

ProtaStructure®

ProtaSteel®

ProtaDetails®



ProtaStructure Suite 2022 – New Features

Version: 4.0

5 January 2022

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

asiasupport@protasoftware.com

globalsupport@protasoftware.com

Limitation of Responsibilities

While Prota endeavors to ensure that 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 of results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- make sure that the software is appropriately used following 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	6
Extended Localization and Code Coverage	7
New Report and UI Languages	7
Imperial Unit System	7
Seamless Switch Between Unit Systems	8
Level Mark, Analysis Length, and Displacement Units	8
Extended Code Coverage	9
Automated Calculation of Snow Loads	9
Stage Construction Analysis with Creep and Shrinkage	10
New Seismic Checks and Calculations	11
New Tools in BIM Integration	16
Bi-directional Communication with SAF Files	16
Improved Revit Integration	16
Streamlined UI and Dialogs	17
Improved Analytical Links with SAP2000	17
Analysis Results in CSV Format	17
Flexible Loading System	18
Assign Loads in Any Direction and Visualize in 3D	18
Manual Loading on Truss Joints	18
Static and Dynamic Soil Thrust on Basement Walls	19
Enhanced Load Visualization	20
User-Defined Vertical Load Cases	20
Roof Live Load, Snow and Rain Load Cases	22
Copy and Paste Loads	22
Better Loading Experience for Corewalls	23
Detailed Load Profile for Infill Walls with Openings	23
Rib Slab Loads –Automatically Decomposed in Building Analysis	24
Automatic Height Adjustment for Infill Wall Loads	24
New Modeling Tools	25
Steel Domes	25
Curved Frame Members	25
Improvements in Brace Insertions	26

New Visual Interrogation Options	26
New Slab Insertion Method: Pick Closed Slab Edge	27
Splitting and Joining Beam and Frame Members	27
Extended Steel Profile Library	27
Enhancements in Performance and Memory Usage	27
Transparency and Quality in Design	28
Design of Steel Domes	28
New Beam Rebar Patterns	28
Modified Beam Rebar Editor	30
New Slab Design User Interface	30
Check Design for User-Defined Column Links	31
New Settings for Slab Design	32
Enhanced Design Diagrams	32
Mat Foundation Punching: Soil Pressure Under Each Column	33
New UI for Rebar Diameter Availability	33
Design Reports with Step-by-Step Calculations	35
New Slab Design Report	36
Comprehensive FE Mat Foundation Design Report	36
Minimum Slab Thickness Checks	37
Member Loads Report	38
Member Properties Report	38
Enhanced Analysis Results Report	39
New Analysis Framework	40
Integrated 64-bit FE Slab and Foundation Module	40
Single Data Representation and Architecture	40
Improved Visualization	40
New FE Floor Analysis Manager	41
Slab Strip Diagrams	42
Manage Multiple Analysis At the Same Time	44
Soil-Structure Interaction in a Single Run	44
Analytical Model Export Options	45
A Single Integrated Post Processor for All Analysis Results	46
FE Floor Analysis vs. Building Analysis Post-Processing	46
Slab Strip Diagrams	48
Other Improvements in Analysis Post Processor	48

Enhancements in RC Detailing	50
Rebar Lap Splice and Anchorage Length Report	50
Improved Quantity Take-Off for Corewalls	50
Rebar Quantity Tables to Excel	50
Unlimited Rebar Annotation Characters	50
Imperial Rebar Labels For Metric Design	50
The New ProtaSteel	51
New Technology and User Experience	51
User-Defined Connections Wizard	52
Intelligent Input Fields on Macro Dialogs	53
New Connection Macros	53
Improvements in Object Grouping, Filtering, and Selection	55
Connection Design Checks and Internal Force Reporting	56
New Program-wide Settings Systematics	56
Other Enhancements in ProtaSteel	57
Overall Improvements on Stability and Performance	58
Thank You	59

Introduction

As Prota, we have been developing leading Structural BIM software for more than 30 years.

ProtaStructure 2021, which we delivered in July 2020, was an important milestone in achieving our mission and commitment. As you know, ProtaStructure 2021 came with 64-bit architecture, multi-threaded operations, and the latest technology platforms.

Despite the challenges of the global pandemic around the world, we were able to continue our development and customer services without significant interruptions during 2020 and 2021. ProtaStructure 2022 is the result of our team's devotion and hard work.

To provide you with a more integrated experience, we have migrated our FE Floor and Foundation Module to 64-bit and integrated it with the main program. We have achieved performance gains of up to 10 folds in modeling, general usage, and visualization.

We have introduced new seismic checks such as Seismic Joint Spacing Calculation, Forces on Non-structural members, Target Displacement Calculation in Pushover Analysis, retrofitting, and member assessment with CFRPs various international design codes.

As a result of user feedback, we have developed new reports and enhanced the existing ones, such as integrated slab and mat foundation design reports. Besides, a more refined user experience awaits you in slab and mat foundation design. The loading system is redesigned from scratch with a more flexible, scalable, and visual approach. Snow load calculation is now automatically done to EC and TS codes. Imperial units, steel domes, arch, and curve-shaped frame members can be counted among the new features.

Additionally, you can create custom beam rebar patterns and create templates that include rebars with more than two layers. Staged Construction that considers the long-term effects of creep and shrinkage is also introduced with ProtaStructure 2022.

Significant enhancements are made, and new features are introduced in data communication with BIM platforms. Bi-directional SAF Data Exchange is the latest addition to our BIM collaboration arsenal.

ProtaSteel 2022 is completely migrated to a 64-bit platform. The user interface is redesigned from scratch, promoting practicality without compromising habits and usability. In-product instructions and wizards are provided for productivity.

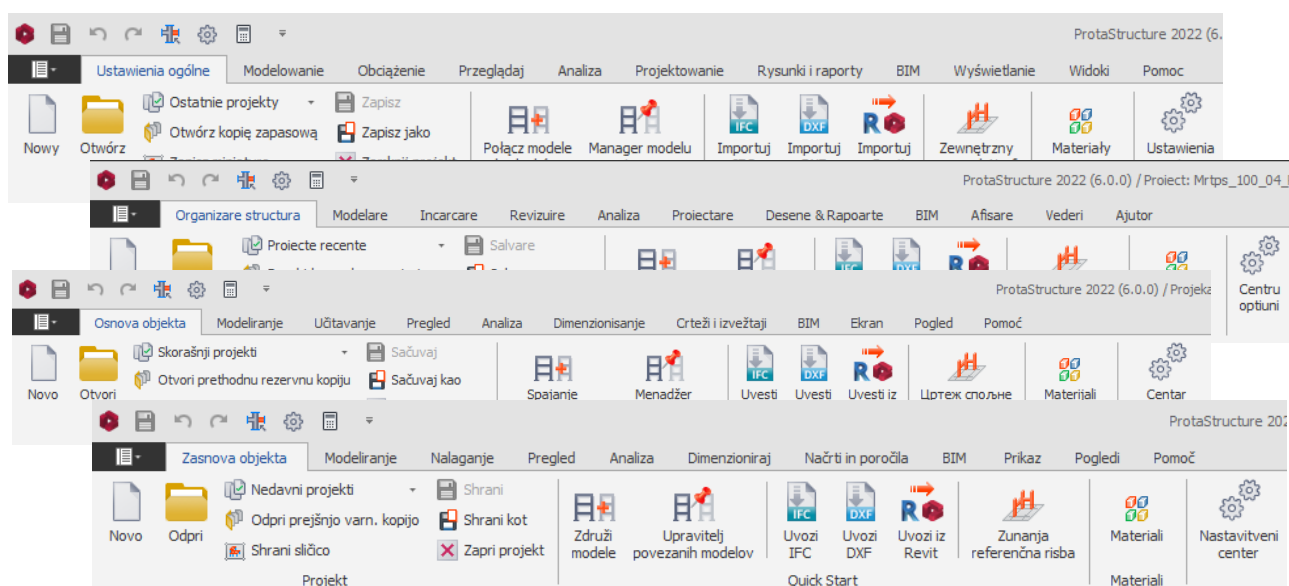
We are confident that you will enjoy and benefit from the new features and improvements in ProtaStructure Suite 2022. You will be able to find the details in the pages to follow.

Thank you for choosing ProtaStructure.

Extended Localization and Code Coverage

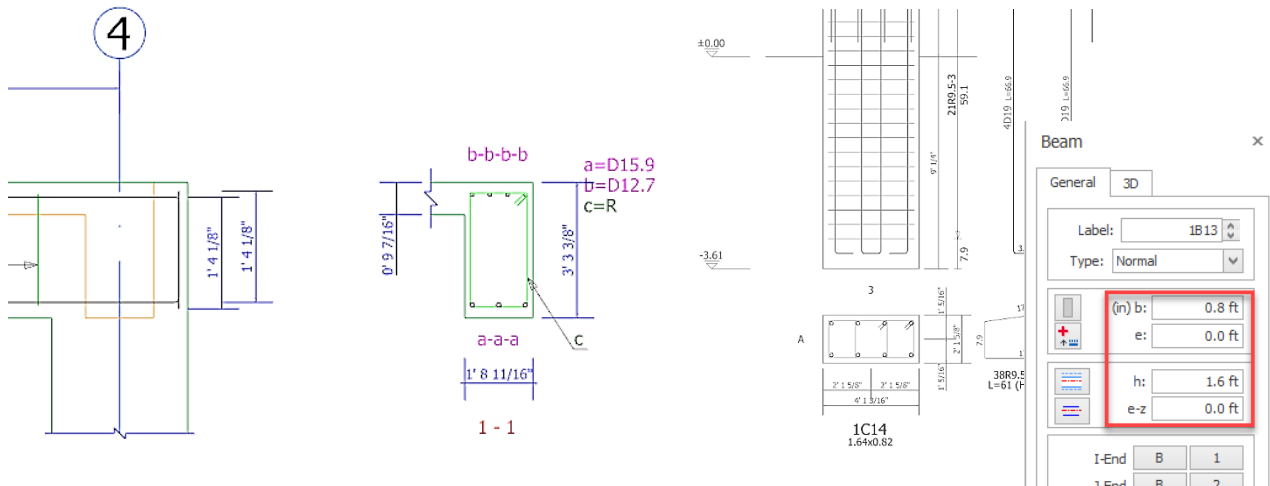
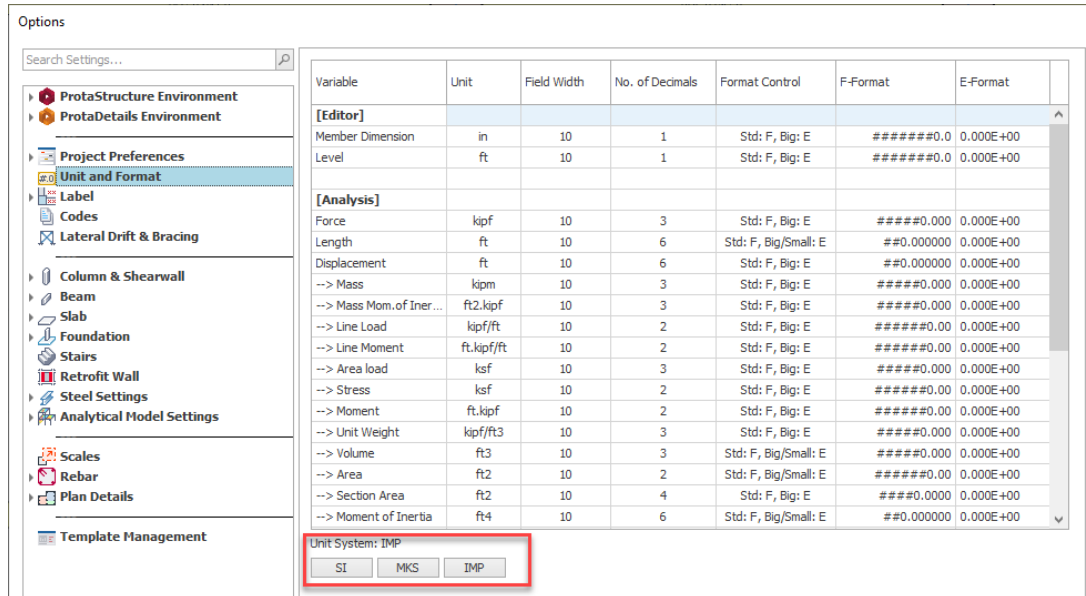
New Report and UI Languages

With ProtaStructure 2022, we have supported Polish, Romanian, Serbian, and Slovene languages. You will be able to set UI and Report languages separately. English, Spanish, Portuguese and Turkish are among the other supported languages.



Imperial Unit System

Imperial units can now be used during modeling, analysis, design, documentation, result review, and detailing.

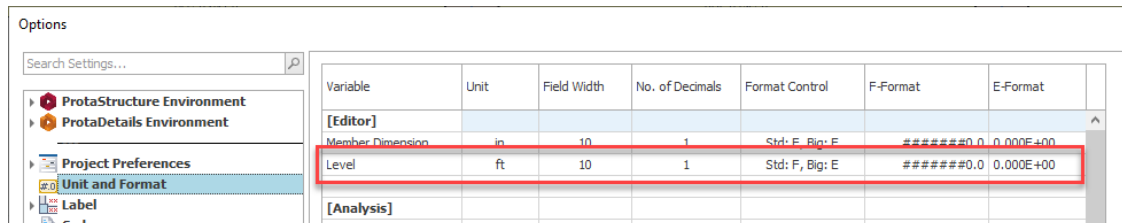


Seamless Switch Between Unit Systems

ProtaStructure 2022 offers three preset unit systems widely adopted by structural engineers worldwide: **SI**, **MKS**, and **Imperial**. The unit system is automatically adjusted based on the selected template and can be changed or fine-tuned later on by the user. Navigate Settings > Unit and Format menu to control the units.

Level Mark, Analysis Length, and Displacement Units

New unit selection entry, “**Level**,” is introduced with ProtaStructure 2022. As a result of user feedback, the level mark unit was differentiated from the member length unit.



Besides, ‘Length’ and ‘Displacement’ units used in the analysis are also exposed to user choice. You can change these units any time you like to see the analysis results in desired units. In ProtaStructure 2022, analysis is done with constant internal units, so you don’t need to repeat and update the analysis after you change any units.

Important Note:

The only limitation is the Post Analysis Reports created during the analysis. After changing the length and displacement units, you will need to repeat the analysis to see these reports in the correct units.

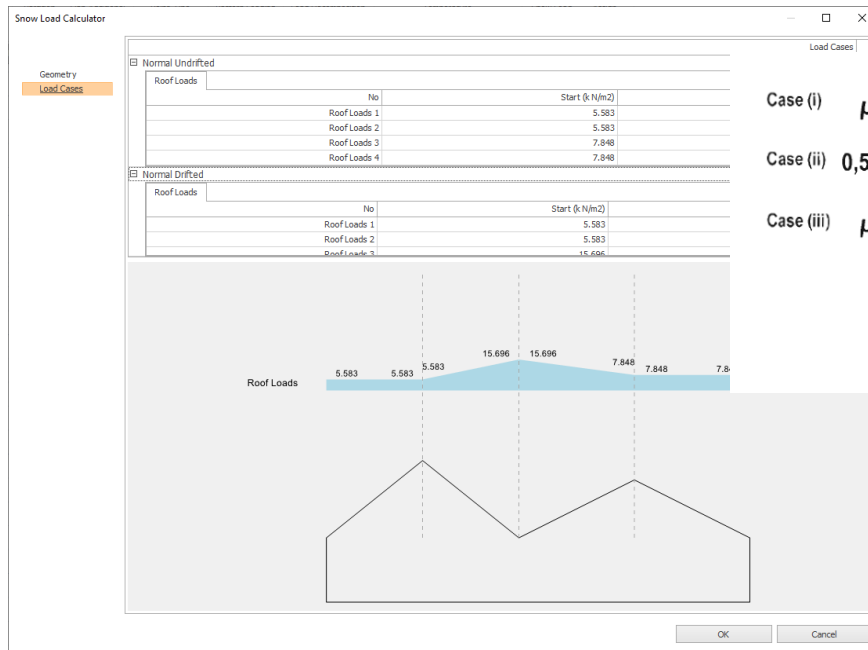
Extended Code Coverage

The following new codes are implemented, and existing code support is extended in ProtaStructure 2022. Design guides for code support can be found in our Help Center.

- Romania Seismic and Snow Loading Codes (EC8, P100, EC1)
- Wind Load Calculations to Colombian Seismic Code Title-B – Reglamento Colombiano de Construcción Sismo Resistente Título B – Cargas (NSR-10)
- Wind Load Calculations to Peruvian Loading Code, NTE020
- Peruvian Seismic Code, NTE030
- Peruvian Concrete Code (RC Beam Design)
- Brazilian Design Codes, NBR (RC Beam Design)
- Indian Design Codes, NBCI (RC Beam Design)
- Indonesia Seismic Code 2019 (SNI1726-2019)
- Indonesia Design Codes 2019, SNI (RC Beam Design)
- Philippines Structural Design Code, NSCP (RC Beam Design)

Automated Calculation of Snow Loads

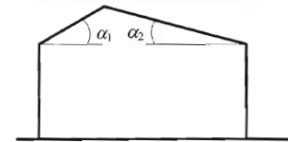
Snow loads can be automatically calculated to **EN1991-1-3 (Base code and Romania Annex)** and **TS498** loading codes.



Case (i) $\mu_1(\alpha_1)$  $\mu_1(\alpha_2)$

Case (ii) $0,5\mu_1(\alpha_1)$  $\mu_1(\alpha_2)$

Case (iii) $\mu_1(\alpha_1)$  $0,5\mu_1(\alpha_2)$



Stage Construction Analysis with Creep and Shrinkage

Staged construction analysis can now consider creep and shrinkage in addition to elasticity variations with time. For this purpose, variation of **time-dependent properties** (Elasticity, Creep, and Shrinkage) of concrete is introduced with ProtaStructure 2022.

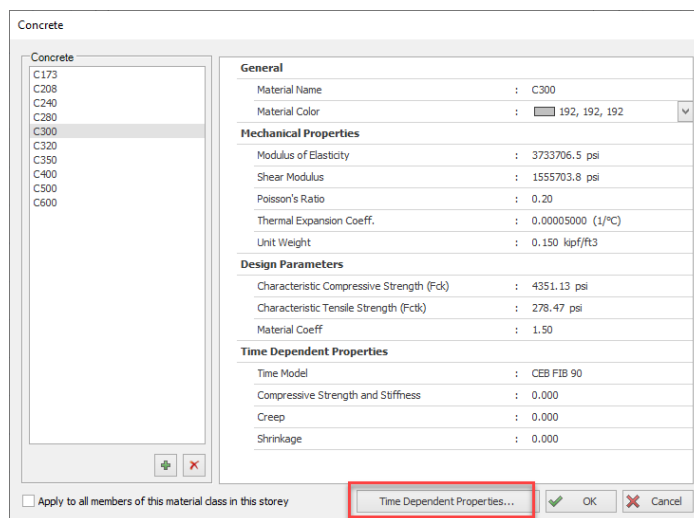
Although highly parametric new models are developed, predicting creep in reinforced concrete structures still maintains its uncertainty. This complexity is due to the material itself and the problem of assessing time-dependent deformations and loss of prestressing forces, which are usually not considered in structural analysis. One can tackle this uncertainty and complexity in creep prediction to a certain degree by adapting the code procedures that give general guidance.

In ProtaStructure, time models specified in **FIB Model Code 90** and **EN1992-1-1:2004** can be used. These time models are used explicitly in staged construction analysis. Creep and shrinkage strains are incorporated in the analysis depending on the time between stages and post-construction.

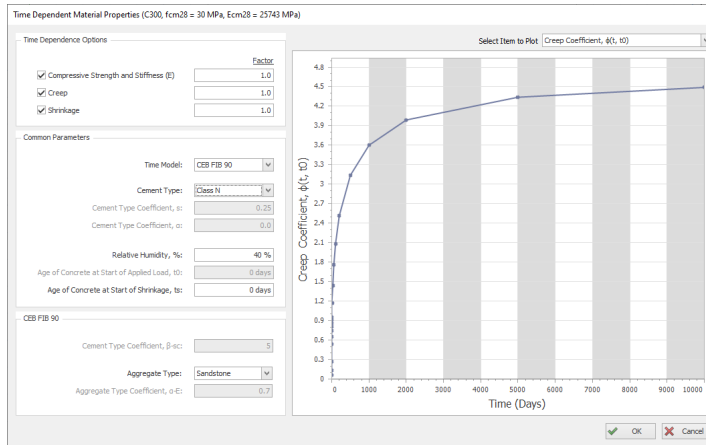
Time-dependent parameters and staged construction data are also communicated to general-purpose analysis software, including SAP2000.

To specify these parameters:

1. Open the concrete material properties dialog.
2. Click on the **“Time-Dependent Properties...”** button.



3. Check the **“Time Dependence Options”** you want to consider in the staged construction analysis. If you wish to partially consider these parameters’ effect, you can enter values **between 0 and 1**. Entering ‘0’ or unchecking the related option will deactivate the parameter.



4. Select the “Time Model” that you want to use. Available options are CEB FIB 90 and EN1992-1-1:2004.
5. Specify other parameters like **Cement Type**, **Relative Humidity**, **Start of Shrinkage**, and **Aggregate Type**.
6. The time models will be automatically considered in staged construction analysis once you assign the concrete material that you have edited to structural members (especially columns and walls)

New Seismic Checks and Calculations

Seismic Forces on Non-Structural Members

ProtaStructure 2022 can calculate forces acting on non-structural members and their connections to the building to ASCE07, Eurocode 8, and TBDY2018. Specify the non-structural members and their properties, and ProtaStructure will automatically calculate the story accelerations and forces depending on the analysis type. **The non-structural Member Force Calculator** is located under the “Loading” ribbon.

Seismic Force

According to **ASCE 7-16 13.3.1** horizontal seismic force, F_p , is defined as, “The horizontal seismic design force (F_p) shall be applied at the component’s center of gravity and distributed relative to the component’s mass distribution”. Calculation details are given in subclause **13.3.1.1** which states the formula (**13.3-1**) given below.

$$F_p = \frac{0.4 \alpha_p S_{DS} W_p}{I_p} \left(1 + 2 \frac{z}{h} \right)$$

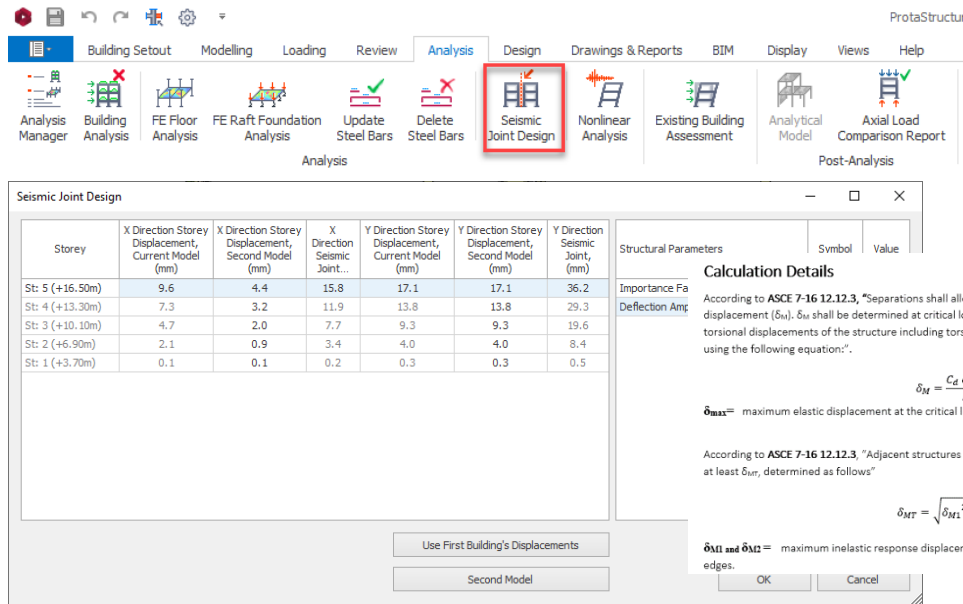
Seismic Coefficient

According to **EC8 4.3.5.2 (2)** seismic coefficient, S_a is defined as, “Seismic coefficient applicable to non-structural elements”. Calculation details are given in subclause **4.3.5.2 (3)** which states the formula (**4.25**) given below.

$$S_a = \alpha S \left[3 \left(1 + \frac{z}{H} \right) / (1 + (1 - T_a/T_1)^2) - 0.5 \right]$$

Seismic Separation of Buildings

To avoid pounding damage in adjacent buildings, we must provide spacing between structures. The calculation procedure is well defined in seismic codes. If you need to design the seismic separation between two buildings, you can use the “Seismic Joint Design” tool in ProtaStructure 2022. It is located under the “**Analysis**” tab on the ribbon. With this tool, you can either load the displacements of an existing ProtaStructure model or enter the second model’s displacement values manually. ProtaStructure can perform the calculations to **ASCE07**, **Eurocode 8**, and **TBDY2018**.



Extended Strong Column-Weak Beam and Joint Shear Checks

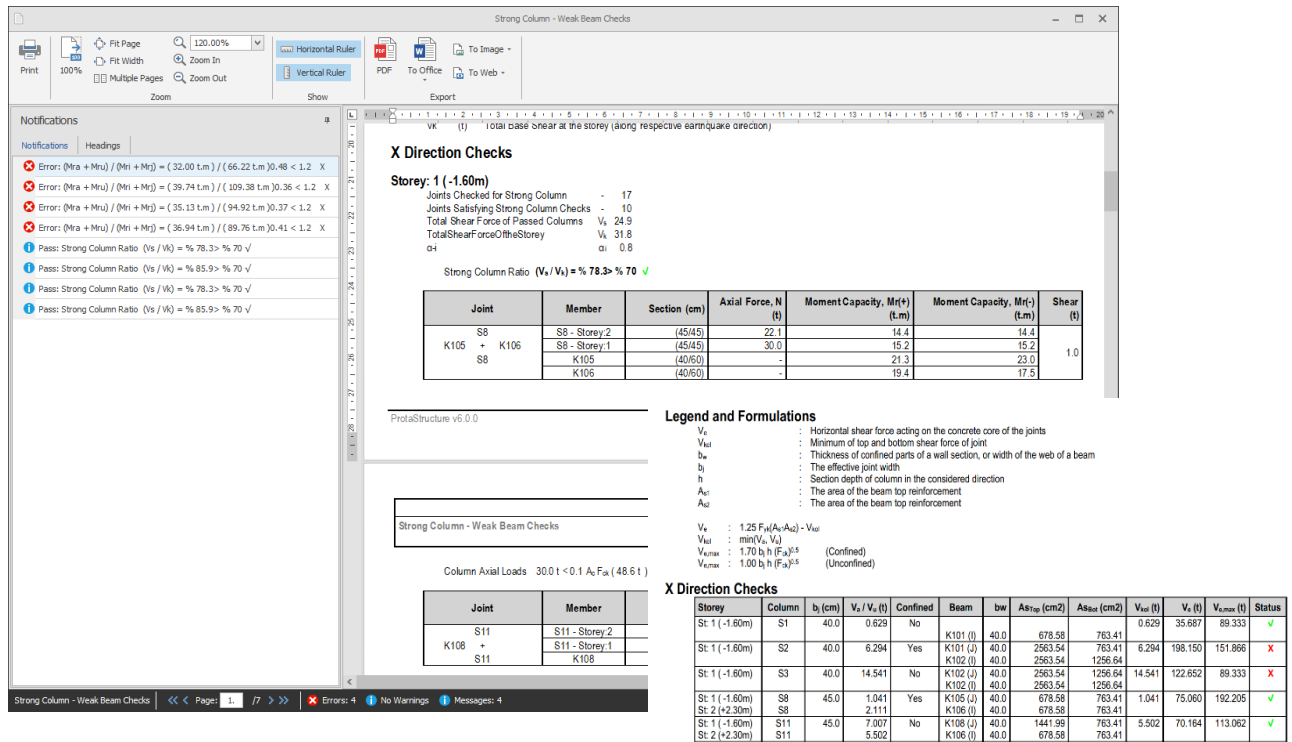
ProtaStructure 2022 can perform strong column checks of RC and steel special moment-resisting frames according to ACI318, Eurocode 8, and TBDY2018.

Checks for Steel Frames

Strong column checks for special moment resisting frames can be done to AISC341, Eurocodes, and Turkish Codes.

Enhanced Reports

In addition to new code coverage, the strong column-weak beam and joint shear check report are rewritten from scratch to include more visual components with intelligent notifications.

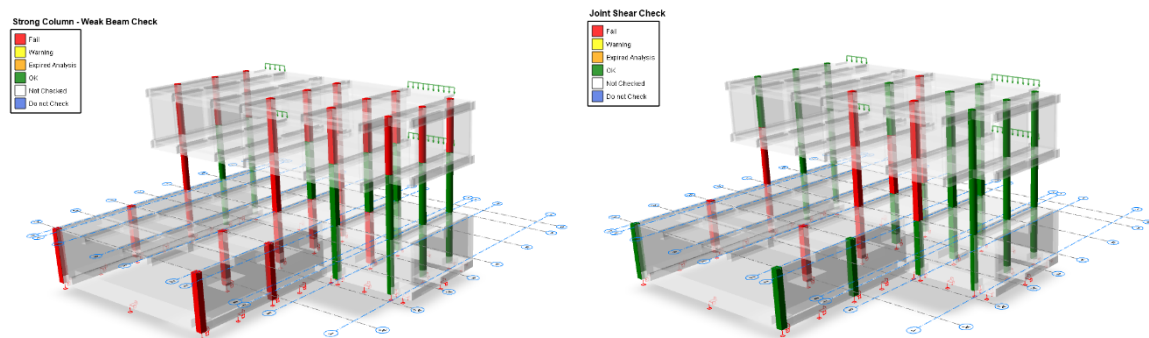


Optimized Workflow

Strong Column-Weak Beam and Joint Shear checks can now be reviewed for a single column and beam while designing their reinforcement. In this way, you can fine-tune the design while keeping an eye on these global seismic checks.

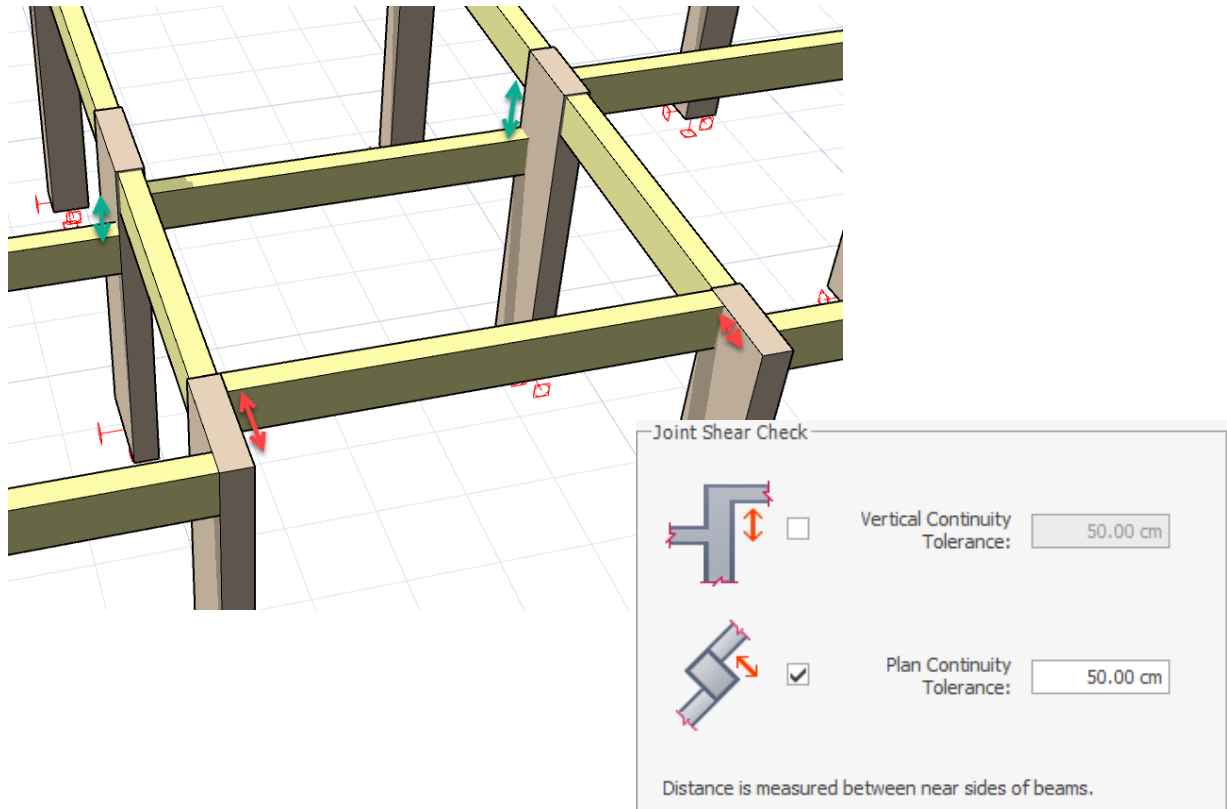
Visual Interrogation

These checks can be reviewed using the new visual interrogation options.



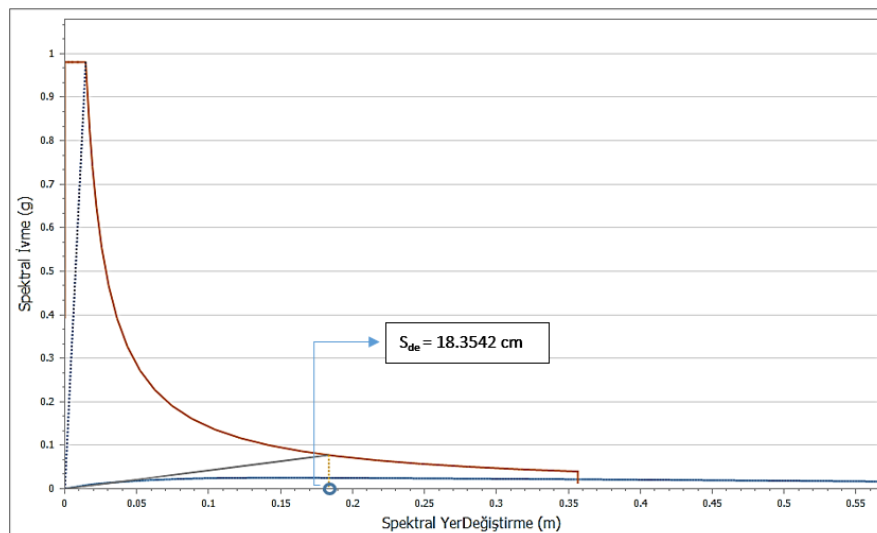
Discontinuous Beams in Joint Shear Check

Discontinuous beam in plan or elevation can now be optionally considered in Joint Shear Checks. You can set the **“Continuity Tolerance”** in plan or elevation. If the distance between beam faces is larger than the tolerance in a joint, they will be considered discontinuous. **“Continuity Tolerance”** settings can be accessed via the **“Settings > Beam > Design”** tab.



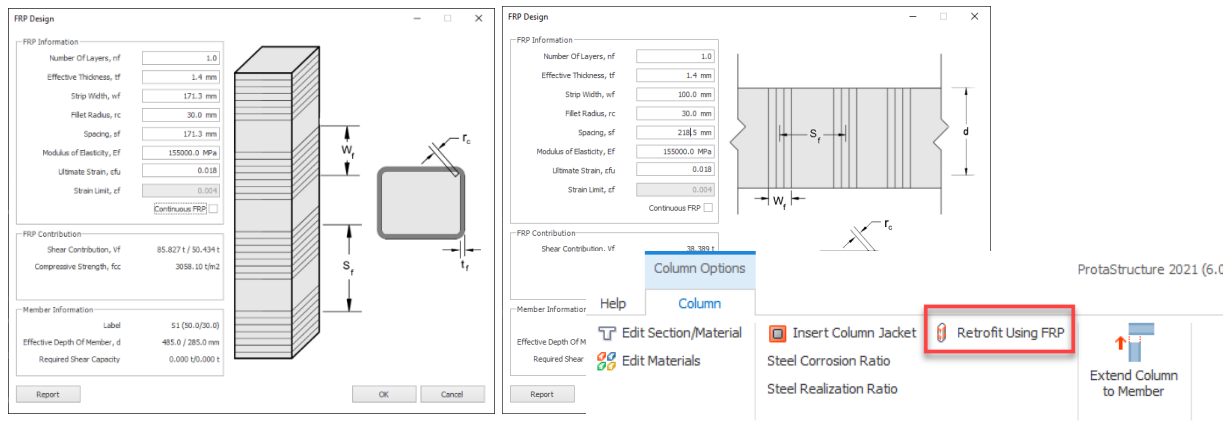
Target Displacement Calculation in Pushover Analysis

A vital step after pushover analysis is to find the demand that a target code spectrum requires (a.k.a, performance point). A target displacement identifies the performance point. After plotting the capacity curve, ProtaStructure 2022 can detect the target displacement value according to **FEMA356**, **Eurocode 8**, and **TBDY2018**.

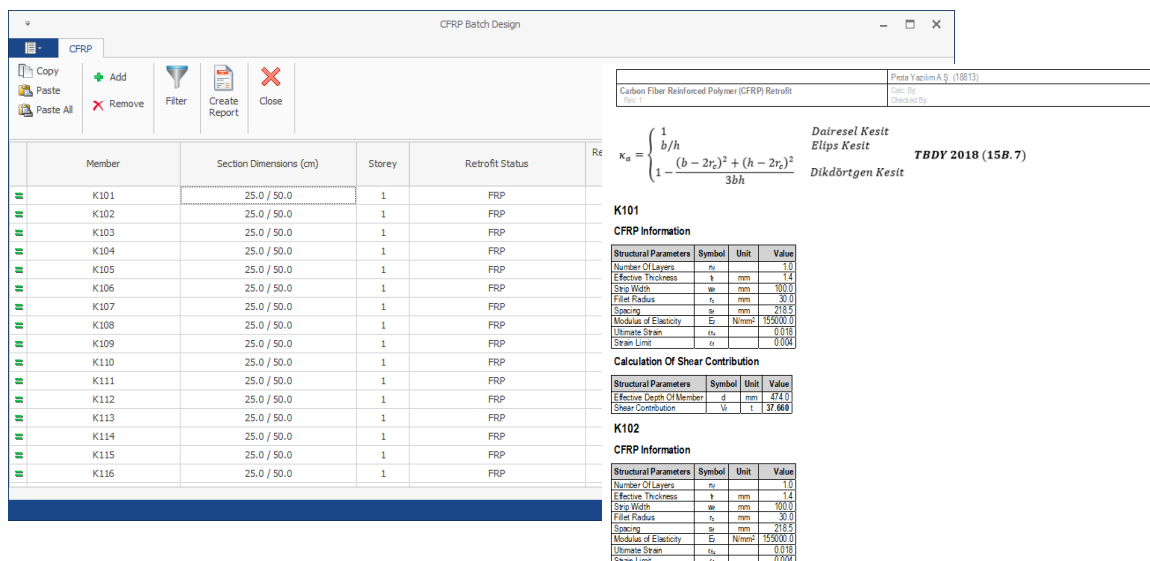
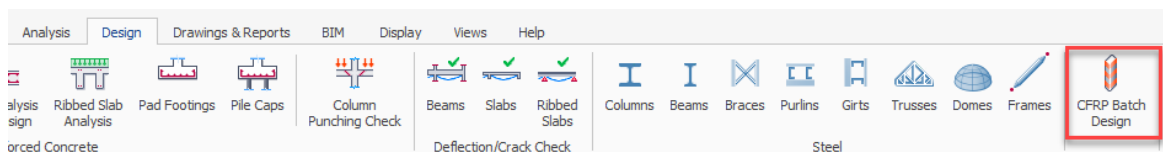


Retrofitting and Assessment with CFRP

ProtaStructure can retrofit and assess beam and column members confined with **Carbon Fiber Reinforced Polymers (CFRPs)**. The CFRP confinement positively affects the shear resistance and axial load capacity of members. Currently, ProtaStructure can consider these effects according to the TBDY2018 seismic code. CFRP definition can be assigned to the members using the “**Retrofit Using FRP**” command on the member’s right-click menu or contextual ribbon tab.



We have developed a batch design interface for fast definition of FRPs and copying these definitions between members. You can manage your FRP retrofit designs collectively and get a detailed design report that includes all retrofitted members. The batch design interface is accessible at **Design Ribbon Tab > CFRP Batch Design**.



New Tools in BIM Integration

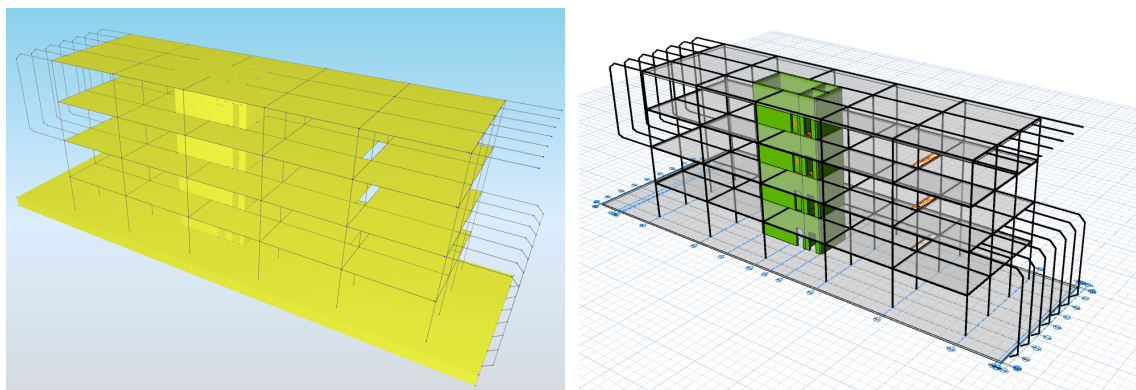
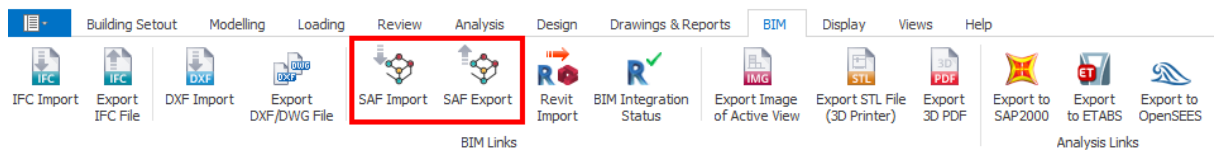
BIM integration is always a significant priority item in the ProtaStructure development roadmap. Projection of the changing norms, digitalization, stringent emission standards, and carbon footprint reduction in structural engineering passes through the application of BIM technology. We strive to improve our BIM collaboration infrastructure and introduce new capabilities with each new release.

Bi-directional Communication with SAF Files

SAF is an initiative from the Nemetschek Group to improve the collaboration between structural engineers by developing an open exchange format for exchanging data between structural analysis software based on the Excel format. It focuses on a practical, easy-to-use design that structural engineers can use in daily practice, and it is meant as an open format.

SAF communication is currently supported by leading structural and architectural software, and Prota is also participating in this workgroup.

The first implementation of the SAF format is provided with ProtaStructure 2022 and more to come with future updates.



Improved Revit Integration

User feedback is implemented, and enhancements are made in our dedicated Autodesk Revit Add-on. It is also adapted to work with **Autodesk Revit 2022**.

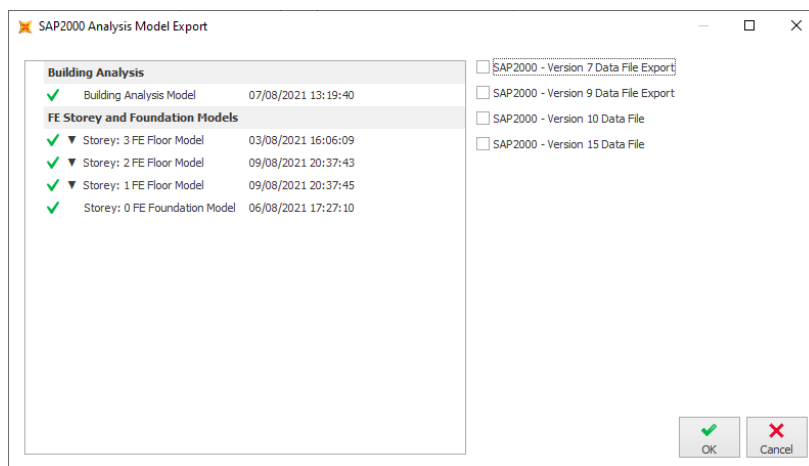
Streamlined UI and Dialogs

All BIM Collaboration dialogs and wizards have been aligned to provide a similar user experience.

Improved Analytical Links with SAP2000

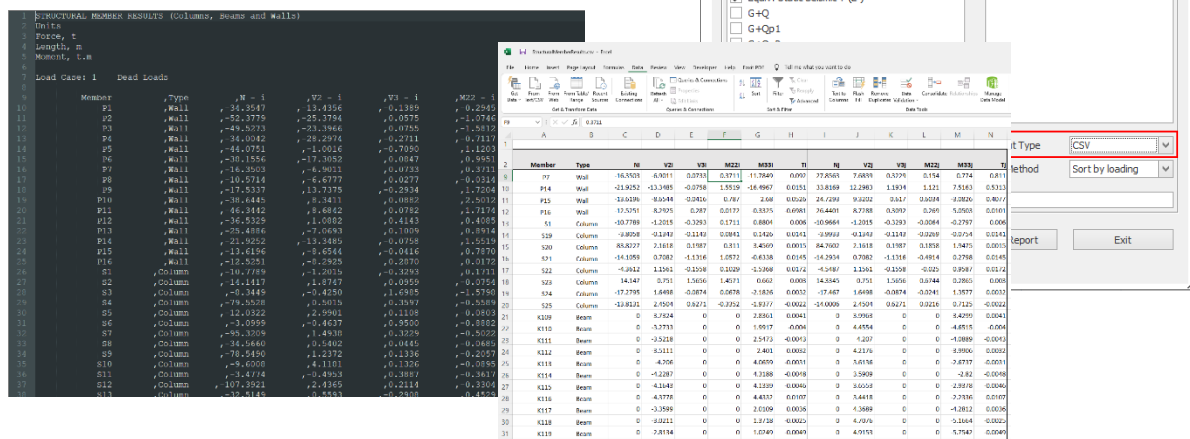
SAP2000 links have been enhanced to export FE Floor and Foundation models in addition to the Building Analysis model. All selected analytical models can now be exported at once.

Time-dependent concrete material properties and **staged construction data** are also communicated to SAP2000.



Analysis Results in CSV Format

We have received much user feedback for exporting the analysis results in comma-delimited (CSV) file format. In ProtaStructure 2022, we have provided this so that you can export and interpret the results in your custom spreadsheets or utility software. You can create the CSV files using the “Analysis Results Report” dialog.



Flexible Loading System

One of the revolutionary features in ProtaStructure 2022 is our new and flexible loading system.

Don't worry! All the standard loads are automatically calculated and decomposed on the members. You may still need to apply additional loads in any orientation and load case. For this purpose, you can use the new interactive load editor.

For example, you can now apply horizontal loads on the beams along their lateral axis.

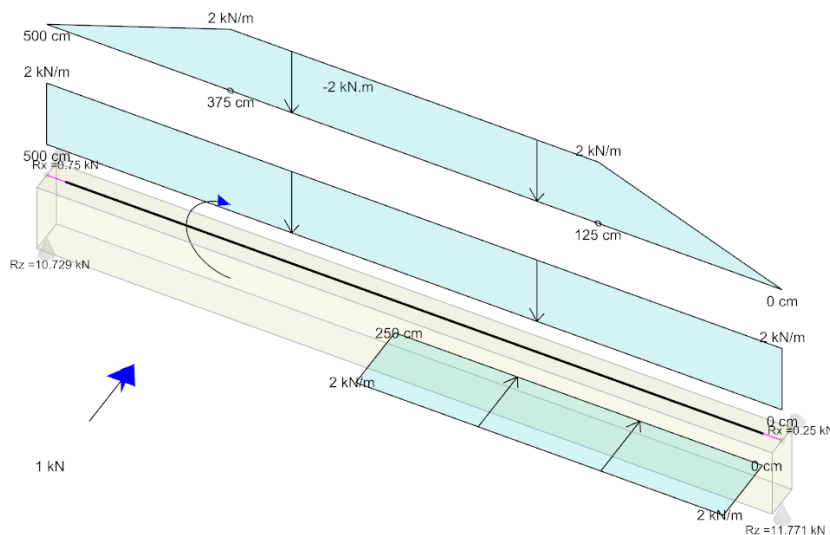
In addition to this, point and uniform loads can be manually applied on truss joints and truss members.

You can also visualize the loads in the structure in 3D. You can view the entire storey with the loads applied and even insert new loads on this view.

We are sure you will find a lot of flexibility and interactivity in our new load editor.

Assign Loads in Any Direction and Visualize in 3D

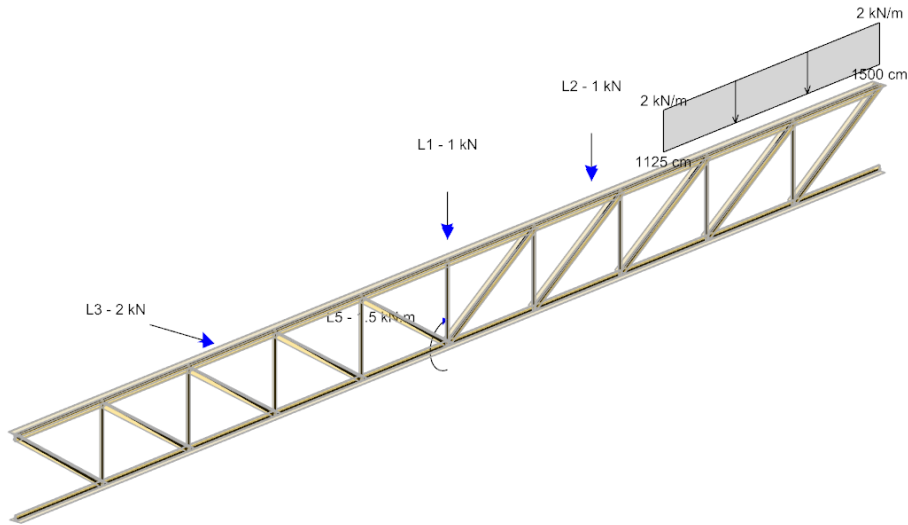
You can now assign loads on members in any direction and any load case with the help of the new **"Load Editor"** and visualize them in 3D for better inspection.



Manual Loading on Truss Joints

In ProtaStructure, trusses usually resist indirect loads automatically transferred from purlins, claddings, frames, and connected beams.

ProtaStructure 2022 can now assign manual loads on truss joints and chord spans.

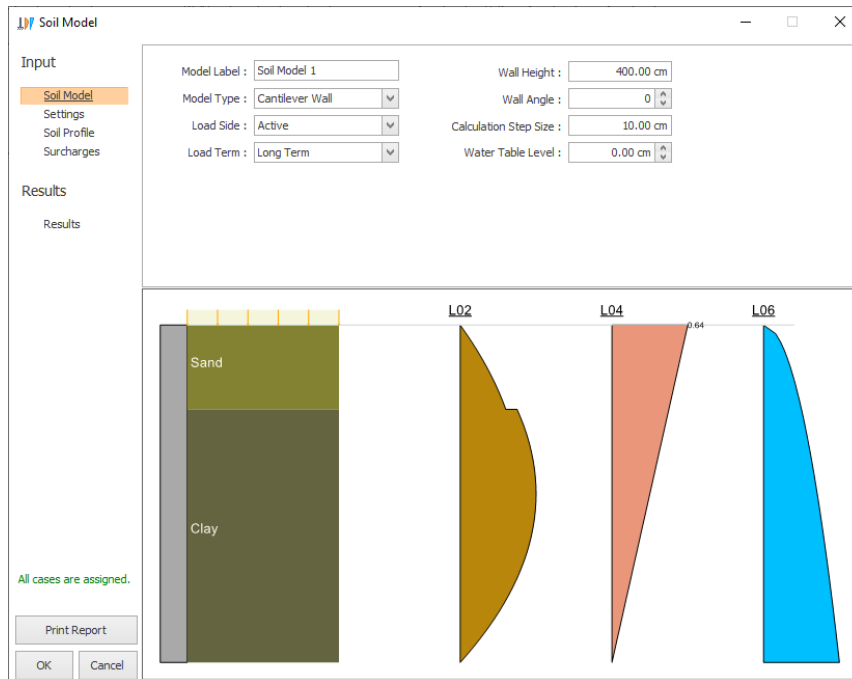


Static and Dynamic Soil Thrust on Basement Walls

In ProtaStructure 2022, we have developed a new soil thrust calculation wizard. It can calculate static and dynamic loads soil with or without water table, including surcharge loads according to geotechnical principles for cantilever and basement walls. Any number of soil layers and water table can be defined.

The dynamic load calculation depends on the selected seismic code if the wall in question is the basement wall. Currently, only TBDY2018 seismic code is supported for dynamic basement loads.

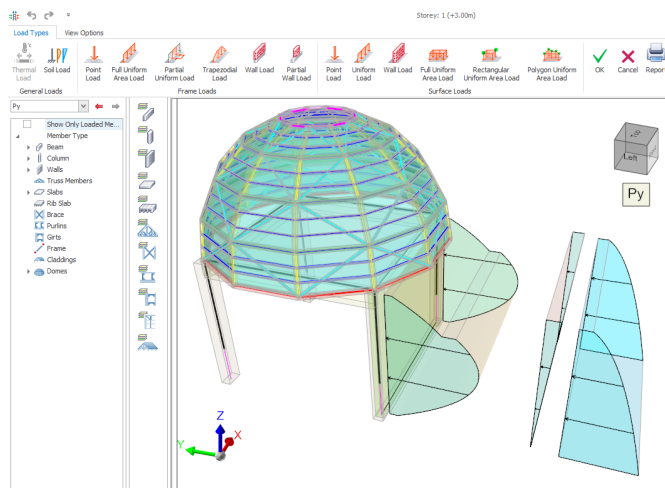
In summary, this wizard can be used for all codes as a means for the calculation of the soil loads on walls following the geotechnical principles. You will see the same wizard powering the pool macro in ProtaDetails.



Enhanced Load Visualization

The most-reported feedback from our users was the lack of 3D visualization of the loads assigned on the structure. Although ProtaStructure had several dialogs and editors to visualize the loads, it was impossible to picture them all together in 3D before the building analysis.

With ProtaStructure 2022, you can visualize the loads on the entire structure or a part of it.



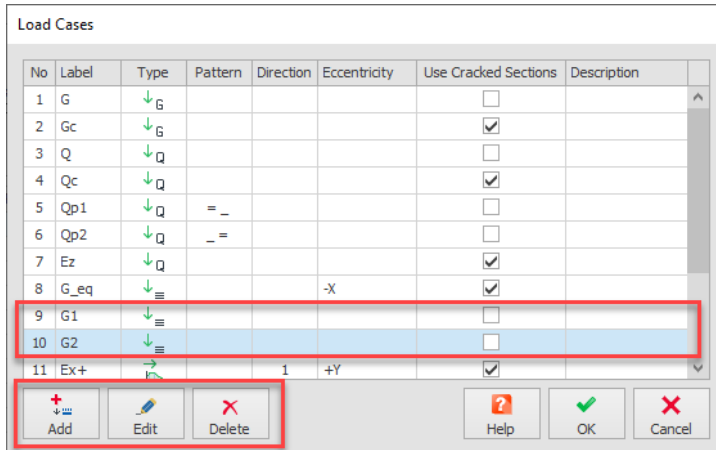
User-Defined Vertical Load Cases

Thanks to the new loading framework, you can now create **more than one Pattern G (Gravity)**, **Pattern L (Live Load)**, and **User-Defined Vertical Load** case and assign loads to them. In addition to this, you can specify if user-defined vertical load cases will contribute to seismic masses or not. In the

previous versions of ProtaStructure, our users could create user-defined vertical load cases but could not assign any loads to them.

User-defined vertical load cases are not automatically considered in the load combinations. So, you must add them manually to the load combinations. To define new additional vertical load cases:

1. Go to the “**Building Analysis > Pre-Analysis > Loading Combinations > Load Cases**” menu
2. Click the “**Add**” button

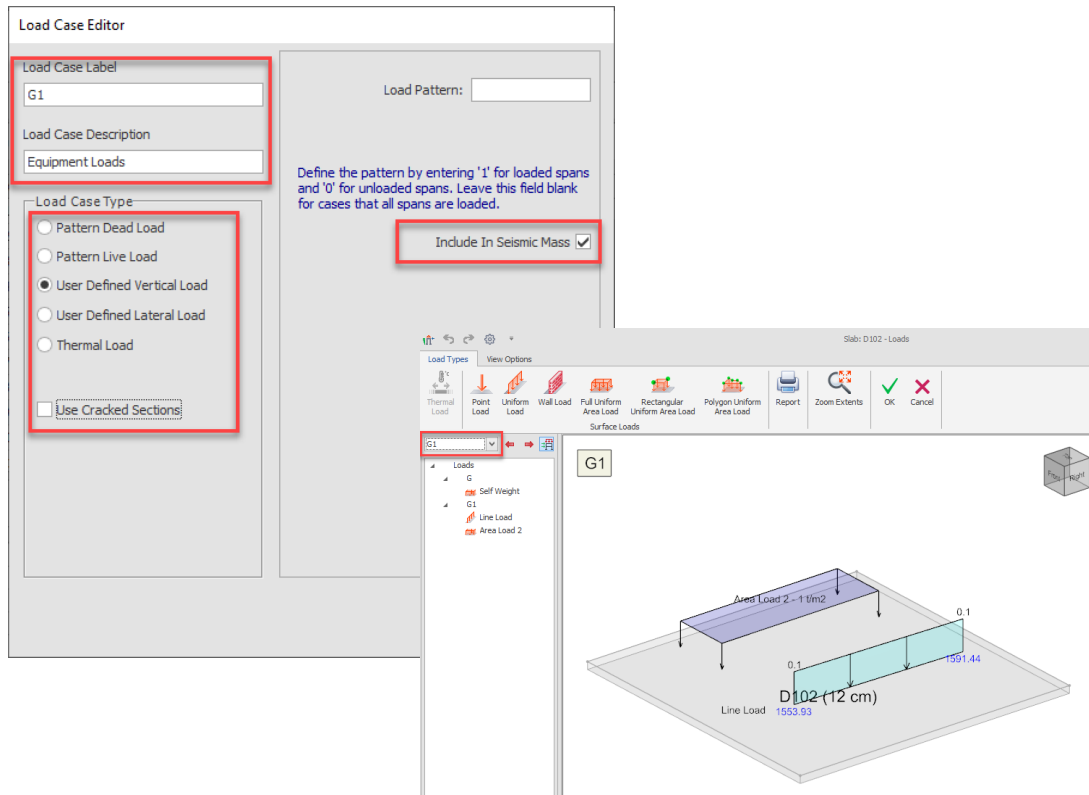


No	Label	Type	Pattern	Direction	Eccentricity	Use Cracked Sections	Description
1	G	↓ G				<input type="checkbox"/>	
2	Gc	↓ G				<input checked="" type="checkbox"/>	
3	Q	↓ Q				<input type="checkbox"/>	
4	Qc	↓ Q				<input checked="" type="checkbox"/>	
5	Qp1	↓ Q	= _			<input type="checkbox"/>	
6	Qp2	↓ Q	_ =			<input type="checkbox"/>	
7	Ez	↓ Q				<input checked="" type="checkbox"/>	
8	G_eq	↓ ≡			-X	<input checked="" type="checkbox"/>	
9	G1	↓ ≡				<input type="checkbox"/>	
10	G2	↓ ≡				<input type="checkbox"/>	
11	Ex+	→		1	+Y	<input checked="" type="checkbox"/>	

3. Change the **Label** and the **Description**.
4. Following load cases are automatically generated by the ‘Loading Generator.’ They can not be edited or deleted.
 - a. Dead Loads
 - b. Live Loads
 - c. Pattern Dead Loads
 - d. Pattern Live Loads
 - e. Dead Loads with Cracked Section (Dc or Gc)
 - f. Live Loads with Cracked Section (Lc or Qc)
 - g. Vertical Earthquake Loads

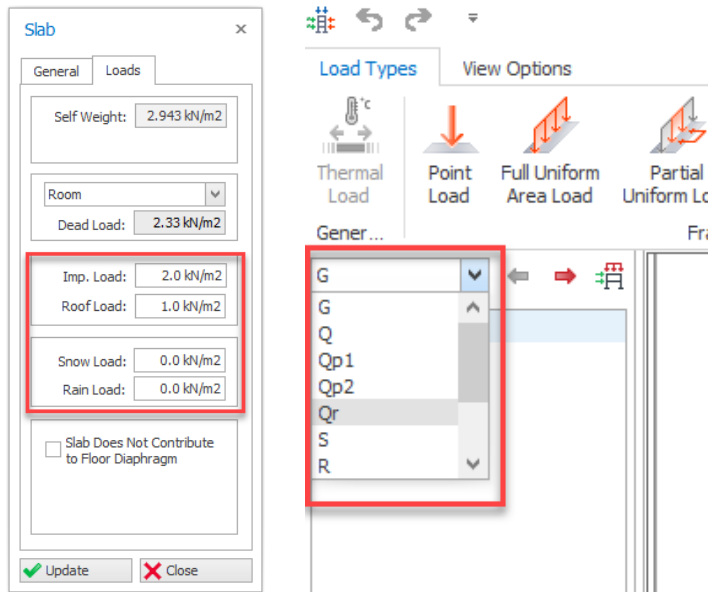
You can not define a second Dead or Live Load case with the same pattern. You can define as many dead or live load cases with different patterns. ‘Dead’ and ‘Pattern Dead’ load cases automatically include the self-weight of members.

If you want to define a new vertical load case without the self-weight of the members, pick the “User Defined Vertical Load Case” option. You can check “**Include In Seismic Mass**” to include the loads in seismic mass calculations.



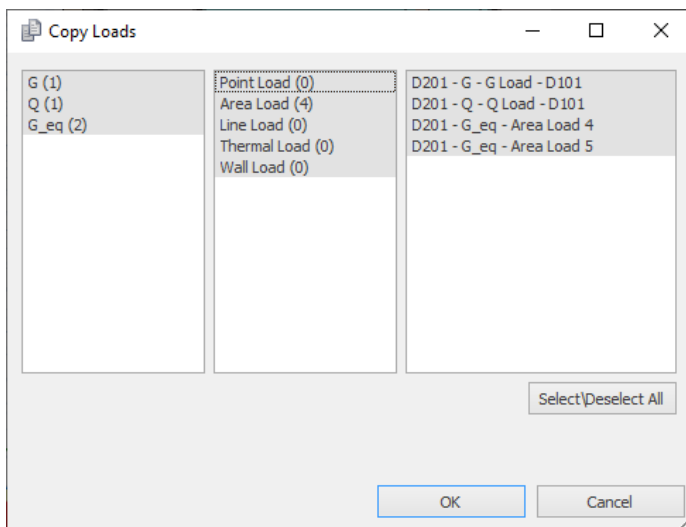
Roof Live Load, Snow and Rain Load Cases

Separate load cases (and combinations) for roof live loads, snow, and rain loads can be automatically created in ProtaStructure 2022. While snow loads can be calculated automatically by the program, other loads can be assigned manually to these load cases.



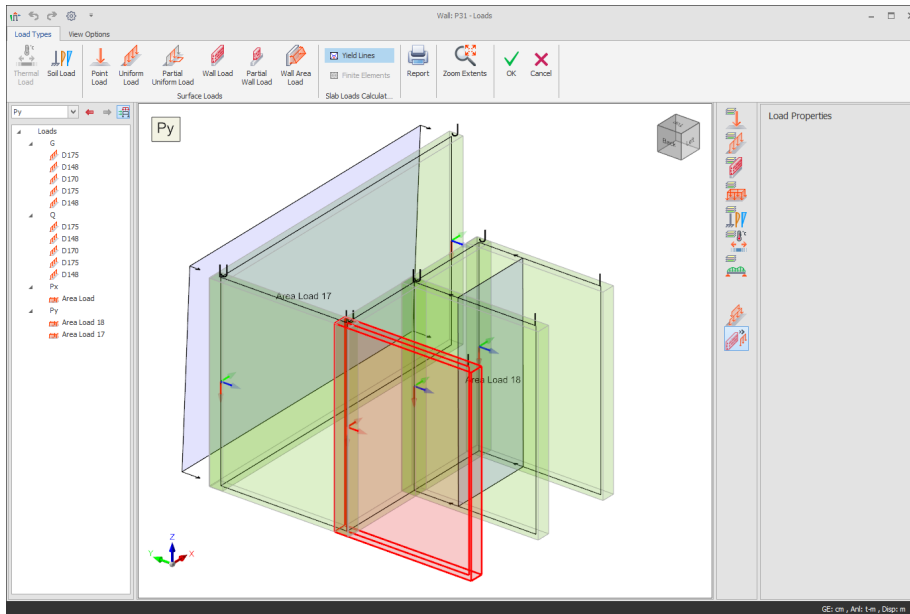
Copy and Paste Loads

As the variety of the available loading options has increased, a more advanced load copying functionality is developed. Previously, you could only copy infill wall loads and user-defined loads on beams. Now, you can copy and paste any type of load between members in any load case using a single user interface.



Better Loading Experience for Corewalls

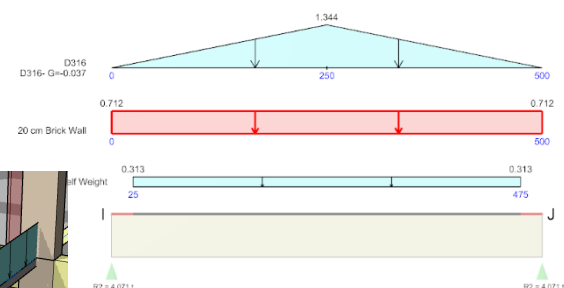
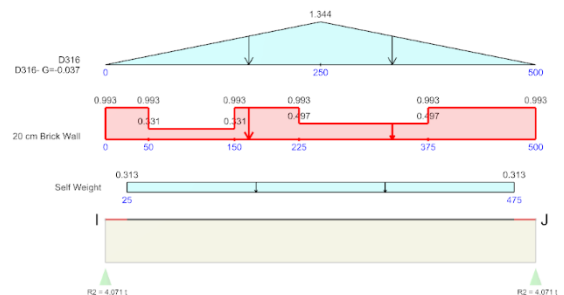
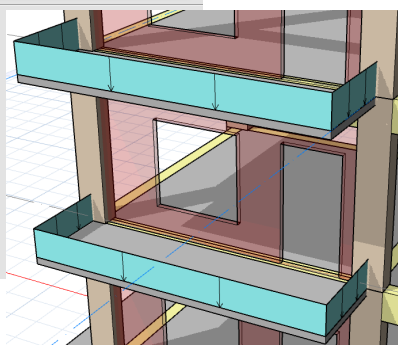
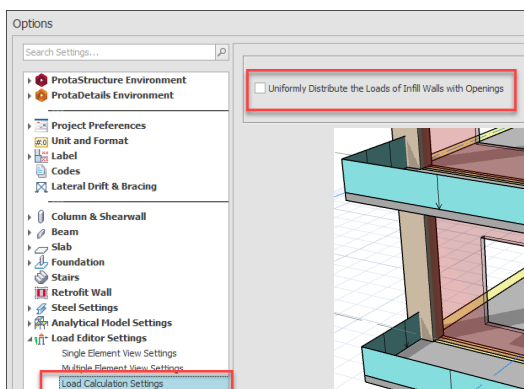
With ProtaStructure 2022, corewalls can be loaded before or after merging the corewall panels. The loads that are defined before the merging are preserved.



Detailed Load Profile for Infill Walls with Openings

ProtaStructure 2022 can optionally calculate a detailed load profile for the infill walls with openings. The exact position and dimension of the openings are considered, and the load profile is calculated accordingly.

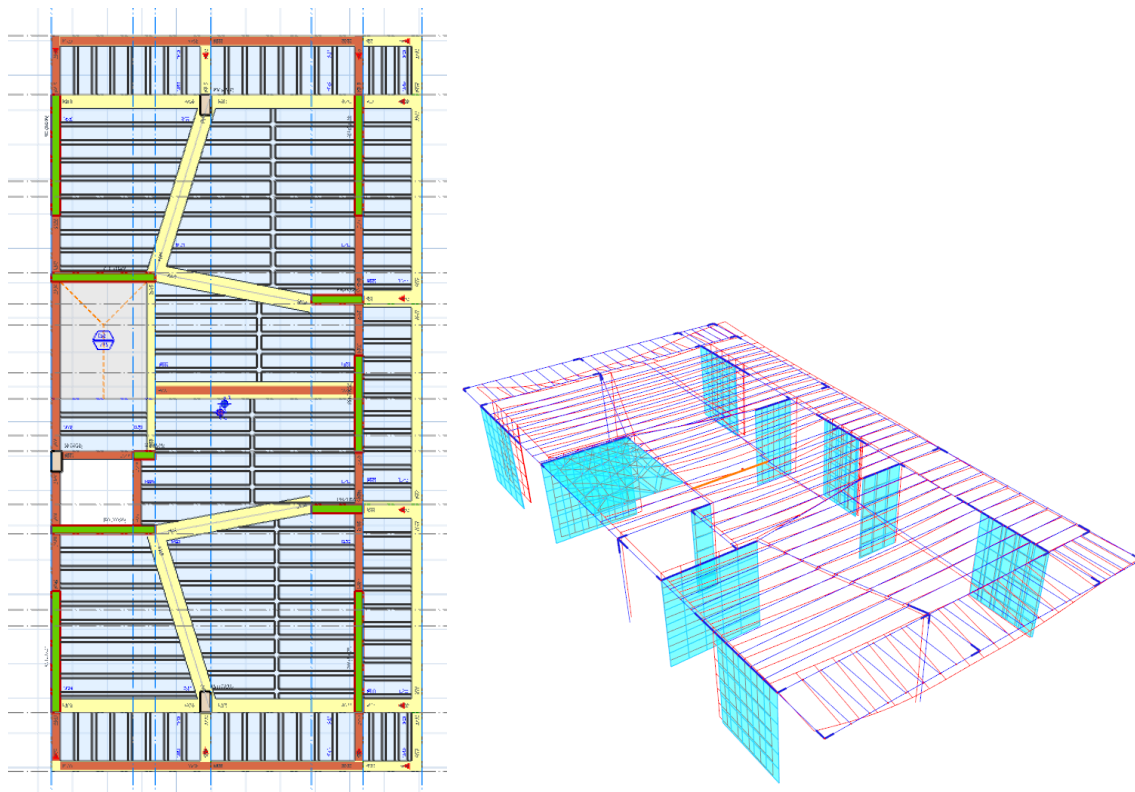
You can activate the detailed load profile calculation by unchecking the “**Uniformly Distribute the Loads of Infill Walls with Openings**” option in “**Load Editor Settings**.”



Rib Slab Loads –Automatically Decomposed in Building Analysis

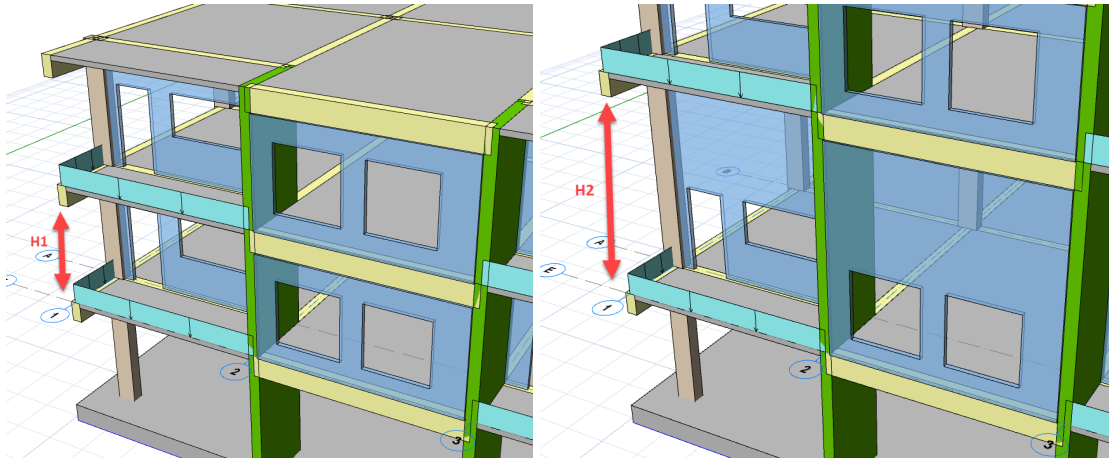
In previous versions of ProtaStructure, ribbed slab loads were automatically decomposed by Yield Lines Method or FE Decomposition Method **'before'** building analysis.

With ProtaStructure 2022, rib beams are **always automatically included** in the building analysis model, and loads are transferred automatically during the analysis using the connectivity information.



Automatic Height Adjustment for Infill Wall Loads

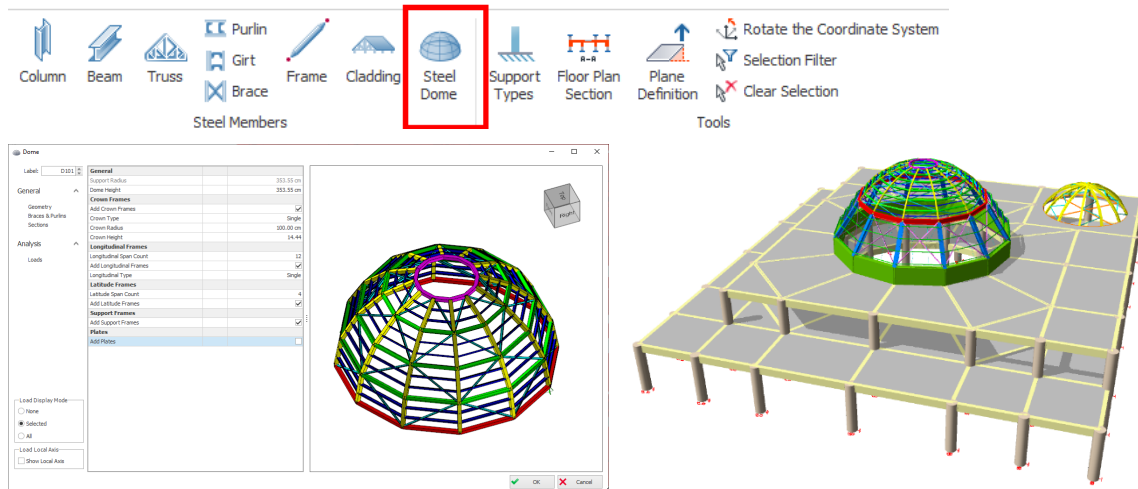
The height of infill walls can now be automatically adjusted when the storey height is changed.



New Modeling Tools

Steel Domes

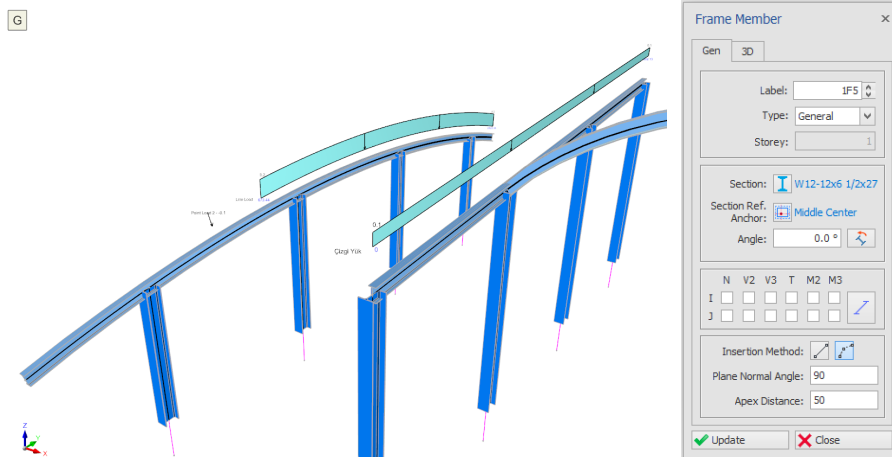
You can now insert steel dome members in your models with the help of a highly parametric wizard. Domes can be inserted by picking two points defining their diameter or selecting a slab hole.



Curved Frame Members

Frames that are curved on horizontal, vertical, or any inclined plane can be inserted with ProtaStructure 2022. To insert such frames:

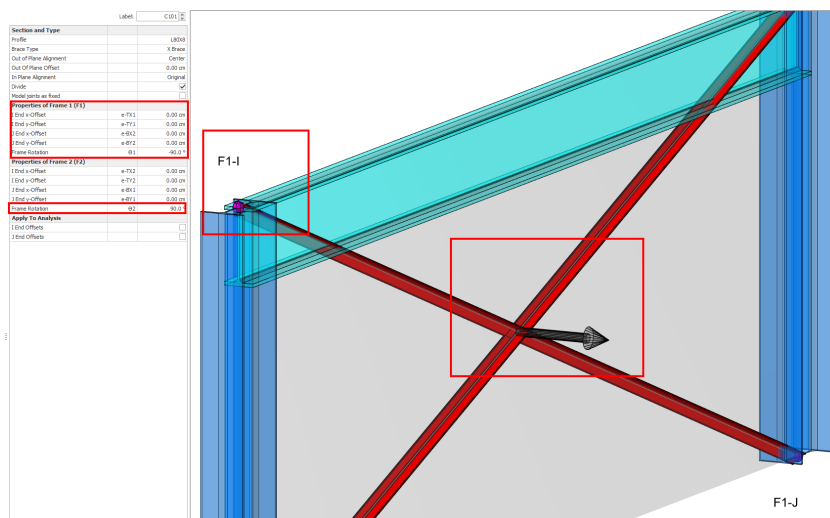
1. Select the **“Curved”** option on the frame properties window,
2. Pick two points on the screen and specify the apex distance.
3. Enter the **“Plane Normal Angle”** to change the 3D orientation of the frame member.



Improvements in Brace Insertions

Significant improvements are made in brace generation and insertion in Protastucture.

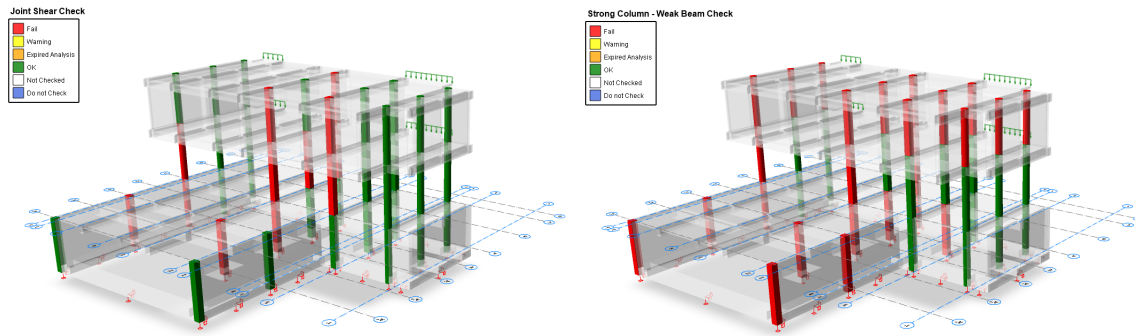
1. Plane Normal is now displayed in 3D view. This visual hint will help you to visualize and locate frame rotation.
2. Connected elements are shown in a translucent manner for a better presentation.
3. Frame labels are displayed, and frame ends are marked in 3D view.
4. The frame's end offsets and in-plane rotations composing a brace can be adjusted independently.



New Visual Interrogation Options

Following new visual interrogation items are now available with Protastucture 2022:

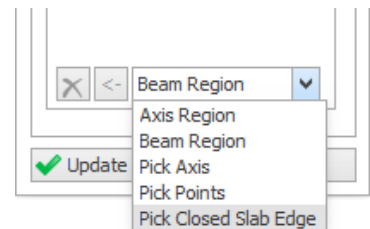
1. Strong Column - Weak Beam Check Status
2. Joint Shear Check Status
3. Member Performance Assessment Status



New Slab Insertion Method: Pick Closed Slab Edge

Slabs can be inserted now by selecting a closed slab edge line on the screen without the need for surrounding grids.

1. Select the **“Pick Closed Slab Edge”** option on the slab properties window and pick a slab edge object on the screen.
2. Alternatively, you can select the **“Beam Region”** option and pick a point inside the closed slab edge object. In both methods, you don’t need the existence of surrounding grids.

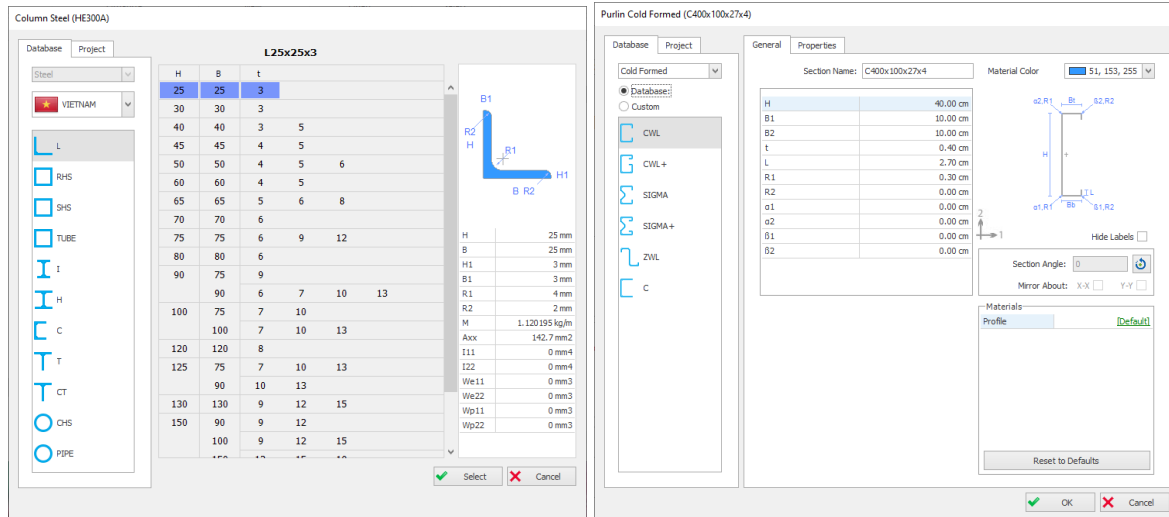


Splitting and Joining Beam and Frame Members

Collinear beams and frame members can now be merged or split for a more straightforward model modification.

Extended Steel Profile Library

The steel profile database is extended for Vietnamese and Malaysian high-strength Z and C profiles.



Enhancements in Performance and Memory Usage

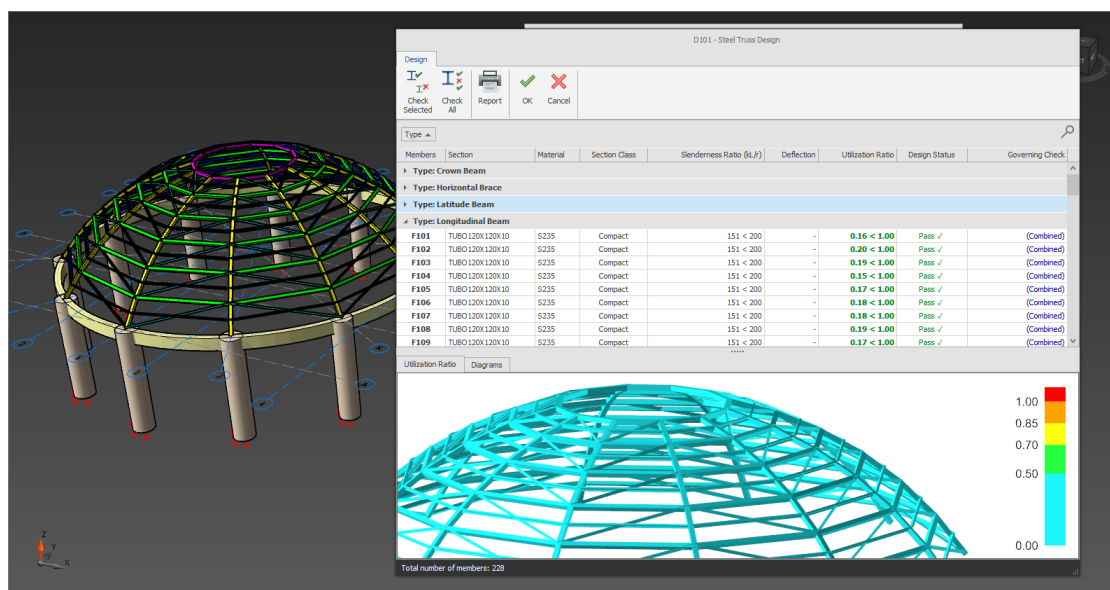
ProtaStructure 2022 has optimized visualization, general usage, and memory management performance. Depending on the project size, we have achieved overall performance gains up to 10x in modeling. The performance improvement can be recognized better in larger projects.

Transparency and Quality in Design

In the past years, we have received a lot of user requests regarding the reports that we generate in ProtaStructure. The following titles summarize what we have done new in our design and documentation system.

Design of Steel Domes

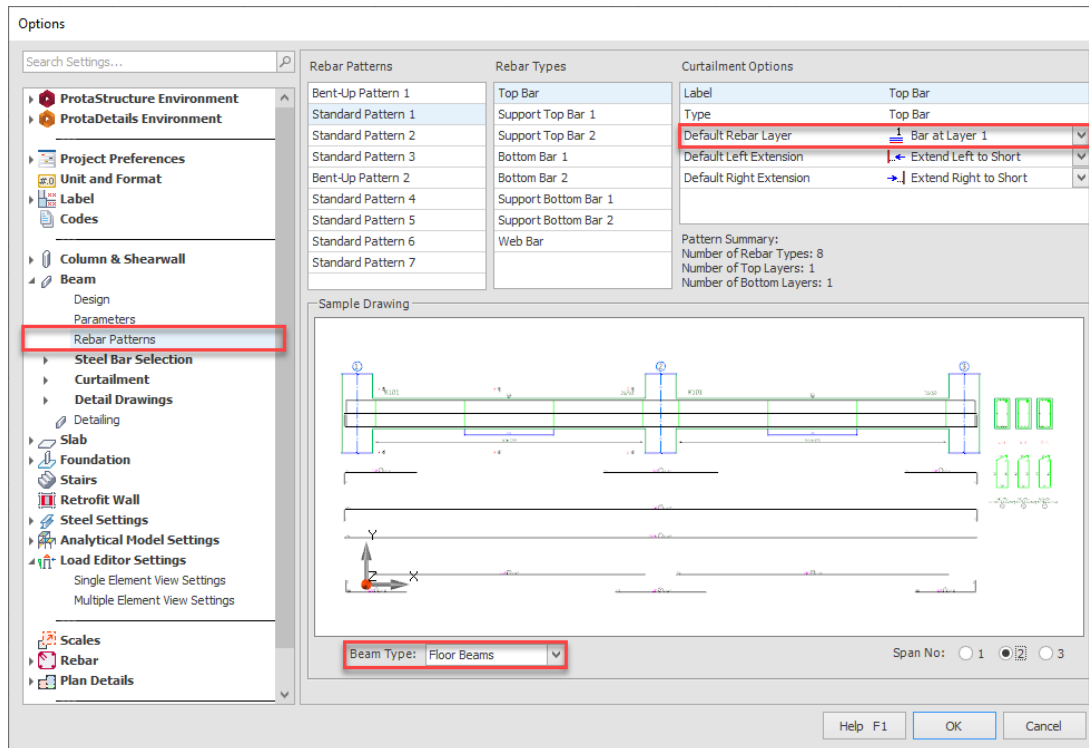
Design of the individual members in a steel dome can be done in ProtaStructure 2022.



New Beam Rebar Patterns

In ProtaStructure 2022, we have developed a flexible rebar patterning system that integrates custom patterns accurately and efficiently. The pattern editor itself is not released to our users yet, but you will be able to benefit from its substantial advantages.

As a result of user feedback and localization requirements, we have placed several preset rebar patterns to meet the beam design and curtailment needs. You can review these patterns via the **“Options > Beam > Rebar Patterns”** menu.

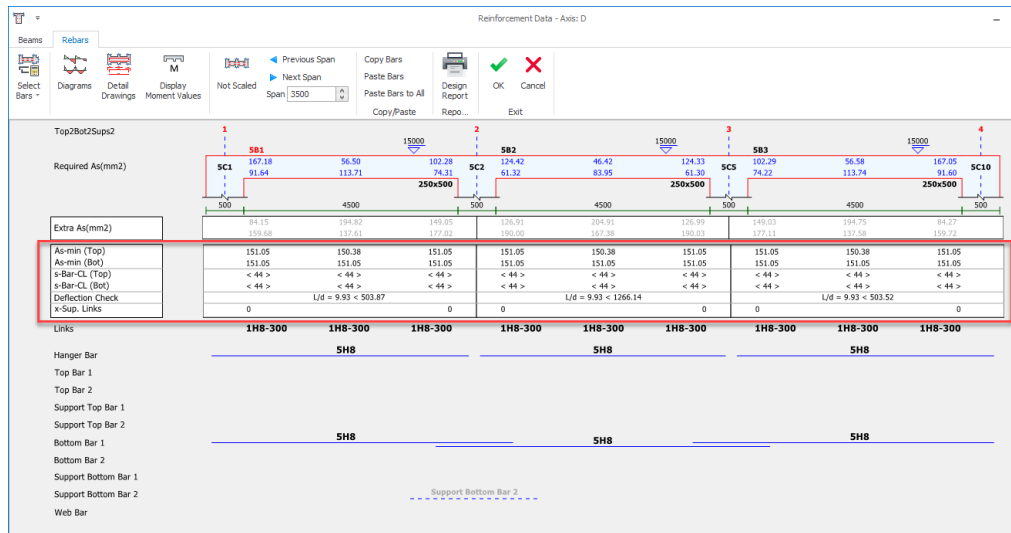


A brief walkthrough on how to use the rebar patterns is as follows:

1. Rebar patterns differ **for Floor, Rib, and Foundation** beam types. This setting can be switched using the “**Beam Type**” list at the bottom of the window.
2. A dynamic live drawing exhibits the selected options on the user interface for easy review. This review can be adjusted for up to three spans.
3. The label and type of rebars are fixed and cannot be edited. The rebar types constitute the backbone of the pattern system, such as **Hanger Bar, Top Bar, Support Top Bar, Bent-Up Bar, Bottom Bar, Support Bottom Bar, Web Bar**, and so forth.
4. Click one of the rebar types. The settings for the selected type will be displayed on the right. **The Default Rebar Layer controls the layer where** a specific rebar type is placed initially. The curtailment algorithm will put the rebars on the same layer starting from the top if given the same layer.
5. If the algorithm fails to place all bars in the first row, it puts the top bar with “Extend Left to Short” or “Extend Right to Short” options on the second row. If “**Default Rebar Row**” is set 2, this rebar is placed directly on the second row.
6. ProtaStructure provides up to 3 default rows. However, this does not mean that number of rows is limited. These are just default rows, and as explained above, the algorithm decides the actual row. It means if you have a deep beam with a narrow width, you can even have six rows of top bars, although you only have one layer of the top bar.
7. Set the default right and left extensions for the selected rebar.

Modified Beam Rebar Editor

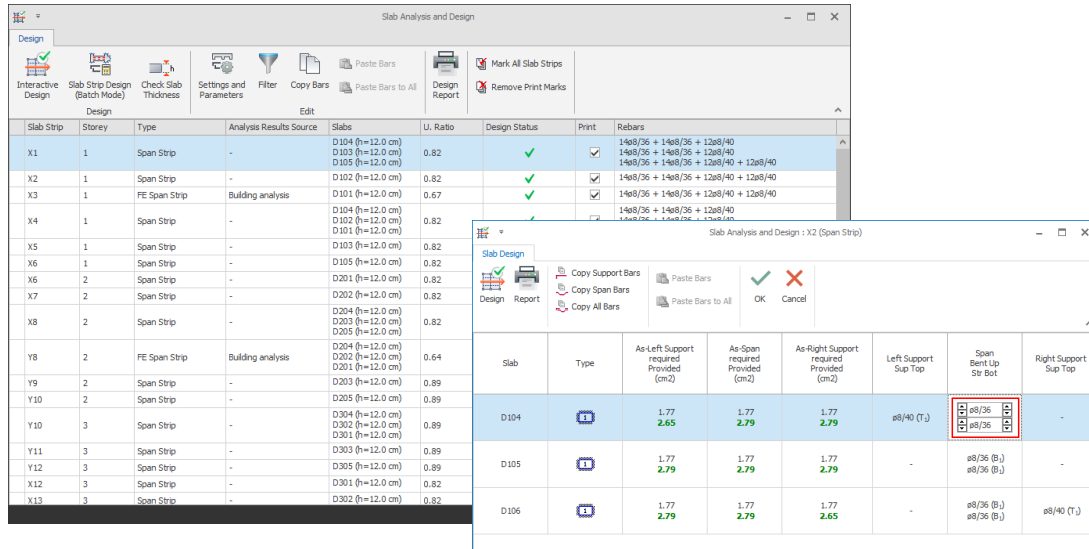
Since now it is possible to add an infinite number of rebar layers, we had to modify the Beam Design interface. The **Minimum Rebar Area** and **Spacing** indicators are moved to the top to allow the rebar groups to expand downwards as necessary.



New Slab Design User Interface

We have redeveloped the slab design user interface to make it more user-friendly and productive, enabling you to copy/paste the bars and see the design details in a drill-down approach.

- The existing “Slab Analysis and Design” button opens up the new design interface.
- To edit the rebars, click on them and use the in-place editor similar to Beam Rebar Editor
- Note that the right support bar of the first slab is the left support bar of the second one. Therefore, they are linked and edited together
- You can copy the support and span bars selectively between the slabs of the same strip



Slab Strip	Storey	Type	Analysis Results Source	Slabs	U. Ratio	Design Status	Print	Rebars
X1	1	Span Strip	-	D104 (h=12.0 cm) D103 (h=12.0 cm) D105 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 14x8/36 + 14x8/36 + 12x8/40 14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X2	1	Span Strip	-	D102 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X3	1	FE Span Strip	Building analysis	D101 (h=12.0 cm)	0.67	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X4	1	Span Strip	-	D104 (h=12.0 cm) D102 (h=12.0 cm) D101 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X5	1	Span Strip	-	D103 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X6	1	Span Strip	-	D105 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X6	2	Span Strip	-	D201 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X7	2	Span Strip	-	D202 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X8	2	Span Strip	-	D204 (h=12.0 cm) D203 (h=12.0 cm) D205 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y8	2	FE Span Strip	Building analysis	D204 (h=12.0 cm) D202 (h=12.0 cm) D203 (h=12.0 cm)	0.64	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y9	2	Span Strip	-	D203 (h=12.0 cm)	0.89	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y10	2	Span Strip	-	D205 (h=12.0 cm)	0.89	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y10	3	Span Strip	-	D304 (h=12.0 cm) D302 (h=12.0 cm) D301 (h=12.0 cm)	0.89	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y11	3	Span Strip	-	D303 (h=12.0 cm)	0.89	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
Y12	3	Span Strip	-	D305 (h=12.0 cm)	0.89	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X12	3	Span Strip	-	D301 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40
X13	3	Span Strip	-	D302 (h=12.0 cm)	0.82	✓	✓	14x8/36 + 14x8/36 + 12x8/40 + 12x8/40

Slab	Type	As-Left Support required Provided (cm2)	As-Span required Provided (cm2)	As-Right Support required Provided (cm2)	Left Support Sup Top	Span Bent Up Str Bot	Right Support Sup Top
D104	Span Strip	1.77 2.65	1.77 2.79	1.77 2.79	ø8/40 (T.)	ø8/36 (B.) ø8/36 (B.)	-
D105	Span Strip	1.77 2.79	1.77 2.79	1.77 2.79	-	ø8/36 (B.) ø8/36 (B.)	-
D106	Span Strip	1.77 2.79	1.77 2.79	1.77 2.65	-	ø8/36 (B.) ø8/36 (B.)	ø8/40 (T.)

Check Design for User-Defined Column Links

ProtaStructure checks whether the user-defined links are sufficient according to the selected concrete design code. By default, ProtaStructure automatically calculates the diameter and spacing of the column links depending on the user settings. However, you may want to do optimizations and customizations in your design. To enter and check user-defined links:

- Go to the “Shear Design” tab on the column design dialog.
- To change the automatically calculated values, check the “Edited” checkbox and enter new values for diameter and spacing,
- ProtaStructure will check the provided links and state the design status on the screen.
- If you click OK and exit the column design dialog, the design status of the column will be set to “Fail” if the links are not sufficient.
- Also, design status will be set to “Fail” if a batch column design is performed with the “Check existing reinforcement (Do not reselect bars) option.

b1 / b2:

50.0 cm

30.0 cm

e1 / e2:

0.0 cm

0.0 cm

L1 / L2:

360.0 cm

360.0 cm

Concrete Cover:

2.5 cm

Update

Loading: ☐ User Defined

Select Marked Combinations as User Defined

Steel Bars

Links

Shear Design

Slenderness

Settings

Shear Design

Dir:1

Dir:2

Vd*D

5.347

4.522

Vc

0.000

0.000

Ve

10.189

5.724

Link

Span:

a8

/

b0 cm

Support:

a8

/

20 cm

Edited ☒

No. of Link Legs -1:

3

Calculate

-2:

4

C25 / S420

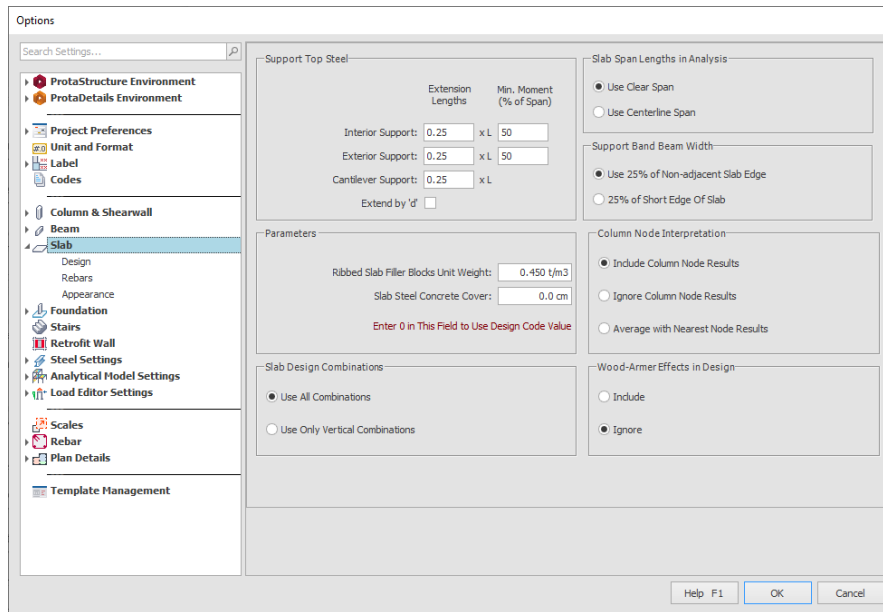


No	N (t)	M11 (t.m)	M22 (t.m)	V1 (t)	V2 (t)	Label
1 -Top	9.042	0.22	3.20	1.274	0.091	G+Q
-Bottom	10.932	-0.10	-1.39	1.274	0.091	
2 -Top	9.042	0.22	3.20	1.274	0.091	G+Qs1
-Bottom	10.932	-0.10	-1.39	1.274	0.091	
3 -Top	9.042	0.22	3.20	1.274	0.091	G+Qs2
-Bottom	10.932	-0.10	-1.39	1.274	0.091	
4 -Top	0.148	-1.46	-1.62	-4.991	-2.771	Gc+Qc+Ez+Ex+
-Bottom	1.762	1.81	3.98	-4.991	-2.771	
5 -Top	2.766	1.56	2.51	5.320	2.817	Gc+Qc+Ez+Ex+
-Bottom	4.381	-1.87	-4.27	5.320	2.817	
6 -Top	0.755	-0.87	-0.65	-2.571	-1.683	Gc+Qc+Ez+Ex-
-Bottom	2.370	1.09	2.04	-2.571	-1.683	
7 -Top	2.158	0.97	1.54	2.900	1.729	Gc+Qc+Ez+Ex-
-Bottom	3.773	-1.16	-2.33	2.900	1.729	
8 -Top	1.108	-2.08	0.15	-0.555	-3.920	Gc+Qc+Ez+Ey+
-Bottom	2.722	2.57	0.43	-0.555	-3.920	
9 -Top	1.806	2.18	0.74	0.884	3.966	Gc+Qc+Ez+Ey+
-Bottom	3.421	-2.64	-0.72	0.884	3.966	
10 -Top	0.797	-2.38	-0.34	-1.792	-4.476	Gc+Qc+Ez+Ey-

New Settings for Slab Design

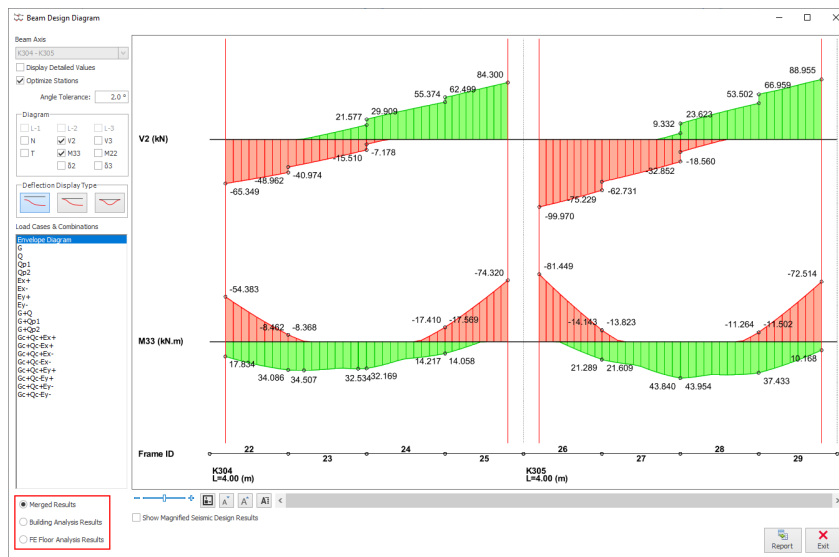
Column Node Interpretation Options, Inclusion of Wood-Armer Effects, and Slab Design Combination Options can now be accessed via “Slab Design Settings.”

Previously these were located in the FE Floor Analysis menu.



Enhanced Design Diagrams

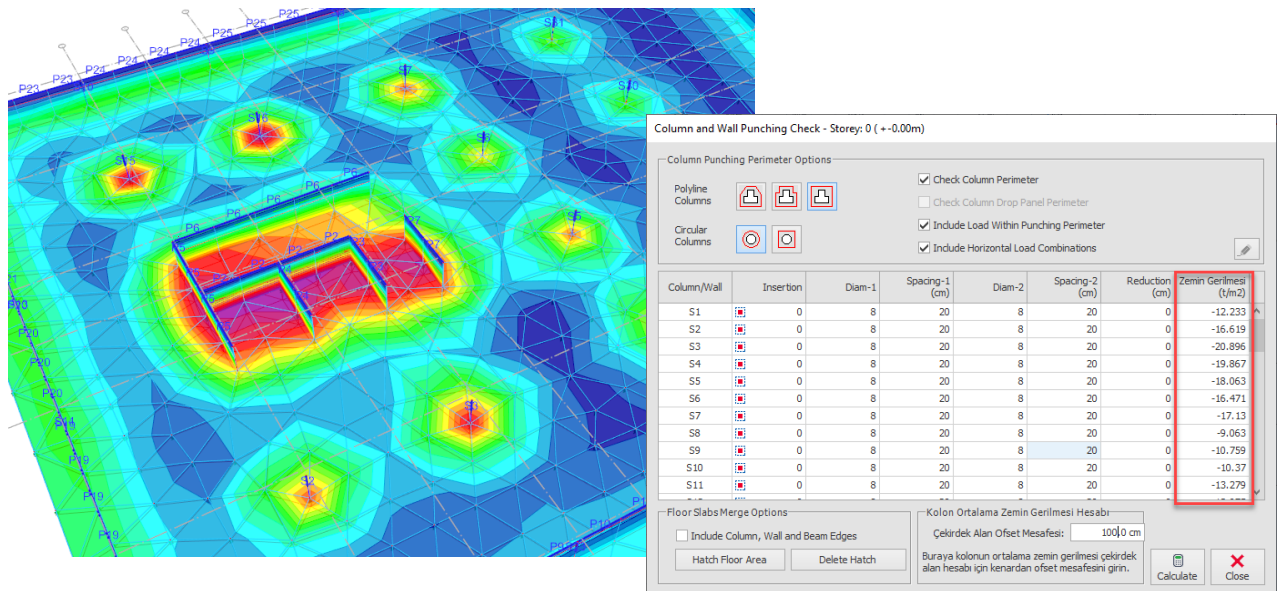
Column, beam, and frame member “**Design Diagrams**” now have an option to display analysis results from “**Building Analysis**”, “**FE Floor Analysis**”, or “**Merged**” results of these two. In the “Merged” mode, vertical load case results are used from “FE Floor Analysis” while the rest of the load case results are collected from the “Building Analysis”. These options are only displayed if an “FE” analysis is performed.



Mat Foundation Punching: Soil Pressure Under Each Column

In the previous versions of ProtaStructure, an overall average soil pressure was used in the mat foundation punching checks. With the new version, average soil pressure is calculated inside the tributary area of each column.

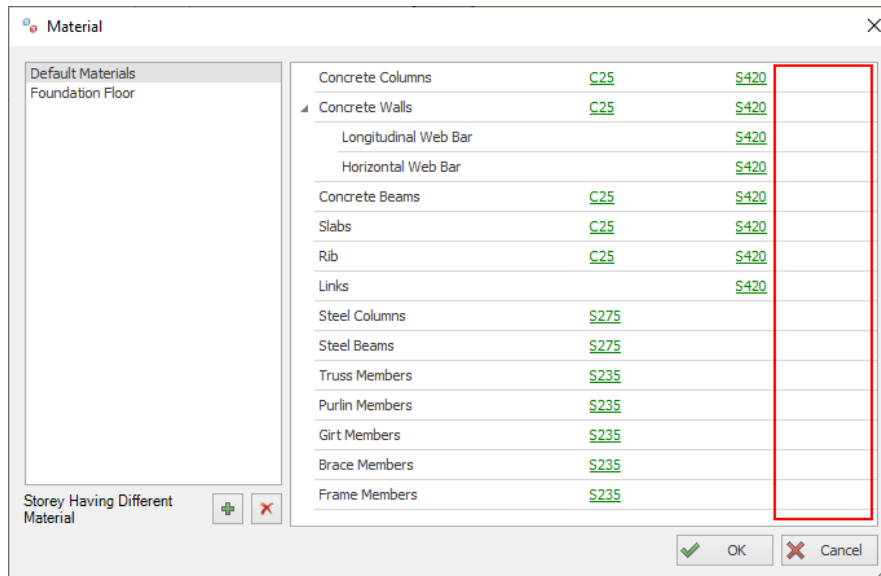
You can specify the dimensions of the tributary area. In this way, punching checks become more accurate.



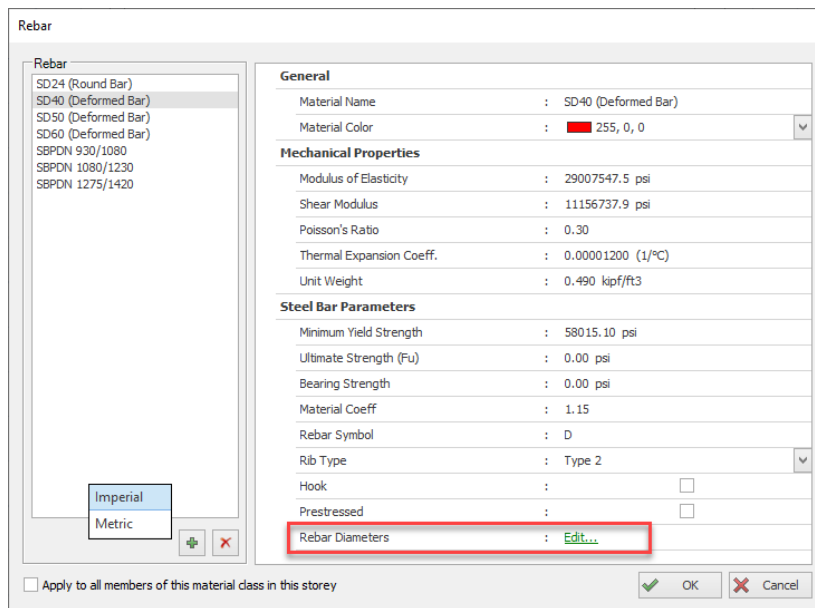
New UI for Rebar Diameter Availability

As a result of the introduction of imperial units and imperial rebar grades, a new user interface and workflow were needed for specifying rebar diameter availability.

First of all, you will not see the **Diameters** column on the **Material** dialog anymore.



Rebar diameter selection can be made using the “**Edit...**” button on the **Rebar Material Properties** dialog.



S420 Rebar Table

☐ Column
☐ Wall
☐ Beam
☐ Slab
☐ Rib Slab
☐ Footing
☐ Links
☐ Longitudinal Web Bar
☐ Horizontal Web Bar
☐ Generic

Label	Diameter (mm)	Area (mm2)																
8	8.0	50.266	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
10	10.0	78.54	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
12	12.0	113.097	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
14	14.0	153.938	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
16	16.0	201.062	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
18	18.0	254.469	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
20	20.0	314.159	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
22	22.0	380.133	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
24	24.0	452.389	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
25	25.0	490.874	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
26	26.0	530.929	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
28	28.0	615.752	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
30	30.0	706.858	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
32	32.0	804.248	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
34	34.0	907.92	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
36	36.0	1017.876	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
38	38.0	1134.115	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
40	40.0	1256.637	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
42	42.0	1385.442	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
44	44.0	1520.531	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
46	46.0	1661.903	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
48	48.0	1809.557	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	

Unit: Metric

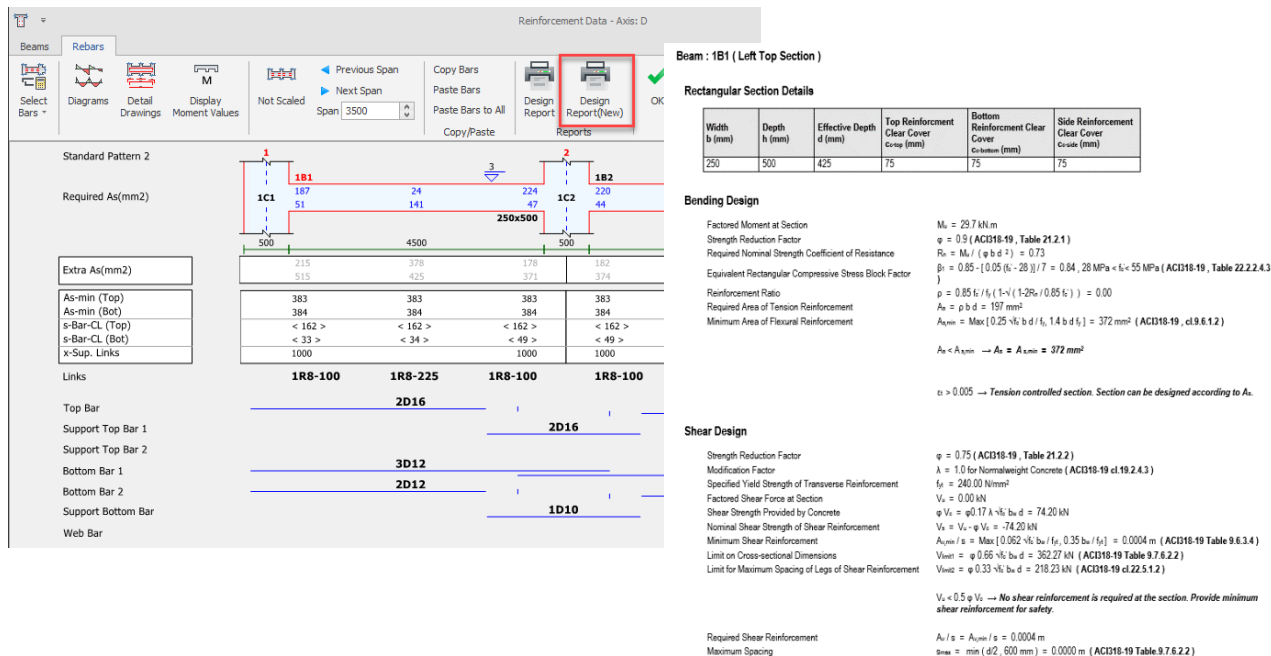
Design Reports with Step-by-Step Calculations

Providing transparent design documentation to our users is the number one priority. We have been investing a lot of time to fulfill this.

This version provides step-by-step design documentation for RC Beams, Pile Caps, and Pad Footings, including code references.

Detailed RC Beam Design Reports

Design reports for RC beams now include more details, including formulas, step-by-step calculations, and code references



Reinforcement Data - Axis: D

Width b (mm)	Depth h (mm)	Effective Depth d (mm)	Top Reinforcement Clear Cover c _{top} (mm)	Bottom Reinforcement Clear Cover c _{bottom} (mm)	Side Reinforcement Clear Cover c _{side} (mm)
250	500	425	75	75	75

Bending Design

Factored Moment at Section $M_u = 29.7 \text{ kNm}$
Strength Reduction Factor $\phi = 0.9$ (ACI318-19, Table 21.2.1)
Required Nominal Strength Coefficient of Resistance $R_n = M_u / (\phi b d^2) = 0.73$
Equivalent Rectangular Compressive Stress Block Factor $\beta_1 = 0.85 - [0.05 (\epsilon_c - 28) / 7] = 0.84$, 28 MPa $\leq \epsilon_c \leq 55 \text{ MPa}$ (ACI318-19, Table 22.2.4.3)
Reinforcement Ratio $\rho = 0.85 f_c' / f_y (1 - \sqrt{1 - (R_n / 0.85 f_c')}) = 0.00$
Required Area of Tension Reinforcement $A_{s,req} = \phi b d = 197 \text{ mm}^2$
Minimum Area of Flexural Reinforcement $A_{s,min} = \text{Max}[0.25 \rho_b b d / f_y, 1.4 b d / f_y] = 372 \text{ mm}^2$ (ACI318-19, cl.9.6.1.2)
 $A_s < A_{s,min} \rightarrow A_s = A_{s,min} = 372 \text{ mm}^2$

$\epsilon_t > 0.005 \rightarrow$ Tension controlled section. Section can be designed according to A_s .

Shear Design

Strength Reduction Factor $\phi = 0.75$ (ACI318-19, Table 21.2.2)
Modification Factor $\lambda = 1.0$ for Normalweight Concrete (ACI318-19 cl.19.2.4.3)
Specified Yield Strength of Transverse Reinforcement $f_{ty} = 340.00 \text{ N/mm}^2$
Factored Shear Force at Section $V_u = 0.00 \text{ kN}$
Shear Strength Provided by Concrete $V_c = V_u - \phi V_s = -74.20 \text{ kN}$
Nominal Shear Strength of Shear Reinforcement $V_{s,req} / s = \text{Max}[0.062 \sqrt{f_c'} b_w / f_y, 0.35 b_w / f_y] = 0.0004 \text{ m}$ (ACI318-19 Table 9.6.3.4)
Minimum Shear Reinforcement $V_{u,req} = \phi 0.66 \sqrt{f_c'} b_w d = 362.27 \text{ kN}$ (ACI318-19 Table 9.7.6.2.2)
Limit on Cross-sectional Dimensions $V_{u,lim} = \phi 0.33 \sqrt{f_c'} b_w d = 218.23 \text{ kN}$ (ACI318-19 cl.22.5.1.2)
 $V_u < 0.5 \phi V_c \rightarrow$ No shear reinforcement is required at the section. Provide minimum shear reinforcement for safety.
Required Shear Reinforcement $A_v / s = A_{s,req} / s = 0.0004 \text{ m}$
Maximum Spacing $s_{max} = \min(d/2, 600 \text{ mm}) = 0.0000 \text{ m}$ (ACI318-19 Table 9.7.6.2.2)

Detailed Pad Footing and Pile Cap Design Report

Design reports for pile caps and pad footings now include more details, including formulas, step-by-step calculations, and code references

$$\Sigma N = N + TW$$

$$\Sigma M_x = M_x + V_x (h - h_{taper}) + E \phi c_1 N$$

$$\Sigma M_y = M_y + V_y (h - h_{taper}) + E \phi c_2 N$$

Corner stresses,

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

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

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

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

Comb	ΣN (kN)	ΣM_x (kN.m)	ΣM_y (kN.m)	σ_1 (kN/m ²)	σ_2 (kN/m ²)	(k
Comb #1	397.55	4.23	0.38	207.51	227.41	2
Comb #2	397.55	4.23	0.38	207.51	227.41	2
Comb #3	397.55	4.23	0.38	207.51	227.41	2

Shear capacity is calculated according to EC-2,

$$(6.2.a) V_{rd1} = 0.12 k (100 \rho f_{ck})^{1/3} + (0.15 * 0.2 * f_{cd} * d_y * d = 144.12 \text{ kN}$$

$$(6.3.N) V_{rd2} = 0.035 (k^{1.5}) (f_{ck}^{0.5}) + (0.15 * 0.2 * f_{cd} * d_y * d = 220.84 \text{ kN}$$

$$V_{rd} = \text{Max}(V_{rd1}, V_{rd2}) = 220.84 \text{ kN}$$

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

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

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

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

New Slab Design Report

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

Slab Analysis and Design

Slab Strip : X1 -- Storey : 1

Materials : C25/S420

Slab/Type	d h cm	g q t/m ²	L ₁ L ₂ cm	C _{Support} M _{Support} t.m	C _{Span} M _{Span} t.m	M _{Left} M _{Right} t.m	A _s L-Required cm ²	A _s M-Required cm ²	A _s R-Required cm ²	Support _{Left} StrTop	Span BentUp StrBot	Support _{Right} StrTop
D101 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.23 0.61	1.77 2.65	1.77 2.79	1.77 10.65	ø8/40 (T _r)	ø8/36 (B _r) ø8/36 (B _r)	ø10/10 (T _r)
D102 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.61 0.61	1.77 10.65	1.77 2.79	1.77 2.79	ø10/10 (T _r)	ø8/36 (B _r) ø8/36 (B _r)	-
D103 1	10.1 12.0	0.589 0.000	475.00 475.00	0.03 0.61	0.03 0.46	0.61 0.23	1.77 2.79	1.77 2.79	1.77 2.65	-	ø8/36 (B _r) ø8/36 (B _r)	ø8/40 (T _r)

Slab Strip : X2 -- Storey : 1

Materials : C25/S420

Slab/Type	d h cm	g q t/m ²	L ₁ L ₂ cm	C _{Support} M _{Support} t.m	C _{Span} M _{Span} t.m	M _{Left} M _{Right} t.m	A _s L-Required cm ²	A _s M-Required cm ²	A _s R-Required cm ²	Support _{Left} StrTop	Span BentUp StrBot	Support _{Right} StrTop
-----------	--------------	----------------------------	--	---	---	--	---	---	---	-----------------------------------	--------------------------	------------------------------------

Comprehensive FE Mat Foundation Design Report

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

Finite Element Foundation Report

Please Select Load Cases and Combinations to be Reported

G
Gc
Q
Qc
Qp1
Qp2
Ez
Ex+
Ex-
Ey+
Ey-
Px
Py

1.4G + 1.6Q
1.4G + 1.6Qp1
1.4G + 1.6Qp2
1Gc + 1Qc + 0.3Ez + 1Ex + 0
1Gc + 1Qc + 0.3Ez - 1Ex + 0
1Gc + 1Qc + 0.3Ez + 1Ex - 0
1Gc + 1Qc + 0.3Ez - 1Ex - 0
1Gc + 1Qc + 0.3Ez + 0.3Ex - 0

Select/Deselect All

General

☒ Add Notification Summary To the Report
☒ Add Column/ShearWall Loads Transferred to Foundation

Analysis Results Source

☐ Building Analysis Results
☒ FE Floor Analysis Results

Contours

☒ Add Base pressure FE Contours
☒ Add Foundation Design Moments FE Contours
☒ Add Required Rebar Area FE Contours (Local)
☒ Add Required Rebar Area FE Contours (Wood-Armer)

FE Informations

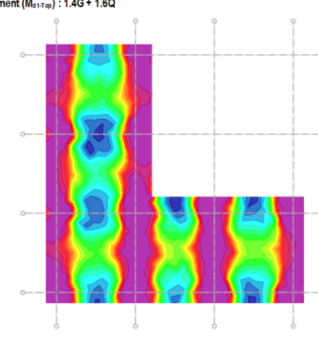
☒ Add Finite Elements Nodes Information

Mat Foundation Design

Prota Yacine A.S. (19813)

Calc. By
Checked By

Direction 1 Top Moment (M_{dir1}) : 1.4G + 1.6Q



M1d-Top
(t.m/m)

2.37
2.07
1.78
1.48
1.18
0.89
0.59
0.3
0

Direction 2 Top Moment (M_{dir2}) : 1.4G + 1.6Q

ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®

Minimum Slab Thickness Checks

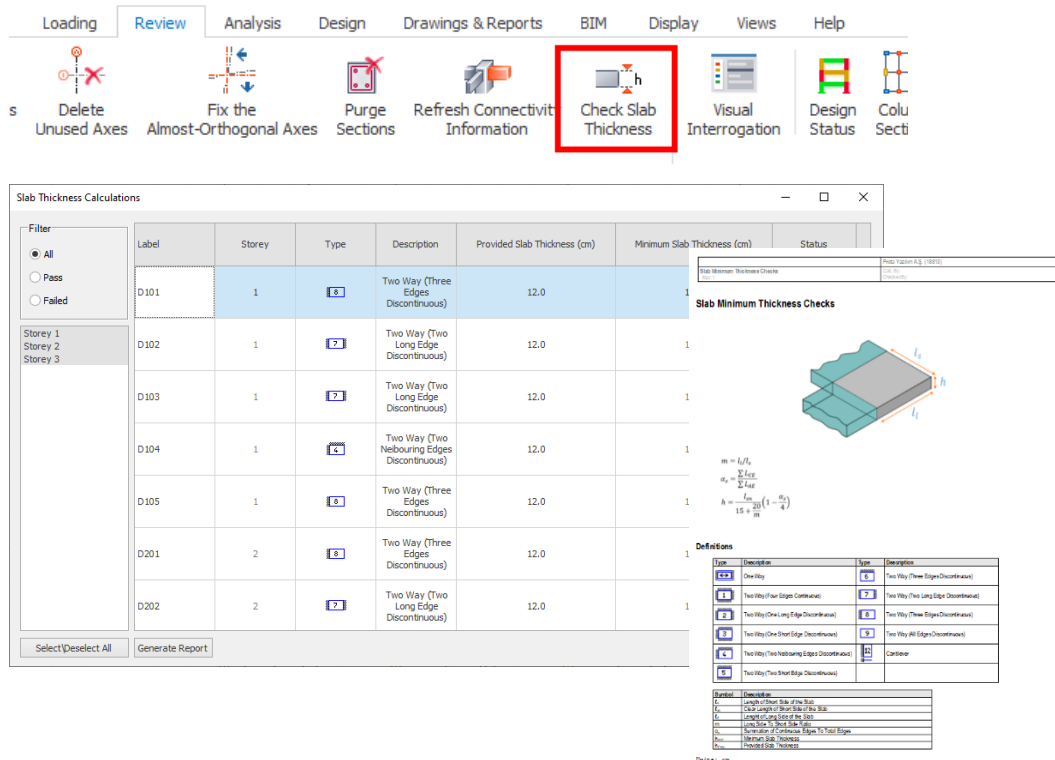
In ProtaStructure, slabs can mainly be designed in two different ways:

1. Code-based Coefficient Method
2. Finite Element Method

Slab types are essential in approximate moment calculation in code-based coefficient methods. Slabs are classified according to their continuous and discontinuous (or free) edges.

Codes also dictate minimum slab thickness values used in the design. ProtaStructure 2022 introduced a new report that summarizes the slab types and minimum thickness requirements to TS codes. US and Eurocode implementations will follow.

You can access this report by clicking the “**Check Slab Thickness**” button on the **Review** or **Drawings&Reports** ribbon tab.



Slab Thickness Calculations

Filter	Label	Storey	Type	Description	Provided Slab Thickness (cm)	Minimum Slab Thickness (cm)	Status
All	D101	1	8	Two Way (Three Edges Discontinuous)	12.0	1	
	D102	1	7	Two Way (Two Long Edge Discontinuous)	12.0	1	
	D103	1	7	Two Way (Two Long Edge Discontinuous)	12.0	1	
	D104	1	6	Two Way (Two Neighbouring Edges Discontinuous)	12.0	1	
	D105	1	8	Two Way (Three Edges Discontinuous)	12.0	1	
	D201	2	8	Two Way (Three Edges Discontinuous)	12.0	1	
	D202	2	7	Two Way (Two Long Edge Discontinuous)	12.0	1	

Slab Minimum Thickness Checks

3D Diagram of a slab with dimensions: l_x , l_y , h .

Formulas:

$$m = l_x/l_y$$

$$m_x = \frac{m}{1 + m}$$

$$m_y = \frac{m}{1 + m}$$

Definitions:

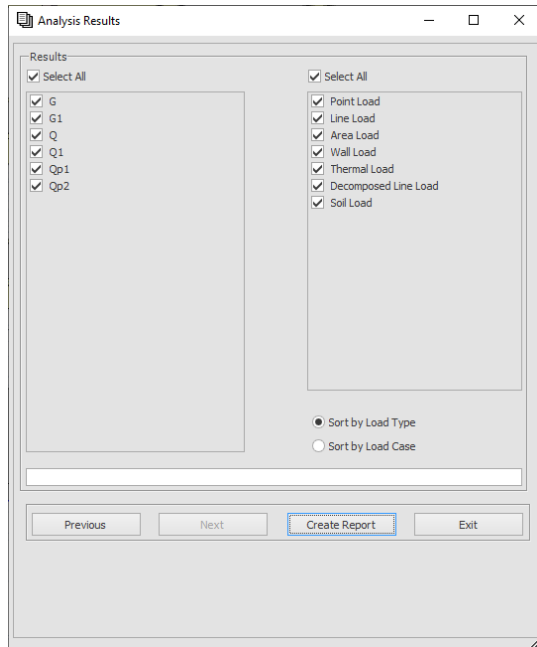
Type	Description	Type	Description
1	One Way	8	Two Way (Three Edges Discontinuous)
2	Two Way (Four Edges Continuous)	7	Two Way (Two Long Edge Discontinuous)
3	Two Way (One Long Edge Discontinuous)	6	Two Way (Three Edges Discontinuous)
4	Two Way (One Short Edge Discontinuous)	5	Two Way (Two Edges Discontinuous)
5	Two Way (Two Neighbouring Edges Discontinuous)	4	Continuous
6	Two Way (Two Short Edge Discontinuous)		

Legend:

- l_x : Length of Short Side of the Slab
- l_y : Length of Long Side of the Slab
- h : Thickness of the Slab
- m : Ratio of Long Side to Short Side
- m_x : Ratio of Long Side to Short Side (X-Direction)
- m_y : Ratio of Long Side to Short Side (Y-Direction)
- h_{min} : Minimum Slab Thickness

Member Loads Report

A detailed report is created that summarizes all the load information in the model. You can generate the report with filtering members and load cases.



Member Loads

S1 (50.0 / 30.0 cm) L: 300.00 cm

Label	Description	Load Type	Load Case	Direction	Position (cm)	Magnitude (t)
Point Load 145		Point Load	G1	Global X	200.00	50.00

S2 (50.0 / 30.0 cm) L: 300.00 cm

Label	Description	Load Type	Load Case	Direction	Magnitude (t/m)
Line Load		Line Load	G1	Global X	10.00

K101 (25.0 / 50.0 cm L: 500.00 cm)

Label	Description	Load Type	Load Case	Direction	Magnitude (t/m)
Self Weight	Self Weight	Line Load	G	Global -Z	3.13

Label	Description	Decomposed By	Load Type	Load Case	Direction	Point	Position (cm)	Magnitude (t/m)
-------	-------------	---------------	-----------	-----------	-----------	-------	---------------	-----------------

Whole structure is considered as a single integrated 3D model for the analysis. Therefore, reactions transferred by the secondary beams are considered naturally to the main beams.

K102 (25.0 / 50.0 cm L: 500.00 cm)

Label	Description	Load Type	Load Case	Direction	Magnitude (t/m)
Self Weight	Self Weight	Line Load	G	Global -Z	3.13

Label	Description	Decomposed By	Load Type	Load Case	Direction	Point	Position (cm)	Magnitude (t/m)
-------	-------------	---------------	-----------	-----------	-----------	-------	---------------	-----------------

Whole structure is considered as a single integrated 3D model for the analysis. Therefore, reactions transferred by the secondary beams are considered naturally to the main beams.

K103 (25.0 / 50.0 cm L: 500.00 cm)

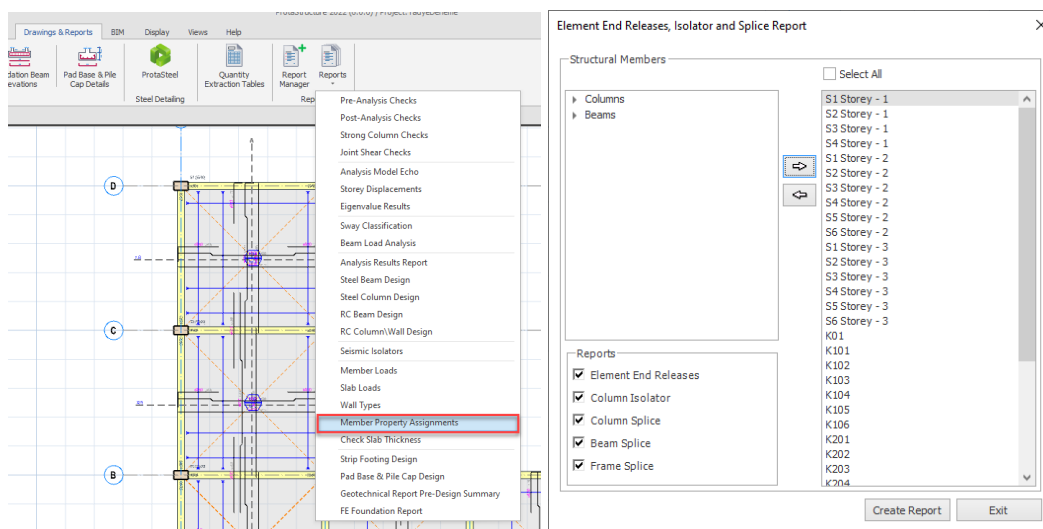
Label	Description	Load Type	Load Case	Direction	Magnitude (t/m)
Self Weight	Self Weight	Line Load	G	Global -Z	3.13

Label	Description	Decomposed By	Load Type	Load Case	Direction	Point	Position (cm)	Magnitude (t/m)
-------	-------------	---------------	-----------	-----------	-----------	-------	---------------	-----------------

Member Properties Report

A new report now summarizes column/beam/frame end release conditions, support assignments, splice locations, and isolator assignments.

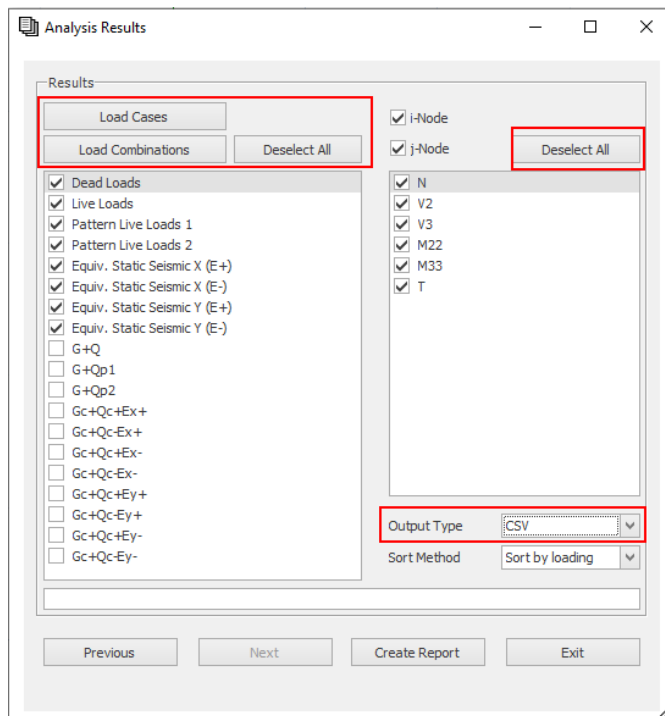
You can access this report by clicking the “Member Properties” button on the Review or Drawings&Reports ribbon tab.



Enhanced Analysis Results Report

The results of “Building Analysis” and “FE Floor and Foundation Analysis” are obtained using the same interface. Go to the “FE analysis Dialog > Post-Analysis Processes and Reports” tab for FE Floor analysis results. For building analysis results, you can use the **“Analysis Results Report”** button under **“Drawings & Reports > Reports.”**

In addition to enhancements for a better user experience, many bug-fix and optimizations have been made on this function. For example, the analysis results can be written in **“CSV”** format.



New Analysis Framework

Analysis, results collection, and review systematics are significantly enhanced in ProtaStructure 2022. These enhancements pay back as an enormous advantage to our users in managing big models, FE mesh creation, practicality, performance, and visualization.

Integrated 64-bit FE Slab and Foundation Module

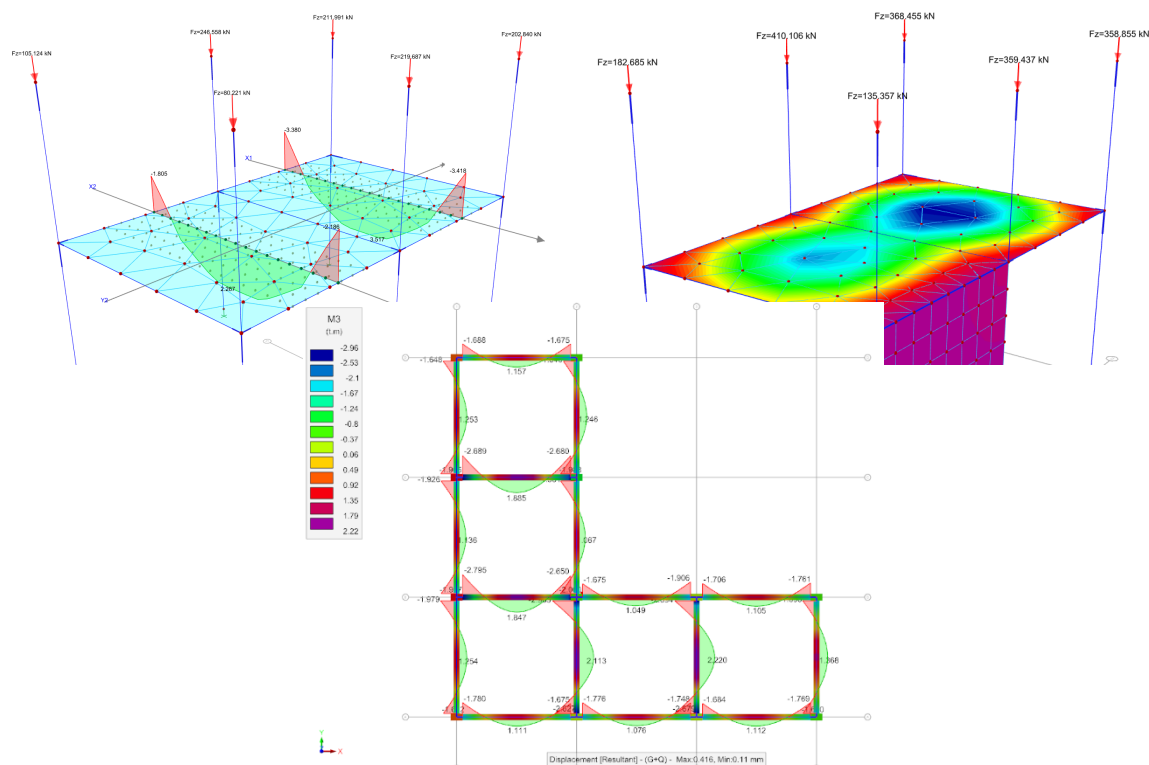
The technology migration we started with ProtaStructure 2021 is now completed with ProtaStructure 2022 by integrating the **FE Floor and Foundation module** into the main program.

Single Data Representation and Architecture

In ProtaStructure 2022, you will see essential outcomes of our platform overhaul. Under the hood, all the analysis data is unified and brought to a common platform. In this way, post-processing and reviewing the results is much easier and more scalable.

Improved Visualization

We have used a more capable and performant 3D renderer while integrating the FE floor and foundation module, resulting in better FE Contour rendering, deflected shape, load, and diagram visualization.

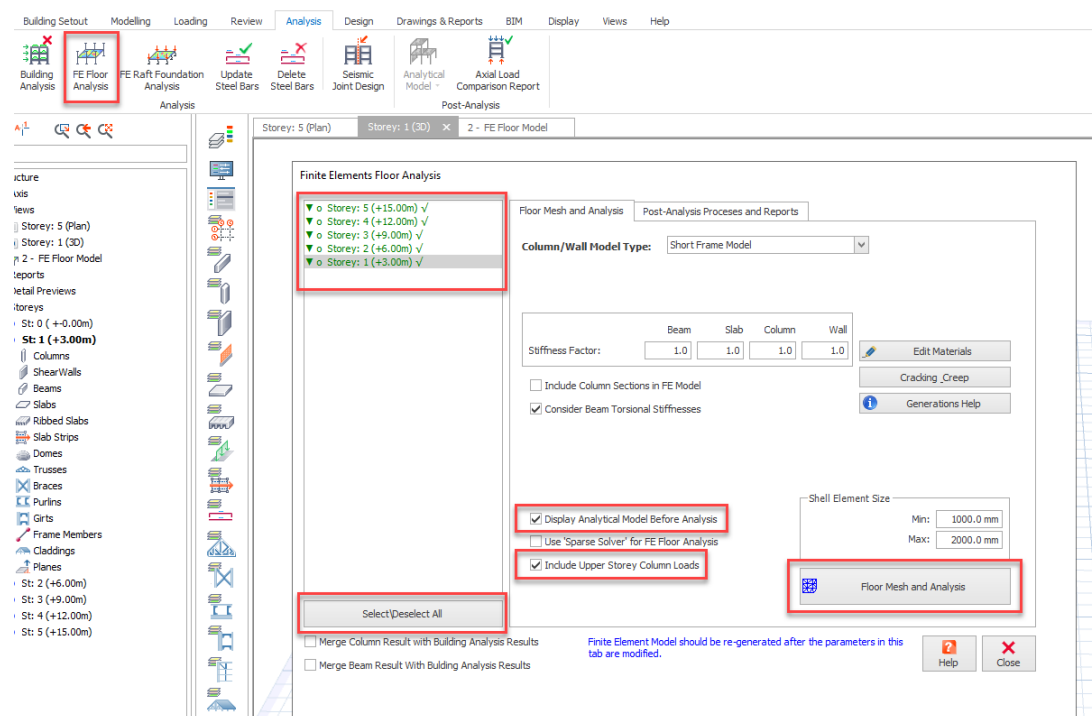


New FE Floor Analysis Manager

ProtaStructure can analyze and design floors under gravity and imposed loads independently from the global 3D building analysis. In this approach, the floor is isolated from the building, and only gravity and imposed loads are applied. This methodology allows engineers to simulate design scenarios where vertical loads govern.

In addition to this, you can conduct a “**FE Chase-down**” analysis, where you analyze the floors (under vertical loads only) top to bottom, iteratively applying the calculated column reactions onto the floor below. This methodology is preferred by engineers, who would like to eliminate the artificial side effects of a global 3D FE Analysis such as the 3D frame effects and differential settlements due to relative axial rigidities. This method simulates the good-old 2D hand calculation methods in a 3D environment.

Having explained this, in ProtaStructure 2022, FE Floor analysis workflow is improved and collected under a single FE Floor Analysis Manager, where you can see and control the parameters for all stories at once. Also, you can perform batch analyses on the selected floors.

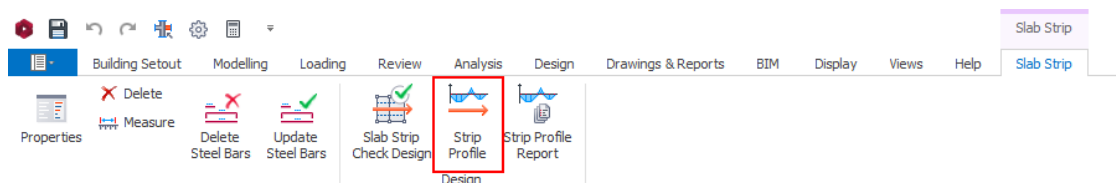


1. FE Floor and Foundation Analysis models are now integrated into the main program. No more reference to the external Meshgen module.
2. Select multiple stories from the left-hand-side list, then click “**Floor Mesh and Analysis**”. All selected floors will be meshed and analyzed at once.
3. If you check “**Include Upper Storey Column Loads**”, a chase-down analysis will be performed starting from the top-most selected story. In this case, a “▼” symbol is displayed next to those stories as a reminder. The same terminology is used in other analysis-related interfaces.

Slab Strip Diagrams

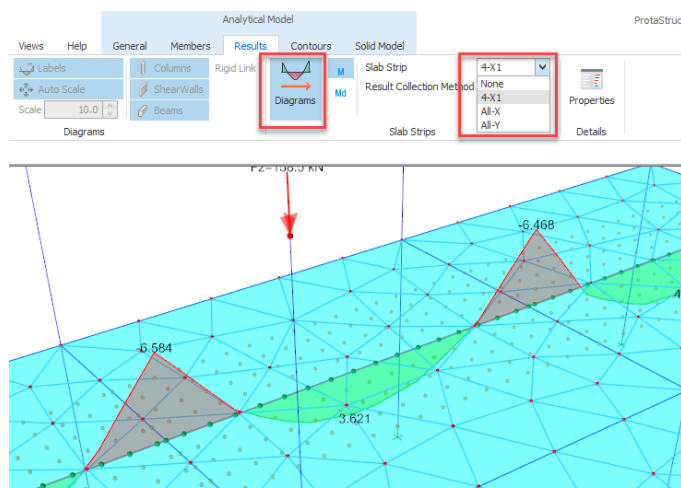
Strip diagrams can be viewed for both Building and FE Floor Analysis Results if both types of analysis have been performed. Slab Strip profiles can be displayed like the beam, column, and frame member diagrams. To review the slab strip diagrams on the graphical editor:

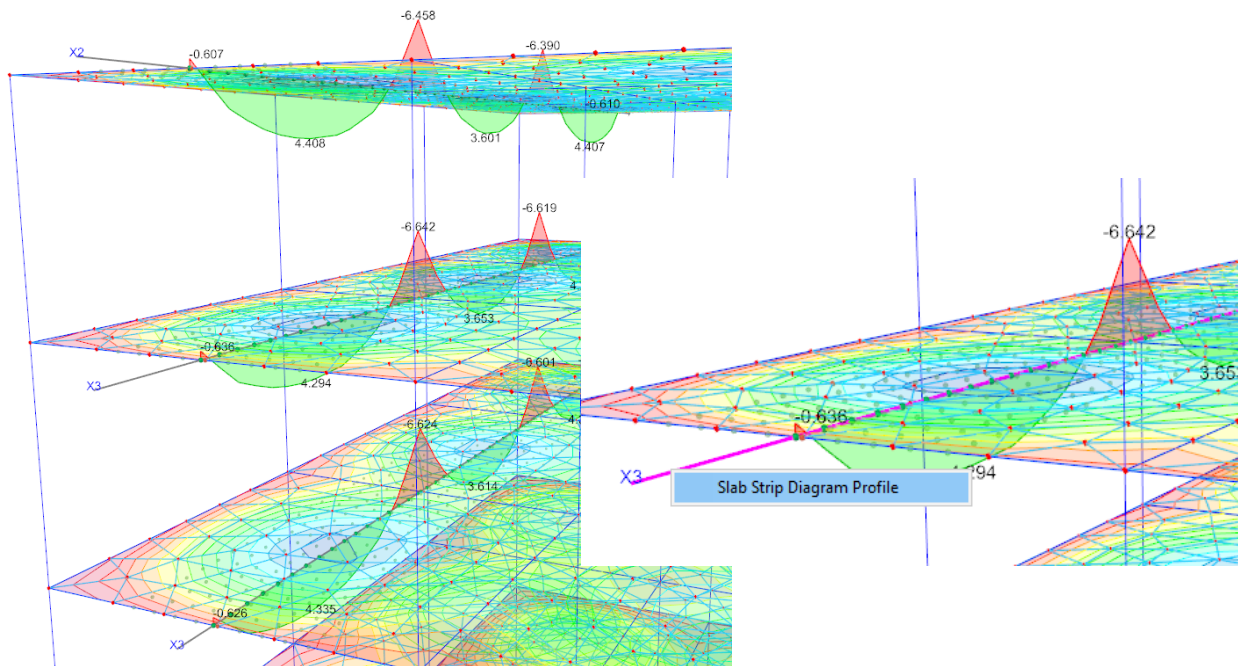
1. Select the slab strip on the model
2. Right-click and run the **“Strip Profile”** command.
3. Alternatively, you can use the same command on the slab strip contextual ribbon tab that appears when the strip is selected.



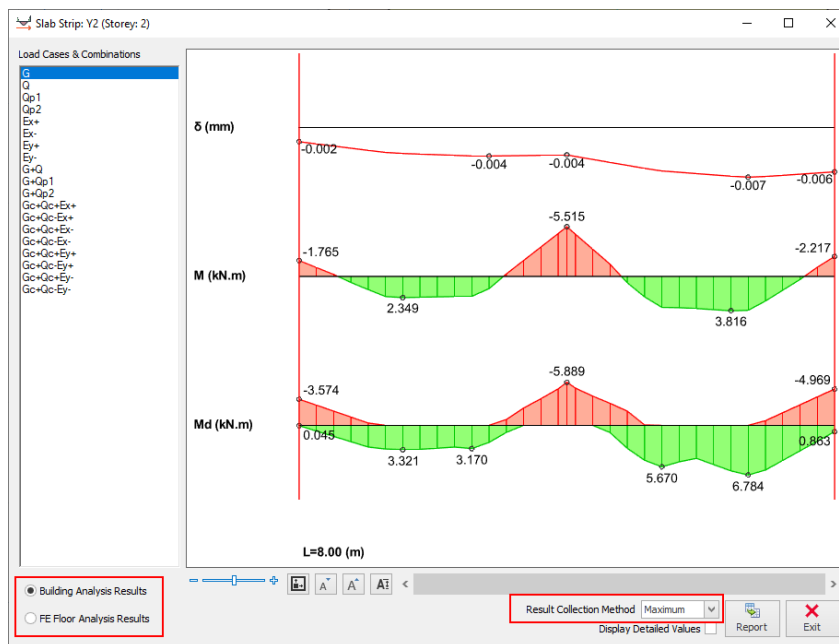
To review the slab strip profiles in analysis post processor:

1. Open the **analytical model** for FE Floor Analysis or Building Analysis,
2. Select the slab strip you want to check from the **Slab Strip list** on the **Results** tab, or alternatively select the strip on the screen
3. The analytical view will display Moment diagrams of selected strips.
4. To see the detailed profile, select a slab strip on the analytical model, right-click and choose **“Slab Strip Diagram Profile.”**





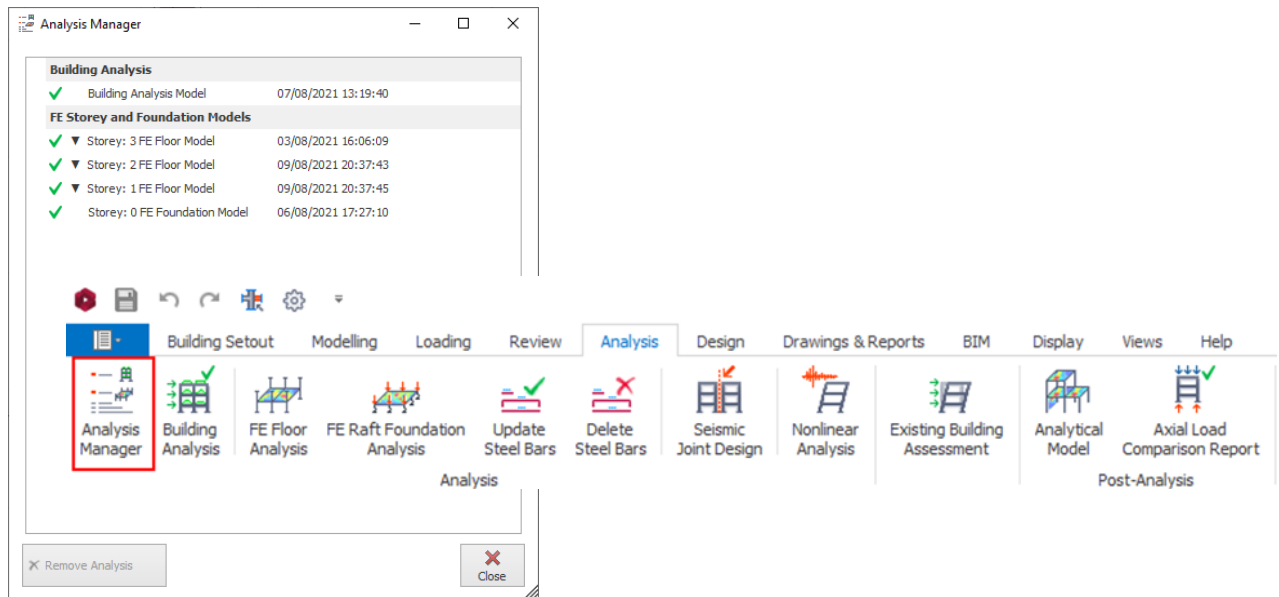
We offer three different result collection methods: **Maximum**, **Integral**, and **Strip Line (Longitudinal Stations)**. In the '**Maximum**' approach, maximum values of the transverse nodes at each station are used in strip result calculation. As its name implies, the '**Integral**' method calculates the integral of transverse node results yielding more economical and smoothened values. The integral approach is only available for "Fixed Band" strips and can not be used for "Span Strip" types. The '**Strip Line**' is solely for checking purposes and does not account for transverse nodes displaying the values precisely at the station nodes.



Manage Multiple Analysis At the Same Time

Building analysis, **FE Floor analysis** on floor sub-structures, and **FE Mat Foundation Analysis** on the foundation can now be managed on a single **Analysis Manager** dialog thanks to the new and flexible analysis infrastructure. For example, FE floor/foundation analysis and building analysis results can be kept and managed together in the same project. This way, results can be easily switched and reviewed on the fly.

Users can interrogate the analysis status of all performed analyses using the “Analysis Manager” under the Analysis ribbon. Any selected analysis can also be deleted using this menu.



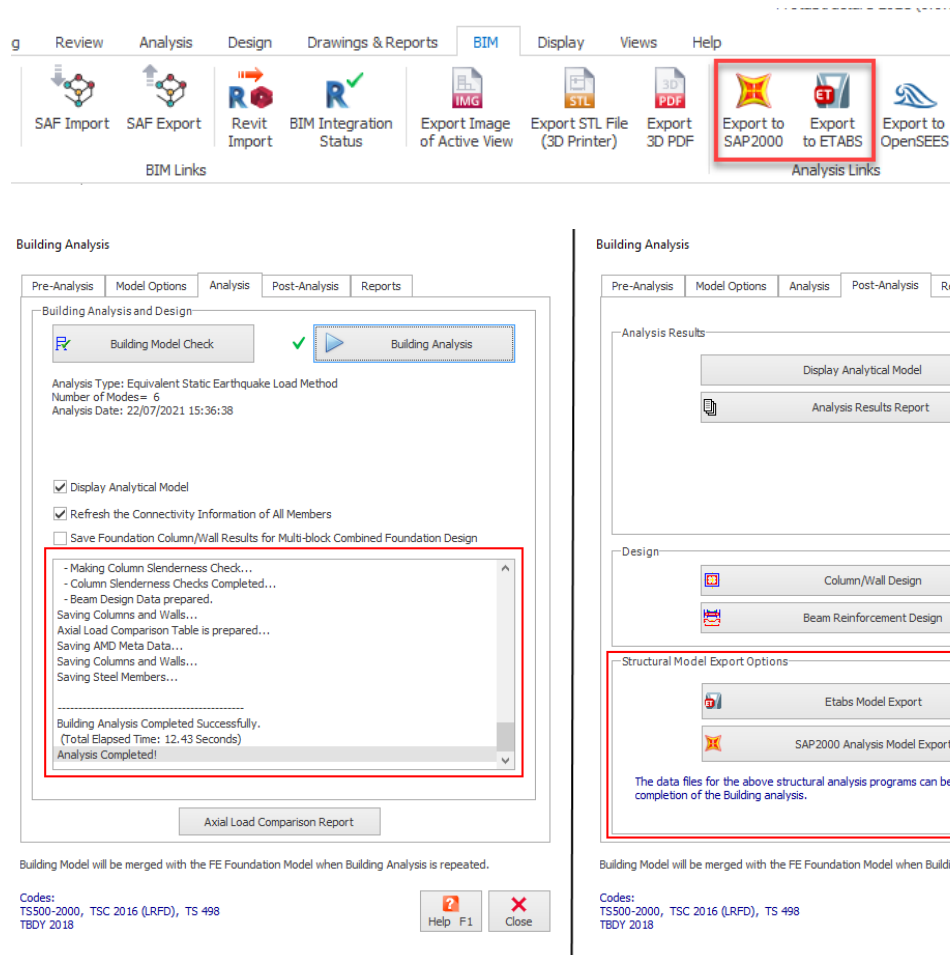
Soil-Structure Interaction in a Single Run

In the previous versions of ProtaStructure, soil-structure interaction analysis (building on elastic foundation) required an iterative process that required the building analysis and foundation analysis done separately in the first step. The building analysis had to be done a third time using the “**Merged Foundation Model**” option.

In ProtaStructure 2022, this is not the case. Once you create the physical model of the foundation, it is sufficient to start the analysis with the “**Merged Foundation Model**” option. No iterative process is needed anymore.

Analytical Model Export Options

We have changed the “Building Analysis” user interface slightly. You will not be able to find the “Export” tab anymore. **Etabs** and **SAP2000** export options are now moved to the “Post-Analysis” and “BIM” tabs.



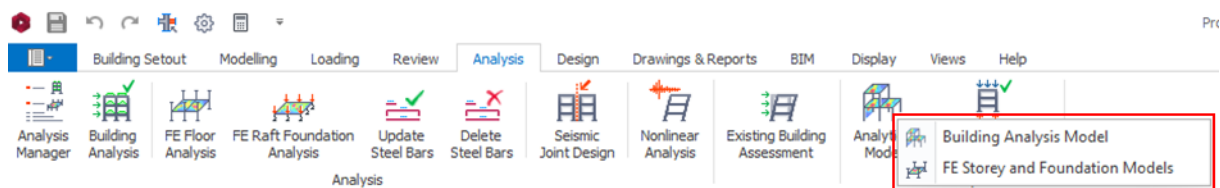
In addition to the above, improvements in the workflow and the messaging pipeline are made in the analysis process. The analysis messages now echo on the **Analysis** tab rather than opening separate progress.

A Single Integrated Post Processor for All Analysis Results

Many enhancements have been applied to APP to cater to the changes in the analysis systematics explained above.

A single analysis post-processor for FE Floor and Building Analysis modules is developed with ProtaStructure 2022. You will be able to check and review analytical models and results through a single interface. To view the analysis model:

1. Click the “**Analytical Model**” button on the **Analysis** ribbon tab to open the analysis post-processor.



2. If you only performed “**Building Analysis**,” clicking the “**Analytical Model**” button will open the analysis post-processor window.
3. However, if you have performed **FE floor or foundation analyses** as well, then clicking the “**Analytical Model**” button will prompt you with two options:
 - Building Analysis Model
 - FE Storey and Foundation Models

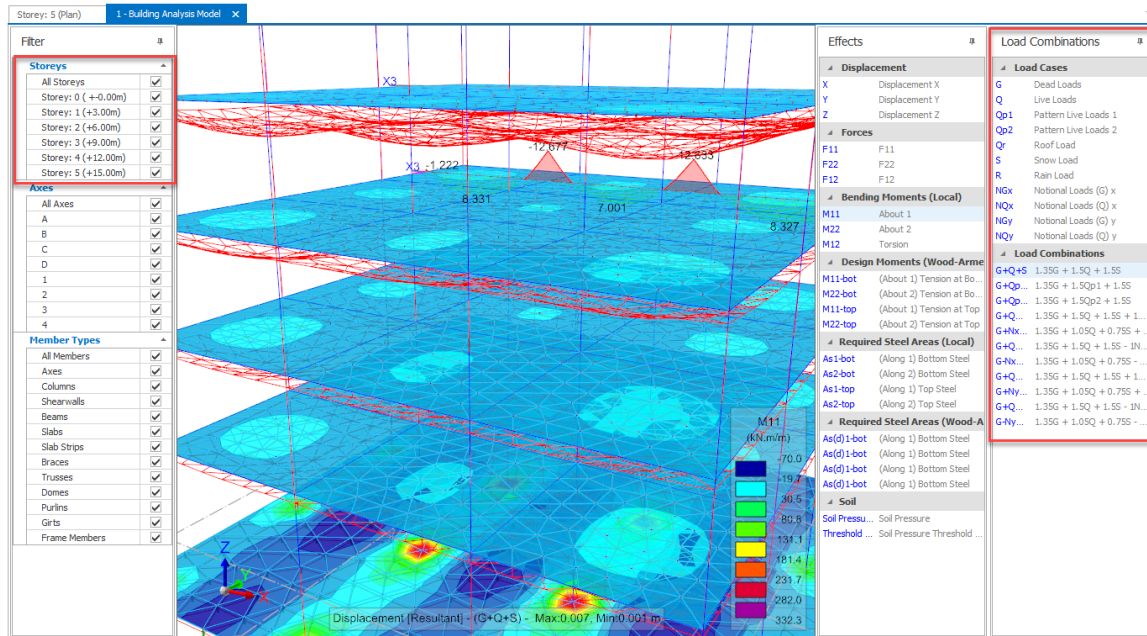
FE Floor Analysis vs. Building Analysis Post-Processing

There are slight differences between FE Floor post-processing mode and Building Analysis post-processing mode.

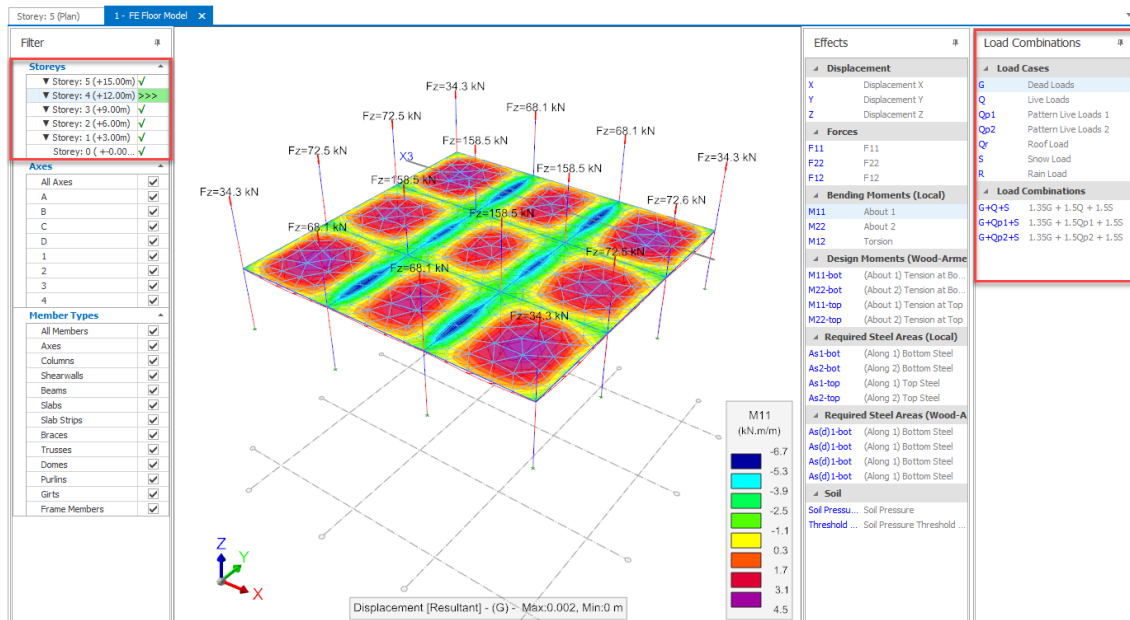
1. A sub-structured (isolated from the building) **FE Floor Analysis** is done under vertical loads only. That’s why you will see only the vertical load cases and combinations in **FE Floor post-processing mode**. On the contrary, a global building analysis considers all load combinations. In this case, the load combination list will list all load cases and combinations in **Building Analysis post-processing mode**.
2. The **Storeys** list acts as a “**Storey Selector**” in **FE Floor post-processing mode**. In other words, you can switch between the results of floors simply by clicking the relevant storey on the list. On the other hand, this lists acts merely as a filter to hide and show the analytical members of the selected level.
3. **How is it used?** In FE Floor post-processing mode, instead of showing/hiding storeys on the list, you can switch between FE models of the selected storeys. A right arrows symbol “>>>” indicates the currently selected floor. A check sign “✓” means that a valid FE Analysis exists

for that floor. A cross sign “X” means that an FE analysis has been performed, but it is not valid anymore. If an FE analysis has not been performed for a specific floor, it will still be listed; however, it will be disabled on the list.

- In both post-processor modes, you can investigate displacements, member and slab strip diagrams, shell contours for different effects, etc.



Building Analysis Post-Processing Mode: Pay attention to Storeys Filter and Combination List

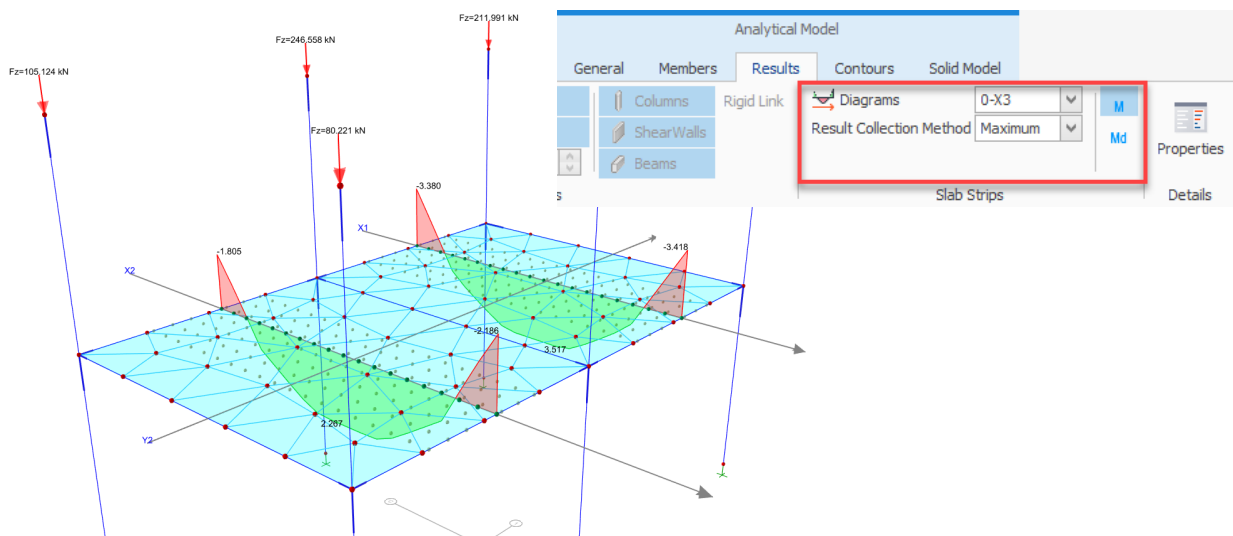


FE Floor Analysis Post-Processing Mode: Pay Attention to Storey Selector and Combination List

Slab Strip Diagrams

Slab strip diagrams can be displayed on Building Analysis post-processing mode and FE Floor post-processing mode. [See Slab Strip Diagrams](#) title for detailed information.

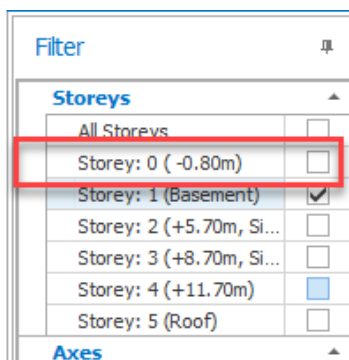
1. To view the diagrams, you need to select either a slab strip on the arena or the in the **Diagrams** list on the same ribbon group. The slab strip list enables you to quickly choose all slab strips in X and Y directions.
2. You can also right-click a strip and show the detailed profile diagram



Other Improvements in Analysis Post Processor

Separate Filter for Foundation Storey

The Foundation floor is now added as a separate filter in the building analysis post-processor.

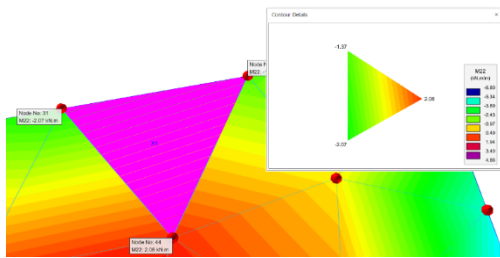


Unified FE Contouring Engine

In the previous versions of ProtaStructure, the FE contouring engine used for building analysis differed from the FE Floor analysis. The FE contouring engine is unified with additional improvements and optimizations thanks to our new infrastructure.

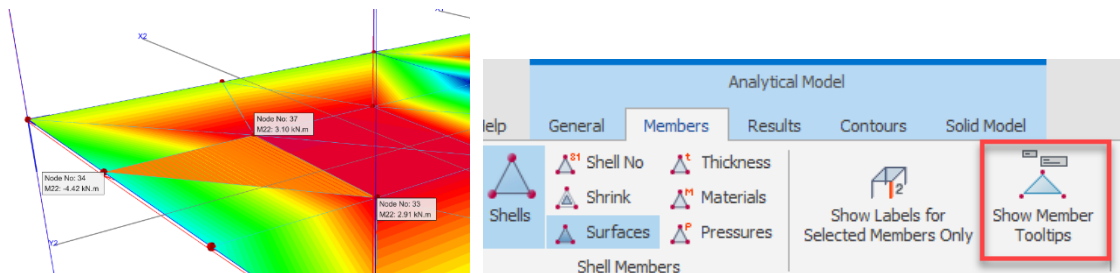
Contour Details Window

Contour details form is now modeless. You can change contour selection, and it updates instantly.



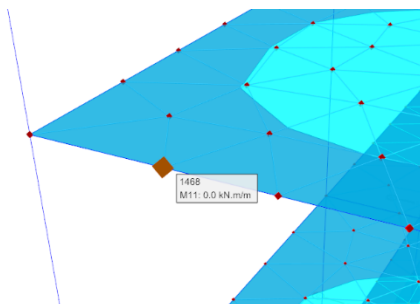
Frame and Shell Highlighting

Frame and shell members are highlighted when the mouse is over them. Tooltip windows will appear, giving identity and connectivity information and analysis results for the selected effect.



Node Magnification

Nodes are magnified when the mouse hovers over them to reveal their identity and analysis results.



Enhancements in RC Detailing

Rebar Lap Splice and Anchorage Length Report

In ProtaDetails 2022, we have added a brand new report that summarizes the step-by-step calculation of lap splices and anchorage lengths with code references included. It can be found under “ProtaDetails > Design Library > Engineering Utilities > Lap and Anchorage Length”.

Lap and Anchorage Length

Lapped Rebar Ratio :	1
Section Depth :	400.0 mm
Concrete Cover :	40.0 mm
Concrete Type :	Normalweight Concrete
Reinforcement Coating :	Uncoated
Anchorage Length :	356.52 mm <input type="checkbox"/> Bar is Under Compression
Lap Length :	445.65 mm <input type="checkbox"/> All Section Under Tension

Print Report

Improved Quantity Take-Off for Corewalls

Quantity take-off for corewalls is improved when elevations are drawn through stories.

Rebar Quantity Tables to Excel

Rebar quantity take-off tables can now be communicated to Excel for further refinement requirements.

Unlimited Rebar Annotation Characters

Rebar annotation character limitation is removed. Any string can be specified for rebar symbols.

Steel Bar Parameters	
Minimum Yield Strength	: 4077.47 kg/cm2
Ultimate Strength (Fu)	: 0.00 kg/cm2
Bearing Strength	: 0.00 kg/cm2
Material Coeff	: 1.15
Rebar Symbol	: DB
Rib Type	: Type 2

Imperial Rebar Labels For Metric Design

Each **rebar material grade** can be assigned to have an **Imperial** or an **SI identity** individually. Being a requirement in some of the countrys’ practices, in ProtaStructure 2022, we have separated the usage of imperial units and assignment of imperial rebar materials to allow our users to define rebar materials (and annotations) in imperial units while the rest of the program uses metric units.

The New ProtaSteel

ProtaSteel 2022 is a **milestone** release with 64-bit architecture, modern user interface, extended connection library, new step-by-step design reports, modernized macro dialogs, in-product wizards, usage and productivity features, the **IntelliConnect**, and much more.

New Technology and User Experience

64-bit Architecture with Improved Technology Infrastructure

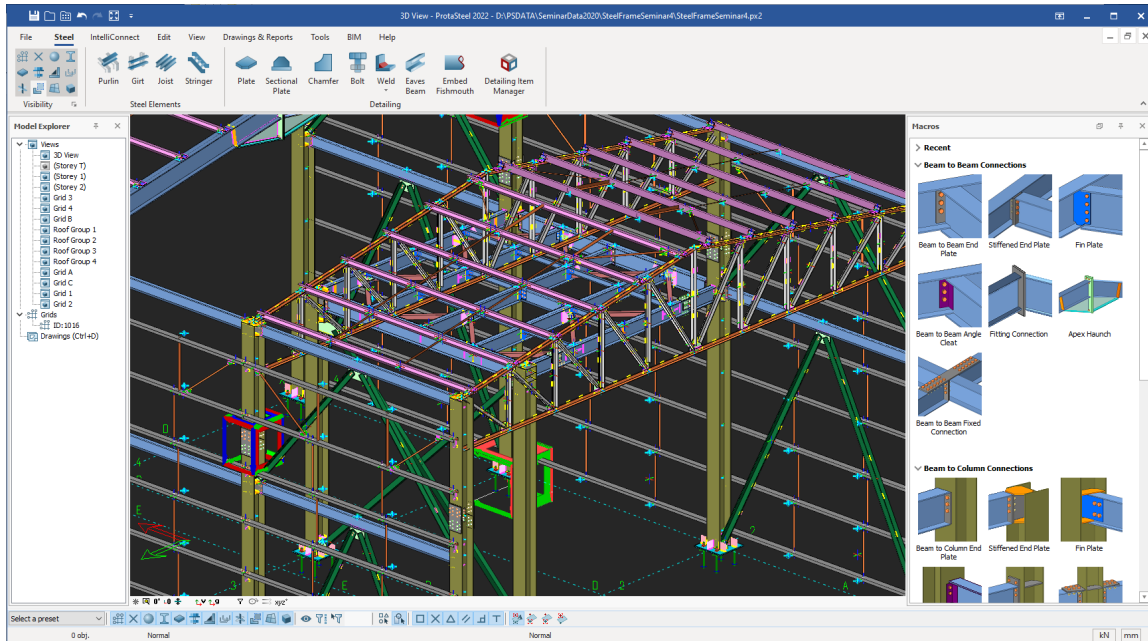
We continually research and invest in new technologies to provide you with the most unique and easy-to-use structural engineering software. ProtaSteel 2022 is now compatible with 64-bit operating systems to bring you significant advantages in the performance and management of big models.

New Ribbon Toolbar For a Streamlined User Experience

ProtaSteel 2022 welcomes you with a redesigned brand-new user interface without breaking usage habits, industry standards, and productivity. The learning curve is almost absent due to the similar layout and commands widely adopted and used in the steel drafting and design industry.

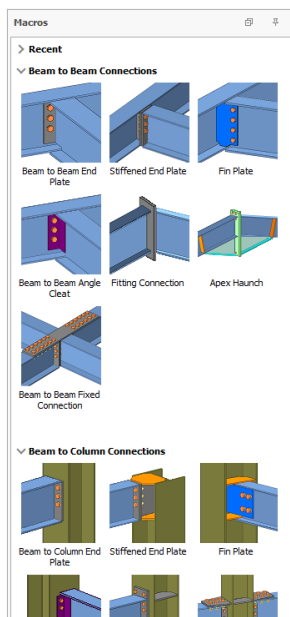
New User Interface

The new ribbon toolbar is shaped around your project workflow and brings new technology and ergonomics together. The new interface is laid out logically, starting from modeling, steel connections, editing, review, and BIM integration to guide you in your project.



New Macro Gallery

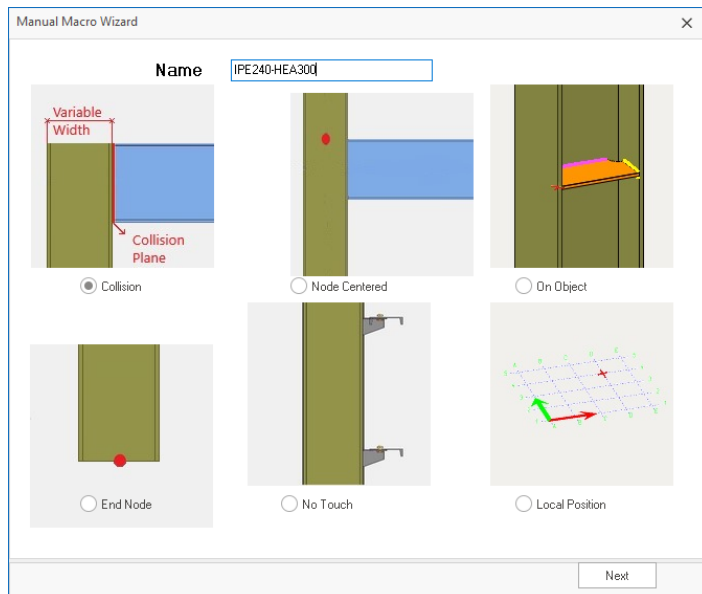
All connection macros are collected and classified under the new **“Macro Gallery”**. The most recent macros are always listed at the top for easy access.



User-Defined Connections Wizard

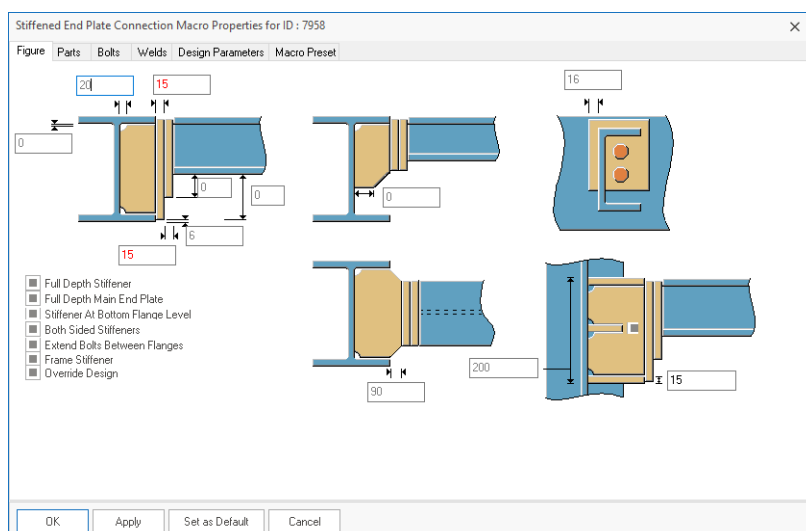
One of the powerful features of ProtaSteel is the ability to create user-defined connections and save them for later use in similar joints.

We have now developed a new wizard that will make your life even easier when you want to create user-defined connections. Just follow the wizard instructions, and you are done.



Intelligent Input Fields on Macro Dialogs

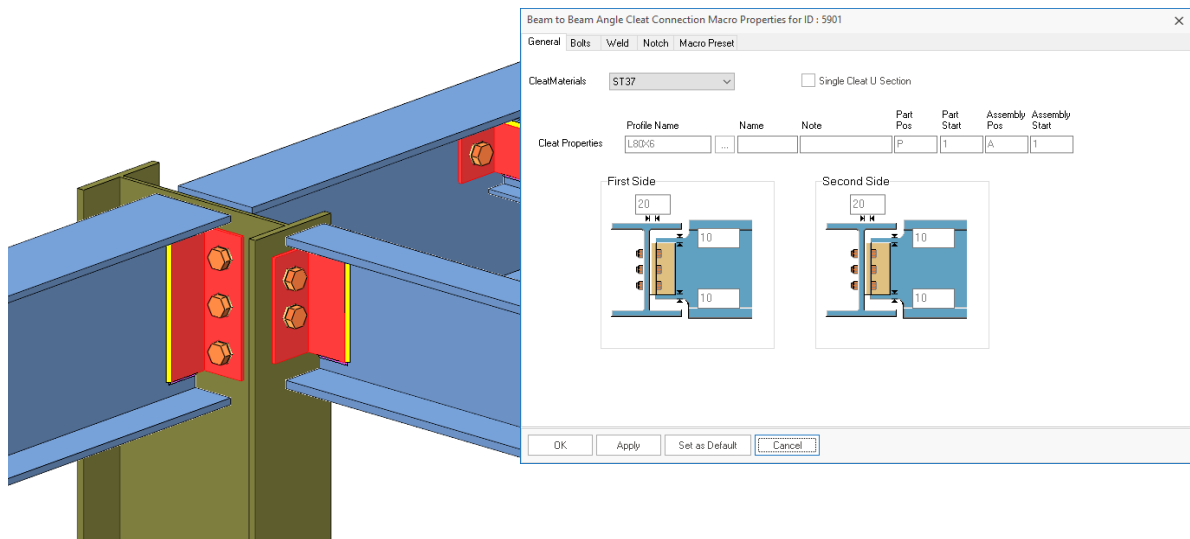
The input fields on macro dialogs now display the actual value used in a particular connection parameter instead of empty. The automatically calculated values are shown in **GRAY**, whereas user values entered and not updated are shown in **RED**. User values that are reflected in the connection macro are shown in **BLACK**. We aim to progressively replace all dialogs in ProtaSteel with this technology with every new release.



New Connection Macros

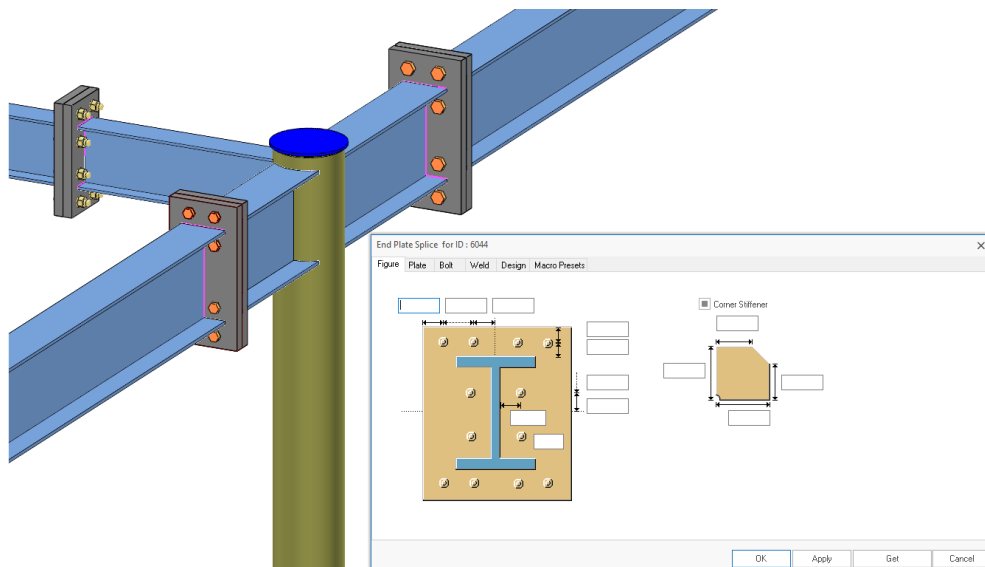
Beam to Column Angle Cleat Connection

In ProtaSteel 2022, the **beam-column angle cleat connection** is introduced. This macro supports a detailed design check report (similar to beam-to-beam angle cleat connection)



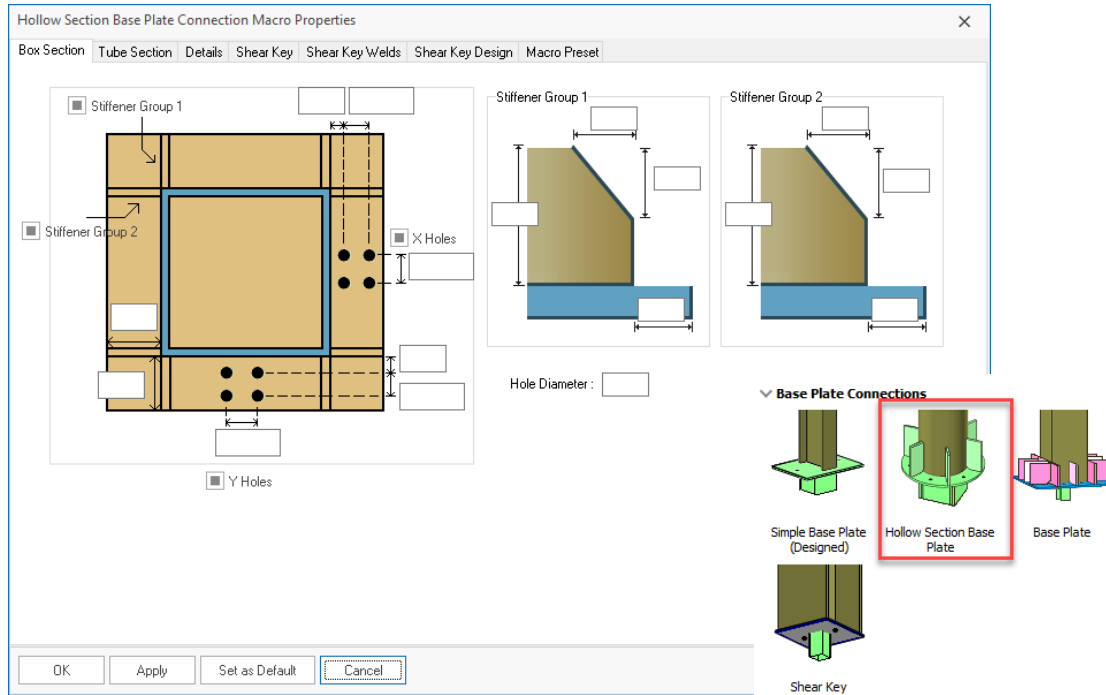
Bolted End Plate Splice Connection

In addition to the standard splice connection, ProtaSteel 2022 introduces the new “Bolted End Plate Splice” connection. This connection may be particularly preferred in the low moment and low tension scenarios.



Base Plate Connection for Hollow Sections

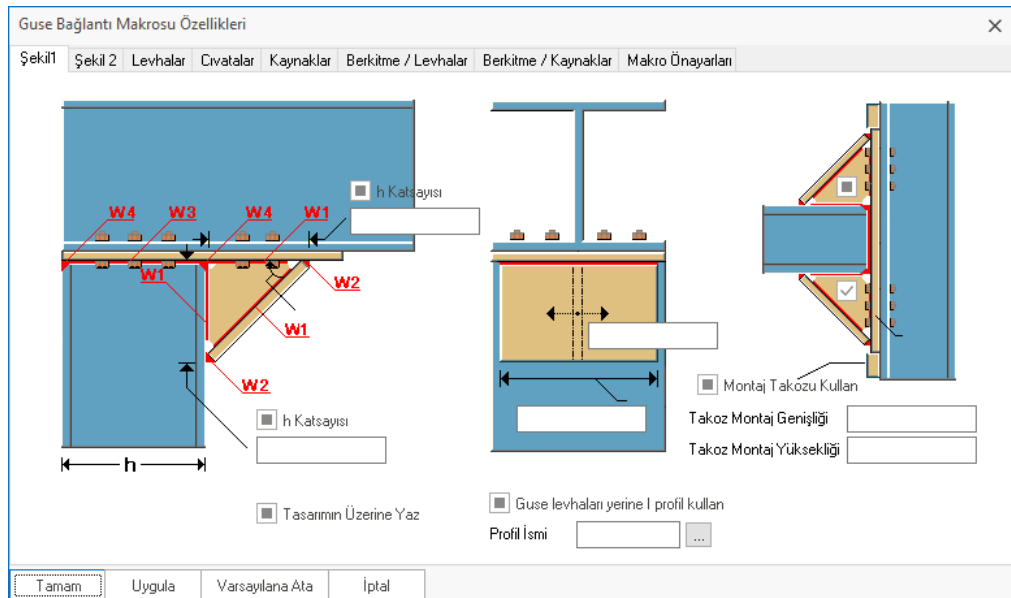
You can now insert a **base plate connection to hollow sections** with ProtaSteel 2022.



Haunch Connection for Continuous Beams

A new type for standard haunch connection is introduced.

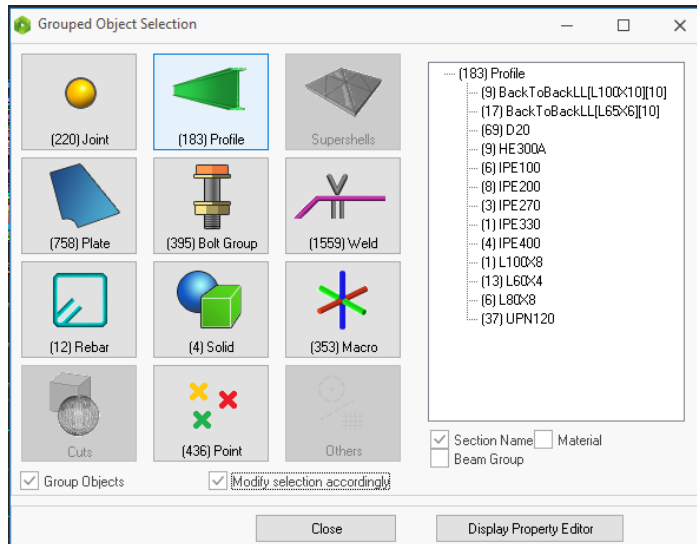
This type is beneficial if your beam needs to be continuous without interruptions, such as beams supporting monorails and such.



Improvements in Object Grouping, Filtering, and Selection

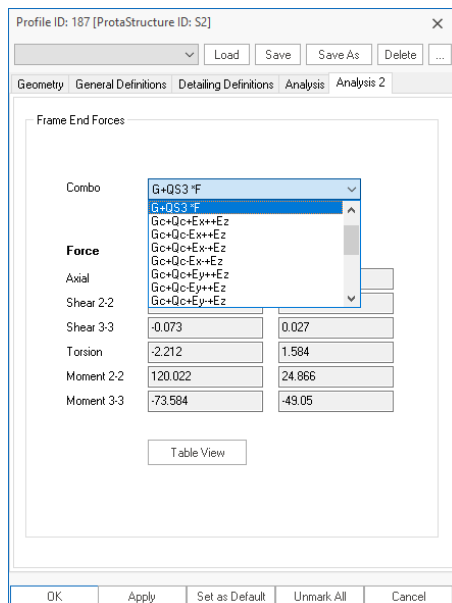
A vital usability improvement comes with object selection. In ProtaSteel 2022, the selection operation **automatically groups** the selected objects.

You can filter the selection **on the fly** to different materials, profile names, plate thicknesses, and several other object attributes.



Connection Design Checks and Internal Force Reporting

Connection design checks now consider each combination instead of using the enveloped values. Member internal forces for each load combination are also listed on the member property window.



New Program-wide Settings Systematics

The settings such as Macro defaults, model filters, sheet settings, model object defaults, used profiles are more consistent all around the software. They are classified into three different levels:

- Global Settings (Factory Defaults)
- Company Settings
- Local Settings (saved in the project folder along with each project)

All settings are copied into the project data folder when you create a new model. In this way, when the model is copied to another computer, all the settings go along.

The settings from these three levels can be summoned in all macro dialog windows capable of changing and saving default settings (such as profiles, connection macro settings, part, assembly drawing settings, sheet settings, etc.). You can select the settings from these three different levels, or you can save and use your custom settings.

This system is designed to be very similar to the method used in industry practice. So, another gap is closed with ProtaSteel 2022 by fulfilling this user feedback.

Other Enhancements in ProtaSteel

Manual Dimensions and Notes in Part and Assembly Drawings

Subsequent automated generation operations preserve manual dimensions and notes you add to part and assembly drawings. This feature is essential in terms of maintaining continuity in your work.

Improvements and Ease of Use in Part and Assembly Drawings

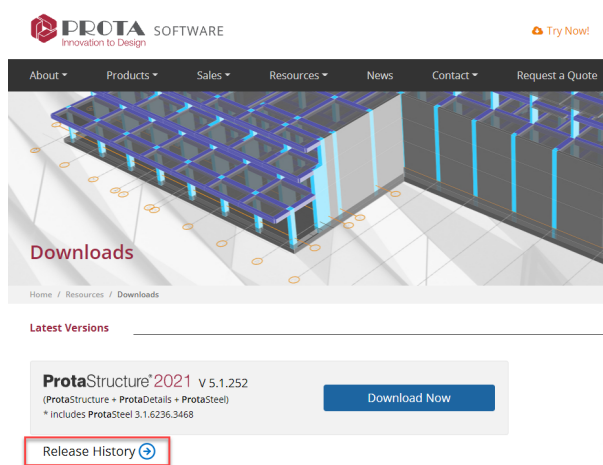
Simplifications are made in part and assembly drawing settings. The Part and Assembly Settings commands are now directly placed on the ribbon toolbar for easier access. Previously they were in the right-click menu, which is harder to find, especially for new learners.

Improved In-Product Learning, Usability, and Productivity

ProtaSteel 2022 includes significant new features and improvements in usability and productivity.

Overall Improvements on Stability and Performance

Due to quality control procedures and user feedback, many stability and performance improvements are implemented in the ProtaStructure 2022 suite of products. Most of the upgrades are already released with the previous version (v2021) updates throughout the year, details of which can be seen in the **“Release Notes”** section in our website’s user area. The release notes also refer to customer ticket numbers that are resolved.



Release History and Notes

Official Releases Beta Releases

Total number of official releases: 18

Release No Date
5.1.252 21.9.2021 [Release Notes -](#)

21 improvements between 5.1.252 - 5.1.250.

Ticket No	Case No	Type	Description
	PS-8020	Improvement	Shearwall analysis model is improved.
#18646	RVT-56	Improvement	Member labels are mapped to the "Mark" field in the Revit plugin.
	RVT-69	Improvement	"ProtaBIM 2021 for Revit 2022" is published. You can download it from the 'Downloads' section of our website.
#24782	PS-9870	Fix	It has been controlled with 5.1.252 and building analysis have been finished properly.
	PS-5870	Fix	Bug fixed in the value of Bw and H1 for Thailand steel sections.

Thank You

Thank you for choosing the ProtaStructure Suite product family.

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

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

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

The Prota Team

