ProtaStructure®

ProtaSteel®

ProtaDetails®

ProtaBIM®

ProtaStructure 2026

Flat Slab Training Manual

Version 4.0 27 Nov 2026

Please get in touch with us for your training and technical support queries

asiasupport@protasoftware.com

globalsupport@protasoftware.com

PROTA

Limitation of Responsibilities

Prota shall not be held responsible for any losses caused by documentation, software, or usage errors.

In addition to Prota License Agreement Terms, it is the responsibility of the user

- to check of results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- make sure that the software is used correctly per the reference manual and documentation,

Intellectual Property

ProtaStructure is a registered trademark of **Prota Software Inc.** All intellectual property rights belong to **Prota Software Inc**. Documentation, training, and reference manuals, and program components can not be copied, distributed, and used in violation of the license agreement.

Trademarks

ProtaStructure®, **ProtaDetails®**, **ProtaSteel®** ve **ProtaBIM®** are registered trademarks of Prota Software Inc. Prota logo is a trademark of Prota Software Inc.

Table of Contents

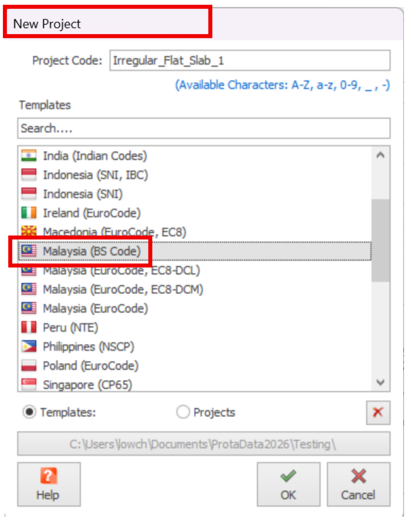
Starting a new project & Import Members

In this example a DXF drawing is imported into ProtaStructure. This drawing can then be used for the following features:-

- Automatic Import of Axis
- Automatic Import of Column Sections
- Use the DXF as Ghost Layer to accurately create the elements and loads
- Snap to locations on the DXF
- Import a different DXF at each level (St) of the structure
- Provides quick effective checking for changes to the scheme by importing revised drawings

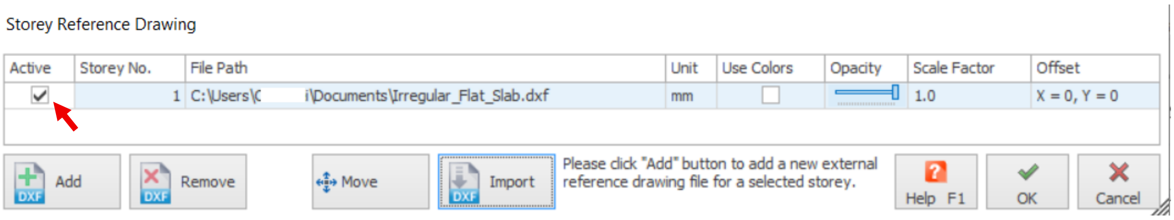
➢ *Create a **New project** with the name "Irregular_Flat_Slab"*

➢ *Select '**Malaysia (BS8110)**' template and **Import** to current project settings*



➢ *Click on the '**External Reference Drawing**' icon in the **Building Setout** tab*

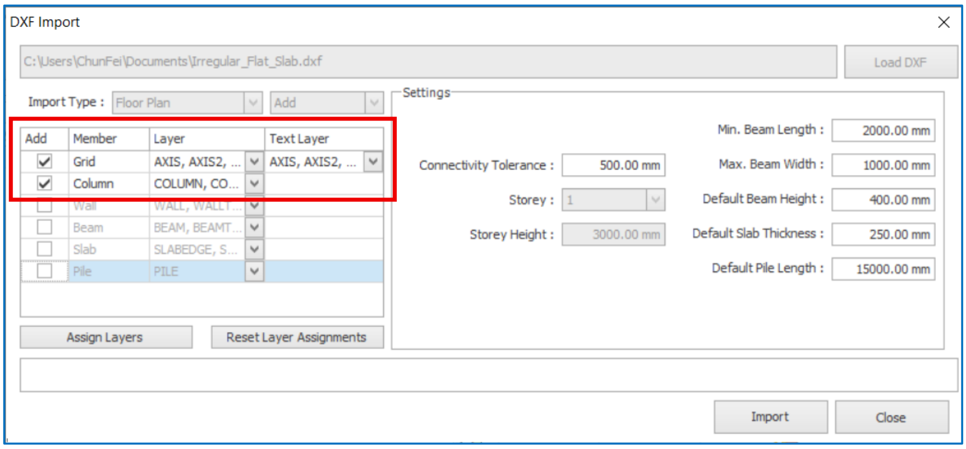
➢ ***Add** the given dxf file '**Irregular_Flat_Slab.dxf**' (DXF drawing can be downloaded from this link)*



➢ *Check **Active***

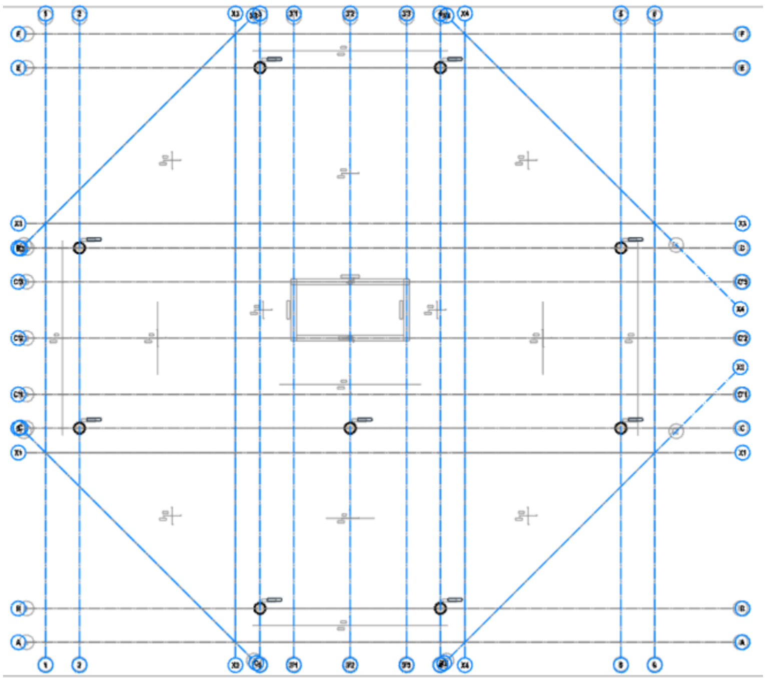
The reference file is shown in the background. We will now import the axes and column using the Import DXF function

➢ *Click **Import***



- *Under Add, check **Grid** and **Column***
- *For **Grid Layer**, choose only **AXIS** & **AXIX2***
- *For **Grid Text Layer**, choose on **AXIS** & **AXIS2***
- *For Column Layer, choose only **COLUMN***
- *Click **Import** **>** **Close >** **OK** to close the dialog*

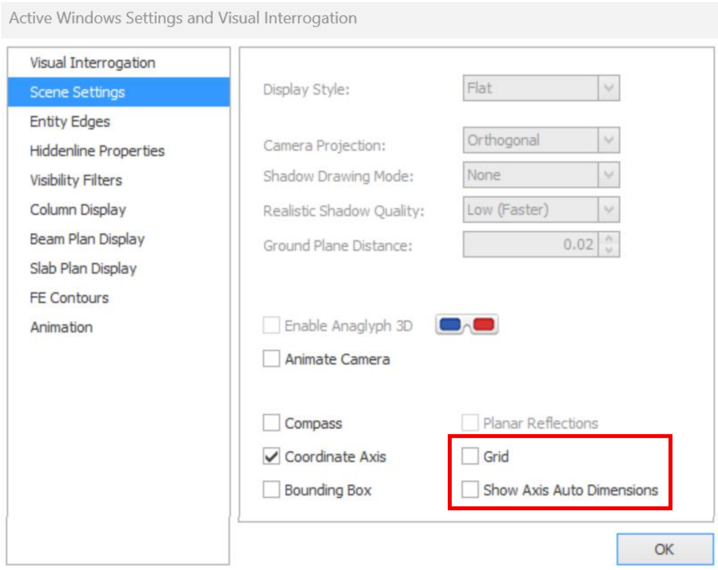
The grids columns will be imported as shown below with external drawing in background.



If the external reference drawing is not shown in background, click **Refresh** icon (leftmost icon just above the structure tree)

To get a less cluttered view, switch off the background grid & grid dimensions.

- *Go to **Review** tab **> Visual Interrogation > Scene Settings >** Uncheck 'Grid' & 'Show Auto Dimensions' > OK*

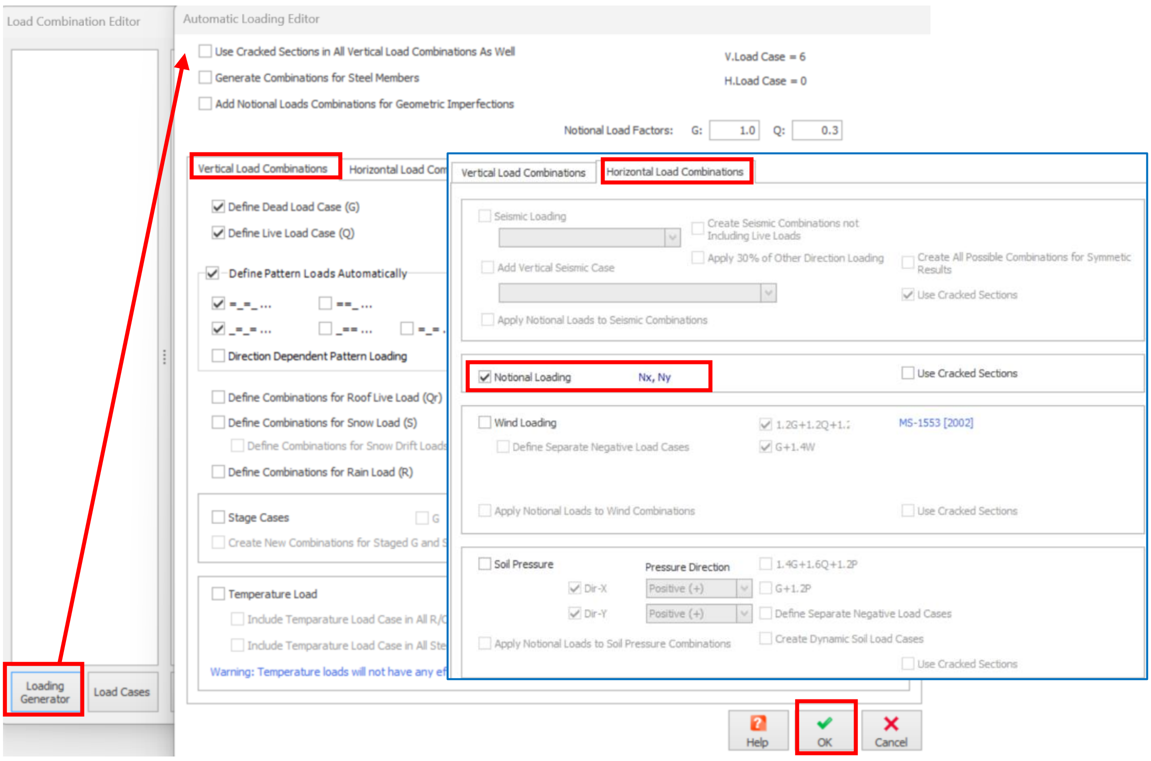


The background grey grids & grid dimensions will be switched off.

Generate Load Combinations

We will now proceed to setup the load combinations before proceeding to create other members.

- ***Loading** tab > **Load Cases and Combinations***
- *Click **Loading Generator** > **Vertical Load Combinations*** > Use all defaults.
- *Go to **Horizontal Load Combinations** > Check "**Notional Loading**" > **OK***



The combinitions will be generated as shown below.

Load Combination Editor

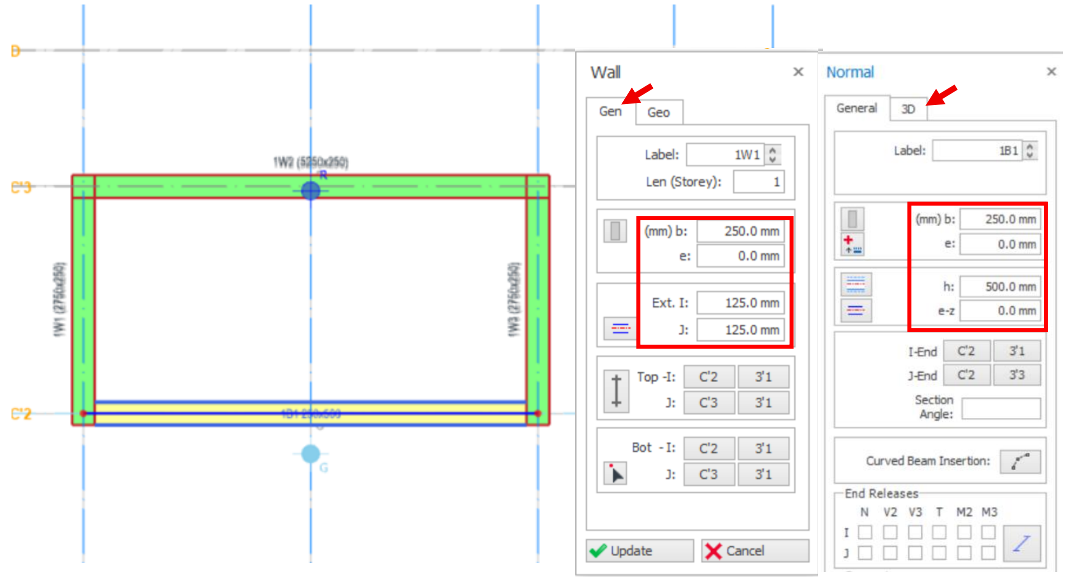
- ULS_RC
- ULS_Steel
- SLS_RC
- SLS_Steel

ID	Label	LL Red	VOM	T...	Load Combinations
1	1.4G+1.6Q	☑	☑	↓	1.4G + 1.6Q
2	G+0.4Gp1+1.6Qp1	☑	☑	↓	G + 0.4Gp1 + 1.6Qp1
3	G+0.4Gp2+1.6Qp2	☑	☑	↓	G + 0.4Gp2 + 1.6Qp2
5	1.2G+1.2Q+NGx+0.3NQx	☑	☐	→	1.2G + 1.2Q + NGx + 0.3NQx
6	1.2G+1.2Q-NGx-0.3NQx	☑	☐	→	1.2G + 1.2Q - NGx - 0.3NQx
7	1.2G+1.2Q+NGy+0.3NQy	☑	☐	→	1.2G + 1.2Q + NGy + 0.3NQy
8	1.2G+1.2Q-NGy-0.3NQy	☑	☐	→	1.2G + 1.2Q - NGy - 0.3NQy

➢ *Click **OK** to save & exit the dialog.*

Create walls & beam

➢ *Create **250mm** thick core walls and a **250mm x 500mm** beam as shown below*



Inserting slabs

There are different techniques for inserting slabs. The options can be changed in **Slab Properties > General tab >** *Insertion*. Each one has its own merits; which to use is your decision.

- **Axis Region**: Insert slab based on the nearest axes as boundary. Hold own **CTRL** & pick several axes region to combine **>** Release CTRL key & single slab in the combined region will be created.
- **Beam Region**: This is of no use for flat slab models due to the absence of beams. This is to be used only for beam & slab model.
- **Pick Axis**: Select the axes one by one consecutively in **clockwise** order to define the boundary of the slab **>** Ensure they are listed in the box **>** Pick back the **first** axis **>** Slab will be auto created.
- **Pick Points**: Pick the intersection of axes that forms the corner of the slab in **clockwise** order **>** Once the loop is closed, the slab will be auto created.
- **Pick Closed Slab Edge**: Insert the slab using closed slab edge line without need for surrounding by grids. Slab edge lines must be created first, e.g. using **Polyline Slab Edge** function.

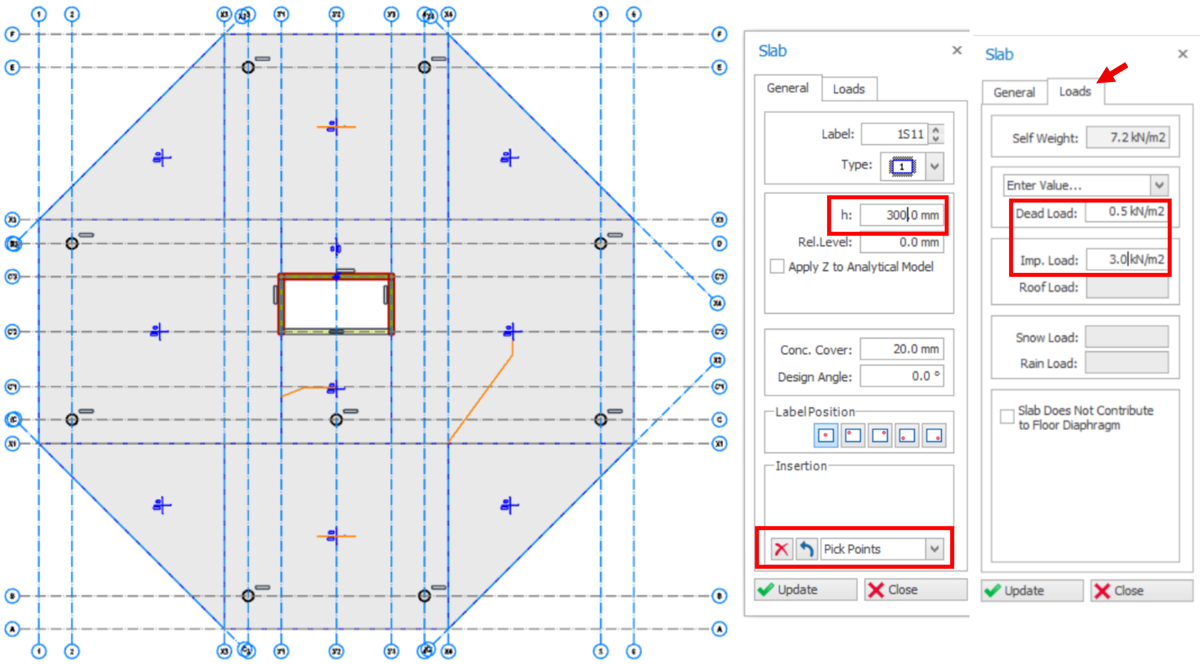
If slabs cannot be inserted by any of the above methods, click "***Delete All Axes***" & start again. Also, try closing the slab properties & start again.

Important! Before creating the slabs in a Flat Slab model it is paramount that the layout of the slab panels is given consideration, and the following guidelines are met :

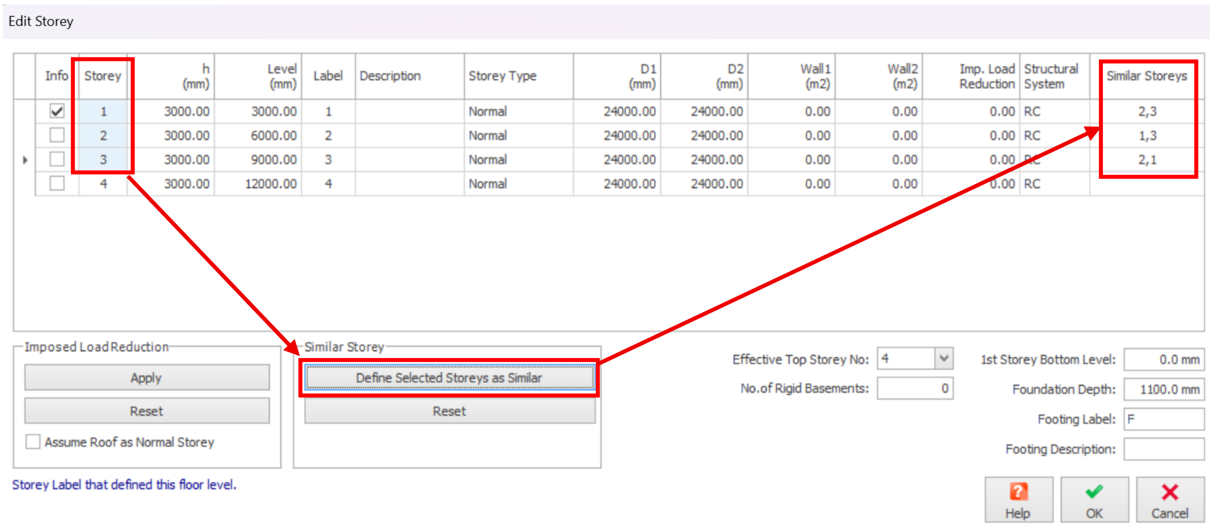
- All Walls and Beams (if any) must lie on slab boundaries or edge; i.e. a wall or beam must not overlap with any slab.
- Columns can sit within slab panels; i.e. a column can overlap with slab.
- Slab boundaries sharing the same grid line will be continuous in the FE model
- Slab panels should be as large as possible (lots of small panels will complicate the FE)
- Slabs should have the minimum edges possible (triangle/square/rectangle). Irregular shaped panels L, etc. should be avoided
- There is No Right or Wrong layout for the slab panels, but by adhering to the above, slab layouts should be simple and effective when entering the FE environment
- Ignore the yield-lines shown on the plan view (they are only valid if the slabs are supported by beams).

➢ *Insert all **300mm** thk. slabs with an **Additional Dead Load** of **0.5kN/m^2**, and an **Imposed Load** of **3kN/m^2**.*
➢ *Proceed to insert **ST04** via **Building Setout** tab > **Storey Operations** > **Insert Storey***

The final model will have a total 4 storeys with each storey height = 3000 mm.

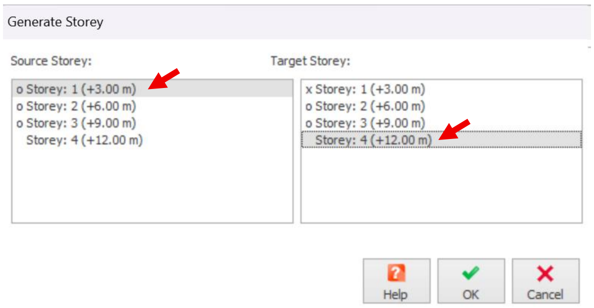


- *Make all ST01, 02 & 03 storeys* ***similar*** *via* ***Building Setout*** *>* ***Storey Operations*** *>* ***Edit Storey*** *(Multiple select 3 Storeys &* ***click Define Selected Storeys as Similar****)*



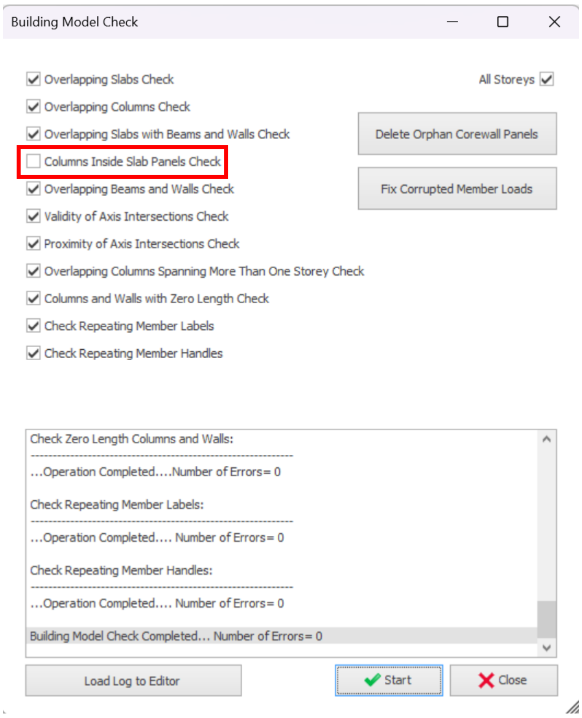
ST04 cannot be similar storey because it's the roof & hence both plan & elevation are not similar to rest of the storeys.

- *For* ***ST04****, go to* ***Building Setout*** *>* ***Storey Operations*** *>* ***Generate Storey*** *> Copy all members from* ***ST01 (Source)*** *to* ***ST04 (Target)***



Before & after copying any storey's information to the other floors, it is suggested to perform the Building Model Check.

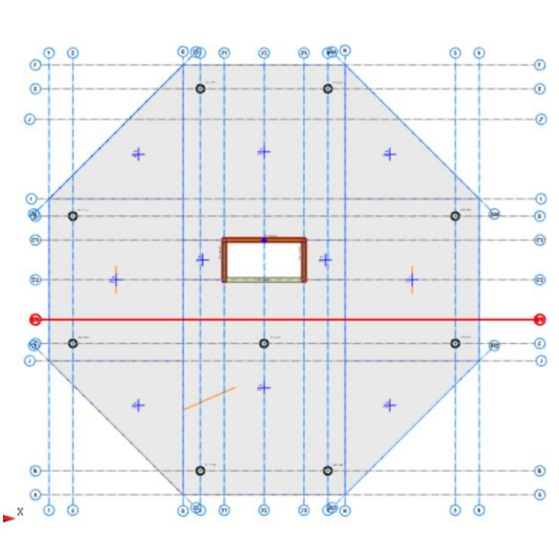
- *Go to the* ***Review*** *tab > Click* ***Building Model Check***

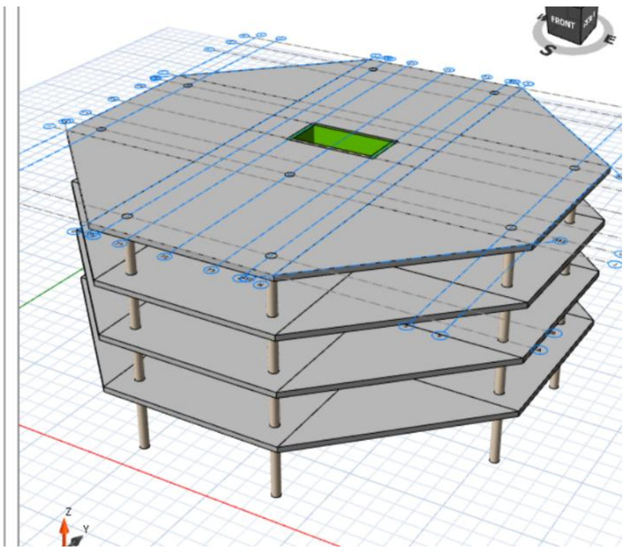


- ***Uncheck*** *"Column Inside Slab Panels"*
  This check is for beam slab layout. For flat slab, the columns can overlap with slabs, hence we uncheck this option.
- *Click* ***Start*** *to check*
  Ensure there are no errors. If there are errors, you can click "Load Log to Editor" to show the errors in separated NotePad so it's easier to refer for correction.
- *Click* ***Close*** *to exit.*

Final Flat Slab Model

This is the Flat Slab Model that will be analysed and designed as shown below :





Consider the model shown above. A core wall system is assumed to brace the structure and therefore the flat slabs and peripheral columns are predominantly to be designed for gravity loads only.

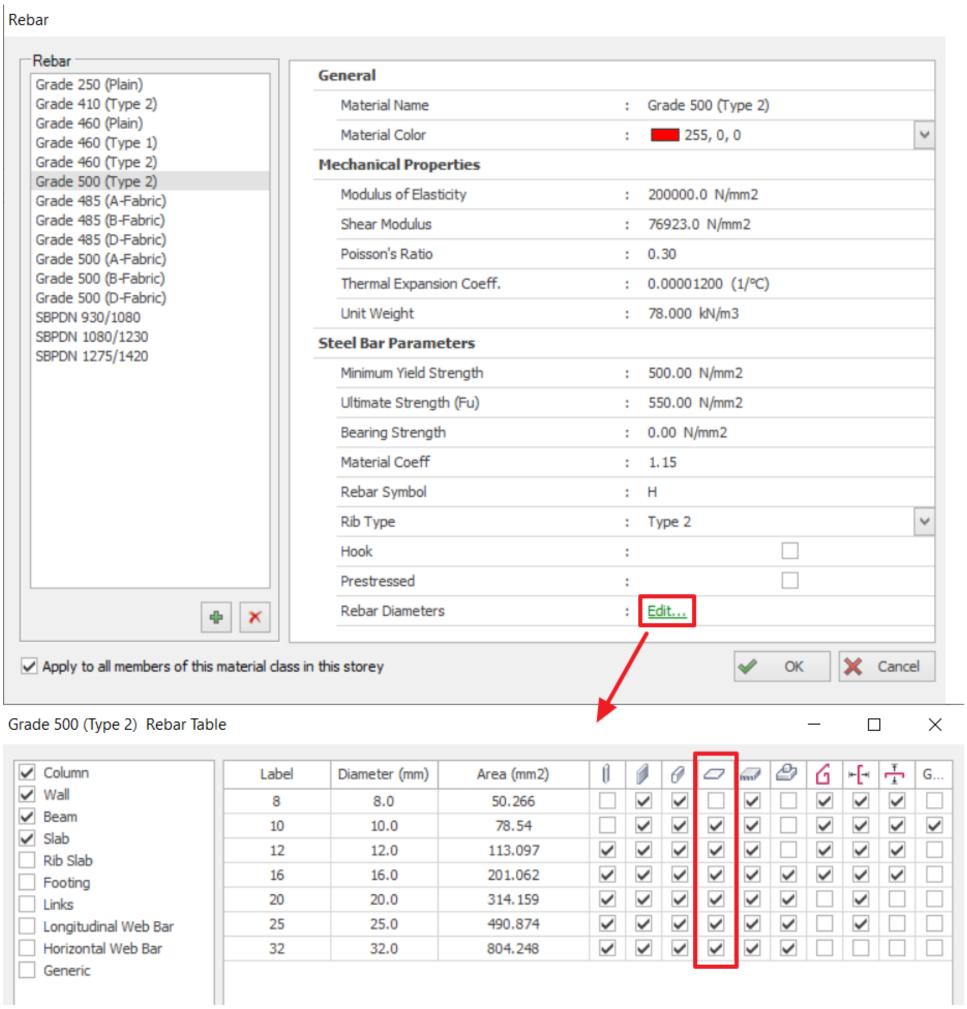
Although it looks like a simple layout, it would be very difficult to apply the idealisation of column and middle strips; hence, it becomes almost impossible to apply the BS8110 code's simplified design methods. Therefore, the ProtaStructure Finite Element Analysis is going to be used to determine the gravity Loads on the slabs, columns and walls.

- Continue with the model you have setup in the former section, or use the given model '***Irregular Flat Slab***'. (Zipped model can be download from link)

Material Properties

- *Go to **Building Setout** tab and click **Materials***
- *Click on the **Concrete Grades** button for Slabs*
- *Select **C40** making sure the Short term **Modulus of Elasticity = 28000 N/mm^2***
- *Make sure the **Unit Weight of Concrete = 24 kN/m^3***
- *Tick the **'Apply to All Member Types'***
- *Click **OK** in the 'Concrete Grade' window*
- *Using similar techniques as used to change the Concrete Grades, specify **Grade 500** steel with a Steel **Material Coefficient of 1.15***
- *Check the **Bar Sizes** for **Slabs** are set to **H10/H12/H16/H20/H25/H32***
- *Click **OK** in the 'Material' window*
- All element types have a Consistent Set of Section Properties

These settings for the Material Properties apply to both the Building Analysis and the Finite Element Analysis.

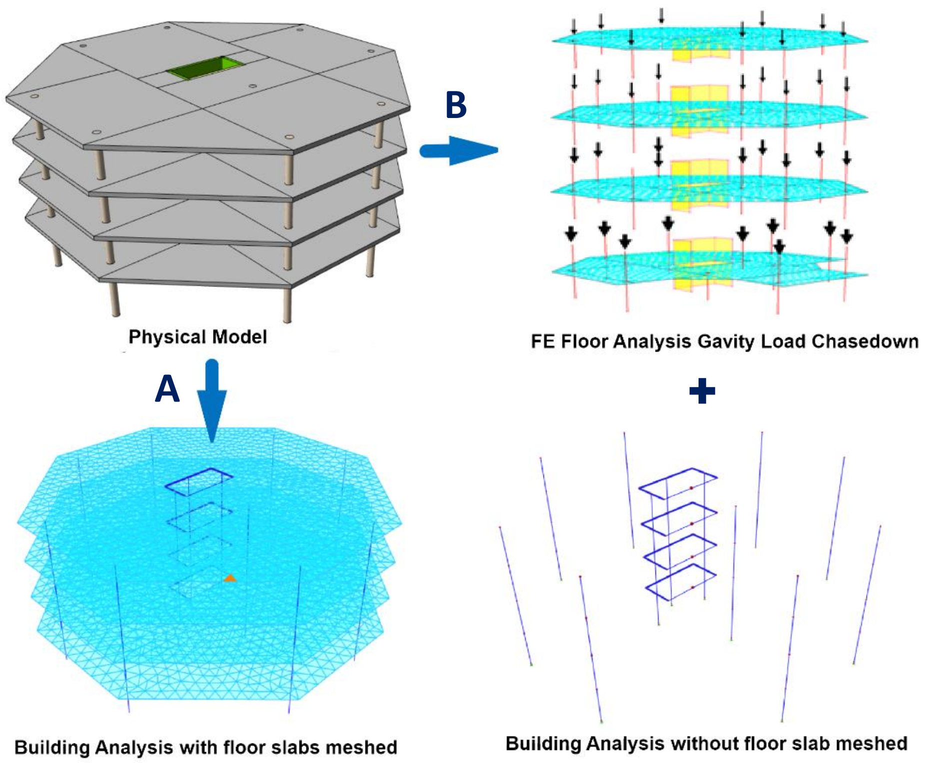


Flat Slab Analysis & Design Flow Chart

Flat slab analysis & design can be done using either of 2 methods :

A. Building Analysis with floor slab meshed
B. Finite Element Floor Analysis Chasedown + Building analysis without floor slab meshed

The difference in the analytical model is as shown & summarized below.

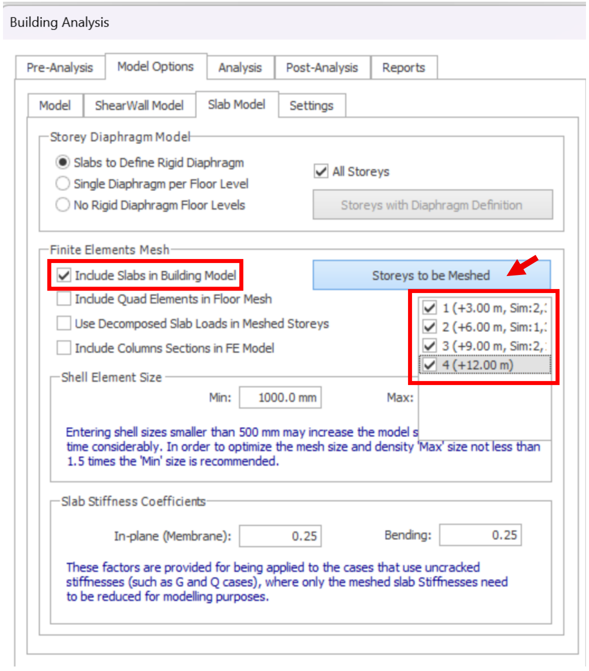


A. Building Analysis with floor slab Meshed	B. FE Floor Analysis Gravity Load Chase down + Building analysis without floor slab meshed
Full 3D model top to bottom storey, with floor slab meshed. There is option to mesh only selected storeys.	FE Floor Analysis analyses 1 storey only, i.e. subframe analysis.
Single analysis to obtain both lateral and gravity load results.	FE Floor analysis gives gravity load results only. Hence, need to run Building Analysis (without meshing floor slabs) to obtain lateral analysis.
3D effects such as differential axial deformation effect or natural sway due to gravity loads included. Hence, results may deviate from traditional or conventional assumption of flat slab.	3D effects are excluded since analysis involves only 1 storey at a time. Hence, result is more in line with traditional or conventional flat slab analysis.
Analytical model is larger & more complex if floor plan is large with many storeys. Hence, may be harder to identify modelling mistakes, as need to examine the full 3D analytical model.	FE floor analytical model is smaller & simpler since only 1 storey is considered at a time. However, more steps are required since both building analysis & FE Floor Gravity Chasedown must be done.
More efficient as single analysis for gravity & lateral loads or for models where only certain floors are flat slab.	More suitable when traditional subframe assumption preferred.

Flat Slab Analysis using Building Analysis with floor slab meshed

The following are summary of steps using this method :

1. **Model** the flat slab model using slabs as outlined in the previous section.
2. **Go to Building Analysis > Model Options > Slab Model**



   - Tick '**Include Slabs in Building Model**' **>** this will mesh the floor slabs
   - Select '**Storeys to be Meshed'** **>** Select all storeys.
   - Review other parameters such as Shell Element size & Slab Stiffness Coefficients
     - The defaults for this model is 0.25 or 25% for In-plane & Bending (out-of plane)
   - For detail explanation of each parameter, kindly refer to Prota Help center : Slab Model

3. Run **Building Analysis**
   - This will give results of gravity (G+Q) and lateral (notional + wind)
   - Go to **Building Analytical Model** to review & verify the results.

➢ *Perform **Building Analysis** with all the settings shown above (without performing any member design).*

After analysis is completed successfully, **drift check** summary will be shown. If the model is set up correctly, the drift check should pass. For details of the drift check, you can generate the **Post Analysis Checks** report.

You should also review **Axial Load Comparison Report** to check whether any gravity loads are lost. This is already covered under the ProtaStructure Basic Training; hence please refer to it if you have not already done so.

Slab Meshing

Long Term Deflections

Adjustment to the slab stiffness to cater for long term deflections are recommended in 2 references:

- **Concrete Society TR58** – Deflections in Concrete Slabs and Beams
- **The Concrete Center (HTFS)** – "How to Series" – How to Design Reinforced Concrete Flat Slabs using Finite Elements Analysis.

HTFS & TR58 – Recommends linear elastic analysis

- ProtaStructure currently uses linear elastic analysis.
- This is the simplest analytical method discussed in the report. So, it should be the simplest for you to use.
- It should only be used to show that deflection is not critical, rather than for providing 'true' deflection estimates.
- For this method a long-term elastic modulus should be used.

HTFS – Recommends adjustment of E

- Creep cracking and shrinkage are all factors that affect the long-term deflection of concrete.
- Short term E (Est) should be reduced to allow for these factors, a long-term value (Elt) should be used in analysis.
- HTFS suggests:
    - For office/ residential : Elt = 1/4 Est
    - For storage/ plant : Elt = 1/6 Est

For more guidance on stiffness adjustments, kindly refer to this article in Prota Help Center : Effective Stiffness Modifiers

Building Analysis Model

➢ *Close the building analysis dialog > go to **Analysis** tab > click **Analytical Model**.*

The building analysis **Analytical Model** shows the analytical wireframe & results of analysis. For details of all the icons of this view, kindly refer to this article in Prota Help Center : Analytical Model

All the floor slabs and shearwalls are meshed using finite element shell. It is highly recommended to carefully examine the **Displacement**, as slabs are the key structural element for flat slab models. Specifically, check if the shape & magnitude of the slab deflection are reasonable.

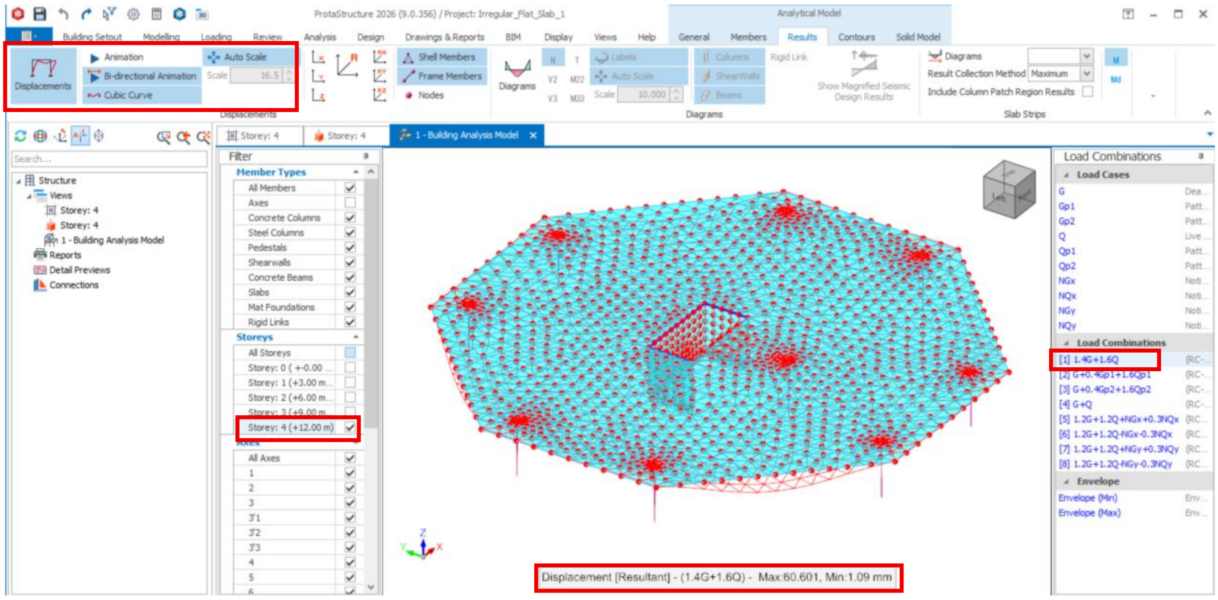
The view is quite congested, hence let's show only Storey 4 elements & the displacements.

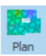
➢ *In the left **Filter** pane, check only **Storey 4**.*

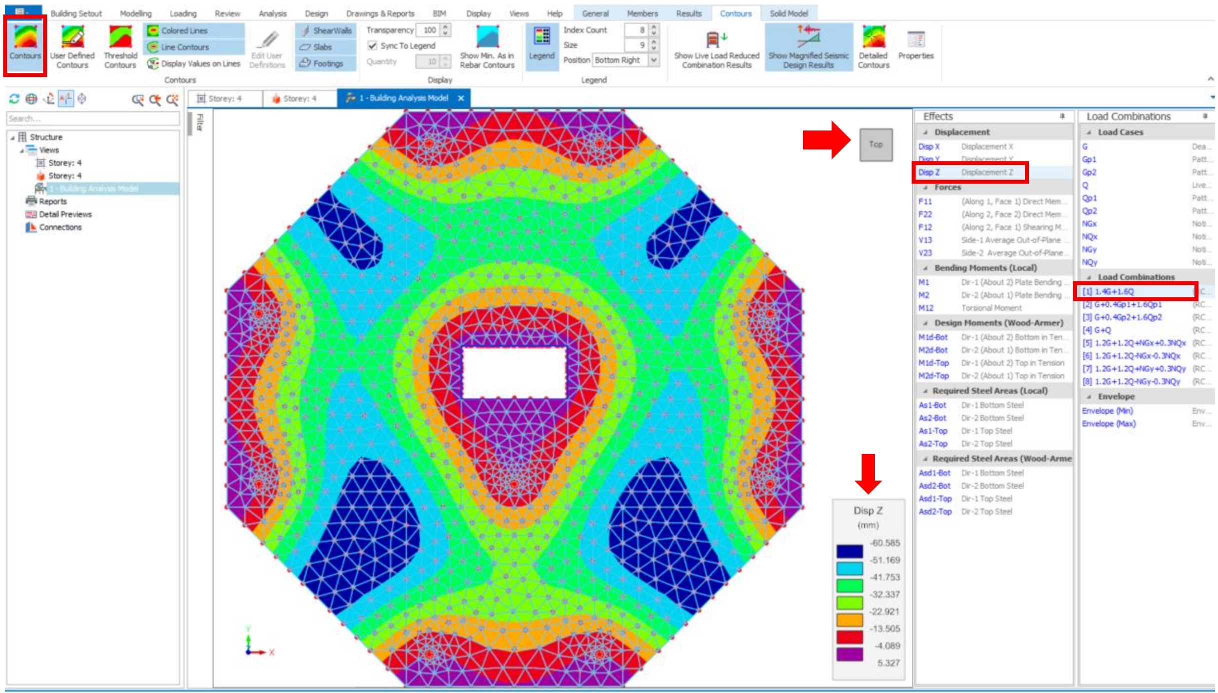
➢ *Go to Results tab, click "Displacements". You can **Increase** or **Reduce Scale** or enter scale after switching off the **Auto Scale**.*

➢ *Pick Load Combination '**[1] 1.4G + 1.6Q'** at the right pane.*

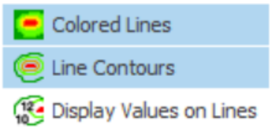
The displacements of the chosen load combination will be rendered in red. The maximum resultant displacement will be show at the bottom.



- *Animation is very useful to visualise the deformation of slab in motion, e.g. observe if the slabs is indeed deforming as connected entities.*
- *Turn off **Displacement** after due verification.*
- *Go to **Contours** tab & Click **Contours** > Under **Effects**, select "**Displacement Z**" (global)*
- *Click "**Top**" on the 3D cube to lock to plan view*
- *Alternatively, click **Plan View***  *(located in **General** tab) to lock the view in plan mode.*



The Contours icon group has options that can be on / off by clicking on button :



- **Coloured Lines**: Add line between contours
- **Line Contours**: Draw line contours
- **Display Values on Lines** : Show the values of line contour

Contours render & display can be changed using **Display** group icons. Legend parameters can be changed and **Legend** group icons.

➢ *Placing the mouse cursor on a slab mesh will display its parameters & the value of the effect.*

Bottom steel reinforcement provision

We will start by assuming that the whole slab is to be reinforced orthogonally in the Global X and Y directions. At this point the slabs reinforcement angles have not been adjusted, so Direction 1 steel is aligned with global X, which runs from left to right in the view below.

➢ ***Select Effect*** *under "****Required Steel Area (Wood-Armer)****", **Asd1-Bot**" & also ensure* ***Standard Contours*** *is activated.*

It is recommended to always refer to plots that include Wood & Armer adjustment i.e. torsional effect of slabs included. These are denoted with "**d**" = design, e.g. As**d**…

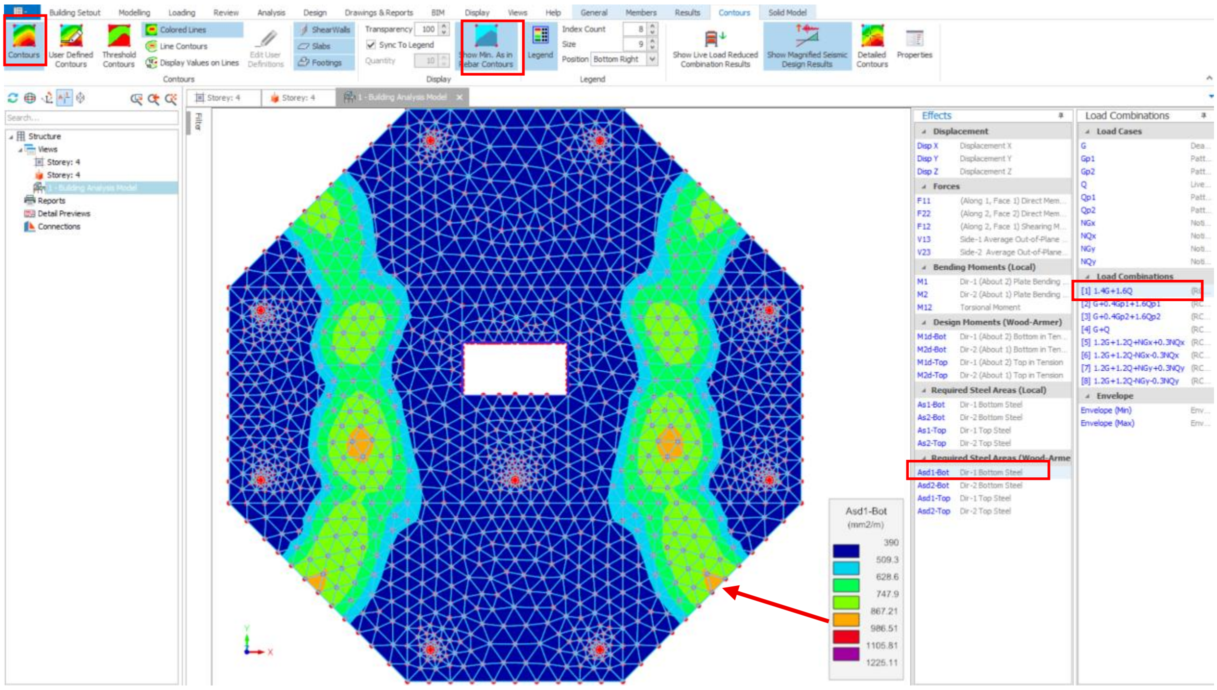
➢ *Under **Load Combinations** select the desired design combination, e.g. "**[1] 1.4G+1.6Q**"*

Alternatively, if you like to design for envelope of all load combination for the bottom steel, choose **Envelope (Max)**

➢ *Click "**Show Min As in Rebar Contours**"* 

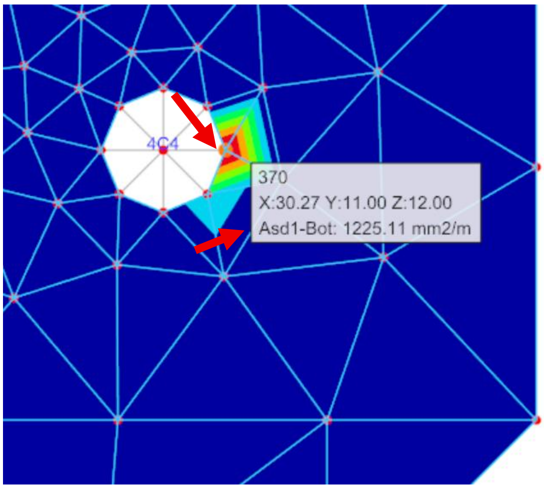
to show minimum required steel for the selected code. We recommend to perform a manual check to ensure this value for extra precaution.

The view (below) shows design bottom steel requirements in direction 1 with the minimum steel of **390mm^2/m** as lowest 1st contour. As might be anticipated the peak requirements are occurring along the longest free edges.



The legend shows the maximum value is **1225 mm2/m**. We can zoom in the contour view to try to find this node, usually maximum value occurs between supports or at the column or wall node.

- *By zooming in the column **4C4** (right corner), we can locate this value by placing the mouse cursor on the column boundary node & reading the tooltip as shown below.*

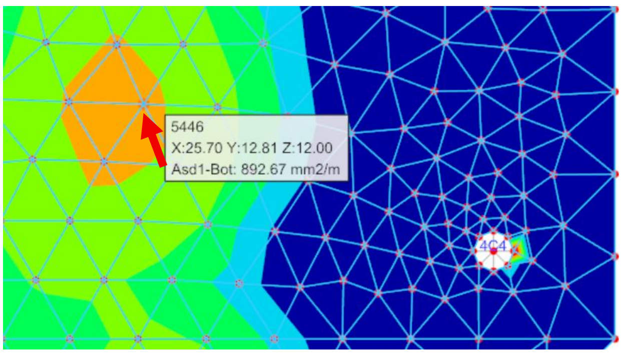


It is unreasonable to expect maximum Asd1-Bot to occur at the column node; it should be at middle of slab where maximum sagging deflection occurs.

This is due circular column being simplified as hexagonal element & the slab shell meshed to the boundary of the column. This causes a local distortion at a single node. If you read the adjacent node, it immediately drops to minimum steel.

This abnormal value can be ignored, as any single abrupt & extreme jump in node value will be deem unreasonable.

- Zoom into the areas between the columns where the maximum Asd-Bot should occur, i.e. locations where maximum slab deflections occurs (switch to **Displacement Z** contour to check)



We can locate this node somewhere between column 4C1 and 4C4. The value is **893 mm^2/m**.

The other similar maximum value is at the lower slanting slab between column 4C3 & 4C4. The value is around **879 mm^2/m**.

We can conclude the reasonable **Asd1-bot** is around **900 mm^2/m.**

- *Select **Effect Asd2-bot***

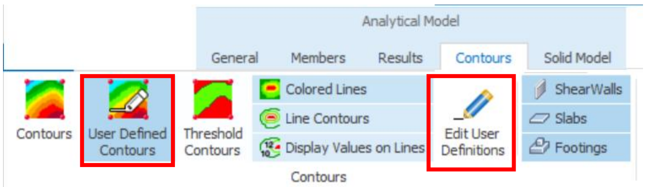
Using the same inspection & reasoning as direction 1, we similarly locate the correct slab nodes values for direction 2. The maximum **Asd2-Bot** value is around **840 mm^2/m** at the slanting slab edge between 4C3 & 4C4. For simplicity, we will use the larger **900 mm^2/m** as maximum of both Asd1-Bot & Asd2-Bot.

User Defined Contours

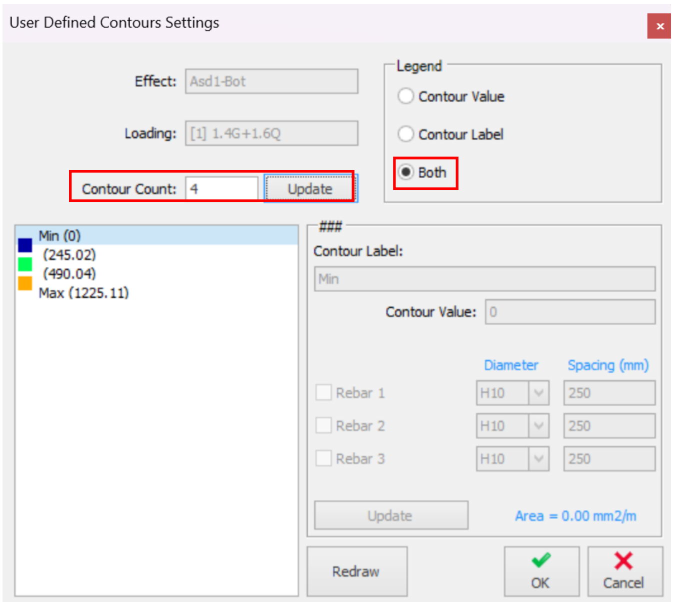
- *Switch back to view **Asd1-bot**.*

We can now use the custom contours option to show where different steel reinforcement provisions would be adequate. In general, the strategy would be to decide on general lower level of reinforcement to be provided continuously throughout the slab and identify the regions where an increased provision is required.

- *Click on the **User Defined Contours** icon and **Edit User Definitions** button.*



- *Set **Number of Contours** to **'4'** and **Legend** to **'Both'** > click **Update** button to re-interpolate contour values. We choose 4 no. because we intend to provide 2 different steel contours (the other 2 no. is auto-calculated min & max contour).*



Effect – This is the brought through from the current view selected in the FE Post Processor.

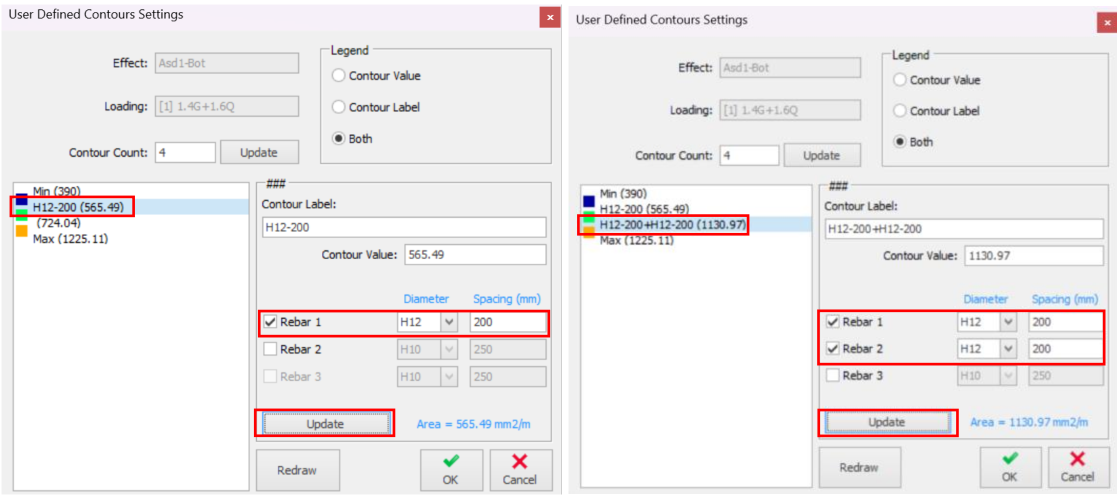
Loading – The current Load Case or Combination selected in the FE Storey and Foundation Models

Legend – This allows the user to choose the information displayed on the legend within the FE Storey and Foundation Models – E.g. Contour Value only (565.5mm2/m) / Contour Label only (H10 – 200) / Both

Contour Count – This allows the user to specify the number of contours to be displayed on the screen. Note that Minimum (0.0) and Maximum cannot be changed.

Contour Label/ Contour Value/ Diameter and Spacing – This area allows the user to select bar sizes and spacing's for the creation of the contours. The labels and values are automatically created based on the selected size and spacing of the reinforcement.

➢ *Enter the contour settings as shown below and click on **Update** for each input.*

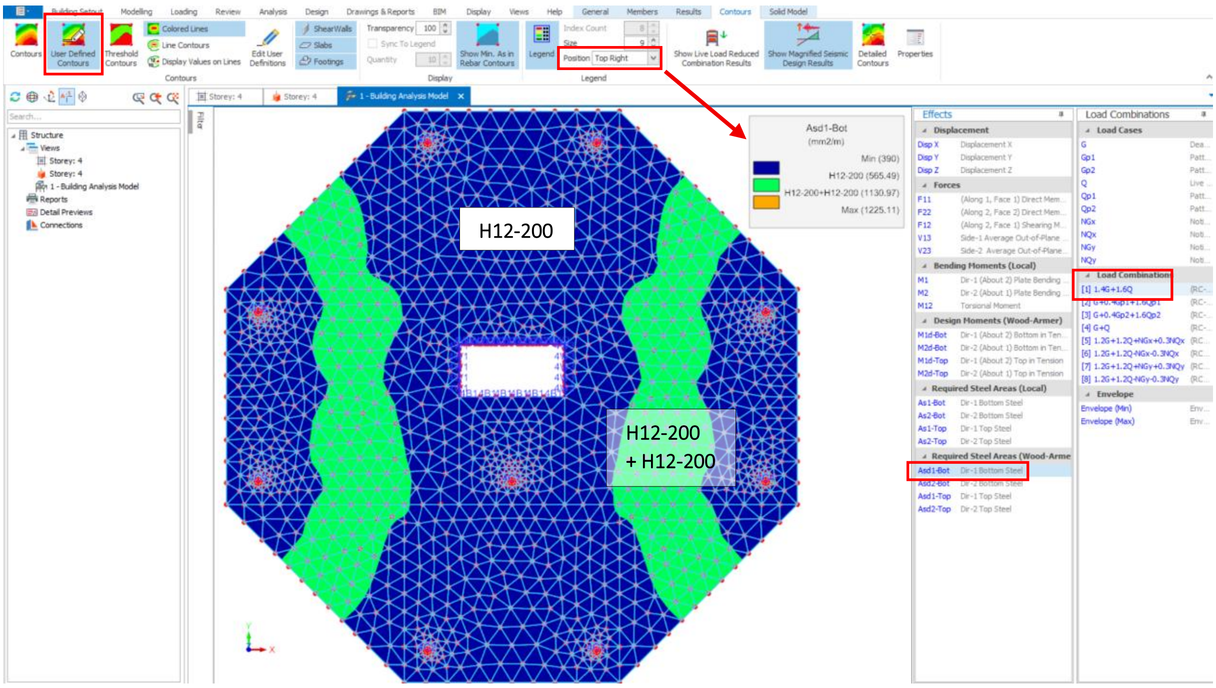


The 1st contour should satisfy As min steel = **390 mm²/m**. **H12-200 = 565 mm²/m** satisfies this.

The 2nd contour must satisfy the Asd-1-Bot maximum = **900 mm²/m**. **H12-200 + H12-200 = 1130 mm²/m** safisfies this. This is effectively **H12-100** but we choose to display as the former.

➢ *Click **OK** to view the Reinforcement Requirements for the Bottom Steel, in Direction 1 (Global X axis in this case).*

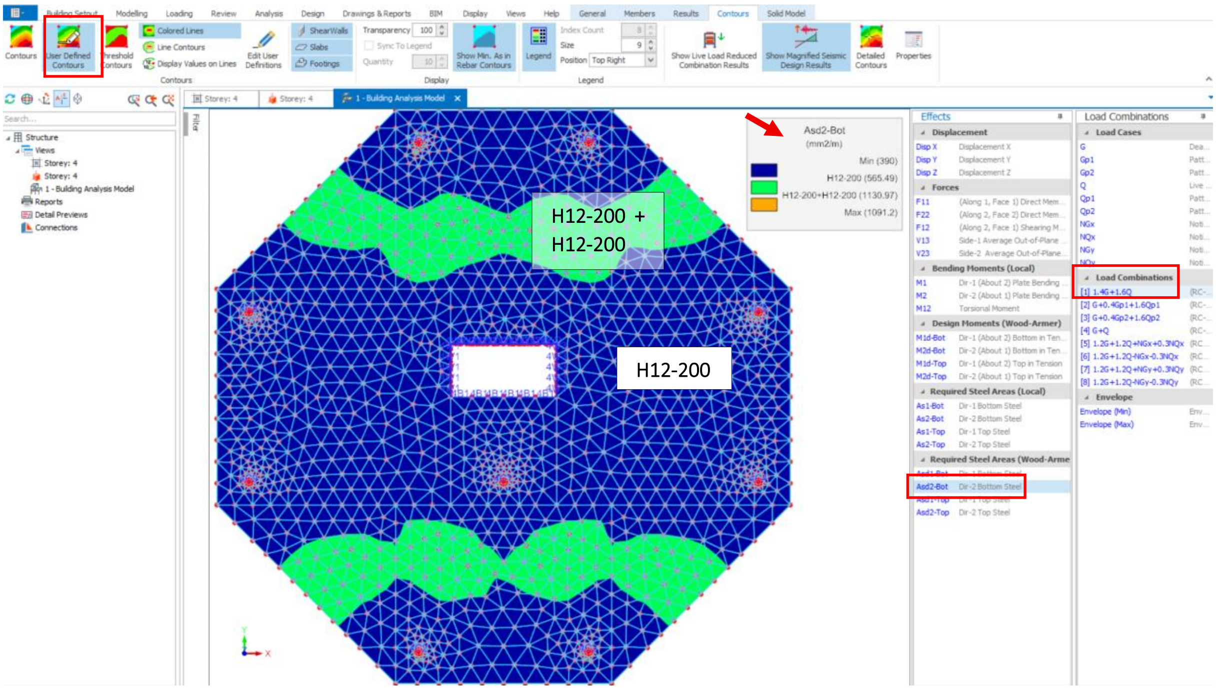
If the contour legend is hidden you can click on "***Position***" under **Legend** icons to shift it to another corner. The legend size can also be increased or decreased.



The "Blue" region is where **H12-200** is required. The "Green" region is where **H12-200 + H12-200** is required. Effectively, this means **H12-200** is placed in in the entire slab, while **H12-200** added in specific "Green" area. This effectively means the "Green region" requires bar **H12-100**. "Amber" region indicates area where the provided steel is insufficient; we can ignore this (if any) as we already determined the reasonable maximum value.

Creating contours (bottom steel provision)

- *Using the same "Edit User Definition" method, create the **Asd2-Bot** contours*
- *Create two contours **H12-200** and **H12-200 + H12-200**.*



In short, **H12-200** & **H12-200 + H12-200** satisfies both the bottom steel in direction 1 and direction 2.

These requirements can be simply communicated to the detailer by display the above contours to plan view and then export to DXF.

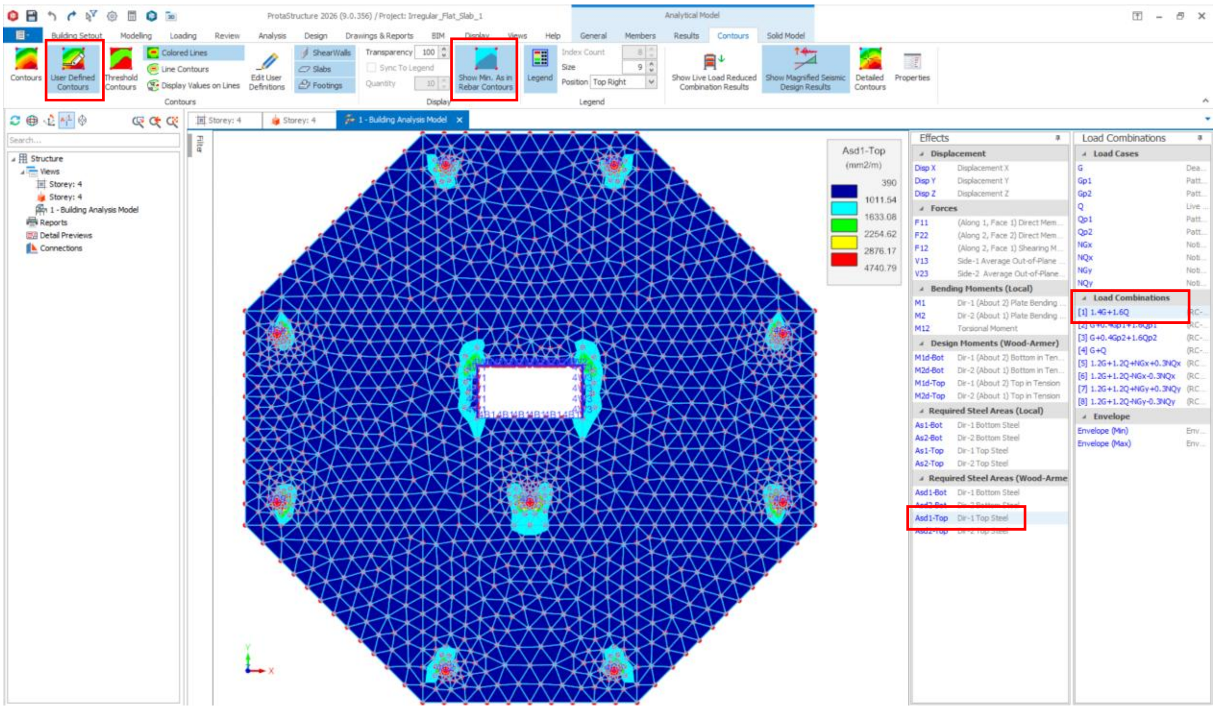
- *Go to **Review** tab, click on **Visual Interrogation** > **FE Contours** , select contours type, effects and load combinations based on preferences.*

Alternatively, you can "***Copy Picture***" or "***Save Picture***" located at ***General*** tab to save the image of the contour by clicking respective icons in top menu.

Although only 2 reinforcement provisions have been shown in the above example, if required many more contours could be introduced to suit specific project requirements.

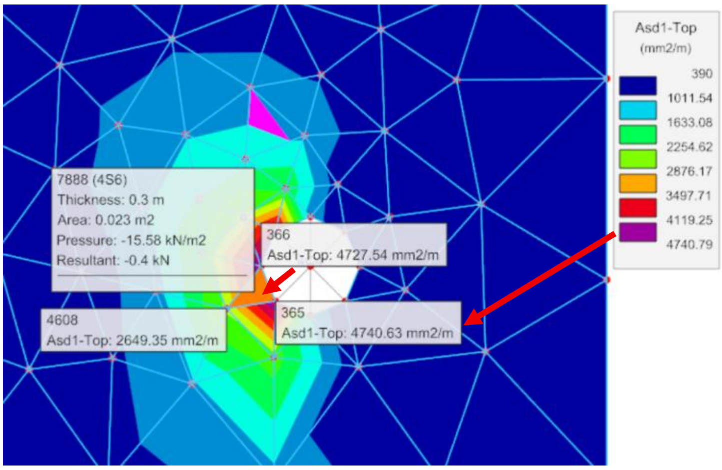
Creating contours (top steel provision) – Asd1-top

➢ *Switch back to view **Contours** for **Asd1-top*** (same factored load comb **1.4G+1.6Q**)



As expected, a review of the top reinforcing requirements shows that in theory, only minimum steel is required over large areas of the slab (blue region) and that the hogging moments intensify rapidly over the column heads & core wall. The minimum steel is **390 mm²/m.** For economy, **H10-200 = 393 mm²/m** would suffice as base top rebar everywhere. We should now locate the maximum top steel by zooming in columns & walls where peak top rebars are expected.

➢ *Zoom in **4C4** & place mouse cursor on slab shell to locate the maximum **Asd1-Top***



The peak As is around **4740 mm²/m**.

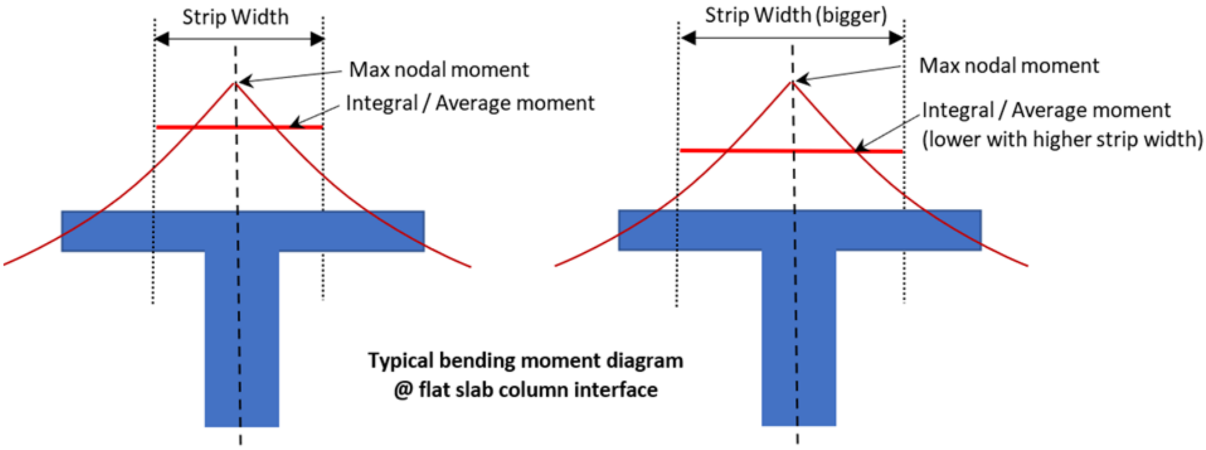
It would be impractical to provide top steel exactly to match this peak maximum steel area as these a very small localised area often exposed when meshing slabs.

The average of this mesh nodes is (4740+4727+2649)/3 = **4038 mm²/m**

If we calculate the average values of all the peak slab shell nodes connected to the column, it is around **2500 mm²/m.**

Manually calculating average of all peak slab shell nodes is tedious. The more practical way to deal with these peak requirements is to integrate (average) results in strips cut across the column & wall heads which is in line with simplified “Equivalent Frame” method stated in most of code practice.

This can be done by using ProtaStrucure **Finite Element Integral Strip**. The purpose is to calculate the design moments using integral approach, i.e. to average out the moment within the strip width, otherwise the maximum moments of a single node within the strip range will be used for rebars design.

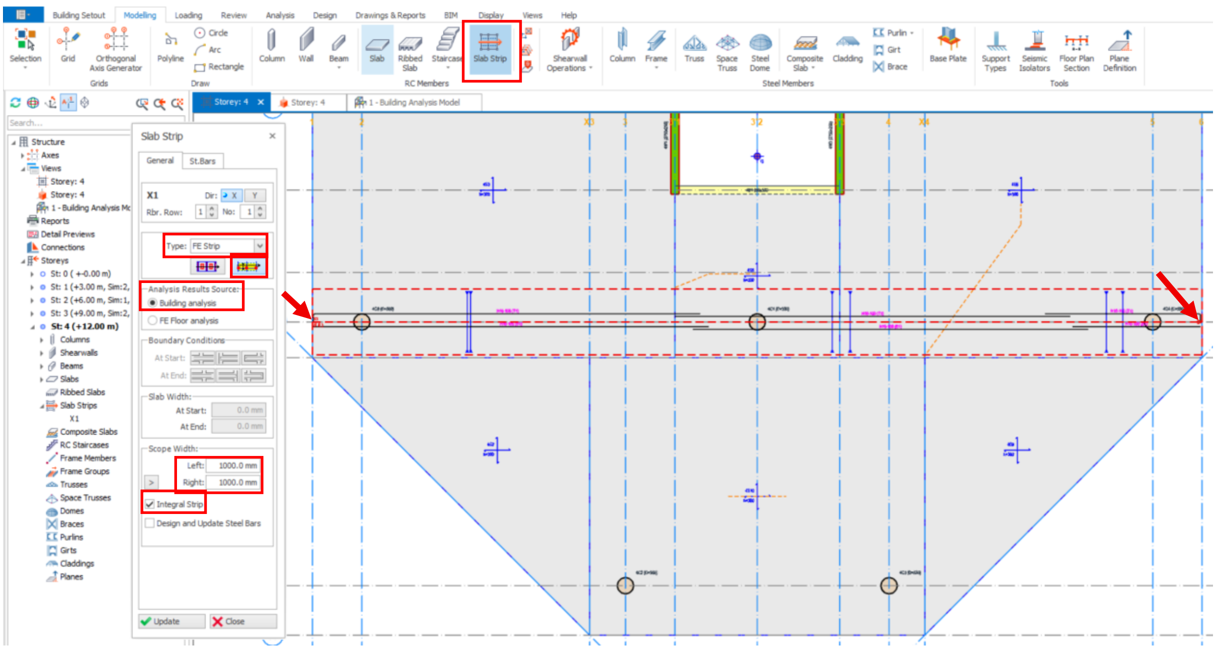


- Refer to the figure above; by default, the maximum peak nodal moment will be used for finite element slab design, i.e. column center line.
- Integral option will average the design moment across the width of the strip.
- The wider the strip, the lower the "averaged" design value, esp. at support region where the moment changes rapidly.
- User should carefully evaluate how wide to create each strip when integral is chosen.

In the discussion below a 2m wide strip is cut across the head of the most critical internal columns.

➢ *Go to* ***Plan view****, ensure the active storey is* ***Storey-04***

➢ *Click "****Slab Strip****" icon in* ***Modelling*** *tab, ensure that below options are set to create an Integral FE slab strip with* ***2m*** *width :*
  - *Type =* ***FE Strip*** *> Select "Fixed Band Strip"*
  - *Analysis Results Source =* ***Building Analysis***
  - *Scope Width Left & Right =* ***1000mm*** *(total 2000mm)*
  - *Check "****Integral****"*

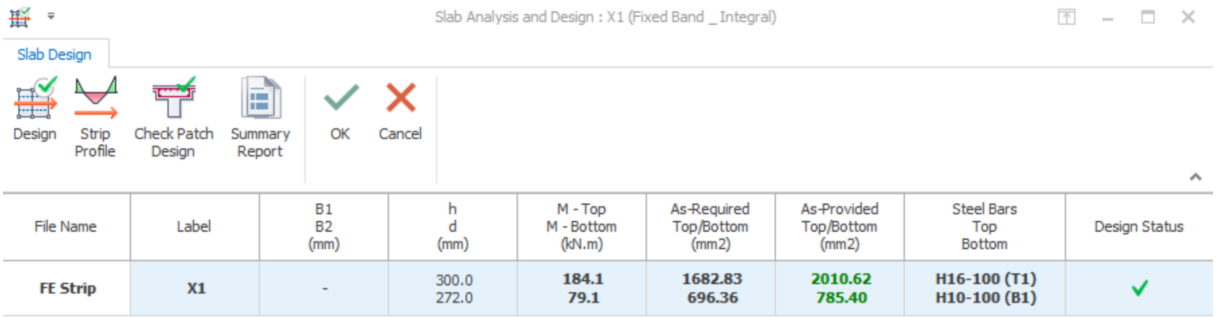


➢ *Click the 1st point & then 2nd point of slab edge, making sure to cut through exactly the middle of columns 4C9,4C1 & 4C4.*

- ➢ *If the slab strip created is according to the settings set as above, program automatically designs & provides H16-100 (top) & H10-100 (bot) to cover the area of the slab strip width.*

  Note: When this steel is provided, **it must** be provided over the full extent of the 2m width strip. The fixed band strip design provides a single top and bottom bar across the entire length & width of the strip. Practically, this is uneconomical design (too conservative) as the required steel contours shows minimum steel is already sufficient for large area of slab. Hence, it's not advisable to adopt this as final steel arrangement, especially the top rebars.

- ➢ *Select the slab strip > Right-click > **Slab Strip Check Design** to launch **Slab Analysis & Design** dialog*



From the above design dialog, the required top steel reinforcement is around **1683 mm^2/m**. This value is significantly lower than the previously manually calculated average mesh nodes connect to the column boundary of **2500 mm^2/m**. This is because the 2 meter slab strip is averaging over many more shells beyond the column boundary. We will now compare the various slab strip width results.

Slab Strip half width	Slab Strip total width	As Required (mm^2/m)	Remarks
1 m	2 m	1683	Likely too wide, as column size is 500mm.
500 mm	1 m	2358	Lower bound, more reasonable
250 mm	500 mm	3200	Upper bound, too conservative?

As expected, the narrower the strip, the higher the required area of steel. The final choice of slab strip width is an engineering judgement and factors to consider are :

- Code of practice & design guidance, which will specify some idealized strip width in accordance with traditional analysis & design method. For example BS8110 suggests a certain percentage of panel width on each side.
- Compare with manual average peak nodes of **2500 mm^2/m**, the 2m total width strip is likely to wide and unconservative. The 2m wide with is 4 times the column size of 500mm.
- Size of support : The size of the column is already 500 mm, hence 500 mm total width is likely too small & over-conservative.

The 1000 mm total strip width As required of **2358 mm^2/m** is close to the manually computed average of **2500 mm^2/m**. This seems to be the most reasonable. The design top steel of **H20-125 = 2513 mm^2/m** is sufficient.

- ➢ Now, apply the same process to horizontal wall area. Create a slab strip across the wall. It is found that the required steel area is around **1515 mm^2/m** when the total slab strip total width is **1000mm**.

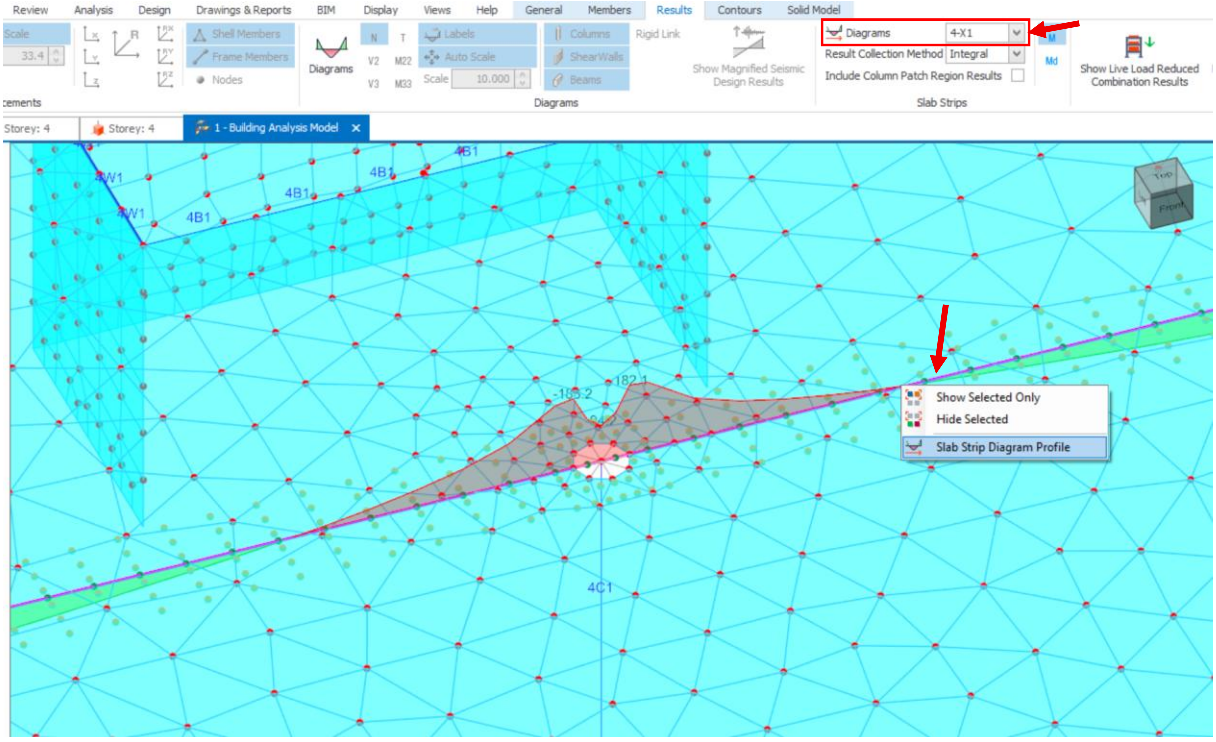
The FE fixed band strip can only be used if cut across slabs with the same thickness. The reason is the design only detects the slab thickness nearest to the first end of the strip. If three are many slabs with different thicknesses, then do not use the FE Slab Strip. Use the other method of design using the As required contours as outlined in this Flat Slab training manual.

Slab Strip Diagram

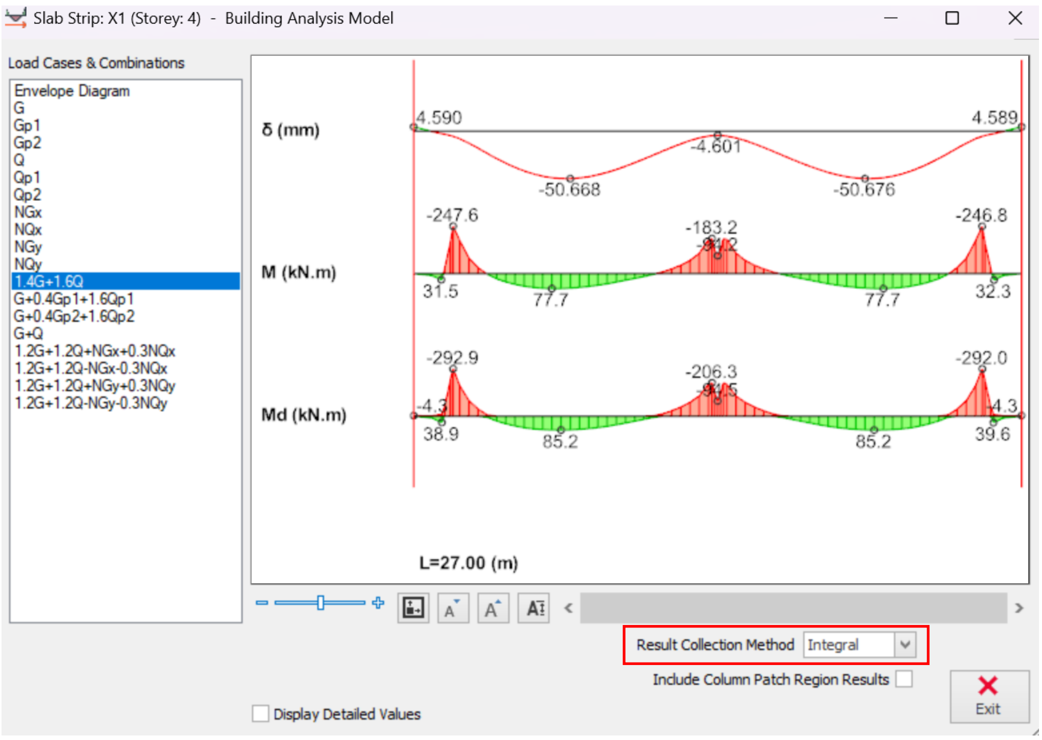
To review slab strip force diagram in Analytical Model :-

- *Go back to the "**Building Analysis Model**" view*
- *Go **Contours** tab, deactivate all the **Contours** to get a clean view.*
- *Go to **Result** tab, select slab strip you want to review from the **Slab Strip List***

  This will display the moment diagrams of selected strip on the analytical view.



- *To view the detailed profile, select a slab strip > Right-click **>** select "**Slab Strip Diagram Profile**".*



Program provides 3 different result collection methods namely Maximum, Integral, and Strip Line.

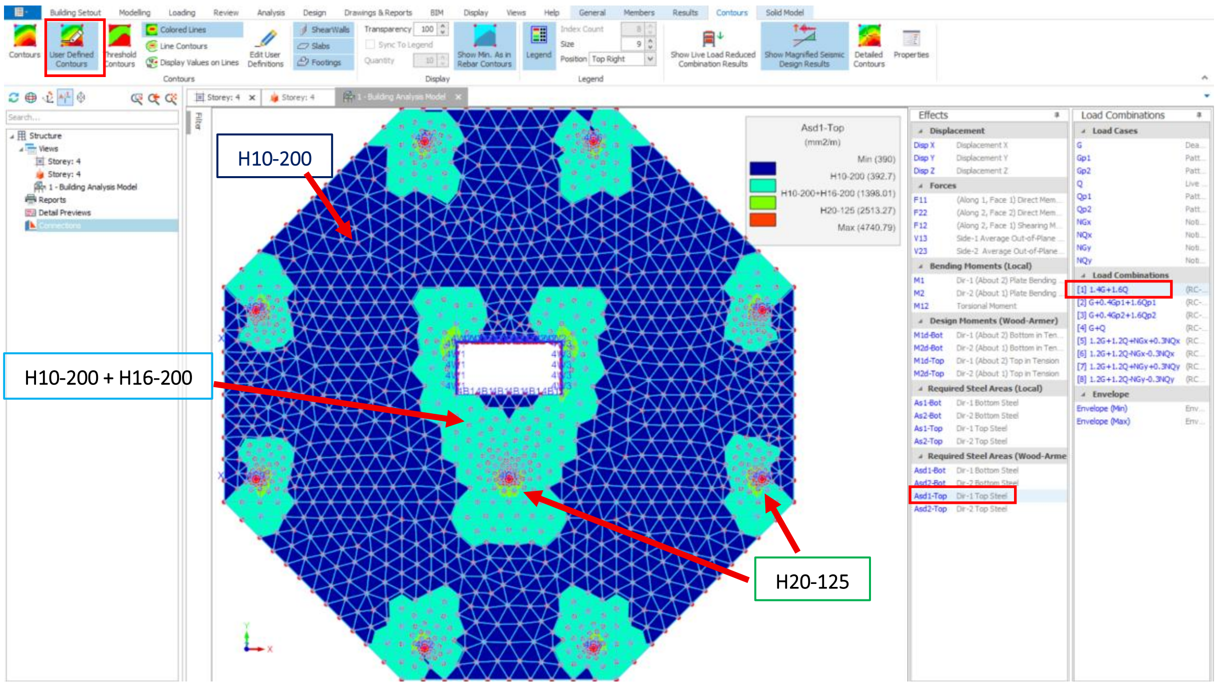
Approach	Description
Maximum	Maximum values of the transverse nodes at each station are used in strip result calculation
Integral	Calculates the integral of transverse node results yielding more economical and smoothened values. **Note:** This method only available for “Fixed Band” strips
Strip Line	This method is solely for checking purposes and doesn’t account for transverse nodes displaying the values exactly at the station nodes

Note : Slab Diagram can also be accessed via the Slab Analysis & Design dialog.

Having gathered all the reinforcement steel area for column and shearwall above, we can add more contour to satisfy the required steel area.

We will add User Defined Contours according to satisfy the top reinforcement required:

- *Click “**Edit User Definitions**” again, and increase Contour count to **5**, and click “**Update**”*
- *Update the 1st contour with reinforcement rebar **H10-200***
- *Update the 2nd contour with reinforcement rebar **H10-200 + H16-200***
- *Update the 3rd contour with reinforcement rebar **H20-125** to satisfy all the areas at shearwalls and columns heads.*



Upon closer inspection on the contour plotted by zooming into columns, you can still see “red” area which suggests the contours created are not adequate. This would be the case if we were considering every peak node value for the floor plate, which we have established it not feasible nor practical.

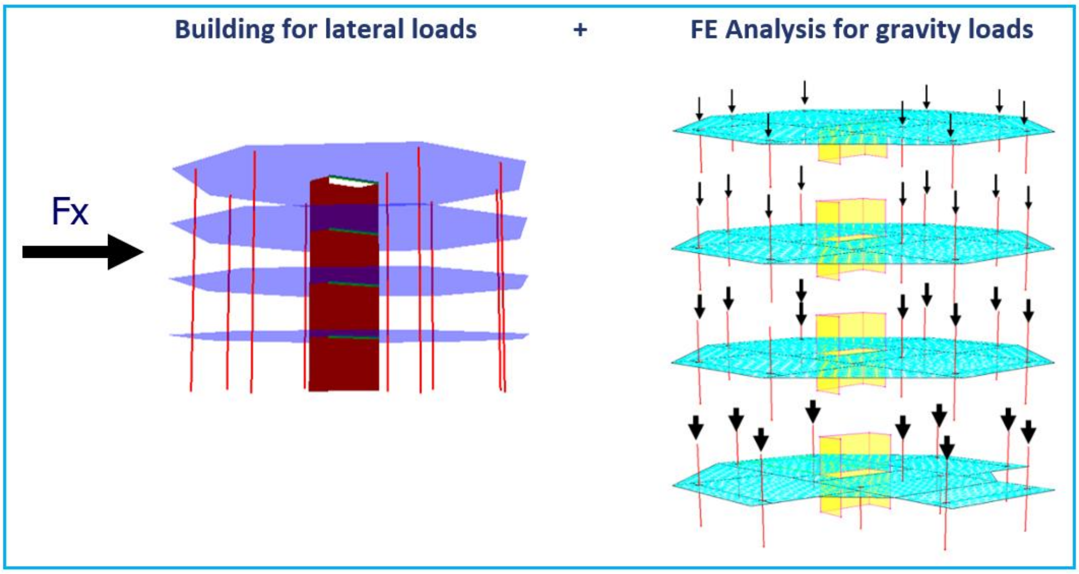
As such, we have chosen to design these areas using average results, the red area can be ignored & we can provide the same maximum reinforcement established earlier.

For practice, the same steps can be repeated for **Asd2-Top** using **User-Defined Contours**.

Flat Slab Analysis using FE Floor Analysis Chasedown

FE Floor Analysis is an alternative to Building Analysis with floor slab meshed. The following are summary of steps using this method :

1. **Model** the flat slab model using slabs as outlined in the previous section.
2. **Go to Building Analysis > Model Options > Slab Model**
   - Untick '**Include Slabs in Building Model**' **>** Floor slabs will not be meshed
3. **Run Building Analysis** (BA)
   - This will give results of gravity (G &Q) and lateral (notional + wind)
   - Gravity results are not valid since BA requires beams to supports slabs
   - Lateral results are valid as lateral loads are transferred via rigid diaphragm formed by slabs
   - Go to **Building Analytical Model** to review & verify the lateral analysis results
4. **Run FE Chasedown Analysis for Gravity Loads**
   - Since BA cannot evaluate flat slab gravity loads, we need to run FE Floor Analysis to obtain gravity load results
   - FE Floor Analysis analyses a single floor with slab meshed. To correctly accumulate column & wall loads, we need to run the process from top to bottom storey (gravity load chase down)
   - Go to **FE Analytical Model** to review & verify the gravity analysis results
5. **Merge** the results of both analyses for column & wall or beam design
   - Lateral loads results will be from Building Analysis
   - Gravity loads results will be from FE Floor Analysis Gravity load Chasedown



Building Analysis to Generate Lateral Design Forces

We will now proceed to analyze the flat slab system using FE Gravity Load chase down method.

We will setup the materials & options before running Building Analysis:

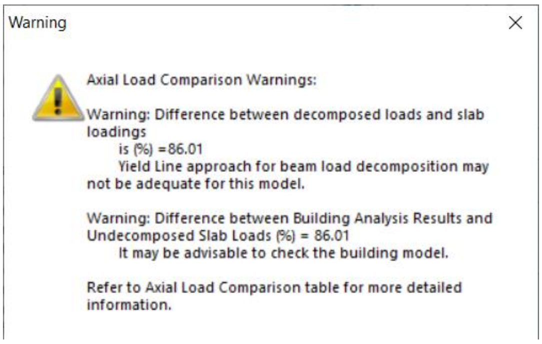
- *Analysis tab > Building Analysis*

- *Go to* ***Model Options*** *>* ***Settings >*** ***ShearwWall Model*** *tab* ***>*** *Pick* ***"Mid-Pier".***

Since we are not meshing the floor slab in Building Analysis, it is not necessary to use FE Shell for walls. This will make the analysis run faster without sacrificing accuracy.

- *Go to* ***Analysis tab*** *> Click* ***Building Analysis*** *>* ***Run Building Analysis***
- *Ensure the Column/Wall and Beam Reinforcement Design is '****Unticked****'*

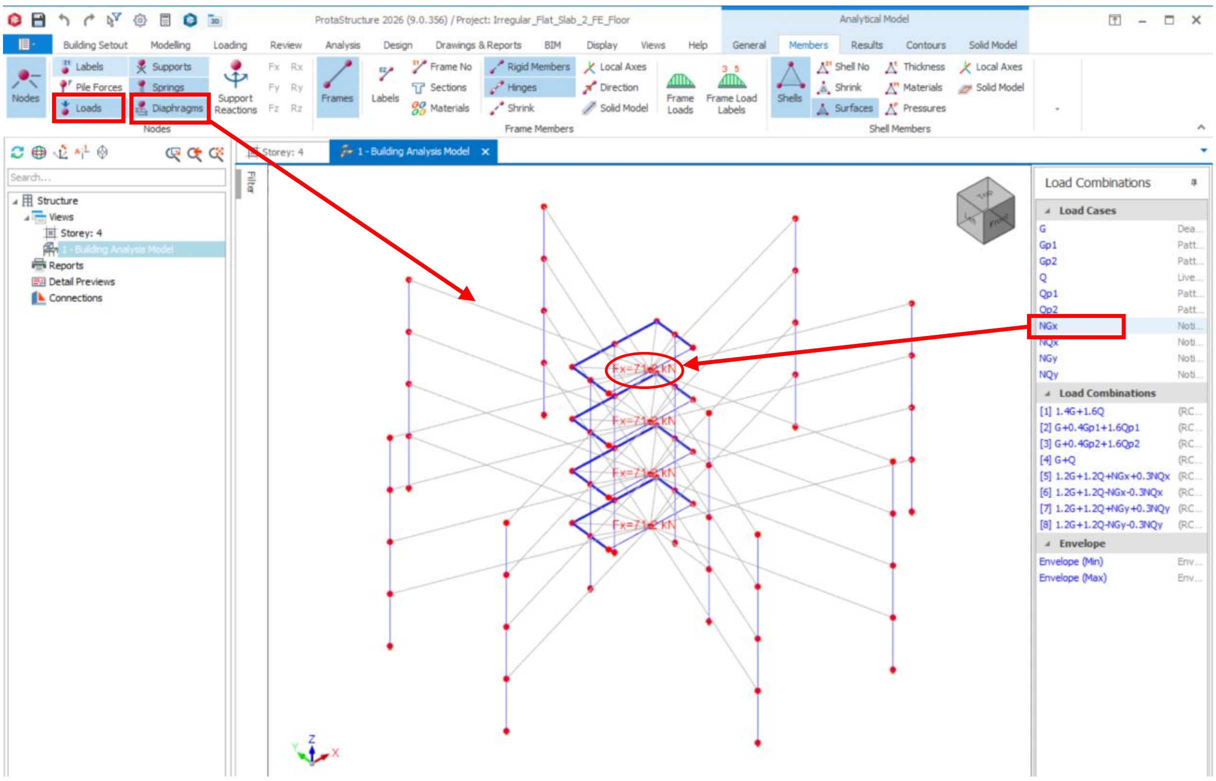
At the end of the analysis process, you should see a warning indicating load has been lost.



This warning will always appear when running Building Analysis for flat slabs. It indicates that an FE analysis is required for the gravity loads. Hence, this message <u>can be ignored</u>.

In the Building Analysis the slabs are replaced with rigid diaphragms to decompose the lateral loads to the Columns/Walls in the structure.

- *In the* ***Analytical Model****, you can switch on "****Diaphragm****" > Select the Notional Horizontal load load case* ***Ngx*** *or* ***Ngy*** *> The nodal loads* ***Fx*** *or* ***Fy*** *will be shown to applied at center of diaphragm of each storey (as shown below).*



Hence, in building analysis ONLY the lateral design forces have been calculated correctly.

As the flat slab has no beams as supports, there is no means of calculating the vertical slab loads onto the Columns/Walls. Therefore, for vertical / gravity loads, we need to perform **FE Analysis Gravity Load Chase Down** for the structure to obtain the correct gravity results.

- *Go to **Analysis tab** > Click **FE Floor Analysis** > **Run Building Analysis***

FE Floor Analysis Settings

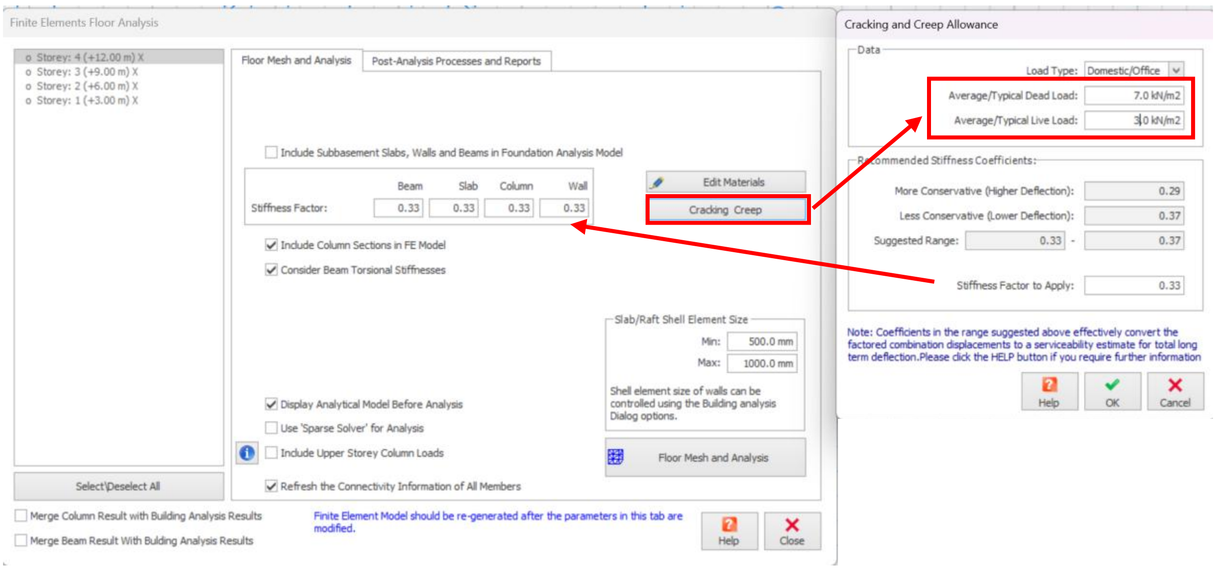
'**Include Column Sections in FE model**' – This allows the Finite Element Mesh to form around the perimeter of the columns, producing moments at the face of columns rather than the centre of the columns. This option significantly reduces the peak hogging moments over the column heads in Flat Slab Construction. This option is automatically activated for flat slab model.

'Consider Beam Torsional Stiffnesses' – This allows the torsional stiffness of any beam members to be considered in the design. Un-ticking this option would assume the beams to have No Torsional Stiffness.

'Include Upper Storey Column Loads' – This option must be selected to allow the Load to be chased down through the model (even at the top storey).

Shell Element Size: This controls the minimum and maximum size of the mesh. Generally, the smaller the mesh, the higher the number of total mesh & hence longer analysis time.

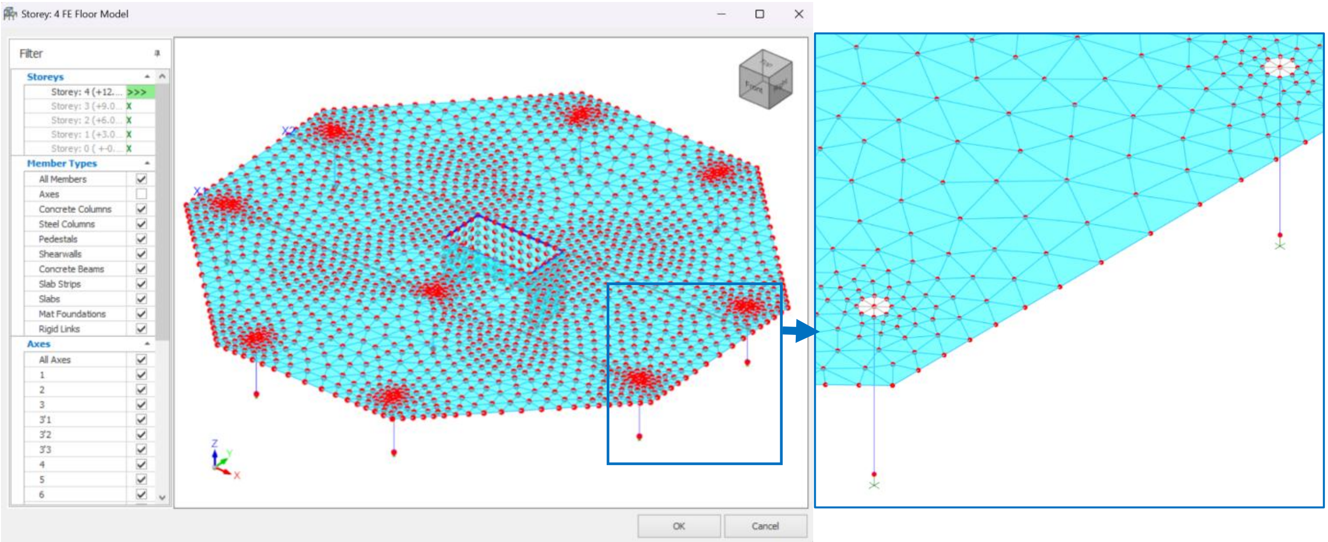
The key objective is to choose an ideal balance of minimum and maximum Shell Element Size and to produce minimum number of plates but still sufficient to accurately replicate the "true" behaviour of the slab (so analysis time is minimized). Generally, try to achieve between **8 plates** between the column heads. The default of minimum 500mm a& maximum 1000mm shell size should be sufficient for most models.



- *Click **Cracking Creep** icon to calculate the stiffness adjustment*
- *Enter **Average Dead Load = 7.0** & **Average Live Load = 3** > **OK***

Stiffness adjustment of **0.33** will be automatically calculated for all members. The theory behind this is explained in this article : [Cracking and Creep Factor Calculator]

- *Tick the option "**Display Analytical Model Before Analysis**"*
- *Click '**Floor Mesh and Analysis**'* button > **FE Analytical** model will be shown as below.



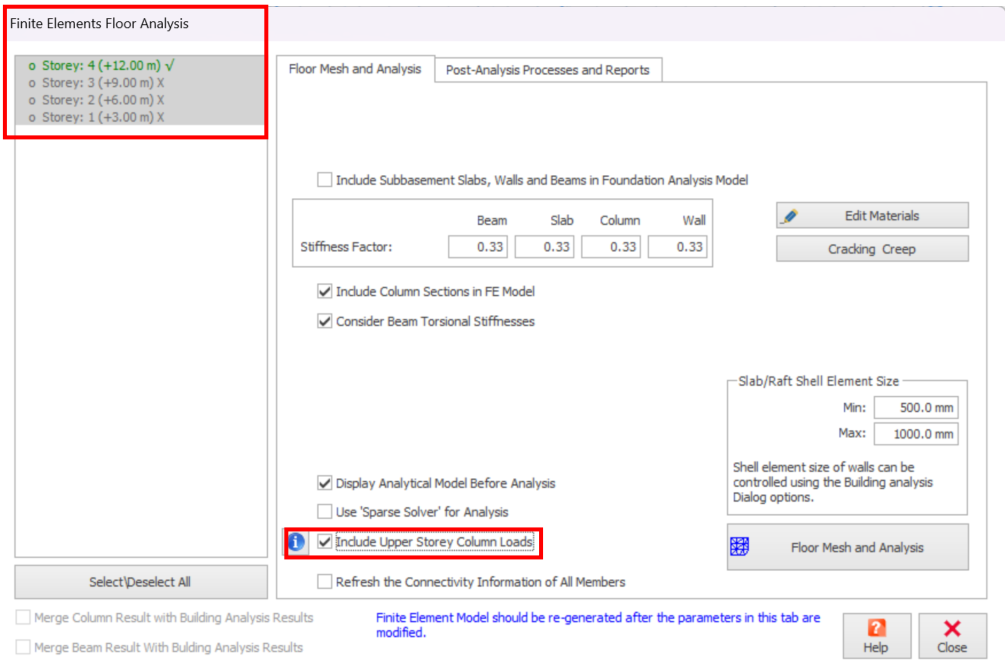
The FE Analytical model is for 1 storey only. You can verify the analytical model by zooming in area of concern, example ensure the elements are connected properly. As mentioned previously, we should have recommended 8 shells between columns or supports. Hence, the default shell size setting is sufficient.

- ➢ *Click **OK** to close & complete the analysis*

Batch FE Chasedown Analysis

The previous operation could be performed at each floor level by analysing and moving to the floor below until the load has been transferred to the foundation level. However, using the 'Batch FE Chasedown' this procedure can be automated:

- ➢ *Select **All Storeys*** and ensure option "***Include Upper Storey Columns Loads***" is ticked.



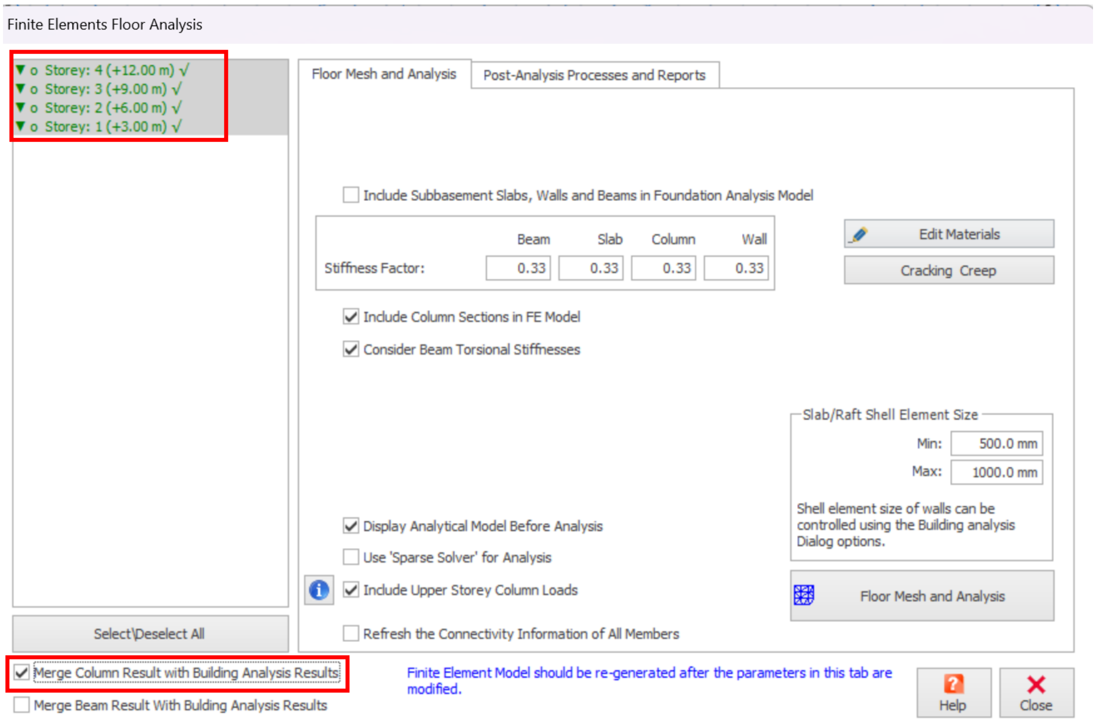
- ➢ *Make the settings as shown above > Click **Floor Mesh and Analysis***
  *Storey will be highlighted as **green colour** when FE Floor Analysis is completed.*

If "FE Chasedown" is performed, "▼" will be displayed next to the storey. Otherwise, it will be greyed.

- *Un-tick the '**Display Analytical Model Before Analysis**' as the floor plates are all the same in this model, so there is no need to check and approve the Mesh at every floor level.*

  It maybe that you choose to "***Display Analytical Model Before Analysis***" for the first FE run of the Load Chase Down. However, after minor changes, you may well un-tick this option.

- *Once FE Chasedown of all storeys is completed successfully, tick '**Merge Column Results with Building Analysis Results**'.*



This ensure that all column and wall gravity (vertical) loads will use FE Floor Analysis results; otherwise, gravity loads will still be based on Building Analysis (wrong!).

The option to also '**Merge Beam Results**…' is optional & is a user decision if there are also beams in the model.

- *Click **Close** to exit the dialog.*

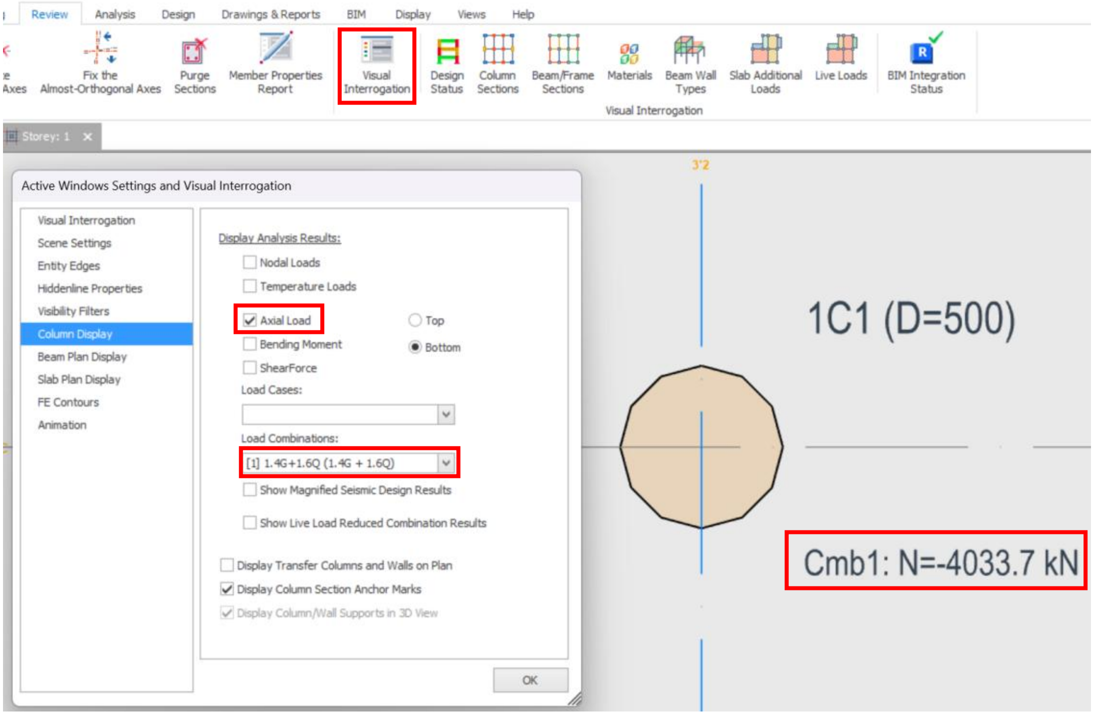
Important!

When "Merge Column / Beam Results with Building Analysis Results" is checked, is mandatory to "**Close**" the dialog for the FE Chasedown Results to ensure all the associated member forces and reports to be updated.

Reviewing Column & Wall Forces on plan view

- *Go to **ST01 Plan** view.*
- *Go to **Review** tab > **Visual Interrogation** > **Column Plan Display** > Check **Axial Load***
- *Under **Load Combination** dropdown > Pick **'[1] 1.4G+1.6Q'** **>** Click **OK***

The Axial load for the selected combination will be shown on plan view as below.



Displaying column / wall forces on the plan view is a convenient way to check member forces. For above column 1C1, the axial load value does seem reasonable when cross-referenced against the Axial Load Comparison Report.

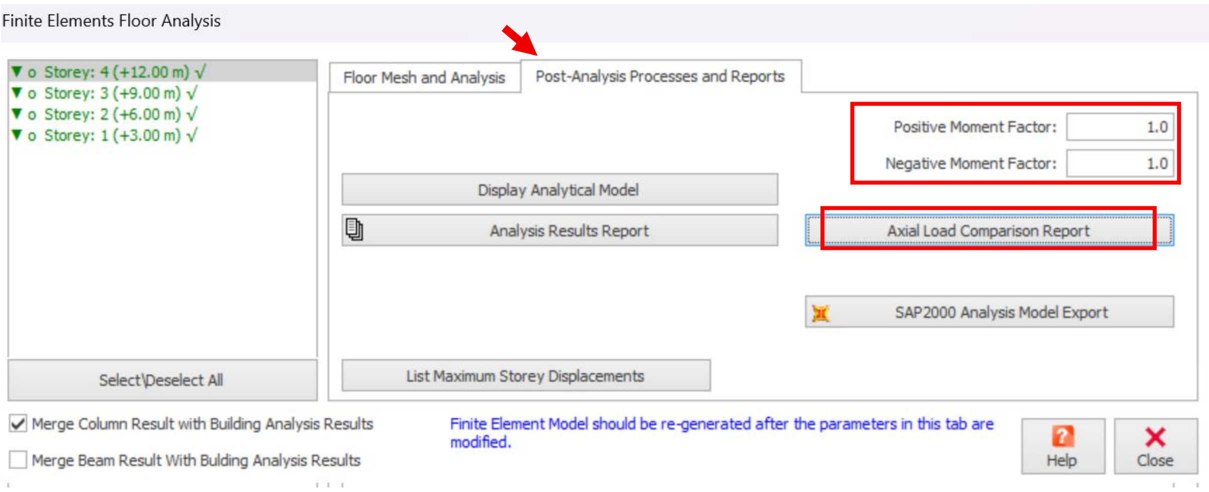
Other methods of reviewing column or wall forces are :

- Select Column / Wall > Right-click > Pick **Analysis Results Diagram** > Dialog shows all the force diagrams of all load cases and combinations.
- Select Column / Wall > Right-click > Pick **Column / Wall Design Forces** > Report will be generated listing down design forces.

Verify the results

When FE Floor Analysis is successfully completed, we can go to "Post Analysis Processes and Report" tab to review or check the results & reports.

- *Go to **Analysis** tab > Click **FE Floor Analysis** to relaunch the FE Floor Analysis dialog.*
- *Click on "**Post-Analysis Processes and Reports**" tab*



Positive Moment Factor

For FE Floor Analysis Models it should be noted that they do not include for the effects of Pattern Loading. It is not feasible to automate pattern loading to generate every possible worst case scenario for every conceivable irregular arrangement. If you are concerned about the effects of Pattern Loading we suggest amplifying the 'Positive Moment Factor' (10 to 20% i.e. 1.1 to 1.2) to allow for these effects. Please note if a factor is applied, "Floor Mesh and Analysis" must be repeated.

- *Leave the factor as* ***1.0*** *for now.*
- *Click* ***Axial Load Comparison Report***
- *Check values of* ***Table 1 : Total Loads*** *(Input Loads) is close to* ***Table 4 : Finite Element Analysis Columns/Wall Axial Loads*** *(Output Loads).*

Axial Load Comparison Report

TOTAL LOADS (Based On Slabs Loads):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total (kN)
4 (+12.00 m)	127.2	193.5	14.3	4412.7	0.0	4747.7
3 (+9.00 m)	127.2	193.5	14.3	4412.7	0.0	4747.7
2 (+6.00 m)	127.2	193.5	14.3	4412.7	0.0	4747.7
1 (+3.00 m)	127.2	193.5	14.3	4412.7	0.0	4747.7
Total						18990.6

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total (kN)
4 (+12.00 m)	0.0	0.0	0.0	1724.7	0.0	1724.7
3 (+9.00 m)	0.0	0.0	0.0	1724.7	0.0	1724.7
2 (+6.00 m)	0.0	0.0	0.0	1724.7	0.0	1724.7
1 (+3.00 m)	0.0	0.0	0.0	1724.7	0.0	1724.7
Total						6898.6

FINITE ELEMENT ANALYSIS COLUMN/WALL AXIAL LOADS:

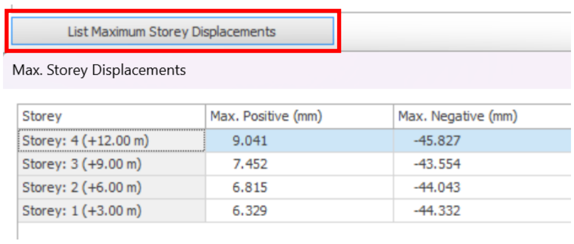
Storey	G (kN)	Delta G (kN)	Q (kN)	Delta Q (kN)
4 (+12.00 m)	6537.4	6537.4	2301.4	2301.4
3 (+9.00 m)	11610.7	5073.3	4129.2	1827.8
2 (+6.00 m)	16681.0	5070.3	5955.9	1826.7
1 (+3.00 m)	18979.9	2298.9	6879.6	923.7
Total		18979.9		6879.6

Note: The values printed in this table are based on the sum of axial load results of columns and walls.

Ignore Table 2 & Table 3 (& the warnings below the tables) as it is only applicable to beam slab model which requires results from Building Analysis.

If there are large discrepancies between sum of Table 1 & Table 4 it may mean that the flat slab model is set up incorrectly & hence loadings are lost.

- *Click "*List Maximum Storey Displacements*" button at the bottom of the dialog.*


List Maximum Storey Displacements

Max. Storey Displacements

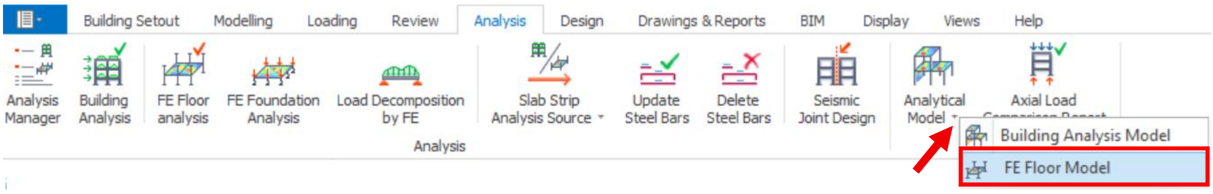
Storey	Max. Positive (mm)	Max. Negative (mm)
Storey: 4 (+12.00 m)	9.041	-45.827
Storey: 3 (+9.00 m)	7.452	-43.554
Storey: 2 (+6.00 m)	6.815	-44.043
Storey: 1 (+3.00 m)	6.329	-44.332

The table shows the maximum Positive and Negative Displacements of all the floors for the 1st combination, [1] 1.4G+1.6Q.

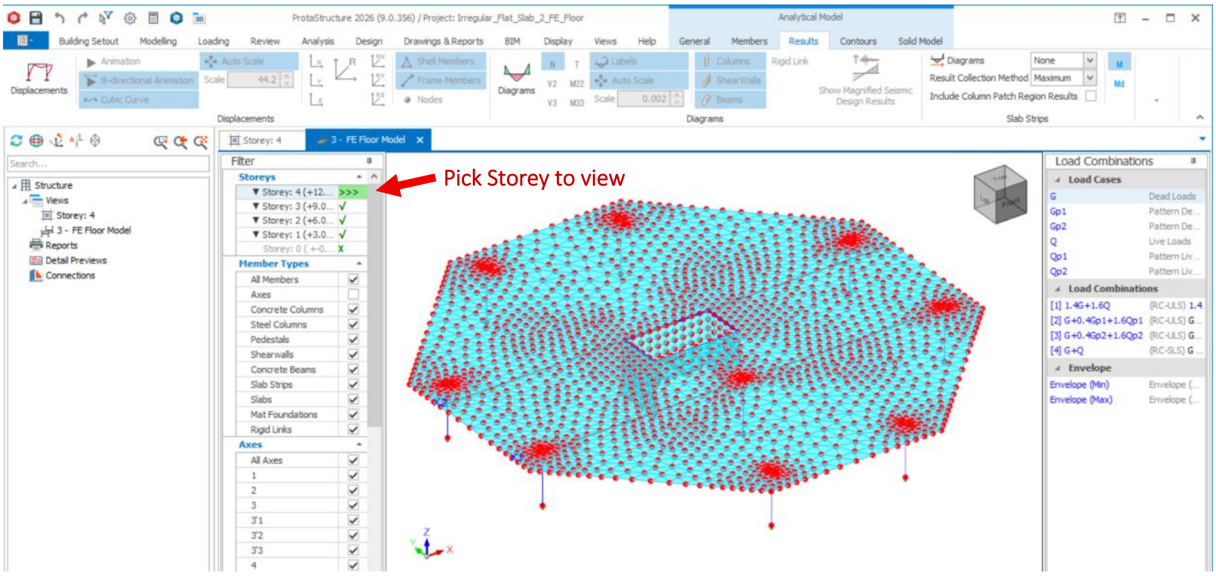
Positive is upwards while negative is downwards, in accordance with the global Z direction.

FE Floor Model

- *Click "*Display Analytical Model*" in the Post Analysis to access the **FE Floor Model**.*
- *Alternatively, go to **Analysis tab** > **Analytical Model** > Pick **FE Floor Model***



This will create and open the FE Model view (as shown below)



This functions & icons is the same as the Building Analysis Model, except that the FE Model is a single storey model. Under the left filter pane, >>> shows the current active storey.

- *Pick on "**Storeys**" number in the **Filter** pane to view another storey.*

The methods to show different contours are exactly as covered previously. Experiment with viewing the various Contours & setting up the various User-Defined Contours, as covered previously.

Column Punching Check

Column punching shear check must be done after the flat slab reinforcement is determined, as required by the design code.

- *Go to **Design** tab > Pick **Column Punching Check*** 

If no column is selected, then all columns will be included in the check. If you only want one or some specific columns, then select multiple the desired columns before clicking on this function.

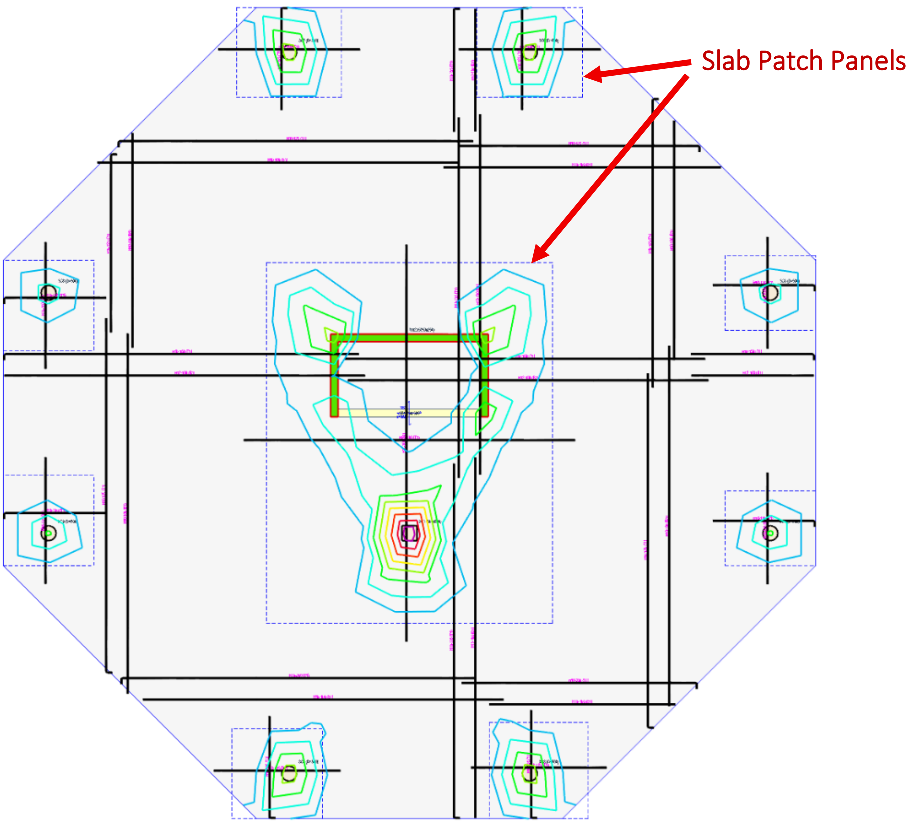
For details on the checks kindly refer to these articles in Prota Help Center :

- Column Punching Check
- Column Punching Shear Checks to BS 8110 and Eurocode 2

Slab Design with Slab Patch Panels

Based on the function of **FE Fixed Band Strip**, ProtaStructure has a new feature that allows users to **apply** a Base Reinforcement throughout the entire floor and design additional top or bottom bars at column support regions. This powerful new feature is called the **Slab Patch Panels**.

The example of the final output that can be achieved is shown below.



For details kindly refer to this article : Flat Slab and Raft Design with Slab Patch Panels

Thank You…

Thank you for choosing the ProtaStructure Suite product family.

It is our top priority to make your experience excellent with our software technology solutions.

Should you have any technical support requests or questions, please do not hesitate to contact us at all times through globalsupport@protasoftware.com and asiasupport@protasoftware.com

Our dedicated online support center together with our responsive technical support team is available to help you get the most out of Prota's technology solutions.

The Prota Team

ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®