

ProtaStructure 2022

Flat Slab Training Manual

Version 1.0

30 March 2022

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

asiasupport@protasoftware.com

globalsupport@protasoftware.com

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

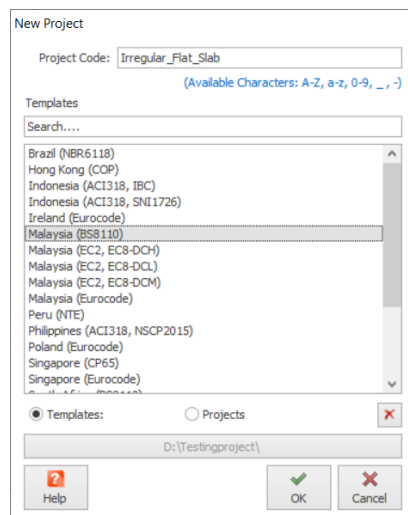
Starting a new project & Import Members.....	4
Generate Load Combinations	6
Create walls & beam.....	6
Inserting slabs.....	7
Final Flat Slab Model.....	9
Material Properties.....	9
Flat Slab Analysis & Design Flow Chart	11
Flat Slab Analysis using FE Floor Analysis Chasedown	12
Building Analysis to Generate Lateral Design Forces	13
Slab Meshing.....	15
Long Term Deflections.....	15
FE Floor Analysis Settings.....	16
Batch FE Chasedown Analysis.....	17
Verify the results.....	18
Reviewing results in the model.....	19
Finite Elements Analysis Results	20
Review the FE results.....	20
Bottom steel reinforcement provision.....	22
User Defined Contours	23
Creating contours (bottom steel provision).....	25
Creating contours (top steel provision) – Asd1-top.....	26
Slab Strip Diagram.....	29
Thank You... ..	31

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

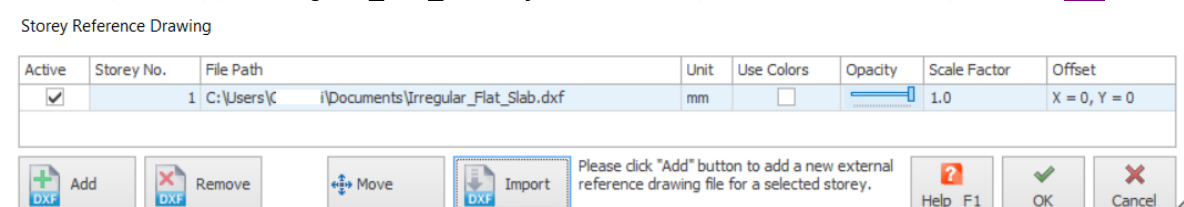
- Create a **New project** with the name "Irregular_Flat_Slab"
- Select '**Malaysia (BS8110)**' template and **Import** to current project settings



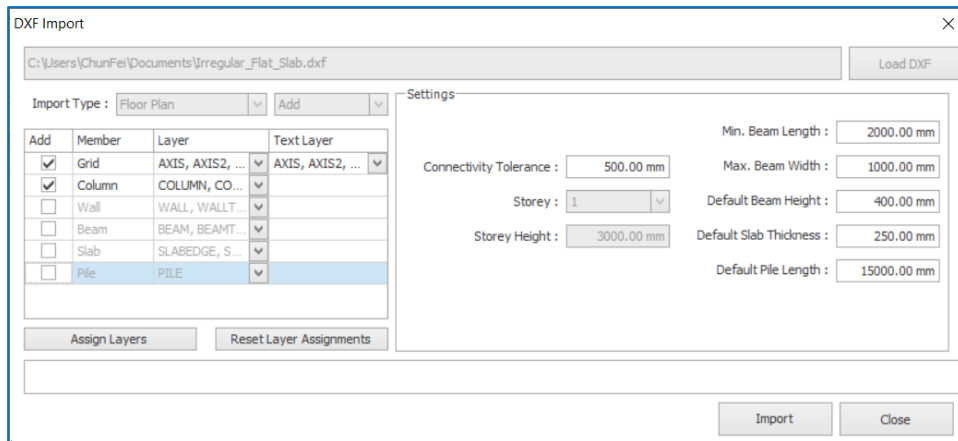
- Click on the '**External Reference Drawing**' icon in the **Building Setout** tab




- Add the given dxf file '**Irregular_Flat_Slab.dxf**' (DXF drawing can be downloaded from this [link](#))



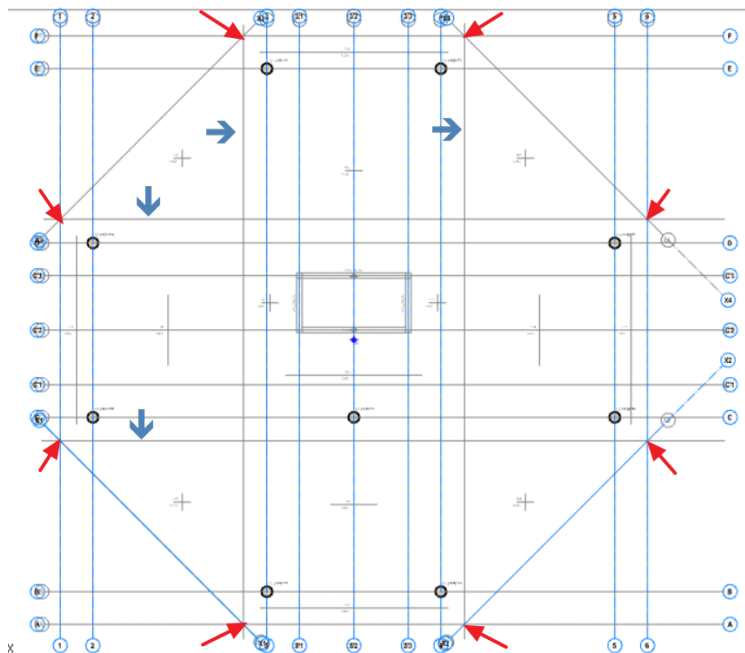
- Check **Active**
The reference file is shown in the background. We will now import the axes and column using the Import DXF function
- Click **Import**



- Under **Add**, check **Grid** and **Column**
 - For **Grid Layer**, choose only **AXIS2** & **AXIS_1**
 - For **Grid Text Layer**, choose on **AXISTEXT_1** & **AXISTEXT_2**
 - For **Column Layer**, choose only **COLUMN**
 - Click **IMPORT** > **Close** > **OK** to close the dialog
- The grids and columns will be imported as shown below
To get a less cluttered view, switch off the background grid.
- Go to **Review** tab > **Visual Interrogation** > **Scene Settings** > **Uncheck Grid** > **OK**
- The background grey grids will be switched off.

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

- Use the **Grid** icon and left click to create the **four** additional axes shown by the **blue** arrows below
(The name of axes does not matter)



TIP: Use the intersection of the existing grids as the start and end point of the new axes (as shown by **red** arrows)

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** > Generate load cases & combinations using > Untick **Create Different Combinations for Steel Design** (since no steel members) > tick **Use Cracked Sections in All Vertical Load Combinations as Well** > rest **all default** > **OK**

Automatic Loading Editor

Create Different Combinations for Steel Member Design

Use Cracked Sections in All Vertical Load Combinations As Well

V.Load Case = 6
H.Load Case = 2

Vertical Load Combinations | Horizontal Load Combinations

Define Dead Load Case (G)
 Define Live Load Case (Q)

Define Pattern Loads Automatically

=_ _ ... ==_ _ ...
 = _ ... _==_ _ ... _=_ _ ...

Direction Dependent Pattern Loading

Create Factored G Combination: No

Create Unfactored G+Q Combination

Favorable. G Coefficient:

Unfavorable. G Coefficient: 1.4

Unfavorable. Q Coefficient: 1.6

No	Combination	LL Red	R/C	Steel	G	Gp1	Gp2	Q	Qp1	Qp2	Nx	Ny
1	G+Q	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.40	0	0	1.60	0	0	0	0
2	(G+Q)p1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	0.40	0	0	1.60	0	0	0
3	(G+Q)p2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	0	0.40	0	0	1.60	0	0
4	G+Q+Nx	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.20	0	0	1.20	0	0	1.00	0
5	G+Q-Nx	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.20	0	0	1.20	0	0	-1.00	0
6	G+Q+Ny	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.20	0	0	1.20	0	0	0	1.00
7	G+Q-Ny	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.20	0	0	1.20	0	0	0	-1.00

Create walls & beam

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

Wall

Gen Drop 3D

Label: 1W1

Len (Storey): 1

(mm) b: 250.0 mm
e: 0.0 mm

Ext. I: 125.0 mm
J: 125.0 mm

Top -I: C'2 3'1
J: C'3 3'1

Bot -I: C'2 3'1
J: C'3 3'1

Update Cancel

Beam

General 3D

Label: 1B1

Type: Normal

(mm) b: 250.0 mm
e: 0.0 mm

h-Bot: 500.0 mm
h-Top: 0.0 mm

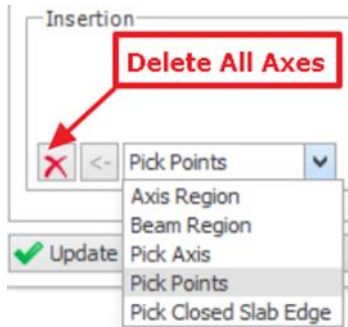
I-End C'2 3'1
J-End C'2 3'3

End Releases:

Update Close

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.



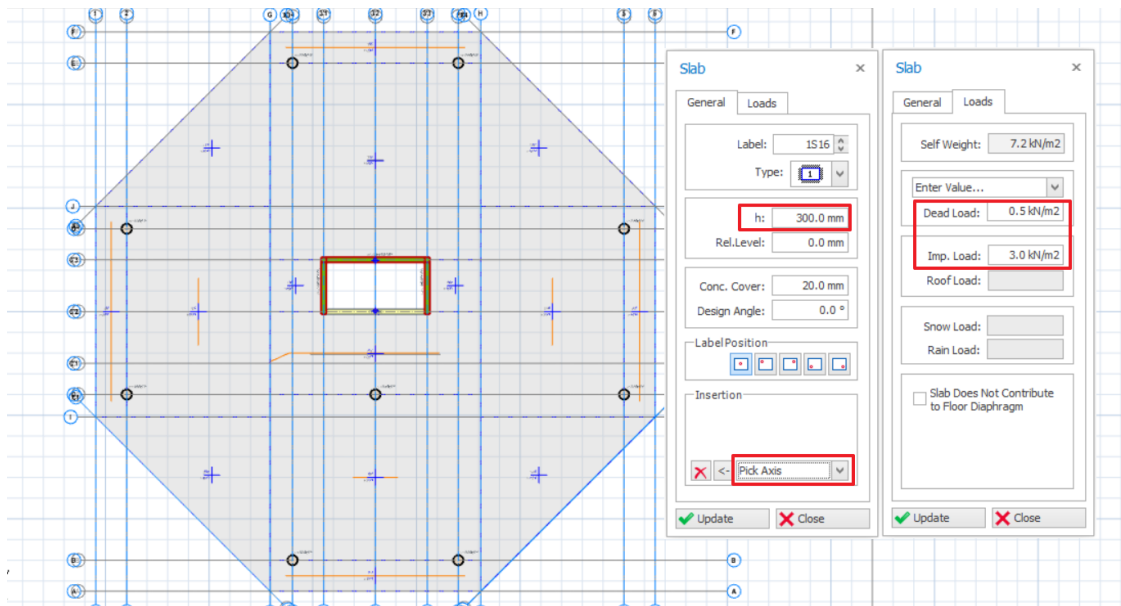
- **Axis Region:** Insert slab based on the nearest axes as boundary. Hold own CTRL & pick several axes region to combine > Release CTRL key & single slab in the combined region will be created.
- **Beam Region:** This is of no use for flat slab models due to the absence of beams. This is to be used 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 – Columns can sit within slab panels
 - 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/m²**, and an **Imposed Load of 3kN/m²**.
- 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.



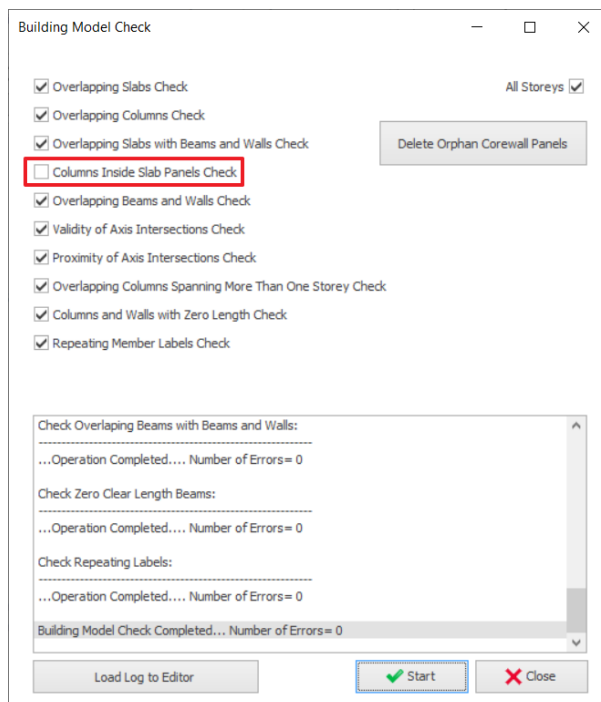
- Make all ST01, 02 & 03 storeys **similar** via **Building Setout > Storey Operations > Edit Storey** (Multiple select 3 Storeys & **click Define Selected Storeys as Similar**)

ST04 cannot be similar storey because it's the roof & hence the elevation is 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 copying this storey's information to the other floors, it is suggested to perform the **Building Model Check**.

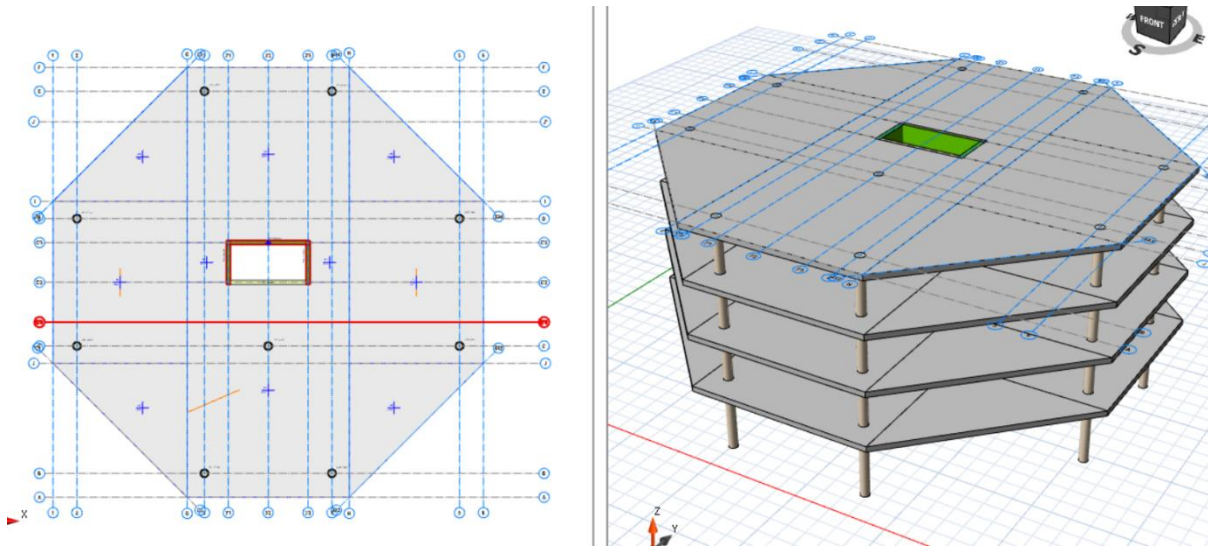
- Go to the **Analysis tab > Building Analysis > Analysis tab > Building Model Check**



- **Uncheck "Column Inside Slab Panels"**
This check is for beam slab layout. For flat slab, the columns can overlap with slabs.
- **Click Start to check**
Ensure there are no errors
- **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 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 and 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 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/mm²**
- Make sure the **Unit Weight of Concrete = 24 kN/m³**
- 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

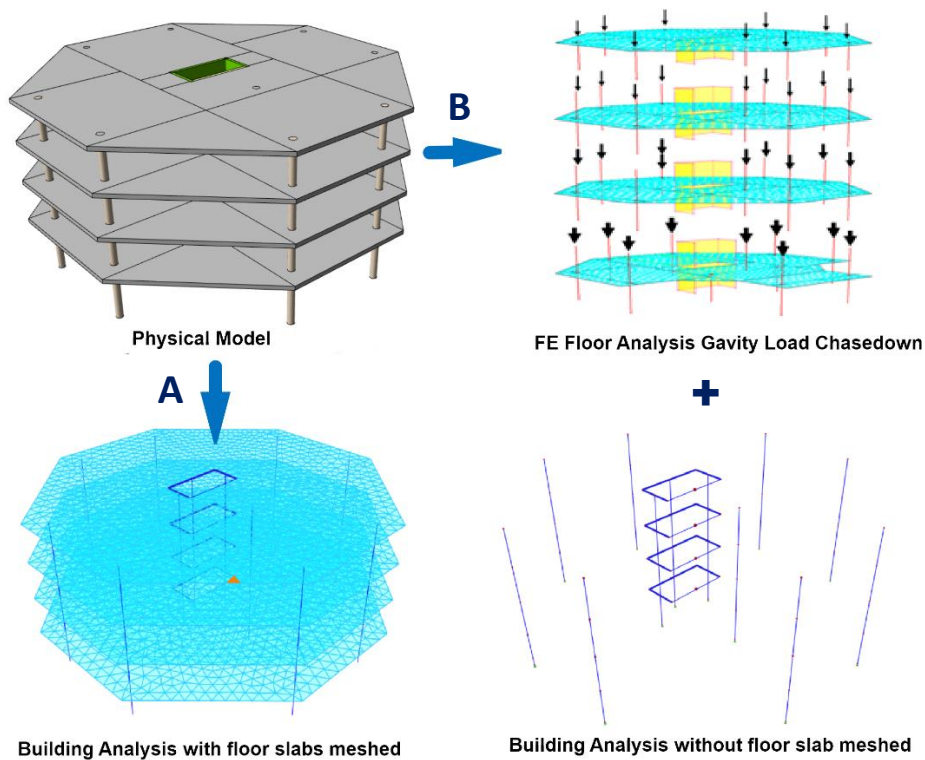
These settings for the Material Properties apply to both the Building Analysis and the Finite Element Analysis.

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.

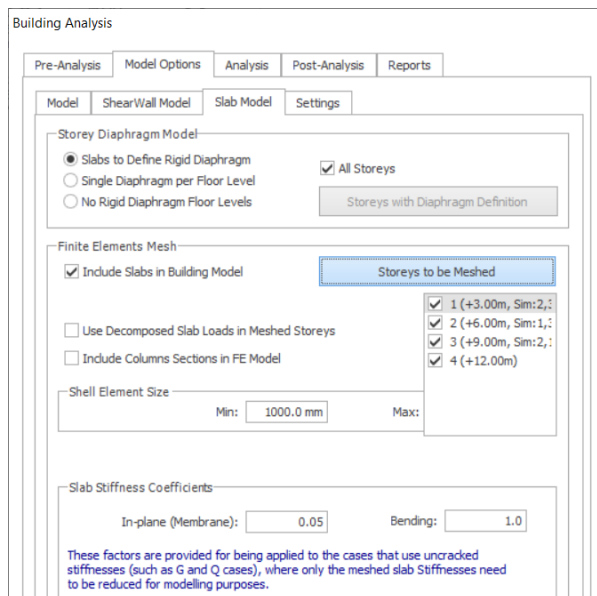


A. Building Analysis with floor slab Meshed	B. FE Floor Analysis Gravity Load Chase down + Building analysis without floor slab meshed
Full 3D model top to bottom storey, with floor slab meshed. There is option to mesh only selected storeys.	FE Floor Analysis analyses 1 storey only, i.e. subframe analysis.
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 small & simple 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 for moderately large models or models where only certain floors are flat slab.	More suitable for very large flat slab models or 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'
 - Review other parameters such as Shell Element size & Slab Stiffness Coefficients
 - 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

Since this method is similar to the basic training guide, we will not cover this in detail in this manual.

Flat Slab Analysis using FE Floor Analysis Chasedown

The following are summary of steps using this method

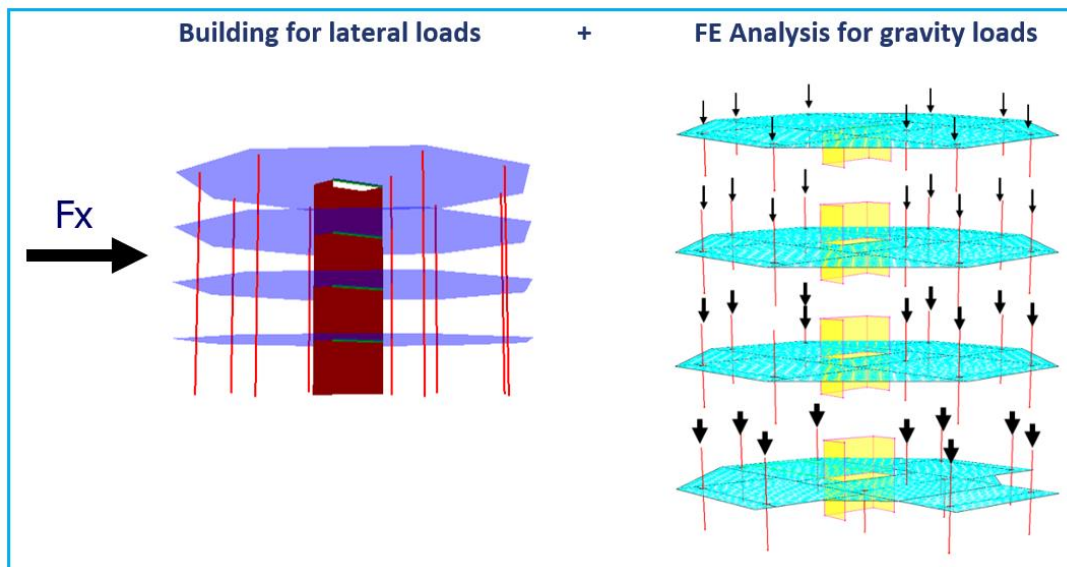
1. **Model** the flat slab model using slabs as outlined in the previous section.
2. **Go to Building Analysis > Model Options > Slab Model**
 - Untick 'Include Slabs in Building Model' > Floor slabs will not be meshed
3. Run **Building Analysis (BA)**
 - This will give results of gravity (G & Q) and 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



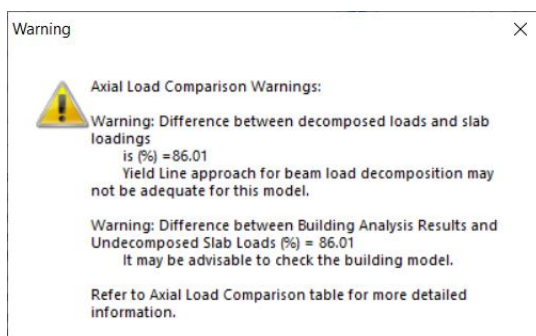
Building Analysis to Generate Lateral Design Forces

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 > For flat slabs, ensure to Use Un-decomposed Slab Loads for the calculation of the storey weight**
- Go to **Analysis tab > Click Building Analysis > Run Building Analysis**
- Ensure the **Column/Wall and Beam Reinforcement Design** is **'Unticked'**

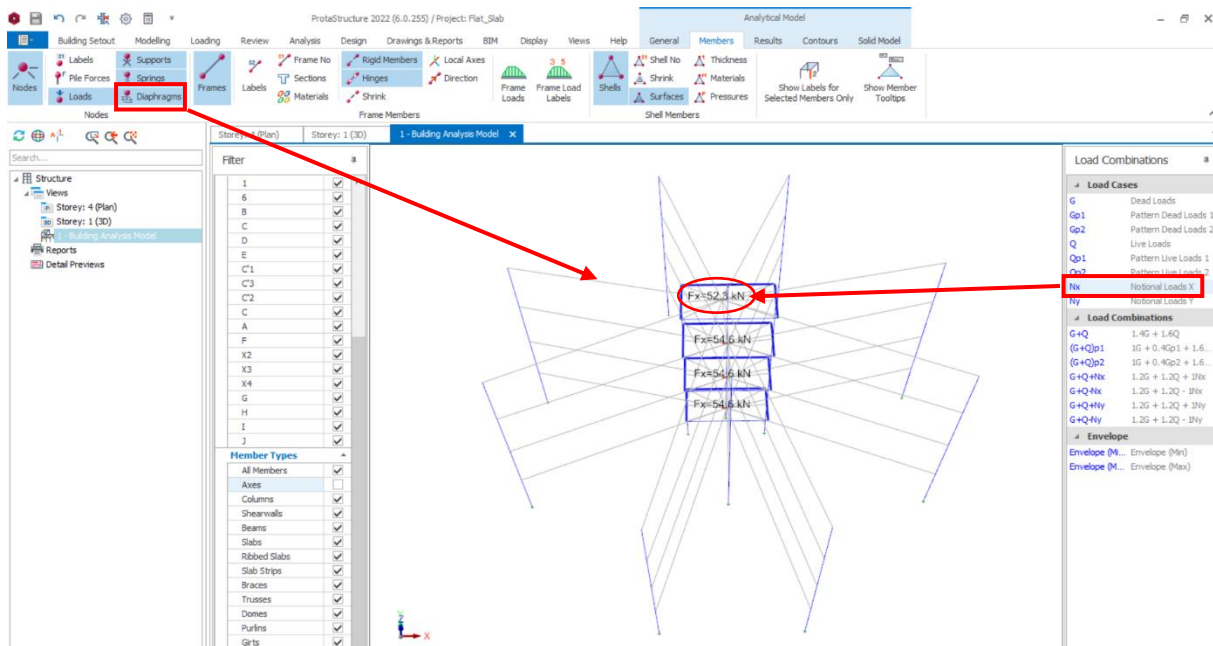
At the end of the analysis process, you should see a warning indicating load has been lost.



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 can be ignored.

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 **Nx** or **Ny** check > 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 no beams are present in the model there is no means of decomposing the vertical 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.

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 : $E_{lt} = 1/4 E_{st}$
 - For storage/ plant : $E_{lt} = 1/6 E_{st}$

In ProtaStructure, you can use the **Cracking and Creep** to estimate the amount of adjustment.

By changing the Properties of ALL elements, we are complying with the code ‘consistent section properties’ and the distribution of load will be unaffected.

If only the ‘Slab Stiffness’ is adjusted the columns will attract more moments, greater than traditional engineering expectation.

In this training manual, the stiffness factors will remain as default value. Users are advised to change the stiffness factor if would like to consider long term deflection in analysis.

To change the stiffness factor, please refer to following steps:-

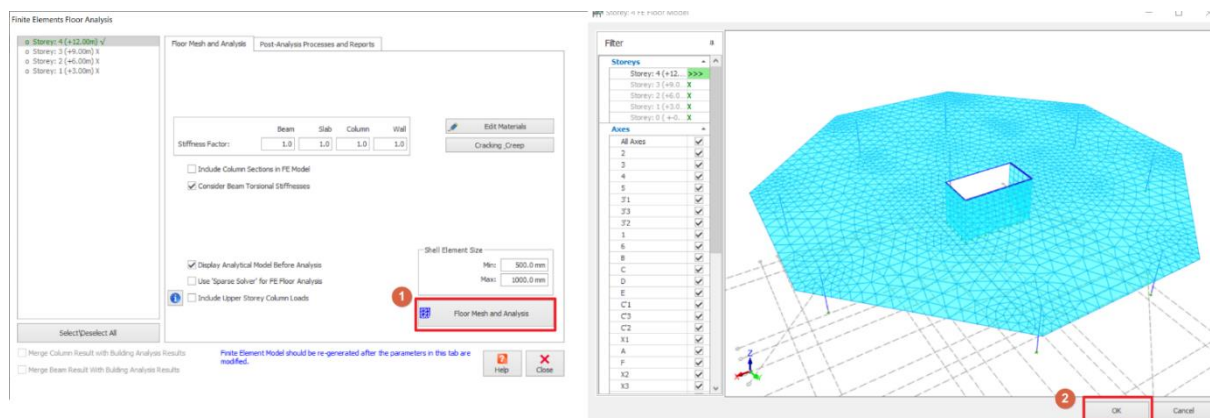
- Go to **Analysis** tab > **FE Floor Analysis** > Select **Top Storey**
- Click the **Cracking and Creep** button
- Enter an Average/Typical Dead Load
- Enter an Average/Typical Live Load
- Click OK

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.

‘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).



- Tick the option **“Display Analytical Model Before Analysis”**
- Input value in Min. & Max Shell Element Size and Click on the **‘Floor Mesh and Analysis’** button.
- Click **OK** to close & complete the analysis

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.

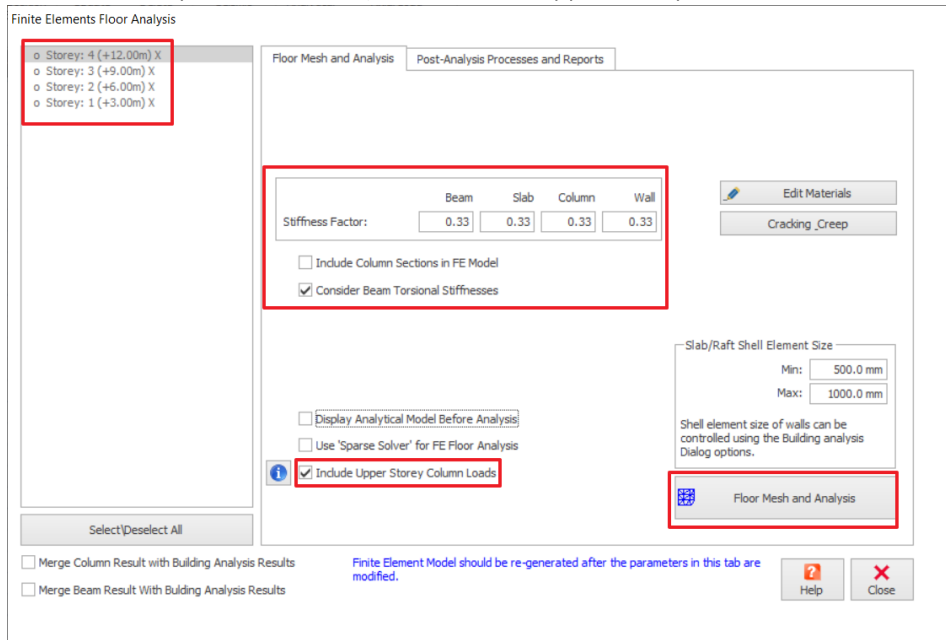
The key objective is to choose an ideal balance of minimum and maximum Shell Element Size and to produce minimum number of plates but still sufficient to accurately replicate the “true” behaviour of the slab (so analysis time is minimized). Generally, try to achieve between **8 plates** between the column heads.

Using minimum 500mm and maximum 1000mm plates size produces a mesh arrangement as shown previously which is clearly more than adequate.

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 may be 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.

Storey	Max. Positive (m)	Max. Negative (m)
Storey: 4 (+12.00m)	0.005	-0.019
Storey: 3 (+9.00m)	0.003	-0.018
Storey: 2 (+6.00m)	0.004	-0.019
Storey: 1 (+3.00m)	0.003	-0.019

When the FE Load Chase Down is complete the Maximum Positive and Negative Displacements can be reached by click **"List Maximum Storey Displacements"**

If there are large displacement, go to the **FE Storey and Foundation Models** to examine the deformation of floor & verify the model is correctly setup.

Finite Elements Floor Analysis

- ▼ Storey: 4 (+12.00m) ✓
- ▼ Storey: 3 (+9.00m) ✓
- ▼ Storey: 2 (+6.00m) ✓
- ▼ Storey: 1 (+3.00m) ✓

Select/Deselect All

Floor Mesh and Analysis
Post-Analysis Processes and Reports

	Beam	Slab	Column	Wall
Stiffness Factor:	0.33	0.33	0.33	0.33

Include Column Sections in FE Model
 Consider Beam Torsional Stiffnesses

Display Analytical Model Before Analysis
 Use 'Sparse Solver' for FE Floor Analysis
 Include Upper Storey Column Loads

Slab/Raft Shell Element Size

Min:

Max:

Shell element size of walls can be controlled using the Building analysis Dialog options.

Merge Column Result with Building Analysis Results
 Finite Element Model should be re-generated after the parameters in this tab are modified.

Merge Beam Result With Bulding Analysis Results

- Before existing the 'Finite Element Analysis Form' tick '**Merge Column Results with Building Analysis Results**'

It is especially important that this is done for flat slab system, 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*.

Verify the results

When FE Chasedown is successfully completed, you can access the **Axial Load Comparison Report** from the **Post-Analysis Processes and Reports** tab of the FE Floor Analysis form.

Finite Elements Floor Analysis

- ▼ Storey: 4 (+12.00m) ✓
- ▼ Storey: 3 (+9.00m) ✓
- ▼ Storey: 2 (+6.00m) ✓
- ▼ Storey: 1 (+3.00m) ✓

Floor Mesh and Analysis
Post-Analysis Processes and Reports

Positive Moment Factor:

Negative Moment Factor:

- Re-enter to **FE Floor Analysis** form
- 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
4 (+12.00m)	127.2	193.5	14.3	4412.8	0.0	4747.8
3 (+9.00m)	127.2	193.5	14.3	4412.8	0.0	4747.8
2 (+6.00m)	127.2	193.5	14.3	4412.8	0.0	4747.8
1 (+3.00m)	127.2	193.5	14.3	4412.8	0.0	4747.8
Total						18991.1

Q - Live Loads:

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

FINITE ELEMENT ANALYSIS COLUMN/WALL AXIAL LOADS:

Storey	G	Delta G	Q	Delta Q
4 (+12.00m)	4748.4	4748.4	1724.7	1724.7
3 (+9.00m)	9488.2	4739.7	3449.3	1724.7
2 (+6.00m)	14227.9	4739.7	5174.0	1724.7
1 (+3.00m)	18967.6	4739.7	6898.6	1724.7
Total		18967.6		6898.6

Ignore Table 2 & Table 3 (& the warnings below the tables) as it is for 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.

Reviewing results in the model

- In the plan view, go to **Review** tab > **Visual Interrogation** > **Column Plan Display** > **Check Axial Load and Loading G, Q, and G+Q** > click **OK**.

The screenshot shows the software interface with the 'Visual Interrogation' menu highlighted. The 'Active Windows Settings and Visual Interrogation' dialog box is open, showing the following settings:

- Display Analysis Results:**
 - Nodal Loads
 - Temperature Loads
 - Axial Load
 - Bending Moment
 - Shear Force
- Loading:**
 - G+Q (G+Q)
 - (G+Q)p1 ((G+Q)p1)
 - (G+Q)p2 ((G+Q)p2)
 - G+Q+Hx (G+Q+Hx)
 - G+Q-Hx (G+Q-Hx)
 - G+Q+Ny (G+Q+Ny)
 - G+Q-Ny (G+Q-Ny)

On the right, a circular diagram of a column (1C1 (D=500)) displays the following axial load results:

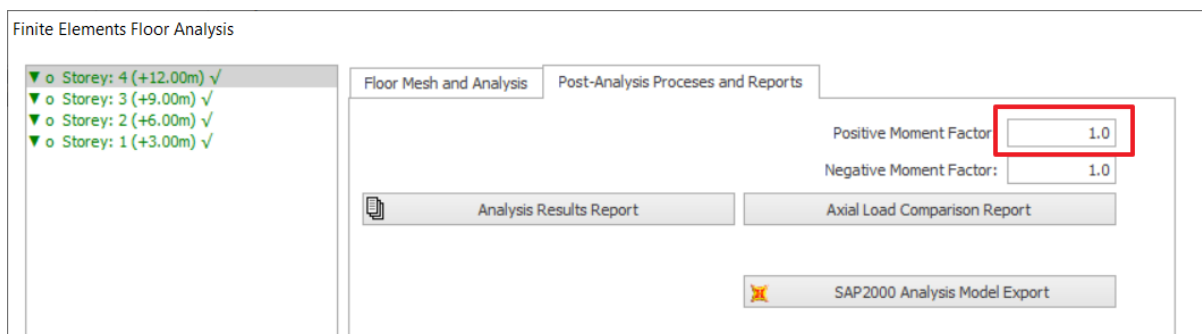
- G: N=2134.8 kN
- Q: N=809.5 kN
- Cmb1: N=4283.8 kN

If you have chosen to design including Imposed Load Reduction Settings for the columns (Edit Storey), the Axial Load values for **G** and **Q** are the Unfactored Dead and Imposed Load.

Any combination results reported graphically include the Imposed Load Reductions if applicable.

Finite Elements Analysis Results

- Double-click **ST: 1** in structure tree to switch active storey view from **ST : 0**
- Return to the **Finite Elements Floor Analysis** form (function is deactivated in **ST: 0** view)
- Select **Storey: ST04 > Post-Analysis Processes and Reports** tab



Positive Moment Factor Adjustment

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 > **Close** the dialog.

Alternatively, you can setup user-defined load pattern using "**Pattern Loading Definition**" under **Loading** tab.

Review the FE results

- Go to **Analysis** tab > **Analytical Model** > **FE Storey and Foundation Models**

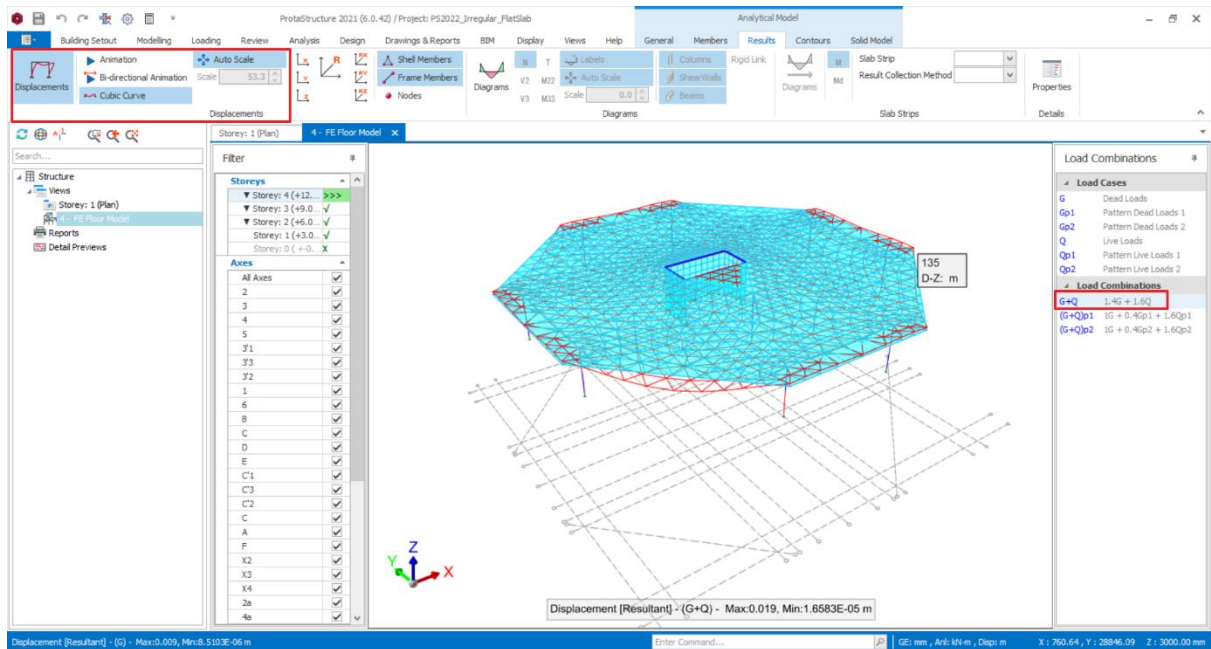


This will open the model in a 3D view with any default selection shown on the slab area. The **Members** tab controls what elements you want to see. Example, if the view is too cluttered, switch off some elements.

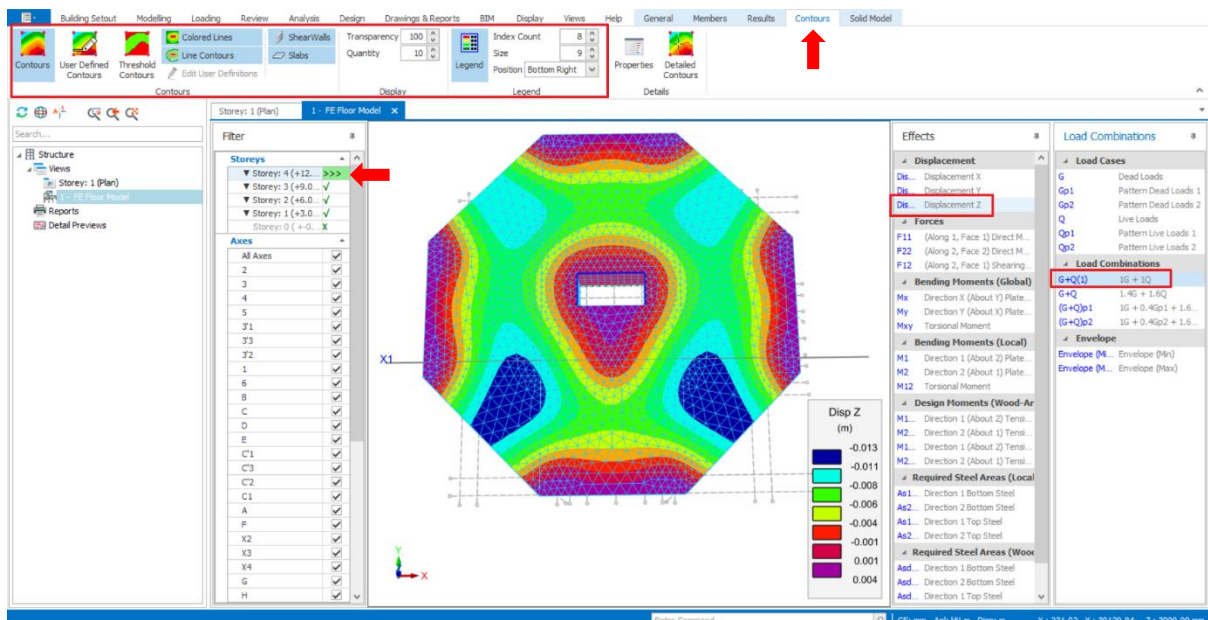
Experiment switching on & off other elements & information icons to find out what each icon does.

It is highly recommended to carefully examine the **Displacement**, as slabs are the key structural element of FE Floor Analysis. Specifically, check if the shape & magnitude of the slab deflection are reasonable.

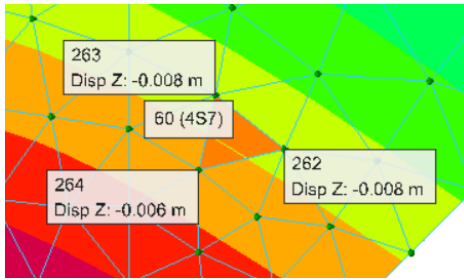
- Choose **Displacement** in the **Result** tab. You can **Increase** or **Reduce Scale** or enter scale after switching off the **Auto Scale**.



- **Animation** is very useful to visualise the deformation of slab in motion, e.g. observe if the slabs is indeed deforming as connected entities.
- Go to **Contours** tab and switching on the **Contours** to display the selected effect in coloured contours. Contours display can be changed in **Display Panel** and **Legend Panel**.



Placing the mouse cursor on any node of the slab will display the value of the effect.




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

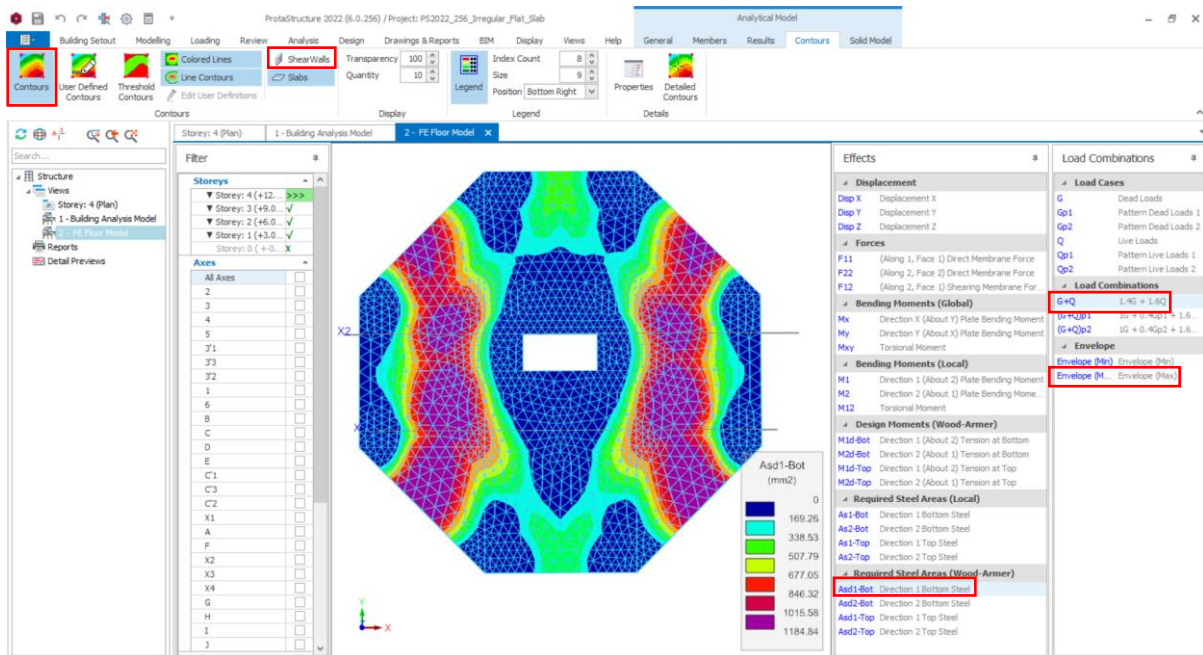
- **Coloured Lines:** Colour line contours
- **Line Contours:** Draw line contours

Because we used the Cracking & Creep tool, we have made adjustments to all of our Element Stiffness Factors. Although these Displacement Contours are based on the G+Q Load Combination, these stiffness factor adjustments mean that the displacements shown are actually **Unfactored**.

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)”**, e.g. **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. Asd...
- Under **Load Combinations** select the desired design combination, e.g. **G+Q (factored)**
Alternatively, if you like to design for envelope of all load combination for the bottom steel, choose **Envelope (Max)**
- Click **Plan View**  (located in **General** tab) to lock the view in plan mode.
- Ensure **shearwall** element is **turn off** to avoid contour generated in slab elements distorted by shearwalls.



The view (above) shows design bottom steel requirements in direction 1 and as might be anticipated the peak requirements are occurring along the longest free edges.

➤ *Select Effect Asd2-bot*

In direction 2, we see a very similar situation and a very similar peak requirement of around 1200mm²/m.

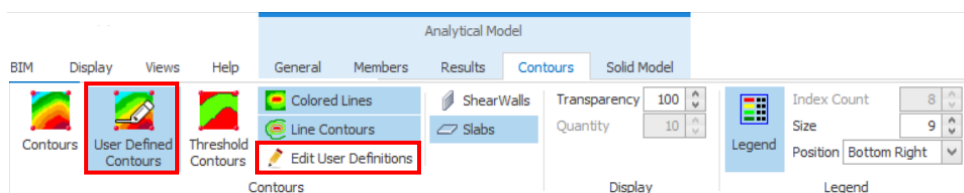
In practice you may want to consider the reinforcing requirements if the main steel along the angled edges is aligned to the edge. This can be done by cutting slab strips at angle, which will be covered later.

User Defined Contours

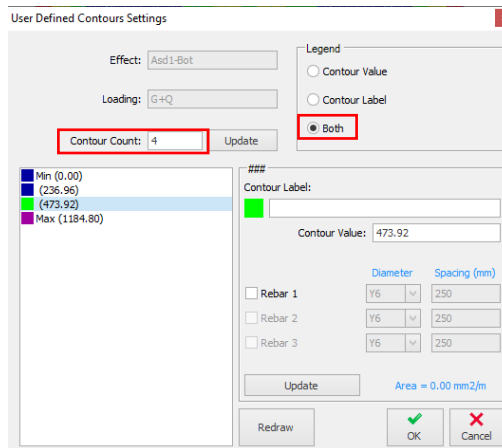
➤ *Switch back to view Asd1-bot.*

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', and click Update button to re-interpolate contour values.*



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

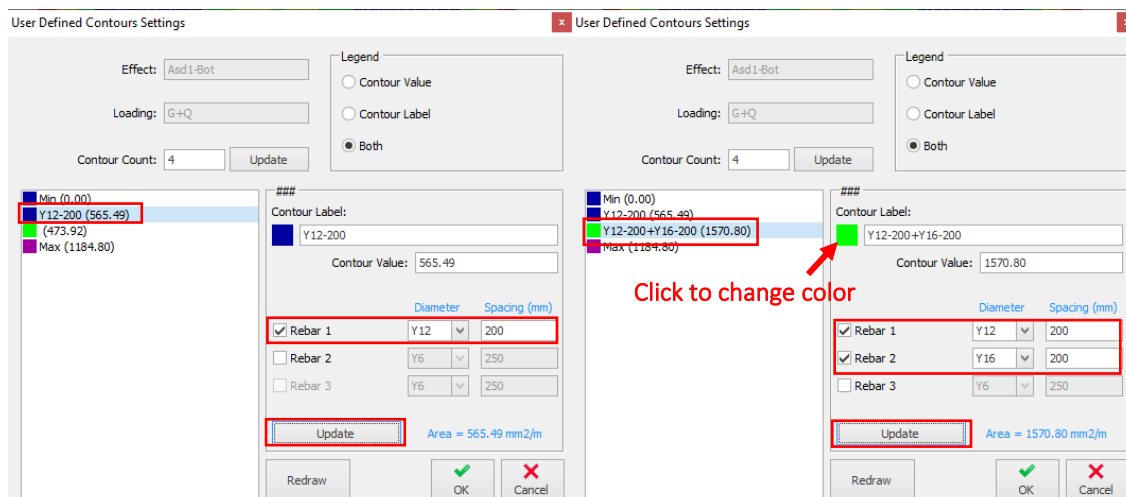
Loading – The current Load Case or Combination selected in the FE Storey and Foundation Models

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. Noted that, the 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.

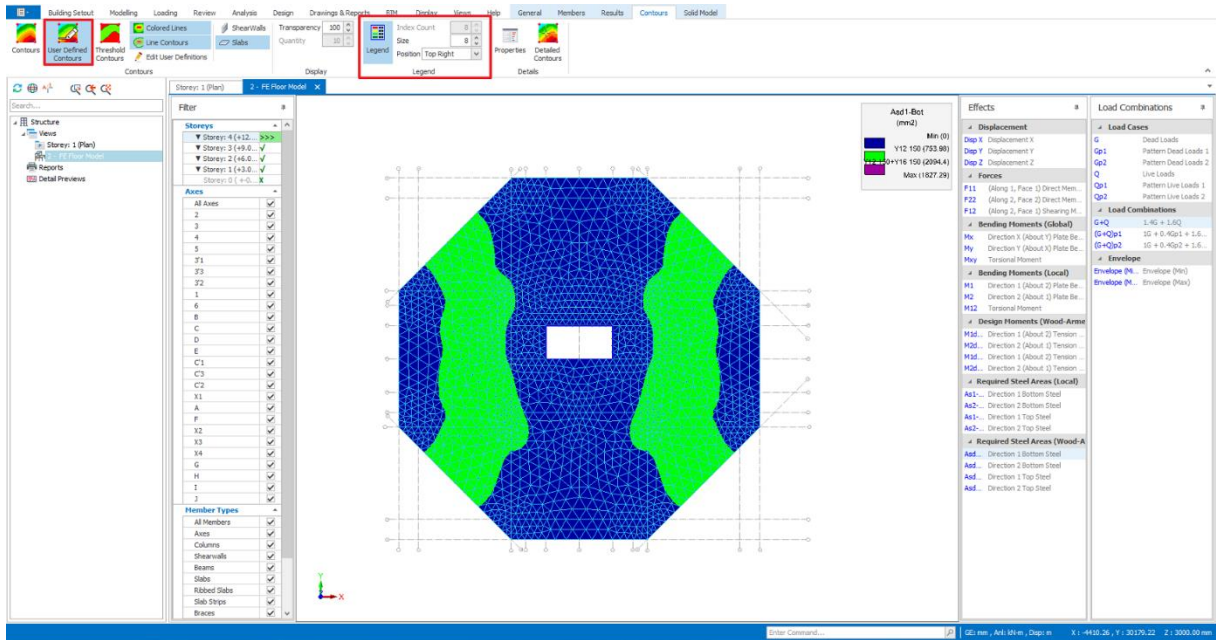
➤ *Make the contour settings as shown below and click on **Update** for each input.*



You can change the contour color by click on the color icon, just below contour label.

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

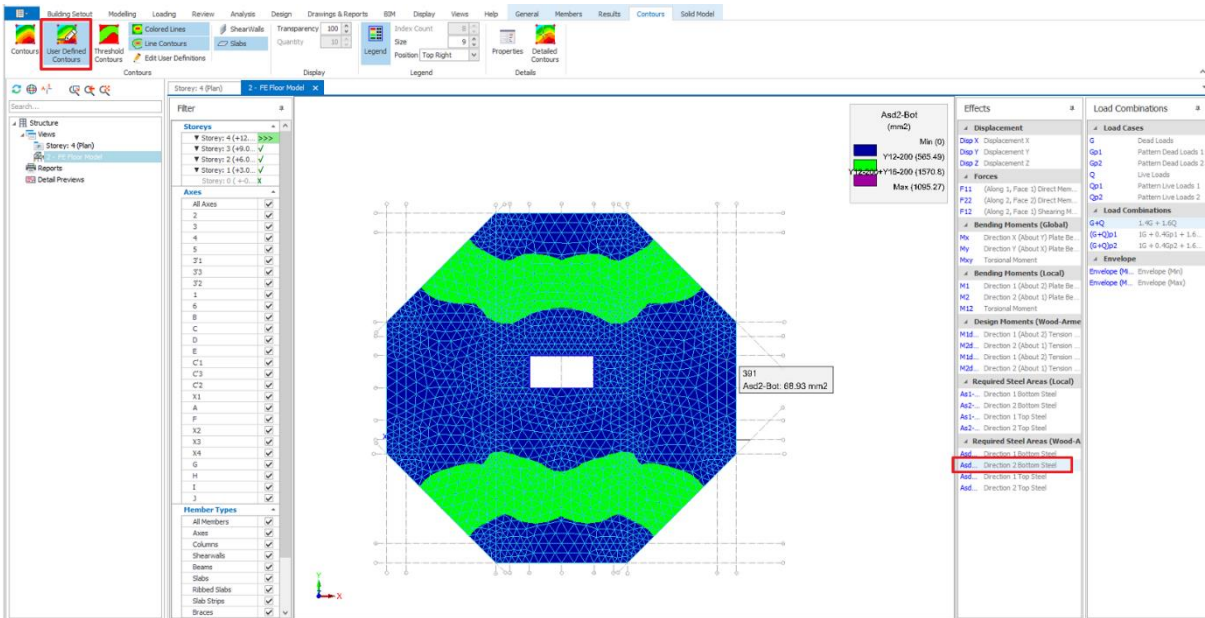
If the contour legend is hidden you can click on **“Position”** shift it to another corner. The legend size can also be increased or decreased.



The “Blue” region is where H12-200 is required. The “Green” region is where H12-200 + H16-200 is required. Effectively, this means H12-200 is placed in the entire slab, while H16-200 added in specific “Green” area. This effectively means the “Green region” requires bar spacing of 100mm. “Purple” region indicates area where the provided steel is insufficient (there are none here).

Creating contours (bottom steel provision)

- Using the same techniques, create the *As(d)2-bot* contours, as shown below
- Create two contours *H12-200* and *H12-200 + H16-200*.



As summary, H12-200 & H12-200 + H16-200 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.

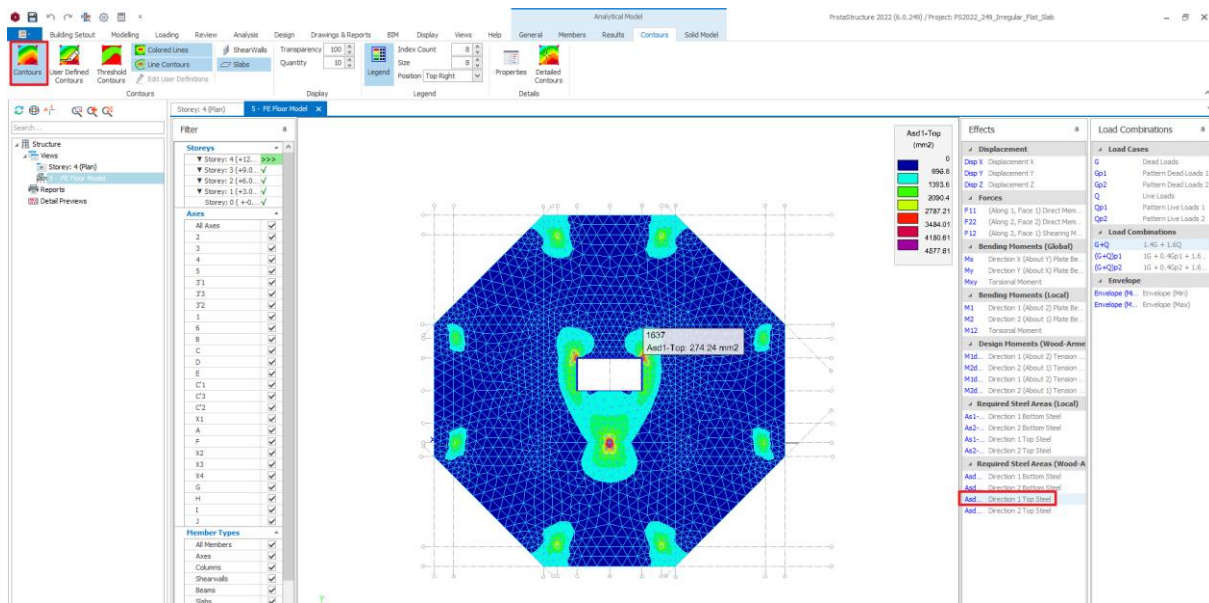
- Go to **Review** tab, click on **Visual Interrogation > FE Contours**, select contours type, effects and load combinations based on preferences.

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.

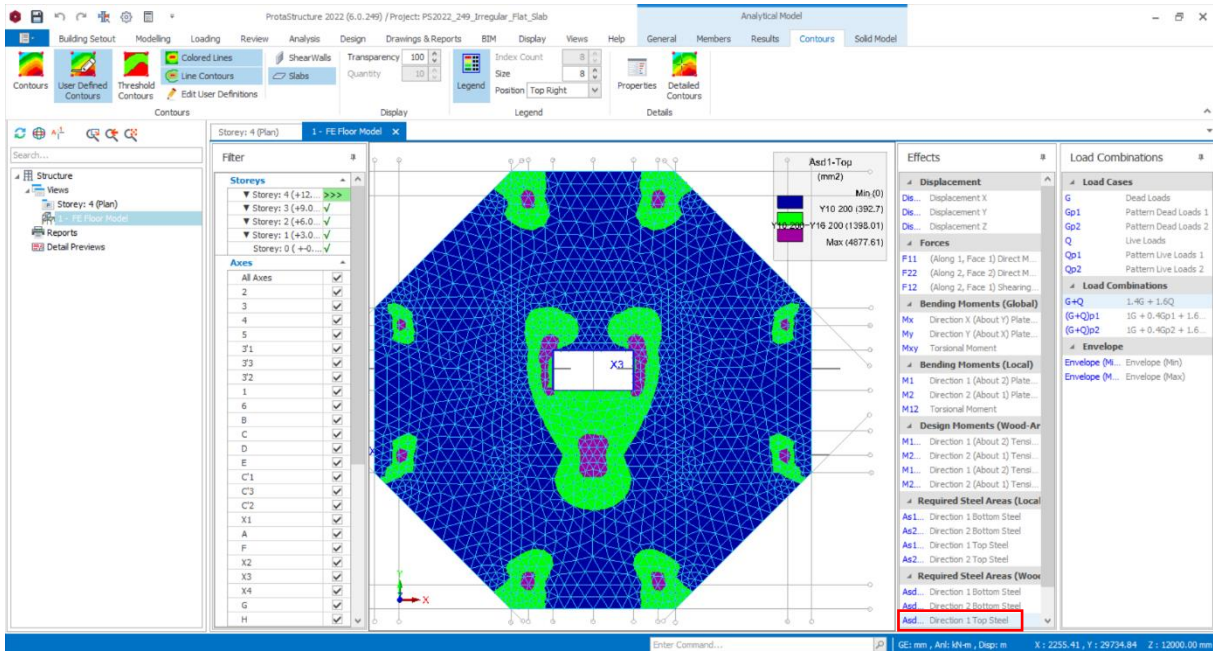
Creating contours (top steel provision) – Asd1-top

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



As expected, a review of the top reinforcing requirements shows that in theory no steel is required over large areas of the slab and that the hogging moments intensify rapidly over the column heads and core wall. However, many engineers would tend to provide minimum reinforcement throughout the top of an irregular flat slab and similarly use user-defined contours to design the steel.

- Create the contours shown below: **H10-200** and **H10-200 + H16-200**.



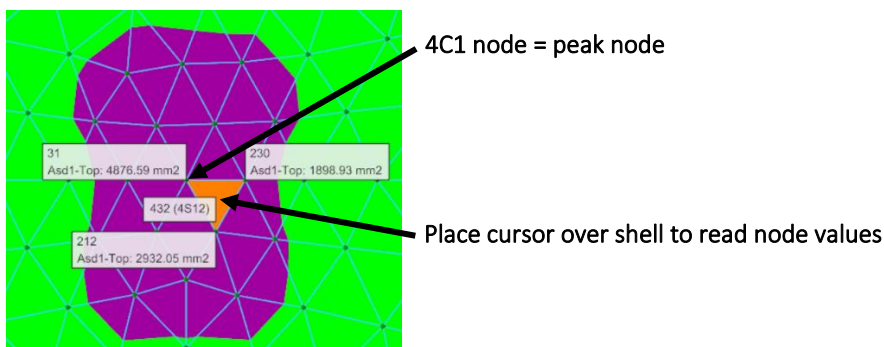
It is noticed that H10-200 will be provided most of the area (blue).

Then we see large areas where H16-200 would need to be added (green) leaving relatively small areas where a higher provision is necessary (purple).

As expected, peak steel requirement is at top of columns & walls. It would be impractical to provide top steel exactly to match the peak maximum steel area as these a very small localised area often exposed when meshing slabs.

The more practical way to deal with these peak requirements is to integrate (average) results in strips cut across the column and wall heads or average out the steel requirements surrounding the peak node. These two methods will be discussed as below.

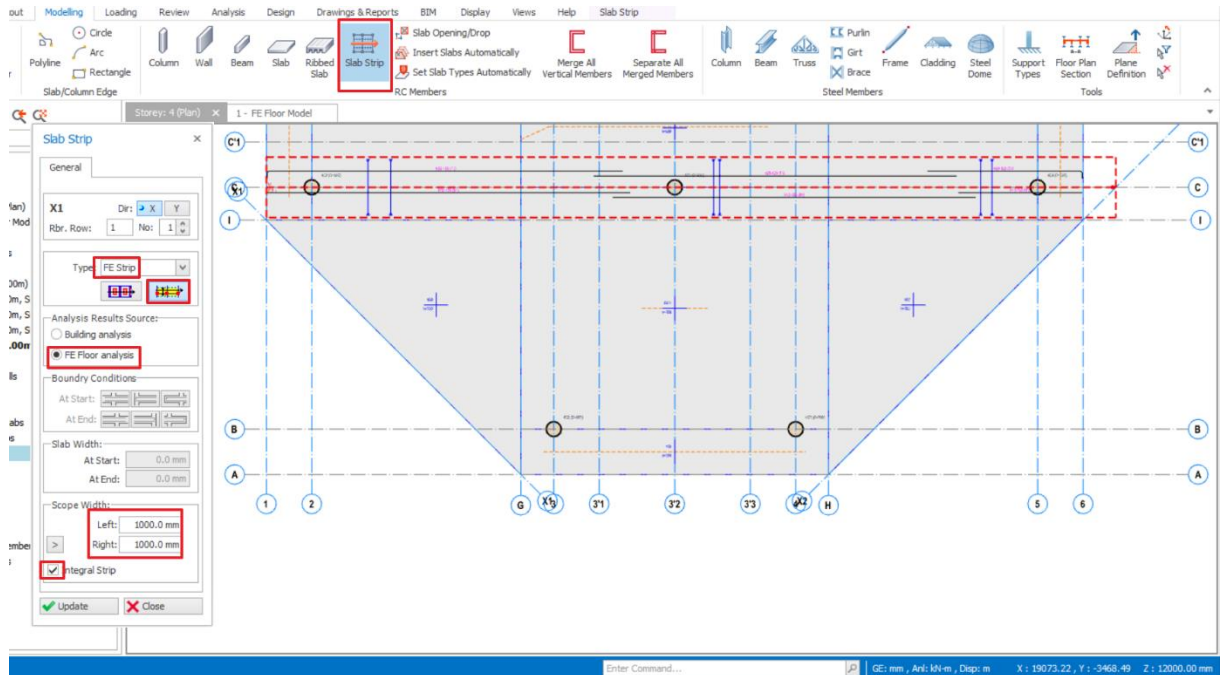
- Place your mouse cursor over at slab mesh at column 4C1 & values at the edges of the shell nodes will be shown (as below)



Once you average out the 6 shells surrounding the peak node, you will find out that the steel requirement is around 3200mm²/m which provides H25-150 to corresponding area will be sufficient.

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

- Go to plan view, ensure the active storey is Storey-04
- Click "Slab Strip" icon, ensure that below options are set to create an integral FE slab strip with 2m wide and draw the slab strip across the column head.



- If the slab strip created is according to the settings set as shown as above, program automatic provides H20-125 cover the area of the slab strip width.

Note: When this steel is provided, it **must** be provided over at least the 2m width of the strip. The fixed band strip design provides a single top and bottom bar across the entire length & width of the strip. Practically, this is not an economical design (too conservative) & 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**.

From the report, it is noted that the required steel reinforcement is around 2300 mm²/m. This value is significantly lower than the manually calculated average shell values around the peak node of 3200 mm²/m. This is because the 2 meter slab strip is averaging over more shells. 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	2300	Likely too wide
500 mm	1 m	2700	Lower bound, less conservative
250 mm	500 mm	3500	Upper bound, more conservative

As expected, the narrower the strip, the higher the required area of steel. The 500mm total strip width is close the manually computed average of 3200 mm²/m. 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.
- Size of support : The size of the column is already 500 mm, hence 500 mm width is likely already conservative as rigid zone effect of column is not considered. The column idealized as single node in FE Analysis.
- The rigid zone of column can be considered by meshing the slab to perimeter of column by checking the option "**Include Column Sections in FE Model**". This will likely result in lower peak moments at the column head.

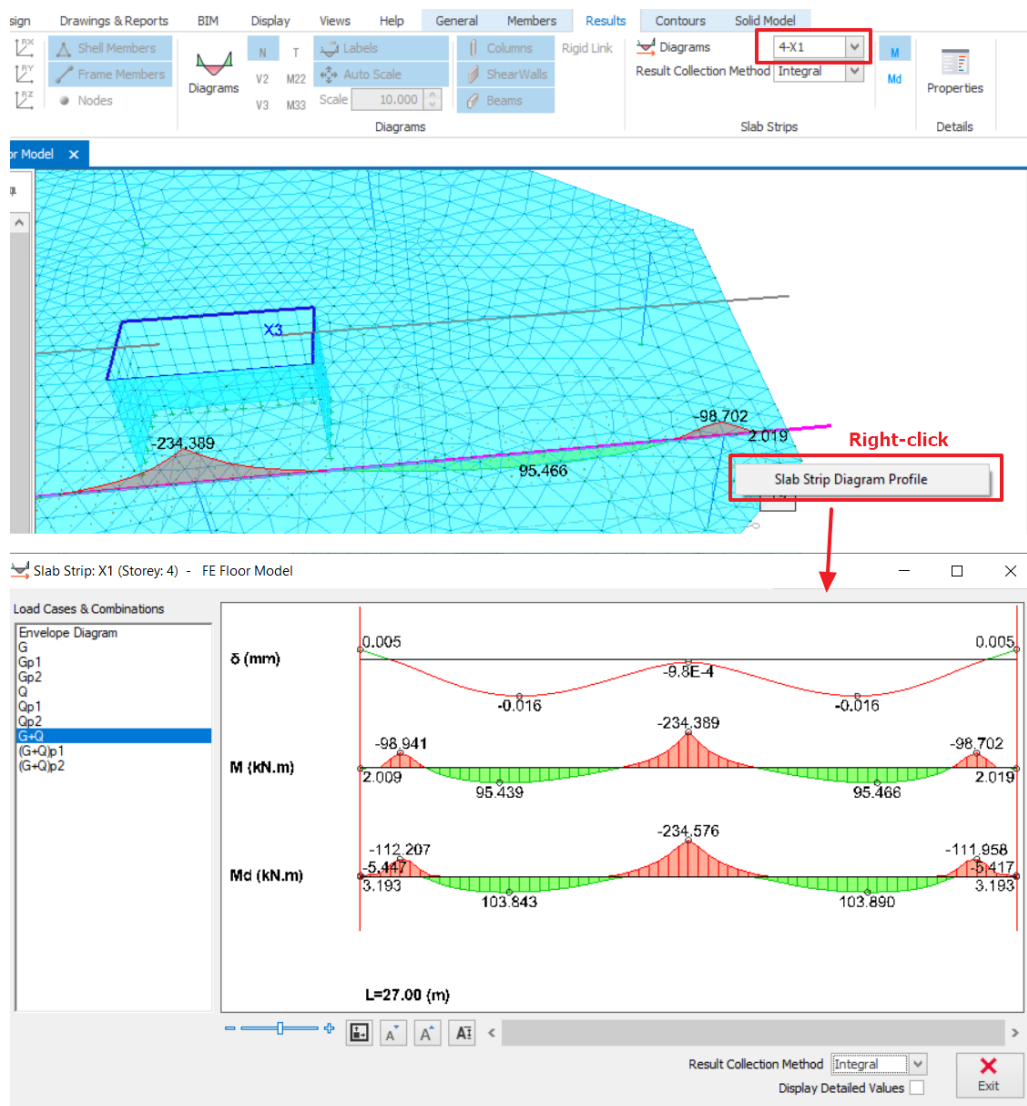
Noted that if a 500mm width strip is adopted, this will require estimated H25-125 over the column area.

Now, apply the same process to wall area. Create a slab strip across the wall. It is found that the required steel area is around 2000 mm² when the total slab strip width is 500mm.

Slab Strip Diagram

To review slab strip diagram in FE Models:-

- 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 on the analytical model, right-click > select "**Slab Strip Diagram Profile**".



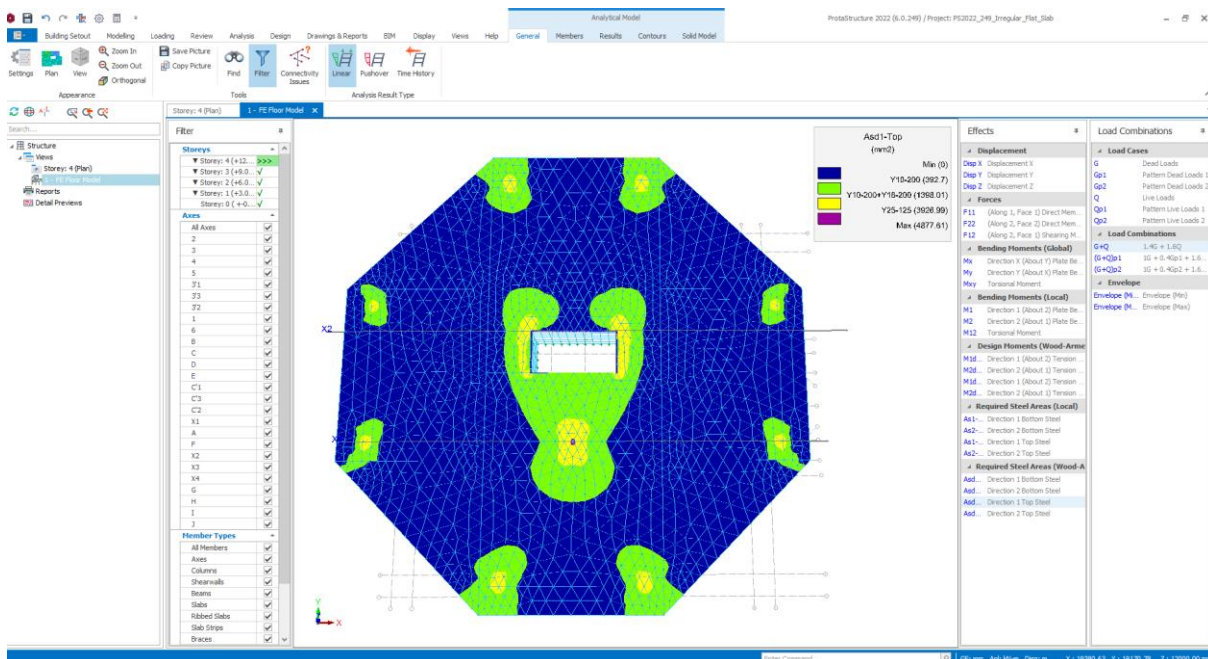
Program provides 3 different result collection methods namely Maximum, Integral, and Strip Line.

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 smoothed 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

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

Below we will add one contour to satisfy both required steel at walls area and all the columns area.

- Click **“Edit User Definitions”** again, and increase Contour count to **5**, and click **“Update”**
- Update the first and second contour with reinforcement rebar **H10-200** and **H10-200 + H16-200** respectively.
- For the third contour, update it with reinforcement rebar **H25-125** to satisfy all the areas (shearwalls and columns).



Upon closer inspection on the contour plotted, you can still see **“purple”** area are shown which is suggesting the contours created are not adequate. This would be the case if we were considering every nodes 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 purple area can be ignored.

The same steps can be repeated for Asd2-Top using User-Defined Contours.

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

