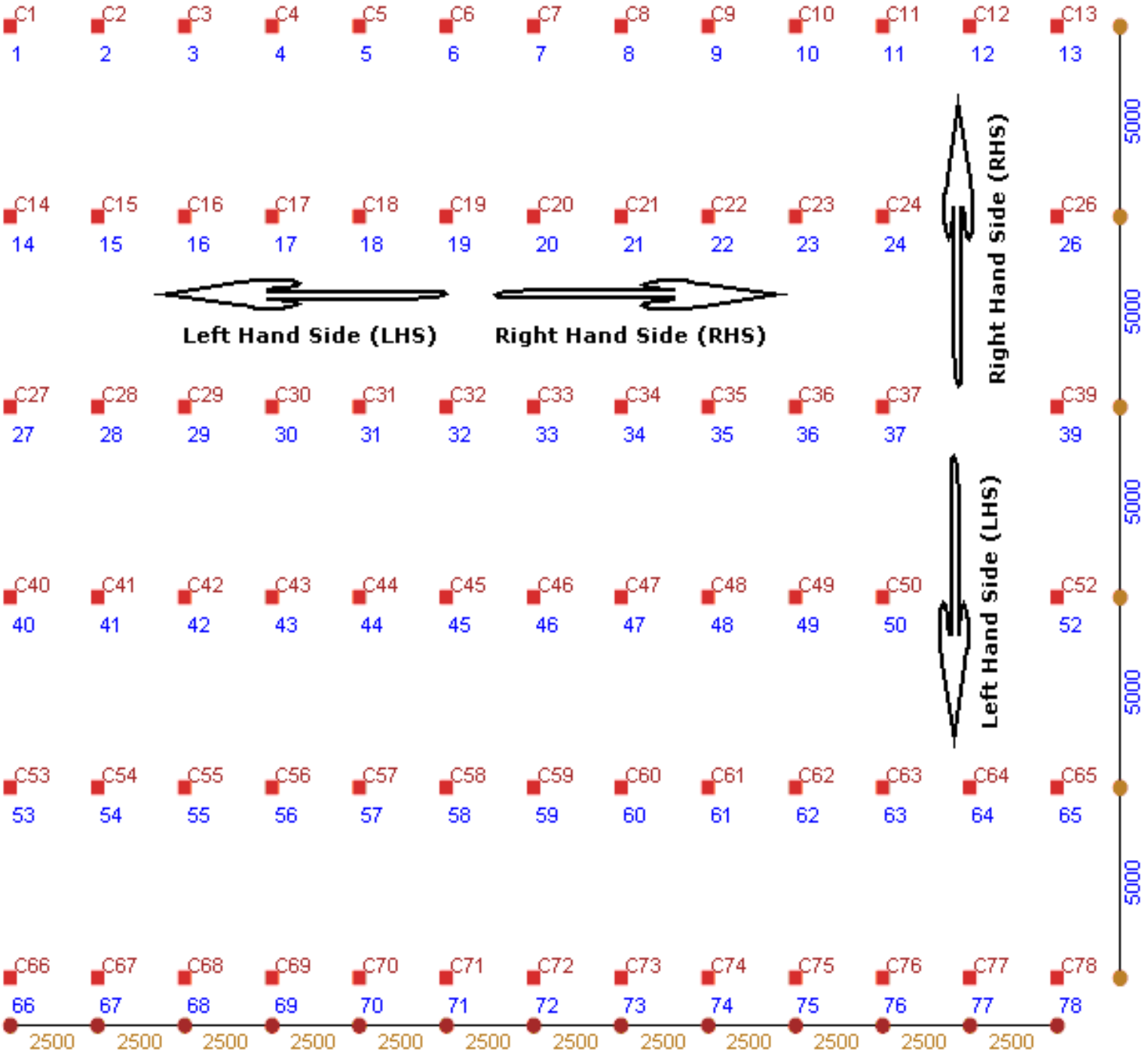
- **Joint + Column Nos**
- **Beam**
- **Beam_H (For Display of Only Horizontal Beams).**
- **Beam_V (For Display of Only Vertical Beams).**
- **Slab + Beam (Beams, Slabs & Columns are displayed).**
- **Slab (Only Slabs & Columns are displayed).**
- **Joints + ALL (For Display of Joints, Columns, Beams & Slabs)**
- **Loads (Display of Slab, Point Loads & Reactions from Secondary Beams, to be used after Analysis, and Design options have been successfully Run).**
- **BMD (Display of Bending Moment Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.**
- **SFD (Display of shear Force Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.**
- **Zoom (Display of part of Floor Plan under Selection).**
- **Continuity (Display of Beams Marked as Continuous.)**

Now Click on " Joint + Col " option.

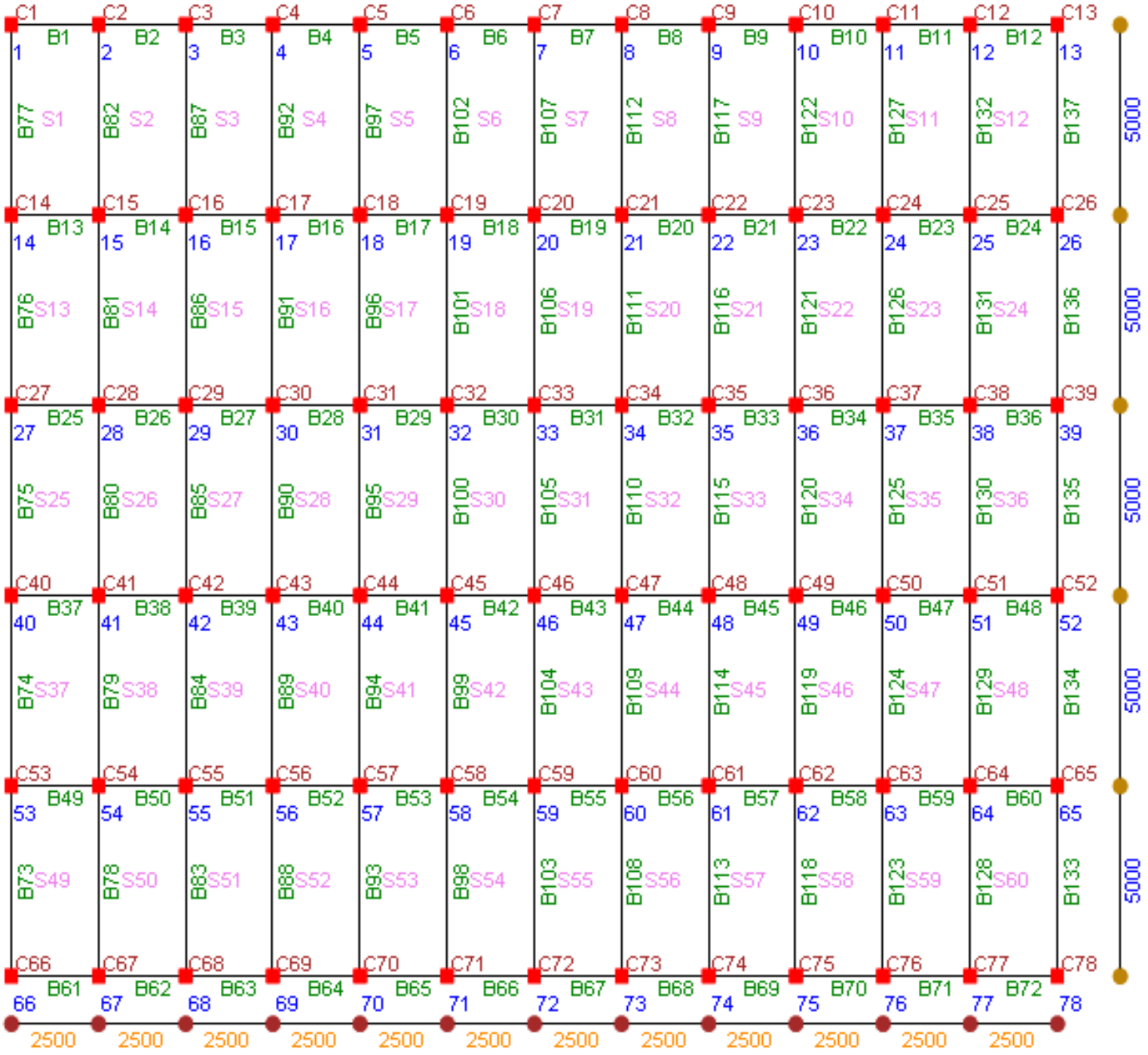
Following Graphics is displayed.



Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.

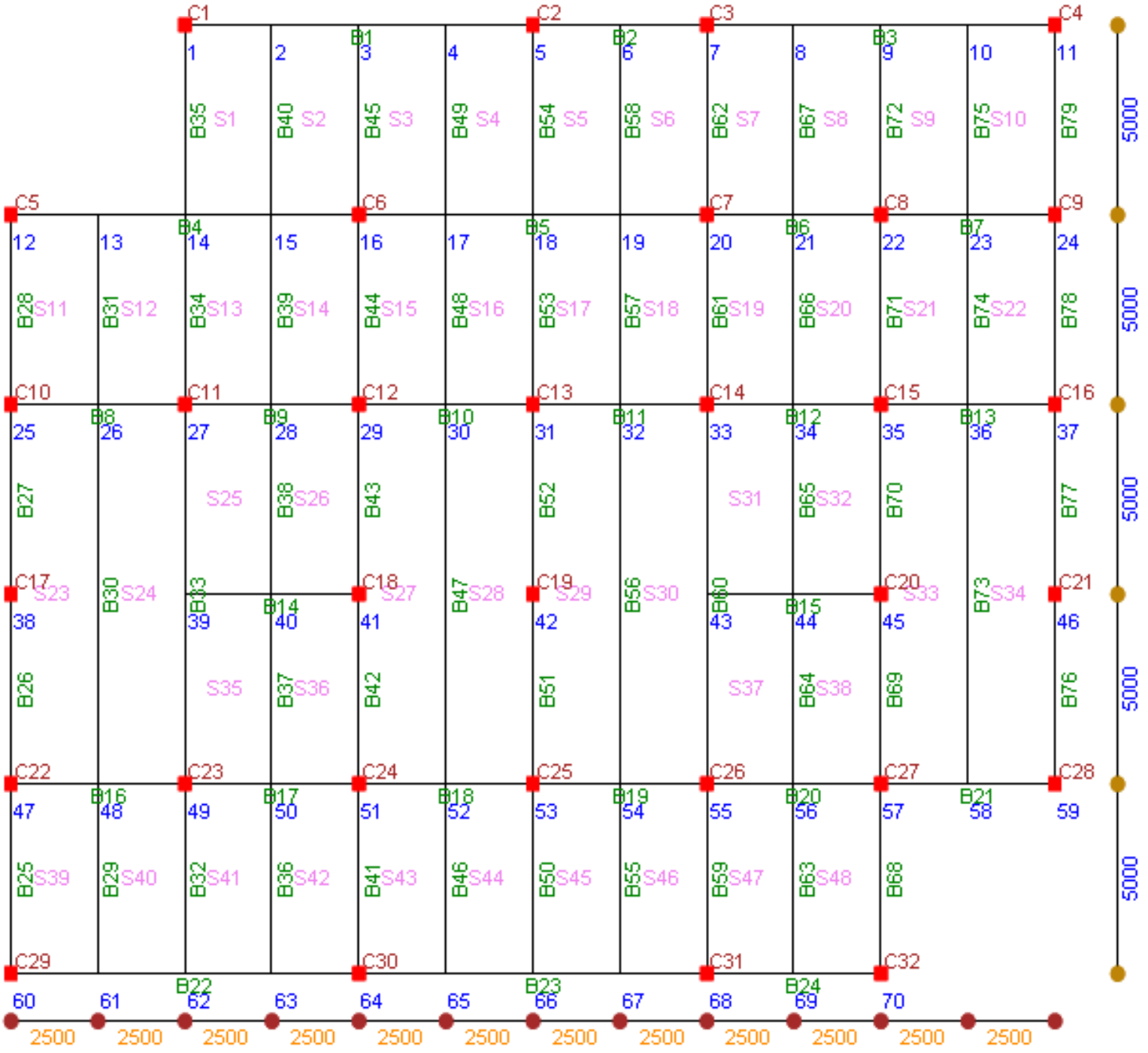


- **Note that Joints Numbers (Including X & Y Co-Ordinates) and Columns are created and displayed automatically at all the intersections of vertical & horizontal grids. Some of the Joint numbers may not be required. A Joint represents a column location or an intersection between 2 beams. The beams are represented by its location in the form of Right Hand Side (RHS) & Left Hand Side (LHS) Joint numbers. The slabs are represented by TOP LEFT & RIGHT BOTTOM joint numbers.**
  Additionally we have displayed above RHS and LHS conventions for Horizontal & Vertical Orientations in the form of Arrows.
  Now click the " Joints + ALL " button, following Graphics will be displayed.



AUTOMATICALLY GENERATED FLOOR PLAN

- **Note that Columns are shown at all the Joints, and Beams are spanning between these columns.**
  This is different than what is the actually required Floor Plan. The intended actual floor plan is reproduced below.



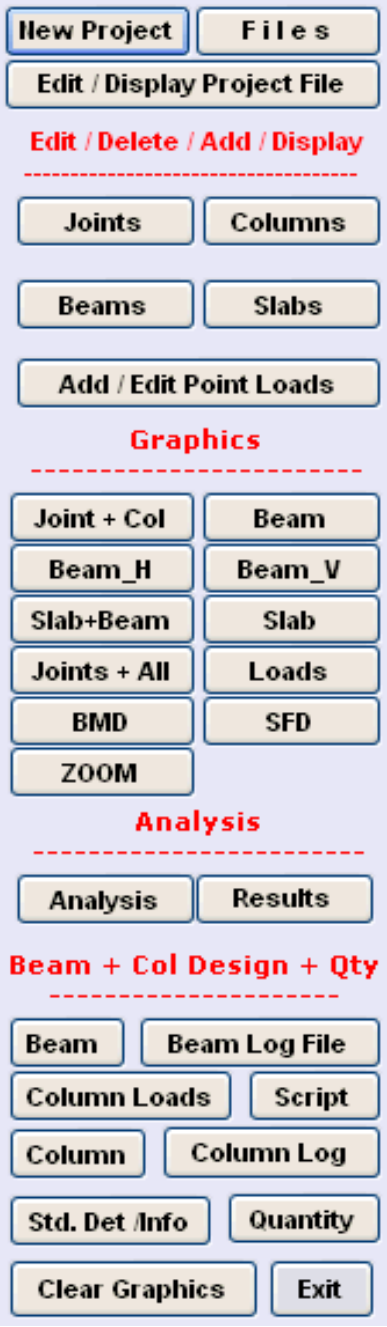
ACTUAL REQUIRED FLOOR PLAN

- **Our Actual Composite Floor Plan has 48 numbers of Slabs, 32 numbers of Columns & 79 numbers of Beams. The automatic generated plan has 60 numbers of Slabs, 78 numbers of Columns and 137 numbers of Beams. Hence we have to delete these extra Slabs, Columns and Beams along with their appropriate Joint numbers.**
  Let us delete these parameters in next step.

STEP NO. 2 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 3 : Delete Un-Wanted Joints

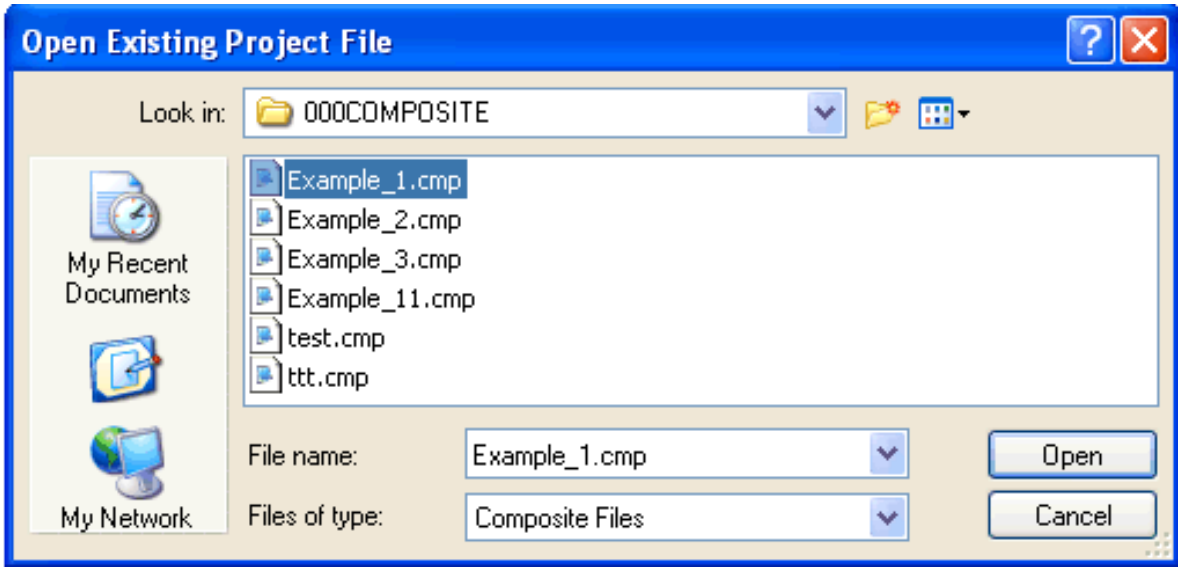


When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Add / Edit Point Loads**

Now Click on " Joints " option.

Following Graphics is displayed.



Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.

DISPLAY / EDIT / ADD JOINT NUMBERS

Note : Origin (0,0) is at Top Left Corner.

Joint No. 1

X Co_Ordinate 0

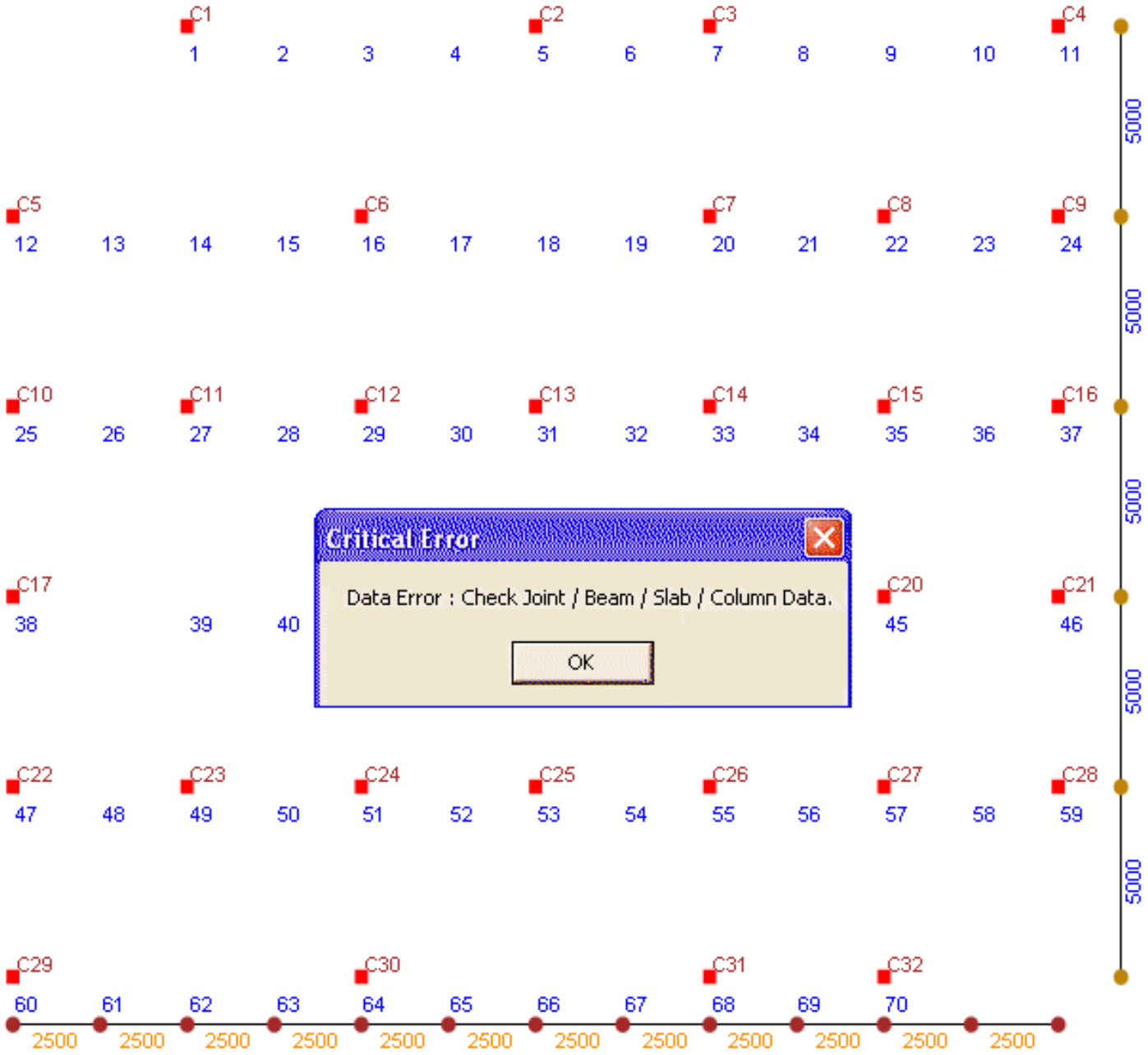
Y Co_Ordinate 0

Record No. : 1 of 78

Read Me | Prev | Next | Paste | Copy | Last | 1 st | Copy All | Update | Go To Rec | Remove | Add Record | Clear | Print | O K

Joint No.	X Co-Ordinate in MM	Y Co-Ordinate in MM
46	15000	15000
47	17500	15000
48	20000	15000
49	22500	15000
50	25000	15000
51	27500	15000
52	30000	15000
53	0	20000
54	2500	20000
55	5000	20000
56	7500	20000
57	10000	20000
58	12500	20000
59	15000	20000
60	17500	20000
61	20000	20000
62	22500	20000
63	25000	20000
64	27500	20000
65	30000	20000
66	0	25000
67	2500	25000
68	5000	25000
69	7500	25000
70	10000	25000
71	12500	25000
72	15000	25000
73	17500	25000
74	20000	25000
75	22500	25000
76	25000	25000
77	27500	25000
78	30000	25000

- **Here we have 78 Joint numbers. The Actual required are only 70. Hence we have to delete the extra joint numbers (78-70 = 8). Select Joint number 1 & press " Remove " button. Joint Number " 1 " is deleted. Now Click " UPDATE ", joints will be re-numbered. Now Select Joint number " 3 " & press " Remove " button. Joint Number " 3 " is deleted. Now Click " UPDATE ", joints will be re-numbered. Similarly select other Joints & press remove & update button till Joint number 70 is reached.**
  You should keep a copy of Required Floor Plan & Automatically generated Floor Plan in front of you.
  Always update after each remove. In this way complete joint numbers can be updated. Now Click on " Joint Nos " option under the Graphics Caption. You will see the revised Joint number Layout as displayed below.



- **Note the Critical Data Error " Check Joint / Beam / Slab / Column data ". What it means is that you have not deleted corresponding Beam (s) / Slab (s) / Column (s) which refers to deleted Joints.**

The " Copy All " button copies data from the selected ROW to all the ROWS. Later on a user can change the values selectively.

Use Copy & Paste Button to copy & paste values to different rows, in case the values are not same.

The " Prev ", " Next ", " Last ", " 1 st ", & " Go to Rec " Buttons are for displaying / Focusing the cursor on Previous, Next, First or required Record Number.

The " Clear " Button clears all values.

The " Print " Button is for printing of values from the Table. Use laser OR Inkjet Printer.

The " Add Record " button is very important one. If a user has deleted any joint by

mistake, than he can easily add the record back by pressing this button. However the Joint number added will be the last + one number. Suppose after deleting a joint, total joints left are 99, then if "Add Record" button is pressed, the next record displayed will be joint number 100.
Remember that a user cannot give joints " X " and " Y " Co-Ordinates outside the boundary limit as set out in the project file (Refer Step No. 1). In our " Example_1 " Project the maximum width is 30000 and maximum length is 25000.

If a user is not comfortable with automatic generation of joint numbers (Co-Ordinates) , then he can use Add Record option to enter complete joint data & corresponding Co-Ordinates manually by first clearing the old data by pressing " Clear " button. Similarly Add Record button can be used for effectively where a floor plan is rather complex, having lots of internal secondary beams in either direction.

- **Now click the " Read Me " button, the following important messages are displayed.**

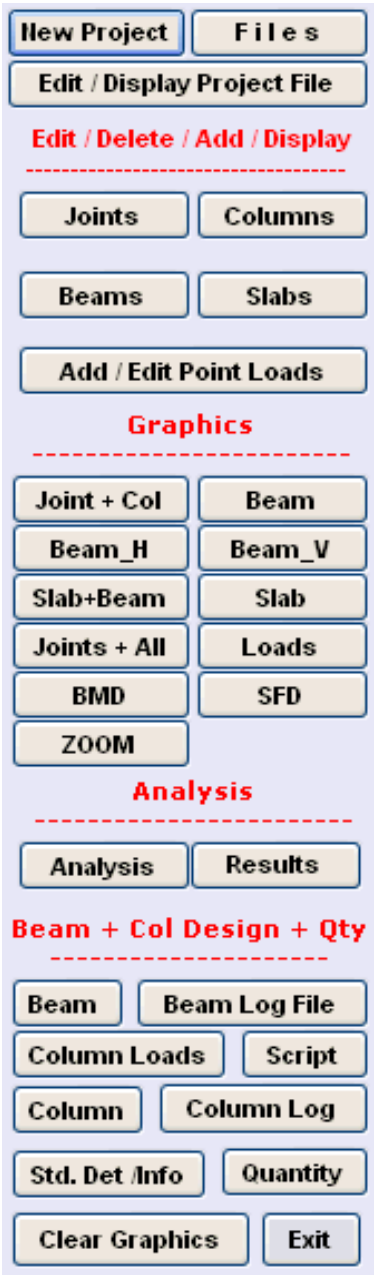
1. Origin (0,0) is at Top Left Hand Corner. Co-Ordinates Cannot be Negative.
2. There shall not be any difference in Maximum Horizontal & Vertical Distance between Project File & Joint File.
3. Joint Number should start with 1 & not 0.
4. Joints Numbers cannot be repeated.
5. Co-ordinates cannot be repeated.
6. Max. Joints Number = Max. Record Number.
7. Joints should be Serially Numbered.
8. Use Add Button to Append Record.
9. Use Update Button to Re-Number & Save Your Work.
10. In case any Joint # is Deleted or Edited then, Do not Forget to Edit Corresponding Column, Beam & Slab to reflect above change.

- **Now we have come to the end of Step # 3.**
  In the next step we will delete the un-wanted Beams.

STEP NO. 3 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 4 : Delete & Edit Beams

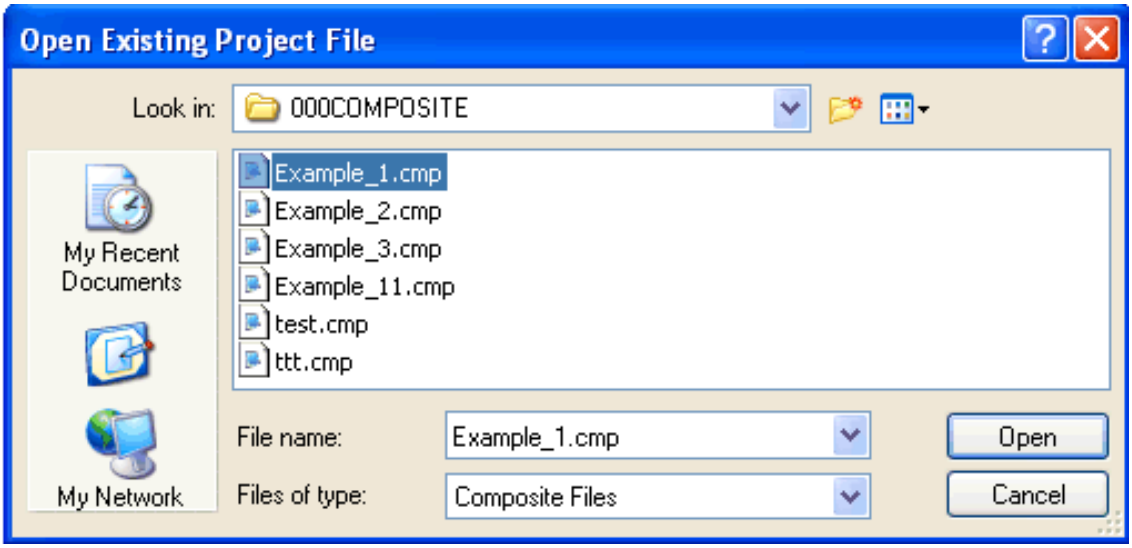


When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Add / Edit Point Loads**

Now Click on " Beams " option.

Following Graphics is displayed.



- **Now select " Example _1 File & Press Open Button.**
  Following Graphics will be displayed.

DISPLAY / EDIT / BEAM DETAILS

Beam #	LHS Joint #	RHS Joint #	Type	Section	Mas Ht	Mas Thk	Extra UDL	Axial
B118	75	62	MB	MB-200				
B119	62	49	MB	MB-200				
B120	49	36	MB	MB-200				
B121	36	23	MB	MB-200				
B122	23	10	MB	MB-200				
B123	76	63	MB	MB-200				
B124	63	50	MB	MB-200				
B125	50	37	MB	MB-200				
B126	37	24	MB	MB-200				
B127	24	11	MB	MB-200				
B128	77	64	MB	MB-200				
B129	64	51	MB	MB-200				
B130	51	38	MB	MB-200				
B131	38	25	MB	MB-200				
B132	25	12	MB	MB-200				
B133	78	65	MB	MB-200				
B134	65	52	MB	MB-200				
B135	52	39	MB	MB-200				
B136	39	26	MB	MB-200				
B137	26	13	MB	MB-200				

Record No. : 1 of 137

Beam # B1 LHS Joint # 1 RHS Joint # 2 Beam Type MB Beam Section MB-200

Net Height of Masonry Wall in M | Thickness of Wall in MM | Additional UDL on Beam in T/M | O K

Un-Factored Axial Load in Tons | Read Me | Prev | Next | Copy | Paste | Span 2500 | Clear

1 st | Last | Copy All | Update | Go To Rec | Remove | Add Record | Print

(All Loads Shall be Un-Factored)

- **Here we have 137 numbers of Beams. Actual required are only 79 numbers of Beams (Refer Step No. 1 - Actual Required Floor Plan). Go down to the last beam number B137 and press " Remove " button. You will notice that Beam B137 is deleted. Similarly delete the next beam, till you reach Beam number B79. I am deleting from the end (Last Beam) for ease of editing, you can even start from the beginning or from any other beam number.**
  Click " Update " button. This will re-number all the beams if required.
  Now let us start editing the RHS & LHS Joint numbers of Beams. Go to first Beam B3 & Select it (Click with Cursor). Now concentrate on the Text Boxes below. Beam # will be shown as B3. LHS Joint # is shown as 3 and RHS joint # is shown as 4. Change LHS Joint # to 7 & RHS Joint # to 11 by editing the text box.
  Again select Beam # B5 or Click " Next " button. LHS Joint # is shown as 5, change it to 16. RHS Joint # is shown as 6, change to 20. Similarly edit the rest of Beam's LHS & RHS Joint numbers as required by our Actual Floor Plan.

  In case you would like to EXIT program after partial editing, first use " Update " button to save your work & then click " OK " button. The program will ask you about exiting, click Yes & quit.

- **All other Beam Parameters Viz; Beam Type, Beam Section, Masonry Height, Masonry Thickness, Any Extra UDL and Axial Load (Compressive OR Tensile { + / -}) can be Added / Edited for individual Beams by clicking at respective Text Boxes. All Loads shall be Un-Factored. Beam Span is displayed in Yellow Text Box.**

- **Now click the " Read Me " button, the following important messages are displayed for guidance.**

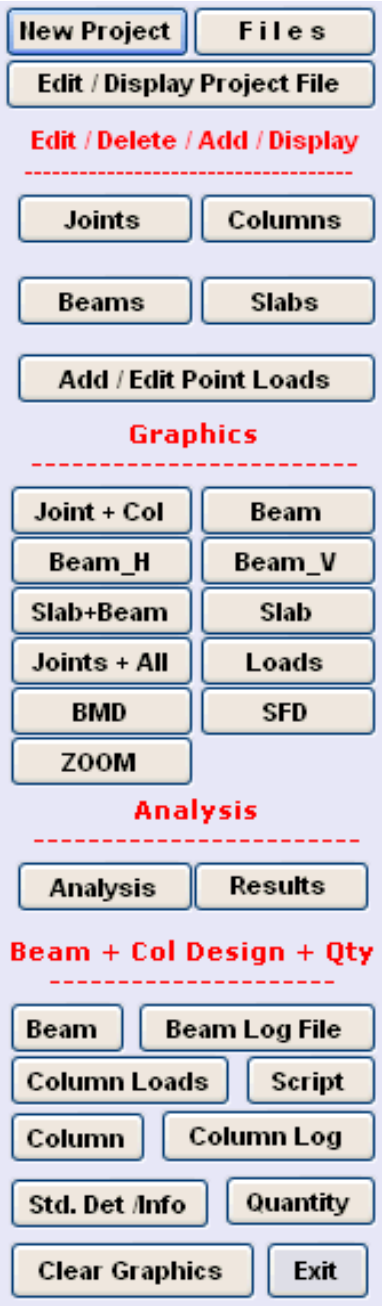
  1. Add Joint Details before Beams.
  2. Beam Number should start with 1 & not 0.
  3. Beam Numbers cannot be repeated.
  4. Beam LHS & RHS Joint #s cannot be repeated.
  5. 2 Types of Beams can be Designed, Viz, ISMC & ISMB.
  6. Max. (LHS or RHS) Beam Joint # cannot > Max. Joint File #.
  7. Use Add Button to Append Record.
  8. Use Update Button to Re-Number & Save Your Work.
  9. Max. Beam Number = Max. Record Number.
  10. Beam Nos. Shall be Numbered Serially.
  11. Beam LHS OR RHS Joint Number Cannot < = 0.0.
  12. LHS : Left Hand Side, RHS : Right Hand Side.
  13. If Beam is Vertical then, LHS Y-Co Ordinate > RHS Y-Co Ordinate.
  14. If Beam is Horizontal then, LHS X-Co Ordinate < RHS X-Co Ordinate.
  15. - Ve Axial Load Indicates Tension.
  16. All Loads Shall be Un-Factored.

- **Now we have come to the end of Step # 4.**
  In the next step we will Delete and Edit un-wanted Columns.

STEP NO. 4 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 5 : Delete & Edit Columns

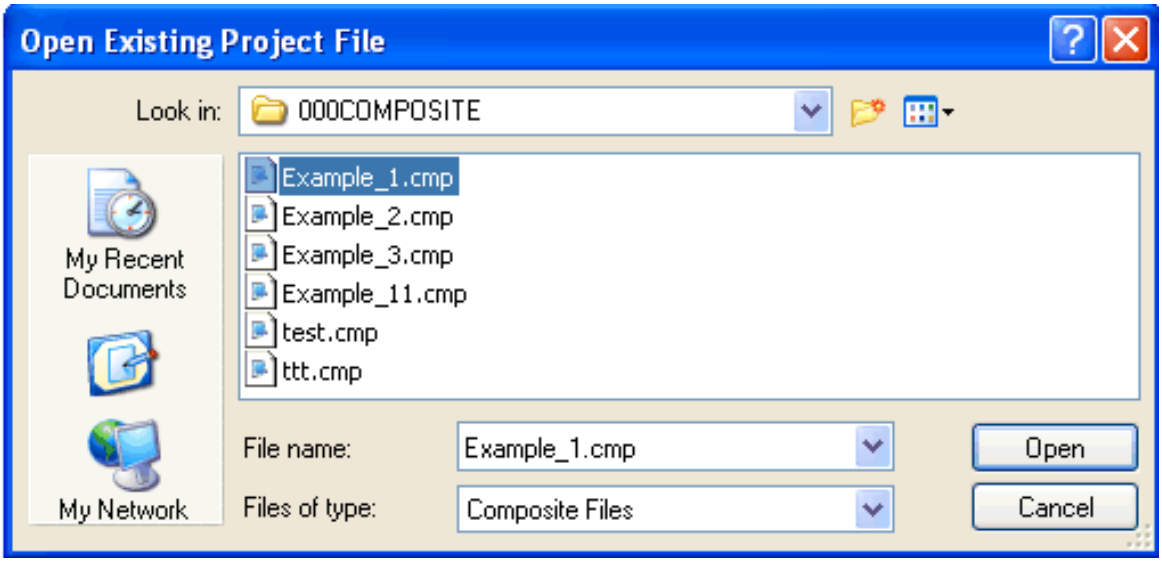


When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Add / Edit Point Loads**

Now Click on " Columns " option.

Following Graphics is displayed.



- **Now select " Example _1 File & Press Open Button.**

DISPLAY / EDIT / BOX COLUMN DETAILS

Column #	Joint #	BM Z-Z	BM Y-Y	Type	Section	Leff Z-Z	Leff Y-Y	Max SF	Ext Load
C11	27	0.697	3.138	2MB T/T	MB-300	3	3	1	
C12	29	0.697	0	2MB T/T	MB-300	3	3	1	
C13	31	0	0	2MB T/T	MB-300	3	3	1	
C14	33	0.697	3.138	2MB T/T	MB-300	3	3	1	
C15	35	0.697	0	2MB T/T	MB-300	3	3	1	
C16	37	2.751	0	2MB T/T	MB-300	3	3	1	
C17	38	0	0	2MB T/T	MB-300	3	3	1	
C18	41	2.054	0	2MB T/T	MB-300	3	3	1	
C19	42	0	0	2MB T/T	MB-300	3	3	1	
C20	45	2.054	0	2MB T/T	MB-300	3	3	1	
C21	46	0	0	2MB T/T	MB-300	3	3	1	
C22	47	2.751	0	2MB T/T	MB-300	3	3	1	
C23	49	0.697	3.138	2MB T/T	MB-300	3	3	1	
C24	51	0.697	0	2MB T/T	MB-300	3	3	1	
C25	53	0	0	2MB T/T	MB-300	3	3	1	
C26	55	0.697	3.138	2MB T/T	MB-300	3	3	1	
C27	57	0.031	0.661	2MB T/T	MB-300	3	3	1	
C28	59	2.085	1.374	2MB T/T	MB-300	3	3	1	
C29	60	3.872	1.374	2MB T/T	MB-300	3	3	1	
C30	64	0	2.036	2MB T/T	MB-300	3	3	1	
C31	68	2.482	2.036	2MB T/T	MB-300	3	3	1	
C32	70	1.389	1.374	2MB T/T	MB-300	3	3	1	

Record No. : 1 of 32

Box Column # C1 | Joint # 1 | UnFactored M Z_Z in t-m 3.872 | UnFactored M Y_Y in t-m 1.376

Box Column Type 2MB T/T | Box Column Section MB-300 | UnFactored External Load in ton

Eff. length along Z_Z in M 3 | Eff. length along Y_Y in M 3 | UnFactored Maximum SF in Tons 1

Read Me | Prev | Next | Last | 1 st | Copy | Paste | Col. Z-Z / Y-Y Axis

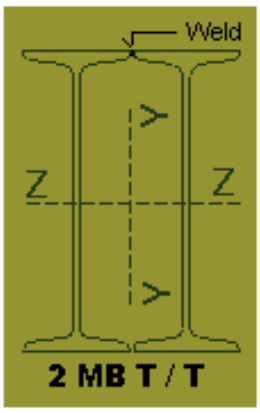
Copy All | Update | Go To Rec | Remove | Add Record | Print | Clear | O K

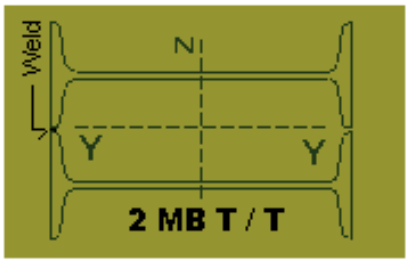
- **You will notice that here we have 78 numbers of Columns. Actual required are only 32 numbers of Columns. (Refer Step No. 1 - Actual Required Floor Plan). Go down to the last Column number C78 and press " Remove " button. You will notice that Column C78 is deleted. Similarly delete the next Column, till you reach Column number C32. I am deleting from the end (Last Column) for ease of editing, you can even start from the beginning or from any other Column number.**
  Click " Update " button. This will re-number all the Columns if required.
  Now let us start editing the Joint numbers of Columns. Go to first Column & Select it (Click with Cursor), or click the " 1 st " button.
  Now concentrate on the Text Boxes below. Column # will be shown as C1. Joint # is shown as 1, which is ok. Again select Column # C4 . Joint # is shown as 4, change it to 11. Similarly edit the rest of Column's Joint numbers as required by our Actual Floor Plan (Refer Step 2).

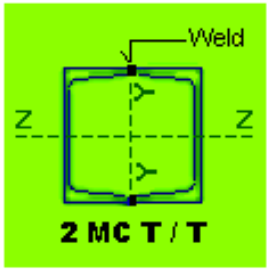
  In case you would like to EXIT program after partial editing, first use " Update " button to save your work & then click " OK " button. The program will ask you about exiting, click Yes & quit.

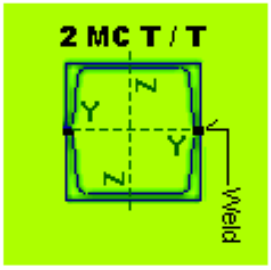
- **All other Box Column Parameters Viz; Column Type, Column Section, Lxx, Lyy, BM_XX & BM_YY can be Added / Edited for individual Columns by clicking at respective Text Boxes. Note that Box Column Type is either 2 MC Toe to Toe OR 2 MB welded Toe to Toe without any Gap.**
  Column Section is either MC or MB.
  External Load is the additional Vertical Load (+ / -) which can be applied to Column.
  Maximum value of SF from both two sides can be entered.
  Click Display Col Z-Z/Y-Y Axis button, following window will open.

COLUMN ORIENTATION WITH RESPECT TO Z-Z & Y-Y AXIS









Under normal orientation Z-Z is Horizontal & Y-Y is Vertical Axis.
If section is rotated, Z-Z Axis will become Vertical & Y-Y will become Horizontal.
Hence user to give MZZ & MYY, LZZ & LYY parameters accordingly.

- **Now click the " Read Me " button, the following important messages are displayed for guidance.**

1. Column Number should start with 1 & not 0.

2. Column Numbers cannot be repeated.

3. Column Joint #s cannot be repeated.

4. All Columns are Deemed Box Columns.

5. Max. Column Joint # cannot > Max. Joint File #.

6. All Columns are assumed as Positioned Centered with respect to Beam / Wall from either Sides.

7. Use Add Button to Append Record.

8. Max. Column Number = Max. Record Number.

9. Columns Shall be Numbered Serially.

10. Use Update Button to Re-Number & Save Your Work.

11. Column Joint Number cannot be < = 0.0.

12. BM Z-Z, BM Y-Y, Leff Z-Z, Leff Y-Y are Bending Moments and Column Effective Lengths about Z-Z / Y-Y axis.

13. Externally Applied Load (+/-) on Column Could be due to Wind/Seismic/Truss/Gantry etc.

14. All Loads Shall be Un-Factored.

15. Under normal orientation Z-Z is Horizontal and Y-Y is Vertical Axis. If section is rotated,
Z-Z Axis will become Vertical & Y-Y will become Horizontal.

16. Hence user to give MZZ & MYY, LZZ & LYY Parameters accordingly.

17. Enter Maximum of SF from either direction.

18. MZZ & MYY are Moments due to Eccentricity of Connections.

19. Minimum Eccentricity at joints is taken as 70 MM + Half of Col. dim.

20. Eccentric Moments MZZ & MYY are calculated automatically by the program.

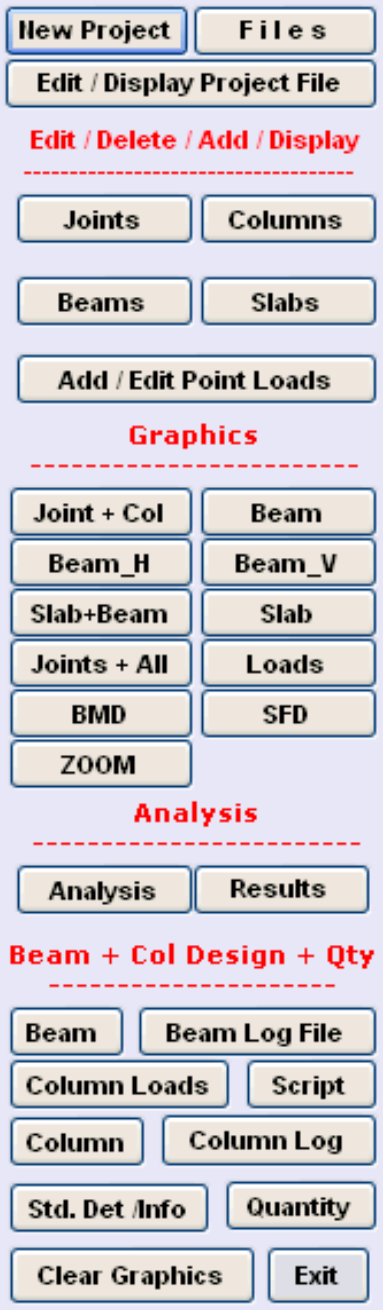
21. Note that there shall be no Bending Moments due to Wind or Earthquake on Frame (Col or Beam) as Structure is designed as Braced / Non-Sway against lateral loading by Using suitable Shear Wall or Cross Bracing System.

- **Now we have come to the end of Step # 5.**
  In the next step we will Delete and Edit un-wanted Slabs.

STEP NO. 5 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 6 : Delete & Edit Slabs

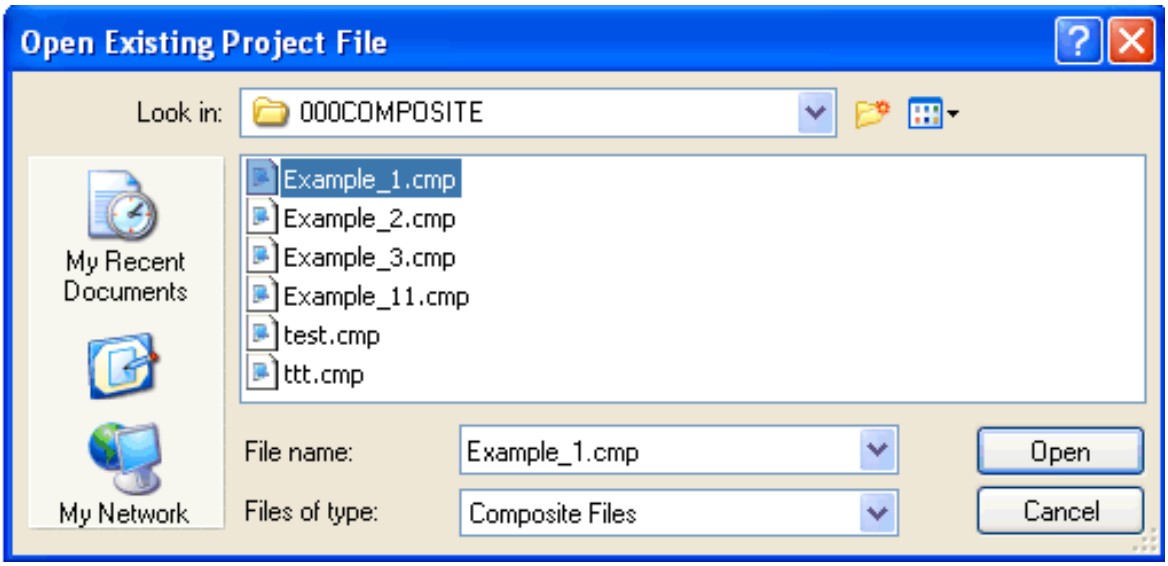


When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Add / Edit Point Loads**

Now Click on " Slabs " option.

Following Graphics is displayed.



Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.

DISPLAY / EDIT / ADD / FLOORING DETAILS

Slab #	Left Bottom Joint #	Right Top Joint #	Floor Dead Weight	Live Load	Span Dir
S27	51	30	430	0.50	X
S28	52	31	430	0.50	X
S29	53	32	430	0.50	X
S30	54	33	430	0.50	X
S31	43	34	430	0.50	X
S32	44	35	430	0.50	X
S33	57	36	430	0.50	X
S34	58	37	430	0.50	X
S35	49	40	430	0.50	X
S36	50	41	430	0.50	X
S37	55	44	430	0.50	X
S38	56	45	430	0.50	X
S39	60	48	430	0.50	X
S40	61	49	430	0.50	X
S41	62	50	430	0.50	X
S42	63	51	430	0.50	X
S43	64	52	430	0.50	X
S44	65	53	430	0.50	X
S45	66	54	430	0.50	X
S46	67	55	430	0.50	X
S47	68	56	430	0.50	X
S48	69	57	430	0.50	X

Record No. : 1 of 48

Slab # S1 | Left Bottom Joint # 14 | Right Top Joint # 2 | Wt of Flooring in Kg/M2 430

Unfactored Total Deck / Slab Dead Load

Unfactored LL on Slab in T/ M2 0.50 | Floor Spanning Along : X Axis

2500 x 5000

Read Me | Prev | Next | Copy | Paste | Last | 1 st | Slab Intensity in T/M2 0.93

Copy All | Update | Go To Rec | Remove | Add Record | Print | Clear | O K

- **Here we have 60 numbers of Slabs. Actual required are only 48 numbers of Slabs (Refer Step No. 1 - Actual Required Floor Plan). Go down to the last Slab number S60 & press**
  " Remove " button. You will notice that Slab S60 is deleted. Similarly delete the next Slab, till you reach Slab number S48. I am deleting from the end (Last Slab) for ease of editing, you can even start from the beginning or from any other Slab number.
  Click " Update " button. This will re-number all the Slabs if required.
  Now let us start editing the LEFT BOTTOM & RIGHT TOP Joint numbers of Slabs. Go to first Slab S11 and Select it (Click with Cursor).
  Now concentrate on the Text Boxes below. Slab # will be shown as S11. Left Bottom Joint # is shown as 24 & Right Top joint # is shown as 12. This is not what we require, hence change Left Bottom Joint to 25. Right Top Joint is OK as "13".
  Similarly edit the rest of Slab's Left Bottom & Right Top Joint numbers as required by our Actual Floor Plan (Refer Step 2).

  In case you would like to EXIT program after partial editing, first use " Update " button to save your work & then click " OK " button. The program will ask you about exiting, click Yes & quit.

- **All other Slab Parameters Viz; Live Load, Floor Weight, and Slab Spanning can be Added / Edited for individual Slabs by clicking at respective Text Boxes. All slabs are Composite Deck Profile Slabs.**
  All Loads are Un-Factored. Slab Spans in either direction is displayed in Golden Color. Similarly Slab Intensity in t/m2 is displayed in Golden Text Box.

- **Now click the " Read Me " button, the following important messages are displayed for guidance.**

  1. Add Joint & Beam Details Before Slab.

  2. Slab Numbers cannot be < = 0.0 & repeated.

  3. Slab LHS & RHS Joint #s cannot be repeated.

  5. Max. Slab Joint # cannot > Max. Joint File #.

  6. Use Add Button to Append Record.

  7. Use Update Button to Re-Number & Save Your Work.

  8. Max. Slab Number = Max. Record Number.

  9. Slabs shall be Numbered Serially.

  10. Slabs Joint left Bottom & right Top (LB/RT) Numbers cannot be < = 0.0.

  11. Flooring could Span in X OR Y Direction only.

  12. Floor Weight = 0.0 Kg / M2 Means Cut-Out / Opening.

  13. All Loads are Un-Factored.

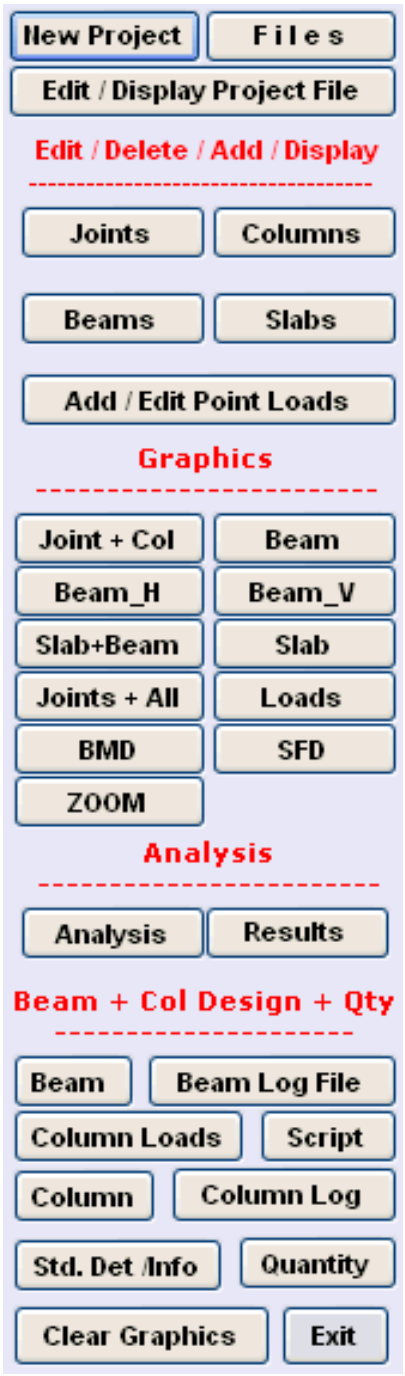
  14. Floor DL shall Include minimum of Deck Profile, RCC, Partition Load (100 Kg / M2), Floor Finish (40 Kg / M2), False Ceiling (20 Kg / M2).

- **Now we have come to the end of Step # 6.**
  In the next step we will Add Point Loads to the Beams.

STEP NO. 6 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 7 : Add & Edit Point Loads to Beam

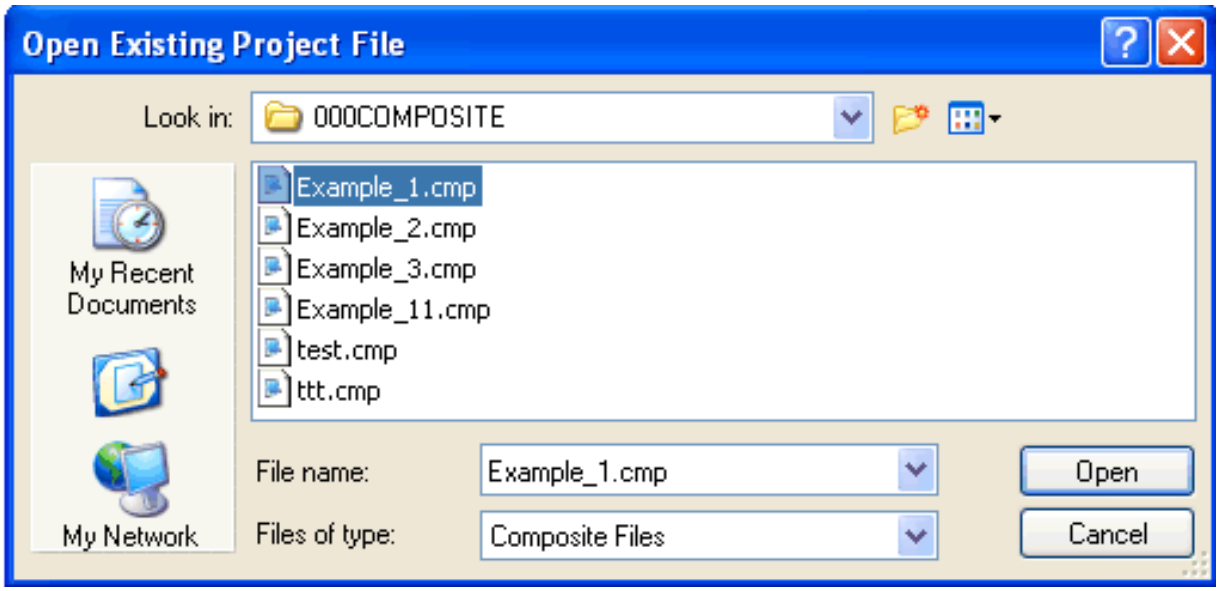


When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.

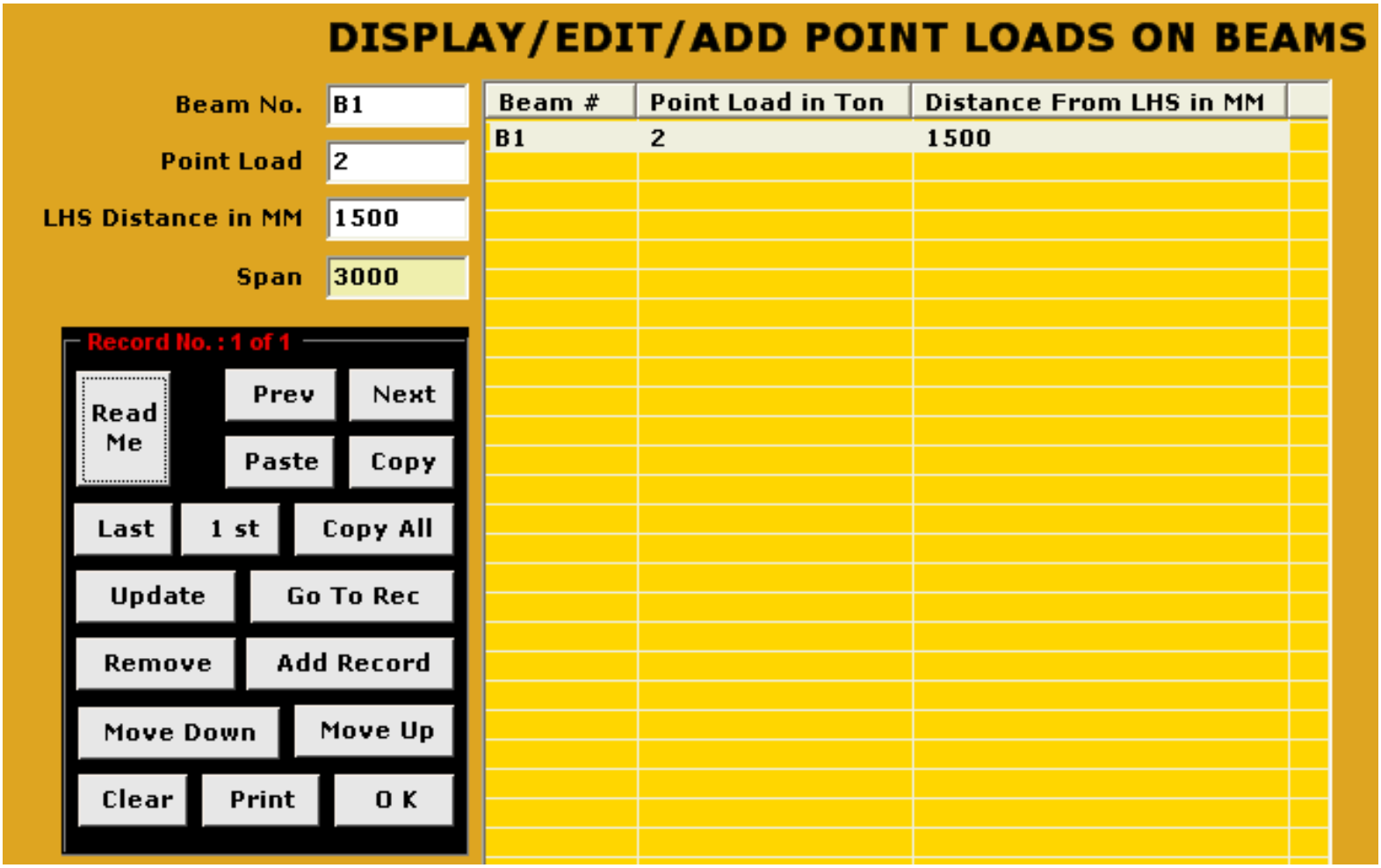
- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Add / Edit Point Loads**

Now Click on " Add / Edit Point Loads " option.

Following Graphics is displayed.



Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.



- **Click " Add Record " button to Add Point Load to a Beam. When Add Record button is clicked,**
  Beam No. Text Box will show B1. You can edit Text Box to change this beam no. Enter required Externally Applied Point Load & its distance from Left. The distance should not exceed the Span as displayed just below.

  Just like any other option, here also a user can Display, Add, Edit & delete the Point Load records at the same time. The " Move Up " and " Move Down " buttons will move the record Up or Down the Table respectively. This is useful if you would like to keep the point loads on the same beam serially.

  You can add any number of point Loads. Do not repeat the same load & location.

  In case you would like to EXIT program after partial Adding / Editing, first use " Update " button to save your work & then click " OK " button. The program will ask you about exiting, click Yes & quit.

- **Now click the " Read Me " button, the following important messages are displayed for guidance.**

  1. LHS & RHS Joint #s cannot be repeated.
  2. Enter Point Loads due to Externally Applied Loads Only.
  3. Reactions due to Secondary Beams are Calculated Automatically.
  4. Use Add Button to Append Record.
  5. Use Update Button to Save Your Work.

- **Now we have come to the end of Step # 7.**
  In the next step we will Mark Beam Continuity.

STEP NO. 7 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 8 : Data Checking Through Graphics

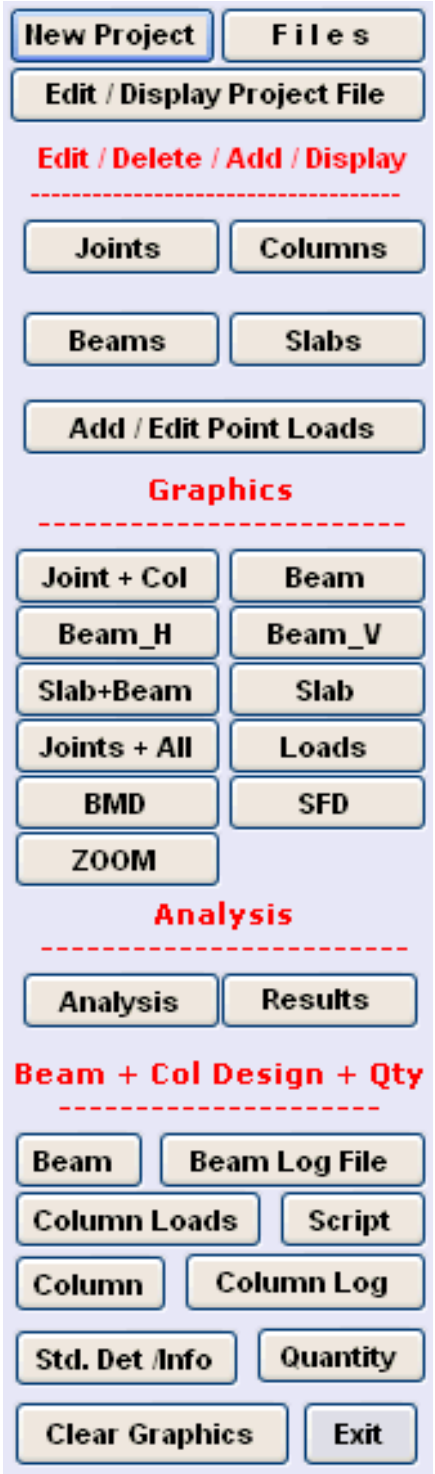
A User should thoroughly check Data Input at all stages. During Adding / Editing Data through tables, Beam " SPAN " and Slab Dimensions (Shorter & Longer) should be constantly monitored for any error.
After DATA Input is over, it should be checked visually & by taking printouts of various Graphics Options. Analysis, Beam Design, Column Loads and Quantities options shall be run (in strict order) after Data Checking is over. If there is any error in DATA, un-expected results will be obtained after running Analysis, Design, Column Loads and Quantities options. Sometimes results obtained are such that it will be difficult to even find out that actually they are wrong due to erroneous data. Any Analysis & Design is as good as its data input. Hence the importance of Data Input cannot be over emphasized.

Note that BMD is drawn on Tension Side which reflects Deflected shape of Beam. BMD, SFD and Load Diagrams are Important from the point of Checking Un-expected Analysis Results & Data Input.
Any un-expected Diagram will reflect Data Error in the form of :

- **Incorrect Geometry (Span, Grid Dimension).**
- **Incorrect Loads (Point Load, Slab Spanning Direction).**
- **Floor Analysis & Beam Design not performed after Editing / Adding Geometry or Loads.**

Under " Column Load " Option Statistical Check is displayed.
Note that the Difference in Loads is due to Maximum Loading On Column, due to External Loads / Beam Continuity / End Moments, i.e. Maximum of Simple Reaction & Continuous Reaction is taken for Column Design.
If the difference is High, a user should look closer at the Data-Input for any Error.

The Most effective check will be when AutoCAD drawing of floor plan is created using Script option.
The script command will be used after Successful Completion of Analysis, Design & Quantity options. In AutoCAD drawing, even the minor error in layout could be identified. We will discuss this in Step no. 12.

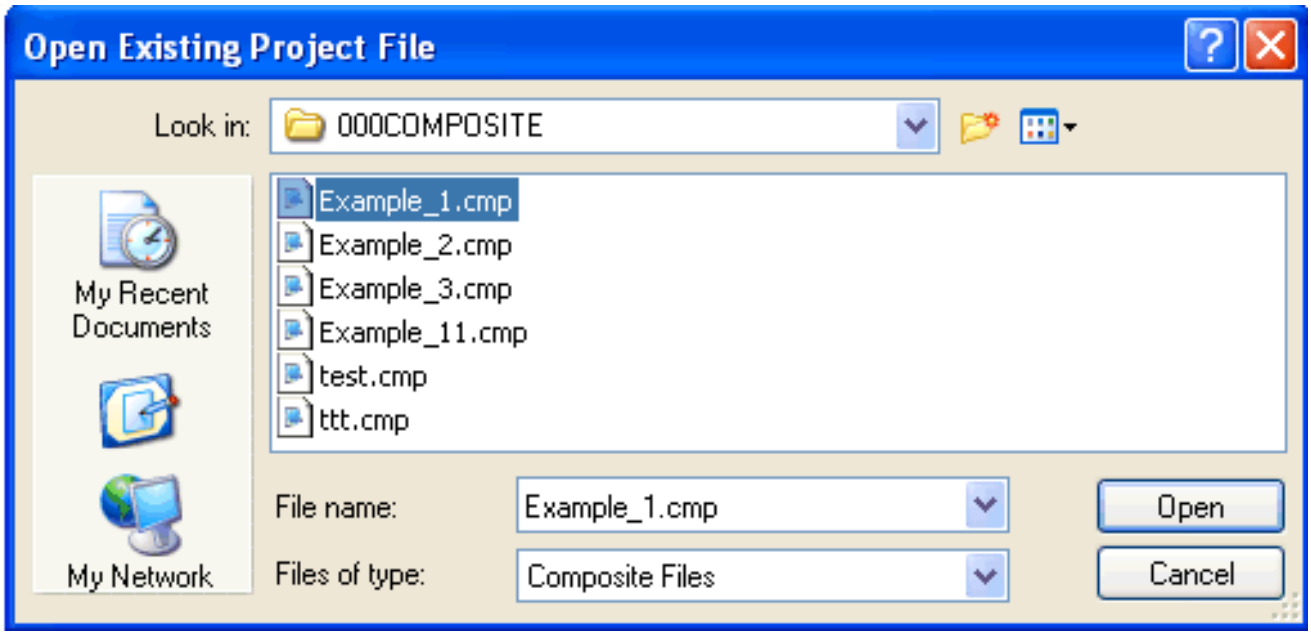

New Project
Files
Edit / Display Project File
Edit / Delete / Add / Display
Joints
Columns
Beams
Slabs
Add / Edit Point Loads
Graphics
Joint + Col
Beam
Beam_H
Beam_V
Slab+Beam
Slab
Joints + All
Loads
BMD
SFD
ZOOM
Analysis
Analysis
Results
Beam + Col Design + Qty
Beam
Beam Log File
Column Loads
Script
Column
Column Log
Std. Det /Info
Quantity
Clear Graphics
Exit

When Program starts, the Menu above is displayed. Under the Graphics Heading following options are displayed.

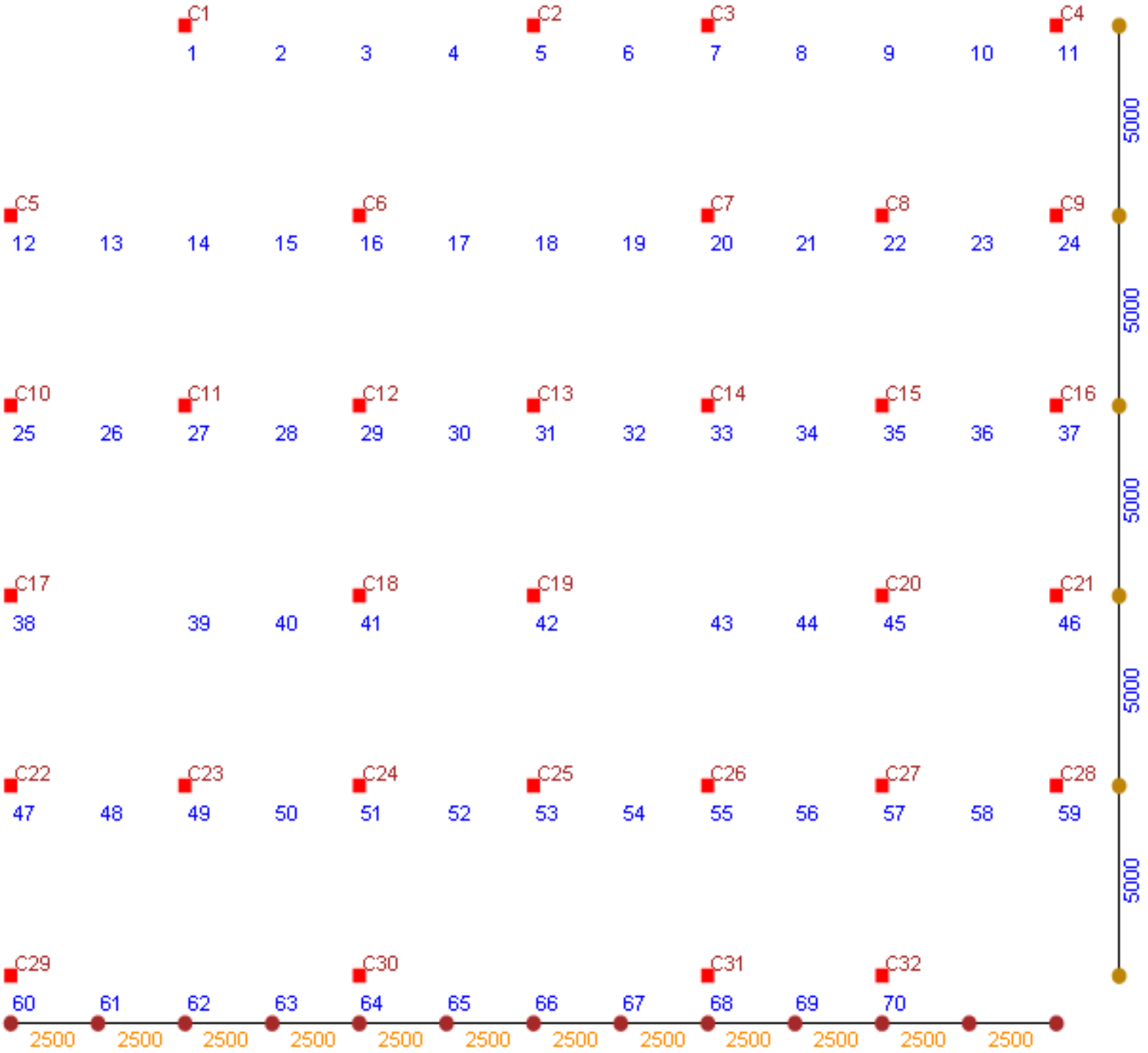
- **Joint + Col**
- **Beam**
- **Beam_H (Only Horizontal Beam numbers will be Displayed).**
- **Beam_V (Only Vertical Beam numbers will be Displayed).**
- **Slab + Beam (Beams, Slabs & Columns are displayed).**
- **Slab (Only Slabs & Columns are displayed).**
- **Joints + ALL (For Display of Joints, Columns, Beams & Slabs)**
- **Loads (Display of Slab, Point Loads & Reactions from Secondary Beams, to be used after Analysis, and Design options have been successfully Run).**
- **BMD (Display of Bending Moment Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.**
- **SFD (Display of shear Force Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.**
- **Zoom (Display of part of Floor Plan under Selection).**

Now Click on " Joint + Col " option.

Following Graphics is displayed.

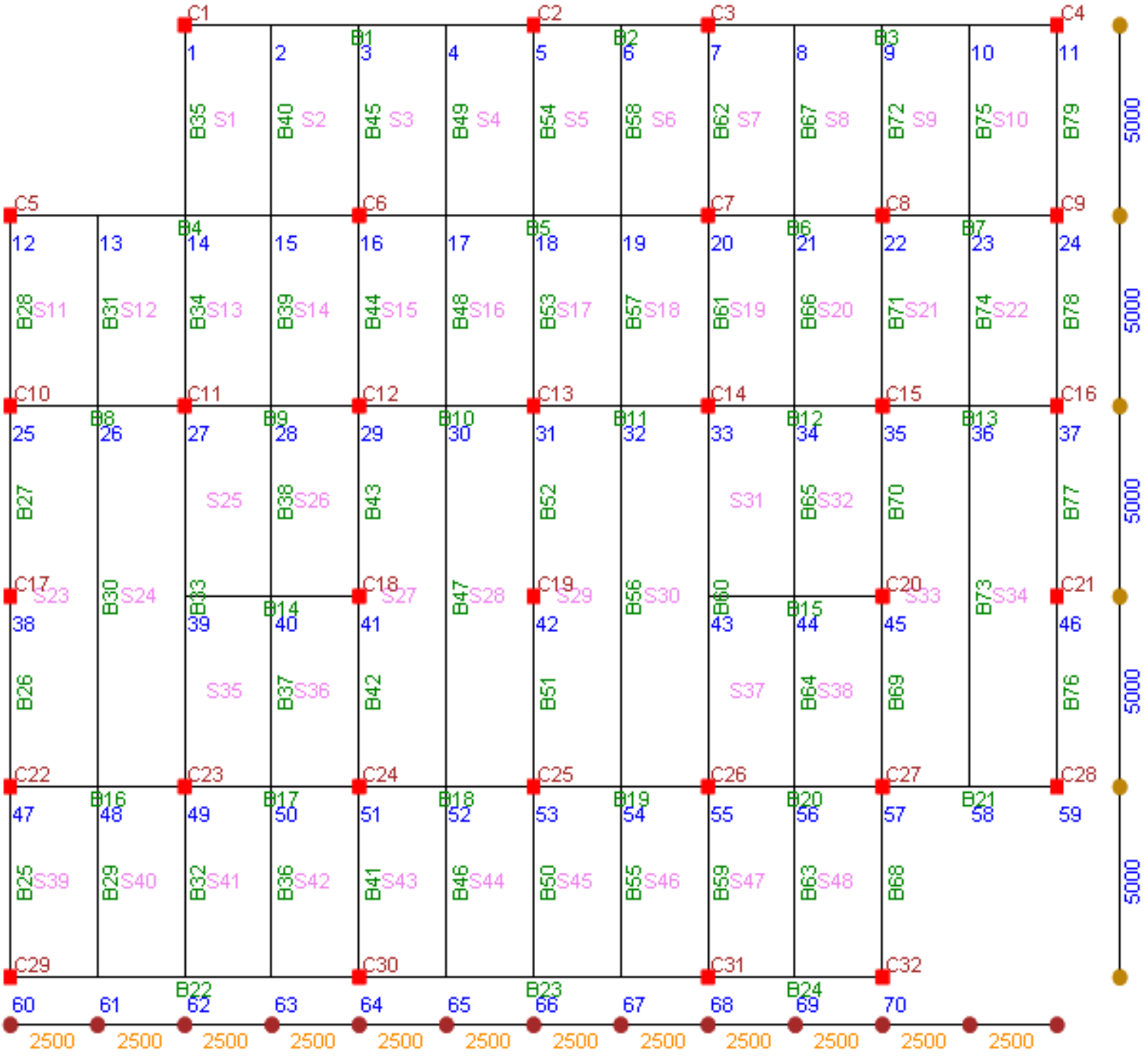


- **Now select " Example _1 File & Press Open Button.**
  Following Graphics will be displayed.



- **The above Graphics displays Joint, Column Numbers as well as Horizontal and Vertical Dimensions.**
  A User should Check the Location of Each Joint & Column & C/C Horizontal & Vertical Grid distance.

Now click the " Joints + ALL " button.
This is the all important Graphics Display, showing Joints numbers, Columns, Beam numbers and Slab numbers. If this display is not very Clear or Congested than use other options such as Beam, Beam_H (Only Horizontal Beam # will be Displayed), Beam_V (Only Vertical Beam # will be Displayed), Slab + Beam (Beams, Slabs & Columns are displayed), Slab (Only Slabs & Columns are displayed) and Zoom Option.
Following Graphics is displayed when " Joints + ALL " button is clicked.



- **Now Click " Beam " button & after display of Graphics click " ZOOM " button. Now Left Click with mouse near the Column C5 & Drag it near the Column C11. You will see change in color in window as mouse is dragged. Now Lift your finger. Following ZOOM Window is displayed. Use Zoom option for more clarity on Floor plan display.**

Note that Graphics Display of :

- **Loads** **(Display of Slab, Point Loads & Reactions from Secondary Beams, to be used after Analysis, and Design options have been successfully Run).**

- **BMD** **(Display of Bending Moment Diagram, to be used after Analysis, Design and Quantity options have been successfully Run.**

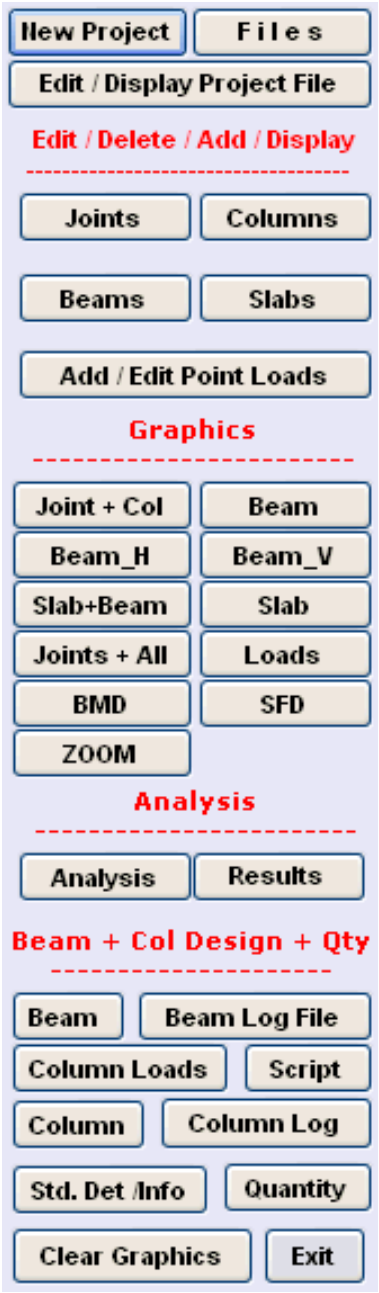
- **SFD** **(Display of shear Force Diagram, to be used after Analysis, Design and Quantity options have been successfully Run.**

Now we have come to the end of Step # 8.
In the next step we will Run " Analysis " option.

STEP NO. 8 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 9 : Analysis & Its Results



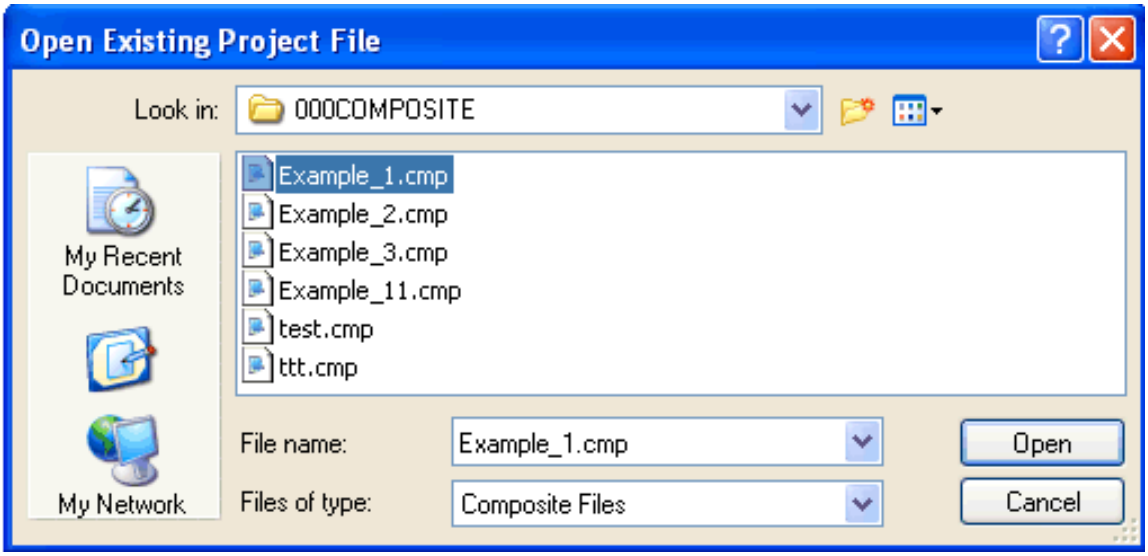
After entering Data & Checking it thoroughly, Relax, let the software do its Job.
The 1st milestone is Analysis.

When Program starts, the Menu above is displayed.
Under the Analysis Heading following options are displayed.

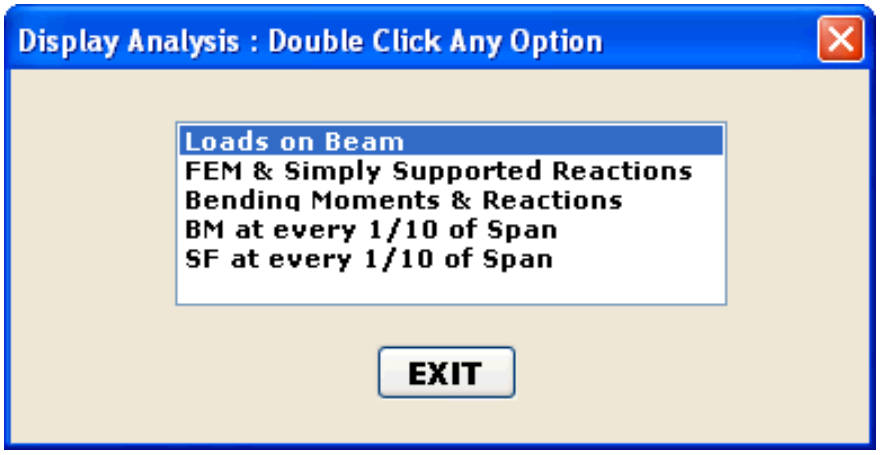
- **Analysis**
- **Results**

Now Click on " Analysis " option.

Following Graphics is displayed.



- **Now select " Example _1 File & Press Open Button.**
  The Analysis will commence. A window will open & it will indicate number of Joints, Columns, Beams and Slabs to be analyzed. The Analysis will take time & will depend up on the file size & computers RAM memory. Minimum Computer RAM memory of 1 to 2 GB is recommended for faster analysis results. After the analysis is over a new message will appear indicating that " Analysis is Successfully Completed ".
- **Now Click the analysis " Results " option, following graphics will be displayed.**



- **Now Double Click on " Loads on Beam " Option. A new window will open displaying various Loads on Beams. Click on " Read Me " button, following important messages are displayed.**

1. UDL is in T / M."
2. RHS_MOM : Right Hand Side Moment is in T-M.
3. LHS_MOM : Left Hand Side Moment is in T-M.
4. Point Load is in Ton."
5. Point Load Could be Externally Applied OR
6. From Reaction of Secondary beam.
7. Dist : is distance of Point Load from Left.
8. NEAR_INT : is Slab Load in T/M Near to LHS of Beam.
9. NEAR_DIST : is Slab Load Distance in M Near to LHS.
10. FAR_INT : is Slab Load in T/M Far from LHS.
11. FAR_DIST : is Slab Load Distance in M Far from LHS.
12. Note that NEAR_INT = FAR_INT, as Slabs are Spanning One Way.

- **Now Double Click on " FEM & SS Reactions " Option. A new window will open displaying Fixed End Moments and Simply Supported Reaction on each Beam. Click on " Read Me " button, following important messages are displayed.**

1. Beam Span in M.
2. LHS SS Reaction : LHS Simply Supported Reaction in Ton.
3. RHS SS Reaction : RHS Simply Supported Reaction in Ton.
4. LFEM : Fixed End Moment at LHS Support in T-M.
5. RFEM : Fixed End Moment at RHS Support in T-M.
6. In order to Sort the Values in Ascending OR
7. Descending Order, Just Click Column Header at Top.

- **Now Double Click on " Bending Moments and Reactions " Option. This is the most Important Option. A new window will open displaying End Moments and Reactions on each Beam. Click on " Read Me " button, following important messages are displayed.**

1. -Ve BM at LHS Support in T-M.
2. -VE BM at RHS Support in T-M.
3. LHS Reaction in Tons.
4. RHS Reaction in Ton.
5. + VE Bending Moment in T-M.
6. Distance of + VE BM from LHS Support in M.
7. In order to Sort the Values in Ascending OR
8. Descending Order, Just Click Column Header at Top.

Shown below is a part Display of Support BM, SF, + Ve BM & Its Distance from Left.

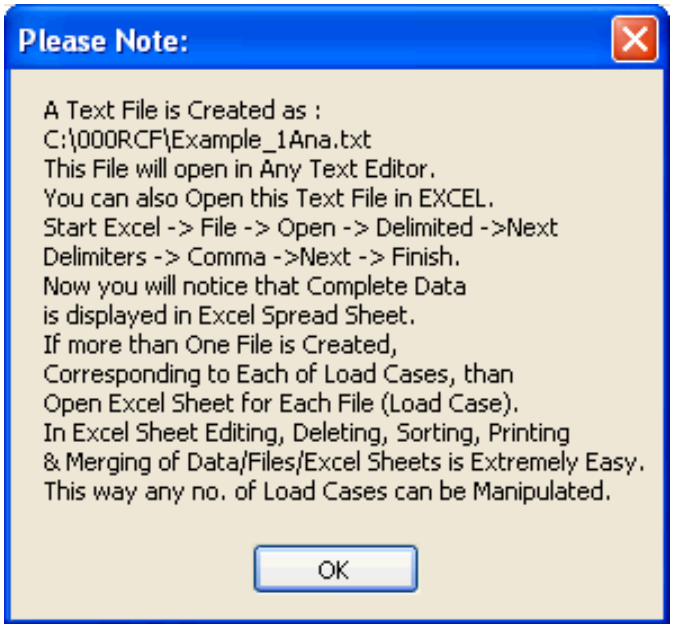
DISPLAYING BENDING MOMENTS AND REACTIONS

DISPLAYING BENDING MOMENTS AND REACTIONS

Column Header : Click Here to Sort A/D

Beam #	-VE BM LHS	- VE BM RHS	LHS Reaction	RHS Reaction	+VE BM	Distance	Load Case
1	0	0	17.212	17.212	54.34	5	DL + LL
2	0	0	6.177	6.177	11.389	2.5	DL + LL
3	0	0	17.212	17.212	54.34	5	DL + LL
4	0	0	21.83	24.766	77.193	5	DL + LL
5	0	0	27.7	27.7	91.866	5	DL + LL
6	0	0	9.112	9.112	18.726	2.5	DL + LL
7	0	0	9.112	9.112	18.726	2.5	DL + LL
8	0	0	12.228	12.228	26.44	2.5	DL + LL
9	0	0	9.129	9.129	18.747	2.5	DL + LL
10	0	0	12.228	12.228	26.44	2.5	DL + LL
11	0	0	12.228	12.228	26.44	2.5	DL + LL
12	0	0	9.129	9.129	18.747	2.5	DL + LL
13	0	0	12.228	12.228	26.44	2.5	DL + LL
14	0	0	9.129	9.129	18.747	2.5	DL + LL
15	0	0	9.129	9.129	18.747	2.5	DL + LL
16	0	0	12.228	12.228	26.44	2.5	DL + LL
17	0	0	9.129	9.129	18.747	2.5	DL + LL
18	0	0	12.228	12.228	26.44	2.5	DL + LL
19	0	0	12.228	12.228	26.44	2.5	DL + LL
20	0	0	9.129	9.129	18.747	2.5	DL + LL
21	0	0	9.269	9.269	19.072	2.5	DL + LL
22	0	0	17.212	17.212	54.34	5	DL + LL
23	0	0	17.212	17.212	54.34	5	DL + LL
24	0	0	6.177	6.177	11.389	2.5	DL + LL
25	0	0	6.108	6.108	7.633	2.5	DL + LL
26	0	0	6.108	6.108	7.633	2.5	DL + LL
27	0	0	6.108	6.108	7.633	2.5	DL + LL
28	0	0	6.108	6.108	7.633	2.5	DL + LL
29	0	0	5.871	5.871	7.335	2.5	DL + LL

Note that Column Headers are all the Titles at Top. Just Click them to Sort.
For Printing Just Click " Print " Button.
When " OK " button is clicked, following Important Message is displayed.



- **The above message describes how any number of Load Cases can be Run & Manipulated once File is Exported to Excel Spread Sheet. Note the File Name Carefully.**
  Similar File is created for " Shear Corrected BM & SF " option.

  1. LHS Reaction in Tons.
  2. RHS Reaction in Ton.
  3. + VE Bending Moment in T-M.
  4. Distance of + VE BM from LHS Support in M.
  5. Load Case is DL + LL Only.
  6. End supports are assumed Hinged.
  7. Wind/EQ BM are resisted by Frame/Shear Wall.
  8. In order to Sort the Values in Ascending OR" Descending Order,
  Just Click Column Header at Top.
  9. All the Result Parameters are Un-factored.

- **Now Double Click on " BM at Every 1 / 10 th of Span " Option. A new window will open displaying Distance from Left and its BM on each Beam. This display is in two (2) Pages. Click on " Read Me " button, following important messages are displayed.**

  1. bm0 = Bending Moment at LHS Support.
  2. d0 = Distance zero from LHS Support.
  3. bm1 = Bending Moment at a distance d1 M. from LHS Support, and so on.
  5. Distances are Multiple of 1 / 10 th of Span.
  6. Bending Moments are in T-M.
  7. In order to Sort the Values in Ascending OR
  8. Descending Order, Just Click Column Header at Top.

- **Now Double Click on " SF at Every 1 / 10 th of Span " Option. A new window will open displaying Distance from Left and its SF on each Beam. This display is in two (2) Pages. Click on " Read Me " button, following important messages are displayed.**

  1. sf0 = Shear Force at LHS Support.
  2. d0 = Distance zero from LHS Support.
  3. sf1 = Shear Force at a distance d1
  4. M. from LHS Support, and so on.
  5. Distances are Multiple of 1 / 10 th of Span.
  6. Shear Forces are in T.
  7. In order to Sort the Values in Ascending OR Descending Order,
  Just Click Column Header at Top.

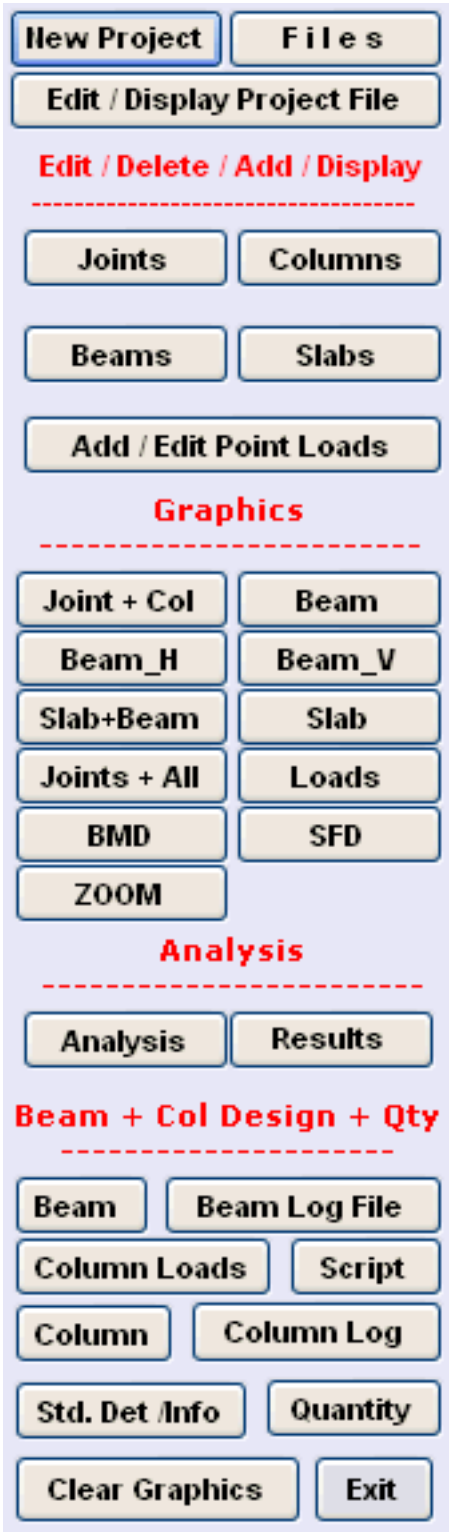
- **Now we have come to the end of Step # 9.**
  In the next step we will Run " Beam Design " Option.

STEP NO. 9 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 10

Beam Design, Quantities, Cost Estimation and Column Loads



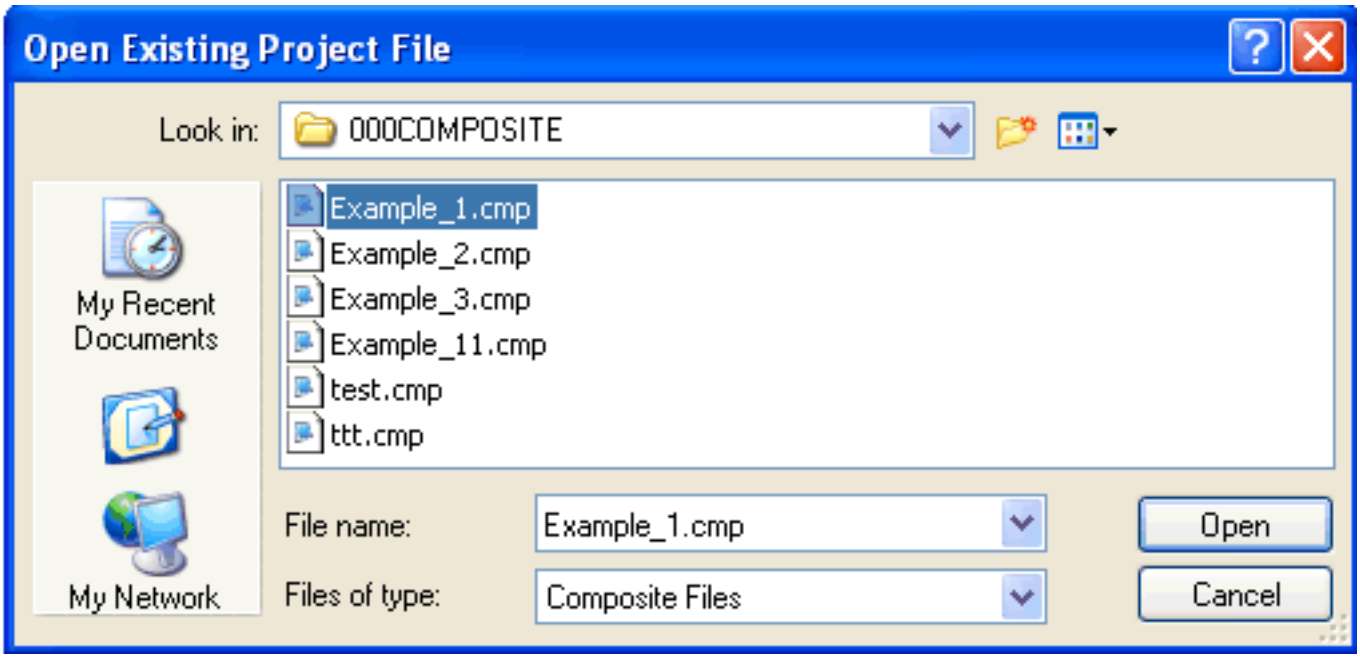
- **After entering Data & Checking it thoroughly, Relax, let the software do its Job.**
  The 1st milestone is Analysis.

When Program starts, the Menu above is displayed.
Under the Beam + Column Design + QTY Heading following options are displayed.

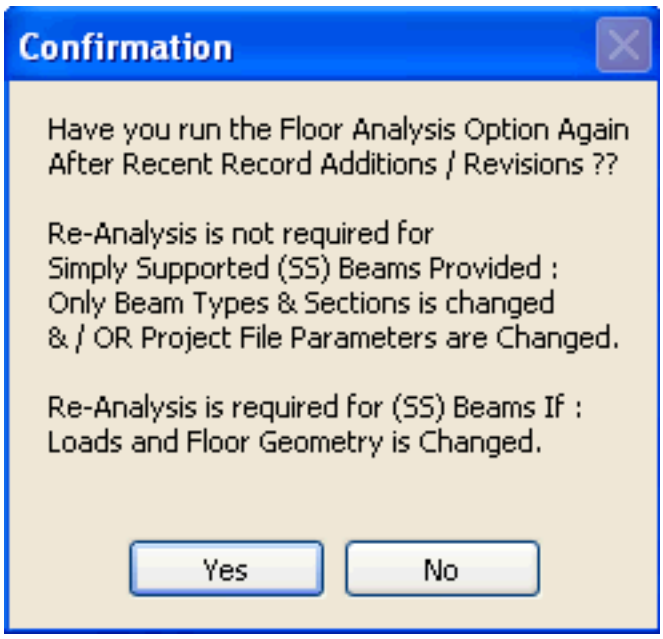
- **Beam**
- **Beam Log File**
- **Column Loads**
- **Script**
- **Column**
- **Column Log**
- **Standard Details**
- **Quantities**

Now Click on " Beam " Option.

Following Graphics is displayed.



- **Now select " Example _1 File & Press Open Button.**
  Following Warning is displayed.



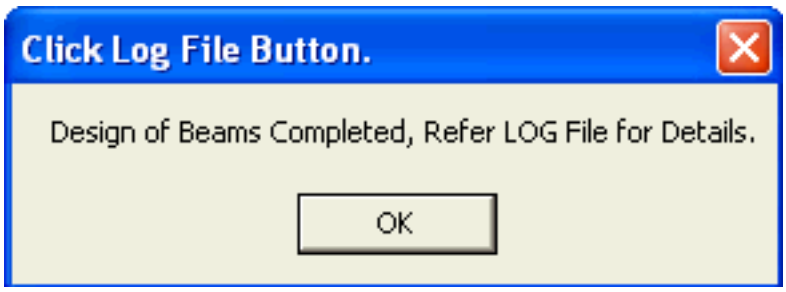
This is a very Important Message. In case a user has edited or added any Joint / Column / Beam or Slab Member after performing analysis then he should re-perform the analysis, else old (in-correct) results will be displayed. However Re-Analysis is not required for Beams Provided:

- **Only Beam Type & Section is Changed.**
- **Project File Parameters are Changed.**

Re-Analysis is required for Beams if Loads & Floor Geometry is Changed.

Click " Yes " if you have not revised any member after analysis or click " No " if you are not sure.

If " Yes " is clicked then following message will be displayed.



- **Now Click the " Log File " Button. Following Design Results (Part) is displayed. The LOG file will display Beam results (Safe / Unsafe) & various parameters on which Beam is evaluated. User should study the each parameter in order to redesign the beam for safety or for economy / Optimization.**

The Composite Beam Design is carried out as per following Codes,

- **Construction Stage:**

- **As per IS 800 : 2007.**

- **Composite Stage:**

(a) Section Classification and BM as per Euro 4.

(b) SF, Axial Load, Deflection, Slenderness, Bolt, Weld Design as per IS 800 : 2007.

- **Profile Deck design by Deck manufacturer.**

Beam No. : B 1
Beam Section : MB-550
Factored BM during Construction Stage in T-M : 20.619
Effective Length during constn. stage in MM : 2500
Effective Length of Comp. Flange during Composite stage in MM : 0
Effective Concrete Flange Width in MM : 1250
Factored Max. BM during Composite Stage in T-M = 81.51
Factored Max. SF during Composite Stage in Tons = 25.536
Factored Max. Axial Force during Composite Stage in Tons = 0
Section Class is : Plastic
b/tf = 4.922
d/tw = 42.446
Mcr in KN-M = 1987.054
Beta_B = 1
Lambda_LT = 0.584
Phi_LT = 0.71
X_LT = 0.895
Extreme Fiber Bending Comp. Stress {fcr} in N/MM2 = 732.695
Design Bending Compressive Stress {fbd} in N/MM2 = 203.616
M_ZZ of Section during Construction Stage in T-M = 55.22
Max. M_ZZ Capcity during Construction Stage in T-M = 64.358
****

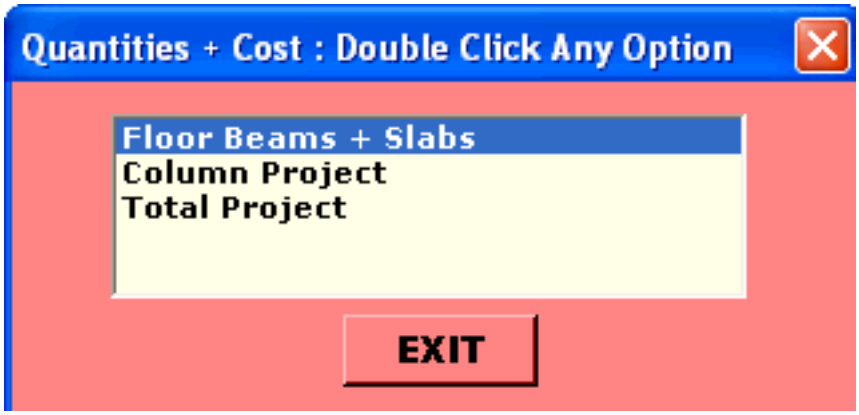
SR_ZZ = 45.126
SR_YY = 67.024
Governing SR during Construction Stage = 67.024
****

Permissible Deflection during Construction Stage in MM = 30
Actual Deflection during Construction Stage in MM = 10.591
****

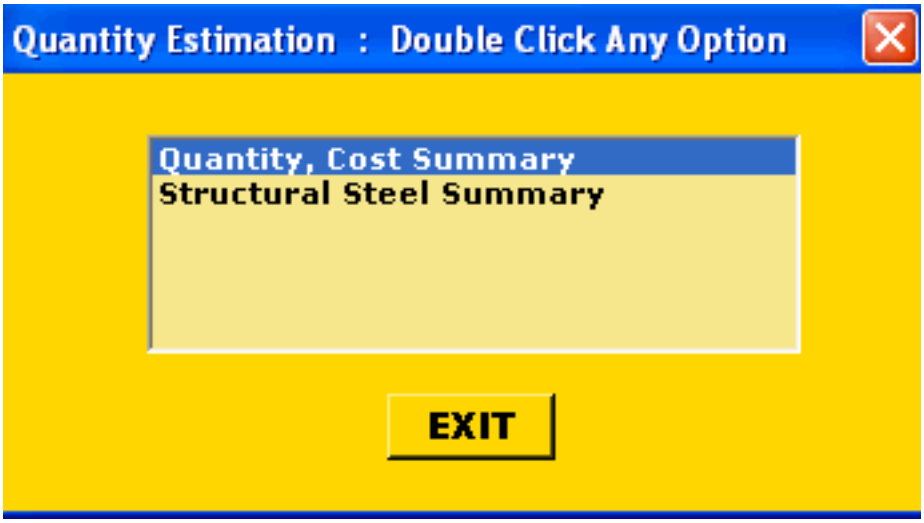
%%
%%%%% Beam B 1 is Safe During Construction Stage %%%%%
%%

```
------------------------------------------------------------------------
00000 Design Check During Composite Stage 00000
------------------------------------------------------------------------
Permissible Span / Depth Ratio = 20
Actual Span / Depth Ratio = 14.705
Moment of Inertia of Un-Cracked Section in CM4 = 144283.4
Permissible Deflection during Composite Stage in MM = 33.333
Actual Deflection during Composite Stage in MM = 18.83
Frequency of Vibration of Floor Beam in Cycles / Sec = 4
**********
Neutral Axis Lies within Flange
Concrete Thickness in MM = 130
Steel Flange Thickness in MM = 19.3
Neutral Axis Depth in MM = 140.16
M_ZZ of Section during Composite Stage in T-M = 101.248
Longitudinal Shear in Ton = 212.5
For Beam B 1 :  Total no. of Single Shear Connectors Reqd. = 81
oooooooooooooooooooooooooooo
Longitudinal Design Shear in T / M = 21.25
Longitudinal Shear Capcity in T / M = 65.076
88888888888888888888888888888
Factored Shear is Less than 0.6 x Shear Str. of Section
0.6 * Shear Strength of Section in Ton = 48.498
**********
Design Compressive Stress for Web in N/MM2 = 103.229
Shear Capacity of Web Under Buckling in Ton = 54.917
**********
Shear Capacity of Web Under Bearing in Ton = 74.645
Assumed bearing of External load on Beam = 200 MM including RCC
**********
Bolt Shear Capacity in Ton = 45.629
Bolt Bearing Capacity in Ton = 64.96
Bolt Capacity in Ton = 45.629
**********
Bolt diameter in MM = 20
Bolt Hole diameter in MM = 24
Bolt Numbers = 10
Shear Connection Plate Size in MM = 310 x 140
```

- **Note that Step by Step design is given for Construction & Composite stage. If Beam is Unsafe, all the parameters will be displayed indicating why beam is unsafe. A designer shall study & correct the various parameters such as Concrete Grade/ Concrete thickness, Profile Depth / Thickness and Beam Size.**
- **Now Click " Quantity " Option. Following Graphics is displayed.**



- **Now Double Click " Floor Beams + Slabs " Option. Following Graphics is displayed. (Quantities and Cost of Column Project and Total Project will be discussed in later Steps).**



- **Now Double Click " Quantity, Cost Summary " Option. Following Graphics is displayed.**

DISPLAYING QUANTITIES AND COST SUMMARY

Item	Quantity	Rate	Cost
Structural Steel Framing in Tons	23.265	50000	1163250
Total Masonry Work in M2	710.85	850	604222.5
Total Plaster in M2	1421.7	400	568680
Total Masonry Painting in M2	1421.7	100	142170
M30 : Concrete Floor Slab in M3	85.4	5000	427000
Tor 8 MM Bars in Kg	2110.769	50	105538.5
Tor 10 MM U Bars in Kg	466.188	50	23309.4
Deck Profile + Studs in M2	700	500	350000
False Ceiling in M2	700	400	280000
Total Door / Windows in M2	92.4	2500	231000
Total Cost of Floor			3895171
Unit Cost of Floor in Rs / M2			5564.529
Unit Cost of Floor in Rs / sqft			517.63
Total Floor Area in M2	700		
Structural Steel Framing in Kg/M2	33.235		
Total Cement Bags Required in Nos.	1433		
Total Sand Consumption in M3	63		
Total Aggregate Consumption in M3	68		

- **The above display gives cost summary as per the Rates Put-In during creation of Project File.**
  Now Double Click " Structural Steel Summary " Option. Following Graphics is displayed.

SUMMARY OF STRUCTURAL STEEL IN KG

MC 100	0
MC 125	0
MC 150	0
MC 200	994.5
MC 250	0
MC 300	0
MC 350	0
MC 400	0
MB 100	0
MB 150	0
MB 200	2667
MB 250	4289.5
MB 300	1326
MB 350	262
MB 400	2156
MB 450	2896
MB 500	0
MB 550	6222
MB 600	2452

TOTAL STRUCTURAL STEEL IN TONS = 23.265

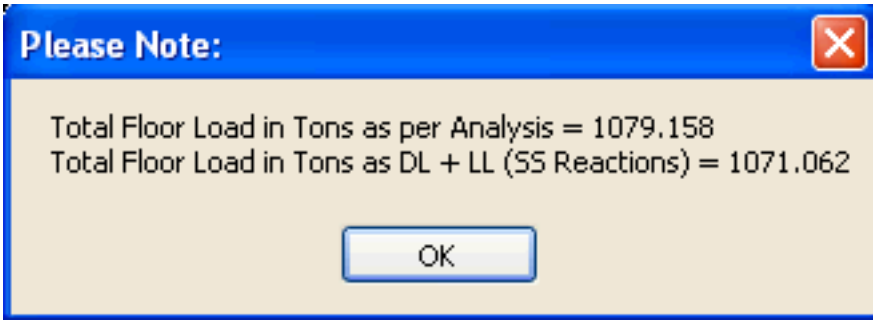
- **Note that the above steel weight does not include end connections & member splice. We have taken C/C distance between columns as member length, which may offset the connection + splice weight.**
  Now Double Click " Column Loads " Option from the Main Menu. Following Graphics is displayed.

UN-FACTORED COLUMN LOADS

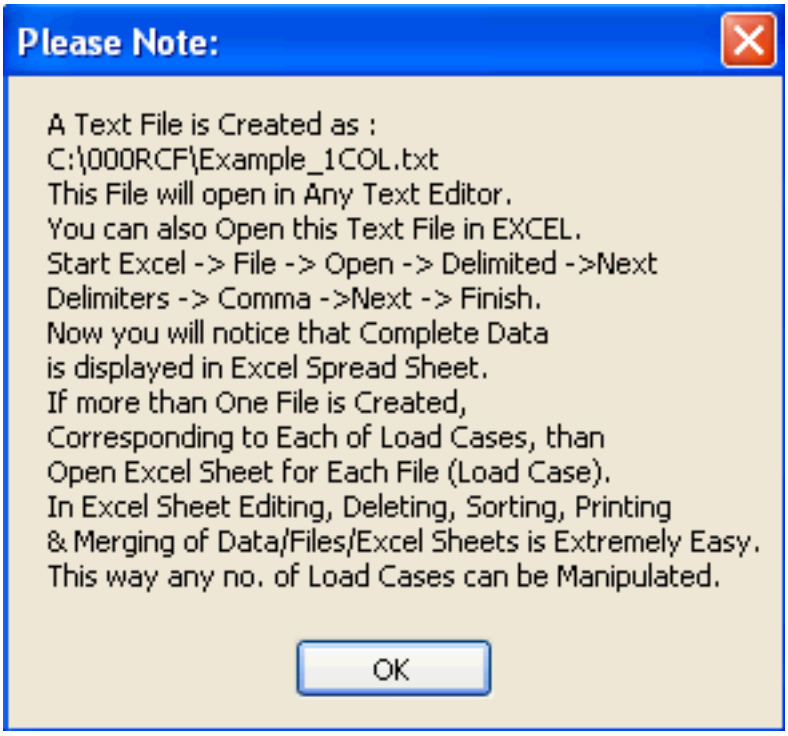
PART DISPLAY

Column No.	Type	Section	Height in M	Load in Tons
C1	2MB T/T	MB-300	3	23.593
C2	2MB T/T	MB-300	3	32.704
C3	2MB T/T	MB-300	3	32.704
C4	2MB T/T	MB-300	3	23.585
C5	2MB T/T	MB-300	3	28.203
C6	2MB T/T	MB-400	3	70.935
C7	2MB T/T	MB-400	3	55.281
C8	2MB T/T	MB-300	3	36.589
C9	2MB T/T	MB-300	3	21.593
C10	2MB T/T	MB-300	3	24.709
C11	2MB T/T	MB-300	3	53.67
C12	2MB T/T	MB-300	3	39.722
C13	2MB T/T	MB-300	3	42.821
C14	2MB T/T	MB-300	3	53.67
C15	2MB T/T	MB-300	3	39.722
C16	2MB T/T	MB-300	3	24.709
C17	2MB T/T	MB-300	3	12.481
C18	2MB T/T	MB-300	3	27.494
C19	2MB T/T	MB-300	3	18.365
C20	2MB T/T	MB-300	3	27.494
C21	2MB T/T	MB-300	3	12.481
C22	2MB T/T	MB-300	3	24.709

- **The above Column Loads Graphics is self explanatory. Self Weight of Column is not included.**
  When " OK " button is clicked following vital Statistical Check is displayed.
  The Difference should not exceed say 10 %.
  The major difference should calls for closer look at the Data-Input.



- **When " OK " button is clicked, following Important Message is displayed.**

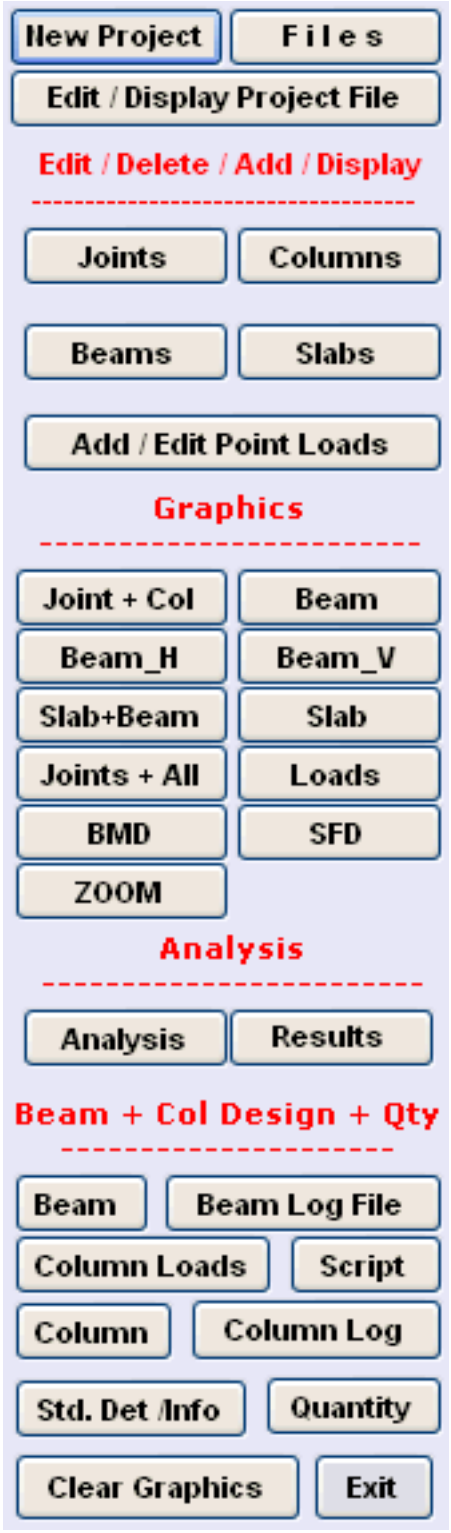


- **The above message describes how any number of Load Cases can be Run & Manipulated once the File is Exported to Excel Spread Sheet. Note the File Name Carefully.**
  Click " OK " button.
  Now we have come to the end of Step # 10.
  Let us proceed to Step No. 11.

STEP NO. 10 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 11 : BENDING MOMENT, SHEAR FORCE DIAGRAM LOAD DISPLAY AND FILES OPTION

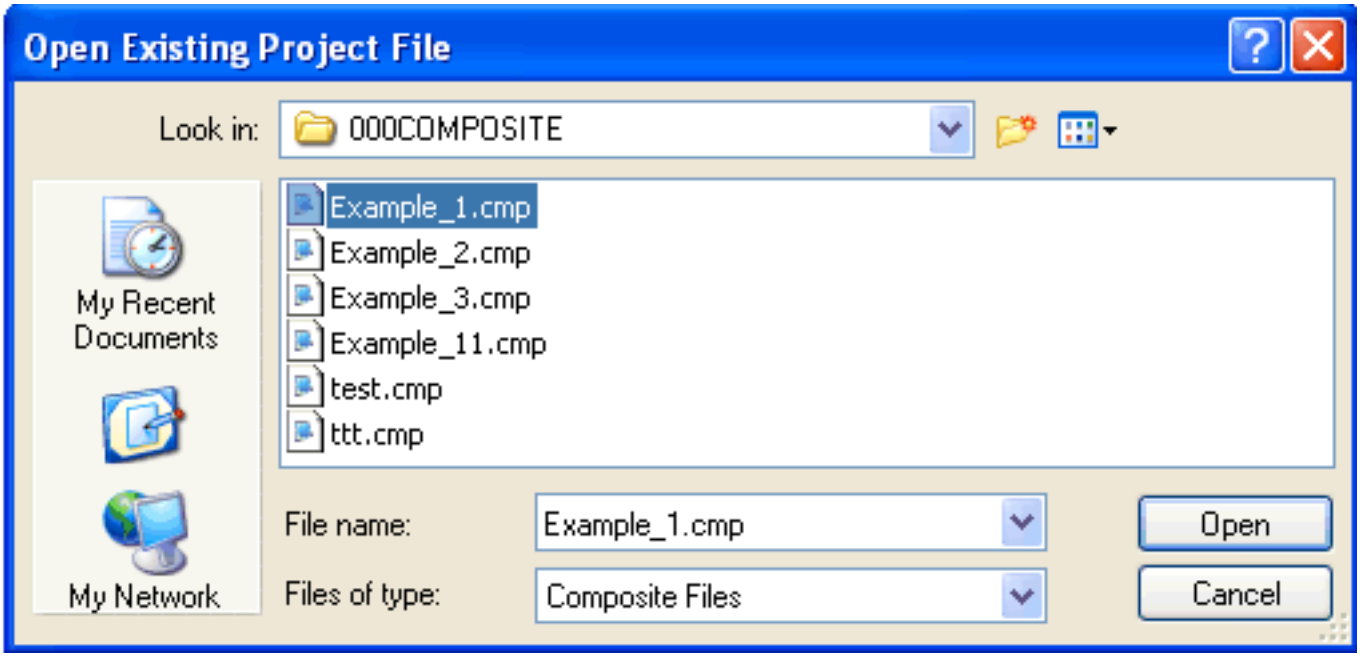


- **When Program starts, the Menu above is displayed. Under the Graphics Heading following options are displayed.**

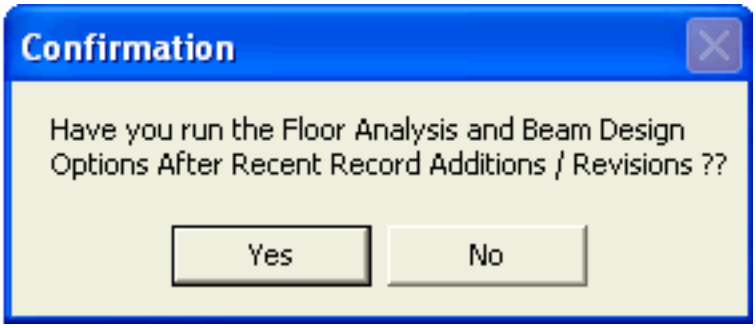
• Joint + Col
• Beam
• Beam_H (Only Horizontal Beam # will be Displayed).
• Beam_V (Only Vertical Beam # will be Displayed).
• Slab + Beam (Beams, Slabs & Columns are displayed).
• Slab (Only Slabs & Columns are displayed).
• Joints + ALL (For Display of Joints, Columns, Beams & Slabs)
• Loads (Display of Slab, Point Loads & Reactions from Secondary Beams, to be used after Analysis, and Design options have been successfully Run).
• BMD (Display of Bending Moment Diagram, to be used after Analysis, Design and Quantity options have been successfully Run.
• SFD (Display of shear Force Diagram, to be used after Analysis, Design and Quantity options have been successfully Run.
• Zoom (Display of part of Floor Plan under Selection).

Now Click on " BMD " option.

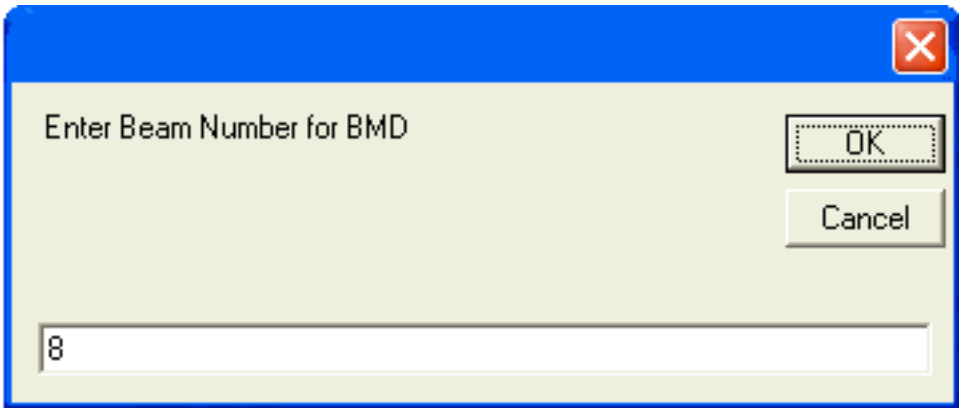
Following Graphics is displayed.



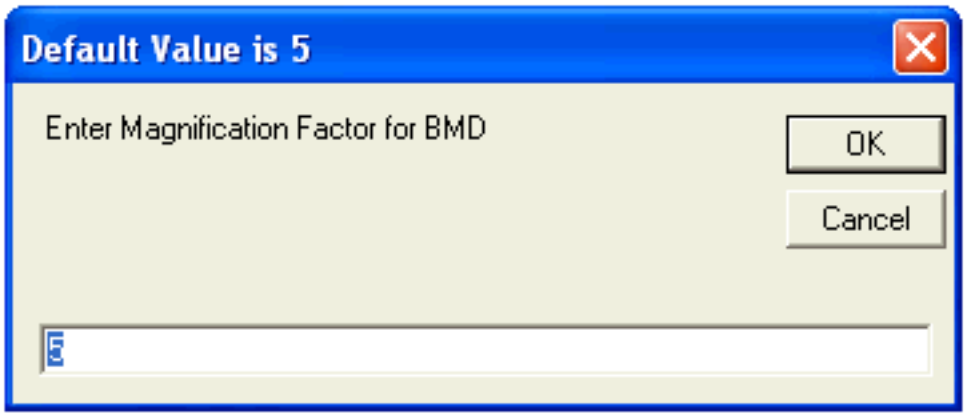
- **Now select " Example _1 File & Press Open Button. Following Warning is displayed.**



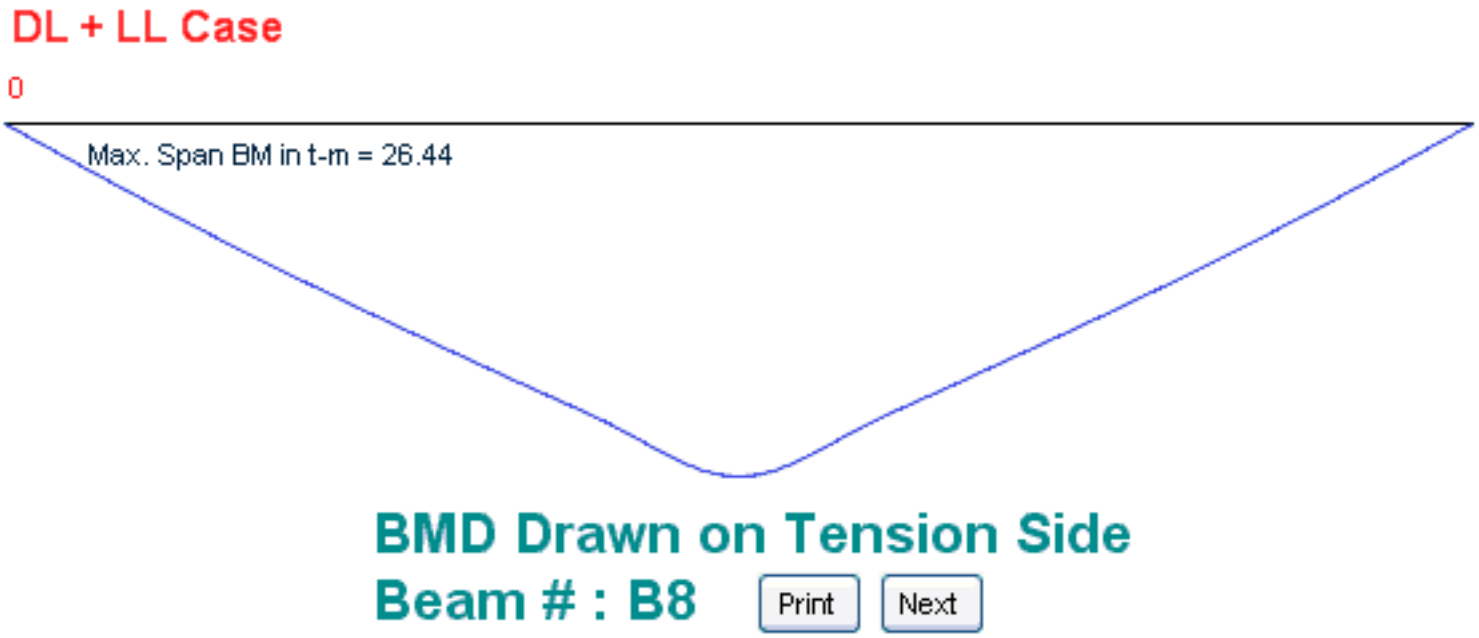
- **This is a very Important Message. In case a user has edited or added any Joint / Column / Beam or Slab Member after performing analysis then he should re-perform the analysis, else old (in-correct) results will be displayed. The Beam Designs are equally important as these options inform you about correctness of Beam Design.**
  Click " Yes " if you have not revised any member after analysis or click " No " if you are not sure.
  If " Yes " is clicked then following graphics will be displayed.



- **Type the Beam # whose BMD, you would like to see. I want to see BMD for B8. Enter " 8 " & Click Ok.**
  Following message is displayed.



- **You are asked to specify Magnification Factor (MF). You have to do trial & error to achieve the required MF for appropriate display on computer screen. Keep the MF of 5. Click OK.**
  Following BMD is displayed.

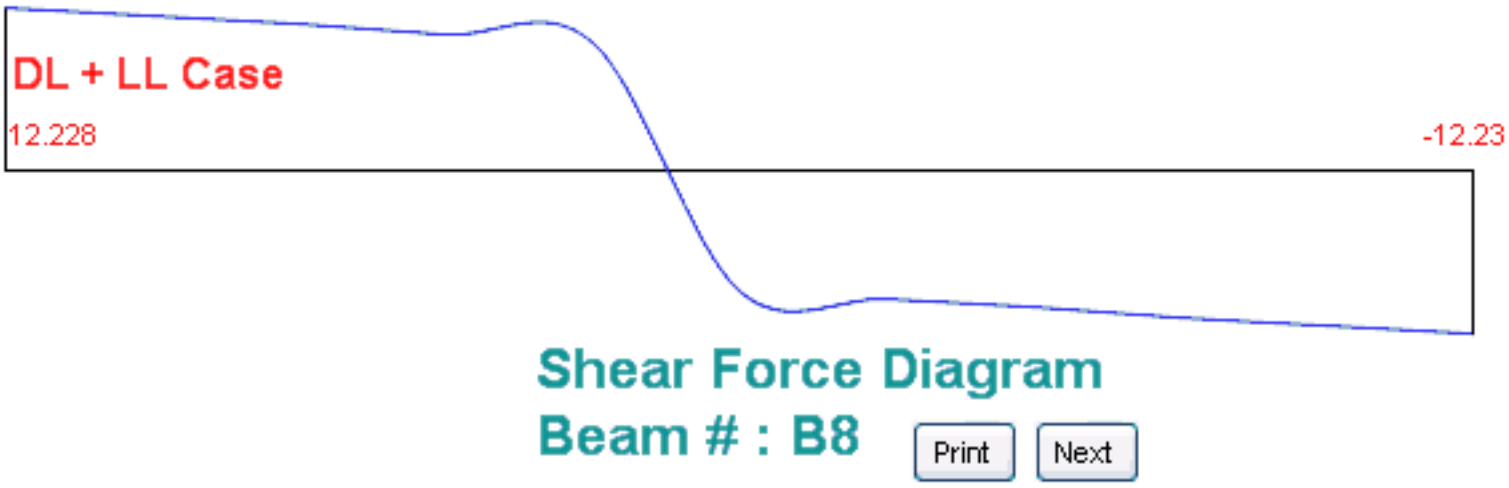


- **Note that BMD is drawn on Tension Side which reflects Deflected shape of Beam. BMD, SFD and Load Diagrams are Important from the point of Checking Results & Data Input. Any un-expected Diagram will reflect Data Error in the form of :**

• Incorrect Geometry (Span, Grid Dimension).
• Incorrect Loads (Point Load, Slab Spanning).
• Floor Analysis and Beam Design not performed after Editing / Adding Geometry or Loads.

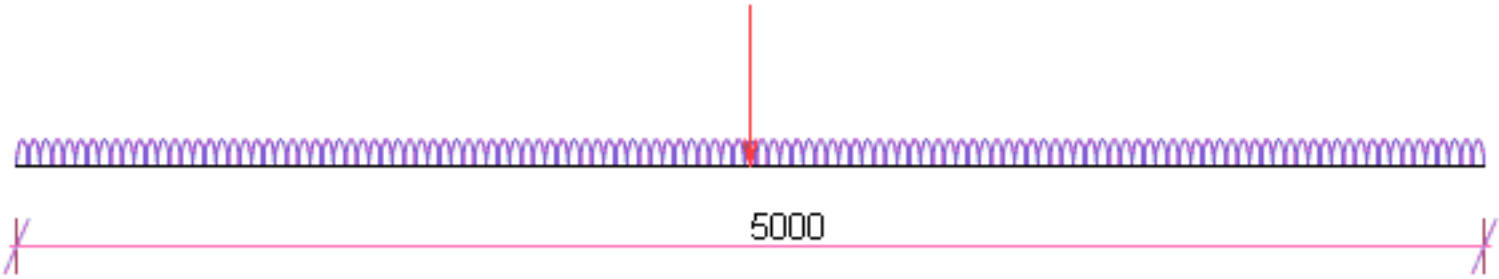
Now Click on " SFD " option. The procedure is exactly same as that of BMD.

SFD is displayed as under. MF = 5.0



- **Now Click " Loads " button. The procedure is exactly same as that of BMD / SFD.**

Load Diagram is displayed for Beam No. 24 as under. MF = 5
Near Intensity & Far Intensity = 0 means no direct Floor load on the beam.
However Point Loads from secondary Beams are present.

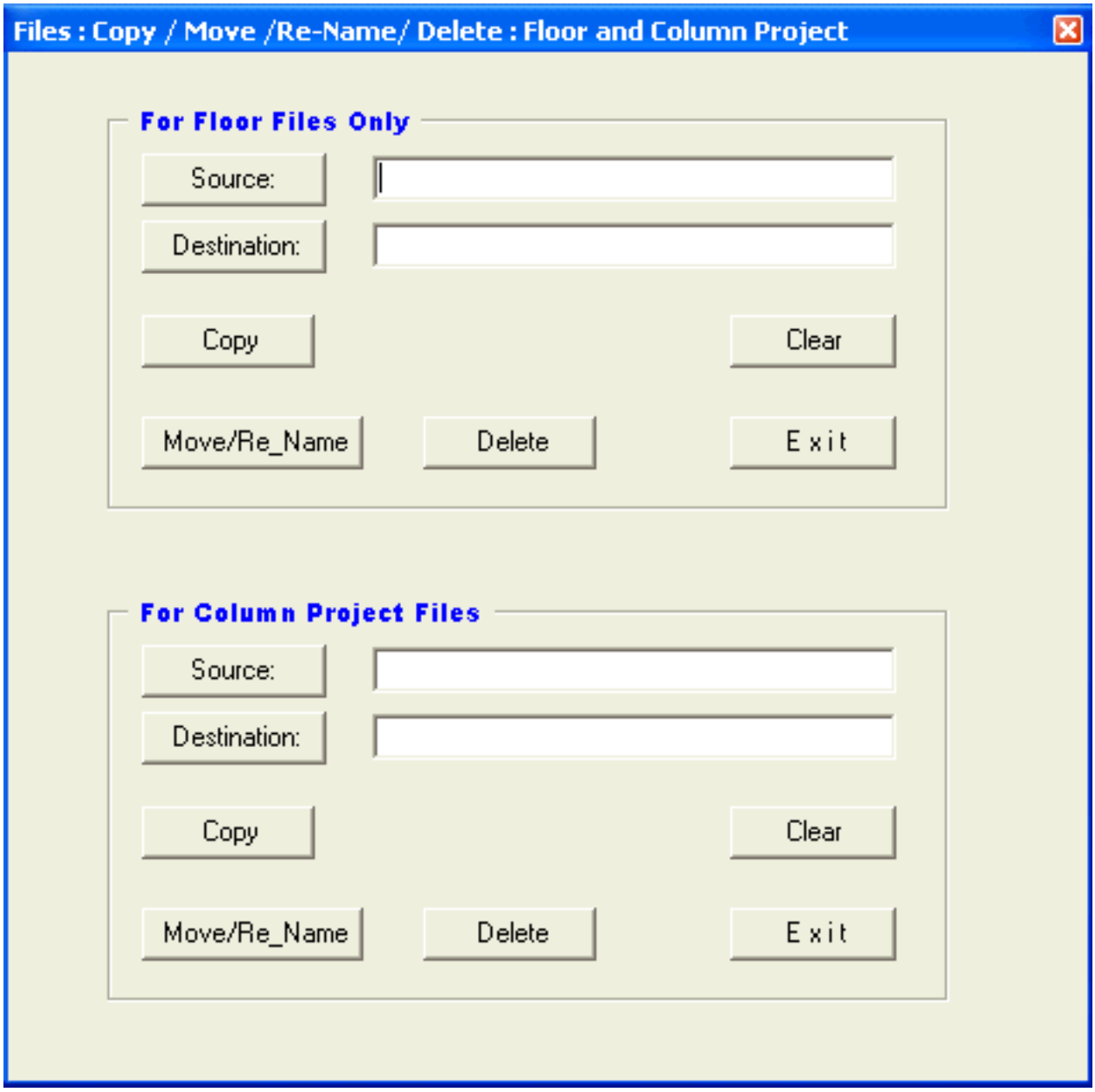


UDL in t/m = 1.29699
Near Int. in t/m = 0 @ dist. of 0 m : Far Int. in t/m = 0 @ dist. of 2.5 m
Near Int. in t/m = 0 @ dist. of 2.5 m : Far Int. in t/m = 0 @ dist. of 5 m
Point Load in t = 5.871 @ dist. of 2.5 m

Display of Loads on Beams

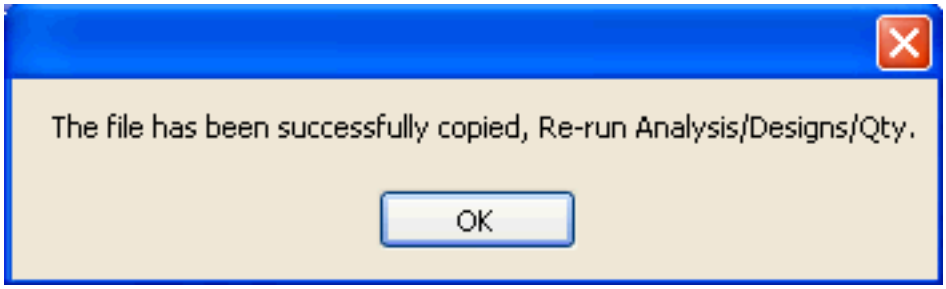
Beam # : B24 Print Next

- **The best way to check data entry is Load Diagram. Check that Loads are Correct in magnitude as well as in Location & Shape. Check the presence or absence of Point Load Reaction from Secondary beams. In the present case the reaction point load is from Beams B9, 10, 12 & B13. Check span with total of slab load distances. All distances are from LHS.**
- **Now Click " Files " button at the top.**
Following window is displayed.



- **Here we have 2 menus, one for Floor file and another for Column Project File.**

Use " For Floor Files Only " option to Copy, Delete & Move / Re-Name Floor Files.
Now we will copy Example_1 file to Example_2 file. Click " Source " Button & select Eample_1 File from the file Dialogue Box. Again Click " Destination " Button & select Eample_2 File from the file Dialogue Box. Click " Copy " button. Following Window is displayed.



- **Similarly we can use Delete Option to Delete Files, however note that there will be no " Destination " file & destination text box shall be empty.**

Note that Floor File extension is " .CMP ", while Column Project File Extension is " .DAT ".

The Column-Foundation File menu is similar to Floor File Menu, only difference is File extension.
Hence Use " For Column " option to Copy, Delete & Move / Re-Name Column and Foundation Files.

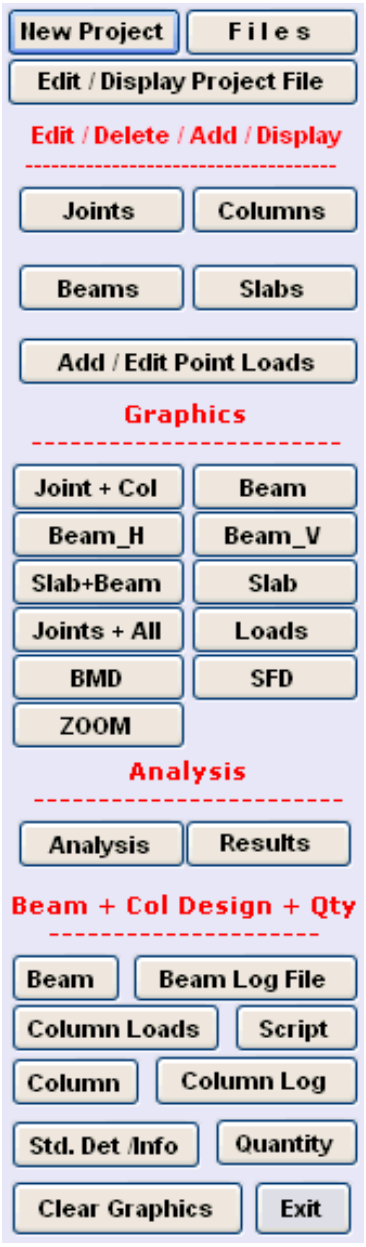
Now we have come to the end of Step # 11.

STEP NO. 11 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP 12 : CREATION OF FLOOR PLAN IN AUTOCAD

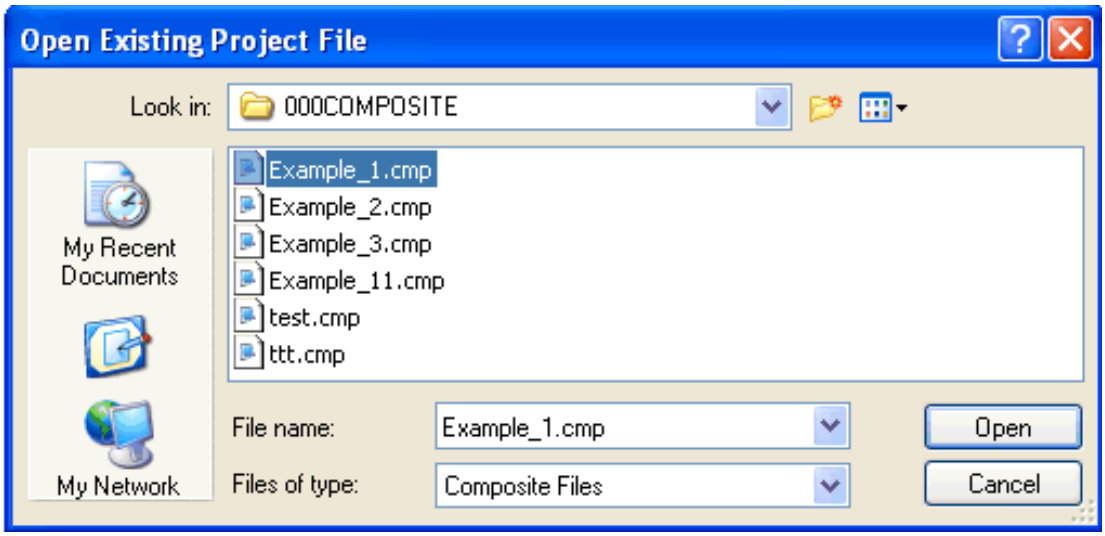
When The Program starts, following Menu below is displayed.



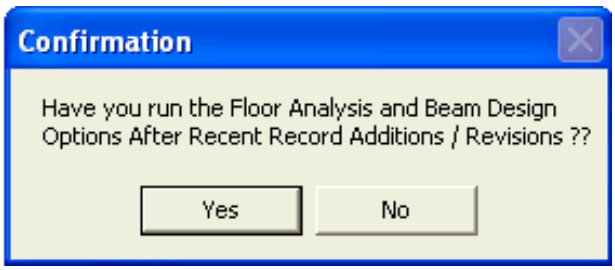
When Program starts, the Menu above is displayed.
Under the Beam + Column Design + QTY Heading following options are displayed.

- **Beam**
- **Beam Log File**
- **Column Loads**
- **Script**
- **Column**
- **Column Log**
- **Standard Details**
- **Quantities**
- **Clear Graphics**
- **Exit**

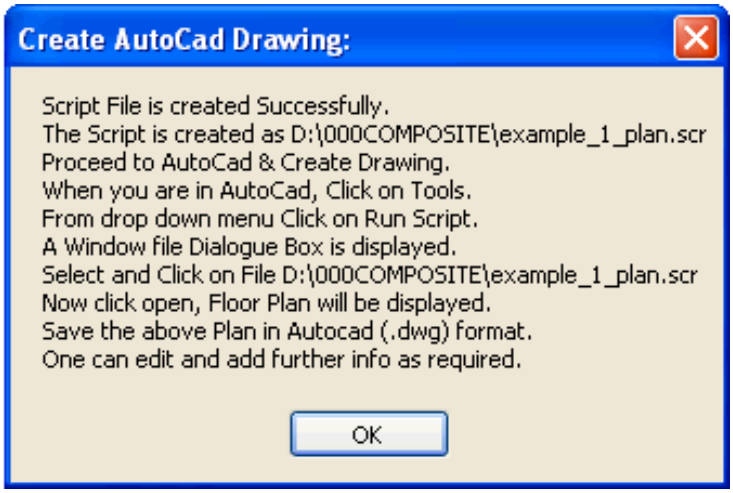
In Order to create an AutoCAD drawing, a script file has to be created first.
To create the script file, click on Script Option.
A window dialogue box appears.
Click on Example_1.cmp file and click on open.



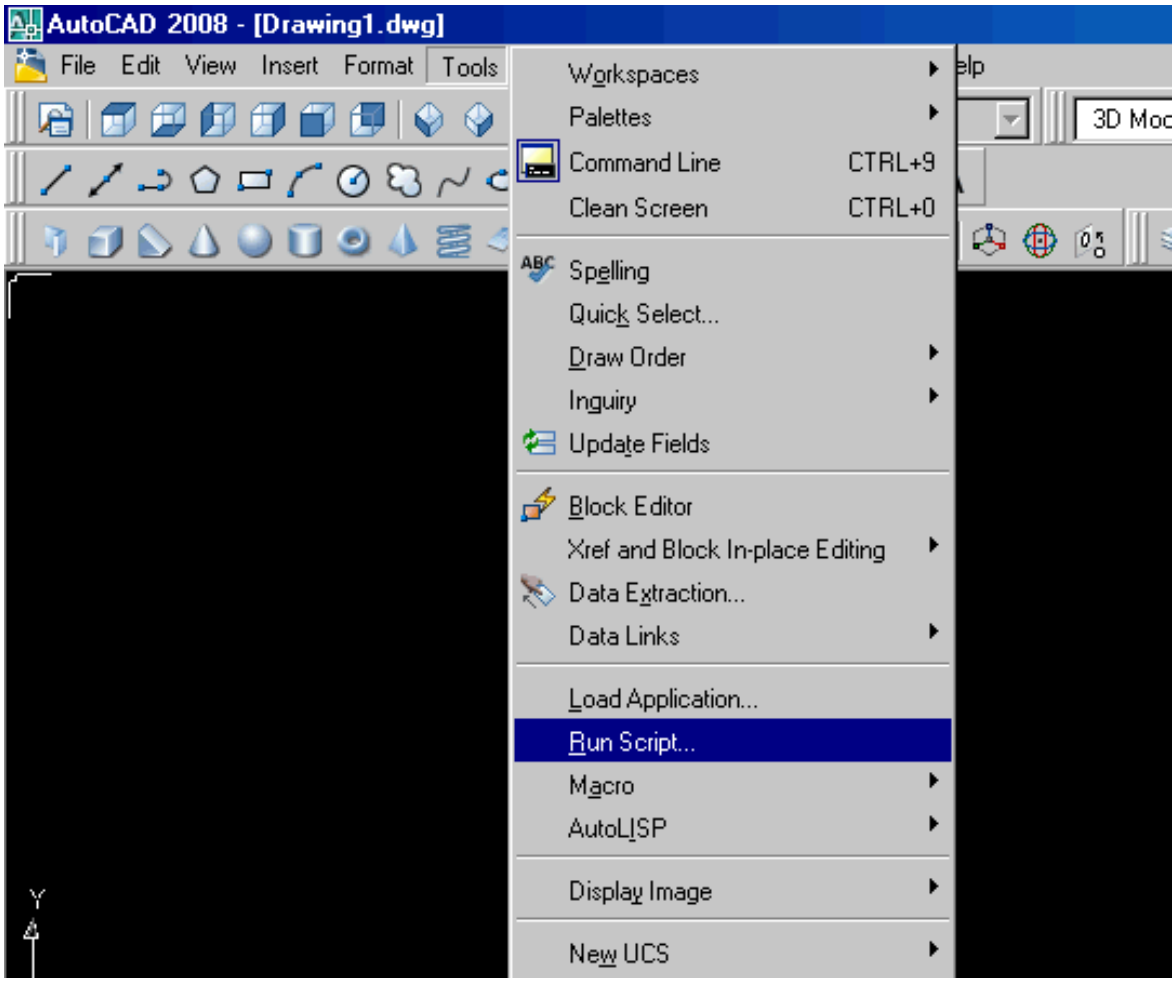
Following graphics is displayed.
Click on Yes if Floor Analysis and Beam Design Options are performed.



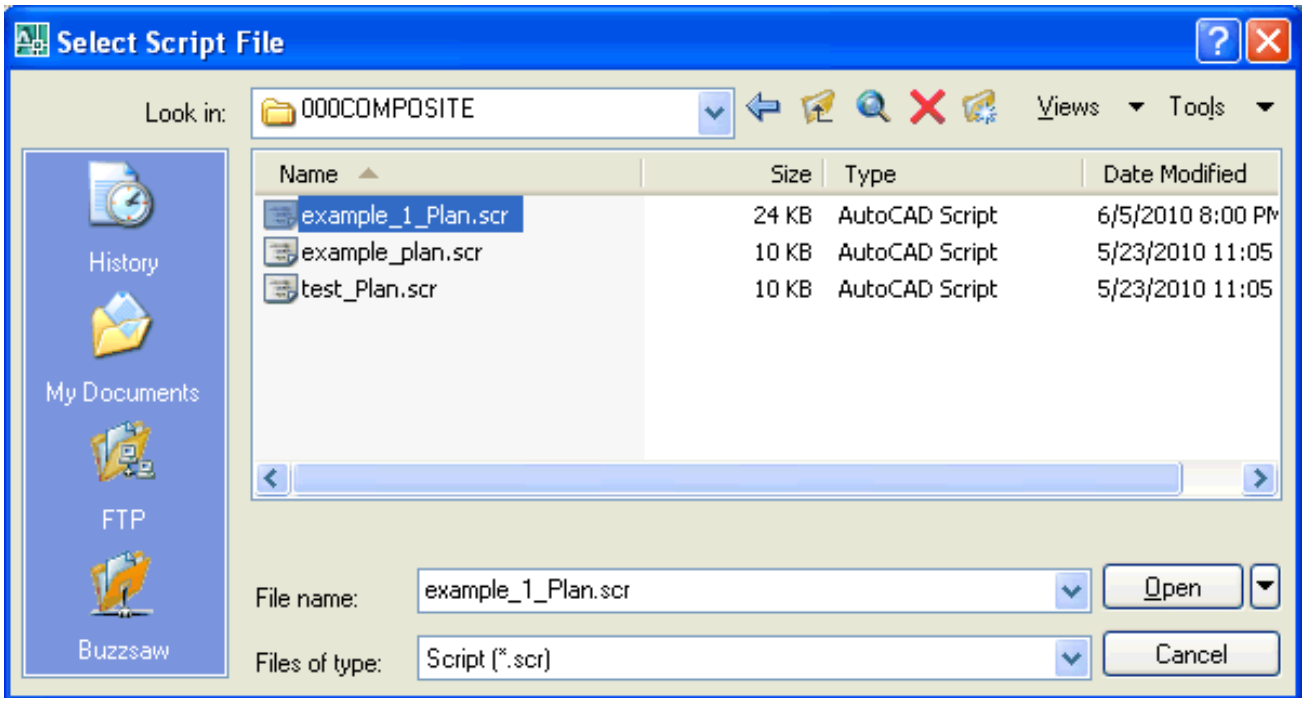
Once Yes is clicked, following graphics is displayed.
The script file is created as Example_1_plan.scr . Note that "_plan" is added to file name. The ".scr" stands for script file and not screen saver.
Now click on OK and Exit from the Program.



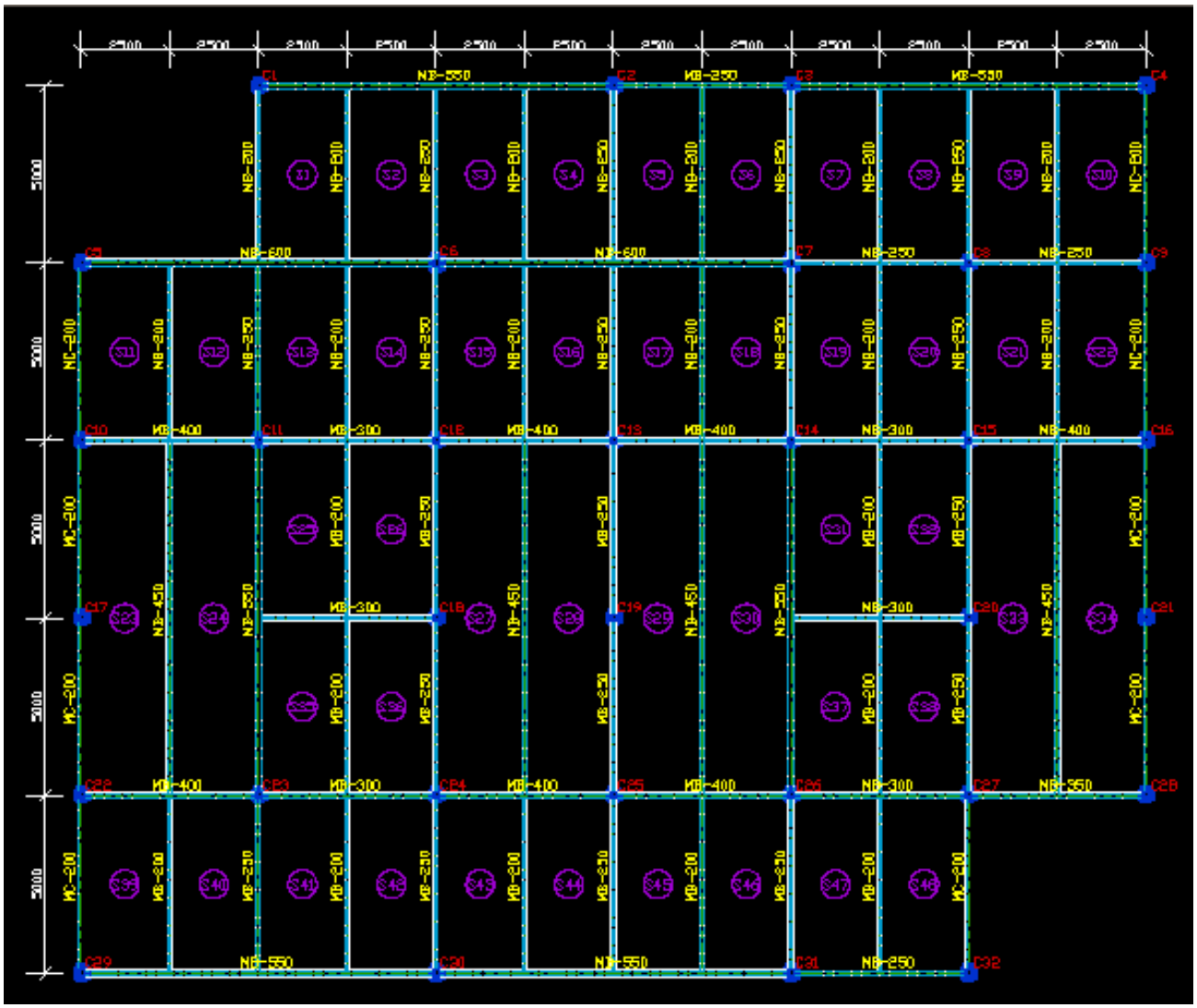
Start AutoCAD.
In AutoCAD click on Tools. From the drop down menu click on Run Script.



A window dialogue box appears .
Click on the required file and click on open.



It will take a few seconds for the script to run, after which the plan will appear in the form of AutoCAD drawing . The display will be as follows.

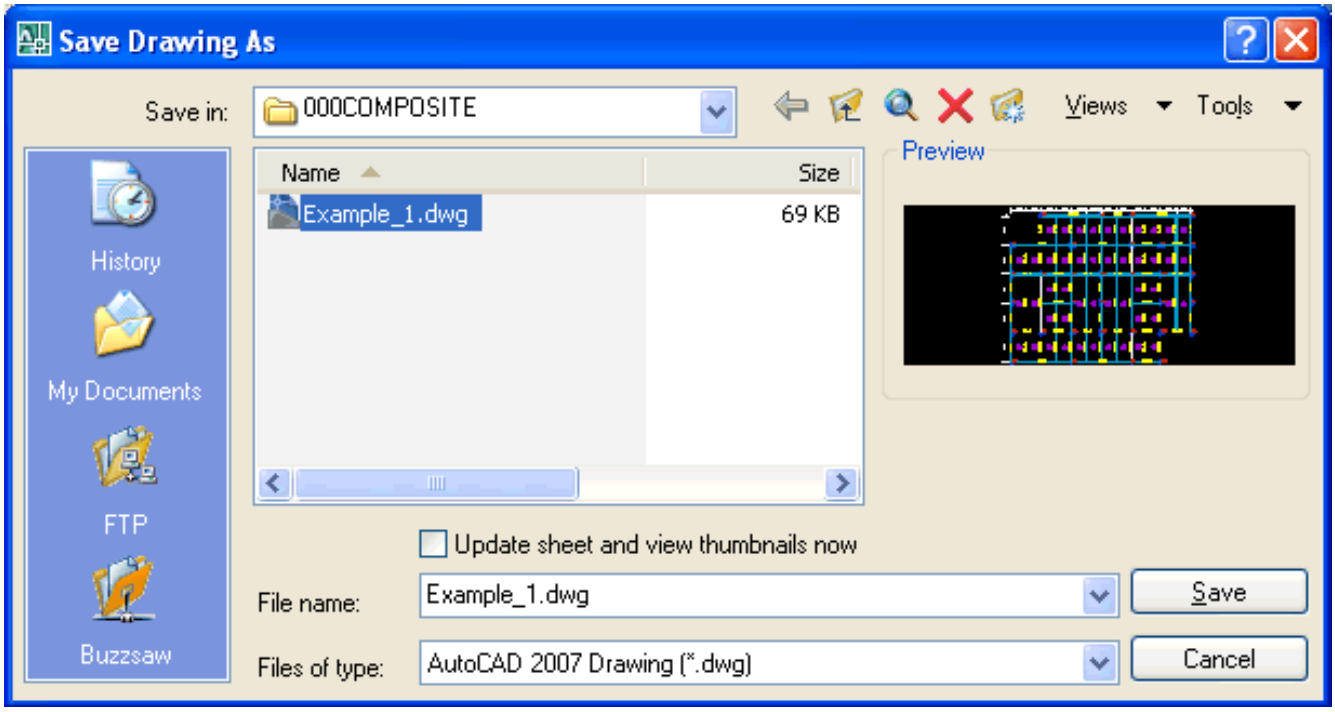


Please note that the above drawing is Editable in AutoCAD.

The above drawing is drawn in the following layers , they are
1) Beam : Denotes beam
2) BeamCen : Denotes center line of the beam
3) Beamtext : Denotes text for beam
4) Column : Denotes Columns
5) Columntext : Denotes text for columns
6) Grids : Denotes dimensions
7) Slabtext : Denotes text for Slab
8) Border : For Web Thickness

The layers can be turned Off/On at any time for convenience.
just go to format option and click on layer from the drop down menu.

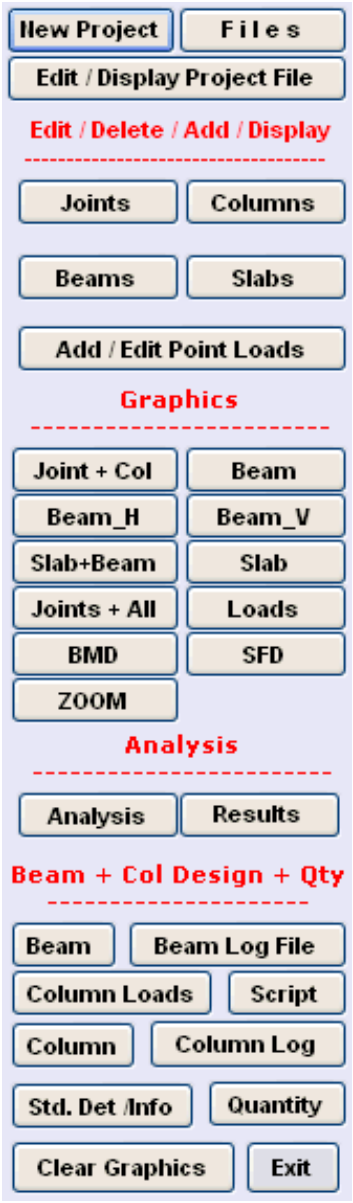
➤ **Save the above Drawing in AutoCAD i.e. (.dwg) format.**



STEP NO. 12 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 13 : Design of Building Columns

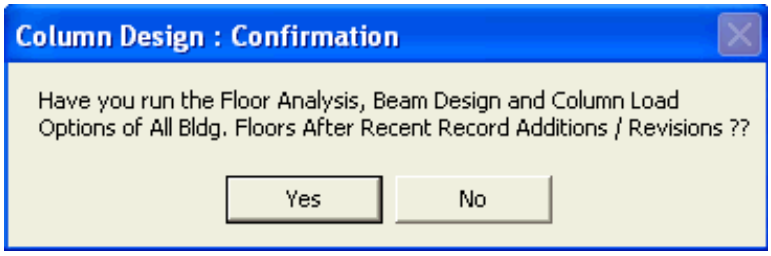


When Program starts, the Menu above is displayed.
Under the Beam + Column Design + QTY Heading following options are displayed.

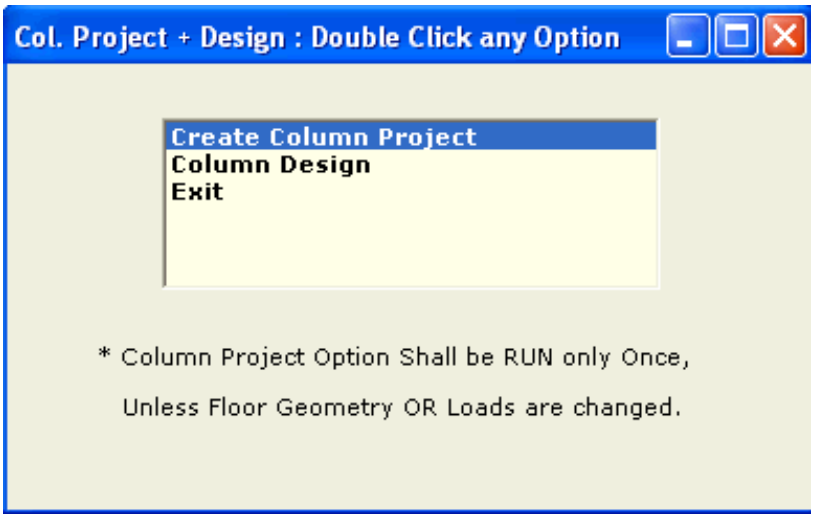
- **Beam**
- **Beam Log File**
- **Column Loads**
- **Script**
- **Column**
- **Column Log**
- **Standard Details / Info**
- **Quantities**
- **Clear Graphics**
- **Exit**

Now Click on " Column " Option.

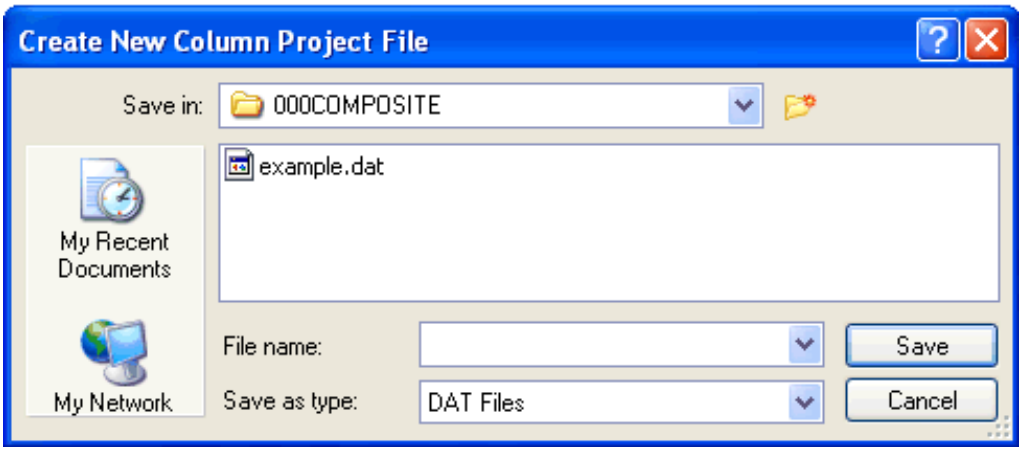
Following Important Message is displayed.



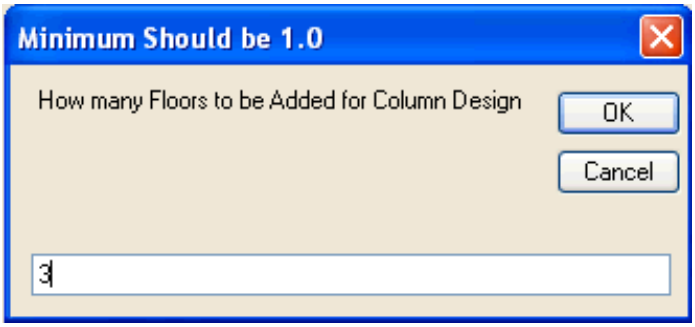
In order to Design all the Columns of Building, we must first RUN the Analysis, Beam Design and Column Loads of individual Floors. If Columns are having External Loads than they should be incorporated using Edit / Delete / Add / Display option (Refer Step 5) before performing floor analysis. Also any changes to column parameters should be carried out at this stage.
If Yes is clicked, following dialogue window is displayed.



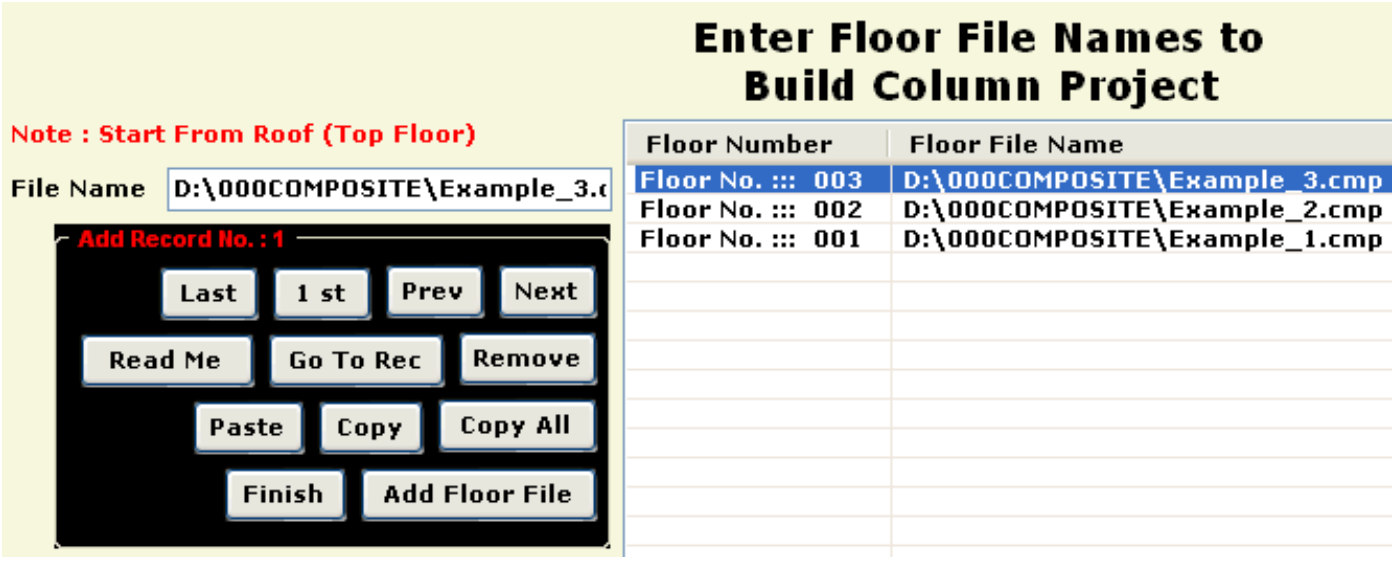
Before Designing all Building Columns, a user has to create Column Project. The Column Project option shall be RUN only once, unless Individual Floor Geometry Or Loads are Changed. Double Click to get following display.



Give a suitable name to Column Project File for Design. I have given "EXAMPLE" as the file name. Note that column project file name is with extension ".DAT", while floor file extensions are with ". CMP". In order to remove any confusion, a user should give different file name to Column project and respective Floor Files. Click save button, following window will appear.



➤ **Enter Total number of Structural Steel Floors in the Building and Click OK button. Here I have a 3 storey building. A new window appears.**



➤ **In order to Build Column Project, I have indicated 3 Floor files, corresponding to 3 floors. Actually Example_1, Example_2 and Example_3 are typical floors. I have created, analyzed and designed only one floor (Example_1) and copied this floor file to Example_2 and Example_3 using Files Option.**

Click Read Me button following relevant info is displayed.

- **This option Develops Column Project File.**
- **User has to specify File name for each floor of the Building. Use Add Floor File Button.**
- **File name for each floor shall be different.**
- **File name for floors cannot be repeated.**
- **The program will add column loads of respective floors for each column.**
- **This option is to be Run after Analysis / Design and Column Load option of each floor (File) has been performed.**
- **Start from the TOP floor.**
- **Floors of the Same Building is to be Added.**
- **Same Floor File can be given repeatedly in case of Typical floors Using Copy All Button, and Later Edited to make different file name.**
- **All Columns on the Floors should be same in Numbers & Location.**
- **Column Designation on all floors should be Same.**
- **Column Addition or Deletion between floors is not Permitted.**
- **Floors will always start from Floor no. -> 001, corresponding to 1st Floor.**
- **However for Building Column project always Enter File Names from Roof / Top Floor Downwards.**

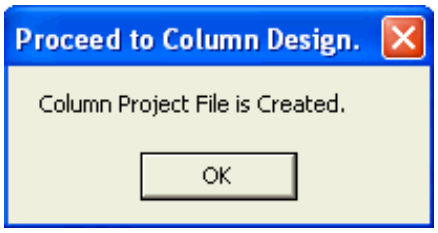
There is no restriction on Number of Floors, except your computer's memory.

Click Finish Button to Proceed. Following window will appear displaying Column Cumulative Loads. {Part}

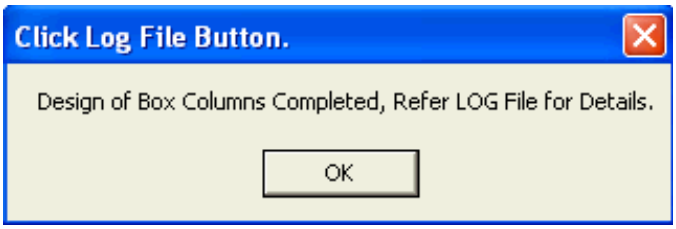
DISPLAYING UNFACTORED COLUMN LOADS OF BUILDING

Column Number	Cummulative Loads
C1 : Floor No. ::: 003	23.551
C1 : Floor No. ::: 002	47.102
C1 : Floor No. ::: 001	70.695
C2 : Floor No. ::: 003	32.661
C2 : Floor No. ::: 002	65.323
C2 : Floor No. ::: 001	98.027
C3 : Floor No. ::: 003	32.661
C3 : Floor No. ::: 002	65.323
C3 : Floor No. ::: 001	98.027
C4 : Floor No. ::: 003	23.543
C4 : Floor No. ::: 002	47.086
C4 : Floor No. ::: 001	70.671
C5 : Floor No. ::: 003	28.161
C5 : Floor No. ::: 002	56.322
C5 : Floor No. ::: 001	84.525
C6 : Floor No. ::: 003	70.789
C6 : Floor No. ::: 002	141.578
C6 : Floor No. ::: 001	212.513
C7 : Floor No. ::: 003	55.135
C7 : Floor No. ::: 002	110.27
C7 : Floor No. ::: 001	165.551
C8 : Floor No. ::: 003	36.547
C8 : Floor No. ::: 002	73.094
C8 : Floor No. ::: 001	109.683
C9 : Floor No. ::: 003	21.551
C9 : Floor No. ::: 002	43.102
C9 : Floor No. ::: 001	64.695
C10 : Floor No. ::: 003	24.667
C10 : Floor No. ::: 002	49.334
C10 : Floor No. ::: 001	74.043
C11 : Floor No. ::: 003	53.628
C11 : Floor No. ::: 002	107.256

Click OK button, Following message is displayed.



Now Double Click Column Design Option. Following message is displayed.



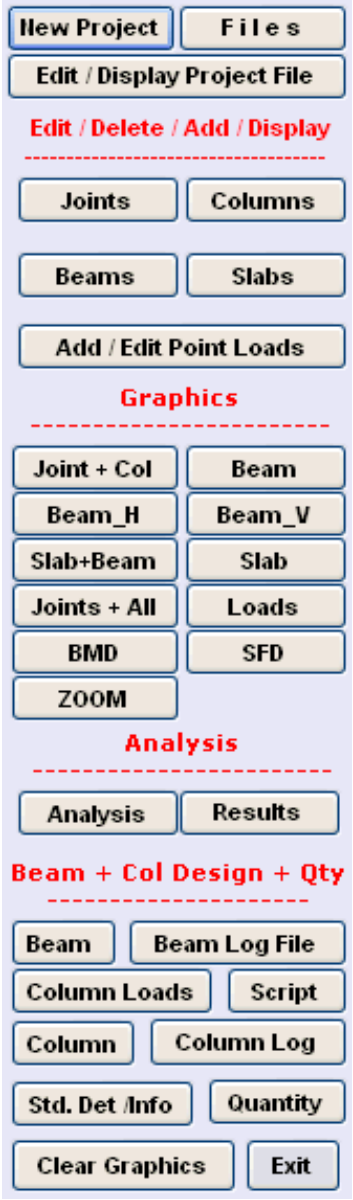
Just Like Beam Design, the Column Design Results (Safe / Un-Safe) are stored in Log File. If you click the Column Log File Button, various parameters on which Column Section is evaluated is displayed. User should study each parameter in order to redesign the Column for safety or for Economy / Optimization.
Double Click Exit to leave Column Option.

Note that Unless all Beams and Columns Designs are Safe there is no Use of Proceeding to Column and Total Building Quantities and Costs.

STEP NO. 13 IS OVER.

LEARN COMPOSITE STEP BY STEP

STEP NO. 14 : Column and Project Quantities And Cost

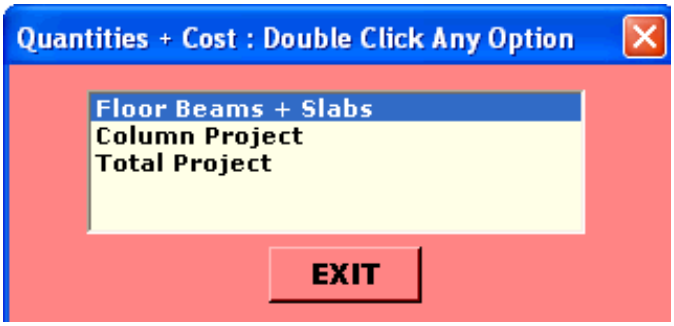


When Program starts, the Menu above is displayed.
Under the Beam + Column Design + QTY Heading following options are displayed.

- **Beam**
- **Beam Log File**
- **Column Loads**
- **Script**
- **Column**
- **Column Log**
- **Standard Details**
- **Quantities**
- **Clear Graphics**
- **Exit**

Now Click on " Quantity " Option.

Following Message is displayed.



- **Now Double Click " Column Project " Option. Following Graphics is displayed. (Quantities and Cost of Floor Beams + Slabs are already Covered in Step 10).**
- **Unless All relevant Floors of the Building are Analyzed and Beam and Columns are Designed, Column Loads worked out, the Calculation of Quantities & Cost of Column / Project have no meaning. The Quantity Option is to be RUN last after all other things are completed. Click Yes if complied. Select Your Project File from Window Dialogue Box (Example.Dat in My case) & Click Open. Following Graphics is Displayed.**

SUMMARY OF STRUCTURAL STEEL COLUMNS IN KG

MC 100	0
MC 125	0
MC 150	0
MC 200	0
MC 250	0
MC 300	0
MC 350	0
MC 400	0
MB 100	0
MB 150	0
MB 200	0
MB 250	13875.53
MB 300	8486.402
MB 350	0
MB 400	739.198
MB 450	0
MB 500	0
MB 550	0
MB 600	0

TOTAL STRUCTURAL STEEL IN COLUMNS IN TONS : 23.101

TOTAL COLUMN COST : 1155050

- **Click OK to Exit.**
  Now Double Click " Total Project " Option. Select your File from Window Dialogue Box & Click Open.
  Following Graphics is displayed.

SUMMARY OF PROJECT QUANTITIES AND COST

File Name : D:\000COMPOSITE\example.dat

Project # : 8912

Date : 05 June 2010

Bldg. ID : Admin

FY : 250

STEEL SUMMARY IN KG

MC100 :	0
MC125 :	0
MC150 :	0
MC200 :	2983.5
MC250 :	0
MC300 :	0
MC350 :	0
MC400 :	0
MB100 :	0
MB150 :	0
MB200 :	8001
MB250 :	26744.03
MB300 :	12464.4
MB350 :	786
MB400 :	7207.198
MB450 :	8688
MB500 :	0
MB550 :	18666
MB600 :	7356

No. of Floors = 3

Total Steel in Tons = 92.896

Total Steel Cost = 4644800

Total Masonry in M2 = 2132.55

Total Masonry Cost = 1812668

Total Plaster in M2 = 4265.1

Total Plaster Cost = 1706040

Total Painting in M2 = 4265.1

Total Painting Cost = 426510

Conc. Flooring in M3 = 256.2

Conc. Flooring Cost = 1281000

Reinforcement in Kg = 7730.871

Reinforcement Cost = 386543.7

Deck Profile + Studs in M2 = 2100

Deck Profile + Studs Cost = 1050000

False Ceiling in M2 = 2100

False Ceiling Cost = 840000

Total Door / Window in M2 = 277.2

Total Door / Window Cost = 693000

Total Cement Bags in Nos. = 4299

Total Sand in M3 = 189

Total Aggregate in M3 = 204

Print OK

Total Project Cost = 12840562

Total Floor Area in M2 = 2100
Cost per M2 = 6114.553
Cost per sft = 568.795
Cement Bags per sft = 0.19
Str. Steel in Kg per sft = 4.115
Str. Steel in Kg / M2 = 44.236
Str. Steel Cost as % of Total = 36.172
Masonry Cost as % of Total = 14.116
Flooring Cost as % of Total = 21.163

This Completes Learning Steps.

STEP NO. 14 IS OVER.

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

2D FRAME ANALYSIS - Discover The Beauty Of Structural Analysis

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.

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