LEARN RCF

A Software for Analysis, Design, Estimation
& Costing of RCC Floors

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 R C F

A Software for Analysis, Design, Estimation, Costing
& Drawing of Reinforced Concrete Building

Introduction	INTRO & LIMITATION
New Project (File) Creation	STEP NO. 1
Scan Joint, Beam, Column & Slab Data from AutoCAD Drawing	STEP NO. 2
OR	
Automatic Joint Number Creation	STEP NO. 2
Delete Un-Wanted Joints	STEP NO. 3
Delete and Edit Beams	STEP NO. 4
Delete and Edit Columns	STEP NO. 5
Delete and Edit Slabs	STEP NO. 6
Add & Edit Point Loads to Beam	STEP NO. 7
Add & Edit Beam Continuity	STEP NO. 8

LEARN RCF STEP BY STEP

INTRO & LIMITATIONS

- **Please take Print Out of Every Step, including this page before commencing Learn. Take a Yellow Marker Pen and Mark the Learning Process while Proceeding further. This is Essential for Learning.**
- **The software performs Analysis, Design, Estimation & Costing of A RCC Floor at a given Uniform Level (2D). Multiple Level Floors (3D) cannot be analyzed. RCF also designs building Columns and Isolated Footings when Column Project file is created by the user. Column & Footing quantities & costs are added to floor costs to arrive at complete building cost.**

  The Software basically requires a User to enter floor data for Joints, Columns, Beams, Slabs, Point loads & Continuity. The rest of the things are taken care of by the software.

  The results are displayed in the form of BM & SF, Beam & Slab Schedule, Quantities, Cost, Bar bending Schedule for Beams & 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). Only Beam Schedule will be Printed in Landscape Orientation.

  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.**
- **Beams / Columns / Slabs shall be along two mutually perpendicular axis (X and Y). Polygonal (Multi-sided) slabs cannot be analyzed.**
- **Only three (3) Load Cases can be analyzed per File, viz :**

  • DL + LL

  • { DL + LL + Wind Or Seismic End Moment (WL1). } * 0.80

  Wind Or Seismic End Moment shall be entered by the user on individual beams having Columns supports.

  • { DL + LL + Wind Or Seismic End Moment (WL2). } * 0.80

  Whatever will the value of WL1, the program will reverse its sign (Multiply it by - 1) and use it as the default 3rd Case.

  • Multiplication Factor of 0.80 will be applied within the Program by default.

  End Moments (Wind or Seismic) 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 Moments.

  More cases can be obtained by manipulating initial values as given in the project file or individual Beam and Slab files. For example Analysis option can be re-run with LL decreased by 0.8 (0.5* 0.8 = 0.4 T/M2). Similarly masonry thickness can be decreased or default Partition load can be changed to get desired loading on the floor. However a designer has to 1st copy the original file in to another file (using Copy Option) & do the above mentioned modifications.

- **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), " Beam Schedule ", " Slab Schedule " & " Column Loads " option for Exporting Results to Excel Spread Sheet & its subsequent Manipulation.

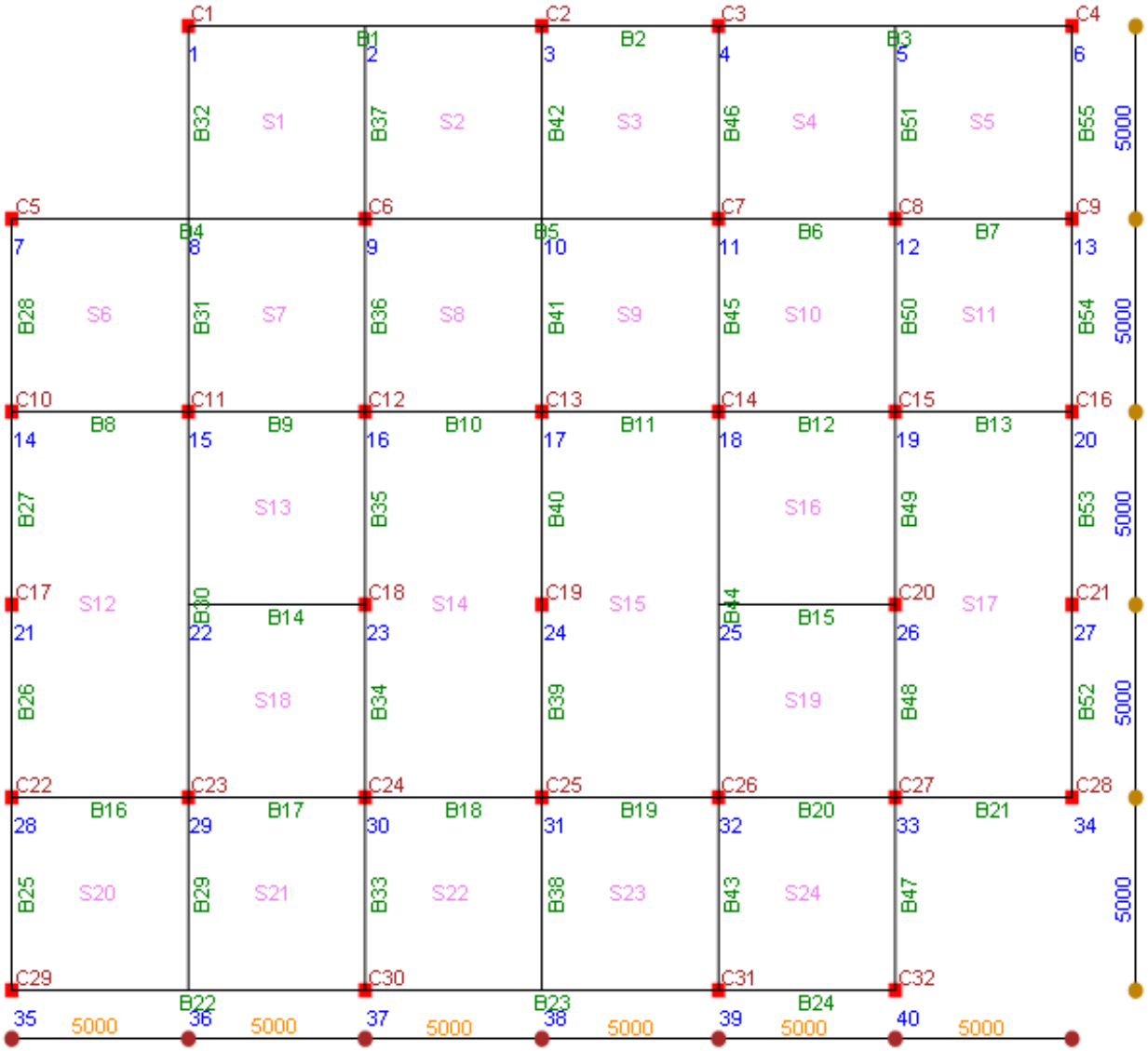
- **Intersecting Joints between two Beams (Main & Secondary) is assumed as Hinged. Hence no Moment transfer is envisaged.**

- **Connection between End Column and Beam is considered as Hinged. Hence no Moment transfer is envisaged between Column and Beam. However End Moments can be applied at joints as mentioned above under Load Case WL1.**
- **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.**
- **Beam & Slab Design is as per IS 456 - 2000. For rigorous Beam / Slab design taking in to account Durability aspect, refer our " Super Civil CD " software.**
- **Beam Bar bending codes and details are as per standard drawings.**
- **Beam Width / Depth < 150 mm not permitted.**
- **Beam Width / Depth > 3500 mm not permitted.**
- **Links (Stirrups) Area > 22.4 cm2 for Beams is not permitted.**
- **Beam Reinforcement > 4 % not permitted.**
- **Slab Thickness > 600 mm not allowed.**
- **Age factor is considered as 1.1 .**
- **skin reinforcement is provided when web depth is more than 750 mm.**

- **All Columns are placed Centrally with respect to Beams in either direction. There is no provision to offset the column in either direction. If the offset is large than user should re-workout the Beam Span. Column design is fully automatic, Input from the User is not required. Column Size is governed by Initial size given by the user and allowable standard size and reinforcement %, Refer Step No. 5 and 16.**
- **All Foundations are designed as Isolated Footings under Pure Compression. No moments are allowed in any direction. User should Analyze all Building frames considering Base as Hinged. Isolated Footing design is fully automatic, Input from the User is not required. Footing Size is governed by allowable bearing capacity (SBC) of soil and the initial size given by the user. Footing is optimized by having offset in either direction from column as equal, hence footing reinforcement in both direction is same. Overlapping footings should be corrected by changing the footing dimension, keeping the required base area constant OR by making Combine footing / Raft / Piles etc.**
- **Minimum Computer RAM memory of 1 GB is recommended.**
- **Use Laser OR Ink Jet Printer.**

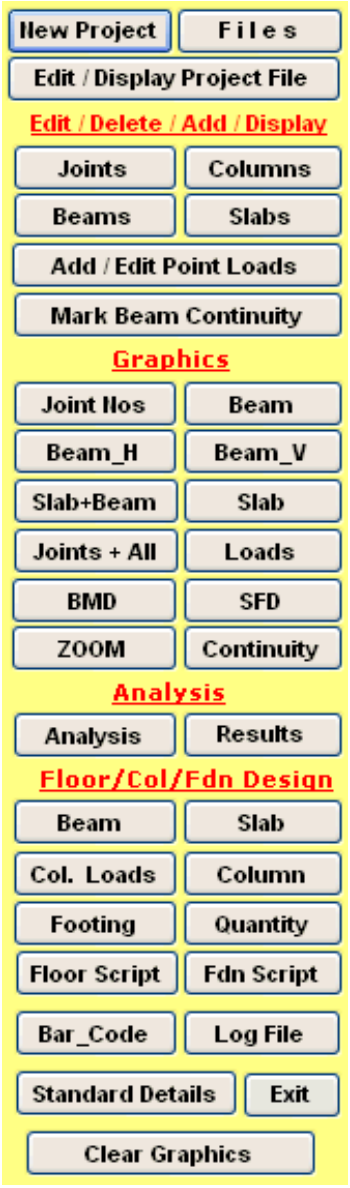
LEARN RCF STEP BY STEP

STEP NO. 1 : New Project (File) Creation



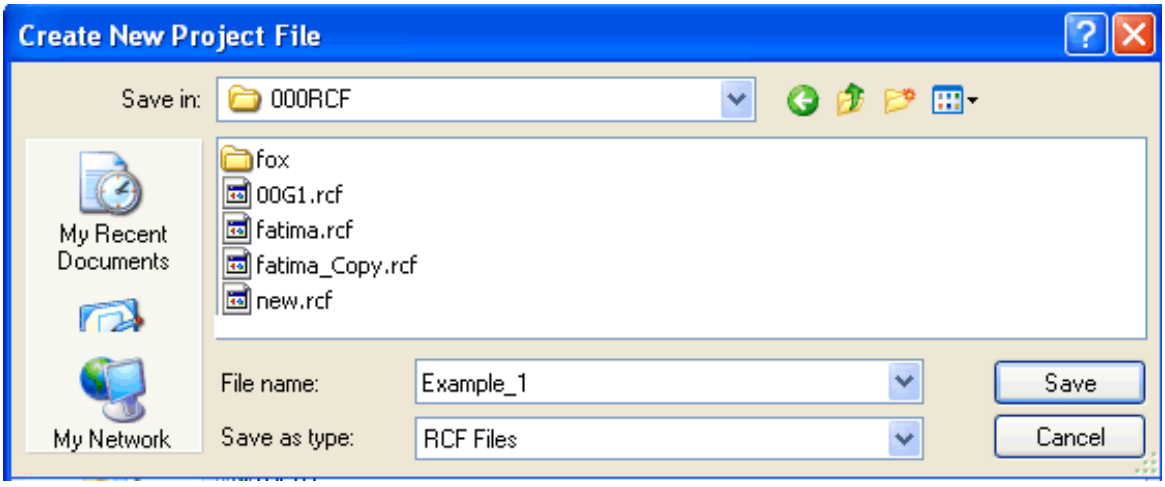
ACTUAL REQUIRED FLOOR PLAN

- **Refer the above Floor Drawing. Our Intention is to Analyze, Design, Estimate, Cost & Prepare BBS for the same. The above floor has 40 # of Joints, 32 # of Columns, 55 # of Beams and 24 # of Slabs.**
  Please go through the following steps carefully, so that we can achieve our object efficiently.

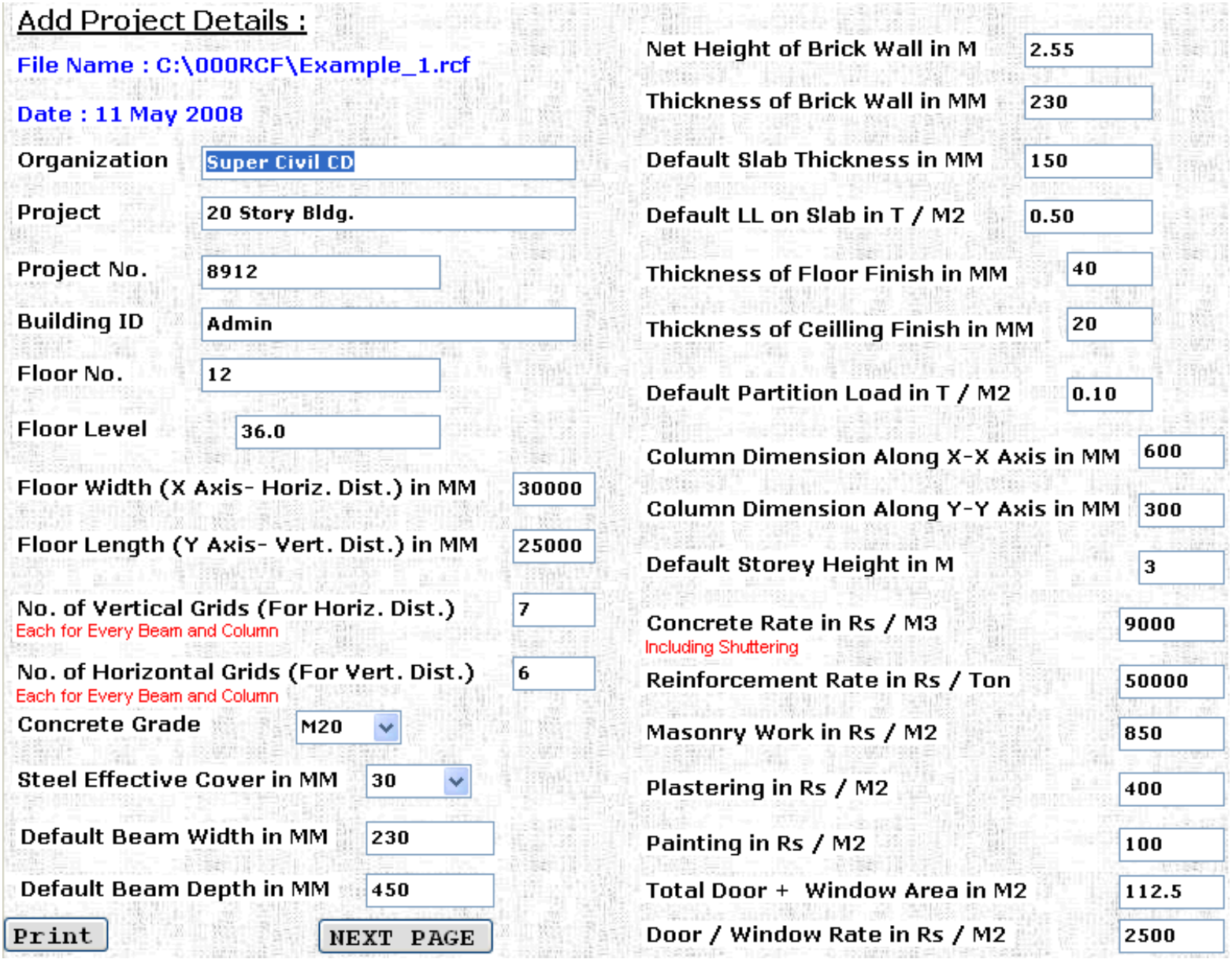


- **When Program starts, the graphics above is displayed. Consider the " New Project Option ".**

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

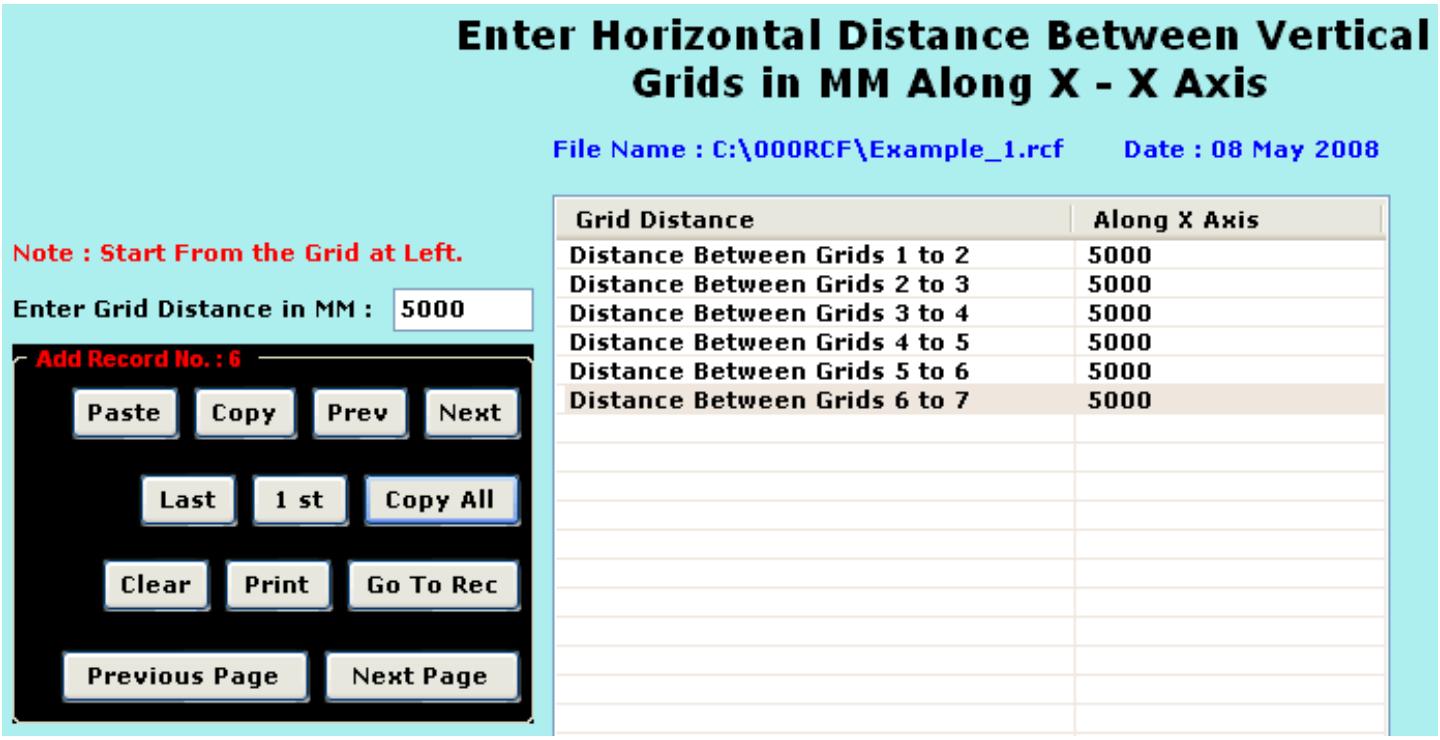


- **You must create a separate Folder / Directory to store your files.**
  I have created a Directory called " 000RCF " in C drive to store my Project files.
  Now go to this folder & give a file name to your project. I have given " Example_1 " as the name of my new project file. Click the save button.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



- **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 can only change Building information, SBC, Foundation Depth and Material Rate values later. Other vital parameters cannot be changed, so be careful while giving initial info.
  The total floor width & length values will be used to tally the sum of individual Vertical and Horizontal Grids.
  The automatic creation of Joint Numbers & Co-Ordinate system depends up on total width, length & No. of vertical & horizontal Grids of floor.

  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 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.

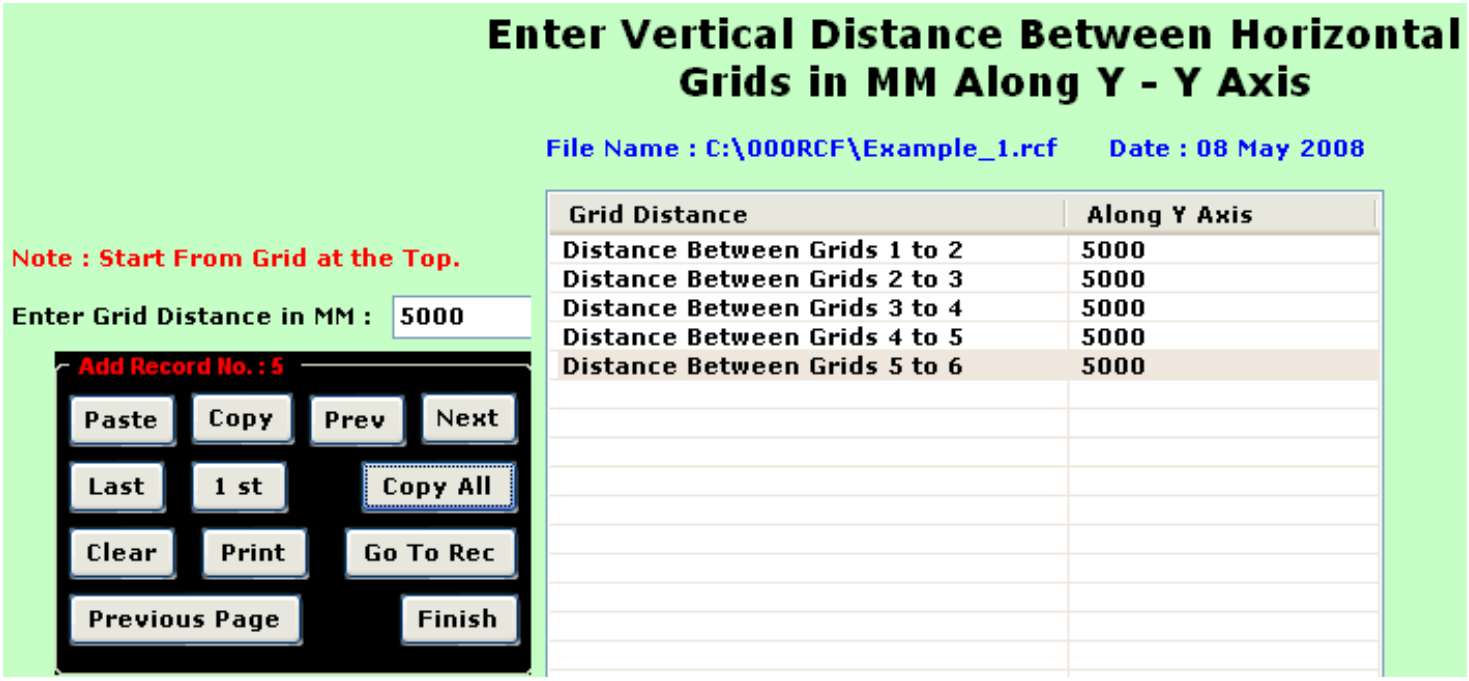
Use Copy & Paste Button to copy & paste values to different rows, in case the grid distances are not same.

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.

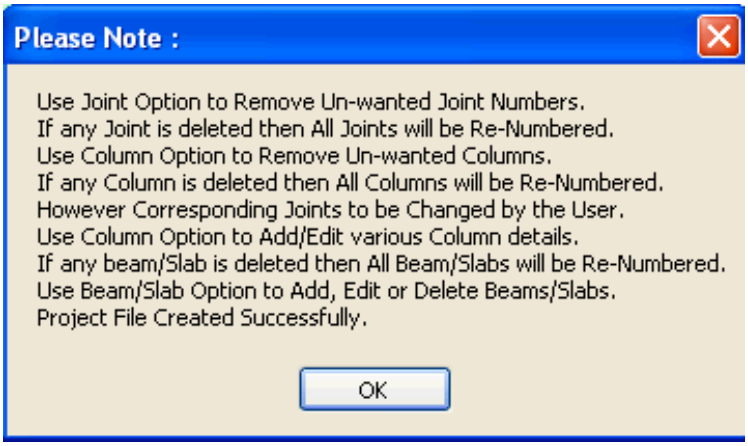
Use Copy & Paste Button to copy & paste values to different rows, in case the grid distances are not same.

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.

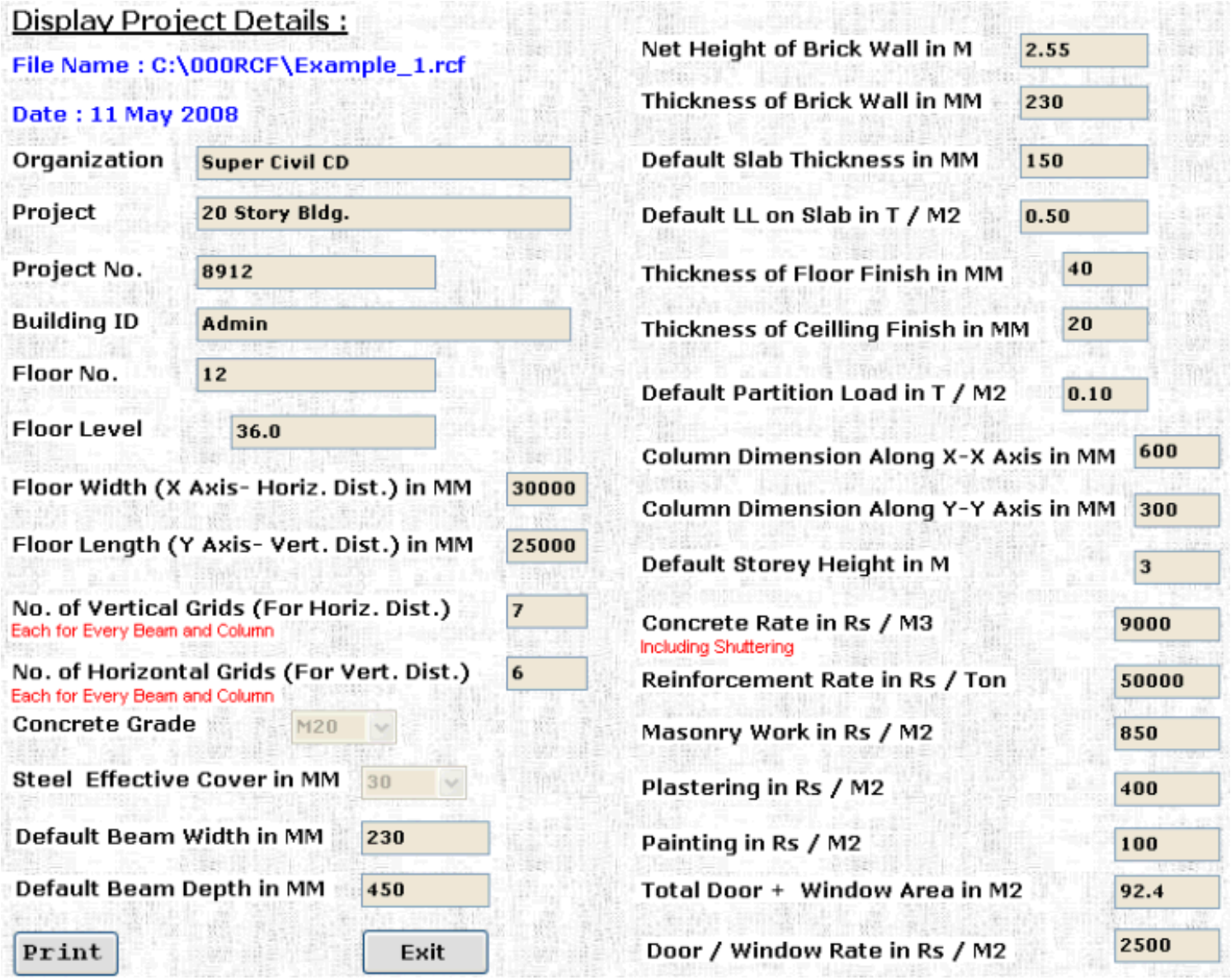
The " Print " Button is for printing of values from the Table. Use laser OR Inkjet Printer.

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 Columns, Beams & Slabs. Now the Columns, Beams & Slabs will be automatically re-numbered.
  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 see the Project File Once again just click " Edit / Display Project File Option ". Following window will display the project file. Note that Only Building information, SBC, Foundation Depth and Material Rates can be edited.**



STEP NO. 1 IS OVER.

LEARN RCF STEP BY STEP

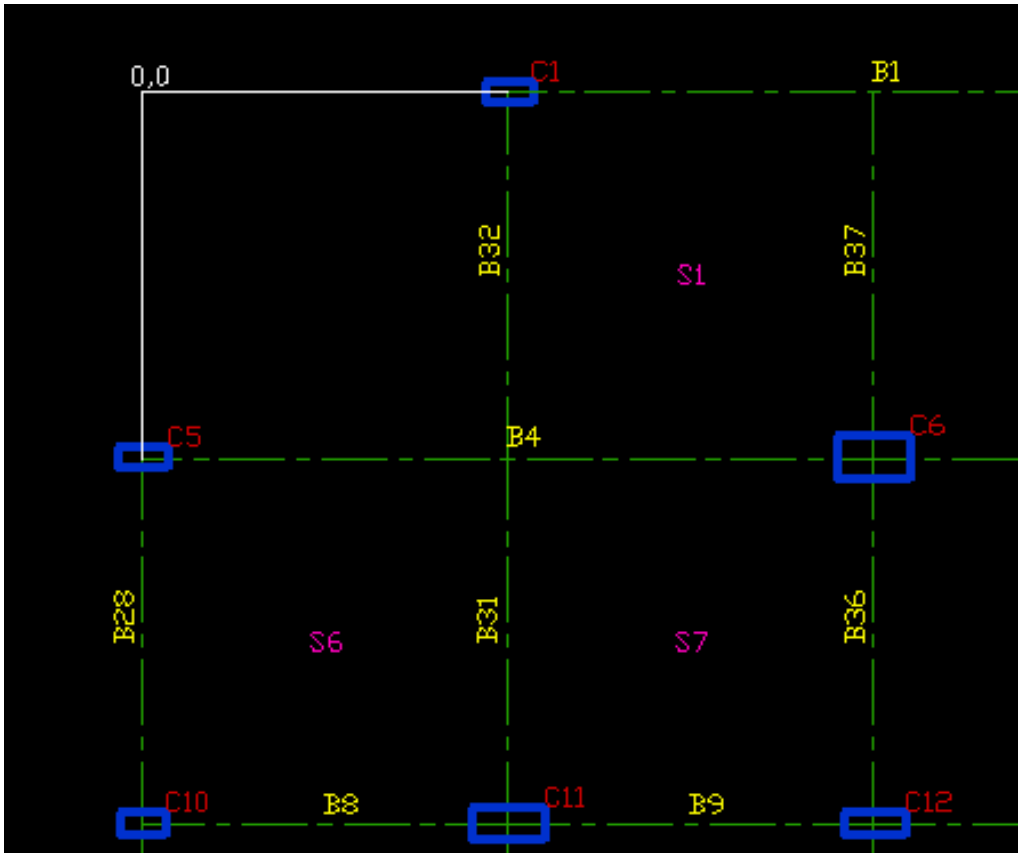
STEP NO. 2 (Alternate) : Scan Joint, Beam, Column & Slab Data from AutoCAD Drawing

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

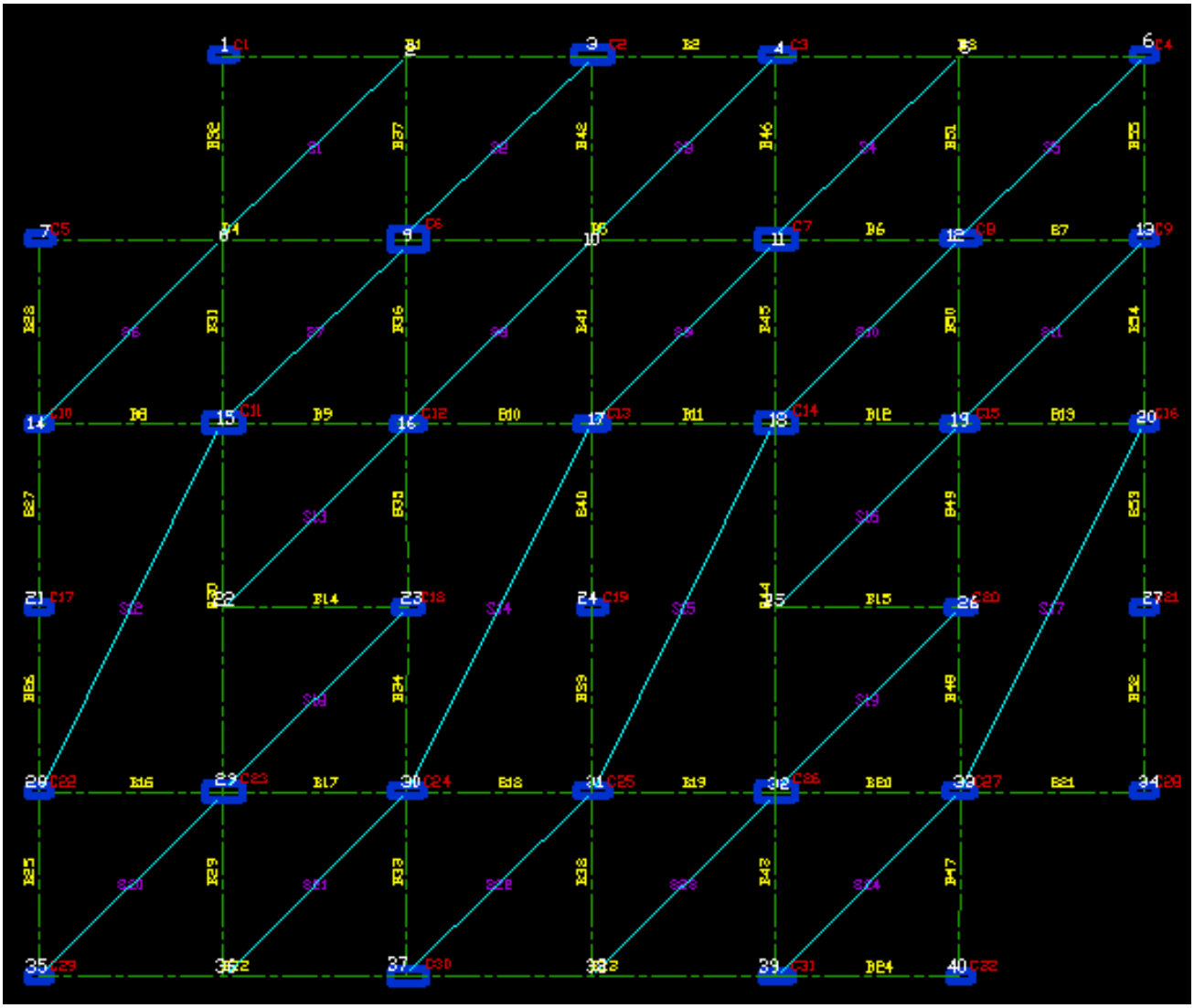
- **The Drawing Components to be drawn to exact scale and in Millimeter (MM).**

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.



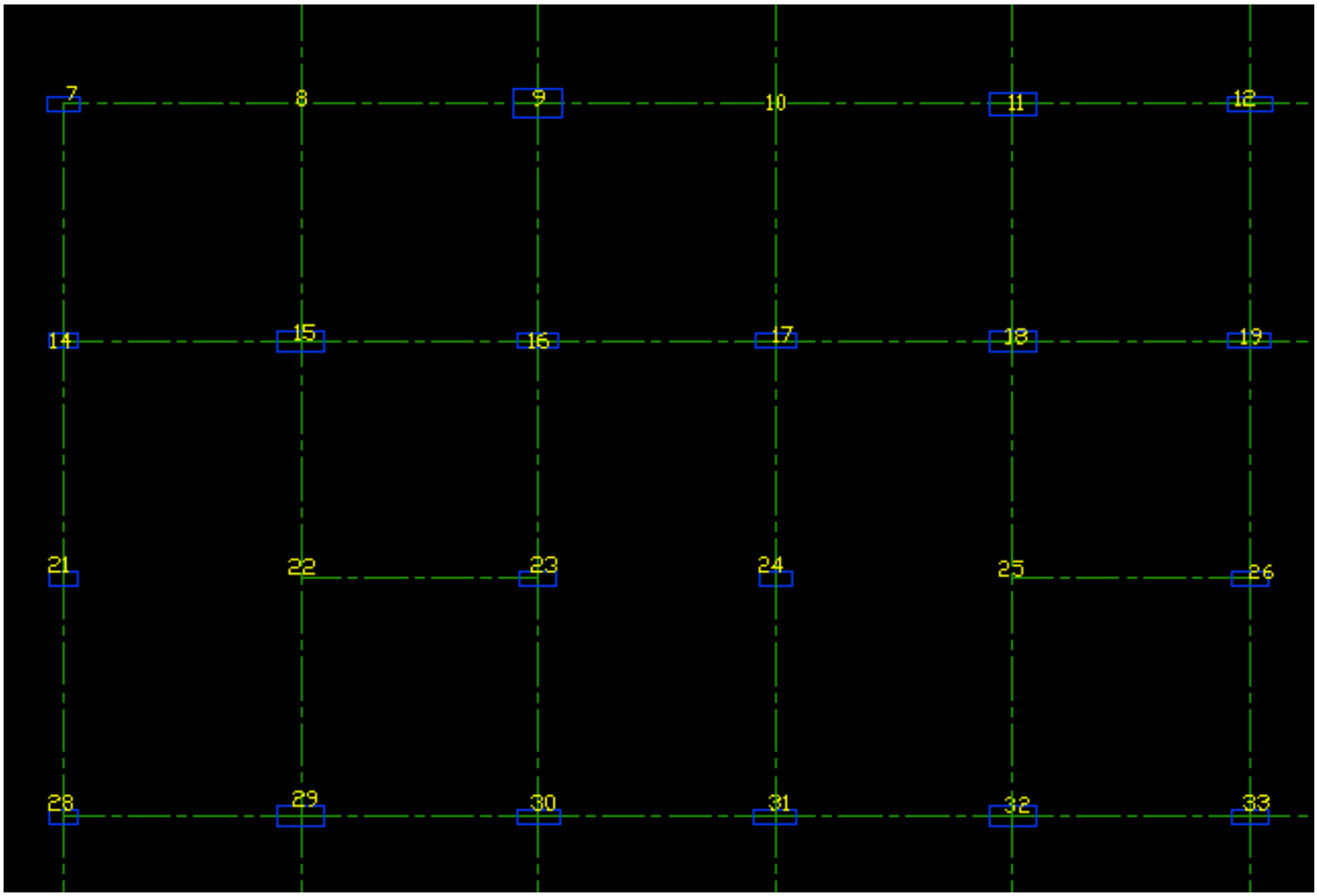
Shown below is a Typical RCC Plan in AutoCAD :



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

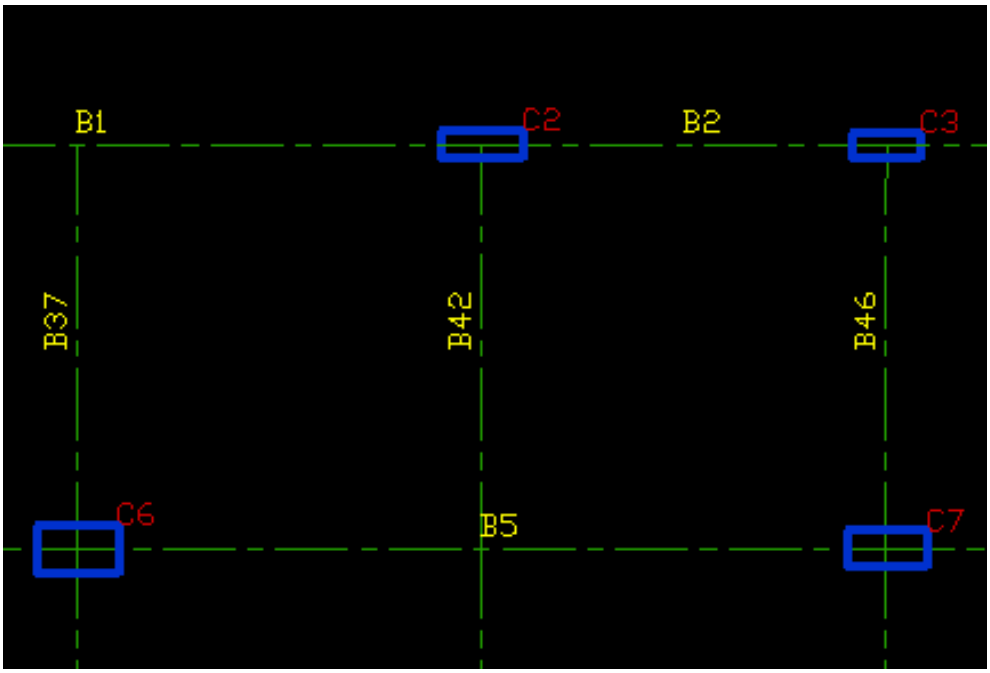


BEAM

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

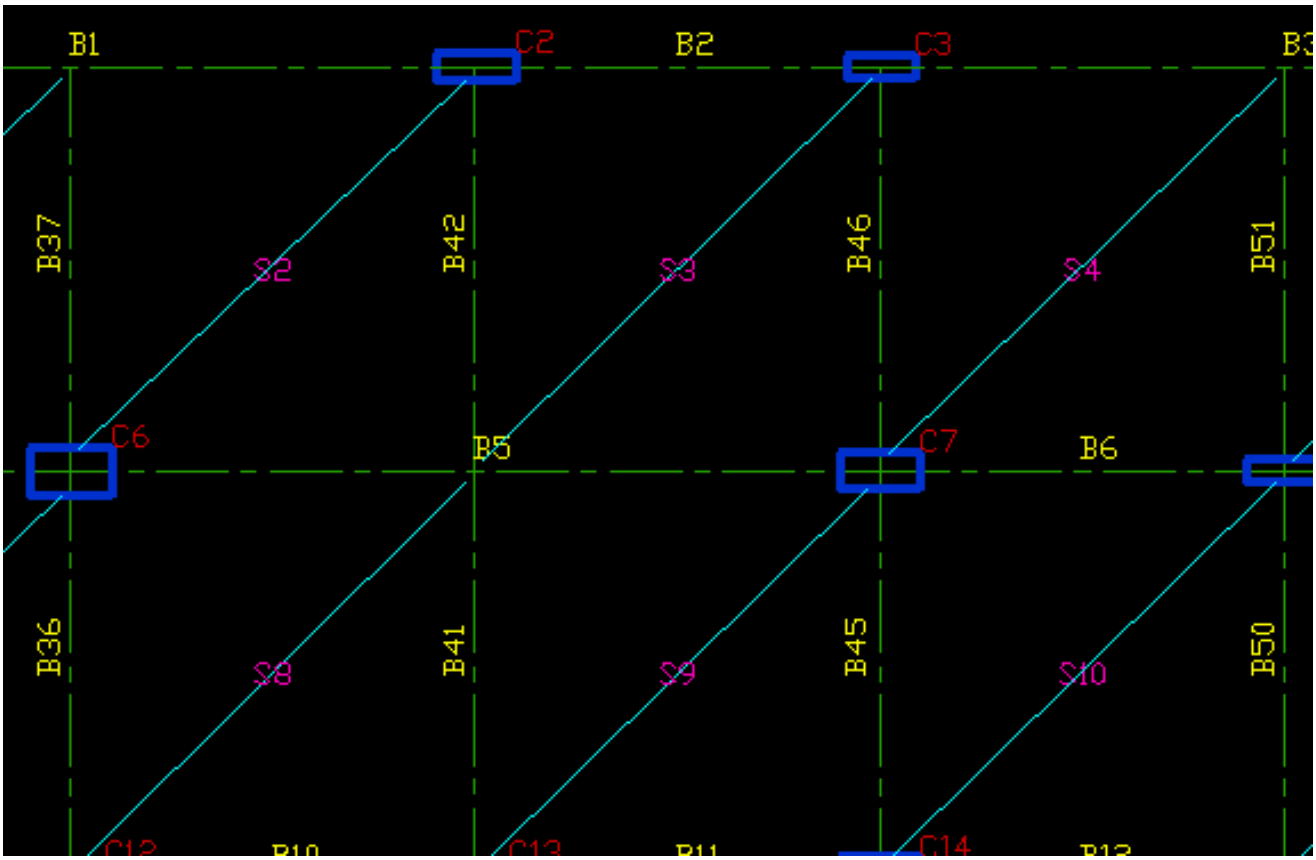
BEAM NUMBERS

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

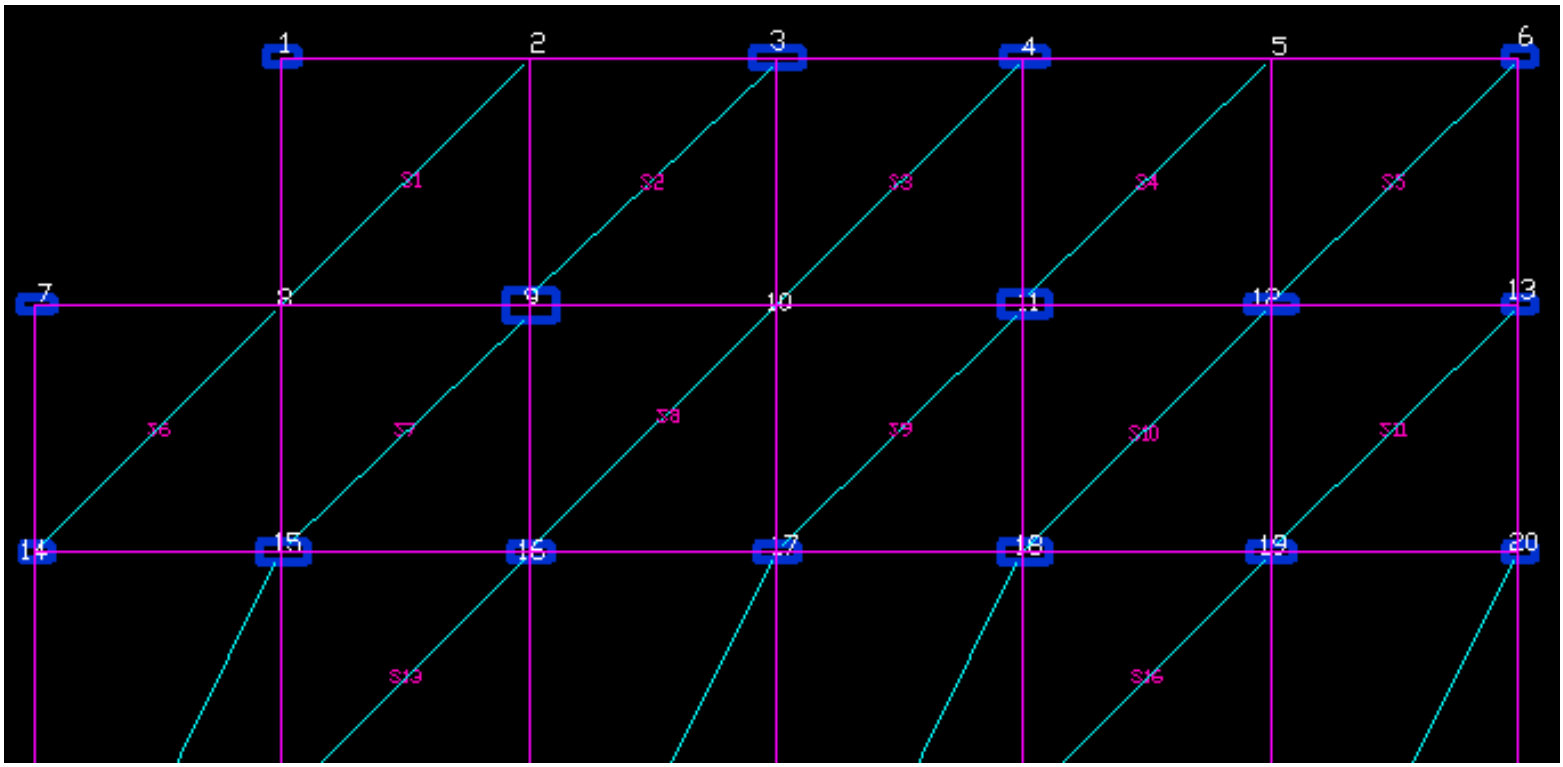
All Slab Numbers should be in the Layer SLABTEXT.
The Slab Text (No.) to be drawn near to the centre of the Slab .
Draw text using 'Single Line Text' option in AutoCAD.
Slab Nos. should not be repeated.
Slabs should be Serially Numbered.
Slab Nos should be prefixed with a "S" (ie. S1, S2)
Slab Text shall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

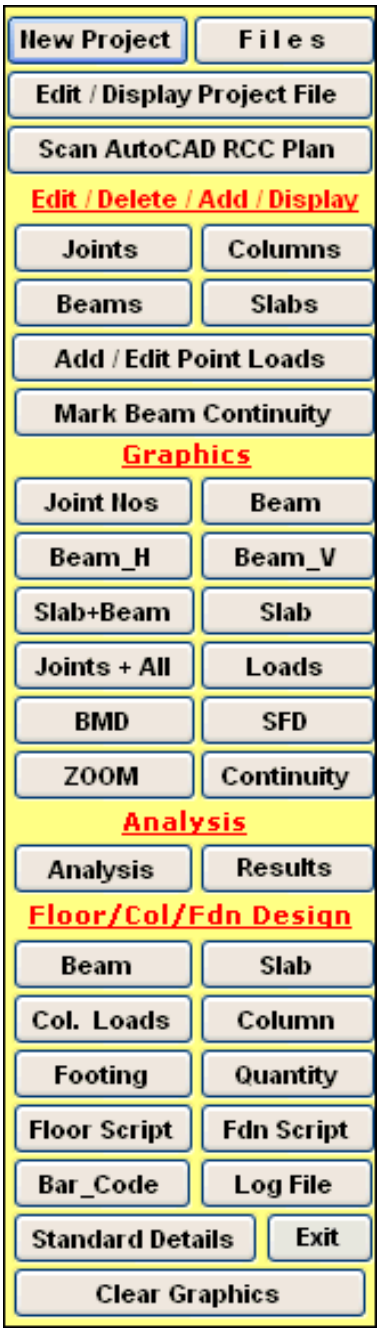
CONTINUITY

All Continuity lines should be drawn in the Layer CNT.
In the Image below, Continuity is marked in magenta.
Beams B1, B2 and B3 are continuous, hence continuiti should be marked from Joint 1 to Joint 6 and not break at any point.
Beams which are not marked as continuous will be treated as simply supported.



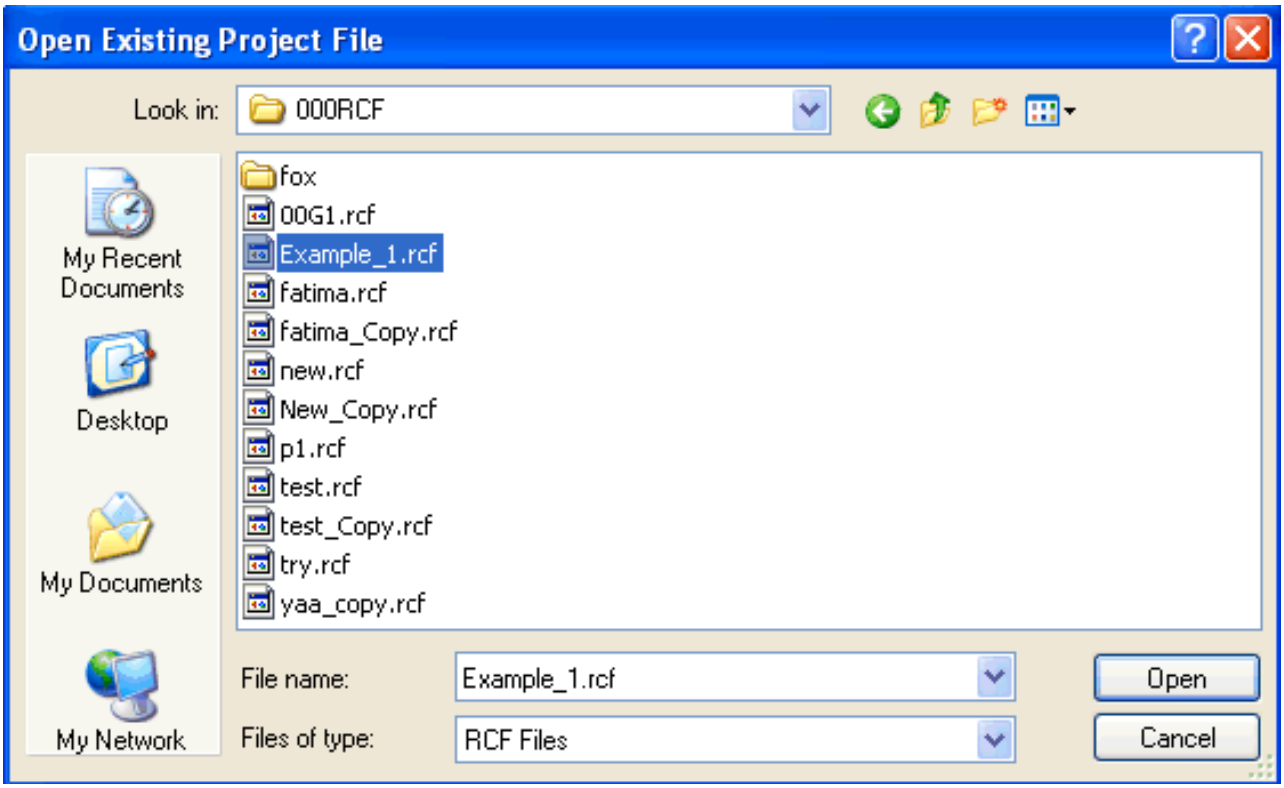
- **Once the drawing is completed, save the drawing in AutoCAD's DXF Format.**

- **Now Start RCF.**

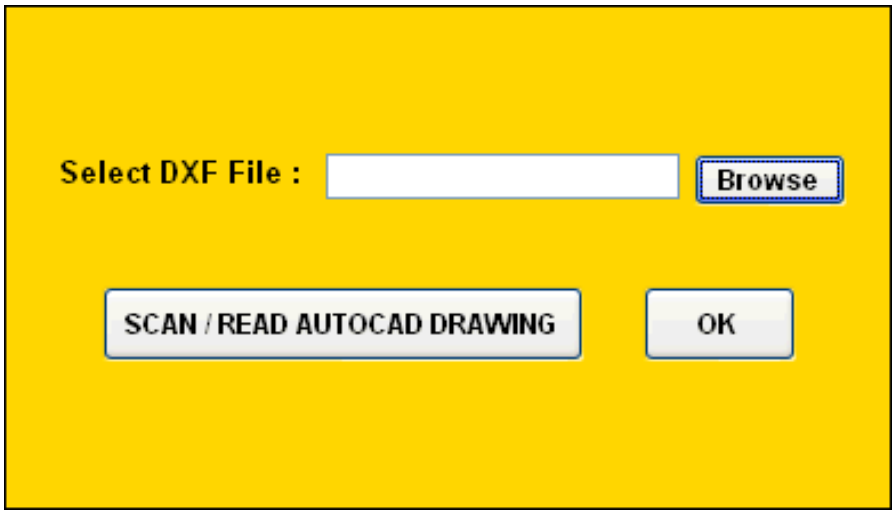


- **When Program starts, the graphics above is displayed.**

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



- **Now select Example _1.rcf 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 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.

Note: **An Architectural Drawing can also be modified and used as an input drawing by making few changes as below :**

1. The Wall Centre line may be used as Beam Centre Line.
Place these lines in CEN Layer.
Draw the Beam Nos in BEAMTEXT layer.

2. Draw the Joints in JOINTS layer.

3. Columns are usually marked in Architectural Plan.
Draw the Column Nos in COLUMNTEXT layer.

4. Draw Slab Diagonal lines in the layer SLAB and Draw the Slab Nos in layer SLABTEXT.

5. Mark Beam Continuity in the layer CNT.

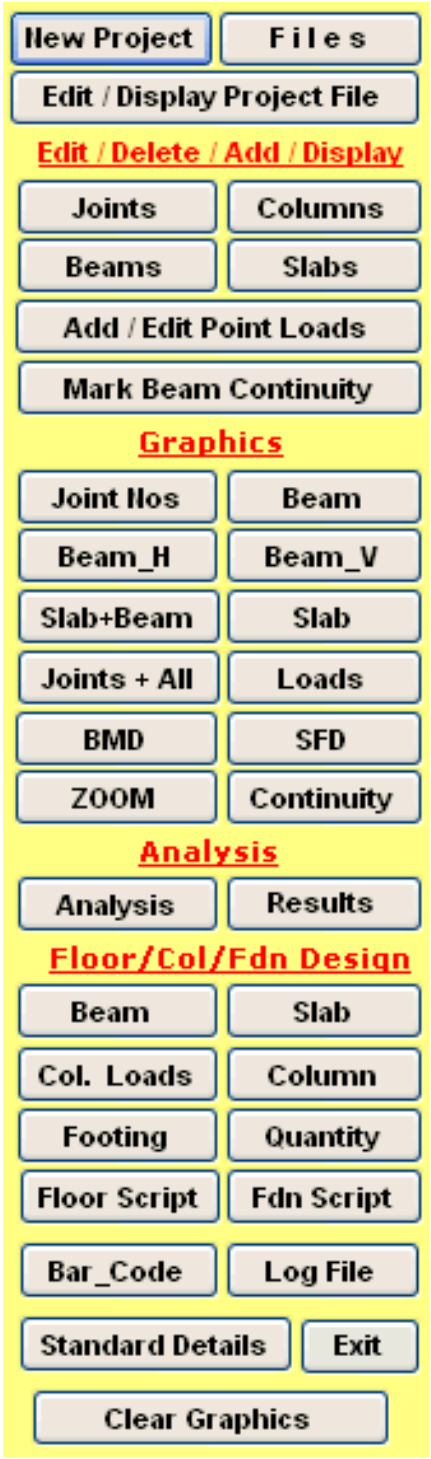
6. Move the Top Left Corner of the Plan to (0,0) Coordinate, by using the 'MOVE' command of AutoCAD.
Save the Drawing in DXF Format.

STEP NO. 2 IS OVER.

go back page top print

LEARN RCF STEP BY STEP

STEP NO. 2 : Automatic Joint Number Creation

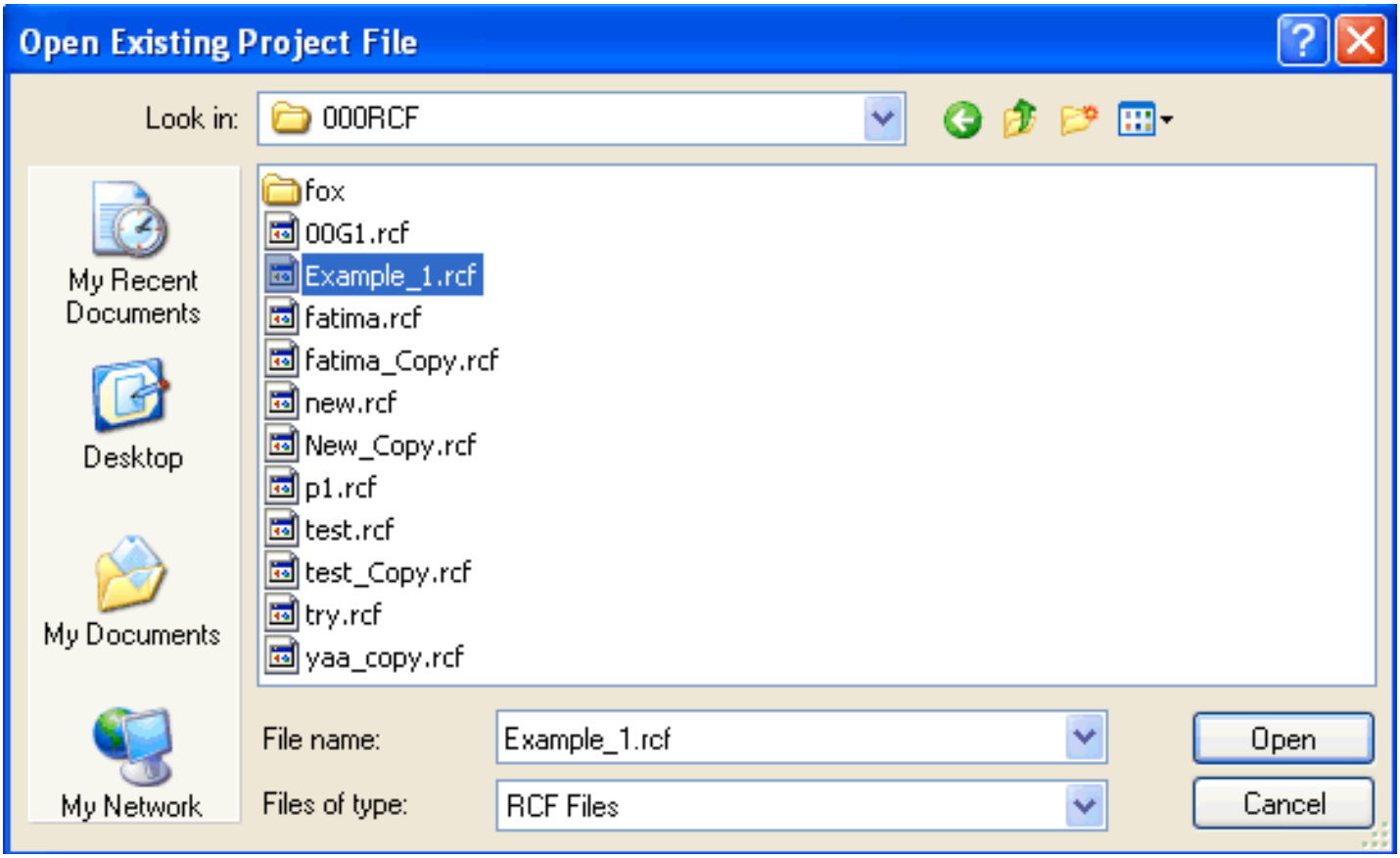


- **When Program starts, the Menu above is displayed. Under the Graphics Heading following options are displayed.**

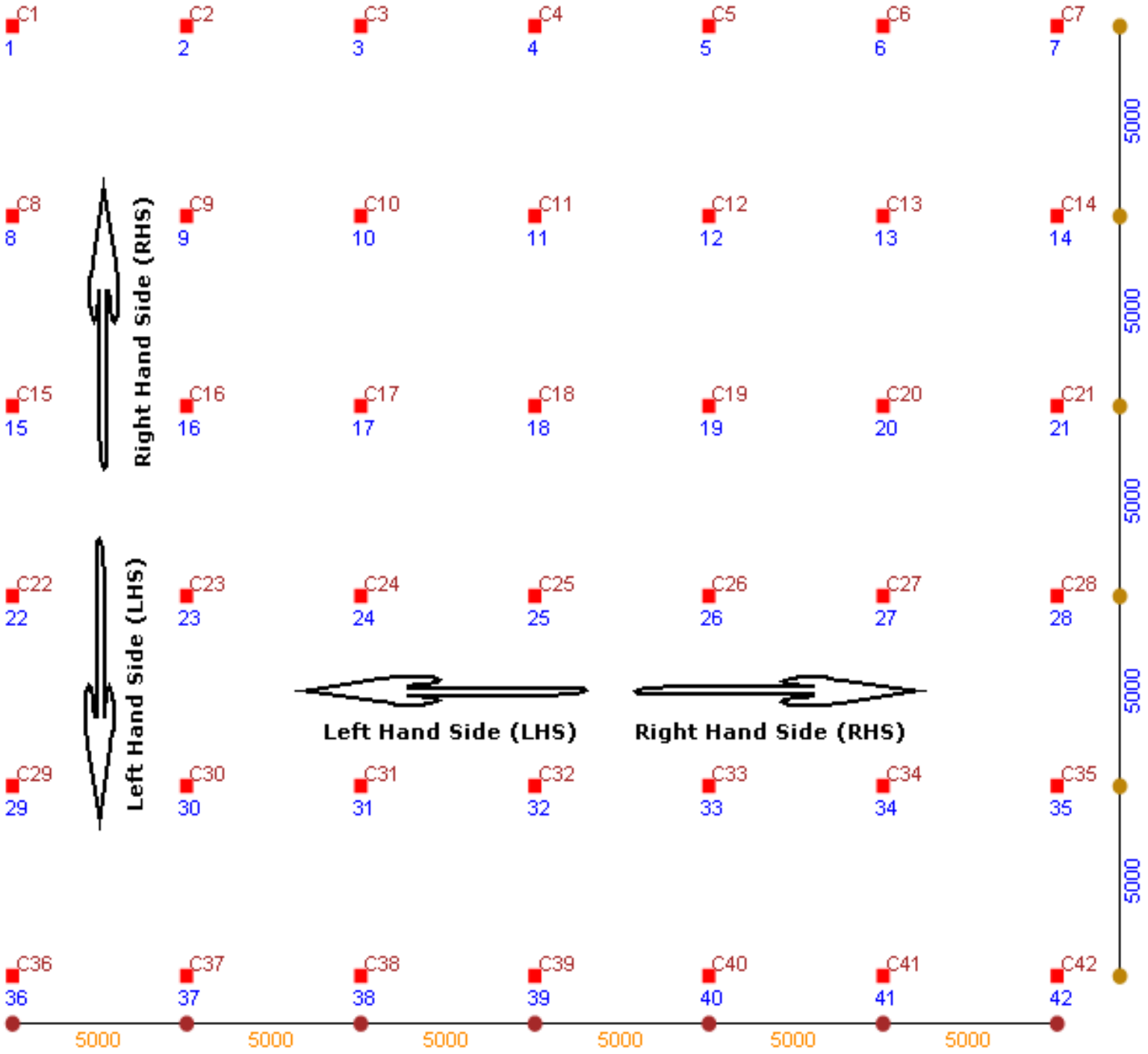
• Joint Nos
• Beam
• Beam_H (For Display of Only Horizontal Beams).
• Beam_V (For Display of Only Vertical Beams).
• Slab + Beam (Beams, Slabs & Columns are displayed).
• Slab (Only Slabs & Columns are displayed).
• Joints + ALL (For Display of Joints, Columns, Beams & Slabs)
• Loads (Display of Slab, 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 Nos " option.

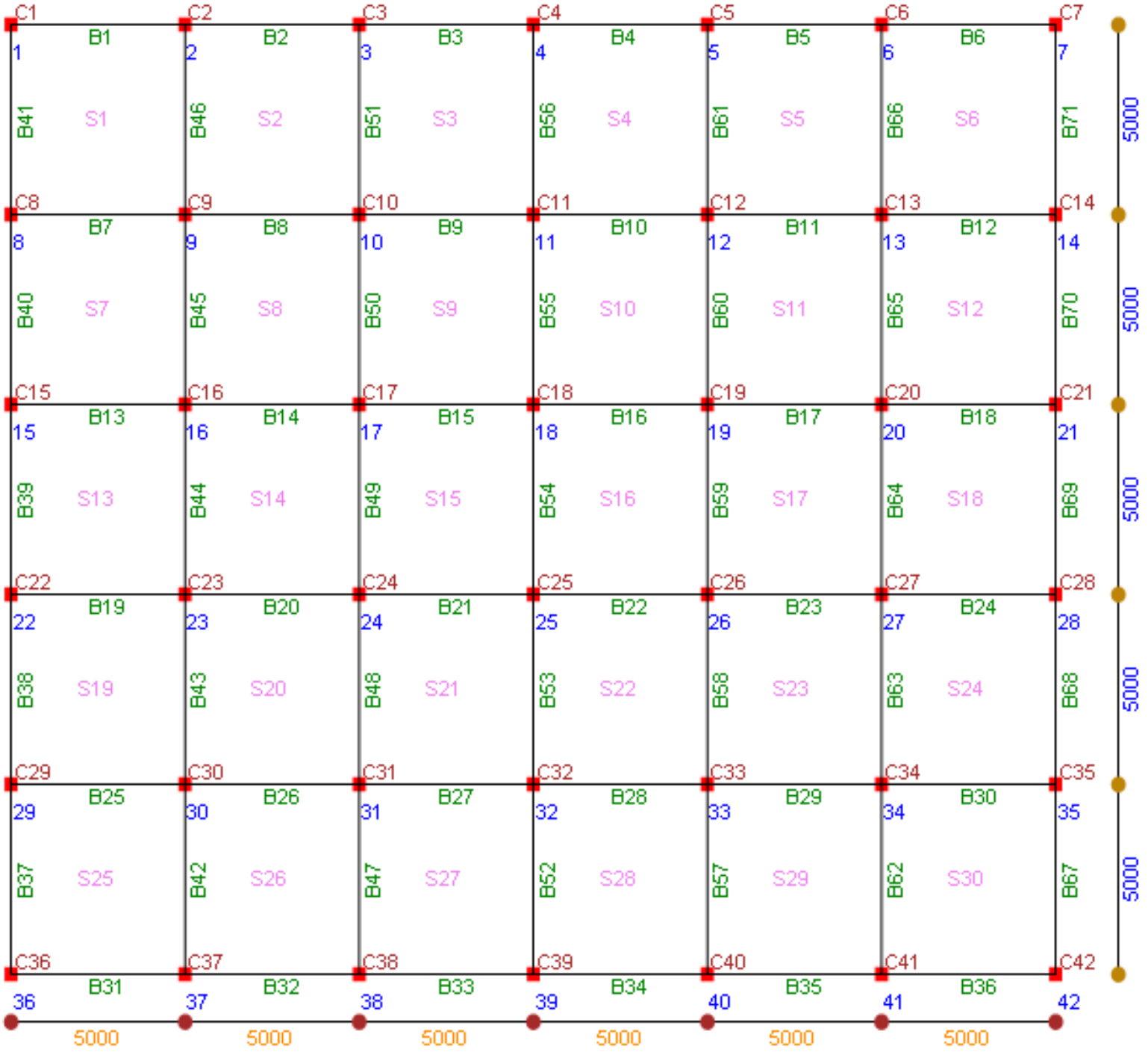
Following Graphics is displayed.



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

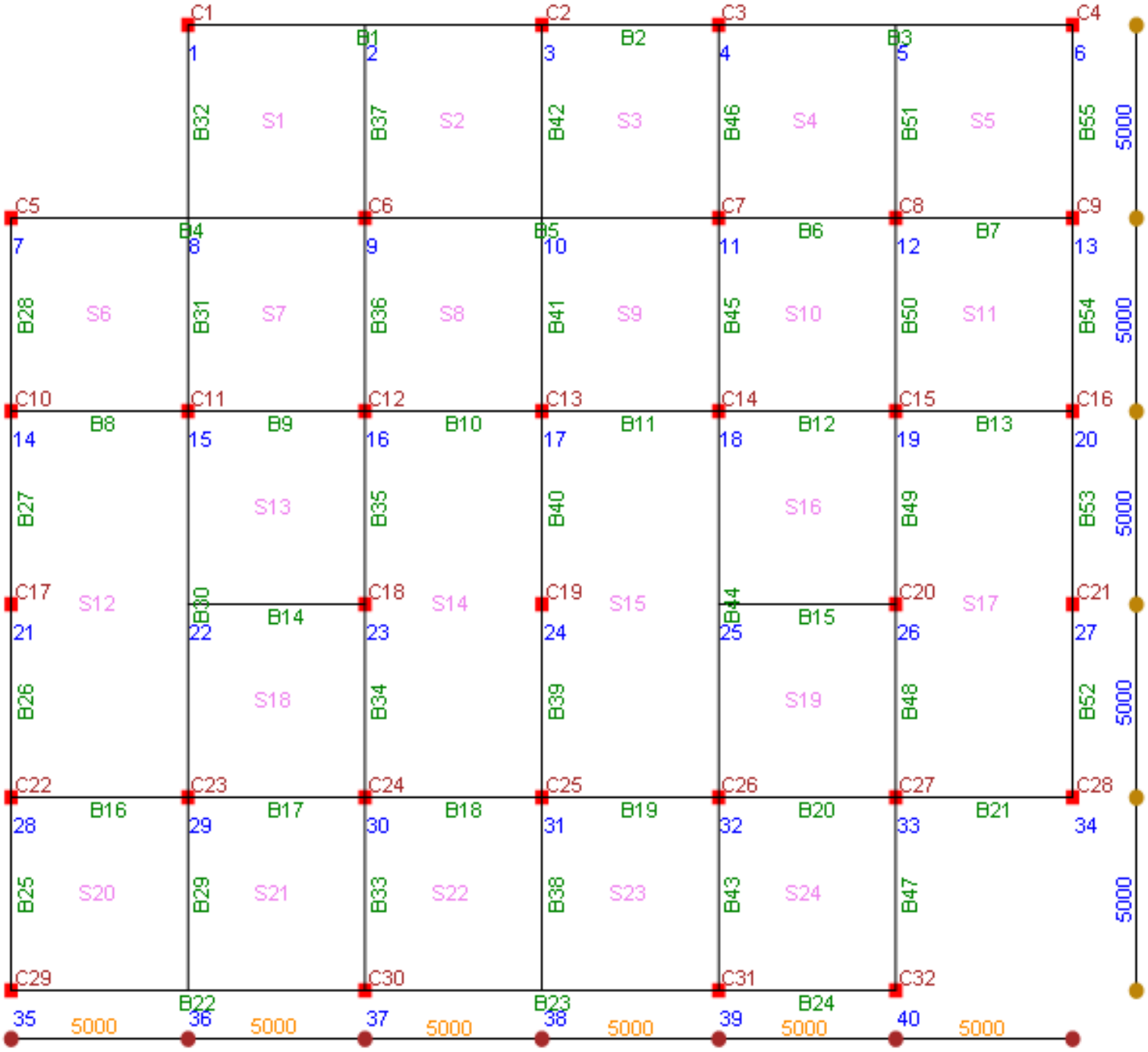


- **Note that Joints Numbers (Including X & Y Co-Ordinates) and Columns are created and displayed automatically at all the intersections of vertical & horizontal grids. Some of the Joint numbers may not be required. A Joint represents a column location or an intersection between 2 beams. The beams are 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 the required Floor Plan. The intended actual floor plan is reproduced below.



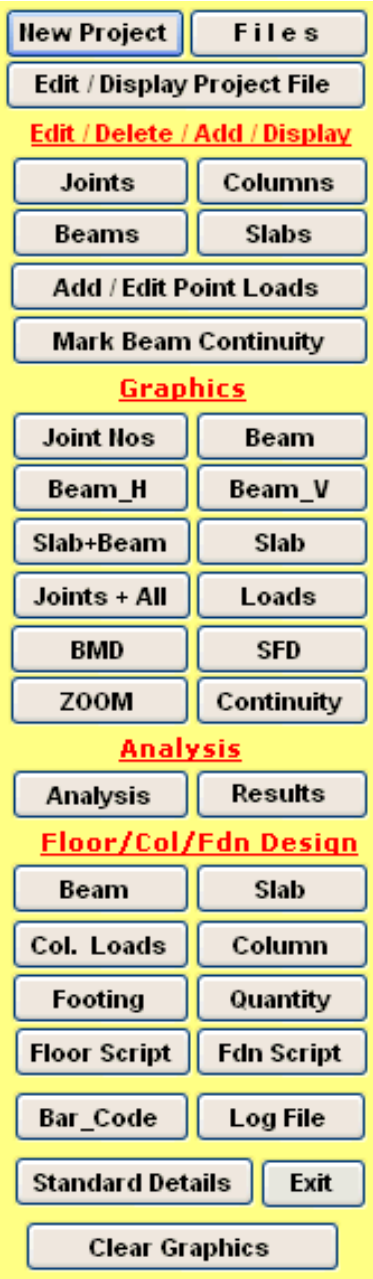
ACTUAL REQUIRED FLOOR PLAN

- **Our Actual RCC Floor Plan has only 24 numbers of Slabs, 32 numbers of Columns and 55 numbers of Beams. The automatic generated plan has 30 numbers of Slabs, 42 numbers of Columns and 71 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 RCF STEP BY STEP

STEP NO. 3 : Delete Un-Wanted Joints

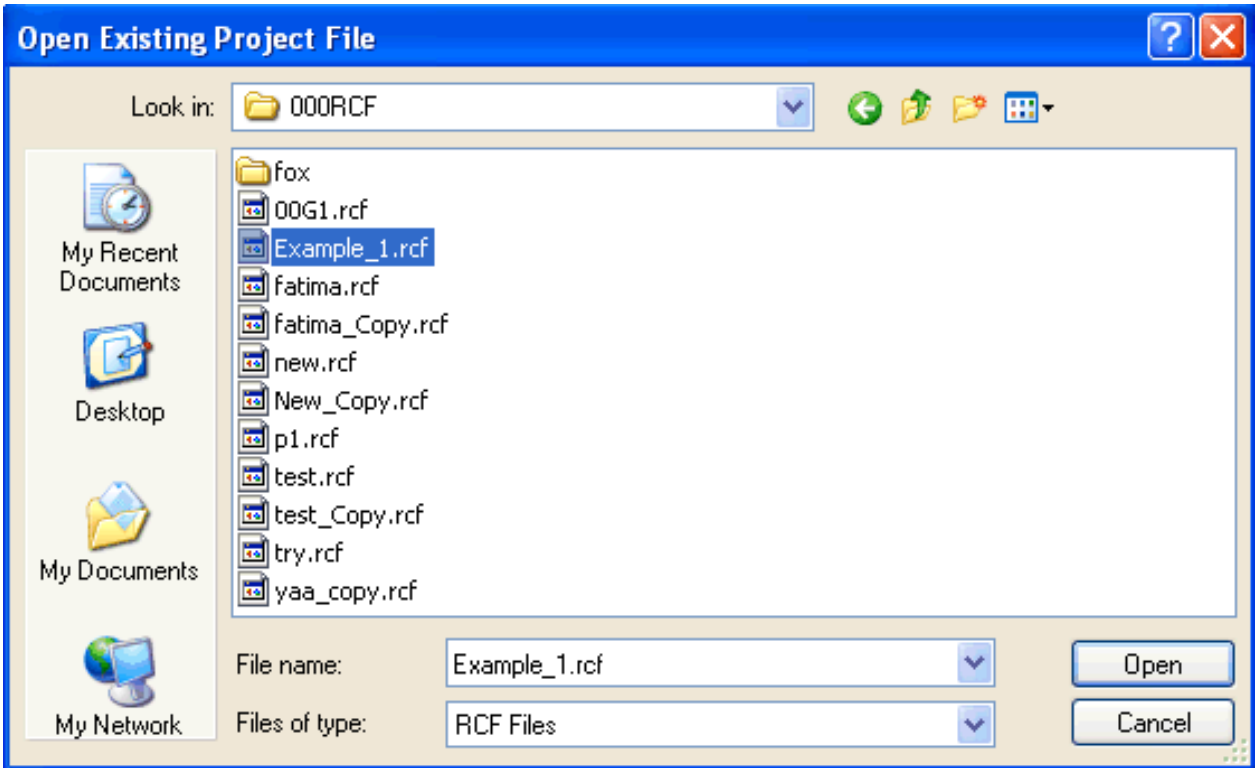


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  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

File Name : C:\000RCF\Example_1.rc Date : 09 May 2008

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

Joint No. 1

X Co_Ordinate 0

Y Co_Ordinate 0

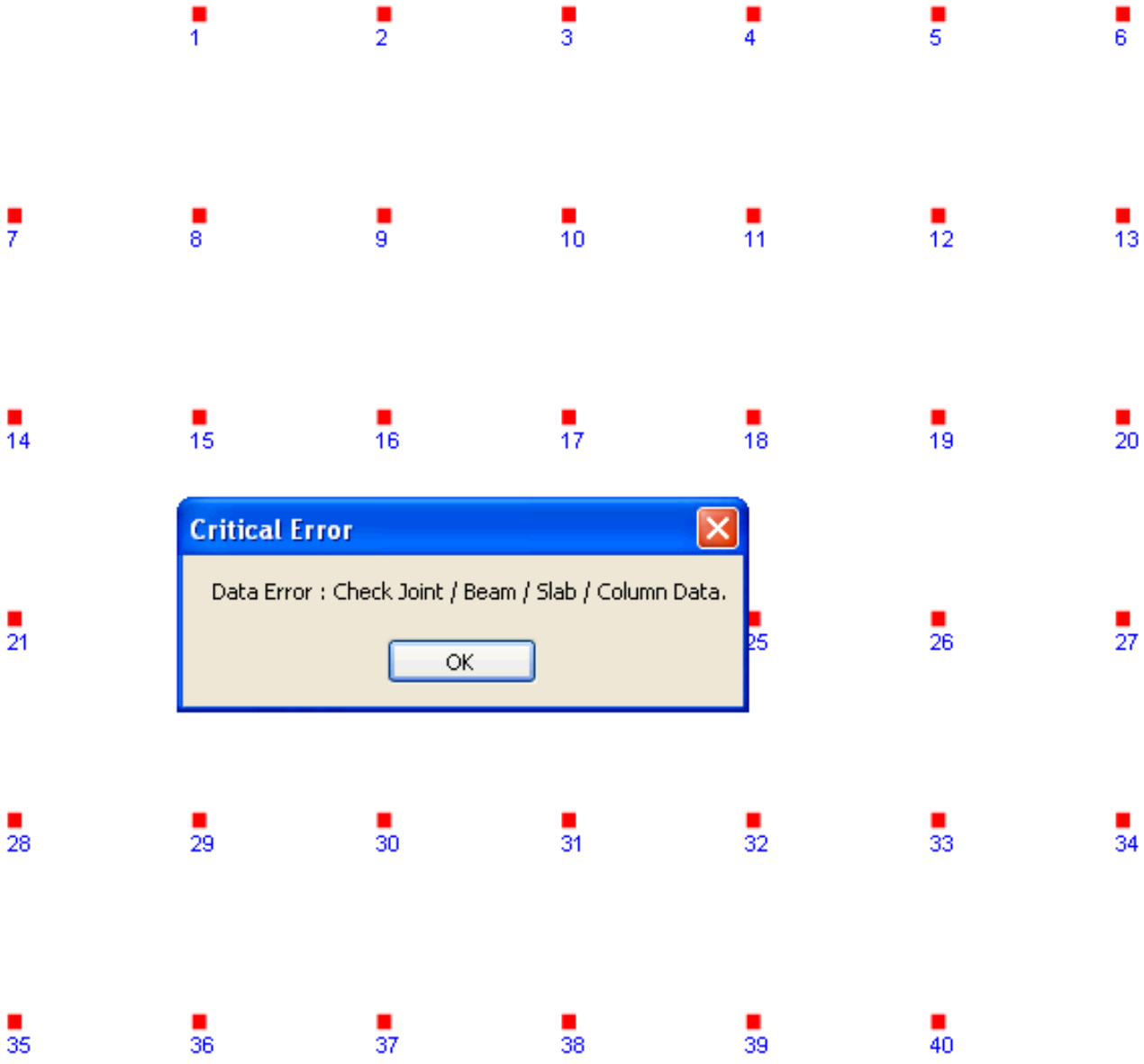
Record No. : 1 of 42

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
1	0	0
2	5000	0
3	10000	0
4	15000	0
5	20000	0
6	25000	0
7	30000	0
8	0	5000
9	5000	5000
10	10000	5000
11	15000	5000
12	20000	5000
13	25000	5000
14	30000	5000
15	0	10000
16	5000	10000
17	10000	10000
18	15000	10000
19	20000	10000
20	25000	10000
21	30000	10000
22	0	15000
23	5000	15000
24	10000	15000
25	15000	15000
26	20000	15000
27	25000	15000
28	30000	15000
29	0	20000
30	5000	20000
31	10000	20000

28	30000	15000
29	0	20000
30	5000	20000
31	10000	20000
32	15000	20000
33	20000	20000

- **We have to delete joint numbers " 1 " and " 42 ". Just Select Joint Number " 1" Row & press " Remove " button. Joint Number " 1 " is deleted. Similarly select Joint Number " 42 " & press remove button. Joint no. " 42 " is deleted. Click Update button, you will notice that all Joints are re-numbered. By repeatedly Deleting & Updating, even a complex floor plan can numbered appropriately. To achieve this a copy of actual & automatic generated plan should be in front of you.**
  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.**

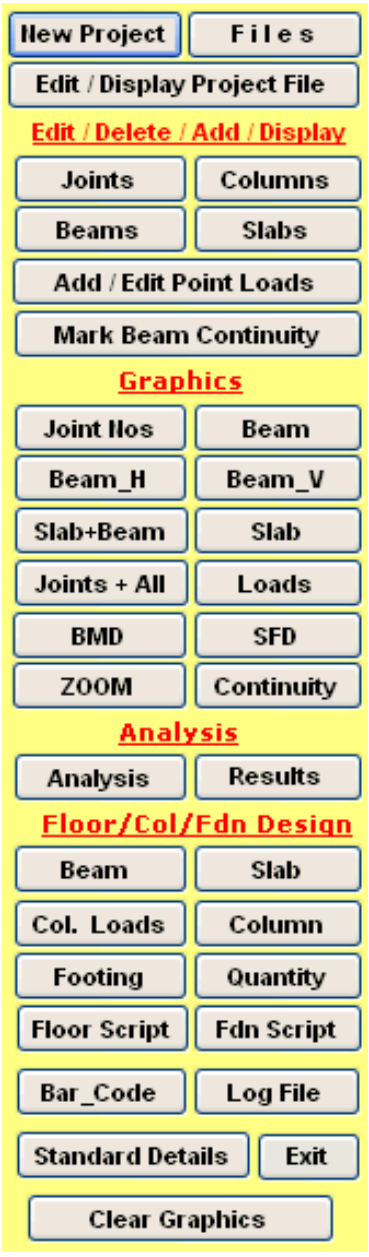
  1. Origin (0,0) is at Top Left Hand Corner. Co-Ordinates Cannot be Negative.
  2. There shall not be any difference in Maximum Horizontal & Vertical Distance between Project File & Joint File.
  3. Joint Number should start with 1 & not 0.
  4. Joints Numbers cannot be repeated.
  5. Co-ordinates cannot 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 RCF STEP BY STEP

STEP NO. 4 : Delete & Edit Beams

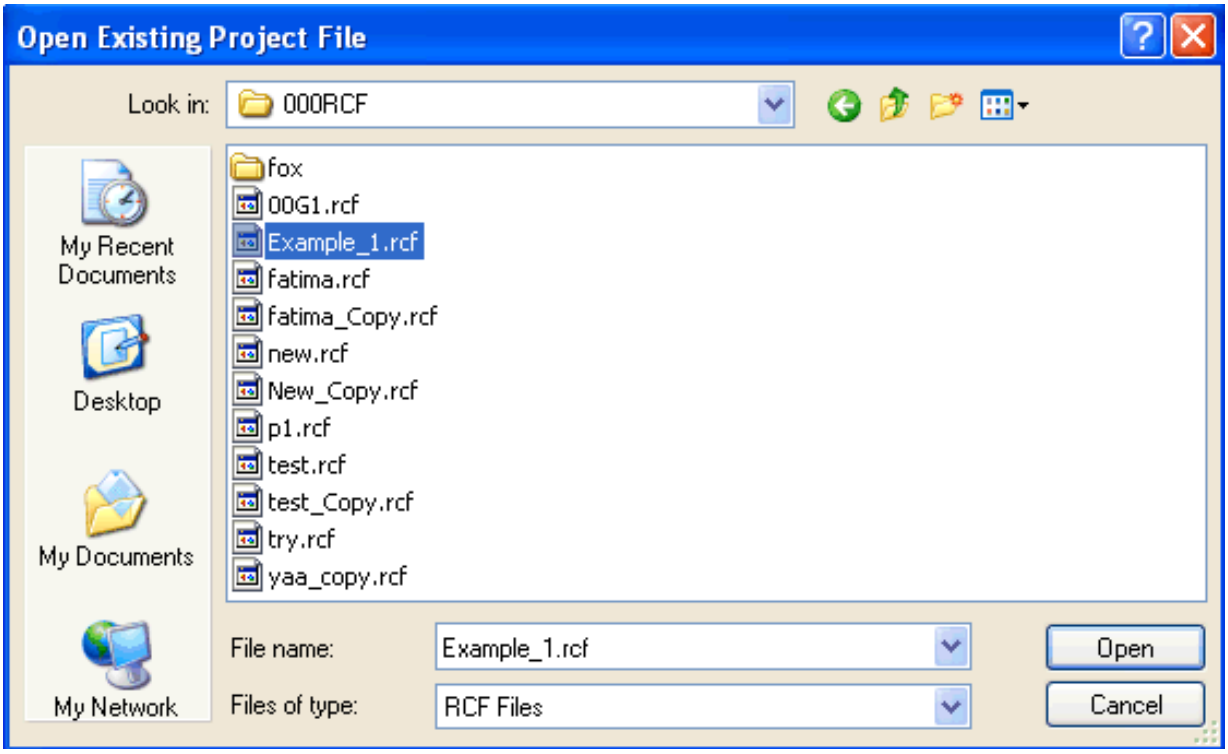


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  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

File Name : C:\000RCF\Example_1.rcf Date : 10 May 2008

Beam #	LHS Joint #	RHS Joint #	Width	Depth	Masonry Ht	Masonry Thk	RHS BM	LHS BM	Extra UDL
B52	39	32	230	450	2.55	230			
B53	32	25	230	450	2.55	230			
B54	25	18	230	450	2.55	230			
B55	18	11	230	450	2.55	230			
B56	11	4	230	450	2.55	230			
B57	40	33	230	450	2.55	230			
B58	33	26	230	450	2.55	230			
B59	26	19	230	450	2.55	230			
B60	19	12	230	450	2.55	230			
B61	12	5	230	450	2.55	230			
B62	41	34	230	450	2.55	230			
B63	34	27	230	450	2.55	230			
B64	27	20	230	450	2.55	230			
B65	20	13	230	450	2.55	230			
B66	13	6	230	450	2.55	230			
B67	42	35	230	450	2.55	230			
B68	35	28	230	450	2.55	230			
B69	28	21	230	450	2.55	230			
B70	21	14	230	450	2.55	230			
B71	14	7	230	450	2.55	230			

Record No. : 71 of 71

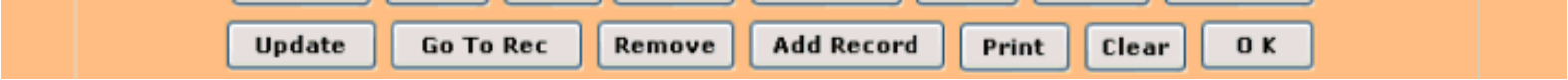
Beam # B71 | LHS Joint # 14 | RHS Joint # 7 | Beam Width in MM 230 | Beam Depth in MM 450

Net Height of Masonry Wall in M 2.55 | Thickness of Wall in MM 230 | Additional UDL on Beam in T/M

RHS End BM due to Wind / Siesmic in T-M | LHS End BM due to Wind / Siesmic in T-M | Span 5000

Read Me | Prev | Next | Copy | Paste | 1 st | Last | Copy All

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



- **Here we have 71 numbers of Beams. Actual required are only 55 numbers of Beams (Refer Step No. 1 - Actual Required Floor Plan). Go down to the last beam number B71 and press " Remove " button. You will notice that Beam B71 is deleted. Similarly delete the next beam, till you reach Beam number B55. 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 B1 & Select it (Click with Cursor), or click the " 1 st " button.
  Now concentrate on the Text Boxes below. Beam # will be shown as B1. LHS Joint # is shown as 1 and RHS joint # is shown as 2. Change RHS Joint # to 3 by editing the text box.
  Again select Beam # B2 or Click " Next " button. RHS Joint # is shown as 2, change it to 3. LHS Joint # is shown as 3, change to 4.
  Similarly edit the rest of Beam's RHS & LHS 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; Width, Depth, Masonry Height, Masonry Thickness, RHS BM, LHS BM and Any Extra UDL can be Added / Edited for individual Beams by clicking at respective Text Boxes.**
  Beam Span is displayed in Yellow Text Box.
- **Now click the " Read Me " button, the following important messages are displayed for guidance.**

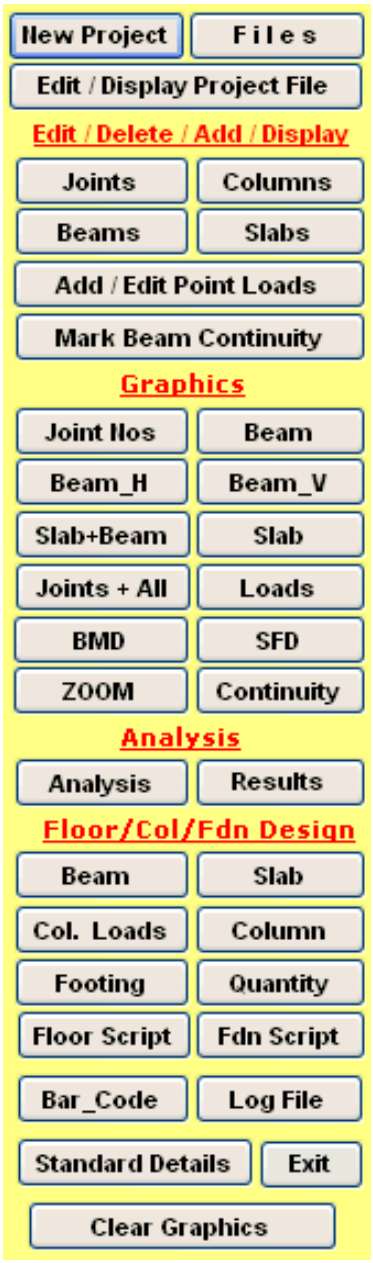
  1. Add Joint Details before Beams.
  2. Beam Number should start with 1 & not 0.
  3. Beam Numbers cannot be repeated.
  4. Beam LHS & RHS Joint #s cannot be repeated.
  5. Beam Width / Depth < 150 mm not allowed.
  6. Beam Width / Depth > 3500 mm not allowed.
  7. Max. (LHS or RHS) Beam Joint # cannot > Max. Joint File #.
  8. Use Add Button to Append Record.
  9. Use Update Button to Re-Number & Save Your Work.
  10. Max. Beam Number = Max. Record Number.
  11. Beam Nos. Shall be Numbered Serially.
  12. Beam LHS OR RHS Joint Number Cannot < = 0.0
  13. LHS : Left Hand Side, RHS : Right Hand Side.
  14. If Beam is Vertical then, LHS Y-Co Ordinate > RHS Y-Co Ordinate.
  15. If Beam is Horizontal then, LHS X-Co Ordinate < RHS X-Co Ordinate.

- **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 RCF STEP BY STEP

STEP NO. 5 : Delete & Edit Columns

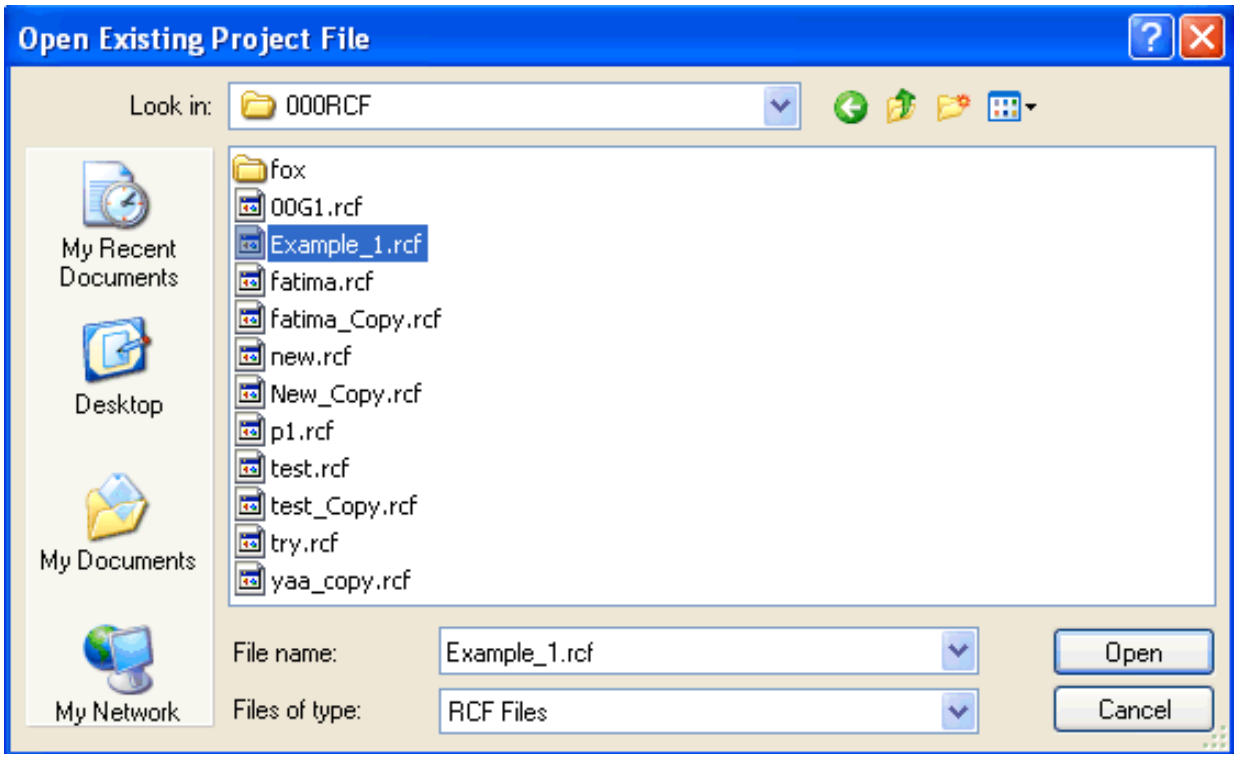


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  Now Click on " Columns " option.

  Following Graphics is displayed.



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

DISPLAY / EDIT / COLUMN DETAILS

File Name : C:\000RCF\Example_1.rcf **Date : 03 January 2009**

Column #	Joint #	BM X-X	BM Y-Y	X-X Dim	Y-Y Dim	Leff X-X	Leff Y-Y	% Steel	Steel Face
C1	1			600	300	3	3	0.8	Y
C2	3			600	300	3	3	0.8	Y
C3	4			600	300	3	3	0.8	Y
C4	6			600	300	3	3	0.8	Y
C5	7			600	300	3	3	0.8	Y
C6	9			600	300	3	3	0.8	Y
C7	11			600	300	3	3	0.8	Y
C8	12			600	300	3	3	0.8	Y
C9	13			600	300	3	3	0.8	Y
C10	14			600	300	3	3	0.8	Y
C11	15			600	300	3	3	0.8	Y
C12	16			600	300	3	3	0.8	Y
C13	17			600	300	3	3	0.8	Y
C14	18			600	300	3	3	0.8	Y
C15	19			600	300	3	3	0.8	Y
C16	20			600	300	3	3	0.8	Y
C17	21			600	300	3	3	0.8	Y
C18	23			600	300	3	3	0.8	Y
C19	24			600	300	3	3	0.8	Y
C20	26			600	300	3	3	0.8	Y
C21	27			600	300	3	3	0.8	Y
C22	28			600	300	3	3	0.8	Y
C23	29			600	300	3	3	0.8	Y

Record No. : 1 of 32

Column # C1 Joint # 1 BM along X_X in t-m BM along Y_Y in t-m

Col. dim. along X_X in MM 600 Col. dim. along Y_Y in MM 300 Assumed % of Reinforcement 0.8

Eff. length along X_X in M 3 Eff. length along Y_Y in M 3 Is Steel Distributed on 2 Faces Y

Read Me | Prev | Next | Last | 1 st | Copy | Paste | Copy All

- **Here we have 42 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 C42 and press " Remove " button. You will notice that Column C42 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 # C2 or Click " Next " button. Joint # is shown as 2, change it to 3.
  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 Column Parameters Viz; X-X Dim, Y-Y Dim, Lxx, Lyy, BM_XX, BM_YY, % of Reinforcement, Steel Face Distribution (2 or 4 Faces : Y / N) & Orientation can be Added / Edited for individual Columns by clicking at respective Text Boxes.**

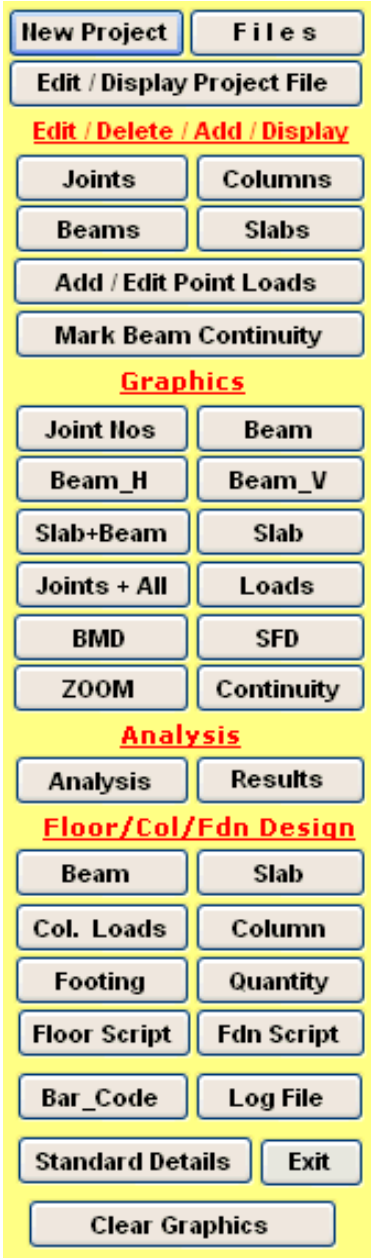
  - **Click Limitations Button. A new window will open up displaying Permissible Column Size & Maximum Reinforcement % Table. The program designs building columns automatically, selecting size and % of reinforcement, within this displayed table.**
- **Now click the " Read Me " button, the following important 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. Column Width / Depth < 200 mm not allowed.
  5. Max. Column Joint # cannot > Max. Joint File #.
  6. Use Add Button to Append Record."
  7. Max. Column Number = Max. Record Number.
  8. Columns Shall be Numbered Serially.
  9. Use Update Button to Re-Number & Save Your Work.
  10. Column Joint Number cannot be < = 0.0.
  11. Column Dimension along X-X means along X Axis (Horizontal).
  12. Column Dimension along Y-Y means along Y Axis (Vertical).
- **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 RCF STEP BY STEP

STEP NO. 6 : Delete & Edit Slabs

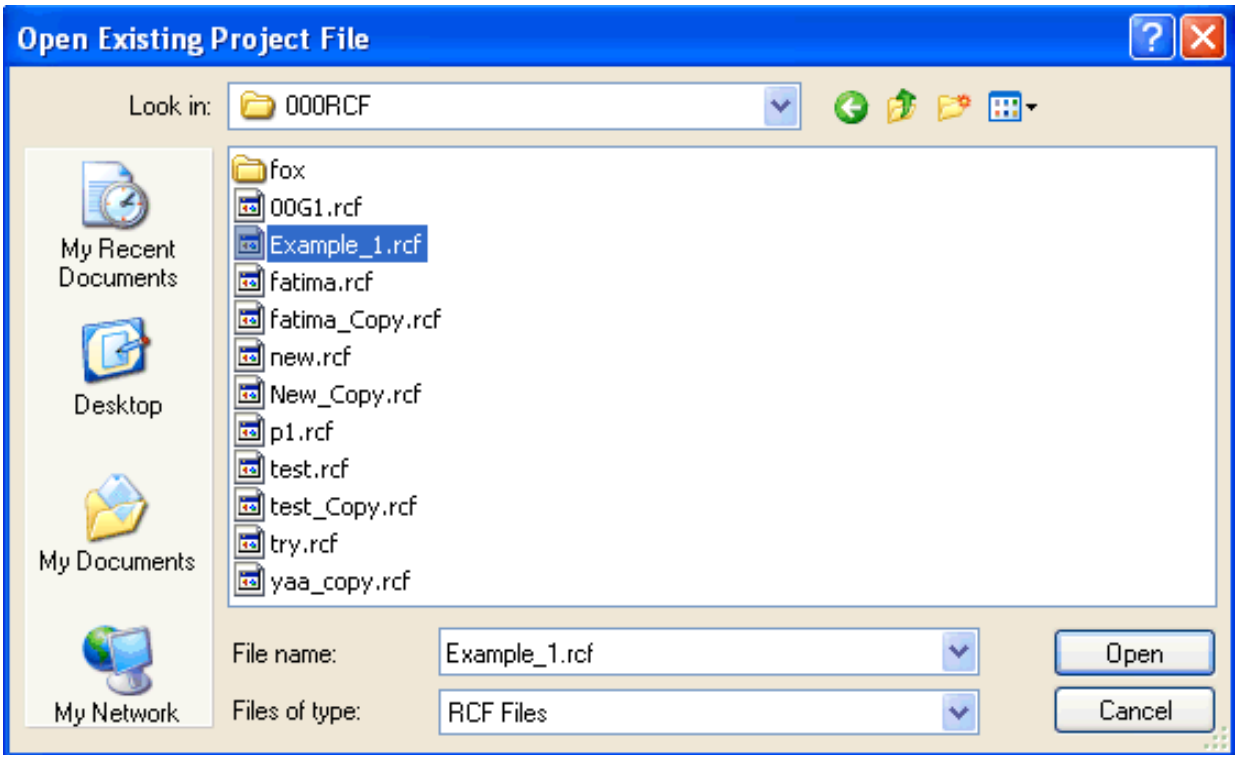


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  Now Click on " Slabs " option.

  Following Graphics is displayed.



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

DISPLAY / EDIT / SLAB DETAILS

File Name : C:\000RCF\Example_1.rcf **Date : 12 May 2008**

Slab #	Left Btm Joint #	Right Top Joint #	Thickness	Live Load	Floor Finish	Ceiling	Partition Load
S1	8	2	150	0.50	40	20	0.10
S2	9	3	150	0.50	40	20	0.10
S3	10	4	150	0.50	40	20	0.10
S4	11	5	150	0.50	40	20	0.10
S5	12	6	150	0.50	40	20	0.10
S6	13	7	150	0.50	40	20	0.10
S7	15	9	150	0.50	40	20	0.10
S8	16	10	150	0.50	40	20	0.10
S9	17	11	150	0.50	40	20	0.10
S10	18	12	150	0.50	40	20	0.10
S11	19	13	150	0.50	40	20	0.10
S12	20	14	150	0.50	40	20	0.10
S13	22	16	150	0.50	40	20	0.10
S14	23	17	150	0.50	40	20	0.10
S15	24	18	150	0.50	40	20	0.10
S16	25	19	150	0.50	40	20	0.10
S17	26	20	150	0.50	40	20	0.10
S18	27	21	150	0.50	40	20	0.10
S19	29	23	150	0.50	40	20	0.10
S20	30	24	150	0.50	40	20	0.10
S21	31	25	150	0.50	40	20	0.10
S22	32	26	150	0.50	40	20	0.10
S23	33	27	150	0.50	40	20	0.10

Record No. : 1 of 30

Slab # S1 | Left Bottom Joint # 8 | Right Top Joint # 2 | Slab Thickness in MM 150 | 5000 x 5000

LL on Slab in T/M2 0.50 | Floor Finish in MM 40 | Ceiling Thk. in MM 20 | Partition Load in T/M2 0.10

Read Me | Prev | Next | Copy | Paste | Last | 1 st | Slab Intensity in T/M2 1.09

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

- **Here we have 30 numbers of Slabs. Actual required are only 24 numbers of Slabs (Refer Step No. 1 - Actual Required Floor Plan). Go down to the last Slab number S30 & press " Remove " button. You will notice that Slab S30 is deleted. Similarly delete the next Slab, till you reach Slab number S24. 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 S1 & Select it (Click with Cursor), or click the " 1 st " button.
  Now concentrate on the Text Boxes below. Slab # will be shown as S1. Left Bottom Joint # is shown as 8 & Right Top joint # is shown as 2. This is what we require, hence there is no change. Similarly no changes are required for slabs S2 to S5.
  Slab S6 requires correction. Change Left Bottom Joint # to 13 and Right Top Joint # to 7 by editing the text box. slab S12 are requires correction to 20,14 from 21, 15.
  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; Thickness, LL, FF, CL and Partition Loads can be Added / Edited for individual Slabs by clicking at respective Text Boxes.**
  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.**

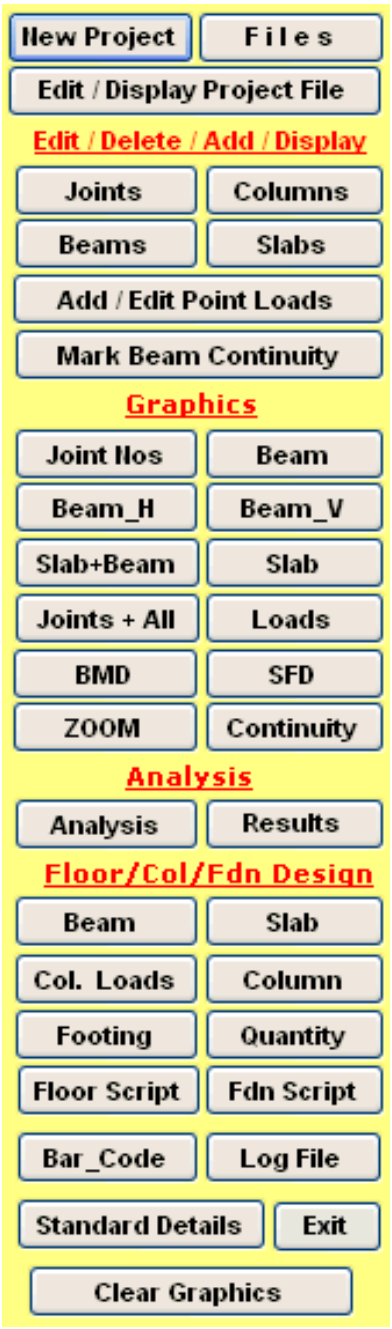
  1. Add Joint & Beam Details Before Slab.
  2. Slab Numbers cannot be < = 0.0 & repeated.
  3. Slab LHS & RHS Joint #s cannot be repeated.
  4. Slab Thickness > 600 mm not allowed.
  5. Max. Slab Joint # cannot > Max. Joint File #.
  6. Use Add Button to Append Record.
  7. Use Update Button to Re-Number & Save Your Work.
  8. Max. Slab Number = Max. Record Number.
  9. Slabs shall be Numbered Serially.
  10. Slabs Joint (LB/RT) Numbers cannot be < = 0.0
  11. Slab Density = 2.5 T/M3, Plaster/FF Density = 2.0 T/M3.
  12. Slab Thickness = 0.0 Means Cut-Out / Opening. Hence shall not contain LL, FF, CL & PL.

- **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 RCF STEP BY STEP

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

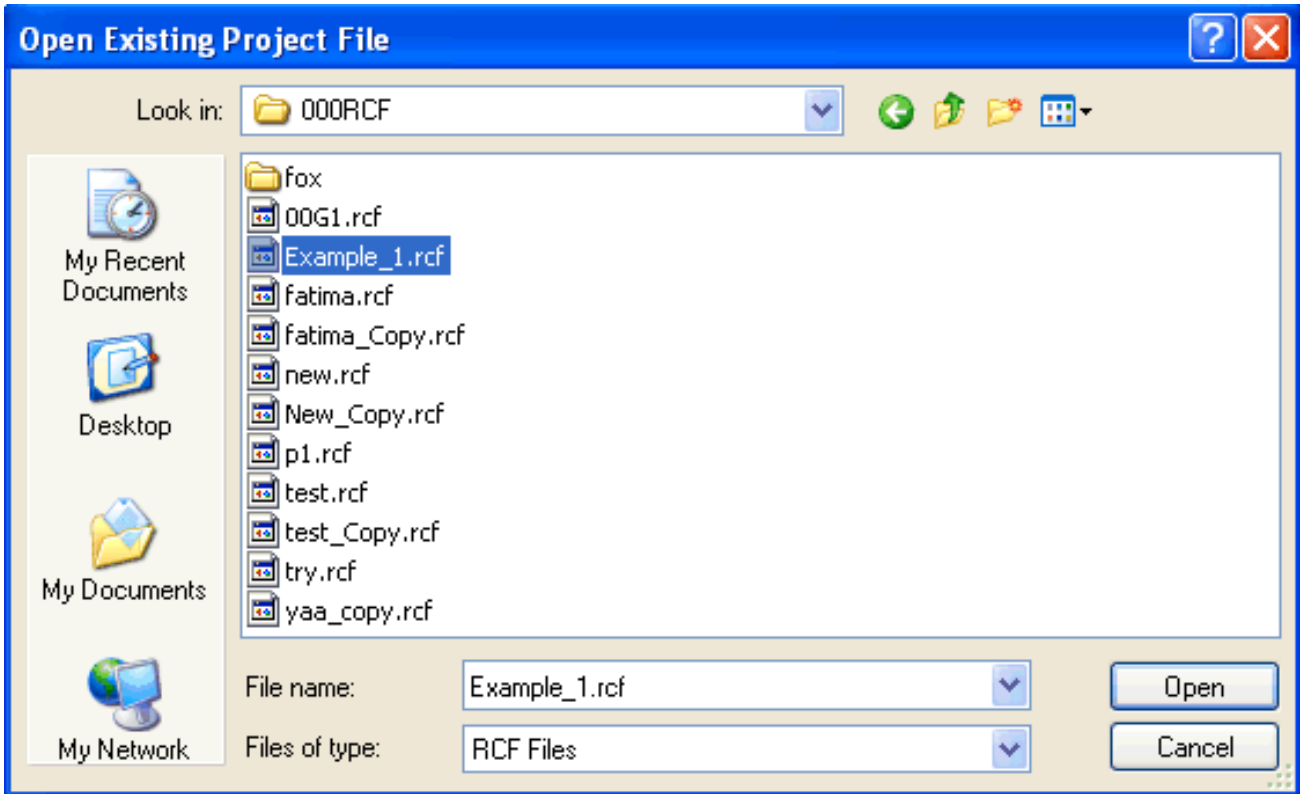


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

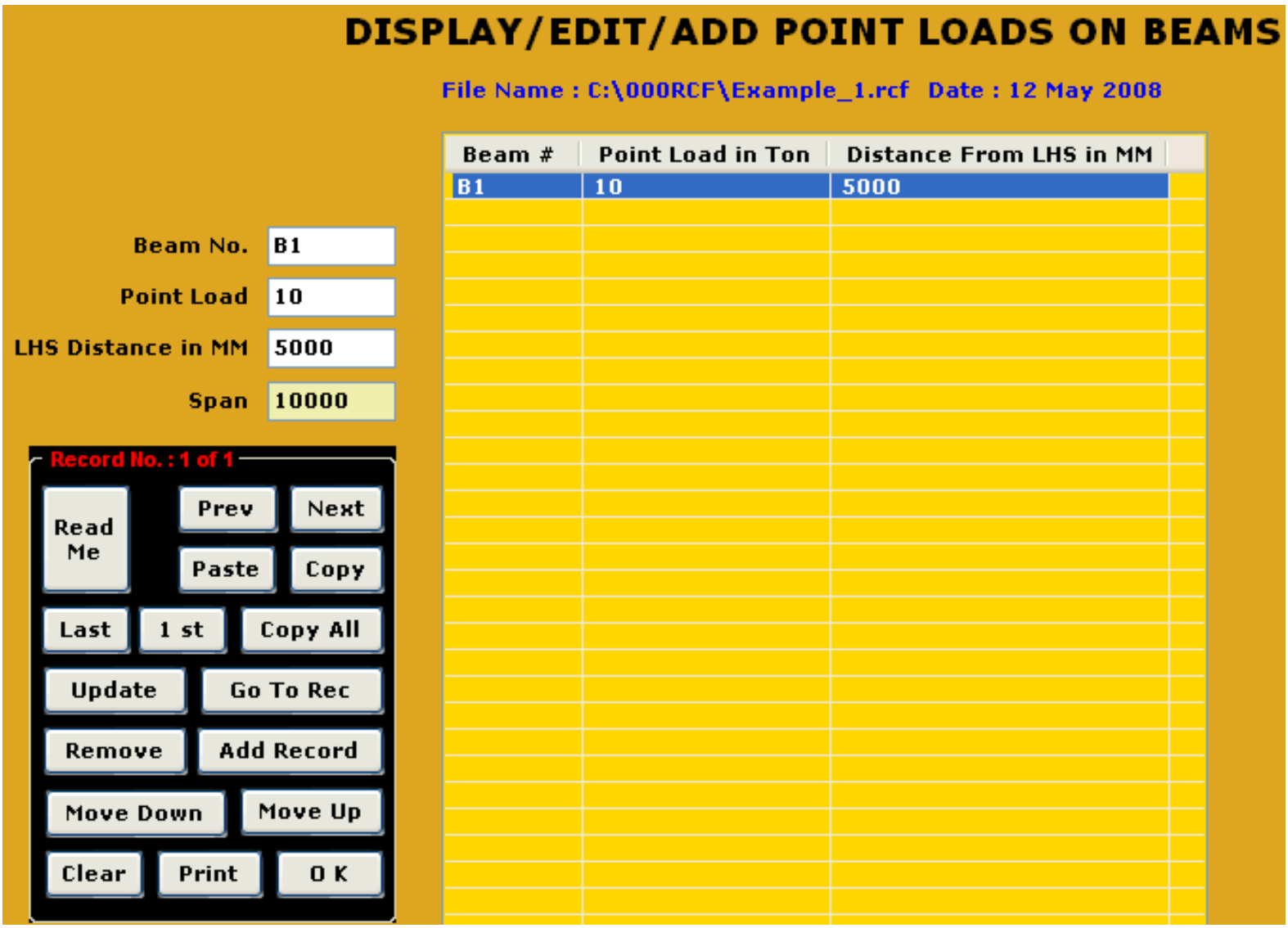
  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  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.**

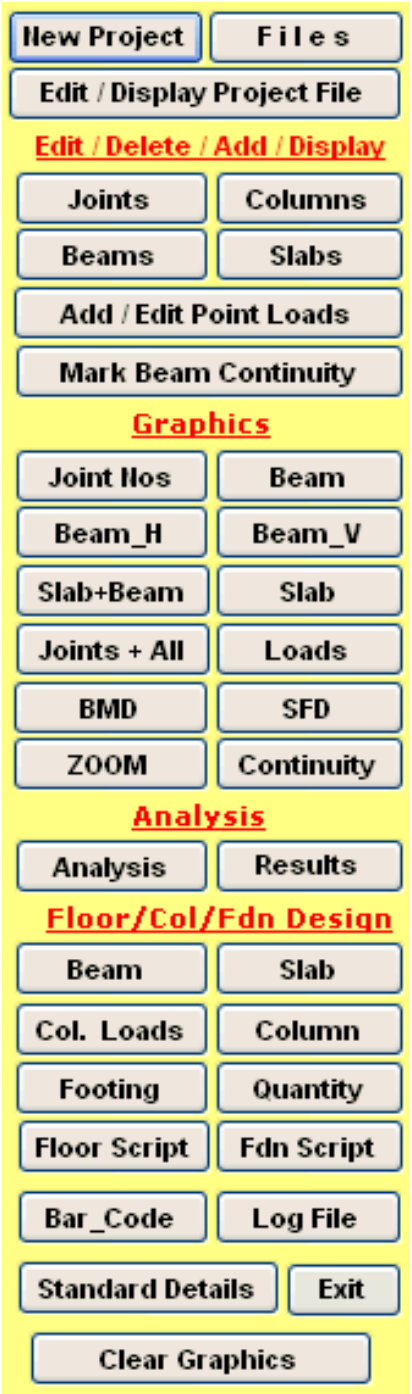
  1. LHS & RHS Joint #s cannot be repeated.
  2. Enter Point Loads due to Externally Applied Loads Only.
  3. Reactions due to Secondary Beams are Calculated Automatically.
  4. Use Add Button to Append Record.
  5. Use Update Button to Save Your Work.

- **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 RCF STEP BY STEP

STEP NO. 8 : Add & Edit Beam Continuity

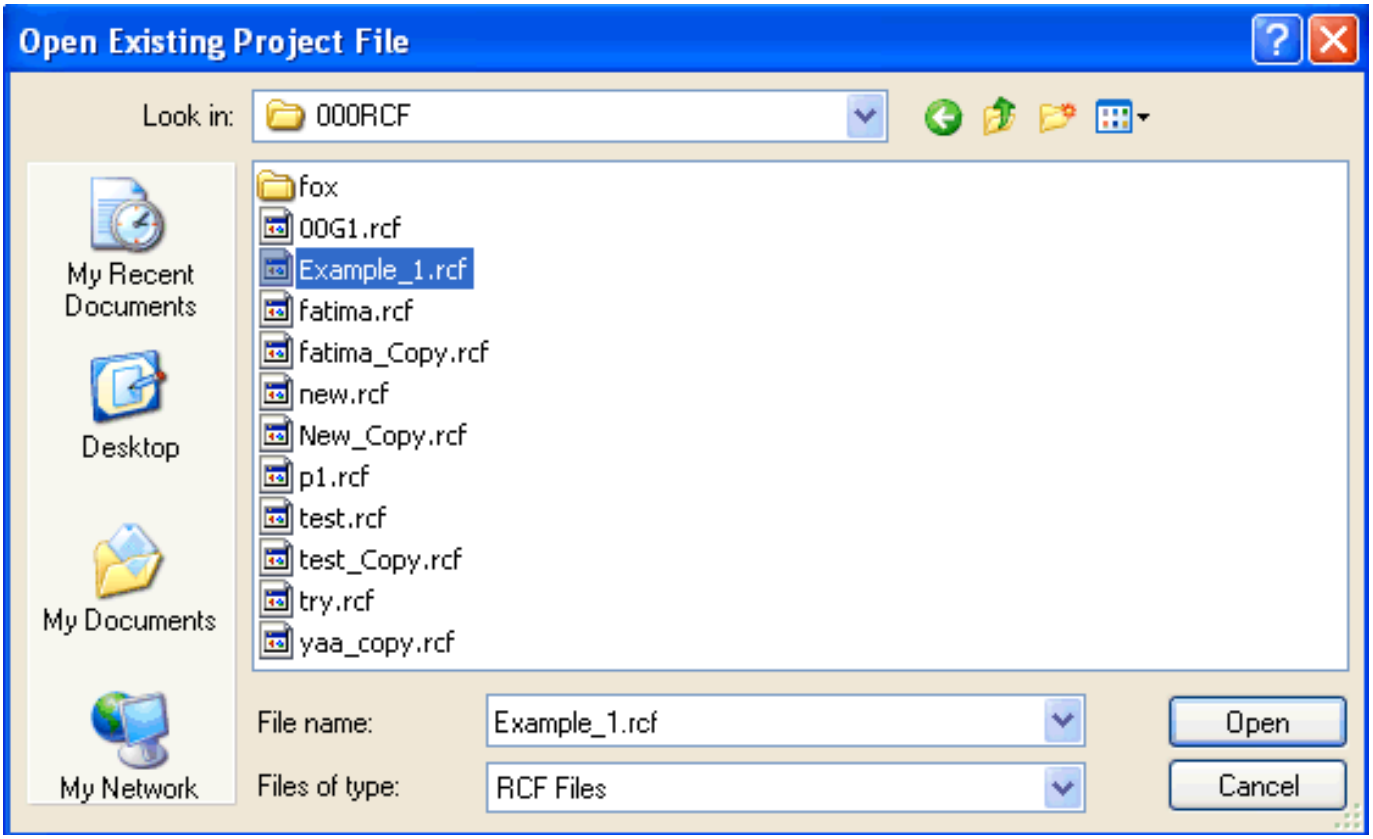


- **When Program starts, the Menu above is displayed. Under the Edit / Delete / Add / Display Heading following options are displayed.**

  - **Joints**
  - **Columns**
  - **Beams**
  - **Slabs**
  - **Add / Edit Point Loads**
  - **Mark Beam Continuity**

  Now Click on " Mark Beam Continuity " option.

  Following Graphics is displayed.



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

DISPLAY/EDIT/MARK BEAM CONTINUITY

File Name : C:\000RCF\Example_1.rcf Date : 12 May 2008

Record # 1

LHS Joint # 1

RHS Joint # 6

Record No. : 1 of 12

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

Serial #	LHS Joint #	RHS Joint #
1	1	6
2	7	13
3	14	20
4	28	34
5	35	40
6	35	7
7	36	1
8	37	2
9	38	3
10	39	4
11	40	5
12	34	6

- **Click " Add Record " button to Add a Continuous Beam. " 1 " will be displayed in the Text Box, When Add Record button is clicked. Refer our Actual Floor Plan (Step No. 2). Enter " 1 " & " 6 " in the corresponding Text Boxes of LHS Joint & RHS Joint. This means that Beams are continuous from Joint numbers 1 to 6 (B1, B2 & B3). Similarly Add all other continuous beams as I have Marked. If any Beam(s) are not marked as continuous than it will be treated as Simply Supported in Analysis. Note that Joints 22, 23 (Beam B14) & 25, 26 (B15) are not Marked as continuous.**

Just like any other option, here also a user can Display, Add, Edit & delete the 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 Records serially.

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. Use Add Button to Append Record.
  3. Use Update Button to Save Your Work.
  4. LHS / RHS Joint Numbers cannot be zero.

- **Now we have come to the end of Step # 8.**
  In the next step we will Check our Data Input Graphically.

STEP NO. 8 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 9 : Data Checking Through Graphics

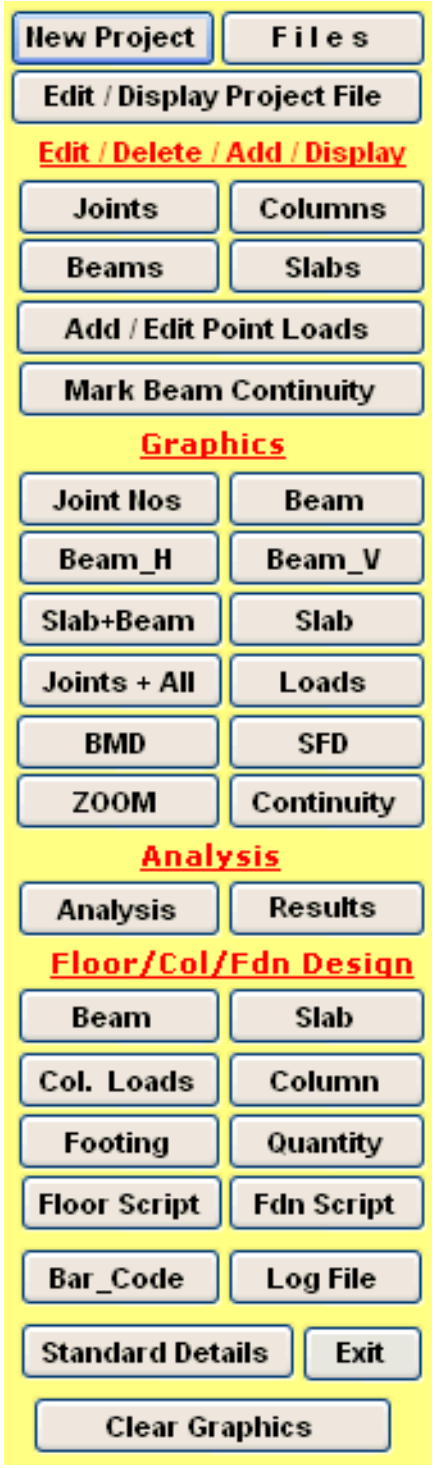
- **A User should thoroughly check Data Input at all stages. During Adding / Editing Data through tables, Beam " SPAN " and Slab Dimensions (Shorter & Longer) should be constantly monitored for any error.**
  After DATA Input is over, it should be checked visually & by taking printouts of various Graphics Options. Analysis, 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, End Moments).**
- **Floor Analysis, Beam & Slab 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, Accounting for Beam Continuity, i.e. Maximum of Simple Reaction & Continuous Reaction is taken for Column Design.
The Difference should not exceed say 10 %. The major difference should calls for closer look at the Data-Input.

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 later chapters.

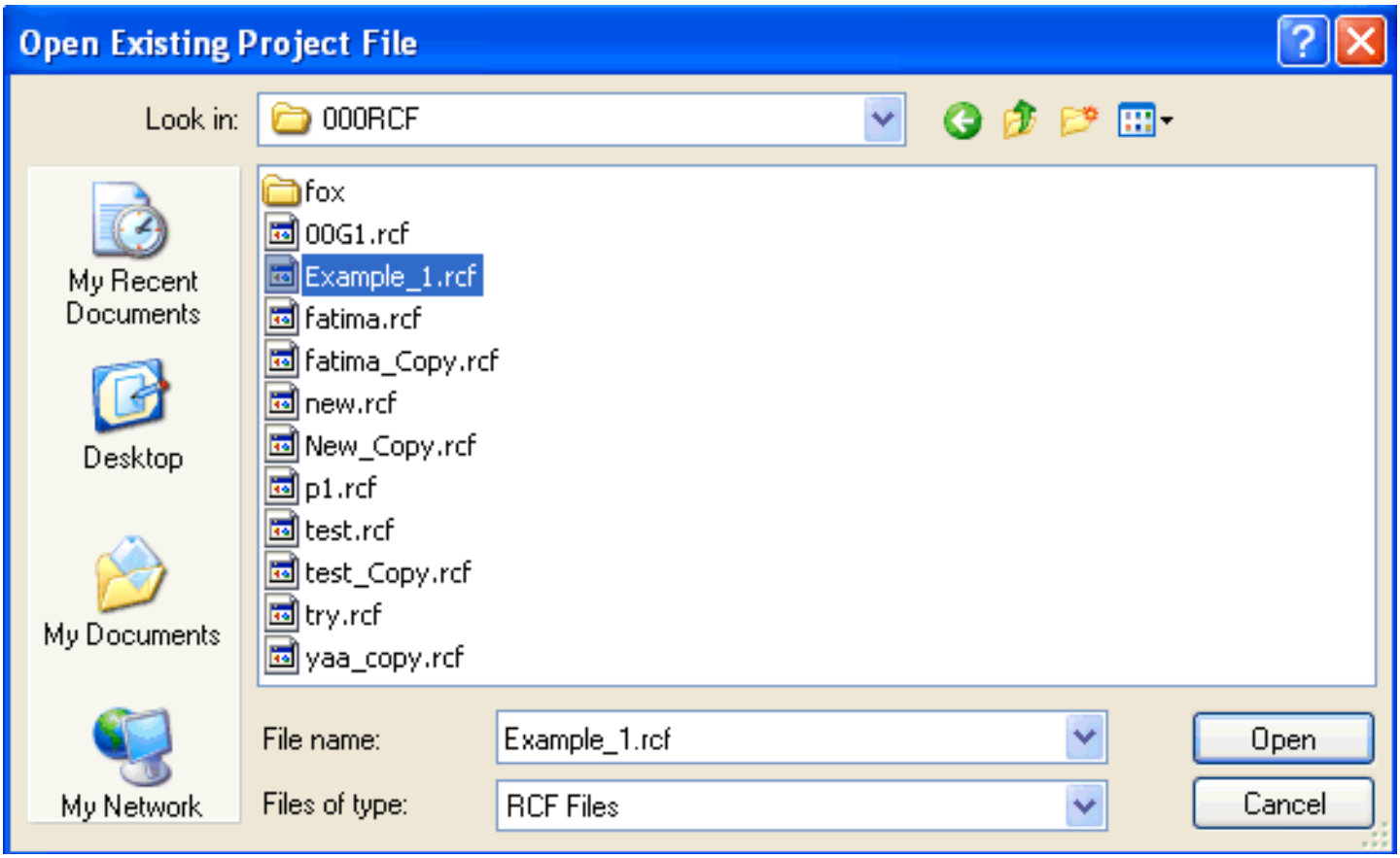

New Project
Files
Edit / Display Project File
Edit / Delete / Add / Display
Joints
Columns
Beams
Slabs
Add / Edit Point Loads
Mark Beam Continuity
Graphics
Joint Nos
Beam
Beam_H
Beam_V
Slab+Beam
Slab
Joints + All
Loads
BMD
SFD
ZOOM
Continuity
Analysis
Analysis
Results
Floor/Col/Fdn Design
Beam
Slab
Col. Loads
Column
Footing
Quantity
Floor Script
Fdn Script
Bar_Code
Log File
Standard Details
Exit
Clear Graphics

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

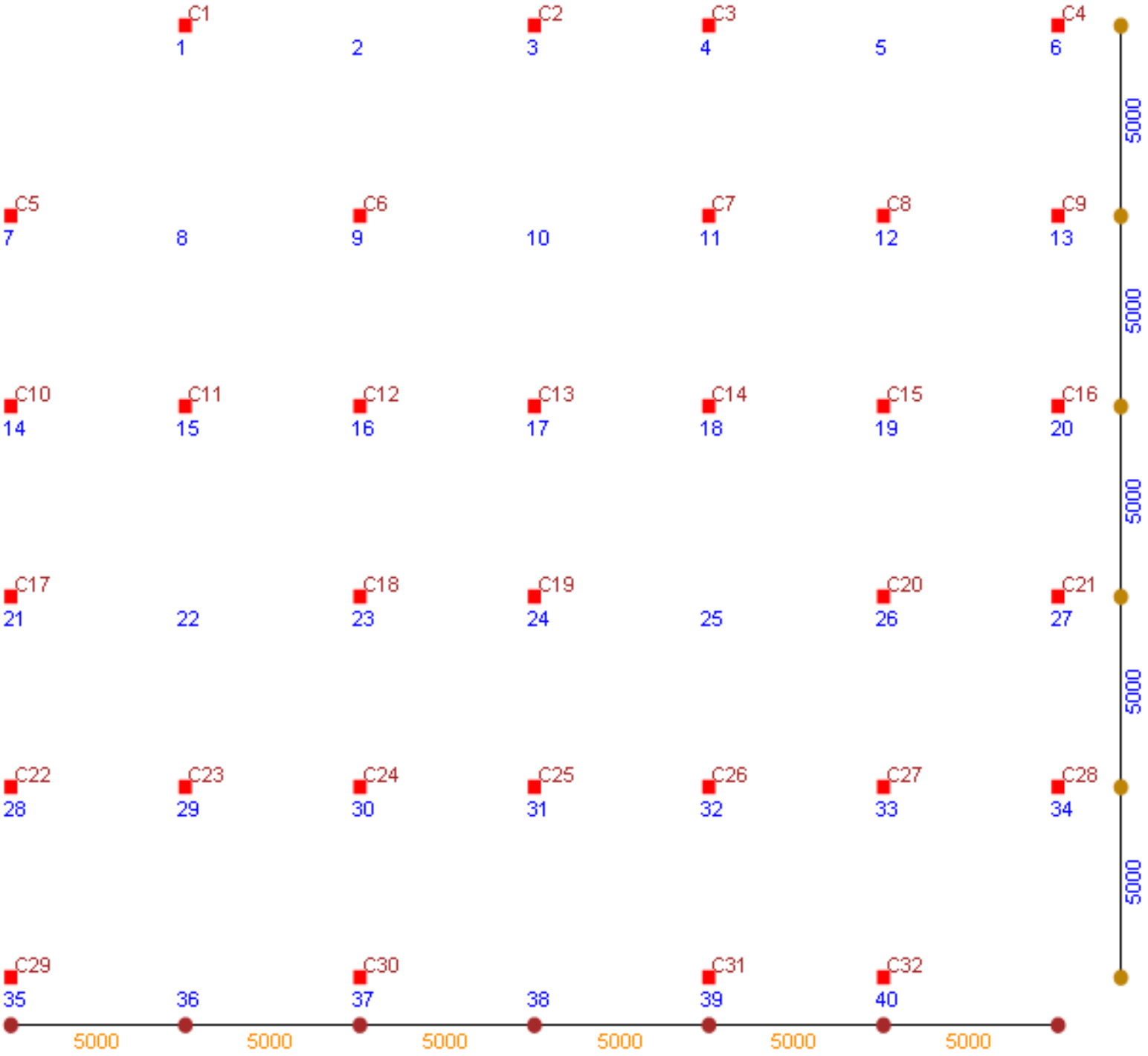
• Joint Nos
• Beam
• Beam_H (Only Horizontal Beam numbers will be Displayed).
• Beam_V (Only Vertical Beam numbers will be Displayed).
• Slab + Beam (Beams, Slabs & Columns are displayed).
• Slab (Only Slabs & Columns are displayed).
• Joints + ALL (For Display of Joints, Columns, Beams & Slabs)
• Loads (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).
• Continuity (Display of Beams Marked as Continuous.)

Now Click on " Joint Nos " option.

Following Graphics is displayed.

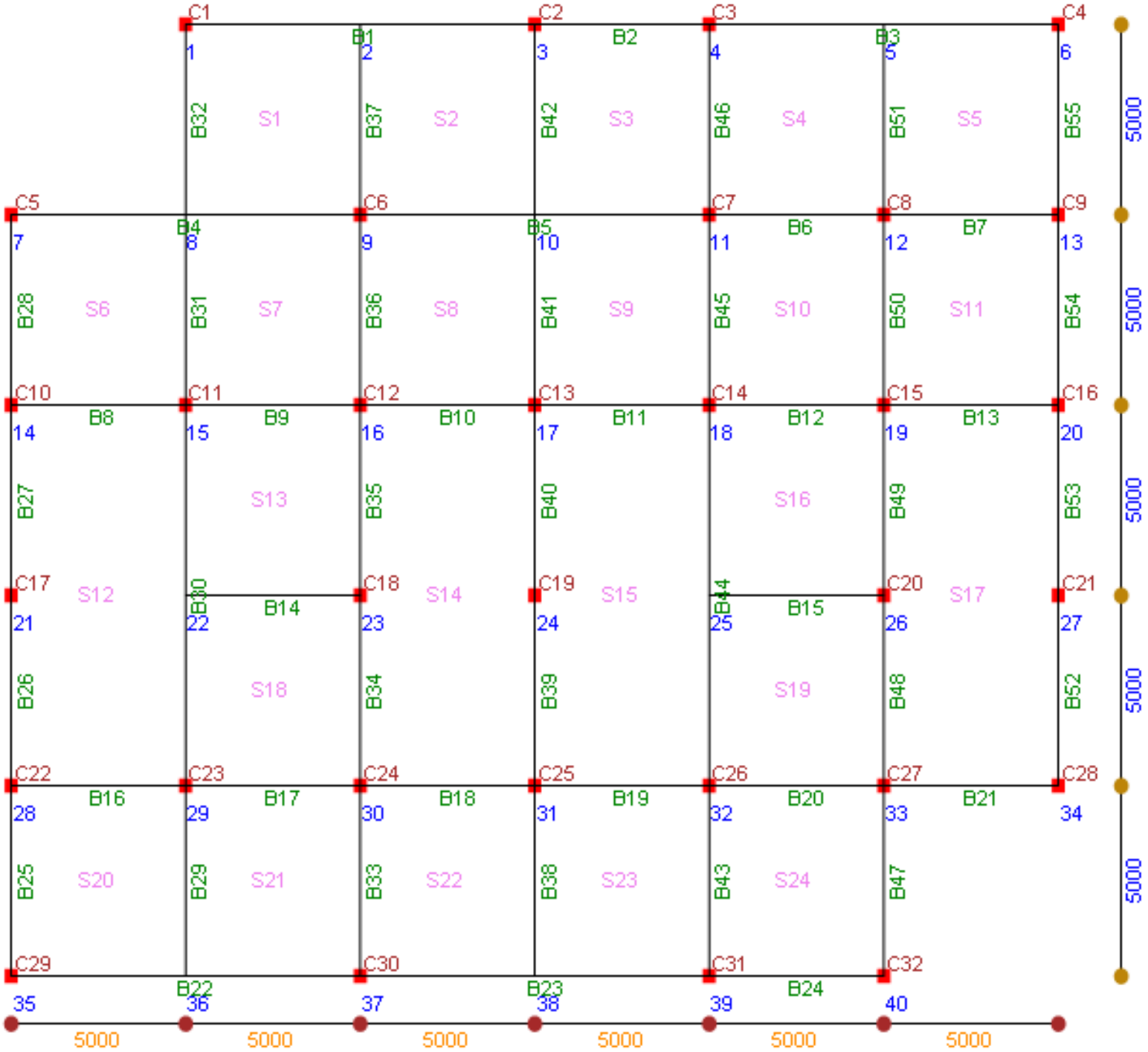


- **Now select " Example _1 File & Press Open Button.**
Following Graphics will be displayed.

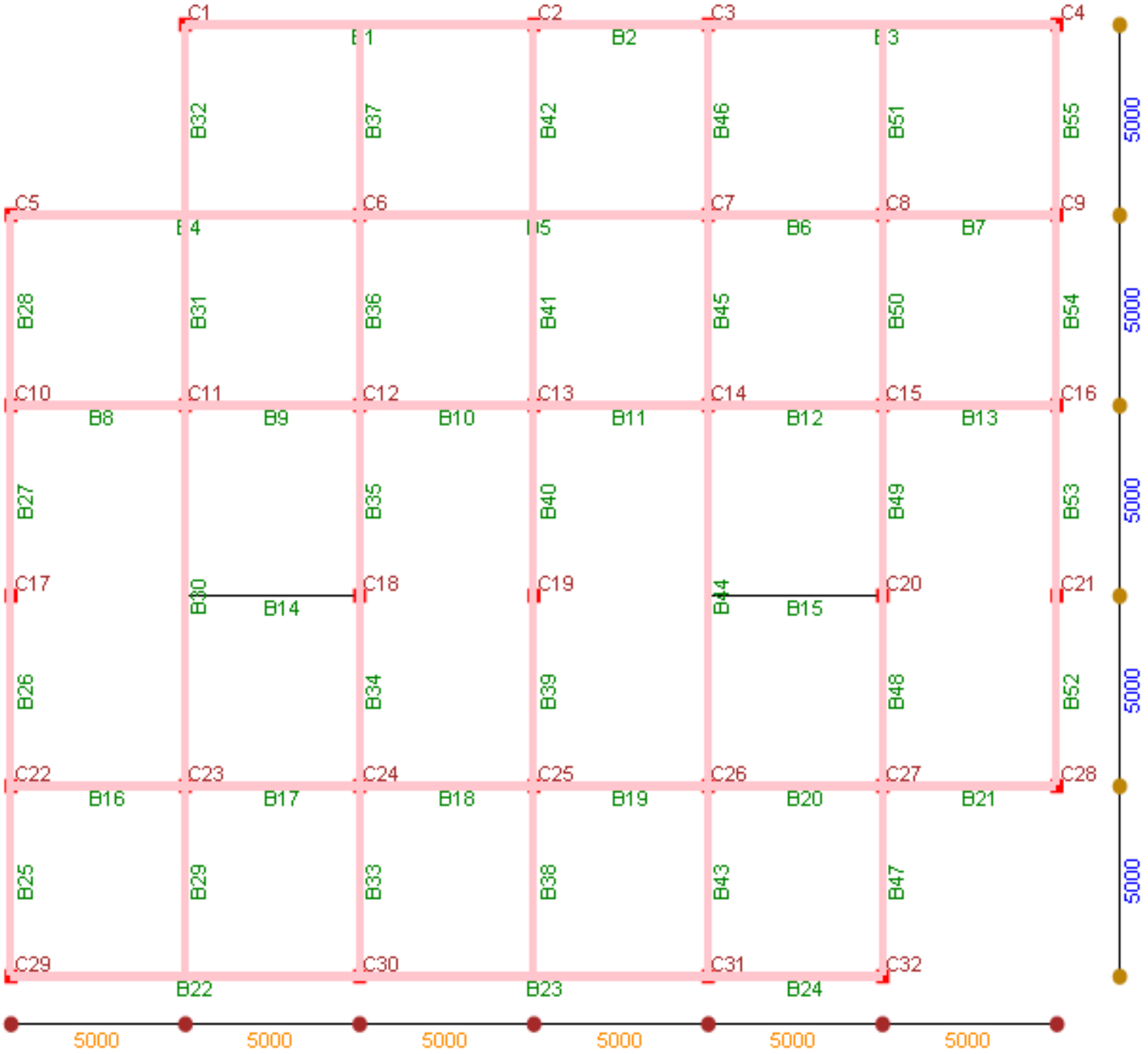


- **The above Graphics displays Joint, Column Numbers as well as Horizontal and Vertical Dimensions.**
  A User should Check the Location of Each Joint & Column & C/C Horizontal & Vertical Grid distance.

- **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.



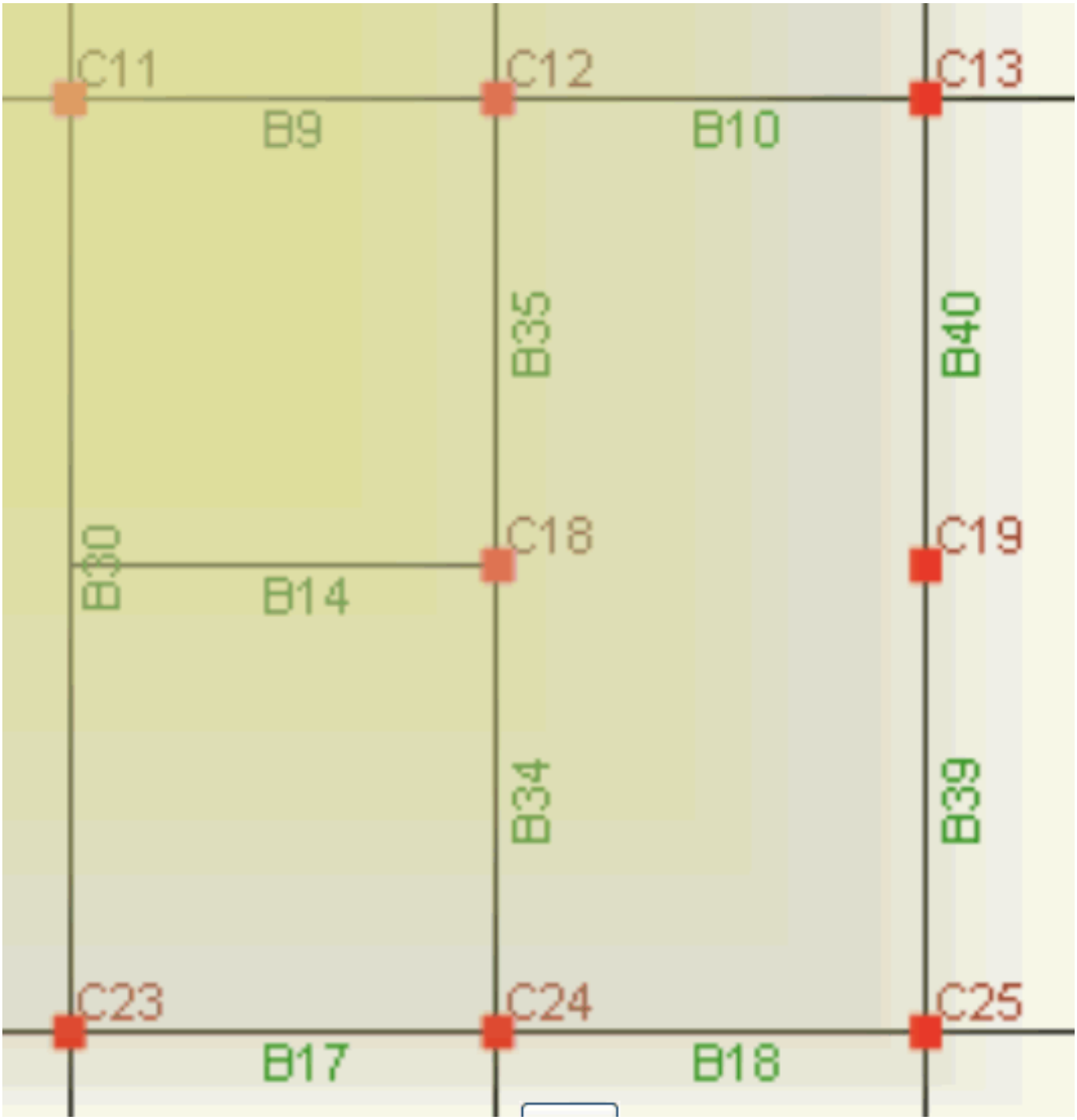
- **Now Click Continuity Button.**
  Following Graphics is displayed.



Continuity of Beams Marked in Pink

Print

- **The Beams not marked in Pink are Simply Supported Beams.**
  Now Click " Beam " button & after display of Graphics click " ZOOM " button.
  Now Left Click with mouse near the Column C11 & Drag it near the Column C25.
  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 :**

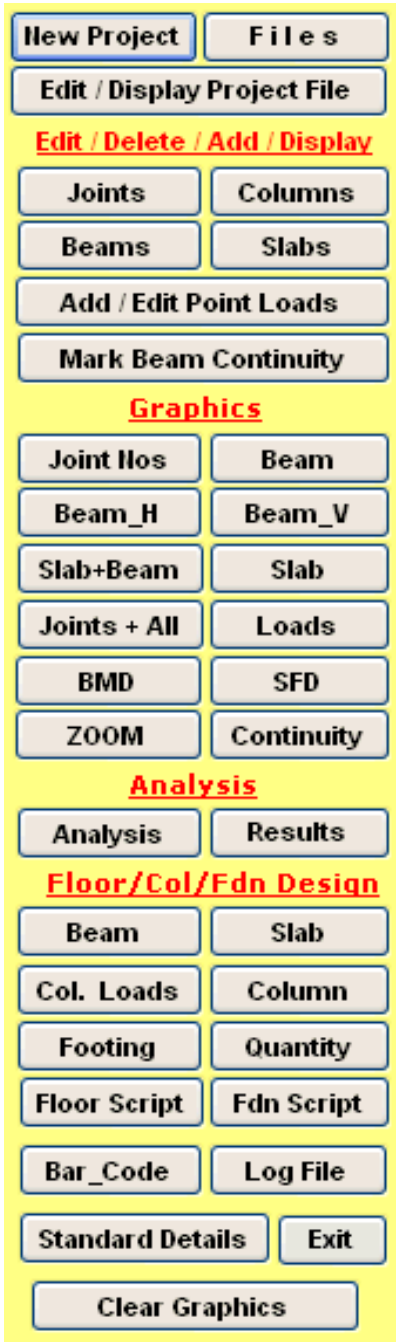
• Loads (Display of Slab, Point Loads & Reactions from Secondary Beams, to be used after Analysis, and Design options have been successfully Run).
• BMD (Display of Bending Moment Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.
• SFD (Display of shear Force Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.

Now we have come to the end of Step # 9.
In the next step we will Run " Analysis " option.

STEP NO. 9 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 10 : Analysis & Its Results



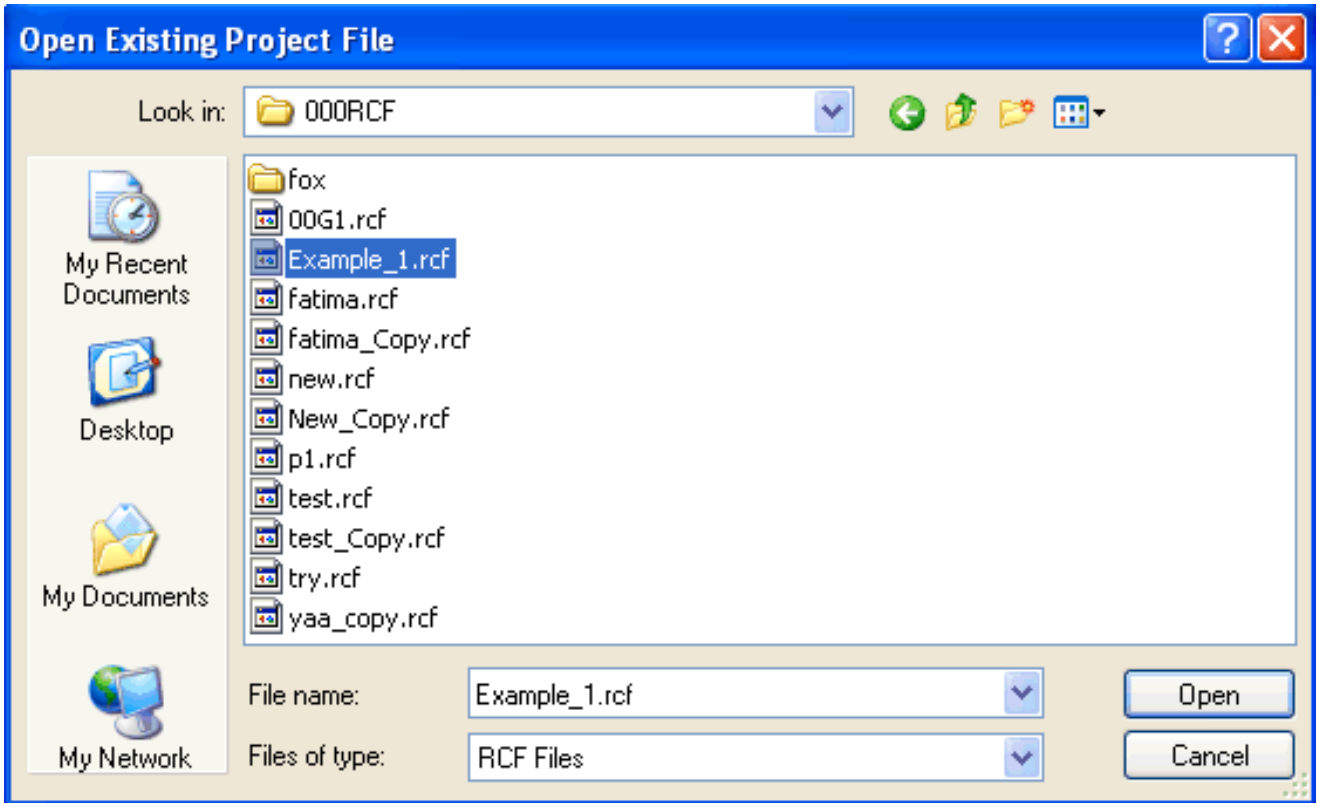
- **After entering Data & Checking it thoroughly, Relax, let the software do its Job. The 1st milestone is Analysis.**

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

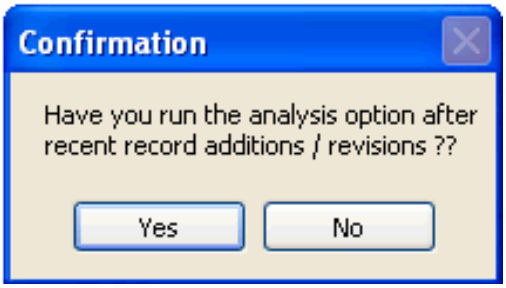
  - **Analysis**
  - **Results**

  Now Click on " Analysis " option.

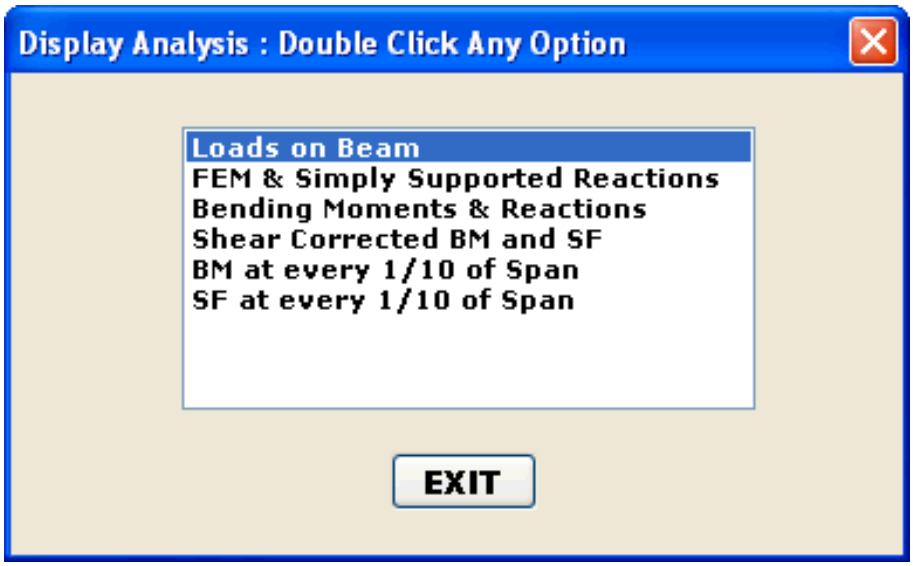
  Following Graphics is displayed.



- **Now select " Example _1 File & Press Open Button.**
  The Analysis will commence. A window will open & it will indicate number of Joints, Columns, Beams 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 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 Warning is displayed after Selecting File from "Open Existing Project File" window.**



- **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.**
  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.



- **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.

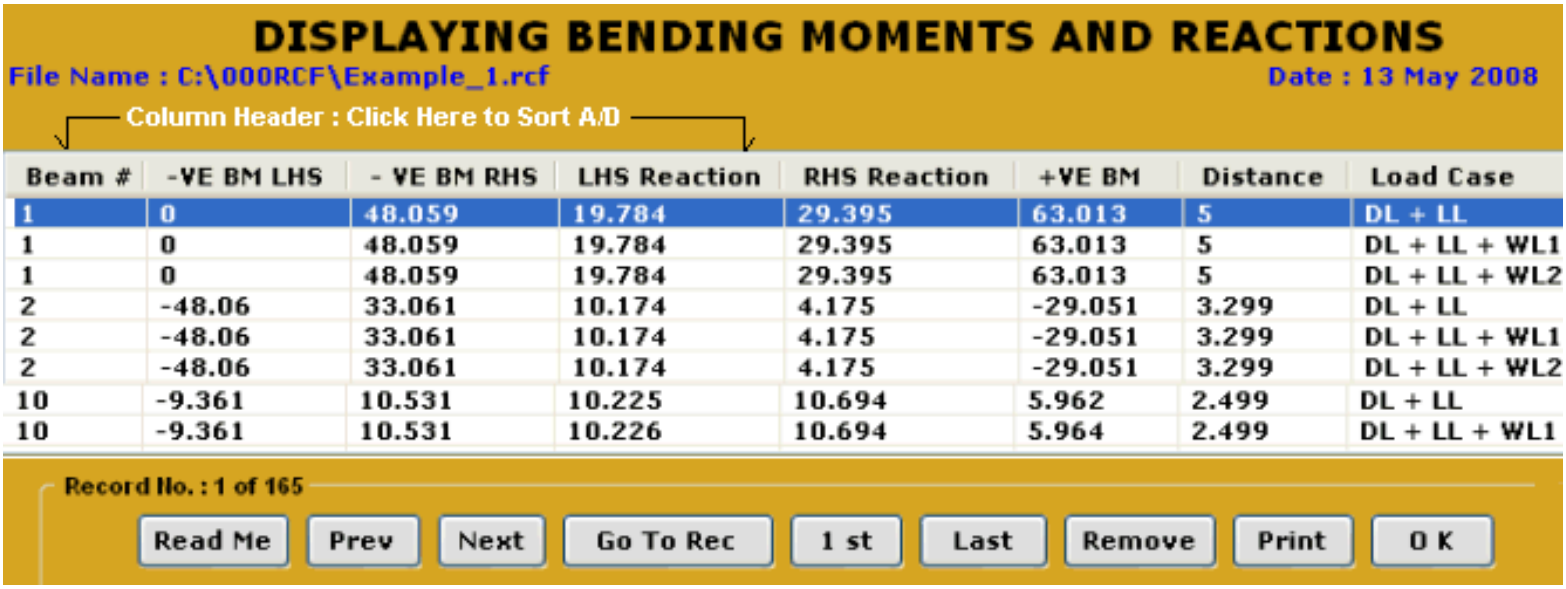
- **Now Double Click on " FEM & SS Reactions " Option. A new window will open displaying Fixed End Moments and Simply Supported Reaction on each Beam. Click on " Read Me " button, following important messages are displayed.**

  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.

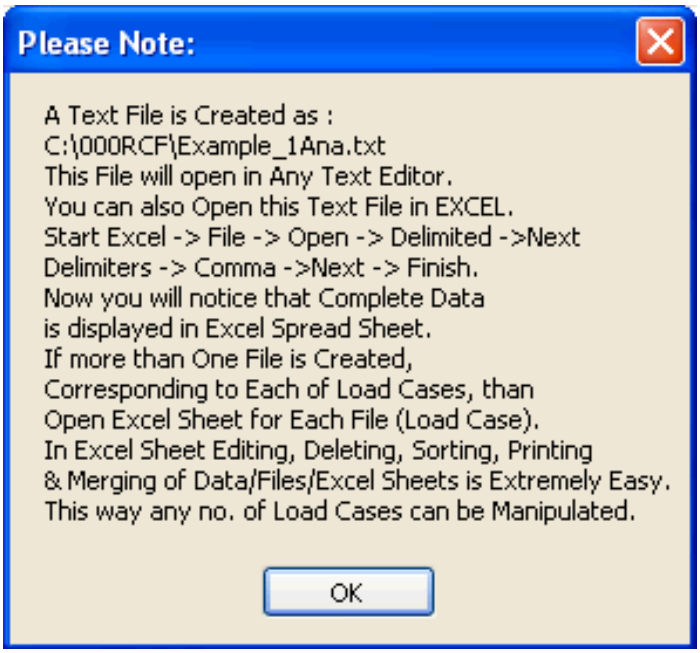
- **Now Double Click on " Bending Moments and Reactions " Option. This is the most Important Option. A new window will open displaying End Moments and Reactions on each Beam. Click on " Read Me " button, following important messages are displayed.**

  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. Load Cases are (a) DL + LL (b) DL + LL + WL1 &
  8. (c) DL + LL + WL2. The Case (c) is automatically
  9. Calculated as -1 x Case (b).
  10. Case (b) & Case (c) are Multiplied by
  11. Reduction factor of 0.80. (During Design)
  12. WL1 is due to Externally applied Wind/EQ Moment.
  13. In order to Sort the Values in Ascending OR
  14. Descending Order, Just Click Column Header at Top.

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



- **Note that Column Headers are all the Titles at Top as Marked in White Color. Just Click them to Sort.**
  The " Remove " Button is placed here for ease of Printing. In case a User wants to Print only DL + LL
  case, then he can simply delete other cases (DL + LL + WL1 WL2). 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.

  Now Double Click on " Shear Corrected BM & SF " Option. These values are used for beam Design. A new window will open displaying Shear Corrected Moments and Shear Forces on each Beam for all the three (3) cases. Click on " Read Me " button, following important messages are displayed.

  1. Beam Width, Depth in MM.
  2. Shear Corrected BM & SF are calculated at Support Face
  3. and At Effective Depth from Support Face Respectively.
  4. LHS / RHS Shear Corrected BM in T-M.
  5. LHS Shear Corrected Shear in Tons.
  6. RHS Shear Corrected Shear in Tons.
  7. Load Cases are (a) DL + LL (b) DL + LL + WL1 &
  8. (c) DL + LL + WL2. The Case (c) is automatically
  9. Calculated as -1 x Case (b).
  10. Case (b) & Case (c) are Multiplied by
  11. Reduction factor of 0.80.
  12. WL1 is due to Externally applied Wind/EQ Moment.
  13. In order to Sort the Values in Ascending OR
  14. Descending Order, Just Click Column Header at Top.

- **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
  4. 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. Load Cases are (a) DL + LL (b) DL + LL + WL1 &
  8. (c) DL + LL + WL2. The Case (c) is automatically
  9. Calculated as -1 x Case (b)
  10. Case (b) & Case (c) are Multiplied by
  11. Reduction factor of 0.80.
  12. WL1 is due to Externally applied Wind/EQ Moment.
  13. In order to Sort the Values in Ascending OR
  14. 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. Load Cases are (a) DL + LL (b) DL + LL + WL1 &
  8. (c) DL + LL + WL2. The Case (c) is automatically
  9. Calculated as -1 x Case (b).
  10. Case (b) & Case (c) are Multiplied by
  11. Reduction factor of 0.80.
  12. WL1 is due to Externally applied Wind/EQ Moment.
  13. In order to Sort the Values in Ascending OR
  14. Descending Order, Just Click Column Header at Top.

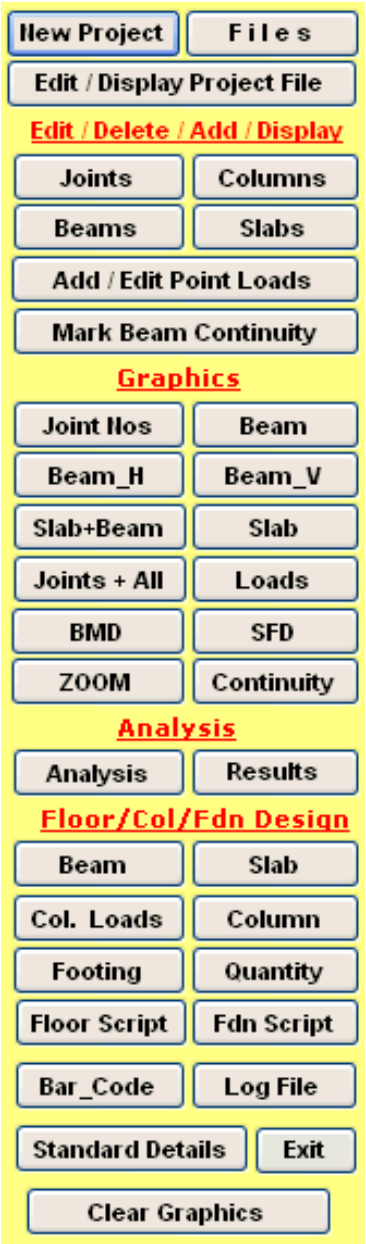
- **Now we have come to the end of Step # 10.**
  In the next step we will Run " Beam Design " Option.

STEP NO. 10 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 11

Beam & Slab Design, Column Loads, Quantities & Cost Estimation

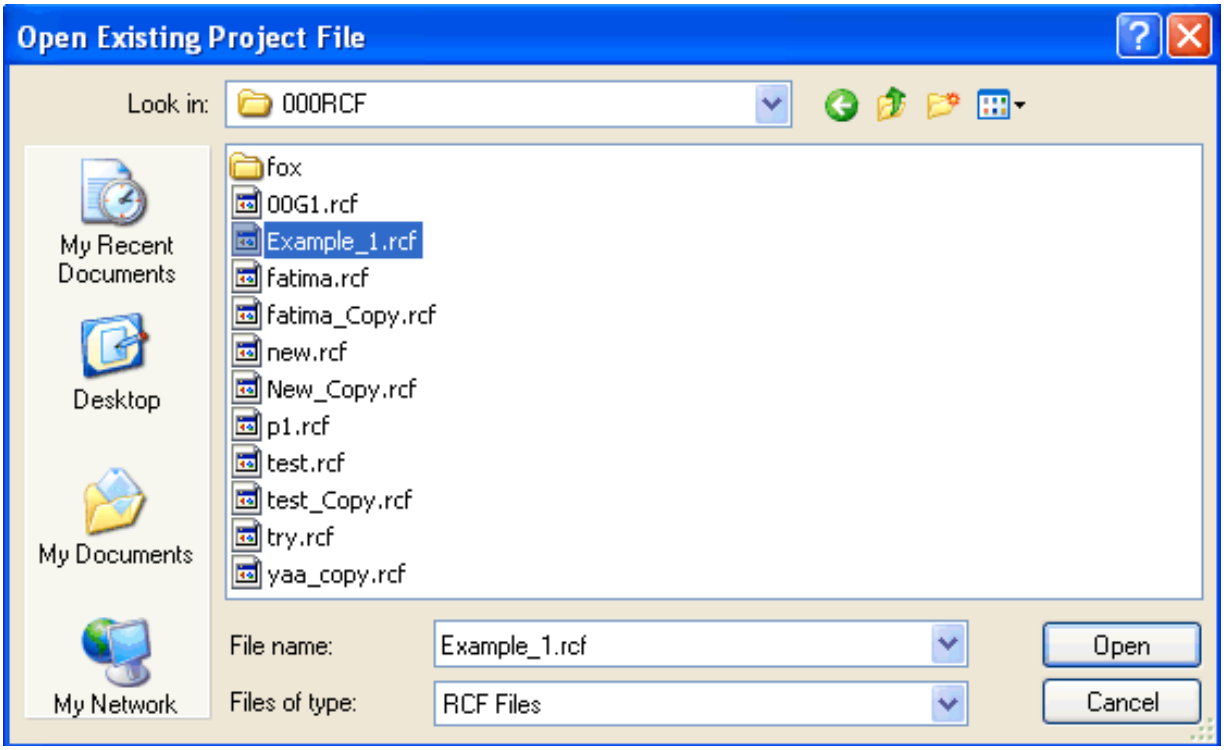


- **When Program starts, the Menu above is displayed. Under the Floor/Col/Fdn Design Heading following options are displayed.**

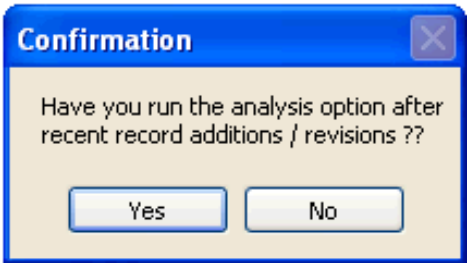
  - **Beam**
  - **Slab**
  - **Column Loads**
  - **Column Design**
  - **Footing Design**
  - **Quantity**
  - **Floor Script for AutoCAD Dwg.**
  - **Foundation Script for AutoCAD Dwg.**

  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.**
  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 Beam Schedule will be displayed. We have displayed the Schedule in two Part as it is too wide due to " Error Message ".

BEAM REINFORCEMENT SCHEDULE

Beam		Bottom Steel			Extra LHS Support Steel				Extra RHS Support Steel	
No.	Width	Depth	Straight	Curtail	Top Steel	#	Top	Bottom	#	Top
1	ERROR : SEE LOG FILE		T		T					
2	ERROR : SEE LOG FILE		T		T					
3	ERROR : SEE LOG FILE		T		T					
4	ERROR : SEE LOG FILE		T		T					
5	ERROR : SEE LOG FILE		T		T					
6	ERROR : SEE LOG FILE		T		T					
7	230	450	2 T 25	2 T 25	3 T 12 + 1 T 20	C8			C9	4 T 12
8	230	450	2 T 20	2 T 20	2 T 10 + 1 T 8	C10	2 T 12 + 2 T 10	2 T 10 + 1 T 12	C11	4 T 20
9	230	450	2 T 16	2 T 10	2 T 8	C11			C12	4 T 16
10	230	450	2 T 16	1 T 20	2 T 8	C12			C13	3 T 20
11	230	450	2 T 16	1 T 20	2 T 8	C13			C14	4 T 16
12	230	450	2 T 16	2 T 10	2 T 8	C14			C15	4 T 20
13	230	450	2 T 20	2 T 20	2 T 10 + 1 T 8	C15			C16	2 T 12 + 2 T 10
14	230	450	2 T 25	2 T 25	3 T 20	B30			C18	2 T 16 + 1 T 12
15	230	450	2 T 25	2 T 25	3 T 20	B44			C20	2 T 16 + 1 T 12
16	230	450	2 T 20	2 T 20	2 T 10 + 1 T 8	C22	2 T 12 + 2 T 10	2 T 10 + 1 T 12	C23	4 T 20
17	230	450	2 T 16	2 T 10	2 T 8	C23			C24	4 T 16
18	230	450	2 T 16	1 T 20	2 T 8	C24			C25	3 T 20
19	230	450	3 T 12	1 T 20	2 T 8	C25			C26	2 T 20 + 2 T 12
20	230	450	2 T 16	1 T 20	2 T 8	C26			C27	3 T 20

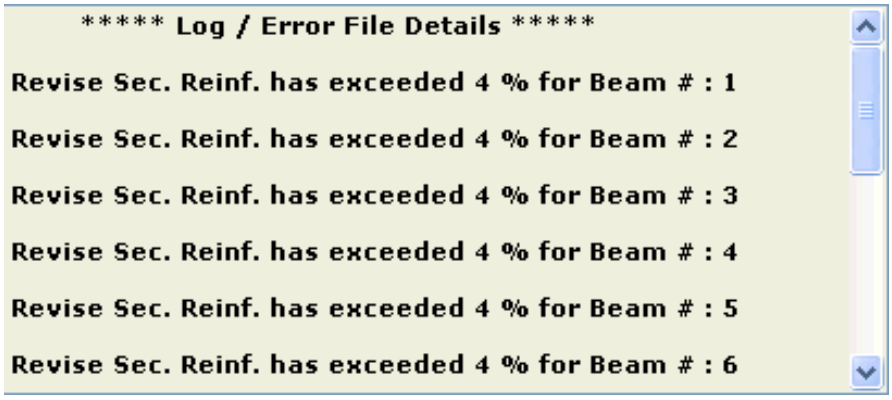
Record No. : 1 of 55

Read Me | Prev | Next | Go To Rec | 1 st | Last | Print | O K

Extra RHS Support Steel			Skin	
#	Top	Bottom	Stirrups	Steel
			T @	
			T @	
			T @	
			T @	
			T @	
			T @	
C9	4 T 12	2 T 12 + 1 T 10	T 8 @ 175	
C11	4 T 20	3 T 8	T 10 @ 200	
C12	4 T 16		T 8 @ 175	
C13	3 T 20		T 8 @ 190	
C14	4 T 16		T 8 @ 190	
C15	4 T 20	3 T 8	T 8 @ 175	
C16	2 T 12 + 2 T 10	2 T 10 + 1 T 12	T 10 @ 200	
C18	2 T 16 + 1 T 12	2 T 12 + 2 T 10	T 8 @ 190	
C20	2 T 16 + 1 T 12	2 T 12 + 2 T 10	T 8 @ 190	
C23	4 T 20	3 T 8	T 10 @ 200	
C24	4 T 16		T 8 @ 175	
C25	3 T 20		T 8 @ 190	
C26	2 T 20 + 2 T 12		T 8 @ 205	
C27	3 T 20		T 8 @ 205	
C28	2 T 10 + 1 T 8	3 T 8	T 8 @ 200	

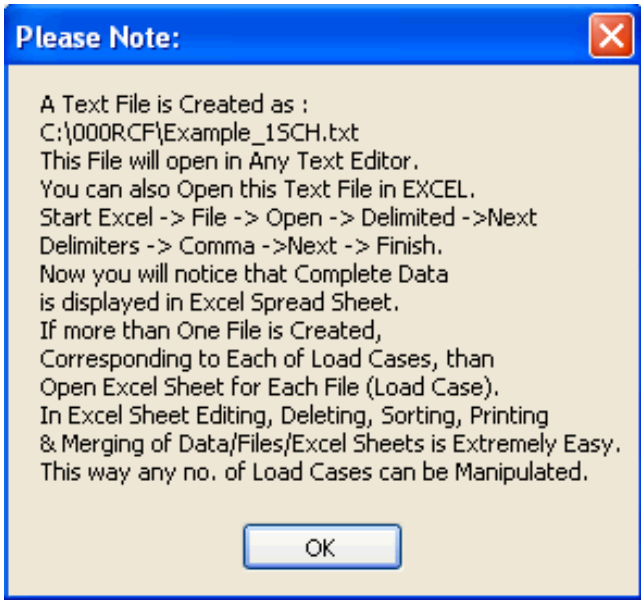
Last | Print | O K

- **Note that in the Beam Schedule " Error : See Log File " is displayed for Beam nos. B1, 2, 3, 4, 5, 6, 22, 23, 29, 30, 31, 43, 44, & B45. Now Click the " Log File " Button (Situated near the End of menu). Following window is displayed.**



- **The error message is clear, Reinforcements has exceeded 4 % for Beam nos. B1, 2, 3, 4, 5, 6, 22, 23, 29, 30, 31, 43, 44, & B45. Change the size of these beams to 300 * 750 MM using " Beams " option under Edit / Delete / Display Caption. Now Re-Run the Analysis File. After analysis is over, again perform Beam Design, now you will find that there is no error in Beam Schedule.**

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.**

When " OK " button is clicked, following Message Regarding Creation of Beam Schedule in AutoCAD is displayed.

- **For Detail explanation refer step no. 14.**
  Now Click " Slab " Option. Following Graphics is displayed.

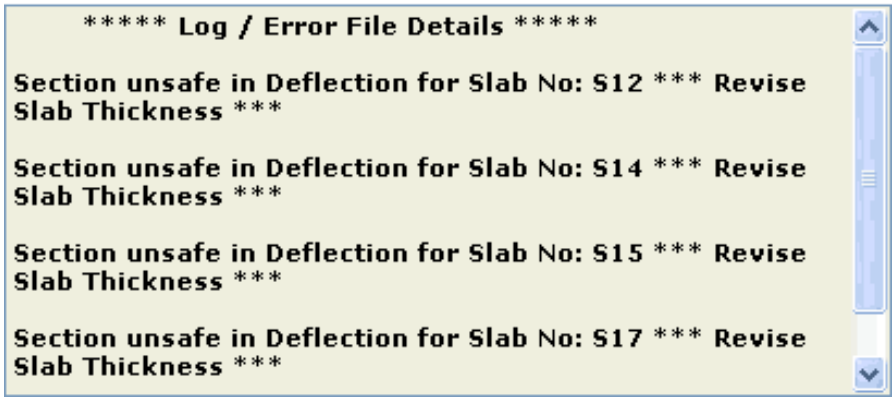
SLAB REINFORCEMENT SCHEDULE

Slab		Steel Along Shorter Direction			Steel Along Longer Direction		
No.	Thickness	Btm Straight	Bottom Cut	@ Support Top	Btm straight	Bottom Cut	@ Support Top
S1	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S2	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S3	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S4	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S5	150	T 8 @ 280	T 8 @ 280	T 10 @ 175	T 8 @ 280	T 8 @ 280	T 10 @ 175
S6	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S7	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S8	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S9	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S10	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S11	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S12	ERROR : SEE ...						
S13	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S14	ERROR : SEE ...						
S15	ERROR : SEE ...						
S16	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S17	ERROR : SEE ...						
S18	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S19	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S20	150	T 8 @ 280	T 8 @ 280	T 10 @ 175	T 8 @ 280	T 8 @ 280	T 10 @ 175
S21	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140

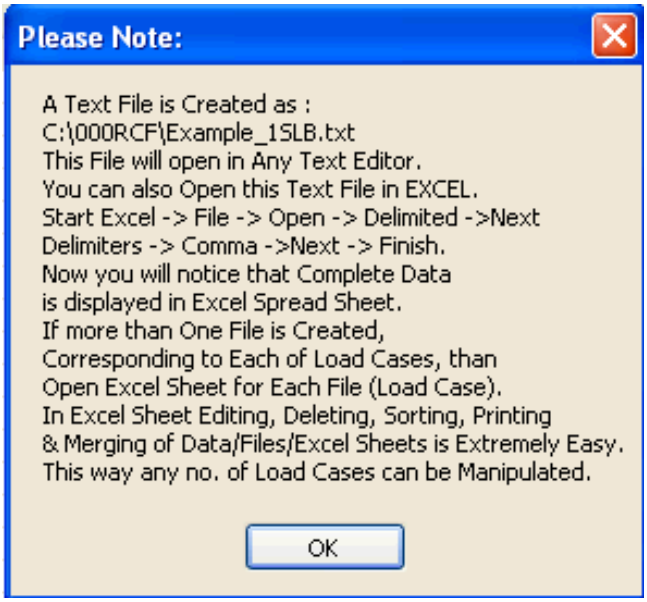
Record No. : 1 of 24

Read Me | Prev | Next | Go To Rec | 1 st | Last | Print | O K

- **Note that in the Slab Schedule " Error : See Log File " is displayed for Slab nos. S12, 14, & S17.**
  Now Click the " Log File " Button (Situated near the End of menu). Following window is displayed.

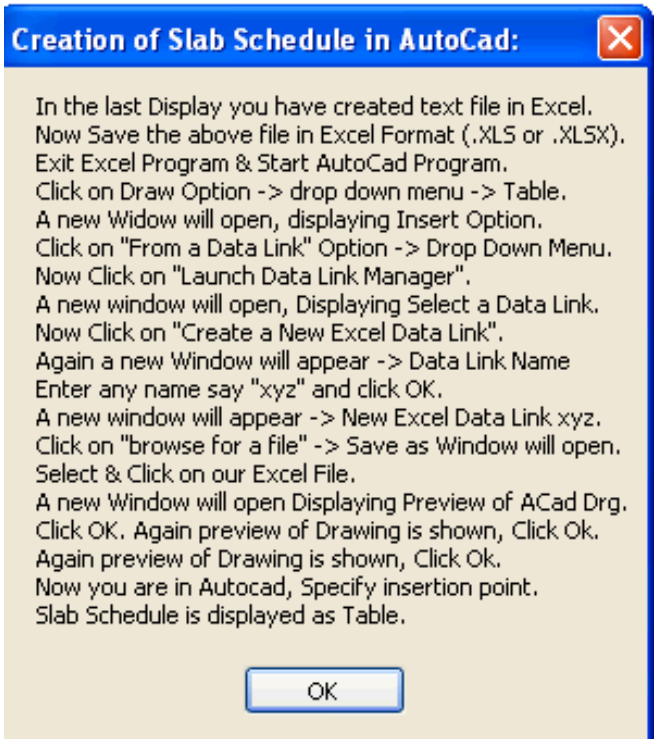


- **The error message is clear, Deflection has exceeded the Permissible Limit for Slabs S12, 14, 15 & S17. Change the Thickness of these Slabs to 175 MM using " Slabs " option under Edit / Delete / Display Caption. Now Re-Run the Analysis File. After analysis is over, again perform Beam and Slab Design, now you will find that there is no error in Slab Schedule.**
  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.**

  When " OK " button is clicked, following Message Regarding Creation of Slab Schedule in AutoCAD is displayed.

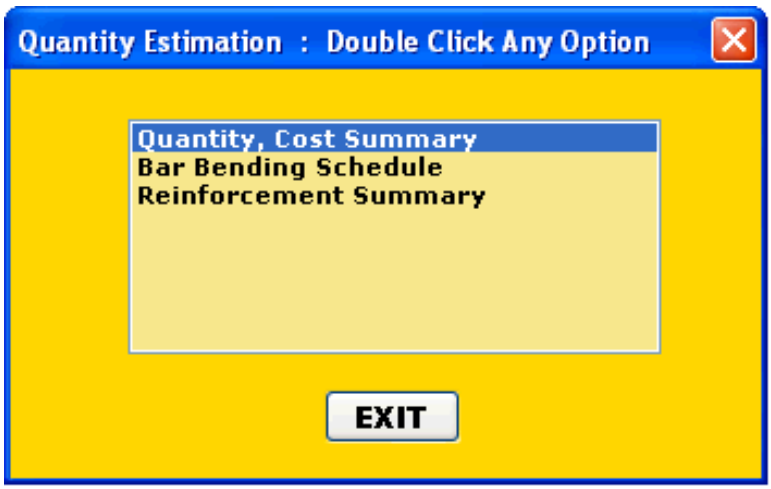


- **For Detail explanation refer step no. 15.**

Now Click " Quantity " Option.
A window will open displaying four options viz;

- **Floor Beams + Slabs**
- **Column Project**
- **Footing Project**
- **Total Project**

Click on Floor Beams + Slabs Option, following Graphics is displayed.



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

DISPLAYING QUANTITIES AND COST SUMMARY

File Name : C:\000RCF\Example_1.rcf Date : 15 May 2008

Item	Quantity	Rate	Cost
M20 Concrete in M3	141.598	9000	1274382
Total Reinforcement in Tons	12.89	50000	644499.9
Total Masonry Work in M2	651.888	850	554104.8
Total Plaster in M2	2337.526	400	935010.4
Total Painting in M2	2337.526	100	233752.6
Total Floor Area in M2 (Flooring)	700	800	560000
Total Door / Windows in M2	92.4	2500	231000
Total Cost of Floor			4432750
Unit Cost of Floor in Rs / M2			6332.499
Unit Cost of Floor in Rs / sqft			589.069
Total Cement Bags Required in Nos.	1729		
Total Sand Consumption in M3	159		
Total Aggregate Required in M3	134		

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

BAR BENDING SCHEDULE

File Name : C:\000RCF\Example_1.rcf Date : 15 May 2008

Beam #	Description	Code	Nos	Dia	Dim_A	Dim_B	Length	Quantity
14	B14-btm steel->st	4	2	25	5.365		5.665	43.636
14	B14-btm steel->cut	1	2	25	3.5		3.5	26.959
14	B14-top steel	4	3	20	5.365		5.604	41.439
14	B14-stirrups	8	25	8	0.4	0.18	1.352	13.33
15	B15-btm steel->st	4	2	25	5.365		5.665	43.636
15	B15-btm steel->cut	1	2	25	3.5		3.5	26.959
15	B15-top steel	4	3	20	5.365		5.604	41.439
15	B15-stirrups	8	25	8	0.4	0.18	1.352	13.33
1	B1-btm steel->st	3	3	32	10.7		10.924	206.796
1	B1-btm steel->cut	1	3	32	7		7	132.513
1	B1-top steel	1	3	25	4.3		4.3	49.683
1	B1-top steel	1	2	20	4.3		4.3	21.198
1	B1-stirrups	8	48	12	0.7	0.25	2.188	93.194
1	B1-lhs btm bar	5	2	16	3.25	0.202	3.404	10.739
1	B1-lhs top bar	5	4	20	3.25	0.39	3.58	35.297
1	B1-rhs btm bar	1	2	20	4.5		4.5	22.184
1	B1-rhs btm bar	1	2	16	4.5		4.5	14.197
1	B1-rhs top bar	1	3	32	4.5		4.5	85.186
1	B1-rhs top bar	1	2	25	4.5		4.5	34.662
2	B2-btm steel->st	1	3	25	5.9		5.9	68.169
2	B2-btm steel->cut	1	2	25	3		3	23.108
2	B2-top steel	1	5	25	1.691		1.691	32.563
2	B2-stirrups	8	14	8	0.7	0.25	2.092	11.55
2	B2-rhs top bar	1	2	32	4.5		4.5	56.791
2	B2-rhs top bar	1	3	20	4.5		4.5	33.276
2	B2-rhs btm bar	1	2	8	4.5		4.5	3.549
4	B4-btm steel->st	3	3	32	10.7		10.924	206.796
4	B4-btm steel->cut	1	3	25	7		7	80.879
4	B4-top steel	1	3	25	4.3		4.3	49.683

- **The BBS should be read in conjunction with " Bar Code " and " Std. Details ".**
  Buttons shown in Main Menu, Just Click to get Display.
  Note the unique style of creating BBS without the Bar Mark. This BBS is only for Beams.
  Now Double Click " Reinforcement Summary " Option. Following Graphics is displayed.

SUMMARY OF REINFORCEMENTS IN KG

6 MM Dia :	0
8 MM Dia :	4600.755
10 MM Dia :	1025.99
12 MM Dia :	913.389
16 MM Dia :	875.21
20 MM Dia :	1600.035
25 MM Dia :	1626.687
32 MM Dia :	2248.077

TOTAL REINFORCEMENT IN TONS = 12.89

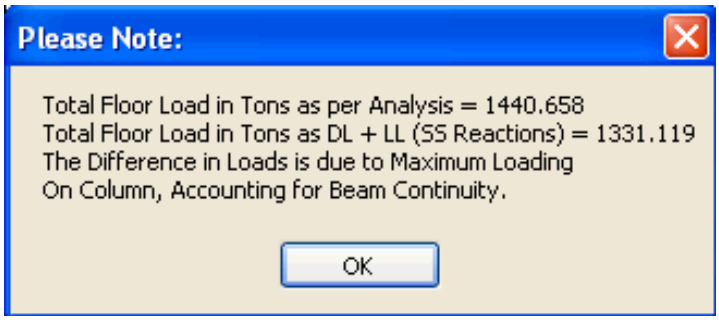
- **The MTO includes total of Beam and Slab steel Quantities. Beam steel quantities are taken from BBS and Slab quantities have been worked out approximately from Slab Schedule.**
  Now Double Click " Column Loads " Option from the Main Menu. Following Graphics is displayed.

COLUMN LOADS

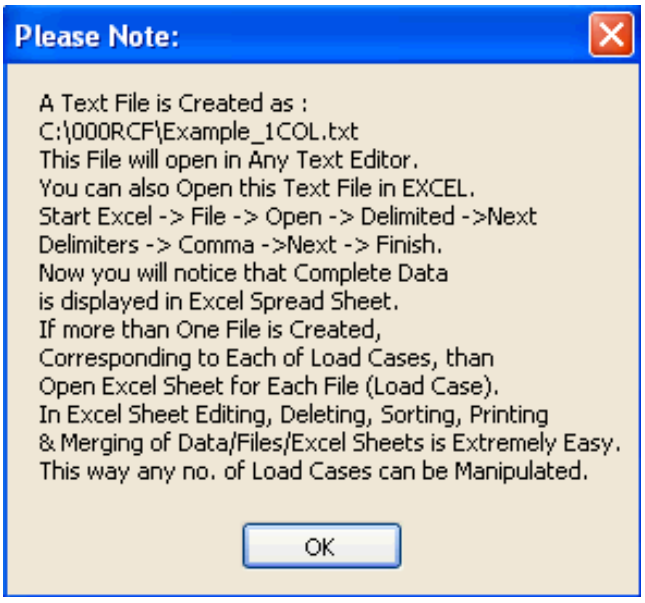
File Name : C:\000RCF\Example_1.rcf Date : 15 May 2008

Column No.	Length in MM	Width in MM	Height in M	Load in Tons
C1	600	300	3	34.713
C2	600	300	3	54.037
C3	600	300	3	44.538
C4	600	300	3	29.715
C5	600	300	3	35.781
C6	600	300	3	102.975
C7	600	300	3	77.393
C8	600	300	3	49.576
C9	600	300	3	28.39
C10	600	300	3	28.751
C11	600	300	3	75.836
C12	600	300	3	43.758
C13	600	300	3	44.794
C14	600	300	3	71.492
C15	600	300	3	47.033
C16	600	300	3	26.727
C17	600	300	3	21.378
C18	600	300	3	38.715
C19	600	300	3	34.241
C20	600	300	3	39.277
C21	600	300	3	24.36
C22	600	300	3	28.751
C23	600	300	3	77.889
C24	600	300	3	46.691
C25	600	300	3	47.217
C26	600	300	3	74.799
C27	600	300	3	41.256
C28	600	300	3	16.268
C29	600	300	3	30.09

- **The above Column Loads Graphics is self explanatory. Self Weight of Column is included.**
  When " OK " button is clicked following vital Statistical Check is displayed.
  Note that the Difference in Loads is due to Maximum Loading On Column, Accounting for Beam Continuity, i.e. Maximum of Simple Reaction & Continuous Reaction is taken for Column Design. 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.**

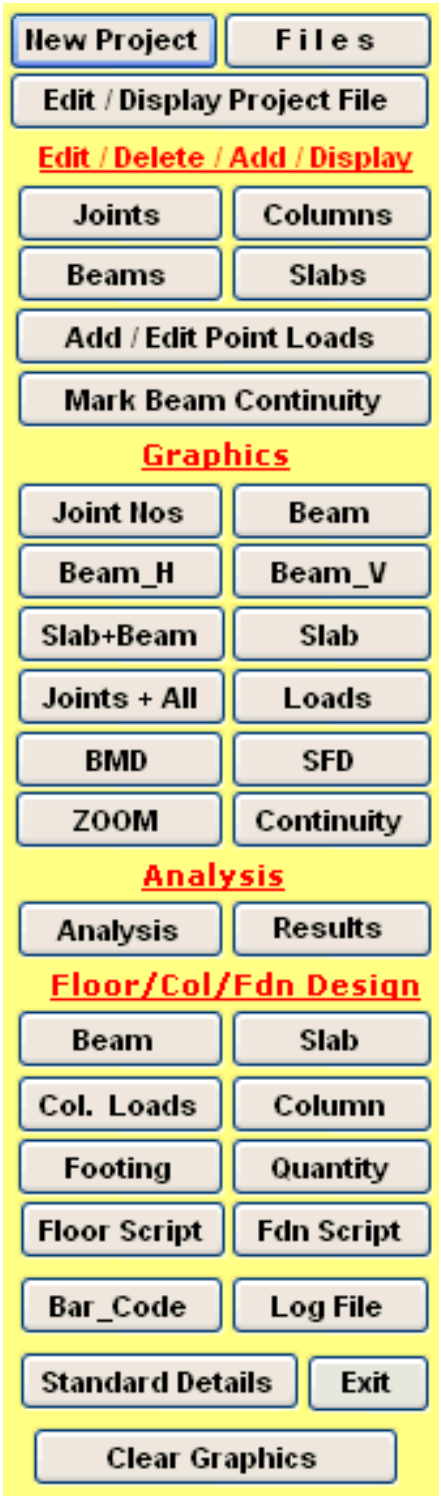


- **The above message describes how any number of Load Cases can be Run & Manipulated once the File is Exported to Excel Spread Sheet. Note the File Name Carefully.**
  Click " OK " button.
  Now we have come to the end of Step # 11.
  Let us proceed to Step No. 12.

STEP NO. 11 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 12 : BENDING MOMENT, SHEAR FORCE DIAGRAM LOAD DISPLAY AND FILES OPTION

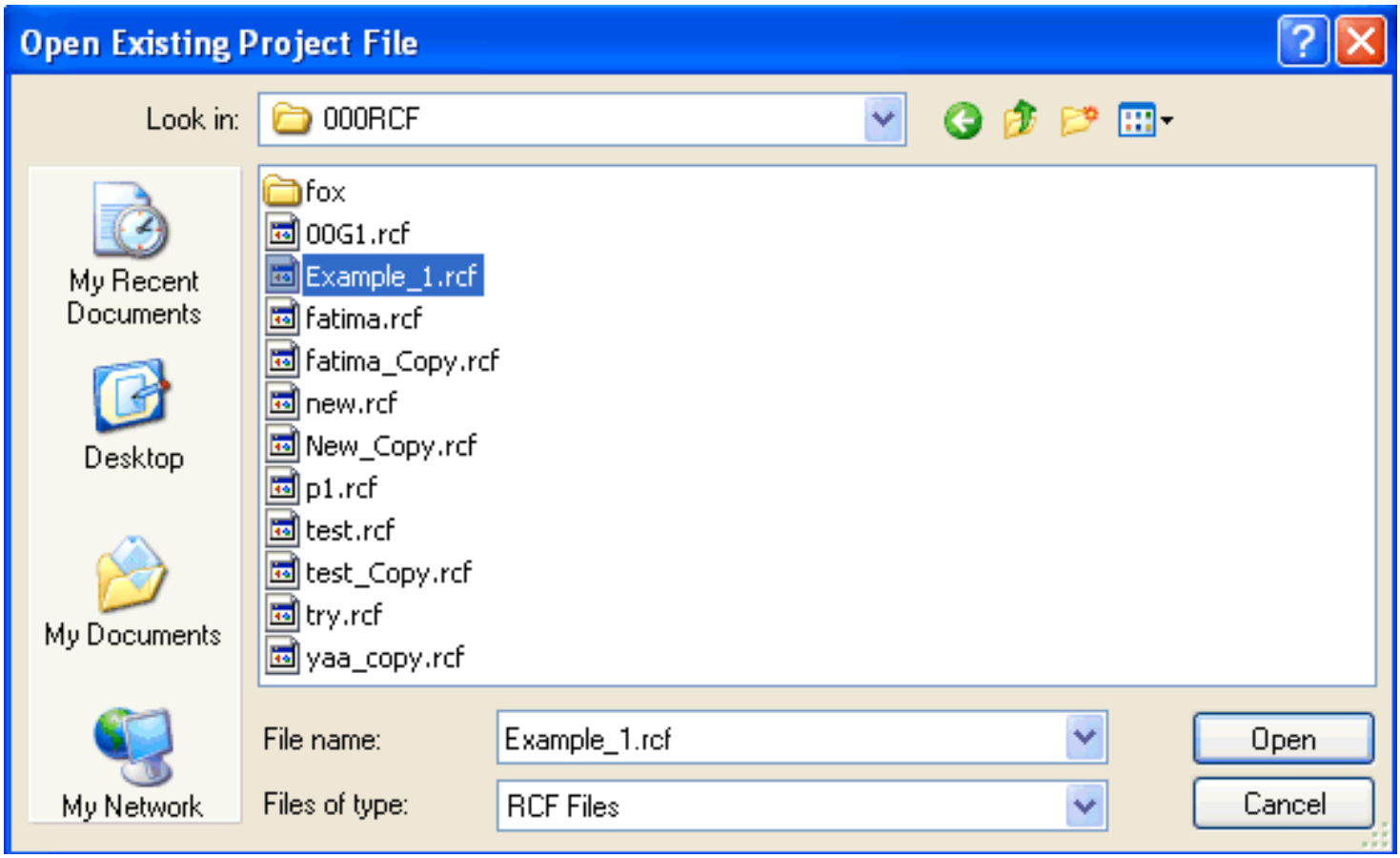


When Program starts, the Menu above is displayed. Under the Graphics Heading following options are displayed.

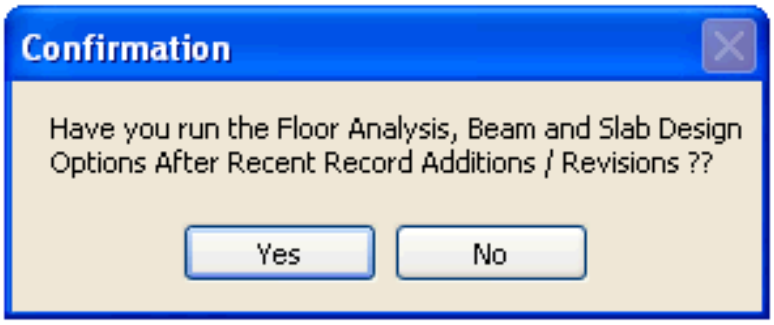
- **Joint Nos**
- **Beam**
- **Beam_H (Only Horizontal Beam # will be Displayed).**
- **Beam_V (Only Vertical Beam # will be Displayed).**
- **Slab + Beam (Beams, Slabs & Columns are displayed).**
- **Slab (Only Slabs & Columns are displayed).**
- **Joints + ALL (For Display of Joints, Columns, Beams & 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 & 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 " BMD " option.

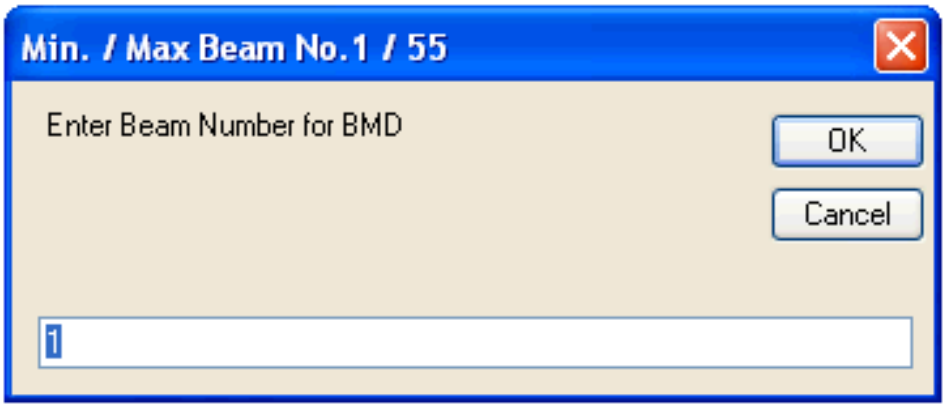
Following Graphics is displayed.



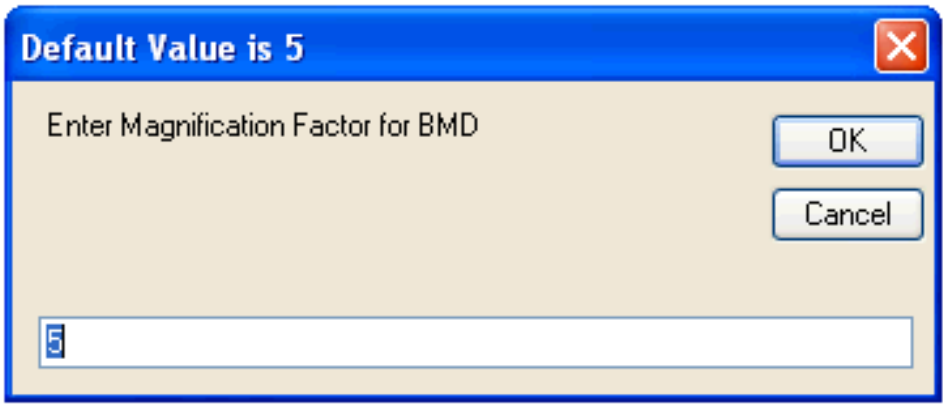
- **Now select " Example _1 File & Press Open Button. Following Warning is displayed.**



- **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 and Slab Designs are equally important as these options inform you about correctness of Beam & Slab 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 B1. 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.**
  Change MF = 1.0
  Following BMD is displayed.

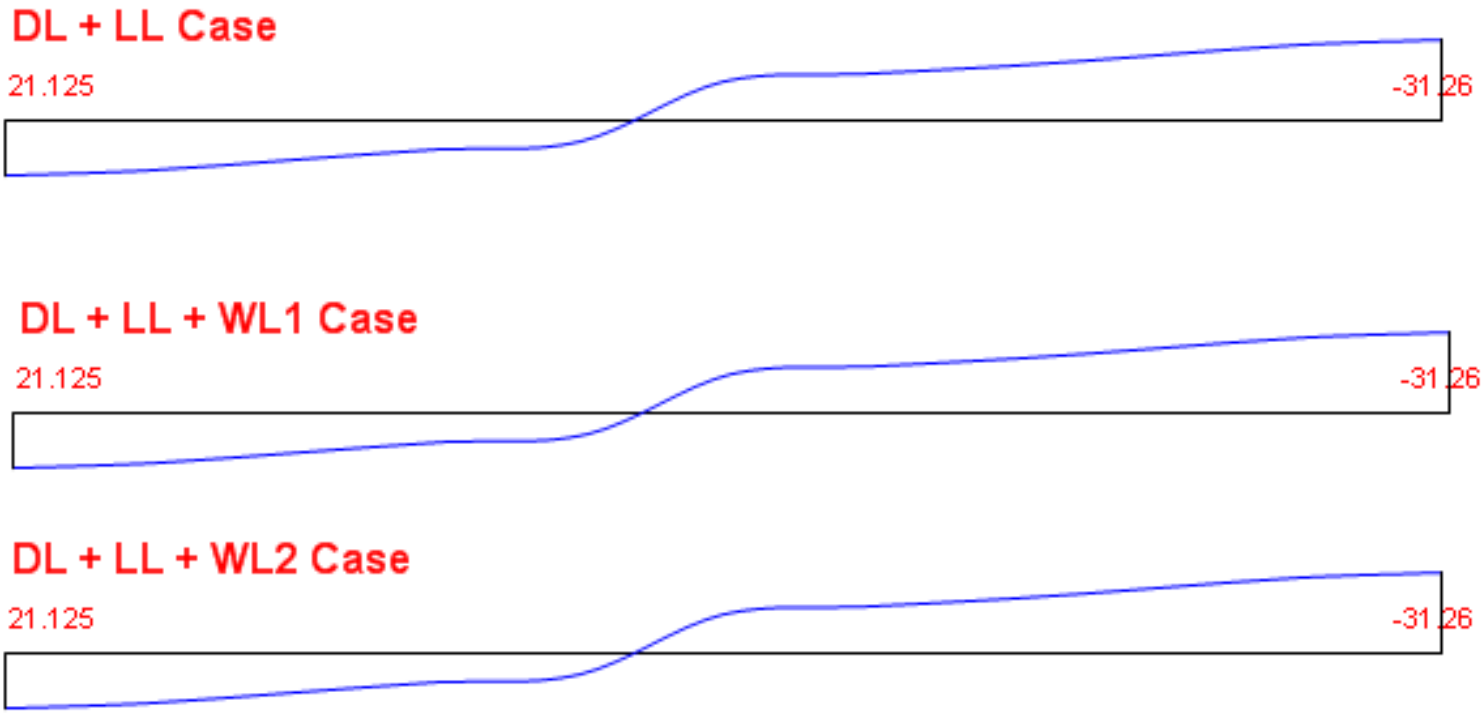
- **Note that BMD is drawn on Tension Side which reflects Deflected shape of Beam. BMD, SFD and Load Diagrams are Important from the point of Checking Results & Data Input. Any un-expected Diagram will reflect Data Error in the form of :**

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

  The 3 Cases are displayed simultaneously. Since we have not given any Externally applied End Moments, all the displays are same. The " Next " button is very useful as it can help you to display continuously the required BMD for a specified Beam.

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

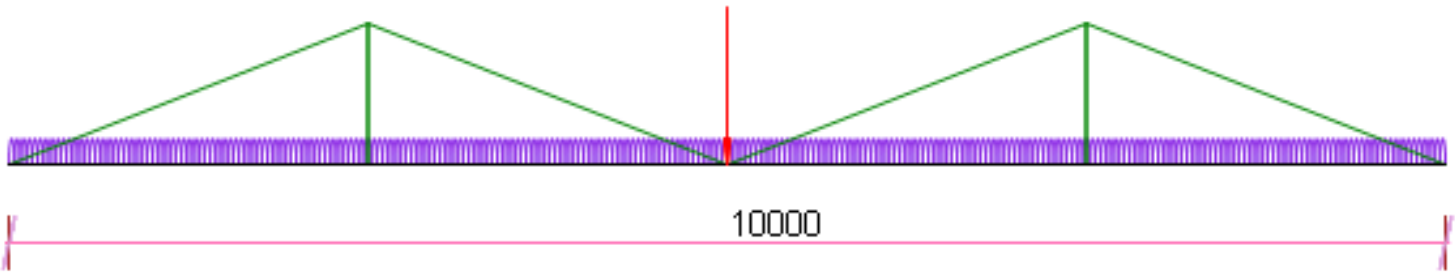
  SFD is displayed as under. MF = 1.0



Print Next **Shear Force Diagram**
Beam # : B1

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

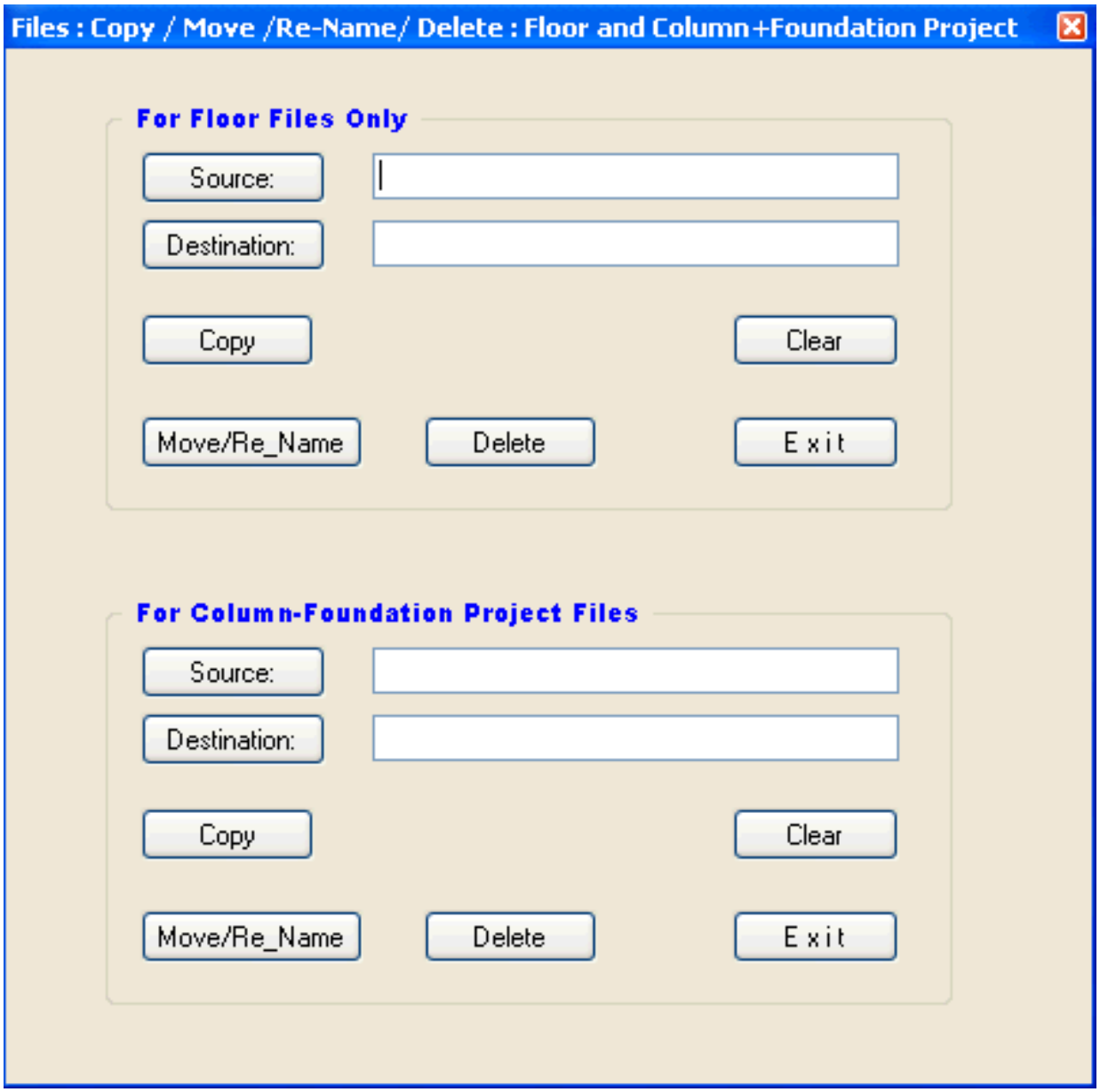
Load Diagram is displayed as under. MF = 5



UDL in t/m = 1.83
Point Load in t = 10 @ dist. of 5 m
Near Int. in t/m = 0 @ dist. of 0 m : Far Int. in t/m = 2.725 @ dist. of 2.5 m
Near Int. in t/m = 2.725 @ dist. of 2.5 m : Far Int. in t/m = 0 @ dist. of 5 m
Near Int. in t/m = 0 @ dist. of 5 m : Far Int. in t/m = 2.725 @ dist. of 7.5 m
Near Int. in t/m = 2.725 @ dist. of 7.5 m : Far Int. in t/m = 0 @ dist. of 10 m
Point Load in t = 10.46 @ dist. of 5 m

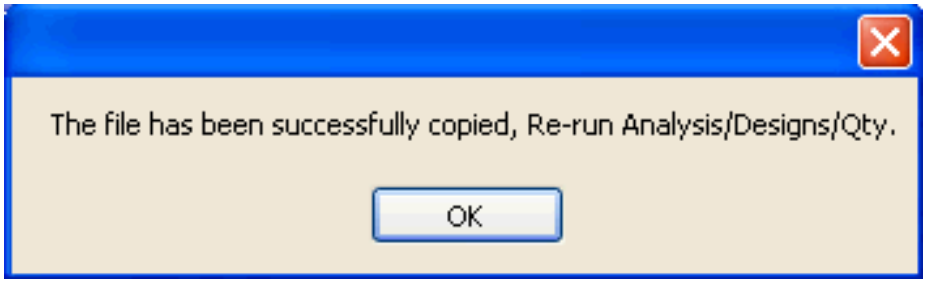
Print Next **Display of Loads on Beams**
Beam # : B1

- **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 beam B37 on B1. 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-Foundation 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 " .RCF ", while Column-Foundation File Extension is " .DAT ".

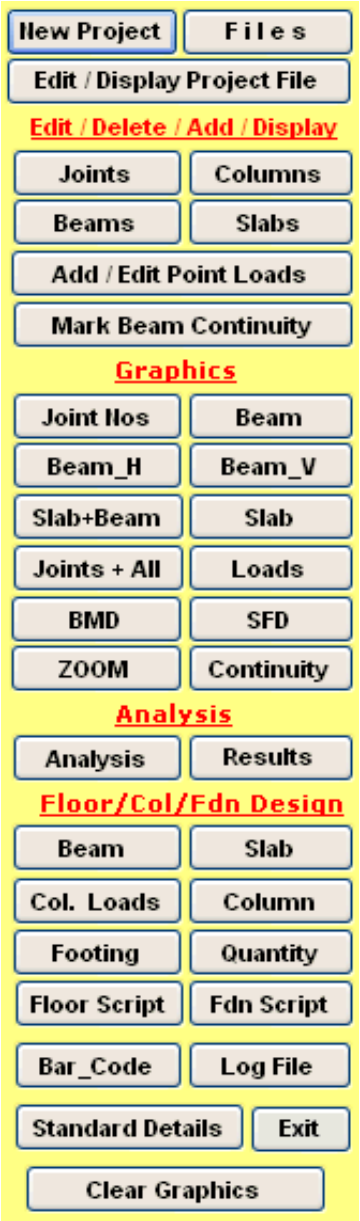
The Column-Foundation File menu is similar to Floor File Menu, only difference is File extension.
Hence Use " For Column-Foundation Project File " option to Copy, Delete & Move / Re-Name
Column and Foundation Files.

Now we have come to the end of Step # 12.

STEP NO. 12 IS OVER.

LEARN RCF STEP BY STEP

STEP 13 : CREATION OF FLOOR AND FOUNDATION PLAN IN AUTOCAD

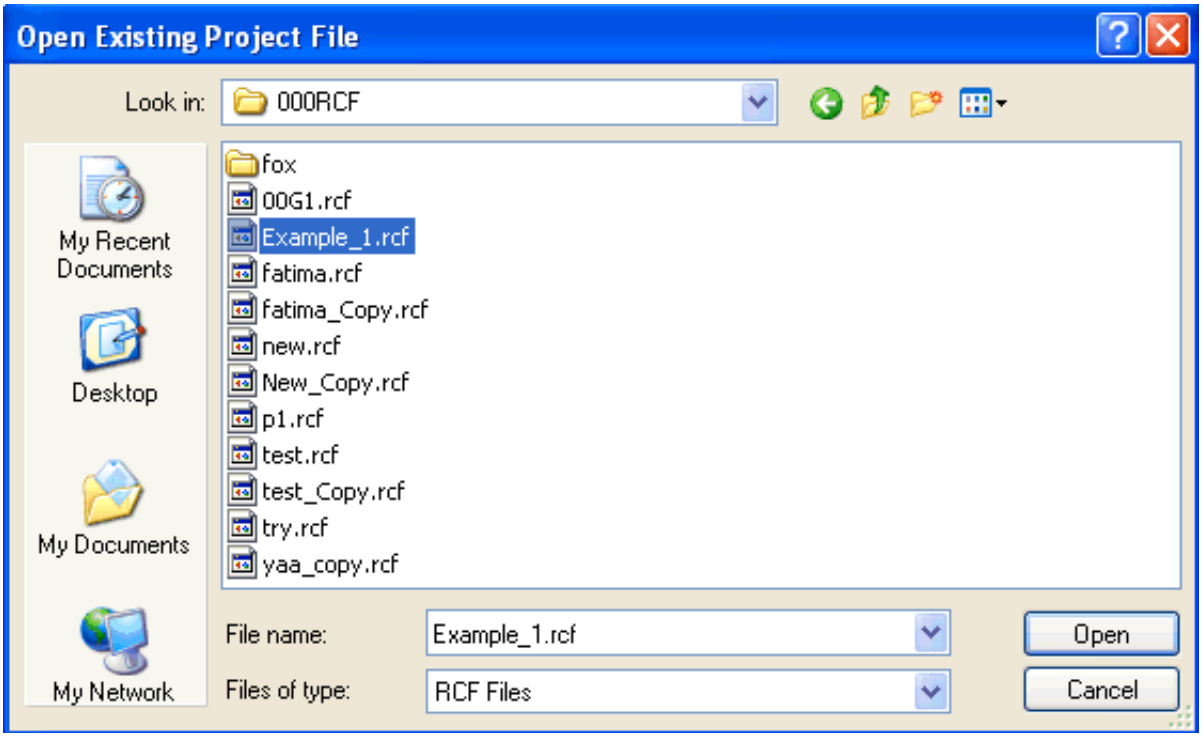


When The Program starts, the above Menu is displayed.

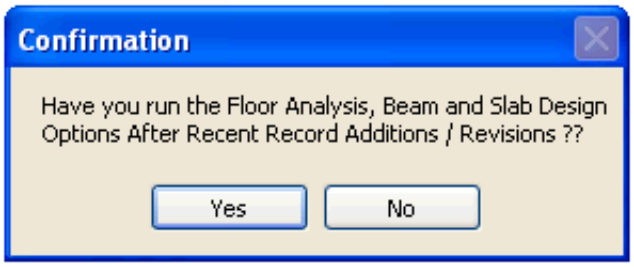
➤ Under the **Floor/Col/Fdn Design** heading following options are displayed.

- Beam
- Slab
- Column Loads
- Column
- Footing
- Quantity
- Floor Script
- Fdn Script

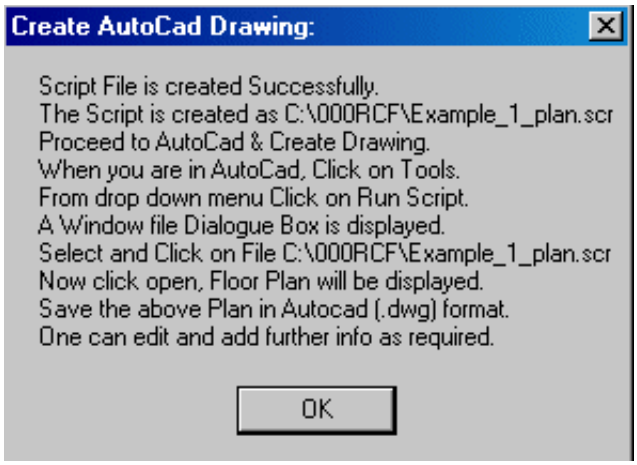
In Order to create an AutoCAD drawing, a script file has to be created first. To create the script file, click on script Option. A window dialogue box appears . Click on the required file and click on open.



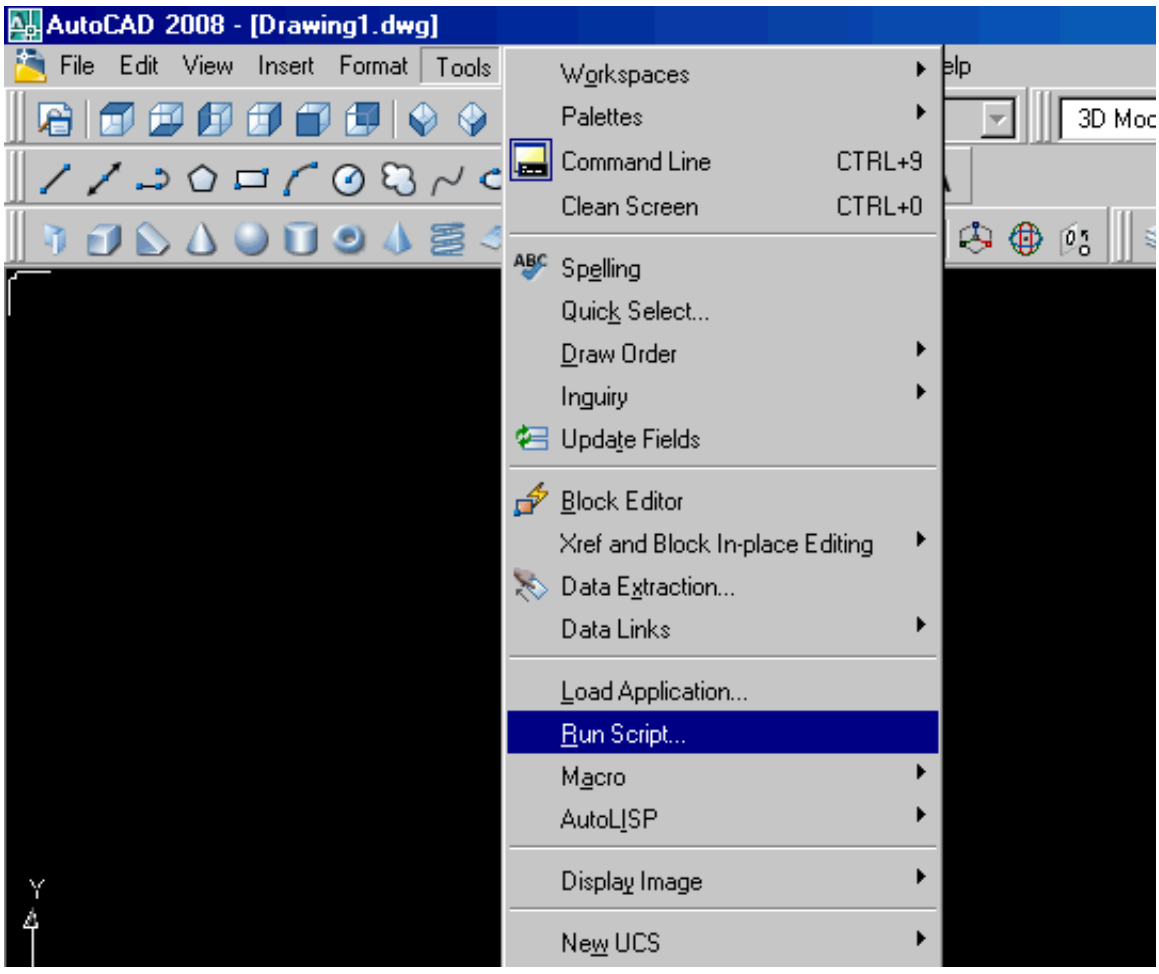
➤ Following graphics is displayed.
Click on Yes if Floor Analysis and Beam Design Options are performed.



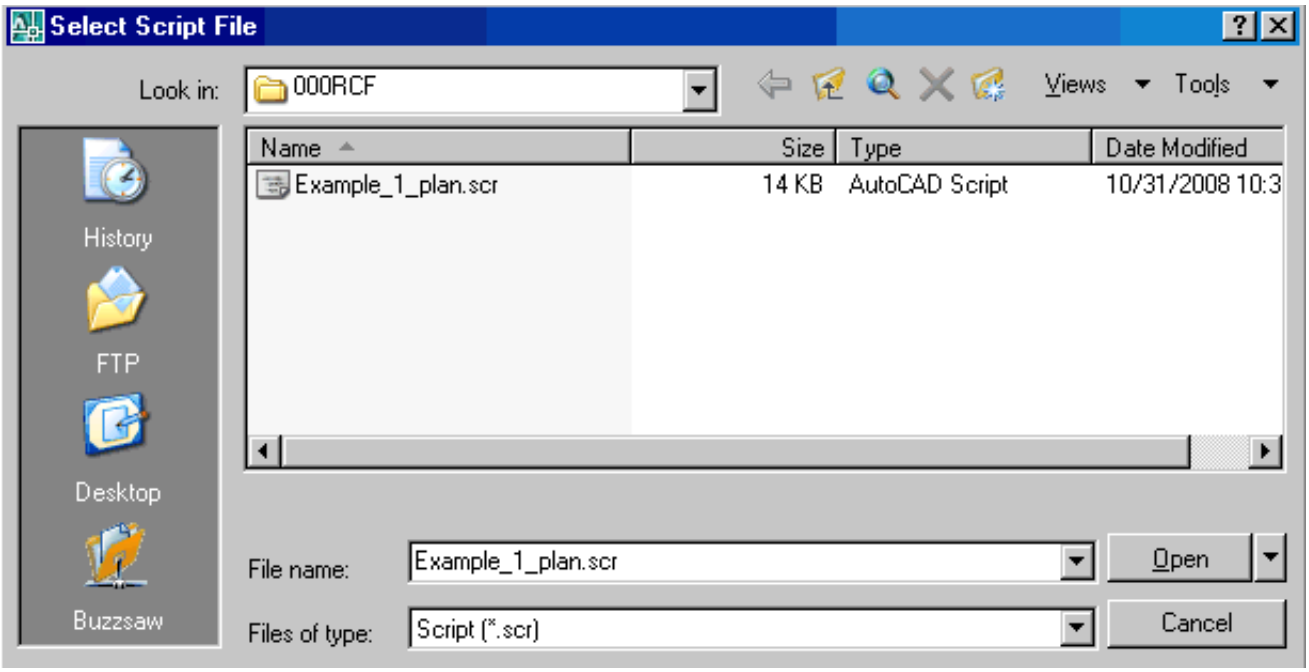
Once Yes is clicked, following graphics is displayed.
➤ The script file is created as Example_1_plan.scr. Note that "_plan" is added to file name and that .scr stands for script file and not screen saver file.
Now click on OK and Exit from the Program.



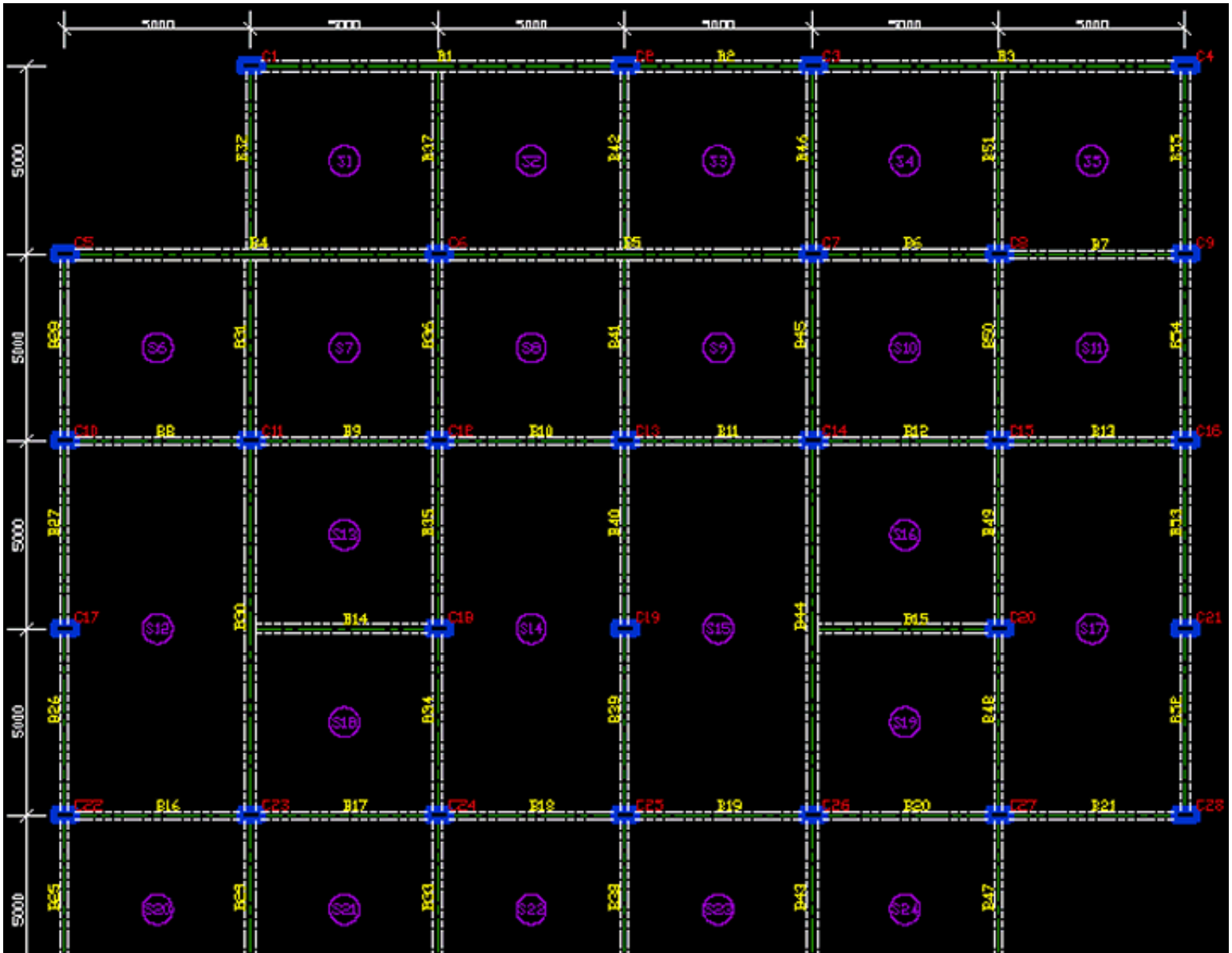
➤ Now Start AutoCAD.
In AutoCAD click on Tools. From the drop down menu click on Run Script.



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

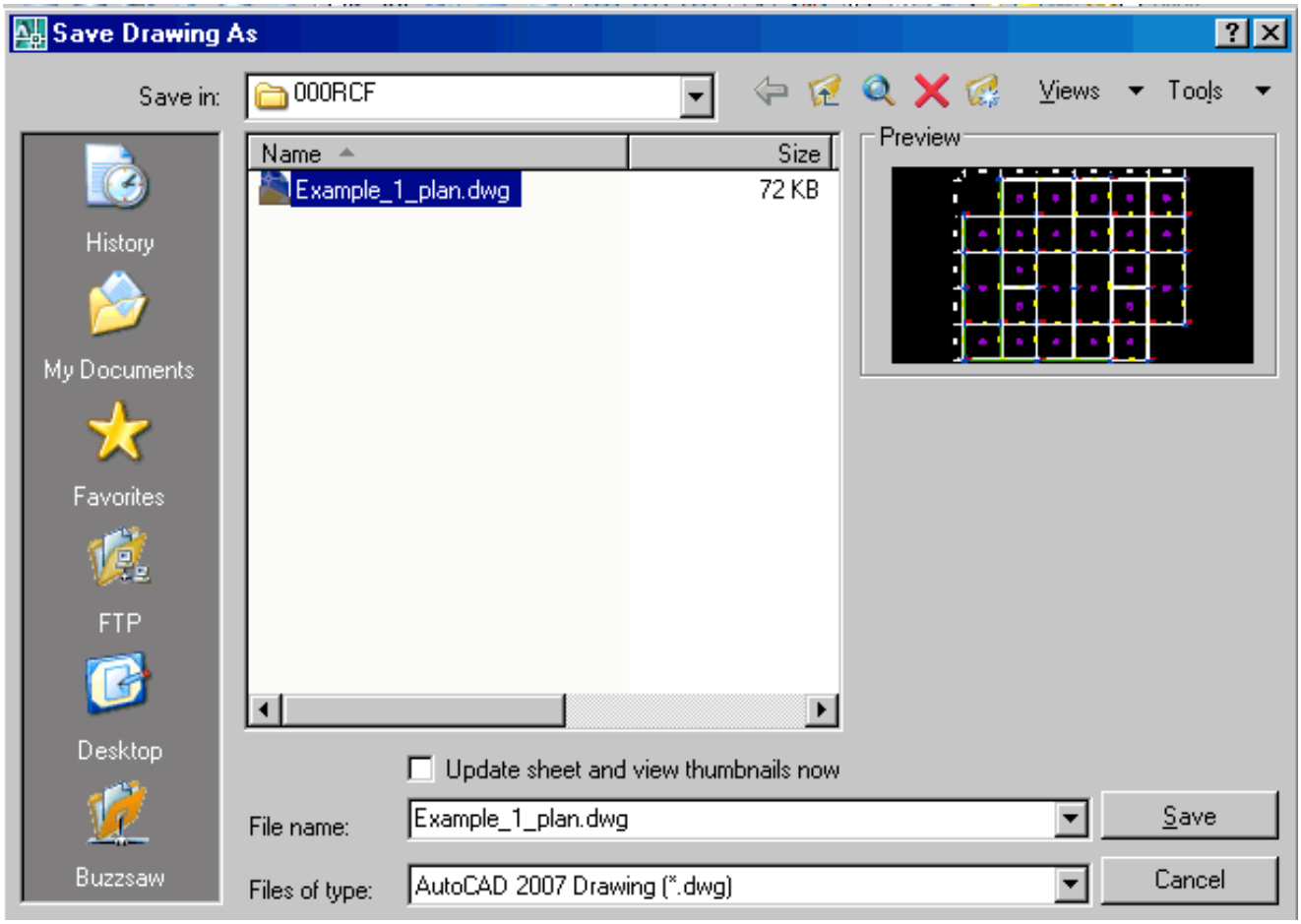


Please note that the above drawing is Editable in AutoCAD.

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

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.



➤ The Procedure for creation of Foundation Plan in AutoCAD is exactly similar to what is described above. Just Click " Fdn Script " button & follow the above procedure.

STEP 13 IS OVER

Now lets have a look on creation of Beam Schedule in the next Step....

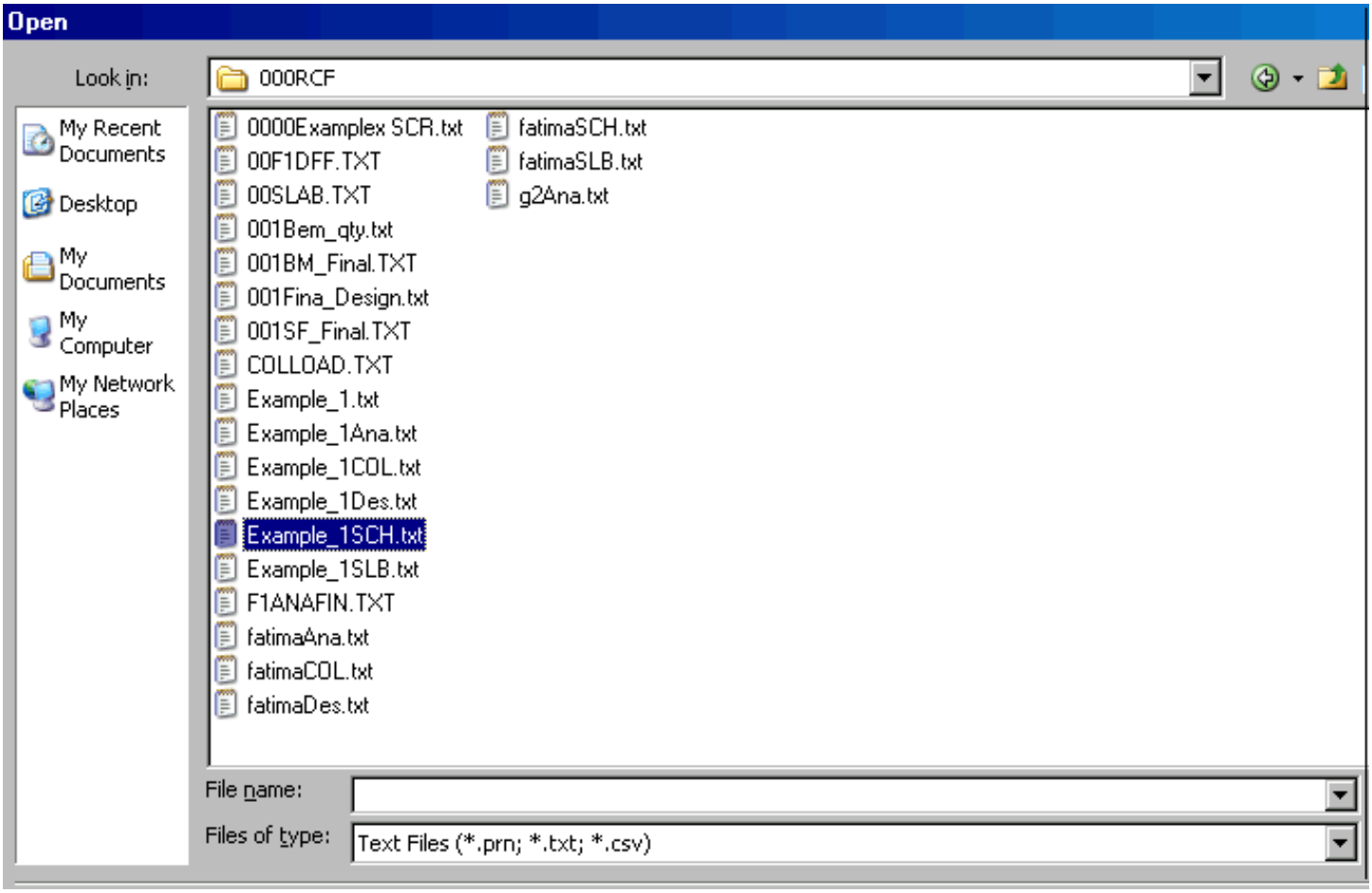
LEARN RCF STEP BY STEP

STEP 14: CREATION OF BEAM / COLUMN / FOUNDATION SCHEDULE IN AUTOCAD

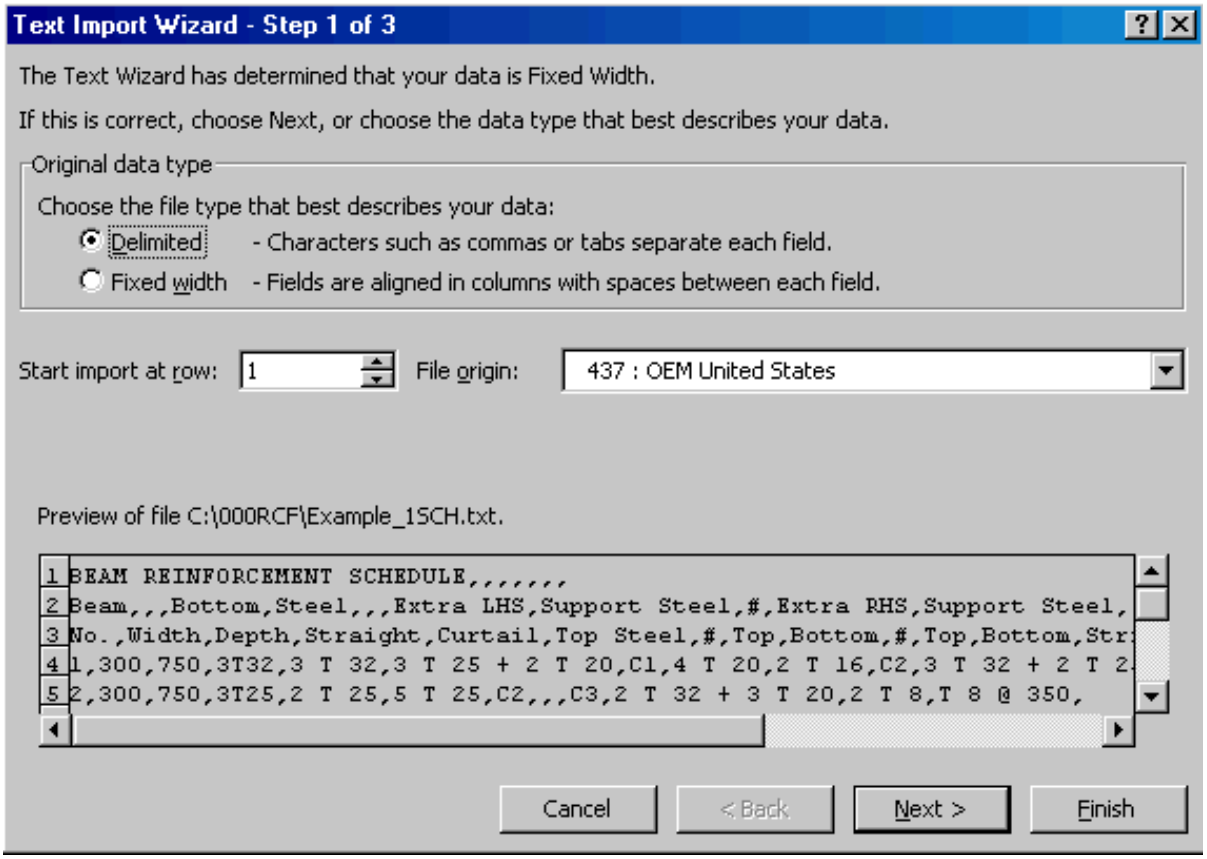
Creation of Beam Schedule in AutoCAD requires going through few steps of Excel And AutoCAD.
Let us have a look.....

- **When you run the Beam Design Option as illustrated in Step No 11, following Graphics is displayed. We will explain this message in details.**

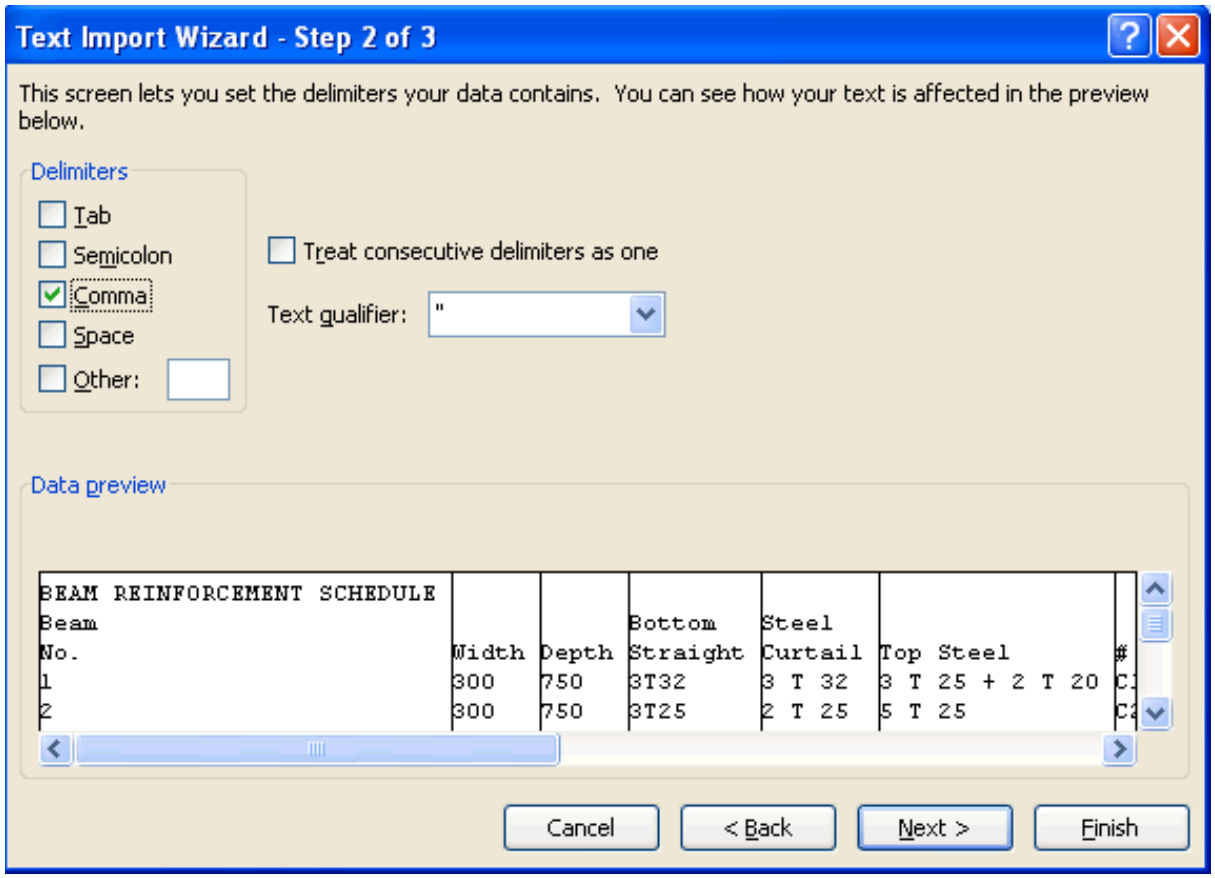
- **Start Microsoft Excel . Click On Open. Following Graphics is Displayed.**



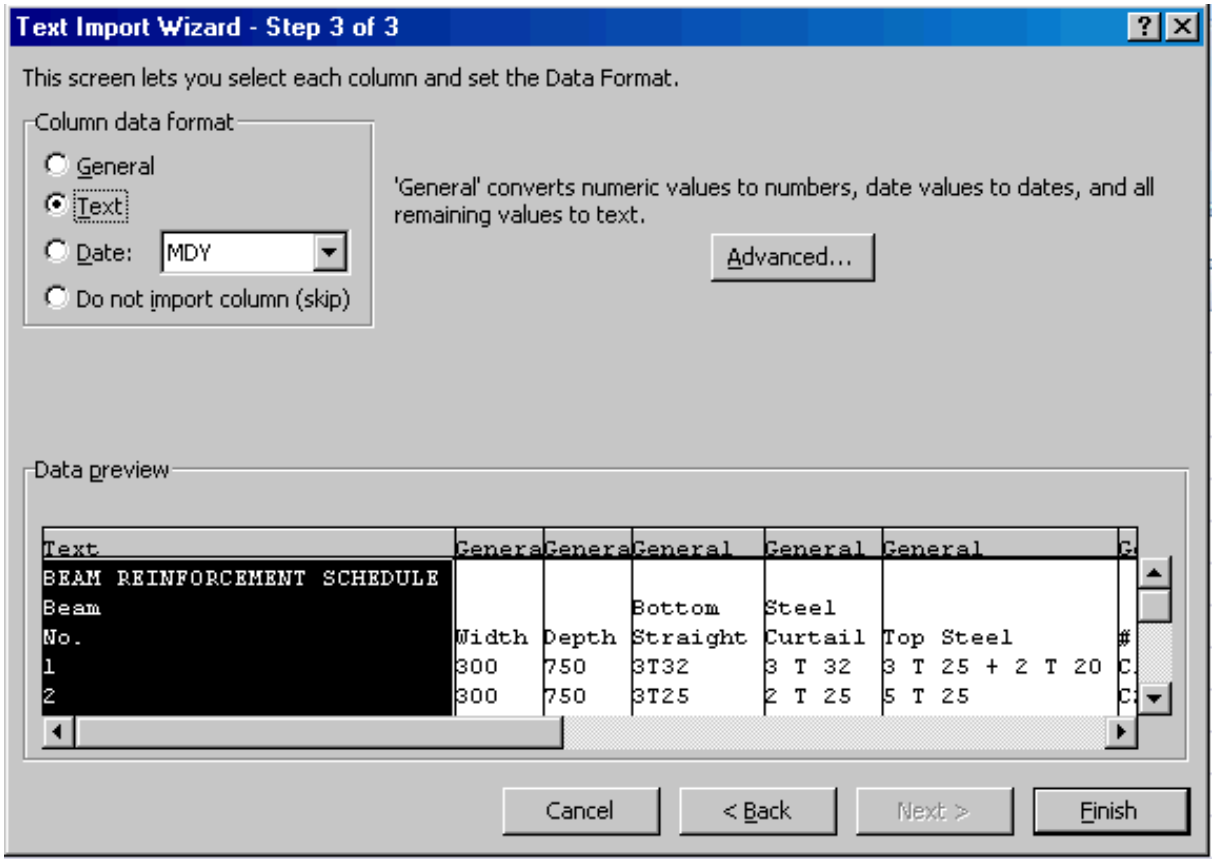
Click on Example_1SCH.txt.
As you can see, the above file is in text format.
In the following steps we will save the file in Excel format.
Once Example1_SCH.txt is clicked, following graphics is displayed.



➤ **As shown Above choose Delimited as your Option. Click On Next.**
You will see the following dialogue box appear.



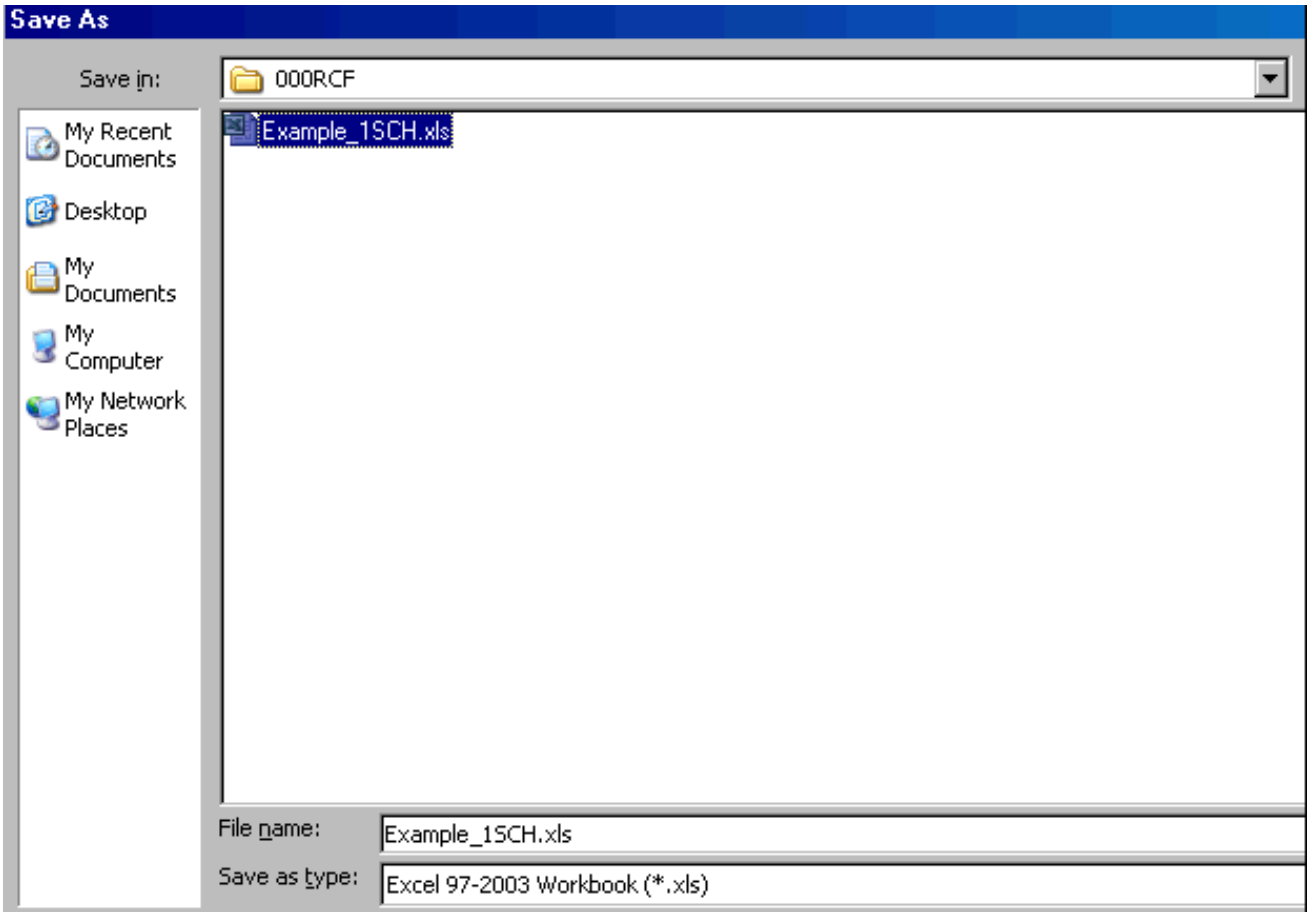
As shown Above choose Comma as Delimiter. Click On Next.
Following graphic is displayed.



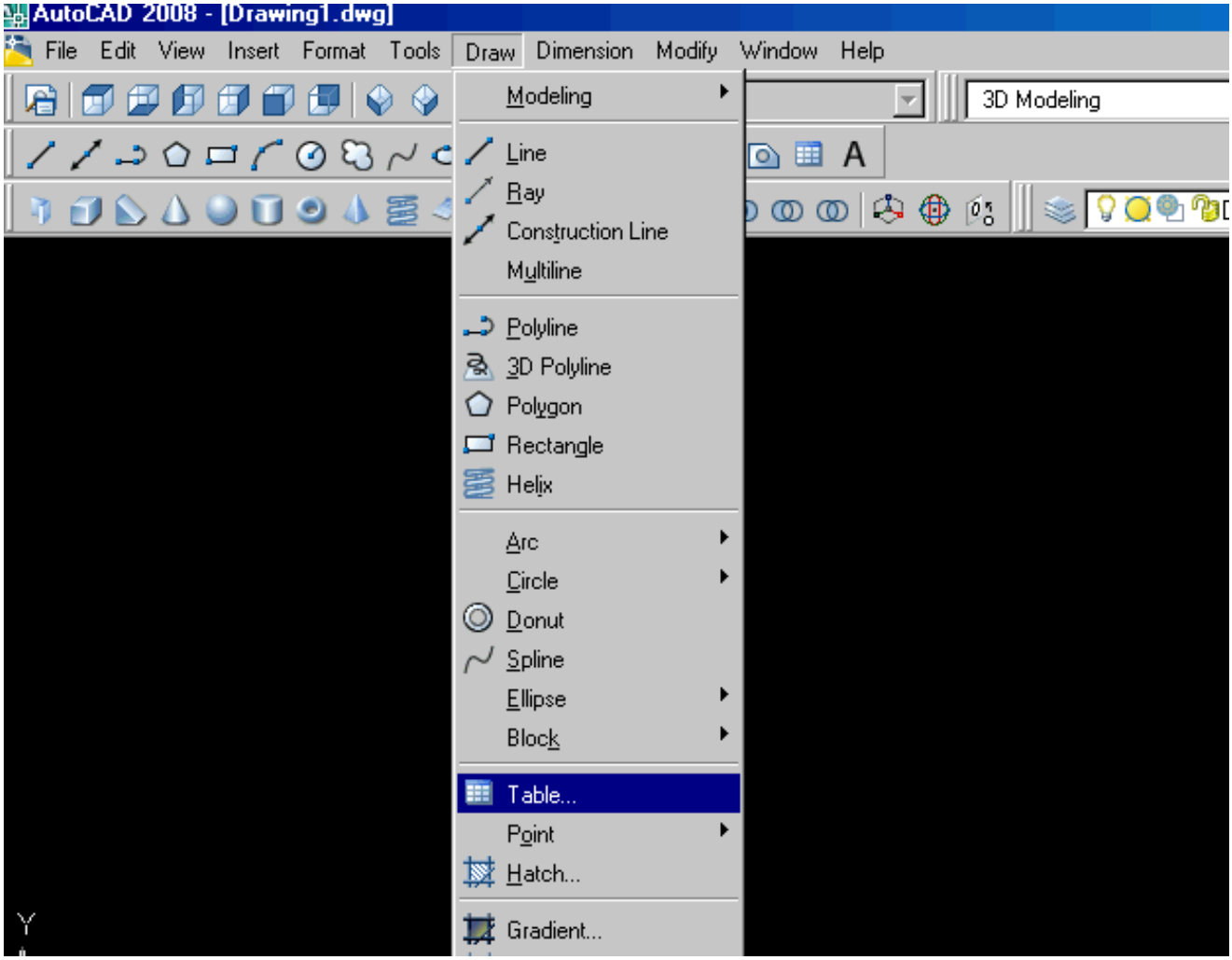
As shown above click on Text and then click on Finish.
Here you will see that Beam schedule appears in Excel . Following is a part display.
Now You can make any number of changes you want within Excel, like changing fonts, alignment of text, Column Width etc..

BEAM REINFORCEMENT SCHEDULE									
Beam			Bottom	Steel			Extra LHS	Support Steel	#
No.	Width	Depth	Straight	Curtail	Top Steel	#	Top	Bottom	#
1	300	750	3T32	3 T 32	3 T 25 + 2 T 20	C1	4 T 20	2 T 16	C2
2	300	750	3T25	2 T 25	5 T 25	C2			C3
3	300	750	3T32	2 T 25	2 T 20 + 2 T 16	C3			C4
4	300	750	3T32	3 T 25	3 T 25	C5	2 T 20 + 2 T 16	2 T 12 + 2 T 10	C6
5	300	750	4T25	2 T 32	4 T 20	C6			C7
6	300	750	2T16	1 T 12	2 T 16 + 1 T 20	C7			C8
7	230	450	2T20	2 T 20	2 T 10 + 1 T 8	C8			C9
8	230	450	2T20	2 T 20	2 T 10 + 1 T 8	C10	2 T 12 + 2 T 10	2 T 10 + 1 T 12	C11
9	230	450	2T16	2 T 10	2 T 8	C11			C12
10	230	450	2T16	1 T 20	2 T 8	C12			C13
11	230	450	2T16	1 T 20	2 T 8	C13			C14
12	230	450	2T16	2 T 10	2 T 8	C14			C15
13	230	450	2T20	2 T 20	2 T 10 + 1 T 8	C15			C16

After making all the required changes, don't forget to save the table in Excel i.e. (in .xls) Format . After having saved the file, you are done with Excel part , Exit from Excel and proceed to AutoCAD.



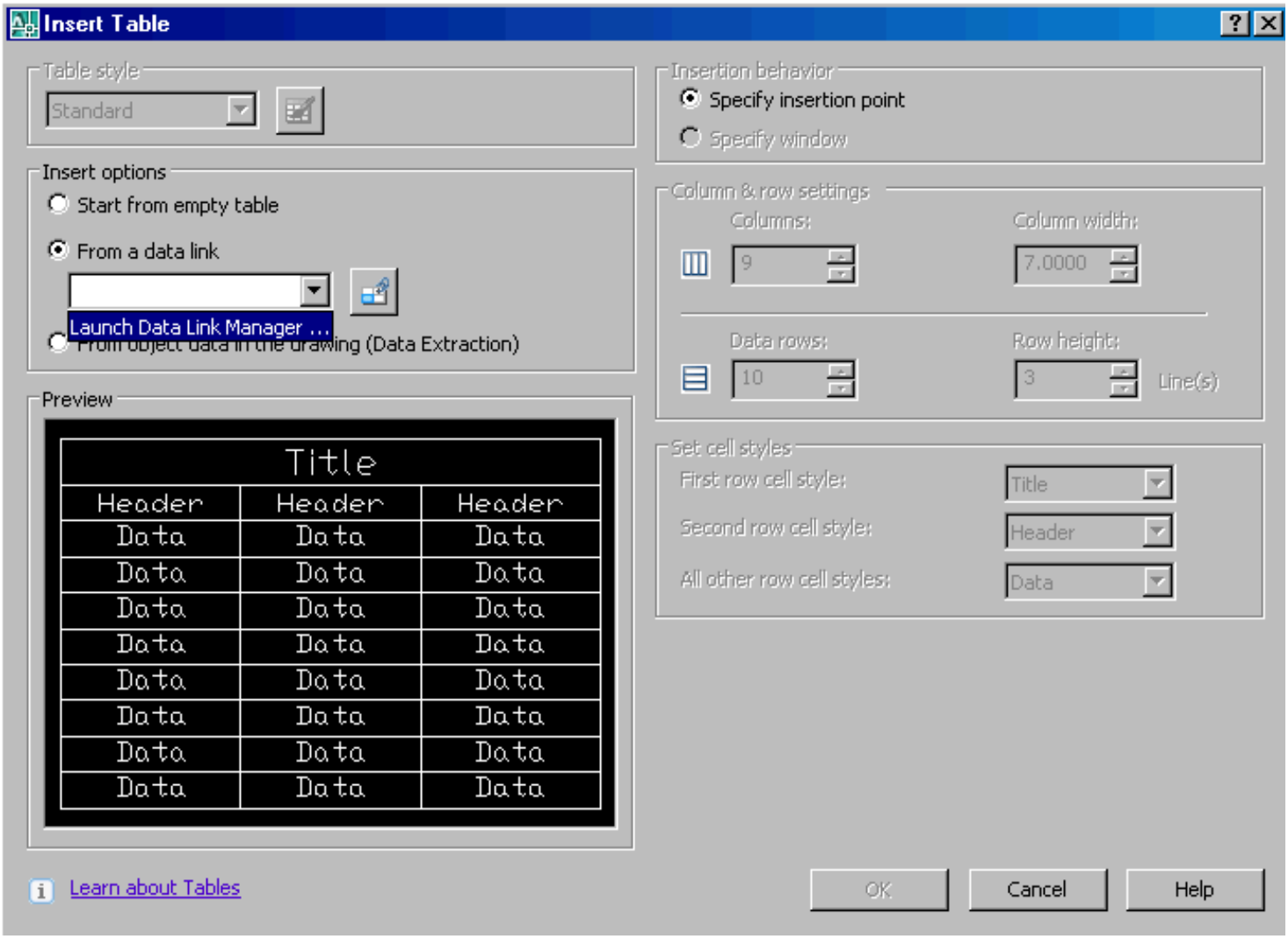
Start AutoCAD. Click on DRAW . From the drop down menu click on Table a shown below.



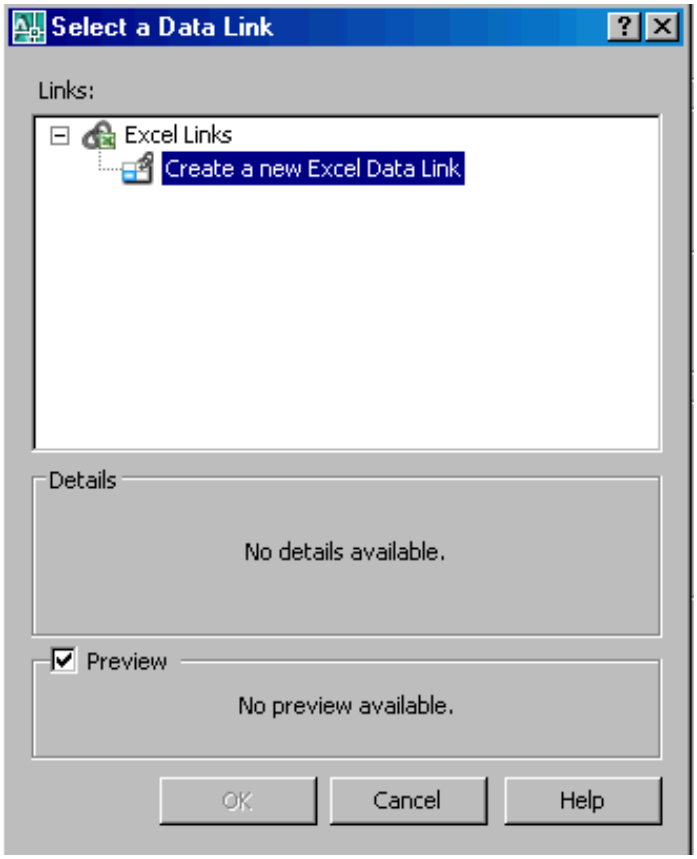
A dialogue box will appear.

As show below click on From a data link .

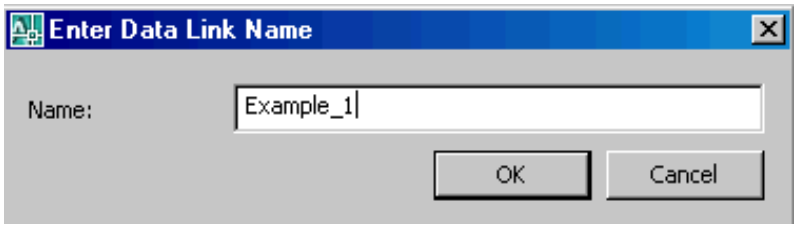
From the drop down menu click on Launch Data Link Manager.



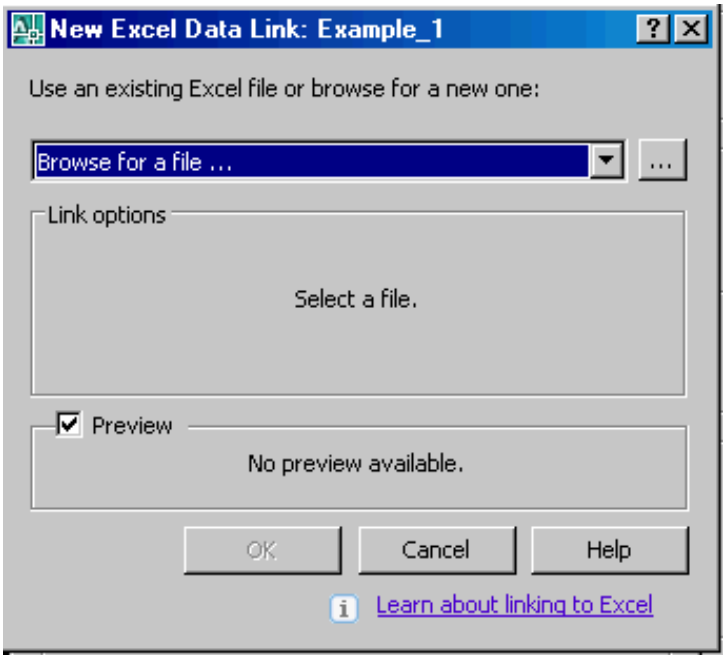
Following graphic is displayed. Click on Create a new Excel Data Link.



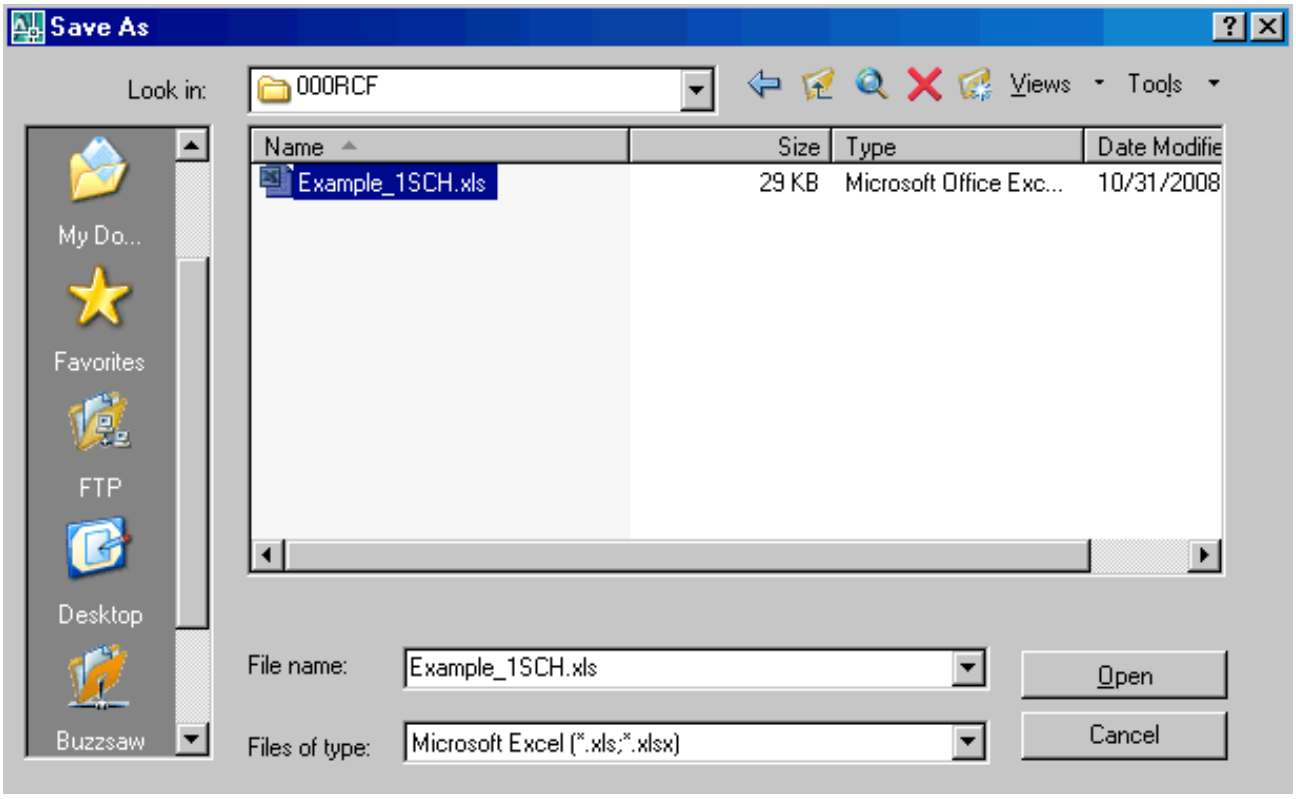
A dialogue box appears asking you to Enter a name.
You can Enter any name for e.g. Example_1.
Click on OK.



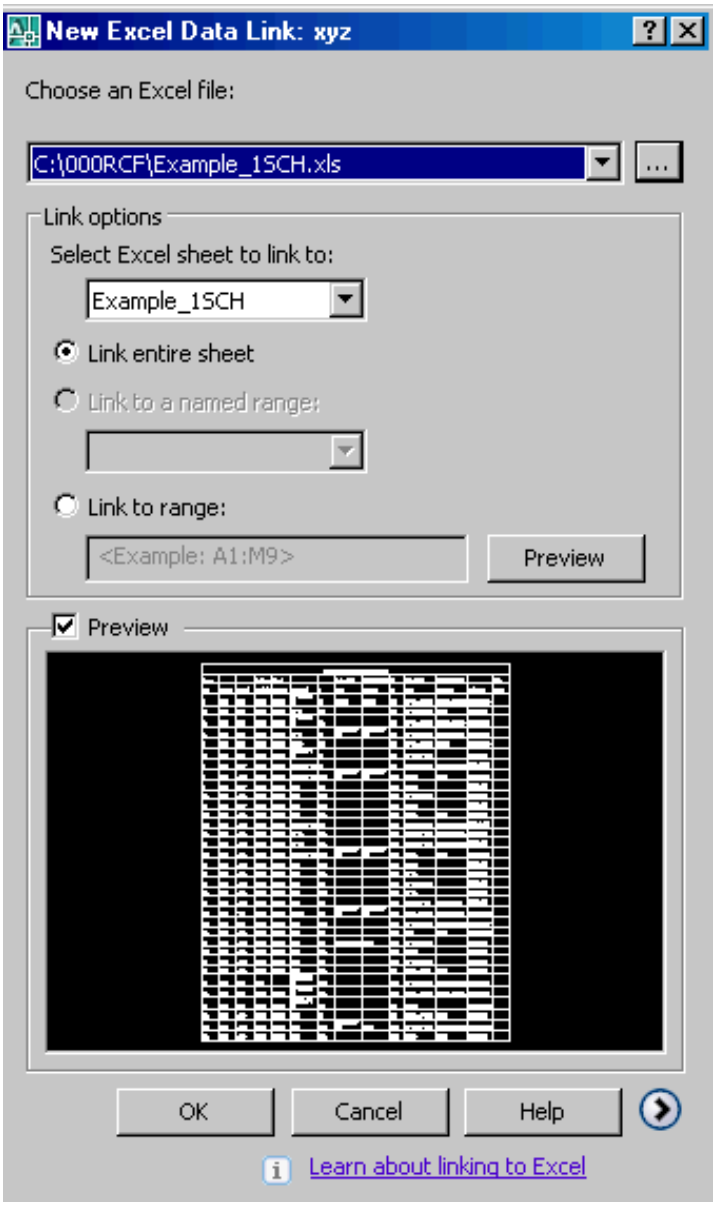
Following Graphics will appear. Click on Browse for a file.



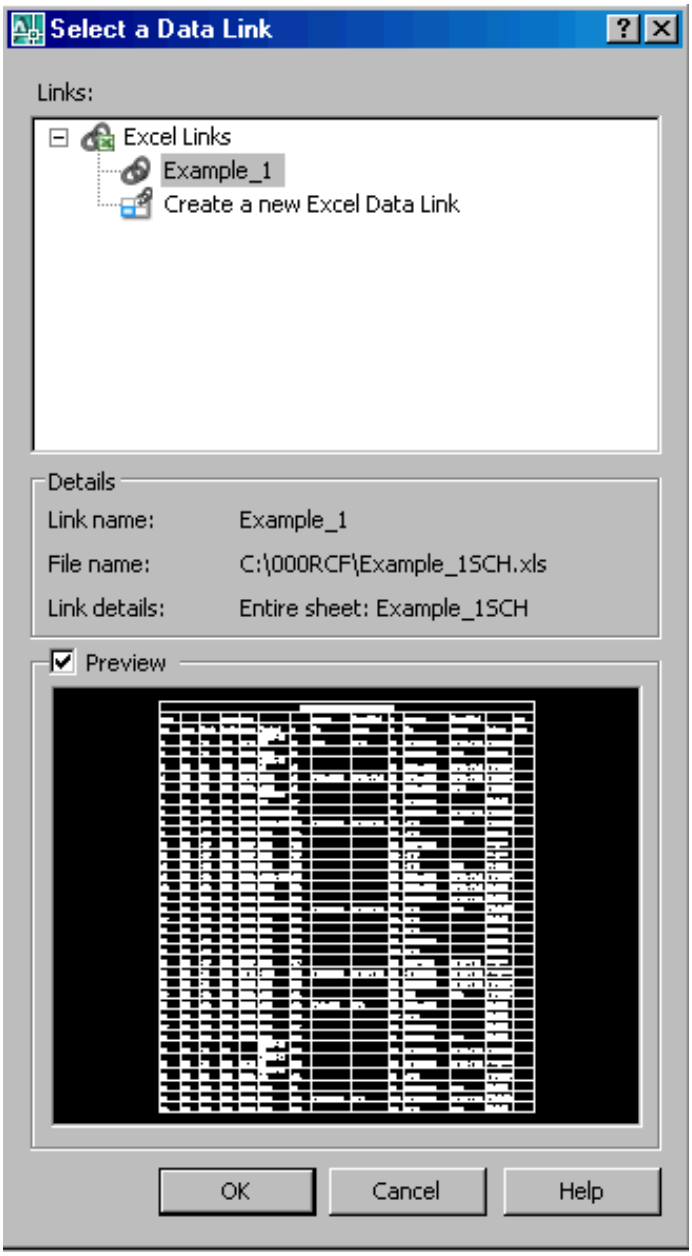
➤ **A Window dialogue box appears. Click on the required file (i.e. the file that we saved previously in Excel format) and click on Open.**



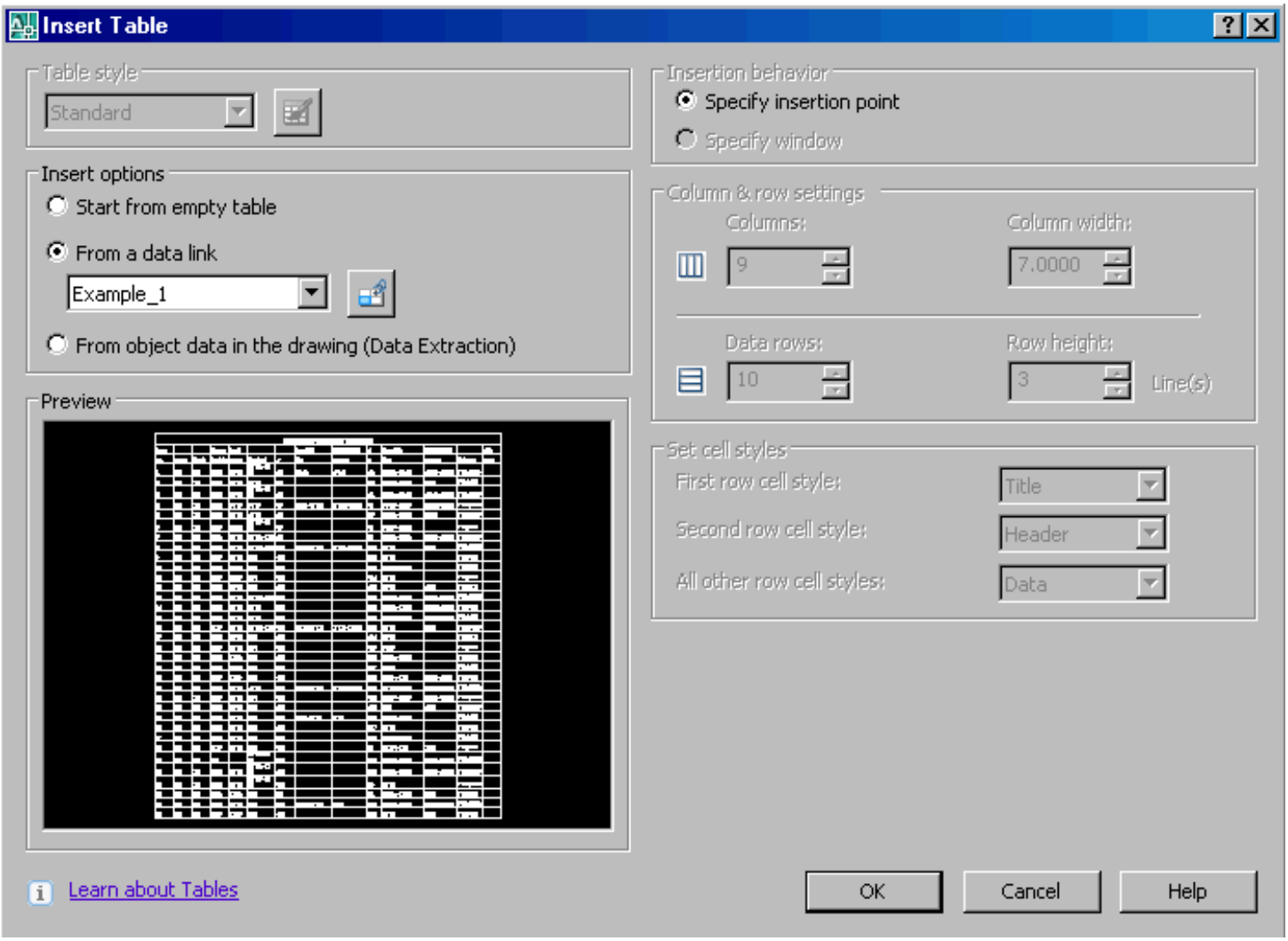
➤ **Following Graphics appears showing the preview of the table in AutoCAD. Click on OK.**



Again a dialogue box appears showing the created link and preview of the table. Click on OK.



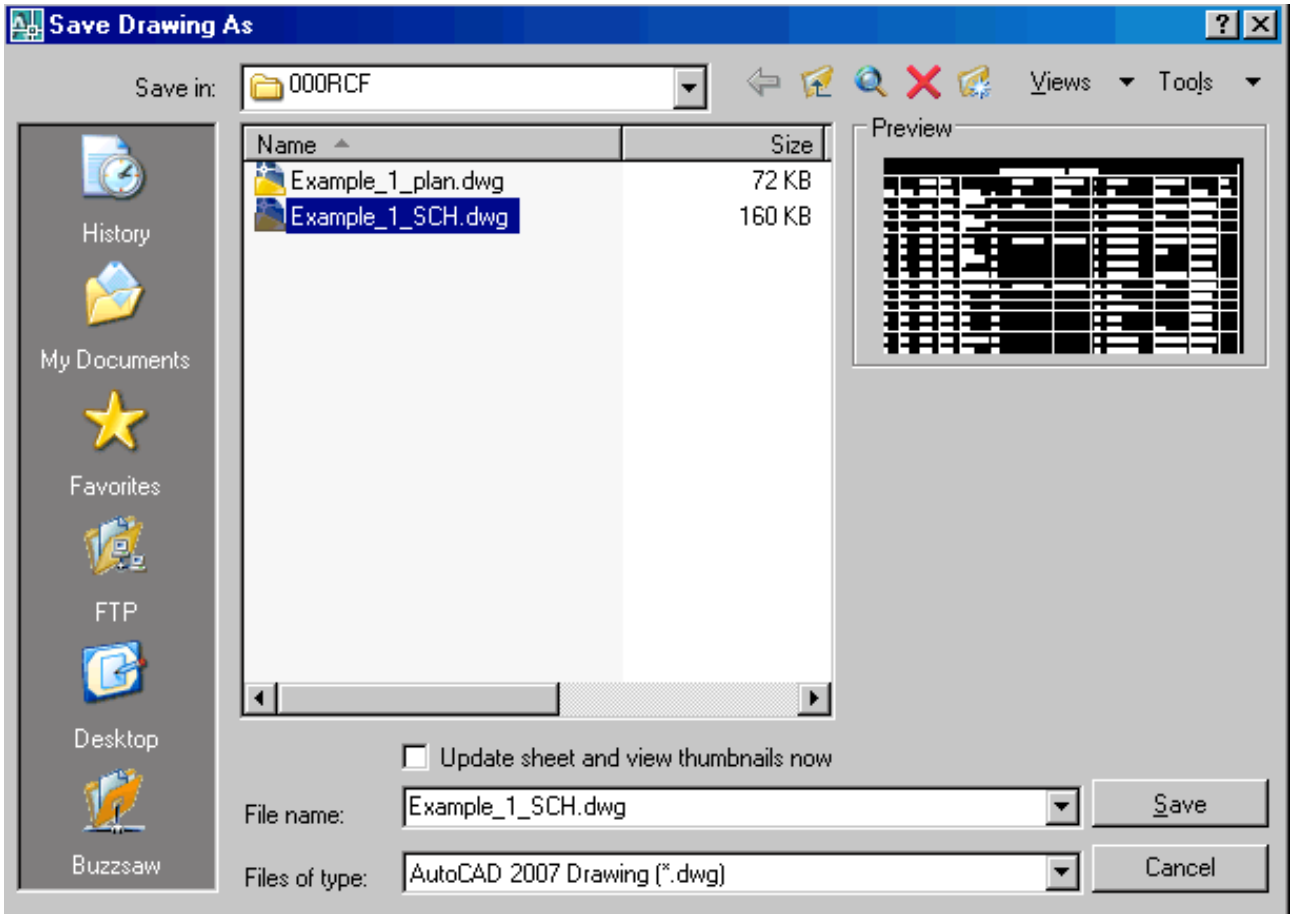
Another dialogue box appears . Click on OK.



Now specify an Insertion Point, after which the Beam Schedule will be displayed in the form of AutoCAD drawing. Following is a part display.

BEAM REINFORCEMENT SCHEDULE								
Beam			Bottom	Steel			Extra LHS	Support Steel
No.	Width	Depth	Straight	Curtail	Top Steel	#	Top	Bottom
1	300	750	3T32	3 T 32	3 T 25 + 2 T 20	C1	4 T 20	2 T 16
2	300	750	3T25	2 T 25	5 T 25	C2		
3	300	750	3T32	2 T 25	2 T 20 + 2 T 16	C3		
4	300	750	3T32	3 T 25	3 T 25	C5	2 T 20 + 2 T 16	2 T 12 + 2 T 10
5	300	750	4T25	2 T 32	4 T 20	C6		
6	300	750	2T16	1 T 12	2 T 16 + 1 T 20	C7		
7	230	450	2T20	2 T 20	2 T 10 + 1 T 8	C8		
8	230	450	2T20	2 T 20	2 T 10 + 1 T 8	C10	2 T 12 + 2 T 10	2 T 10 + 1 T 12
9	230	450	2T16	2 T 10	2 T 8	C11		

Finally, don't forget to save the above drawing in AutoCAD (i.e. . dwg) format.



The Procedure for creation of Column and Foundation Schedule in AutoCAD is exactly similar to what is described above.

STEP 14 IS OVER

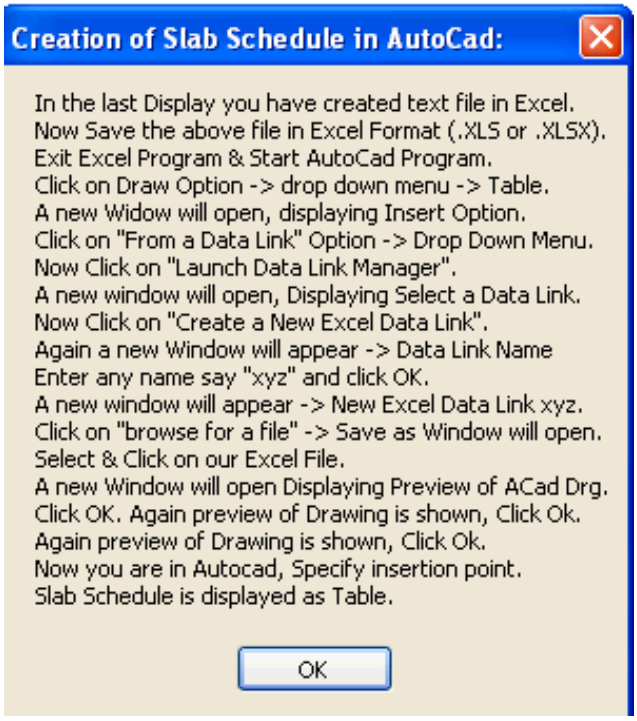
Now lets have a look on creation of Slab Schedule in the next Step....

LEARN RCF STEP BY STEP

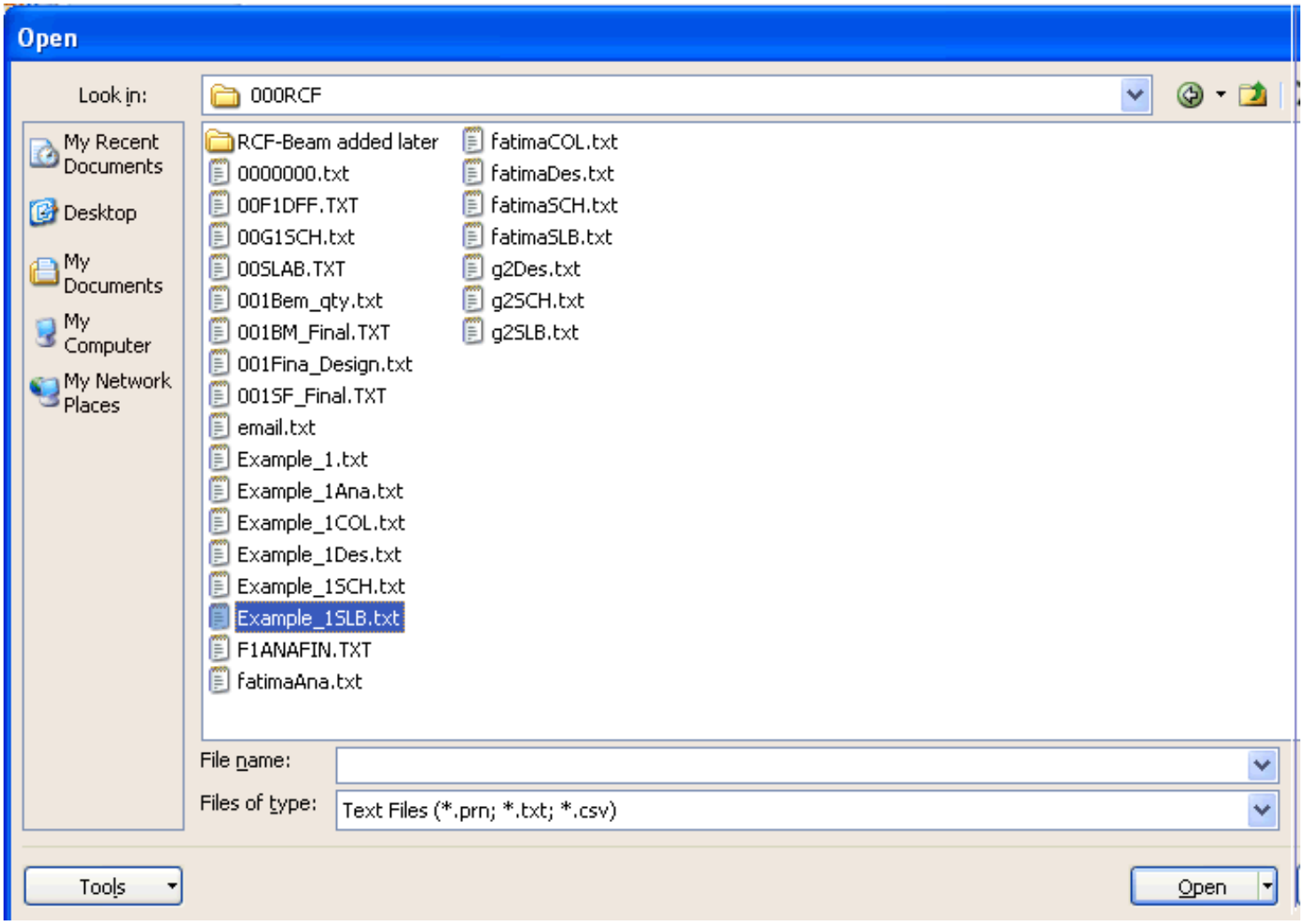
STEP 15: CREATION OF SLAB SCHEDULE IN AUTOCAD

Creation of Slab Schedule in AutoCAD is almost same as that of beam schedule with just a few changes here and there ....

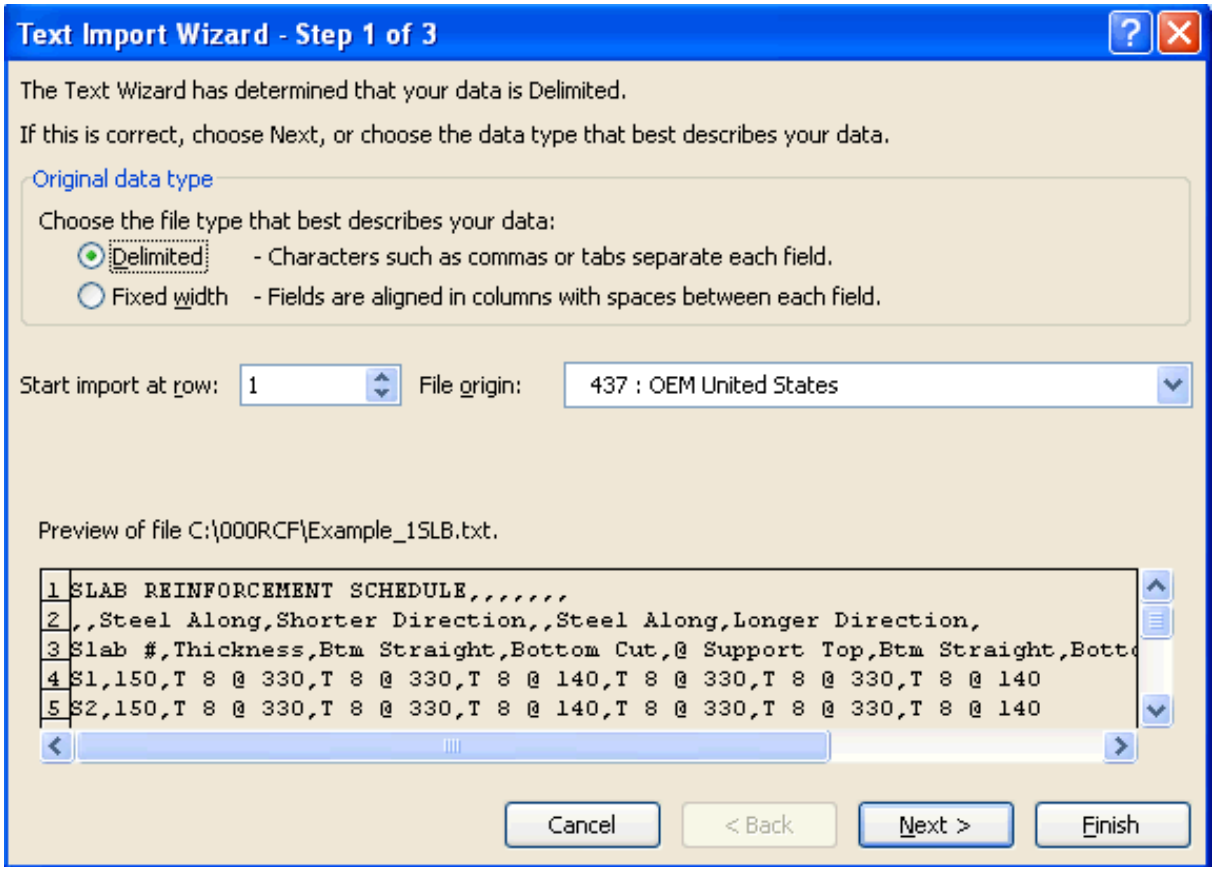
- **When you run the Slab Design Option as illustrated in Step No 11 , following Graphics is displayed. We will explain this message in detail.**



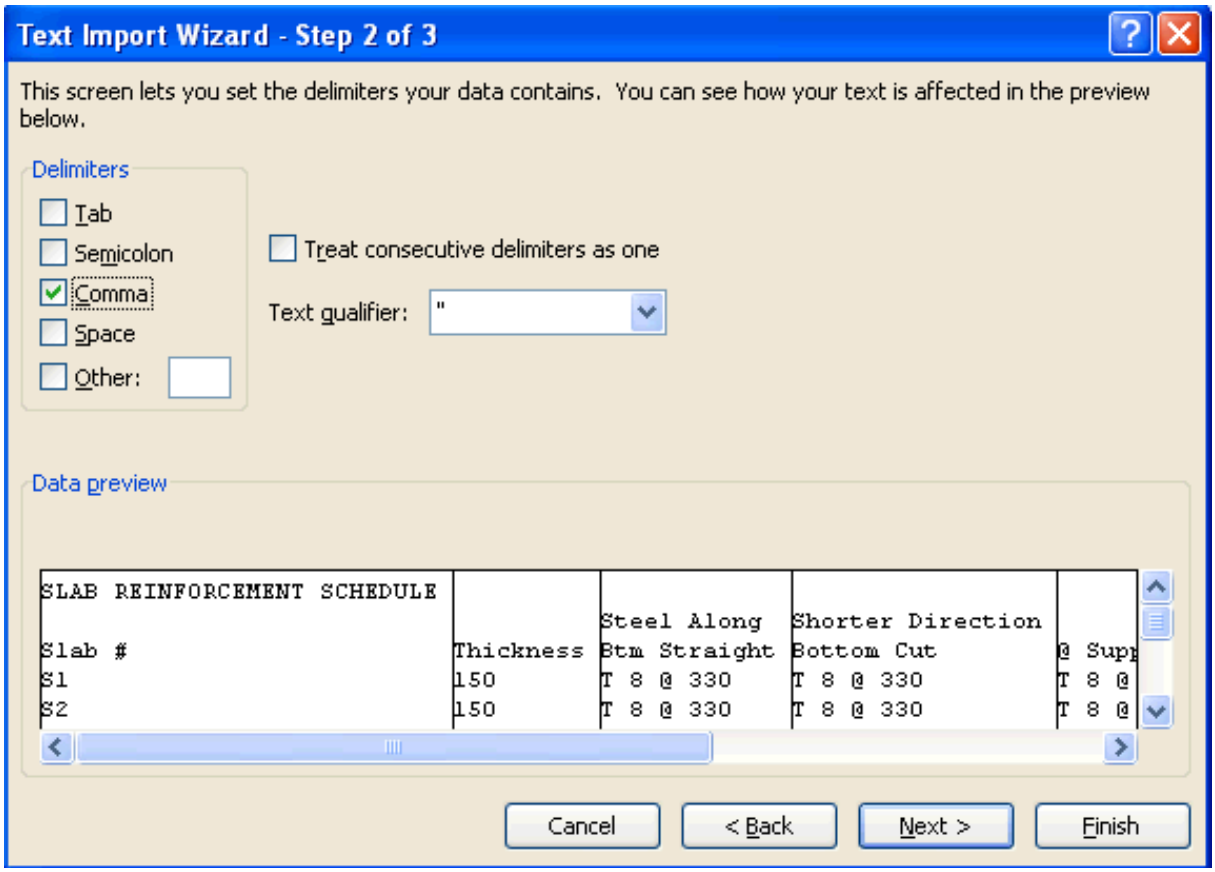
- **Start Microsoft Excel . Click On Open. Following Graphics is Displayed.**



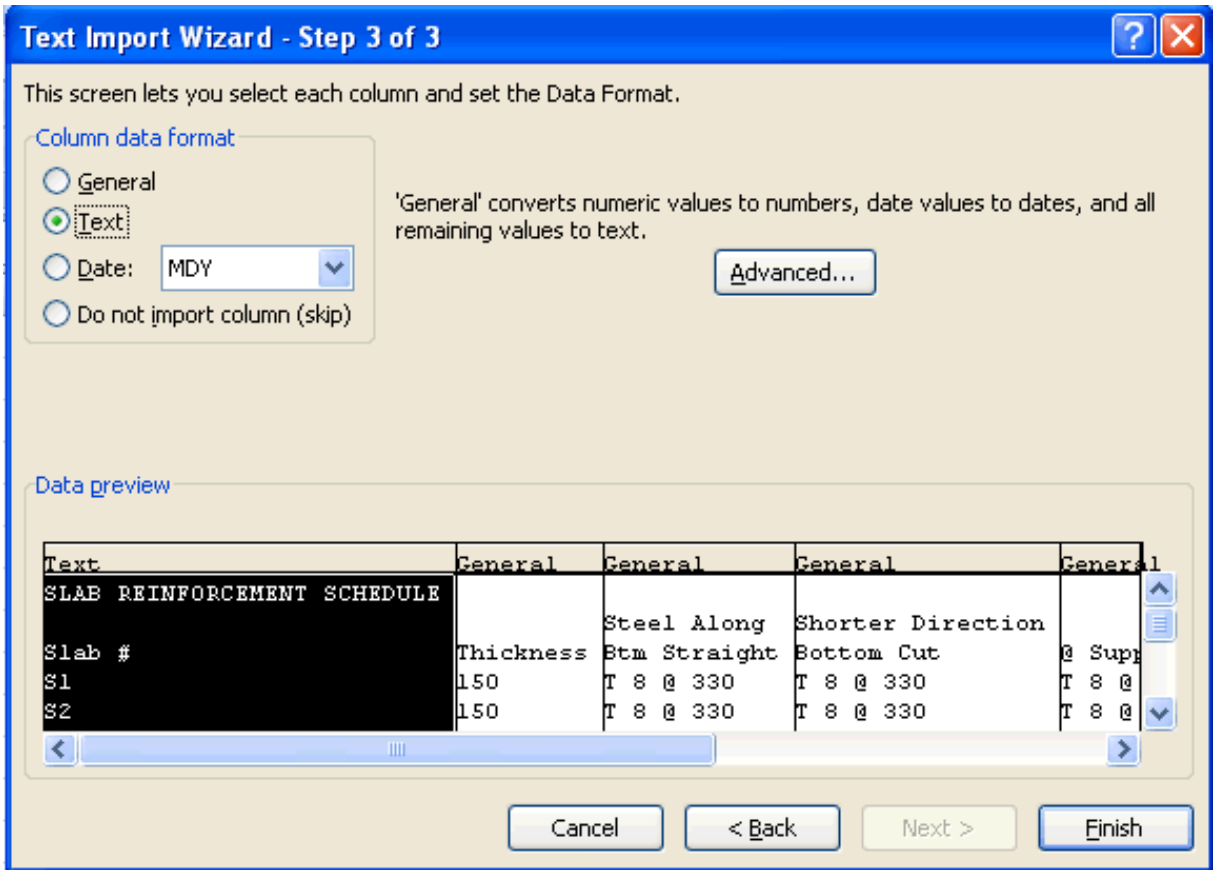
Click on Example_1SLB.txt.
As you can see, the above file is in text format.
In the following steps we will save the file in Excel format.
Once Example1_SLB.txt is clicked, following graphics is displayed.



As shown Above choose Delimited as your Option. Click On Next.
You will see the following dialogue box appear.



As shown Above choose Tab and comma as Delimiters. Click On Next. Following graphics are displayed.

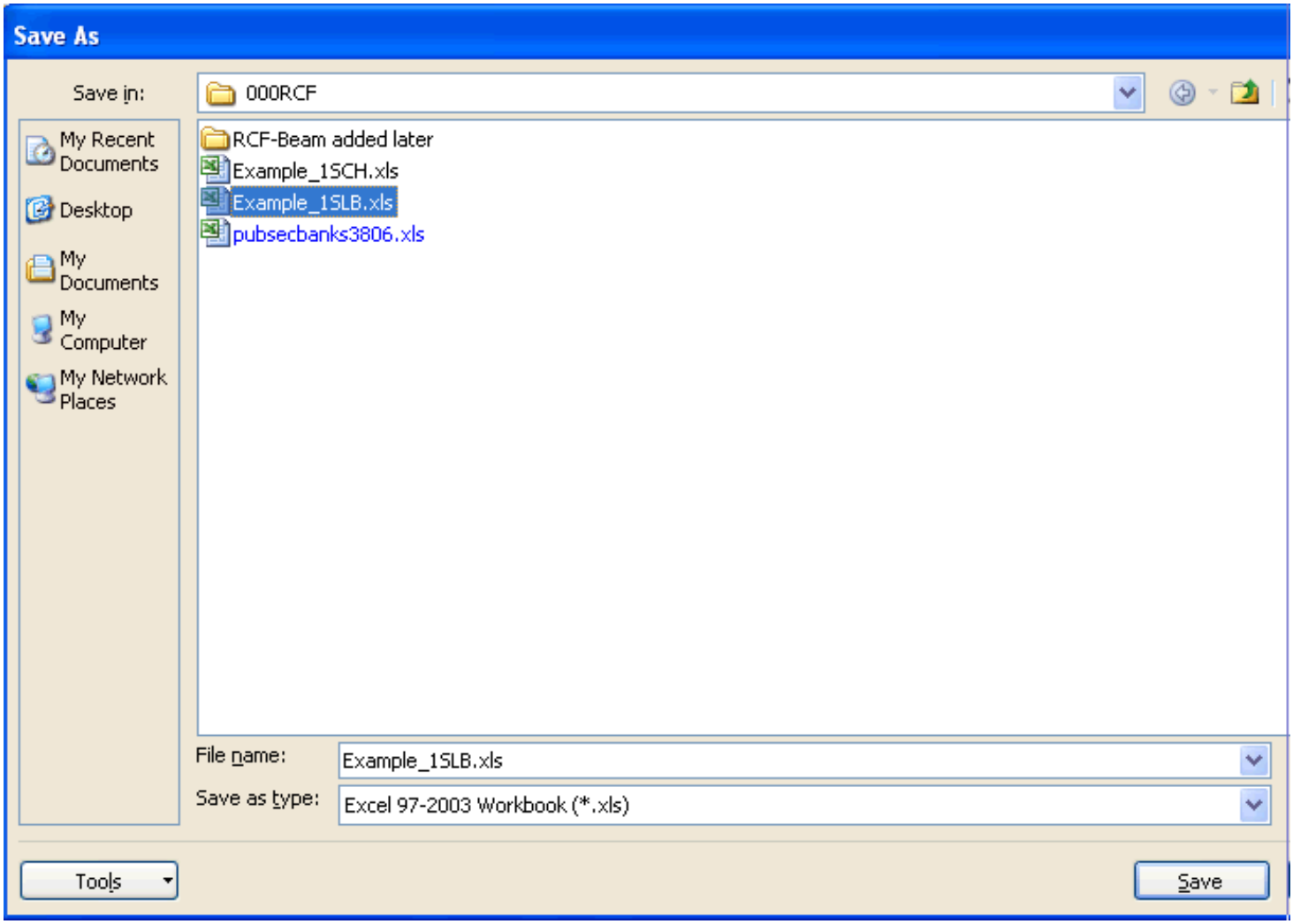


As shown above click on Text and then click on Finish.
Here you will see that Slab schedule appears in Excel . Following is a part display.
Now You can make any no of changes you want within Excel, like changing fonts, alignment of text, Column Width etc..

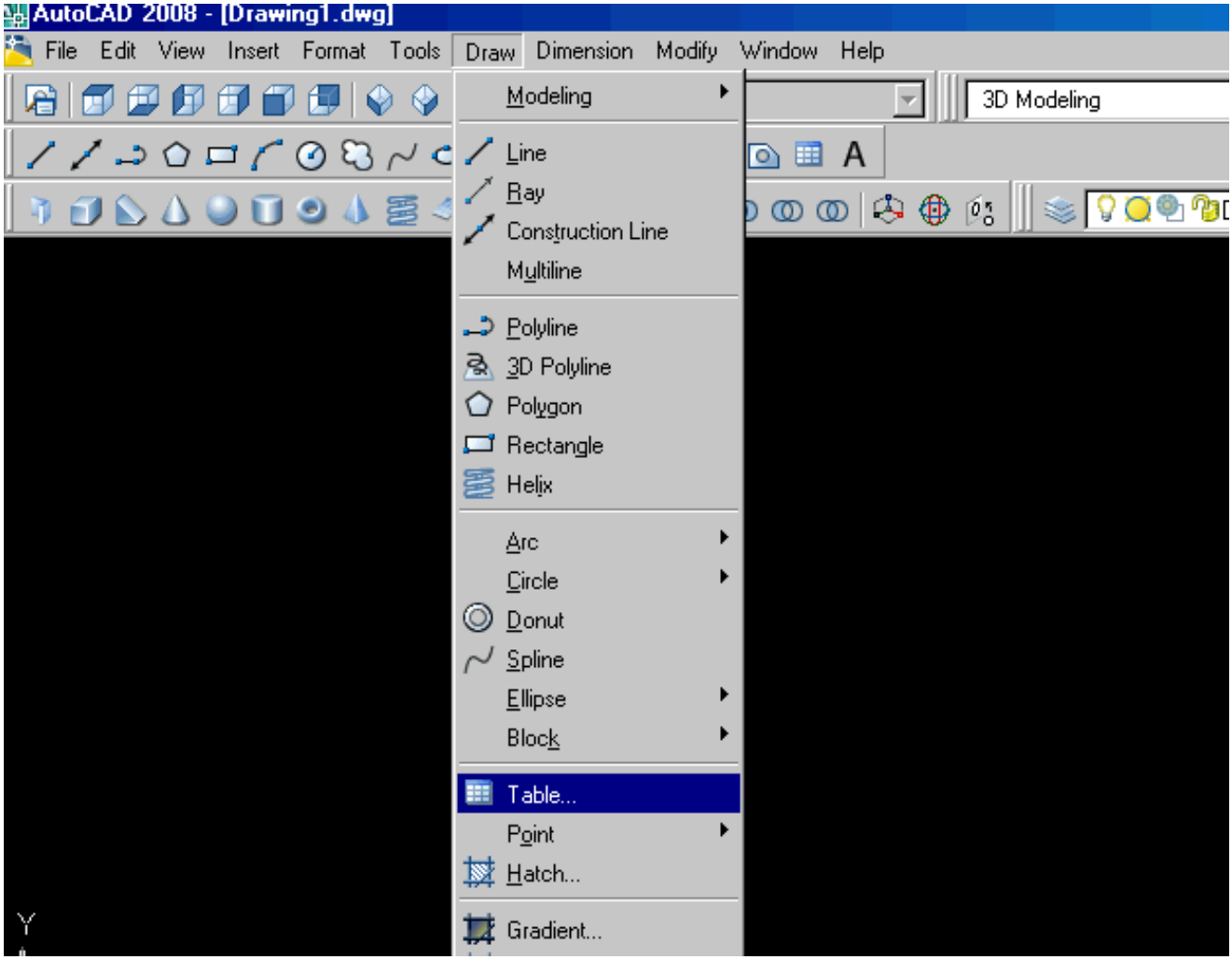
I27

A	B	C	D	E	F	G	H
SLAB REINFORCEMENT SCHEDULE							
		Steel Along	Shorter Direction		Steel Along	Longer Direction	
Slab #	Thickness	Btm Straight	Bottom Cut	@ Support Top	Btm Straight	Bottom Cut	@ Support Top
S1	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S2	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S3	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S4	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S5	150	T 8 @ 280	T 8 @ 280	T 10 @ 175	T 8 @ 280	T 8 @ 280	T 10 @ 175
S6	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S7	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S8	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S9	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S10	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S11	150	T 8 @ 330	T 8 @ 330	T 8 @ 140	T 8 @ 330	T 8 @ 330	T 8 @ 140
S12	175	T 8 @ 180	T 8 @ 180	T 12 @ 150	T 8 @ 400	T 8 @ 400	T 8 @ 165
S13	150	T 8 @ 200		T 8 @ 165	T 8 @ 200		T 8 @ 165
S14	175	T 8 @ 240	T 8 @ 240	T 10 @ 140	T 8 @ 400	T 8 @ 400	T 8 @ 165

After making all the required changes, don't forget to save the table in Excel i.e. (in .xls) Format . After having saved the file, you are done with Excel part , Exit from Excel and proceed to AutoCAD.



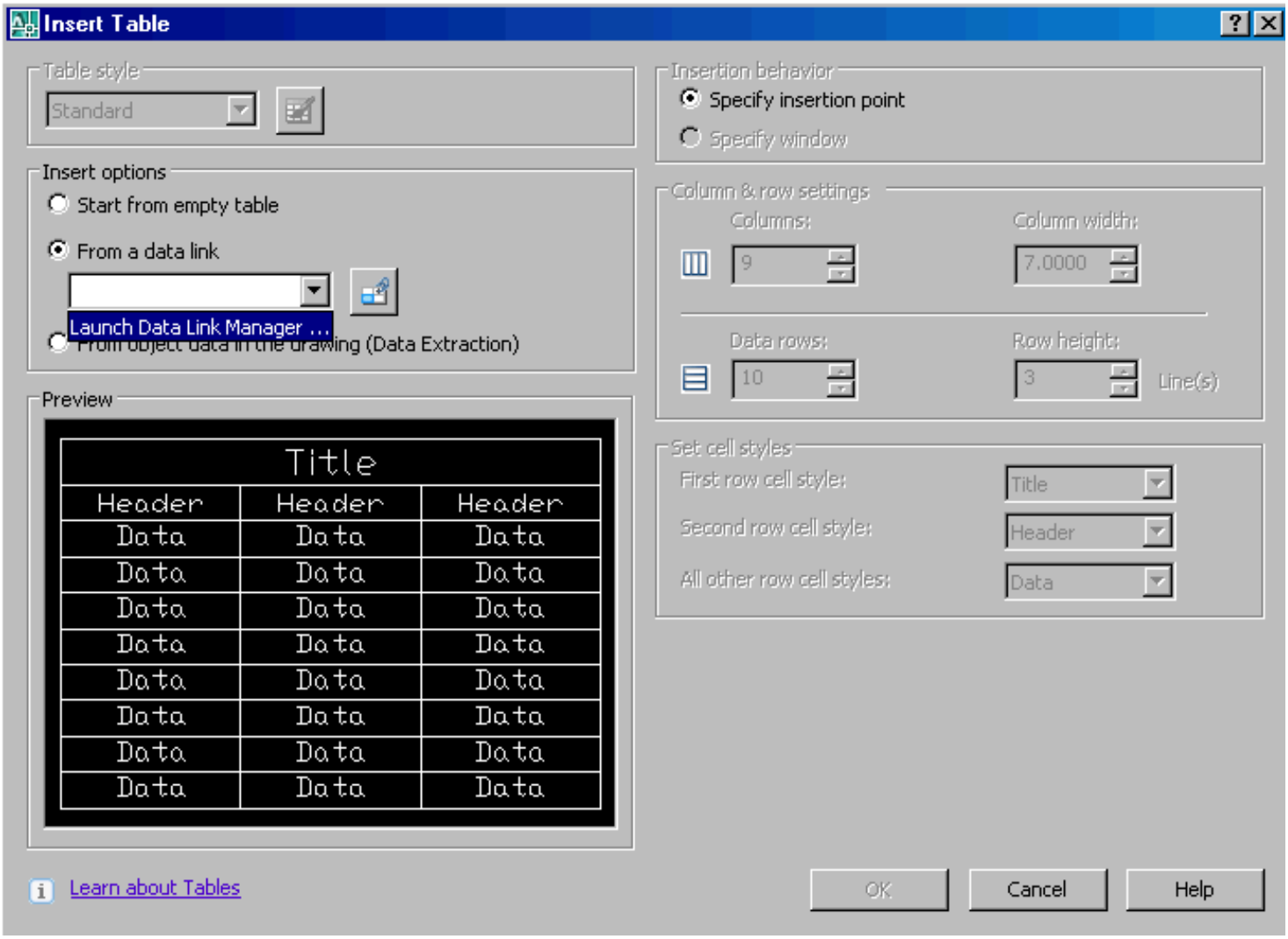
➤ **Start AutoCAD. Click on DRAW . From the drop down menu click on Table a shown below.**



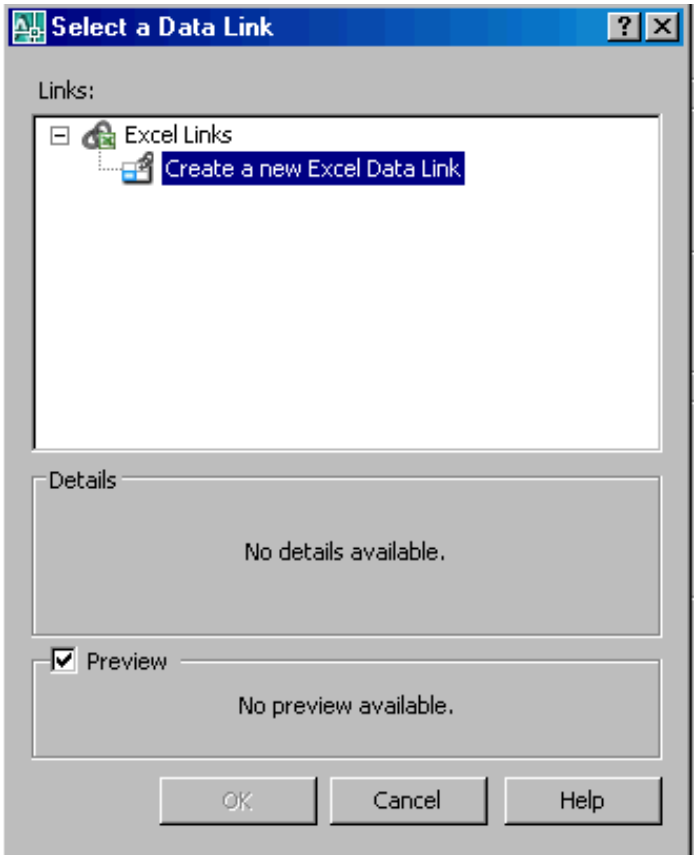
A dialogue box will appear.

As show below click on From a data link .

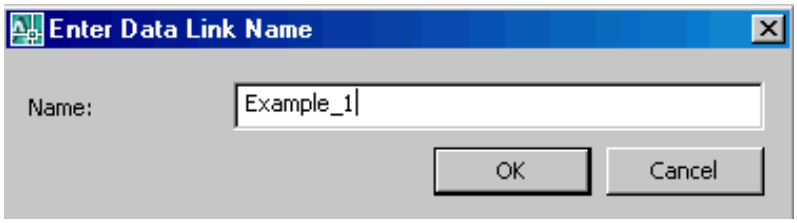
From the drop down menu click on Launch Data Link Manager.



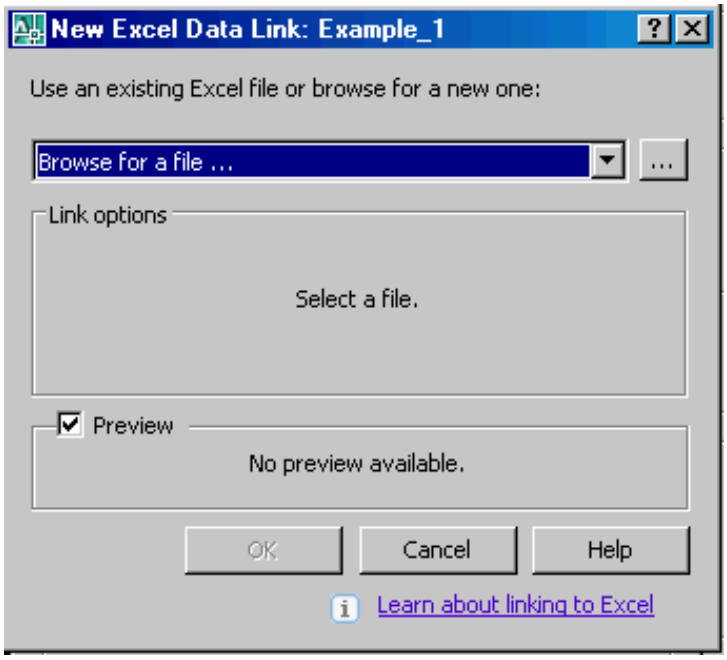
Following graphics are displayed. Click on Create a new Excel Data Link.



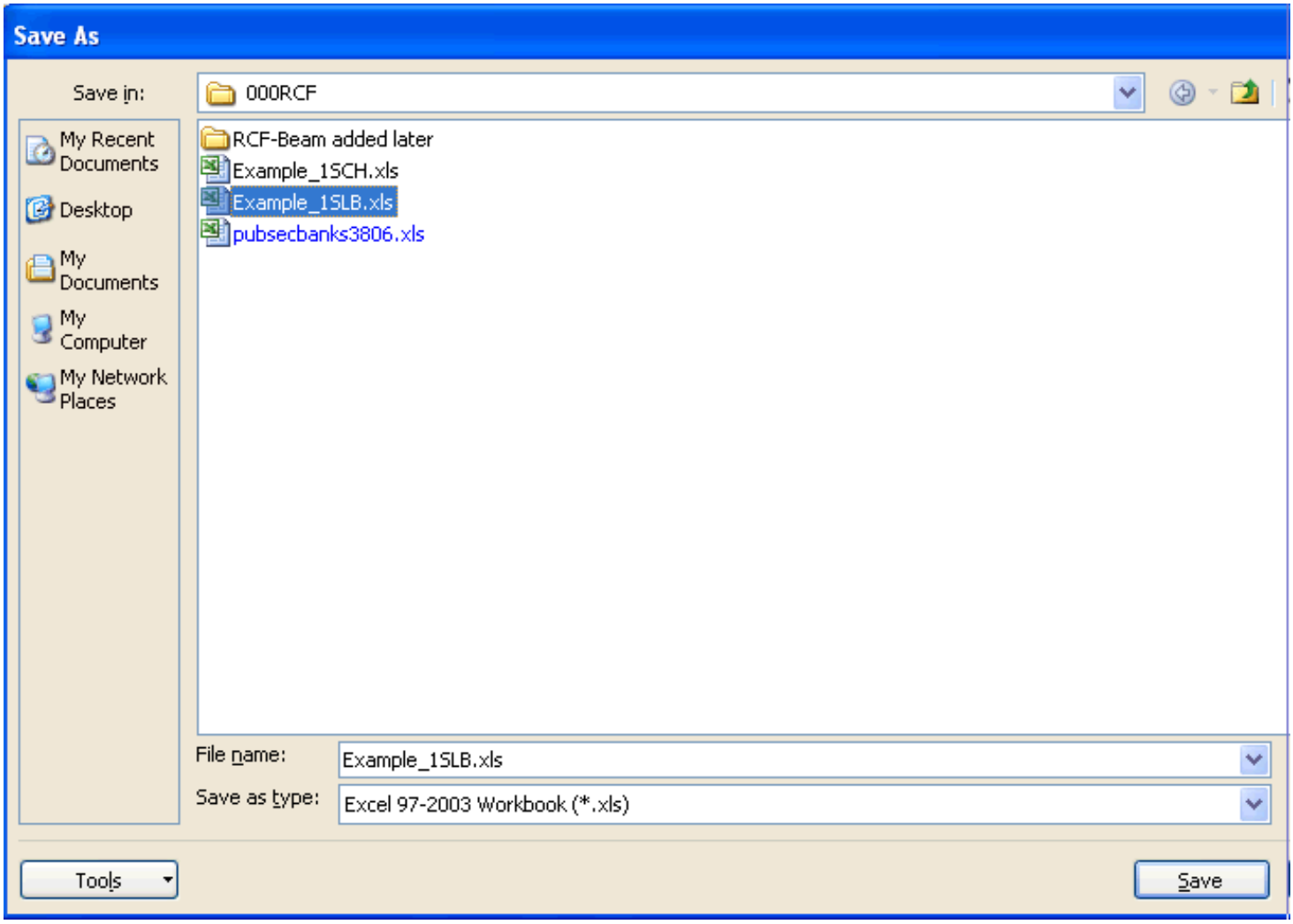
A dialogue box appears asking you to Enter a name.
You can Enter any name for e.g. Example_1.
Click on OK.



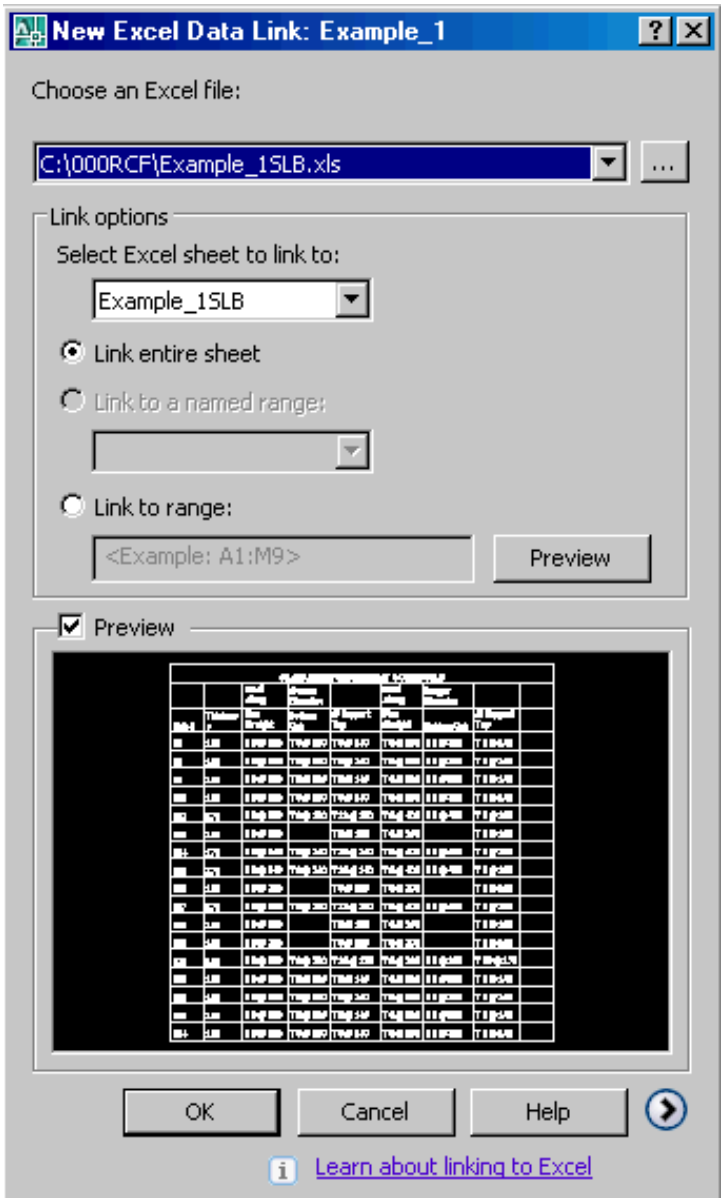
Following Graphics will appear. Click on Browse for a file.



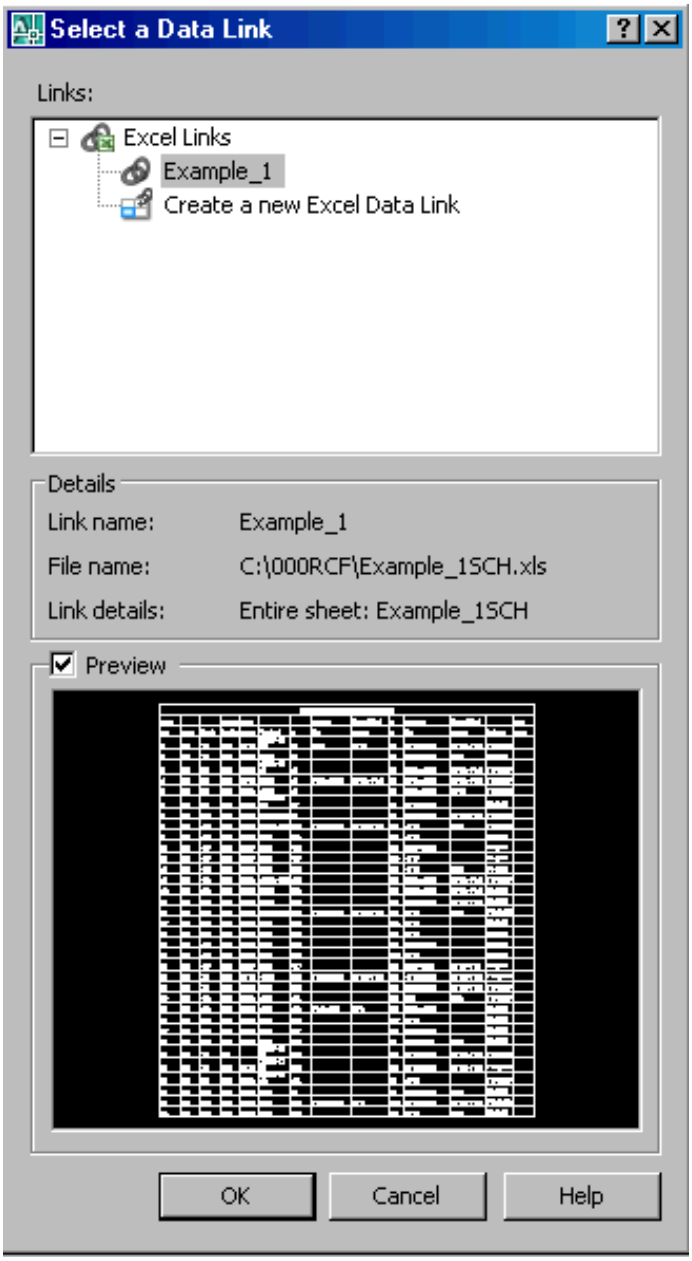
➤ **A Window dialogue box appears. Click on the required file (i.e. the file that we saved previously in Excel format) and click on Open.**



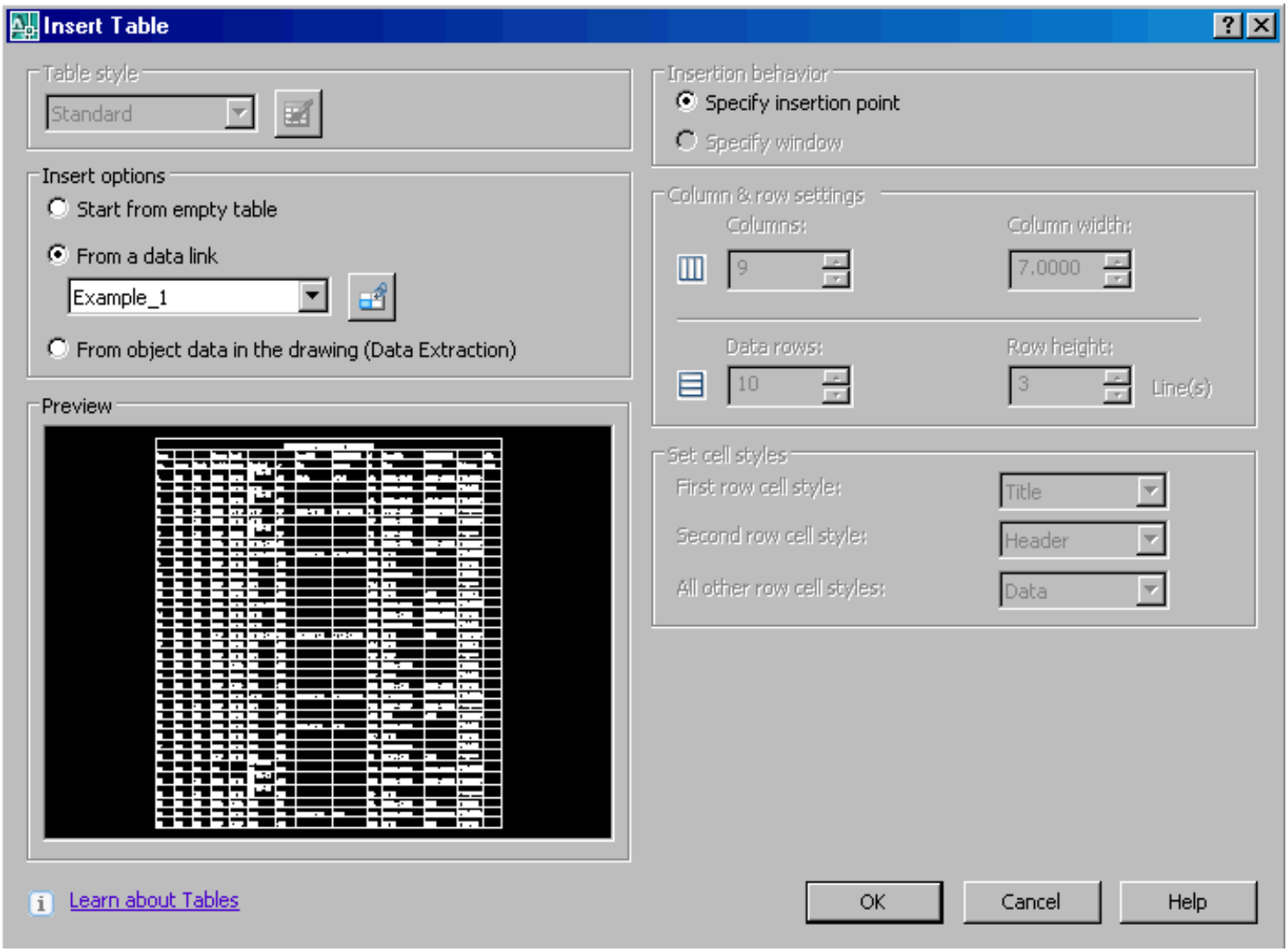
➤ **Following Graphics appears showing the preview of the table in AutoCAD. Click on OK.**



Again a dialogue box appears showing the created link and preview of the table. Click on OK.



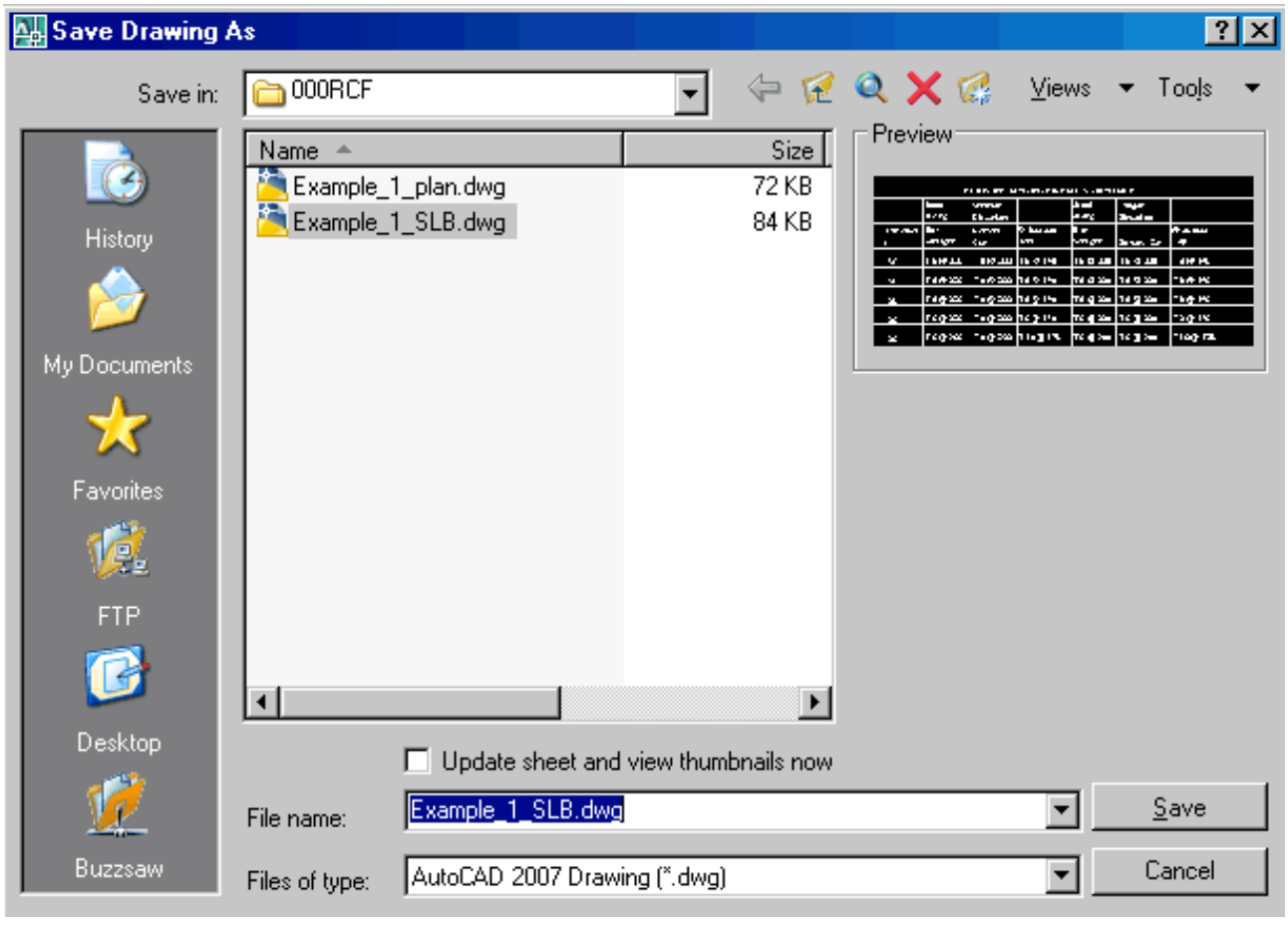
Another dialogue box appears . Click on OK.



Now specify an Insertion Point, after which the Slab Schedule will be displayed in the form of AutoCAD drawing. Following is a part display.

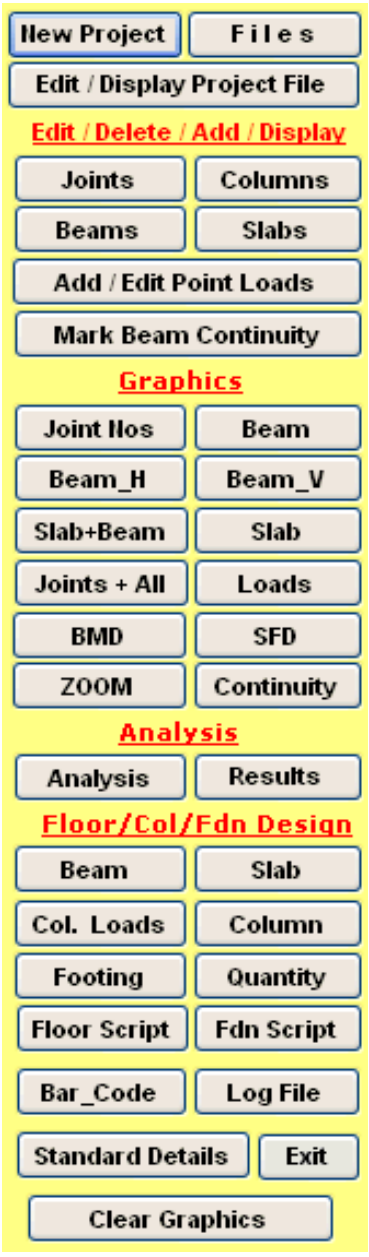
SLAB REINFORCEMENT SCHEDULE						
		Steel Along	Shorter Direction		Steel Along	Longer Direction
Slab #	Thickness	Btm Straight	Bottom Cut	@ Support Top	Btm Straight	Bottom Cut
S1	150	T8 @ 330	T8 @ 330	T8 @ 140	T8 @ 330	T8 @ 330
S2	150	T8 @ 330	T8 @ 330	T8 @ 140	T8 @ 330	T8 @ 330
S3	150	T8 @ 330	T8 @ 330	T8 @ 140	T8 @ 330	T8 @ 330
S4	150	T8 @ 330	T8 @ 330	T8 @ 140	T8 @ 330	T8 @ 330
S5	150	T8 @ 280	T8 @ 280	T10 @ 175	T8 @ 280	T8 @ 280
S6	150	T8 @ 330	T8 @ 330	T8 @ 140	T8 @ 330	T8 @ 330
S7	150	T8 @ 200		T8 @ 165	T8 @ 200	
S8	150	T8 @ 200		T8 @ 165	T8 @ 200	
S9	150	T8 @ 200		T8 @ 165	T8 @ 200	

➤ **Finally, don't forget to save the above drawing in AutoCAD (i.e. dwg) format.**



LEARN RCF STEP BY STEP

STEP NO. 16 : Design of Building Columns

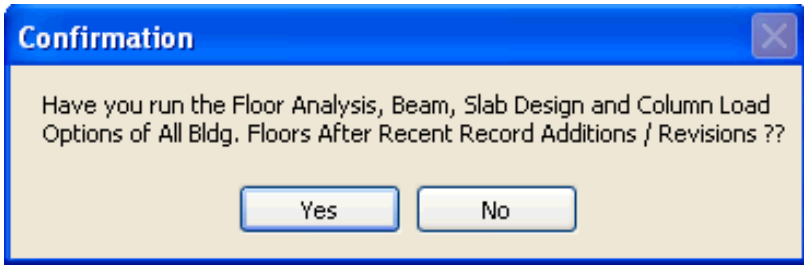


- **When Program starts, the Menu above is displayed. Under the Floor/Col/Fdn Design Heading following options are displayed.**

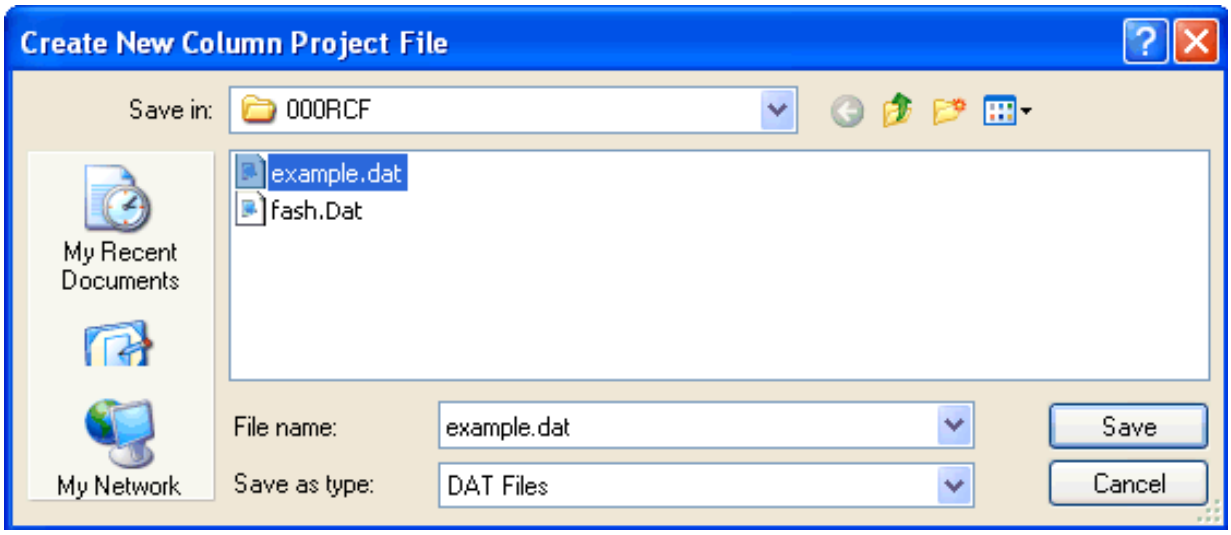
  - **Beam**
  - **Slab**
  - **Column Loads**
  - **Column Design**
  - **Footing Design**
  - **Quantity**
  - **Floor Script for AutoCAD Dwg.**
  - **Foundation Script for AutoCAD Dwg.**

  Now Click on " Column " Option.

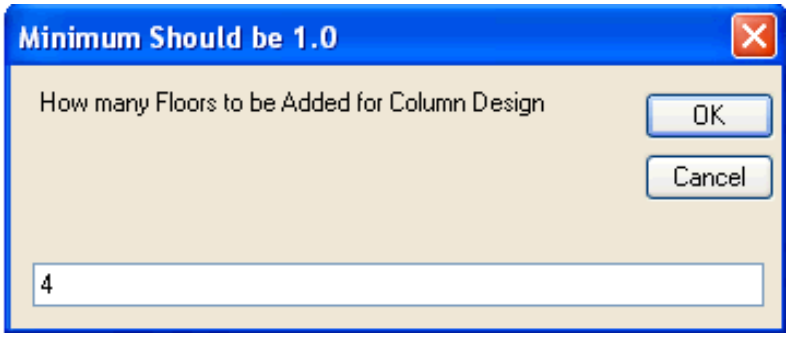
  Following Important Message is displayed.



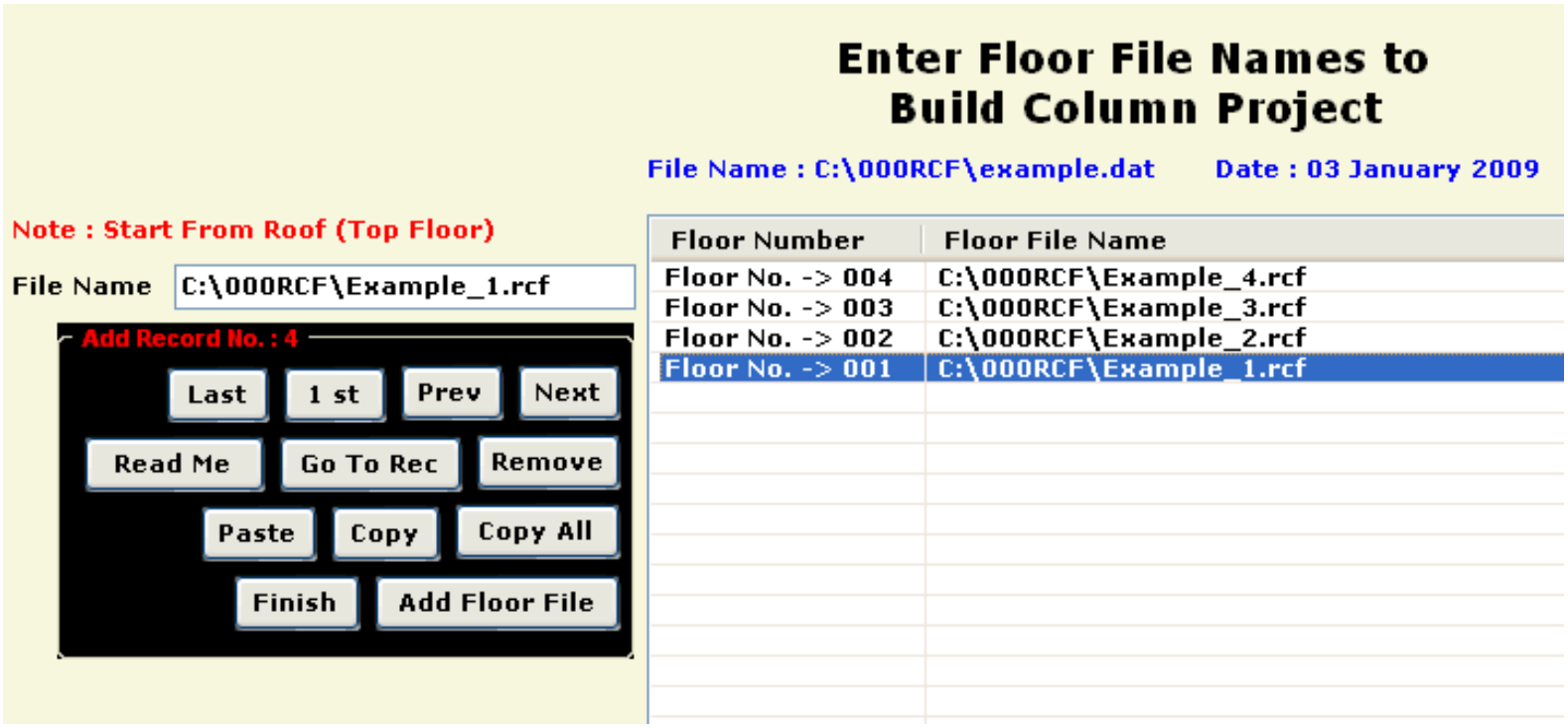
- **In order to Design all the Columns of Building, we must first RUN the Analysis, Beam, Slab Design and Column Loads of individual Floors. If Columns are having Moments than they should be incorporated using Edit / Delete / Add / Display option (Refer Step 5) before performing floor analysis. Also any changes to column parameters should be carried out at this stage, for example I have changed Steel Face to "N" for all columns (Reinforcement distributed on all four faces).**
  If Yes is clicked, following dialogue window is displayed.



- **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 ".RCF". 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.**



- **No. of Floors can be no. of storey of multi-story building OR no. of storeys + 1, to account for ground floor. I have indicated no. of floors as four (4). A typical floor can be converted in to G. F. by deleting all slabs (Refer Step # 6) and re-analyzing the floor. Thus all floor beams are Plinth Beams having only Masonry Wall Load. Click OK button. A new window appears.**



- **In order to Build Column Project, I have indicated 4 Floor files, corresponding to 4 floors. Actually Example_4, Example_3, and Example_2 are typical floors. I have created, analyzed and designed only one floor (Example_4) and copied this floor file to Example_3, Example_2 and Example_1.**
  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.
  • This Floor no. -> 001 could be G. F. of Bldg. having only Plinth Beams + Walls, but no Slab.

  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.

DISPLAYING COLUMN LOADS OF BUILDING

File Name : C:\000RCF\example.dat Date : 03 January 2009

Record No. : 1

Last | 1 st | Prev | Next

Print | Go To Rec | OK

Column Number	Cummulative Loads
C1 : Floor No. -> 004	34.713
C1 : Floor No. -> 003	69.426
C1 : Floor No. -> 002	104.139
C1 : Floor No. -> 001	138.965
C2 : Floor No. -> 004	54.037
C2 : Floor No. -> 003	108.074
C2 : Floor No. -> 002	162.111
C2 : Floor No. -> 001	217.047
C3 : Floor No. -> 004	44.538
C3 : Floor No. -> 003	89.076
C3 : Floor No. -> 002	133.614
C3 : Floor No. -> 001	178.714
C4 : Floor No. -> 004	29.715
C4 : Floor No. -> 003	59.43
C4 : Floor No. -> 002	89.145
C4 : Floor No. -> 001	118.859
C5 : Floor No. -> 004	35.774
C5 : Floor No. -> 003	71.548
C5 : Floor No. -> 002	107.322
C5 : Floor No. -> 001	143.321
C6 : Floor No. -> 004	102.909
C6 : Floor No. -> 003	205.818
C6 : Floor No. -> 002	308.727
C6 : Floor No. -> 001	414.785
C7 : Floor No. -> 004	77.34
C7 : Floor No. -> 003	154.68
C7 : Floor No. -> 002	232.019
C7 : Floor No. -> 001	311.385
C8 : Floor No. -> 004	49.548
C8 : Floor No. -> 003	99.096
C8 : Floor No. -> 002	148.644
C8 : Floor No. -> 001	198.98

- **Click OK button, Column Schedule is displayed as under. Note that Column design is fully automatic, Input from the User is not required.**

COLUMN REINFORCEMENT SCHEDULE

File Name : C:\000RCF\example.dat

Date : 03 January 2009

Project # : 8912

Conc. Grade : M20

Bldg ID : Admin

Client ID : Super Civil CD

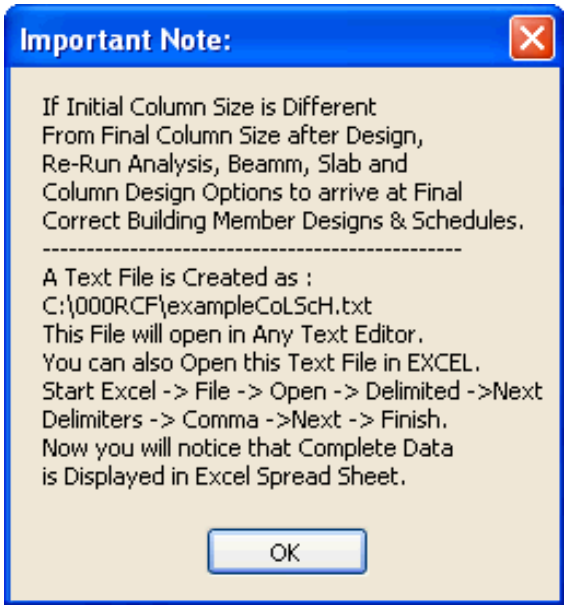
Description	X-X Dim	Y-Y Dim	Reinforcement	Links
C1 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C1 : Floor No. -> 003	600	300	8 T 16	d 6 @ 255 - 2 Nos
C1 : Floor No. -> 002	600	300	8 T 16	d 6 @ 255 - 2 Nos
C1 : Floor No. -> 001	650	300	8 T 16	d 6 @ 255 - 2 Nos
C2 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C2 : Floor No. -> 003	600	300	8 T 16	d 6 @ 255 - 2 Nos
C2 : Floor No. -> 002	750	300	4 T 12 + 8 T 16	d 6 @ 190 - 3 Nos
C2 : Floor No. -> 001	1000	350	4 T 12 + 8 T 20	d 6 @ 190 - 3 Nos
C3 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C3 : Floor No. -> 003	600	300	8 T 16	d 6 @ 255 - 2 Nos
C3 : Floor No. -> 002	650	300	8 T 16	d 6 @ 255 - 2 Nos
C3 : Floor No. -> 001	850	300	4 T 12 + 8 T 16	d 6 @ 190 - 3 Nos
C4 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C4 : Floor No. -> 003	600	300	8 T 16	d 6 @ 255 - 2 Nos
C4 : Floor No. -> 002	600	300	8 T 16	d 6 @ 255 - 2 Nos
C4 : Floor No. -> 001	600	300	8 T 16	d 6 @ 255 - 2 Nos
C5 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C5 : Floor No. -> 003	600	300	8 T 16	d 6 @ 255 - 2 Nos
C5 : Floor No. -> 002	600	300	8 T 16	d 6 @ 255 - 2 Nos
C5 : Floor No. -> 001	700	300	8 T 12 + 4 T 16	d 6 @ 190 - 3 Nos
C6 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos
C6 : Floor No. -> 003	950	300	12 T 16	d 6 @ 255 - 3 Nos
C6 : Floor No. -> 002	1000	450	12 T 20	d 6 @ 285 - 3 Nos
C6 : Floor No. -> 001	1000	600	12 T 25	T 8 @ 380 - 3 Nos
C7 : Floor No. -> 004	600	300	8 T 16	d 6 @ 255 - 2 Nos

- **Click Read Me Button, following info + design philosophy is described.**

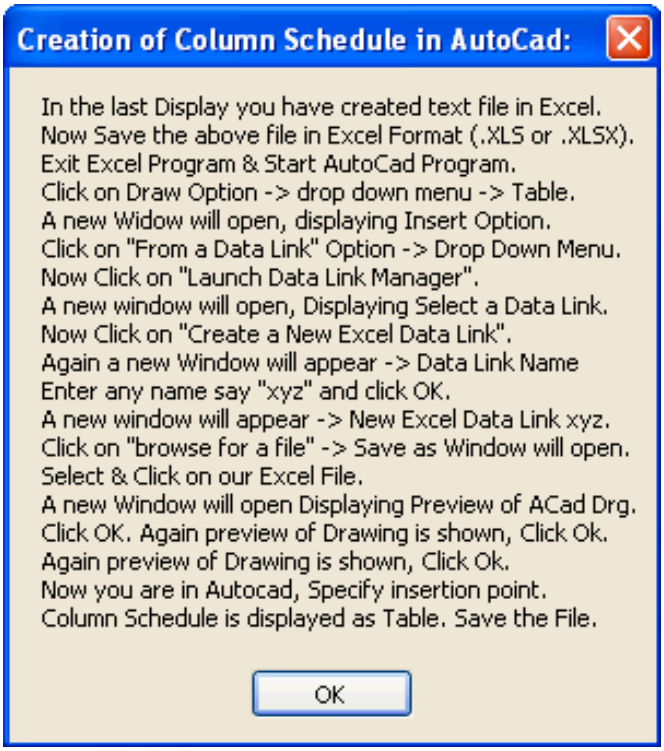
1. Column Size in MM.
2. Max. Size = 600 x 1800 MM.
3. T Indicates Tor Steel.
4. d Indicates Mild Steel.
5. @ means Spacing of Bars in MM C/C.
6. Nos Means Links per Set."
7. ERROR : Indicates Design Error,
8. Refer LOG File for Details.
9. Always Run Analysis, Beam and
10. Slab Design Options, before running
11. Column Design to avoid ERRORS.
12. Refer Std. Column Details for Reinforcement Details.
13. Column design is based on keeping the steel %
14. fixed at 0.80 %, for a given column section.
15. If the section is inadequate, longer dimension
16. will increase by 50 up to 1000 MM. If still the sec.
17. is UnSafe, shorter dim. will increase up to 600 MM.
18. After reaching 600 x 1000 MM section, if still col.
19. sec. is unsafe than longer dim. will increase upto
20. 1800 MM. If still sec. is unsafe then steel % will
21. increase by 0.1 upto 3.0 %, after reaching this limit
22. Error Message will be Flashed.
23. If a user wants to keep the Column size fixed then

24. he/she should give initial steel % appropriately high.

Now click OK button to proceed.



- **The above Message regarding Re-Analyzing, Designing & Column Load calculation option is very important for arriving at Correct Floor Design. The other Message is regarding exporting Column Schedule file to EXCEL, which will be used for Re-Exporting file to AutoCAD. Click OK Button.**

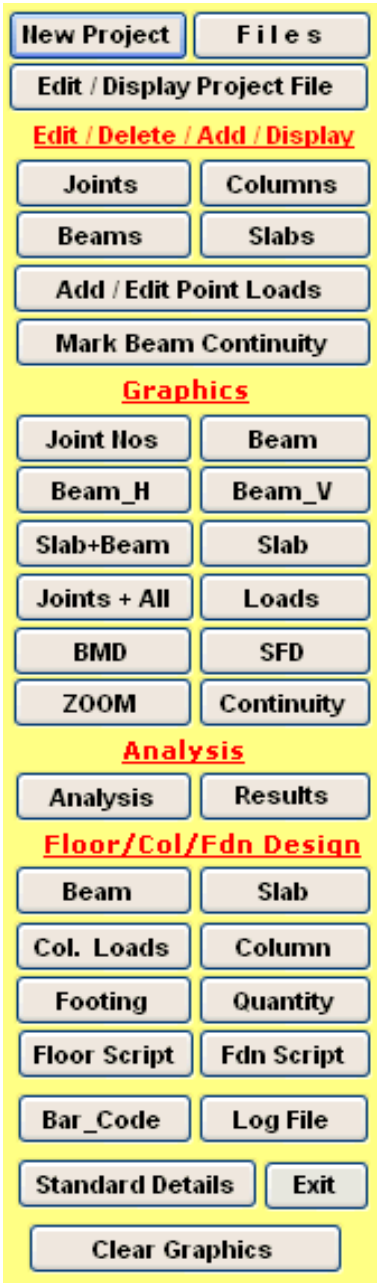


- **The above message describes creation of column schedule in AutoCAD. The Steps required to create Column Schedule in AutoCAD is similar to that of Beam / Slab, Refer Step No. 14 and 15 for details. Now we have come to the end of Step # 16.**
  In the next step we will Design Isolated Footings.

STEP NO. 16 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 17 : Design of Isolated Footings

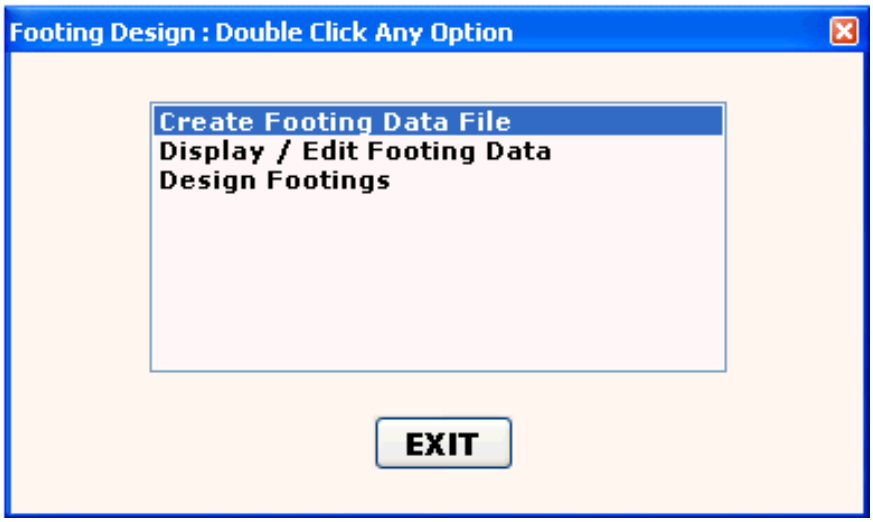


- **When Program starts, the Menu above is displayed. Under the Floor/Col/Fdn Design Heading following options are displayed.**

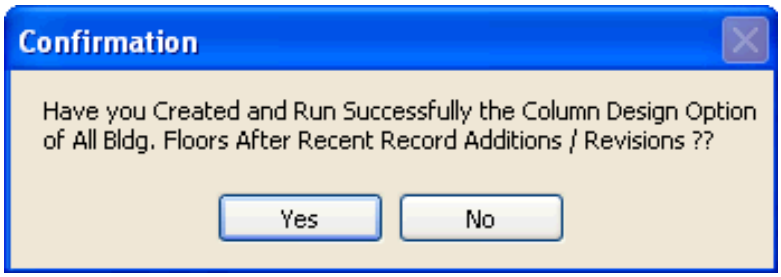
  - **Beam**
  - **Slab**
  - **Column Loads**
  - **Column Design**
  - **Footing Design**
  - **Quantity**
  - **Floor Script for AutoCAD Dwg.**
  - **Foundation Script for AutoCAD Dwg.**

Now Click on " Footing " Option.

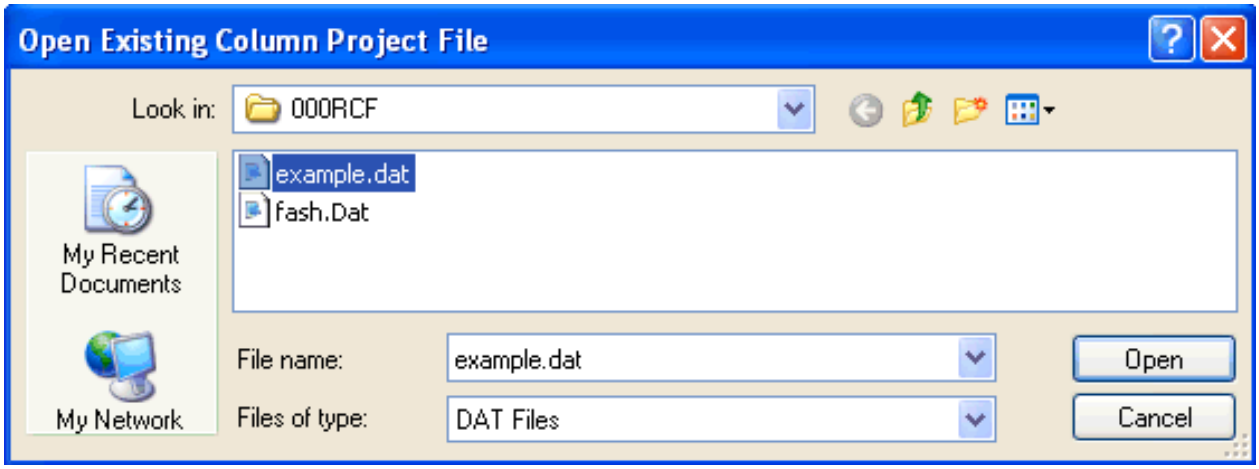
Following Foundation Menu is displayed.



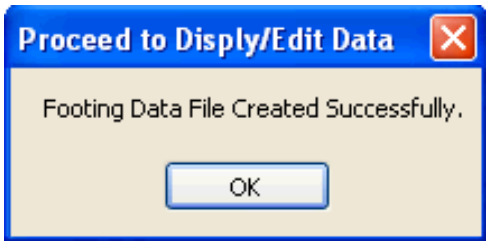
- **Double Click the 1st Option, " Create Footing Data File ".**
  Following Confirmation Window is displayed.



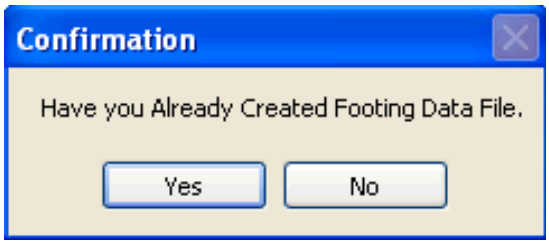
- **In order to Design Foundation of Building, we must first RUN the Column Design Option, which calculates cumulative Floor Loads for Each Column, apart from designing them. The Cumulative Load of 1st Floor Columns are taken as Compressive Load on foundation. Tensile Load or Bending Moments from either direction is not allowed. Isolated Footings could be Tapered or Uniform. Usually Tapered Footings are 40 % cheaper than Uniform footings, however quality of construction suffers due to Taper Shape. A offset of 50 - 100 MM is provided when Footing's taper meets column.**
  If Yes is clicked, following dialogue window is displayed.



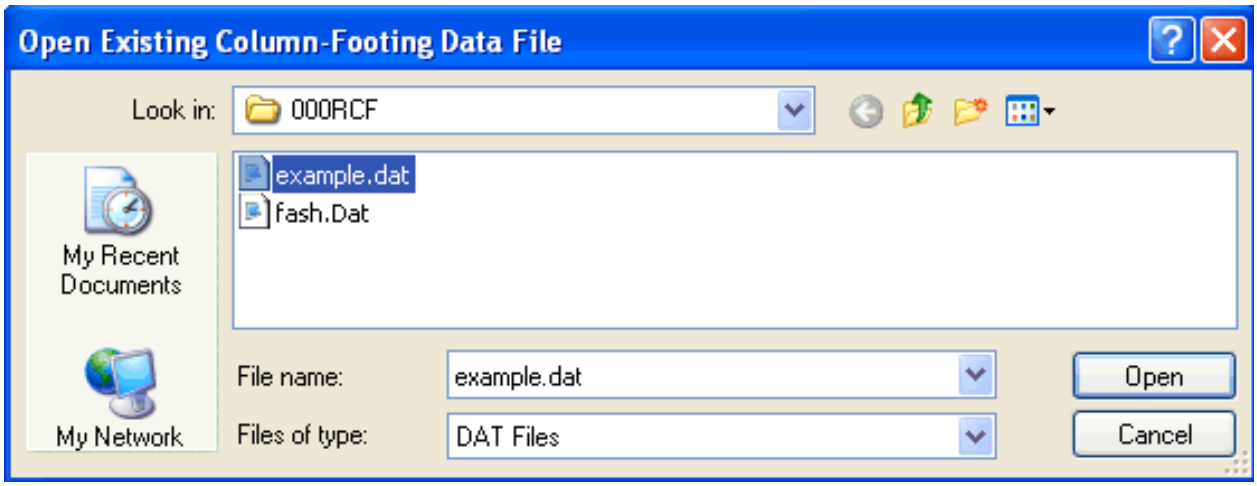
- **Open the existing Column " Example.Dat " File. Following message will appear.**



- **Click the OK Button. Now Proceed to Display / Edit Footing Data from Foundation Menu. Double Click this option. Following Confirmation message will open up.**



- **Click YES, if it is true. Following dialogue window will appear.**



- **Open the Existing Column-Footing Data File. Following table is displayed.**

DISPLAY / EDIT / FOOTING INPUT DATA

File Name : C:\000RCF\example.dat Date : 06 January 2009

Col. #	Load	Col. X-X Dim	Col. Y-Y Dim	Ftg X-X Dim	Ftg Y-Y Dim	Type	Edge Thk
C1	138.965	650	300	2.85	2.5	Tapered	150
C2	217.047	1000	350	3.67	3.02	Tapered	150
C3	178.714	850	300	3.3	2.75	Tapered	150
C4	118.859	600	300	2.62	2.32	Tapered	150
C5	143.321	700	300	2.92	2.52	Tapered	150
C6	414.785	1000	600	4.8	4.4	Tapered	150
C7	311.385	1000	450	4.27	3.72	Tapered	150
C8	198.98	950	300	3.52	2.87	Tapered	150
C9	113.456	600	300	2.57	2.27	Tapered	150
C10	116.303	600	300	2.6	2.29	Tapered	150
C11	309.133	1000	450	4.25	3.7	Tapered	150
C12	176.971	850	300	3.3	2.75	Tapered	150
C13	182.855	850	300	3.35	2.8	Tapered	150
C14	291.597	1000	450	4.15	3.6	Tapered	150
C15	190.351	900	300	3.42	2.82	Tapered	150
C16	108.328	600	300	2.5	2.19	Tapered	150
C17	88.096	600	300	2.27	1.97	Tapered	150
C18	157.885	750	300	3.07	2.62	Tapered	150
C19	142.934	700	300	2.92	2.52	Tapered	150
C20	160.321	750	300	3.1	2.65	Tapered	150
C21	100.507	600	300	2.42	2.12	Tapered	150
C22	116.299	600	300	2.6	2.29	Tapered	150
C23	317.273	1000	450	4.3	3.75	Tapered	150

Record No. : 1 of 32

Column # C1 Load in Tons 138.965 Col. dim. along X_X in MM 650 Col. dim. along Y_Y in MM 300

Footing dimension along X-X in M 2.85 Footing dimension along Y-Y in M 2.5

Footing Type : Uniform / Tapered Tapered Footing Thickness at Edge in MM 150

Prev | Next | Last | 1 st | Go To Rec | Update | Print | Clear | O K

Read Me | Typ. Footing | All Uniform | All Tapered | Copy All Edge Thk.

- **It may be noted from above table that you can edit Load on Footing, Column (Pedestal) size, Footing size, Edge thickness and type of footing (Uniform or Tapered). The isolated footing size is arrived by the program after considering applied Column Load, SBC and foundation depth.**

Click Read Me Button, following info + design philosophy is described.

1. **All Footings Correspond to respective Column Nos.**
2. **Column Loads in Tons.**
3. **Column X-X Dim -> Col. dimension along x-x axis.**
4. **Column Y-Y Dim -> Col. dimension along y-y axis.**
5. **Footing X-X Dim -> Footing dimension along x-x axis.**
6. **Footing Y-Y Dim -> Footing dimension along y-y axis.**
7. **Minimum Footing/Edge thickness shall not be < 150 MM.**
8. **Copy All Edge Thickness -> Will copy the selected edge**
9. **thickness to all column footings.**
10. **Click on All Uniform Button will make**
11. **all footing type as uniform.**
12. **Click on All Tapered Button will make**

13. all footing type as tapered.
14. A user can Increase but not Decrease Column
15. Loads, Column Size & Footing Size.
16. RCF will not design Footings Under Tension and Moments.
17. Isolated Footings under Compression will be designed.
18. While performing Frame Analysis of Bldg. A user
19. Shall make all base supports as hinged.
20. Refer our 2D Frame Analysis software or
21. Any other 2D / 3D Frame Analysis Software.

Now click OK button to proceed. A user will be asked to confirm that he/she has created & Edited Footing Data File. Click Ok and again select & Open "Example" file. Following Foundation Schedule will be displayed.

SCHEDULE OF FOUNDATION

File Name : C:\000RCF\example.dat

Date : 06 January 2009

Concrete Grade : M20

Depth Below GL in M. = 1.5

Project # : 8912

SBC in t/m2 = 20

Col. #	Col. X-X	Col. Y-Y	Ftg. X-X	Ftg Y-Y	Thickness	Edge Thk	Steel X-X	Steel Y-Y
C1	650	300	2.85	2.5	835	150	Tor 10 @ 105 c/c	Tor 10 @ 115 c/c
C2	1000	350	3.67	3.02	1020	150	Tor 12 @ 125 c/c	Tor 12 @ 140 c/c
C3	850	300	3.3	2.75	935	150	Tor 12 @ 130 c/c	Tor 10 @ 100 c/c
C4	600	300	2.62	2.32	770	150	Tor 10 @ 115 c/c	Tor 10 @ 120 c/c
C5	700	300	2.92	2.52	845	150	Tor 10 @ 105 c/c	Tor 10 @ 110 c/c
C6	1000	600	4.8	4.4	1455	150	Tor 16 @ 165 c/c	Tor 16 @ 175 c/c
C7	1000	450	4.27	3.72	1245	150	Tor 12 @ 105 c/c	Tor 12 @ 110 c/c
C8	950	300	3.52	2.87	980	150	Tor 12 @ 125 c/c	Tor 10 @ 100 c/c
C9	600	300	2.57	2.27	755	150	Tor 10 @ 120 c/c	Tor 10 @ 125 c/c
C10	600	300	2.6	2.29	755	150	Tor 10 @ 115 c/c	Tor 10 @ 125 c/c
C11	1000	450	4.25	3.7	1240	150	Tor 12 @ 105 c/c	Tor 12 @ 115 c/c
C12	850	300	3.3	2.75	930	150	Tor 12 @ 135 c/c	Tor 10 @ 105 c/c
C13	850	300	3.35	2.8	955	150	Tor 12 @ 130 c/c	Tor 10 @ 100 c/c
C14	1000	450	4.15	3.6	1200	150	Tor 12 @ 110 c/c	Tor 12 @ 115 c/c
C15	900	300	3.42	2.82	970	150	Tor 12 @ 130 c/c	Tor 10 @ 100 c/c
C16	600	300	2.5	2.19	730	150	Tor 10 @ 120 c/c	Tor 10 @ 130 c/c
C17	600	300	2.27	1.97	645	150	Tor 10 @ 140 c/c	Tor 10 @ 150 c/c
C18	750	300	3.07	2.62	880	150	Tor 12 @ 140 c/c	Tor 10 @ 105 c/c
C19	700	300	2.92	2.52	845	150	Tor 10 @ 105 c/c	Tor 10 @ 115 c/c
C20	750	300	3.1	2.65	895	150	Tor 12 @ 140 c/c	Tor 10 @ 105 c/c
C21	600	300	2.42	2.12	695	150	Tor 10 @ 125 c/c	Tor 10 @ 135 c/c
C22	600	300	2.6	2.29	755	150	Tor 10 @ 115 c/c	Tor 10 @ 125 c/c
C23	1000	450	4.3	3.75	1260	150	Tor 12 @ 105 c/c	Tor 12 @ 110 c/c
C24	900	300	3.42	2.82	965	150	Tor 12 @ 130 c/c	Tor 10 @ 100 c/c
C25	900	300	3.45	2.85	970	150	Tor 12 @ 125 c/c	Tor 10 @ 100 c/c

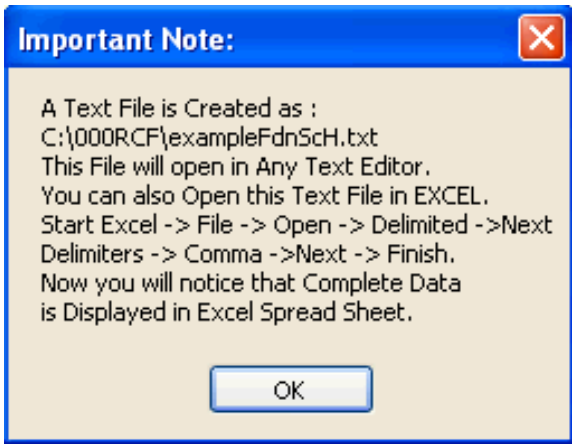
Record No. : 1 of 32

Read Me | Prev | Next | Last | 1 st | Go To Rec | Print | O K

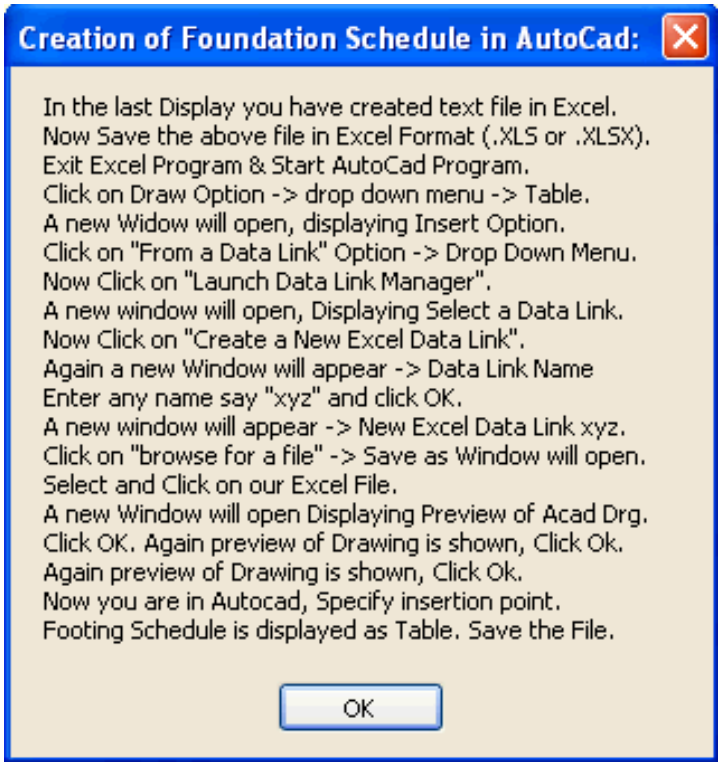
- **Click Read Me Button, following info is described.**

1. Steel X-X -> Reinforcement along X-X Direction.
2. Steel Y-Y -> Reinforcement along Y-Y Direction.
3. Effective Cover is taken as 60 mm.
4. PCC of 50 MM below each Footing is Assumed.
5. Fy = 415 : Load factor = 1.5
6. For design of foundation under tension & bending,
7. refer our SUPER CIVIL CD software.
8. Design will not check overlapping of footings,
9. It has to be checked manually.
10. The 1st Floor (G. F.) Concrete Grade and size of
11. columns will be taken for design of footings.
12. Spacing of Reinforcement Exceeding 1000 mm will be
13. restricted to 1000 mm. Spacing < 100 mm, will
14. not be allowed, instead Bar diameter is increased.
15. Bar diameter exceeding 25 mm is will not be allowed,
16. Error message will be displayed. Refer LOG File.
17. Hence Spacing is restricted to above 100 MM
18. and Below 1000 MM. Isolated Footing Pressure exceeding
19. SBC will not be allowed.
20. Min. steel = 0.12 % in each direction.
21. For Tapered Footing Average thickness is taken for
22. calculation of minimum steel.
23. For Uniform Footing Edge thickness = Total Thickness.

Click OK Button to Proceed.



- **The other Message is regarding exporting Footing Schedule file to EXCEL, which will be used for Re-Exporting file to AutoCAD. Click OK Button.**

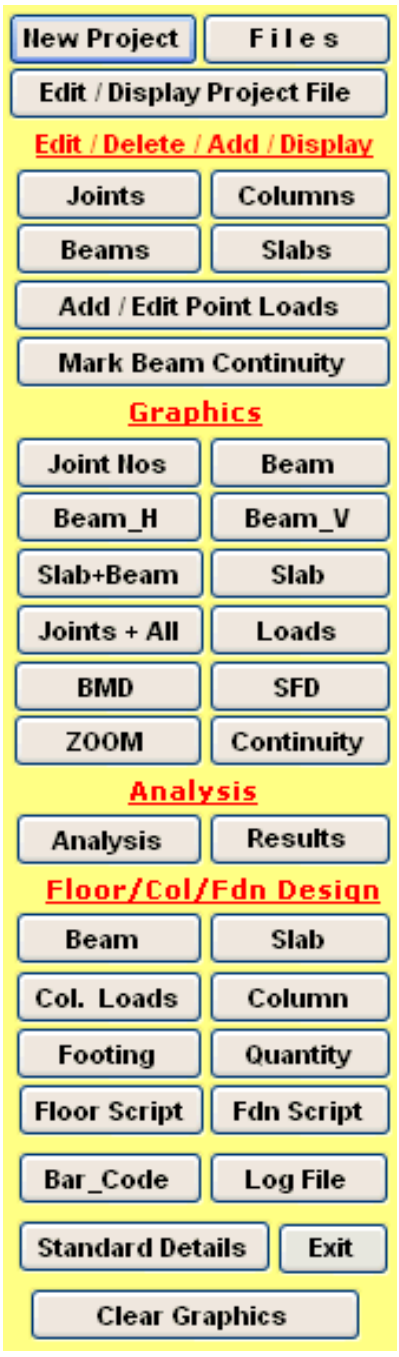


- **The above message describes creation of Foundation Schedule in AutoCAD. The Steps required to create Foundation Schedule in AutoCAD is similar to that of Beam / Slab, Refer Step No. 14 and 15 for details. Now we have come to the end of Step # 17.**

STEP NO. 17 IS OVER.

LEARN RCF STEP BY STEP

STEP NO. 18 : Column, Footing and Project Quantities And Cost

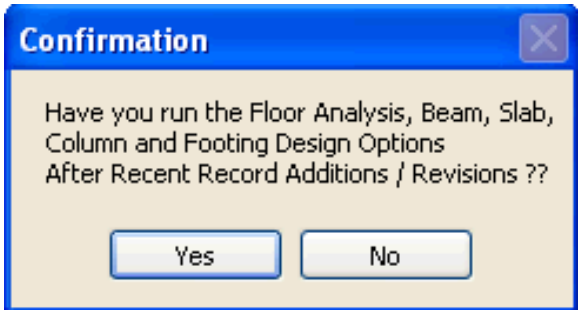


- **When Program starts, the Menu above is displayed. Under the Floor / Col / Fdn Design Heading following options are displayed.**

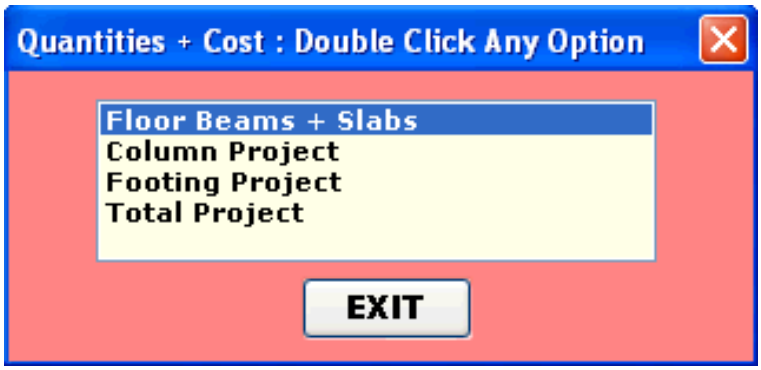
  - **Beam**
  - **Slab**
  - **Column Loads**
  - **Column Design**
  - **Footing Design**
  - **Quantity**
  - **Floor Script for AutoCAD Dwg.**
  - **Foundation Script for AutoCAD Dwg.**

  Now Click on " Quantity " Option.

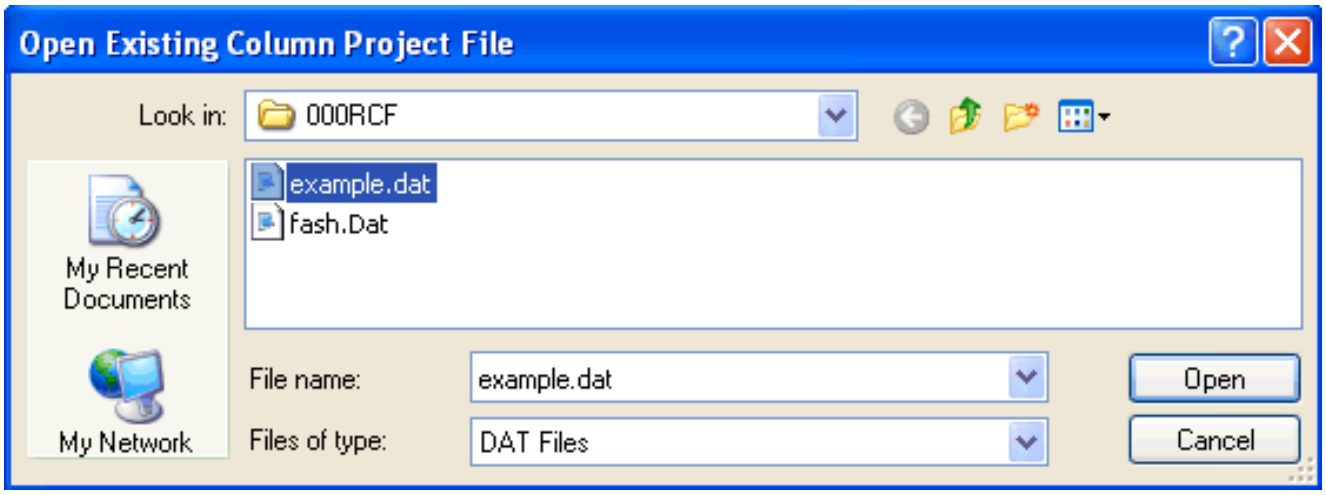
  Following Confirmation Message is displayed.



- **Unless All relevant Floors of the Building are Analyzed and Beam, Slabs, Columns, Foundation are Designed, Column Loads worked out, the Calculation of Quantities & Cost have no meaning. The Quantity Option is to be RUN last after all other things are completed. Click Yes if complied. Following ALL QTY menu is displayed.**



- **Note that we have already covered " Floor Beams + Slabs " in Step No. 11.**
  Now double click " Column Project " option. Following is displayed.



- **Open the existing Column " Example.Dat " File. Summary of Complete Building Column Quantities and Cost will be displayed as under.**

SUMMARY OF COLUMN QUANTITIES AND COST

File Name : C:\000RCF\example.dat **Date : 06 January 2009**

Total Concrete in M20 Grade in M3 = 83.505
Total Reinforcement in Tons = 8.832
Total Plaster / Painting in M2 = 764.7
Total Cement in Bags = 716
Total Aggregate in M3 = 66.804
Total Sand in M3 = 48.696
Total Concrete Cost = 751545
Total Reinforcement Cost = 441600
Total Plaster Cost = 305880
Total Painting Cost = 76470
Total Column Cost = 1575495

SUMMARY OF REINFORCEMENTS IN KG

6 MM Dia :	1576.898
8 MM Dia :	27.472
10 MM Dia :	0
12 MM Dia :	417.458
16 MM Dia :	5374.381
20 MM Dia :	1243.098
25 MM Dia :	192.955
32 MM Dia :	0

Print O K

- **Click the OK Button, and Double Click Footing Project. Open the Existing Column-Footing " Example " file. Summary of Complete Building Isolated Footing Quantities and Cost will be displayed as under.**

SUMMARY OF FOUNDATION QUANTITIES AND COST

File Name : C:\000RCF\example.dat Date : 06 January 2009

Total Concrete in M20 Grade in M3 = 148.025
Total Reinforcement in Tons = 4.035
Total Excavation and Re-Filling in M3 = 446.45
Total Cement in Bags = 1066
Total Aggregate in M3 = 118.42
Total Sand in M3 = 59.21
Total Concrete Cost = 1332225
Total Reinforcement Cost = 201750
Total Excavation and Re-Filling Cost = 66967.5
Total Foundation Cost = 1600943

SUMMARY OF REINFORCEMENTS IN KG

6 MM Dia :	0
8 MM Dia :	47.007
10 MM Dia :	1306.486
12 MM Dia :	2289.273
16 MM Dia :	392.31
20 MM Dia :	0
25 MM Dia :	0
32 MM Dia :	0

Print O K

- **Click the OK Button, and Double Click Total Project. Confirm Compliance of Message shown and Open the Existing Column-Footing " Example " file. Summary of Complete Building Quantities and Cost (Floors + Columns + Isolated Footings) will be displayed as under.**

SUMMARY OF PROJECT QUANTITIES AND COST

File Name : C:\000RCF\example.dat

Date : 06 January 2009

Project # : 8912

Bldg. ID : Admin

Concrete Grade : M20

Fy = 415

SBC in T/M2 : 20

Fdn. below GL in M = 1.5

No. of Floors = 4

Effective Cover - Beams = 30 MM
Effective Cover - Slabs = 20 MM
Effective Cover - Columns = 50 MM
Effective Cover - Foundation = 60 MM

Total Concrete in M20 Grade in M3 = 796.218
Total Concrete Cost = 7165962
Total Reinforcement in Tons = 64.747
Total Reinforcement Cost = 3237350
Total Masonry in M2 = 2604.7
Total Masonry Cost = 2213995
Total Plaster in M2 = 10109.1
Total Plaster Cost = 4043642
Total Painting in M2 = 10109.1
Total Plaster Cost = 1010910

Total Flooring in M2 = 2800
Total Flooring Cost = 2240000
Total Door / Window in M2 = 369.6
Total Door / Window Cost = 924000
Total Excavation+Refilling in M3 = 446.45
Total Excavation+Refilling Cost = 66967.5
Total Cement Bags in Nos. = 8686
Total Sand in M3 = 739.906
Total Aggragates in M3 = 721.224
Total Project Cost = 20902828
Total Floor Area in M2 = 2800
Cost per M2 = 7465.296
Cost per sft = 694.446
Cement Bags per sft = 0.288
Reinforcement in Kg per sft = 2.151
Steel in Kg / M3 of Concrete = 81.318
Conc. Cost as % of Total = 34.282
Steel Cost as % of Total = 15.487
Masonry Cost as % of Total = 10.591

REINFORCEMENT SUMMARY IN KG

6 MM Dia :	1576.898
8 MM Dia :	18527.89
10 MM Dia :	5144.84
12 MM Dia :	6940.259
16 MM Dia :	9045.808
20 MM Dia :	7789.438
25 MM Dia :	7730.955
32 MM Dia :	7992.78

Print

O K

- **Note that above data describes Technical & Managerial parameters, essential for cost comparison.**

This Completes Step # 18.

STEP NO. 18 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

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

Q T Y - Quantity Estimation & Cost, Project Control

SUPER REAL VALUATION - A Software For Immovable Properties

ROADS - Pavement Design & Rate Analysis Of Road Items

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

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

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

BILLING JI - A Database Management Software For General Billing

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

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

BID ANALYSIS - A Software For Technical & Commercial Tender Analysis

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

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

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

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

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

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

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

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

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

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

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