LEARN RAFT

A Software for Analysis, Design, Estimation,
Costing & Drawing of RCC Raft Foundation

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 RAFT

A Software for Analysis, Design, Estimation, Costing & Drawing of Reinforced Concrete Rigid Raft Foundation

Introduction + Excerpts from IS 2950 🔴 **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 Raft Beams 🔴 **STEP NO. 4**

Delete and Edit Columns 🔴 **STEP NO. 5**

Delete and Edit Raft Slabs 🔴 **STEP NO. 6**

Add & Edit Raft Beam Continuity 🔴 **STEP NO. 7**

Data Checking Through Graphics 🔴 **STEP NO. 8**

LEARN RAFT STEP BY STEP

INTRO & LIMITATIONS +
EXCERPTS FROM IS : 2950 - PART 1 - 1981

● Please take Print Out of Every Step, including this page before commencing Learn. <mark>Take a Yellow Marker Pen and Mark the Learning Process while Proceeding further.</mark> This is Essential for Learning.

● The software performs Analysis, Design, Estimation, Costing & AutoCAD Drawing of RCC Rigid Raft Foundation.

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

The results are displayed in the form of BM & SF, Raft Beam & Slab Schedule, Quantities, Cost & Approximate Bar bending Schedule for Beams.

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 Foundation Plan, Joint, Beam, Column and Slab Numbers. A Joint represents a Column location or an intersection between 2 Raft 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 Raft Foundation Plan.

● Cantilever Raft beams cannot be analyzed.

- **Beams / Columns / Slabs shall be along two mutually perpendicular axis (X and Y). Polygonal (Multi-sided) slabs cannot be analyzed.**
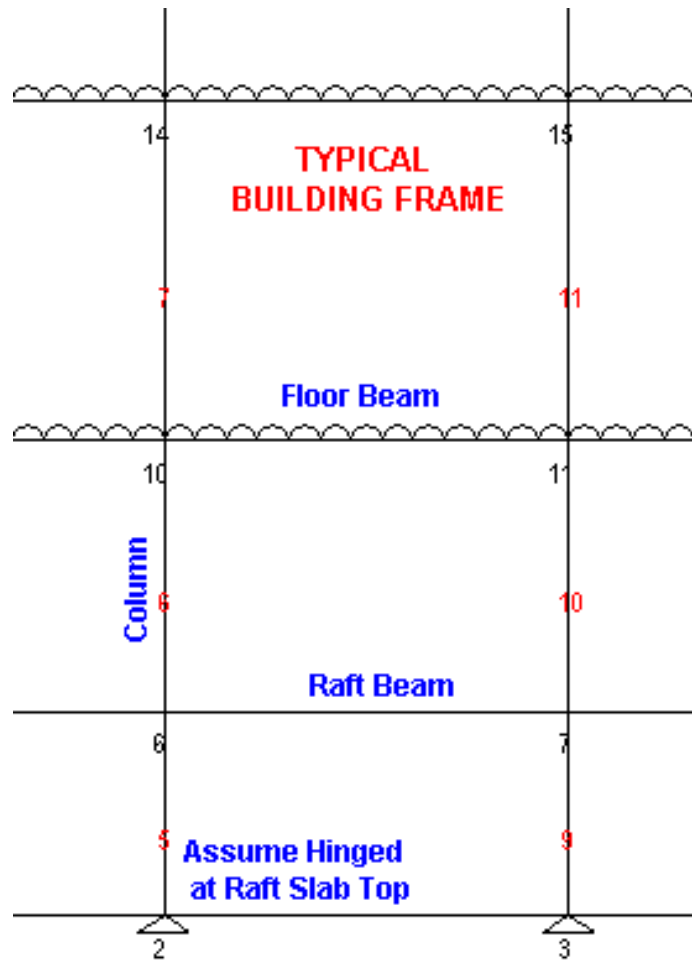
- **Analyze the Structure, assuming Raft Top (Column Base) as Hinged (Refer Graphics above). Hence there shall be no Moments on Raft Top. Raft Beam shall Carry Moments due to DL + LL and also Due to Wind and Seismic Shear.** Structure shall be Analyzed in Either direction. Hence User shall enter 3 Column Load Cases, i.e.

  ● **DL + LL**

  ● **DL + LL + Wind / Seismic Load in Ton (From X-X Direction)**

  ● **DL + LL + Wind / Seismic Load in Ton (From Y-Y Direction)**

  End Column Reactions on Raft Beams (DL + LL + 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 Reactions.

- Only three (3) Load Cases can be analyzed per File for Each Raft Beam, viz :

  ● DL + LL (Due to Upward Soil Pressure, Calculated by Program Automatically)

  ● { 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 on Raft Beams (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). However a designer has to 1st copy the original file in to another file (using Copy Option) & do the above mentioned modifications.

- Provide 300 x 300 MM Haunch between All Raft Slabs & Beams.
  In absence of 300 x 300 MM Haunch, Raft Slab may not be safe in Shear.

- **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), " Raft Beam Schedule " and " Raft Slab Schedule 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 Raft Beam joints as mentioned above under Load Case { DL + LL + Wind Or Seismic End Moment (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.

- Raft Beam & Slab Design is as per IS 456 - 2000. Refer IS 2950 : Part 1 - 1981 / 1965 For Rigid Raft Design / Relative Stiffness Factor.

- Beam Width / Depth < 150 mm not permitted.

- Beam Width / Depth > 3500 mm not permitted.

- Links (Stirrups) Area > 44.8 cm2 for Beams is not permitted.

- Beam Reinforcement > 4 % not permitted.

- 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.

- Minimum Computer RAM memory of 1 GB is recommended.

- **Use Laser OR Ink Jet Printer.**

EXCERPTS FROM IS : 2950 - PART 1 - 1981

- **IS : 2950 - Part 1 is CP for Design & Construction of Raft Foundation.**

- **In granular Soil, the SBC of Raft is generally very high, however for rafts placed at considerable depth (Like Basements) the punching mode of failure shall be investigated. The Influence of soil compressibility & related scale effects should be assessed.**
  For Rafts on cohesive soils, the stability against deep seated failures & effect of long term settlement shall be taken in to consideration.
  The Depth of Raft Foundation shall not be Less than 1.0 M.

- **Rigidity of Foundation tends to iron out uneven deformations & thereby modifies the contact pressure distribution. High order of rigidity is characterized by large moments & relative small uniform settlements. A rigid Foundation may also generate high secondary stresses in structural members. The effects of Rigidity shall be taken in to account in the analysis.**

- **Rigid foundation design is based on the assumption of linear distribution of contact pressure. The basic assumption of this method are :**
  (a) The Foundation is rigid relative to the supporting soil & the compressible soil layer is relatively shallow.
  (b) The contact pressure is assumed as planner, such that the centroid of the contact pressure coincides with the line of action of the resultant force of all loads acting on the foundation.

  This method may be used when either of the following conditions is satisfied.
  (a) The structures behaves as rigid (due to combine action of the superstructure & the foundation) with a relative stiffness factor K > 0.50. (Refer Appendix C)
  (b) The Column spacing is less than 1.75 / $\tilde{A}$. (Refer Appendix C)

  The Raft is analyzed as a whole in of the two perpendicular direction & analysis is based on statics.

LEARN RAFT STEP BY STEP

STEP NO. 1 : New Project (File) Creation
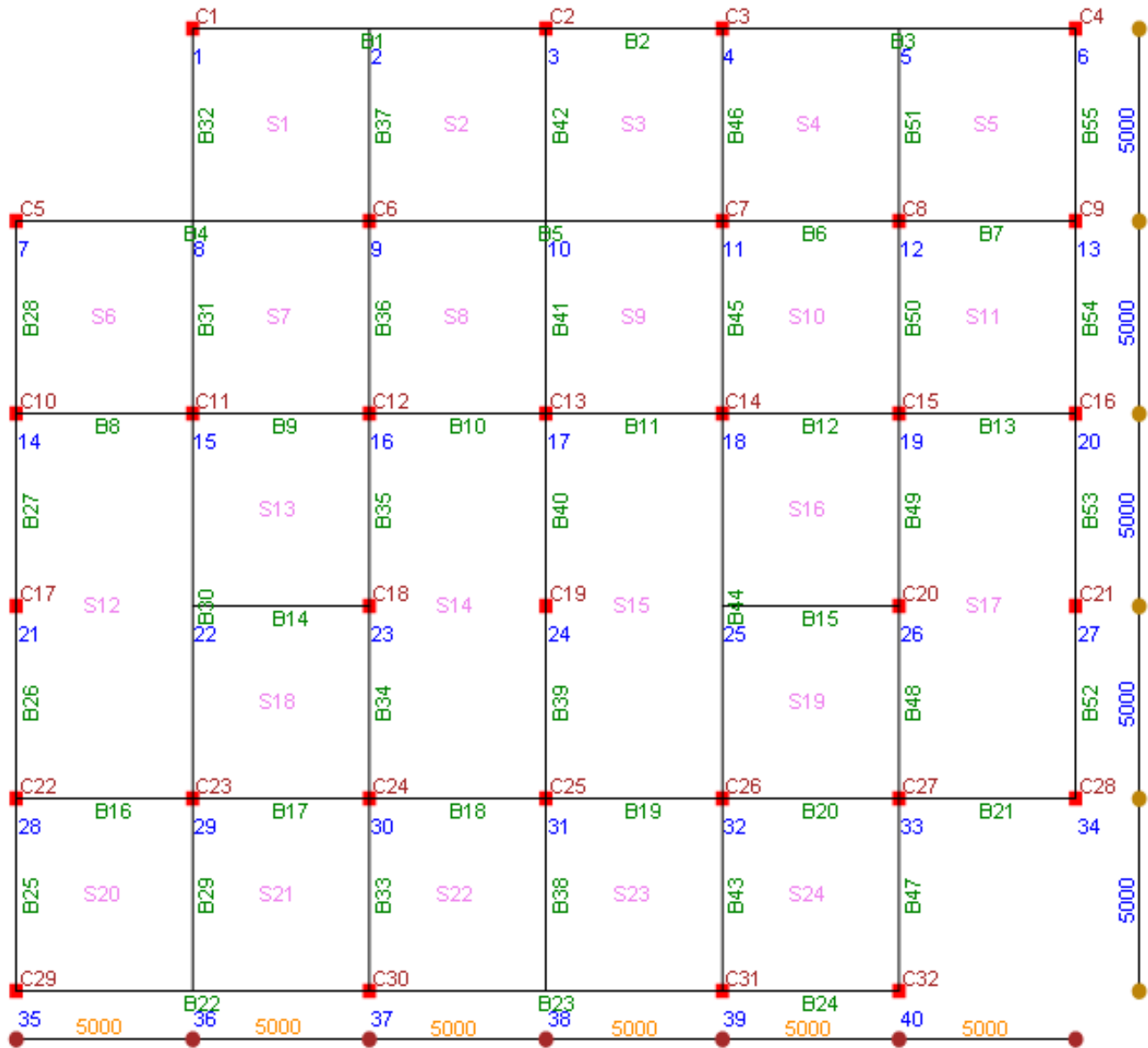
ACTUAL REQUIRED FLOOR PLAN

- 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.

New Project	F i l e s
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN
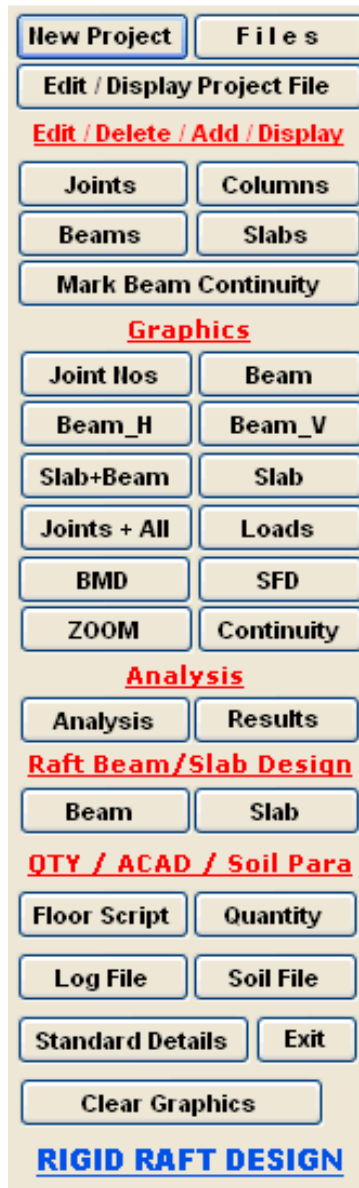
🔴 **When Program starts, the graphics above is displayed. Consider the " New Project Option ".**

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

Create New Project File

Save in: 000RAFT

000Raft_Example_1.rft
000Raft_Example_1Copy.rft
test.rft
test_Copy.rft

My Recent Documents

My Network

File name: Example_1   Save

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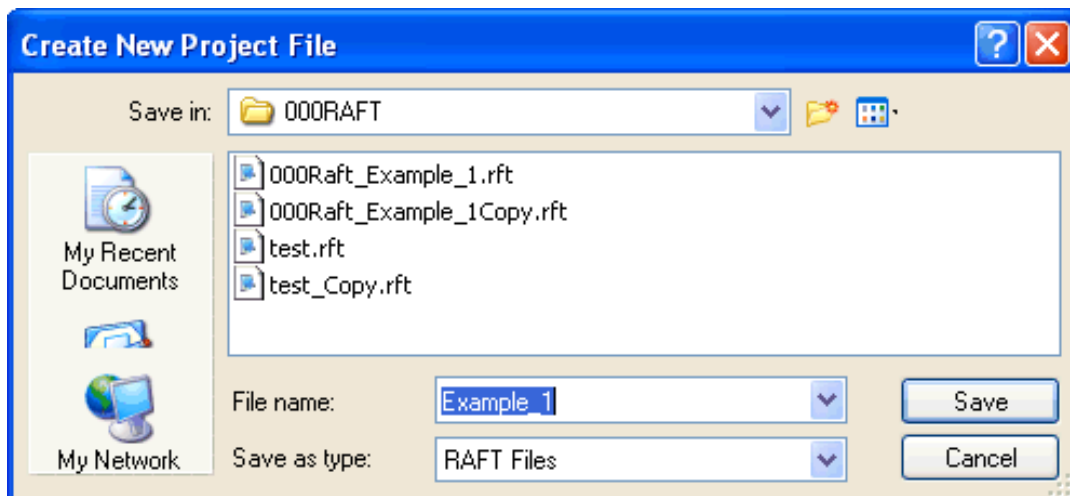
- You must create a separate Folder / Directory to store your files.
  I have created a Directory called " 000RAFT " in C drive to store my Project files.
  Now go to this folder & give a file name to your project. I have given " Example_1 " as the name of my new project file. Click the save button.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

Add Project Details :

File Name : D:\000RAFT\Example_1.rft

Date : 11 August 2009

Organization	Super Civil CD	Default Raft Thickness in MM	250
Project	20 Story Bldg.	Default LL on Grade Slab in T / M2	0.50
Project No.	8912	Thickness of Floor Finish on Grade Slab	40
Building ID	Admin	Thk. of Grade / Basement Slab in MM	100
Floor Width (X Axis- Horiz. Dist.) in MM	30000	Partition Load on G.F. Slab in T / M2	0.10
Floor Length (Y Axis- Vert. Dist.) in MM	25000	Column Dimension Along X-X Axis in MM	750
No. of Vertical Grids (For Horiz. Dist.)	7	Column Dimension Along Y-Y Axis in MM	400
Each for Every Beam and Column		Concrete Rate in Rs / M3	5000
No. of Horizontal Grids (For Vert. Dist.)	6	Including Shuttering	
Each for Every Beam and Column		Reinforcement Rate in Rs / Ton	50000
Concrete Grade	M25	Depth of Water Table Below G.L. in M	1.0
Raft Beam Steel Effective Cover in MM	75	Total Max. Lateral Shear Due to Wind OR Seismic on Raft Top in Ton	100.0
Default Raft Beam Width in MM	300	Coefficient of Friction between Soil/ Raft	0.30
Default Raft Beam Depth in MM	1000	Excavation + Refilling Rate in Rs/M3	150
Net SBC in T/ M2	20	Thickness of Soil Filling up to	1450
Raft Depth below G.L. in M	1.5	Plinth OR Basement Level in MM	

EXIT | PRINT | NEXT PAGE

- 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.

Enter Horizontal Distance Between Vertical Grids in MM Along X – X Axis

Note : Start From the Grid at Left.

File Name : D:\000RAFT\Example_1.rft   Date : 11 August 2009

Enter Grid Distance in MM :   5000

Add Record No. : 6

| Paste | Copy | Prev | Next |

| Last | 1 st | Copy All |

| Clear | Go To Rec |

| Previous Page | Next Page |

Grid Distance	Along X Axis
Distance Between Grids 1 to 2	5000
Distance Between Grids 2 to 3	5000
Distance Between Grids 3 to 4	5000
Distance Between Grids 4 to 5	5000
Distance Between Grids 5 to 6	5000
Distance Between Grids 6 to 7	5000

● 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 Raft 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.

Enter Vertical Distance Between Horizontal Grids in MM Along Y – Y Axis

File Name : D:\000RAFT\Example_1.rft   Date : 11 August 2009

Note : Start From Grid at the Top.

Enter Grid Distance in MM : 5000

Add Record No. : 5

| Paste | Copy | Prev | Next |

| Last | 1 st | | Copy All |

| Clear | | | Go To Rec |

| Previous Page | | | Finish |

Grid Distance	Along Y Axis
Distance Between Grids 1 to 2	5000
Distance Between Grids 2 to 3	5000
Distance Between Grids 3 to 4	5000
Distance Between Grids 4 to 5	5000
Distance Between Grids 5 to 6	5000

● 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.

Please Note :

Use Joint Option to Remove Un-wanted Joint Numbers.
If any Joint is deleted then All Joints will be Re-Numbered.
Use Column Option to Remove Un-wanted Columns.
If any Column is deleted then All Columns will be Re-Numbered.
However Corresponding Joints to be Changed by the User.
Use Column Option to Add/Edit various Column details.
If any beam/Slab is deleted then All Beam/Slabs will be Re-Numbered.
Use Beam/Slab Option to Add, Edit or Delete Beams/Slabs.
Project File Created Successfully.

OK

- 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 Text Boxes which as Grayed Cannot be Edited.

Edit Project Details :

File Name : D:\OOORAFT\Example_1.rft

Date : 11 August 2009

Organization	Super Civil CD	Default Raft Thickness in MM	250
Project	20 Story Bldg.	Default LL on Grade Slab in T / M2	0.50
Project No.	8912	Thickness of Floor Finish on Grade Slab	40
Building ID	Admin	Thk. of Grade / Basement Slab in MM	100
Floor Width (X Axis- Horiz. Dist.) in MM	30000	Partition Load on G.F. Slab in T / M2	0.10
Floor Length (Y Axis- Vert. Dist.) in MM	25000	Column Dimension Along X-X Axis in MM	750
No. of Vertical Grids (For Horiz. Dist.) Each for Every Beam and Column	7	Column Dimension Along Y-Y Axis in MM	400
		Concrete Rate in Rs / M3 Incuding Shuttering	5000
No. of Horizontal Grids (For Vert. Dist.) Each for Every Beam and Column	6	Reinforcement Rate in Rs / Ton	50000
Concrete Grade	M25	Depth of Water Table Below G.L. in M	1.0
Raft Beam Steel Effective Cover in MM	75	Total Max. Lateral Shear Due to Wind OR Seismic on Raft Top in Ton	100.0
Default Raft Beam Width in MM	300	Coefficient of Friction between Soil/ Raft	0.30
Default Raft Beam Depth in MM	1000	Excavation + Refilling Rate in Rs/M3	150
Net SBC in T/ M2	20	Thickness of Soil Filling up to	1450
Raft Depth below G.L. in M	1.5	Plinth OR Basement Level in MM	

EXIT   PRINT

STEP NO. 1 IS OVER.

LEARN RAFT STEP BY STEP

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

In order to Read the AutoCAD drawing in RAFT , 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.
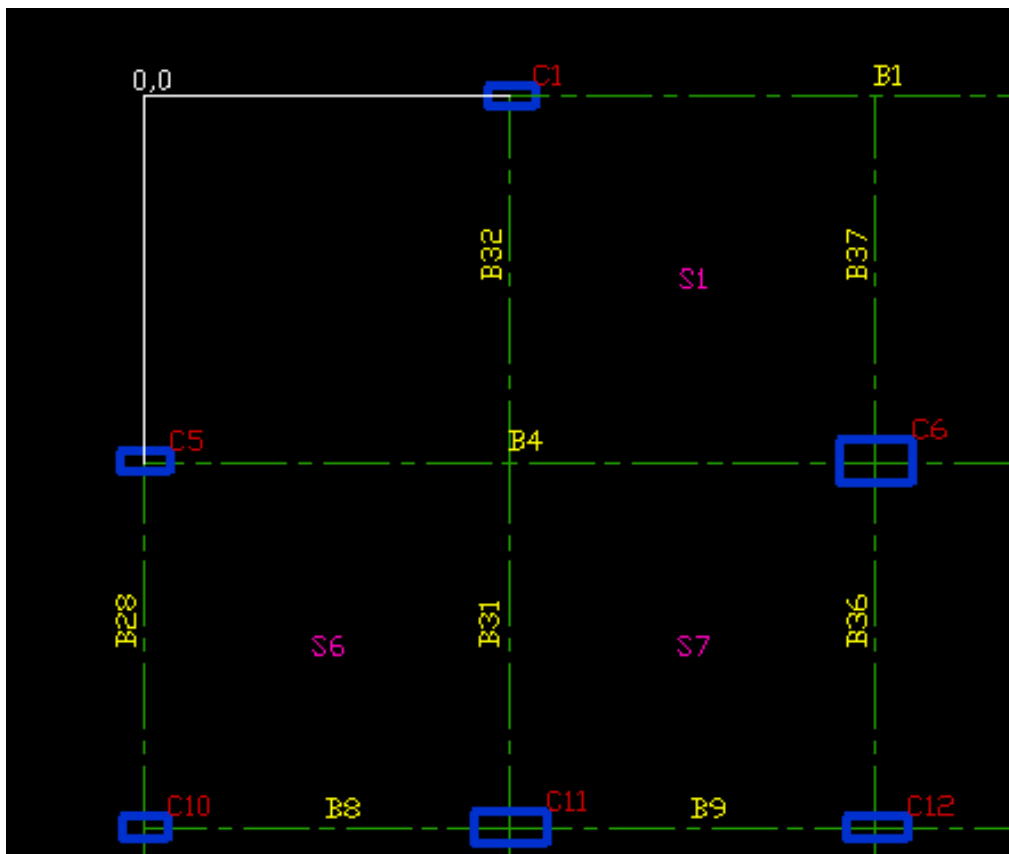
Shown below is a Typical RCC Plan in AutoCAD :
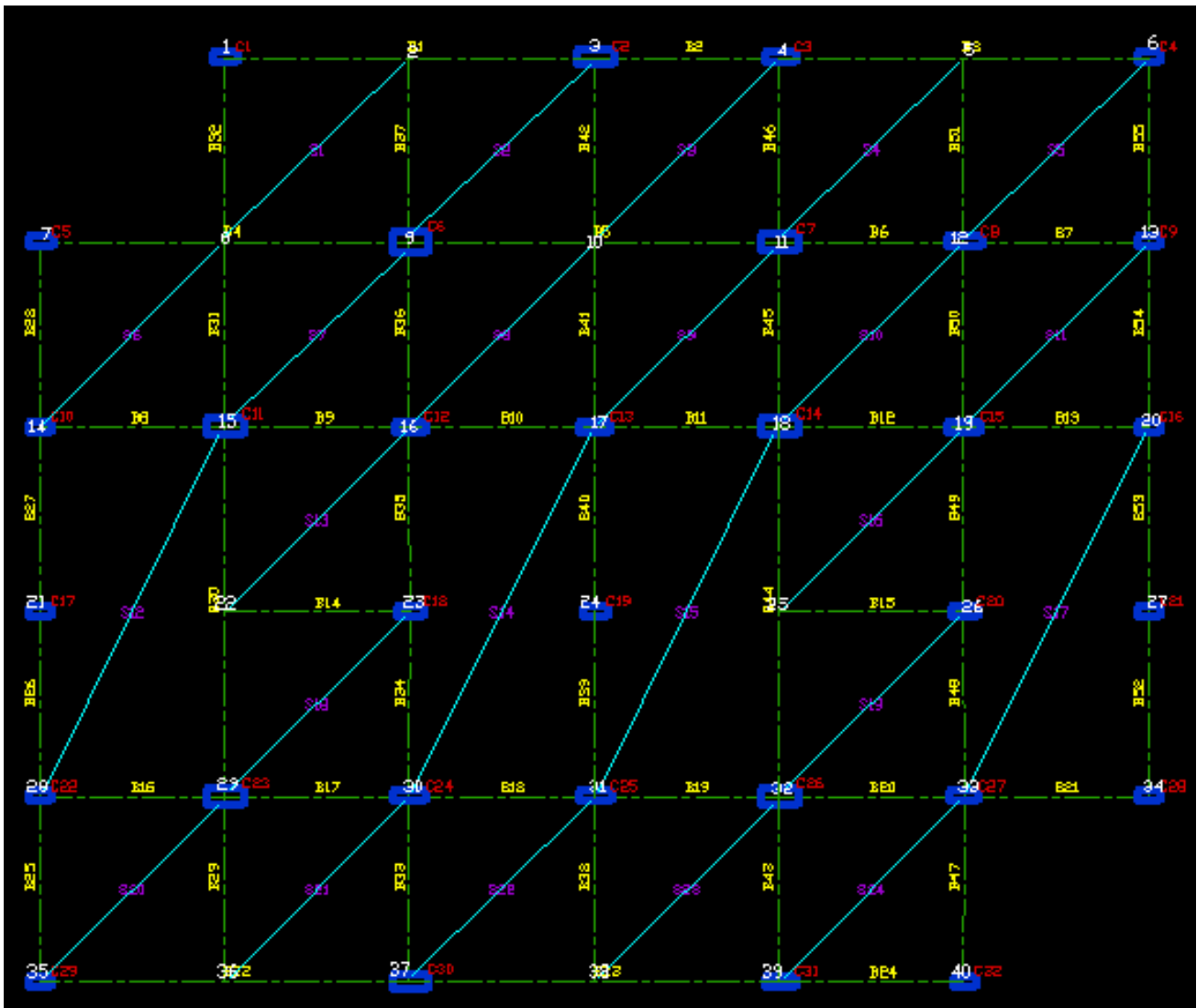
The Layers are explained as follows:

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
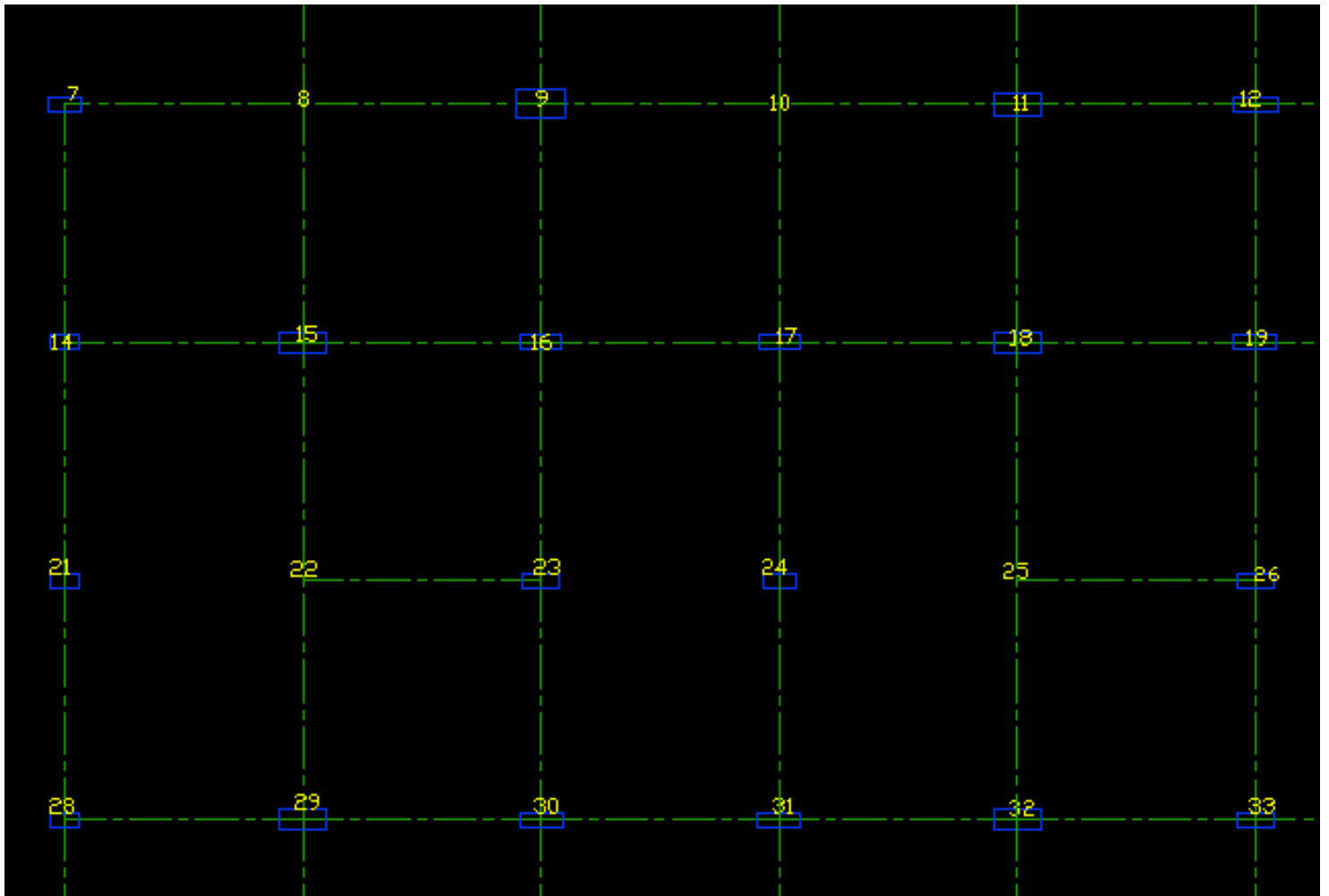
BEAM

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)
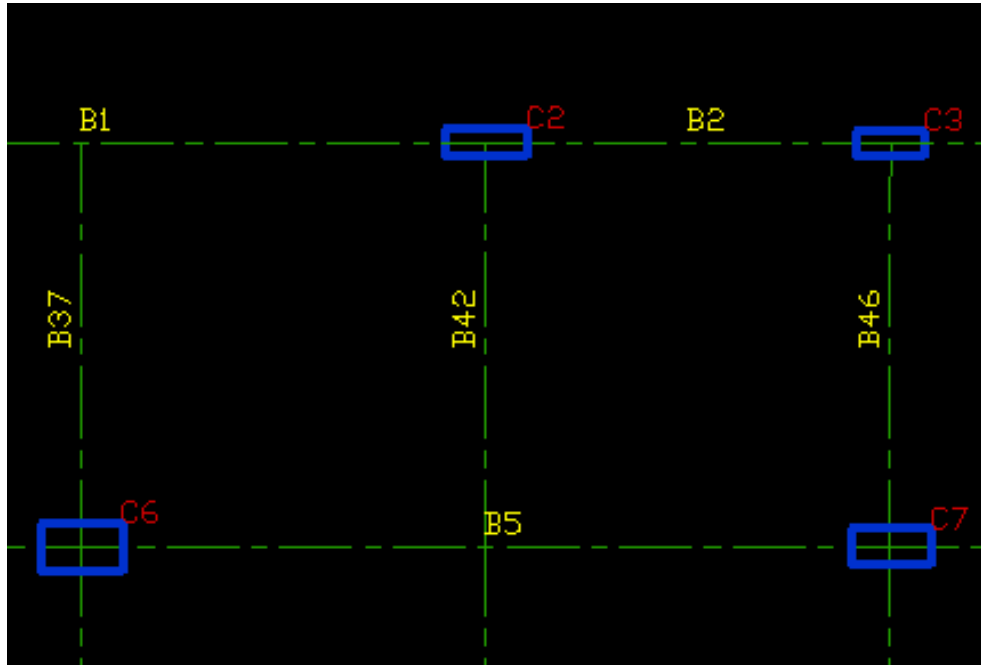
SLAB

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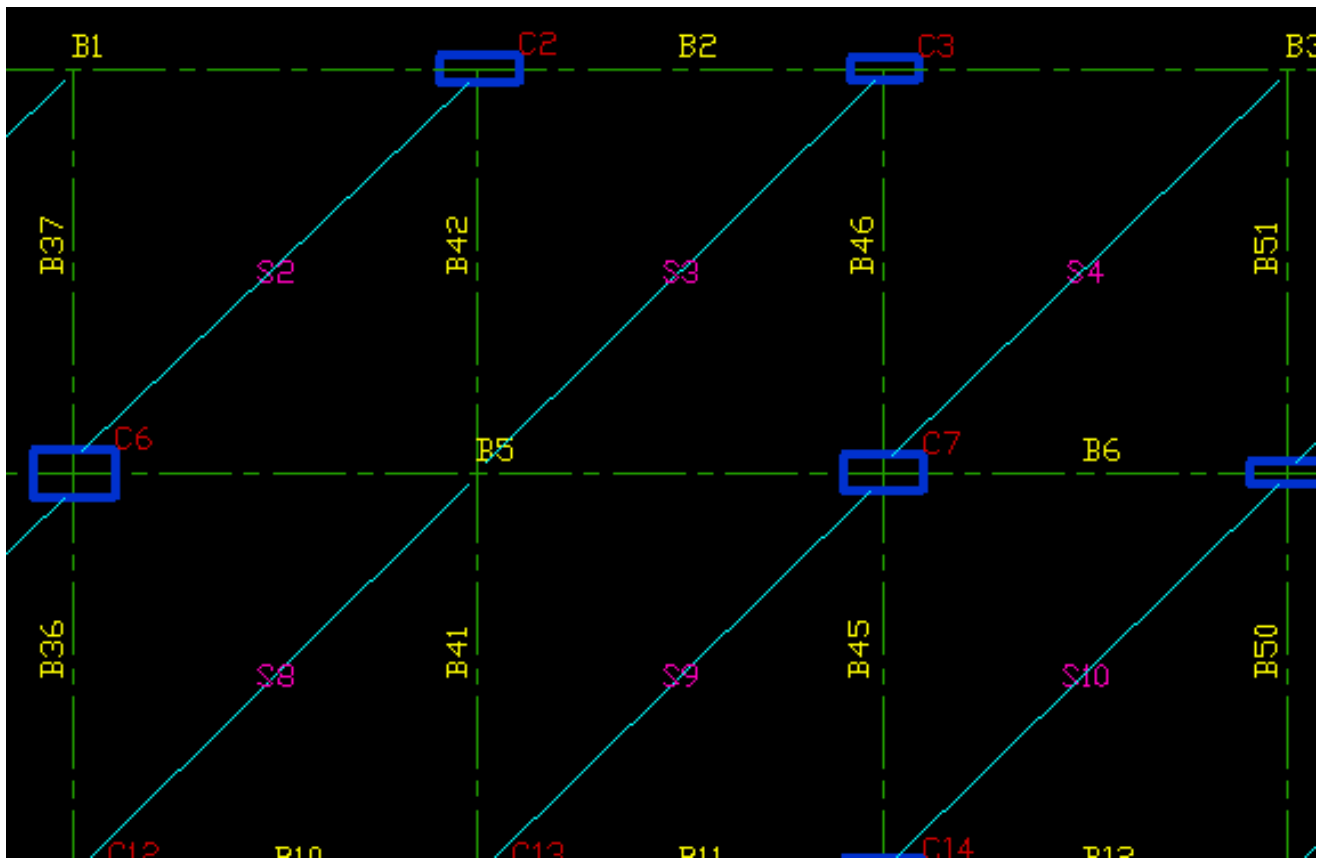
All Slab Numbers should be in the Layer SLABTEXT.
The Slab Text (No.) to be drawn near to the centre of the Slab .
Draw text using 'Single Line Text' option in AutoCAD.
Slab Nos. should not be repeated.
Slabs should be Serially Numbered.
Slab Nos should be prefixed with a "S" (ie. S1, S2)
Slab Text shall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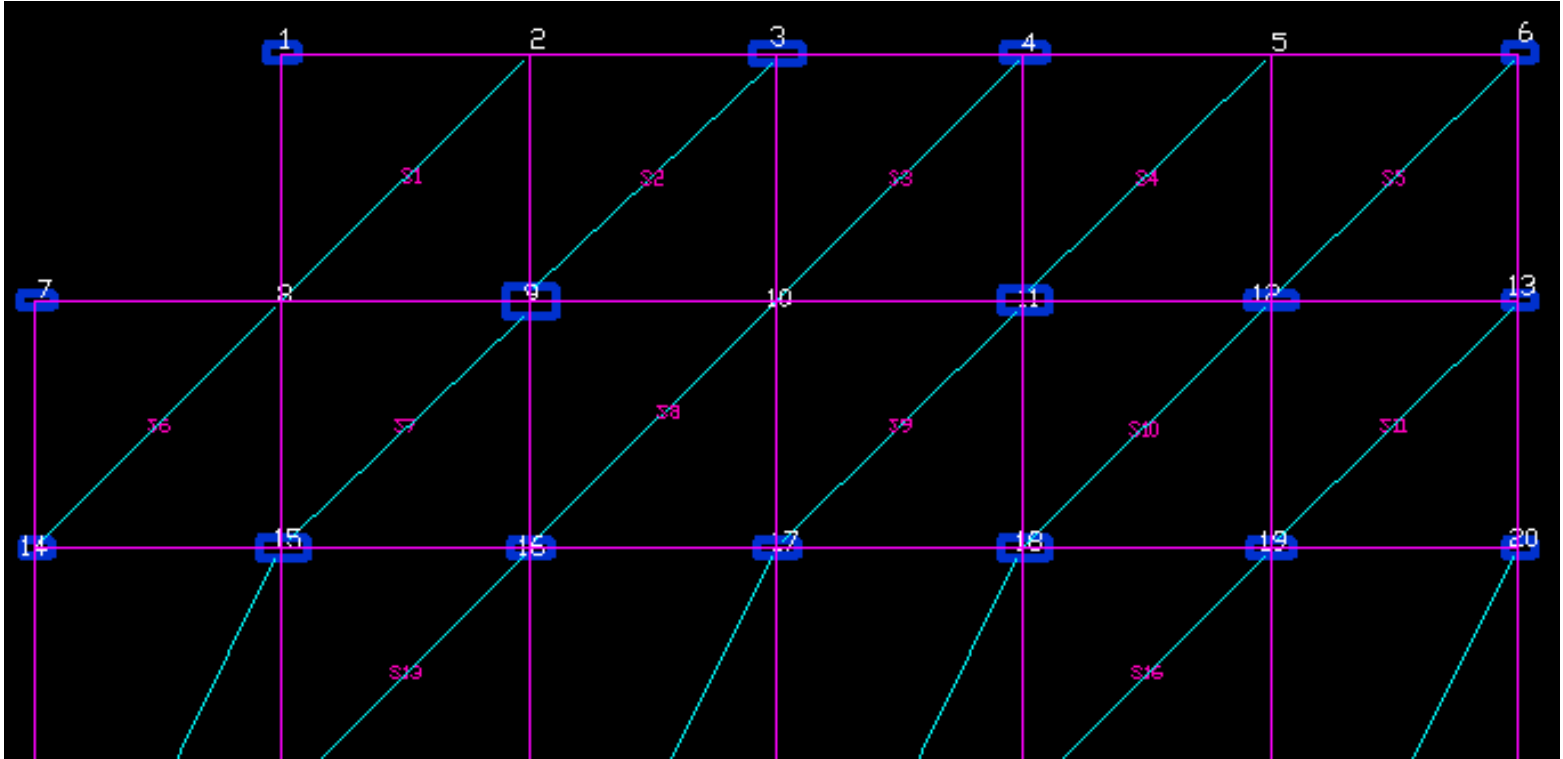
- **Once the drawing is completed, save the drawing in AutoCAD's DXF Format.**

- **Now Start RAFT.**

New Project	F i l e s

Edit / Display Project File

Scan AutoCAD RCC Plan

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs

Mark Beam Continuity

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit

Clear Graphics

RIGID RAFT DESIGN

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

**Click the " Scan AutoCAD RCC Plan" option in the MENU bar.
The following window will open.**
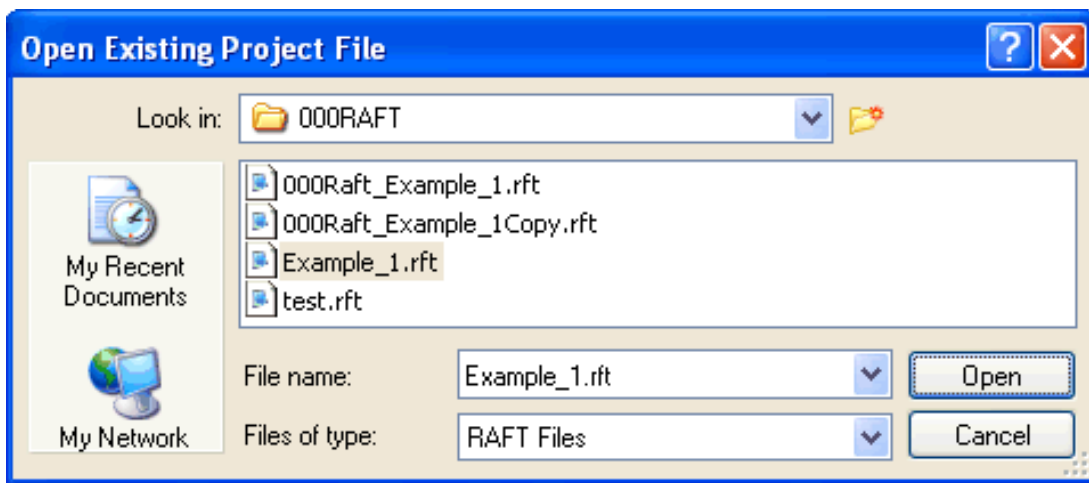
- **Now select Example _1.rft File.**
  Following Graphics will be displayed.
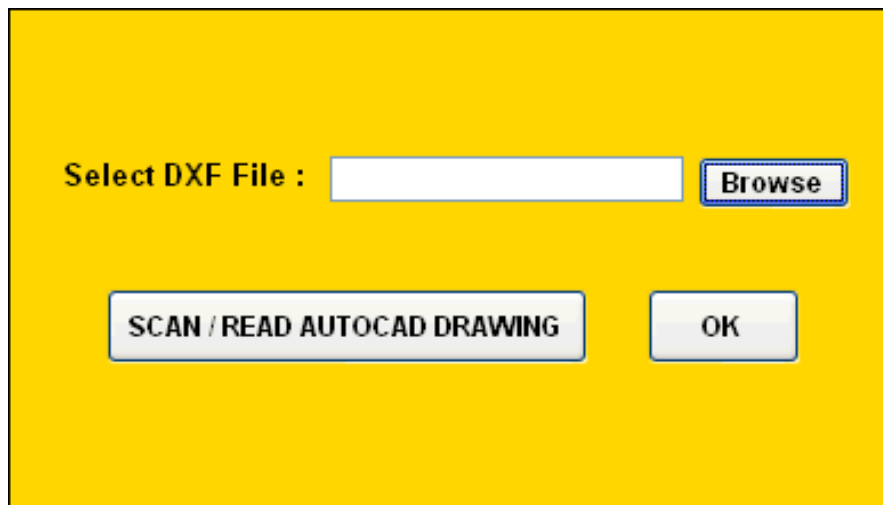
- **Click on browse to select the AutoCAD Drawing.**
  Next click on "Scan/ Read AutoCAD Drawing" button.

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

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.

LEARN RAFT STEP BY STEP

STEP NO. 2 : Automatic Joint Number Creation
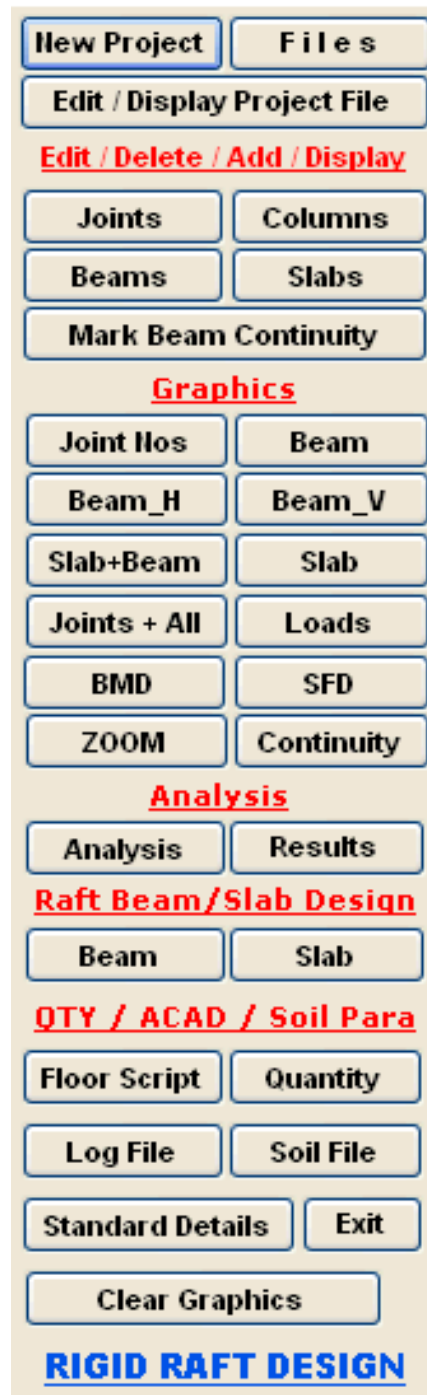
New Project	F i l e s
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

- **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 (Raft 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.**

Right Hand Side (RHS)

Left Hand Side (LHS)

Left Hand Side (LHS)   Right Hand Side (RHS)

C1 1	C2 2	C3 3	C4 4	C5 5	C6 6	C7 7
C8 8	C9 9	C10 10	C11 11	C12 12	C13 13	C14 14
C15 15	C16 16	C17 17	C18 18	C19 19	C20 20	C21 21
C22 22	C23 23	C24 24	C25 25	C26 26	C27 27	C28 28
C29 29	C30 30	C31 31	C32 32	C33 33	C34 34	C35 35
C36 36	C37 37	C38 38	C39 39	C40 40	C41 41	C42 42

5000 5000 5000 5000 5000 5000 5000

5000 5000 5000 5000 5000

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.
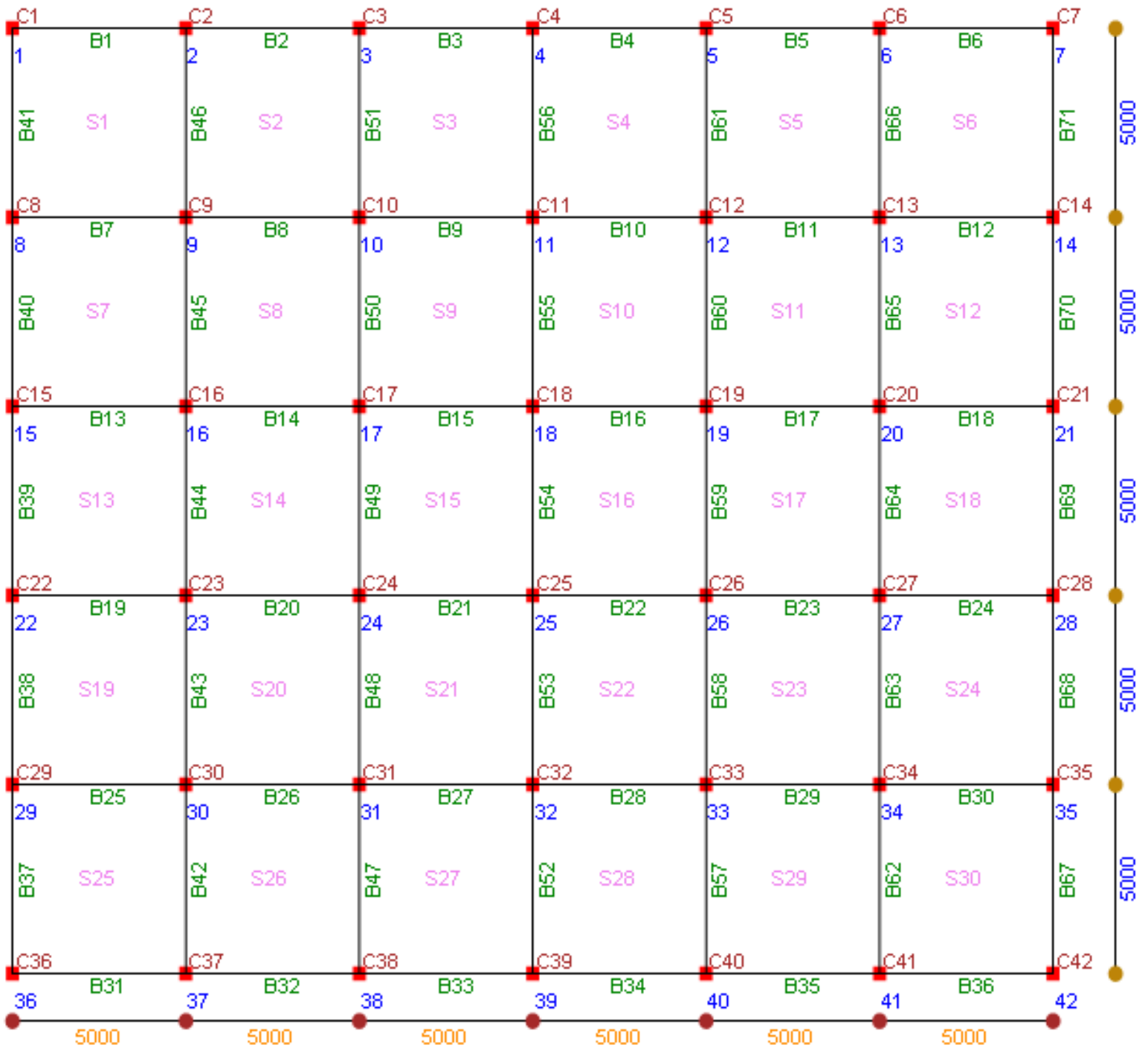
AUTOMATICALLY GENERATED FLOOR PLAN

- 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.
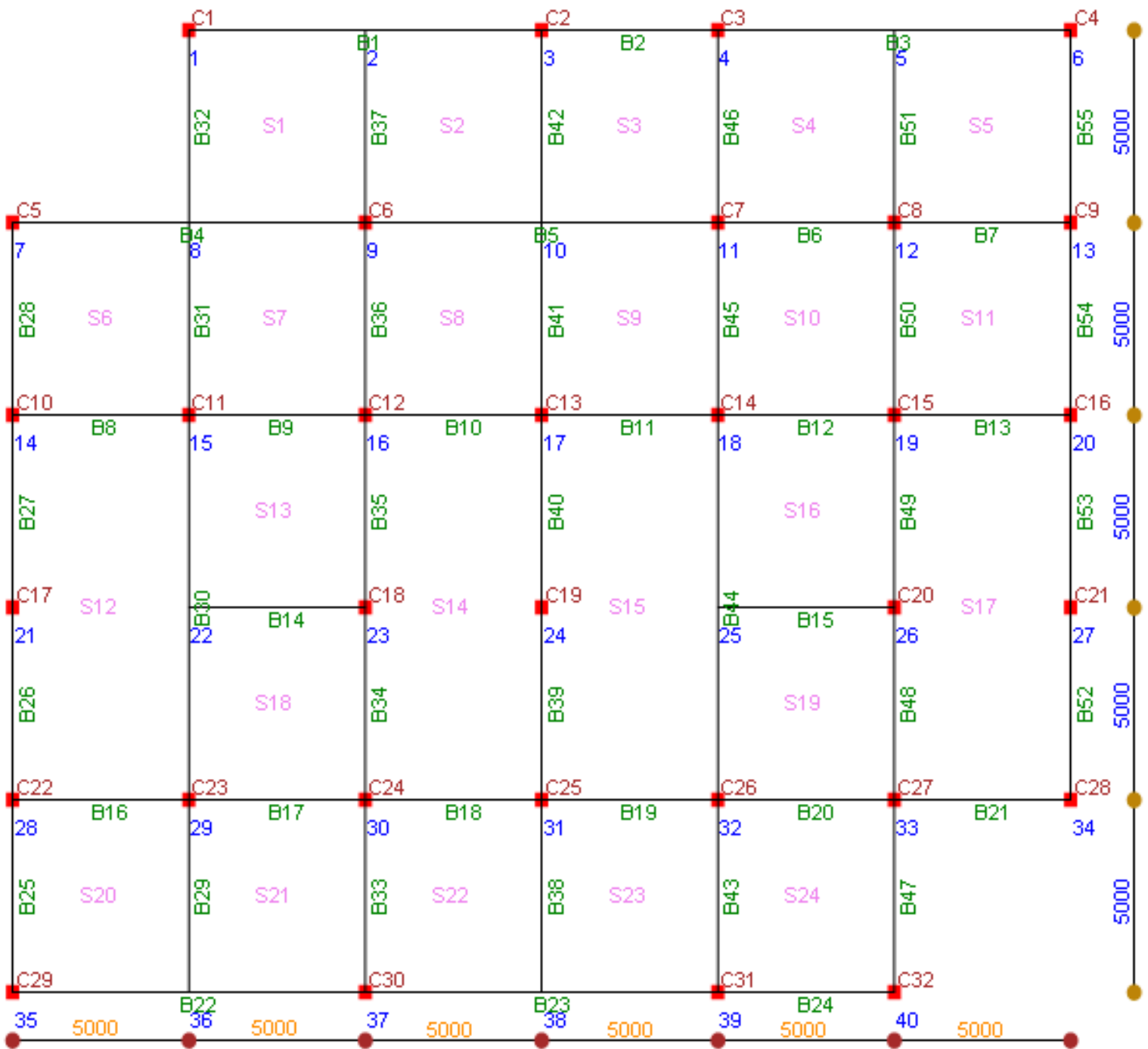
ACTUAL REQUIRED FLOOR PLAN

- Our Actual Raft 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 RAFT 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**
- **Mark Beam Continuity**

Now Click on " Joints " option.

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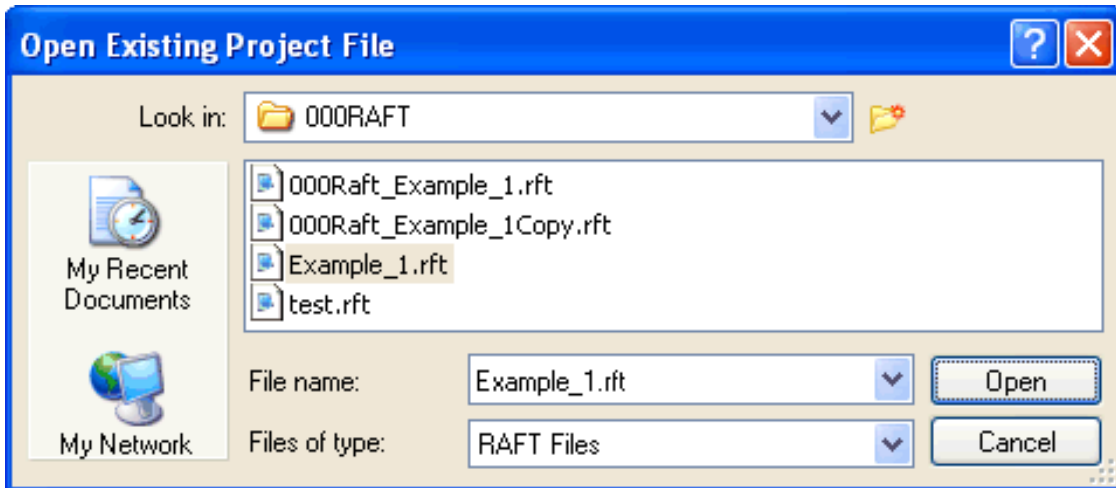
Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.

DISPLAY / EDIT / ADD JOINT NUMBERS

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

- 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.

Critical Error

Data Error : Check Joint / Beam / Slab / Column Data.

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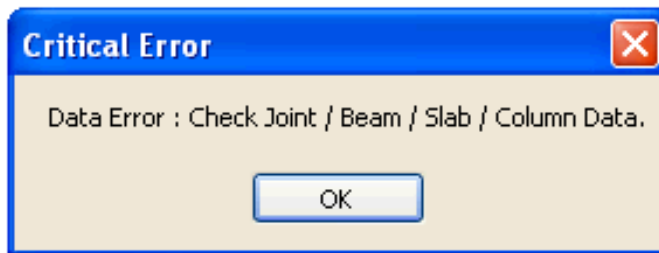
● **Note the Critical Data Error " Check Joint / Beam / Slab / Column data ".**
What it means is that you have not deleted corresponding Beam (s) / Slab (s) / Column (s) which refers to deleted Joints.

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 RAFT STEP BY STEP

## STEP NO. 4 : Delete & Edit Raft Beams

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

- ● **Joints**
- ● **Columns**
- ● **Beams**
- ● **Slabs**
- ● **Mark Beam Continuity**

Now Click on " Beams " option.

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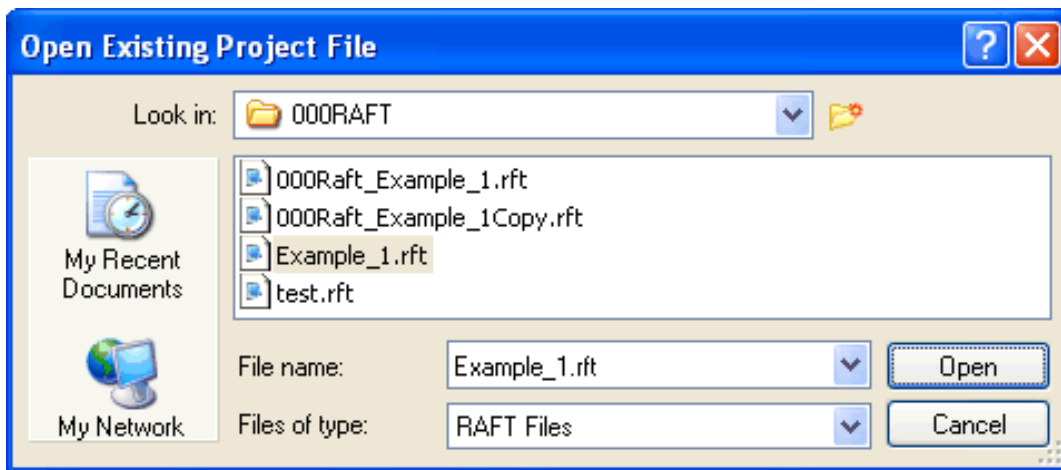
- **Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.**

DISPLAY / EDIT / RAFT BEAM DETAILS

File Name : D:\000RAFT\Example_1.rft   Date : 11 August 2009

Raft Beam #	LHS Joint #	RHS Joint #	Width	Depth	RHS BM	LHS BM	Extra UDL
B51	10	3	300	1000			
B52	39	32	300	1000			
B53	32	25	300	1000			
B54	25	18	300	1000			
B55	18	11	300	1000			
B56	11	4	300	1000			
B57	40	33	300	1000			
B58	33	26	300	1000			
B59	26	19	300	1000			
B60	19	12	300	1000			
B61	12	5	300	1000			
B62	41	34	300	1000			
B63	34	27	300	1000			
B64	27	20	300	1000			
B65	20	13	300	1000			
B66	13	6	300	1000			
B67	42	35	300	1000			
B68	35	28	300	1000			
B69	28	21	300	1000			
B70	21	14	300	1000			
B71	14	7	300	1000			

Record No. : 1 of 71

Beam # **B1**   LHS Joint # **1**   RHS Joint # **2**   Beam Width in MM **300**   Beam Depth in MM **1000**

RHS End BM DL + LL + Wind / EQK in T-M [ ]   LHS End BM DL + LL + Wind / EQK in T-M [ ]   Span **5000**

Read Me   Prev   Next   Copy   Paste   1 st   Last   Additional UDL on Beam in T/M [ ]

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, 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.
I have performed Frame Analysis in either direction & Entered the Maximum End Moments for Beam Nos. 1 and 2 Only.

- 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.
  16. Additional UDL on Raft Beam Could be due to Masonry Wall etc.
  17. Analyze the Structure, assuming Raft Top as Hinged.
  18. Hence there shall be no Moments on Raft Top.
  19. Raft Beam shall Carry Moments due to DL+ LL and also Due to Wind and Seismic Shear.
  20. Structure shall be Analyzed in Either direction.

- 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 RAFT STEP BY STEP

STEP NO. 5 : Delete & Edit Columns

New Project	F i l e s
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

| Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

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

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Mark Beam Continuity**

Now Click on " Columns " option.

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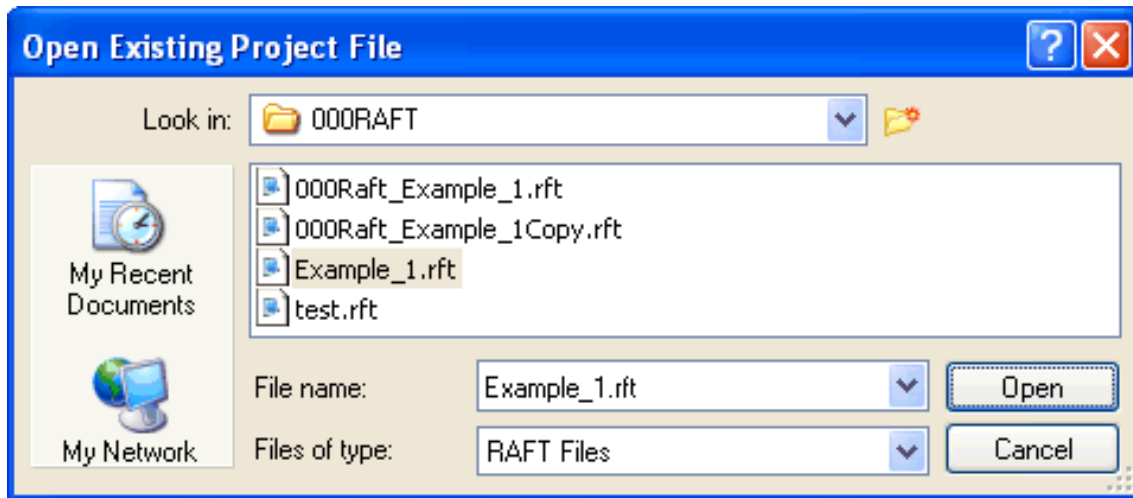
● **Now select " Example _1 File & Press Open Button. Following Graphics will be displayed.**

DISPLAY / EDIT / COLUMN DETAILS

Col. #	Joint #	X-X Dim	Y-Y Dim	DL+LL	DLL+LL+WLX	DL+LL+WLY
C1	1	650	300	255	300	200
C2	3	650	300	255	400	290
C3	4	650	300	255	220	190
C4	6	650	300	255	255	260
C5	7	650	300	255	355	255
C6	9	650	300	255	444	333
C7	11	650	300	255	333	444
C8	12	650	300	255	200	200
C9	13	650	300	255	200	200
C10	14	650	300	255	200	200
C11	15	650	300	255	200	200
C12	16	650	300	255	300	250
C13	17	650	300	255	400	400
C14	18	650	300	255	190	180
C15	19	650	300	255	150	110
C16	20	650	300	255	140	110
C17	21	650	300	255	222	113
C18	23	650	300	255	155	165
C19	24	650	300	255	333	222
C20	26	650	300	255	300	300
C21	27	650	300	255	300	300
C22	28	650	300	255	300	300
C23	29	650	300	255	300	300

Record No. : 1 of 32

Column # `C1`   Joint # `1`   Col. dim. along X_X in MM `650`   Col. dim. along Y_Y in MM `300`

DL + LL on Column in ton `255`   DL + LL + Wind / Seismic Load in Ton (From X-X Direction) `300`

DL + LL + Wind / Seismic Load on Column in Ton (From Y-Y Direction) `200`

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

⬤ **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 "

30

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, DL + LL on Column, DL + LL + Wind / Seismic Load (From X-X Direction) and DL + LL + Wind / Seismic Load in Ton (From Y-Y Direction) can be Added / Edited for individual Columns by clicking at respective Text Boxes.
I have Changed Column Size & added Column Loads for the respective 3 Load Cases.

- 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 < 230 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.
12. Column Dimension along Y-Y means along Y axis.
13. Load on Column Could be + ve OR - ve.
14. Analyze the Structure, assuming Raft Top as Hinged.
15. Hence there shall be no Moments on Raft Top.
16. Raft Beam shall Carry Moments due to DL+ LL and also Due to Wind and Seismic Shear.
17. Structure shall be Analyzed in Either direction.

- 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 RAFT STEP BY STEP

STEP NO. 6 : Delete & Edit Raft Slabs

New Project	F i l e s
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

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

- **Joints**
- **Columns**
- **Beams**
- **Slabs**
- **Mark Beam Continuity**

Now Click on " Slabs " option.

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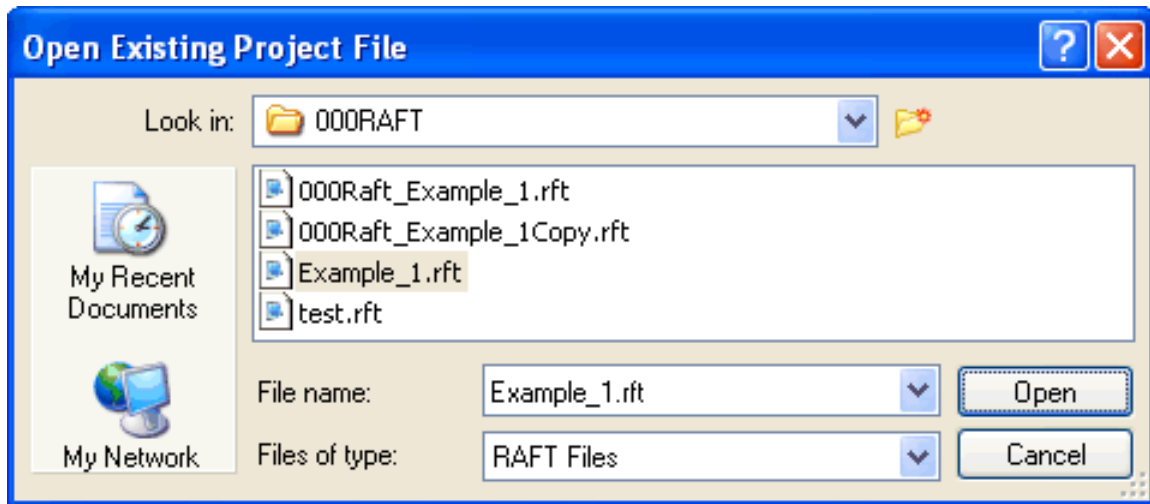
Now select " Example _1 File & Press Open Button.
Following Graphics will be displayed.

DISPLAY / EDIT / RAFT SLAB DETAILS

Raft Slab #	Left Btm Joint #	Right Top Joint #	Raft Slab Thickness
S1	8	2	300
S2	9	3	300
S3	10	4	300
S4	11	5	300
S5	12	6	350
S6	14	8	300
S7	15	9	300
S8	16	10	300
S9	17	11	300
S10	18	12	300
S11	19	13	300
S12	28	15	550
S13	22	16	300
S14	30	17	660
S15	31	18	660
S16	25	19	300
S17	33	20	550
S18	29	23	300
S19	32	26	300
S20	35	29	350
S21	36	30	300
S22	37	31	300
S23	38	32	300

Record No. : 1 of 24

Slab # `S1`   Left Bottom Joint # `8`   Right Top Joint # `2`   Raft Slab Thickness in MM `300`

Read Me	Prev	Next	Copy	Paste	Last	1 st	Copy All

Go To Rec	Update	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.

● **Raft Slab Thickness can be Added / Edited for individual Slabs by clicking at respective Text Boxes.**
Slab Spans in either direction is displayed in Golden Color.

● **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. Max. Slab Joint # cannot > Max. Joint File #.
  5. Use Add Button to Append Record.
  6. Use Update Button to Re-Number & Save Your Work.
  7. Max. Slab Number = Max. Record Number.
  8. Slabs shall be Numbered Serially.
  9. Slabs Joint (LB/RT) Numbers cannot be < = 0.0
  10. Slab Density = 2.5 T/M3.
  11. 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 Continuity to the Raft Beams.

STEP NO. 6 IS OVER.

LEARN RAFT STEP BY STEP

STEP NO. 7 : Add & Edit Raft Beam Continuity

New Project	Files
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

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

- Joints
- Columns
- Beams
- Slabs
- Mark Raft 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

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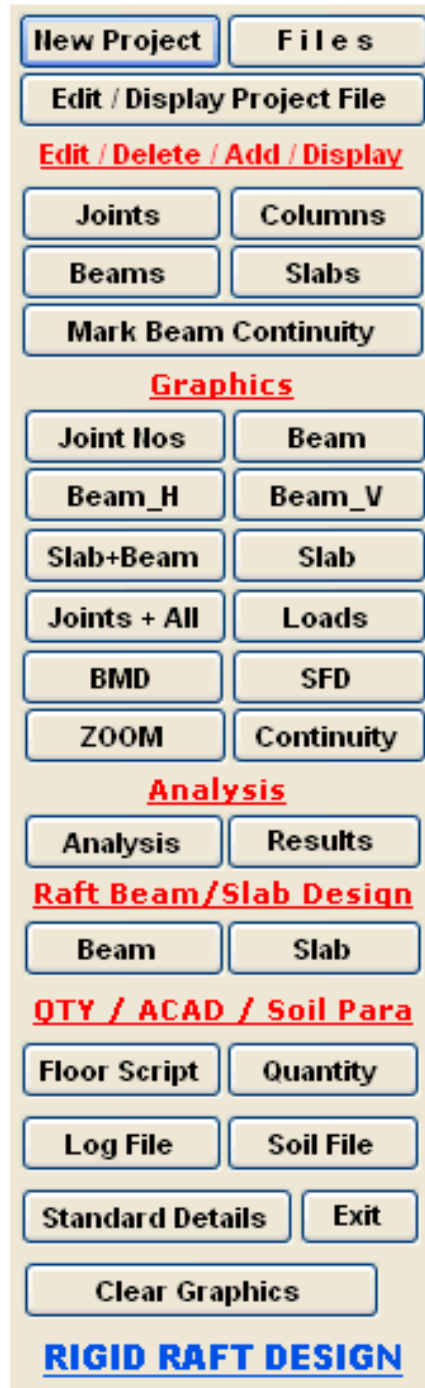
In case you would like to EXIT program after partial Adding / Editing, first use " Update " button to save your work & then click " OK " button. The program will ask you about exiting, click Yes & quit.

- **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 # 7.**
  In the next step we will Check our Data Input Graphically.

STEP NO. 7 IS OVER.

LEARN RAFT STEP BY STEP

STEP NO. 8 : Data Checking Through Graphics

● A User should thoroughly check Data Input at all stages. During Adding / Editing Data through tables, Raft 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 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 (End Moments).
● Raft Analysis, Beam & Slab Design not performed after Editing / Adding Geometry or Loads.

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	F i l e s

Edit / Display Project File

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs

Mark Beam Continuity

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit

Clear Graphics

RIGID RAFT DESIGN

🔴 **When Program starts, the Menu above is displayed. Under the <u>Graphics</u> 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.

C1 1   2   C2 3   C3 4   5   C4 6

C5 7   8   C6 9   10   C7 11   C8 12   C9 13

C10 14   C11 15   C12 16   C13 17   C14 18   C15 19   C16 20

C17 21   22   C18 23   C19 24   25   C20 26   C21 27

C22 28   C23 29   C24 30   C25 31   C26 32   C27 33   C28 34

C29 35   36   C30 37   38   C31 39   C32 40

5000   5000   5000   5000   5000   5000   5000

Vertical dimensions: 5000, 5000, 5000, 5000, 5000

- 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.
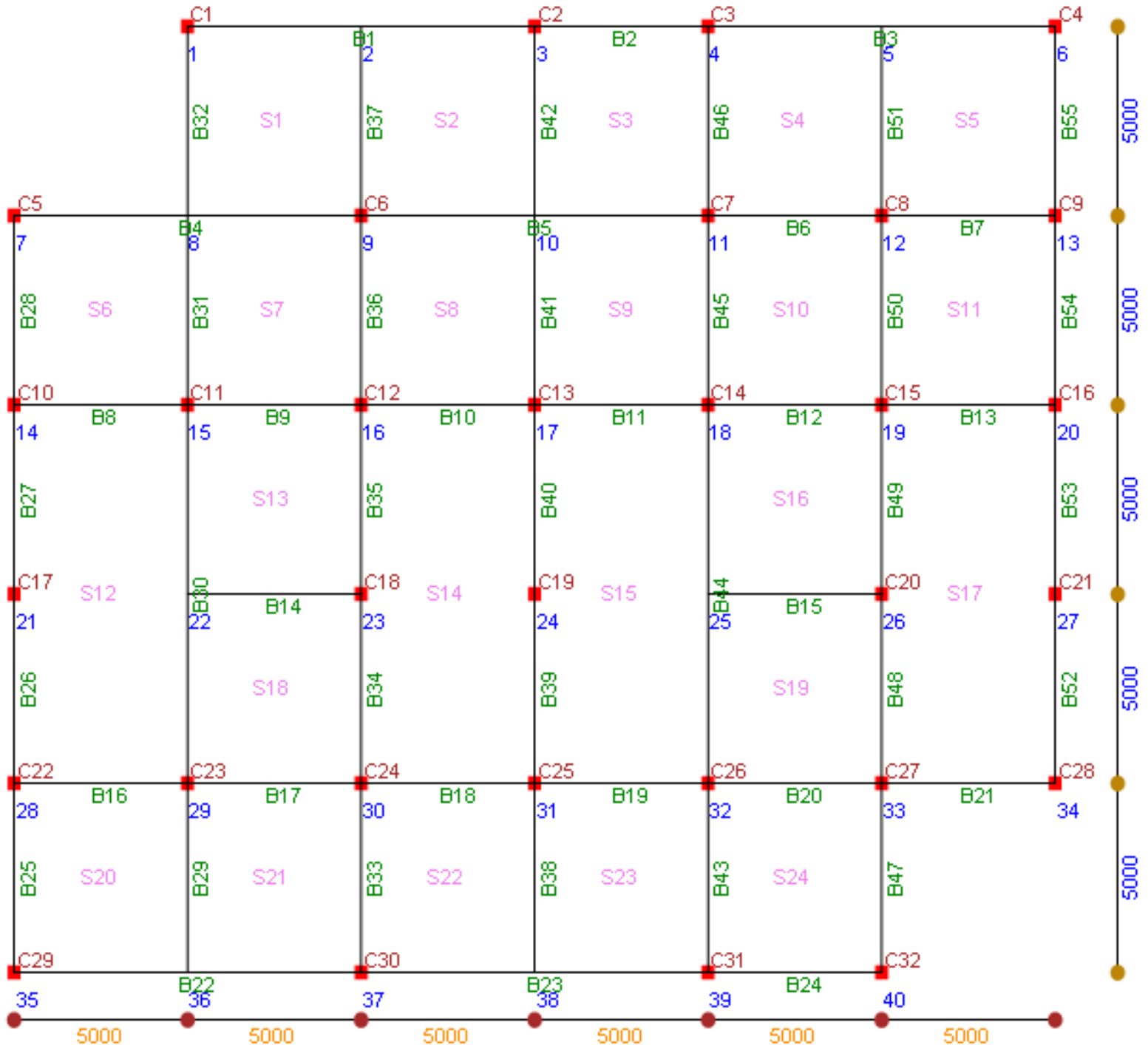
- **Now Click Continuity Button.**
  Following Graphics is displayed.
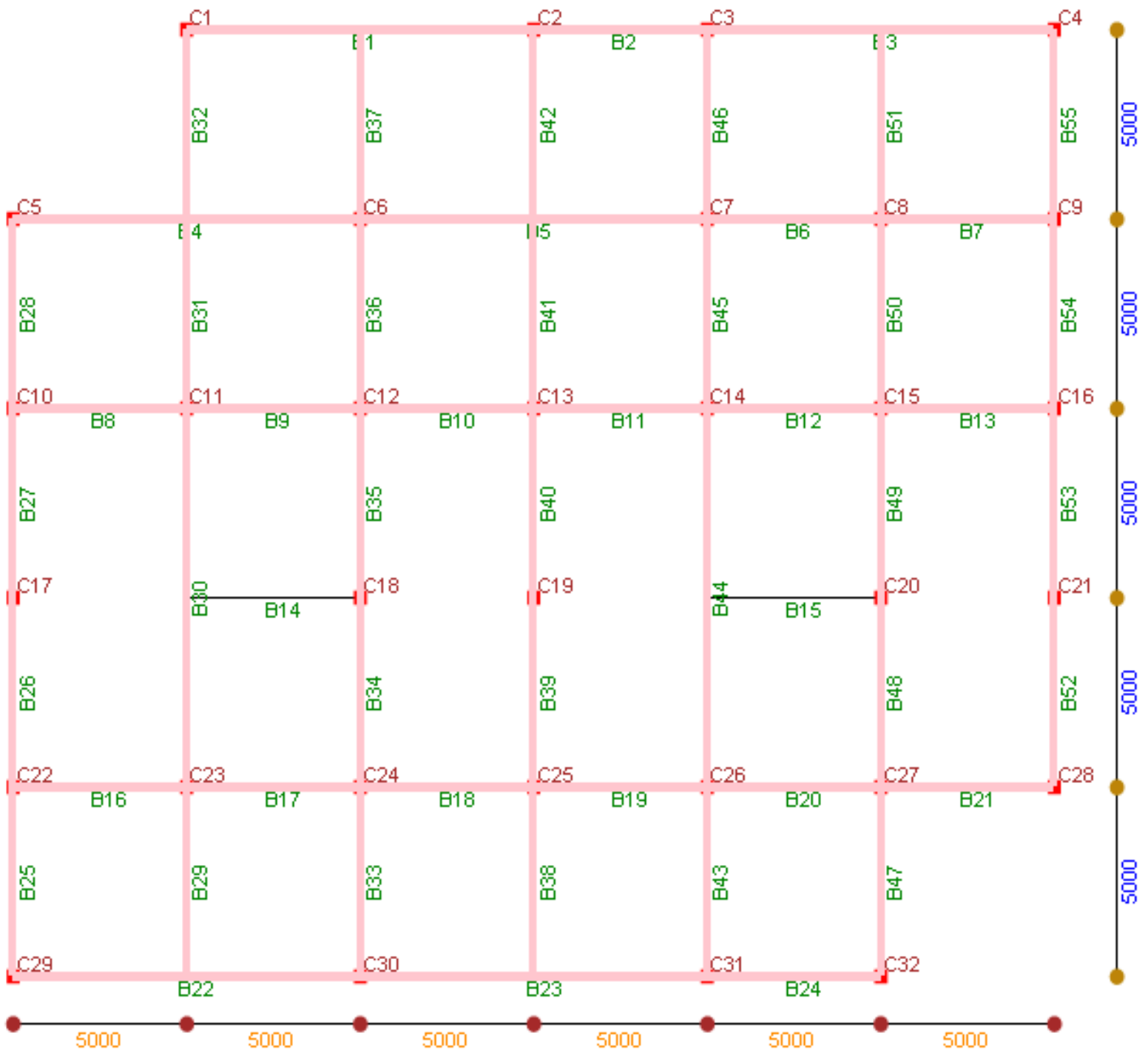
Continuity of Beams Marked in Pink

● 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.
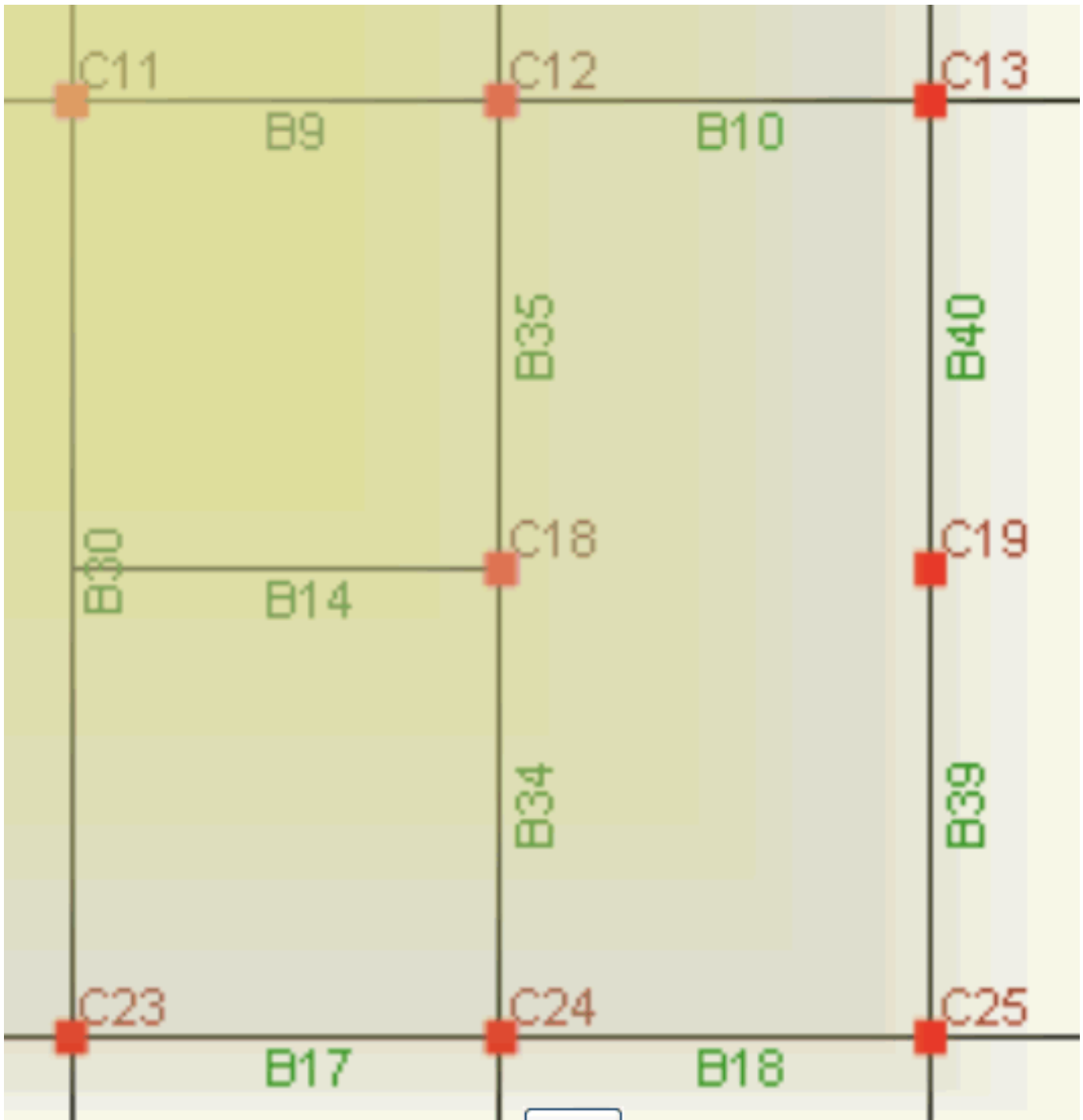
● **Note that Graphics Display of :**

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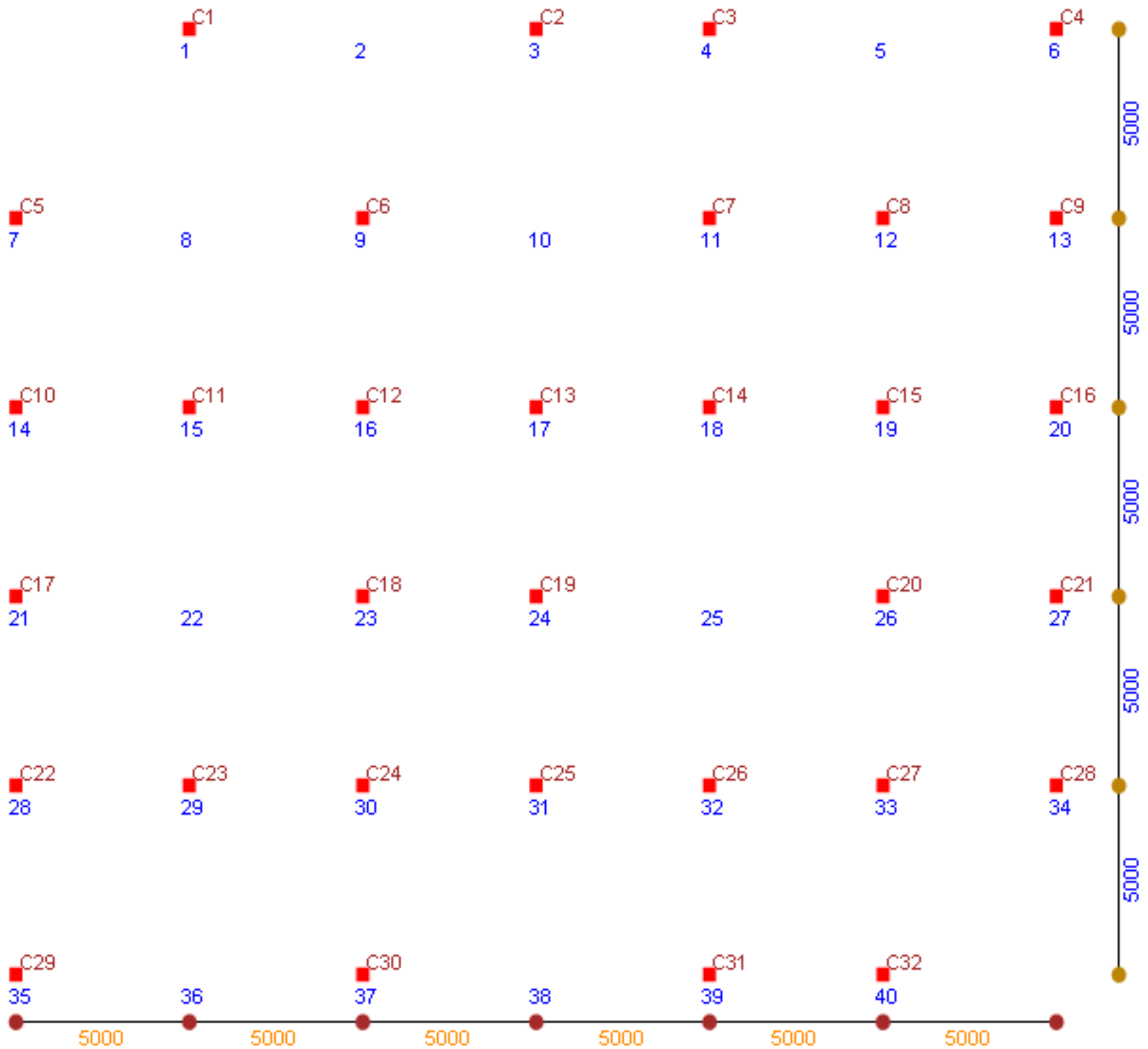● **SFD** (Display of shear Force Diagram, to be used after Analysis, Design & Quantity options have been successfully Run.

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

STEP NO. 8 IS OVER.

LEARN RAFT STEP BY STEP

STEP NO. 9 : Analysis & Its Results + Soil File Option

New Project	F i l e s
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

● 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.
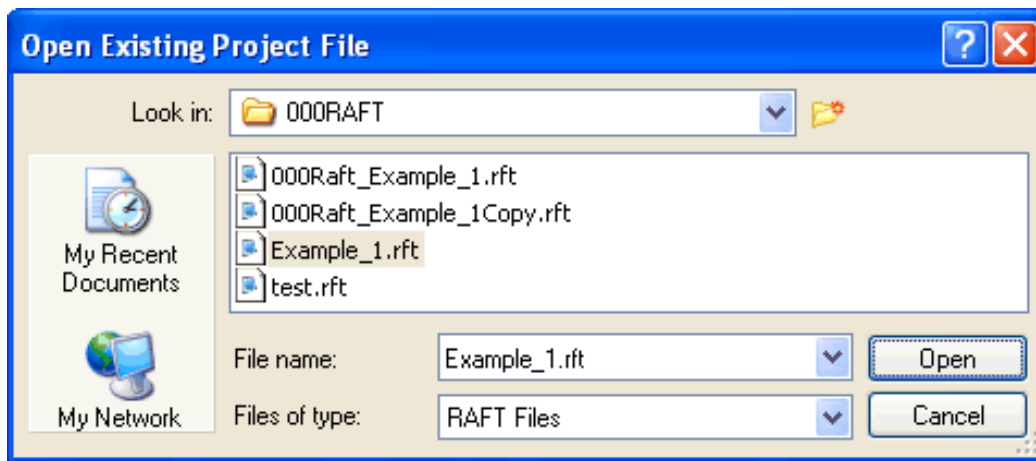
- **Now select " Example _1 File & Press Open Button.**
  The Analysis will commence. A window will open & it will indicate number of Joints, Columns, Beams Slabs to be analyzed. The Analysis will take time & will depend up on the file size & computers RAM memory. Minimum Computer RAM memory of 1 GB 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.**
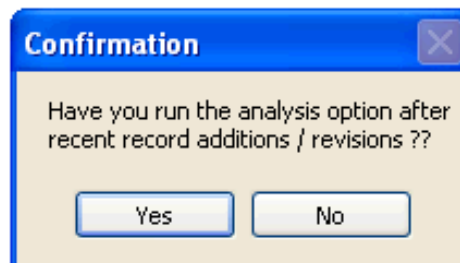
- **This is a very Important Message. In case a user has edited or added any Joint / Column / Beam or Slab Member after performing analysis then he should re-perform the analysis, else old (in-correct) results will be displayed.**
  Click " Yes " if you have not revised any member after analysis or click " No " if you are not sure.
  If " Yes " is clicked then following graphics will be displayed.
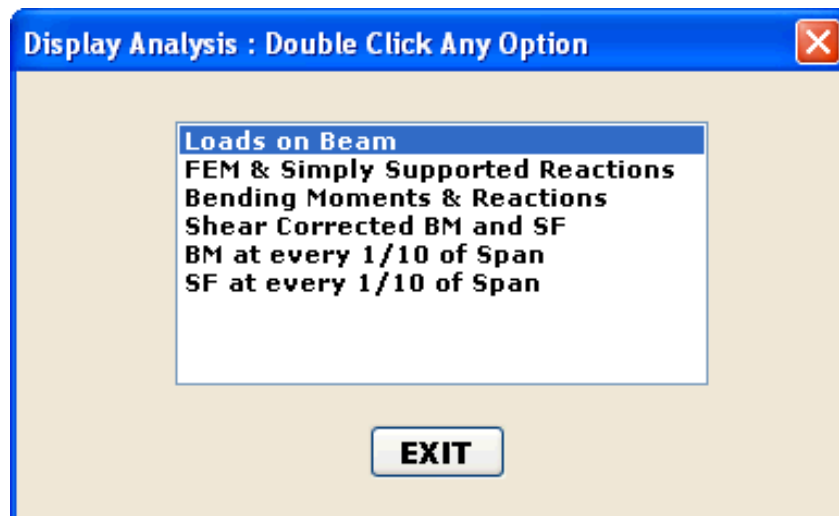
- **Now Double Click on " Loads on Beam " Option. A new window will open displaying various Loads on Beams. Click on " Read Me " button, following important messages are displayed.**

  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 Loads are Reaction of Secondary beam.
  6. Dist : is distance of Point Load from Left.
  7. NEAR_INT : is Slab Load in T/M Near to LHS of Beam.
  8. NEAR_DIST : is Slab Load Distance in M Near to LHS.
  9. FAR_INT : is Slab Load in T/M Far from LHS.
  10. 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. LHS Support in T-M.
  2. RHS Support in T-M.
  3. LHS Reaction in Tons.
  4. RHS Reaction in Ton.
  5. Span Bending Moment in T-M.
  6. Distance of Span 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/Seismic 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, Span BM & Its Distance from Left.

RAFT BEAM BENDING MOMENTS AND REACTIONS

Beam #	LHS Supp BM	RHS Supp BM	LHS Reaction	RHS Reaction	Mid Span BM	Dist	Load Case
1	0	159.127	105.246	137.072	333.593	5	DL + LL
1	25	179.127	100.746	141.572	336.093	5	DL + LL + WL1
1	-25	139.127	109.746	132.572	331.093	5	DL + LL + WL2
2	-159.127	159.285	40.066	40.13	-90.245	2.5	DL + LL
2	-126.127	194.285	26.466	53.73	-88.367	2.1	DL + LL + WL1
2	-192.127	124.285	53.666	26.53	-86.429	3	DL + LL + WL2
3	-159.285	0	137.173	105.315	333.818	5	DL + LL
3	-159.285	0	137.172	105.316	333.812	5	DL + LL + WL1
3	-159.285	0	137.172	105.316	333.812	5	DL + LL + WL2
4	0	642.779	96.535	270.319	308.788	5	DL + LL
4	0	642.779	96.535	270.319	308.788	5	DL + LL + WL1
4	0	642.779	96.535	270.319	308.788	5	DL + LL + WL2
5	-642.779	227.203	307.297	224.181	483.931	5	DL + LL
5	-642.779	227.203	307.297	224.181	483.931	5	DL + LL + WL1
5	-642.779	227.203	307.297	224.181	483.931	5	DL + LL + WL2
6	-227.203	105.033	111.14	62.272	-15.631	2.9	DL + LL
6	-227.203	105.033	111.14	62.272	-15.631	2.9	DL + LL + WL1
6	-227.203	105.033	111.14	62.272	-15.631	2.9	DL + LL + WL2

Record No. : 1 of 165

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

● **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.
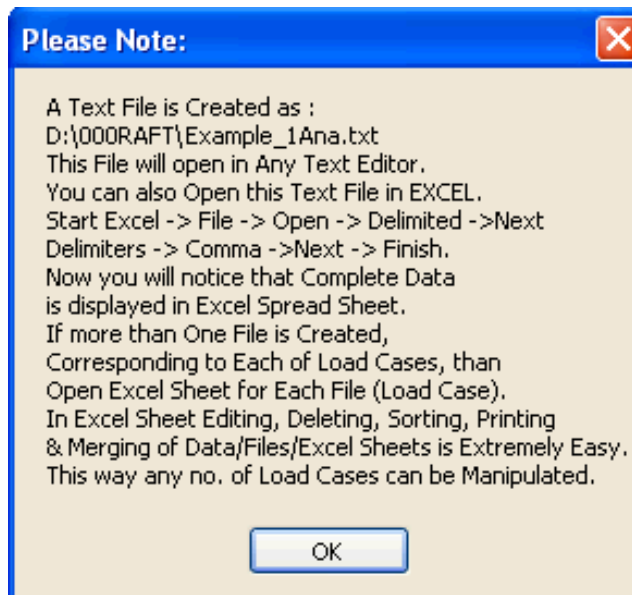
Please Note:

A Text File is Created as :
D:\000RAFT\Example_1Ana.txt
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.

OK

● 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/Seismic 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 Click on Soil File Option, situated near the base of Main Menu. After Analysis this the most important option, as it displays Raft Geometry, Area, Inertia, Bearing Pressures, Raft Stability, Total Loads etc. Following crucial info is displayed.**

● **SOIL PRESSURES ON RIGID RAFT FOUNDATION :**
^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^^
Max. Raft Dimension Along X-X Axis in MM = 30000
Max. Raft Dimension Along Y-Y Axis in MM = 25000
CG of Raft Along X-X Axis in MM = 15000
CG of Raft Along Y-Y Axis in MM = 12500
Area of Raft in M2 = 700
Moment of Inertial of Raft IXX in M4 = 33958.34
Moment of Inertial of Raft IYY in M4 = 47291.66
Extreme Fiber Distance Along X-X Axis in M = 15
Extreme Fiber Distance Along Y-Y Axis in M = 12.5
CG of Column Loads Along X-X for DL + LL Case (1) in MM = 15625
CG of Column Loads Along Y-Y for DL + LL Case (1) in MM = 12812.5
CG of Column Loads Along X-X for DL + LL + WL/EQ Case (2) = 15055.37
CG of Column Loads Along Y-Y for DL + LL + WL/EQ Case (2) = 12681.39
CG of Column Loads Along X-X for DL + LL + WL/EQ Case (3) = 15464.66
CG of Column Loads Along Y-Y for DL + LL + WL/EQ Case (3) = 13059.17
Column Load on Raft in Ton (DL + LL) = 8160
Column Load on Raft in Ton (DL + LL + WL/EQ : X_X) = 8669
Column Load on Raft in Ton (DL + LL + WL/EQ : Y_Y) = 7909
Weight of Raft Slab in Ton = 683.75
Weight of Raft Beams in Ton = 527.125
Total Additional UDL on Raft Beams in Tons = 105
Weight of Soil in Ton = 1573.47
Wt. of Grade/Basement Slab + FF + LL + Partition = 511
Total Grade Weight Acting on CG of Raft = 3400.345
Upward Thrust of Water in Ton = 0
Total Vertical Load on Raft in Ton (DL + LL) = 11560.34
Total Vertical Load on Raft (DL + LL + WL/EQ : X_X) = 12069.34
Total Vertical Load on Raft (DL + LL + WL/EQ : Y_Y) = 11309.34
BM @ X_X in T-M (DL + LL) = 7225.216
BM @ Y_Y in T-M (DL + LL) = 3612.608
BM @ X_X in T-M (DL + LL + WL/EQ : X_X) = 668.281
BM @ Y_Y in T-M (DL + LL + WL/EQ : X_X) = 2189.301
BM @ X_X in T-M (DL + LL + WL/EQ : Y_Y) = 5255.002
BM @ Y_Y in T-M (DL + LL + WL/EQ : Y_Y) = 6323.878
Lateral Frictional Resistance of Foundation in Ton = 4624.138
Maximum Gross Soil Pressure Under DL + LL in T/M2 = 20.136
Minimum Gross Soil Pressure in T/M2 = 10.393
Maximum Gross Soil Pressure Under DL + LL + WL/EQ - X_X in T/M2 = 18.26
Maximum Gross Soil Pressure Under DL + LL + WL/EQ - Y_Y in T/M2 = 20.151
Net Design Pressure on Raft in T/M2 = 14.473
Max. Permissible Column Spacing in M along X-X Dir (Rigidity Consideration) = 13.86
Max. Permissible Column Spacing in M along Y-Y Dir (Rigidity Consideration) = 13.08
Notes : Extend Raft Slab at least 300 MM from Beam / Column Face.
All Column Loads Shall be Design Loads, i.e. Without Load Factor.
Design Pressure is Reduced by 0.80 Factor Under DL + LL + WL/EQ Condition.
Keep Depth of All Raft Beams Uniform as far as Possible.
Suggested Minimum Raft Beam Size is 300 x 600 MM.
Minimum of .2 % of Reinforcement is Provided in Raft Beams & Slabs.
Effective Cover to Raft is assumed as 50 MM.
Provide a minimum of 40 MM Blinding Layer (PCC) below RC Raft Slab.
40 MM thick Blinding Layer (PCC) Shall be atleast of M 7.5 Grade.
Provide 300 x 300 MM Haunch between All Raft Slabs & Beams.
In absence of 300 x 300 MM Haunch, Raft Slab may not be safe in Shear.
Refer IS 2950:Part/1/1981 For Rigid Raft Design/Col. Spacing/Relative Stiffness Factor.

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

STEP NO. 9 IS OVER.

LEARN RAFT STEP BY STEP

STEP NO. 10
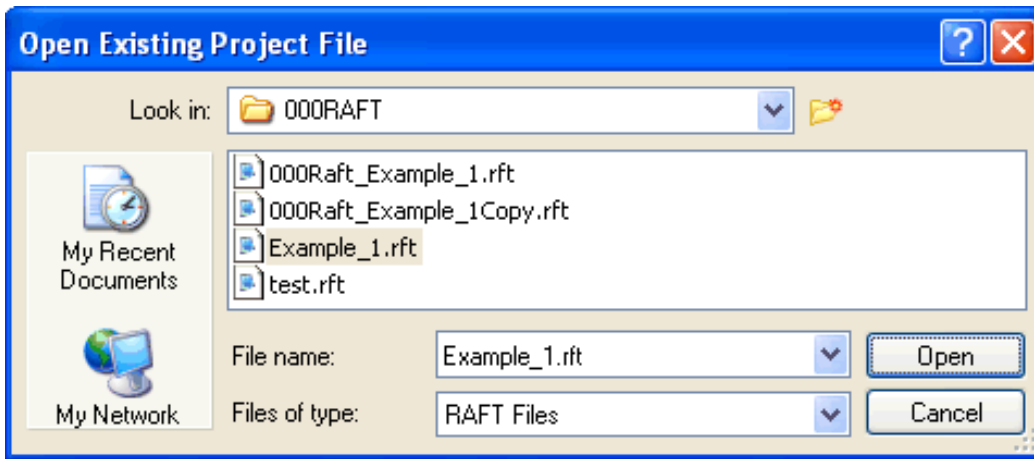### Raft Beam & Slab Design, Quantities & Cost Estimation

● When Program starts, the Menu above is displayed. Under the <u>Raft Beam/Slab Design</u> Heading following options are displayed.

- ● **Beam**
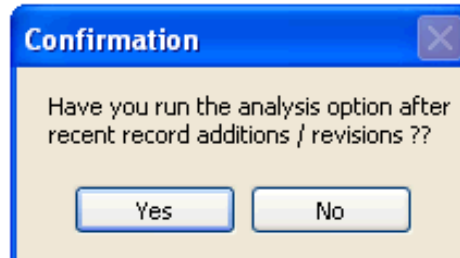- ● **Slab**
- ● **Quantity**
- ● **Floor Script for AutoCAD Drawing**
- ● **Soil File**

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. I have purposely created an error in Beam Size of B1 to demonstrate Error. Change the Beam Depth to 1000 MM & Re-run Analysis.**
  Now run Beam Design. Following Beam Schedule will be displayed with Error.

BEAM REINFORCEMENT SCHEDULE

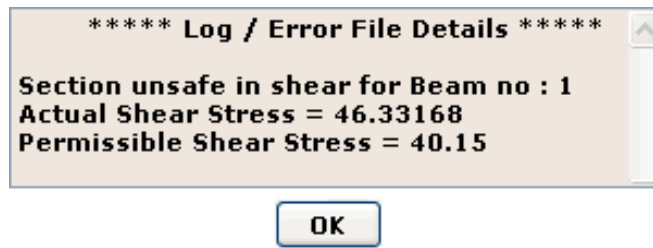**Project # : 8912**   **Conc. Grade : M35**

Bldg ID : Admin   **Note : All Links are 4 Legged.**

No.	Width	Depth	Top Steel	Bottom Steel	Stirrups	Skin Steel
1	ERROR...		T	T	T @ - 4 Legged	
2	400	1200	17 T 32	15 T 32	T 8 @ 100 - 4 Legged	8 T 8
3	500	1500	16 T 32	3 T 32 + 2 T 25	T 12 @ 105 - 4 Legged	8 T 10
4	750	2000	9 T 32	17 T 32	T 12 @ 75 - 4 Legged	18 T 10
5	800	2000	15 T 32	4 T 25 + 2 T 32	T 12 @ 60 - 4 Legged	18 T 10
6	500	1500	2 T 25 + 2 T 20	2 T 32 + 3 T 25	T 10 @ 120 - 4 Legged	8 T 10
7	400	1200	4 T 32 + 2 T 25	3 T 20 + 1 T 12	T 12 @ 105 - 4 Legged	8 T 8
8	400	1200	3 T 32 + 3 T 25	3 T 32 + 2 T 25	T 12 @ 100 - 4 Legged	8 T 8
9	400	1200	4 T 25	5 T 25	T 10 @ 90 - 4 Legged	8 T 8
10	400	1200	4 T 25 + 1 T 20	2 T 32 + 3 T 25	T 10 @ 100 - 4 Legged	
11	400	1200	4 T 25 + 1 T 20	4 T 25 + 1 T 20	T 10 @ 95 - 4 Legged	
12	400	1200	4 T 25	4 T 25 + 2 T 32	T 10 @ 95 - 4 Legged	8 T 8
23	500	1500	8 T 32	2 T 32 + 3 T 25	T 12 @ 85 - 4 Legged	8 T 10
24	400	1200	2 T 20 + 2 T 16	2 T 20 + 2 T 16	T 10 @ 95 - 4 Legged	8 T 8

Record No. : 1 of 55
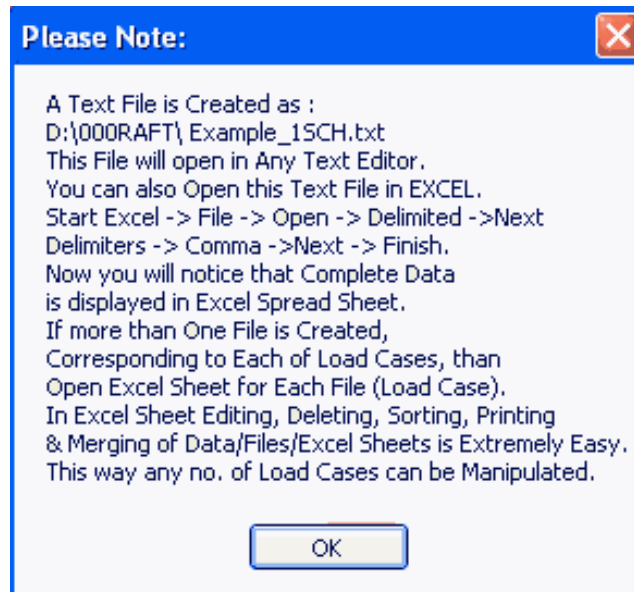
| Read Me | Prev | Next | Go To Rec | 1 st | Last | Print | O K |

- Note that in the Beam Schedule " Error : See Log File " is displayed for Beam no. B1. Now Click the " Log File " Button (Situated near the End of menu). Following window is displayed.

```
***** Log / Error File Details *****

Section unsafe in shear for Beam no : 1
Actual Shear Stress = 46.33168
Permissible Shear Stress = 40.15
```

OK

- The error message is clear, Beam No. 1 is Un-safe in Shear.
  Change the Section Size to to 500 * 1500 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.

Please Note:

A Text File is Created as :
D:\000RAFT\ Example_1SCH.txt
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.

OK

- 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.

Creation of Beam Schedule in AutoCad:

In the last Display you have created text file in Excel.
Now Save the above file in Excel Format (.XLS or .XLSX).
Exit Excel Program & Start AutoCad Program.
Click on Draw Option -> drop down menu -> Table.
A new Widow will open, displaying Insert Option.
Click on "From a Data Link" Option -> Drop Down Menu.
Now Click on "Launch Data Link Manager".
A new window will open, Displaying Select a Data Link.
Now Click on "Create a New Excel Data Link".
Again a new Window will appear -> Data Link Name
Enter any name say "xyz" and click OK.
A new window will appear -> New Excel Data Link xyz.
Click on "browse for a file" -> Save as Window will open.
Select & Click on our Excel File.
A new Window will open Displaying Preview of ACad Drg.
Click OK. Again preview of Drawing is shown, Click Ok.
Again preview of Drawing is shown, Click Ok.
Now you are in Autocad, Specify insertion point.
Beam Schedule is displayed as Table.

[OK]

- **For Detail explanation refer step no. 13.**
  Now Consider " Slab " Option. I have purposely created an error in Slab Size of S12 to demonstrate Error. Change the Slab Depth to 400 MM. & run Slab Design option. Following Graphics is displayed.

SLAB REINFORCEMENT SCHEDULE

No.	Thickness	Shorter Top & Bottom Steel	Shorter Haunch Steel	Longer Top & Bottom Steel	Longer Haunch Steel
S1	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S2	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S3	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S4	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S5	350	T 20 @ 120	T 20 @ 120	T 20 @ 120	T 20 @ 120
S6	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S7	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S8	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S9	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S10	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S11	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S12	400: ERROR ...	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S13	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S14	660	T 16 @ 120	T 16 @ 120	T 16 @ 150	T 16 @ 150
S15	660	T 16 @ 120	T 16 @ 120	T 16 @ 150	T 16 @ 150
S16	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S17	550	T 20 @ 110	T 20 @ 110	T 16 @ 175	T 16 @ 175
S18	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S19	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S20	350	T 20 @ 120	T 20 @ 120	T 20 @ 120	T 20 @ 120
S21	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S22	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S23	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125

Record No. : 1 of 24

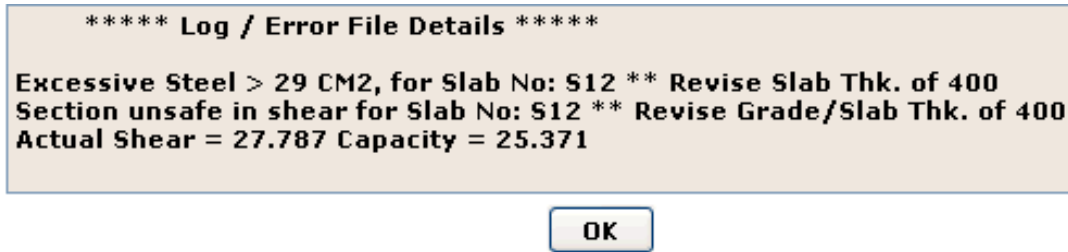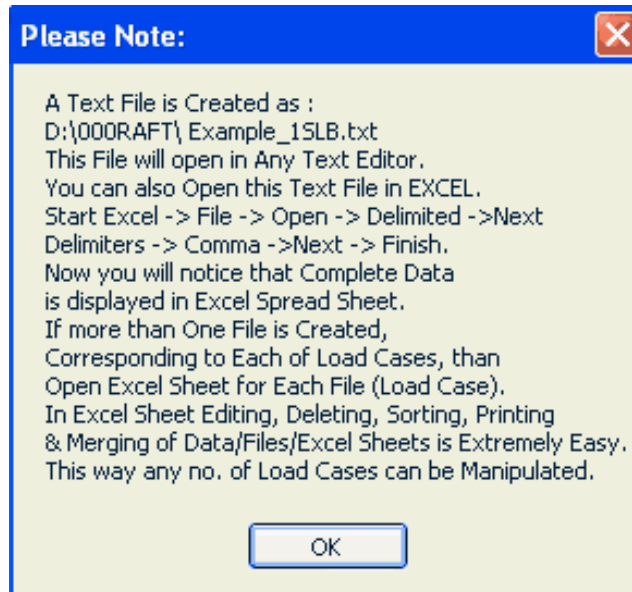[Read Me] [Prev] [Next] [Go To Rec] [1 st] [Last] [Print] [O K]

- 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.
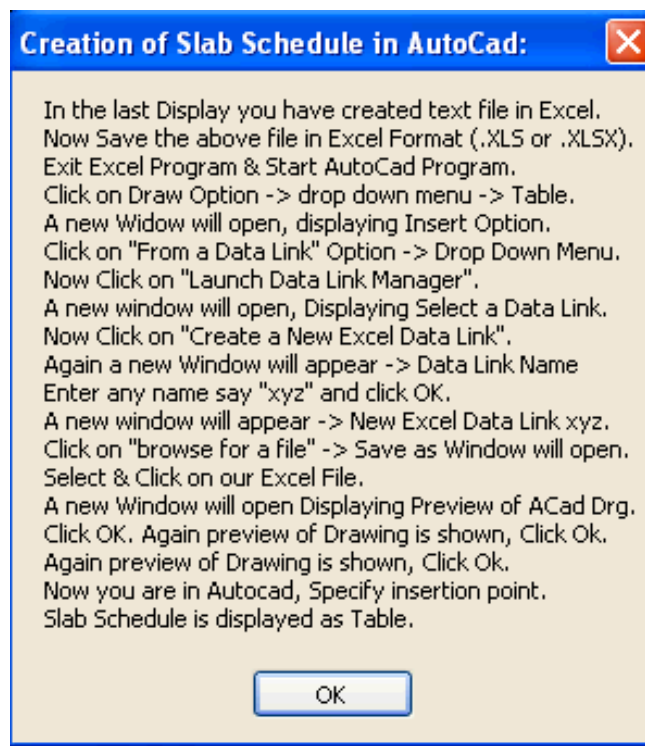
***** Log / Error File Details *****

Excessive Steel > 29 CM2, for Slab No: S12 ** Revise Slab Thk. of 400
Section unsafe in shear for Slab No: S12 ** Revise Grade/Slab Thk. of 400
Actual Shear = 27.787 Capacity = 25.371

OK

- The error message is clear, Excessive Steel & Unsafe in Shear for Slabs S12. Change the Thickness of S12 Slab to 550 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.

Please Note:

A Text File is Created as :
D:\000RAFT\ Example_1SLB.txt
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.
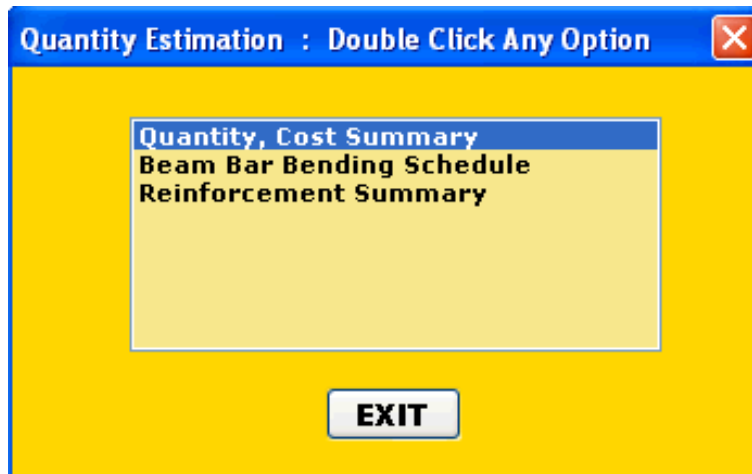This way any no. of Load Cases can be Manipulated.

OK

- 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.

Creation of Slab Schedule in AutoCad:

In the last Display you have created text file in Excel.
Now Save the above file in Excel Format (.XLS or .XLSX).
Exit Excel Program & Start AutoCad Program.
Click on Draw Option -> drop down menu -> Table.
A new Widow will open, displaying Insert Option.
Click on "From a Data Link" Option -> Drop Down Menu.
Now Click on "Launch Data Link Manager".
A new window will open, Displaying Select a Data Link.
Now Click on "Create a New Excel Data Link".
Again a new Window will appear -> Data Link Name
Enter any name say "xyz" and click OK.
A new window will appear -> New Excel Data Link xyz.
Click on "browse for a file" -> Save as Window will open.
Select & Click on our Excel File.
A new Window will open Displaying Preview of ACad Drg.
Click OK. Again preview of Drawing is shown, Click Ok.
Again preview of Drawing is shown, Click Ok.
Now you are in Autocad, Specify insertion point.
Slab Schedule is displayed as Table.

OK

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

Now Click " Quantity " Option.
Following Graphics is displayed.

Quantity Estimation : Double Click Any Option

Quantity, Cost Summary
Beam Bar Bending Schedule
Reinforcement Summary

EXIT

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

RAFT QUANTITIES AND COST SUMMARY

Item	Quantity	Rate	Cost	
M35 Concrete in M3	457.975	9000	4121775	
Total Reinforcement in Tons	107.235	50000	5361750	
PCC Flooring + FF (Raft Area in M2)	700	540	378000	
Excavation & Re-Filling	900.525	150	135078.8	
Total Cost of Raft			9996604	
Unit Cost of Raft in Rs / M2			14280.86	
Unit Cost of Raft in Rs / sqft			1328.452	
Total Cement Bags Required in Nos.	5783			
Total Sand Consumption in M3	199			
Total Aggregate Required in M3	387			

- 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.

APPROXIMATE BAR BENDING SCHEDULE

Beam #	Description	Code	Nos	Dia	Dim_A	Dim_B	Length	Quantity
1	B1-Top steel->st	1	15	32	12.112		12.112	1146.427
1	B1-Btm steel->st	1	4	25	11.65		11.65	179.476
1	B1-Btm steel->st	1	2	32	12.112		12.112	152.857
1	B1-Stirrups->4 Legged	8	190	12	1.45	0.225	3.638	613.364
1	B1-skin reinf.	1	8	10	10.55		10.55	52.009
2	B2-Top steel->st	1	3	32	7.112		7.112	134.633
2	B2-Top steel->st	1	3	25	6.65		6.65	76.836
2	B2-Btm steel->st	1	8	32	7.112		7.112	359.022
2	B2-Stirrups->4 Legged	8	36	8	1.15	0.175	2.842	40.35
2	B2-skin reinf.	1	8	8	5.55		5.55	17.511
3	B3-Top steel->st	1	15	32	12.112		12.112	1146.427
3	B3-Btm steel->st	1	2	32	12.112		12.112	152.857
3	B3-Btm steel->st	1	3	25	11.65		11.65	134.607
3	B3-Stirrups->4 Legged	8	190	12	1.45	0.225	3.638	613.364
3	B3-skin reinf.	1	8	10	10.55		10.55	52.009
4	B4-Top steel->st	1	9	32	12.112		12.112	687.856
4	B4-Btm steel->st	1	17	32	12.112		12.112	1299.284
4	B4-Stirrups->4 Legged	8	252	12	1.95	0.35	4.888	1093.034
4	B4-skin reinf.	1	18	10	10.55		10.55	117.021
5	B5-Top steel->st	1	15	32	12.112		12.112	1146.427
5	B5-Btm steel->st	1	4	25	11.65		11.65	179.476
5	B5-Btm steel->st	1	2	32	12.112		12.112	152.857
5	B5-Stirrups->4 Legged	8	314	12	1.95	0.375	4.938	1375.886
5	B5-skin reinf.	1	18	10	10.55		10.55	117.021
6	B6-Top steel->st	1	2	25	6.65		6.65	51.224
6	B6-Top steel->st	1	2	20	6.32		6.32	31.156
6	B6-Btm steel->st	1	2	32	7.112		7.112	89.755
6	B6-Btm steel->st	1	3	25	6.65		6.65	76.836
6	B6-Stirrups->4 Legged	8	74	10	1.45	0.225	3.59	163.706

- **BBS is approximate, do not use cutting length for fabrication.**

Now Double Click **" Reinforcement Summary "** Option. Following Graphics is displayed.

APPROXIMATE SUMMARY OF REINFORCEMENTS IN KG

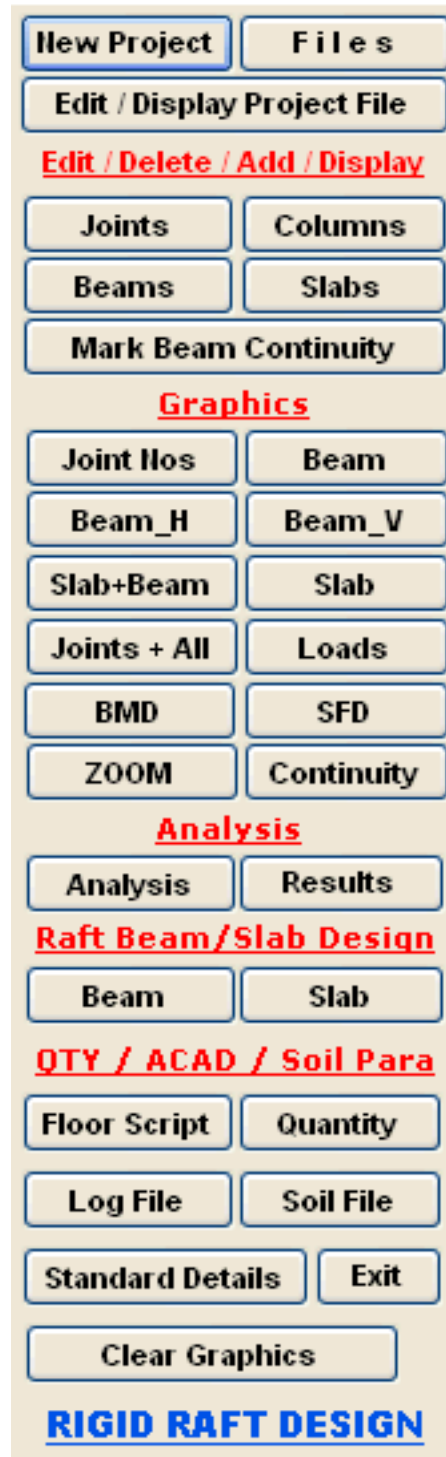|	
6 MM Dia :	0
8 MM Dia :	1902.602
10 MM Dia :	4234.76
12 MM Dia :	11291.79
16 MM Dia :	7268.882
20 MM Dia :	56222
25 MM Dia :	6057.874
32 MM Dia :	20256.77

TOTAL REINFORCEMENT IN TONS = 107.235

- The MTO includes total of Beam and Slab steel Quantities. Beam steel quantities are taken from Approx. BBS and Slab quantities have been worked out **approximately** from Slab Schedule.

- Now we have come to the end of Step # 10.
  Let us proceed to Step No. 11.

STEP NO. 10 IS OVER.

LEARN RAFT STEP BY STEP

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

New Project	Files
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

When Program starts, the Menu above is displayed. Under the <u>**Graphics**</u> 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.
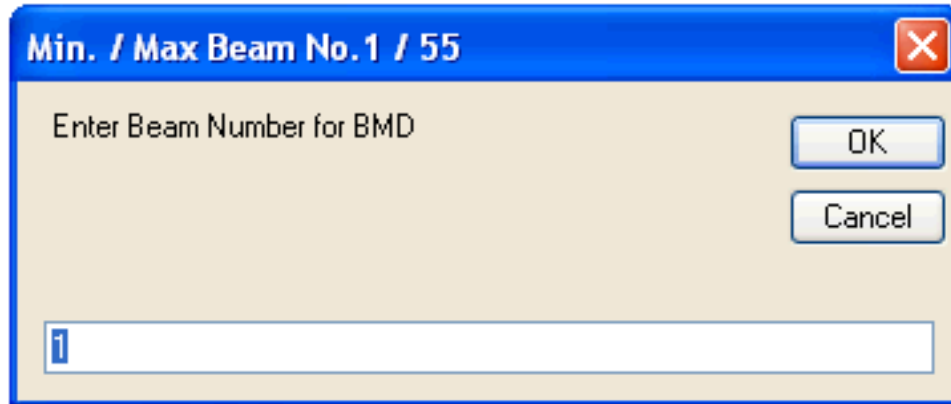
Following Graphics is displayed.
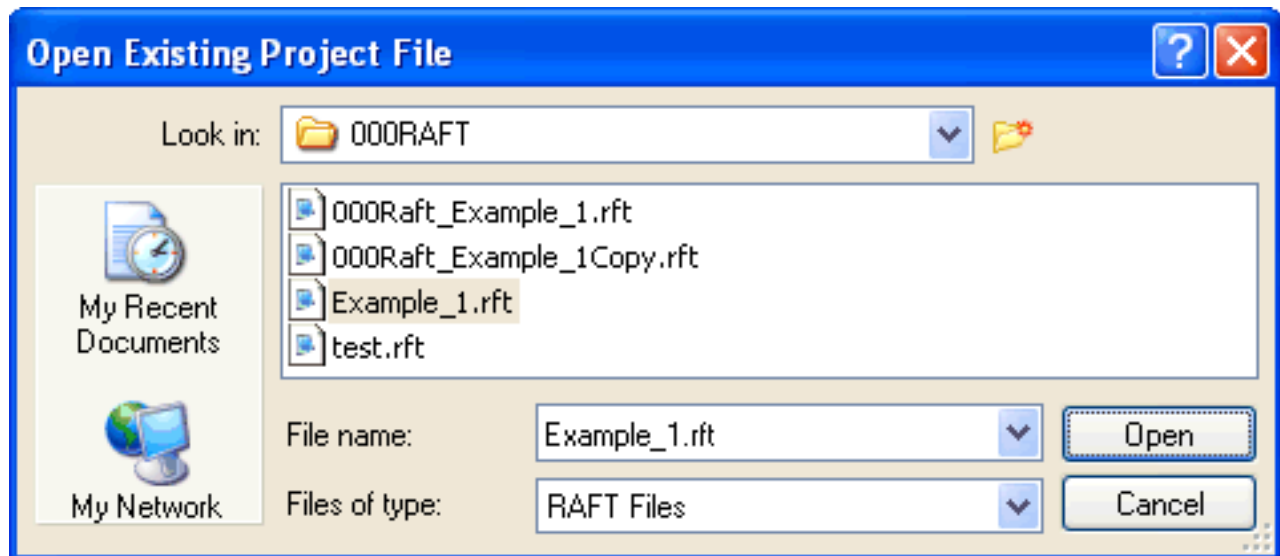
🔴 **Now select " Example _1 File & Press Open Button. Following Warning is displayed.**
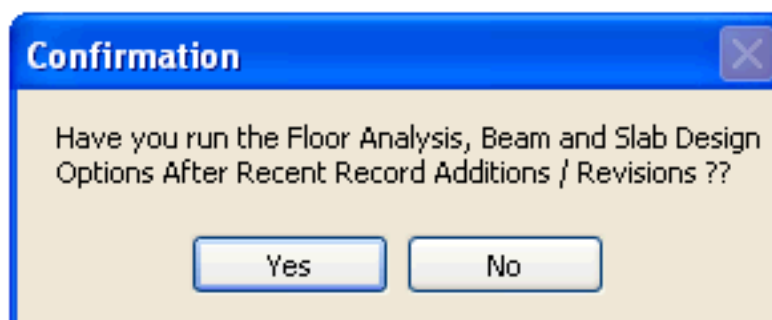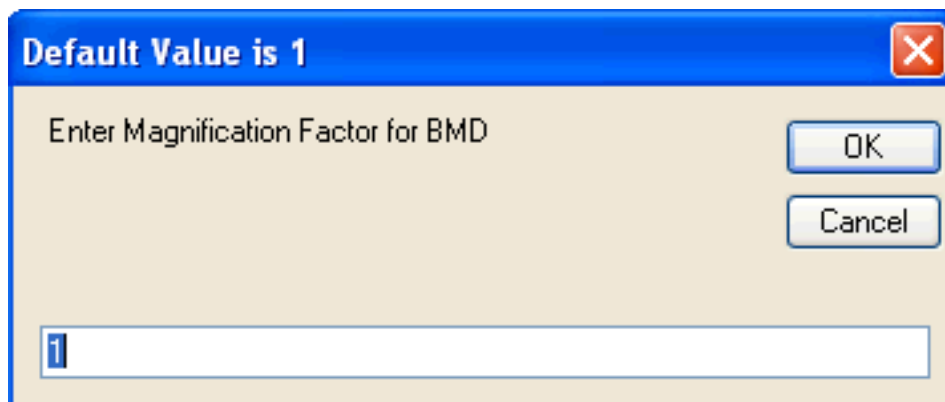
- **This is a very Important Message. In case a user has edited or added any Joint / Column / Beam or Slab Member after performing analysis then he should re-perform the analysis, else old (in-correct) results will be displayed. The Beam and Slab Designs are equally important as these options inform 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. Click OK.**
Following BMD is displayed.
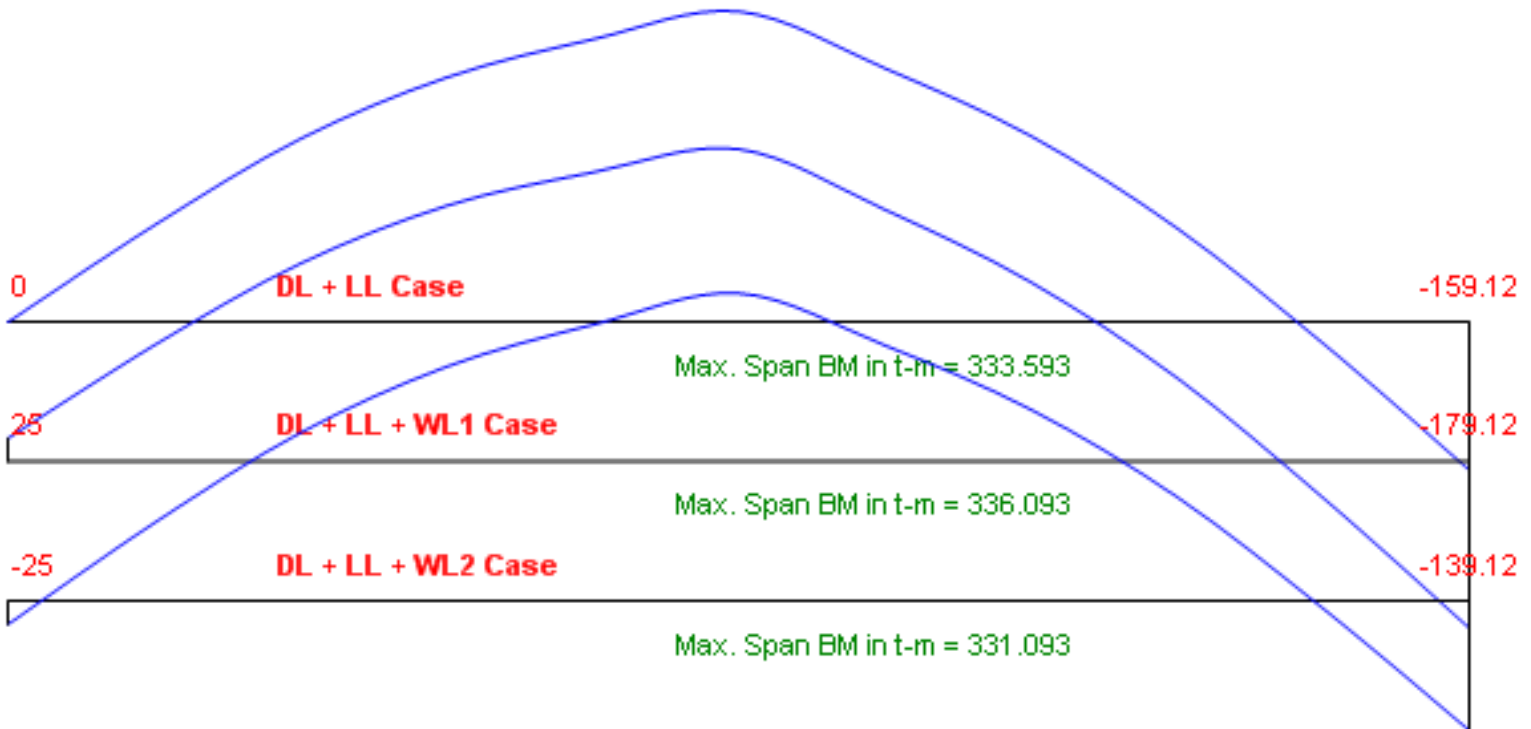
BMD Drawn on Tension Side
Beam # : B1

| Print | Next |

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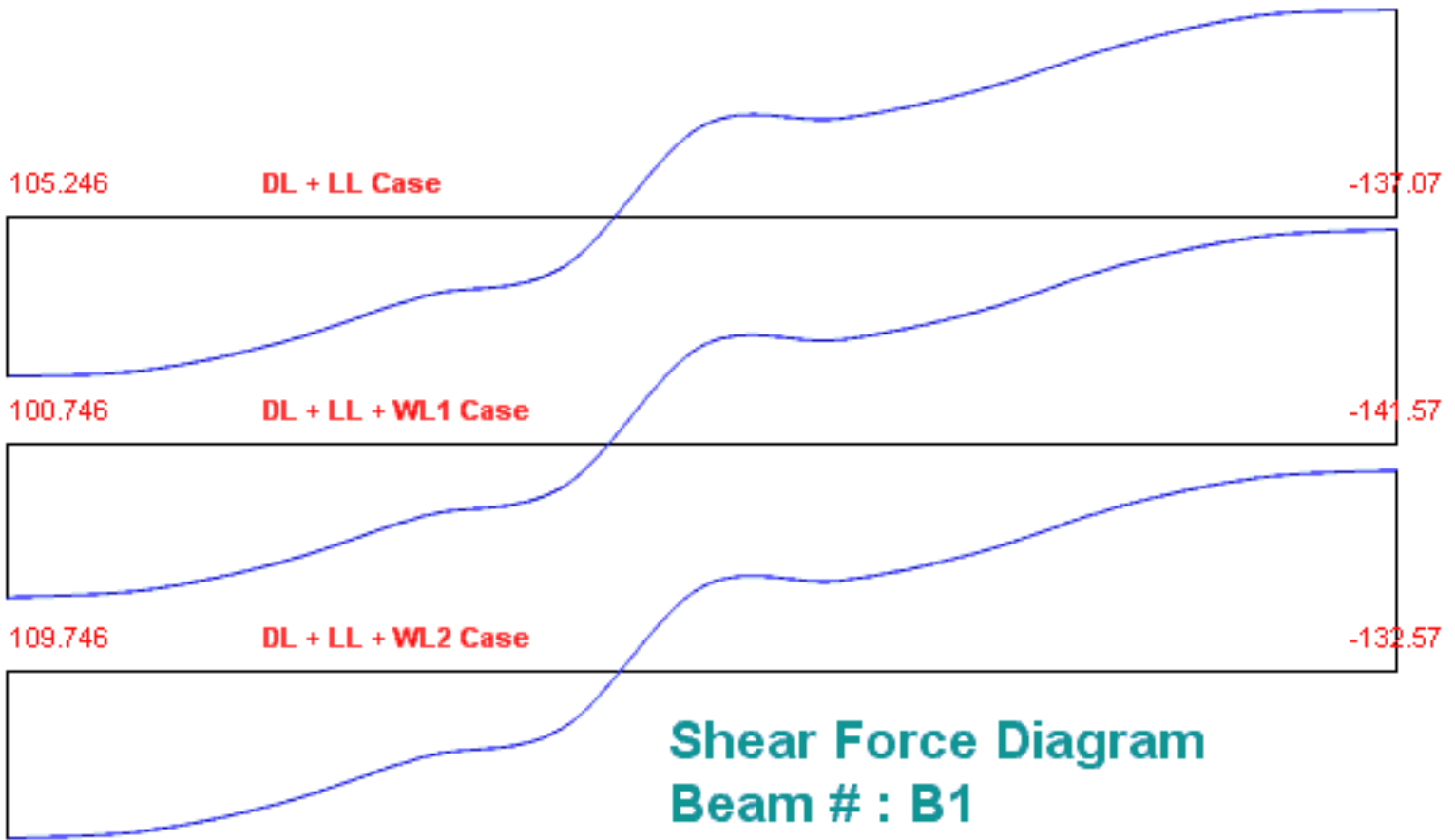
● Incorrect Geometry (Span, Grid Dimension).
● Incorrect Loads (Point Load, End Moments).
● Floor Analysis, Beam & Slab Design not performed after Editing / Adding Geometry or Loads.

The 3 Cases are displayed simultaneously.
The " Next " button is very useful as it can help you to display continuously the required BMD for a specified Beam.
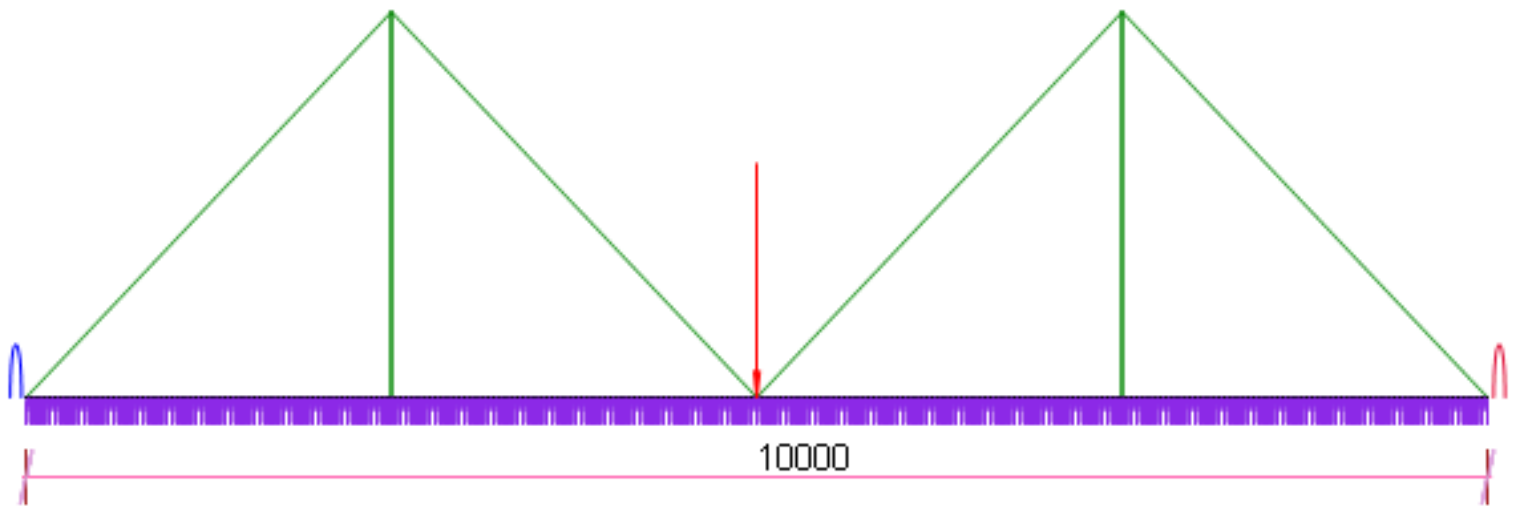
Now Click on " SFD " option. The procedure is exactly same as that of BMD.

SFD is displayed as under. MF = 1.0

105.246   **DL + LL Case**   -137.07

100.746   **DL + LL + WL1 Case**   -141.57

109.746   **DL + LL + WL2 Case**   -132.57

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. Read the Diagram upside down.

UDL in t/m = -2.68
RHS Moment in t-m = 20
LHS Moment in t-m = 25
UDL in t/m =
Near Int. in t/m = 0 @ dist. of 0 m : Far Int. in t/m = 36.182 @ dist. of 2.5 m
UDL in t/m =
Near Int. in t/m = 36.182 @ dist. of 2.5 m : Far Int. in t/m = 0 @ dist. of 5 m
UDL in t/m =
Near Int. in t/m = 0 @ dist. of 5 m : Far Int. in t/m = 36.182 @ dist. of 7.5 m
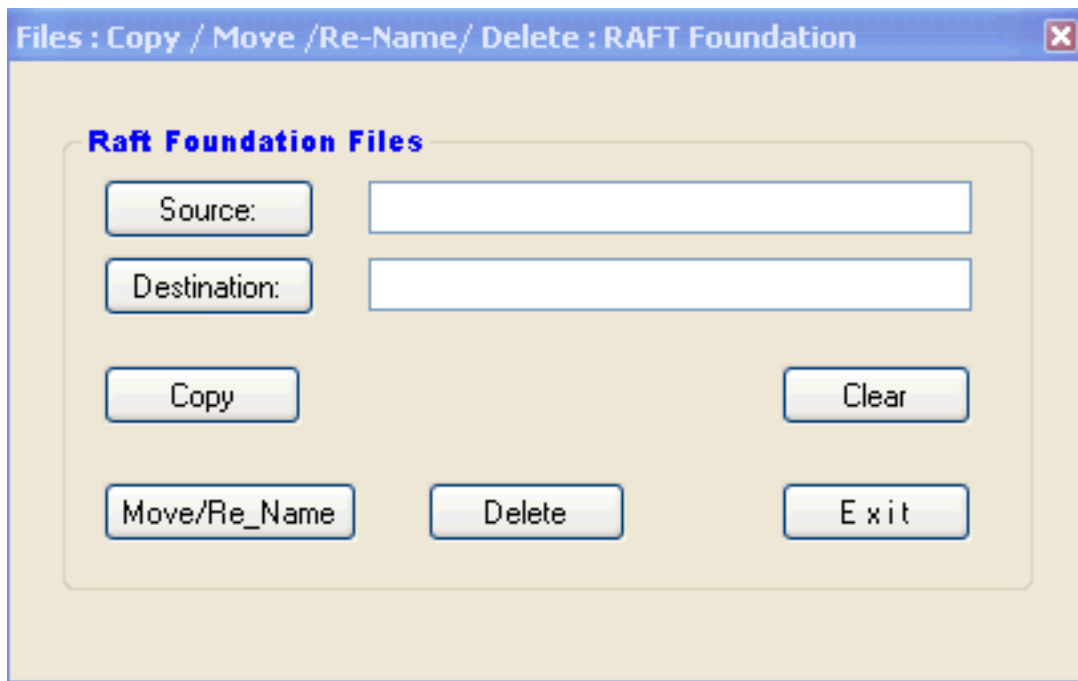UDL in t/m =
Near Int. in t/m = 36.182 @ dist. of 7.5 m : Far Int. in t/m = 0 @ dist. of 10 m
UDL in t/m =
Point Load in t = 88.206 @ dist. of 5 m

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.

Files : Copy / Move /Re-Name/ Delete : RAFT Foundation

Raft Foundation Files

Source: [   ]

Destination: [   ]

Copy   Clear

Move/Re_Name   Delete   E x i t

- **Use this 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.

The file has been successfully copied, Re-run Analysis/Designs/Qty.

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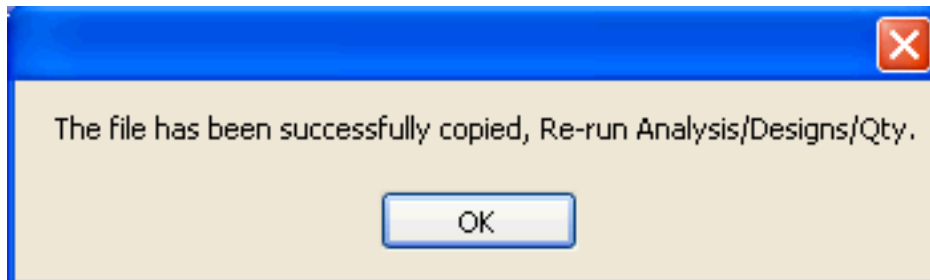
- **Similarly we can use Delete Option to Delete Files, however note that there will be no**
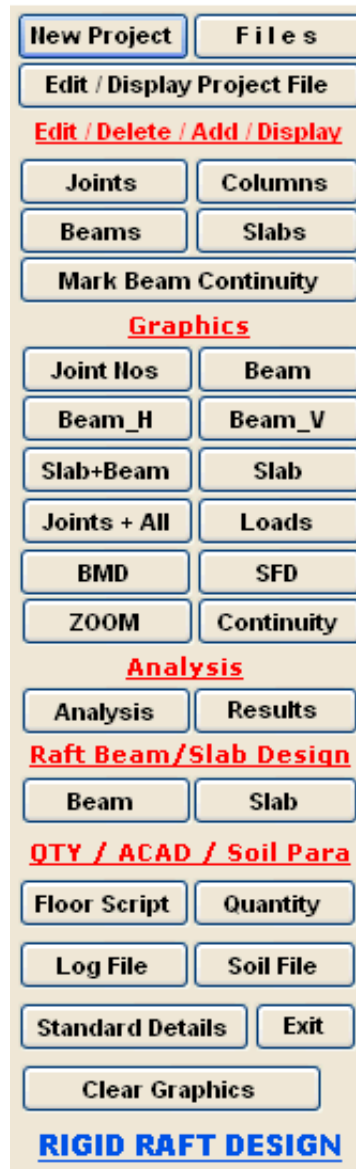 **" Destination " file & destination text box shall be empty.**

  Note that Raft Foundation File extension is " .RFT ".

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

STEP NO. 11 IS OVER.

LEARN RAFT STEP BY STEP

STEP 12 : CREATION OF RAFT FOUNDATION PLAN IN AUTOCAD

New Project	**F i l e s**
Edit / Display Project File	

Edit / Delete / Add / Display

Joints	Columns
Beams	Slabs
Mark Beam Continuity	

Graphics

Joint Nos	Beam
Beam_H	Beam_V
Slab+Beam	Slab
Joints + All	Loads
BMD	SFD
ZOOM	Continuity

Analysis

Analysis	Results

Raft Beam/Slab Design

Beam	Slab

QTY / ACAD / Soil Para

|	
Floor Script	Quantity
Log File	Soil File
Standard Details	Exit
Clear Graphics	

RIGID RAFT DESIGN

➤ When The Program starts, the above Menu is displayed.
Under the **QTY / ACAD / Soil Para** heading following options are displayed.

- Floor Script
- Quantity
- Log File
- Soil File
- Standard Details
- Exit
- Clear Graphics

In Order to create an AutoCAD drawing, a script file has to be created first.
To create the script file, click on Floor Script Option. A window dialogue box appears .
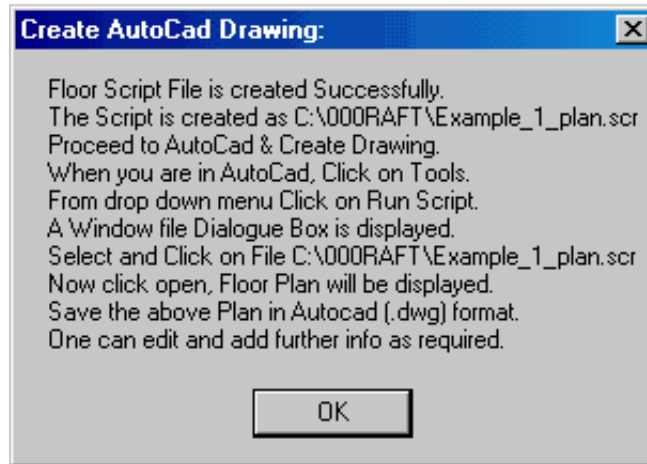Click on the required file and click on open.
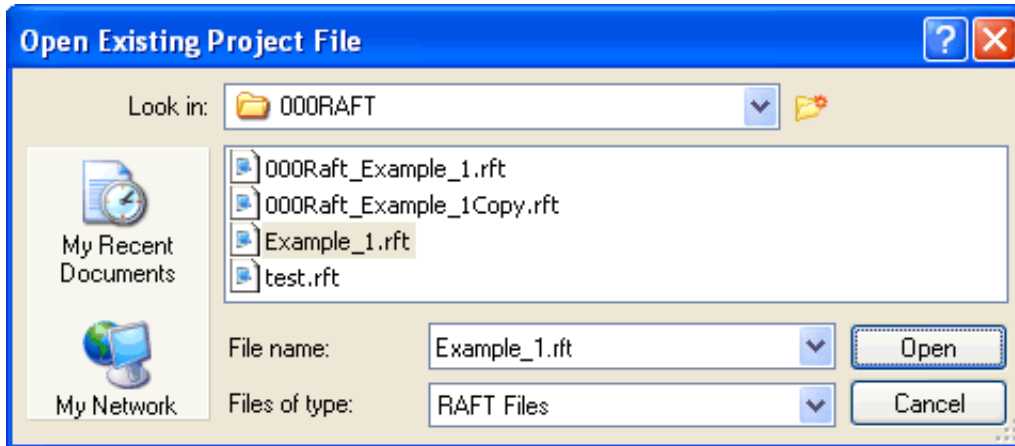
➤ Following graphics is displayed.
Click on Yes if Floor Analysis and Beam Design Options are performed.
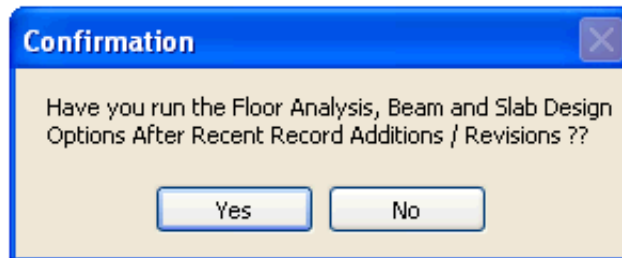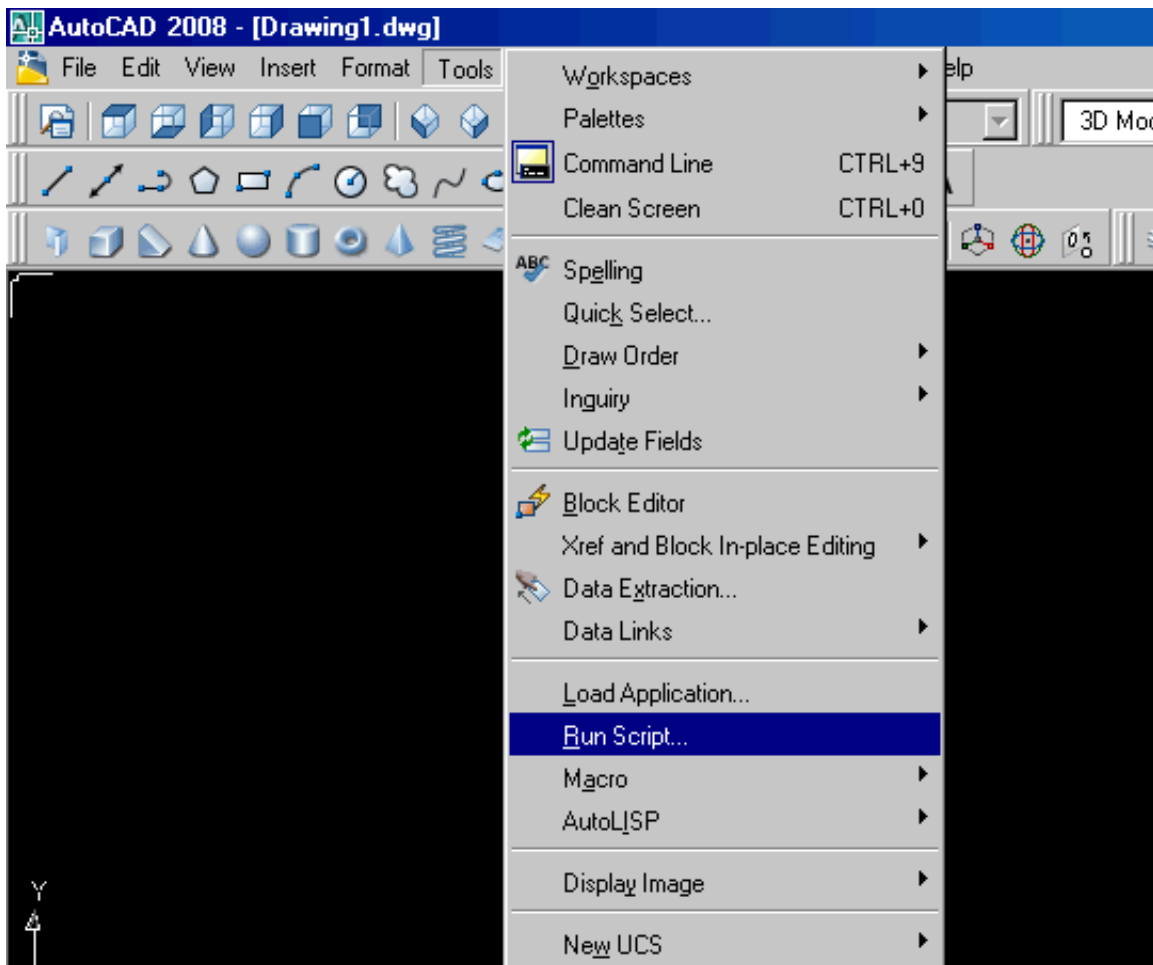

Once Yes is clicked, following graphics is displayed.

➤ 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.

Create AutoCad Drawing:

Floor Script File is created Successfully.
The Script is created as C:\000RAFT\Example_1_plan.scr
Proceed to AutoCad & Create Drawing.
When you are in AutoCad, Click on Tools.
From drop down menu Click on Run Script.
A Window file Dialogue Box is displayed.
Select and Click on File C:\000RAFT\Example_1_plan.scr
Now click open, Floor Plan will be displayed.
Save the above Plan in Autocad (.dwg) format.
One can edit and add further info as required.

OK

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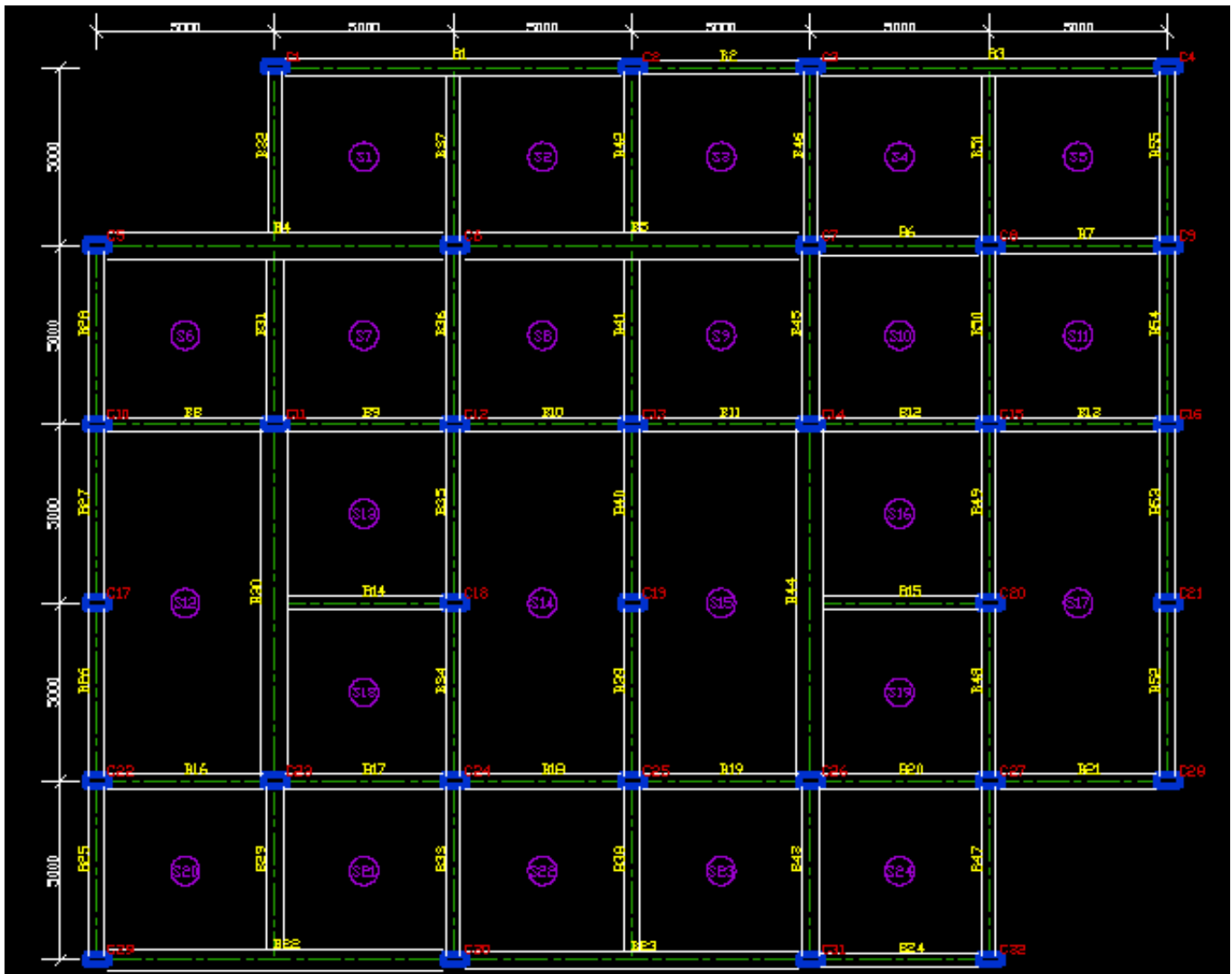In AutoCAD click on Tools. From the drop down menu click on Run Script.

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

72
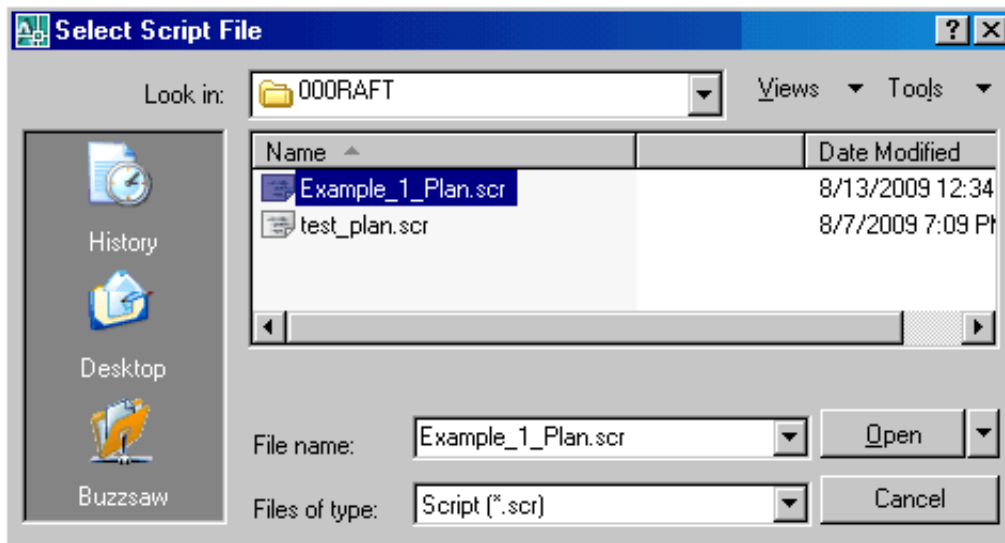
➤ It will take a few seconds for the script to run, after which the plan will appear in the form of AutoCAD drawing . The display will be as follows.


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.

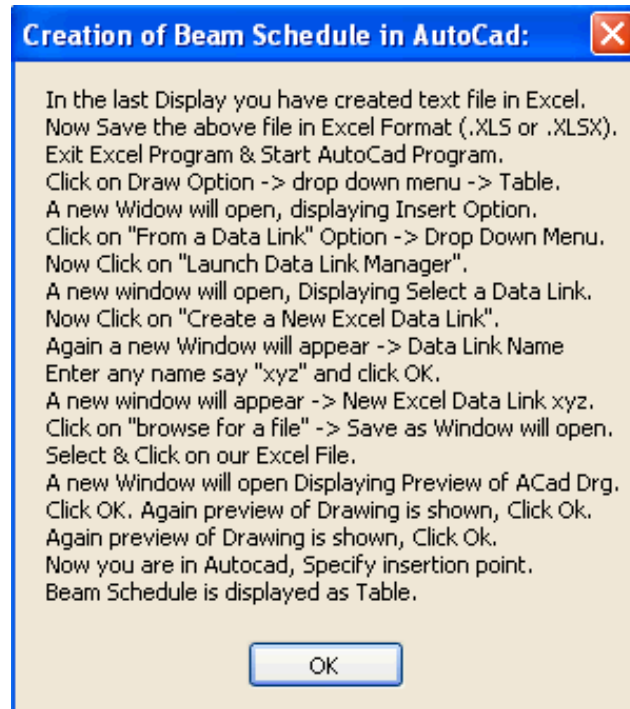Save the above Drawing in AutoCAD i.e. (.dwg) format.
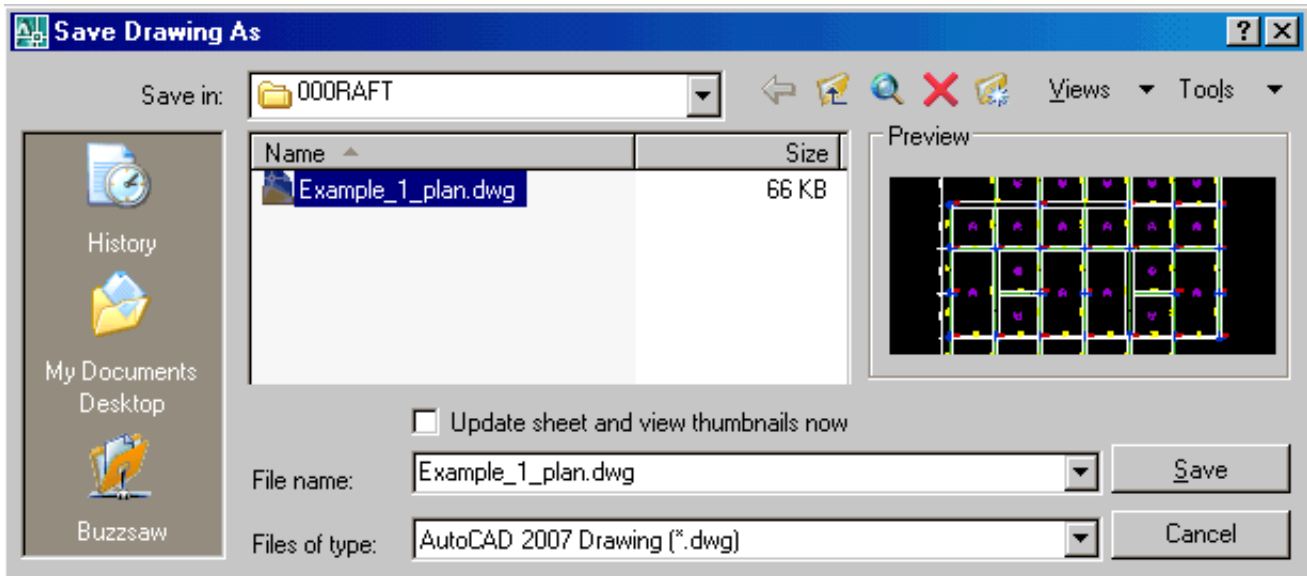
STEP 12 IS OVER
Now lets have a look on creation of Beam Schedule in the next Step....
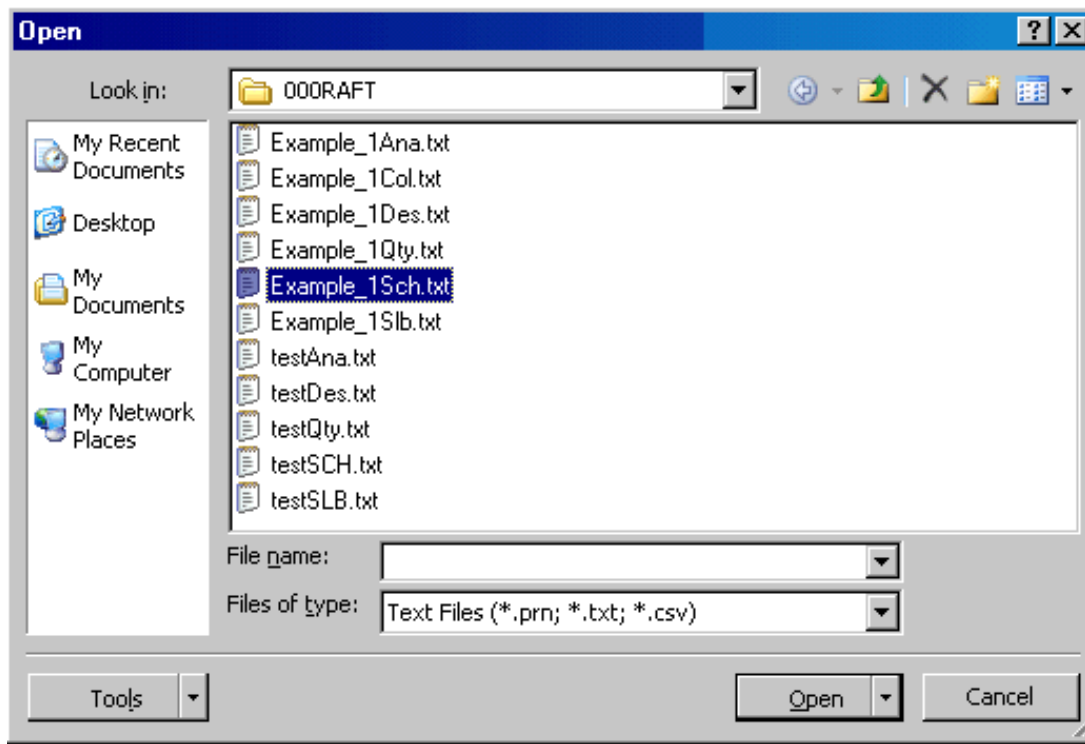
LEARN RAFT STEP BY STEP

STEP 13: CREATION OF BEAM 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 10, following Graphics is displayed. We will explain this message in details.**

Creation of Beam Schedule in AutoCad:

In the last Display you have created text file in Excel.
Now Save the above file in Excel Format (.XLS or .XLSX).
Exit Excel Program & Start AutoCad Program.
Click on Draw Option -> drop down menu -> Table.
A new Widow will open, displaying Insert Option.
Click on "From a Data Link" Option -> Drop Down Menu.
Now Click on "Launch Data Link Manager".
A new window will open, Displaying Select a Data Link.
Now Click on "Create a New Excel Data Link".
Again a new Window will appear -> Data Link Name
Enter any name say "xyz" and click OK.
A new window will appear -> New Excel Data Link xyz.
Click on "browse for a file" -> Save as Window will open.
Select & Click on our Excel File.
A new Window will open Displaying Preview of ACad Drg.
Click OK. Again preview of Drawing is shown, Click Ok.
Again preview of Drawing is shown, Click Ok.
Now you are in Autocad, Specify insertion point.
Beam Schedule is displayed as Table.

[OK]

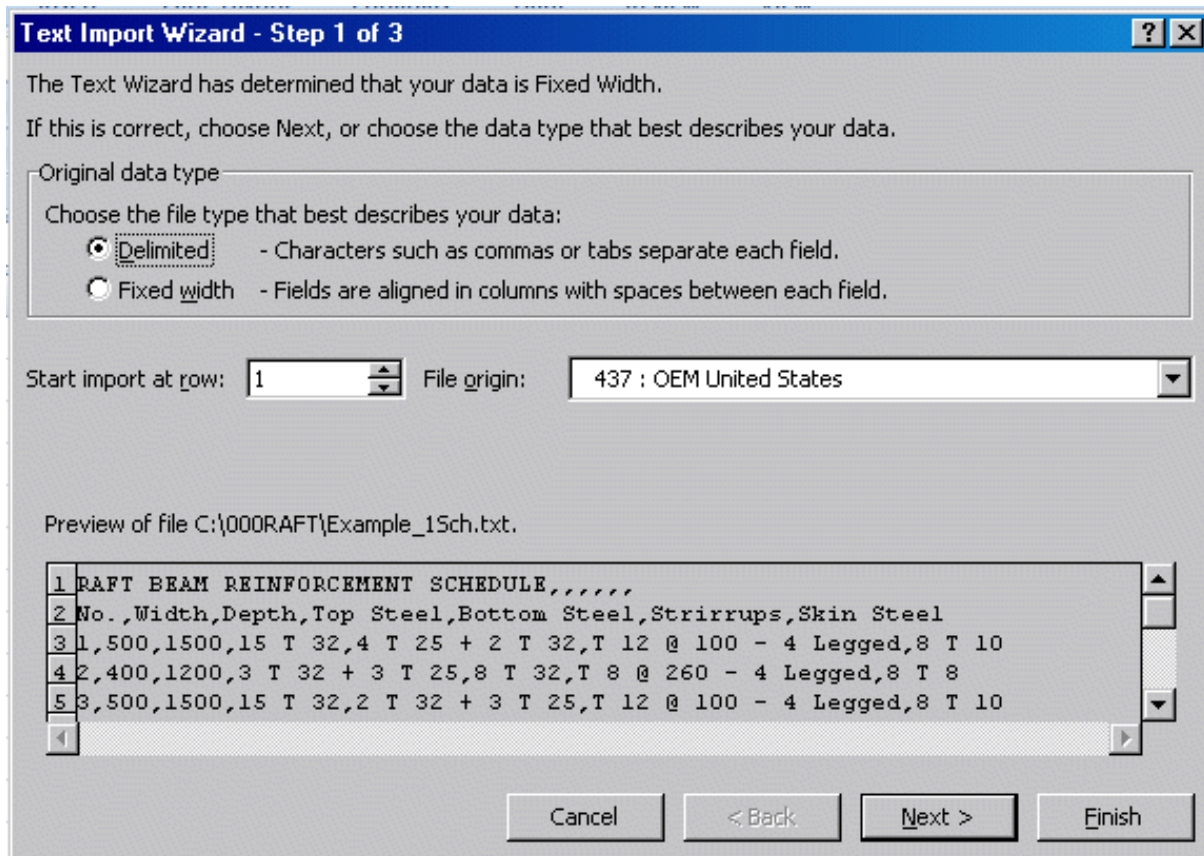➤ **Start Microsoft Excel . Click On Open. Following Graphics is Displayed.**

Click on Example_1Sch.txt.

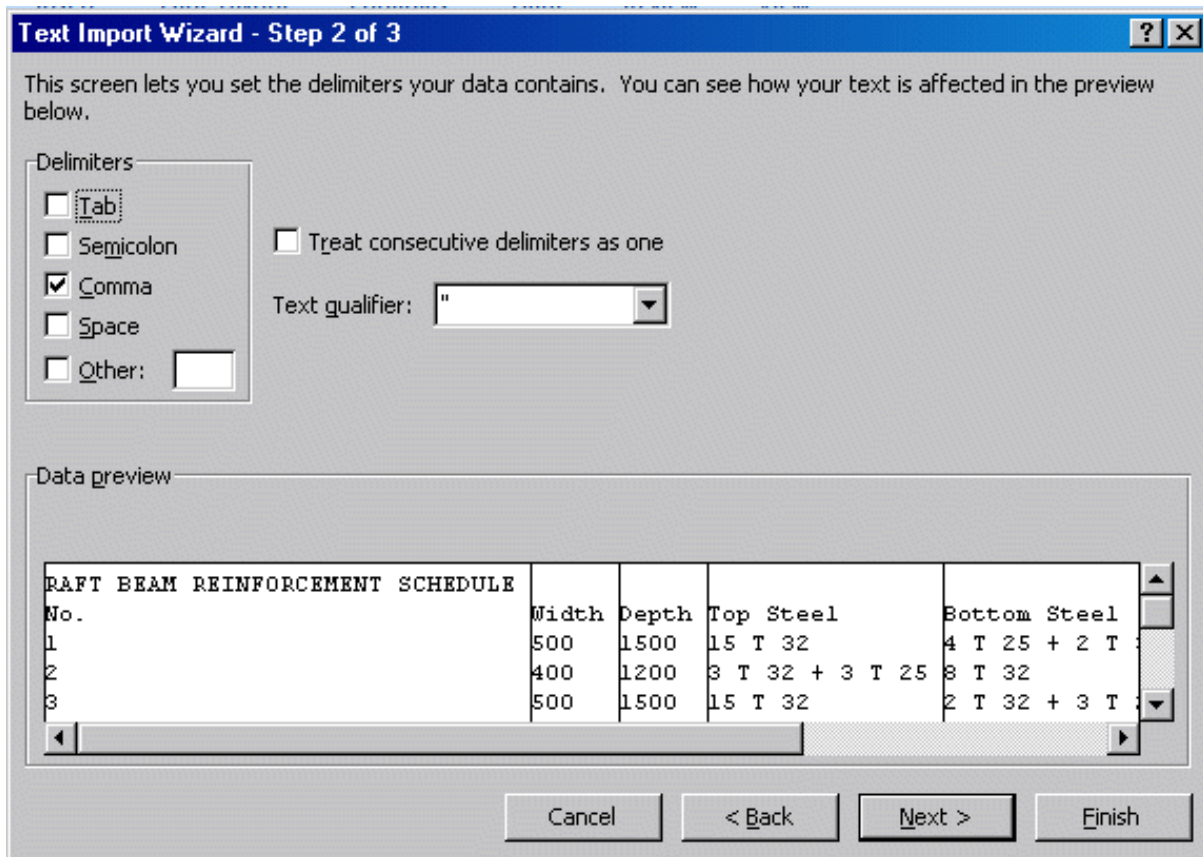➤ **As you can see, the above file is in text format.**
In the following steps we will save the file in Excel format.
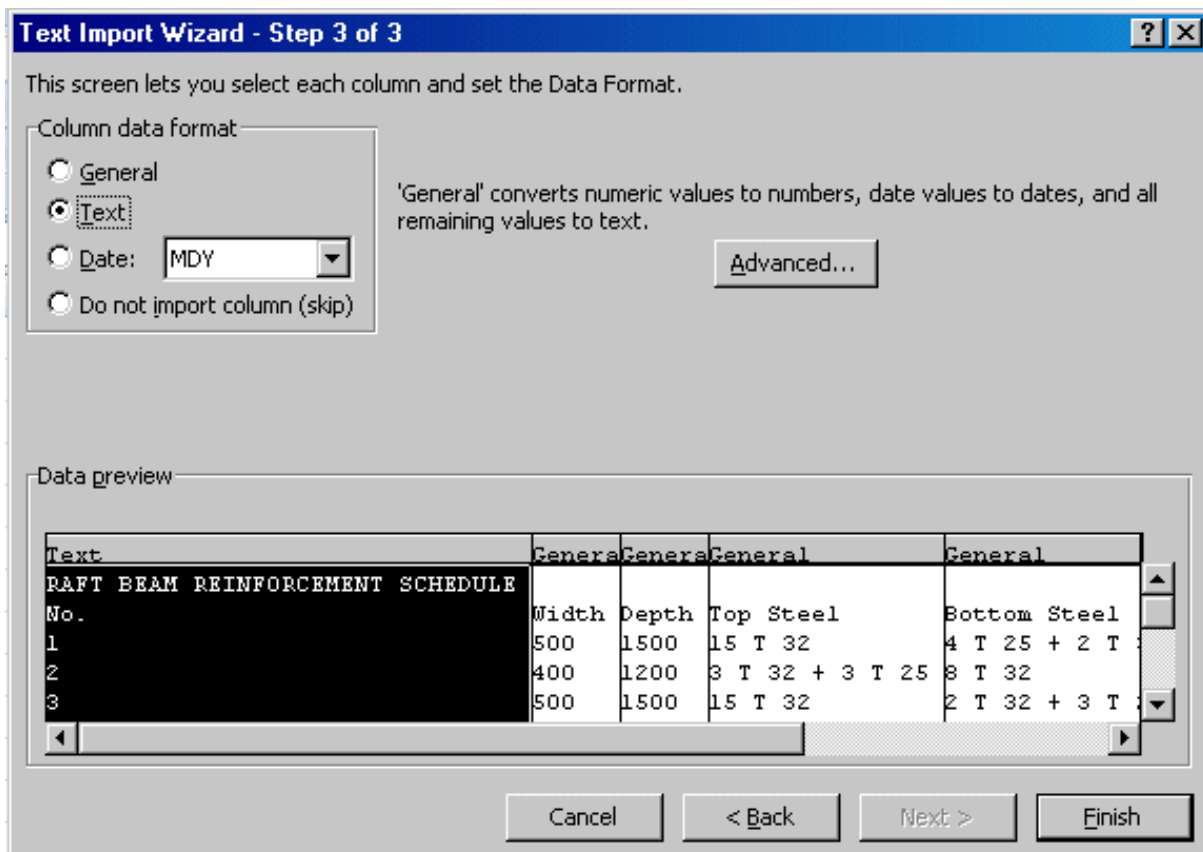**Once Example1_Sch.txt is clicked, following graphics is displayed.**

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

Text Import Wizard - Step 2 of 3   ? ☒

This screen lets you set the delimiters your data contains. You can see how your text is affected in the preview below.

Delimiters
- ☐ Tab
- ☐ Semicolon
- ☑ Comma
- ☐ Space
- ☐ Other: [　]

☐ Treat consecutive delimiters as one

Text qualifier: [" ▼]

Data preview

```
RAFT BEAM REINFORCEMENT SCHEDULE
No.                               Width  Depth  Top Steel          Bottom Steel
1                                 500    1500   15 T 32            4 T 25 + 2 T
2                                 400    1200   3 T 32 + 3 T 25    8 T 32
3                                 500    1500   15 T 32            2 T 32 + 3 T
```

[Cancel]   [< Back]   [Next >]   [Finish]

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

Text Import Wizard - Step 3 of 3   ? ☒

This screen lets you select each column and set the Data Format.

Column data format
- ☐ General
- ☑ Text
- ☐ Date: [MDY ▼]
- ☐ Do not import column (skip)

'General' converts numeric values to numbers, date values to dates, and all remaining values to text.

[Advanced...]

Data preview

```
Text                              General General General            General
RAFT BEAM REINFORCEMENT SCHEDULE
No.                               Width  Depth  Top Steel          Bottom Steel
1                                 500    1500   15 T 32            4 T 25 + 2 T
2                                 400    1200   3 T 32 + 3 T 25    8 T 32
3                                 500    1500   15 T 32            2 T 32 + 3 T
```
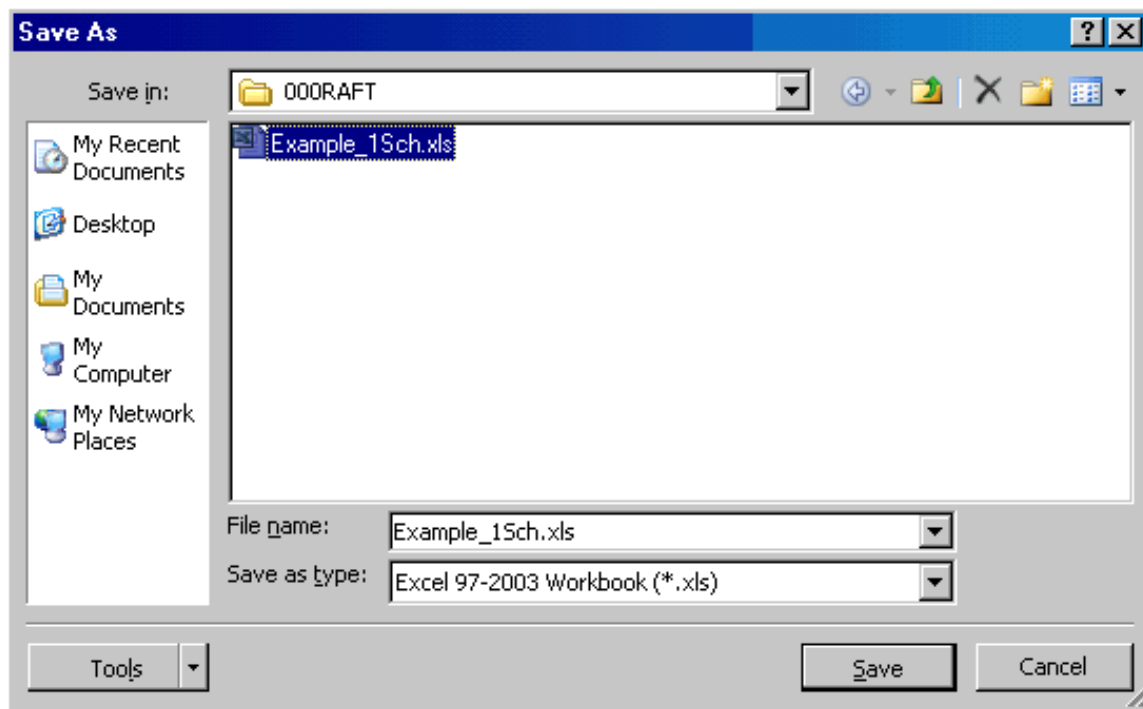
[Cancel]   [< Back]   [Next >]   [Finish]

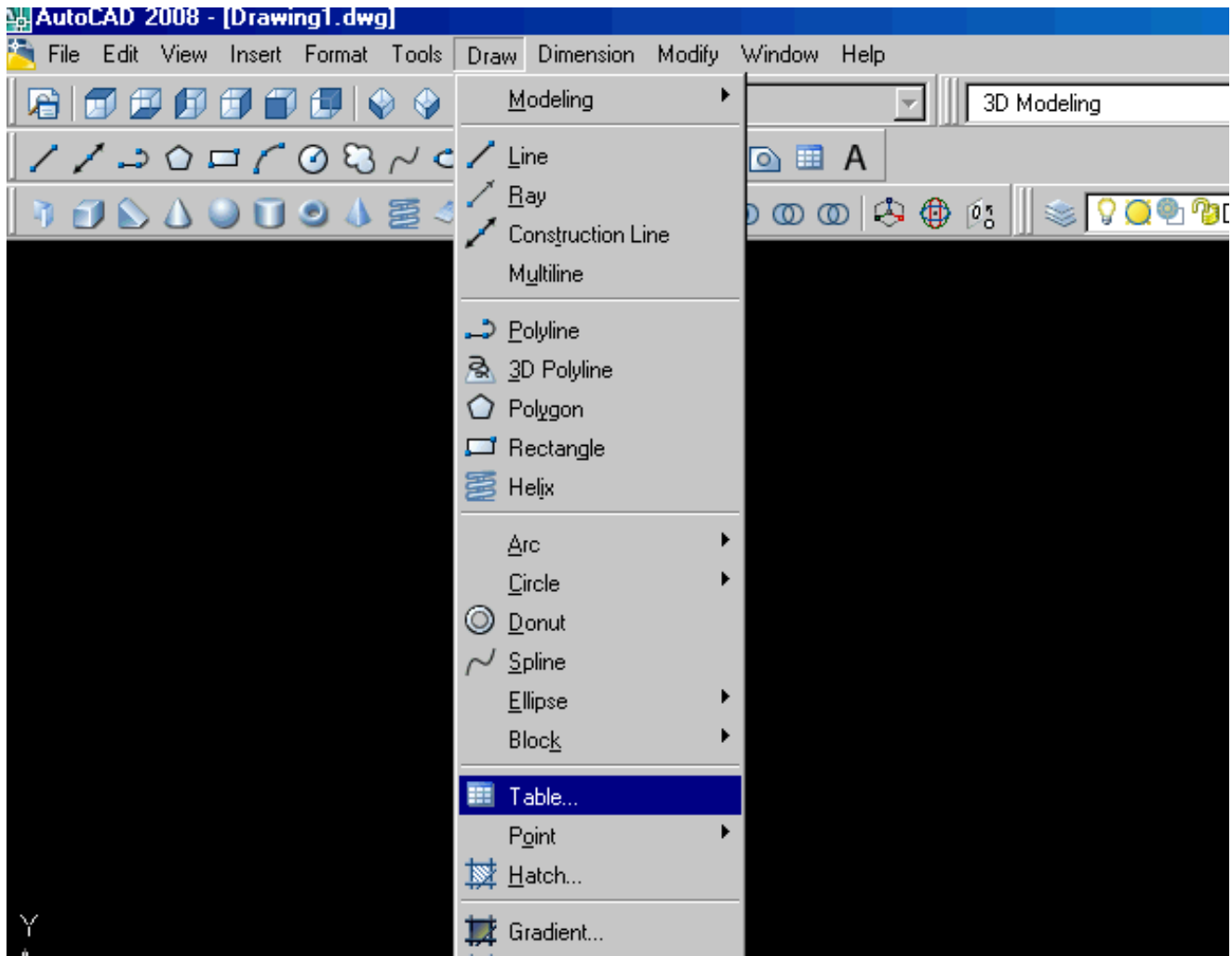**As shown above click on Text and then click on Finish.**
Here you will see that Beam schedule appears in Excel . Following is a part display.
Now You can make any number of changes you want within Excel, like changing fonts, alignment of text, Column Width etc..

RAFT BEAM REINFORCEMENT SCHEDULE

No.	Width	Depth	Top Steel	Bottom Steel	Strirrups	Skin Steel
1	500	1500	15 T 32	4 T 25 + 2 T 32	T 12 @ 100 - 4 Legged	8 T 10
2	400	1200	3 T 32 + 3 T 25	8 T 32	T 8 @ 260 - 4 Legged	8 T 8
3	500	1500	15 T 32	2 T 32 + 3 T 25	T 12 @ 100 - 4 Legged	8 T 10
4	750	2000	9 T 32	17 T 32	T 12 @ 75 - 4 Legged	18 T 10
5	800	2000	15 T 32	4 T 25 + 2 T 32	T 12 @ 60 - 4 Legged	18 T 10
6	500	1500	2 T 25 + 2 T 20	2 T 32 + 3 T 25	T 10 @ 120 - 4 Legged	8 T 10
7	400	1200	4 T 32 + 2 T 25	3 T 20 + 1 T 12	T 12 @ 105 - 4 Legged	8 T 8
8	400	1200	3 T 32 + 3 T 25	3 T 32 + 2 T 25	T 12 @ 100 - 4 Legged	8 T 8
9	400	1200	4 T 25	5 T 25	T 10 @ 90 - 4 Legged	8 T 8
10	400	1200	4 T 25 + 1 T 20	2 T 32 + 3 T 25	T 10 @ 100 - 4 Legged	
11	400	1200	4 T 25 + 1 T 20	4 T 25 + 1 T 20	T 10 @ 95 - 4 Legged	
12	400	1200	4 T 25	4 T 25 + 2 T 32	T 10 @ 95 - 4 Legged	8 T 8
13	400	1200	3 T 32 + 3 T 25	2 T 20 + 2 T 16	T 12 @ 100 - 4 Legged	8 T 8
14	400	1200	8 T 32	2 T 25 + 2 T 20	T 8 @ 80 - 4 Legged	8 T 8
15	400	1200	8 T 32	2 T 25 + 2 T 20	T 8 @ 80 - 4 Legged	8 T 8

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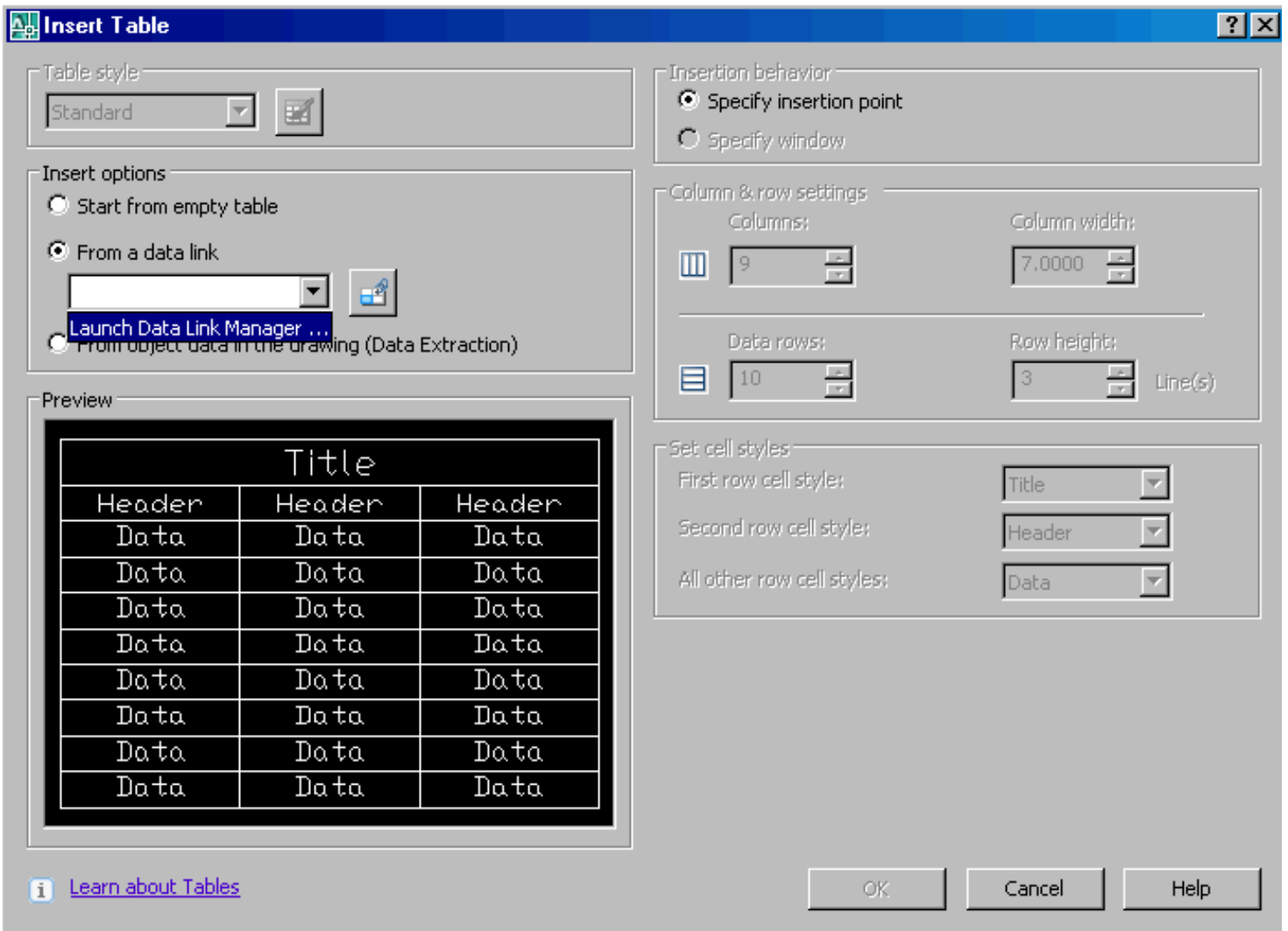
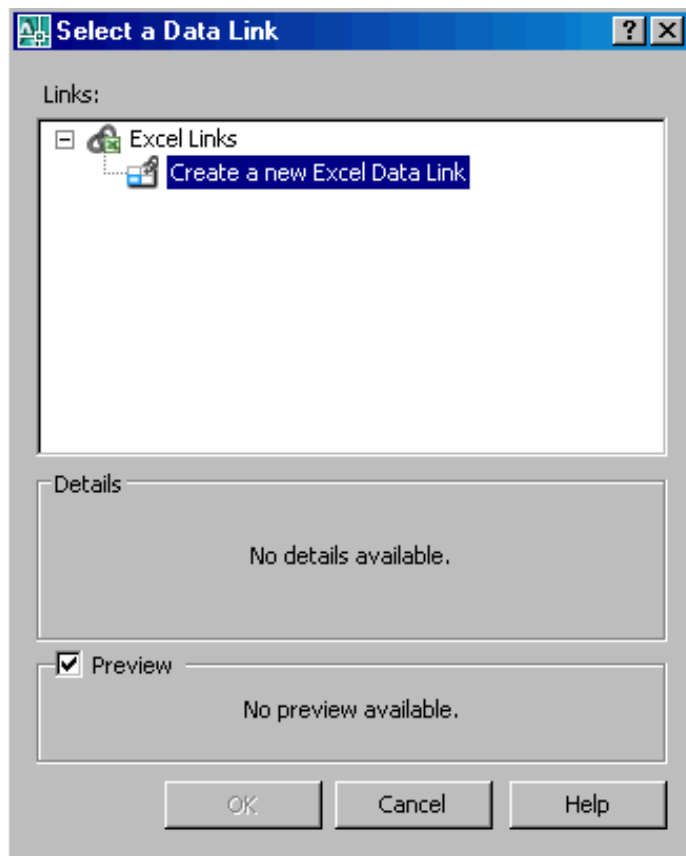**Start AutoCAD. Click on DRAW . From the drop down menu click on Table a shown below.**

A dialogue box will appear.
➤ As show below click on From a data link .
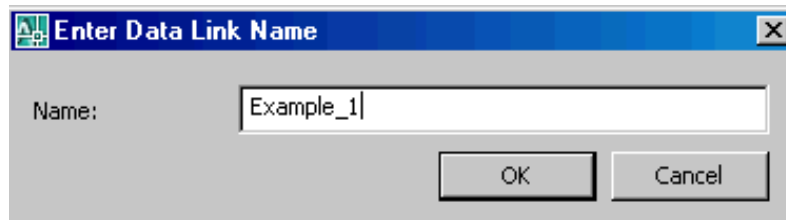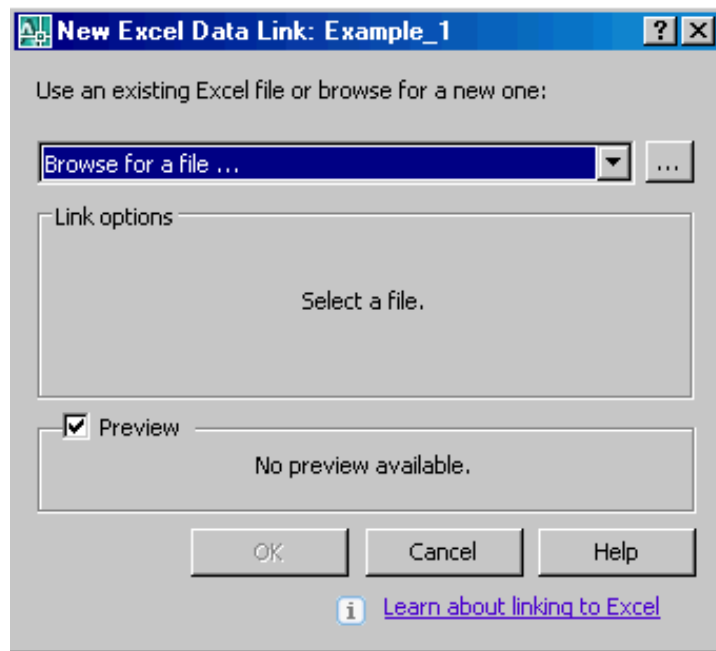From the drop down menu click on Launch Data Link Manager.

Insert Table

Table style
Standard

Insert options
- ○ Start from empty table
- ⦿ From a data link

  Launch Data Link Manager ...
- ○ From object data in the drawing (Data Extraction)

Preview

Title		
Header	Header	Header
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data

Insertion behavior
- ⦿ Specify insertion point
- ○ Specify window

Column & row settings
- Columns: 9
- Column width: 7.0000
- Data rows: 10
- Row height: 3 Line(s)

Set cell styles
- First row cell style: Title
- Second row cell style: Header
- All other row cell styles: Data

Learn about Tables

OK   Cancel   Help

➤ **Following graphic is displayed. Click on Create a new Excel Data Link.**
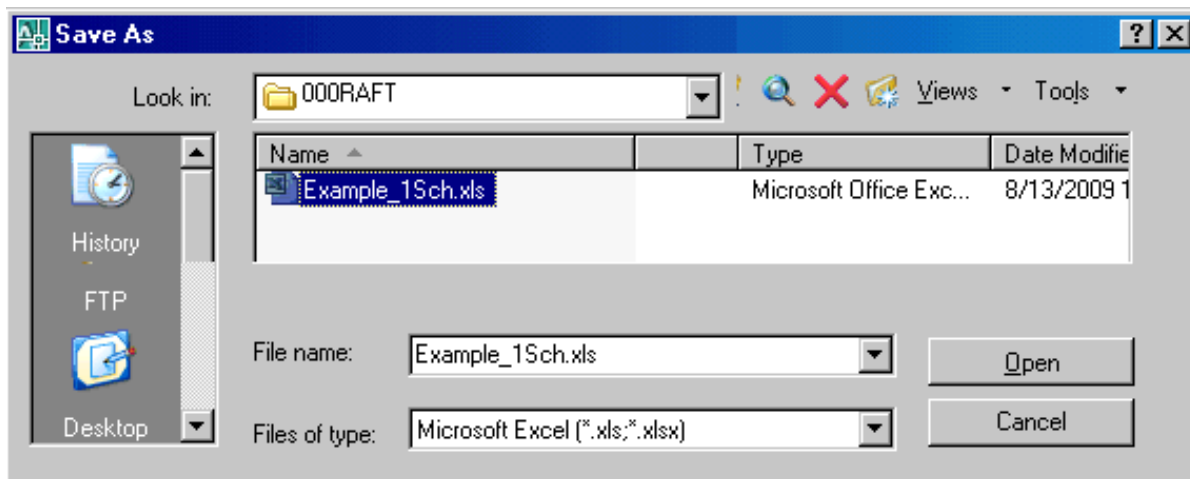
A dialogue box appears asking you to Enter a name.
You can Enter any name for e.g. Example_1.
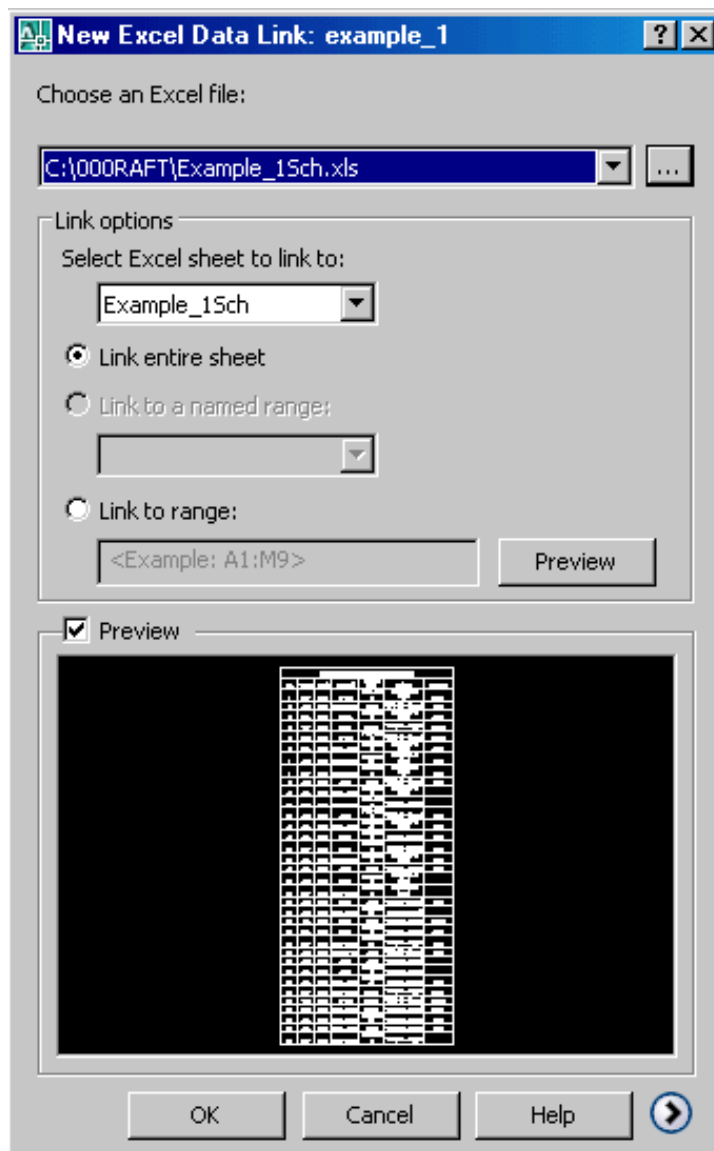**Click on OK.**

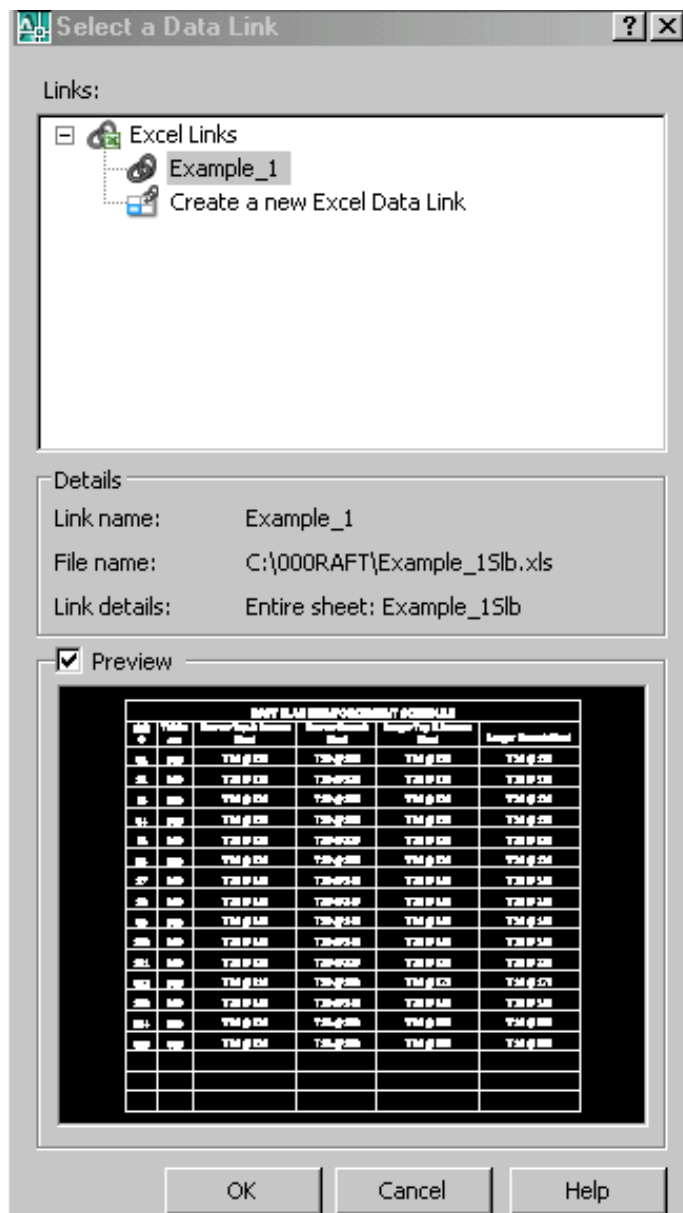**Following Graphics will appear. Click on Browse for a file.**

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.
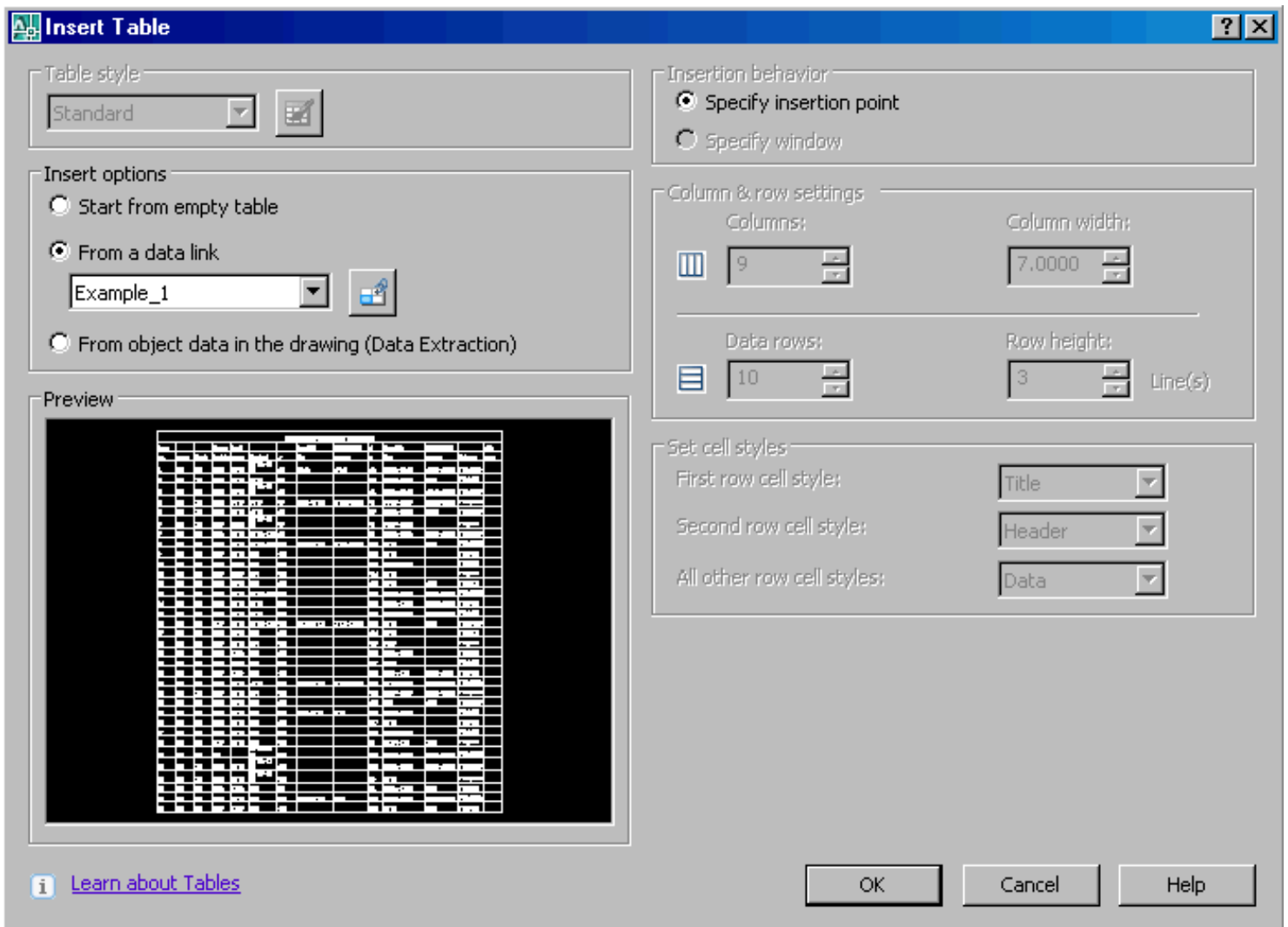

Following Graphics appears showing the preview of the table in AutoCAD. Click on OK.

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

Another dialogue box appears . Click on OK.

Insert Table dialog box

Table style	Insertion behavior
Standard	● Specify insertion point
	○ Specify window

Insert options
- ○ Start from empty table
- ● From a data link
  - Example_1
- ○ From object data in the drawing (Data Extraction)

Column & row settings

	Columns:	Column width:
	9	7.0000

	Data rows:	Row height:
	10	3 Line(s)

Set cell styles

First row cell style:	Title
Second row cell style:	Header
All other row cell styles:	Data

Preview

ℹ Learn about Tables

[OK] [Cancel] [Help]

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

RAFT BEAM REINFORCEMENT SCHEDULE

No.	Width	Depth	Top Steel	Bottom Steel	Strirrups	Skin Steel
1	500	1500	15 T 32	4 T 25 + 2 T 32	T 12 @ 100 - 4 Legged	8 T 10
2	400	1200	3 T 32 + 3 T 25	8 T 32	T 8 @ 260 - 4 Legged	8 T 8
3	500	1500	15 T 32	2 T 32 + 3 T 25	T 12 @ 100 - 4 Legged	8 T 10
4	750	2000	9 T 32	17 T 32	T 12 @ 75 - 4 Legged	18 T 10
5	800	2000	15 T 32	4 T 25 + 2 T 32	T 12 @ 60 - 4 Legged	18 T 10
6	500	1500	2 T 25 + 2 T 20	2 T 32 + 3 T 25	T 10 @ 120 - 4 Legged	8 T 10
7	400	1200	4 T 32 + 2 T 25	3 T 20 + 1 T 12	T 12 @ 105 - 4 Legged	8 T 8
8	400	1200	3 T 32 + 3 T 25	3 T 32 + 2 T 25	T 12 @ 100 - 4 Legged	8 T 8
9	400	1200	4 T 25	5 T 25	T 10 @ 90 - 4 Legged	8 T 8

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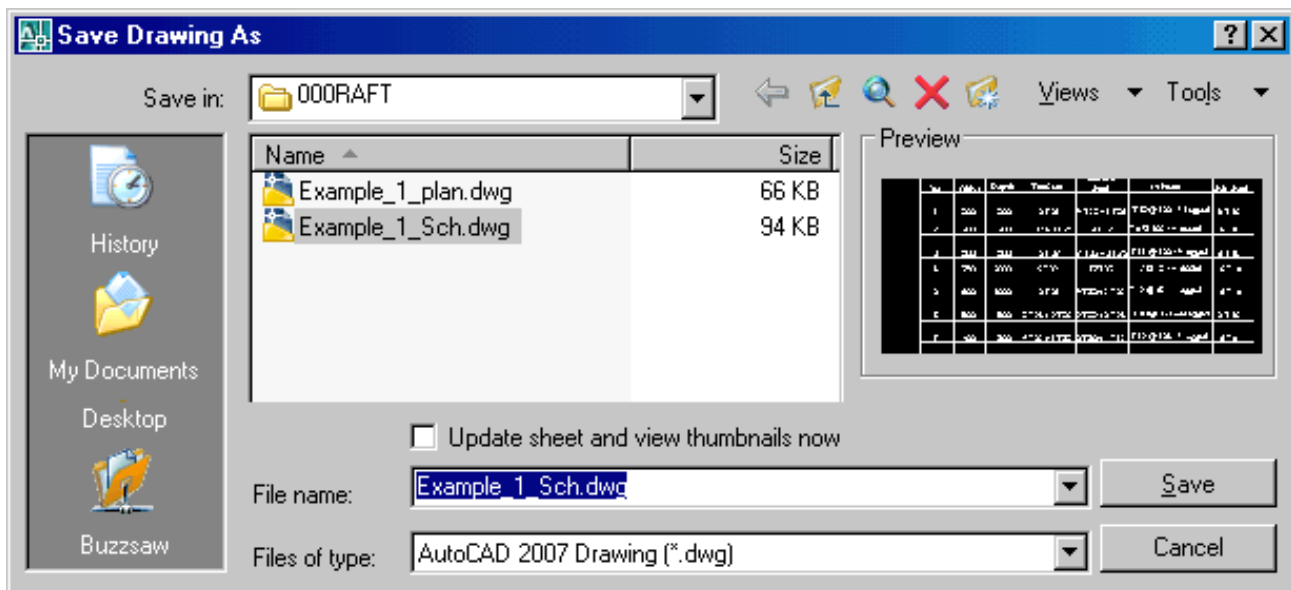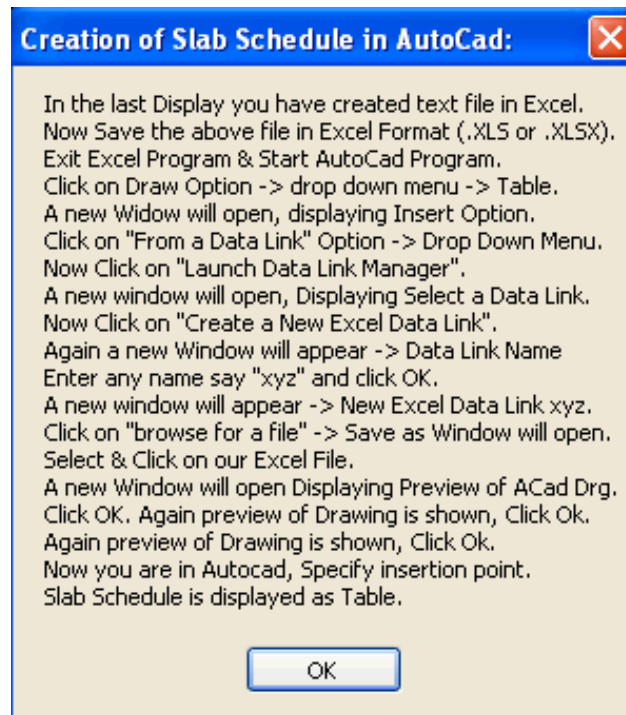

STEP 13 IS OVER

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

LEARN RAFT STEP BY STEP
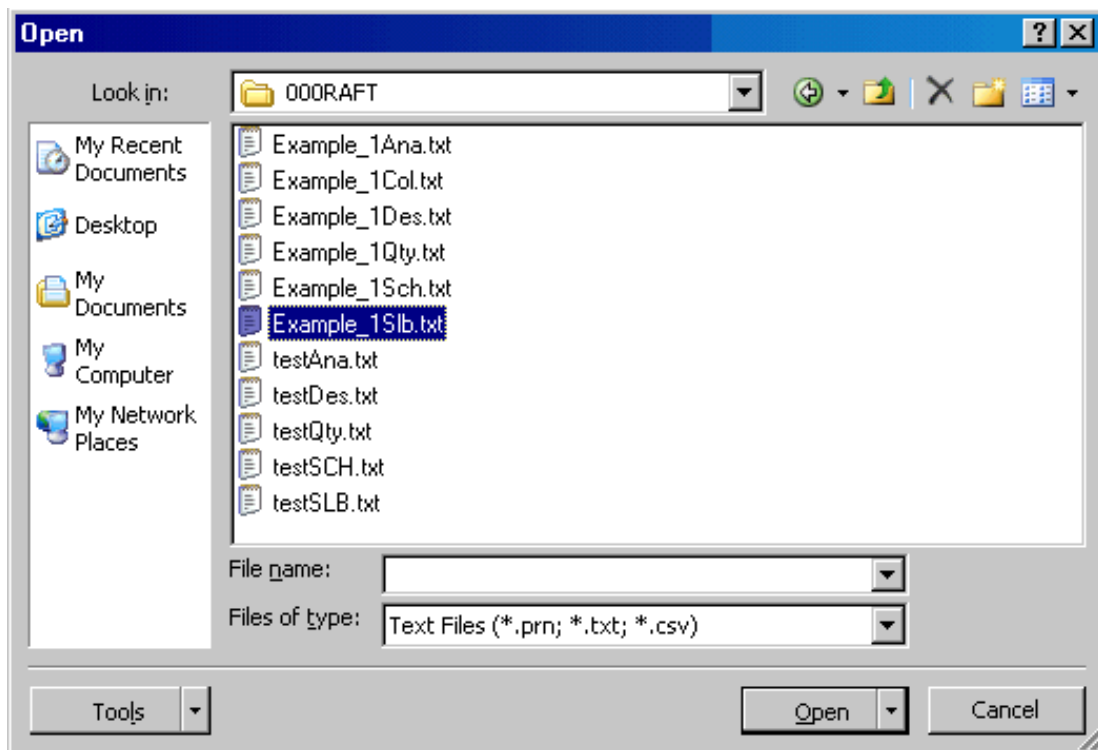
STEP 14: CREATION OF SLAB SCHEDULE IN AUTOCAD

**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.**

Creation of Slab Schedule in AutoCad:

In the last Display you have created text file in Excel.
Now Save the above file in Excel Format (.XLS or .XLSX).
Exit Excel Program & Start AutoCad Program.
Click on Draw Option -> drop down menu -> Table.
A new Widow will open, displaying Insert Option.
Click on "From a Data Link" Option -> Drop Down Menu.
Now Click on "Launch Data Link Manager".
A new window will open, Displaying Select a Data Link.
Now Click on "Create a New Excel Data Link".
Again a new Window will appear -> Data Link Name
Enter any name say "xyz" and click OK.
A new window will appear -> New Excel Data Link xyz.
Click on "browse for a file" -> Save as Window will open.
Select & Click on our Excel File.
A new Window will open Displaying Preview of ACad Drg.
Click OK. Again preview of Drawing is shown, Click Ok.
Again preview of Drawing is shown, Click Ok.
Now you are in Autocad, Specify insertion point.
Slab Schedule is displayed as Table.

OK

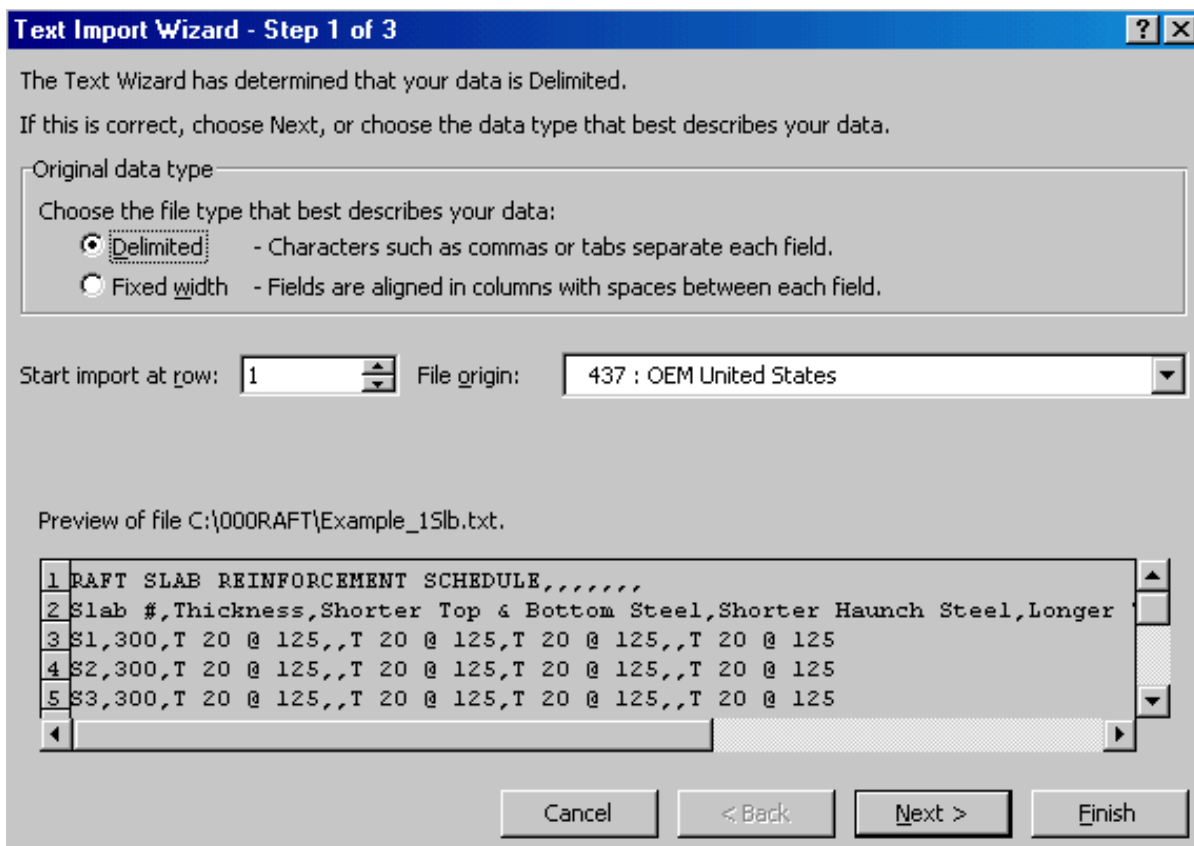➤ **Start Microsoft Excel . Click On Open. Following Graphics is Displayed.**

Click on Example_1Slb.txt.
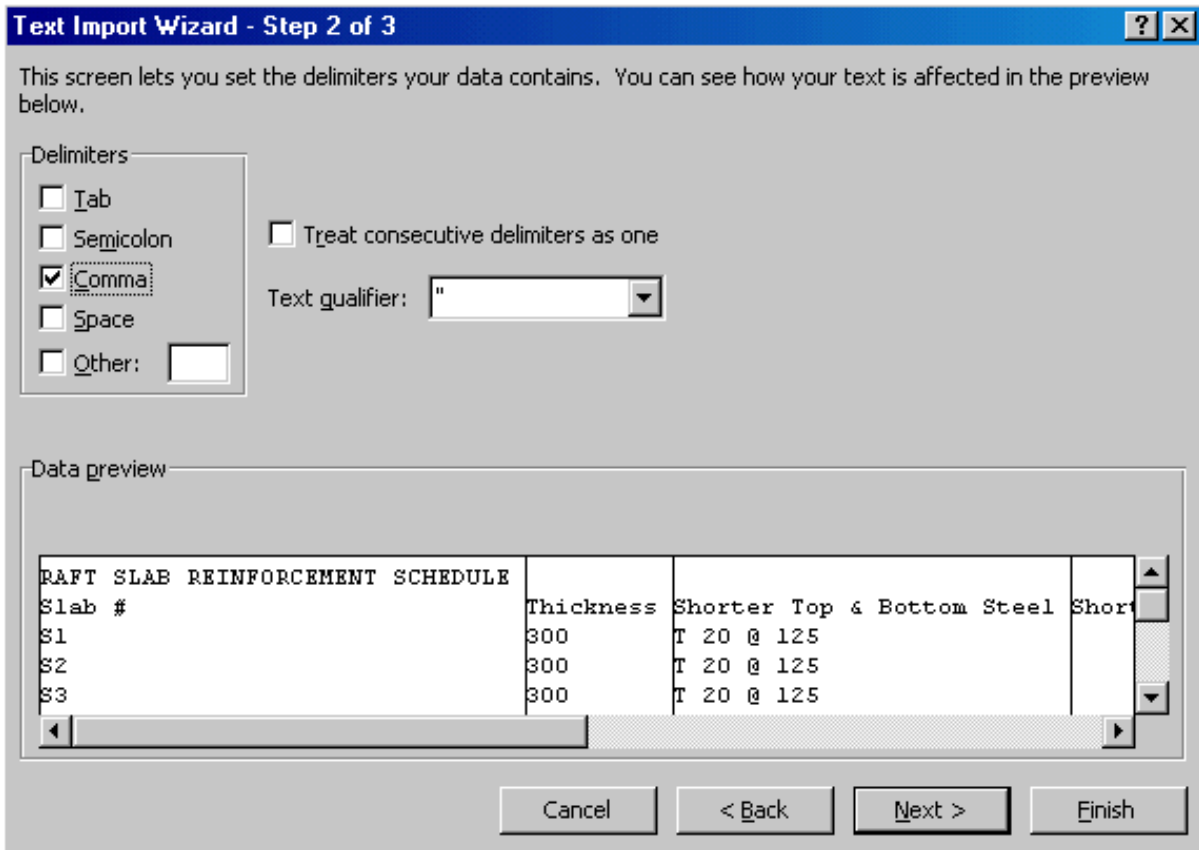**As you can see, the above file is in text format.**
➤ **In the following steps we will save the file in Excel format.**
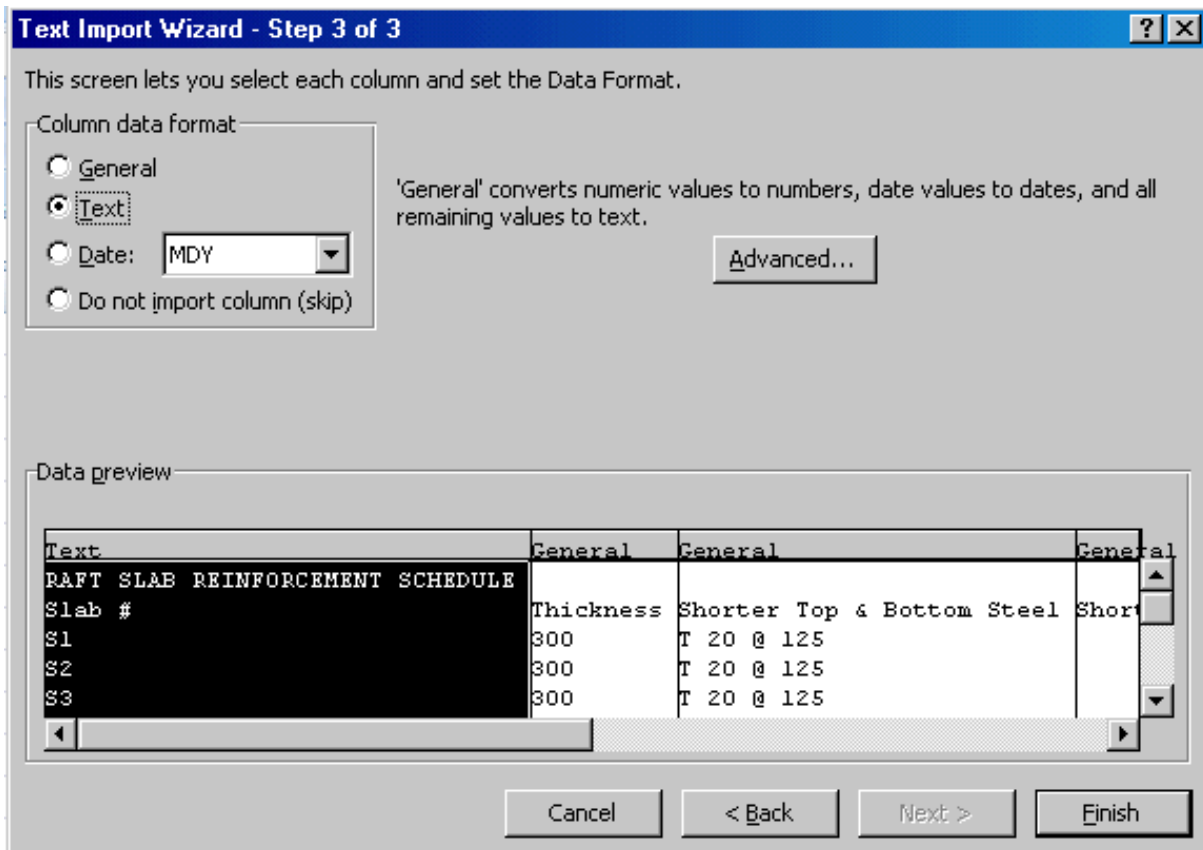**Once Example1_Slb.txt is clicked, following graphics is displayed.**

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

Text Import Wizard - Step 2 of 3

This screen lets you set the delimiters your data contains. You can see how your text is affected in the preview below.

Delimiters

☐ Tab
☐ Semicolon   ☐ Treat consecutive delimiters as one
☑ Comma
☐ Space   Text qualifier: [" ▼]
☐ Other: [   ]

Data preview

```
RAFT SLAB REINFORCEMENT SCHEDULE
Slab #                            Thickness Shorter Top & Bottom Steel Short
S1                                300       T 20 @ 125
S2                                300       T 20 @ 125
S3                                300       T 20 @ 125
```

[Cancel]   [< Back]   [Next >]   [Finish]

➤ **As shown Above choose Comma as Delimiter. Click On Next. Following graphics are displayed.**

Text Import Wizard - Step 3 of 3

This screen lets you select each column and set the Data Format.

Column data format

○ General
● Text   'General' converts numeric values to numbers, date values to dates, and all remaining values to text.
○ Date: [MDY ▼]
○ Do not import column (skip)   [Advanced...]

Data preview

```
Text                              General   General                    General
RAFT SLAB REINFORCEMENT SCHEDULE
Slab #                            Thickness Shorter Top & Bottom Steel Short
S1                                300       T 20 @ 125
S2                                300       T 20 @ 125
S3                                300       T 20 @ 125
```

[Cancel]   [< Back]   [Next >]   [Finish]

89

**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..

RAFT SLAB REINFORCEMENT SCHEDULE

Slab #	Thickness	Shorter Top & Bottom Steel	Shorter Haunch Steel	Longer Top & Bottom Steel	Longer Haunch Steel
S1	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S2	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S3	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S4	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S5	350	T 20 @ 120	T 20 @ 120	T 20 @ 120	T 20 @ 120
S6	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S7	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S8	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S9	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S10	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S11	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S12	550	T 20 @ 110	T 20 @ 110	T 16 @ 175	T 16 @ 175
S13	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S14	660	T 16 @ 120	T 16 @ 120	T 16 @ 150	T 16 @ 150
S15	660	T 16 @ 120	T 16 @ 120	T 16 @ 150	T 16 @ 150

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.
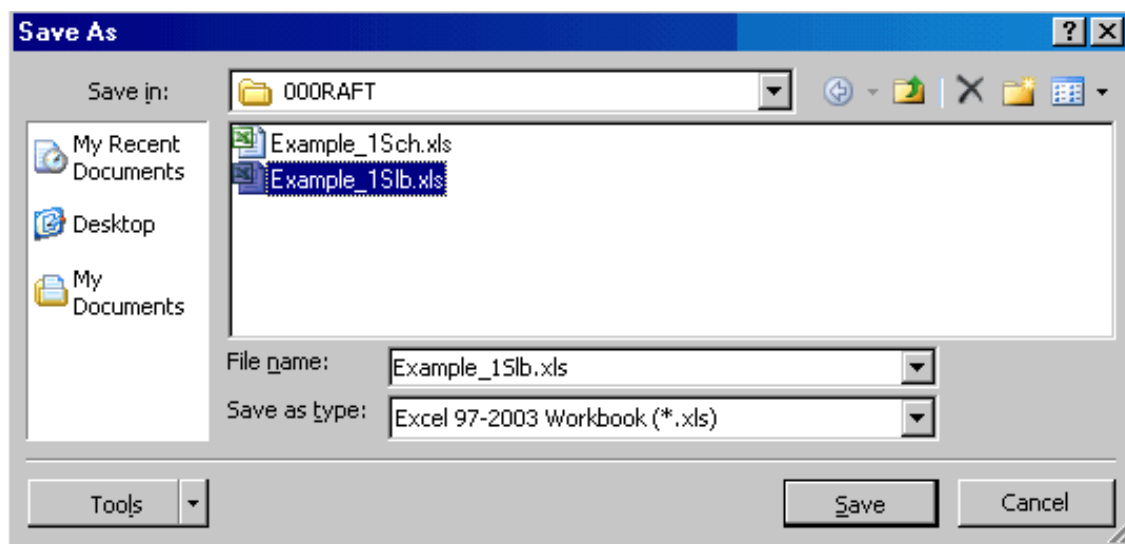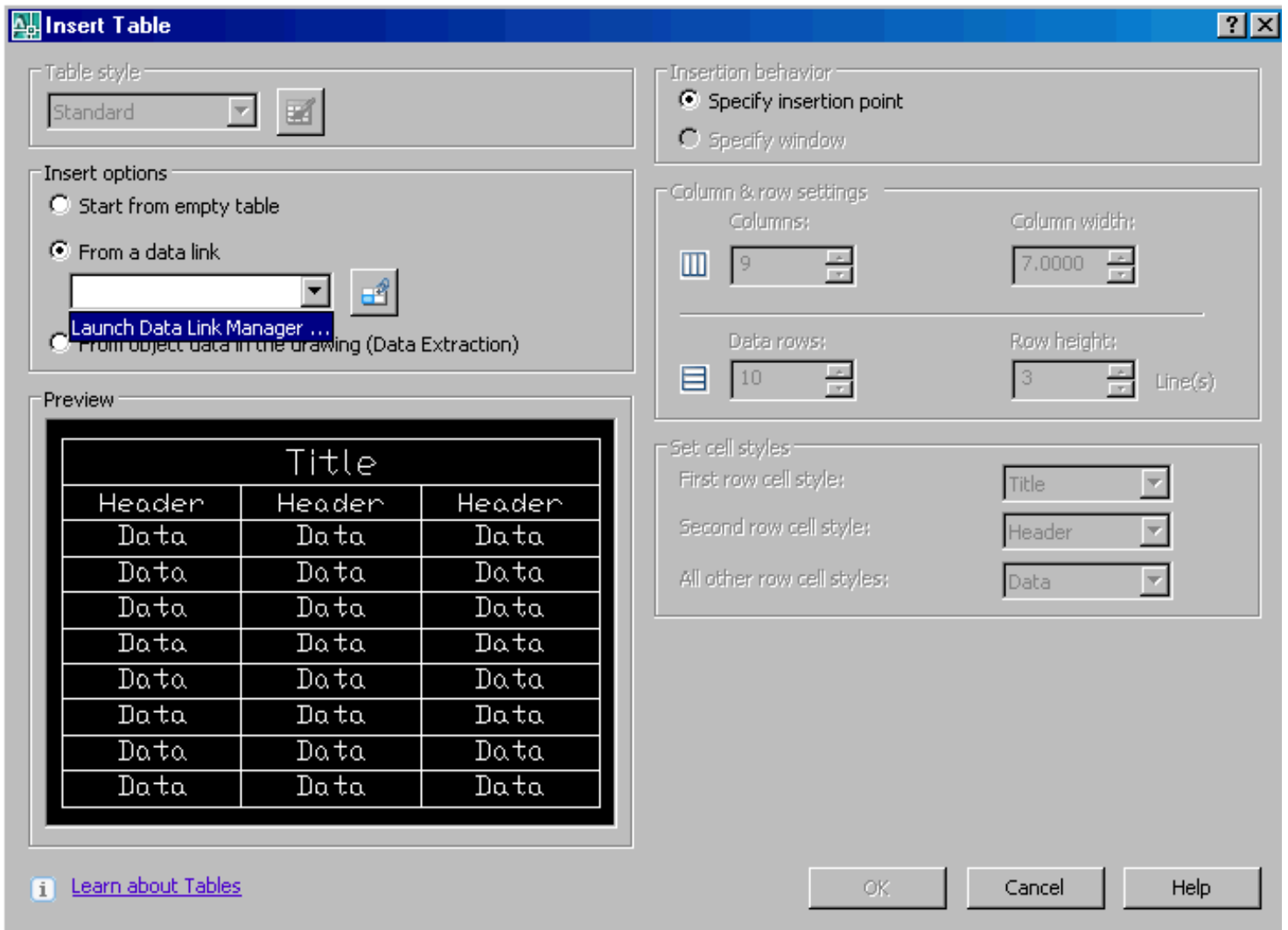

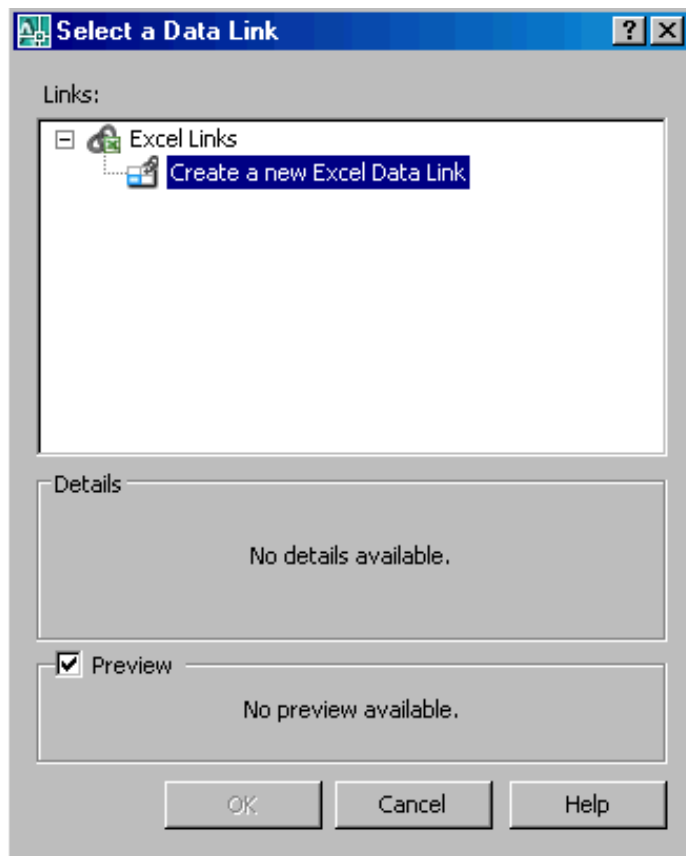**Start AutoCAD. Click on DRAW . From the drop down menu click on Table a shown below.**
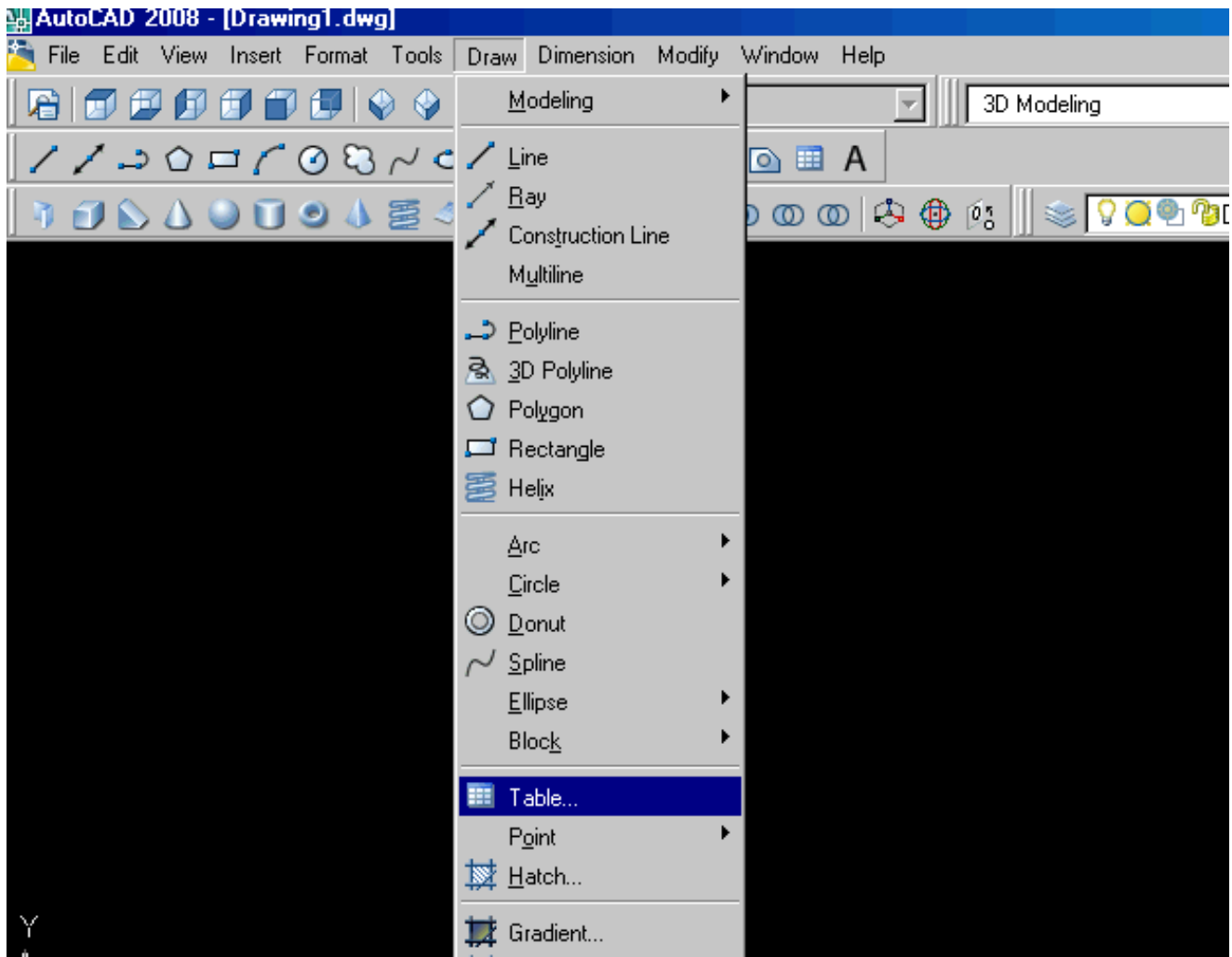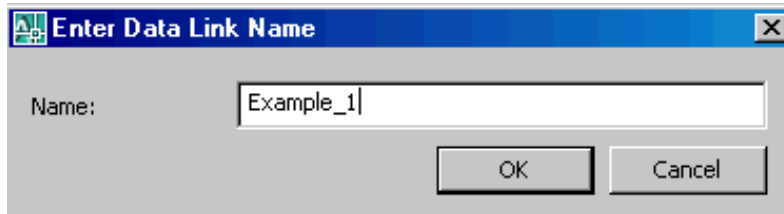
A dialogue box will appear.

➤ As show below click on From a data link .

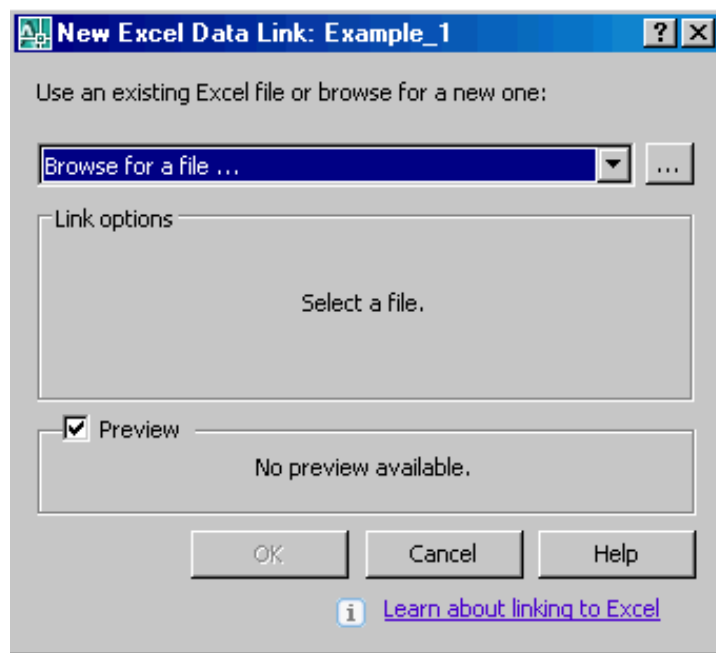From the drop down menu click on Launch Data Link Manager.

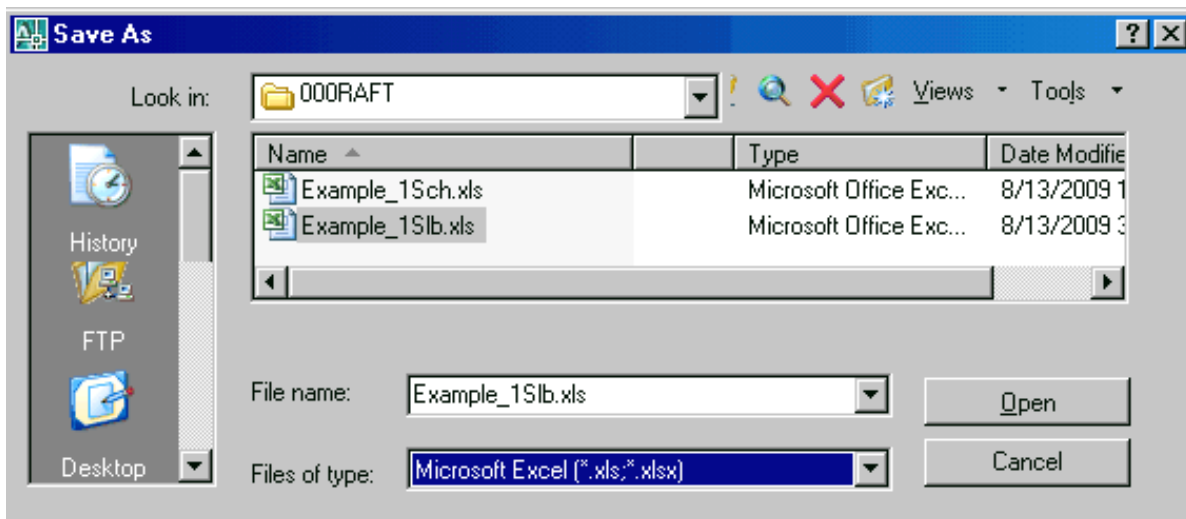**Following graphics are displayed. Click on Create a new Excel Data Link.**

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.
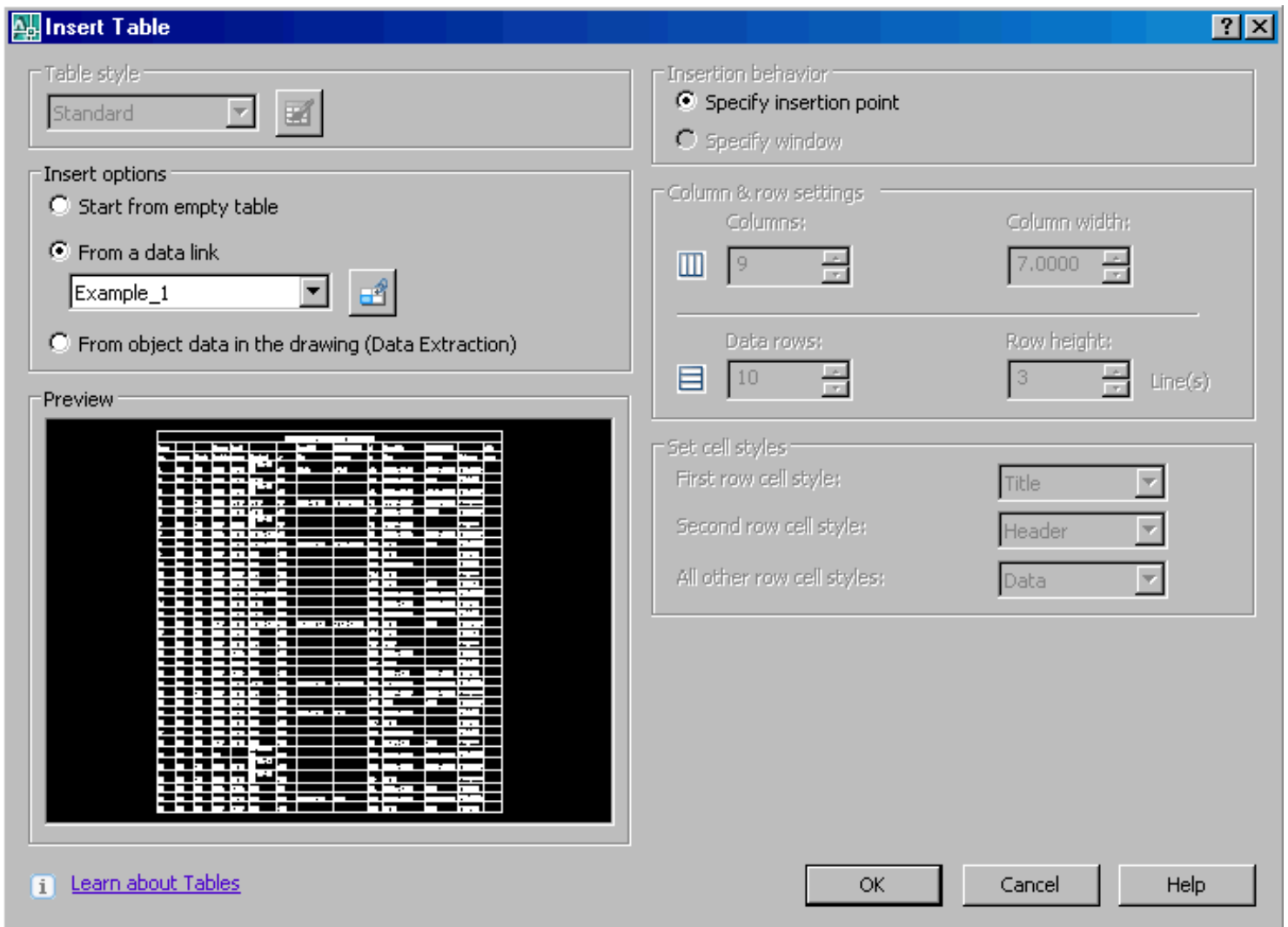

**Following Graphics appears showing the preview of the table in AutoCAD. Click on OK.**

**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.**

RAFT SLAB REINFORCEMENT SCHEDULE

Slab #	Thickness	Shorter Top & Bottom Steel	Shorter Haunch Steel	Longer Top & Bottom Steel	Longer Haunch Steel
S1	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S2	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S3	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S4	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S5	350	T 20 @ 120	T 20 @ 120	T 20 @ 120	T 20 @ 120
S6	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S7	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S8	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S9	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S10	300	T 20 @ 145	T 20 @ 145	T 20 @ 145	T 20 @ 145
S11	300	T 20 @ 125	T 20 @ 125	T 20 @ 125	T 20 @ 125
S12	550	T 20 @ 110	T 20 @ 110	T 16 @ 175	T 16 @ 175

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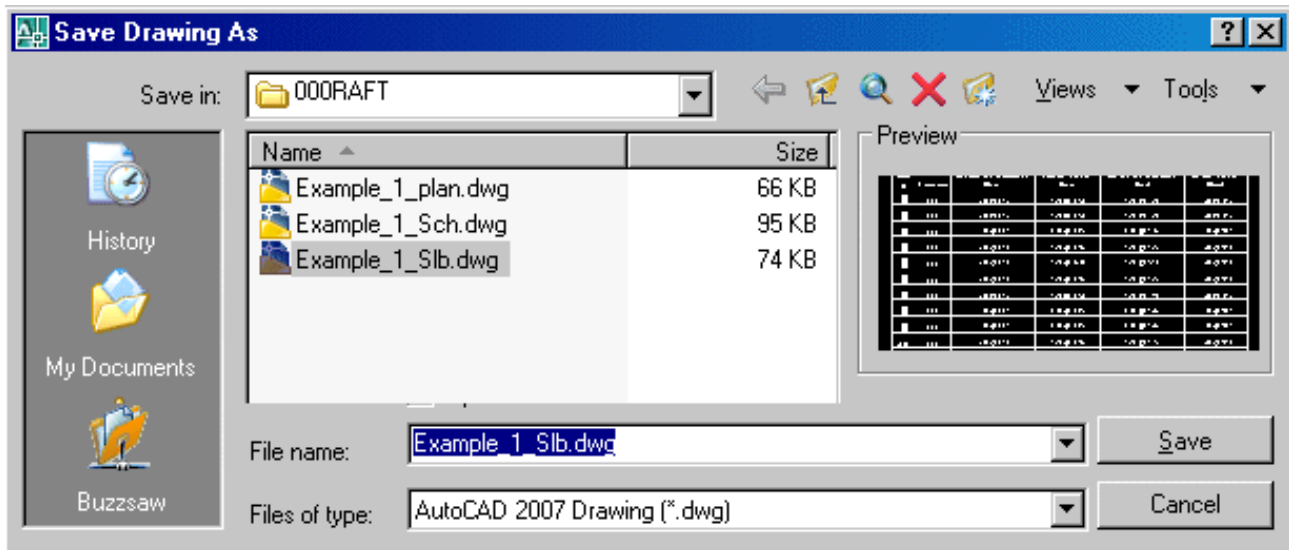
Learn RAFT is over.

OTHER SOFTWARES:

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

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

2D FRAME ANALYSIS - Discover The Beauty Of Structural Analysis

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

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

Q T Y - Quantity Estimation & Cost, Project Control

SUPER REAL VALUATION -   A Software For Immovable Properties

ROADS -   Pavement Design & Rate Analysis Of Road Items

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

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

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

BILLING JI   -   A Database Management Software For General Billing

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

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

BID ANALYSIS -   A Software For Technical & Commercial Tender Analysis

STEEL_2OO7 - 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