ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®

ProtaStructure Suite 2024 – New Features

Version: 3.0 November 2023

Please get in touch with us for your training and technical support queries.
asiasupport@protasoftware.com | globalsupport@protasoftware.com

Published by PROTA

Limitation of Responsibilities

While Prota ensures every new update is tested, 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 results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- Ensure the software is appropriately used according to the reference manual and documentation.

Intellectual Property

ProtaStructure is a registered trademark of **Prota Yazılım Bilişim ve Mühendislik A.Ş.,** and all intellectual property rights belong to **Prota Yazılım Bilişim ve Mühendislik A.Ş**. Documentation, training, reference manuals, and any program component cannot be copied, distributed, and used in violation of the license agreement.

Trademarks

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

Contents

Introduction

As Prota, we have been developing leading structural BIM software for over 30 years.

Priority on our product strategy has always been:

1. To provide new practical modeling approaches
2. To introduce innovative design technologies that will add value to your business and extend your toolset in your daily engineering practice.
3. To improve on the existing functionalities
4. To introduce more localization in our products, so that you can benefit from our technology much better.

ProtaStructure 2024 is our brand-new release, a significant step taken to fulfill this strategy, raise the bar for competition, and meet expectations. Developing a comprehensive structural BIM analysis, design, and detailing solution is challenging teamwork in terms of meeting user and industry expectations. We sincerely thank all our users for their trust in our products.

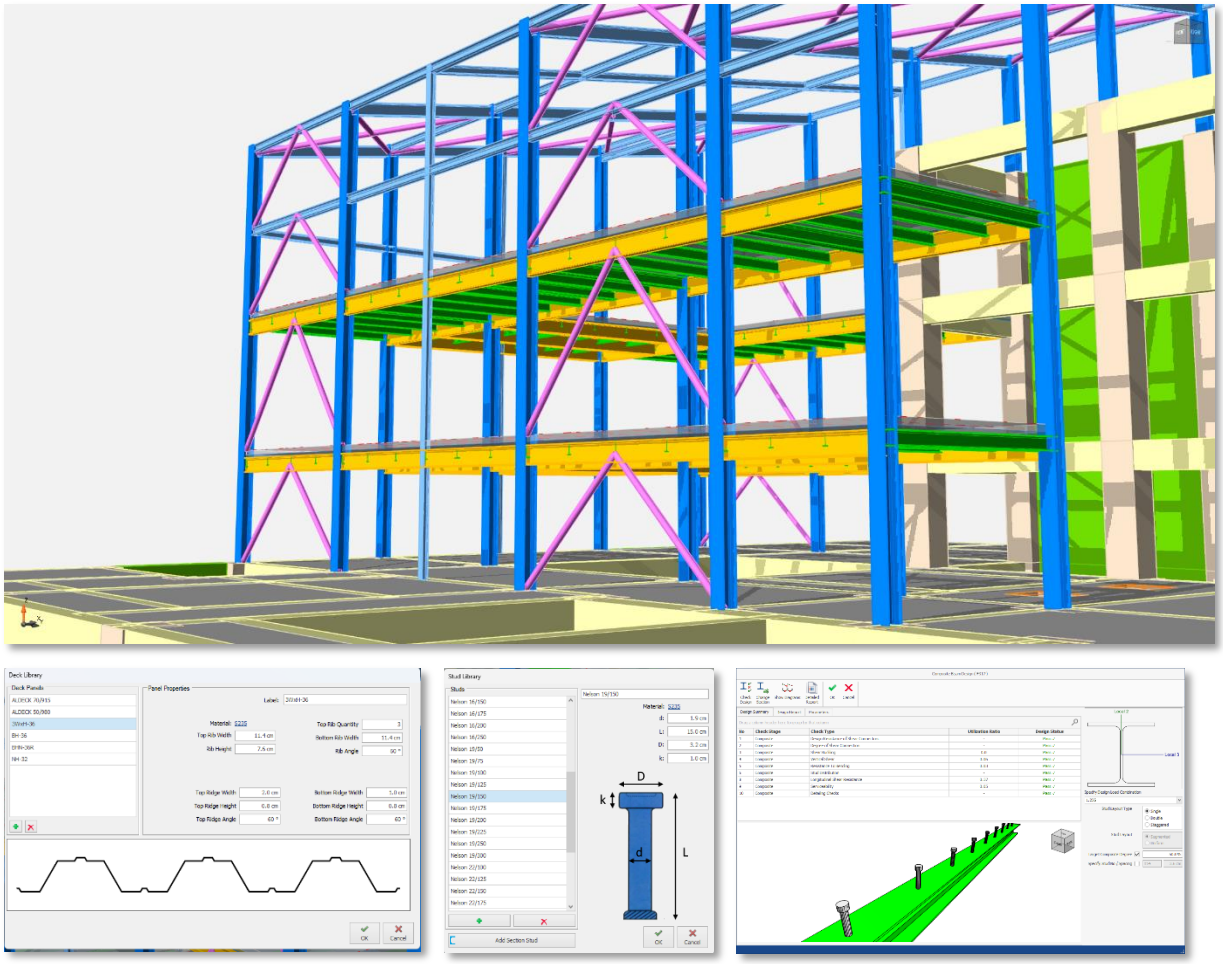
We are confident that you will enjoy and benefit from the features and improvements in ProtaStructure Suite 2024. You can find the details in the pages to follow.

Thank you for choosing ProtaStructure.

New Design Automation

Composite Slab Design

The top-rated request from our users was the ability to design composite slabs with profiled metal decks. We listened to our users, and this feature now complements the steel design solution offered in ProtaStructure.



The overview of the composite slab modeling, analysis, and design in ProtaStructure is as follows:

1. Metal decks can be chosen from a **Metal Deck Library**.
2. Shear studs can be specified from a **Stud Library**.
3. **Primary** and **Secondary Composite Frames** can be modeled freely to create any structural layout.
4. Composite Slab can be inserted on top of primary and secondary frames using flexible methods.
5. The **effective widths** of composite frames are automatically calculated, and **load distribution** is automatically done.
6. **Transformed section properties** are automatically calculated.

7. The **Construction Stage** and **Final Composite Stage** analysis are automatically performed considering construction stage imposed loads, final stage imposed loads, dead loads, etc. Different mechanical properties and end conditions are automatically considered in these analyses.

8. Steel profile checks and deflection checks are done in the construction stage.

9. The composite design is done according to a default **Composite Action Ratio**. You can change this ratio and optimize your design.

10. **Uniform** or **Segmented Stud Layout** can be used depending on the moment diagram of the member. You have complete control over the designed shear studs.

11. Checks are done for the mesh reinforcement inside the topping concrete.

12. Final deflection and vibration checks are done.

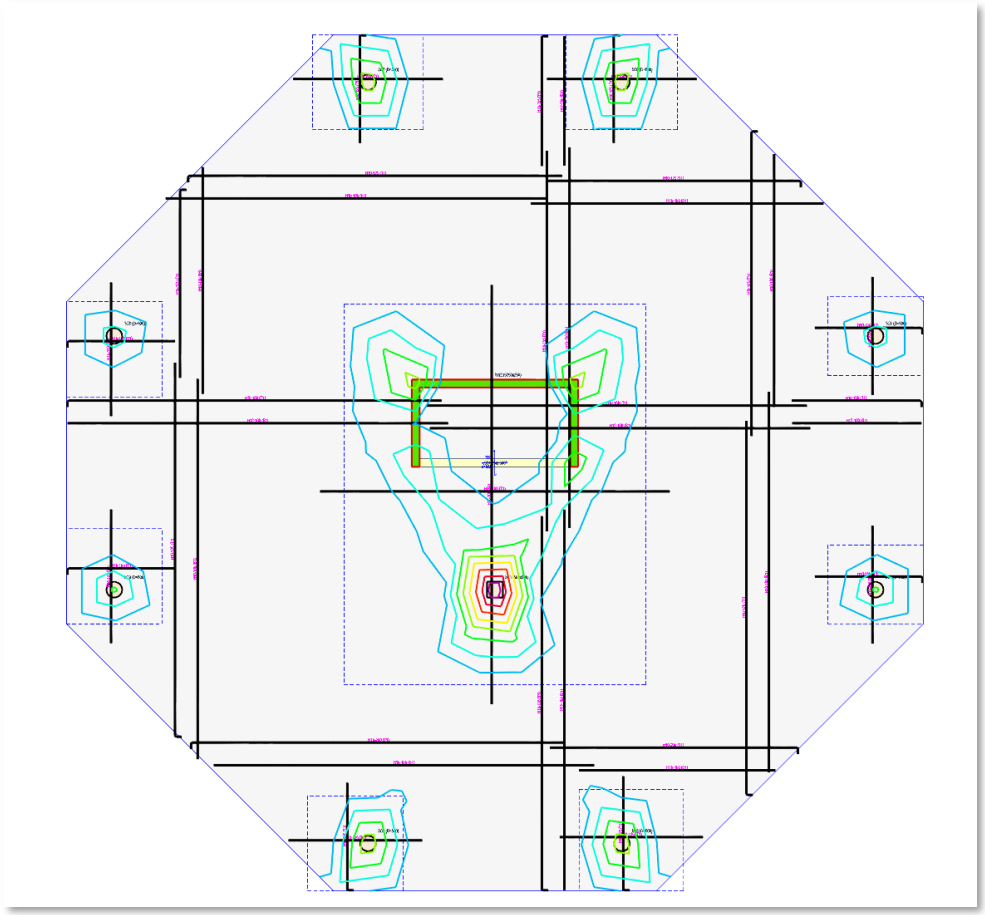
13. AISC360, Eurocode 4, and Turkish Steel Design codes are supported.

14. Composite frame sections and layout can be easily changed and optimized.

For detailed information on how to use this feature, please refer to:
"**ProtaStructure Design Guide – Composite Slab Design General Principles**".

Slab Design with Slab Patch Panels

Another most asked feature for flat slabs and raft foundations was to use a base reinforcement in the entire floor and calculate additional support bars at column regions. In **ProtaStructure 2024**, you can achieve this with the help of new **Slab Patch Panels**. See the quick peek at what the final output looks like.



Flat Slab reinforcement design with Base Reinforcements and Additional Slab Patch Panel Reinforcements. FE Contours for Direction-1 Top Reinforcement (Asd1-Top) are also displayed on the floor plan.

The New Feature: Slab Patch Panels

With **ProtaStructure 2024,** we introduced a new feature that allows our users to use a **Base Reinforcement throughout the entire floor** and design **additional top or bottom bars at column support regions**. This feature is called the **Slab Patch Panels**. The typical workflow is outlined below.

Recommended Workflow for using Slab Patch Panels

1. (Optional Step) Perform an **FE Floor Analysis** or a **Building Analysis** with the slabs meshed and examine reinforcement contours to decide on the slab patch panel areas where you will put additional rebars.

2. (Required Step) Insert **FE Fixed Band strips** for base reinforcement calculation. Fixed band strips allow you to decide and calculate the base reinforcement regions. Hence, it is important to first decide how many base reinforcement regions you will use.

3. (Required Step) Insert **slab patch panels**. Do not worry about the size of the patch panels yet. Ideally, they should cover the support moment contours. You can use an approximate size of 1.5x15 m or 2x2 m to start with. An FE Fixed Band strip should completely or partially cover slab patch panels in its tributary area.

4. (Optional Step) View the FE Reinforcement Contours on the physical floor plan view (such as $A_{sd1\text{-}top}$, $A_{sd2\text{-}top}$ for the upper floors, $A_{sd1\text{-}bot}$, and $A_{sd2\text{-}bot}$ for raft foundations). This will help you understand whether the patch sizes are large enough to cover the negative support moment regions.

5. (Required Step) Perform an **FE Floor Analysis** or a **Building Analysis** with the slabs meshed.

6. (Required Step) Design the slab reinforcements using the **Design > Slab** ribbon menu. The designed rebars will also be automatically drawn on the physical floor plan view with the correct lap splices and extensions. The additional bars in the patch panels will be automatically extended outside the designated patch area by anchorage length.

7. (Optional Step) Modify the size of slab patch panels if you need to. The most usual scenario would be to extend the patch panel region so that it covers more negative support moment contours. Optimizing the path panel size can further reduce the base reinforcement (not less than the minimum required, A_{smin})

8. (Optional Step) Redesign the reinforcement using the **Design > Slabs** ribbon menu.

9. (Optional Step) Modify the base reinforcement and/or additional reinforcements on the **Design > Slabs** ribbon menu if you need to. If the edited reinforcement is not sufficient, ProtaStructure will issue a warning.

Important Remark

Fixed band strips should completely or partially cover the slab patch panels. Otherwise, the Fixed Band strips will continue to collect bending moment results from **column nodes,** which may yield an uneconomical design. If you think a column support region does not have a significant negative moment, you have the freedom not to insert a patch panel for that column.

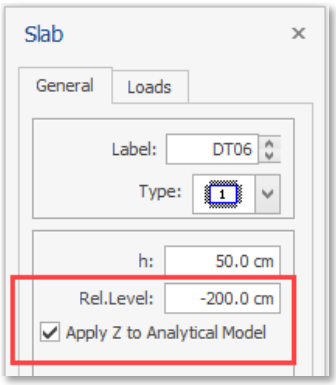
If you choose not to insert patch panels around a column, you must cover the column area with an FE Fixed Band. If you leave a particular column region uncovered by a Patch Panel or an FE Fixed Band Strip, the results in the column region will not be considered, and that particular column region will not be designed. **You may end up with an unsafe design**.

For detailed information on how to use this feature, please refer to:
"**ProtaStructure Design Guide - Slab Design with Slab Patch Panels**".

Stepped Foundations and Elevator Pits

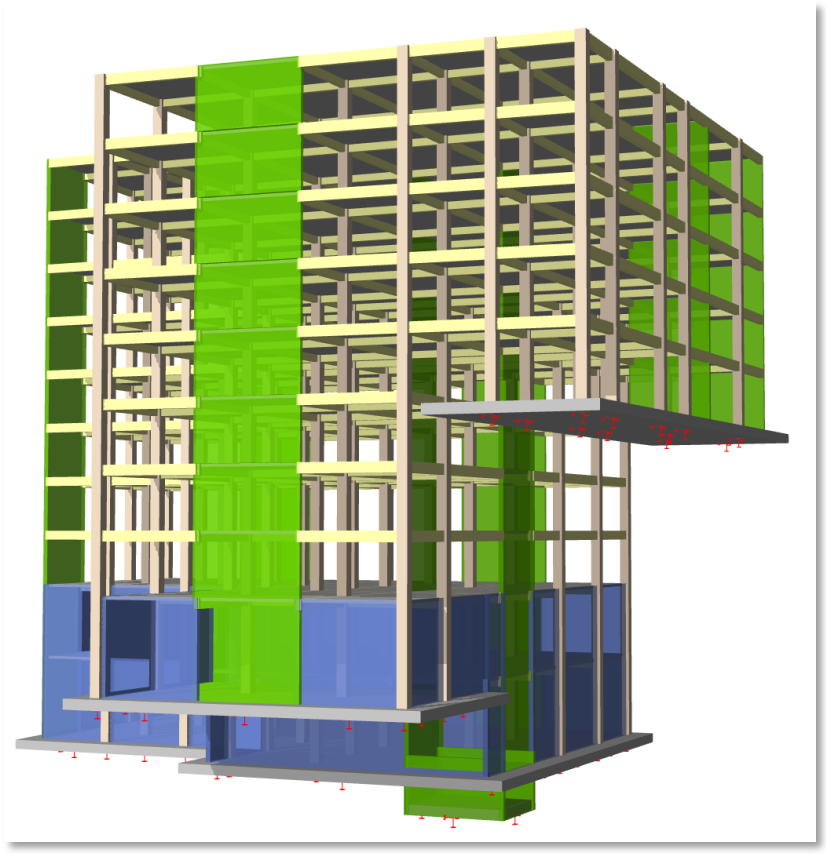
You can now model and analyze **foundations at different elevations** and **elevator pits** with ProtaStructure 2024.

1. Insert slabs in foundation storey as usual.
2. Enter the relative elevation difference in the "**Rel. Level**" field on the Slab **Properties** window. Positive values will move the slab upwards.
3. To consider the "**Rel. Level**" parameter in the analytical model, check the "**Apply Z to Analytical Model**" option.
4. You can create as many elevation differences for different slabs as you need.
5. **Elevator Pits** can also be modeled this way.



Important Tip

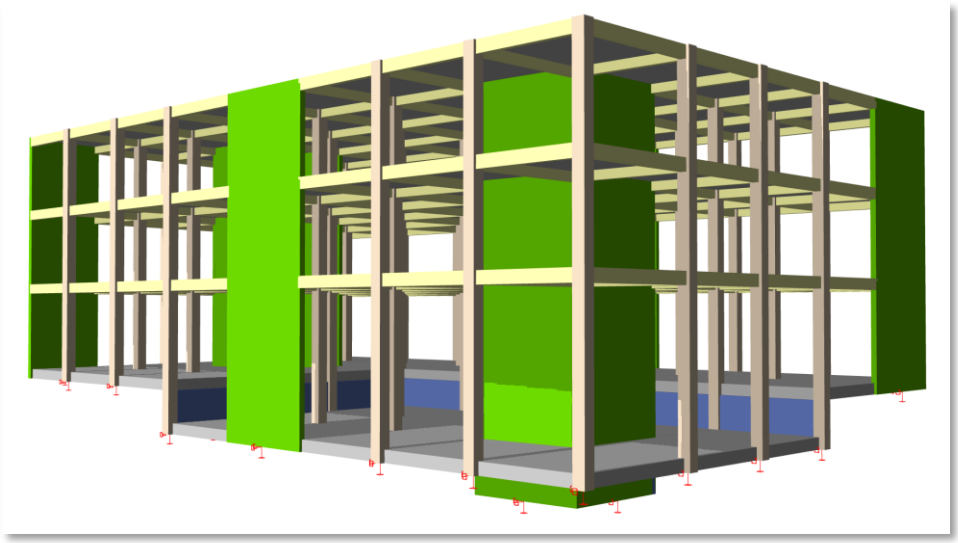
You can modify Del-Z values of shear walls in the upper storeys to create the foundation walls and elevator pit side walls. Similarly, column bottom DelZ values can be modified to match the slab elevations. Do not forget to assign supports to columns and walls that are not modeled in the first storey.

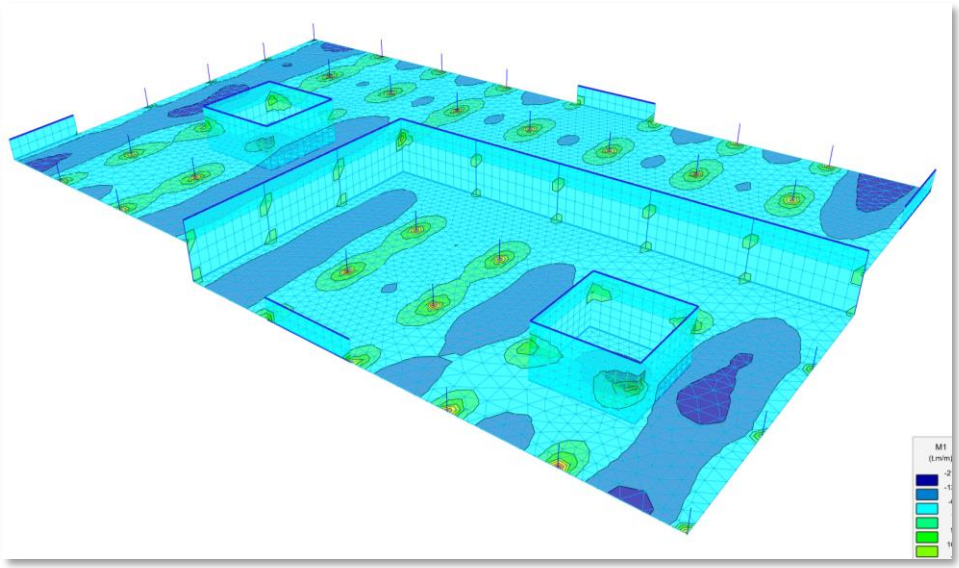


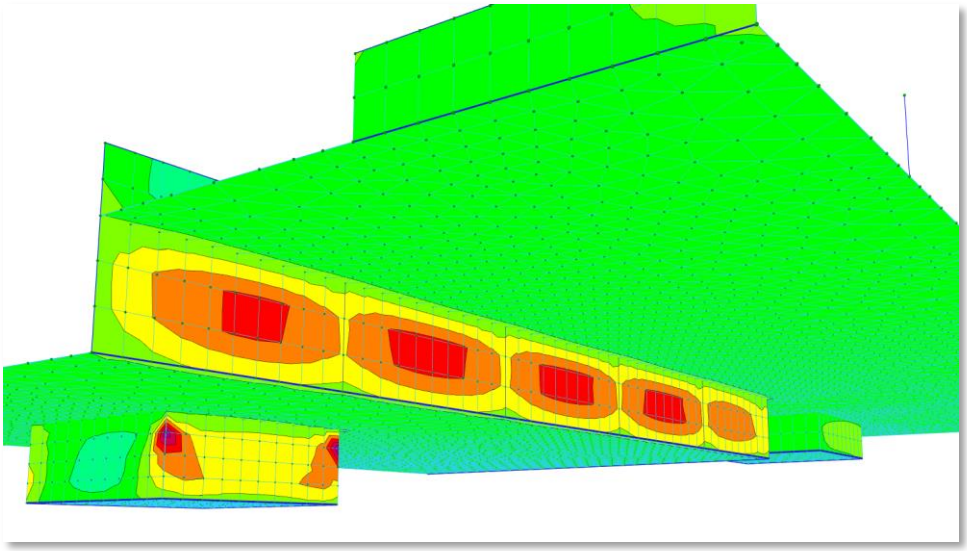
Physical model of a structure with four different foundation levels. Analytical model of the same model is also shown with foundation integrated with super structure.

Important Note

If the elevation difference in the foundation is more than a storey height, it is recommended to use "**Merged Foundation Model**" in the building analysis.

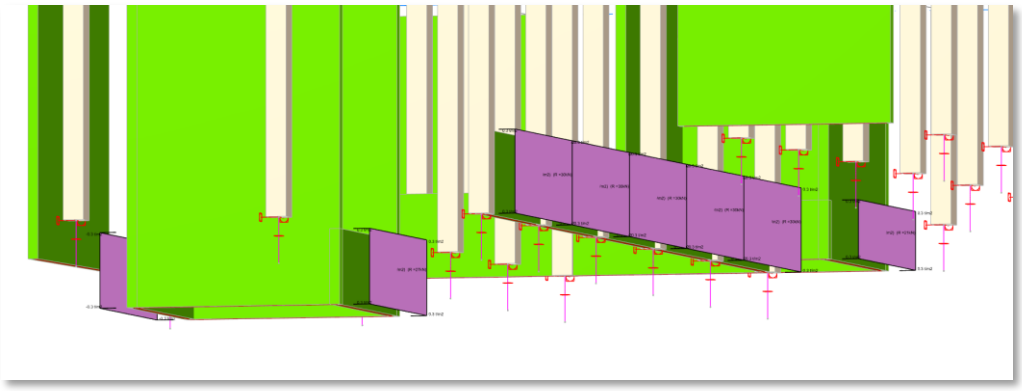






In the pictures above, you see a foundation model with an elevation difference smaller than a storey height. The slabs at the right are lowered by -200 cm relative level. Elevator pit slabs are assigned an

additional -180 cm. The blue colored foundation walls are inserted in the first storey; their top edges are lowered by -510 cm and bottom edges are lowered by -200 cm. Foundation walls and elevator pit walls are assigned horizontal area loads using the load editor.

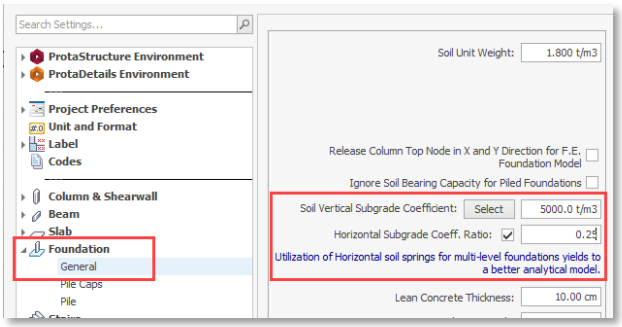


Loads can be assigned to foundation and elevator pit walls using the interactive load editor.

Horizontal Soil Springs in Raft Foundation

Horizontal soil springs may be necessary, especially for stepped foundations under vertical and lateral loads. In ProtaStructure 2024, you can now optionally use horizontal soil springs in addition to vertical ones.

Horizontal springs can be defined as a percentage of vertical springs. The default value is 25% of the vertical spring stiffness. This percentage can be changed using the **Settings > Foundation > General > Horizontal Subgrade Coefficient Ratio** field.



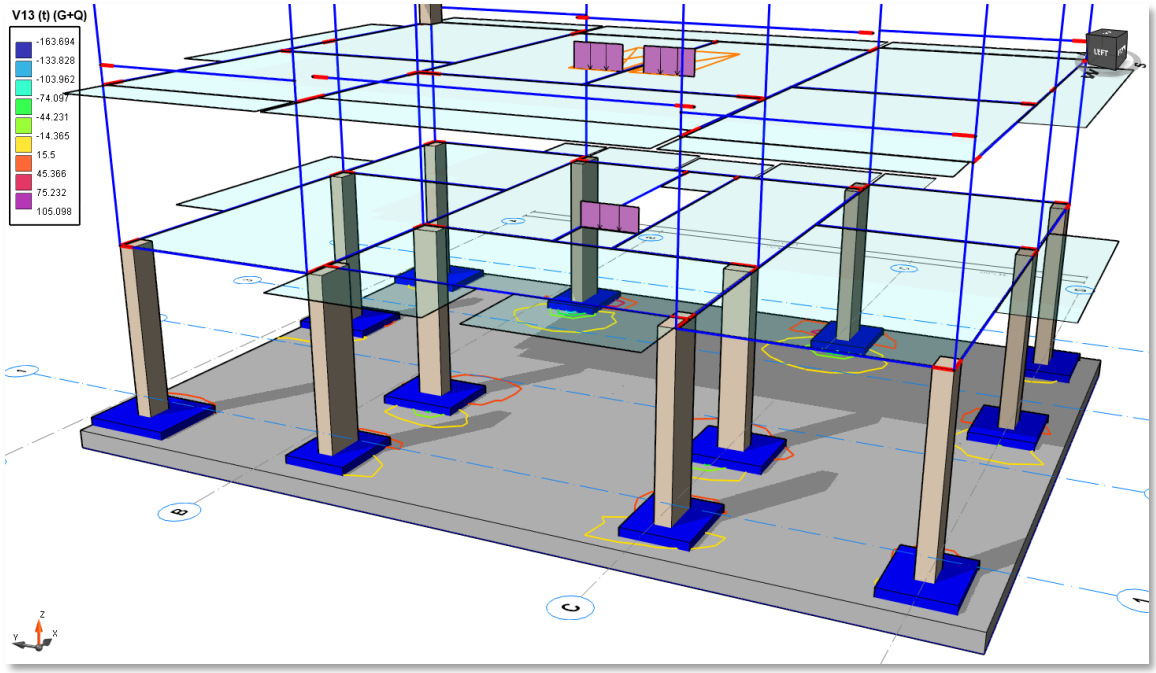
Important Note

If you want to consider horizontal springs in the foundation, it is recommended to use the "**Merged Foundation Model**" in the building analysis.

In the FE Raft Foundation Analysis module, where the building and foundation are analyzed separately, shear forces are not transferred to the foundation and in-plane degrees of freedom are restrained. So, you may not get the expected results. In the Merged Foundation Model, the lateral loads and super-structure stiffness is inherently considered in the analysis, so, usage of horizontal springs will yield more realistic results.

Column Drops in Foundation

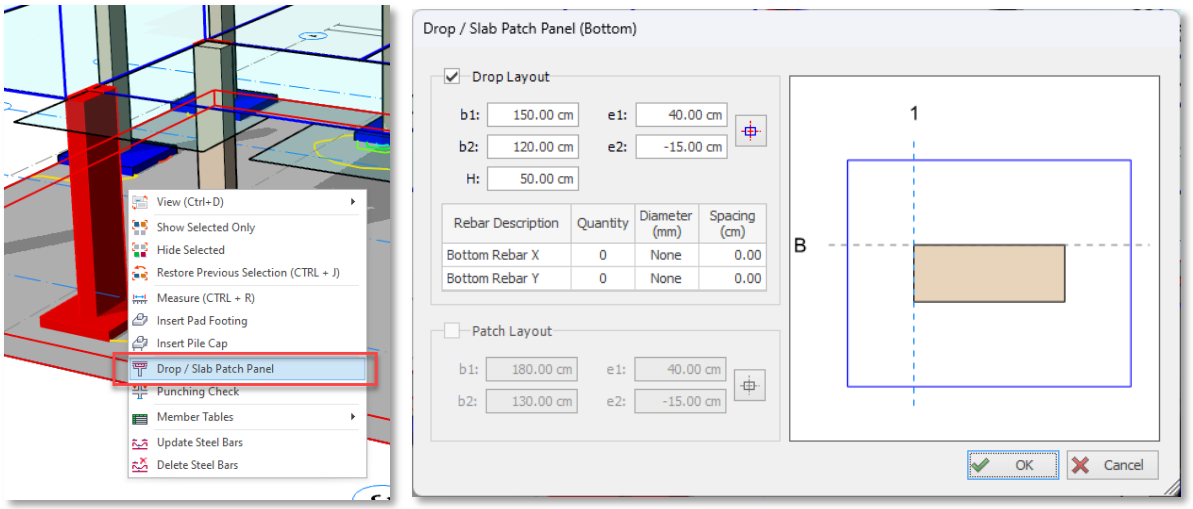
In the previous versions of ProtaStructure, it was possible only to assign drop-heads to the top of the columns. With ProtaStructure 2024, you can assign drop-heads to the bottom ends of the columns sitting on a raft foundation.



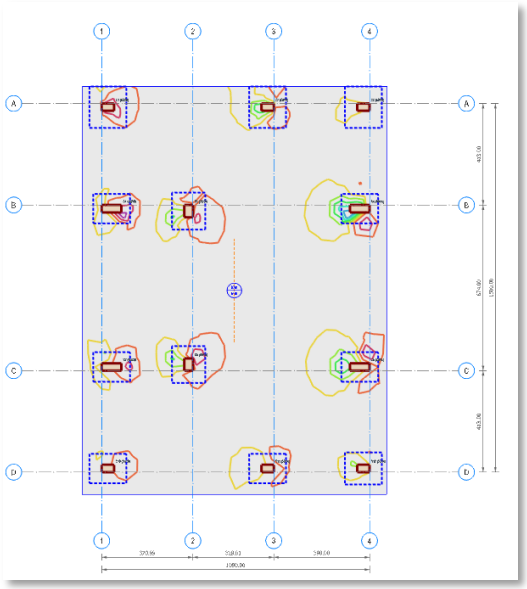
3D Physical model showing the column drops on the foundation. Upper floors are displayed in "Line View" and V13 contours are drawn on the foundation level.

To insert a drop at the bottom end of the column:

1. Switch to the foundation story level (St.00)
2. Select a column on the screen.
3. Pick the "Drop / Slab Patch Panel" command on the right-click menu.



4. The "**Drop / Slab Patch Panel (Bottom)**" window will open.
5. Check the "**Drop Layout**" button to start inserting a drop-head at the bottom of the column.
6. Enter the drop dimensions and eccentricities. A preview will be shown on the screen as you change the values.
7. Click **OK** when you are done. The drop will be shown with thick dashed lines on the plan view. 3D views will also display the drop on the physical model.



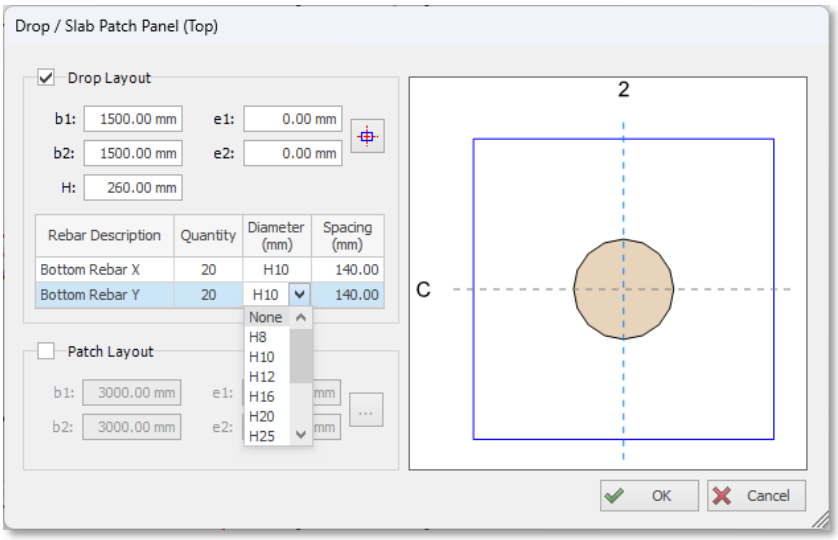
Foundation floor plan with column drops and V13 contours.

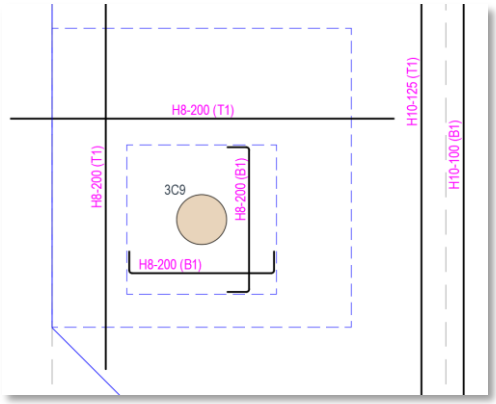
Column Drop Reinforcements

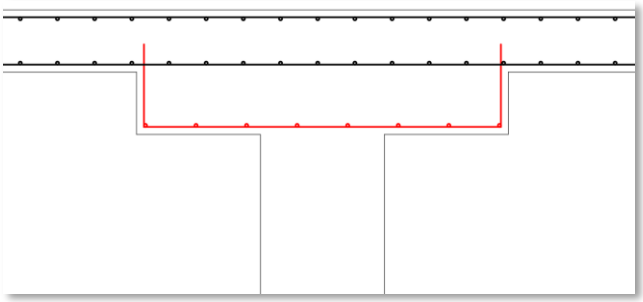
You can now manually specify reinforcements inside the column drop panels. When the drop panel is inserted into the column, the reinforcements will also be drawn automatically on the floor plan. To assign the drop reinforcements:

1. Select a column on the screen and run the "**Drop/Slab Patch Panel**" command.
2. On the drop/slab patch panel window, check the "**Drop Layout**" option and specify the drop dimensions. ProtaStructure will assign a drop panel to the column to be used in punching design.
3. As a new feature, you can now specify **drop panel bottom rebars** manually.
4. Specify the bottom **rebar quantity**, **diameter,** and **spacing** for X and Y directions. Spacing and quantity fields will mutually adjust as you enter information in any of them.
5. In the diameter list, select "**None**" to remove the drop panel rebars. Otherwise, select a diameter.

6. Click **OK** to reflect the changes to the floor plan.







Automated Rebar Layout for Complex Core Walls

ProtaStructure's **polyline column/wall editor** is powerful in the sense that:

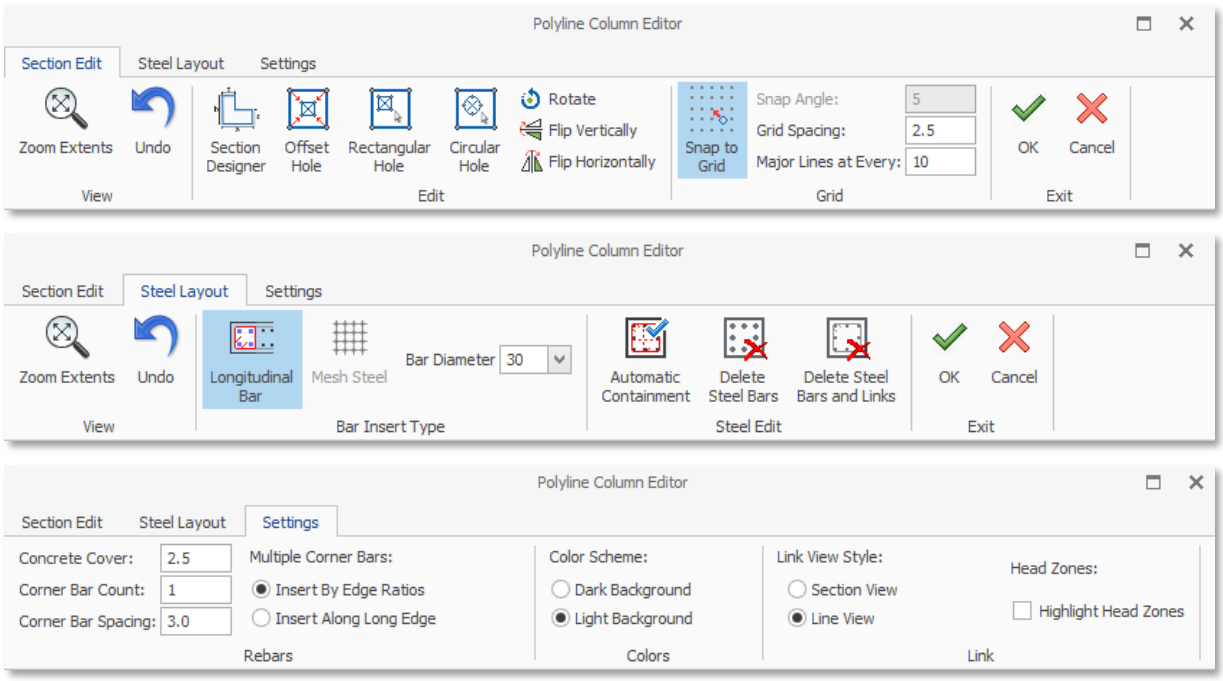
1. You can automatically lay out the rectangular or core wall reinforcements with end zones (boundary elements)
2. You can manually insert rebars, links and cross ties anywhere on the column/wall section interactively.
3. You can edit the section of a column or wall to create any irregular complex shape.

In the previous versions of ProtaStructure, the rebar layout of the core walls with known shapes was parametrically done (sections like U, I, T, E, L). For unclassified complex shapes, only the bars at the link corners were put and intermediate regions were left empty.

ProtaStructure 2024 is now capable of putting reinforcements automatically both in the end zones and intermediate regions.

Minimalistic Ribbon Toolbar

The toolbar in the polyline column/wall editor is greatly simplified to provide a seamless user experience. For rebar layout, just click the "**Auto Containment**" button, and let ProtaStructure work its magic.



Well-Known Shapes Without End Zones

ProtaStructure provides a parametric approach for placing rebars inside the well-known sections like **"I, C, H, E, T, L, +"** if end zones (boundary elements) are not used.

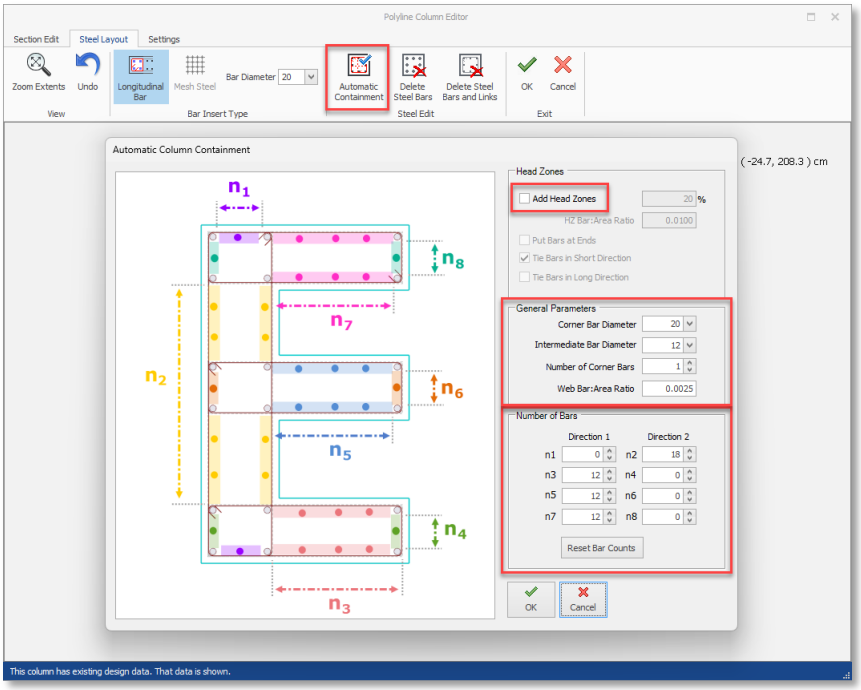
1. Click "**Auto Containment Button**". Automatic Column Containment window will be launched for known shapes.

2. Uncheck "**Add Head Zones**" option to put rebars without end zones. The "**Number of Bars**" section will be enabled. Since the shape is known, the number of bars for each segment can be explicitly specified.

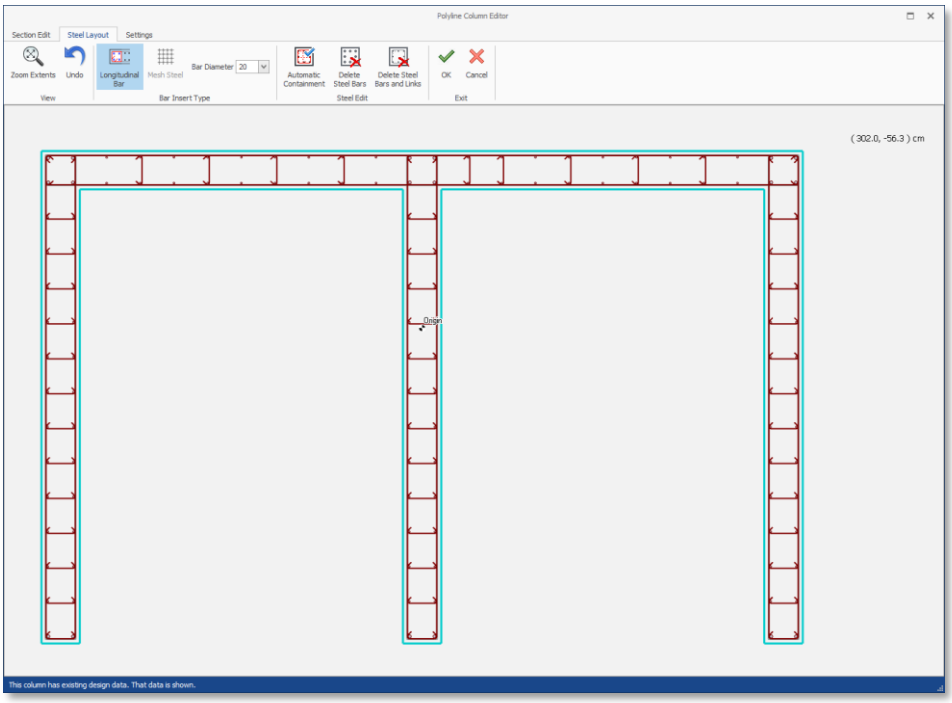
3. Specify "**Corner Bar Diameter**", "**Intermediate Bar Diameter**", and "**Number of Corner Bars**". "**Web Bar Area Ratio**" is optional.

4. Specify "**Number of Bars**" for different segments of the column/wall section for directions 1 and 2. The parameter notation is given on the left panel.

5. If you don't want to specify the number of bars explicitly, you can click the "**Reset Bar Counts**" button. ProtaStructure automatically calculates the number of bars using the "**Web Bar Area Ratio**" you have specified. It is a quick way to put bars without specifying numbers.



Polyline Column/Wall Editor with Auto Containment Window opened for known shapes.



A known E-Shaped core wall parametrically reinforced without end zones in ProtaStructure.

Well-Known Shapes with End Zones (Boundary Elements)

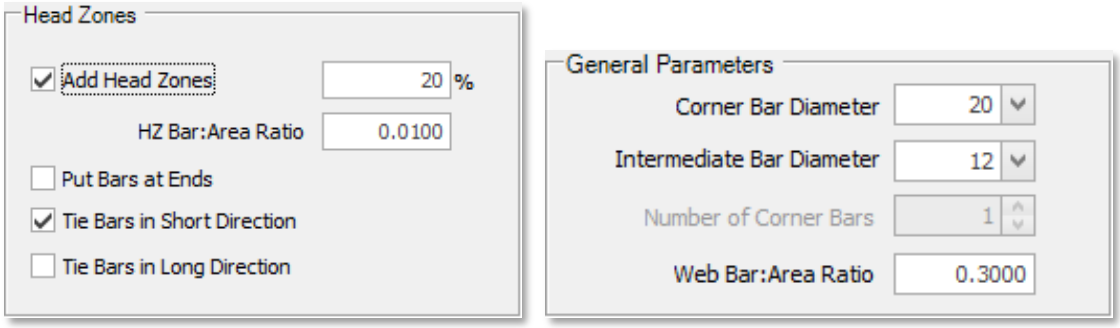
Please refer to the following title for well-known shapes with end zones. The procedure is the same as complex shapes.

Complex Shapes

In the previous versions of ProtaStructure, a shape had to be of a known type to lay out the bars automatically. As a new feature in ProtaStructure 2024, you can now automatically lay out the bars with or without end zones, even if the shape cannot be classified as a known shape.

If you want to reinforce an unknown complex shape with end zones, you must use the "**Head Zone Bar Area Ratio**" and "**Web Bar Area Ratio**" fields, rather than the "**Number of Bars**".

If you don't want end zones, uncheck the "**Add Head Zones**" option. The rest is the same.



1. Check "**Add Head Zones**" option to put rebars with end zones. The "**Number of Bars**" section will be disabled.

2. Specify the **End Zone Length** as a percentage of the wall segment length. ProtaStructure automatically identifies the wall segments and puts the end zone region using the percentage value you have specified. Wall corners and intersections are also identified, and end zones are placed accordingly. Typical value for end zone length is **20% of the segment** for the walls in critical building height (potential wall plastic region) and **10%** outside the critical height. These values change with different seismic codes. You can fine tune this value depending on your requirement.

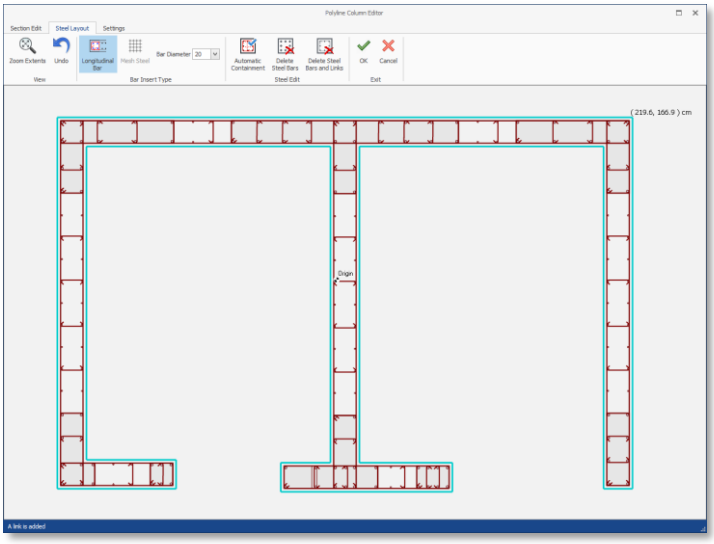
3. Specify the reinforcement ratio that will be put in the end zones using the "**Head Zone Bar Area Ratio**" field. The ratio you specify here is multiplied with the end zone area (not gross area), and the number of bars will be placed using the "**Corner Bar Diameter**" you have specified. Typical value is 1% and may change with different seismic codes.

4. Specify the "**Corner Bar Diameter**". This field controls the diameter of the bars in the end zones. If you are not using end zones, this field controls the diameter of the bars only at the link corners.

5. Specify the "**Intermediate Bar Diameter**". This field controls the diameter of the bars between in the web regions (regions between end zones).

6. "**Number of Corner Bars**" specify the number of adjacent corner bars when end zones are not used. This field is not effective if end zones are used.

7. **"Web Bar Area Ratio"** specifies the reinforcement ratio that will be put in the web regions. The ratio you specify here is multiplied with the web area of individual wall segment (whatever is left between end zones) and the number of bars will be placed using the "**Intermediate Bar Diameter**" you have specified. Typical value is **%0.25** and may change with different seismic codes.

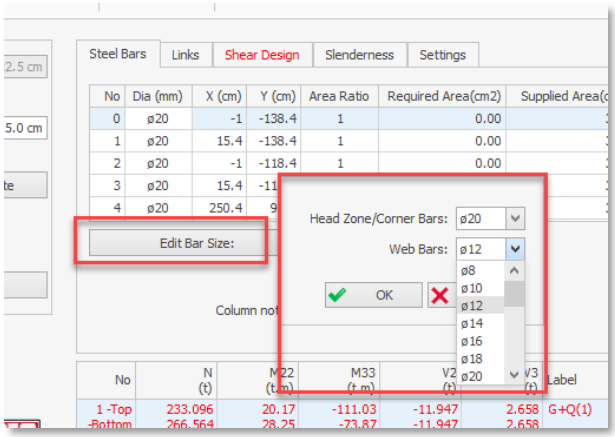


Quick Diameter Setting in Wall / Core Wall Design

After setting the reinforcement layout in polyline column/wall editor, you must proceed with design. The initially specified rebar diameters may change during the design, because only after analysis and design can it be determined whether the rebars will be sufficient or not.

Although ProtaStructure designs the column/wall rebars automatically, further optimization may be an iterative process. A large core wall section may contain dozens of rebars, and it may be difficult to change their diameters one-by-one in the design window. ProtaStructure 2024 provides a quick way to specify the end zone and web bar diameters with a few clicks. To change the head zone and web bars:

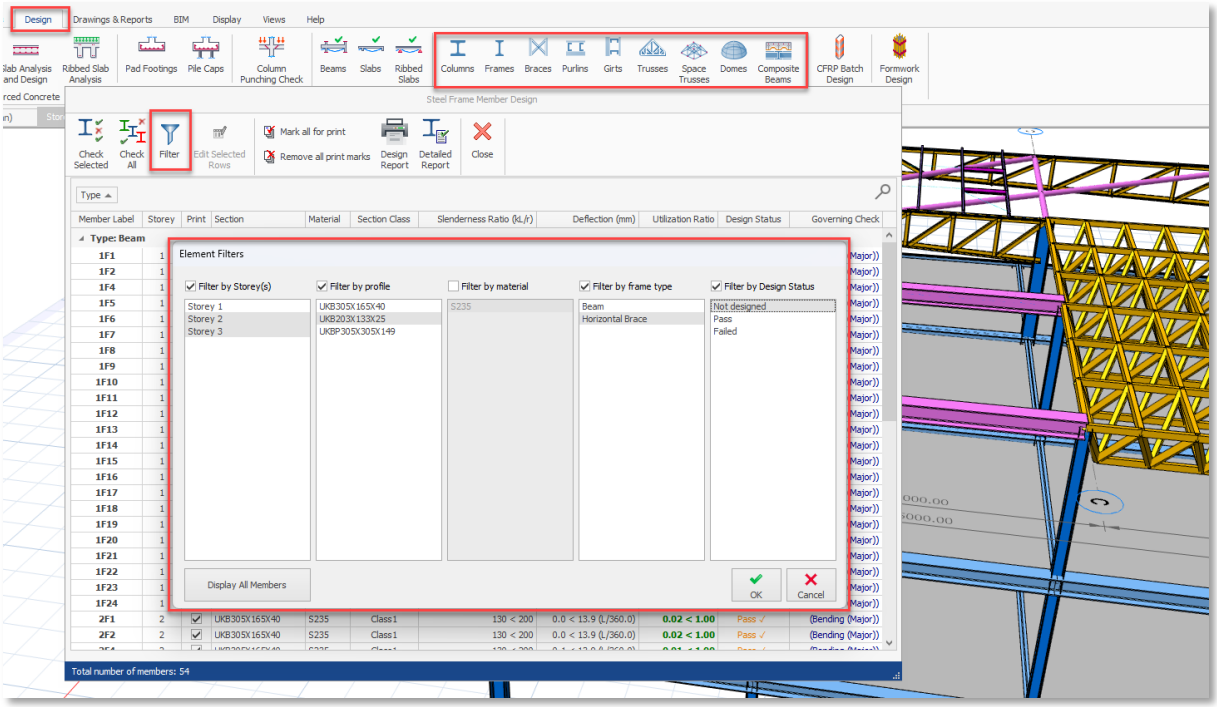
1. Click "**Edit Bar Size**" button.
2. On the pop-up panel, enter the **head zone** and **web bar** diameters and click OK.

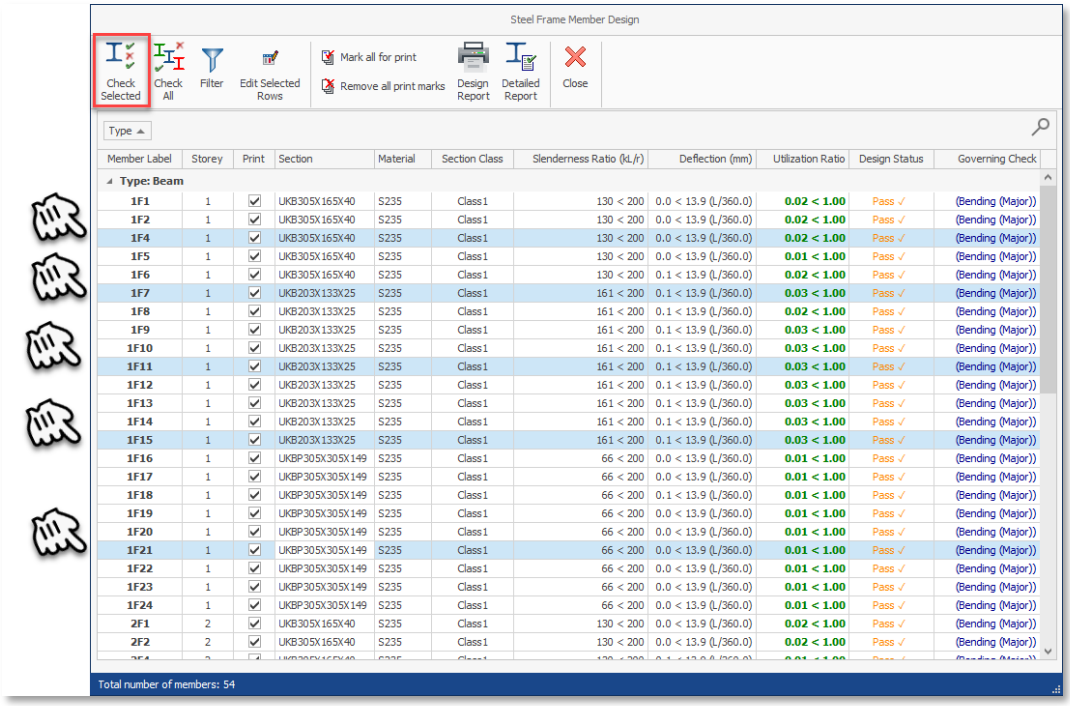


Filtering and Multi-Select in Steel Design

To increase practicality and productivity in steel design, we have developed the filtering functionality in steel design window. You can now filter the members with respect to Storey, Profile, Material, Frame Type, Design Status or their combination.

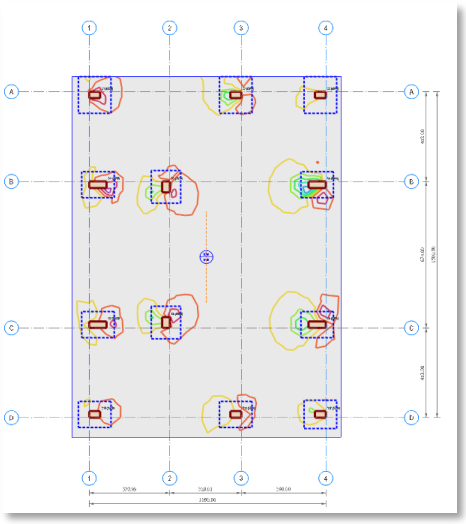
Another important improvement is the multi-selection feature in the steel design window. You can select multiple members on the table and check only them.

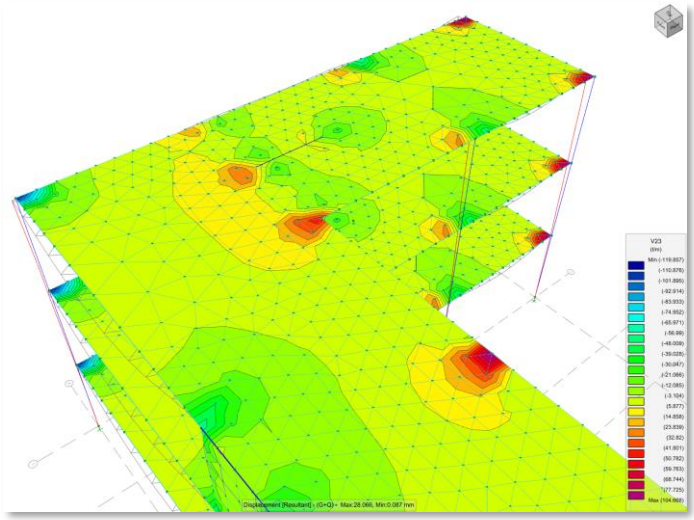




Out-of-Plane Shear Contours (V13, V23)

ProtaStructure 2024 can display the out-of-plane shear contours. You can review them in analysis post-processor and display them in floor plan if you like.

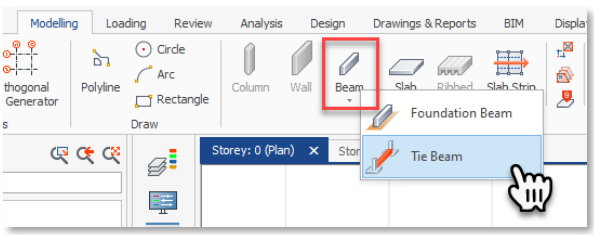


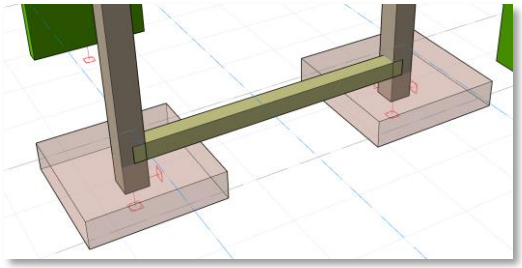


Tie Beams in Foundation

Foundation tie beams are often used to connect pile caps or pad footings to prevent relative vertical and lateral movement. The new **Foundation Tie Beams** in ProtaStructure 2024 are designed considering the maximum axial loads in connecting columns. To insert a tie beam:

1. Switch to the foundation storey (Storey 0)
2. Click "**Modeling > Beam > Tie Beam**" button on the ribbon toolbar.
3. The "**Tie Beam**" property window will be opened.
4. Specify the section size, eccentricity and the elevation information.
5. Place the beam between two columns/walls.





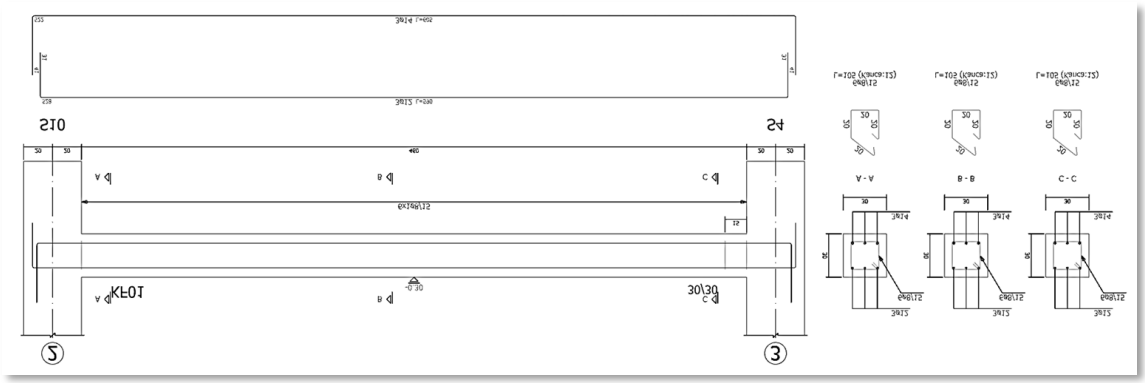
The axial force considered in the design of tie beam is calculated with:

$$N_b = 0.10 S_{DS} N_k \quad (TBDY2018 - 16.8.5 - Eqn.16.13)$$

N_b: Axial force considered in tie beam design.

S_{DS}: Short Period Spectral Acceleration Coefficient

N_k: Maximum axial load of connecting columns or walls.



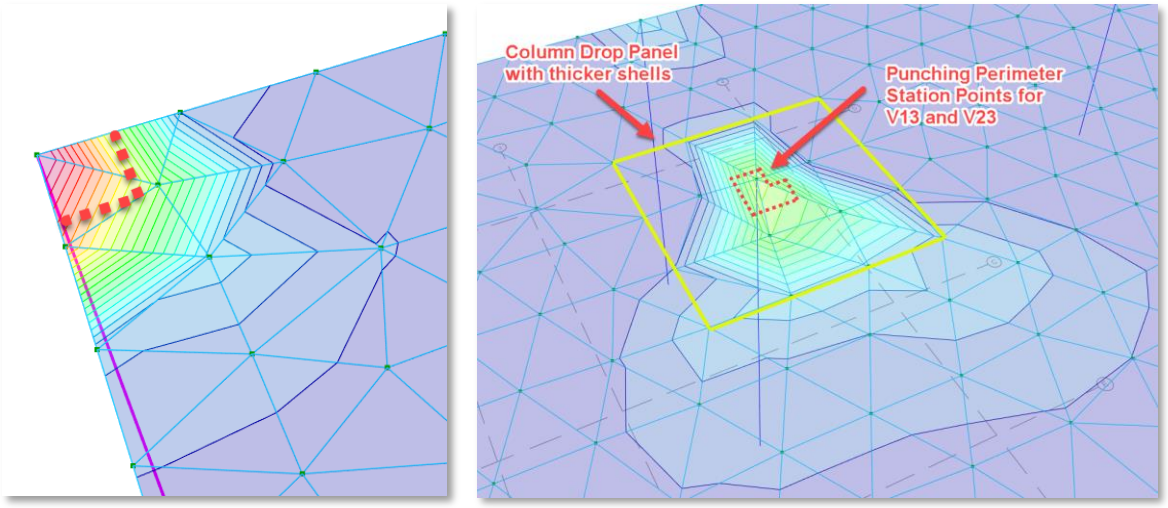
Important Remark:

Foundation Tie Beam design is currently only done to Turkish Seismic Code (TBDY 2018)

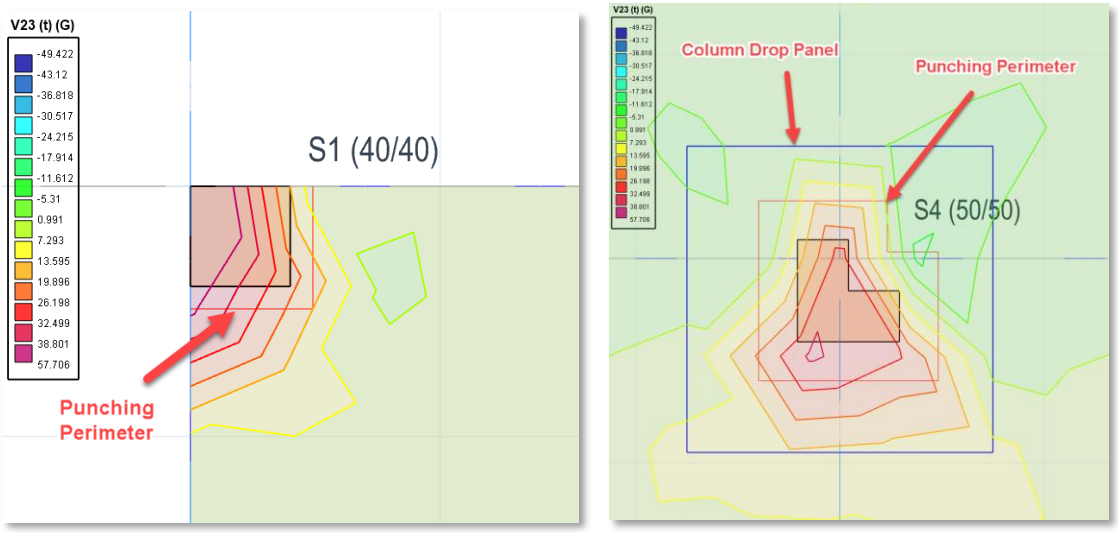
Punching Checks with FE Shell Results

ProtaStructure 2024 is now capable of performing punching checks using the maximum out-of-plane average shear forces (V13 and V23)

The maximum value of shear stress is obtained by collecting results from the punching perimeter which can be rectangular, circular or irregular depending on the column and slab geometry. Results are collected for each combination and the envelope value is used.



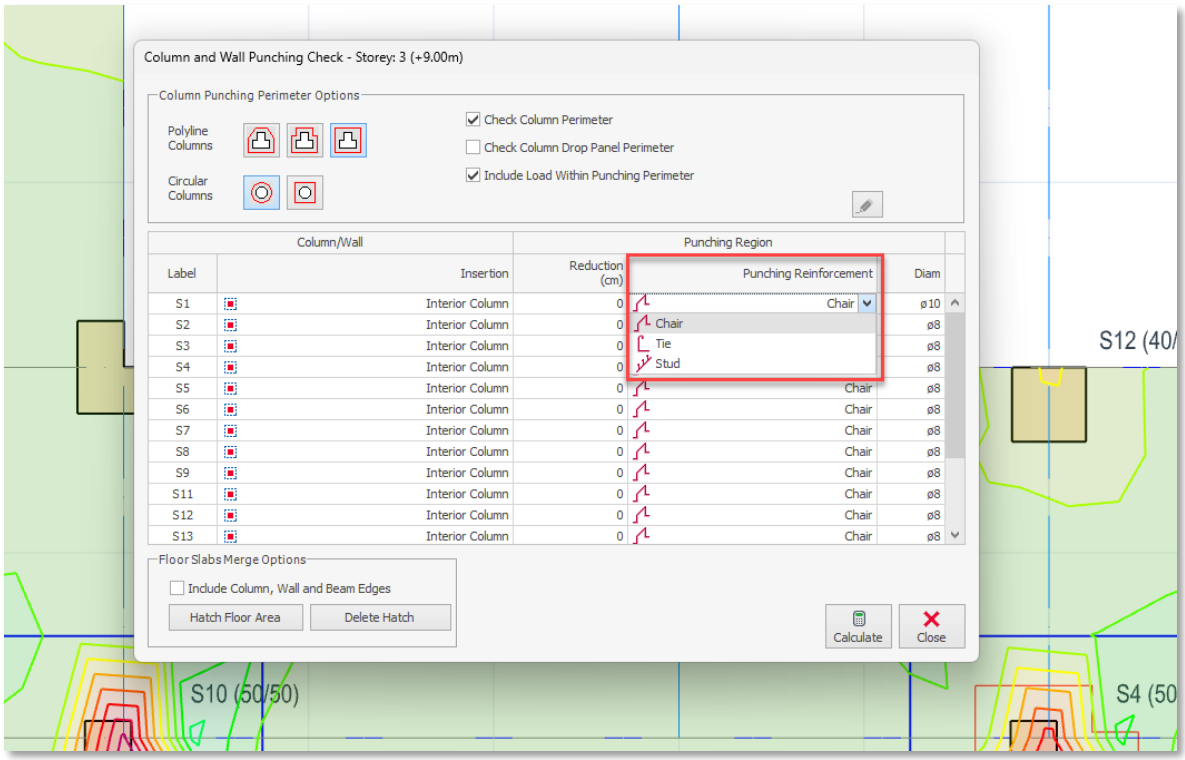
V23 shear contours and punching station points for envelope combinations in analysis post-processor.



Column drop panels, punching perimeters and shear contours shown simultaneously on physical floor plan

If the concrete tensile strength is solely enough to counteract the punching stresses around the punching perimeter, then no additional punching reinforcement is calculated.

However, if the punching stress is larger than concrete tensile strength, and the slab thickness is larger than 250 mm, then the punching reinforcement is calculated by ProtaStructure. Type of the punching reinforcement can be selected on the "**Column and Wall Punching Check**" design window.



Column: S10 (Storey: 3)

V_{13} (t)	V_{23} (t)	V_{pd} (t)	Critical Combination
62.351	61.209	62.351	7 - Gc+Qc+Ez-Ex-

Punching Check: (At d/2 from face)

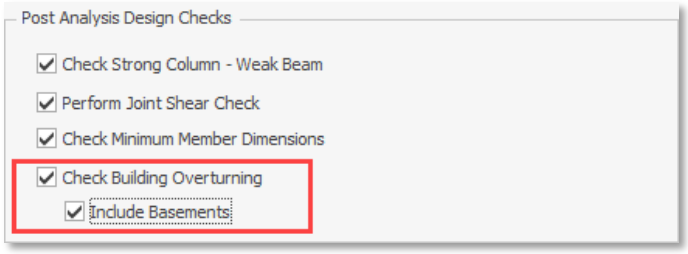
Slab: h = 40.0 d = 38.1 cm
Punching Region: Perimeter (u)= 352.4 cm,
Area (A)= 7136.61 cm^2,
Punching Area (u.d)= 13426.44 cm^2
$\tau_{pd} = V_{pd} / d = 163.650\ t/m^2 > F_{ctd} = 116.667\ t/m^2$
Min. Required Chair Reinforcement Section Area: **ρ = 28.84** cm^2/m^2
$\tau_{pr} = 0.5\ F_{ctd} + \rho\ F_{yd} = 163.650\ t/m^2 \leq 1.5\ F_{ctd} = 175.000\ t/m^2$ ✓
Chair Reinforcement Quantity per Unit Area: **28ø8**

Important Remark:

Punching checks with finite element shell results is currently done to Turkish Seismic Code (TBDY 2018). The analytical approach to other design codes is still available.

Raft Foundations and Basement Storeys in Overturning Checks

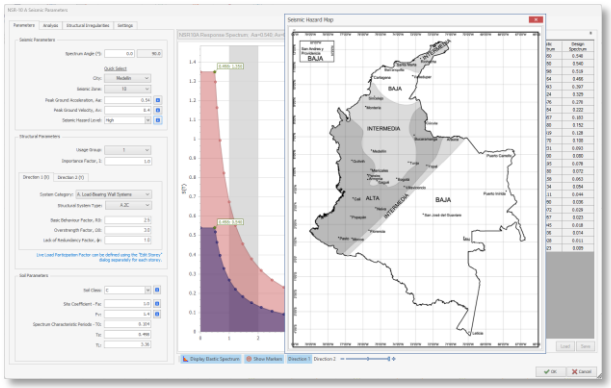
With ProtaStructure 2024, you can consider raft foundations and basement storeys in the overturning checks.

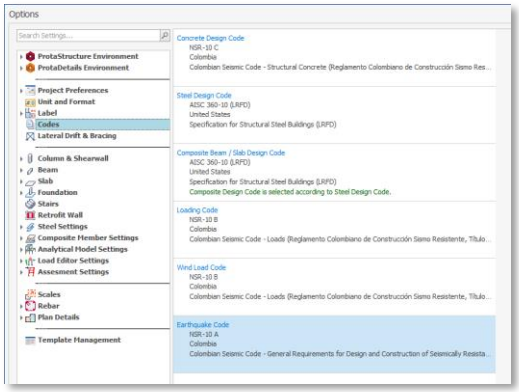


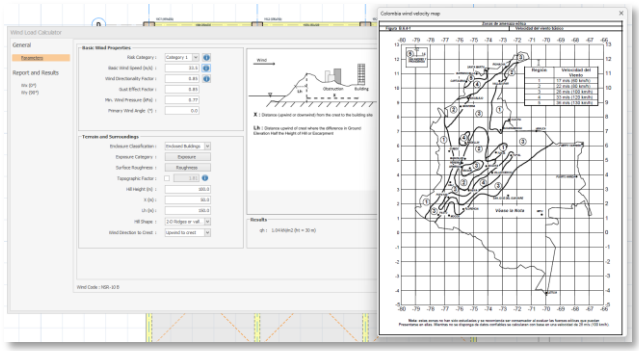
New Codes

Colombian Codes

ProtaStructure 2024 now supports **Colombian Codes** for concrete design, wind loads and seismic loads. RC beam design, wind load calculation, elastic and design earthquake spectrum generation, base shear calculations, equivalent static and response spectrum analysis, structural irregularities are included. The seismic parameters are customized according to NSR10 provisions. More to come with the upcoming releases.

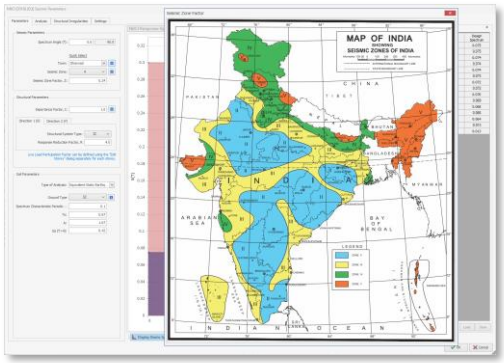


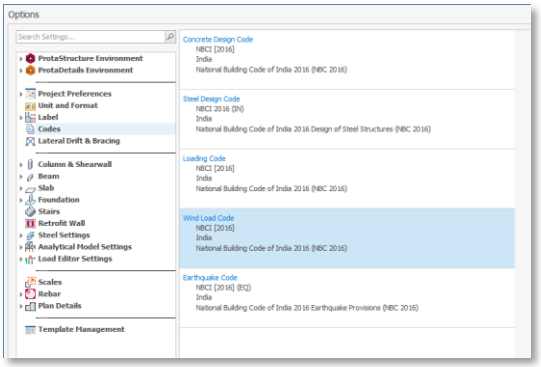




Indian Standards (IS)

With the latest ProtaStructure, the first implementation of **Indian Standards** is now included. RC member design, steel member design, loading code, wind load calculation, seismic load calculation, seismic irregularity, and building checks and detailing are supported.





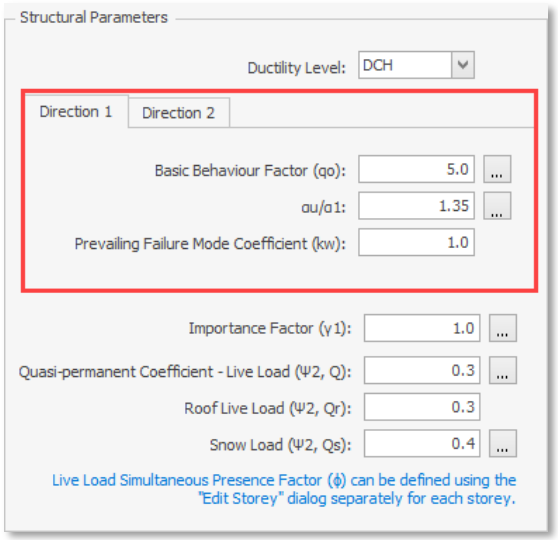
New Seismic Capabilities

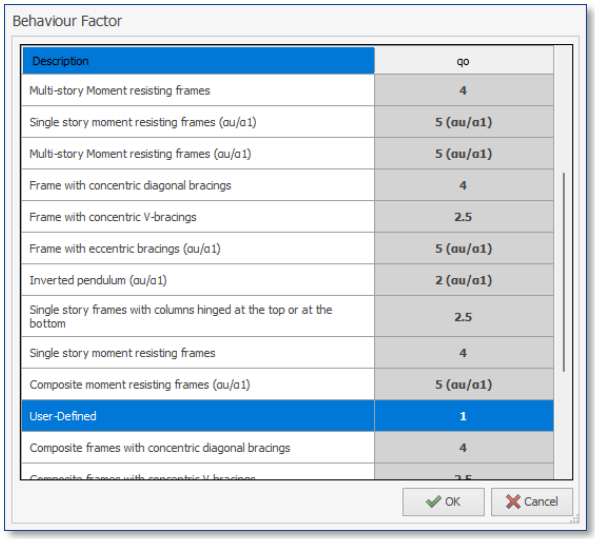
Eurocode 8: Different 'q' Factors in Two Orthogonal Directions

One of the mostly asked features was to introduce different behavior factors in two different orthogonal directions when Eurocode 8 is selected. You can now achieve this with ProtaStructure 2024. Different behavior factors and modification factors can be specified for each direction.

With the new version, torsional flexibility is automatically calculated using relative drifts (or decided by the user). However, unlike the previous versions, the behavior factor is not automatically changed anymore. The initial behavior factor, q_o, selected by the user is multiplied with factors like α_u/α_1 and **irregularity reduction factor** as necessitated by the code, and is not changed during the analysis.

If **User-Defined Structural System** is selected, the selected behavior factor is used without any manipulation.





Members not Part of Lateral Load Resisting System

In seismic design practice, seismic provisions may not be considered in the design of some members, or redistribution of seismic forces in the analytical model may be simulated. The members that are considered not to be a part of the lateral load resisting system, should be included in the seismic analysis with reduced stiffness, but they should be fully active for other load cases. They are also designed only for some of the combinations. With ProtaStructure 2024:

- You can set any member as a "**Vertical Only**" member.
- You can set some combinations as "**Vertical Only**" combinations. In this way, members tagged as "**VOM**" are designed only for those combinations.
- Members with VOM tags use reduced bending stiffness in "Seismic" load cases, while they are modeled with their full stiffness properties in other load cases. In other words, for a G+Q+E

combination, G and Q load cases use a different stiffness matrix than the E load case. (Similar to cracked stiffness properties for seismic load cases)

Designating Columns and Beams as Vertical-Only Member

You can designate selected columns and beams as '**Vertical-Only Member (VOM)**' in ProtaStructure.

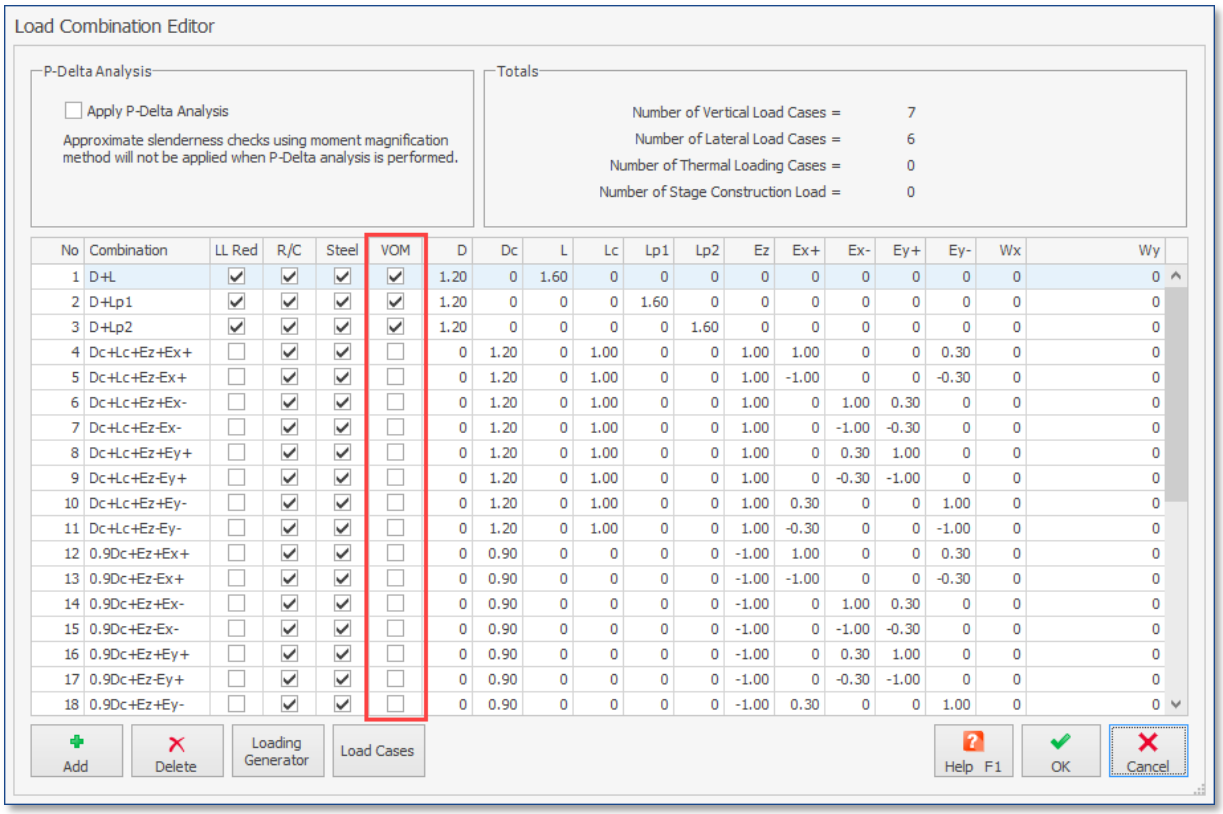
1. Select a column or a beam in modeling screen and open the Properties windows through the right-click menu.

2. Check "**Vertical-Only Member (VOM)**" option to designate the member as a vertical-only (not part of the lateral load resisting system) member.

Designating Vertical-Only Load Combinations

Designating selected members as "Vertical-Only (VOM)" is not sufficient. Idea behind this feature is designing vertical-only members only for non-seismic combinations, whereas the rest of the structure is designed for all combinations.

By default, ProtaStructure will designate only the gravity combinations as 'vertical-only'. This means that the members designated as 'vertical-only' will only use these combinations in design. You can set any combination as vertical-only by checking the VOM option next to a combination in the load combination editor.

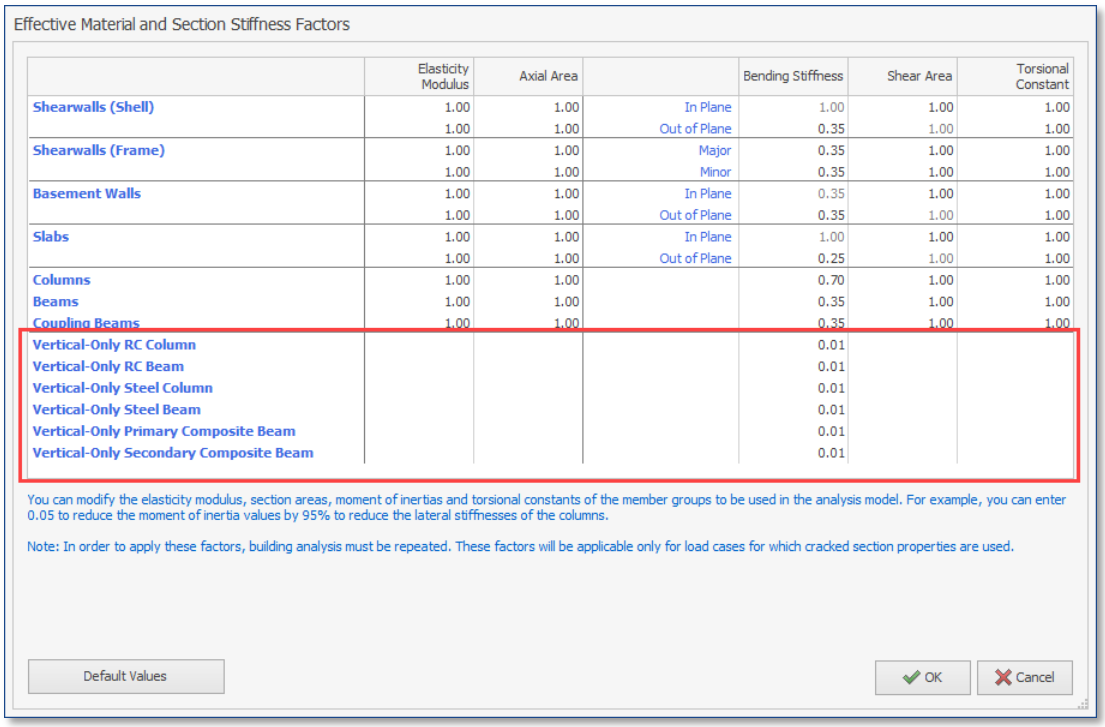


No	Combination	LL Red	R/C	Steel	VOM	D	Dc	L	Lc	Lp1	Lp2	Ez	Ex+	Ex-	Ey+	Ey-	Wx	Wy
1	D+L	☑	☑	☑	☑	1.20	0	1.60	0	0	0	0	0	0	0	0	0	0
2	D+Lp1	☑	☑	☑	☑	1.20	0	0	0	1.60	0	0	0	0	0	0	0	0
3	D+Lp2	☑	☑	☑	☑	1.20	0	0	0	0	1.60	0	0	0	0	0	0	0
4	Dc+Lc+Ez+Ex+	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	1.00	0	0	0.30	0	0
5	Dc+Lc+Ez-Ex+	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	-1.00	0	0	-0.30	0	0
6	Dc+Lc+Ez+Ex-	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	0	1.00	0.30	0	0	0
7	Dc+Lc+Ez-Ex-	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	0	-1.00	-0.30	0	0	0
8	Dc+Lc+Ez+Ey+	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	0	0.30	1.00	0	0	0
9	Dc+Lc+Ez-Ey+	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	0	-0.30	-1.00	0	0	0
10	Dc+Lc+Ez+Ey-	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	0.30	0	0	1.00	0	0
11	Dc+Lc+Ez-Ey-	☐	☑	☑	☐	0	1.20	0	1.00	0	0	1.00	-0.30	0	0	-1.00	0	0
12	0.9Dc+Ez+Ex+	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	1.00	0	0	0.30	0	0
13	0.9Dc+Ez-Ex+	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	-1.00	0	0	-0.30	0	0
14	0.9Dc+Ez+Ex-	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	0	1.00	0.30	0	0	0
15	0.9Dc+Ez-Ex-	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	0	-1.00	-0.30	0	0	0
16	0.9Dc+Ez+Ey+	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	0	0.30	1.00	0	0	0
17	0.9Dc+Ez-Ey+	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	0	-0.30	-1.00	0	0	0
18	0.9Dc+Ez+Ey-	☐	☑	☑	☐	0	0.90	0	0	0	0	-1.00	0.30	0	0	1.00	0	0

Stiffness Modifiers for Vertical-Only Members

Bending stiffness of Vertical-Only members should not contribute to structural behavior for seismic load cases. However, they should contribute with their full bending stiffness to other load cases. This is possible, thanks to ProtaStructure's **Powerful Stiffness Modification System per Load Case**. This feature in ProtaStructure can utilize different stiffness matrices for different load cases in a single analysis run.

By default, ProtaStructure reduces bending stiffness of vertical-only members by **0.01** for seismic load cases. You can modify this value to suit your needs on "**Effective Material and Section Stiffness Factors**" table.



User-defined Periods in EQS Analysis

To determine the natural modes of vibration, ProtaStructure automatically performs a 3D eigenvalue analysis. After the vibration modes are detected, they are sorted with respect to their mass participation in three degrees of freedom, namely, **Translation X**, **Translation Y** and **Rotation Z**. The dominant modes in X and Y direction are selected as candidate modes to be used in equivalent static earthquake load method.

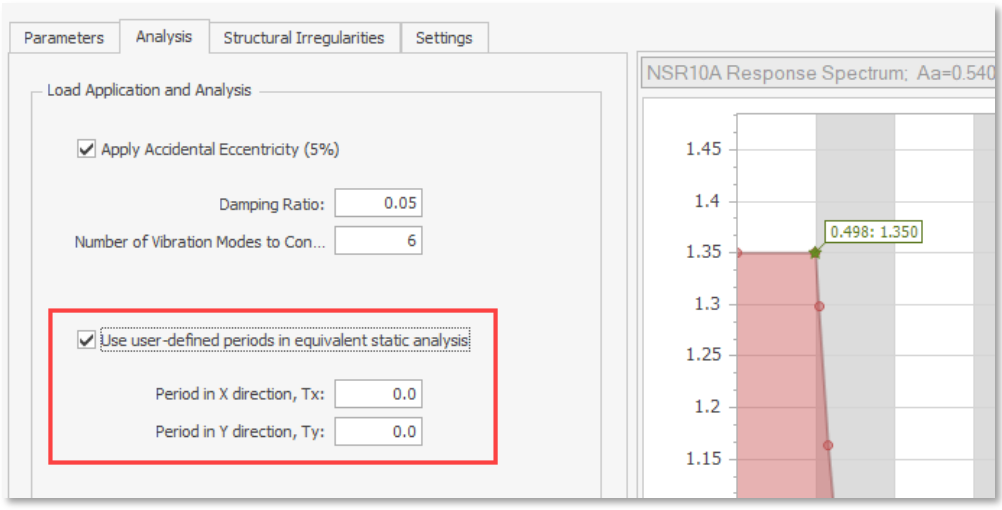
However, the period of vibration from these modes can not be directly used in EQS method. Seismic codes always put an upper limit to these values by empirical formulas. The most conditions are:

1. $T_{eigen} \leq$ Rayleigh formula
2. $T_{eigen} \leq C_u.C_t.h_n^x$
3. $T_{eigen} \leq 0.1N$

ProtaStructure automatically checks these conditions and their variation depending on the selected seismic code.

With ProtaStructure 2024, we allowed our users to directly impose user-defined periods to be used EQS analysis. To enter user-defined periods:

1. Navigate to "**Seismic Parameters > Analysis**" tab.
2. Check "**Use User-Defined Periods in Equivalent Static Analysis**" option.
3. Enter period values in X and Y direction.
4. Exit the window by clicking OK button.



Important Note

The user-defined periods will be directly used in equivalent static base shear calculation without applying any upper limit restrictions.

Transparency and Quality in Design

Step-by-Step Detailed Design Reports

We consider it highly important and prioritize building more confidence in our users' designs. That's why in ProtaStructure 2024, we provide you with reports that include **formulas, code references, and detailed calculation steps**. In fact, we provided the first example of this with Pad Foundations and Pile caps in the previous version. Now, we are taking this even further in ProtaStructure 2024 and will continue to expand this scope with every update and new version.

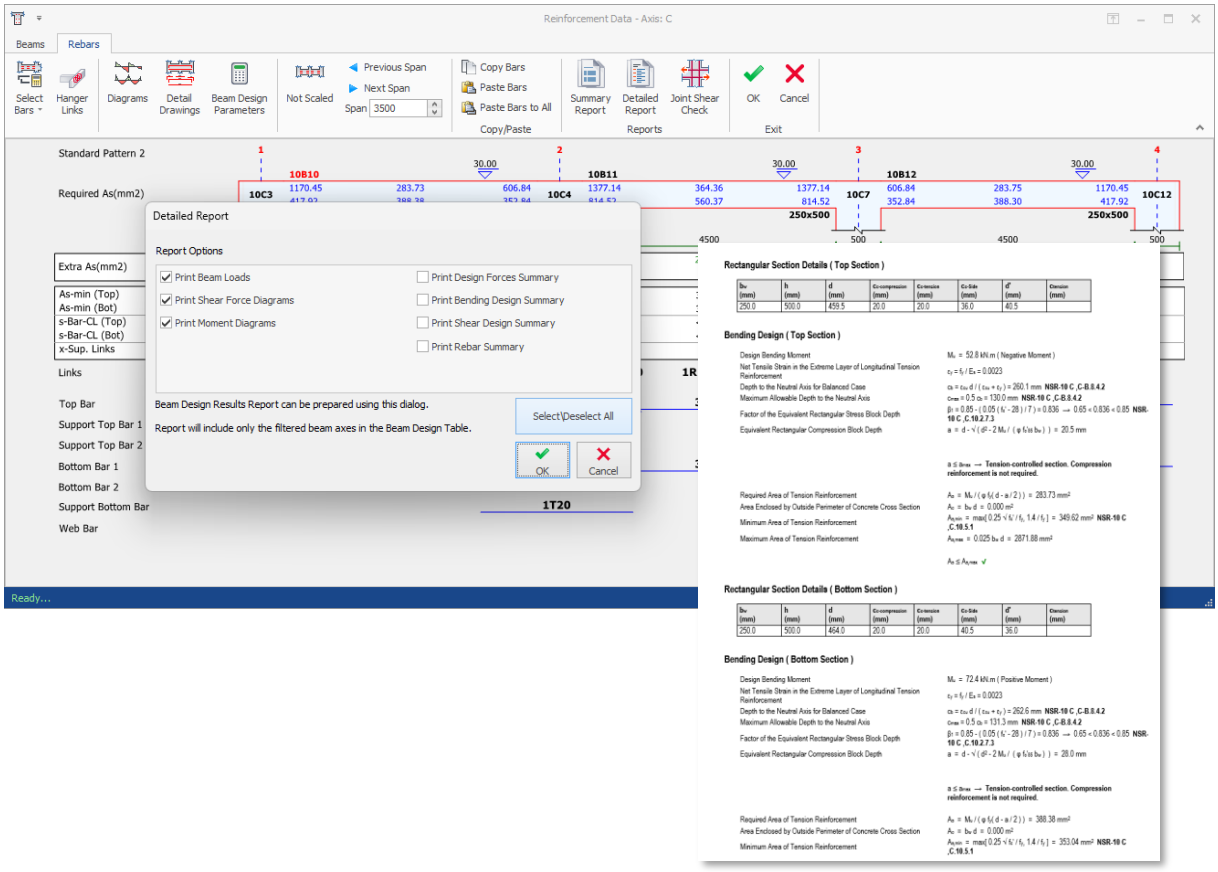
Summary Report vs. Detailed Report

When delivering your project, you can continue to use existing summary reports to reduce the number of printed outputs. However, for the check of certain structural member, we have also added Detailed Report options to the design menus to allow you to obtain detailed reports on an member-by-member basis.

Detailed RC Beam Design Reports

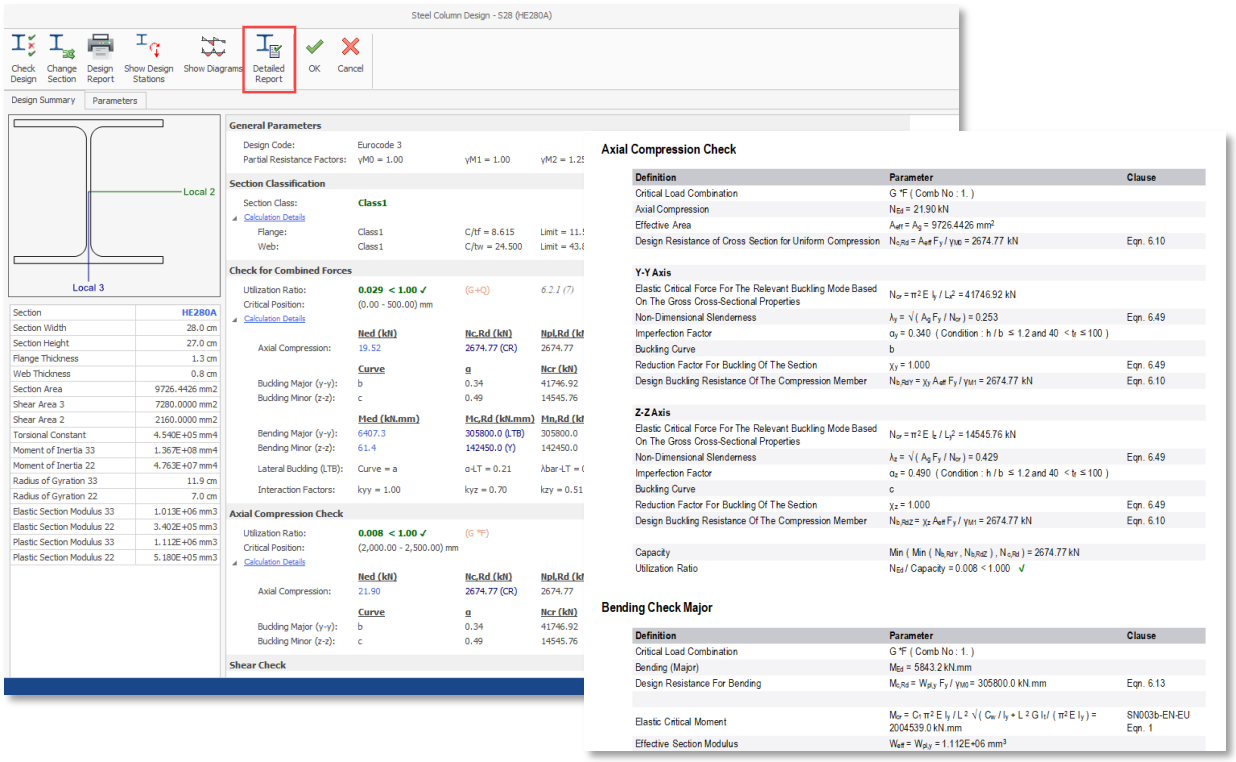
The design reports for reinforced concrete beams now include formulas, more detailed calculation steps, and code references. This feature was intended to be released in the previous version but has been published after comprehensive testing with ProtaStructure 2024.

Our detailed design guides for reinforced concrete beam design have been published with the new version.



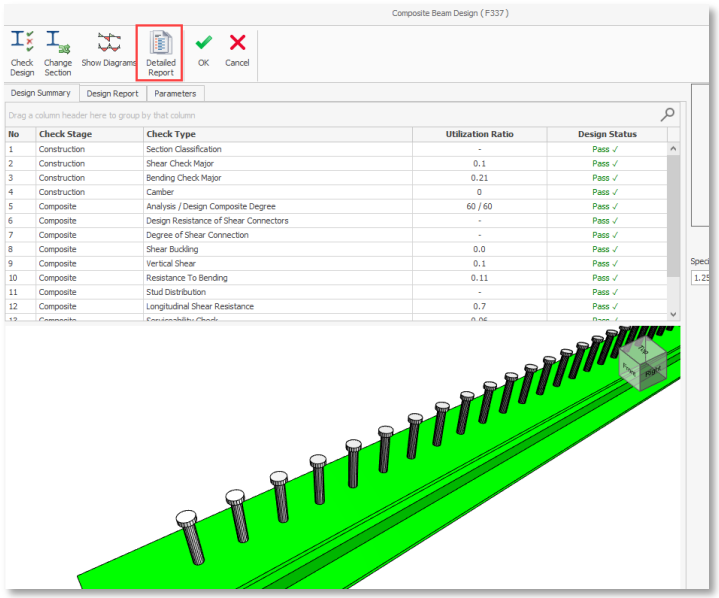
Detailed Steel Design Reports

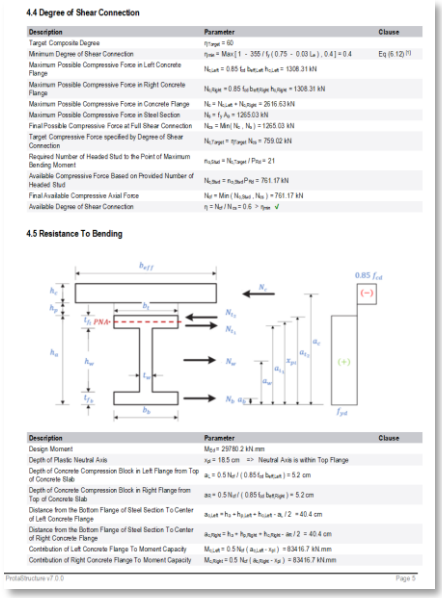
Detailed reports can also be obtained for steel member design. Summary report is still available as a separate option.



Detailed Composite Beam Design Report

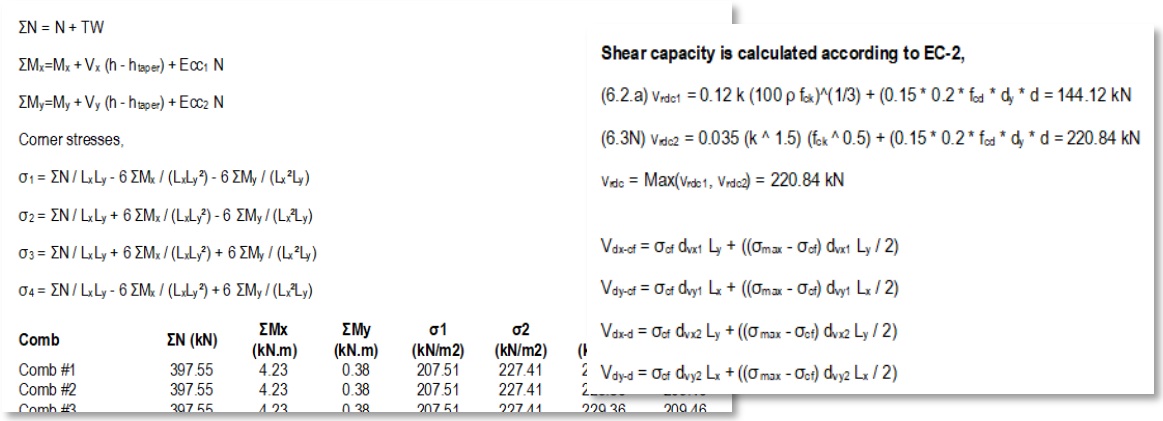
The Composite Beam Design, first introduced in ProtaStructure 2024, results in a detailed report that includes step-by-step checks for construction and service stages.





Detailed Pad Footing and Pile Cap Design Report

The design reports for pad footings and pile foundations now include formulas, more detailed calculation steps, and regulatory provisions. This feature has been added in ProtaStructure 2022.



$\Sigma N = N + TW$

$\Sigma M_x = M_x + V_x\,(h - h_{taper}) + E_{cc_1}\,N$

$\Sigma M_y = M_y + V_y\,(h - h_{taper}) + E_{cc_2}\,N$

Corner stresses,

$\sigma_1 = \Sigma N / L_x L_y - 6\,\Sigma M_x / (L_x L_y^2) - 6\,\Sigma M_y / (L_x^2 L_y)$

$\sigma_2 = \Sigma N / L_x L_y + 6\,\Sigma M_x / (L_x L_y^2) - 6\,\Sigma M_y / (L_x^2 L_y)$

$\sigma_3 = \Sigma N / L_x L_y + 6\,\Sigma M_x / (L_x L_y^2) + 6\,\Sigma M_y / (L_x^2 L_y)$

$\sigma_4 = \Sigma N / L_x L_y - 6\,\Sigma M_x / (L_x L_y^2) + 6\,\Sigma M_y / (L_x^2 L_y)$

Comb	ΣN (kN)	ΣMx (kN.m)	ΣMy (kN.m)	σ1 (kN/m2)	σ2 (kN/m2)
Comb #1	397.55	4.23	0.38	207.51	227.41
Comb #2	397.55	4.23	0.38	207.51	227.41
Comb #3	397.55	4.23	0.38	207.51	227.41

Shear capacity is calculated according to EC-2,

(6.2.a) $v_{rdc1} = 0.12\ k\ (100\ \rho\ f_{ck})^{(1/3)} + (0.15 * 0.2 * f_{cd} * d_y * d = 144.12$ kN

(6.3N) $v_{rdc2} = 0.035\ (k^{1.5})\ (f_{ck}^{0.5}) + (0.15 * 0.2 * f_{cd} * d_y * d = 220.84$ kN

$v_{rdc} = \text{Max}(v_{rdc1}, v_{rdc2}) = 220.84$ kN

$V_{dx\text{-}cf} = \sigma_{cf}\ d_{vx1}\ L_y + ((\sigma_{max} - \sigma_{cf})\ d_{vx1}\ L_y / 2)$

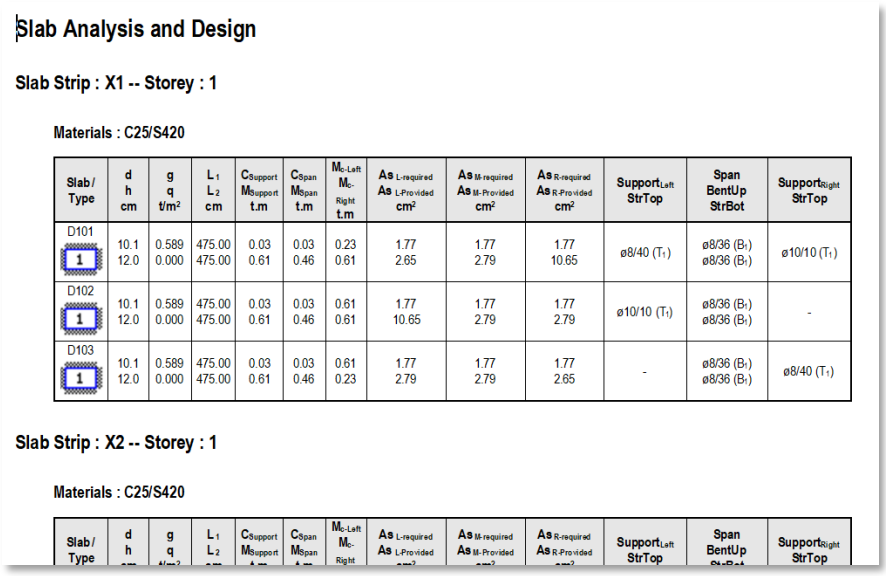
$V_{dy\text{-}cf} = \sigma_{cf}\ d_{vy1}\ L_x + ((\sigma_{max} - \sigma_{cf})\ d_{vy1}\ L_x / 2)$

$V_{dx\text{-}d} = \sigma_{cf}\ d_{vx2}\ L_y + ((\sigma_{max} - \sigma_{cf})\ d_{vx2}\ L_y / 2)$

$V_{dy\text{-}d} = \sigma_{cf}\ d_{vy2}\ L_x + ((\sigma_{max} - \sigma_{cf})\ d_{vy2}\ L_x / 2)$

The New Slab Design Report

We have modernized the slab design report and decorated it with visually appealing tables and intelligent notifications. Slab design reports are now more comprehensive and automatically created in batch mode for selected slab strips.



Slab Analysis and Design

Slab Strip : X1 -- Storey : 1

Materials : C25/S420

Slab / Type	d h cm	g q t/m²	L1 L2 cm	CSupport MSupport t.m	CSpan MSpan t.m	Mc-Left Mc-Right t.m	As L-required As L-Provided cm²	As M-required As M-Provided cm²	As R-required As R-Provided cm²	SupportLeft StrTop	Span BentUp StrBot	SupportRight StrTop
D101 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.23 0.61	1.77 2.65	1.77 2.79	1.77 10.65	ø8/40 (T1)	ø8/36 (B1) ø8/36 (B1)	ø10/10 (T1)
D102 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.61 0.61	1.77 10.65	1.77 2.79	1.77 2.79	ø10/10 (T1)	ø8/36 (B1) ø8/36 (B1)	-
D103 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.61 0.23	1.77 2.79	1.77 2.79	1.77 2.65	-	ø8/36 (B1) ø8/36 (B1)	ø8/40 (T1)

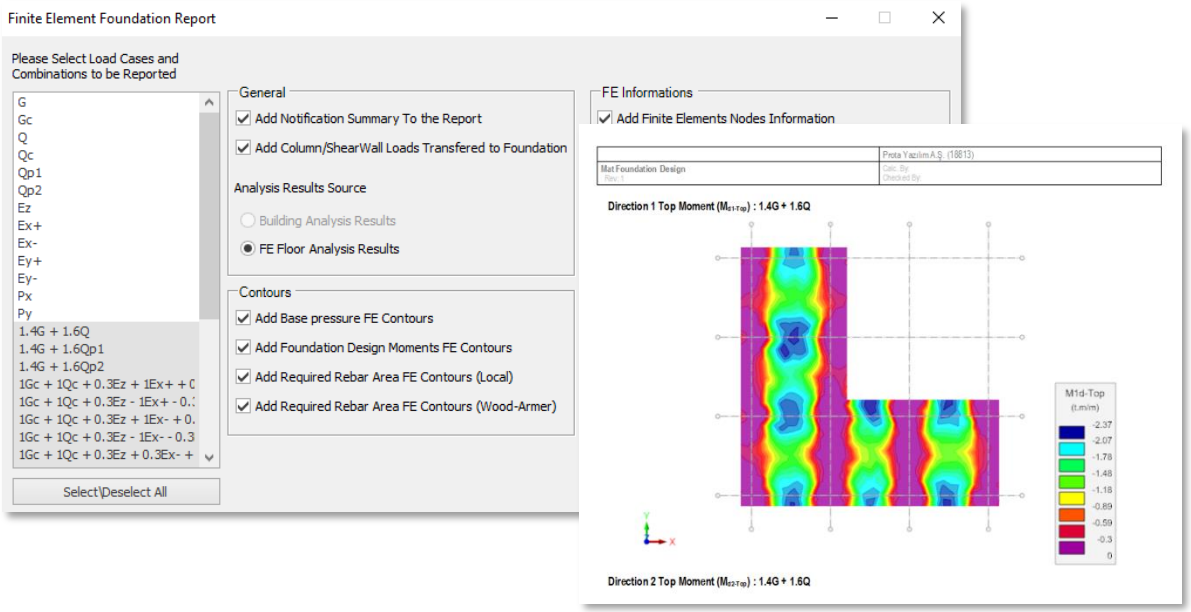
Slab Strip : X2 -- Storey : 1

Materials : C25/S420

Slab / Type	d h	g q	L1 L2	CSupport MSupport	CSpan MSpan	Mc-Left Mc-Right	As L-required As L-Provided	As M-required As M-Provided	As R-required As R-Provided	SupportLeft StrTop	Span BentUp StrBot	SupportRight StrTop

Detailed Mat Foundation Design Report

We have developed a detailed report that combines all aspects of a mat foundation design, including FE contours, soil pressures, rebar design, punching checks, etc.



Improved BIM Integration

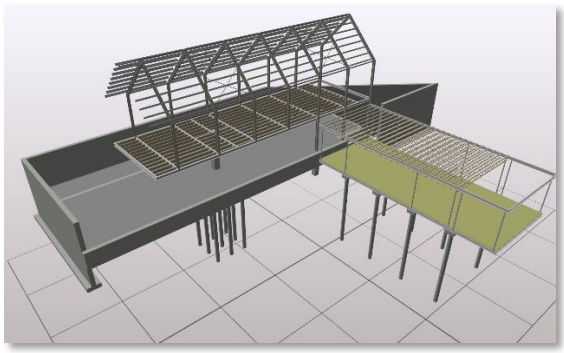
Better IFC Import and Export

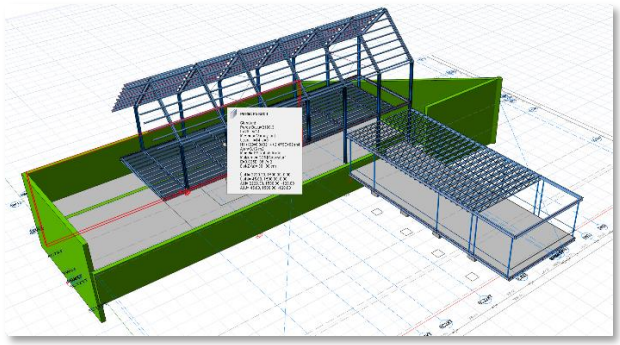
Importing and exporting a IFC format in a consistent manner is a challenging task. There are various adaptations and elements employed by different architectural and structural software on the market. Unfortunately, data representation and consistency are a big problem in a seamless IFC communication.

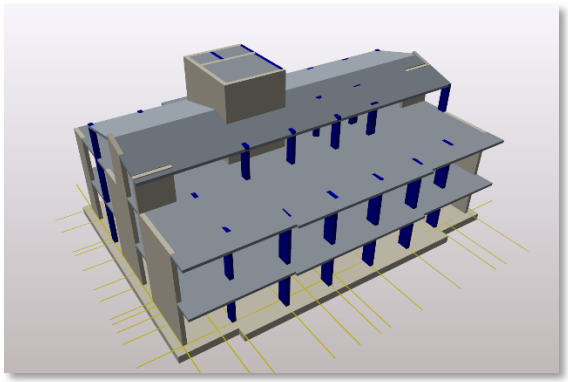
In ProtaStructure 2024, we have come a long way in IFC handling. We have almost redeveloped our IFC interpretation library to handle more scenarios more robustly and flexibly. A wider range of IFC files and elements can now be recognized and converted to ProtaBIM Hub.

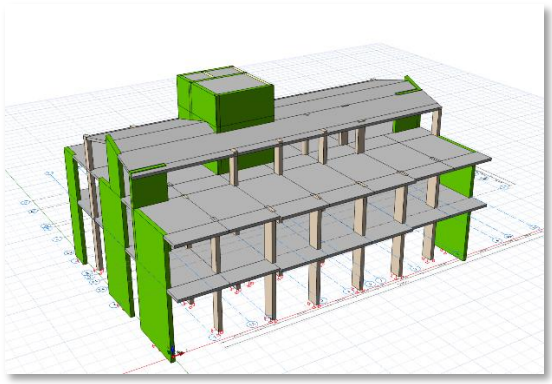
As a result of this, most of the reported issues are fixed in ProtaStructure 2024. As improvement, BREP, Tessellation Objects, Custom Sections can now be imported into ProtaStructure.

We are keen to enhance IFC communication with every update.



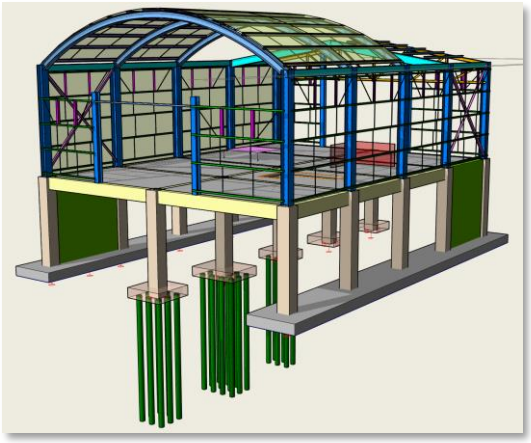


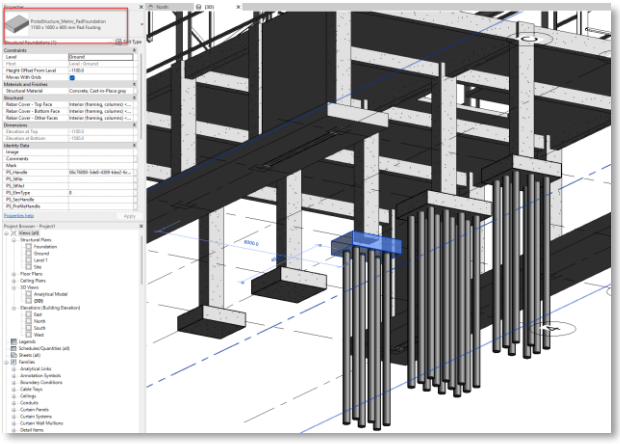


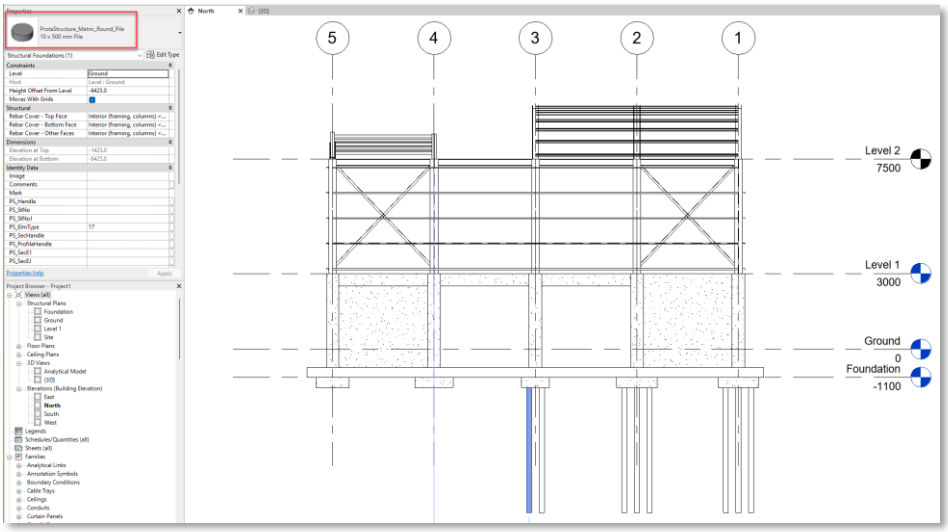


Pile Caps and Pad Footings to Revit

Pad footings and rectangular pile caps can be communicated to Revit via corresponding Revit families. The latest ProtaBIM2022_forRevit2023 must be used to benefit from this feature. The setup file can be downloaded on our website. Starting the Revit project with the provided ProtaStructure metric Revit template is recommended.





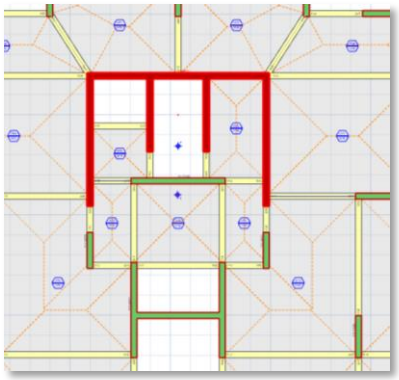


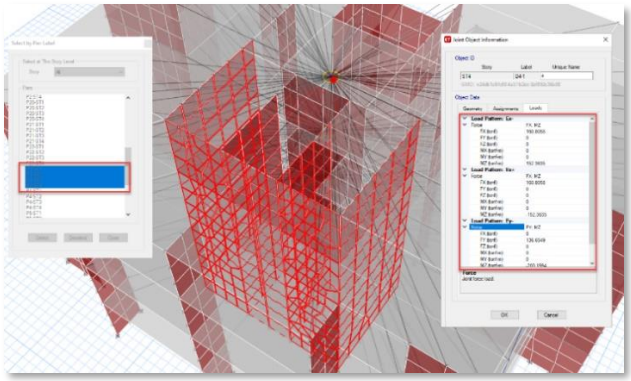
Improvements in ETABS Export

The ability to export analytical and physical models to ETABS is one of ProtaStructure's many BIM communication features. We have been continually improving the ETABS link to better suit our user's needs.

Previous improvements included:

- Export of equivalent static forces to diaphragm master joints to increase accuracy (instead of automatic calculation in ETABS)
- Automatic assignment of PIER definitions to core walls and rectangular walls modeled with FE Shell elements.
- Export of shell section modifiers to improve consistency and accuracy.





With the latest update, the export of frame members (beams, columns) is improved to make the communicated model more design-ready in ETABS.

1. The material type is now set to 'Concrete' for concrete frame members. Previously, it was set to "Other".
2. Concrete section details are now introduced. Default bars are assigned to columns. (16 mm for longitudinal bars, 10 mm for stirrups)
3. The concrete column section design type is set to "P-M2-M3" by default.
4. The concrete beam section design type is set to "M3" by default.

Modeling and Loading

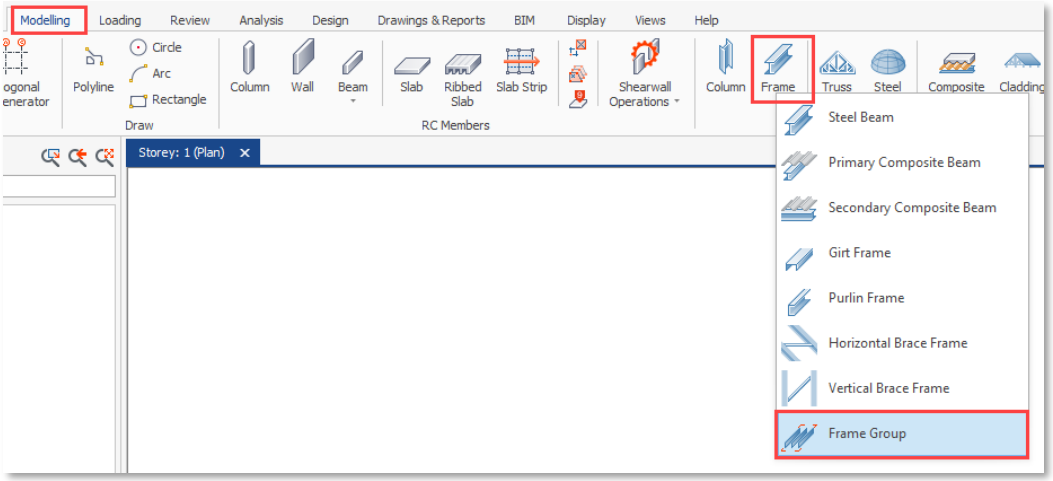
Frame Groups

For steel floor and decking systems, "**Frame Groups**" have been developed to quickly insert secondary beams or parallel steel beams. In previous versions, the feature of insertion of multiple steel frames parallel to each other was available. However, these members did not behave as a group, and after being inserted, the parameters of the group layout could not be adjusted.

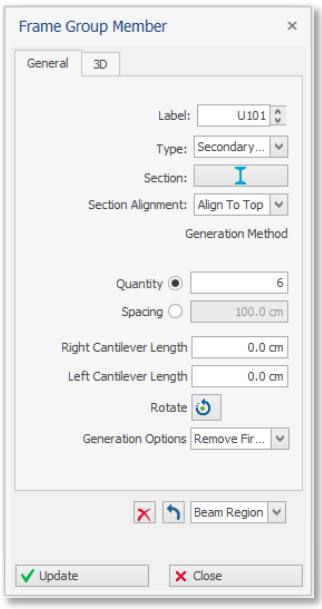
With **ProtaStructure 2024**, steel frames can be inserted in groups, and the properties of the inserted frames and their layout options can be changed from a single properties window.

To insert a frame group:

1. Click the **Model > Frame > Frame Group** command in the ribbon toolbar.

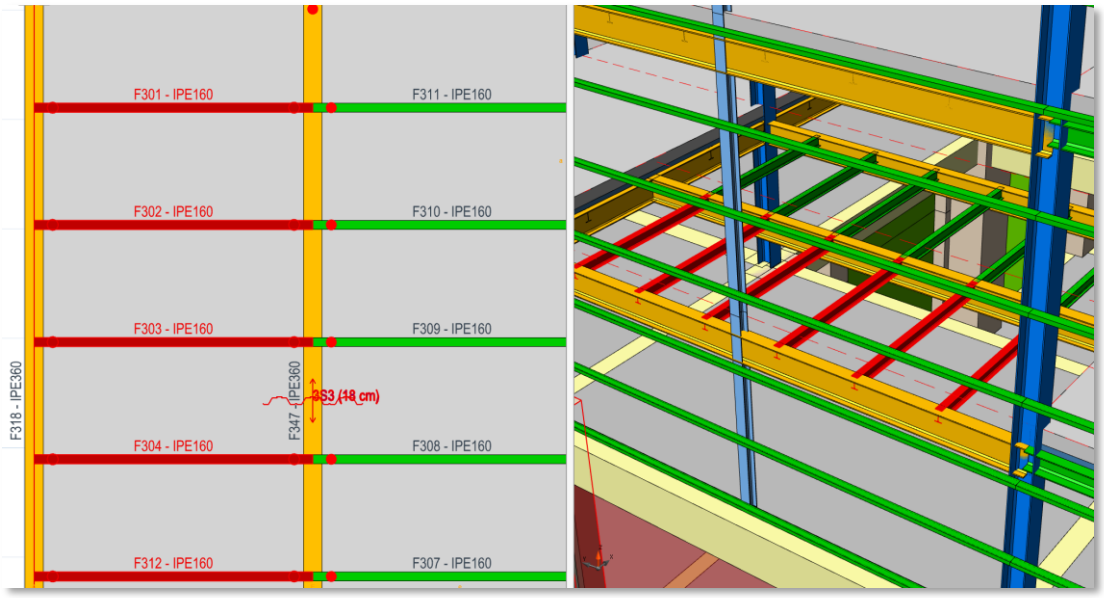


2. The **Frame Group Properties** window will open.



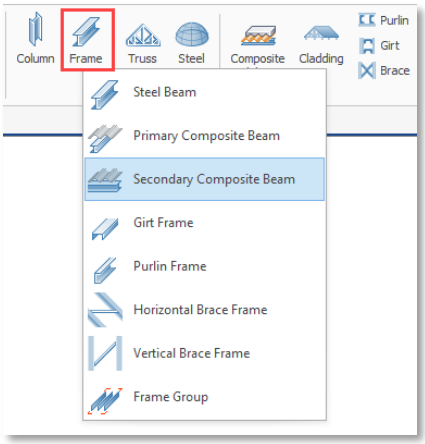
3. Using this window, you can specify the **type, section, section alignment, quantities** or **spacing, cantilever extension lengths, group angle, insertion option, element offsets,** and **end conditions** of the members to be placed.

4. The inserted frame group will now behave as a single object. You can edit the group parameters at any point in your work.



New Frame Types

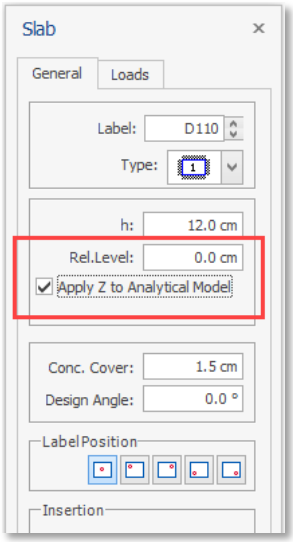
In ProtaStructure 2024, the steel element types that can be defined with frame members have been expanded. In the **Model > Frame** menu in the Ribbon toolbar, you can access frame members of the desired type such as **Steel Beam**, **Primary Composite Beam**, **Secondary Composite Beam**, **Brace**, **Purlin**, **Horizontal Brace**, **Vertical Brace**, and **Frame Group**.



Frame members are a new generation of elements that are independent of the axis, allowing for easy manipulation of definition points and offsets, providing great convenience in modeling. **Finite element method can be used to transfer loads** to frame elements from slabs, and **restraints** and **end-releases** can be assigned at both ends. The desired **load profile** can be defined on frame members with ProtaStructure's interactive load editor. In addition, frame members also support advanced duplication options such as **copying**, **mirroring**, and automatic generation between storeys.

These members are automatically assigned to different drawing layers according to their types and are displayed in different colors in the model. Furthermore, in ProtaSteel, suitable connections and detail drawings can be created for these types.

Practical Modeling of Slab Elevations



In ProtaStructure, you can use **plane definitions** to bring the slabs and member ends to a different 3D orientation from their current floor level. Plane definition has been available in ProtaStructure for a long time. Slabs with altered elevations or slopes are modeled in the analysis using these plane definitions.

In ProtaStructure 2024, we have developed an even more practical way than plane definitions to facilitate **shifting the slabs parallel to a different 3D orientation**. This eliminates the need to use plane definitions. Here's how:

1. Select a slab on the screen and load the **Properties** window.
2. Enter the relative elevation difference from the floor level in the "**Rel. Level**" field. Negative values will move the slab downwards.
3. Check the "**Apply Z to Analytical Model**" option. This way, the elevation difference you entered will be applied to the analysis model. Otherwise, the slab will be modeled at the floor level in the analysis.

Important Note

Stepped foundations on the foundation floor should be modeled with this option.

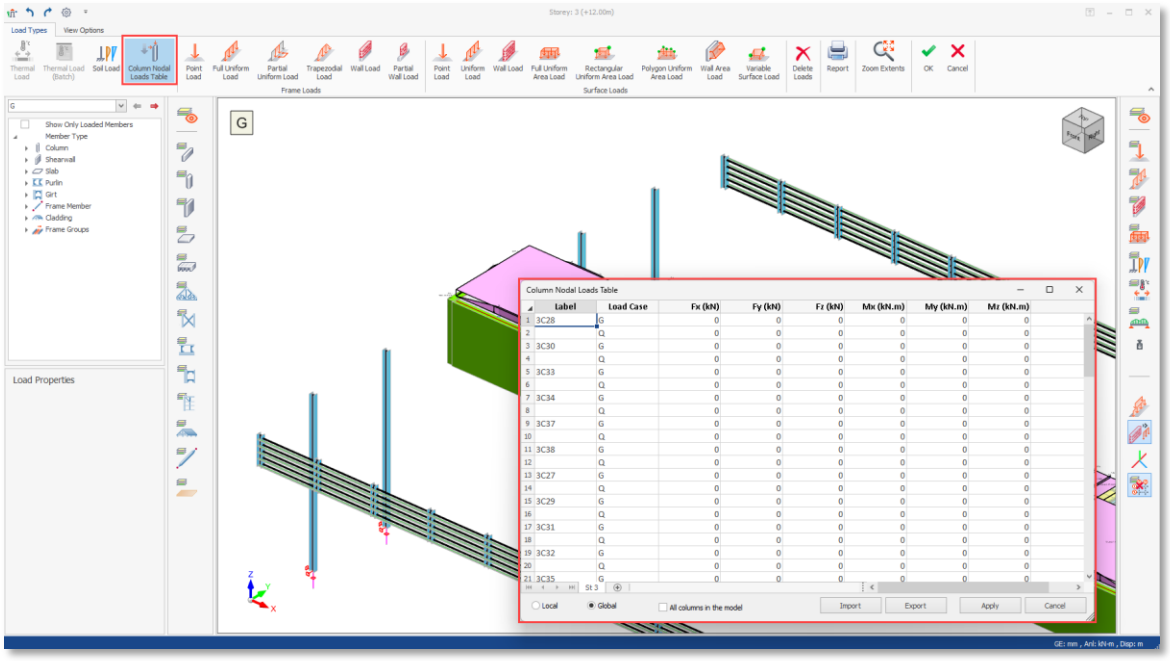
Assigning Point Loads via Spreadsheet

The **Interactive Load Editor** in ProtaStructure provides powerful and visual tools for loading elements with different load types. In **ProtaStructure 2024**, we have taken this a step further.

If you have multiple point loads or concentrated moments prepared in a tabulated manner in an external Excel sheet, you can now quickly assign them to columns in ProtaStructure in a batch. Here's how:

Open the **Loading Editor** for the active floor.

Click on the "**Column Nodal Loads Table**" button on the toolbar in the loading editor. An Excel-like table containing all columns on the floor and load cases will open within ProtaStructure.



First and Easiest Method: Direct Value Entry

1. Use the table to enter the **Fx, Fy, Fz, Mx, My, Mz** values for the desired columns in local or global coordinates.

2. Assign the loads by clicking the **Apply** button.

Second Method: Exporting the File

1. If you prefer to work within Excel, click the "**Export**" button to open the table in Excel.

2. After entering the values in Excel, save the file.

3. Return to ProtaStructure and click the "**Import**" button. ProtaStructure will automatically recognize the file you exported and bring the values automatically.

4. Assign the loads by clicking the **Apply** button.

Third Method: Copy/Paste

If you already have load values ready at your disposal in a different place, you can paste them into the table in ProtaStructure using the copy/paste command.

Important Note

For all three methods, the table format you introduce to ProtaStructure must match the format expected by ProtaStructure. Therefore, it is recommended to first export the empty table and work from there. If you attempt to transfer data in a different format, you may not be successful.

Easier Merging of Identical Blocks

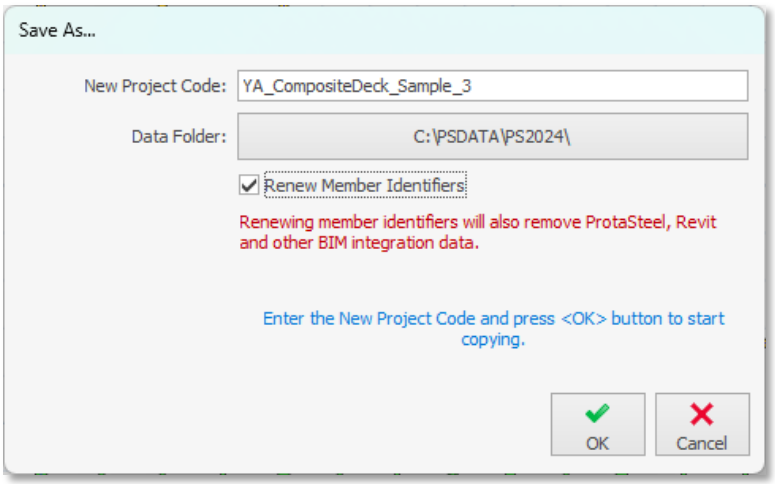
One of the most important features that increase productivity in ProtaStructure is the merging of different blocks in a single project. This allows multiple engineers to work in different parts of a project and then merge their models later.

In ProtaStructure, you can merge different blocks for the following purposes:

1. To include only the axis system from a different block
2. To merge different floors of the same block
3. To merge only the foundation columns and analysis results of different blocks in a common foundation design
4. To merge different blocks completely in the same project

In previous versions of ProtaStructure, it was necessary to remodel identical blocks from scratch to merge them in the same project. Otherwise, even if duplicated with the "Save As New" option, they could not be merged. This is no longer necessary.

1. Duplicate the building to create the twin block with "Save As New" command.
2. Check the "Renew Element Identifiers" option in the save window.
3. Complete the saving process. All member identifiers will be changed in the newly saved model. However, if there is existing BIM integration information in this project, it will be lost.



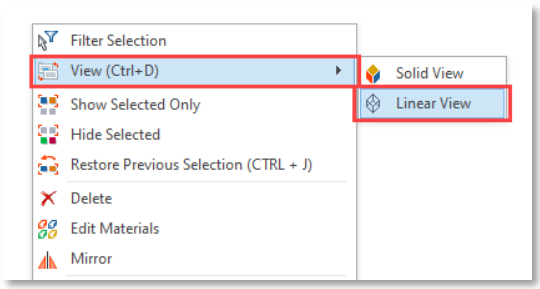
Line View in 3D Physical Model

As is known, ProtaStructure is a powerful **Structural BIM** program that allows modeling, analysis, design, and detailing with physical structural members. Therefore, the analytical model is automatically generated from the physical model.

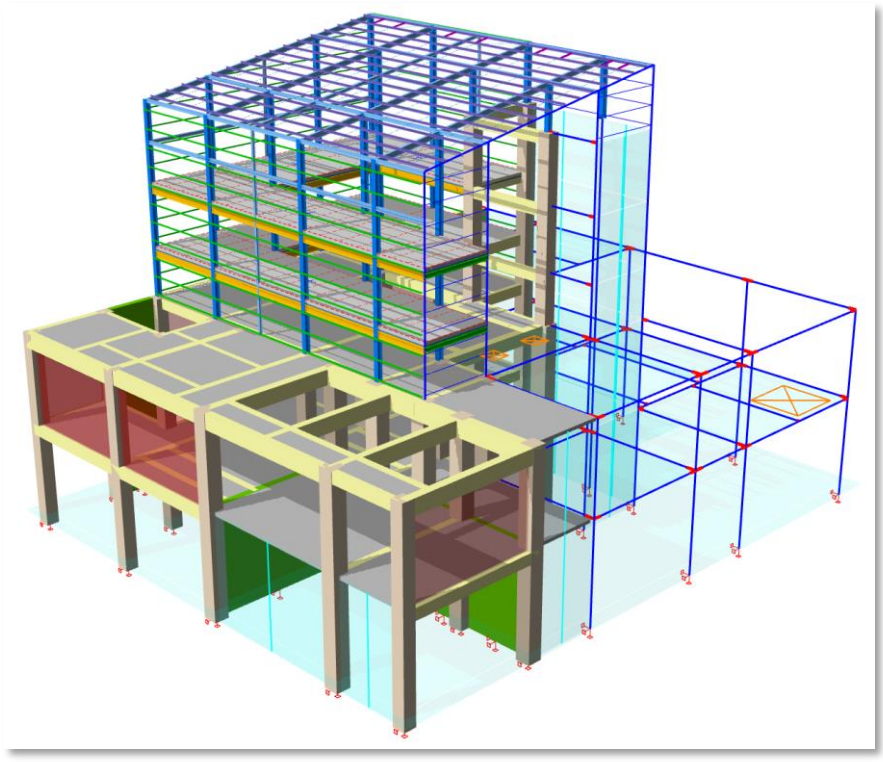
When creating a 3D physical model and before performing analysis, it is important to understand how the analytical model will be created, how the elements will be connected to each other, the joints, rigid zones, and similar details.

In ProtaStructure 2024, you can now switch quickly between **solid** and **linear view modes** while modeling. You can display selected elements or the entire model in linear mode.

1. Select the elements you want to view in wireframe mode.
2. Choose the "**View > Linear View**" command from the right-click menu.
3. The selected elements will begin to be displayed in wireframe mode. This way, you will obtain a hybrid view showing how the elements are connected to each other.



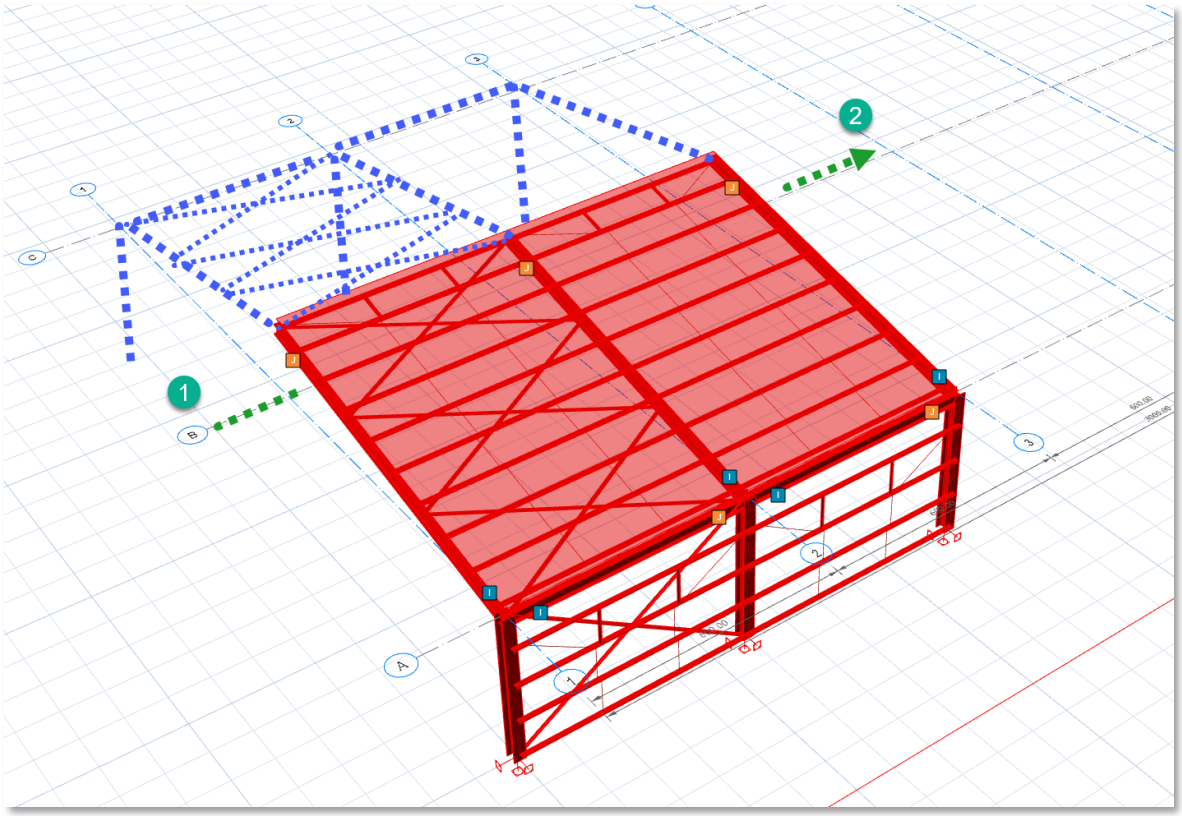
4. You can also quickly switch between view modes by pressing the **F11** key repeatedly. However, this way, the view mode of the entire model will change, and the view properties of the selected elements will be reset.



Mirroring Steel Members

With ProtaStructure 2024, steel members can be mirrored around any two points. Steel columns, beams, frame members, frame groups, braces, trusses, purlins, girts, and claddings can be mirrored.

When combined with **Copy** and **Move** operations, you will now be able to create your models much faster with the **Mirroring** command.



Advanced Earthquake Engineering

Nonlinear Seismic Isolators

By leveraging our expertise in seismic isolation, we have completely re-developed the modeling and analysis of seismic isolators in ProtaStructure 2024. We have developed new tools to ensure the final precise design of seismically isolated buildings.

The design of earthquake-resistant structures requires **expertise in advanced earthquake engineering**. Seismic isolation is used to extend the period of the structure above the isolation layer, reducing earthquake forces, storey acceleration and relative displacements. Seismic isolators also introduce a certain level of damping to the structure. However, there are important points to consider in the construction and project phases of these structures.

Several key phases in the design of earthquake-resistant structures:

1. **Defining the structure's architecture and function:** The building's function should be designed with the seismic isolators in mind during the architectural and mechanical design stages.

2. **Shaping the structural system:** It is crucial to determine the placement of isolators and design the members that will safely transfer forces.

3. Identifying **site-specific earthquake hazards**, determining appropriate **ground motions**, and **target spectra**.

4. Modeling the structure and making preliminary designs to **determine the types of isolator units**. Linear elastic analysis may be sufficient at this stage, and the types of isolators are generally decided in collaboration with the manufacturers.

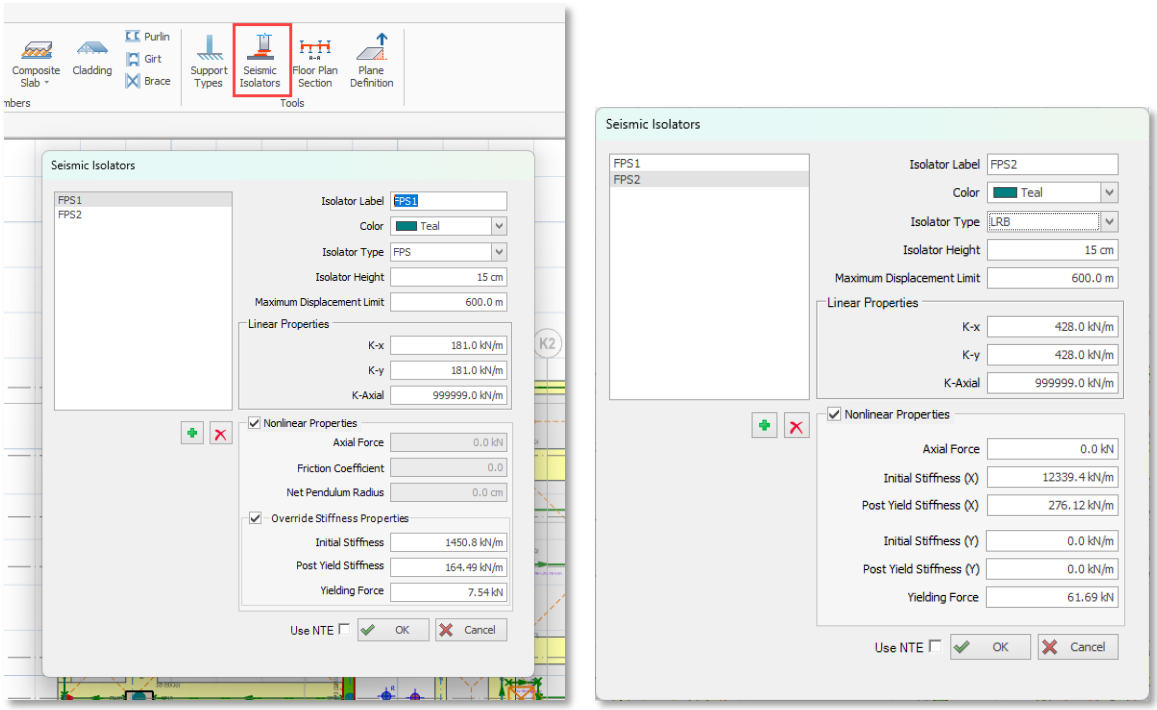
5. **Finalizing the design** of the structure with the determined isolator types. Design, displacement, and period checks are performed using lower and upper bound values for isolators for different earthquake levels.

Seismic Isolator Library

The types of seismic isolators can now be defined with the help of the library located in the **Modeling > Seismic Isolators** ribbon toolbar.

In addition to the Linear Properties of the isolation units, the **Nonlinear Properties** can now also be defined. A change made in the library is automatically reflected to all assigned types.

Nonlinear properties can be automatically calculated from parameters such as **Axial Force, Friction Coefficient**, and **Net Pendulum Radius**. for friction pendulum (FPS) isolator units. Force-deformation relationship can also be manually defined by entering values such as **Initial Stiffness, Post-Yield Stiffness**, and **Yield Force**. For **Lead Rubber Bearings**, nonlinear properties can be defined by entering values for Initial Stiffness, Post-Yield Stiffness, and Yield Force differently in both directions.



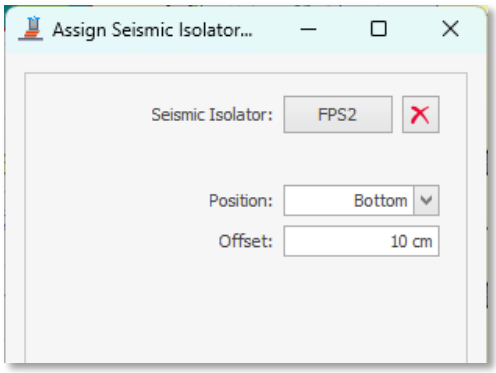
Assigning Seismic Isolators to Members

ProtaStructure 2024 brings innovations and improvements in insertion and analytical modeling of seismic isolators.

Assigning Seismic Isolators to Columns

To assign seismic isolators to columns:

1. Select a column on the screen and use the "Assign Isolator" command in the right-click menu.
2. Select one of the isolator types previously defined in the library from the "**Seismic Isolator**" list.
3. Enter the **Position** and **Offset** values and click the **OK** button.
4. An isolator will be inserted on the column's analytical frame at the specified position.



Assigning Seismic Isolators to Rectangular Walls and Core Walls

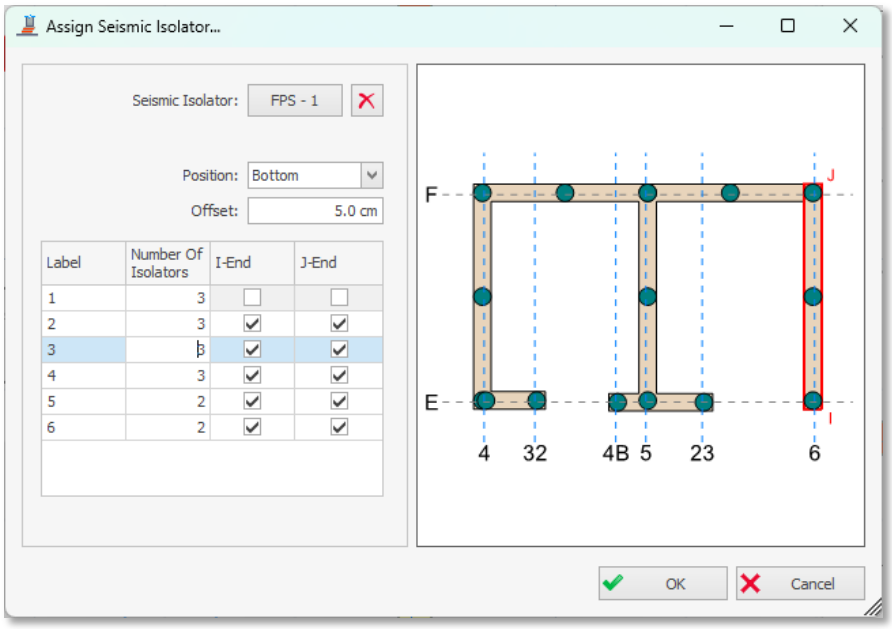
In previous versions of ProtaStructure, only one seismic isolator could be inserted to rectangular walls or core walls. If multiple isolators needed to be defined, their linear properties had to be combined to create a single lumped isolator by the user. However, in practice, there is a need to define multiple isolators for walls, and the approach of combining all isolator properties will not be valid (or sufficient) in non-linear analyses. This is because it will become impossible to determine the tensile forces that may occur when the isolators under the walls are lumped.

With ProtaStructure 2024, you can individually define any number of isolators for wall panels. To do this:

Select a rectangular wall or a core wall on the screen and use the "**Assign Isolator**" command in the right-click menu.

On the isolator assignment window, a preview of the isolator placement will be shown.

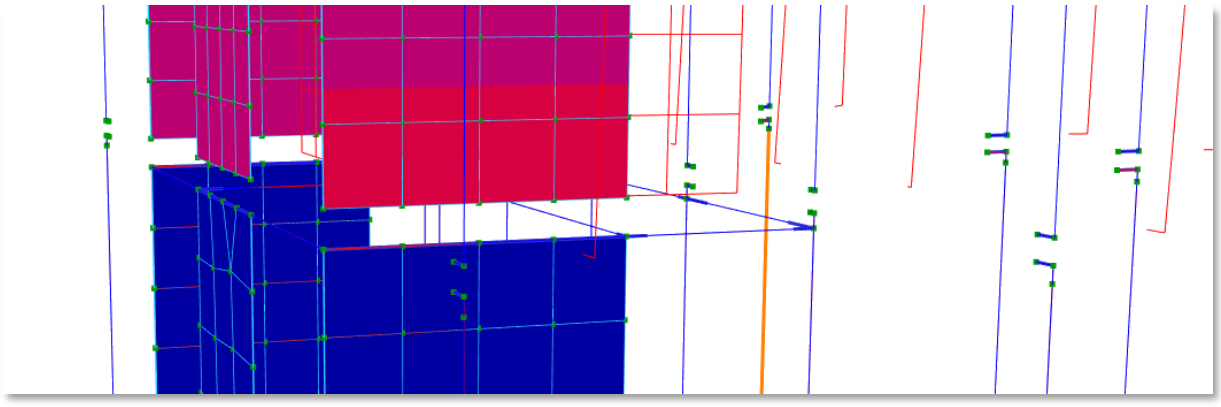
You can place the isolators to correspond to the I and J insertion points of the wall panel. Alternatively, you can also ensure equal distribution across the section. In core walls, the wall legs will be automatically detected and listed. You can define isolators separately for each wall leg.



Analytical Modeling of Isolators Assigned to Shearwalls

In previous versions of ProtaStructure, shearwalls with seismic isolators had to be modeled with the "**Mid-pier**" option. With **ProtaStructure 2024**, shearwalls with isolators can be modeled with the "**Finite Element Shell**" option.

The shell elements are divided by the isolator height to create an analytical model, and the specified number of isolator elements are analytically placed in between.

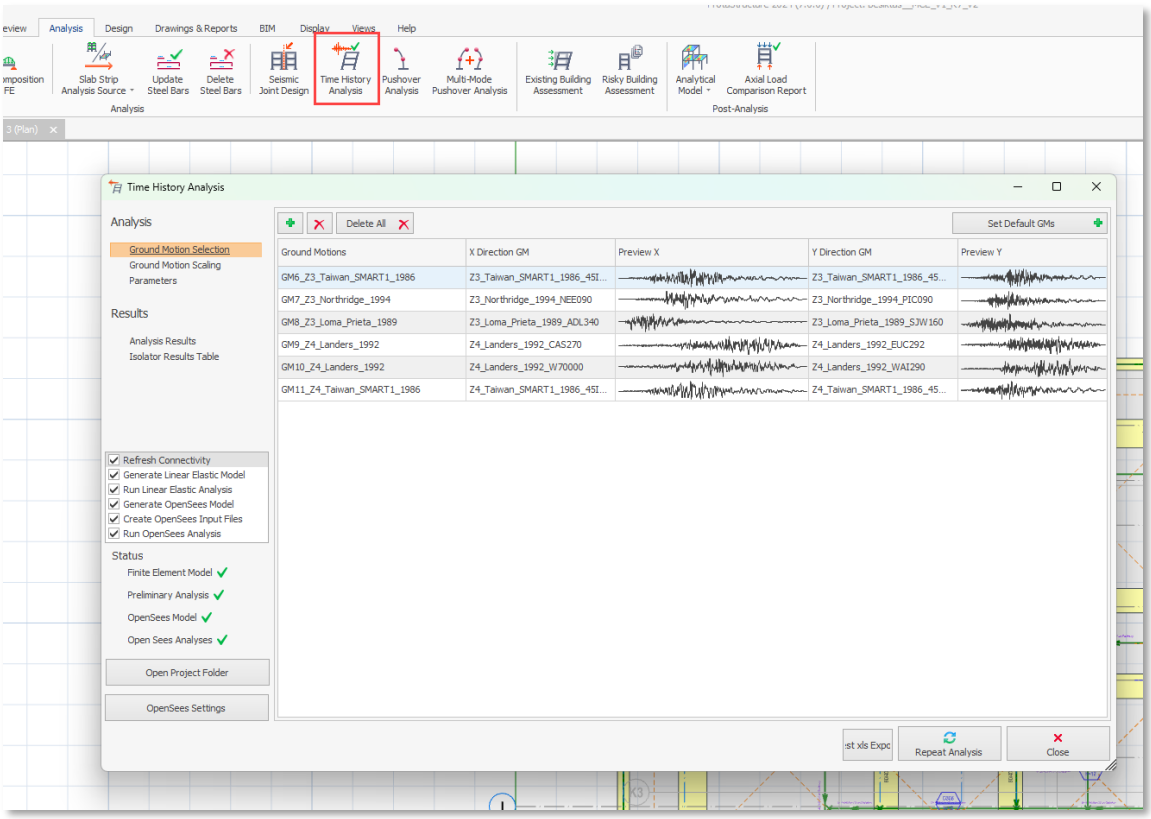


Nonlinear Time-History Analysis

Nonlinear characteristics of isolators will be automatically considered in a **nonlinear time-history analysis** performed in ProtaStructure. In such an analysis, all members except isolators are assumed to be linearly elastic, and no material plasticity is assigned.

1. To perform a time-history analysis, click the "**Analysis > Time-History Analysis**" button in the ribbon toolbar.

2. You can perform operations such as **selecting** and **scaling ground motions** before initiating the analysis.

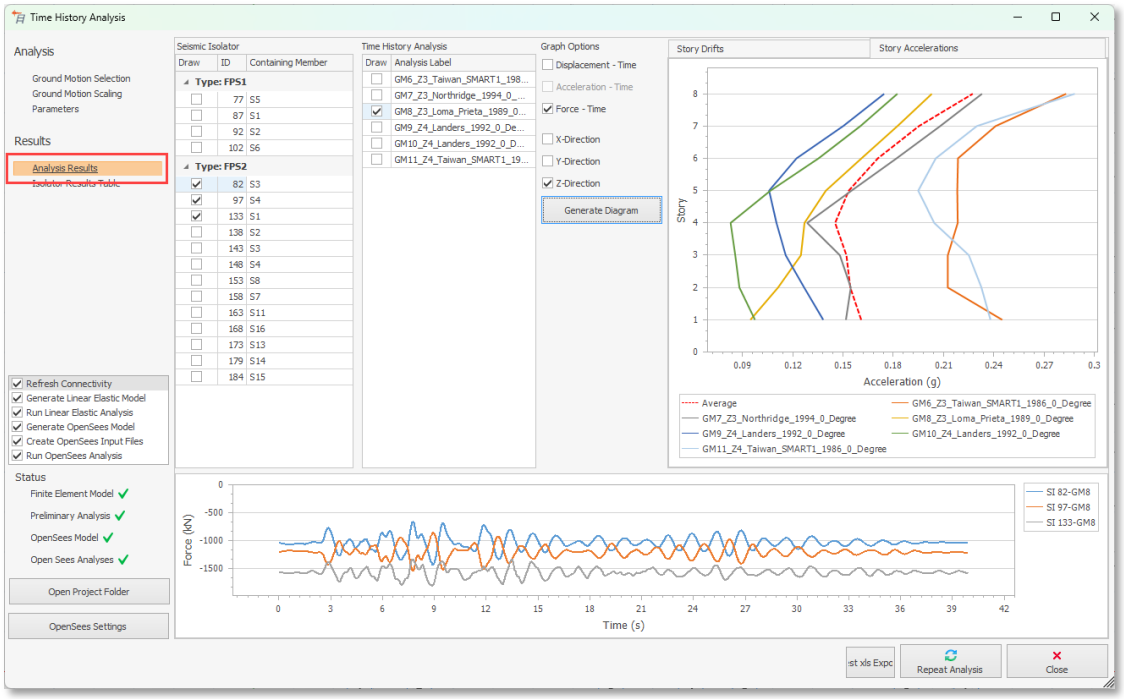


Analysis Output

ProtaStructure 2024 will provide you with the detailed outputs for each isolator and the overall building after non-linear time-history analysis. You can use these outputs to generate your own custom design report for your isolated building.

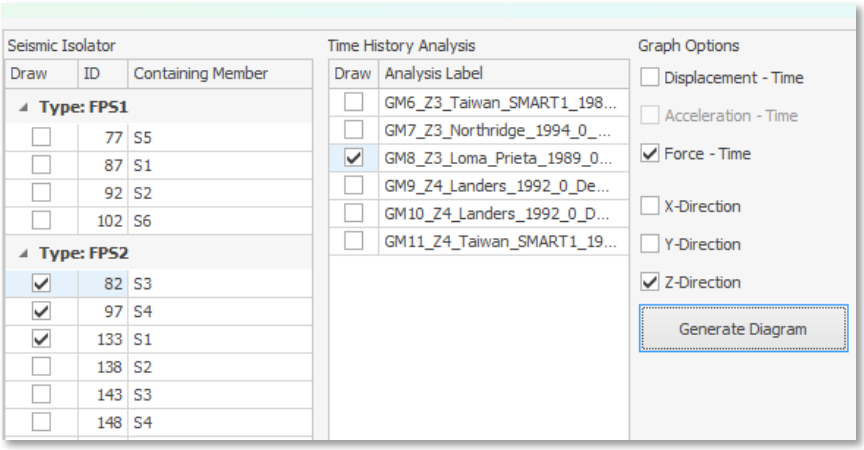
Analysis Results

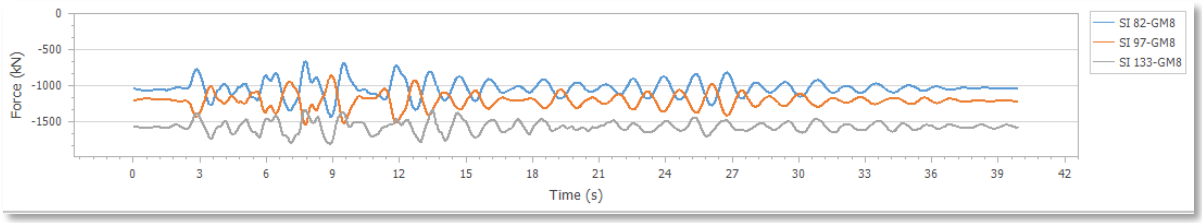
You can visualize the isolator response history for each ground motion on the charts provided at the Analysis Results tab.



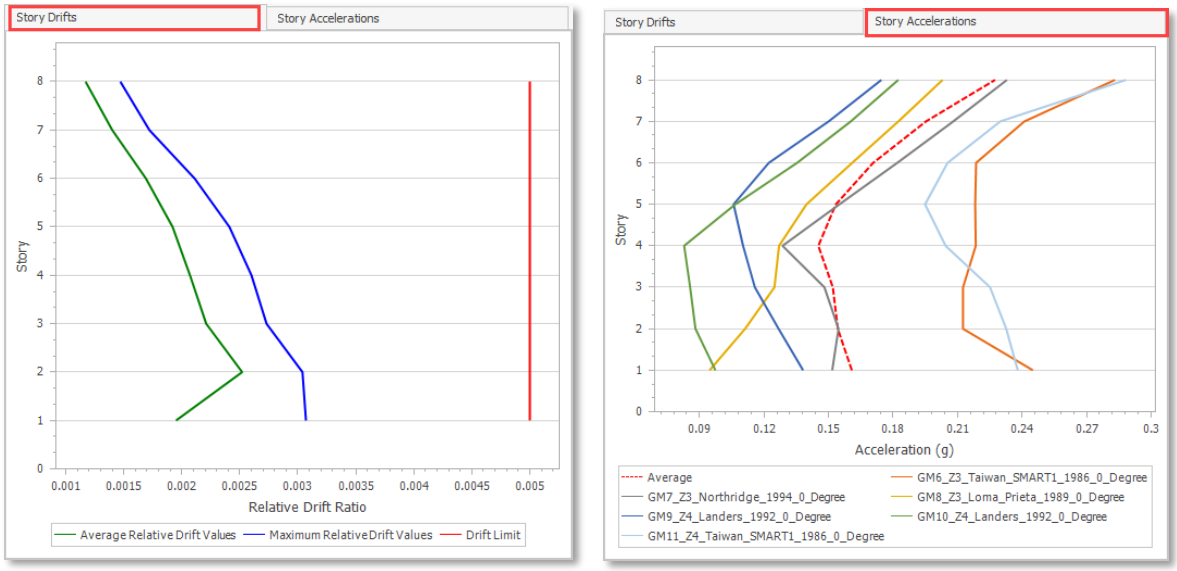
1. You can obtain **Displacement**, **Acceleration** and **Force Response History** for **X, Y,** and **Z** directions. The requested results will be plotted on the same graph.

2. To refresh the plot according to the selections you have made, click "**Generate Diagram**" button.



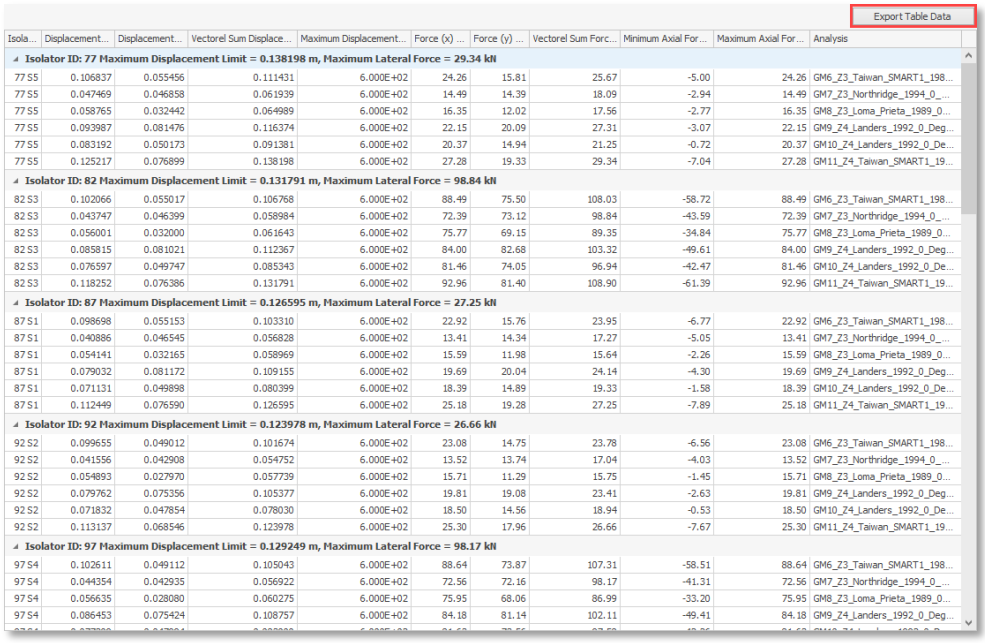


3. **Storey Accelerations** and **Relative Drift Ratios** for the upper structure will also be plotted for all ground motions, their maximum and minimum responses.



Isolator Results Table

All isolator results will be tabulated in a detailed manner when you click "**Isolator Results Table**" tab. You can export this table to Excel and use it in your design reports.



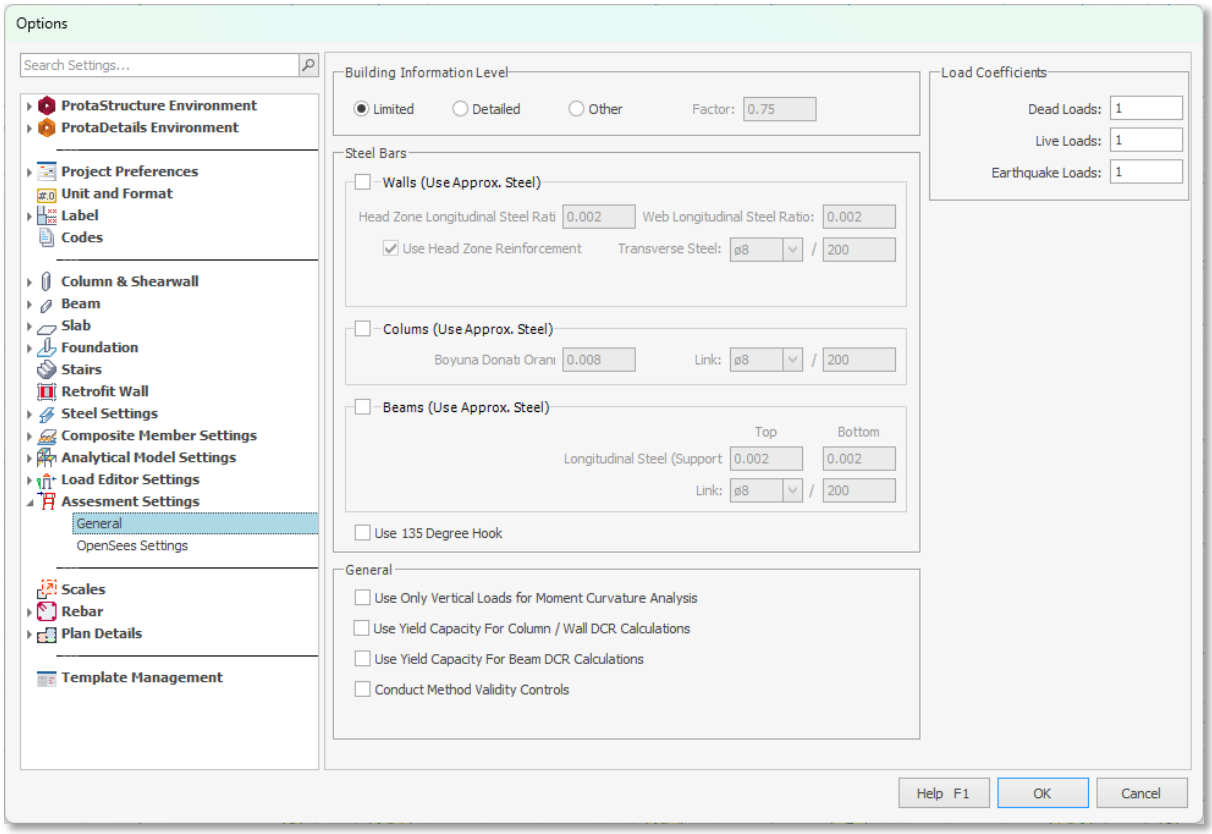
Isola...	Displacement...	Displacement...	Vectorel Sum Displace...	Maximum Displacement...	Force (x) ...	Force (y) ...	Vectorel Sum Forc...	Minimum Axial For...	Maximum Axial For...	Analysis
Isolator ID: 77 Maximum Displacement Limit = 0.138198 m, Maximum Lateral Force = 29.34 kN										
77 S5	0.106837	0.055456	0.111431	6.000E+02	24.26	15.81	25.67	-5.00	24.26	GM6_Z3_Taiwan_SMART1_198...
77 S5	0.047469	0.046858	0.061939	6.000E+02	14.49	14.39	18.09	-2.94	14.49	GM7_Z3_Northridge_1994_0_...
77 S5	0.058765	0.032442	0.064989	6.000E+02	16.35	12.02	17.56	-2.77	16.35	GM8_Z3_Loma_Prieta_1989_0...
77 S5	0.093987	0.081476	0.116374	6.000E+02	22.15	20.09	27.31	-3.07	22.15	GM9_Z4_Landers_1992_0_Deg...
77 S5	0.083192	0.050173	0.091381	6.000E+02	20.37	14.94	21.25	-0.72	20.37	GM10_Z4_Landers_1992_0_De...
77 S5	0.125217	0.076899	0.138198	6.000E+02	27.28	19.33	29.34	-7.04	27.28	GM11_Z4_Taiwan_SMART1_19...
Isolator ID: 82 Maximum Displacement Limit = 0.131791 m, Maximum Lateral Force = 98.84 kN										
82 S3	0.102066	0.055017	0.106768	6.000E+02	88.49	75.50	108.03	-58.72	88.49	GM6_Z3_Taiwan_SMART1_198...
82 S3	0.043747	0.046399	0.058984	6.000E+02	72.39	73.12	98.84	-43.59	72.39	GM7_Z3_Northridge_1994_0_...
82 S3	0.056001	0.032000	0.061643	6.000E+02	75.77	69.15	89.35	-34.84	75.77	GM8_Z3_Loma_Prieta_1989_0...
82 S3	0.085815	0.081021	0.112367	6.000E+02	84.00	82.68	103.32	-49.61	84.00	GM9_Z4_Landers_1992_0_Deg...
82 S3	0.076597	0.049747	0.085343	6.000E+02	81.46	74.05	96.94	-42.47	81.46	GM10_Z4_Landers_1992_0_De...
82 S3	0.118252	0.076386	0.131791	6.000E+02	92.96	81.40	108.90	-61.39	92.96	GM11_Z4_Taiwan_SMART1_19...
Isolator ID: 87 Maximum Displacement Limit = 0.126595 m, Maximum Lateral Force = 27.25 kN										
87 S1	0.098698	0.055153	0.103310	6.000E+02	22.92	15.76	23.95	-6.77	22.92	GM6_Z3_Taiwan_SMART1_198...
87 S1	0.040886	0.046545	0.056828	6.000E+02	13.41	14.34	17.27	-5.05	13.41	GM7_Z3_Northridge_1994_0_...
87 S1	0.054141	0.032165	0.058969	6.000E+02	15.59	11.98	15.64	-2.26	15.59	GM8_Z3_Loma_Prieta_1989_0...
87 S1	0.079032	0.081172	0.109155	6.000E+02	19.69	20.04	24.14	-4.30	19.69	GM9_Z4_Landers_1992_0_Deg...
87 S1	0.071131	0.049898	0.080399	6.000E+02	18.39	14.89	19.33	-1.58	18.39	GM10_Z4_Landers_1992_0_De...
87 S1	0.112449	0.076590	0.126595	6.000E+02	25.18	19.28	27.25	-7.89	25.18	GM11_Z4_Taiwan_SMART1_19...
Isolator ID: 92 Maximum Displacement Limit = 0.123978 m, Maximum Lateral Force = 26.66 kN										
92 S2	0.099655	0.049012	0.101674	6.000E+02	23.08	14.75	23.78	-6.56	23.08	GM6_Z3_Taiwan_SMART1_198...
92 S2	0.041556	0.042908	0.054752	6.000E+02	13.52	13.74	17.04	-4.03	13.52	GM7_Z3_Northridge_1994_0_...
92 S2	0.054893	0.027970	0.057739	6.000E+02	15.71	11.29	15.75	-1.45	15.71	GM8_Z3_Loma_Prieta_1989_0...
92 S2	0.079762	0.075356	0.105377	6.000E+02	19.81	19.08	23.41	-2.63	19.81	GM9_Z4_Landers_1992_0_Deg...
92 S2	0.071832	0.047854	0.078030	6.000E+02	18.50	14.56	18.94	-0.53	18.50	GM10_Z4_Landers_1992_0_De...
92 S2	0.113137	0.068546	0.123978	6.000E+02	25.30	17.96	26.66	-7.67	25.30	GM11_Z4_Taiwan_SMART1_19...
Isolator ID: 97 Maximum Displacement Limit = 0.129249 m, Maximum Lateral Force = 98.17 kN										
97 S4	0.102611	0.049112	0.105043	6.000E+02	88.64	73.87	107.31	-58.51	88.64	GM6_Z3_Taiwan_SMART1_198...
97 S4	0.044354	0.042935	0.056922	6.000E+02	72.56	72.16	98.17	-41.31	72.56	GM7_Z3_Northridge_1994_0_...
97 S4	0.056635	0.028080	0.060275	6.000E+02	75.95	68.06	86.99	-33.20	75.95	GM8_Z3_Loma_Prieta_1989_0...
97 S4	0.086453	0.075424	0.108757	6.000E+02	84.18	81.14	102.11	-49.41	84.18	GM9_Z4_Landers_1992_0_Deg...

New Workflow in Building Assessment

In ProtaStructure 2024, we have introduced innovations and significant improvements to the building assessment workflow. This will make your assessment and retrofit work much easier and more efficient.

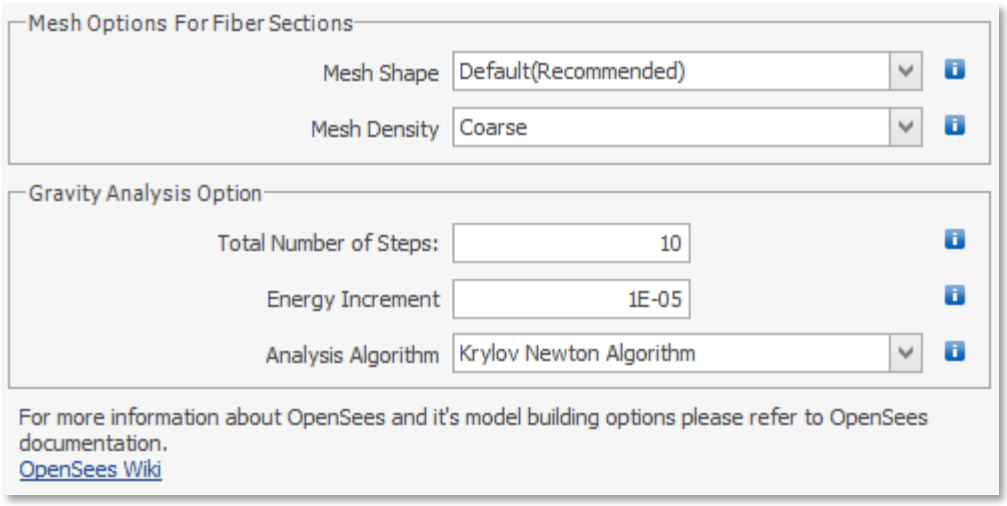
Building Assessment Settings

Parameters such as the **Building Information Level**, **Estimated Reinforcements**, **Load Coefficients**, and **OpenSees Settings** used in the assessment and retrofit of existing buildings have been consolidated into a single location for easier access and control. You can access these parameters from the "**Settings > Assessment Settings**" menu. These settings have been removed from the interface where existing building assessment analyses are performed.



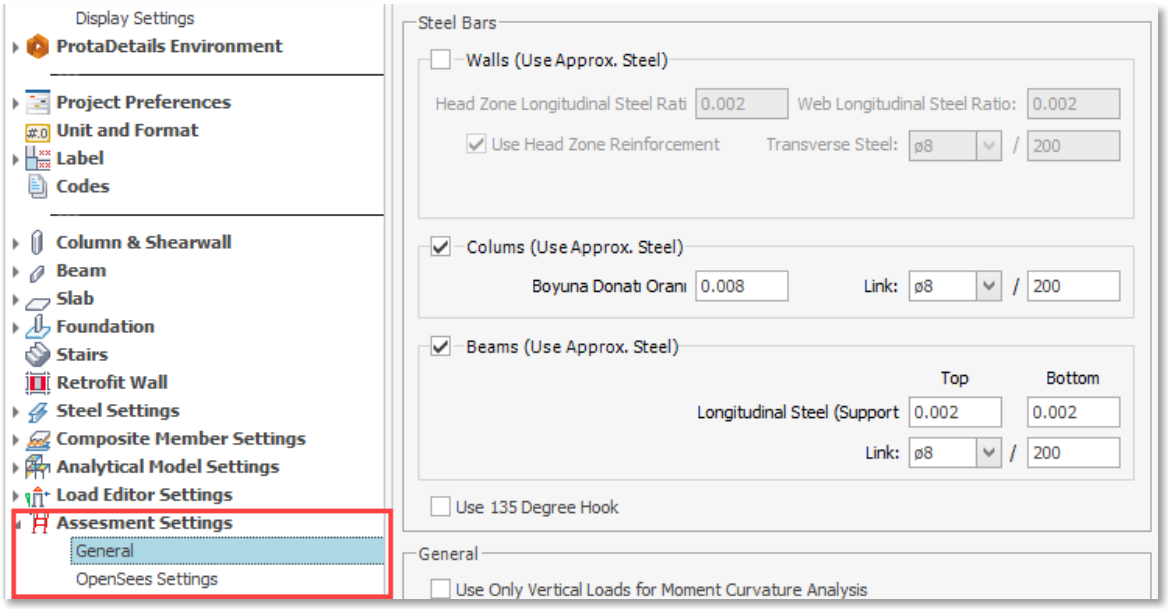
OpenSees Settings

OpenSees settings have been expanded by adding additional settings. **Fiber Section Mesh Options** and **Gravity Analysis Options** can be controlled. The default values are adjusted for the optimum solution. These values can be changed depending on the conditions of the structure you are evaluating.



Default Estimated Reinforcements for the Entire Building

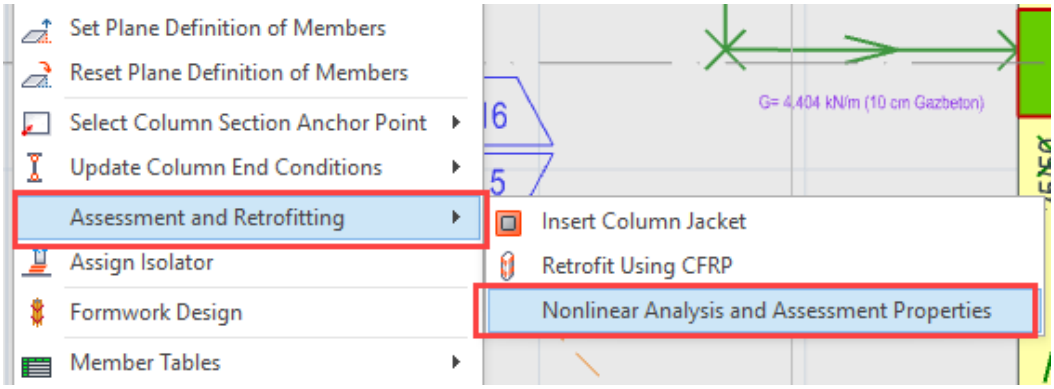
You can define estimated reinforcements for columns, walls and beams using **Settings > Assessment Settings** section. To activate the estimated reinforcements, you must check the corresponding options for member types and enter the estimated values.



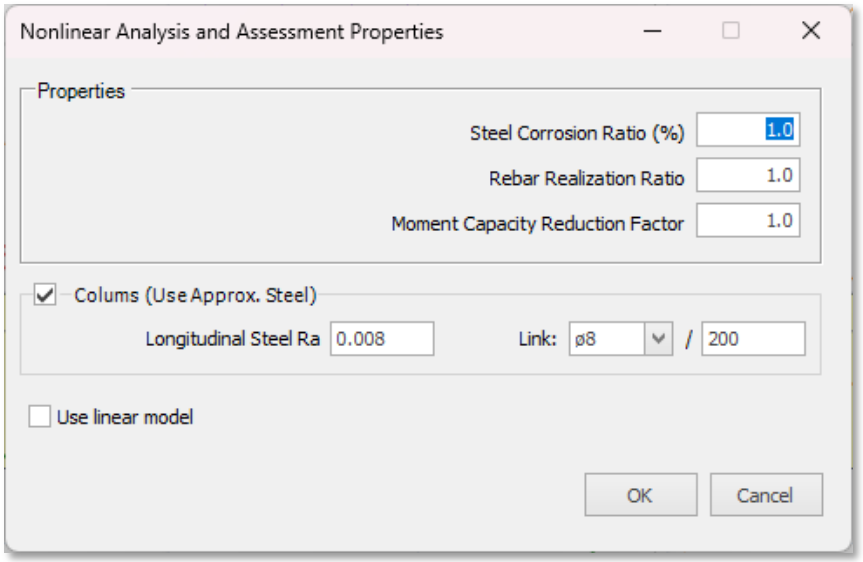
Estimated Reinforcements for Individual Members

As a new feature in ProtaStructure 2024, you can assign e**stimated reinforcement values to individual members**.

1. Select the member and right click to load the contextual menu.
2. The functionalities related to assessment are collected under **Assessment and Retrofitting** subcategory.
3. Select the **Nonlinear Analysis and Assessment Properties** command.



4. **Estimated Reinforcement, Corrosion Factor, Rebar Realization Factor** and **Moment Capacity Reduction Factor** can be entered for the selected member.
5. <u>You must check the estimated reinforcement option and enter the value</u> in order to activate it for the selected member.



Important Note

A **hierarchical approach** is adopted in the building assessment process for the use of estimated reinforcements.

The estimated reinforcement assigned to individual members have the priority in the assessment analysis. If no estimated reinforcement is assigned to individual members, then the globally defined values in the assessment settings for the entire building will be used. If global settings do not specify any estimated reinforcement, then the actual detailed reinforcements on the members defined via design menus will be used.

Priority: Member-specific estimated reinforcement values in the section "**Nonlinear Analysis and Assessment Properties**" in the right click menu.

Second Priority: Global Estimated Reinforcement Values in the "**Settings > Assessment Settings**"

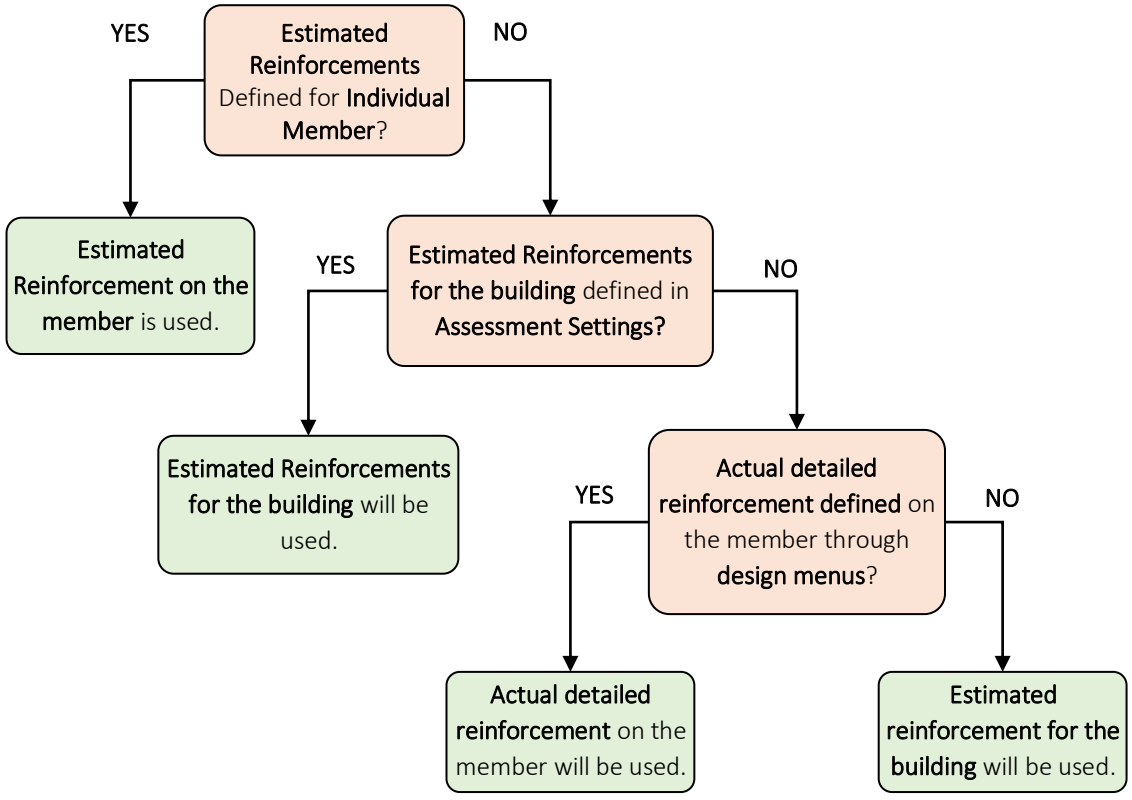
Third Priority: Actual detailed reinforcements defined through design menus directly on the member itself.

If no actual reinforcement is defined on the member itself, ProtaStructure will automatically use the default estimated reinforcement in the global settings, even if estimated reinforcement usage is not requested.

Members such as **Retrofit Wall** and **Column Jackett** will never use estimated reinforcement. Reinforcements on these members must always be entered through design menus.

Flowchart for Estimated and Detailed Reinforcement Usage

The following flowchart demonstrates how ProtaStructure will use **Estimated** and **Detailed Reinforcements** on the members.



Using Linear Model for Specific Members

A new option is introduced in ProtaStructure to consider any member as "**Linear Elastic**" during assessment. This option is accessible through **Right Click > Assessment and Retrofitting > Nonlinear Analysis and Assessment Properties > Use Linear Model** setting.

There may be cases where you want to assume that a beam, column or wall member will behave linearly elastic during assessment, or simply, you want to ignore the material nonlinearity for that member. ProtaStructure will automatically assume following members as linear elastic during a nonlinear assessment:

1. Basement walls
2. Cantilever beams
3. Members with end releases (I or J ends, asymmetrically)
4. Secondary joists supported with beams at I or J ends.
5. Members marked as Vertical-Only by the user.
6. Short beams (Beam length smaller than two times the plastic hinge length)
7. Shallow beams (Beam height less than or equal to the adjacent slab thickness)

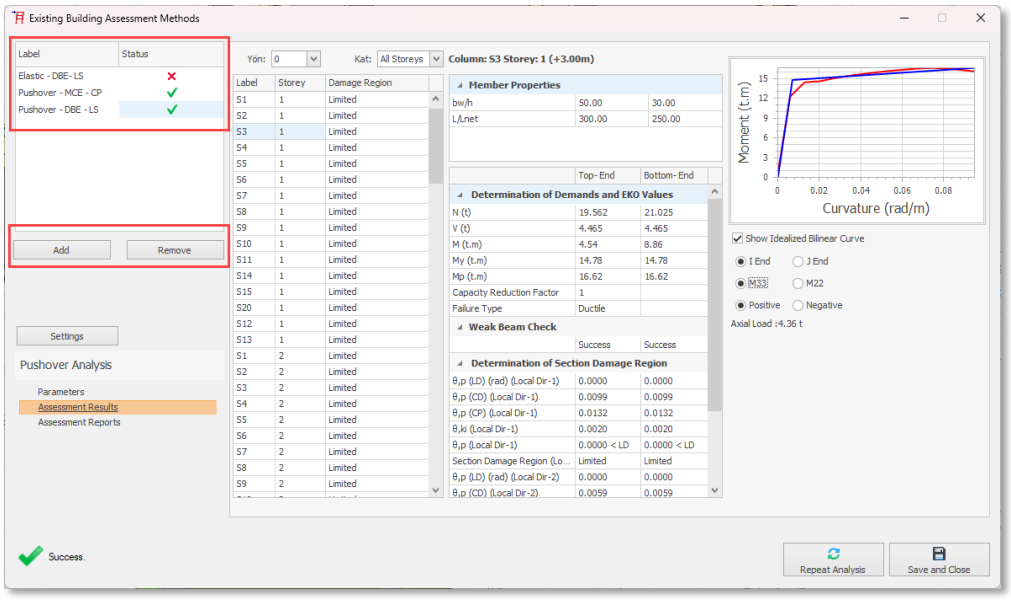
You can use the new "**Use Linear Model**" setting to force any member to behave linearly elastic during assessment. These members will be modelled with **elastic beam-column elements** in OpenSees integration.

New Building Assessment Interface

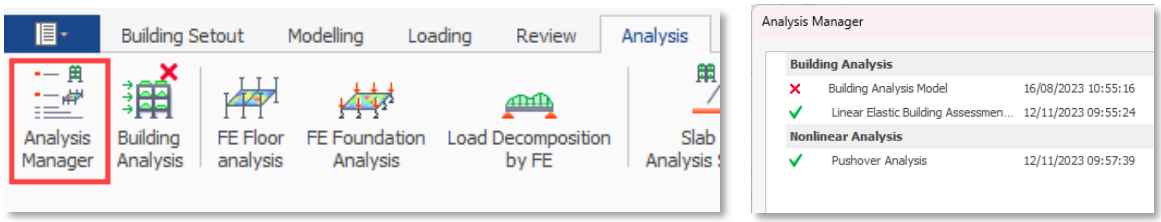
Multiple Analysis Management

In ProtaStructure 2024, the existing building assessment interface has also been renewed. Now, you can perform **as many separate assessment analyses as you want** with different parameters. For example, you can perform an Elastic Assessment that needs to ensure Controlled Damage under the DBE earthquake, and a separate Nonlinear Pushover Analysis that needs to ensure Collapse Prevention under the MCE earthquake by adding them to the list.

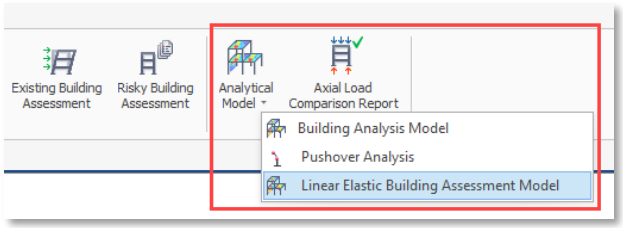
All the parameters and results of these analyses are kept ready in memory for you to access at any time. In previous versions, evaluation results were always overwritten and only a single type of analysis could be performed each time (only Elastic Evaluation or only Pushover, for instance).



Separate assessment analyses can also be managed through the **Analysis Manager**.



Similarly, analytical model and analysis results can be reviewed for each assessment analysis.

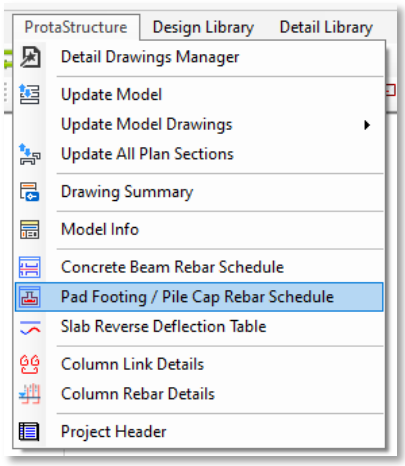


ProtaDetails

Pad Footing/Pile Cap Rebar Schedule

ProtaDetails 2024 now provides the reinforcement schedule for pad footings and pile caps. This table, like the beam reinforcement table, summarizes the foundation dimensions, associated columns, and reinforcement quantities. To create the rebar schedule:

1. Open ProtaDetails.
2. Select **ProtaStructure > Pad Footing / Pile Cap Rebar Schedule** command.
3. Show the top-left corner of the table on the screen.



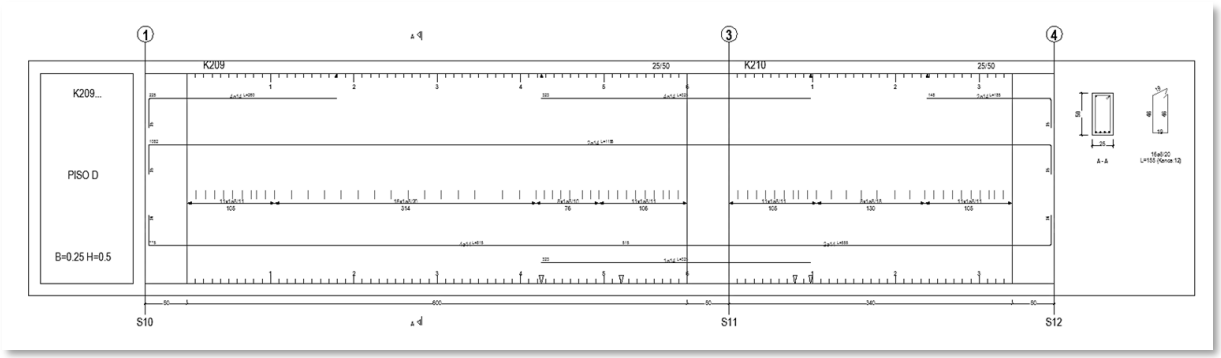
PAD FOOTING AND PILE CAP REBAR SCHEDULE

Member						Top Reinforcements		Bottom Reinforcements		Distribuion Rebars	
	Lx (cm)	Ly (cm)	H (cm)	Quantity	Columns	Top Rebar (X-Dir)	Top Rebar (Y- Dir)	Rebar X	Rebar Y	Distribution Rebar (X-Dir)	Distribution Rebar (Y-Dir)
F-S8	2300.0	2300.0	400.0	1	S8	1x12ø8/20	1x12ø8/20	1x12ø16/20	1x12ø16/20	1x1ø8/20	1x1ø8/20
F-S4	1800.0	2000.0	400.0	1	S4	1x10ø8/20	1x9ø8/20	1x10ø16/20	1x9ø16/20	1x1ø8/20	1x1ø8/20
F-S5	2300.0	2300.0	400.0	1	S5	1x12ø8/20	1x12ø8/20	1x12ø16/20	1x12ø16/20	1x1ø8/20	1x1ø8/20
F-S1	1400.0	1400.0	400.0	1	S1	1x7ø8/20	1x7ø8/20	1x7ø16/20	1x7ø16/20	1x1ø8/20	1x1ø8/20
F-S2	1800.0	1800.0	400.0	1	S2	1x9ø8/20	1x9ø8/20	1x9ø16/20	1x9ø16/20	1x1ø8/20	1x1ø8/20
F-S3	1300.0	1300.0	400.0	1	S3	1x7ø8/20	1x7ø8/20	1x7ø16/20	1x7ø16/20	1x1ø8/20	1x1ø8/20
F-S6	2500.0	2500.0	400.0	1	S6	1x13ø8/20	1x13ø8/20	1x13ø16/20	1x11ø16/20	1x1ø8/20	1x1ø8/20
PC-S7	1600.0	2100.0	500.0	1	S7	1x11ø16/20	1x8ø16/20	1x11ø18/20	1x8ø18/20	1x3ø16/23	1x3ø16/23
PC-S10	1100.0	1600.0	500.0	1	S10	1x8ø16/20	1x6ø16/20	1x8ø18/20	1x6ø18/20	1x3ø16/23	1x3ø16/23
PC-S11	1600.0	1600.0	400.0	1	S11	1x8ø16/20	1x8ø16/20	1x8ø18/20	1x8ø18/20	1x2ø16/23	1x2ø16/23
PC-S12	1600.0	1500.0	400.0	1	S12	1x8ø16/20	1x8ø16/20	1x8ø18/20	1x8ø18/20	1x2ø16/23	1x2ø16/23
PC-S9	1600.0	2600.0	600.0	1	S9	1x13ø16/20	1x8ø16/20	1x13ø18/20	1x8ø18/20	1x3ø16/23	1x3ø16/23

Ruler (Box) Style Beam Elevations

We have developed a new drawing style for beam elevations, which we call the "**Ruler or Box Style**". In this representation, instead of the schematic view of the beam axis, rebars are drawn inside a box and a ruler is shown marking the bar end points together with the links.

This style is preferred in some country practices. It is optional and can be turned on/off any time using **Settings > Beam > Detail Drawings > Drawing Style**. You can also use two different styles in the same drawing.

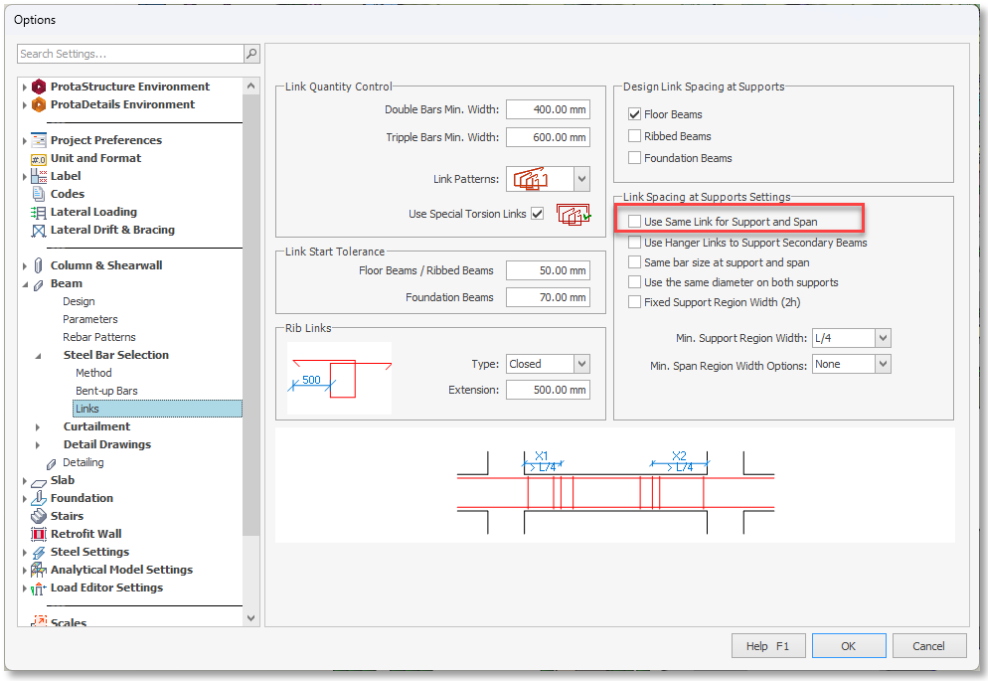


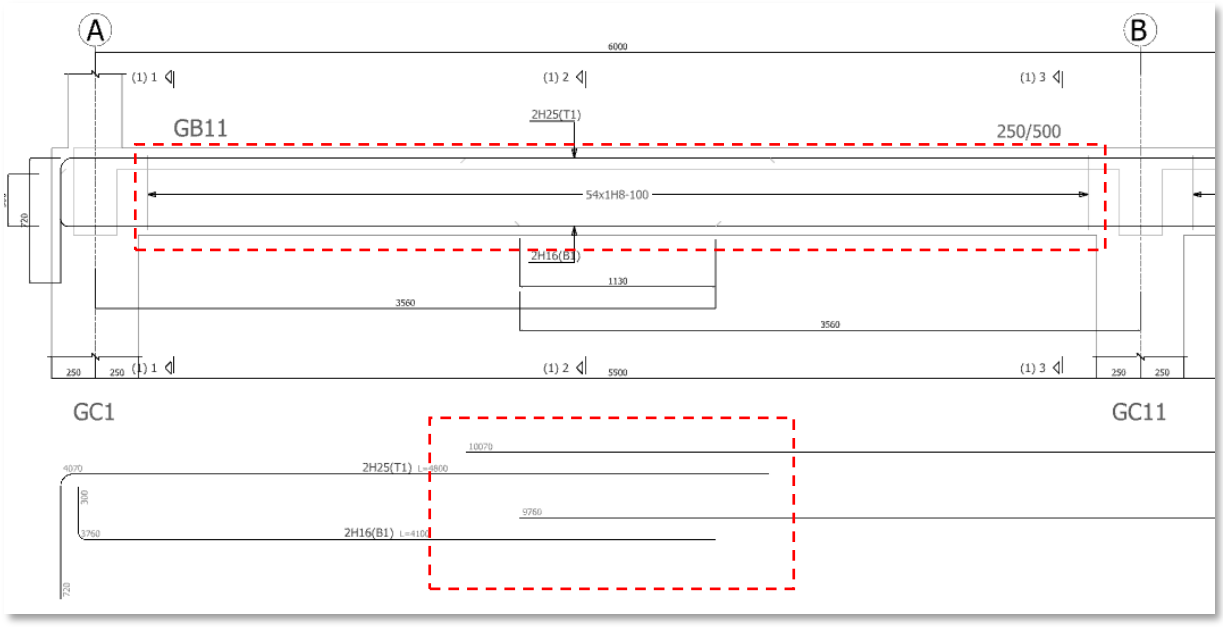
Use the Same Links for Beam Span and Support Regions

When “Standard Pattern 7” is used for RC beam rebar curtailment, the top and bottom lap splices will always be made in the mid-one-third region of the beams.

An additional setting is now introduced for confined link spacing in the span region. When the option is checked, the support link spacing will also be applied to span links, resulting in an overall confined beam in all spans.

The setting is accessible from the “**Settings/Beam/Steel Bar Selection/Links**” section.



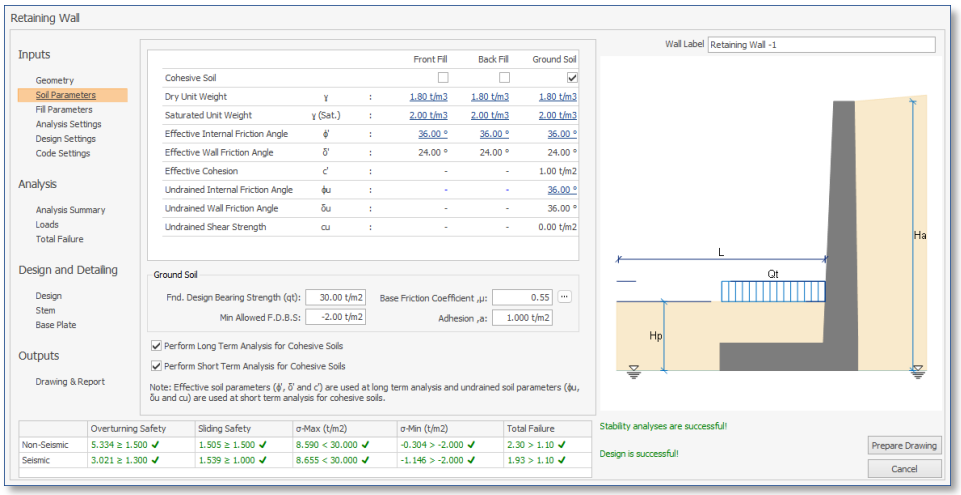


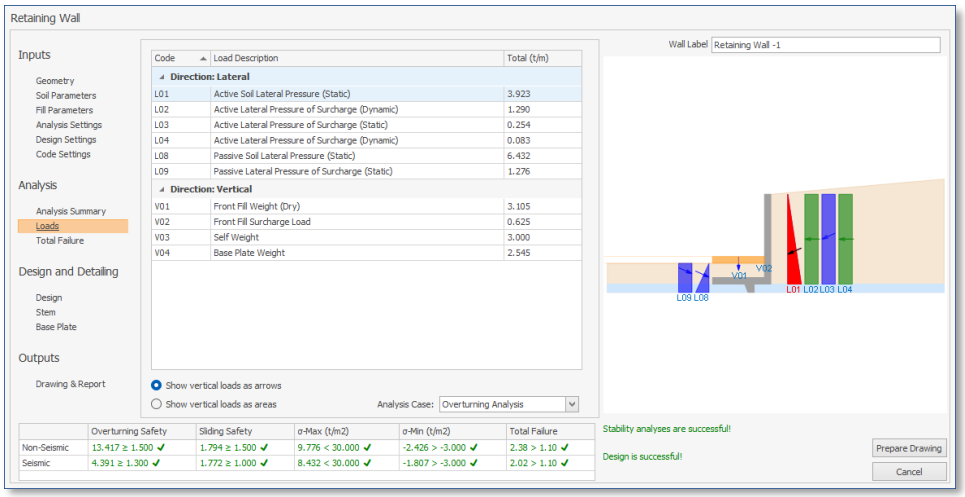
Design of Retaining Walls with No Heel Base Plate

The retaining wall design is more challenging when the heel base plate is not used. It is usually preferable to design with the heel for an optimum design. That's because the stability and soil pressure calculations will not be able to make use of the favorable effect of the backfill when the heel base plate is absent.

There may be situations when you need to design a retaining wall with no heel base plate, such as the problematic excavation of the backfill and no practical way to build the base plate.

With the latest update of ProtaStructure, the retaining wall design algorithm and data input is revised to allow the design of retaining walls with no heel. However, you must note that the stability and foundation soil pressure checks may be harder to satisfy. You may need to take additional precautions, such as designing a **more extended toe base plate** or **a shear key**.

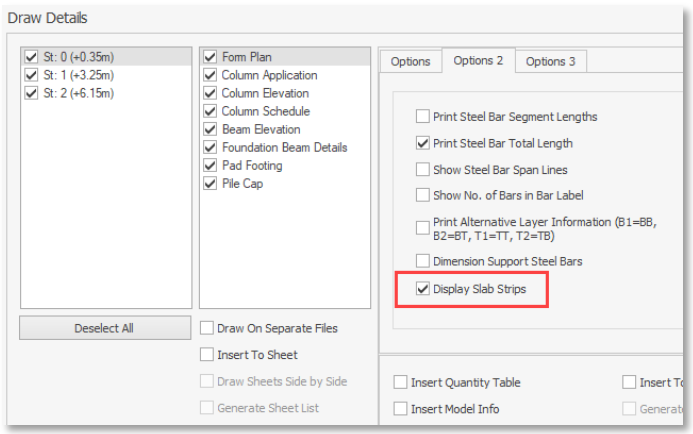


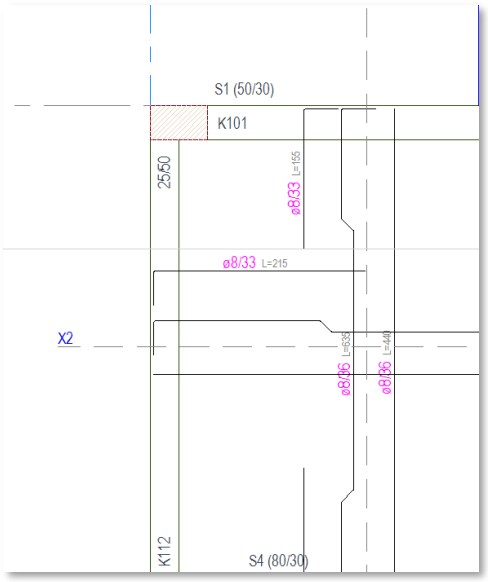


Normally, this feature has been introduced with the ProtaStructure 2022 6.0.512 maintenance update. It is repeated here since it was a high voted user request.

Slab Strips in Floor Plans

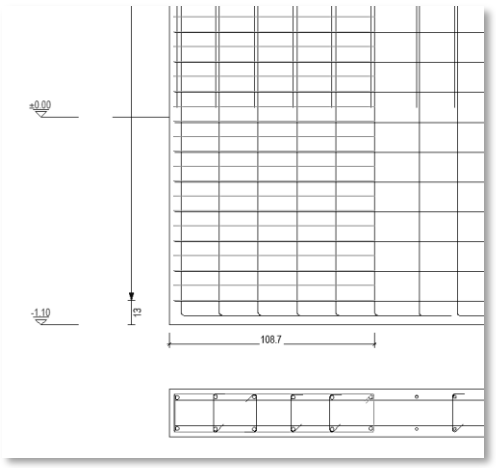
As per our users' request, the slab strips are now included in the floor plan drawings. You can turn on/off this feature using the **Detail Drawing Manager > Form Plan > Options 2 > Display Slab Strips** option.

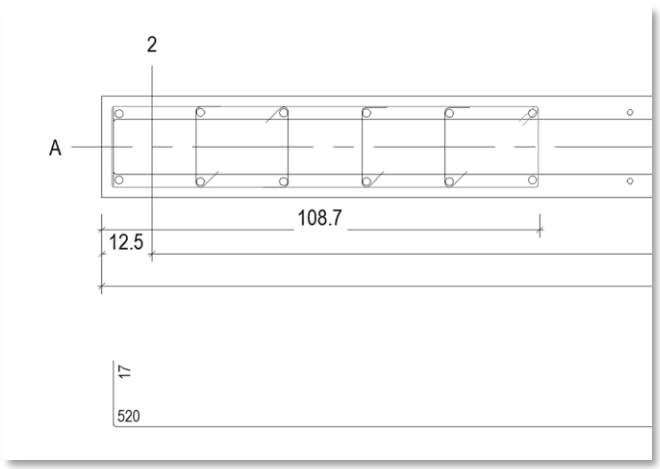




Shearwall End Zone Dimensions

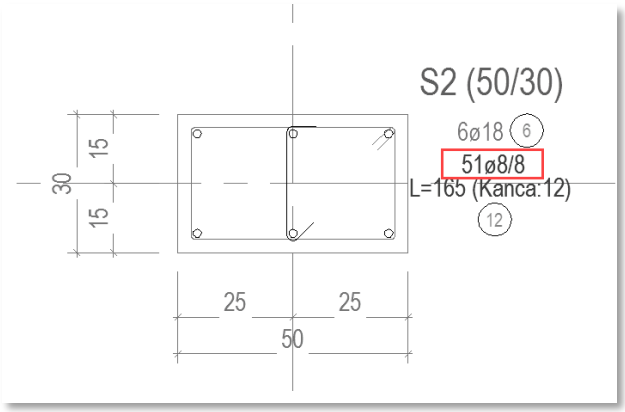
Shearwall end zones can now be dimensioned in column application plans and shearwall elevation drawings.





Link Labels in Column Application Drawings

Link labels are now indicated in column application plans in ProtaDetails 2024.



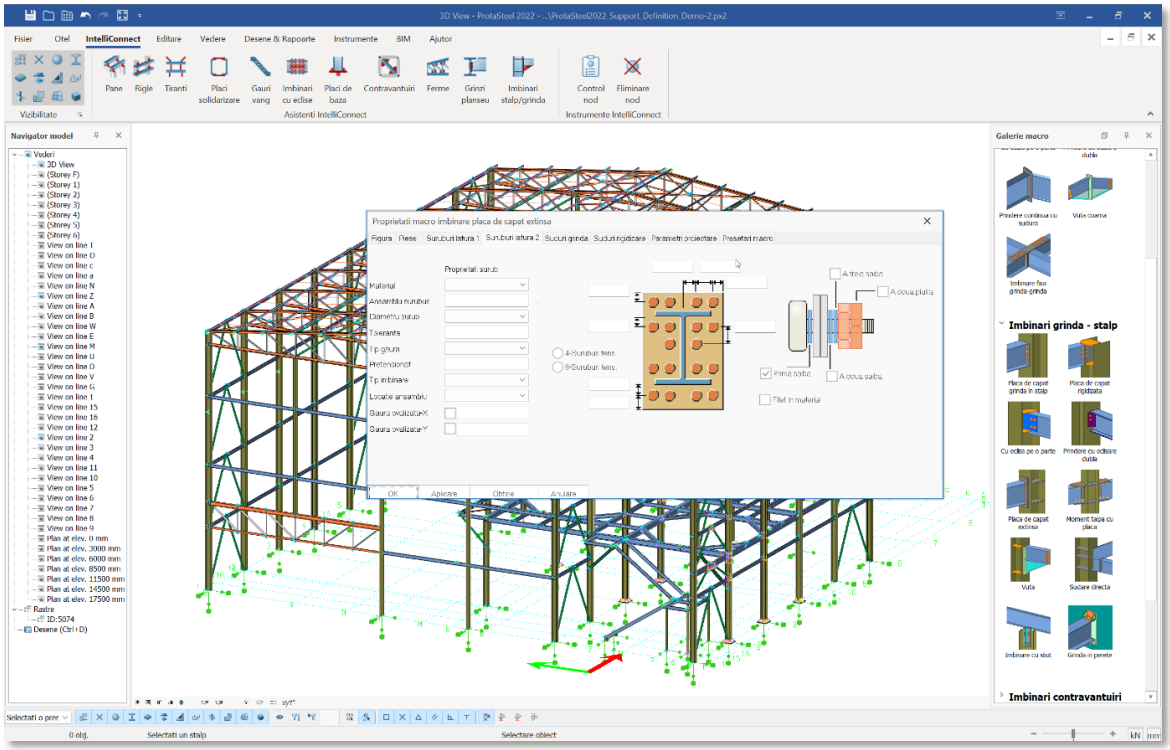
Stability and Performance Improvements in ProtaDetails

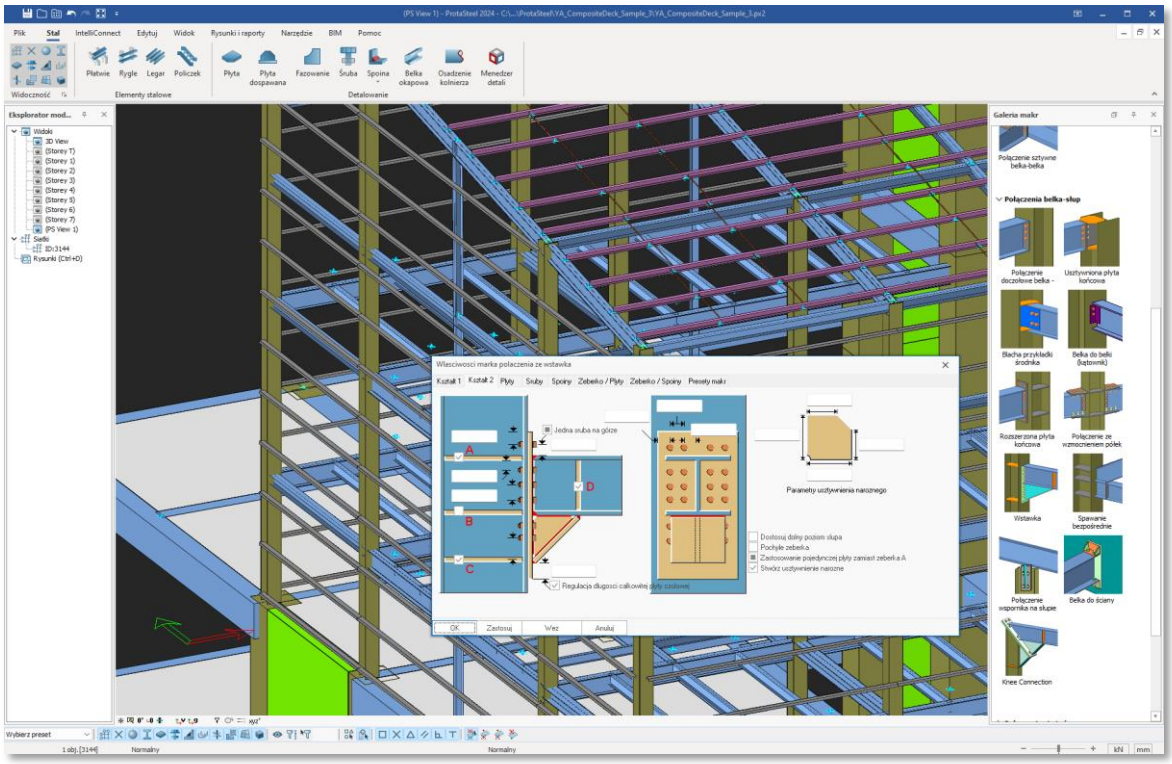
With ProtaDetails 2024, performance and stability improvements have been made in detail drawings based on user feedback and test studies. While working on ProtaDetails 2024, improvements made last year have also been delivered to our users without delay through ProtaStructure 2022 updates.

ProtaSteel

Romanian and Polish Language Support in ProtaSteel

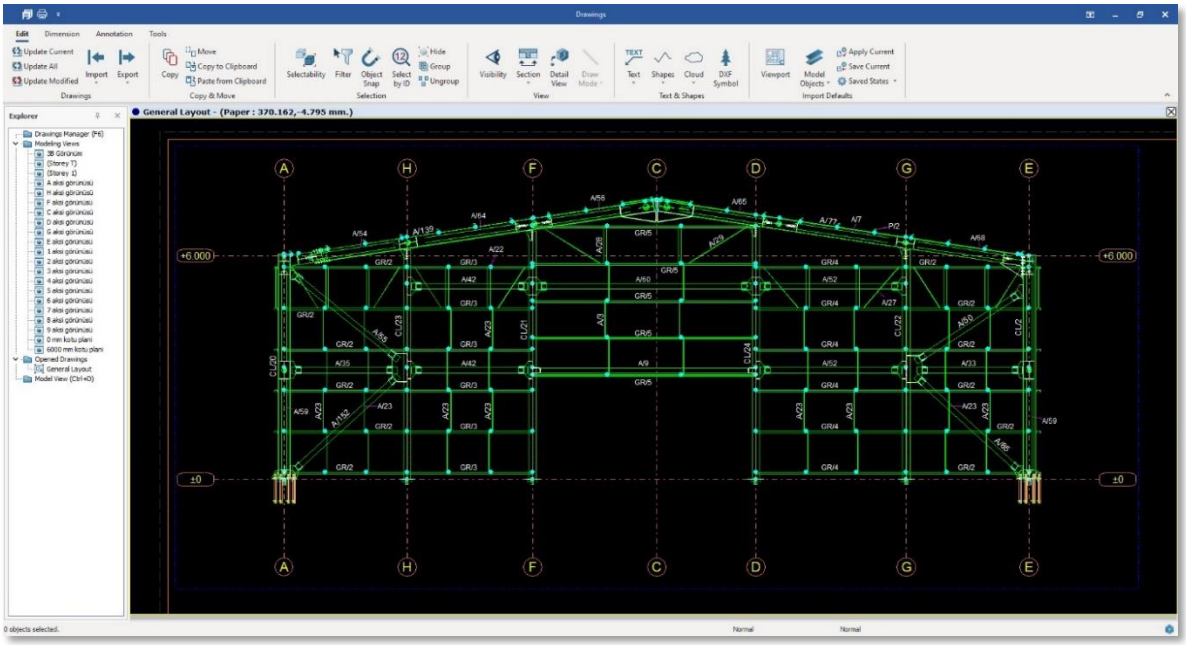
ProtaSteel now supports the Romanian and Polish language. We have achieved a translation percentage of 90%, which covers the most used functions and user interfaces, such as toolbar commands, tooltips, drawing manager UI, connection macro windows, clash check, assembly, part drawing settings, etc.





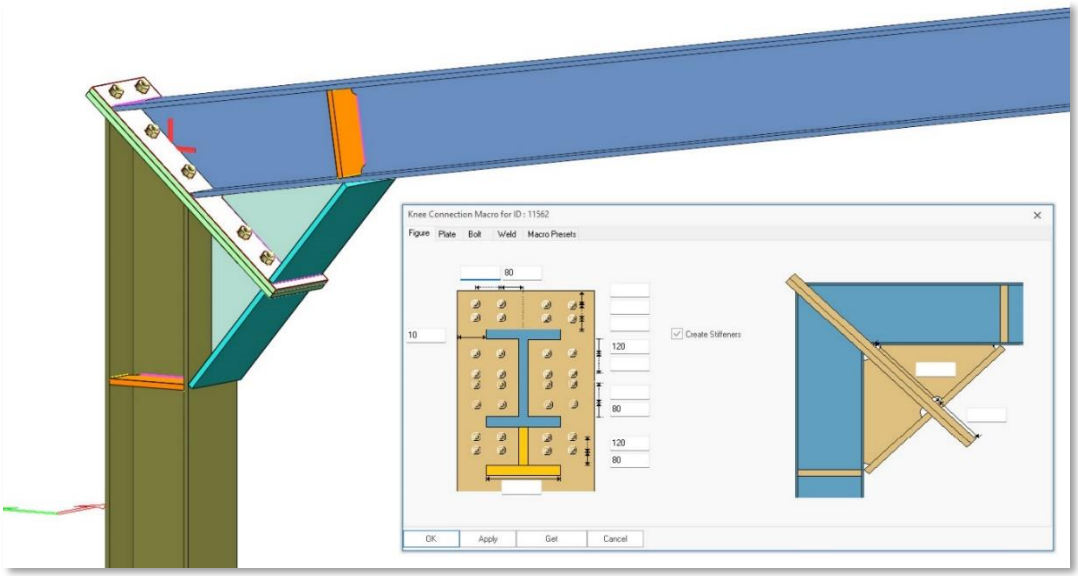
New Drawing Module

ProtaSteel drawing module is now much more user-friendly and good looking with a well-organized ribbon toolbar. It is now easier than ever to create general arrangement drawings, put annotations and dimensions, and review fully automated part and assembly drawings.



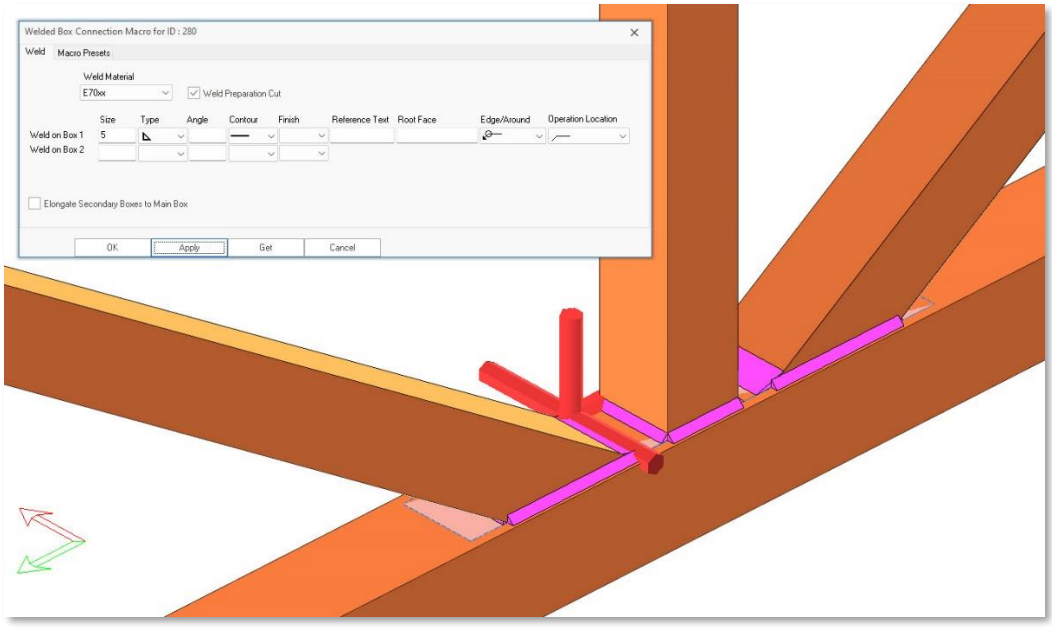
New Macro: Knee Connection

The new Knee Connection macro is introduced with ProtaSteel 2024. It is similar to apex haunch connection. But you can use this to connect portal rafters to columns where you don't want to use a standard haunch connection.



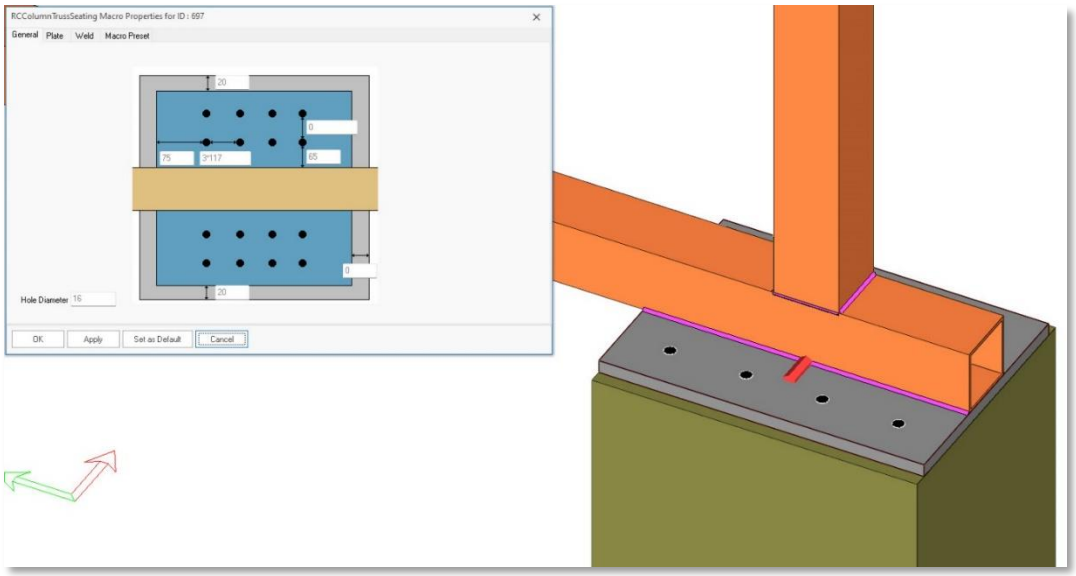
New Macro: Welded Box Profile Connections

Previously we had welded connection for pipe profiles. With ProtaSteel 2024, you can now connect box sections with welded connection. This is especially useful in trusses where the members are usually designed with box profiles.



New Macro: RC Column – Truss Seating Connection

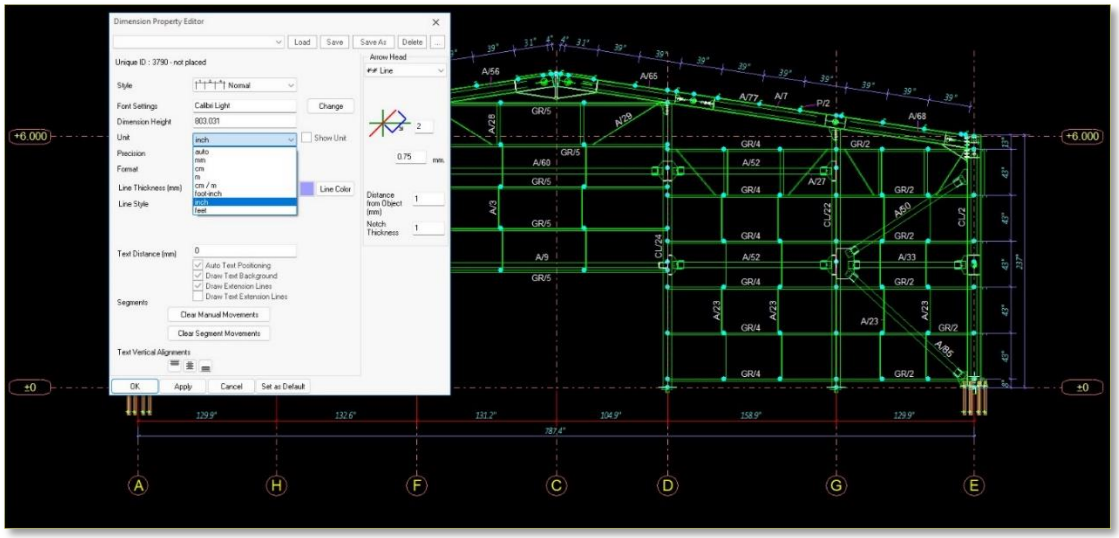
There are cases where you need to support trusses on RC columns. Now, ProtaSteel 2024 has you covered with the new RC Column – Truss Seating Connection.



Imperial Units in Drawing Module

With ProtaSteel 2024, you can now change the dimension display to imperial units. You can choose one of inch, feet or foot-Inch fractional options.

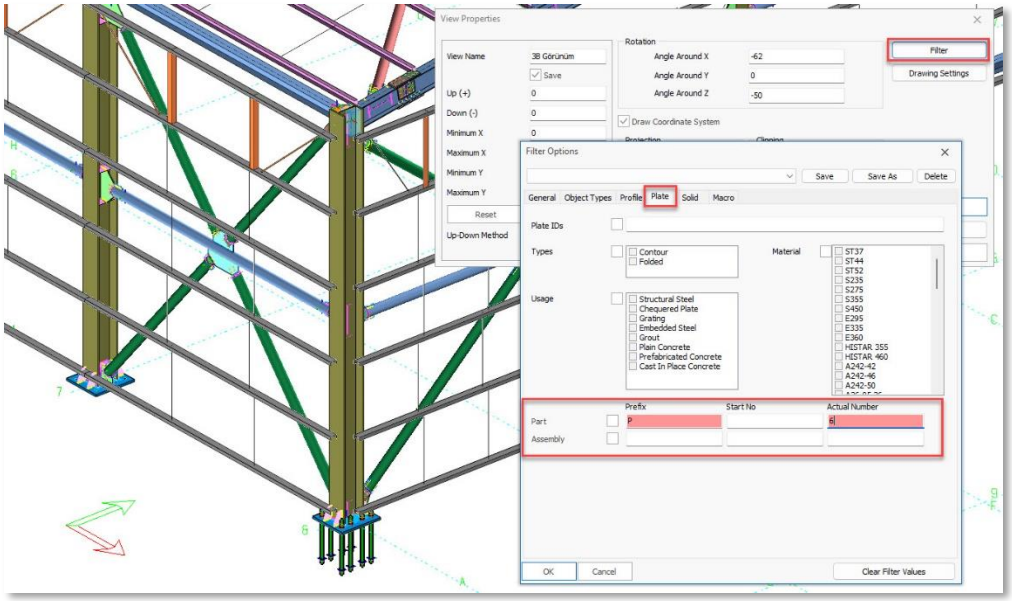
The last choice is remembered, and dimensions are updated automatically.



Filtering Parts

For a typical steel project, the number of different parts can increase significantly, and it is easy to lose track of components.

With the new version, you can now filter the parts with their assigned prefixes and numbers. So that, you can see where they are used. It is a powerful way to review your project.

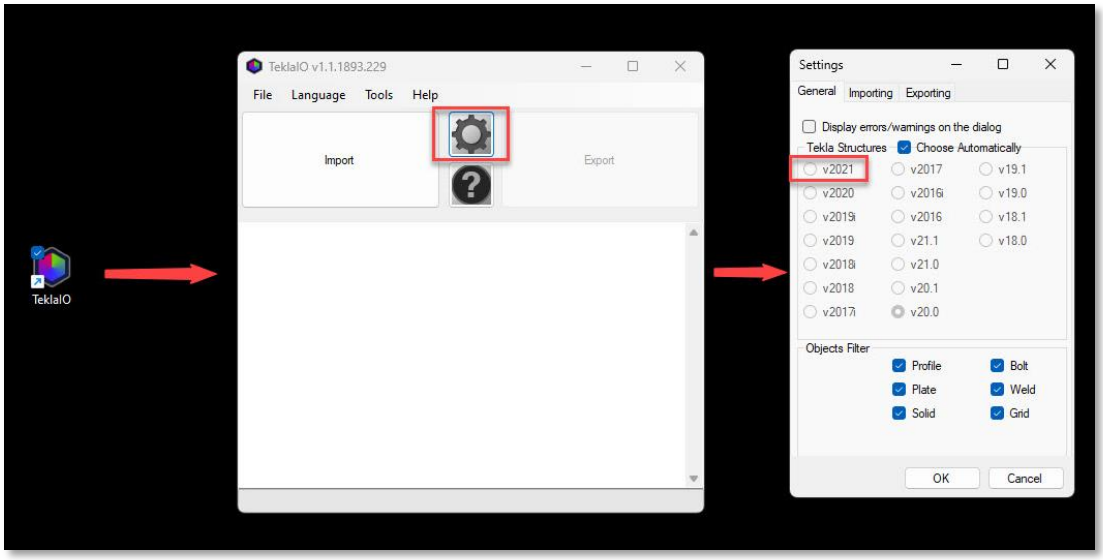


Updated Tekla IO

With ProtaSteel, you can create NC files and go to fabrication directly.

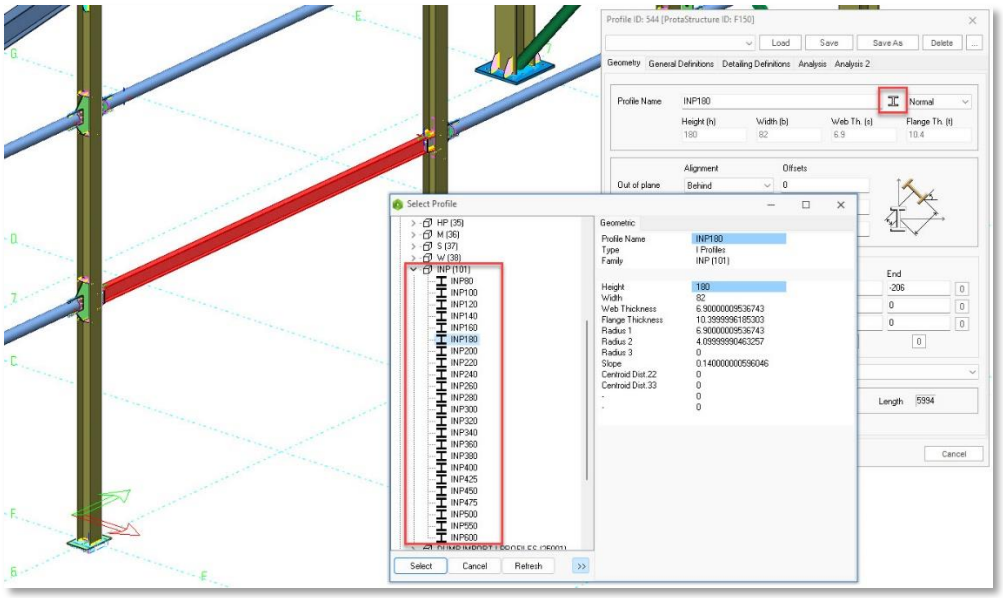
However, one of the powerful features of ProtaSteel is the ability of communicating a fully connected constructable model to other fabrication platforms in the industry.

With the latest ProtaSteel, you can export the fully connected model to Tekla Structures 2021. All you must do is to download and install our latest free plugin.



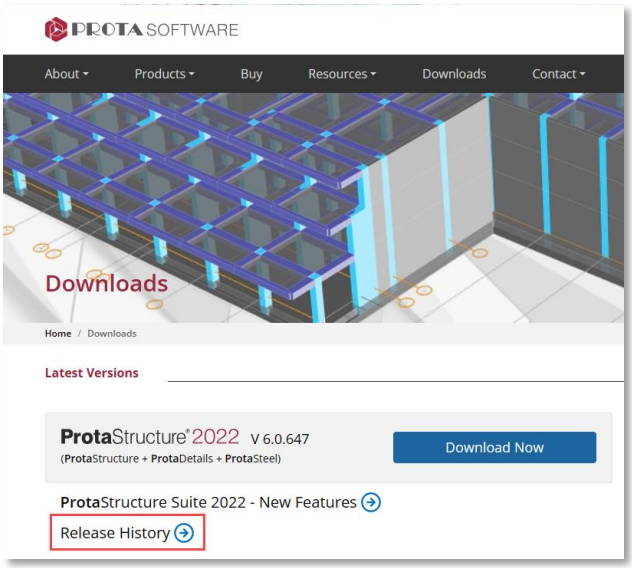
INP Profile Catalog

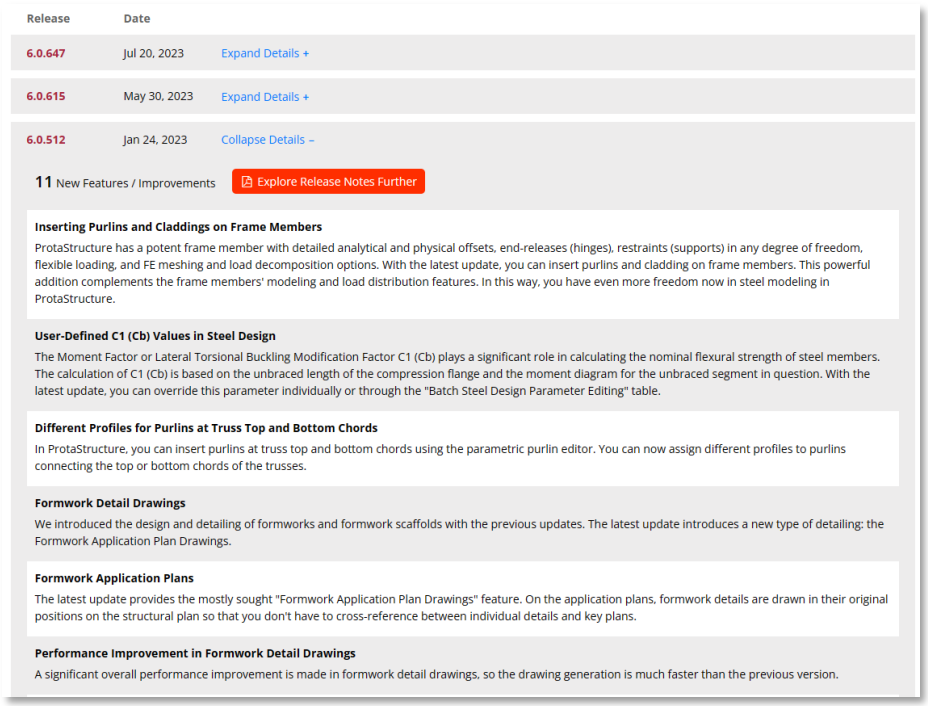
INP profile catalogue is now included in ProtaSteel 2024. You can model the steel members with INP profiles in ProtaStructure and send them over to ProtaSteel. They will be automatically recognized.



General Stability and Performance Improvements

Based on our quality control procedures and user feedback, significant stability and performance improvements have been made to all products in the **2024 family**. Most of these improvements and new features have been delivered to you throughout the year with the ProtaStructure 2022 maintenance updates. The details of these updates can be found in the "**Release Notes**" section on our website with references to relevant customer ticket numbers.





New Design Guides and Videos

We have been working hard to provide extensive documentation, videos, and webinars on numerous topics to improve your experience and knowledge of ProtaStructure.

New design guides have been published with ProtaStructure 2024 release.

Visit Prota Help Center's Knowledge Base and our website's 'Whitepaper' section to learn about the latest design guides.

Prota Help Center Knowledge Base:
https://support.protasoftware.com/

Protasoftware.com Whitepapers:
https://www.protasoftware.com/whitepaper

In addition to design guides, we have been conducting new webinars and publishing the recordings. Also, our technical team is always busy creating video content on ProtaStructure features, usage, and theory. Please subscribe to our YouTube channel to get notified of new content.

https://www.youtube.com/c/ProtaStructure

Thank You

Thank you for choosing the ProtaStructure Suite product family.

At Prota, it is our continual aim to provide you with user-friendly, industry-leading technology for building design and documentation.

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

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

The Prota Team

ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®