ProtaStructure Suite 2025 – New Features

Version: 2.0 July 2024

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

asiasupport@protasoftware.com | globalsupport@protasoftware.com

Publisher

PROTA SOFTWARE

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 40 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 2025 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 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 2025**. You can find the details on the pages to follow.

Thank you for choosing ProtaStructure.

ProtaStructure 2025: A Wealth of New Features

In our latest major release, ProtaStructure 2025, we're excited to introduce a host of new features that enhance user experience and boost efficiency.

In ProtaStructure 2025, our primary focus has been on extending existing code support while enhancing stability, performance, and addressing user feedback.

Here are the **top highlights** of ProtaStructure 2025:

Comprehensive Support for RC Design and Detailing: We now offer more comprehensive support for non-seismic and seismic reinforced concrete (RC) design and detailing in various countries, including the US, Colombia, Peru, Dominican Republic, Philippines, Indonesia, India, and Thailand. Detailed calculation reports are provided.

Seismic Checks for Steel Buildings: Our software now includes extended support for seismic checks in steel buildings, following standards such as AISC341 and TBDY2018.

Shearwall Moment and Shear Seismic Envelopes (Eurocode 8): ProtaStructure 2025 incorporates shearwall moment and shear seismic envelopes (tension shift) based on Eurocode 8.

Vertical Modal Earthquake Analysis: Specifically designed for Eurocode 8 and other codes, this feature accurately analyzes the vertical effect of the earthquake for special cases such as large span beams, slabs, cantilevers, discontinuities, and transfer members.

Dedicated ULS and SLS Combination Groups: We've introduced dedicated combination groups for ultimate limit state (ULS) and serviceability limit state (SLS) checks in both steel and RC members. Additionally, the load combination editor has been improved to provide better user experience and visual presentation.

Orthotropic Shell Elements for One-Way Slabs: You can model and analyze cast-in-place or prefabricated one-way slabs (such as hollow core slabs) using the orthotropic shell elements. This is a significant improvement that will accelerate your designs.

Multi-Mode Pushover Analysis (MPA): Assess irregular buildings with higher mode effects using our new multi-mode pushover analysis feature.

ProtaSteel: The Best Just Getting Better: ProtaSteel now supports IdeaStatica Export, features new connection macros, allows partial model updates, and offers extended localization.

New Modeling Tools: Explore our new modeling tools, including Space Trusses, Castellated Beams, Copy/Move Special and Match Properties.

Improved Analysis Post-Processor: The analysis post-processor has been enhanced with features like pushover and time-history result visualization, window selection, improved solid model view, and show/hide members functionality. Performance improvements are also part of this update.

Delving Into Key Areas…

This major release introduces a wealth of new features, providing users with an even more powerful and efficient experience. Below, we'll delve into the key areas of improvement:

Enhanced Code Support

- **RC Seismic and Non-Seismic Design and Detailing:** Improved support for ACI318 and similar standards.
- **User-Defined Strength Reduction Factors:** Enter user-defined strength reduction factors to cater for versions and variations of ACI318.
- **Long-Term RC Deflection Checks:** Calculate vertical deflections by considering existing reinforcement, cracking status and long-term load factors, and compare them to code limits.
- **Steel Seismic Design Checks:** Extended support for AISC341 and TBDY2018 to check additional seismic provisions such as seismic compactness, eccentric braces and more.
- **Indian Standards:** Improved support for RC column and shearwall design and detailing (including seismic provisions) according to Indian Standards.
- **Eurocode 8:** Tension shift, bending moment and shear envelopes for DCH and DCM structures.
- **Thailand RC Design Standards:** Customized strength reduction factors compliant with Thailand RC design codes.
- **Drift Scaling in RSA and EQS in All Applicable Codes:** Drift calculations and scaling are subject to different rules in different seismic codes. The outlined procedures are now applied more rigorously in ProtaStructure
- **Shearwall End Zones (Boundary Elements):** Batch editing inside and outside the critical wall height.

Design Improvements

- **Improved Load Combination Editor:** Remembers combination generation preferences for and efficient workflow supported by a fresh and neat visual representation.
- **Separate Combination Groups for RC and Steel:** Separate groups for RC and steel members, covering ULS and SLS combinations.
- **SLS Combinations for Serviceability Checks:** User-defined SLS combinations to cover more serviceability scenarios.
- **Notional Load Improvements:** They are now added optionally to all combinations for considering imperfections.

- **Orthotropic Shell Elements:** Model and analyze one-way slabs with orthotropic shell elements capturing a more accurate behaviour.
- **Purlin/Girt Design Option:** Design purlins/girts for non-seismic combinations only.
- **Slab Design Enhancement:** Include manual strip rebars in the report.
- **RC Beam Design Seismic Checks:** Detailed design reports are improved to cover more information on RC beam seismic checks according to ACI318 and similar codes and Indian Standards.
- **Include RC Beam Batch Design Messages in Beam Design UI:** Batch design messages are at your disposal on the ribbon bar for quick access.

Advanced Earthquake Engineering

- **Vertical Modal Earthquake Analysis:** Accurately assess seismic behavior.
- **Multi-Mode Pushover Analysis:** Assess irregular buildings with higher mode effects using our new multi-mode pushover analysis feature.
- **Stability and Performance Improvements:** Comprehensive enhancements due to user feedback, more variety of existing building scenarios and testing.

New Modeling Tools

- **Space Trusses:** Easily create, analyze and design space truss structures.
- **Castellated/Honeycomb Beams (Without Design):** Explore innovative beam designs.
- **Copy/Move Special:** Streamline model manipulation.
- **Match Member Properties:** Equate the member properties using the new Format Painter tool.
- **Quick Load Assignment for Claddings:** Simplify load assignment.
- **Storey Visibility Filter in Right-Click Menu:** Isolating the active storey is now a breeze with a single click.
- **Cantilever Mark in 3D View:** The cantilever marks shown on plan view are now available in 3D.
- **More Flexible Column End Releases:** Assign independent end releases in two different directions.
- **Continuous purlins:** Model purlins and girt so that they are automatically cut at a predefined member length. In between the cut points, they are treated as continuous, and end releases are assigned accordingly. They are also exported to ProtaSteel in this manner.
- **More Girt Section Alignment Options:** Girt section alignment options are extended to support more variations such as 'Centered', 'Align to Top', 'Align to Bottom'.

- **Insert Purlins Using Multiple Members:** You can now directly select the primary members which will support the purlins for more precision.
- **Relabel Axes:** Use auto axis relabeling to get things back in order when the axis labels mix up as you introduce new ones during modeling.
- **Twin Face-to-Face L Sections in Steel Profile Catalog:** Face-to-face twin L-profiles are the new addition to our compound steel profile catalog.
- **Modeling Improvements:** Enjoy a more intuitive modeling process.

Advanced BIM Integration Tools

- **IdeaStatica Export:** Communicate steel connections to IdeaStatica from ProtaSteel to perform a CBFEM analysis on the connections you create in ProtaSteel.
- **Improved IFC, SAF and DXF Communication:** Enhancements are made considering wider range of user scenarios and feedback.
- **Quantity Take-Off by Concrete Grade:** Get a categorized quantity take-off in your projects using multiple concrete grades.

Analysis Post-Processing

- **Visualization of Pushover and Time-History Results:** View step-by-step nonlinear analysis results with animated time-history displacement and plastic hinge mechanism. Hinge plastic states are color coded in real time.
- **Window Selection and Show Values for Selected Only:** Select multiple members or nodes by boxing around them. Review the displacement values of only what you have selected.
- **Solid View Around Analytical Lines:** See the physical members as faint extrusions around the analytical lines. This will give a better grasp of how analytical model integrates with the physical model.
- **Displaced Solid View with FE Contours:** See the displacements on the solid model with FE contours.
- **Hide/Show Selected Members:** Filter any selected elements out for a more focused review of the analysis results.
- **Show/Hide Minimum A_s in Rebar Contours**

RC Detailing

- **Ruler (Box) Style Beam Elevations:** Employ a new beam elevation style where the bars and splices are shown against a ruler in a scaled box.
- **Adjustable Beam Critical Zone Length:** Enter user-defined values for beam plastic region lengths for more control over ductile design and detailing.
- **Top and Bottom Bar Splices 2H Away From Supports:** For US codes and similar, ductile beam detailing is improved with new detailing options.
- **New Beam Rebar Pattern:** Standard Pattern 8 offers more control over beam rebar curtailment.
- **Rebar Table for Pad Footings and Pile Caps:** Include the pile cap and pad footing rebars in a tabulated manner in your detail drawings.
- **Export Quantity Tables to Excel with Shape Codes:** Enjoy a direct export to Excel with bar shapes automatically displayed.
- **Stability and Performance Improvements:** Comprehensive enhancements due to user feedback, scenarios and testing.

ProtaSteel Enhancements

- **IdeaStatica Export:** Communicate steel connections to IdeaStatica from ProtaSteel to perform a CBFEM analysis on the connections you create in ProtaSteel.
- **Exposed Base Plate with Extended Anchor Bolts for Cruciform Steel Columns:** You can now assign base plates to king-cross steel columns. This connection can be used for other profile types as well and does not support auto design yet.
- **Curved Beams End Plate Connection:** Assign pin and moment connections at the ends of the curved beams with these new macros.
- **Extended End Plate Connection with Additional Tension Bolts:** This prequalified seismic connection is improved to support additional tension bolts.
- **Partial Model Update Between ProtaSteel and ProtaStructure:** No worries if you need to update only a part of your model between ProtaStructure and ProtaSteel. This feature has you covered in your large-scale projects when you need to update only a small part.

Enhanced Code Support

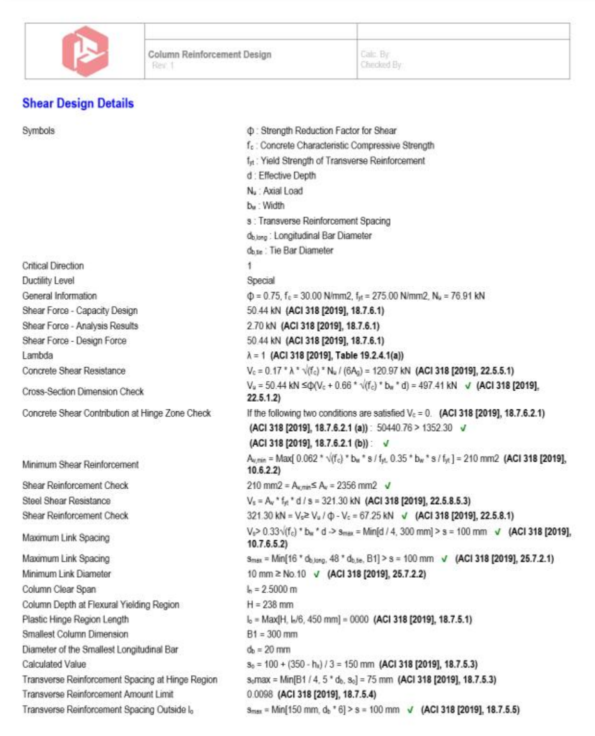
RC Column, Wall and Beam Design

ProtaStructure 2025 is now more capable in terms of supporting seismic and non-seismic design provisions of international seismic and design codes such as ACI318 (including Chapter 18), Colombia, Peru, Dominican Republic, India, Indonesia, Philippines and Thailand.

We are now more rigorously applying ductile/non-ductile design principles and clauses.

Our advanced seismic analysis and design technology ensures compliance with demanding earthquake regulations, automating checks for irregularities and capacity design.

But that's not all—our step-by-step calculation reports provide transparency and confidence throughout the design process.



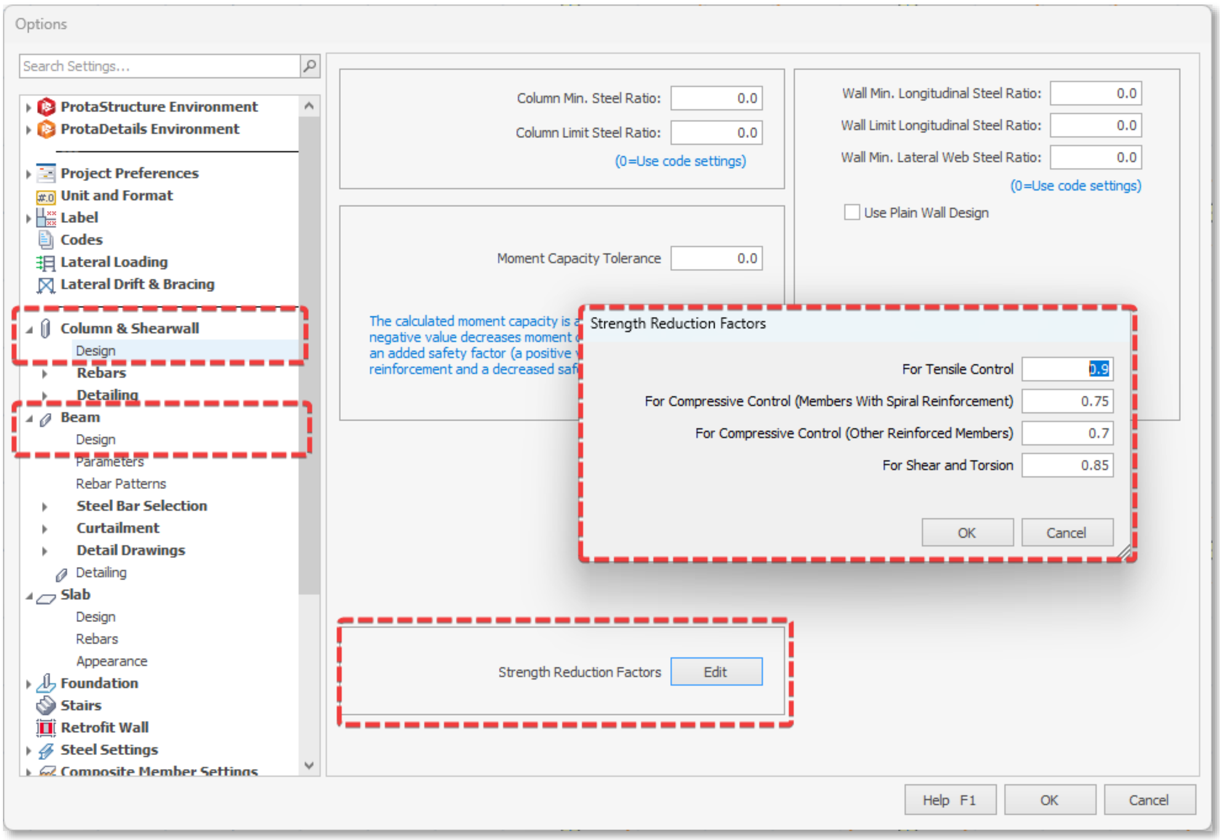
	Column Reinforcement Design Rev: 1	Calc. By: Checked By:

Shear Design Details

Symbols	Φ : Strength Reduction Factor for Shear f_c : Concrete Characteristic Compressive Strength f_{yt} : Yield Strength of Transverse Reinforcement d : Effective Depth N_u : Axial Load b_w : Width s : Transverse Reinforcement Spacing $d_{b,long}$: Longitudinal Bar Diameter $d_{b,tie}$: Tie Bar Diameter
Critical Direction	1
Ductility Level	Special
General Information	$\Phi = 0.75$, $f_c = 30.00$ N/mm2, $f_{yt} = 275.00$ N/mm2, $N_u = 76.91$ kN
Shear Force - Capacity Design	50.44 kN **(ACI 318 [2019], 18.7.6.1)**
Shear Force - Analysis Results	2.70 kN **(ACI 318 [2019], 18.7.6.1)**
Shear Force - Design Force	50.44 kN **(ACI 318 [2019], 18.7.6.1)**
Lambda	$\lambda = 1$ **(ACI 318 [2019], Table 19.2.4.1(a))**
Concrete Shear Resistance	$V_c = 0.17 * \lambda * \sqrt{(f_c)} * N_u / (6A_g) = 120.97$ kN **(ACI 318 [2019], 22.5.5.1)**
Cross-Section Dimension Check	$V_u = 50.44$ kN $\leq \Phi(V_c + 0.66 * \sqrt{(f_c)} * b_w * d) = 497.41$ kN ✓ **(ACI 318 [2019], 22.5.1.2)**
Concrete Shear Contribution at Hinge Zone Check	If the following two conditions are satisfied $V_c = 0$. **(ACI 318 [2019], 18.7.6.2.1)** **(ACI 318 [2019], 18.7.6.2.1 (a))** : 50440.76 > 1352.30 ✓ **(ACI 318 [2019], 18.7.6.2.1 (b))** : ✓
Minimum Shear Reinforcement	$A_{v,min} = Max[\,0.062 * \sqrt{(f_c)} * b_w * s / f_{yt}, 0.35 * b_w * s / f_{yt}\,] = 210$ mm2 **(ACI 318 [2019], 10.6.2.2)**
Shear Reinforcement Check	210 mm2 = $A_{v,min} \leq A_v = 2356$ mm2 ✓
Steel Shear Resistance	$V_s = A_v * f_{yt} * d / s = 321.30$ kN **(ACI 318 [2019], 22.5.8.5.3)**
Shear Reinforcement Check	321.30 kN = $V_s \geq V_u / \Phi - V_c = 67.25$ kN ✓ **(ACI 318 [2019], 22.5.8.1)**
Maximum Link Spacing	$V_s > 0.33\sqrt{(f_c)} * b_w * d \rightarrow s_{max} = Min[d / 4, 300 mm] > s = 100$ mm ✓ **(ACI 318 [2019], 10.7.6.5.2)**
Maximum Link Spacing	$s_{max} = Min[16 * d_{b,long}, 48 * d_{b,tie}, B1] > s = 100$ mm ✓ **(ACI 318 [2019], 25.7.2.1)**
Minimum Link Diameter	10 mm ≥ No.10 ✓ **(ACI 318 [2019], 25.7.2.2)**
Column Clear Span	$l_n = 2.5000$ m
Column Depth at Flexural Yielding Region	H = 238 mm
Plastic Hinge Region Length	$l_o = Max[H, l_n/6, 450 mm] = 0000$ **(ACI 318 [2019], 18.7.5.1)**
Smallest Column Dimension	B1 = 300 mm
Diameter of the Smallest Longitudinal Bar	$d_b = 20$ mm
Calculated Value	$s_o = 100 + (350 - h_x) / 3 = 150$ mm **(ACI 318 [2019], 18.7.5.3)**
Transverse Reinforcement Spacing at Hinge Region	$s_o max = Min[B1 / 4, 5 * d_b, s_o] = 75$ mm **(ACI 318 [2019], 18.7.5.3)**
Transverse Reinforcement Amount Limit	0.0098 **(ACI 318 [2019], 18.7.5.4)**
Transverse Reinforcement Spacing Outside l_o	$s_{max} = Min[150 mm, d_b * 6] > s = 100$ mm ✓ **(ACI 318 [2019], 18.7.5.5)**

User-Defined Strength Reduction Factors

ACI318 is broadly adopted by different countries. However, a different version of ACI318 is in effective use in these countries. In addition to that, slight variations exist based on local requirements and practice. To support this better, we have now introduced the “User-Defined Strength Reduction Factors” for ACI318 to be used in design. This is accessible from **Settings > Column&Shearwall > Design** or **Settings > Beam > Design** both controlling the same parameters.



Long-Term RC Deflection Checks

ProtaStructure now supports **long-term detailed RC deflection checks** according to ACI318-2014, ACI318-2019, NSR10-C, NTE060, SNI2847, NSCP2015 (and Indian Standards, IS456-200 work in progress as of 29.07.2024).

In this check, the effective section inertias are calculated considering the provided reinforcement and design moments. Then, support and span cracking status are determined. Deflections are obtained using the effective inertias according to the SLS combinations defined by the user and the serviceability conditions given in the codes.

Long term deflections are also calculated using code-specified time-dependent factors and sustained load durations. Results are given in a detailed tabulated report.

Concrete Beam Deflection Check

7B4 - Storey: 7

Flexural Member Type	: Beam
Span Type	: Exterior Span
Clear Span / Height	: 4700 / 500 mm = 9 ≤ 18.63 (ACI318-19 Table 9.3.1.1)

Section Dimensions:	250 / 500 mm	d = 442 mm	d' = 58 mm
Flange Dimensions:	b = 908 mm	h_f = 120 mm	
Uncracked Section:	I_c = 4.446E-03 m^4	y(b) = 324 mm	y(t) = 176 mm
Concrete:	f_c' = 30.00 N/mm^2	f_r = 5.10 N/mm^2	E_c = 25743.0 N/mm^2
Steel:	f_y = 415.00 N/mm^2	E_s = 200000.0 N/mm^2	E_s/E_c = 8

	Left Support	Right Support	Span
$A_s(t)/A_s(b)$	982 / 402 mm^2	982 / 402 mm^2	982 / 402 mm^2
M(g) / M(g+q)	-42.4 / -56.2 kN.m	-30.0 / -38.9 kN.m	27.3 / 36.0 kN.m
M_{cr}	128.5 kN.m	128.5 kN.m	70.1 kN.m
Cracking Status (G)	NO	NO	NO
Cracking Status (G+Q)	NO	NO	NO
Cracked Inertia (G)	-	-	-
Cracked Inertia (G+Q)	-	-	-
Effective Inertia (G)	4.446E-03 m^4	4.446E-03 m^4	4.446E-03 m^4
Effective Inertia (G+Q)	4.446E-03 m^4	4.446E-03 m^4	4.446E-03 m^4

Effective Inertia (G)	4.401E-03 m^4
Effective Inertia (G+Q)	4.401E-03 m^4
Immediate Def. (G)	6.8997E-04 m
Immediate Def. (G+Q)	9.0830E-04 m
Immediate Def. (Q)	2.1832E-04 m
Loading Time	FIVE YEARS or MORE
Total Time Dep. Deflection	0.002 m

Flexural Member and Position: Storey/Roof Member Supporting Partition (Sensitive Load)
Total Deflection = 1.959 mm ≤ 9.792 mm (4700/480) ... **Adequate** ✓

Serviceability Limit State Combination Results

	Left Support	Right Support	Span
D+L M	-56.2 kN.m	-38.9 kN.m	36.0 kN.m
M_{cr}	128.5 kN.m	128.5 kN.m	70.1 kN.m
Cracking Status	NO	NO	NO
Cracked Inertia	9.395E-04 m^4	9.395E-04 m^4	5.180E-04 m^4
Effective Inertia	4.446E-03 m^4	4.446E-03 m^4	4.446E-03 m^4
Effective Inertia	4.401E-03 m^4		
Immediate Def.	0.006 m		
Total Time Dep. Deflection	0.007 m		

ACI318-19 Table 24.2.2

Table 24.2.2—Maximum permissible calculated deflections

Member	Condition		Deflection to be considered	Deflection limitation
Flat roofs	Not supporting or attached to nonstructural elements likely to be damaged by large deflections		Immediate deflection due to maximum of L_r, S, and R	ℓ/180[1]
Floors			Immediate deflection due to L	ℓ/360
Roof or floors	Supporting or attached to nonstructural elements	Likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements, which is the sum of the time-dependent deflection due to all sustained loads and the immediate deflection due to any additional live load[2]	ℓ/480[3]
		Not likely to be damaged by large deflections		ℓ/240[4]

ACI318-19 Table 24.2.4.1.3

Table 24.2.4.1.3—Time-dependent factor for sustained loads

Sustained load duration, months	Time-dependent factor ξ
3	1.0
6	1.2
12	1.4
60 or more	2.0

Steel Seismic Design Checks

Another important undertaking for us was to implement the seismic checks of steel members according to AISC341 and Turkish codes.

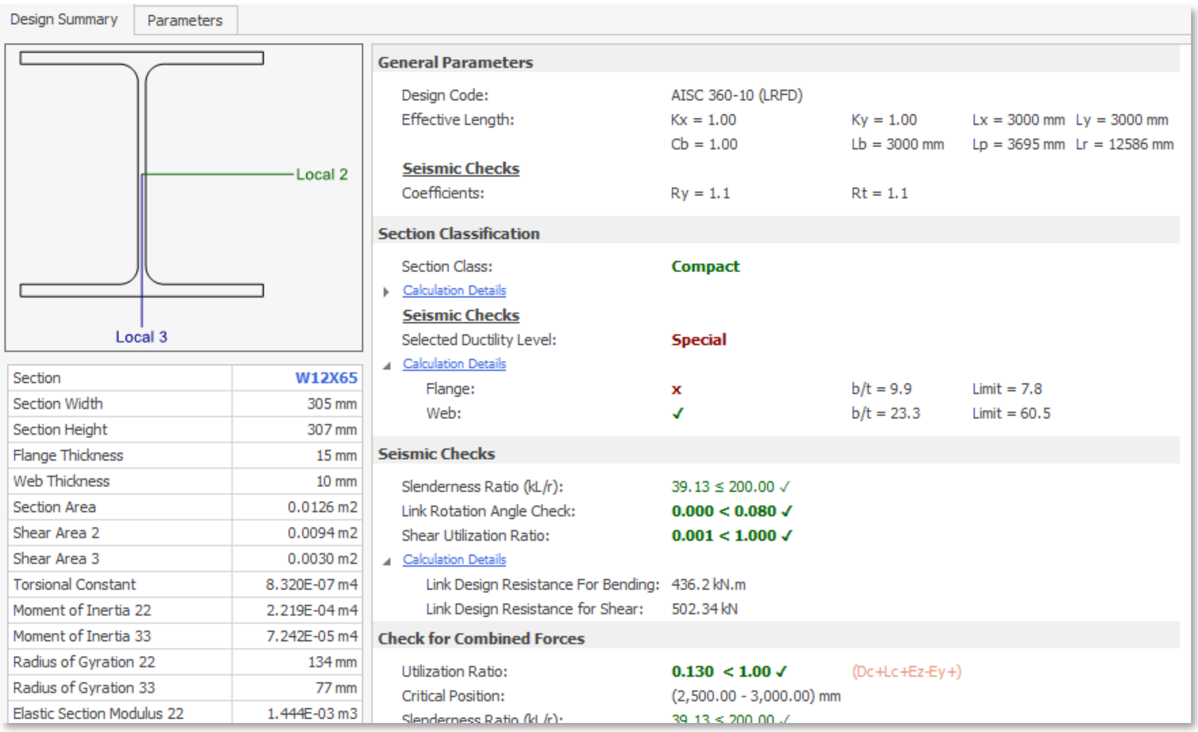
Seismic design checks of Eurocode for steel members are not fully supported yet and it is work in progress.

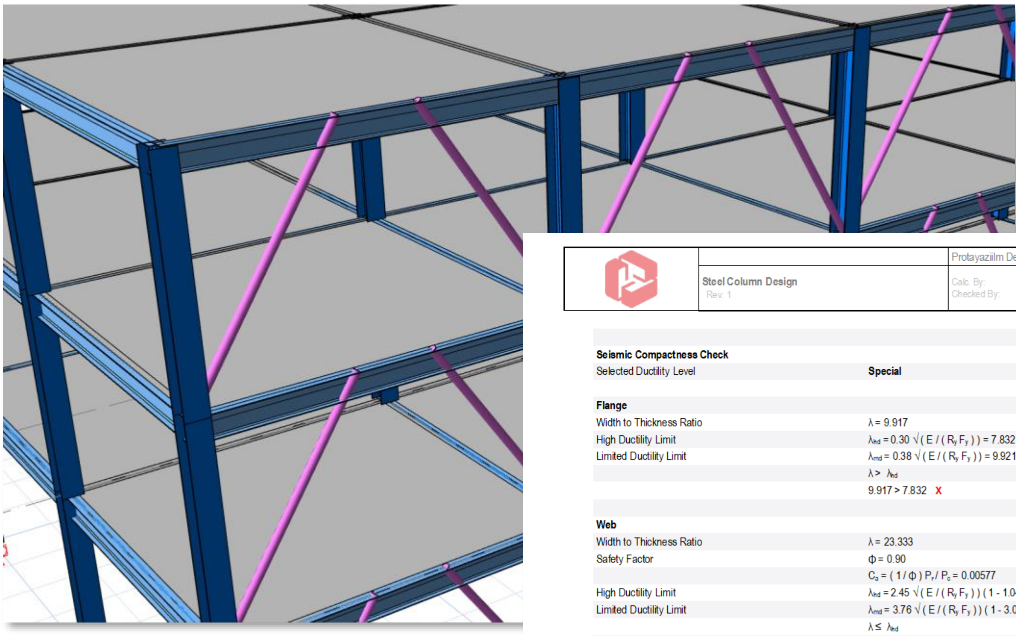
Seismic provisions limit us in usage of steel profiles. For Special Moment Frames, compact sections must be used in design. ProtaStructure will now check this and fail the design if the requirement is not met.

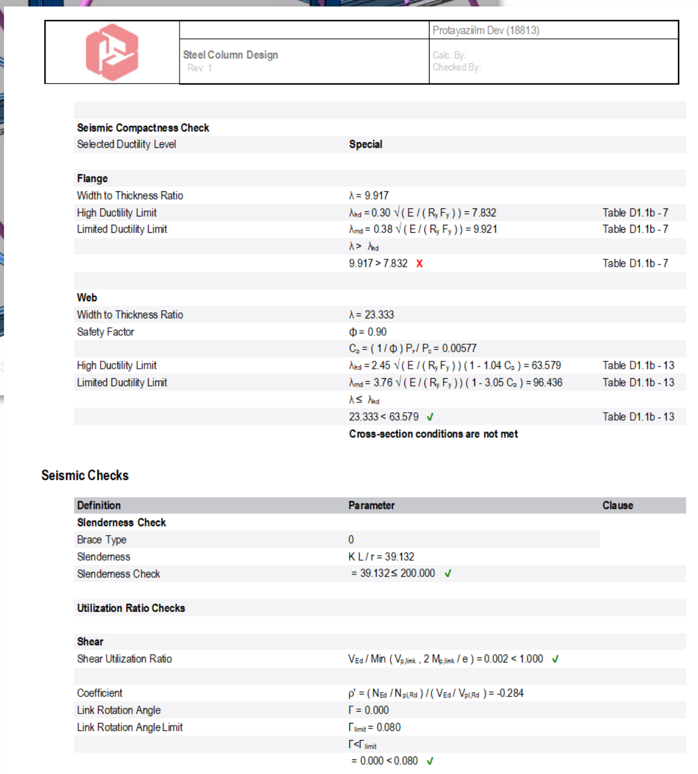
In addition to this, in eccentrically braced frames, seismic link checks must be performed, and these are also now in the software.

Results are reported with their associated clauses in the detailed report.

ProtaStructure does not perform the seismic checks on members which are not part of lateral load resisting system such as planar trusses, space trusses, steel domes, purlins, girts, and secondary beams with no connected columns at both ends







		Protayazilm Dev (18813)
	Steel Column Design Rev: 1	Calc. By: Checked By:

Seismic Compactness Check		
Selected Ductility Level	**Special**	
Flange		
Width to Thickness Ratio	$\lambda = 9.917$	
High Ductility Limit	$\lambda_{hd} = 0.30 \sqrt{(E / (R_y F_y))} = 7.832$	Table D1.1b - 7
Limited Ductility Limit	$\lambda_{md} = 0.38 \sqrt{(E / (R_y F_y))} = 9.921$	Table D1.1b - 7
	$\lambda > \lambda_{hd}$	
	9.917 > 7.832 X	Table D1.1b - 7
Web		
Width to Thickness Ratio	$\lambda = 23.333$	
Safety Factor	$\Phi = 0.90$	
	$C_a = (1 / \Phi) P_r / P_c = 0.00577$	
High Ductility Limit	$\lambda_{hd} = 2.45 \sqrt{(E / (R_y F_y))} (1 - 1.04 C_a) = 63.579$	Table D1.1b - 13
Limited Ductility Limit	$\lambda_{md} = 3.76 \sqrt{(E / (R_y F_y))} (1 - 3.05 C_a) = 96.436$	Table D1.1b - 13
	$\lambda \leq \lambda_{hd}$	
	23.333 < 63.579 ✓	Table D1.1b - 13
	Cross-section conditions are not met	

Seismic Checks

Definition	Parameter	Clause
Slenderness Check		
Brace Type	0	
Slenderness	$K L / r = 39.132$	
Slenderness Check	$= 39.132 \leq 200.000$ ✓	
Utilization Ratio Checks		
Shear		
Shear Utilization Ratio	$V_{Ed} / \text{Min} (V_{p,link}, 2 M_{p,link} / e) = 0.002 < 1.000$ ✓	
Coefficient	$\rho' = (N_{Ed} / N_{pl,Rd}) / (V_{Ed} / V_{pl,Rd}) = -0.284$	
Link Rotation Angle	$\Gamma = 0.000$	
Link Rotation Angle Limit	$\Gamma_{limit} = 0.080$	
	$\Gamma < \Gamma_{limit}$	
	$= 0.000 < 0.080$ ✓	

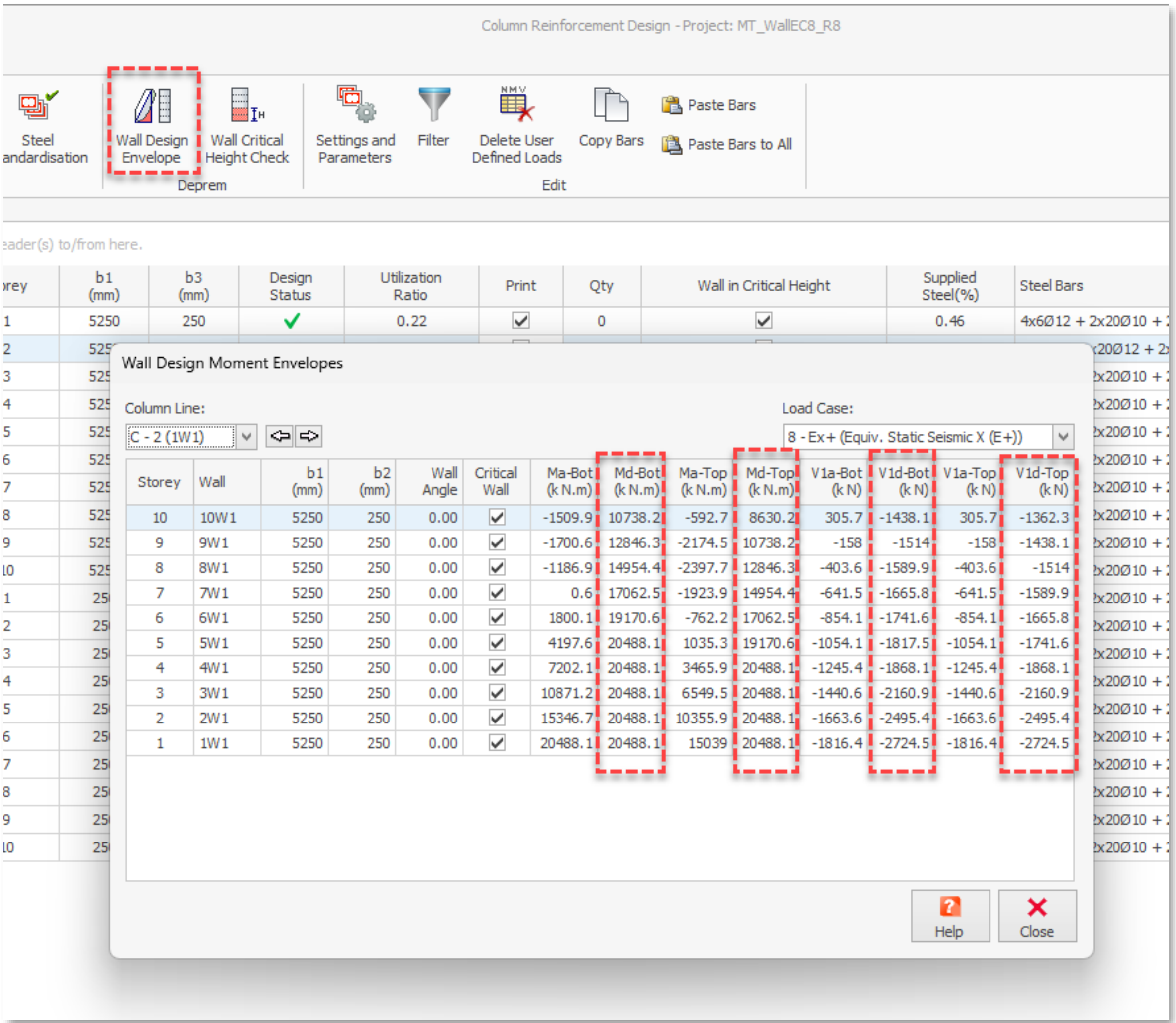
Eurocode 8- Shearwall Moment & Shear Design Envelope

Another important improvement in the new version is the use of bending moment and shear force design envelopes in shearwalls.

To ensure that the ductile shearwalls, with $H_w/L_w \geq 2$, remain elastic above the base hinge considering the uncertainties in the structure dynamic behavior, the design bending moments and shear forces obtained from analysis are magnified.

ProtaStructure will now automatically calculate the tension shift and apply bending moment and shear force envelopes according to the procedure outlined in **Eurocode 8 - 5.4.2.4.**

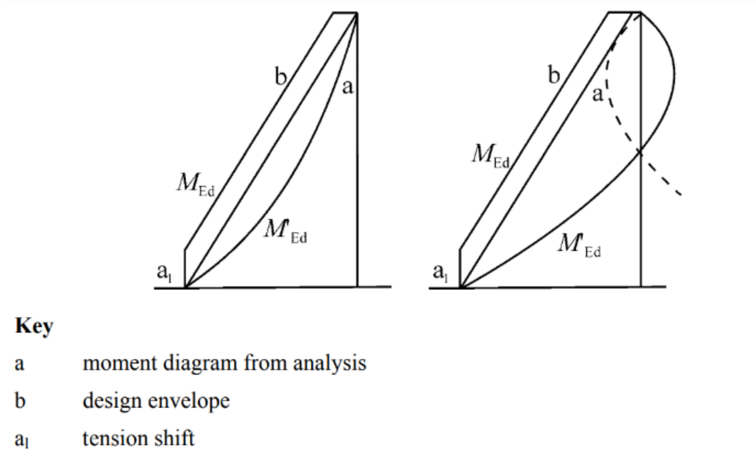
You can review and change the design envelope values from **Column Reinforcement Design > Wall Design Envelope** window.

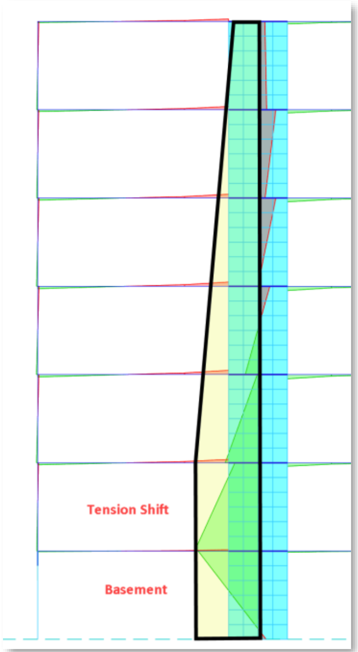


Bending Moment Design Envelope

From the bending moment diagram obtained from the analysis, a linear envelope can be established.

This diagram must be shifted upwards by a distance a_1, designated tension shift in Eurocode 8 - 5.4.2.4 (5), consistent with the strut inclination adopted in the Ultimate Limit State verification for shear.





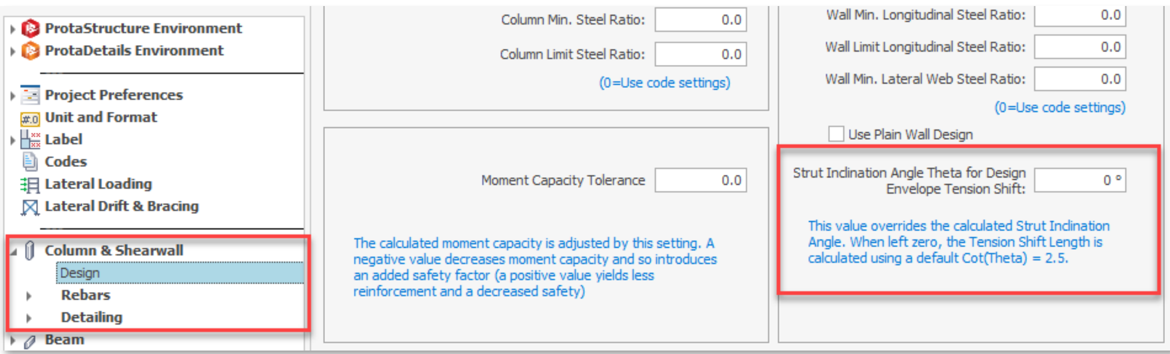
User-defined Strut Inclination Angle, θ, for Tension Shift Length

The default **Strut Inclination Angle, θ,** for tension shift calculation in ProtaStructure is 21.8°, which yields **cot(θ) = 2.5**.

The tension shift length is calculated with the formula **$a_1 = L_w \cot(\theta)$**.

The value of strut inclination angle varies from 21.8 to 45 degrees. To leave room for individual interpretation, we have introduced a new setting, "**Strut Inclination Angle for Design Envelope Tension Shift**". It is accessible from **Settings > Column & Shearwall > Design**.

The value entered here will override the calculated strut inclination angle. When left zero, the tension shift length is calculated using the default **cot(θ) = 2.5.**

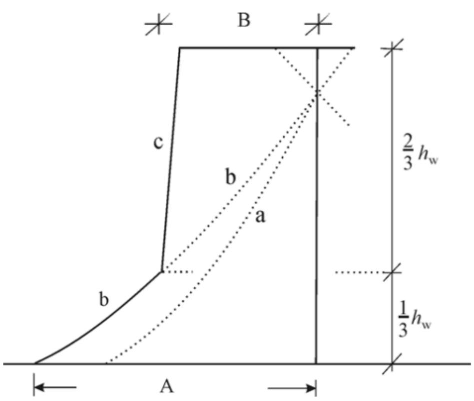


Shear Design Envelope

According to **Eurocode 8 – 5.4.2.4 (6),** the possible increase in shear forces after yielding at the base of a primary seismic wall, must be considered. This requirement may be satisfied if the design shear forces are increased by 50% (for DCM) than the analysis shear forces in the critical wall height extending Hw/3 from the building base or basement level. For DCH structures, the shear amplification factor is calculated by **Eurocode 8 Eqn. 5.25**.

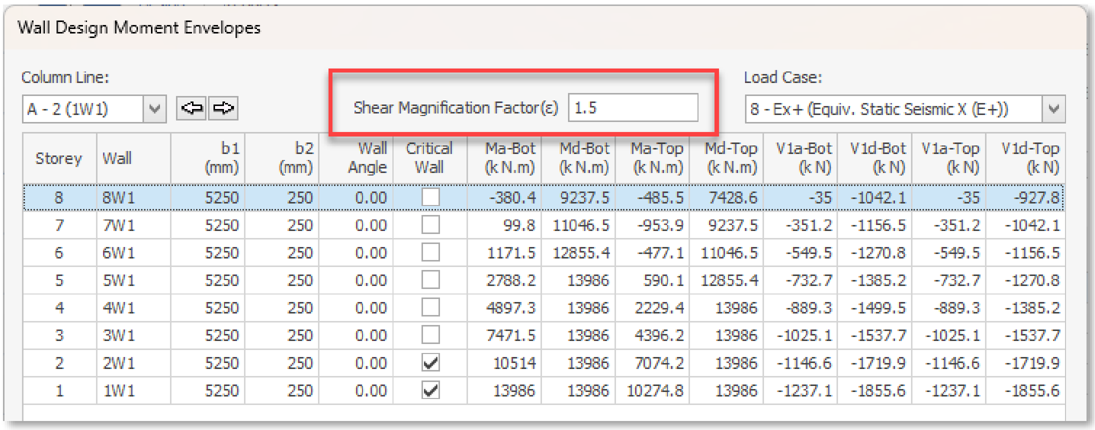
User-Defined Shear Amplification factor, ε

ProtaStructure provides a user-defined parameter for shear amplification factor which can be edited for each wall line. The default and the minimum value is **1.5** and the maximum value is the '**q**' factor used in the analysis in the wall direction. ProtaStructure will magnify the analysis shear forces are by this factor inside the critical height. The shear diagram is linearly connected to the value at the top, which is $V_{wall,base}/2$.



Key

- a shear diagram from analysis
- b magnified shear diagram
- c design envelope
- A $V_{wall,base}$
- B $V_{wall,top} \geq V_{wall,base}/2$



Wall Design Moment Envelopes

Column Line: A - 2 (1W1)

Shear Magnification Factor(ε) 1.5

Load Case: 8 - Ex+ (Equiv. Static Seismic X (E+))

Storey	Wall	b1 (mm)	b2 (mm)	Wall Angle	Critical Wall	Ma-Bot (kN.m)	Md-Bot (kN.m)	Ma-Top (kN.m)	Md-Top (kN.m)	V1a-Bot (kN)	V1d-Bot (kN)	V1a-Top (kN)	V1d-Top (kN)
8	8W1	5250	250	0.00	☐	-380.4	9237.5	-485.5	7428.6	-35	-1042.1	-35	-927.8
7	7W1	5250	250	0.00	☐	99.8	11046.5	-953.9	9237.5	-351.2	-1156.5	-351.2	-1042.1
6	6W1	5250	250	0.00	☐	1171.5	12855.4	-477.1	11046.5	-549.5	-1270.8	-549.5	-1156.5
5	5W1	5250	250	0.00	☐	2788.2	13986	590.1	12855.4	-732.7	-1385.2	-732.7	-1270.8
4	4W1	5250	250	0.00	☐	4897.3	13986	2229.4	13986	-889.3	-1499.5	-889.3	-1385.2
3	3W1	5250	250	0.00	☐	7471.5	13986	4396.2	13986	-1025.1	-1537.7	-1025.1	-1537.7
2	2W1	5250	250	0.00	☑	10514	13986	7074.2	13986	-1146.6	-1719.9	-1146.6	-1719.9
1	1W1	5250	250	0.00	☑	13986	13986	10274.8	13986	-1237.1	-1855.6	-1237.1	-1855.6

Drift Calculation in RSA and EQS Analyses

Seismic codes usually allow relative drift checks to ignore Minimum Base Shear and Maximum Period limitations in the equivalent static load approach. This is based on observations in the past earthquakes and on the fact that tall buildings are more drift-controlled.

Following are the excerpts from ASCE07 exemplifying the requirement.

12.8.6.1 Minimum Base Shear and Load Combination for Computing Displacement and Drift The elastic analysis of the seismic force-resisting system for computing displacement and drift shall be made using the prescribed seismic design forces of Section 12.8 using a load factor of 1.0 on E_h, in the presence of expected gravity loads. Expected gravity loads shall be taken as no less than $1.0D + 0.5L$, L shall be taken as $0.8L_0$ for live loads that exceed 100 lb/ft^2 (4.79 kN/m^2) and $0.4L_0$ for all other live loads, and L_0 is the unreduced design live load (see Table 4-1).

EXCEPTION: Equation (12.8-6) need not be considered for computing drift.

12.8.6.2 Period for Computing Displacement and Drift For determining displacements and drifts, it is permitted to determine the elastic displacements, δ_e, using seismic design forces based on the computed fundamental period of the structure without the upper limit, $C_u T_a$, specified in Section 12.8.2.

Lower Bounds for C_s for Both Methods: C_s shall not be less than

$$C_s = 0.044 S_{DS} I_e \geq 0.01 \quad (12.8\text{-}6)$$

12.8.2.1 Approximate Fundamental Period The approximate fundamental period (T_a), in seconds, shall be determined from the following equation:

$$T_a = C_t h_n^x \quad (12.8\text{-}8)$$

ProtaStructure will now more elaborately consider seismic code clauses that outlines the scaling and limitation of drifts.

Response Spectrum Analysis

When a response spectrum analysis is performed, the forces and displacements are now magnified with different scale factors. While **Force Magnification** will consider Minimum Base Shear and Maximum Periods, **Displacement Magnification** will not (if applicable in the code)

For example, in India and Peru seismic standards, displacements are not scaled when RSA is performed. On the other hand, ASCE07 requires scaling of displacements with a different factor calculated by ignoring minimum base shear and limiting periods.

The term, scaling is used here intentionally. Because ProtaStructure will always start the analysis with an unmodified behavior factor considering minimum base shear and applying period limitations. Results are always post-processed after the analysis.

Force Scaling and Displacement Scaling Factors Separated

In ProtaStructure 2025, different factors are calculated for force scaling and displacement scaling.

Following is the output from the Post-Analysis Checks Report in ProtaStructure 2025 when an RSA analysis is performed. The RSA-EQS comparison section of this report is extended to provide new information on displacement scaling.

		Protayaziilm Dev (18813)
	Post-Analysis Checks Rev: 1	Calc. By: Checked By:

Post-Analysis Checks

COMPARE RSA RESULTS TO EQUIVALENT STATIC LOAD ANALYSIS BASE SHEAR

(IBC - Cl. 12.9.1.4)

Earthquake Direction: 1 (Angle From X 0.00 Deg)

γ = 1.00 will be used.

Base Shear Forces:

	V_{tx-X} (kN)	V_{tx-Y} (kN)	V_{tx} (kN)
Ex-	92.36	8.35	92.74
Ex+	92.36	8.35	92.74
Vt-Static:	135.47	0.00	135.47

Total Earthquake Load:

Response Spectrum Analysis V_{tx} = 92.74 kN
Equivalent Static Load Analysis V_{tE} = 135.47 kN

Warning: $V_{tx} < \gamma V_{tE}$ (135.47 kN)
RSA Results will be multiplied with the factor: $\beta_{tE} = \gamma_E V_{tE} / V_{tx} = 1.461$

Spectral Acceleration (Sa) = 0.007 g @ Period (Dir: 1) = 1.519 sec
(Limit Period and Min. Spectral Acceleration are not applied for calculating the displacement magnification factor.)
Base Shear (Equivalent Static Load Analysis, V_{tE}) = 113.93 kN

Warning: $V_{tx} < \gamma V_{tE}$ (113.93 kN)
RSA Displacements will be multiplied with the factor. $\beta_{tE} = \gamma_E V_{tE} / V_{tx} = 1.229$

Earthquake Direction: 2 (Angle From X 90.00 Deg)

Warning: $V_{tx} < \gamma V_{tE}$ (135.47 kN)
RSA Results will be multiplied with the factor: $\beta_{tE} = \gamma_E V_{tE} / V_{tx} = 1.461$

Spectral Acceleration (Sa) = 0.007 g @ Period (Dir: 1) = 1.519 sec
(Limit Period and Min. Spectral Acceleration are not applied for calculating the displacement magnification factor.)
Base Shear (Equivalent Static Load Analysis, V_{tE}) = 113.93 kN

Warning: $V_{tx} < \gamma V_{tE}$ (113.93 kN)
RSA Displacements will be multiplied with the factor. $\beta_{tE} = \gamma_E V_{tE} / V_{tx} = 1.229$

Equivalent Static Load Analysis

The same principle applies if an equivalent static load analysis is performed. It is not likely that EQS will suffice for tall buildings, for which the minimum base shear and maximum period conditions govern. But nevertheless, we have added a new section in Post-Analysis Checks Report to report the details for Equivalent Static Analysis Displacement Reduction.

Post-Analysis Checks

Equivalent Static Analysis Displacement Reduction

(IBC - Cl. 12.9.1.4)

Building Analysis is completed using the Limiting Period.
Therefore, for the calculation of Seismic Displacements a factor will be calculated by dividing the spectral acceleration value at the fundamental period to spectral acceleration at Limit Period.

Earthquake Direction: 1 (Angle From X 0.00 Deg)

Spectral Acceleration (SA) = 0.007 g @ Period (Dir. 1) = 1.519 sec
Spectral Acceleration (SA) = 0.008 g @ Maximum Period (Dir: 1) = 0.903 sec

Displacement Reduction Factor = 0.007 / 0.008 = 0.846

Earthquake Direction: 2 (Angle From X 90.00 Deg)

Spectral Acceleration (SA) = 0.007 g @ Period (Dir. 2) = 1.547 sec
Spectral Acceleration (SA) = 0.008 g @ Maximum Period (Dir: 2) = 0.903 sec

Displacement Reduction Factor = 0.007 / 0.008 = 0.846

Shearwall End Zone (Boundary Elements) Batch Editing

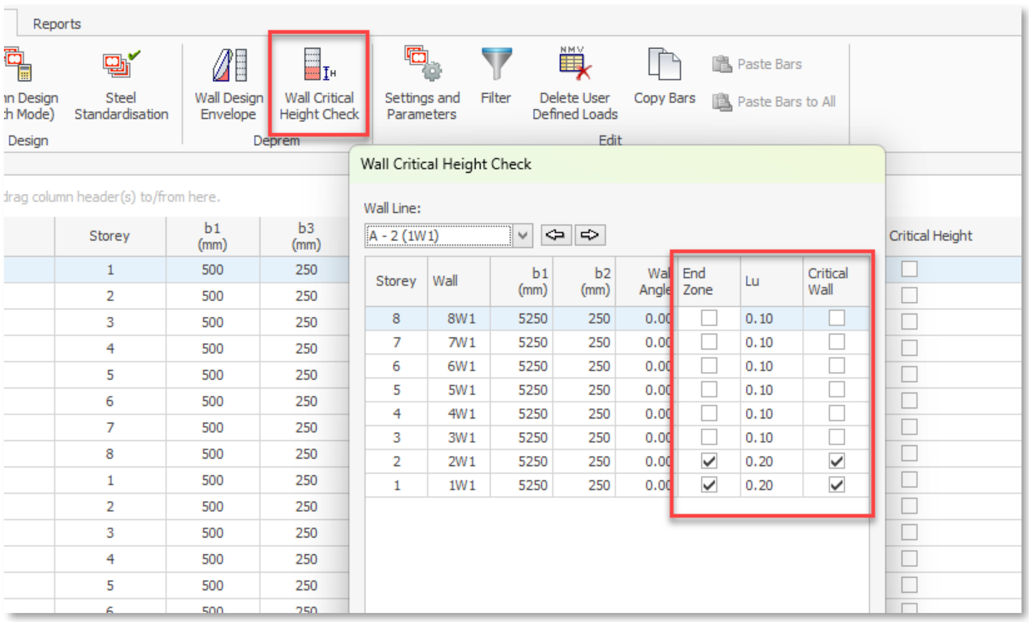
Shearwall end zones or boundary elements are essential in ensuring a ductile behavior especially in critical plastic regions of shearwalls.

Each seismic code has different regulations in calculation and curtailment of end zones.

ACI and Eurocode follow a strain-based approach in end-zone length calculation while Turkish code requires a minimum ratio of the wall length.

Also, for some codes, end zones may not be necessary outside the critical height, while a reduced end zone length is used in the other codes.

ProtaStructure has been supporting shearwall and corewall endzones (boundary elements) for a long time now. In the new version it just improved. We now allow you to set the **critical height**, **end zone usage** and **end zone length** easily in a batch mode editor.



ProtaStructure does not automatically calculate the length of the boundary elements. For rectangular walls default end zone length in critical region is **0.2Lw** and **0.1Lw** outside the critical region. These values can be edited using the newly introduced editor.

For core walls, user is expected to decide on the end zone layout using the automated tools in Polyline Column Editor.

Design Improvements

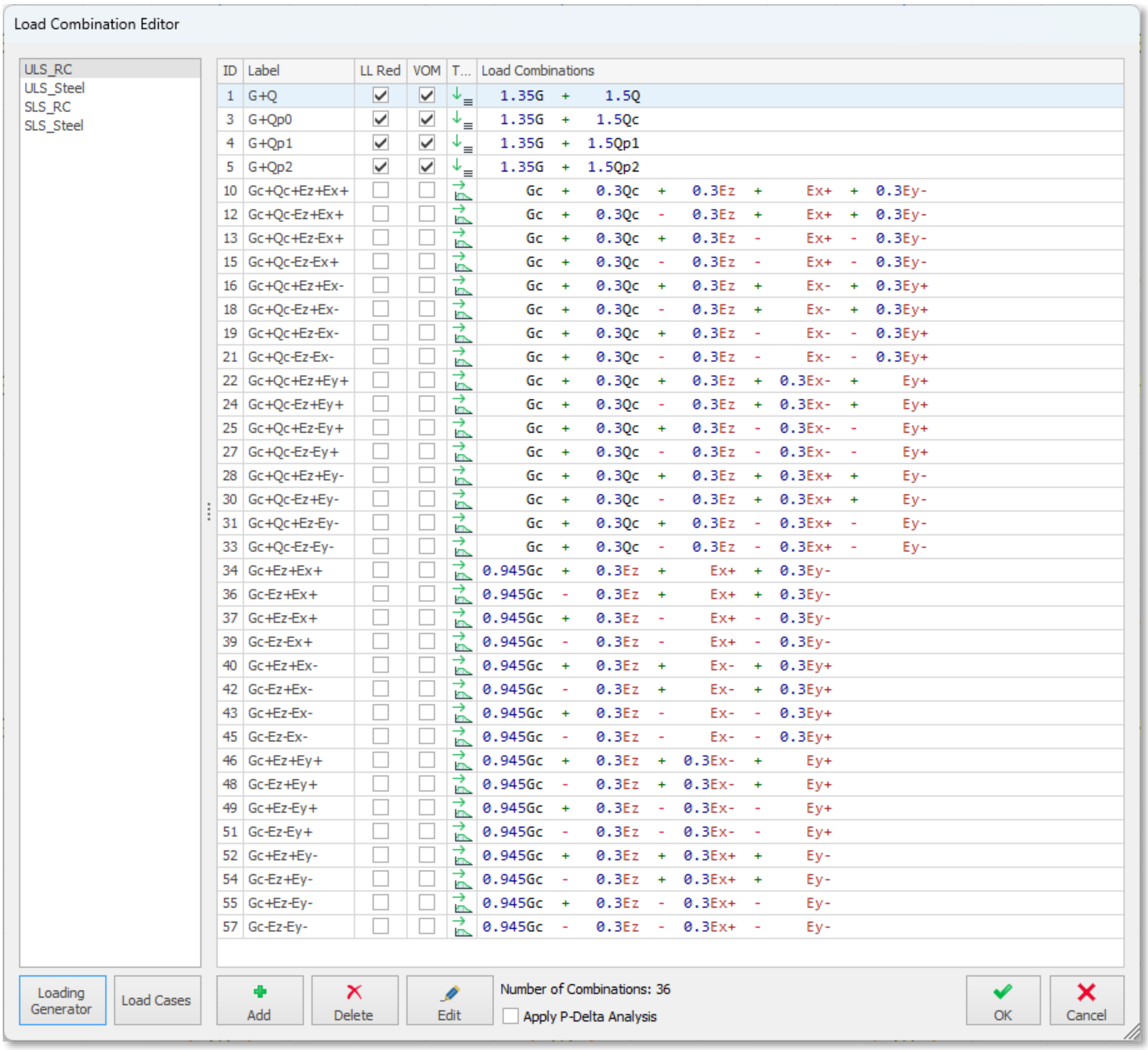
Improved Load Combination System

The load combination system in ProtaStructure is very flexible and allows you to define load cases and load combinations freely and automatically with the factors and permutations compliant to design and seismic codes.

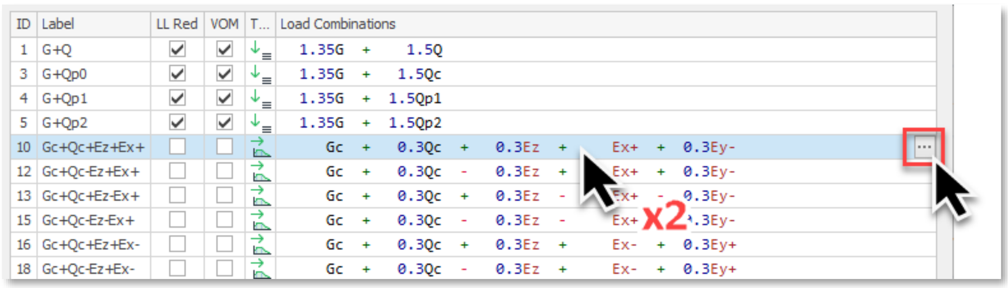
In the new version, we have focused on load combination system to make the design process more seamless and practical.

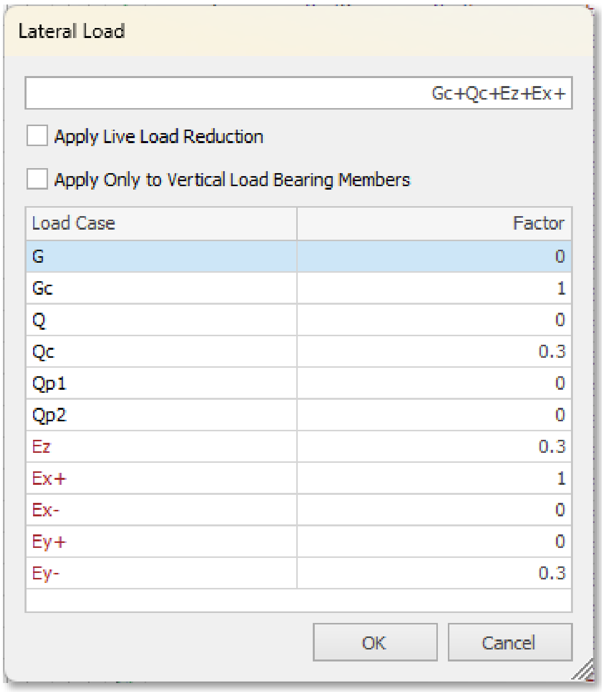
A New User Interface with Improved User Experience

First, we started with improving the user interface and user experience. Instead of the previous large matrix-style table, you will now find a more readable and clear presentation that color codes the load cases and their combination factors.



Editing the combinations is also easy. Double clicking one of them or clicking on the edit button appearing at the right-hand side of the row, allows you to customize the load case coefficients, combination name and other combination options as you wish.

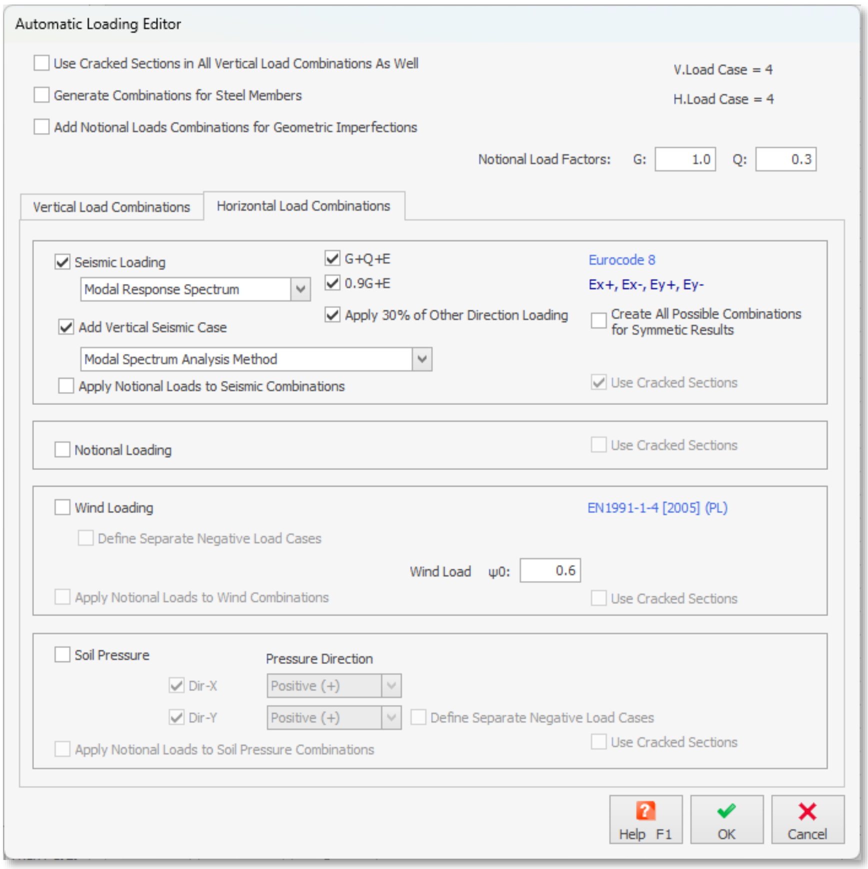




Improved Loading Generator – Remembering Previous Choices

ProtaStructure provides a powerful loading generator allowing you to automatically generate the necessary load combinations in a wizard like manner by selecting several options.

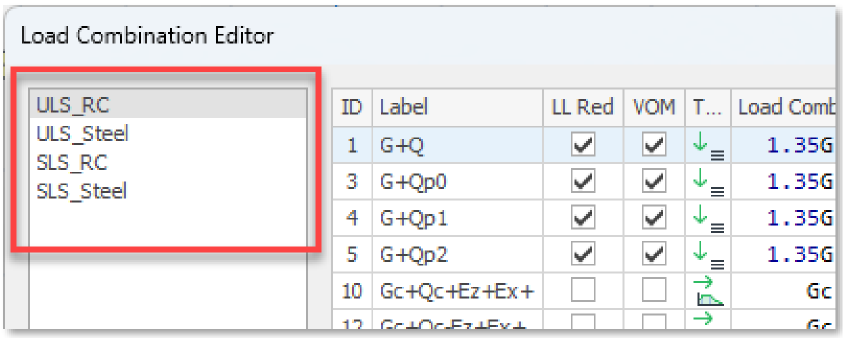
In the previous versions of ProtaStructure, the wizard options you have selected were always reset to factory defaults when you reopen the loading generator. ProtaStructure 2025 now will remember the loading generator options you have selected for creating the combinations and will automatically bring the same options when you reopen it.



Separate Combinations for RC and Steel

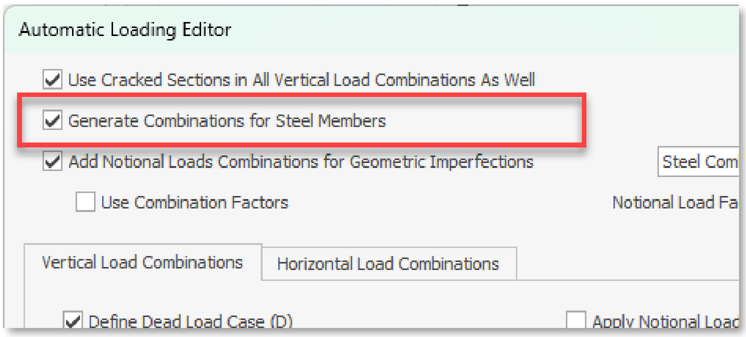
Steel and Concrete design and serviceability checks require different combinations. Knowing this, we followed a more structured approach in ProtaStructure 2025, and we created different groups for Steel and RC Ultimate Limit State (ULS) and Serviceability Limit State (SLS) combinations

You will now be able to access these combinations through their respective category in Load Combinations Editor.



To create Steel ULS and SLS combinations together with RC:

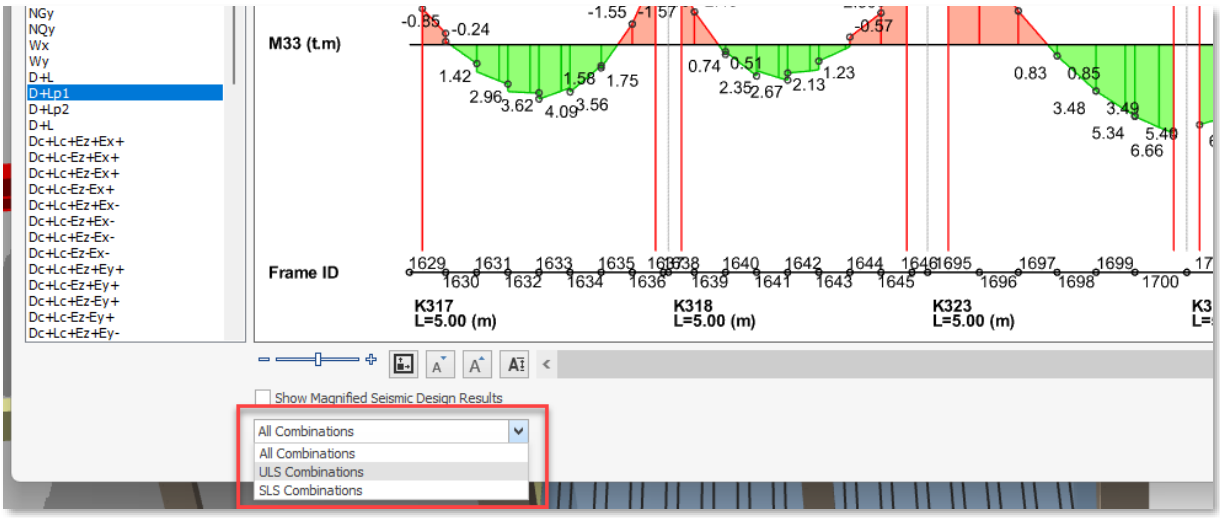
1. Open **Load Combination Generator** on the **Load Combinations Editor** window.
2. Check the "**Generate Combinations for Steel Members**" option.



Steel members will only honor the steel combinations during the design. Similarly, RC members will only use the Concrete combinations.

Reviewing Analysis Result Diagrams

Analysis result diagram window was organized for easy access to combination groups. First, the diagram window will show **Steel** or **RC** combinations depending on the member's material type. To review **ULS**, **SLS** or **All Combination** results, choose the relevant combination type from the list.



User-Defined SLS Combinations

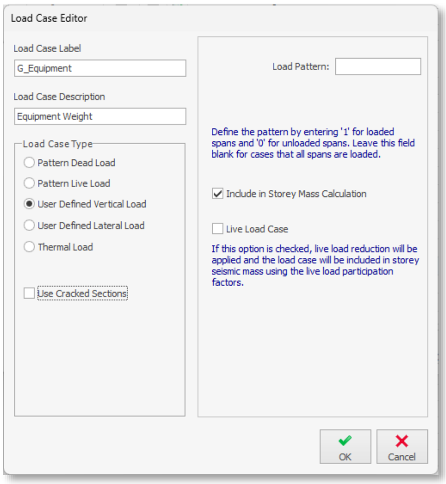
The new combination system in ProtaStructure is much more flexible. We now support **separate ULS** and **SLS groups** for steel and RC members.

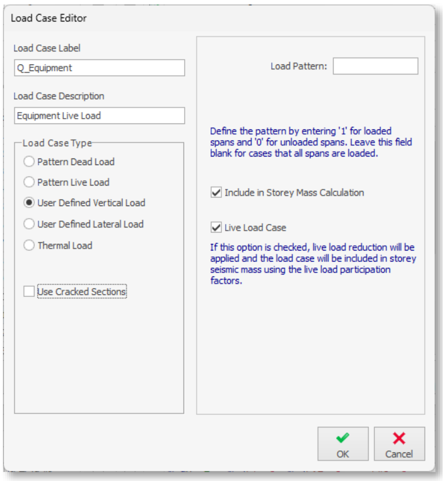
A valuable addition to this is the ability to create **User-defined SLS Combinations**. The SLS combinations you create will be automatically used in **Steel and RC Vertical Deflection** checks.

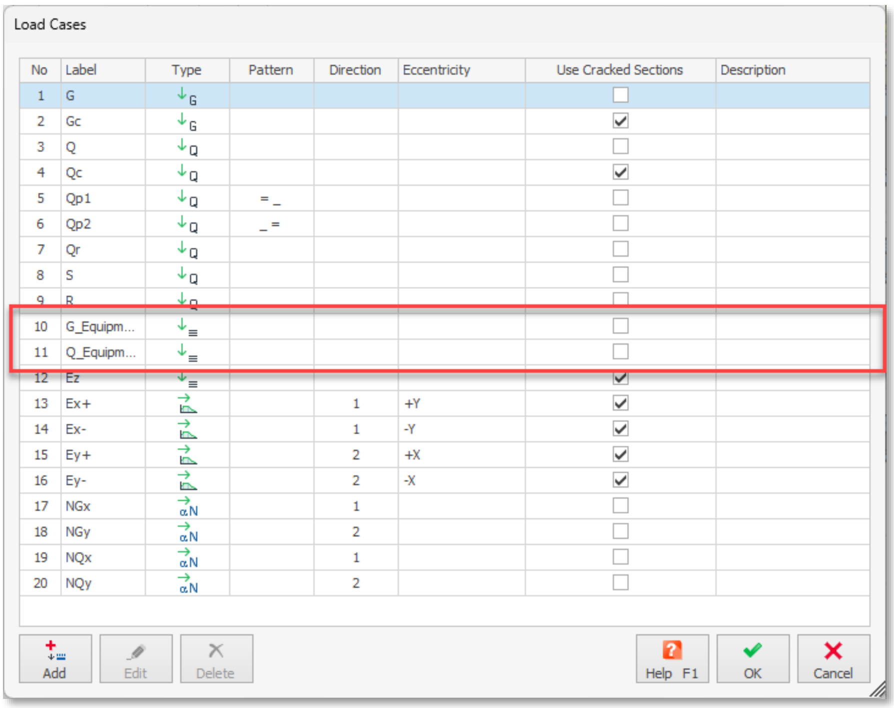
You can create user-defined serviceability combinations using the automatically generated load cases or you can create new user-defined load cases and use them in the new load combinations.

Creating User Defined Load Cases

ProtaStructure allows you to create user-defined vertical and lateral load cases. In the Load Cases window, click **Add** and create new load cases such as dead and live loads for an equipment.

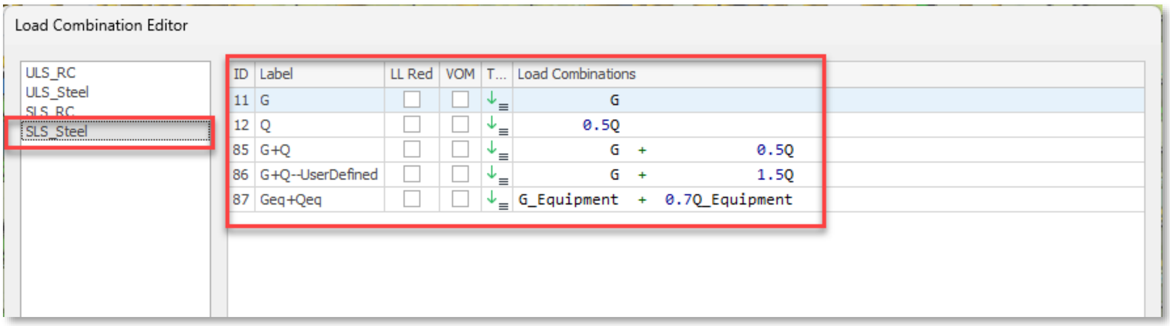




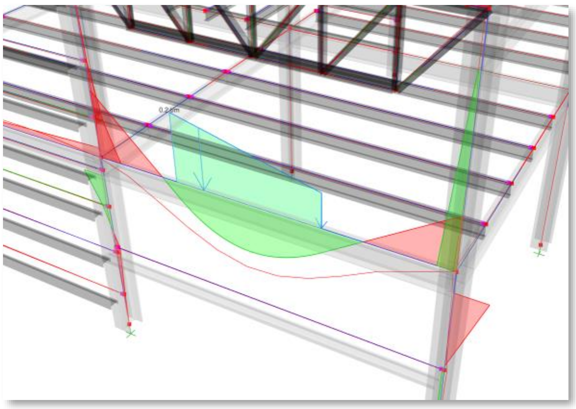


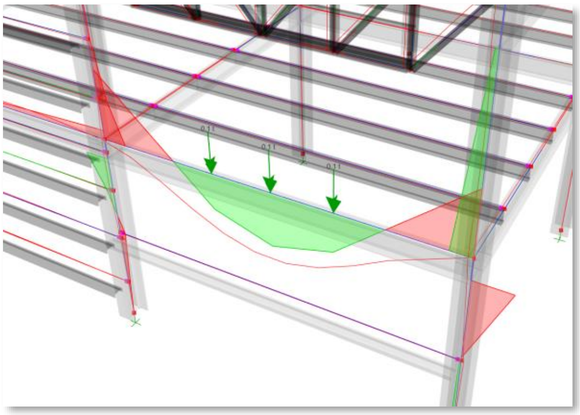
Creating User Defined SLS Combinations

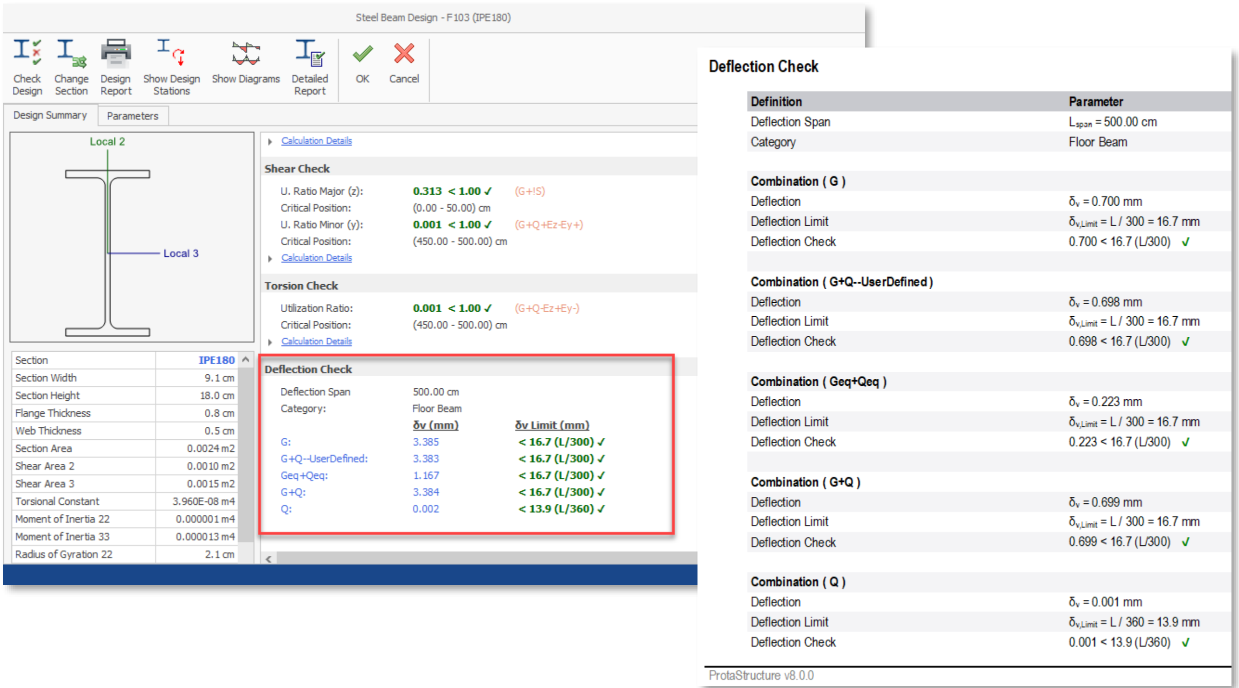
Once you create the load combinations using the automated **Loading Generator**, all related ULS and SLS combinations will be generated automatically. You can create additional SLS combination by clicking Add button and specifying the name and coefficients of the new combination.



You can then carry on with your model and assign loads on members of interest.







Notional Load Combinations

In the countries where wind loads or seismic loads do not govern the design, minimum lateral loads, so called the Notional Loads, are applied to the structure at storey levels.

The storey loads are usually calculated as a fraction of dead and live load.

But notional loads have another use for us.

This is not straightforward because Notional Load Case coefficients differ for each loading code. Some specify custom coefficients whereas some of the codes, notional loads use the same coefficients with dead and live load cases.

Besides, the DEAD or LIVE Load Fraction that is used to calculate the load magnitude also needs to be customizable.

All these requirements are now met in ProtaStructure. You can now flexibly apply notional loads to any combination type with any load case coefficient.

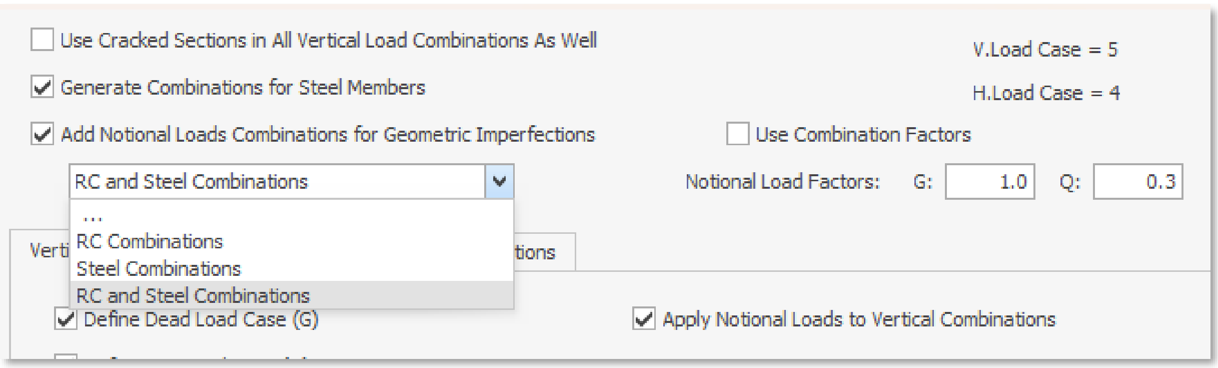
Adding Notional Loads to Combinations for Geometric Imperfections

In addition to their role as acting as minimum lateral load, we also need to consider them in some of the combinations or all combinations to cater for the geometric imperfections.

To add the notional loads to combinations:

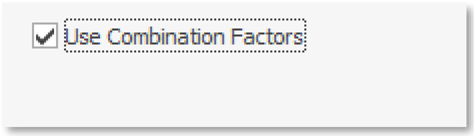
1. Check "**Add Notional Loads Combinations for Geometric Imperfections**" option.

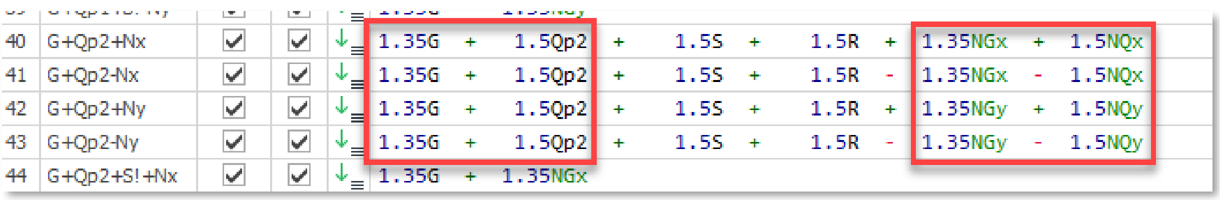
2. Specify whether you want to add notional loads to **RC Combinations**, **Steel Combinations** or **RC and Steel Combinations**.



Using Combination Factors

If you check "**Use Combination Factors**", the Notional Load Case factors will be the same as dead and live load coefficients in the combination.

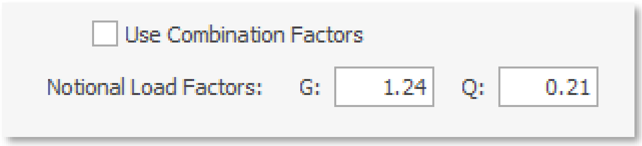


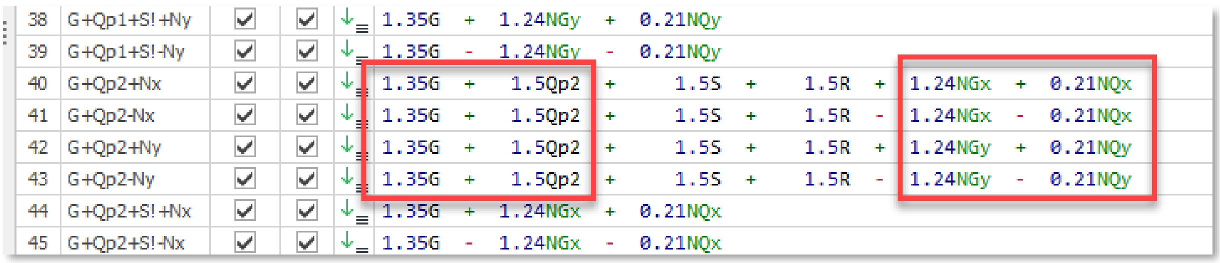


40	G+Qp2+Nx	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R + 1.35NGx + 1.5NQx
41	G+Qp2-Nx	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R - 1.35NGx - 1.5NQx
42	G+Qp2+Ny	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R + 1.35NGy + 1.5NQy
43	G+Qp2-Ny	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R - 1.35NGy - 1.5NQy
44	G+Qp2+S!+Nx	✓	✓	↓	1.35G + 1.35NGx

Using Custom Load Factors

By unchecking "**Use Combination Factors**" option, you can specify custom load factors.

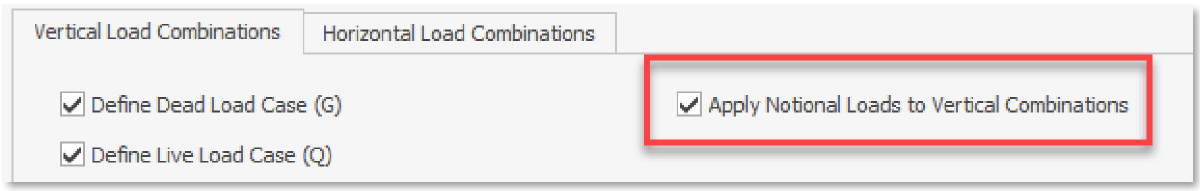


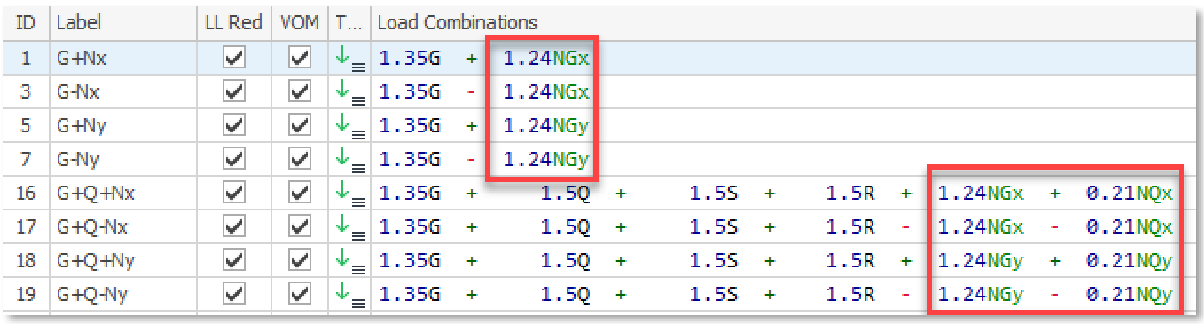


38	G+Qp1+S!+Ny	✓	✓	↓	1.35G + 1.24NGy + 0.21NQy
39	G+Qp1+S!-Ny	✓	✓	↓	1.35G - 1.24NGy - 0.21NQy
40	G+Qp2+Nx	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R + 1.24NGx + 0.21NQx
41	G+Qp2-Nx	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R - 1.24NGx - 0.21NQx
42	G+Qp2+Ny	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R + 1.24NGy + 0.21NQy
43	G+Qp2-Ny	✓	✓	↓	1.35G + 1.5Qp2 + 1.5S + 1.5R - 1.24NGy - 0.21NQy
44	G+Qp2+S!+Nx	✓	✓	↓	1.35G + 1.24NGx + 0.21NQx
45	G+Qp2+S!-Nx	✓	✓	↓	1.35G - 1.24NGx - 0.21NQx

Apply Notional Loads to Vertical Combinations

When this option is checked, the notional loads will be added to the vertical combinations with the factors you have specified with "**Use Custom Load Factors**" option, as explained above.

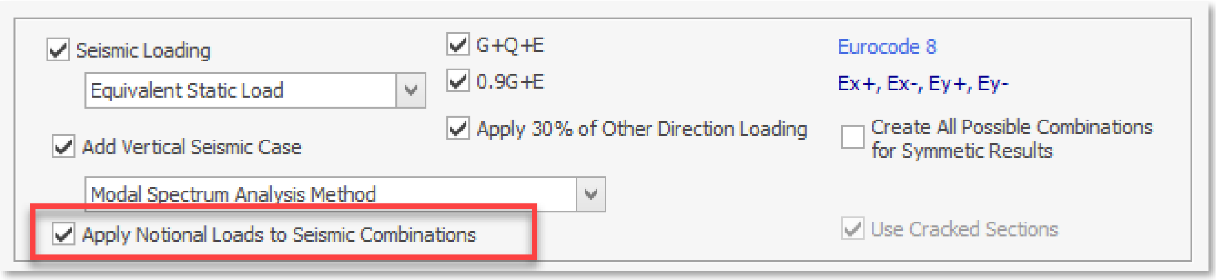




ID	Label	LL Red	VOM	T...	Load Combinations
1	G+Nx	✓	✓	↓	1.35G + 1.24NGx
3	G-Nx	✓	✓	↓	1.35G - 1.24NGx
5	G+Ny	✓	✓	↓	1.35G + 1.24NGy
7	G-Ny	✓	✓	↓	1.35G - 1.24NGy
16	G+Q+Nx	✓	✓	↓	1.35G + 1.5Q + 1.5S + 1.5R + 1.24NGx + 0.21NQx
17	G+Q-Nx	✓	✓	↓	1.35G + 1.5Q + 1.5S + 1.5R - 1.24NGx - 0.21NQx
18	G+Q+Ny	✓	✓	↓	1.35G + 1.5Q + 1.5S + 1.5R + 1.24NGy + 0.21NQy
19	G+Q-Ny	✓	✓	↓	1.35G + 1.5Q + 1.5S + 1.5R - 1.24NGy - 0.21NQy

Apply Notional Loads to Seismic Combinations

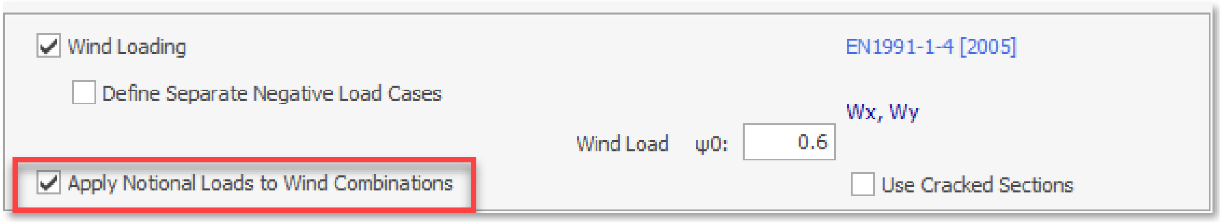
When this option is checked, the notional loads will be added to the seismic combinations with the factors you have specified with “**Use Custom Load Factors**” option, as explained above.



123	G+Q+Ez+Ex+				G	+	Q	+	0.3Ez	+	Ex+	+	0.3Ey-	+	1.24NGx	+	0.21NQx
125	G+Q-Ez+Ex+				G	+	Q	-	0.3Ez	+	Ex+	+	0.3Ey-	+	1.24NGx	+	0.21NQx
126	G+Q+Ez-Ex+				G	+	Q	+	0.3Ez	-	Ex+	-	0.3Ey-	-	1.24NGx	-	0.21NQx
128	G+Q-Ez-Ex+				G	+	Q	-	0.3Ez	-	Ex+	-	0.3Ey-	-	1.24NGx	-	0.21NQx
129	G+Q+Ez+Ex-				G	+	Q	+	0.3Ez	+	Ex-	+	0.3Ey+	+	1.24NGx	+	0.21NQx
131	G+Q-Ez+Ex-				G	+	Q	-	0.3Ez	+	Ex-	+	0.3Ey+	+	1.24NGx	+	0.21NQx
132	G+Q+Ez-Ex-				G	+	Q	+	0.3Ez	-	Ex-	-	0.3Ey+	-	1.24NGx	-	0.21NQx

Apply Notional Loads to Wind Combinations

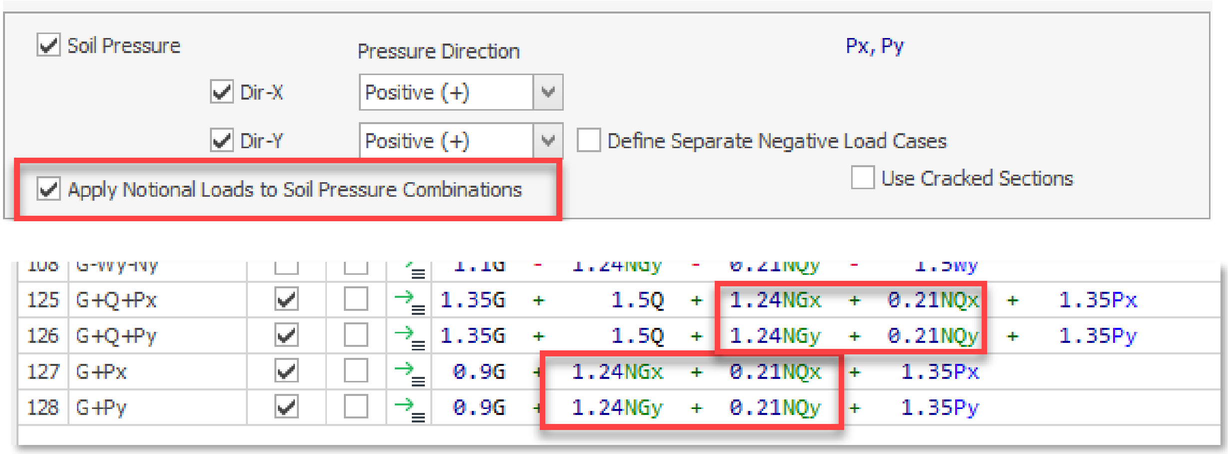
When this option is checked, the notional loads will be added to the wind combinations with the factors you have specified with “**Use Custom Load Factors**” option, as explained above.



93	G+Q+Wx+Nx	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	+	1.24NGx	+	0.21NQx	+	0.9Wx
94	G+Q-Wx-Nx	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	-	1.24NGx	-	0.21NQx	-	0.9Wx
95	G+Q+Wy+Ny	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	+	1.24NGy	+	0.21NQy	+	0.9Wy
96	G+Q-Wy-Ny	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	-	1.24NGy	-	0.21NQy	-	0.9Wy
97	G+Q+S!+Wx+Nx	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	+	1.24NGx	+	0.21NQx	+	0.9Wx
98	G+Q+S!-Wx-Nx	✓			1.35G	+	1.5Q	+	1.5S	+	1.5R	-	1.24NGx	-	0.21NQx	-	0.9Wx

Apply Notional Loads to Soil Pressure Combinations

When this option is checked, the notional loads will be added to the soil pressure combinations with the factors you have specified with “**Use Custom Load Factors**” option, as explained above.



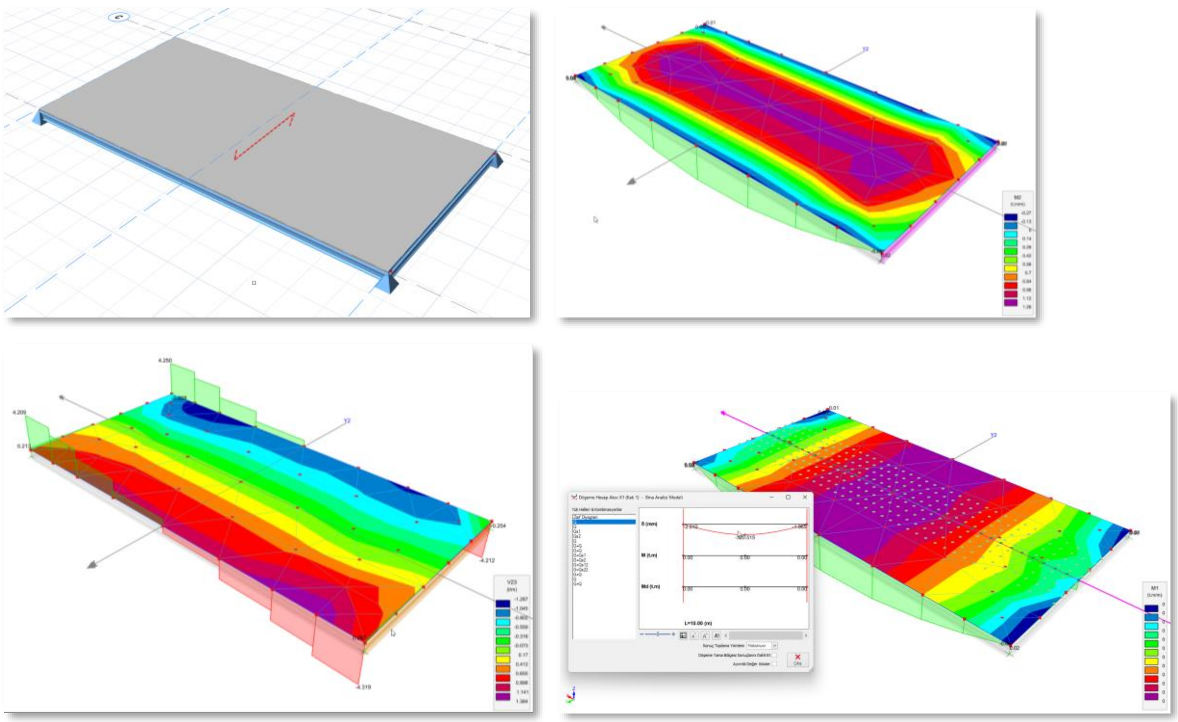
One-Way Slabs with Orthotropic Shell Elements

ProtaStructure 2025 has a significant enhancement in finite element modeling of one-way slabs.

Normally, a regular isotropic slab will exhibit a two-way behavior when analyzed with finite elements. This is usually sufficient for cast-in-place RC slabs. However, when it comes to **one-way prefabricated**, **pretensioned** or **hollow core slabs**, the one-way behavior must be accurately reflected to the analytical model.

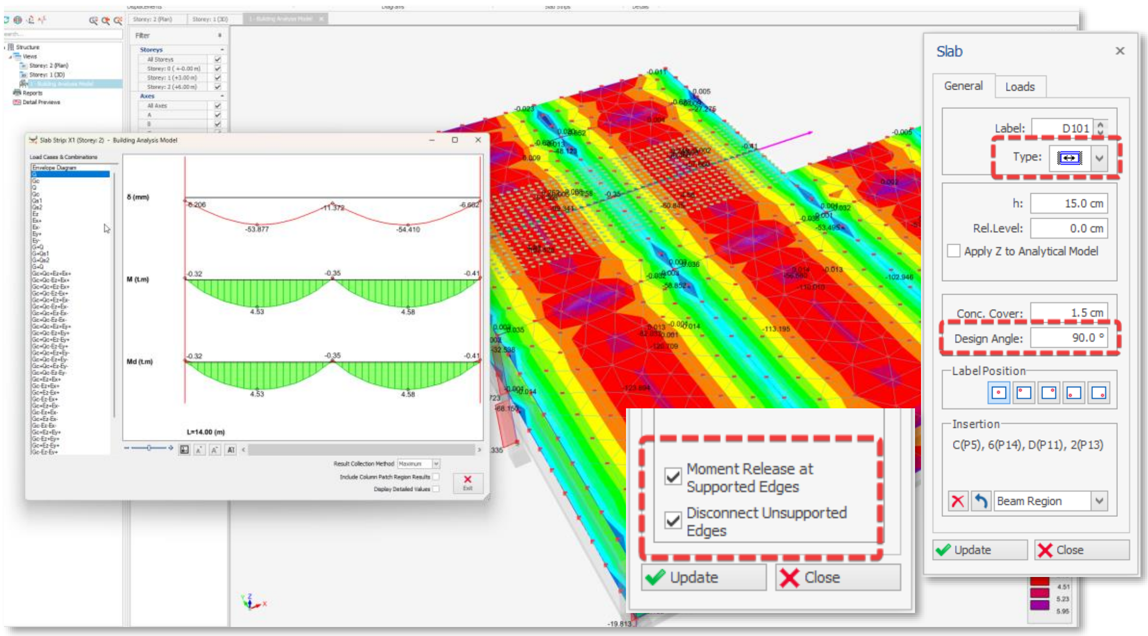
Hence, in the new version we have developed the **Orthotropic Shell Element** which has a significantly lower **Modulus of Elasticity, Shear Modulus** and **Poisson's Ratio** in the secondary direction.

All you must do is to specify the slab as a one-way slab and ProtaStructure will automatically adjust the analytical properties.



One-Way Slab Edge Releases

In addition to orthotropic material properties, you can also apply **End Releases** to one-way slabs. This will help you model scenarios where the slab is sitting on the supports without any moment connection or non-load transferring edges are not supported in z direction.



Moment Release at Supported Edges

The load transferring edges of the one-way slabs will be hinged, i.e., they will not transfer any bending moments to the supports.

Disconnect Unsupported Edges

The non-load transferring edges will be disconnected from the neighbouring slabs and supports, allowing them to deform independently.

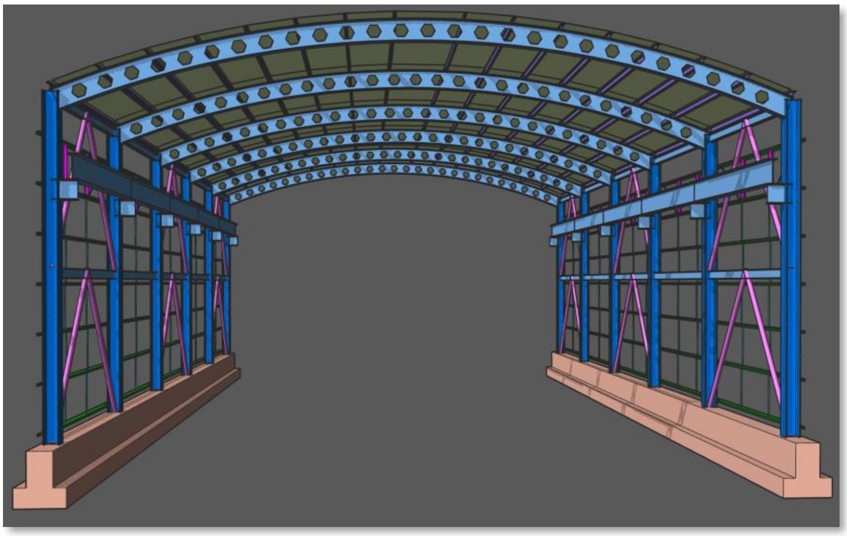
Important Remark:

The moment release and disconnection are achieved by duplicating the nodes and utilizing equal constraints in relevant degrees of freedom. This approach works best in slabs whose edges are parallel to global X and Y coordinate axes.

If you are trying to model a hollow core slab or any other slab system with these features, you are expected to enter the correct thickness and material properties. ProtaStructure will not automatically calculate those properties.

Castellated Beam Design

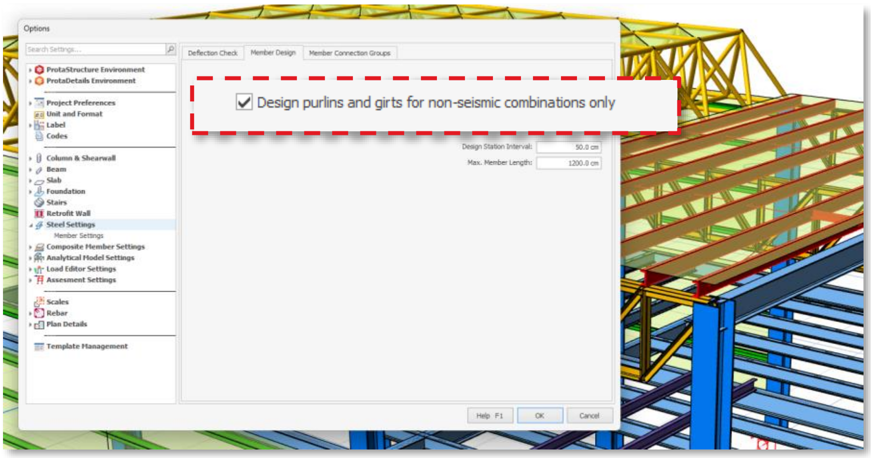
A new addition in ProtaStructure 2025 is the castellated beams. You can now consider them modeling, analysis, design and detailing.



Please refer to our new design guide on castellated beams for detailed explanations, modeling tips and more:
ProtaStructure Design Guide – Castellated Beams

Non-Seismic Purlin/Girt Design

One of the mostly asked features was to design the purlins and girts for non-seismic combinations only. And that's what we have delivered in ProtaStructure 2025. You can now ignore seismic combinations in purlin and girt design. The setting is accessible from **Settings > Steel Settings > Member Design**



Apart from seismic combinations, ProtaStructure does not perform the seismic checks on Purlins and Girts.

Manual Slab Strips in Slab Design Report

Another effective addition is the manual slab strips included in the design report. In addition to automatically designed slab rebars, we may need to manually put rebars in the slabs as necessary. Previously the manual strip rebars were not included in the design report. Now with the new version they are for a more complete design submission.

		Protayazilim Dev (18813)
	Slab Reinforcement Design Rev: 1	Calc. By: Checked By:

Slab Reinforcement Design

Eurocode 2 (PL) (EN 1992) Design of Concrete Structures (Poland Annex)

Notation	Definitions
d	Slab's Effective Depth
h	Slab Total Depth
q.max	Maximum Load Combination
L_1	Width of the Slab Along the Strip Direction
L_2	Width of the Slab Perpendicular to the Strip Direction
C	Moment Coefficient ($C = M / (p_d L^2)$)
M	Ultimate Moment
As	Reinforcment Area

Slab Strip : X1 -- Storey : 1

Materials : C25/30/Grade 500 (Type 2)

Slab	h/d (mm) Angle (°)	q.max max Combination (kN/m²)	L_1 L_2 (mm)	Support C M (kN.m)	Span C M (kN.m)	M_c Left Right (kN.m)	As-Left Required Provided (mm²)	As-Span Required Provided (mm²)	As-Right Required Provided (mm²)	Left- Support Straight	Span Bars BentUp Straight	Right- Support Straight
1S4 1	120.0/90.0 0	7.38 1G + 1Q	5000.00 4750.00	0.0320 5.3	0.0240 4.0	0.6 5.3	230.85 392.70	230.85 392.70	230.85 392.70	Ø10-200 (T_1)	- Ø10-200 (B_1)	Ø10-200 (T_1)
1S5 1	120.0/90.0 0	7.38 1G + 1Q	5000.00 4750.00	0.0320 5.3	0.0240 4.0	5.3 5.3	230.85 392.70	230.85 392.70	230.85 392.70	Ø10-200 (T_1)	- Ø10-200 (B_1)	Ø10-200 (T_1)
1S6 1	120.0/90.0 0	7.38 1G + 1Q	5000.00 4750.00	0.0320 5.3	0.0240 4.0	5.3 0.6	230.85 392.70	230.85 392.70	230.85 392.70	Ø10-200 (T_1)	- Ø10-200 (B_1)	Ø10-200 (T_1)

Manual Strip : X3 -- Storey : 1

Label	h d (mm)	Steel Bars Top/Bottom
X3	120.0 120.0	Top: Ø10-250 (B) Bottom: Ø10-250 (T)

Advanced Earthquake Engineering

Modal Vertical Earthquake Analysis

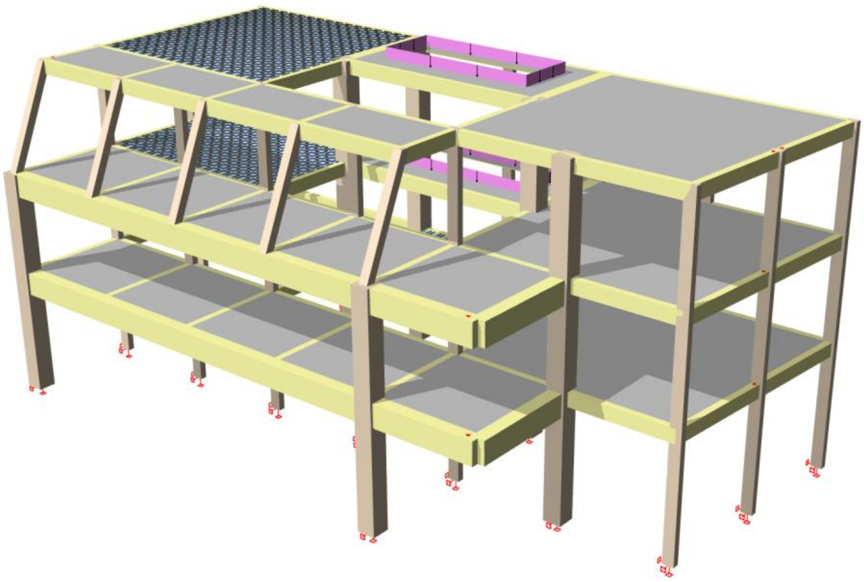
ProtaStructure can reflect the effects of vertical earthquake action in the design by providing two methods.

1. Approximate Static Approach
2. Modal Spectrum Analysis Method

Approximate Static Approach

In the static approach, the result from gravity load case is multiplied with a fraction of horizontal spectral acceleration such as $0.2S_{DS}$. Vertical vibration modes and a dedicated vertical spectrum is not considered in this approach.

However, static vertical earthquake calculation may not be sufficient (or allowable by the code) for the buildings with transfer columns, beams or slabs covering large spans, long cantilevers or slanting columns.



Certain seismic codes require the use of modal vertical earthquake analysis in these scenarios. For example, Eurocode only allows the use of modal vertical earthquake analysis.

Modal Spectrum Analysis Method

In modal spectrum method, vertical vibration modes of the structure are considered together with a vertical acceleration spectrum. With ProtaStructure 2025, you are now able to perform modal vertical earthquake analyses.

Please refer to our new design guide on vertical earthquake analysis for detailed explanations, modeling tips and more:
ProtaStructure Design Guide – Vertical Earthquake Effects

Modal Pushover Analysis

Traditional pushover analysis, which utilizes the primary vibrational mode of the structure, is viable for a limited range of buildings with a relatively regular geometry, little torsional irregularity, a very dominant primary mode, and low to mid rise height.

For buildings suffering from higher mode effects or irregularities, single-mode approaches will not be sufficient.

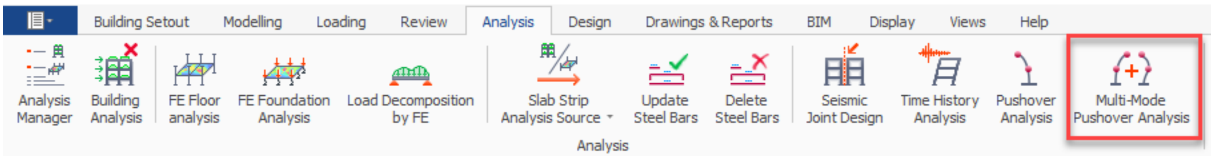
In these cases, nonlinear time history analysis is used. But this analysis is computationally demanding considering the number of accelerograms used, acceleration time steps, material nonlinearity and convergence conditions. Moreover, they often introduce unnecessary complexity to modeling and analysis.

Knowing these facts, a middle ground between single mode approaches and a full nonlinear time-history analysis is the modal pushover analysis (MPA). MPA is the non-linear static equivalent of the response spectrum analysis (RSA) in the linear domain.

In the new version of ProtaStructure, we have adopted the Chopra & Goel's (2002) Modal Pushover methodology. In simple terms, we perform separate pushover analyses for each viable mode up to a predetermined cumulative mass participation and combine their results using complete quadratic combination (CQC). The target displacement for each analysis instance is determined automatically using the target response spectrum and related modal properties.

The CQC procedure leaves us with combined results for all 4 orthogonal directions, which allows us to perform an assessment in a similar fashion to the traditional pushover analysis. The force, displacement and plastic rotation results for each analysis instance, as well as combined results can be viewed.

The Modal Pushover Analysis is accessible from **Analysis > Multi-Mode Pushover Analysis** on the ribbon toolbar.



Enhancements in Building Assessment and P.B.D

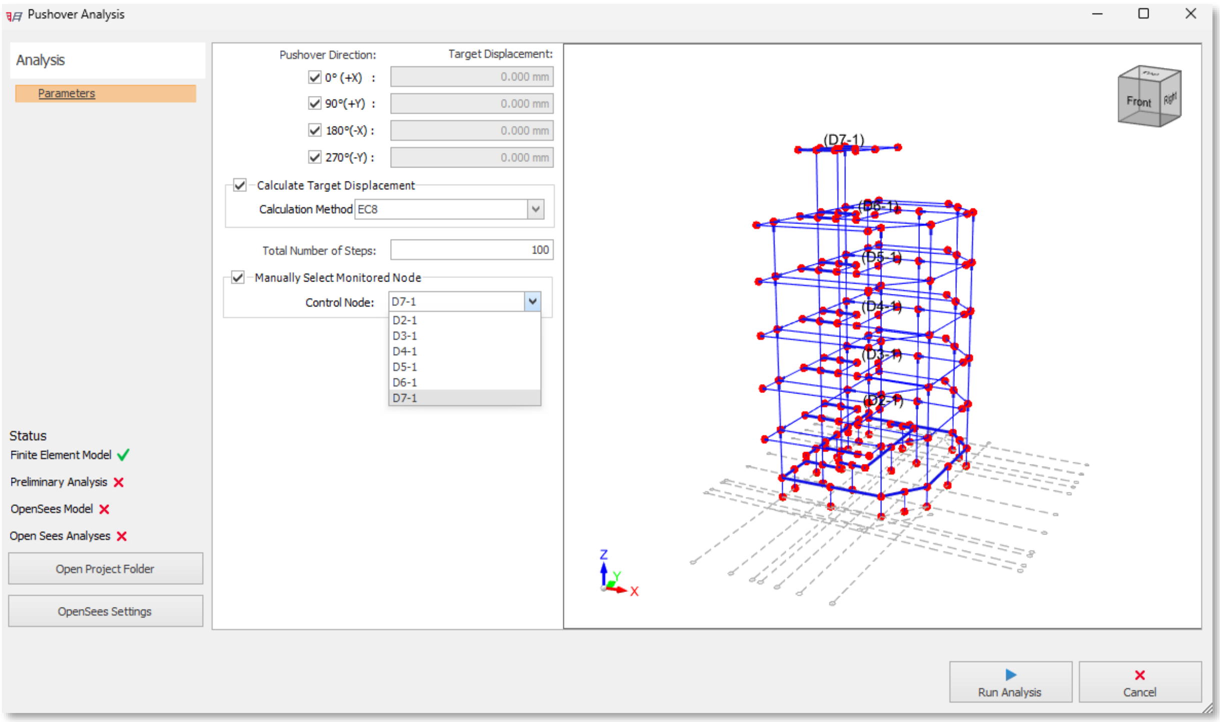
In addition to the new methodologies we developed in the new version, we have worked hard to make further optimizations and streamlining to the underlying system itself.

Efficiency in Reading Assessment Results

ProtaStructure is now more capable in result reading and performance in non-linear analyses

Pushover Analysis Screen and Monitored Node

We have made improvements for Pushover Parameters Screen. It provides a much better flow for selecting monitored node.



Member-Specific Nonlinear Modeling Options

There are now member-wise override options for non-linear modelling parameters.

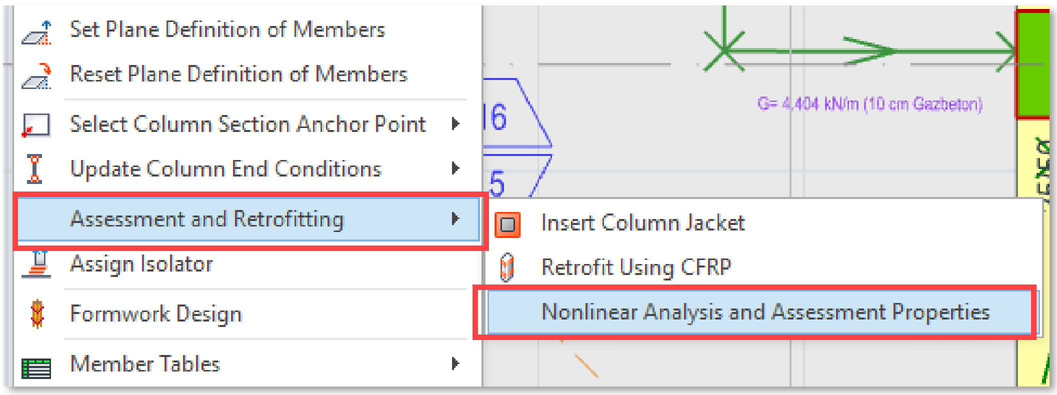
Estimated Reinforcements for Individual Members

As a new feature in ProtaStructure 2024 and ProtaStructure 2025, you can assign **estimated reinforcement values to individual members**.

1. Select the member and right click to load the contextual menu.

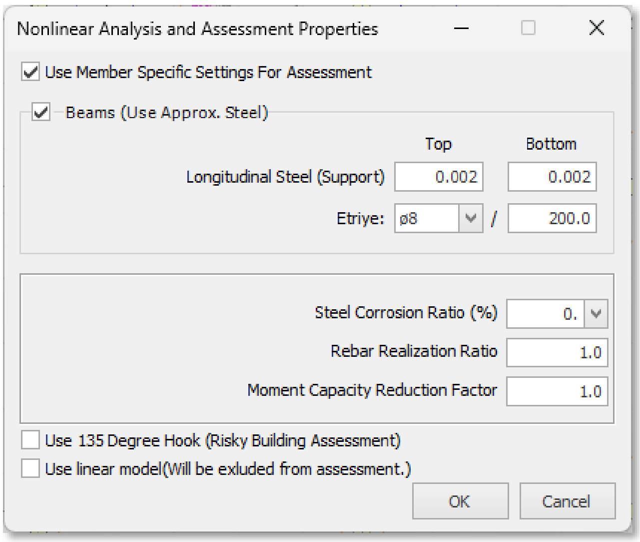
2. The functionalities related to assessment are collected under **Assessment and Retrofitting** subcategory.

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.

<u>The estimated reinforcement assigned to individual members</u> 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 <u>for the entire building</u> will be used. If global settings do not specify any estimated reinforcement, then the <u>actual detailed reinforcements on the members defined via design menus</u> will be used.

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

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

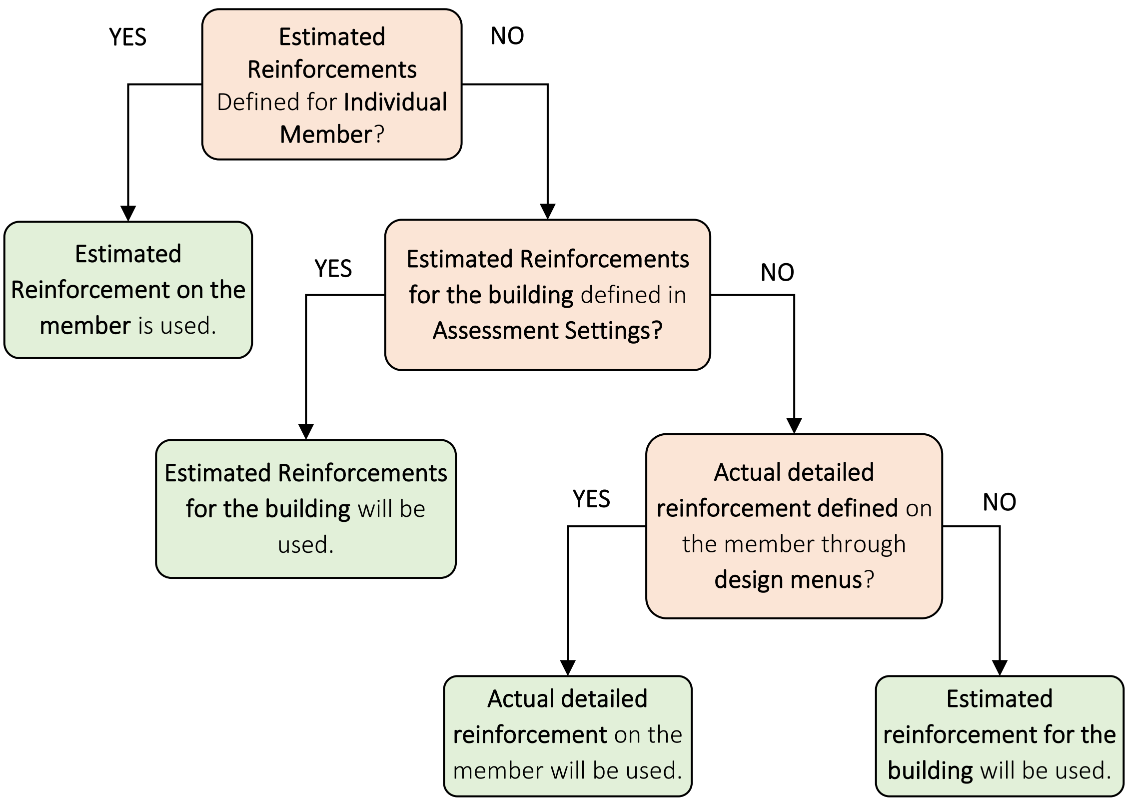
<u>Third Priority:</u> Actual detailed reinforcements defined through design menus directly on the member itself.

<u>If no actual reinforcement is defined</u> 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

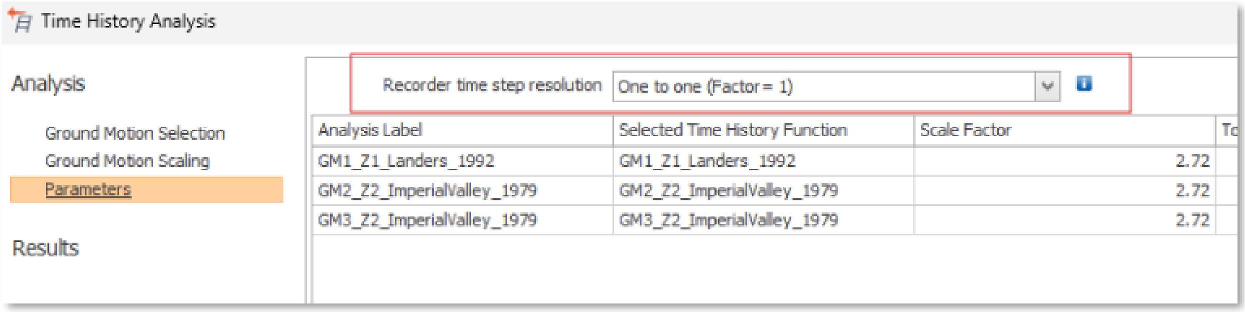
A new option is introduced in ProtaStructure to consider any member as "**Linear Elastic**" during assessment. This option is accessible through **Right Click > Assessment and Retrofitting > Nonlinear Analysis and Assessment Properties > Use Linear Model** setting.

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.

More Control Over Time-History Analysis Result Recorders

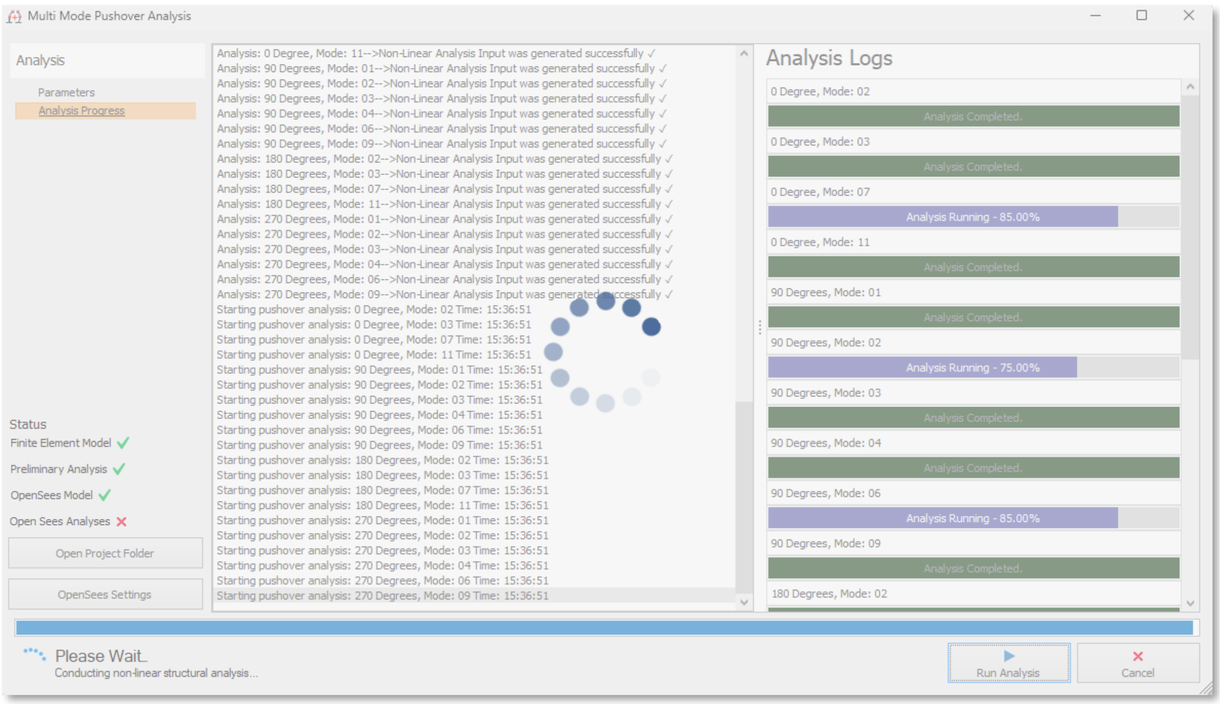
You will now have better control over time-history analysis by changing recorder resolution.

> This denotes the relationship between ground motion time step and analysis result time step. A higher factor means a smaller number of results will be generated which increases the performance and reduce the memory requirements. Factors lower than 1 will result in higher convergence rate and accuracy at the cost of memory and performance.



Parallel Processing in OpenSees Integration

We have seamlessly integrated OpenSees with parallel processing for better UI representation and responsiveness of analysis progress.



Modeling Improvements for Better Convergence

We have implemented modelling improvements for better convergence. 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.

Performance Improvements

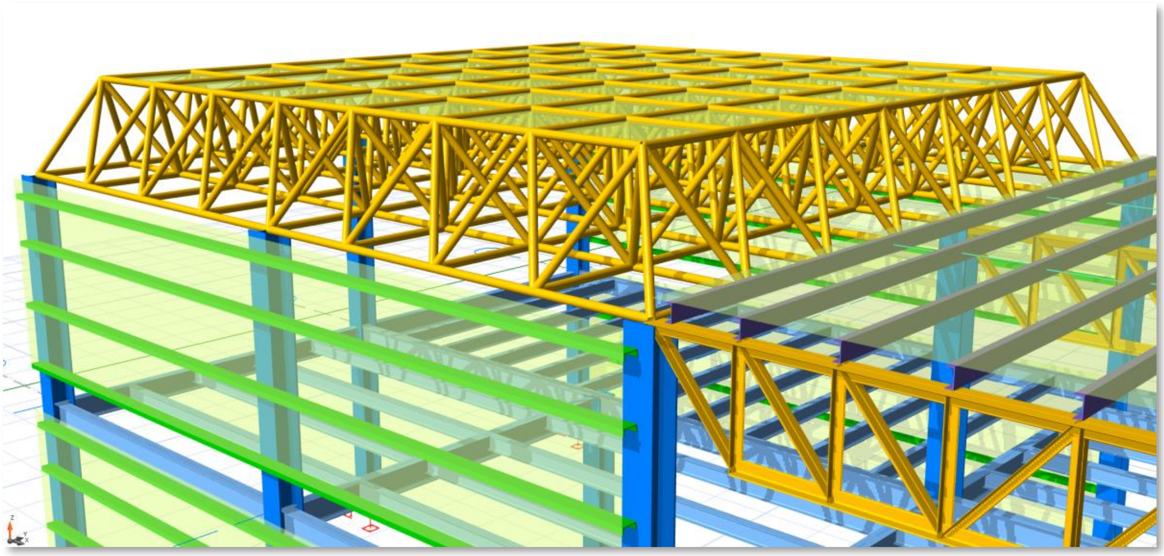
And finally, we have made performance tweaks that will create difference.

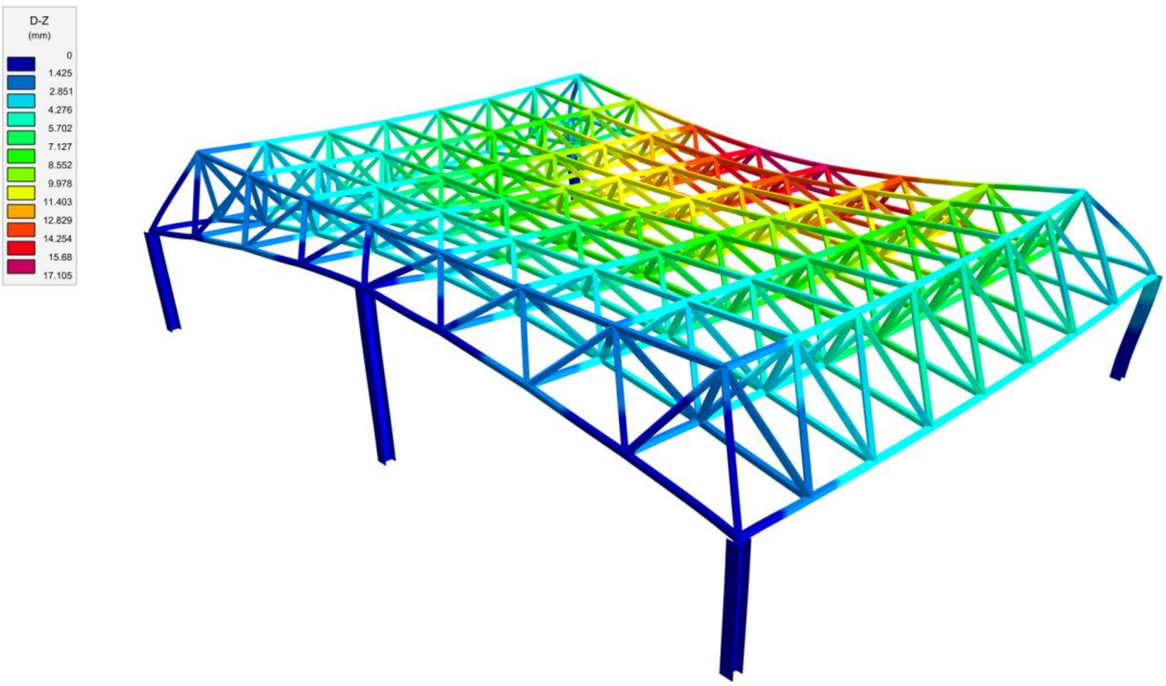
New Modeling Tools

Space Trusses

ProtaStructure offers powerful steel modeling macros like trusses, purlins, girts, braces, steel domes, etc. With these macros, creating a fabrication-ready steel model with accurate eccentricities and member positions is very easy.

In ProtaStructure 2025, we have developed **Space Truss** macro. This is a powerful addition to our steel modeling, analysis and design toolset.





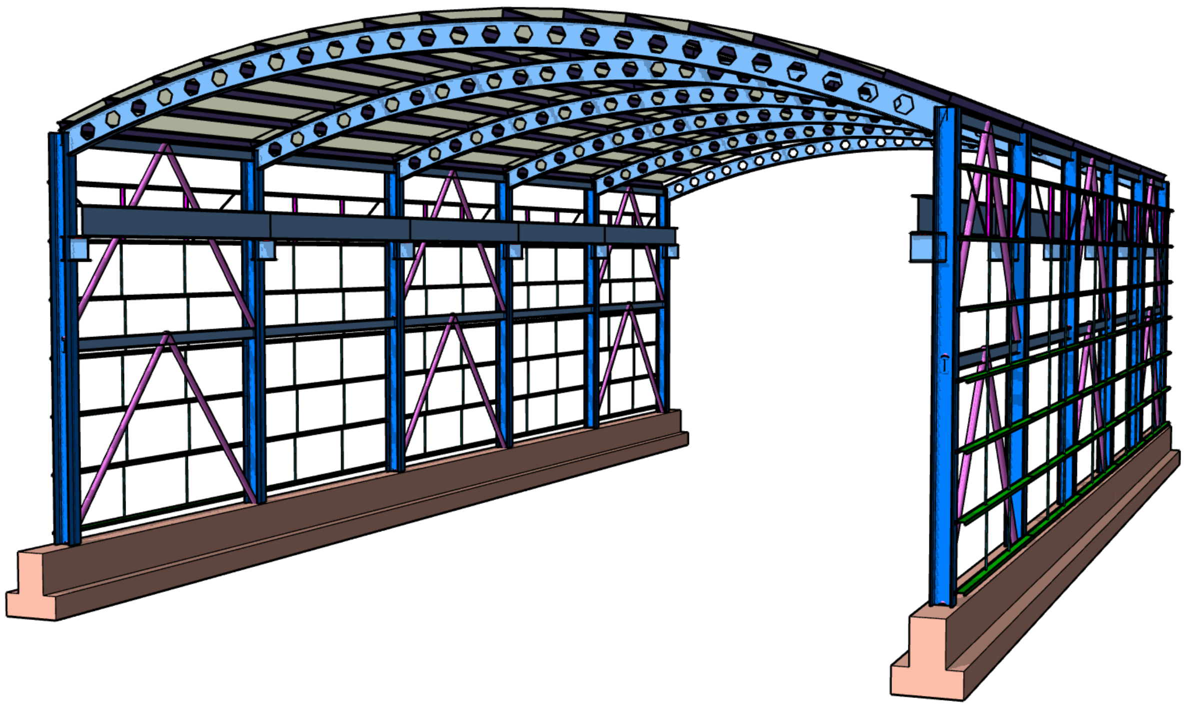
Please refer to our new design guide on space trusses for detailed explanations, modeling tips and more:
ProtaStructure Design Guide – Space Trusses

Castellated Beams

A castellated beam is created by slicing an I-beam longitudinally along its web in a specific pattern. The goal is to separate and then rejoin the beam with a deeper web. These beams are categorized based on the shape of the openings in the web section, with common shapes including hexagonal, circular (also known as cellular), octagonal, and diamond.

Castellated beams offer increased depth compared to the original section, while maintaining nearly the same weight. By welding square or rectangular plates between the cut halves, an even deeper castellated beam with octagonal holes can be formed. The key advantage of castellated beams is their ability to increase depth without adding weight, making them highly efficient for enhancing load-bearing capacity. They provide greater moment-carrying capacity without additional steel.

ProtaStructure 2025 now allows you to model, analyze and design castellated steel beams.



Please refer to our new design guide on castellated beams for detailed explanations, modeling tips and more:
ProtaStructure Design Guide – Castellated Beams

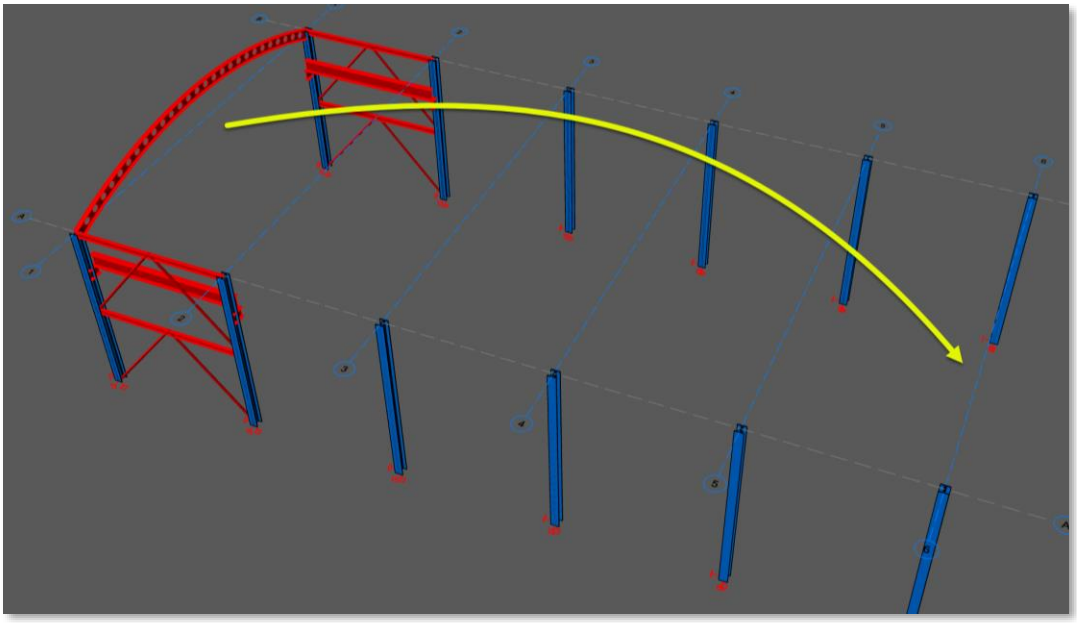
Copy/Move Special Command

Copy/Move Special

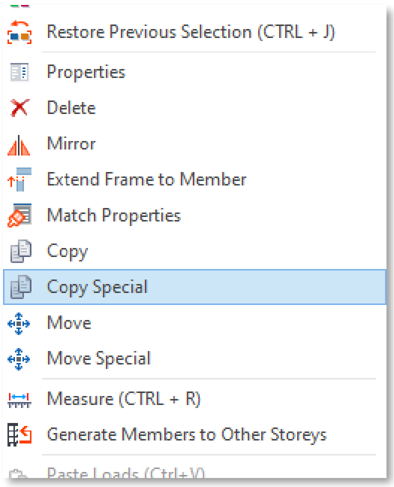
ProtaStructure already provides an **Interactive Copy/Move** command which requires picking a **Base Point** and specifying the **Destination Point** by snapping to available points on the modeling screen.

Unlike the Interactive Copy/Move command, **Copy/Move Special** provides a dedicated UI for entering relative copy/move distances in **global X, Y** and **Z** axes. In addition to this, you can specify the **number of copies** to be generated.

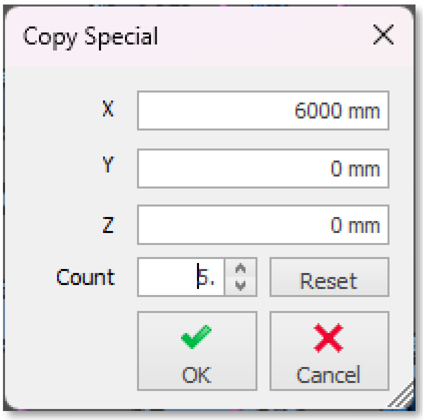
1. Select the members to be copied/moved multiple times.

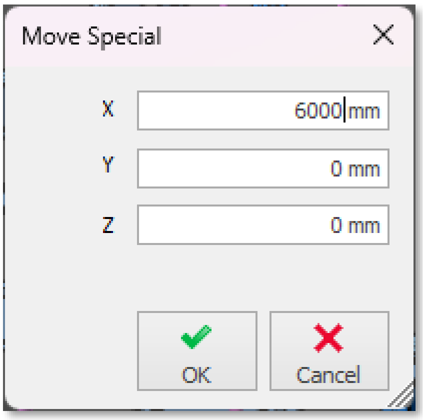


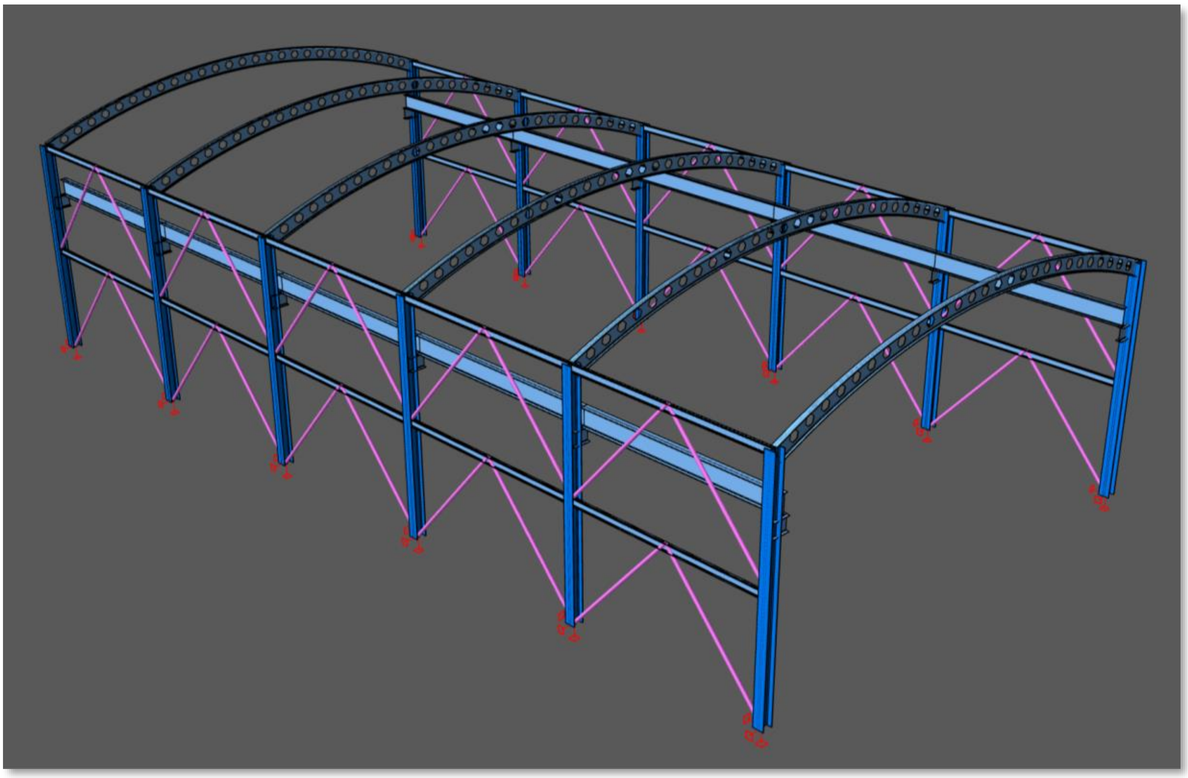
2. Right click and pick **Copy Special** or **Move Special** command depending on what you want to do.



3. A small window will pop-up for specifying relative translations in **Global X, Y** and **Z** coordinate axes. For the copy command, the **Count** field will be visible for specifying number of copies to be generated with the same relative translations.







Summary of Member Replication Tools

ProtaStructure supports several member replication and manipulation tools for different members. Not all these tools are applicable to all the members, depending on how complicated the member definition is and how tightly the member is dependent to its adjacent members.

Latest Addition to these tools is **Copy Special, Move Special** and **Match Properties**. The table below summarizes the tools and their applicability on different member types.

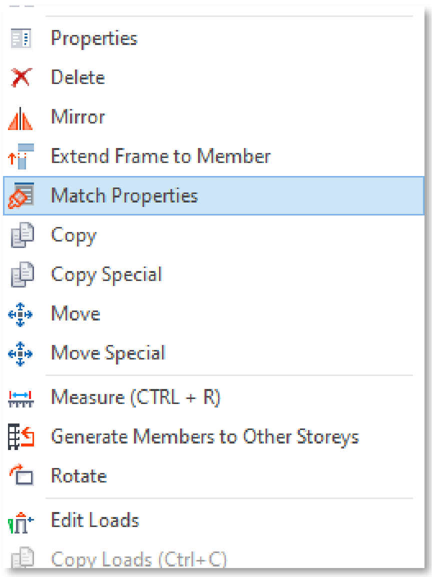
Member	Copy	Move	Mirror	Copy Special	Move Special	Match Properties	Use Last Member Properties
RC Columns	-	+	+	-	-	+	+
RC Beams	-	+	+	-	-	+	+
RC Slabs	-	-	+	-	-	+	+
RC Rib/Waffle Slabs	-	-	+	-	-	+	+
Steel Columns	-	+	+	-	-	+	+
Steel Beams (Frame Member)	+	+	+	+	+	+	+
Steel Frame Members	+	+	+	+	+	+	+
Steel Frame Groups (Macro Member)	+	+	+	+	+	+	+
Brace Members (Macro Member)	-	-	+	-	-	+	+
Purlins (Macro Member)	-	-	+	-	-	+	+
Girts (Macro Member)	-	-	+	-	-	+	+
Trusses (Macro Member)	+	+	+	+	+	+	+
Space Trusses (Macro Member)	+	+	-	+	+	-	+
Steel Domes (Macro Member)	+	+	-	+	+	-	+

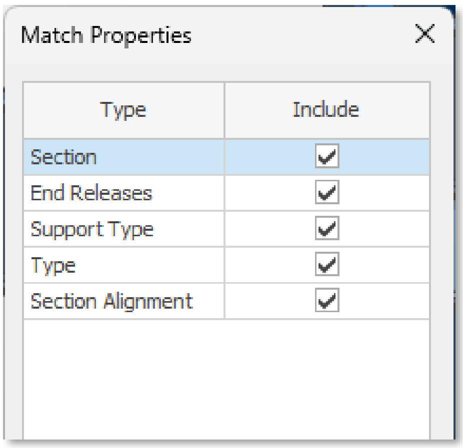
Match Member Properties

Suppose that you have two or more structural members, and you want them to have the same key properties.

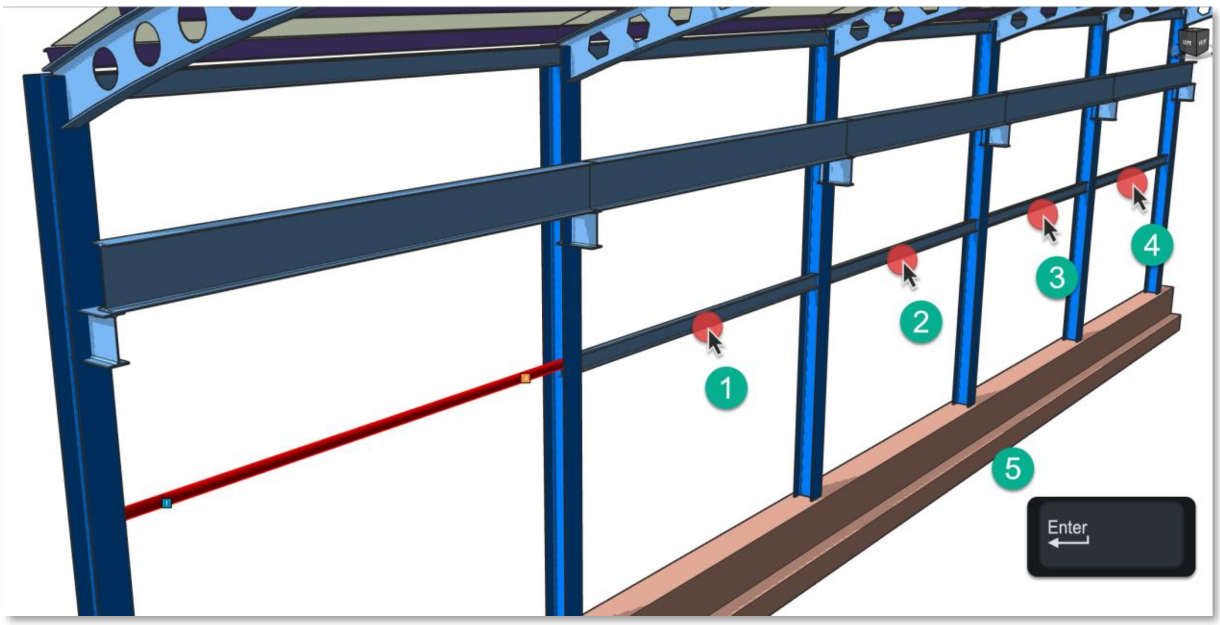
ProtaStructure 2025 will help you in such a case. You can use the new "**Match Properties**" command to copy the properties from one source element to other target elements.

1. Select the target member and pick the **Match Properties** command on the right click menu.
2. On the pop-up window, select the properties you want to equate and click **OK**. You can select different properties depending on the member type you have chosen.





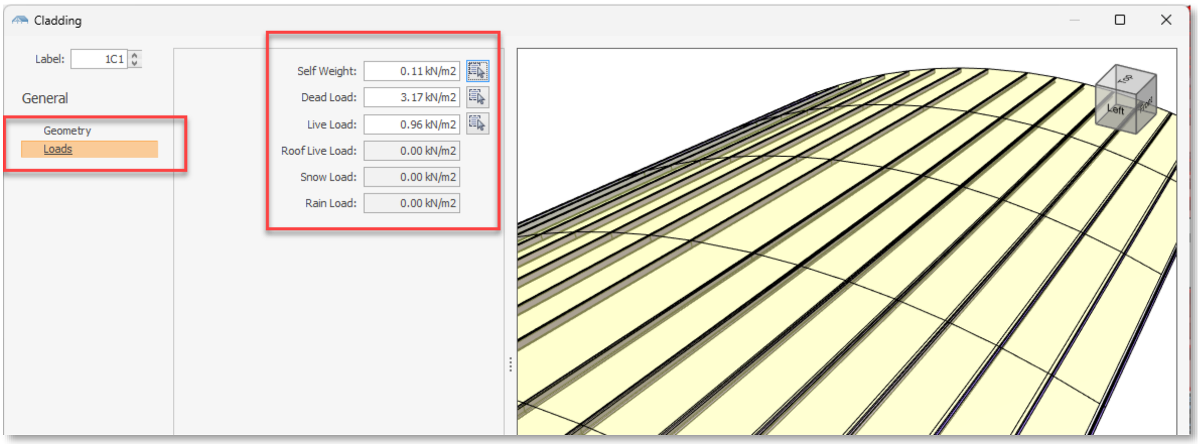
3. **Pick the target members** on the screen and hit **ENTER** on the keyboard when you are finished.



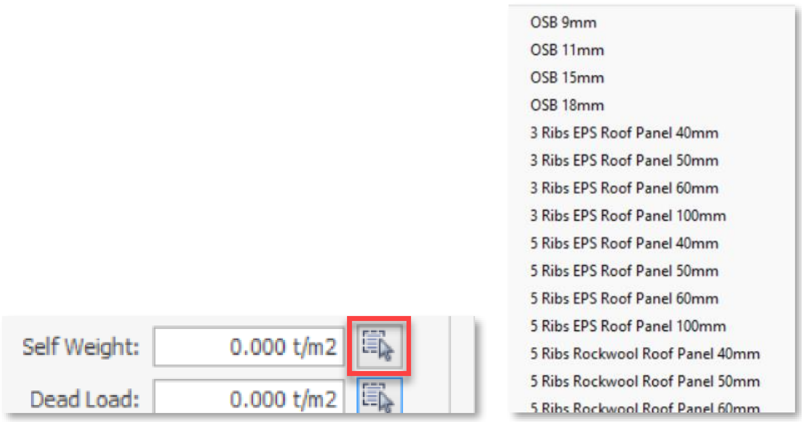
Quick Load Assignment to Claddings

Although **ProtaStructure Load Editor** already provides flexible tools for easy to load application in any direction on claddings (or any member types whatsoever), there is always room for improvement.

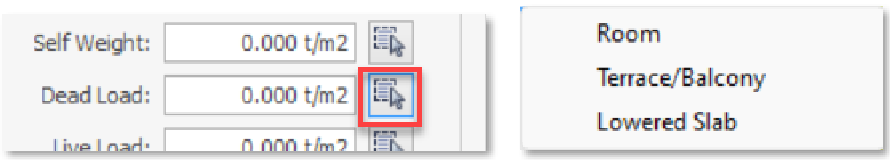
With ProtaStructure 2025, you can now directly assign area loads on claddings using their **Properties Window**.



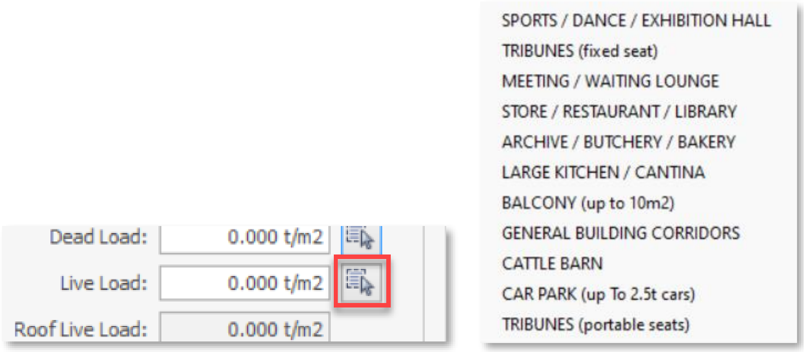
Self-Weight: Enter the self-weight of the cladding in this field. The self-weight will depend on the type of the cladding profile. We have provided a **library of cladding loads** for easy application of cladding self-weight. You can access the cladding load library by clicking the small button next to Self-Weight field.



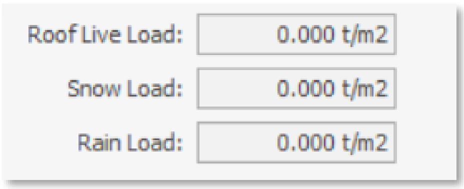
Dead Load: Enter the additional dead loads (finishes etc.) in this field. ProtaStructure already has a library of additional dead loads. You can access the cladding load library by clicking the small button next to Dead Load field. The library editor is in **Building Setout > Slab Additional Loads** ribbon tab.



Live Load: Enter the live loads in this field. You can access a quick list of imposed load values by clicking the small button next to Live Load field.

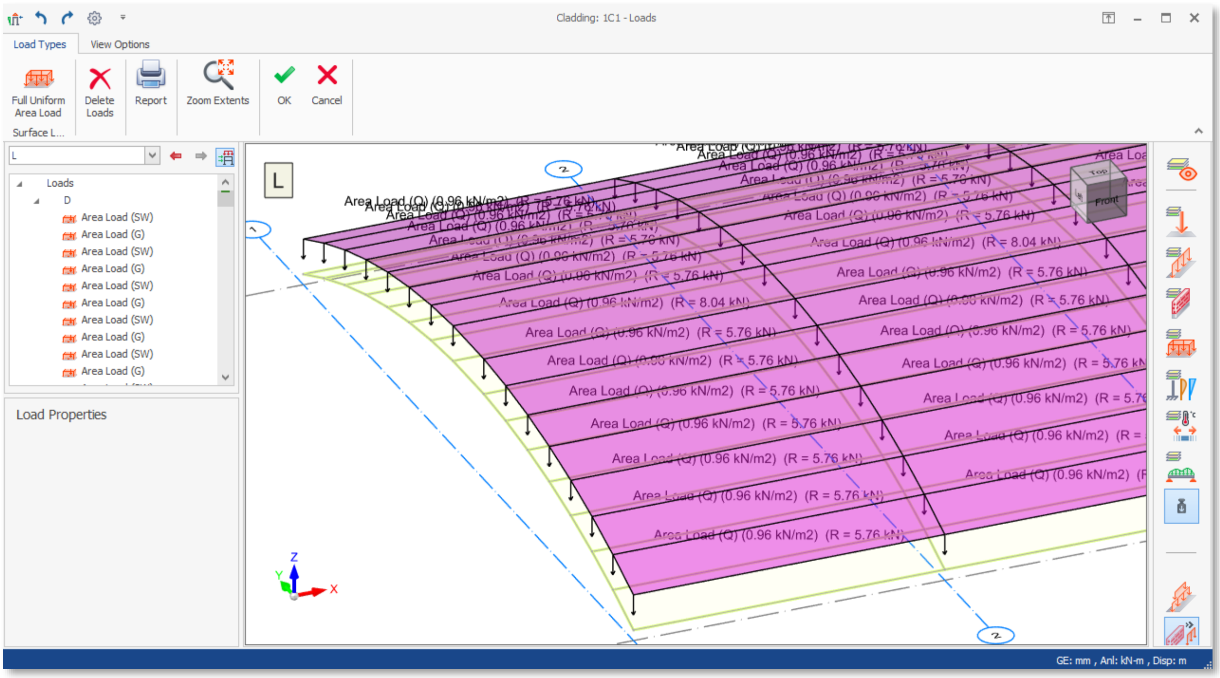


Roof Live Load, Snow Load and **Rain Load** fields will be activated if the associated load cases are defined in the Load Combinations Editor.



You can always open the load editor to verify the automated load assignment or assign new loads on the cladding.

If you have created user-defined load cases, you must use the Load Editor to assign loads to these load cases. The quick load assignment is only applicable for standard dead loads, live loads, roof live loads, snow and rain loads.

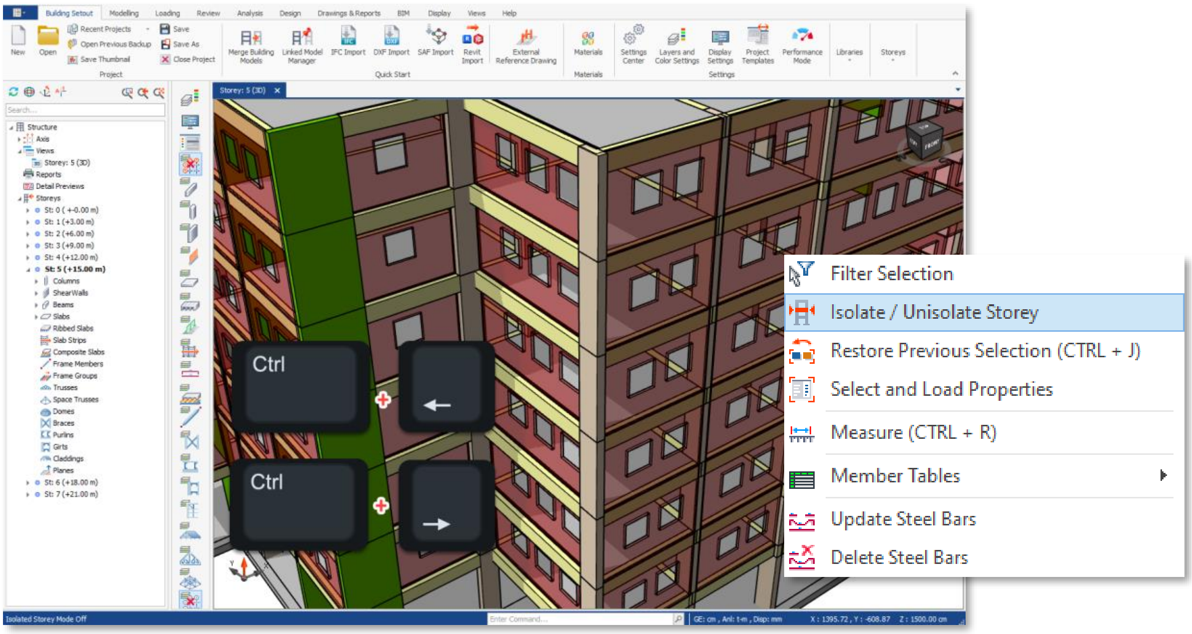


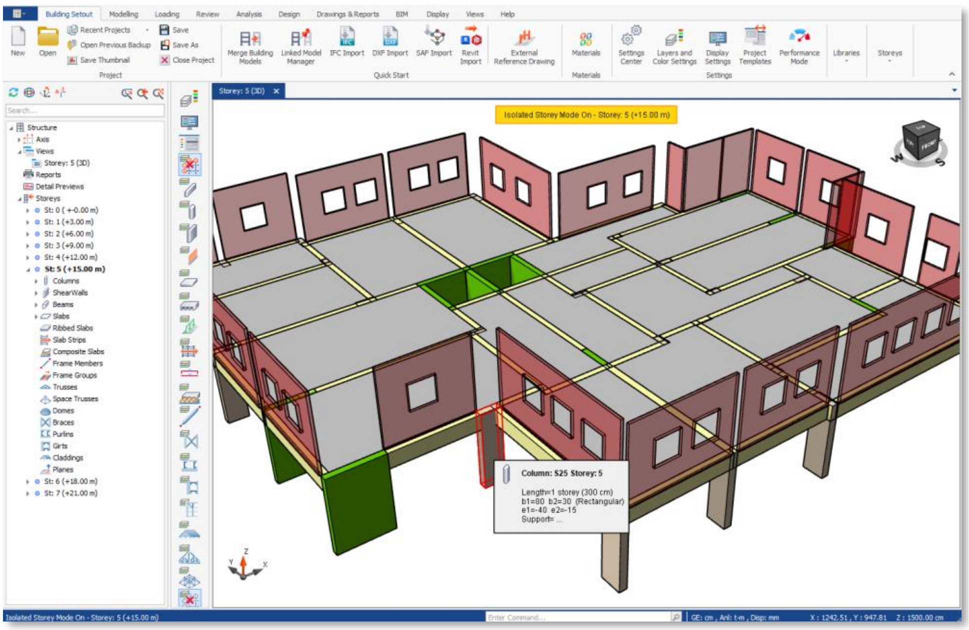
Storey Visibility in Right-Click Menu

3D view in ProtaStructure is essential as it helps us visualize and manipulate the whole model in 3D. There are some cases where you may want to focus on a storey and walk through it without the distraction of all the elements in the rest of the model.

For such cases you can now use the **"Isolate/Unisolate Storey"** option in the right click menu. You can also use **CTRL+LEFT ARROW** or **CTRL+RIGHT ARROW** key combinations on the keyboard to isolate and unisolate the current storey.

In addition to this, while in the Isolated Mode, you can use **CTRL+UP** and **CTRL+DOWN** key combinations to switch between storeys.

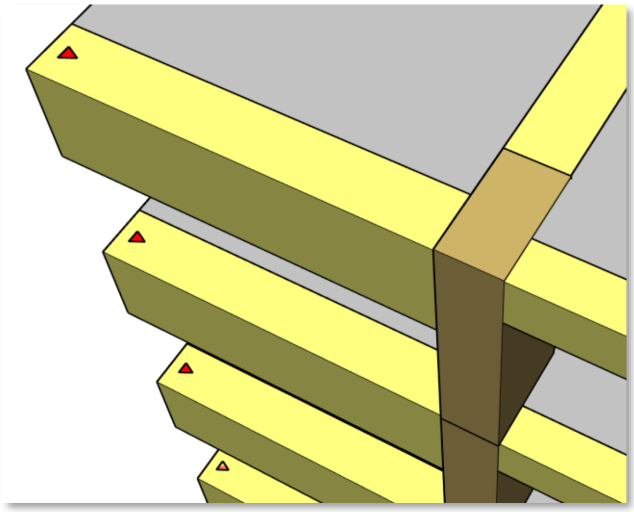


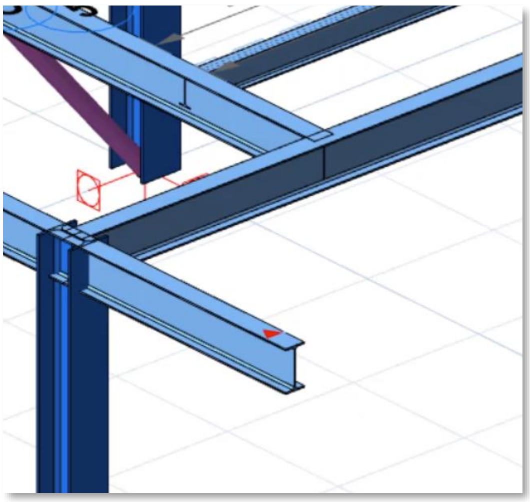


Cantilever Marks in 3D

In ProtaStructure, cantilever beams are designed and detailed exclusively. During modeling, it becomes very handy to see the cantilever ends of beams during inspection of a complicated model.

ProtaStructure now allows you to spot cantilever ends in 3D modelling environment. It was already available in Plan View in the previous versions, but we carried this feature to 3D view as well.





More Flexible Column End Releases

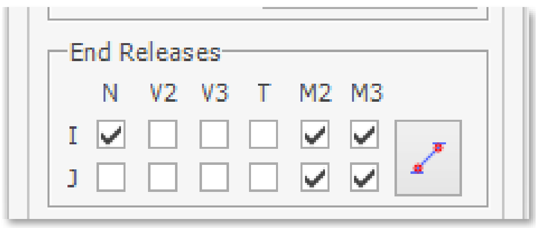
Another improvement has been made in the specification of **Column End Releases**. In ProtaStructure 2025, you can set the column end releases **for any of the 6 degrees of freedoms** at each end.

Previously, only **M22** and **M33** could be released at the same time. **Axial**, **Shear** or **Torsional Releases** <u>were not</u> available and could not be assigned independently.

ProtaStructure creates the analytical model using a central physical Structural BIM model. This means that, ProtaStructure will automatically reflect the industry best practices and years of structural engineering expertise to the automated creation of analysis model. While doing so, ProtaStructure also provides sufficient flexibility to allow users to override decisions taken by the software. As Prota software we always seek for the correct blend of automation and user interference.

To adjust the column end releases:

1. Open the **Column Properties** window.
2. In the **End Releases** section, adjust the end releases the way you want. You can use the **Quick Setting** button on the right to apply moment releases in two directions quickly.



Increased flexibility in column end releases is mostly driven by the steel design requirements where there are various connection and support types that can be applied to steel columns.

For example, a wind column can be hinge connected to the rafter in two directions with a slotted plate with no axial load transfer from the roof.

We hope that this feature will help you build more optimized models for different scenarios.

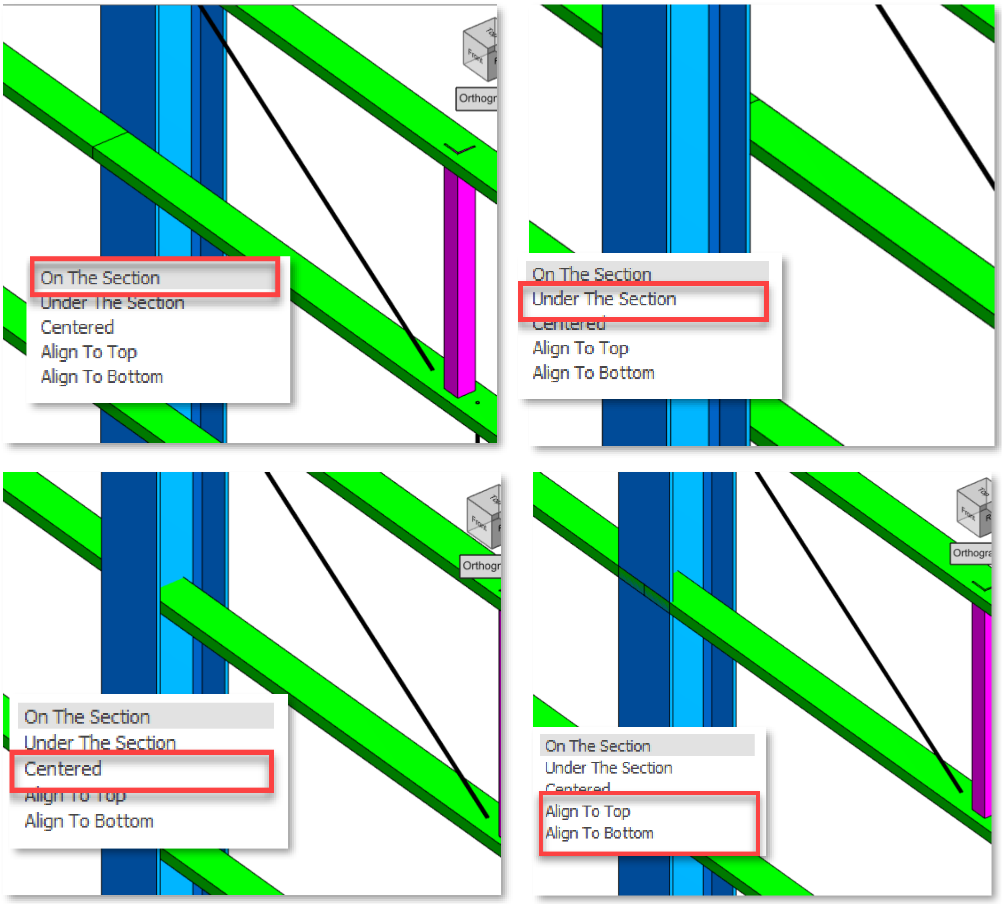
New Girt Alignment Options

One important strength of ProtaStructure is providing powerful tools to accurately create the physical **structural model**.

For this purpose, we have developed **macros** for steel members such as purlins, trusses, girts, steel domes, frame groups, so on and so forth.

These macro members aim to provide a practical modeling experience as well as a **seamless detailing** where you won't need any further editing.

Girt section alignment options are now extended to support more variations such as **Centered**, **Align to Top** and **Align to Bottom**.

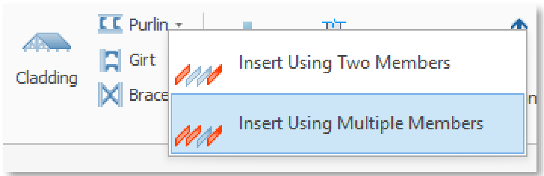


Insert Purlins Using Multiple Members

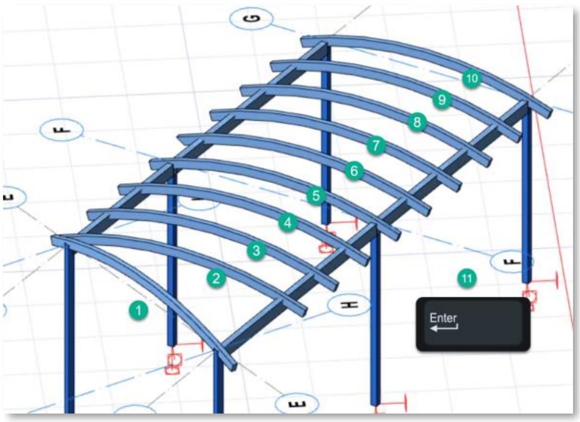
While inserting purlins in ProtaStructure, it is usually sufficient to pick the first and last supporting member. All the intermediate members will be detected, and purlin will be inserted automatically considering the intermediate member orientation.

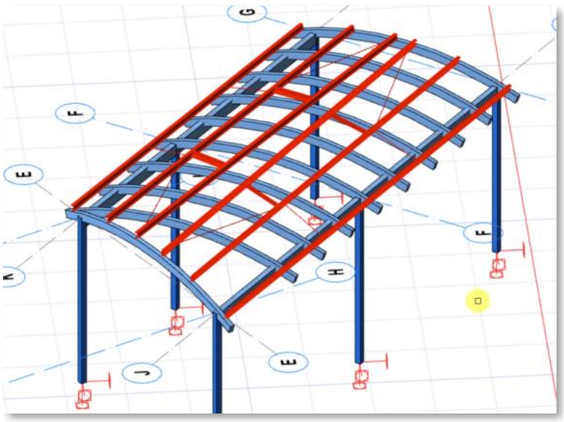
For some cases, you may need to pick the supporting members manually for more precision, simply because the automatically detected intermediate members may be redundant or ProtaStructure may not pick the one you wanted when there is more than one supporting member in the same location.

That's why we developed a new insertion method for purlins: **Insert Using Multiple Members**



1. Run the Insert Using Multiple Members command under **Modeling > Purlin** tab.
2. **Pick the supporting members one by one** on the modeling environment.
3. Hit **ENTER** on the keyboard when you are finished selecting.
4. **Purlin Properties** window will pop up. Specify the purlin parameters and click **OK** to finish insertion.

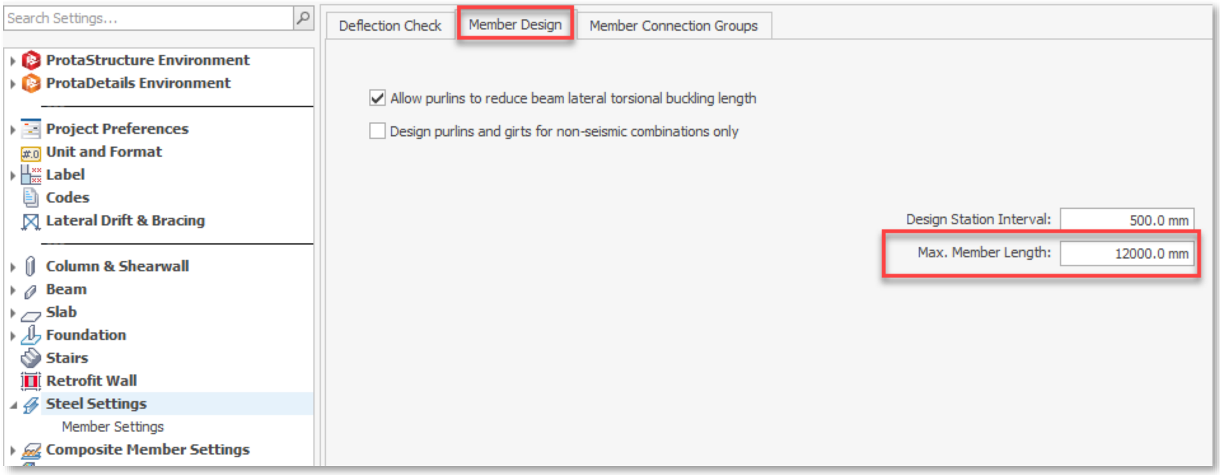




Continuous Purlins

In previous versions of ProtaStructure, purlins were automatically cut at every support and a new purlin frame was inserted after that.

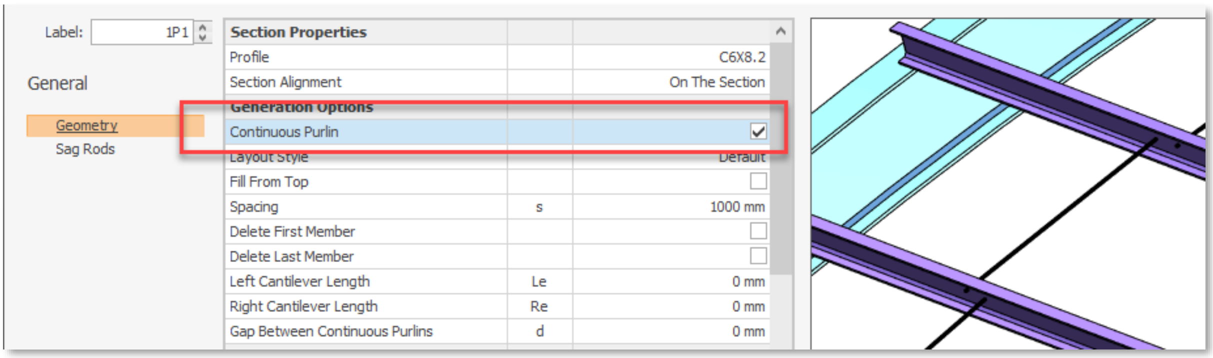
With ProtaStructure 2025, you can now specify a **Maximum Member Length (Cut Length)** for purlins. You can access this setting from **Steel Settings > Member Design**.

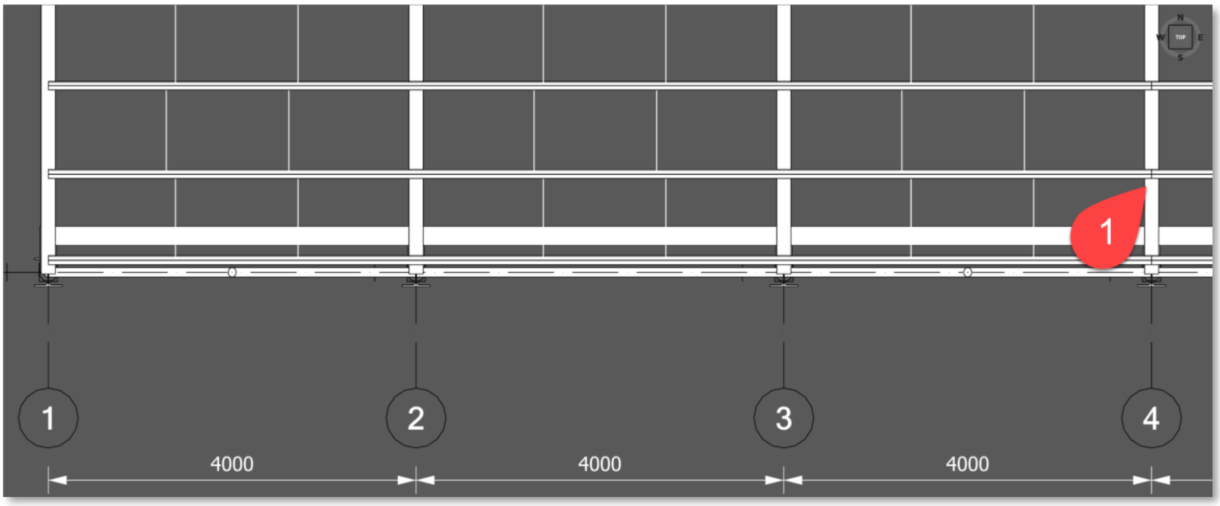


To model a continuous purlin:

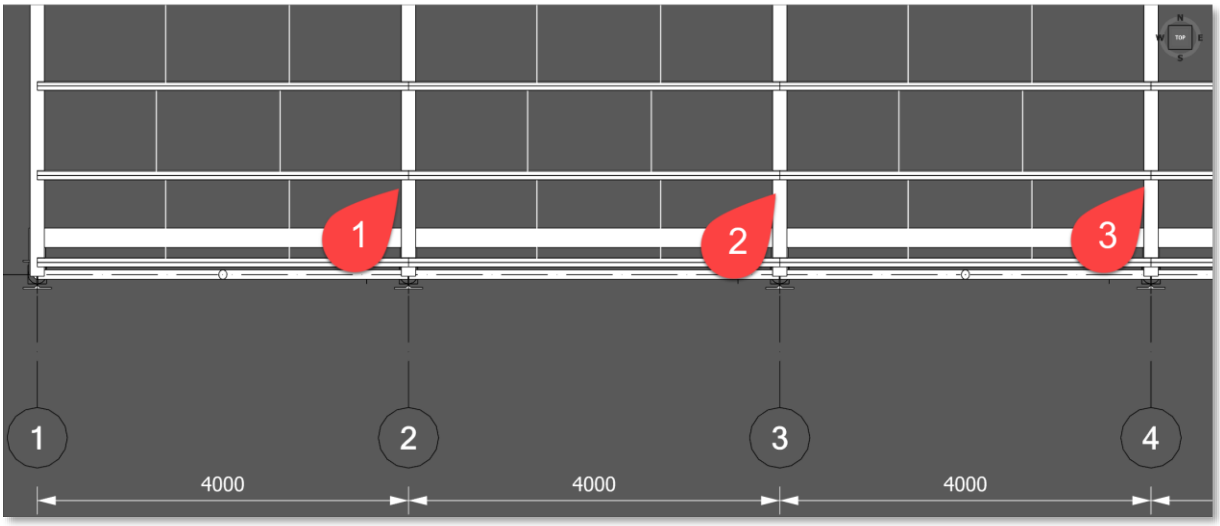
1. Specify the cut length using the **Steel Settings > Member Design > Max. Member Length** field. The default value is 12 m.

2. While insert the purlin, check the **Continuous Purlin** option on the **Purlin Properties Window**.





Continuous Purlin Option Checked: *The purlin is only cut at third support. Cut length is specified as 12 m in a 4 m bay frame*



Continuous Purlin Option Unchecked: *The purlin is cut at each support*

The purlins will be cut from the closest support to the given cut length, ensuring they don't exceed the manufactured steel length.

Between the cut points they will be treated continuous and end releases will be assigned accordingly. They are also communicated to **ProtaSteel** with the correct cut length and end releases so that you have a **fabrication-compatible** structural model.

This feature is not supported for Girts yet. We are planning to apply this to the girts with the upcoming maintenance releases.

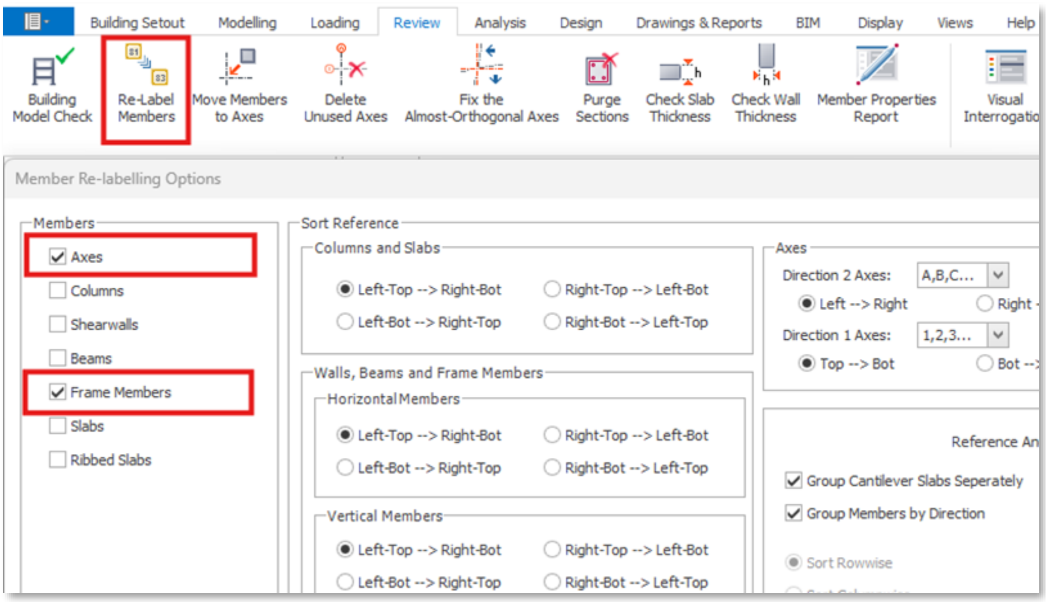
Other Modeling Improvements

The improvements and user feedback we have incorporated in ProtaStructure 2025 are not limited to the ones explained above. We are committed to focusing on each small user feedback once we believe that it will benefit many of our users in ProtaStructure's general concept and coherence.

Following are several additional improvements we have made in modelling in ProtaStructure 2025:

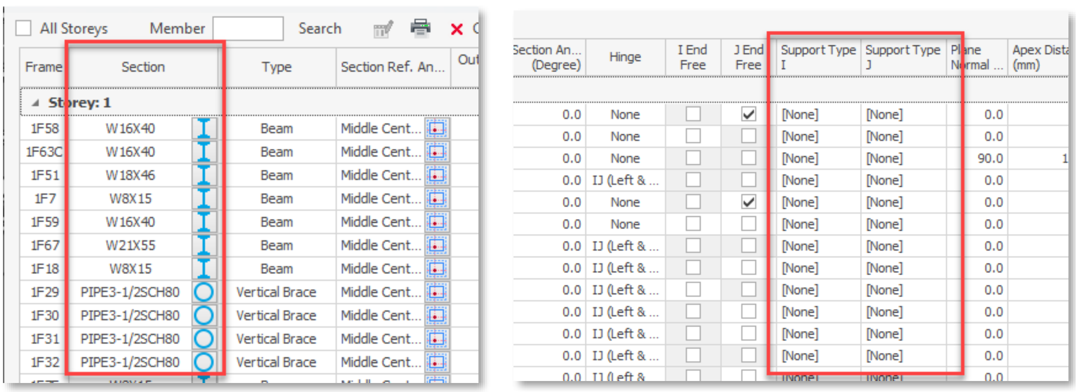
Relabel Members

You can now relabel axes and frame members using the **Re-Label Members** command. Relabeling options for other steel members are coming soon.



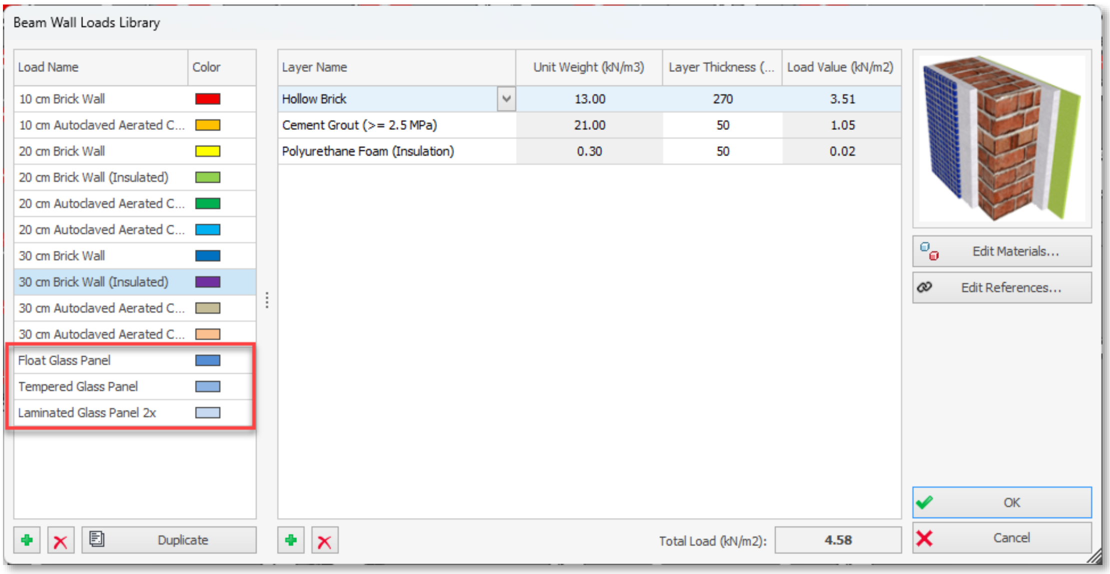
Frame Member Tables

Member tables provide a convenient and indispensable **spreadsheet-style batch editing** functionality. We continuously improve the Member Tables as we introduce new member types or new features related to the members. With ProtaStructure 2025, we have introduced the batch editing of **Sections** and **Support Types** in this table.



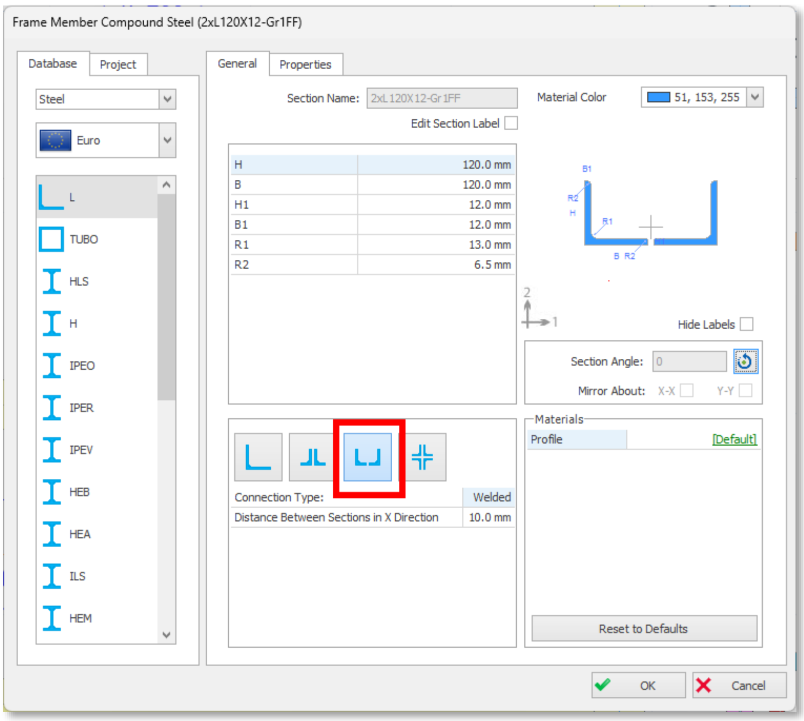
Glass Material in Wall Loads Catalog

Glass is a widely practiced material for facades of high-rise buildings and office partition walls. We have added glass material layer and different preset glass wall loads to the partition wall load library. You can quickly assign glass wall loads on slabs and other primary load carrying members now.



Face-to-Face Twin L Profiles

We have introduced Face-to-Face Twin L-profiles to our compound steel profile catalog with ProtaStructure 2025.



Advanced BIM Integration Tools

Export Steel Connections to IdeaStatica

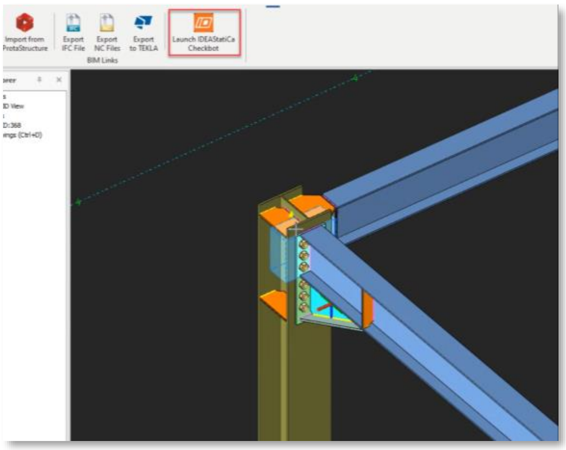
ProtaSteel provides detailed design reports for a large variety of steel connections using analytical methods. Analyzing complicated or user-defined steel connections with Component Based Finite Element Analysis may be required in your project.

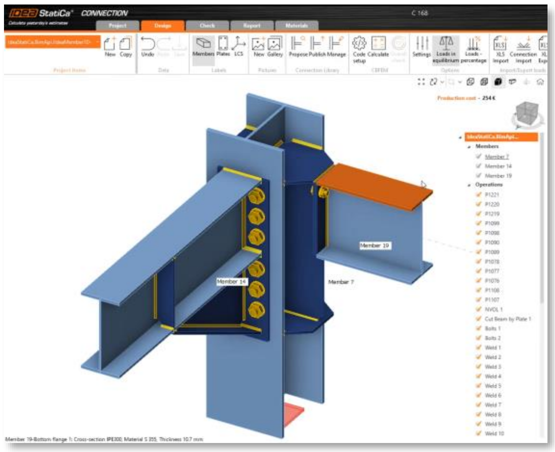
In Prota Software, we believe in open collaboration. In each new version, we strive to create links between our products and other software solutions and file formats in the industry.

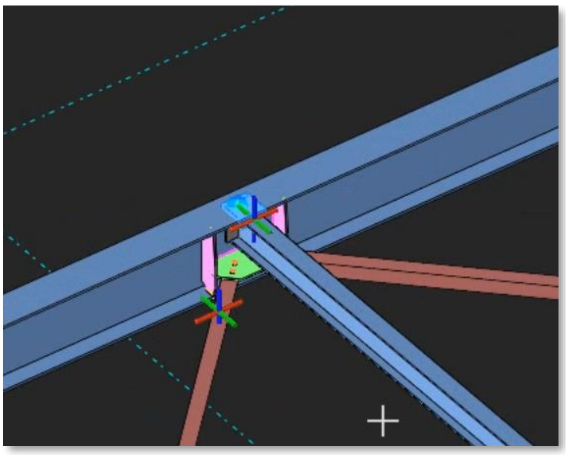
Our last addition is the **IdeaStatica CAD Links**. You can now export your automatic or user-defined connections to IdeaStatica and perform CBFEM analysis if you need to.

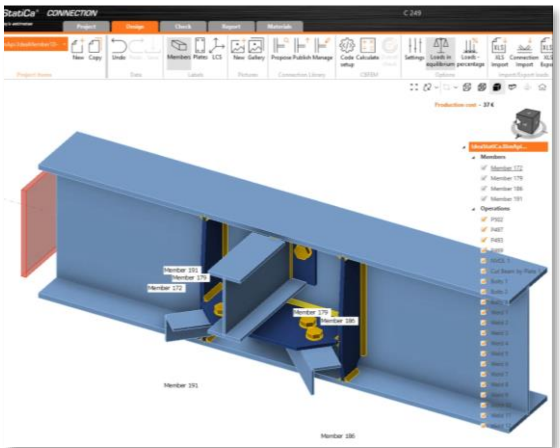
All you must do is to initiate **IdeaStatica Check Bot** in ProtaSteel and select the primary members and connection components by following the instructions on the wizard window.

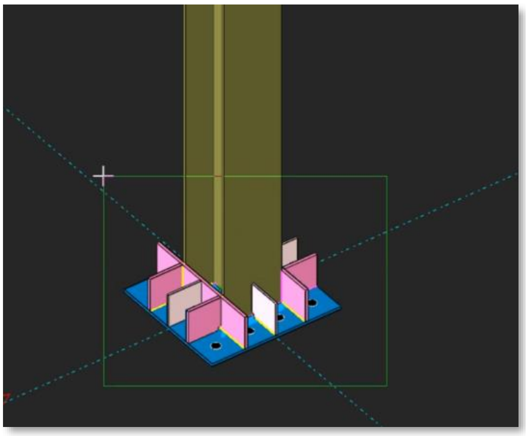
ProtaSteel is a sophisticated fabrication software which provides design and detailing to our users, covering a large scope. The IdeaStatica CAD Link API is designed for CAD information communication rather than internal forces and design information. We are working on a few workarounds to transfer of internal forces.

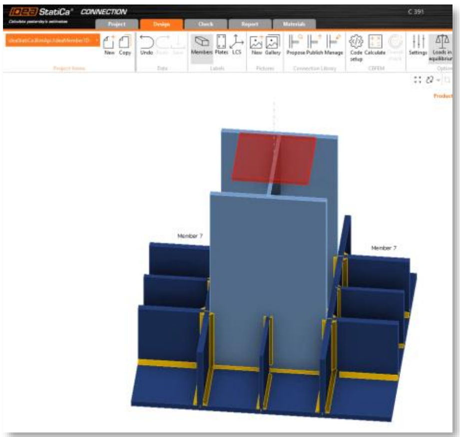










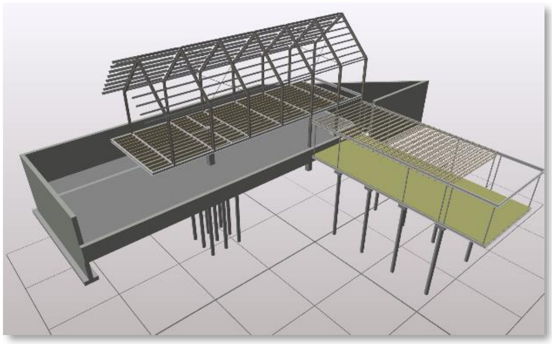


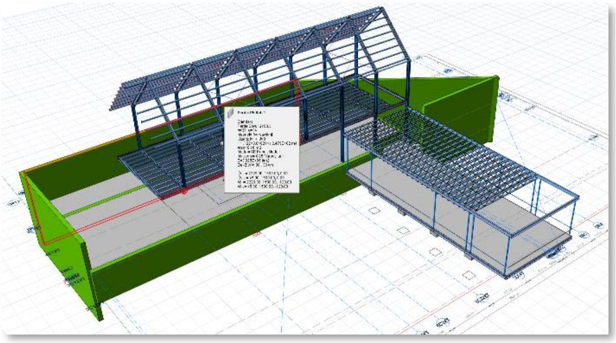
Enhancements in IFC Import and Export

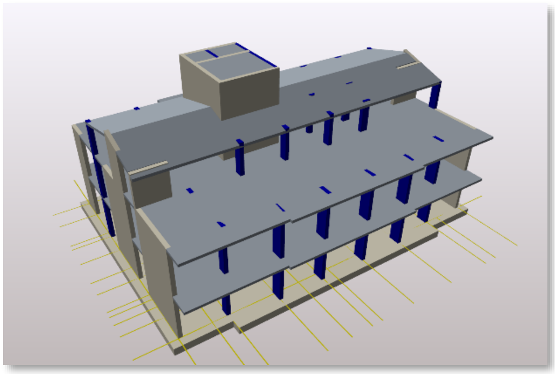
Importing and exporting a consistent IFC format is a challenging task.

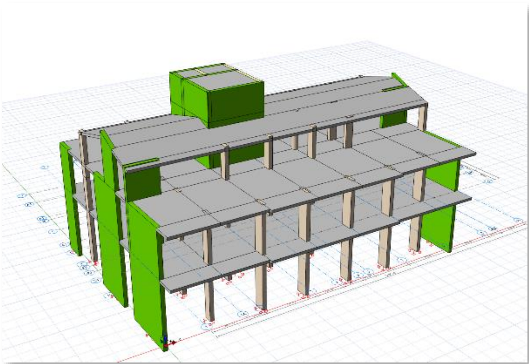
There are various adaptations and objects employed by different architectural and structural software on the market. Unfortunately, data representation and consistency are a big problem in a seamless IFC communication.

With ProtaStructure 2025, we keep our commitment to continuously develop our IFC import and export links. We have continued fine-tuning our IFC support with wide range of models we received from our users.



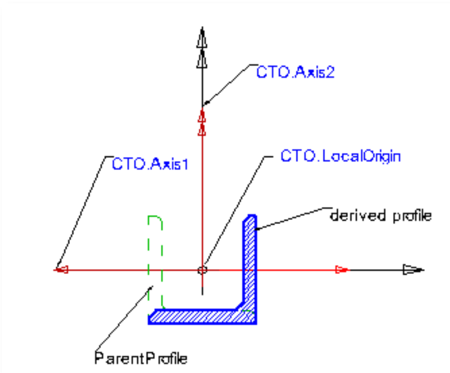


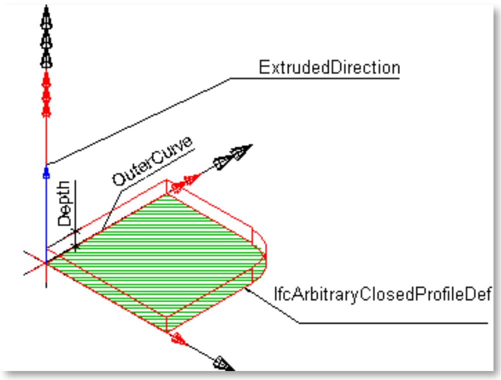


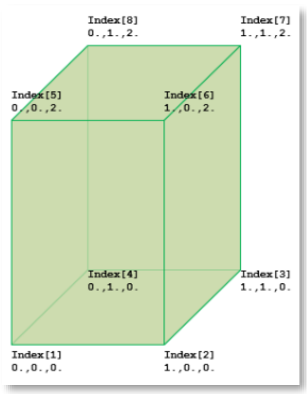


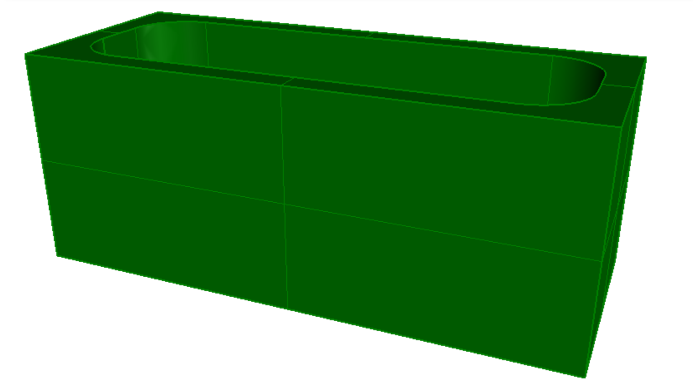
New Object Type Definitions

While trying to support all the model variations received from our users, we found ourselves adding more and more object types into our definitions database. A few examples for these are **IFCDerivedProfile**, **IFCPlate**, **IFCPolygonalFaceSet** and **IFCBooleanResults** objects.



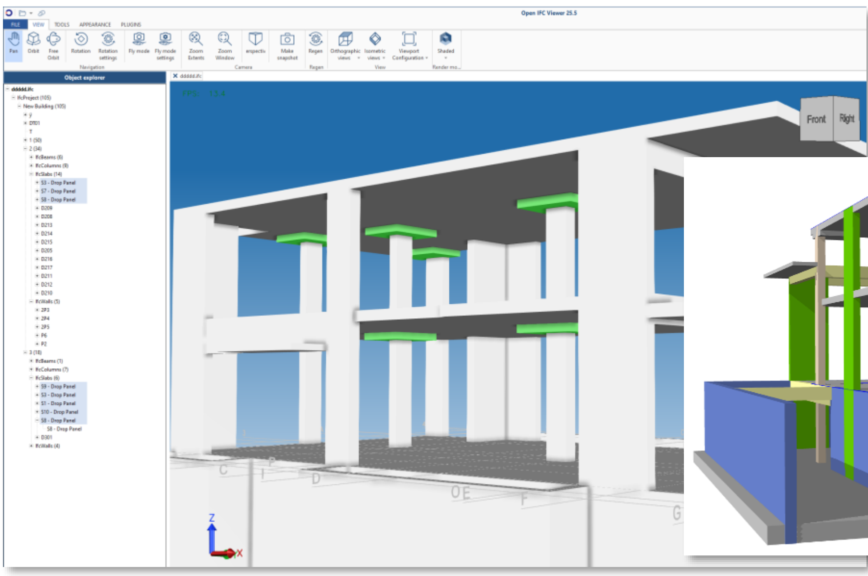


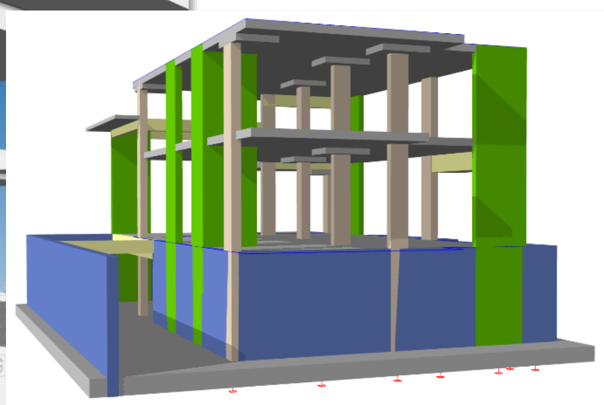




Column Drop Panels to IFC

As a new addition, we can now export column drop panels to IFC.





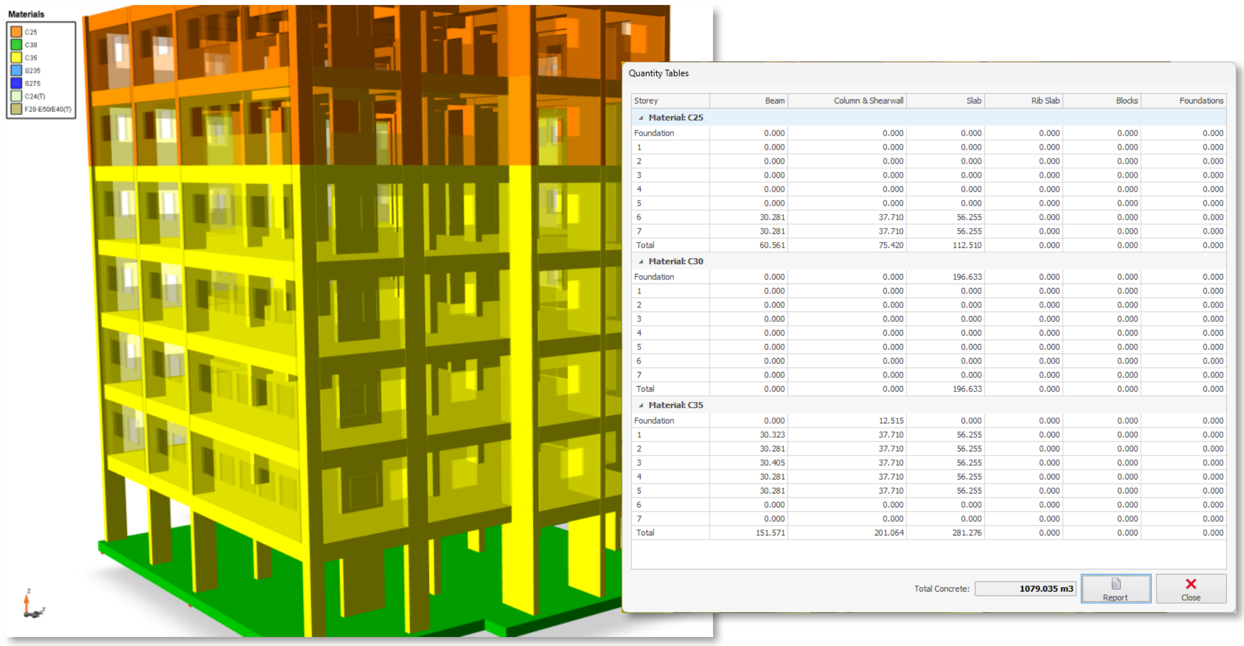
Improved Concrete Quantities

Thanks to the hierarchical material system in ProtaStructure, you can assign **different concrete grades** to **different stories** or **individual members** in a drill-down approach. There is also a **default concrete grade** that is used by members with no material overrides.

In the ProtaStructure Suite workflow, we normally obtain rebar quantity take-off and other detailing output in ProtaDetails. One exception is that you can get the concrete and formwork quantity take-off for the whole model in ProtaStructure. In ProtaStructure 2025, we have made a few improvements here:

You can obtain quantity take-off in your projects categorized with respect to any different concrete grades in the project. So, you will know how much the contractor needs to order for each grade of concrete for each floor.

Another improvement is that the concrete quantity take-off now includes **column drop heads** as well.



Storey	Beam	Column & Shearwall	Slab	Rib Slab	Blocks	Foundations
Material: C25						
Foundation	0.000	0.000	0.000	0.000	0.000	0.000
1	0.000	0.000	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000	0.000	0.000
6	30.281	37.710	56.255	0.000	0.000	0.000
7	30.281	37.710	56.255	0.000	0.000	0.000
Total	60.561	75.420	112.510	0.000	0.000	0.000
Material: C30						
Foundation	0.000	0.000	196.633	0.000	0.000	0.000
1	0.000	0.000	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000	0.000	0.000
Total	0.000	0.000	196.633	0.000	0.000	0.000
Material: C35						
Foundation	0.000	12.515	0.000	0.000	0.000	0.000
1	30.323	37.710	56.255	0.000	0.000	0.000
2	30.281	37.710	56.255	0.000	0.000	0.000
3	30.405	37.710	56.255	0.000	0.000	0.000
4	30.281	37.710	56.255	0.000	0.000	0.000
5	30.281	37.710	56.255	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000	0.000	0.000
Total	151.571	201.064	281.276	0.000	0.000	0.000

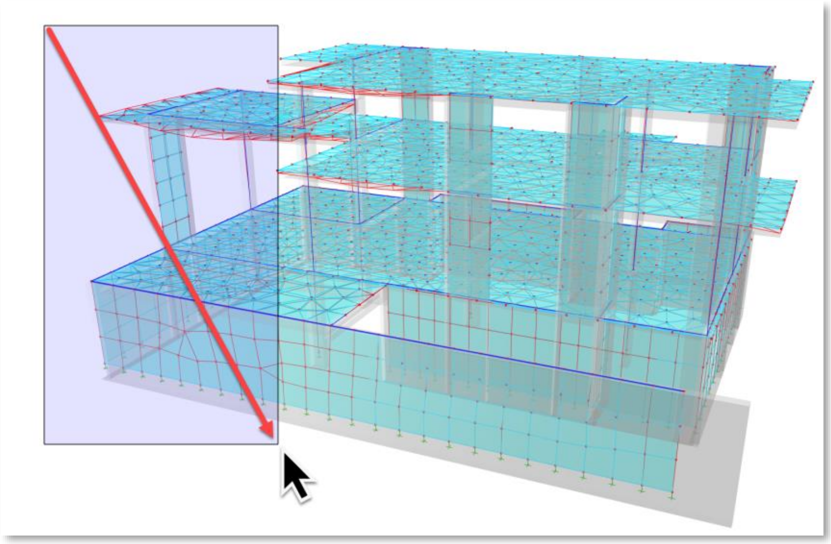
Analysis Post Processing

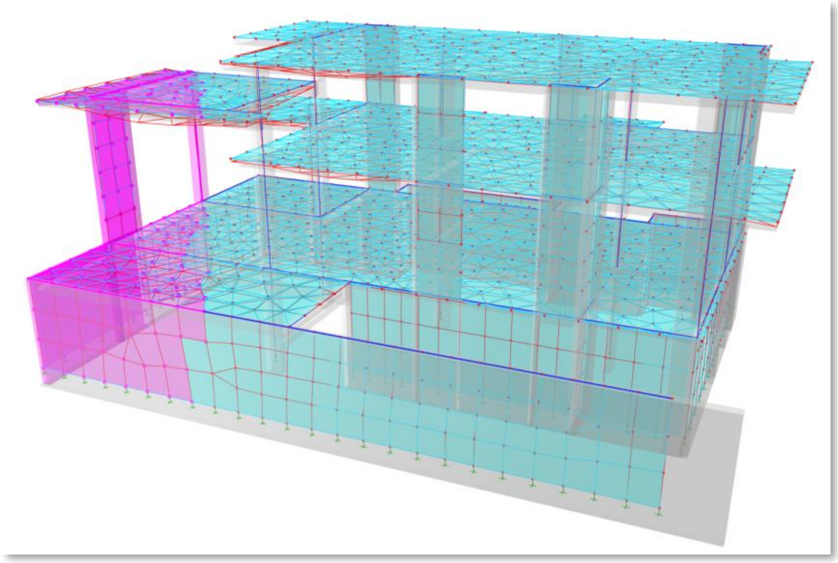
User Interaction Enhancements

With the latest update, we have made significant enhancements to user interactions in the Analytical Post Processor.

Window Selection

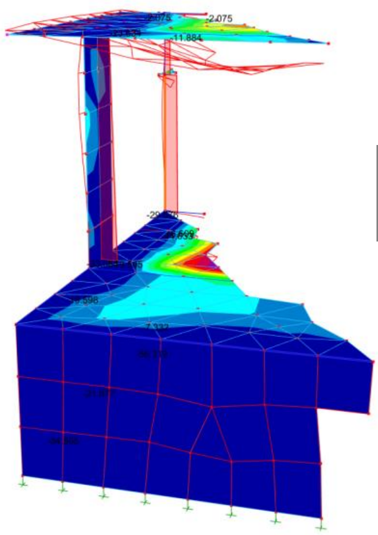
You can now select a part of the model by **boxing around** them.

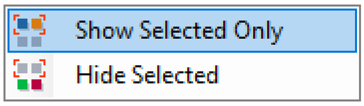




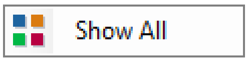
Isolating Selected Members

After selecting one or more members with window selection, right click and pick **Show Selected Only** command. The results will be displayed only for selected part of the model.



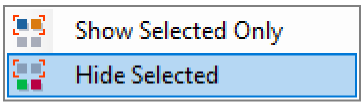


To view the entire model again, right click and pick **Show All** command.



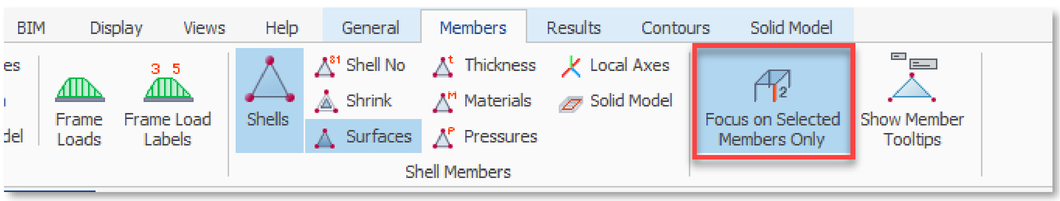
Hiding the Selected Members

Instead of showing only the selected members, you can choose to hide them. For that purpose, select one or more members, right click and pick **Hide Selected** command.

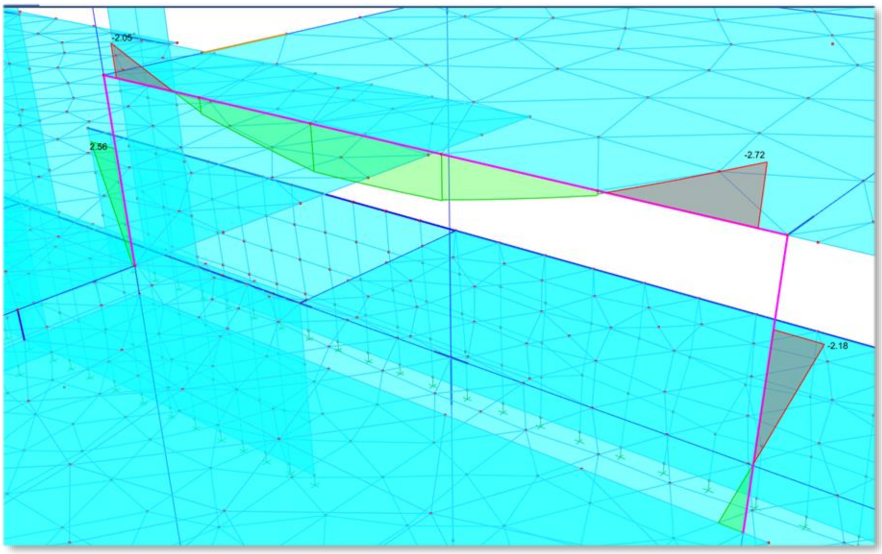


Focus on Selected Members Only

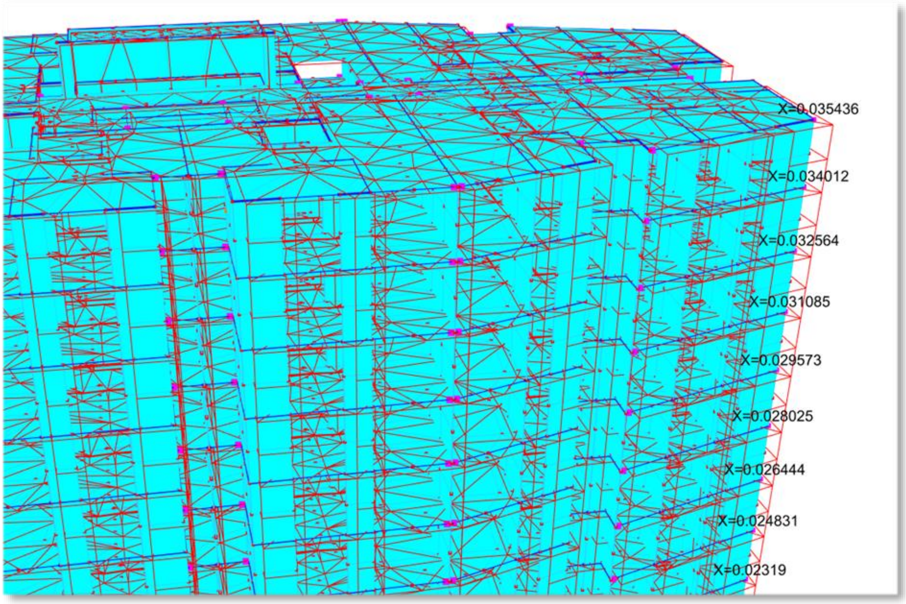
If you want to concentrate on the results of specific nodes or members while displaying all members, you can do so by activating **Focus on Selected Members Only** option on **Analytical Model > Members** ribbon tab.



In the picture below, the moment diagram is displayed only for the selected frames.

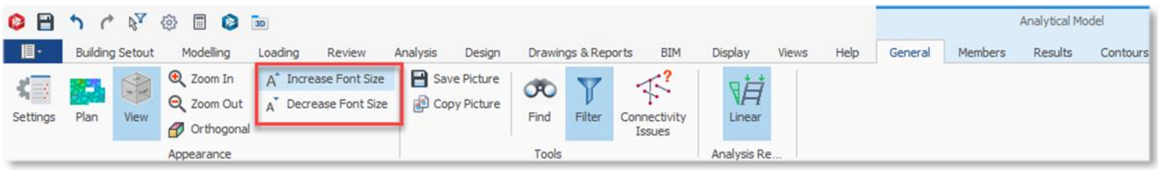


In the example below, corner storey nodes are selected and only their displacement values are shown.



Increase/Decrease Font Size

Another handy enhancement is the easy adjustment of font size during inspection. We have placed two buttons for this purpose under **Analytical Model> General** ribbon tab.



Pushover and Time History Results Visualization

With ProtaStructure 2025, you can now visualize the results for **Nonlinear Pushover** and **Nonlinear Time History** analyses.

Performing Nonlinear Analyses

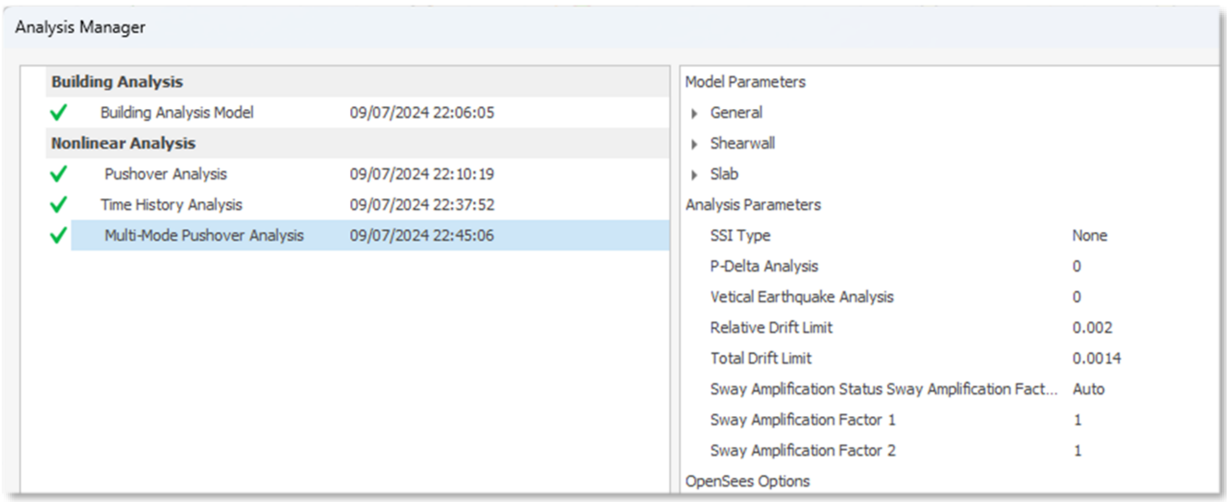
Time History Analysis, **Static Pushover** and **Modal Pushover** analyses are accessible under **Analysis** ribbon tab. Details of these analyses are out of scope of this document. We will focus on visualizing the results.

You can also perform these analyses using the dedicated **Existing Building Assessment** menu. When you use this menu, a detailed member-by-member assessment report is generated after the analyses are completed.

Existing building assessment report is currently created only for **Turkish Seismic Code**.

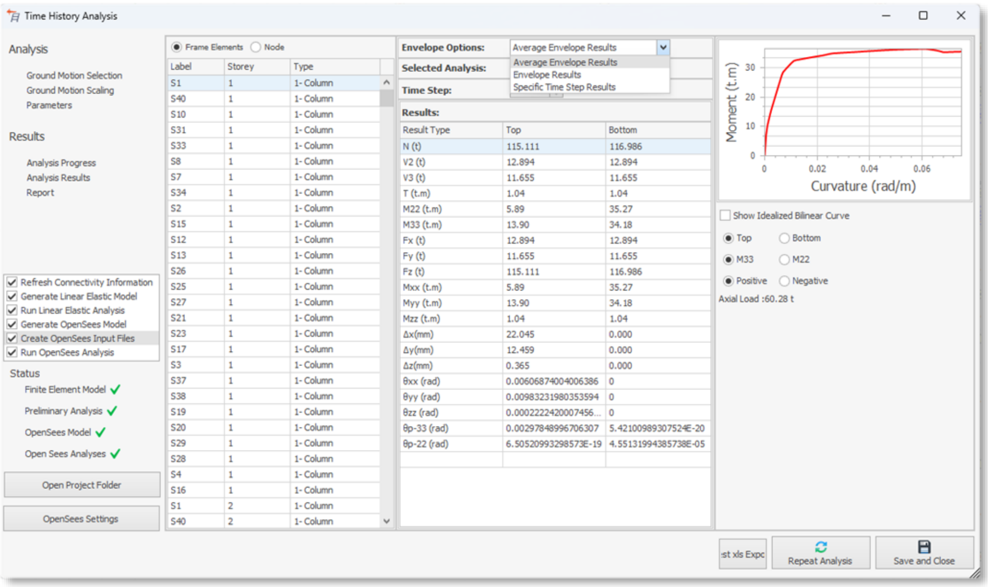
Analysis Manager

Analyses that are performed on the structure are listed under **Analysis Manager**. Here you can review the model options used for each analysis or delete them. When you delete an analysis, the related results will also be removed.



Reviewing Results via the Interface

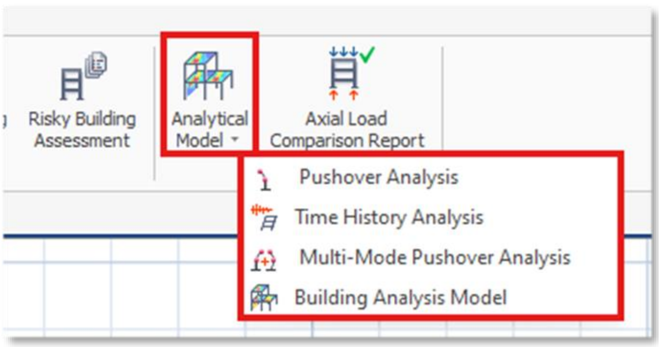
Static Pushover, Modal Pushover and Time History Analysis results can be reviewed on their associated user interfaces by clicking the **Analysis Results** tab on the navigation panel.



Reviewing results for Time History Analysis using the Time History Analysis UI

Reviewing Nonlinear Analysis Results via the Analysis Post Processor

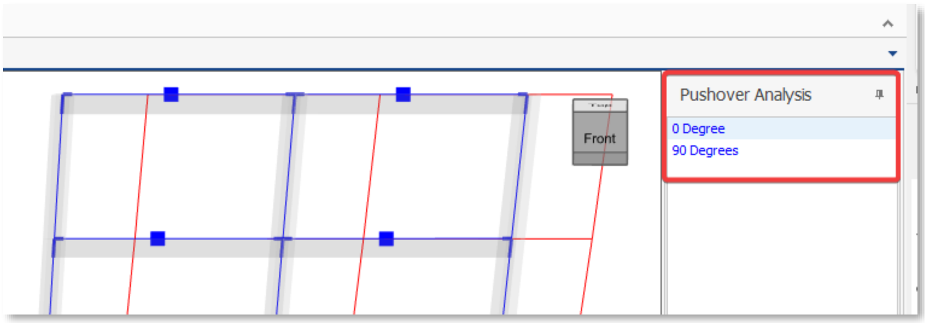
As you perform different analyses, new options will be added under the Analytical Model button on the ribbon toolbar. You select the appropriate analysis type the review its results visually.



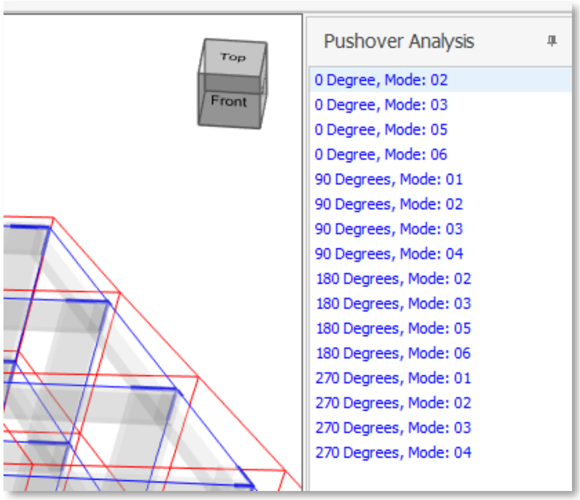
To view the displacements and plastic hinge mechanism for nonlinear performance analyses:

1. Select **Pushover, Multi-Mode Pushover,** or **Time-History Analysis** under the **Analytical Model** button (If they exist). The options will not be visible if a successful nonlinear assessment analysis is not present.
2. On the right-hand side of the screen, the Pushover Analysis panel will be filled with the Pushover analysis cases done for the building. PO cases will usually belong to different analyses done in different directions such as **0 Degrees, 180 Degrees**, **90 degrees** and **270 Degrees**. Choose the PO case that you want to review.

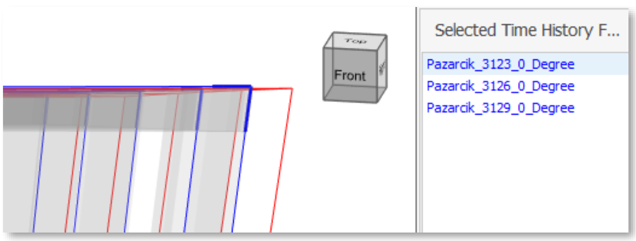
3. If you have performed a Modal Pushover or Time-History Analysis, the listed options will be different. For modal pushover, the mode results in different directions will be listed. For Time-History Analysis, different accelerograms will be shown.



Result options shown for static pushover analysis

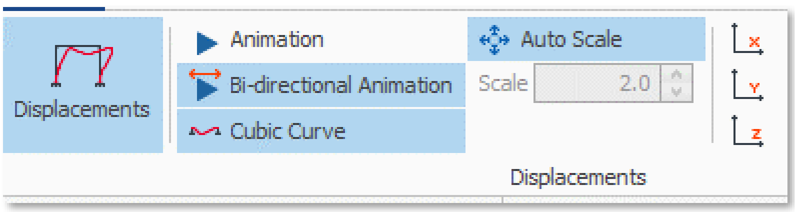


Result options for Modal Pushover Analysis. Individual modes and analysis directions are listed as separate cases

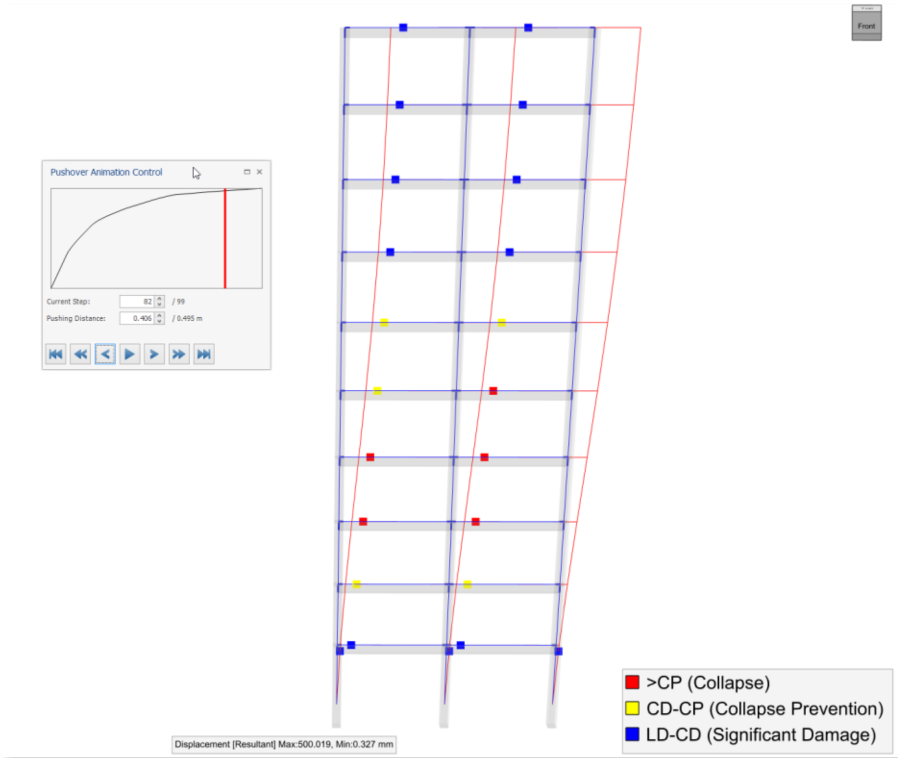


Result options for Time-History Analysis. Each simultaneous analysis shown as a single item.

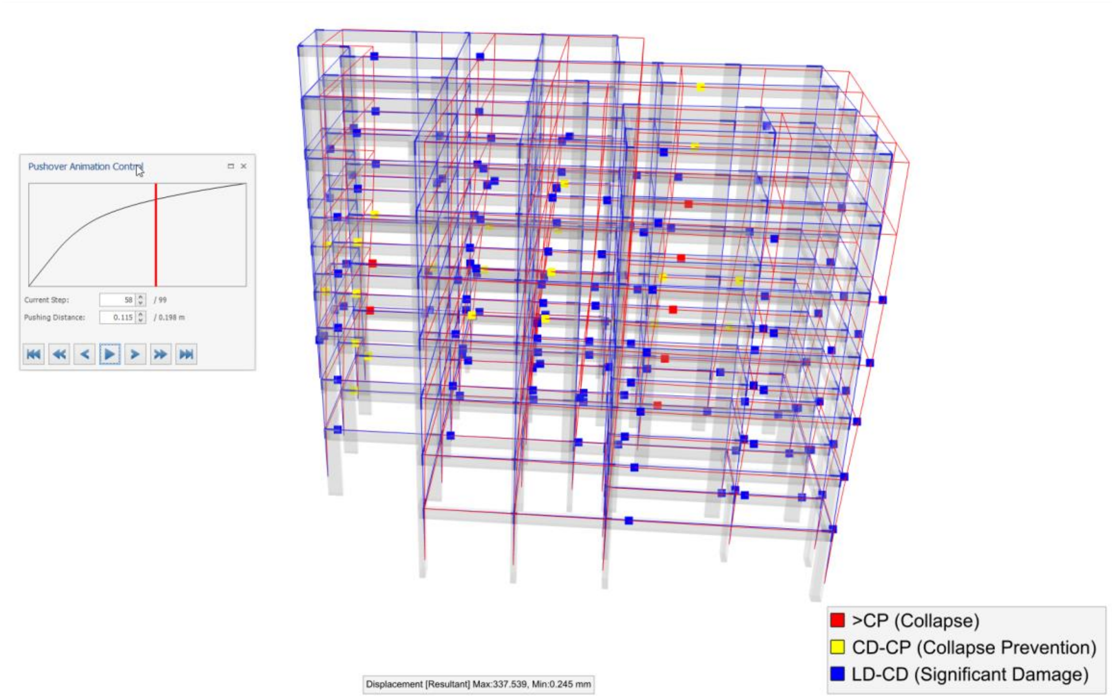
4. Click on **Animation** button to start the animation.



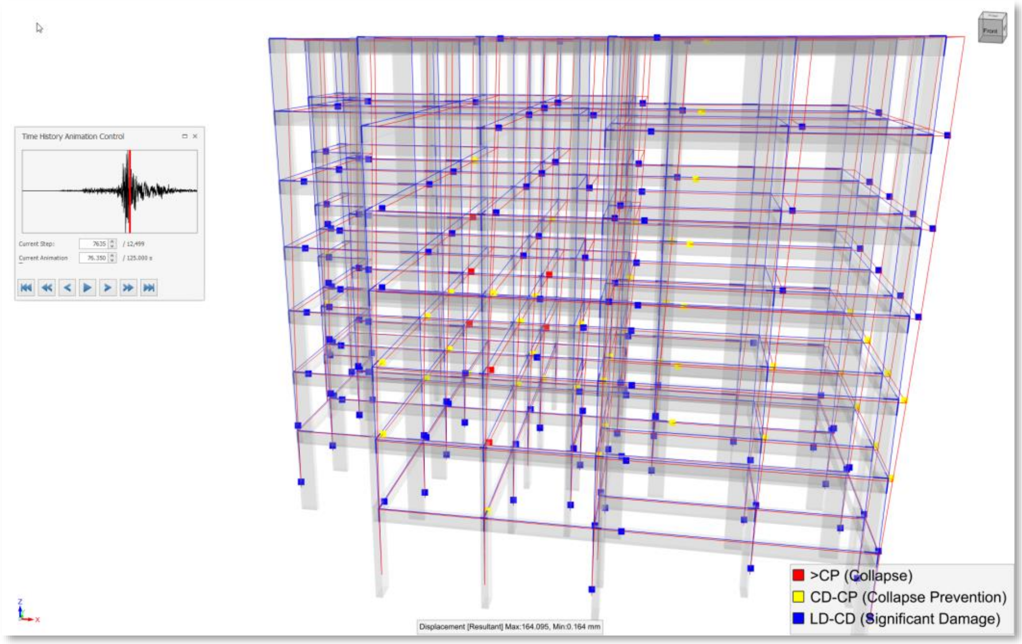
5. The **Pushover Curve** will be displayed on an animation panel showing the step-by-step hinge mechanism in real time on the model. The plastic hinge damage boundaries with their associated colours can be seen on the legend at the bottom-right of the screen. You can control the animation parameters yourself using the **Pause, Previous, Next** buttons on the panel.



Pushover Analysis Results on a 2D concrete frame



Pushover Analysis Results on a 3D concrete building

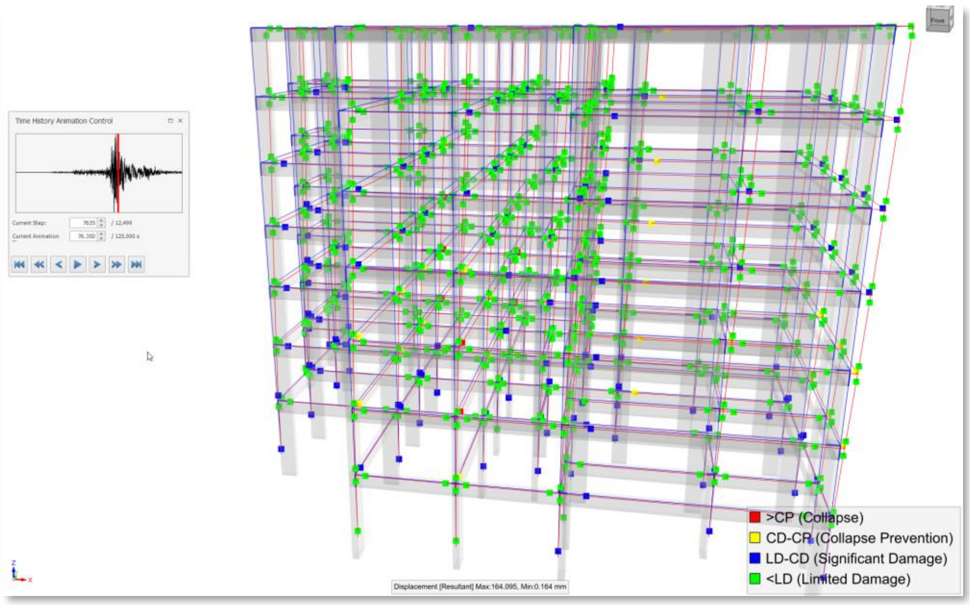


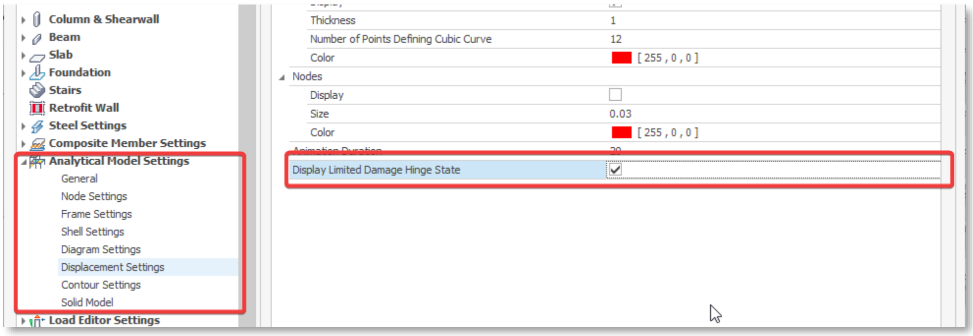
RC building under February 2023 Mw7.7 Turkey/Pazarcık Earthquake.
Accelerograms from Antakya Stations 3123, 3126 and 3129

Show/Hide Members in Immediate Occupancy Region

By default, the plastic hinges not exhibiting plastic behavior in a particular pushover step or time step (i.e. plastic hinges which are still in immediate occupancy region) are not displayed to achieve a simpler view. If you also want to see the unyielded plastic hinges in a particular pushover step or time step:

1. Navigate to **Settings > Analytical Model Settings > Displacement Settings**
2. Check the "**Display Limited Damage Hinge State**" option.
3. Analytical view will start showing plastic hinges which are still in elastic range and the legend will also adapt accordingly.



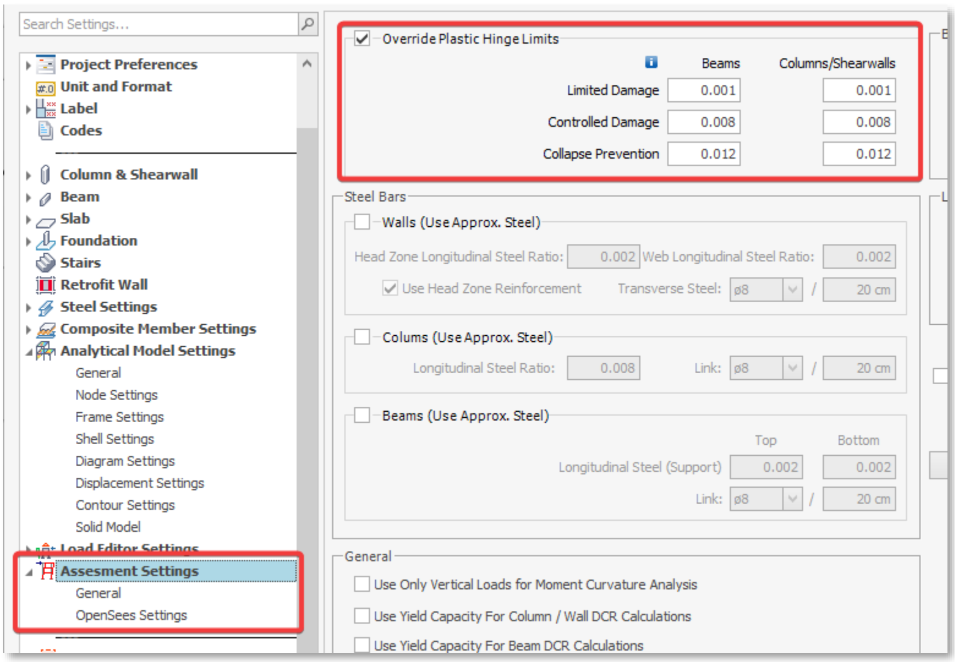


Plastic Hinge Damage Boundaries

By default, ProtaStructure will calculate the plastic hinge damage boundaries according to the selected seismic code requirements. And the limits usually differ for each individual element having different plasticity characteristics.

However, you can override these damage boundaries for the sake of visualization. To customize the boundaries:

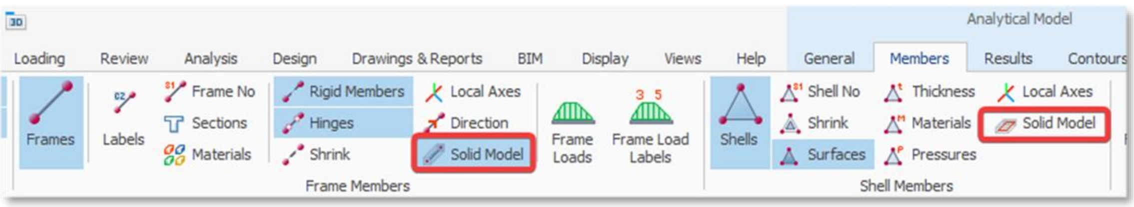
1. Open **Settings > Assessment Settings**
2. Check the "**Override Plastic Hinge Limits**" option.
3. Enter the plastic rotation limits in radians for each performance level.
4. The terminology you see on the settings window may change with the selected code. For example, on this screenshot Limited Damage corresponds to the Immediate Occupancy limit according to the Turkish Seismic Code.

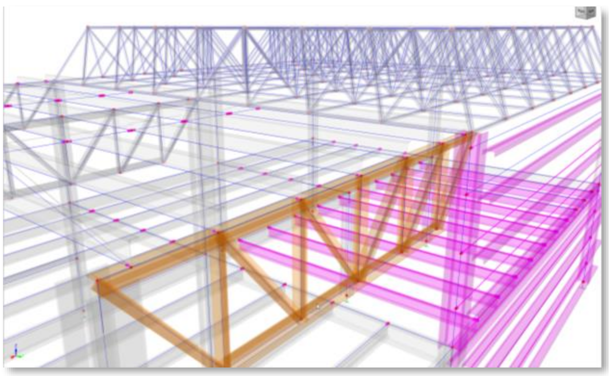


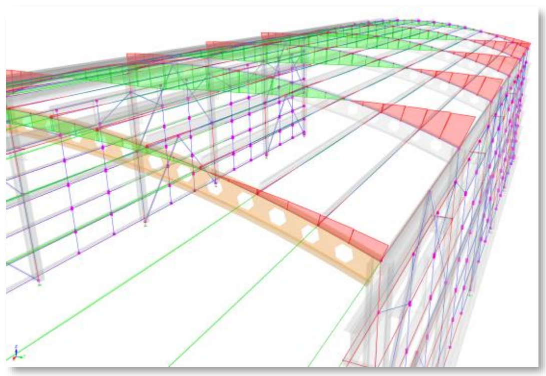
Solid View Around Analytical Lines

ProtaStructure is a structural BIM solution that automatically creates the analytical model from one physical 3D central model. In the latest version, we have introduced **solid member views** to see the physical members as faint extrusions around the analytical lines. This will give a better grasp of how analytical model integrates with the physical model.

To turn on/off the physical member view around analytical lines, check or uncheck in **Analytical Model > Members > Frames > Solid Model** or **Analytical Model > Members > Shells > Solid Model** buttons on the ribbon toolbar.







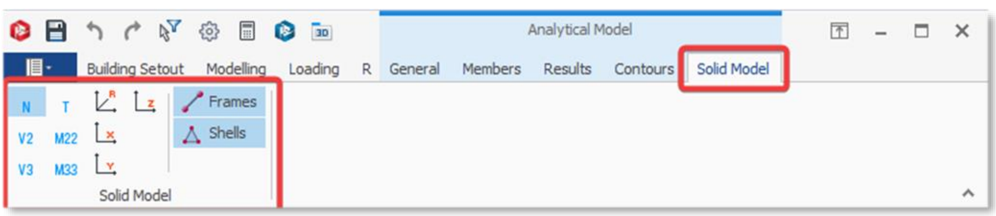
Solid Deflected Shape with FE Contours

We had introduced the solid analytical model in the previous versions. In this view style, instead of analytical lines, nodes or shells, you can see the physical model with the internal forces or displacements overlaid with colour coding.

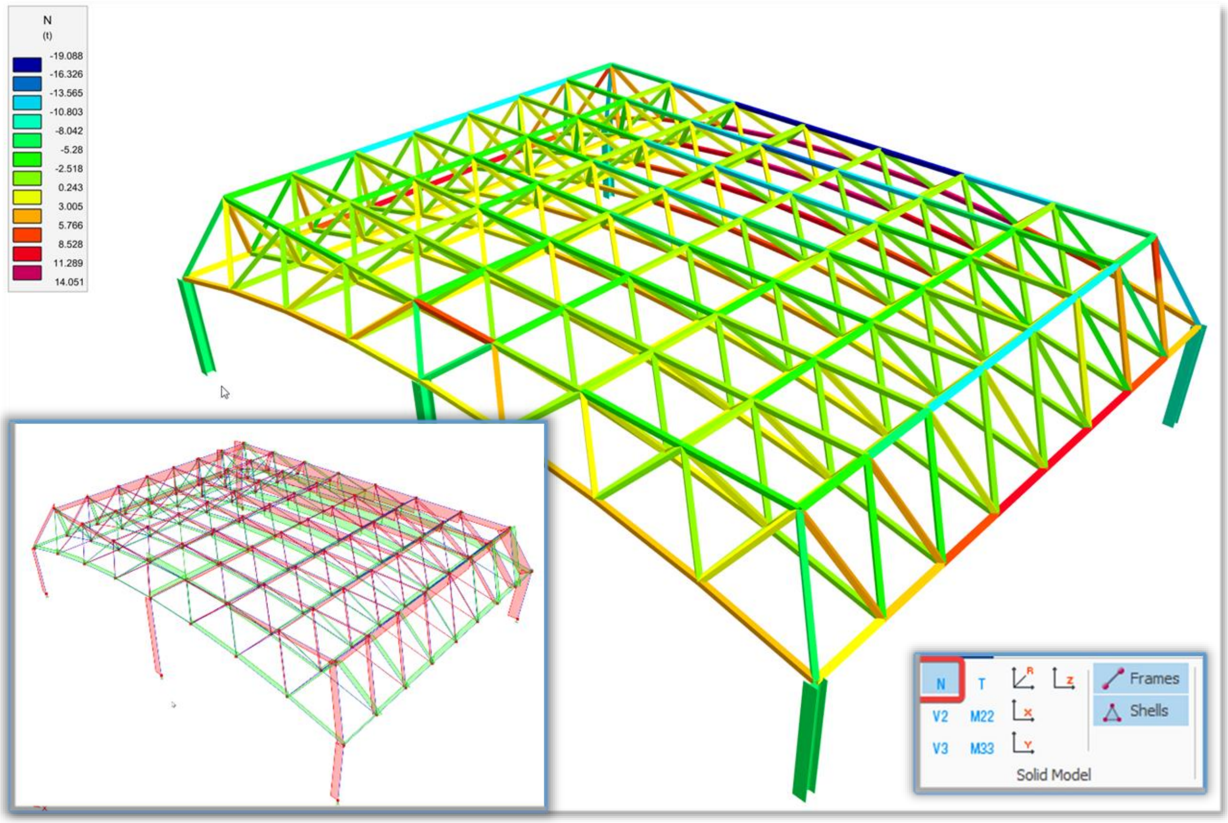
With ProtaStructure 2025, now you can see the **solid deflected shape** with displacement values colour-coded on the members.

To activate the **Solid Analytical View**:

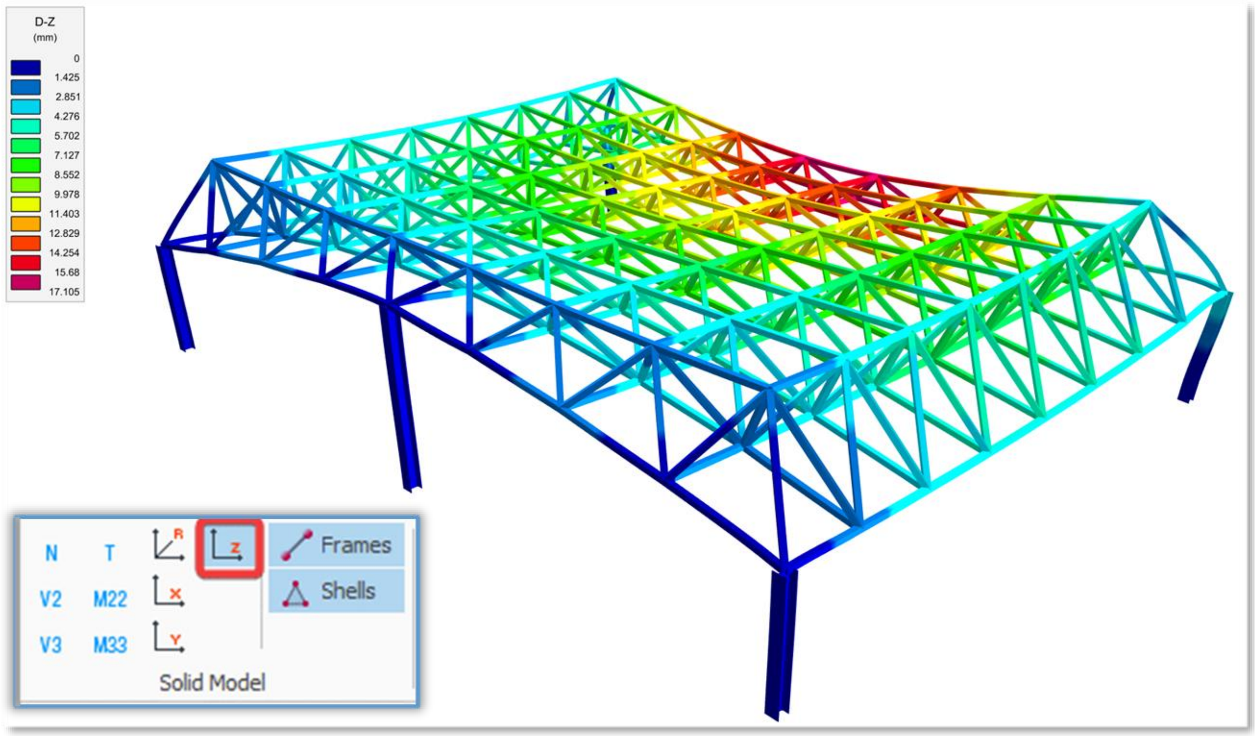
1. Switch to **Solid Model** tab on the **Analytical Model** ribbon bar.



2. Select N, V2, V3, T, M22 or M33 to view the internal forces with a coloured legend.



3. Select R, X, Y or Z to see the displacements. Once you activate one of these, you will see the <u>solid model also deflecting</u>.



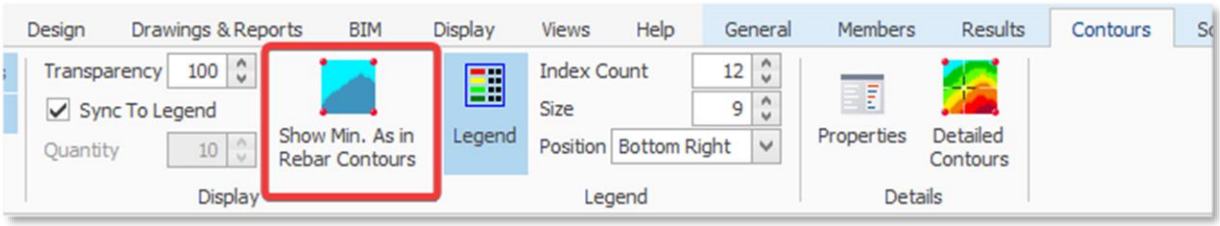
Show Minimum Rebar Area in FE Contours

ProtaStructure can calculate the rebar area contours and plot them on the analytical model. The following rebar contours are plotted:

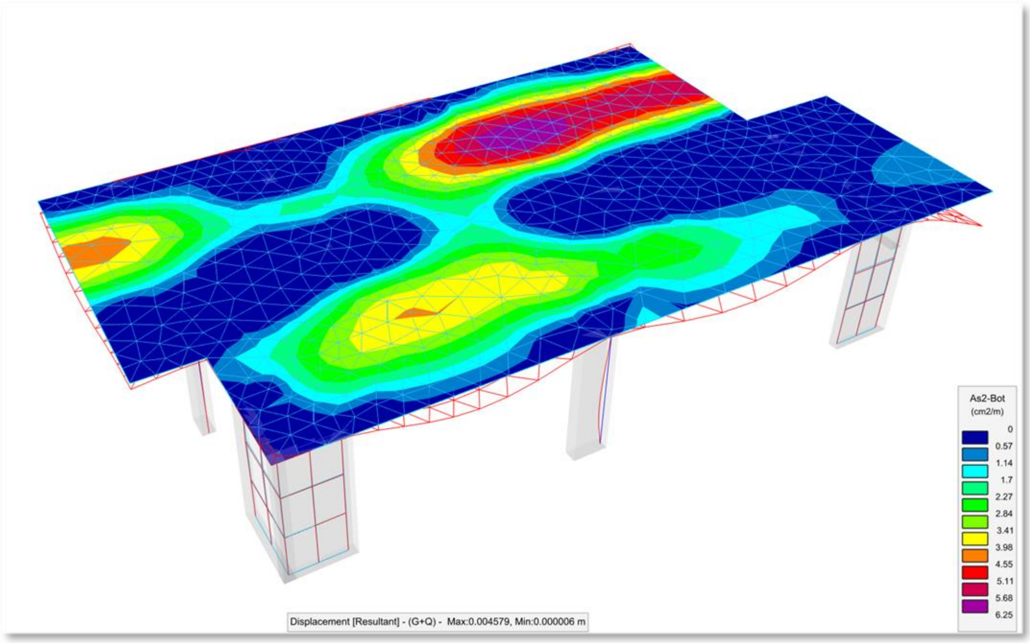
- **As1-Bot** (Bottom Rebar Area in local axis 1 direction)
- **As2-Bot** (Bottom Rebar Area in local axis 2 direction)
- **As1-Top** (Top Rebar Area in local axis 1 direction)
- **As2-Top** (Top Rebar Area in local axis 2 direction)
- **Asd1-Bot** (Bottom Rebar Area in local axis 1 direction including Wood-Armer Effects)
- **Asd2-Bot** (Bottom Rebar Area in local axis 2 direction including Wood-Armer Effects)
- **Asd1-Top** (Top Rebar Area in local axis 1 direction including Wood-Armer Effects)
- **Asd2-Top** (Top Rebar Area in local axis 2 direction including Wood-Armer Effects)

In the previous versions of ProtaStructure, the Required Rebar Area was shown on the FE contours. With ProtaStructure 2025, we have introduced an option to impose the minimum rebar area on the contour values. To use this option:

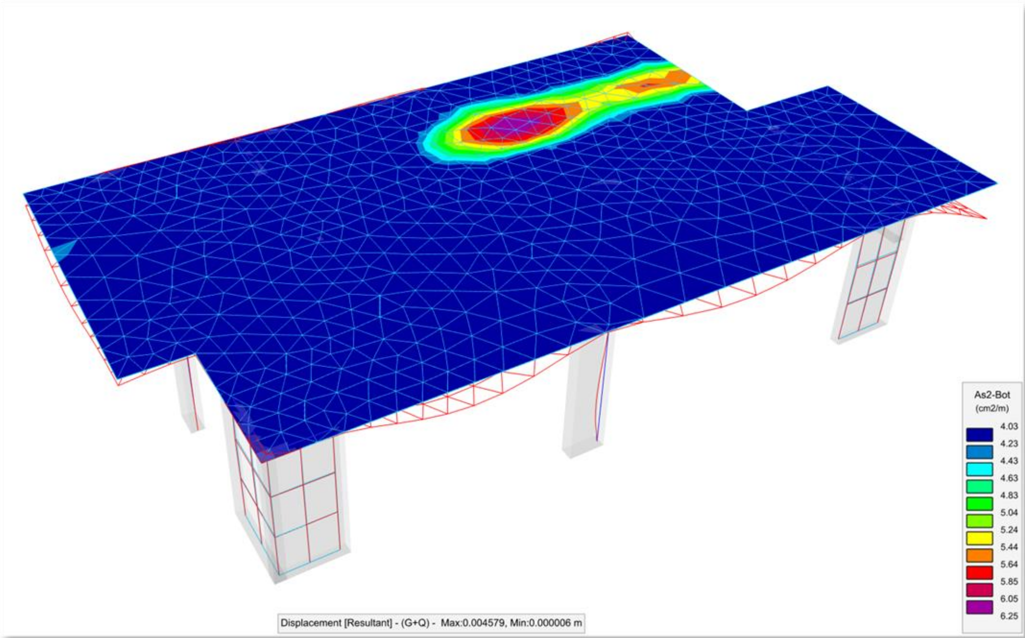
1. Open the analytical model of the structure.
2. Display one of the rebar contours.
3. Check "**Show Min. As in Rebar Contours**" option on the **Contours** ribbon tab.



If this option is checked, the minimum rebar area will be calculated for each node using the selected design code formulations and the associated shell thickness. If the required rebar area is greater than the minimum, then the required rebar area will be plotted. Otherwise, the minimum rebar area will be used for that node. This means that, if the required rebar area is smaller than the minimum for all nodes, you will see all the shells colored with a single color.



The above picture shows **Top Rebar Required Area** in local axis direction 2. Note that the maximum value is 6.25 cm^2/m and minimum value is zero. The minimum value is zero because the contours are showing only the top bars. Positive moment regions are shown with zero value. The **Show Min. As in Rebar Contours** option is unchecked.



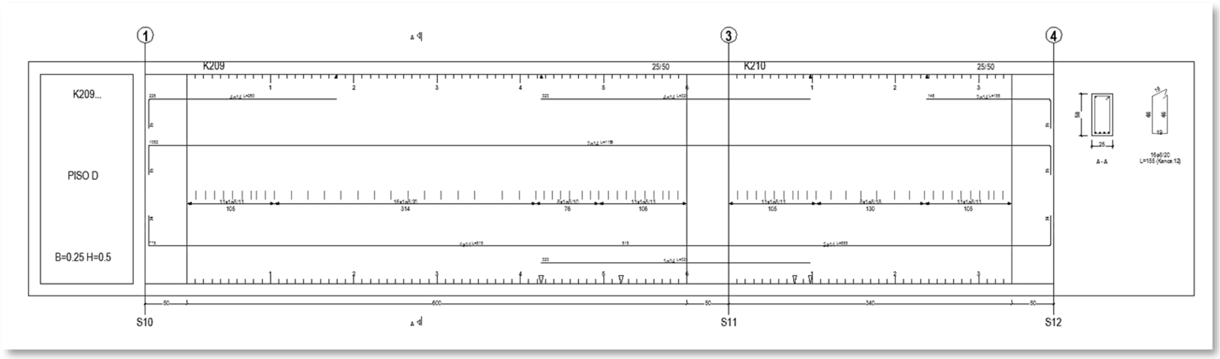
In the picture above, the **Show Min. As in Rebar Contours** option is checked. Note that the minimum value in the legend is 4.03 cm^2/m which is the code minimum value for the slab. The required area below the minimum value is suppressed and shown by a blue color (the color for minimum). The required rebar area values above the minimum value are colored according to the legend.

ProtaDetails

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.

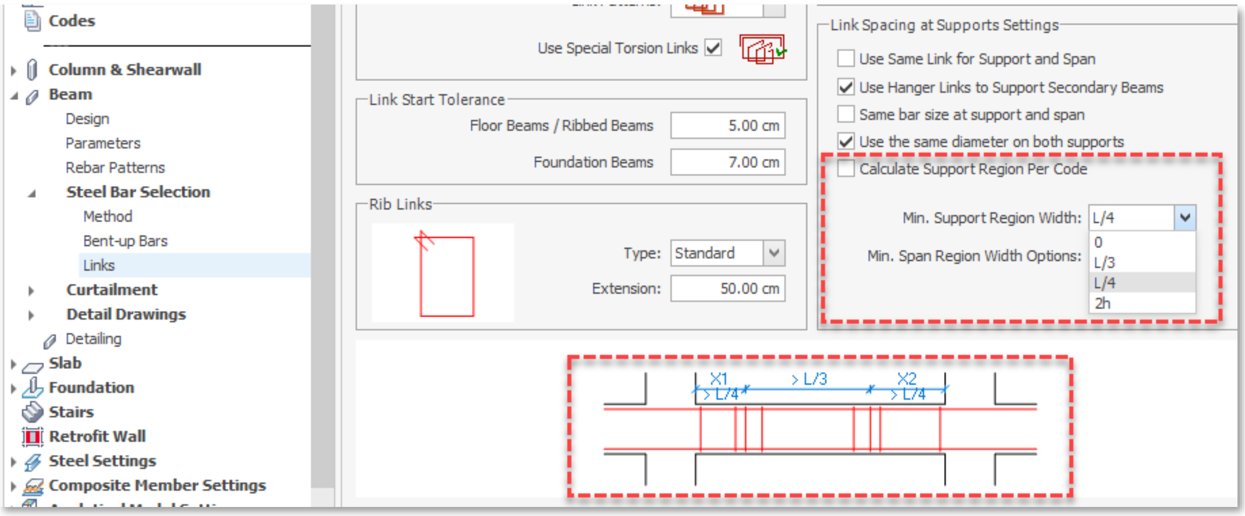
This style is preferred in some country practices. It is optional and can be turned on/off any time using **Settings > Beam > Detail Drawings > Drawing Style**. You can also use two different styles in the same drawing.

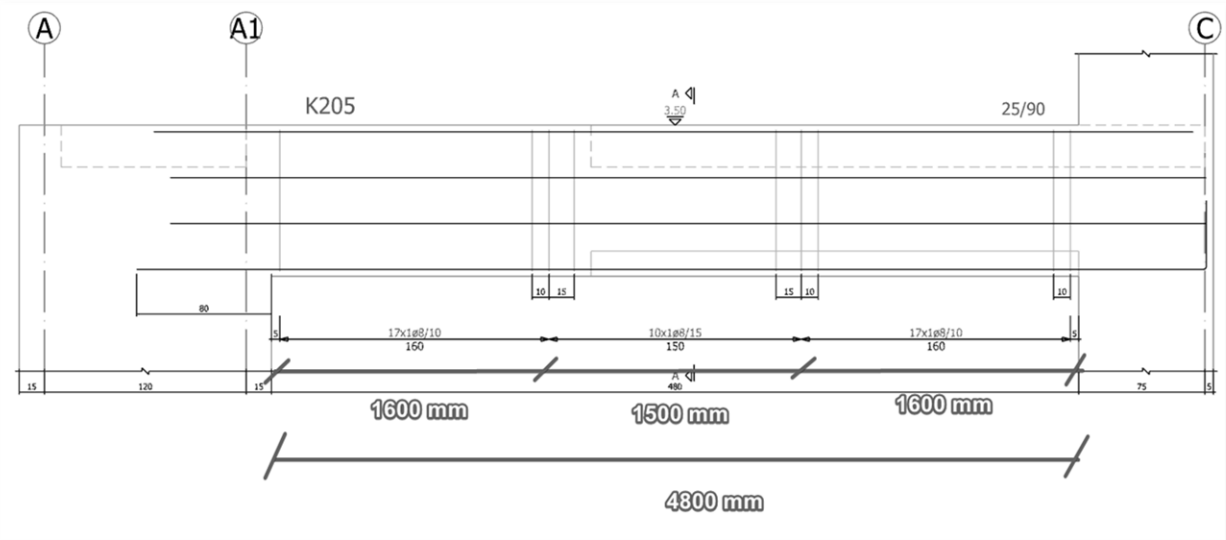


Adjustable Beam Critical Zones

With the new version, you can now specify the “**Minimum Support Region Width**” in terms of fraction of the span length (**0, L/3, L/4**) or **2H**.

1. Open the **Settings > Beam > Steel Bar Selection > Links** settings page.
2. Uncheck “**Calculate Support Region Per Code**” option.
3. Specify a support region width from the “**Min. Support Region Width**” list
    a. **0** means no confinement zone will be used even if the support links are specified.
    b. The confinement zone (critical region) length will be calculated according to the one of selected L/**3, L/4** or **2H**. Here, **L** is the beam clear length and **H** is the gross section depth.
4. Please note that, **Min. Span Region Width Options** also dictates the curtailment. You can choose the minimum span zone length to be considered.



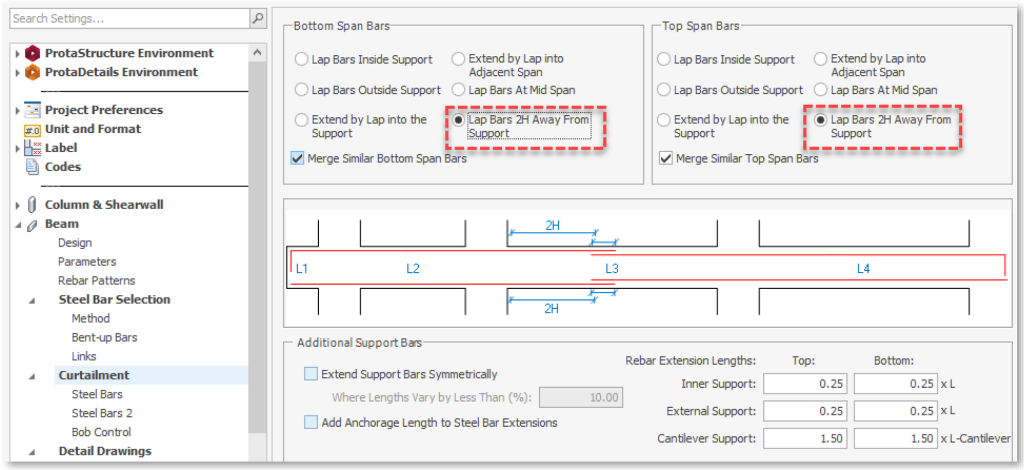


The **Minimum Support Zone Length** has priority over **Minimum Span Region Length**. ProtaStructure will make sure that support zone length preference is reflected to detailing first.

In the detail drawing above, the beam clear length is **4800 mm**. Minimum support length is specified as **L/3** so the critical zone detailing is done to **1600 mm**. Although the minimum span length was set to **L/2**, ProtaStructure did not have any choice but the use the remaining **1500 mm** as the support length. The residual **100 mm** accounts for the offset of the first and last links from the support face.

Beam Rebar Splices 2H Away From Critical Zones

Another new option was introduced for lapping the top and bottom bar splices **2H** away from the beam supports. This curtailment option is necessary and useful especially for the special and intermediate moment resisting frames. The setting can be accessed via **Settings > Beam > Curtailment > Steel Bars**.

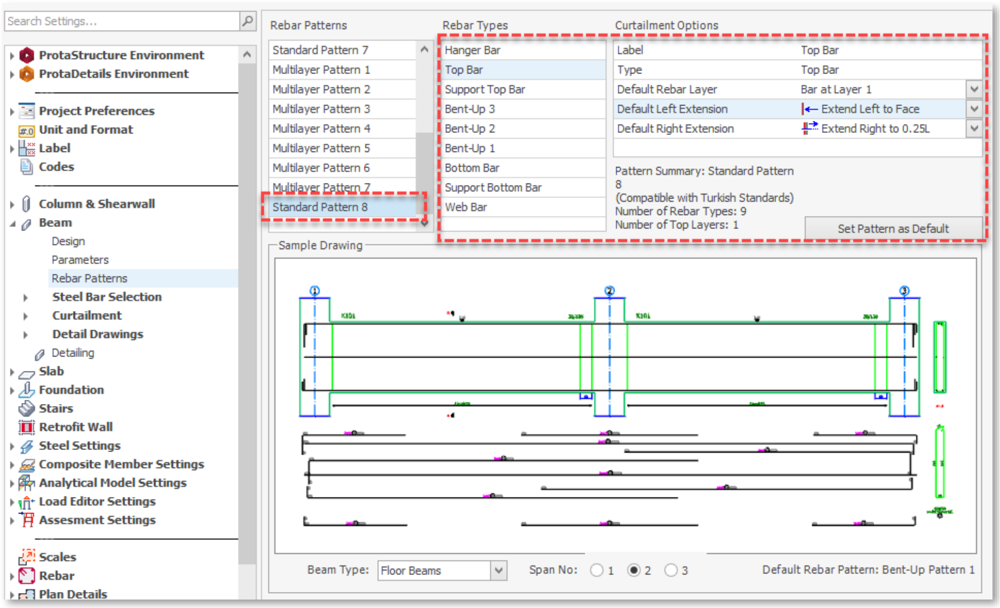


New Beam Rebar Pattern: Standard Pattern 8

A new beam rebar pattern, Standard Pattern 8, was introduced with the new version. This new pattern is similar to Bent-Up Pattern, but it does not use bent-up bars.

This is essentially a non-caging two-layer pattern where top bars are extended by 0.25L and bottom bars are extended by lap distance to adjacent spans. In this pattern, hanger bars are included, and web bars are automatically distributed to the sides if necessary.

As it is the case for all patterns, you can customize this pattern as well, to suit your needs.



Rebar Schedule for Pad Footings and Pile Caps

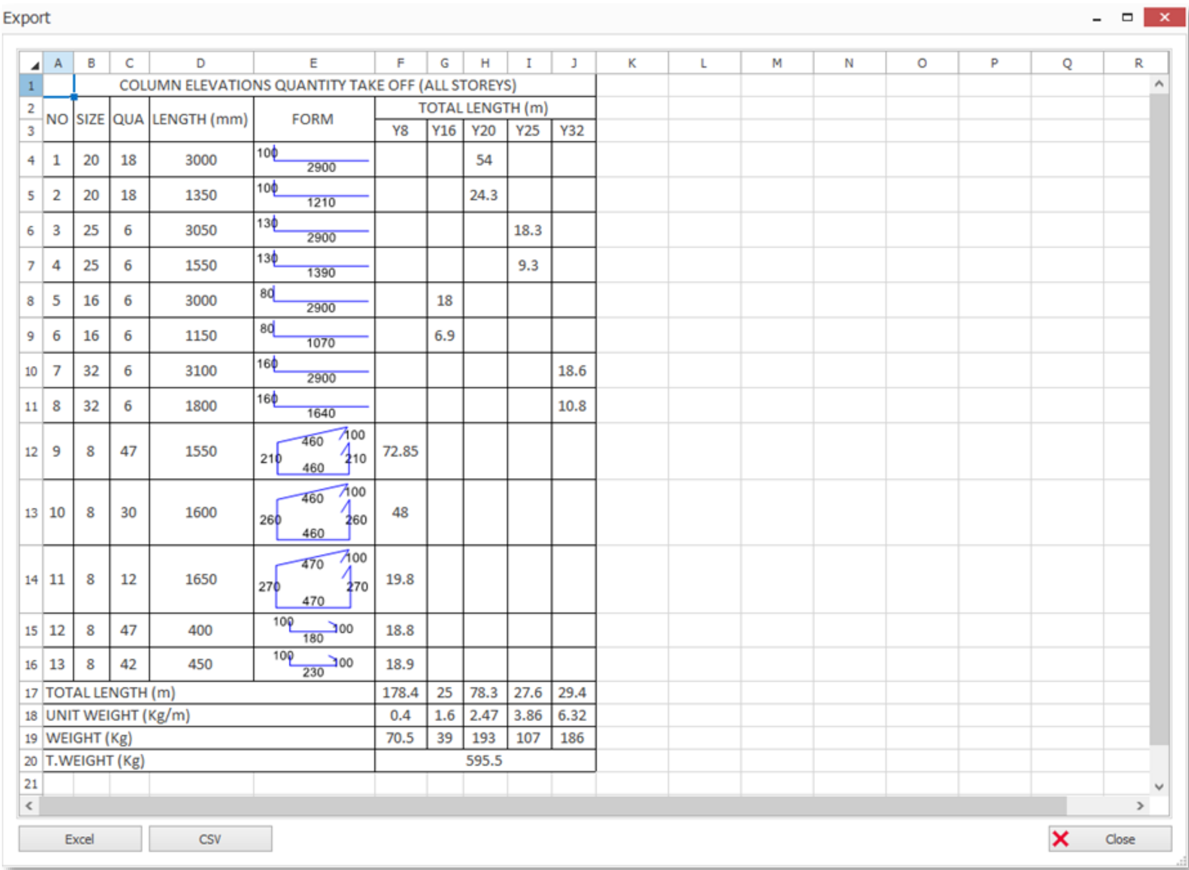
It is now possible to generate a detailed reinforcement schedule for pile caps and pad footings similar to the one that is already available for beams.

PILE CAP REBAR SCHEDULE											
Member						Top Reinforcements		Bottom Reinforcements		Distribuion Rebars	
	Lx (mm)	Ly (mm)	H (mm)	Quantity	Columns	Top Rebar (X-Dir)	Top Rebar (Y- Dir)	Rebar X	Rebar Y	Side Rebar (X-Dir)	Side Rebar (Y-Dir)
PC-1C1	1100.0	1100.0	500.0	1	1C1	1x6T12-200	1x6T12-200	1x5T20-250	1x5T20-250	1x3T12-225	1x3T12-225
PC-1C3	1600.0	1500.0	500.0	1	1C3	1x8T12-200	1x8T12-200	1x6T20-250	1x7T20-250	1x3T12-225	1x3T12-225
PC-1C6	1100.0	1100.0	500.0	1	1C6	1x6T12-200	1x6T12-200	1x5T20-250	1x5T20-250	1x3T12-225	1x3T12-225
PC-1C4	1500.0	1600.0	500.0	1	1C4	1x8T12-200	1x8T12-200	1x7T20-250	1x6T20-250	1x3T12-225	1x3T12-225
PC-1C8	1600.0	2100.0	500.0	1	1C8	1x11T12-200	1x8T12-200	1x9T20-250	1x7T20-250	1x3T12-225	1x3T12-225
PC-1C2	1350.0	1350.0	500.0	1	1C2	1x7T12-200	1x7T12-200	1x6T20-250	1x6T20-250	1x3T12-225	1x3T12-225

Export Quantity Tables to Excel

We have enhanced the **Rebar Quantity Take-off Export** functionality further in ProtaStructure 2025.

Previously, we were only supporting Excel export via CSV files. With the new version you can enjoy a native export which also includes the shapes of the rebars drawn inside the cells in Excel.



COLUMN ELEVATIONS QUANTITY TAKE OFF (ALL STOREYS)									
NO	SIZE	QUA	LENGTH (mm)	FORM	TOTAL LENGTH (m)				
					Y8	Y16	Y20	Y25	Y32
1	20	18	3000	100 2900			54		
2	20	18	1350	100 1210			24.3		
3	25	6	3050	130 2900				18.3	
4	25	6	1550	130 1390				9.3	
5	16	6	3000	80 2900		18			
6	16	6	1150	80 1070		6.9			
7	32	6	3100	160 2900					18.6
8	32	6	1800	160 1640					10.8
9	8	47	1550	460 100 210 210 460	72.85				
10	8	30	1600	460 100 260 260 460	48				
11	8	12	1650	470 100 270 270 470	19.8				
12	8	47	400	100 100 180	18.8				
13	8	42	450	100 100 230	18.9				
TOTAL LENGTH (m)					178.4	25	78.3	27.6	29.4
UNIT WEIGHT (Kg/m)					0.4	1.6	2.47	3.86	6.32
WEIGHT (Kg)					70.5	39	193	107	186
T.WEIGHT (Kg)					595.5				

ProtaSteel

Export Steel Connections to IdeaStatica

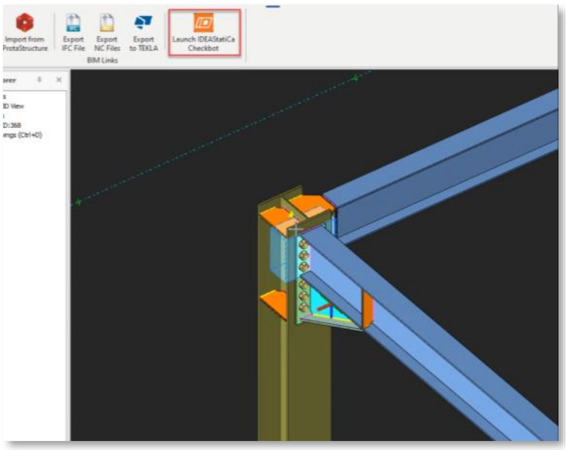
ProtaSteel provides detailed design reports for a large variety of steel connections using analytical methods. Analyzing complicated or user-defined steel connections with Component Based Finite Element Analysis may be required in your project.

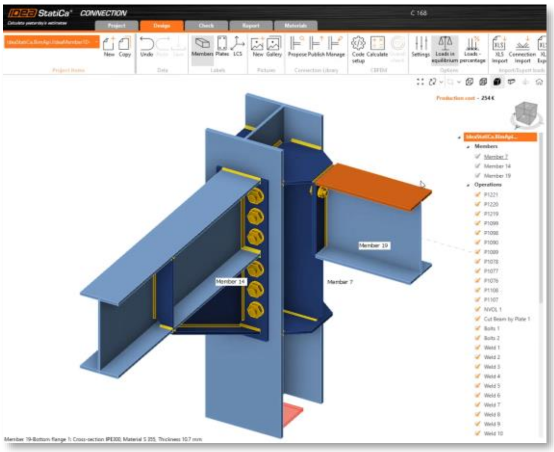
In Prota Software, we believe in open collaboration. In each new version, we strive to create links between our products and other software solutions and file formats in the industry.

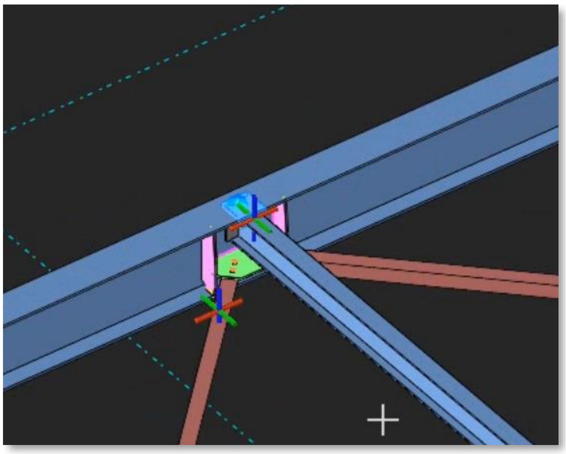
Our last addition is the **IdeaStatica CAD Links**. You can now export your automatic or user-defined connections to IdeaStatica and perform CBFEM analysis if you need to.

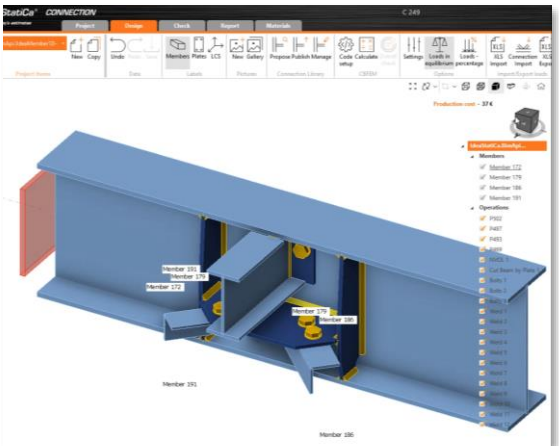
All you must do is to initiate **IdeaStatica Check Bot** in ProtaSteel and select the primary members and connection components by following the instructions on the wizard window.

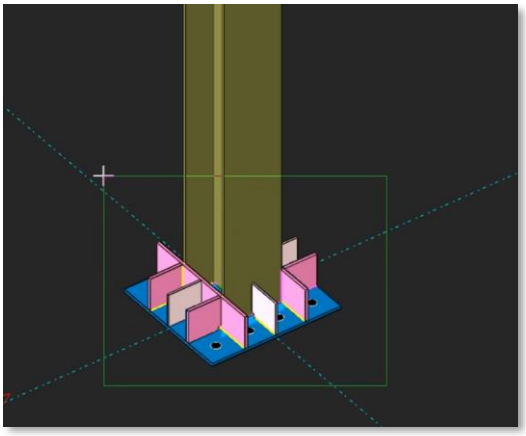
ProtaSteel is a sophisticated fabrication software which provides design and detailing to its users, covering a large scope. The IdeaStatica CAD Link API is designed for CAD information communication rather than internal forces and design information. We are working on a few workarounds to transfer of internal forces.

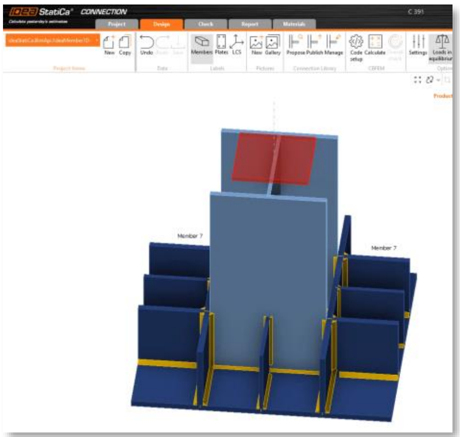










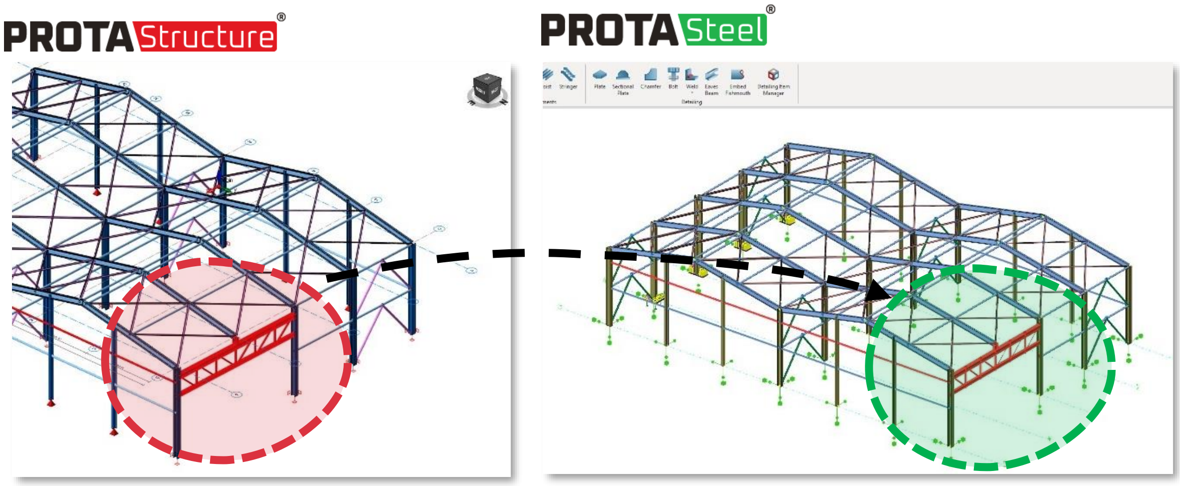


Partial Model Update from ProtaStructure

ProtaStructure and ProtaSteel enjoy a seamless data communication and update system. You can update the model in ProtaStructure and reflect the changes to ProtaSteel seamlessly even if you finished connecting and editing your model.

However, as the size of the model increases, the update operation can start taking time. That's because all the members and connections need to be checked. And connections need to be adjusted to their new location or size if an update was necessary.

With the latest ProtaSteel, you can now select a subset of members in ProtaStructure and update only selected members in ProtaSteel. That will accelerate your design process when you need small partial updates.



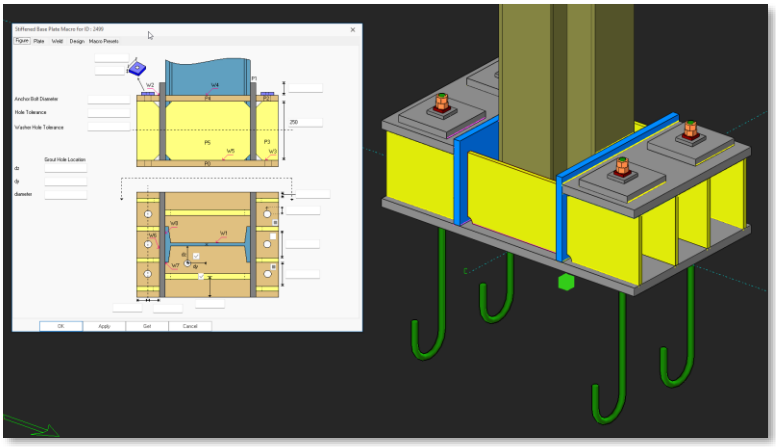
Stiffened Base Plate (Exposed Base Plate)

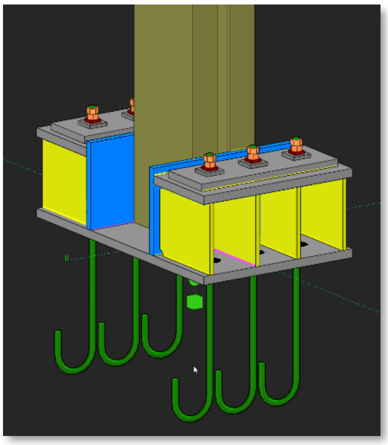
Column Base Connections are one of the most critical components of Special Steel Moment Frames (SMFs).

This type of connection is common in seismic areas, especially for industrial facility buildings. This method aims at concentrating all plasticity in the extended region of the anchor bolts.

Especially in the intense load scenarios, conventional base plate thickness may be unpractically high. To overcome this design issue, exposed base plates can be used to reduce the base plate thickness.

This connection can be used for any type of profiles including face-to-face or back-to-back twin channels, cruciform columns, and regular I profile. They are particularly useful for cruciform columns which tend to be used to counter high bending and axial stresses.

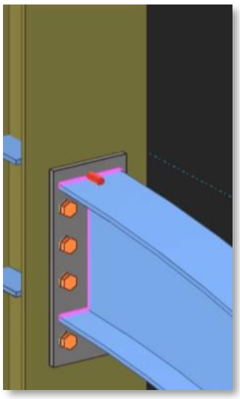


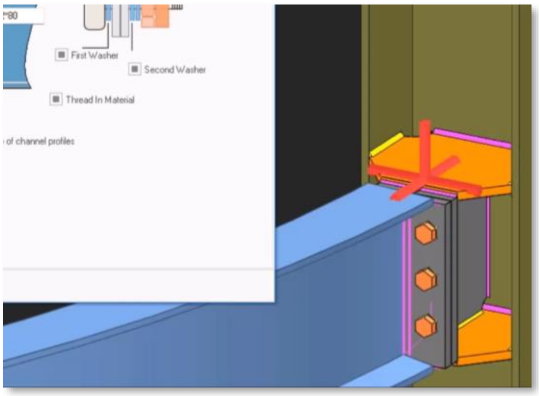


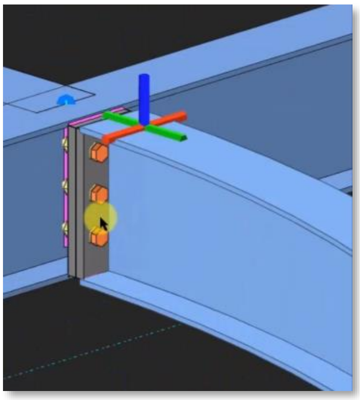
End Plate Connections for Curved Beams

ProtaSteel now supports **Beam to Beam** and **Beam to Column End Plate** and **Stiffened End Plate Connections** for **Curved Frames**.

The frame approach angles must be close to perpendicular for these connection types.





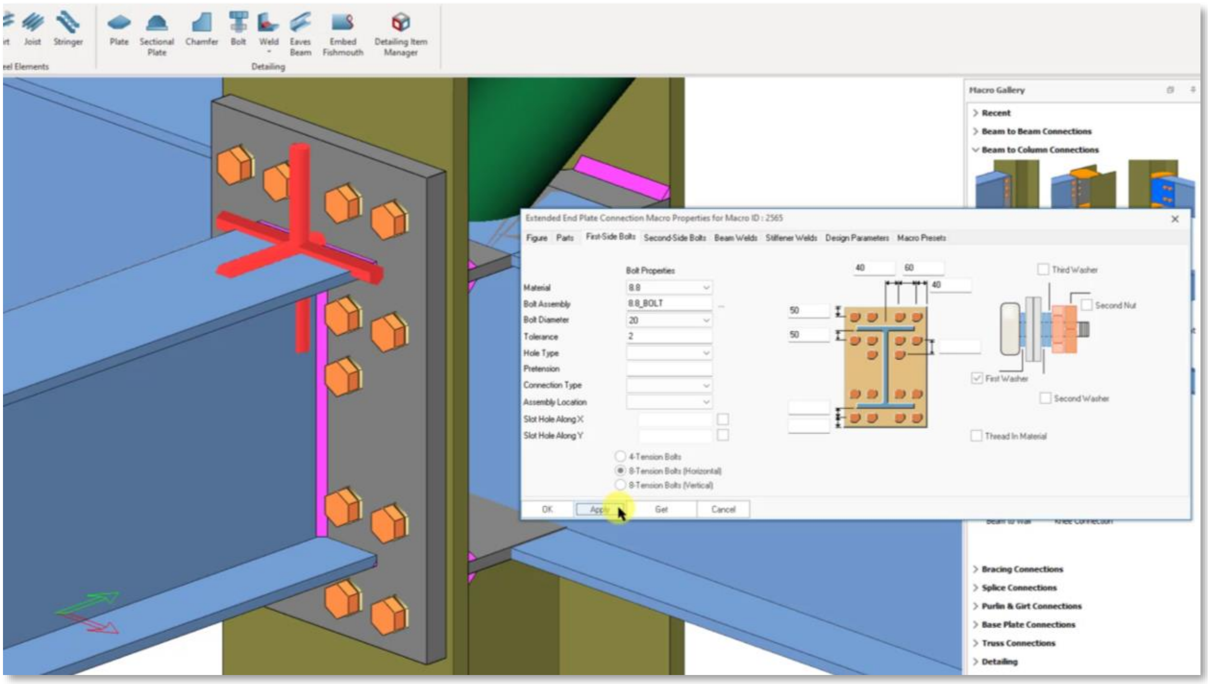


Extended End Plate with Additional Tension Bolts

Extended end plates can be used as prequalified seismic connections. They are designed to create a rigid and moment-resisting connection between a steel beam and a column. Extended end-plate connections provide excellent performance in resisting seismic loads, especially in high-risk areas.

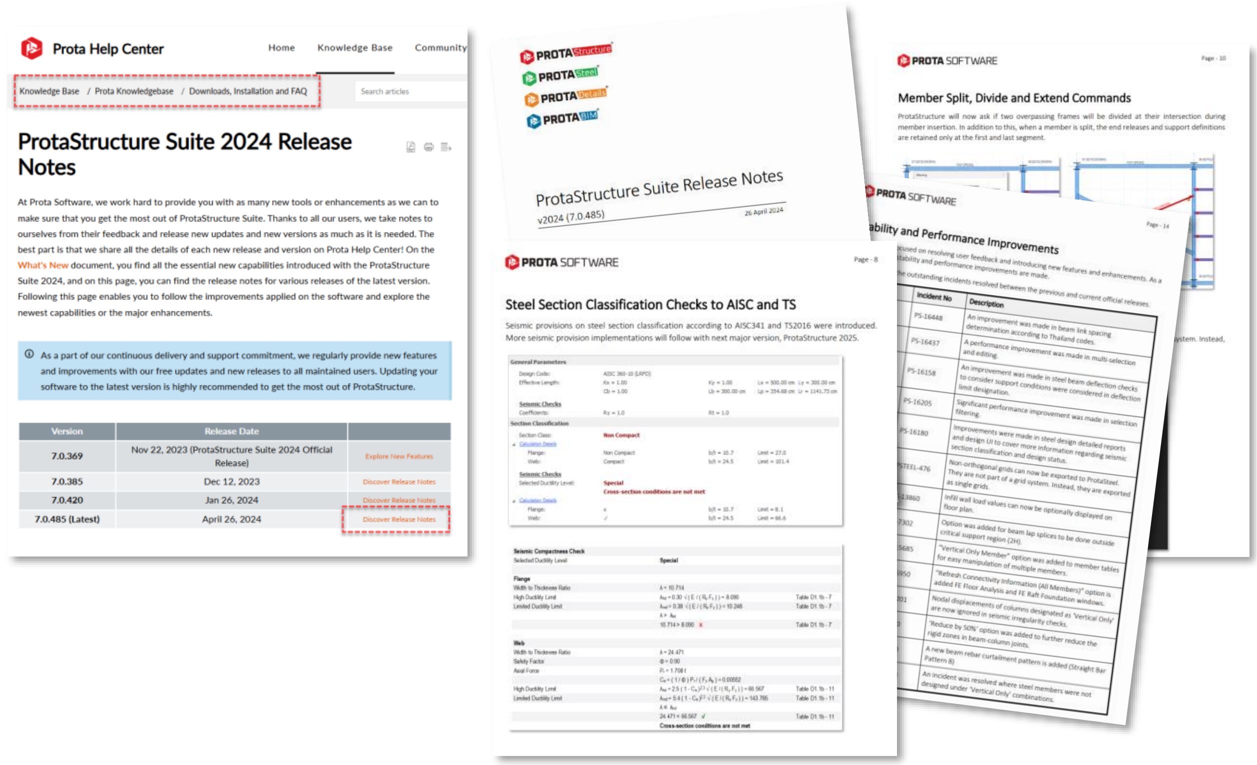
Bolts are used to attach the end plate to the beam and column flanges. The end plate extends beyond the beam flanges allowing the use of bolts Other than the interior ones.

In the new ProtaSteel, you can now increase the number of bolts and introduce additional ones in the plate extension.



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 2024 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 2025 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

What’s Next?

While we strive to include as many improvements as possible in each major version, some items may be delayed beyond the initial release. Rest assured; we're committed to delivering them in subsequent updates to our user with an active maintenance.

Thank You

Thank you for choosing the ProtaStructure Suite product family.

At Prota, our continual aim is 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