ProtaStructure Design Guide

Assessment & PBD to Eurocode 8 Part 3

Version 2.0 August 2025

Please contact us for your training and technical support queries

asiasupport@protasoftware.com

globalsupport@protasoftware.com

Publisher

PROTA SOFTWARE

Table of Contents

Introduction

Eurocode 8: *Design of structures for earthquake resistance – Part 3: Assessment and retrofitting of buildings* (EN 1998-3:2005) is a fundamental reference for the seismic evaluation of existing buildings. It provides a structured methodology for assessing the seismic performance of reinforced concrete, steel, and masonry buildings, and for designing appropriate retrofitting measures. Unlike the design of new buildings, the assessment of existing structures involves greater uncertainty due to limited documentation, potential damage history, or outdated construction practices—therefore requiring tailored safety factors, analysis procedures, and knowledge-level definitions.

This design guide aims to walk users through the complete workflow of evaluating reinforced concrete buildings using ProtaStructure's EC8 module. The system supports both strengthening of undamaged buildings and repair of earthquake-damaged structures, while ensuring compliance with Eurocode principles. Through practical tools and codified checks, ProtaStructure enables engineers to efficiently carry out performance-based assessments, select intervention strategies, and generate comprehensive evaluation reports.

Eurocode 8 Building Assessment

The seismic assessment of existing buildings in **ProtaStructure** is based on the guidelines defined in **Eurocode 8 – Part 3 (EN 1998-3:2005)** and **Eurocode 8 – Part 1 (EN 1998-1:2004)**. Together, these standards provide a consistent and performance-based approach for evaluating structural safety under seismic loading.

EN 1998-3 focuses specifically on the **assessment and retrofitting of existing buildings**, offering rules for both **linear and nonlinear analysis methods**, performance limit states (**Damage Limitation – DL**, **Significant Damage – SD**, **Near Collapse – NC**), and the application of **confidence factors** based on the quality of available data. It also outlines strategies for retrofitting, whether for **strengthening undamaged structures** or **repairing earthquake-damaged buildings**.

Complementing this, EN 1998-1 defines the **general seismic action models** used in analysis, such as **response spectra**, **soil categories**, **importance factors**, and other fundamental design parameters. These are essential when setting up the seismic loading conditions in the model and are directly integrated into ProtaStructure's seismis parameters.

By combining the provisions of both parts, ProtaStructure enables users to perform member-level and global performance checks under seismic demand. This includes:

- Applying elastic spectra from EN 1998-1
- Conducting performance checks and deformation verifications per EN 1998-3
- Utilizing detailed or approximate reinforcement information
- Selecting appropriate knowledge levels and confidence factors

This dual-standard framework ensures that both the seismic input and assessment logic remain fully compliant with the Eurocode methodology, providing a robust foundation for safe and effective structural evaluations.

Performance Requirements and Limit States

The fundamental performance requirements for seismic assessment are defined in terms of structural damage, categorized into three Limit States (LS): **Near Collapse (NC)**, **Significant Damage (SD)**, and **Damage Limitation (DL)**. Each Limit State characterizes a distinct level of damage and associated structural behavior under seismic action.

- **LS of Near Collapse (NC)** (*EC8- Part 3 2.1 (1)P)*: The structure sustains severe damage, exhibiting minimal residual lateral strength and stiffness. While vertical elements may still support gravity loads, most non-structural components collapse, and significant permanent drifts occur. The structure is unlikely to survive another seismic event and is considered to be on the verge of collapse.
- **LS of Significant Damage (SD)** (*EC8- Part 3 2.1 (1)P)*: The structure experiences notable damage with some remaining lateral strength and stiffness. Vertical elements continue to carry loads,

and non-structural components such as infills and partitions are damaged but generally remain in place. Moderate permanent drifts occur, and the structure may be uneconomical to repair.

- LS of Damage Limitation (DL) (*EC8- Part 3 2.1 (1)P)*: The structure incurs only minor damage. Structural components do not yield significantly and retain most of their strength and stiffness. Non-structural elements may develop distributed cracking, but the damage is repairable and permanent drifts are negligible. The structure remains functional without requiring intervention.

These Limit States are linked to specific seismic return periods, which define the level of protection required. For ordinary buildings, the following return periods are typically used: (*EC8- Part 3 2.1 (3)P*)

- NC: 2475 years (2% probability of exceedance in 50 years),
- SD: 475 years (10% in 50 years),
- DL: 225 years (20% in 50 years).

The return periods to be adopted may vary based on national provisions and are specified in the corresponding National Annexes. By selecting the appropriate return period for each Limit State, the assessment ensures that the building meets the required level of safety and performance.

Knowledge Levels and Confidence Factors (EC8-3, *Clause 3.3*)

For the purpose of choosing the adlnissible type of analysis and the appropriate confidence factor values, the fo11owing three knowledge levels are defined:

KL I : Limited knowledge

KL2 : Normal knowledge

KL3 : Full knowledge

KL1 – Limited Knowledge (*EC8- Part 3, Clause 3.3.2*)

- Geometry: Overall geometry and member dimensions are known either from a site survey or from original outline drawings. If outline drawings are used, a representative sample of dimensions must be verified on-site. Significant discrepancies necessitate a more complete survey.
- Details: Structural detailing is not available from construction drawings and is assumed based on typical design practices of the construction period. Limited inspections in critical regions must be carried out to validate assumptions. Otherwise, extended in-situ investigations are required.

- **Materials**: No direct information on material properties is available. Default values based on construction era standards should be assumed. Limited testing in critical elements must be performed.

Analysis based on KL1 should use **linear methods** (either static or dynamic). The information gathered must be sufficient to verify element capacities and build a reliable linear model.

KL2 – Normal Knowledge (*EC8- Part 3, Clause 3.3.3*)

KL2 corresponds to a state of intermediate knowledge, where:

- **Geometry**: The overall geometry and member sizes are known either
  (a) from an extended survey, or
  (b) from outline construction drawings used for both the original structure and any modifications.
  In case of using drawings, representative dimensions must be verified on-site. If discrepancies are found, a more detailed dimensional survey is required.

- **Details**: Structural details are known either from comprehensive on-site inspection or from incomplete detailed construction drawings. If drawings are incomplete, limited on-site inspections in critical elements must be conducted to verify correspondence.

- **Materials**: Mechanical properties of materials are known either through extended in-situ testing or from original design specifications. In the latter case, limited verification testing should be performed.

The data collected at KL2 is sufficient to verify element capacities and to construct a structural model suitable for **either linear or nonlinear** analysis (static or dynamic).

KL3 –Full Knowledge (EC8- Part 3,Clause 3.3.4)

KL3 corresponds to the highest level of knowledge, defined as follows:

- **Geometry**: The overall structural geometry and member sizes are known either
  (a) from a comprehensive survey, or
  (b) from a complete set of outline construction drawings covering both the original construction and any subsequent modifications.
  In case (b), a sufficient sample of both global geometry and member sizes should be verified on site. If significant discrepancies are found, a full dimensional survey must be carried out.

- **Details**: Structural details are available either from comprehensive in-situ inspection or from a complete set of detailed construction drawings. In the latter case, limited inspection of the most critical elements should be conducted to verify the consistency of the available information with the actual construction.

- **Materials**: Information on the mechanical properties of materials is available either through comprehensive in-situ testing or from original test reports. If the latter is used, limited in-situ verification testing should be performed.

This knowledge level provides the most reliable data for analysis and allows for the use of **nonlinear analysis methods** with the **lowest confidence factor (CF).**

CF – Confidence Factors (*EC8- Part 3, 3.3.1*)

Confidence Factors Table is given Figure 3.1.

Table 3.1: Knowledge levels and corresponding methods of analysis (LF: Lateral Force procedure, MRS: Modal Response Spectrum analysis) and confidence factors (CF).

Knowledge Level	Geometry	Details	Materials	Analysis	CF
KL1		Simulated design in accordance with relevant practice ***and*** from **limited** *in-situ* inspection	Default values in accordance with standards of the time of construction ***and*** from **limited** *in-situ* testing	LF- MRS	CF_{KL1}
KL2	From original outline construction drawings with sample **visual** survey ***or*** from **full** survey	From incomplete original detailed construction drawings with **limited** *in-situ* inspection ***or*** from **extended** *in-situ* inspection	From original design specifications with **limited** *in-situ* testing ***or*** from **extended** *in-situ* testing	All	CF_{KL2}
KL3		From original detailed construction drawings with **limited** *in-situ* inspection ***or*** from **comprehensive** *in-situ* inspection	From original test reports with **limited** *in-situ* testing ***or*** from **comprehensive** *in-situ* testing	All	CF_{KL3}

NOTE The values ascribed to the confidence factors to be used in a country may be found in its National Annex. The recommended values are CF_{KL1} = 1,35, CF_{KL2} = 1,20 and CF_{KL3} = 1,00.

Figure 1 Confidence Factors From Table 3.1 (EC-8 Part 8 3.3)

Knowledge Level	Applicable Analysis	Confidence Factor (CF)
KL1	Linear	1.35
KL2	All Analysis Type	1.20
KL3	All Analysis Type	1.00

Methods of Analysis (*EC8- Part 3, 4.4*)

The assessment of existing buildings according to EN 1998-3:2005, the effects of seismic actions combined with permanent and variable loads can be evaluated using the following structural analysis methods:

- Lateral Force Analysis (Linear)
- Modal Response Spectrum Analysis (Linear)
- Nonlinear Static Analysis (Pushover)
- Nonlinear Time-History Analysis (Dynamic)
- q-Factor Approach (Force-Based Linear Analysis with Reduction Factor)

Unless the q-factor approach is used, the seismic input must be based on the **elastic (unreduced)** response spectrum defined in EN 1998-1:2004, Clause 3.2.2.2. In the q-factor method, a **reduced seismic spectrum** (based on ductility assumptions) is applied as outlined in Clause 4.2(3)P.

For all methods, mean material properties (e.g., mean concrete strength, steel yield) shall be used when constructing the structural model.

Each analysis method has specific conditions and limitations for its application. These are detailed in Clauses 4.4.2 through 4.4.5 of EN 1998-3 and must be considered, particularly for reinforced concrete or masonry buildings.

Lateral Force Analysis *(EC8- Part 3, 4.4.2)*

Lateral Force Analysis (LFA) is a **linear elastic method** that assumes the structural response is governed solely by the fundamental mode of vibration. The method distributes seismic actions as lateral forces along the height of the building, based on the mass and the mode shape.

Applicability Conditions (EN 1998-1:2004, Clause 4.3.3.2.1)

- The structural response must be **predominantly governed by the first mode**. That is, the influence of higher modes should be negligible.
- The following two conditions must be satisfied simultaneously:
    - The fundamental period T_1 must be **less than the spectral limit period** (typically denoted as *Tc*
    - The structure must be **vertically regular**, in accordance with Clause 4.2.3.3.

Calculation Steps

Base Shear (Fb)

The total base shear is calculated using the elastic response spectrum (Se), not the reduced design spectrum (Sd), as per EN 1998-1:2004, Clause 4.3.3.2.2:

$$F_b = S_e(T_1) \cdot m \cdot A \quad \text{(Eq. 4.5)}$$

A: Correction factor (0.85 or 1.0 depending on number of stories and periods)

m: Total seismic mass of structure

$S_e(T_1)$: Spectral acceleration from the elastic response spectrum at period T_1.

Lateral Load Distribution to Stories *(EN 1998-1:2004, Clause 4.3.3.2.3)*

The base shear is distributed to the floors either:

- Proportional to story height (linear distribution), or
- According to the shape of the fundamental mode:

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\sum z_j \cdot m_j} \quad Eq\ (4.11)$$

Where:

- F_i: Lateral force applied at floor *i*
- z_i: Height of floor *i*
- m_i: Mass at floor *i*

If the plan is irregular, torsional effects must be considered by introducing **accidental eccentricity** in the application of lateral forces.

Demand/Capacity (D/C) Check for Assessment Purposes *(EN 1998-3:2005, Clause 4.4.2)*

For all **ductile primary elements**, the following **demand-to-capacity ratio** must be computed:

$$p_i = \frac{D_i}{C_i}$$

Where:

- D_i: Seismic demand obtained from the analysis
- C_i: Capacity of the structural element

The ratio should generally remain between **2.0 and 3.0**. The **recommended value** is **2.5**.

This ratio serves as a **measure of demand imbalance** between ductile elements and must be checked especially at **beam–column joint regions** where plastic hinges are expected to form.

For **brittle elements**, demand/capacity ratios should also be verified. Capacities should be computed using material properties reduced by the corresponding **Confidence Factor (CF)**.

Multi-Modal Response Spectrum Analysis *(EC8- Part 3, 4.4.3)*

To determine the seismic response of the structure by superimposing the contributions of multiple vibration modes, using the elastic response spectrum. This method provides a more accurate representation of structural behavior for irregular, tall, or torsionally sensitive buildings compared to single-mode methods.

Scope of Applicability

This method is required when the structural response **cannot be accurately represented by the fundamental mode alone**. According to *EN 1998-1:2004 Clause 4.3.3.3.1*, the following conditions apply:

- The method shall be used when **lateral force analysis is not sufficient**, such as in:
    - Irregular structures (plan or vertical),
    - Structures with significant higher-mode effects,
    - Tall or torsionally unbalanced systems.
- The **sum of the effective modal masses** considered in the analysis must:
    - Account for at least **90% of the total mass** in each principal direction, **OR**,
    - Include all modes contributing more than **5% effective modal mass** each.

Input Requirements

- Full 3D structural model with defined:
    - Mass distribution,
    - Stiffness matrix (K),
    - Damping ratio (typically 5% critical),
    - Boundary conditions.
- **Elastic response spectrum** Se(T), defined in *EN 1998-1:2004 Clause 3.2.2.2*, based on site class, importance factor, and return period.

Procedure

1. **Modal Analysis**
    - Solve the eigenvalue problem to determine natural periods T_i and mode shapes ϕ_i.
    - Calculate the **effective modal mass** for each mode in X, Y, and Z directions.
2. **Spectral Response Calculation**
    - For each mode *i*, obtain the spectral acceleration $S_e(T_i)$ from the elastic spectrum.
    - Compute the corresponding forces and displacements.

3. **Modal Combination**

   Combine modal responses using appropriate statistical methods:

   - **SRSS (Square Root of Sum of Squares)**: when modal periods are well-separated.
   - **CQC (Complete Quadratic Combination)**: when modal periods are close.

4. **Torsional Effects**

   - Include accidental eccentricity effects in each mode if plan irregularities are present (Clause 4.3.3.3.3).

5. **Output**

   - Story shears, floor displacements, element forces (axial, shear, moment), and torsional effects.

Nonlinear Static (Pushover) Analysis *(EC8- Part 3, 4.4.4)*

This analysis method is used to evaluate the nonlinear behavior of a structure under seismic loading. Gravity loads are applied in combination with monotonically increasing lateral loads. The resulting capacity curve provides insight into the structure's strength, deformation capacity, plastic hinge development, and collapse mechanism.

Scope of Applicability

According to **EN 1998-3:2005 Clause 4.4.4.1** and **EN 1998-1:2004 Clause 4.3.3.4.2**:

- **Irregular buildings in plan** must be analyzed using **3D spatial models** to account for torsional and three-dimensional effects.
- **Plan-regular buildings** (as per EN 1998-1 Clause 4.2.3.2) may be analyzed using **two separate 2D models**, one for each principal direction.
- The method is applicable for both **new designs** and **performance assessment of existing structures**.

Input Requirements

- **Structural model**:
  - Nonlinear behavior defined at the element level (e.g., moment-rotation and axial deformation capacities),
  - Realistic mass and stiffness distribution,
  - Material properties based on **mean values** f_{cm}, f_{ym},
  - Explicit definition of plastic hinge zones.
- **Loading**:
  - Constant gravity loads,

- Monotonically increasing lateral load patterns.

- **Spectral data**:
  - **Elastic acceleration spectrum Se(T)** as defined in EN 1998-1:2004, Clause 3.2.2.2.

Analysis Procedure

1. Lateral Load Patterns (EN 1998-3:2005 Clause 4.4.4.2)

At least two lateral load patterns shall be applied:

- **Uniform distribution**: Constant across the height, proportional to mass only.
- **Modal distribution**: Based on the fundamental mode shape of the structure.

Loads shall be applied at the mass centers of each floor. **Accidental eccentricity** must be considered.

2. Generation of the Capacity Curve *(EN 1998-3:2005 Clause 4.4.4.3)*

As lateral loads increase, the relationship between **base shear (Vb)** and **control node displacement (δ)** is recorded. This relationship defines the **capacity curve** of the structure.

3. Determination of Target Displacement *(EN 1998-3:2005 Clause 4.4.4.4)*

The **target displacement** is defined as the intersection point between the elastic displacement demand and the capacity curve.

It can be calculated according to **EN 1998-1:2004 Informative Annex B**:

$$\delta_t = \frac{S_d(T)}{g} \cdot \Gamma \cdot \Phi$$

- Sd(T): Spectral displacement,
- Γ: Modal participation factor,
- Φ: Mode shape value at the the control node.

4. Torsional and Higher Mode Effects *(EN 1998-3:2005 Clause 4.4.4.5)*

- For buildings with **plan asymmetry**, torsional effects shall be estimated as per EN 1998-1:2004, Clause 4.3.3.4.2.7(1)-(3).
- If higher modes contribute significantly to the global response, the following options may be considered:
  - **Nonlinear time-history analysis** (Clause 4.4.5),
  - **Advanced pushover procedures** (e.g., Adaptive or multi-mode Pushover), which allow translation from global measures (e.g., interstory drift) to local deformation demands (e.g., plastic rotations).

Expected Outputs

- Global and interstory displacements,
- Base shear vs. displacement (capacity curve),
- Sequence and locations of plastic hinge formation,
- Ability of the structure to reach target displacement,
- Estimated performance level (Damage Limitation – DL, Significant Damage – SD, Near Collapse – NC)

Nonlinear Time History Dynamic Analysis *(EC8 - Part 3, 4.4.5)*

This analysis method aims to compute the nonlinear dynamic response of structures under seismic actions by directly integrating the differential equations of motion over time. The analysis is carried out using time-varying acceleration inputs (accelerograms), allowing the direct modeling of energy dissipation, loading/unloading cycles, and damage accumulation at the element level.

EN 1998-3:2005 Clause 4.4.5(1)P: The method shall be applied in accordance with EN 1998-1:2004, Clauses 4.3.3.4.3(1)–(3).

EN 1998-1:2004 Clause 4.3.3.4.3: The analysis involves evaluating the time-dependent structural response to specified ground motion records (accelerograms).

Input Requirements

- Ground Motion (Seismic Input):
  - At least 3, preferably 7 records (natural or artificial) shall be used, in accordance with EN 1998-1:2004 Clause 3.2.3.1.
  - The records shall be scaled or selected to match the site-specific response spectrum.
- Structural Model:
  - Nonlinear material and connection behavior shall be defined for all structural elements.
  - Energy dissipation shall be modeled using hysteretic, viscous, or combined damping mechanisms.
- Material Properties:
  - Mean material properties shall be used.
  - Constitutive models shall reflect energy dissipation during unloading and reloading cycles.

Analysis Procedure

1. **Numerical Integration of Differential Equations**
    - The structure's time-dependent response is computed via direct numerical integration of its governing equations under time-varying acceleration input.
2. **Modeling of Cyclic Loading/Unloading Behavior**
    - Structural elements must be capable of dissipating energy during unloading and subsequent reloading.
    - This behavior should be defined using cyclic plasticity models, including stiffness degradation.
3. **Evaluation of Design Effects**
    - If **7 or more ground motion records** are used, the **average** of the resulting internal forces shall be taken.
    - If **3 to 6 records** are used, the **most unfavorable result** shall be adopted as the design value.

q-factor Approach (EC8 - Part 3, 4.4.6)

The q-factor approach is a method that accounts for the nonlinear seismic behavior of structures **indirectly**, by reducing the results of elastic analysis using a **behavior factor (q)**. This factor represents the expected ductility level and energy dissipation capacity of the structure.

EN 1998-3:2005, Clause 4.4.6: This method is permitted and shall be used in conjunction with the corresponding linear analysis procedures.

EN 1998-1:2004, Clauses 4.3.3.2–4.3.3.3: The method can be applied with Lateral Force and Modal Response Spectrum analysis procedures.

The selection of **q-values** is based on the **structural system type** and **ductility class**, as defined in **EN 1998-1:2004, Table 5.1** and **Clause 5.2.2.1**.

Analysis Procedure

The **elastic spectrum** $S_e(T)$ is divided by the q-factor to obtain the **design spectrum**:

$$S_d(T) = \frac{S_e(T)}{q}$$

This spectrum is applied to the structural model for use in linear analysis procedures.

Ductility Classes and Reference q-Values

DCL – Low Ductility Class

Reference: EN 1998-1:2004, Clause 5.2.2.1 and Table 5.1

Structural System	Ductility Class	q_o
Moment-resisting frame	DCL	1.5
Wall system	DCL	1.5
Dual system (frame + wall)	DCL	1.5–2.0

- Structures designed with DCL shall be detailed to accommodate **limited inelastic deformation**.
- Therefore, the **q-factor is generally limited to 1.5**. Higher values are not permitted due to reduced ductility demands.

DCM – Medium Ductility Class

Reference: EN 1998-1:2004, Clause 5.2.2.1(2)

- Structures in this class must be detailed for **controlled plastic behavior**.
- Critical regions shall provide adequate moment–rotation capacity through:
    - Stirrup confinement,
    - Strong column–weak beam provisions,
    - Proper anchorage and lap splice detailing.
- The q-factor is typically selected in the **3.0–4.0** range.
- Detailed rules are provided in **EN 1998-1:2004, Sections 5.4–5.7**.

DCH – High Ductility Class

Reference: EN 1998-1:2004, Clause 5.2.2.1(3)

- DCH structures are expected to develop **extensive plastic deformation**.
- The most stringent detailing rules apply:
    - Closely spaced stirrups in plastic hinge regions,
    - Strong column–weak beam principle,
    - Anti-buckling reinforcement,
    - Energy dissipation through ductile connections.
- The q-factor is selected between **4.5 and 6.0** depending on the structural configuration.

Summary Table – Behavior Factor Selection

Ductility Class	Description	q_0 Value	Notes
DCL	Low Ductility	1.5	Limited energy dissipation, minimal detailing
DCM	Medium Ductility	3.0 – 4.0	Controlled plasticity, moderate detailing
DCH	High Ductility	4.5 – 6.0	Extensive plasticity, stringent detailing requirements

Note:
These values apply to **structures that are regular in elevation**.
If the structure is irregular, the selected q-factor shall be reduced by **20%**, as per **EN 1998-1:2004, Clause 4.2.3.4**.

Assessment

Beams, Columns and Walls: Shear

This section defines the procedure for verifying the shear capacity of reinforced concrete elements (beams, columns, and walls) in seismic performance assessments according to EN 1998-3:2005, Annex A. The check applies to all limit states (NC, SD, DL), with shear demand and capacity evaluated per the provisions of A.3.3.1 and A.3.3.2.

- Mandatory for all performance levels:
    - Near Collapse (NC)
    - Significant Damage (SD)
    - Damage Limitation (DL)
- Follows **EN 1998-3:2005, Annex A.3.3.1 (primary)** and A.3.3.2 (interpretation note for SD/DL).
- Also complies with **EN 1998-1:2004**, Sections 5.4 and 5.5, and **EN 1992-1-1** for ultimate shear capacity limits.

Shear Capacity Calculation

- The shear resistance V_R is calculated using the following expression from EN 1998-3:2005 (Equation A.12):

$$V_R = \frac{1}{\gamma el}[\frac{h-x}{2\,L_v} min\,(N; 0.55 \cdot Ac \cdot fc) + (1 - 0.05 \cdot min\,(5; \mu_\Delta^{pl})) \cdot$$

$$[(0.16 \cdot max\,(0.5; 100 \cdot \rho_{tot}) \cdot (1 - 0.16 \cdot min\,(5; \tfrac{L_v}{h}))\sqrt{f_c} \cdot A_c + V_w)]] \quad (A.12)$$

Symbol Definition

γ_{el} Model uncertainty factor (typically taken as 1.15).

h Total depth of the cross-section. For circular sections, h = D (diameter).

x Compression zone depth in the cross-section.

N Axial compressive force (taken as positive). If tension exists, it is considered as zero.

L_V Shear span, often taken as the moment/shear ratio M/V at the section of interest.

Ac Concrete area of the cross-section:
- For rectangular sections: Ac = b × d
- For circular sections: Ac = π·(Dc)²/4

where Dc = D − 2c − $2d_{bw}$

fc Concrete compressive strength. For primary seismic elements, this should be divided by the partial safety factor γc as per EN 1998-1:2004 Clause 5.2.4

ρ_{tot} Total longitudinal reinforcement ratio

f_{ywd} Design yield strength of the transverse reinforcement.

A_{sw} Area of transverse reinforcement (stirrups) per unit spacing.

V_w Shear resistance contribution from transverse reinforcement, calculated separately.

μ_{Δ}^{pl} Plastic rotation ductility.

Transverse shear resistance:

a-) For cross-sections with rectangular web of width (thickness) b_w:

$$V_w = p_w \, b_w z \, f_{yw} \quad \text{(A.13)}$$

Symbol Definitions

ρ_w : **Transverse reinforcement ratio**: Defined as: ρw = (Asw / s·bw) where Asw is the area of stirrups, s is the spacing, and bw is the web width.

z : **Internal lever arm**, typically taken as z ≈ 0.9·d, where d is the effective depth.

f_{ywd} : **Design yield strength** of transverse reinforcement. For **primary seismic elements**, divide by partial safety factor for steel (γs) in accordance with **EN 1998-1:2004 Clause 5.2.4**.

b-) For circular cross-sections:

$$V_w = \frac{\pi}{2} \frac{A_{sw}}{s} f_{yw} \, (D - 2c) \quad \text{(A.14)}$$

Symbol Definitions

D : **Outer diameter** of the circular cross-section.

A_{sw} **Cross-sectional area of one circular stirrup.**

s **Centerline spacing** between circular stirrups.

f_{ywd} **Design yield strength** of the transverse reinforcement. Divided by γs for primary seismic elements.

c **Concrete cover thickness** (from outer concrete surface to stirrup centerline).

c-) The strength of a concrete wall:

$$V_{R,\max} = \frac{0.85\left(1 - 0.06 \cdot \min\left(5; \mu_{\Delta}^{pl}\right)\right)}{\gamma_{el}} \left(1 + 1.8 \cdot \min\left(0.15; \frac{N}{A_c f_c}\right)\right) (1 + 0.25 \cdot \max(1.75; 100 \cdot \rho_{\text{tot}})) \left(1 - 0.2 \cdot \min\left(2; \frac{L_v}{h}\right)\right) \sqrt{f_c} \cdot b_w \cdot z$$

Reference: Expression (A.15), EN 1998-3:2005

Symbol Definitions

γ_{el}	Importance factor: 1.15 for **primary seismic elements**, 1.0 for **secondary**.
μ_{Δ}^{pl}	Plastic rotation ductility. Capped at 5.
N	Axial compressive force (positive; tension taken as 0).
A_c/f_c	Axial capacity of the cross-section (concrete).
ρl	Longitudinal reinforcement ratio.
Lv	Shear span (moment/shear ratio).
h	Section height (total depth).
fc	Concrete compressive strength (MPa).
bw	Web width (thickness of the wall).
z	Lever arm, often taken as ≈ 0.9·d (effective depth).
Units	All lengths in meters, forces in MN, stress in MPa.

Concrete Columns – Shear Strength after Flexural Yielding ($V_{R,max}$)

If in a concrete column the shear span ratio, *Lv / h,* at the end section with the maximum of the two end moments less or equal to 2,0, its shear strength, VR, should not be taken greater than the value corresponding to failure by web crushing along the diagonal of the colunn after flexural yielding, VR,max, which under cyclic loading may be calculated from the expression (with units: MN and meters):

$$VRmax = \frac{4}{7} \cdot \frac{\left(1 - 0.02 \cdot \min\left(5; \mu_{\Delta}^{pl}\right)\right)}{\gamma_{el}} \left(1 + 1.35 \cdot \frac{N}{A_c f_c}\right) (1 + 0.45 \cdot (100 \cdot \rho_{tot})) \sqrt{\min(40; f_c)} \cdot b_w \cdot z \cdot \sin(2\delta)$$

Symbol Definitions

γ_{el}	Importance factor: 1.15 for **primary seismic elements**, 1.0 for **secondary**.
μ_{Δ}^{pl}	Plastic rotation ductility. Capped at 5.
N	Axial compressive force (positive; tension taken as 0).
$A_c f_c$	Axial capacity of the cross-section (concrete).
ρ_{tot}	Longitudinal reinforcement ratio.
h	Section height (total depth).
f_c	Concrete compressive strength (MPa).
b_w	Web thickness (width of column cross-section).
z	Lever arm, often taken as ≈ 0.9·d (effective depth).

Symbol Definitions

δ	Angle between diagonal and column axis: $\tan\delta = h/L_v$
Units	All lengths in meters, forces in MN, stress in MPa.

Shear Verification and Ductility Classification Specification

To evaluate whether a reinforced concrete element satisfies shear capacity requirements and to classify it as ductile or brittle based on the result.

Verification Condition

The element passes the shear check if the following inequality holds:

$$V_{Ed} \leq V_R$$

- V_{Ed}: Shear force demand from structural analysis (LFA, pushover, or nonlinear time history).
- V_R: Shear capacity calculated using Equation (A.12) in EN 1998-3:2005 Annex A.

Always take the minimum of:

- Shear capacity from EN 1998-3:2005 (Annex A.3.3.1 / A.3.3.2),
- Shear capacity from EN 1992-1-1 (Eurocode 2, Clause 6.2).

Ductility Classification Rule

Criterion	Classification
$V_{Ed} \leq V_R$	Ductile
$V_{Ed} > V_R$	Brittle

- Ductile Elements:
  - May undergo inelastic deformation.
  - Allowed to proceed to chord rotation verification.
  - Can form plastic hinges under seismic loading.
- Brittle Elements:
  - Shear capacity is insufficient.
  - Failure is expected prior to flexural yielding.

Must be retrofitted or excluded from nonlinear deformation checks.

Applicable Standards

- EN 1998-3:2005
  - A.3.3.1 – Shear verification for ductile elements
  - A.3.3.2 – Verification under SD and DL performance levels
- EN 1992-1-1:2004
  - Clause 6.2 – Shear strength of reinforced concrete members
- EN 1998-1:2004
  - Clause 5.4 and 5.5 – Detailing and material reduction factors

Beam, columns and walls under flexure with and without axial force

To verify whether the **chord rotation demand** from seismic analysis remains within the **rotation capacity (θ_m)** of the structural element under cyclic loading at the **Near Collapse (NC)** performance level.

Verification Condition

The element satisfies the flexural (rotation) check if:

$$\vartheta_{Ed} \leq \vartheta_{um}$$

Where:

- ϑ_{Ed} : Chord rotation demand at the end section (from pushover, nonlinear time history, etc.).
- ϑ_{um}: Ultimate rotation capacity from **Annex A – Equation (A.1)** in **EN 1998-3:2005**.

Chord Rotation Capacity – θ_{um} Formula (Near Collapse (NC)

$$\theta_{um} = \frac{1}{\gamma_{el}} \cdot 0.016 \cdot (0.3)^{\nu} \left[\frac{\max(0.01;\omega')}{\max(0.01;\omega)} f_c \right]^{0.225} \cdot \left(\min \left(9; \frac{L_v}{h}\right)\right)^{0.35} \cdot 25^{\left(\alpha_{psx} \frac{f_{yw}}{f_c}\right)} \cdot\cdot (1.25^{100\rho_d}) \quad \text{(A.1)}$$

Special Case For Walls:

Multiply θ_{um} by **0.58** according to **EN 1998-3:2005 – A.3.2.2(2)**.

Symbol Descriptions

ϑ_{um}	Ultimate chord rotation (total deformation capacity including elastic + inelastic)
γ_{el}	Element classification factor: 1.5 for **primary**, 1.0 for **secondary** seismic elements
ω'	Mechanical reinforcement ratio for **compression** longitudinal reinforcement
ω	Mechanical reinforcement ratio for **tension** longitudinal reinforcement (incl. web steel)
fc	Mean concrete compressive strength
L_v	Shear span = moment/shear at yielding end (distance to contraflexure point)
h	Section depth

Symbol Descriptions

α	Confinement effectiveness factor (see Eq. A.2)
p_{sx}	$A_{sx}/(b_w*s_h)$ ratio of transverse steel parallel to the direction *x* of loading stirrup spacing,
f_{yw}	Yield strength of transverse reinforcement (stirrups), mean value
ρ_d	Diagonal reinforcement ratio (if any)

Confinement Effectiveness Factor (α)

Formula:

$$\alpha = \left(1-\frac{s_h}{2b_o}\right)\left(1-\frac{s_h}{2h_o}\right)\left(1-\frac{\sum b_i^2}{6h_o b_o}\right)\ (A.2)$$

Symbol Description

α	Confinement effectiveness factor
s_h	Center-to-center spacing of transverse hoops or spirals (stirrups)
b_o	Dimension of the confined core measured to the centerline of the hoops in x-direction
h_o	Dimension of the confined core measured to the centerline of the hoops in y-direction
b_i	Center-to-center spacing of longitudinal bars restrained by corner stirrups or cross-ties along perimeter
$\sum b_i^2$	Sum over all longitudinal bars laterally supported by hoops/ties

Chord Rotation Plastic Part – θ_{um}^{pl} Formula (Near Collapse (NC))

The **plastic** component of the chord rotation capacity under cyclic loading for beams, columns, and walls is given by:

$$\theta_{um}^{pl} = \theta_{um} - \theta_y = \frac{1}{\gamma_{el}} \cdot 0.0145\,(0.25)^{v}\left[\frac{max(0.01;\omega')}{max(0.01;\omega)}\right]^{0.3} f_c^{0.2}\cdot\left(min\left(9;\frac{L_v}{h}\right)\right)^{0.35}\cdot 25^{\left(\alpha_{psx}\frac{f_{yw}}{fc}\right)}\ (1.25^{100\rho_d})\ (A.3)$$

Special Case For Walls:

Multiply θ_{um}^{pl} by **0.6** according to **EN 1998-3:2005 – A.3.2.2(2)**.

where the chord rotation at yielding, Θy, should be calculated in accordance with **A.3.2.4,** γel is equal to 1,8 for primary seismic elements and to 1,0 for secondary seismic ones and all other variables are defined as for expression (A.1).

Special Provisions for Chord Rotation Capacity Adjustments

1-Lack of Earthquake Detailing (Undetailed Members)

- **Scope**: Members not detailed for seismic resistance (e.g. existing old buildings).
- **Modification**:

- Both **total chord rotation** (*ϑ_{um}*) and **plastic part** (*ϑ_{pl}*) shall be **divided by 1.2**.
- *Source*: EN 1998-3:2005 – Clause A.3.2.1(3)

2-Deformed (High Bond) Bars with Lap Splicing at End Regions

- **Scope**: If longitudinal bars are spliced at yielding region (common in columns/walls at floor level).
- **Modifications**:
  - The **compression reinforcement ratio ω'** shall be **doubled**.
  - If **lap length lo < lou,min**, then:
    - Plastic part of rotation (*ϑ_{pl}*) is **multiplied by (l_o / $l_{ou,min}$)**
    - Yield rotation *ϑ_y* must be updated using A.3.2.4(3)
- **Minimum lap length**:

$$l_{ou,min} = \frac{d_{bL} f_{yL}}{\left(1.05 + 14.5 \cdot \alpha_1 \cdot \rho_{sx} \cdot f_{yw}/f_c\right) \cdot \sqrt{f_c}}$$

Source: EN 1998-3:2005 – Clause A.3.2.1 (4)

$$\alpha_1 = (1 - s_h / (b_o)(1 - s_h / (2h_o))n_{restr}/n_{tot}$$

n_{restr} : number of lapped longitudinal bars laterally restrained by a stirrup

corner or a cross-tie, and

n_{tot}: total number of lapped longitudinal bars along the cross-section

perimeter.

3-Plain Bars With Lapping and Hooks (lo ≥ 15dbL)

- **Scope**: Lapped smooth bars with standard hooks.
- **Modifications**:
  - Reduce **shear span Lv** by lap length (**Lv - lo**)
  - **Total chord rotation capacity (*ϑ_{um}*)**:

$$\theta um\ (1)\&(3)\ =\cdot\ 0.019 \cdot (10 + \min\,(40, \mathrm{lo}/d_{bL})$$

  - **Total chord rotation capacity (*ϑ_{um}^{pl}*)**:

$$\theta_{um}^{pl}\ (1)\&(3)\ =\cdot\ 0.019 \cdot \min\,(40, \mathrm{lo}/d_{bL})$$

Source: EN 1998-3:2005 – Clause A.3.2.1(5)

Ultimate Chord Rotation Capacity – Alternative Approach

Expression for Ultimate Chord Rotation (A.4)

$$\theta_{um} = \frac{1}{\gamma_{el}}\left(\theta_y + (\varphi_u - \varphi_y)L_{pl}\left(1 - \frac{0.5L_{pl}}{L_v}\right)\right)(A.4)$$

Symbol Definitions

θy	Chord rotation at yielding (A.10 or A.11)
ϕu	Ultimate curvature at end section
ϕy	Yield curvature at end section
Lp	Length of plastic hinge
Lv	Shear span length
γel	Importance factor (1.5 for primary, 1.0 for secondary seismic members)

Chord Rotation Capacity – θ_u (Significant Damage (SD))

The **chord rotation capacity** corresponding to the **Significant Damage (SD)** limit state, denoted as ϑ_{SD} represents the deformation threshold beyond which structural elements are considered to have sustained significant inelastic damage but have not yet reached the ultimate failure level.

According to **EN 1998-3:2005, Clause A.3.2.3(1)**:

$$\theta_{SD} = \frac{3}{4} \cdot \theta_{u(CP)}$$

Where:

- $\theta_{u(CP)}$: Ultimate chord rotation capacity determined per Clause **A.3.2.2** using:
  - Expression (A.1) for total rotation
  - Expression (A.3) for plastic rotation
  - Or alternative expressions (A.4)–(A.9) based on curvature and plastic hinge length.

Yield Rotation – θ_y (Damage Limitation (DL))

The **yield rotation** ϑ_y corresponds to the chord rotation at the **onset of yielding**, used as the deformation limit for the **Damage Limitation (DL)** performance level. It defines the capacity threshold beyond which permanent (plastic) deformations are expected.

According to **EN 1998-3:2005, Clause A.3.2.4**, the yield chord rotation ϑ_y is calculated based on the **geometry, axial force, and reinforcement** condition of the member.

For Beams and Columns (Rectangular Sections)

$$\theta_y = \phi_y \cdot \frac{L_v + a_v \cdot z}{3} + 0.0014\left(1 + 1.5 \cdot \frac{h}{L_v}\right) + \frac{\varepsilon_y}{d - d'} \cdot \frac{d_{bl} \cdot f_y}{6\sqrt{f_c}} \quad (A.10a)$$

Or the simplified alternative expression:

$$\theta_y = \phi_y \cdot \frac{L_v + a_v \cdot z}{3} + 0.0013 + \frac{\varepsilon_y}{d - d'} \cdot \frac{d_{bl} \cdot f_y}{6\sqrt{f_c}} \quad (A.10b)$$

For Walls (Rectangular, T- or Barbelled Sections)

$$\theta y = \phi_y \cdot \frac{L_v + a_v \cdot z}{3} + 0.0013 + \frac{\varepsilon_y}{d - d'} \cdot \frac{d_{bl} \cdot f_y}{6\sqrt{f_c}} \quad (A.11a)$$

Or the simplified alternative expression:

$$\theta y = \phi_y \cdot \frac{L_v + a_v \cdot z}{3} + 0.0013 + \phi_y \cdot \frac{d_{bl} \cdot f_y}{8\sqrt{f_c}} \quad (A.11b)$$

Symbol	Description
ϕy	Yield curvature at the end section
a_vz	Tension shift of the bending moment diagram (EN 1992-1-1:2004, 9.2.1.3(2))
L_v	Shear span (defined as moment/shear ratio M/V at the end section)
a_v	Indicator of shear cracking precedence: av = 1 → If flexural yielding occurs after shear cracking at the end section av = 0 → If flexural yielding occurs before shear cracking
z	Length of internal lever arm: • For beams, columns, or walls with barbelled/T-section: z = d - d' • For rectangular wall sections: z = 0.8h
ε_y	Yield strain of reinforcing steel εy=fy/Es
d_{bl}	Diameter of the longitudinal reinforcement bars
f_y	Yield strength of reinforcing steel (MPa)
fc	Compressive strength of concrete (MPa)
d	Effective depth of the cross-section
d'	Distance from extreme compression fiber to centroid of compression steel

Chord Rotation – Special Considerations for Lap-Spliced and Smooth Bars

For members with **longitudinal bars without lapping** in the end region where yielding is expected:

- If longitudinal bars are **deformed and lapped starting at the end section** (e.g., columns/walls at floor level), expressions A.10 & A.11 must use:
    - A **doubled compression reinforcement ratio** within the lap region.
    - **Yield moment (My)** and **yield curvature (ϕy)** calculated using:

$$f_{y,lap} = f_y \cdot \left(\frac{l_o}{l_{o,min}}\right)$$

Where:

- l_o is lap length,
- $l_{o,min} = 0.3 \cdot d_{bL} \cdot \frac{f_{yL}}{\sqrt{fc}}$
- The **steel yield strain εy** used in A.10a and A.11a is:

$$\varepsilon_y = \frac{f_{yL}}{E_s} \cdot \left(\frac{l_o}{l_{o,min}}\right)$$

- The second term of A.10 / A.11 (anchorage slip) is:
  - **Multiplied by the ratio of modified My** (with lap) to **unmodified My** (outside lap region).
- a_V^z term's contribution is determined by comparing:

$$L_v V_{R,c}\ , M_y^{(lap)}$$

If $My(lap) > Lv \cdot VR, cM_y^{(lap)} > L_v \cdot V_{R,c}$ then av = 1 otherwise av=0.

Beam-Column Joint Verification

To assess the shear safety of beam-column joints under seismic actions using demand values from structural analysis and capacity values derived according to **EN 1998-1:2004** and **EN 1998-3:2005**.

Shear Demand Calculation (Vj,Ed)

The design shear demand on the joint core is determined based on:

- Most adverse effects from seismic actions
- Capacity design conditions for beams (EN 1998-1 §5.5.2.3(1)P)
- Lowest compatible shear values in other framing elements
- Overstrength factor γRd to consider strain hardening
  γRd = 1.2 (EN 1998-1 §5.5.2.3(2))

Formula:

$$\mathrm{V_{j,Ed}} = \gamma_{\mathrm{Rd}} \cdot \mathrm{F_{yd}} \cdot (\mathrm{A_{s1}} + \mathrm{A_{s2}}) - \mathrm{V_c}\ (5.22)$$

- A_{s1},As2 Beam longitudinal reinforcement (top and bottom)
- F_{yd}: Design yield strength of steel
- Vc: Column shear from analysis

Shear Capacity Calculation (Vj,Rd)

a) Confinement Provided (EN 1998-1 5.5.3.3(1)):

$$V_{j,Rd,max} = \eta \cdot f_{cd} \cdot \left(1 - \frac{v_d}{\eta}\right)^{1/2} \cdot b_j \cdot h$$

b) Without Confinement (EN 1998-1 5.5.3.3(2)b):

$$V_{j,Rd,max} = 0.8 \cdot \eta \cdot f_{cd} \cdot \left(1 - \frac{v_d}{\eta}\right)^{1/2} \cdot b_j \cdot h_c$$

With:

- $\eta = 0.6 \cdot \left(1 - \frac{f_{ck}}{250}\right)$ Concrete confinement factor based on fck (EN 1998-1 §5.5.3.3)
- $f_{cd} = \frac{f_{ck}}{\gamma_c}$, design compressive strength of concrete
- $v_d = \frac{N_{Ed}}{f_{cd} \cdot b \cdot h}$ normalized axial load
- b_j: Effective joint width (EN 1998-1 §5.5.3.3)
- h_c: Column cross-section depth

Verification Criterion

The joint satisfies shear verification if:

$$V_{j,Ed} \leq V_{j,Rd,max}$$

- For **interior joints**: both A_{s1} and A_{s2} considered,for **exterior joints**: only the contributing rebar side.Use **conservative values** for reinforcement areas where detailed information is unavailable

References

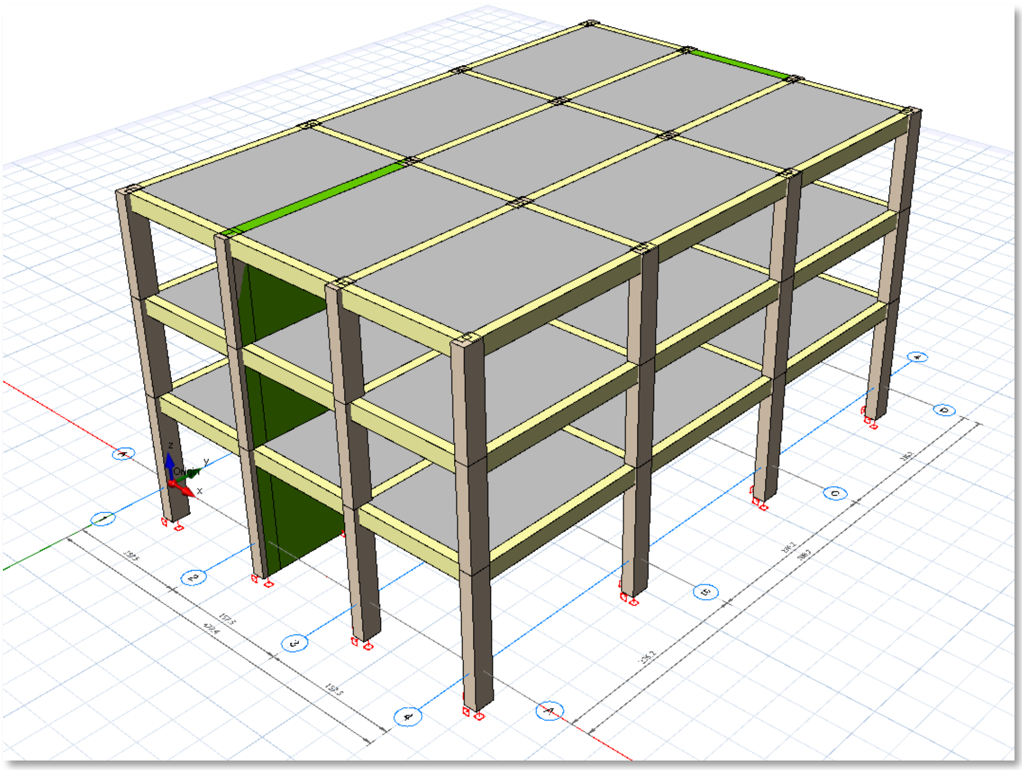
- **EN 1998-1:2004**
  - §5.5.2.3 – Beam-column joints: shear demand
  - §5.5.3.3 – Beam-column joints: shear capacity
- **EN 1998-3:2005**
  - §A.3.4 – Beam-column joints in assessment context

Application of Eurocode 8 – Part 3 in ProtaStructure

Structural Model Properties

Number of Stories		3
Storey Height		3 x 3 m = 9 m
Direction X Span Length		3 x 400 cm = 12 m
Direction Y Span Length		3 x 600 cm = 18 m
Column	Width	50 cm
	Depth	25 cm
	Concrete Material	C320
	Steel Material	SD60
Beam	Width	30 cm
	Depth	50 cm
	Concrete Material	C300
	Steel Material	SD60
Wall	Width	6 m
	Depth	40 cm
	Concrete Material	C300
	Steel Material	SD60
	Longitudinal Web Bar	SD60
	Horizontal Web Bar	SD60

A three-dimensional view of the model is also shown below.



Existing Reinforcement

The reinforcement details of the identified columns and walls are presented in the table below. Before any performance assessment, you must define the reinforcement in the RC members. There are two ways to accomplish this:

1. Defining detailed reinforcements via RC column, shearwall and beam design module
2. Defining reinforcements by estimated approximate ratios

Defining Reinforcements with RC Design Module

You can use the RC design module in ProtaStructure to specify the longitudinal and transverse reinforcement to each member. You may be using this approach for following different reasons:

1. **Performance assessment of an existing building:** If the blueprints are available and you know the exact reinforcements in each member with reasonable site verification.
2. **Performance-based design of a new building:** You performed a preliminary design and designed the reinforcements yourself before validating the performance.

Remark:

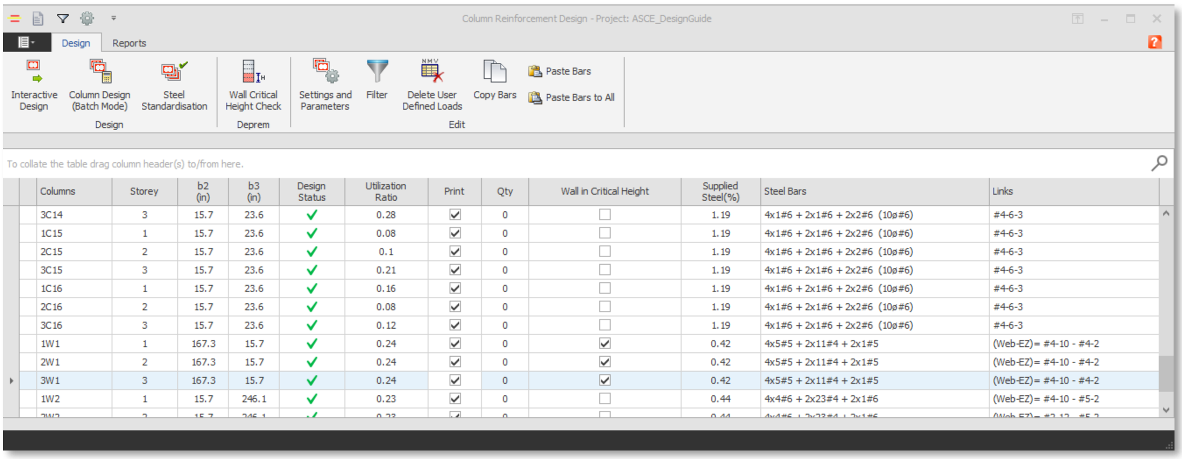
If you are specifying the detailed reinforcements for an existing building, the design status in the design modules may indicate a FAILURE with a RED CROSS sign. Do not worry about this and make sure that you have entered longitudinal and transverse reinforcement as you see in the bluprints. Click OK to exit the design module WITHOUT trying to redesign any members. If you attempt to redesign the members, ProtaStructure will attempt to reselect reinforcement which is not what we want.

If you are specifying the detailed reinforcement for performance-based design of a new building, then you must perform a preliminary design with proper parameters and make ProtaStructure select the reinforcement for you. In this case you can expect the design status to be SUCCESS with GREEN TICK sign.

Information:

Once you define the detailed reinforcement for a particular member, you can copy and paste the reinforcement information to all similar members.

In this example study, a **uniform reinforcement layout** was assigned to all column elements to simplify the evaluation process. This approach aims to demonstrate the application of acceptance criteria independently of the specific demand on each individual element.



Defining Approximate Reinforcement Ratios

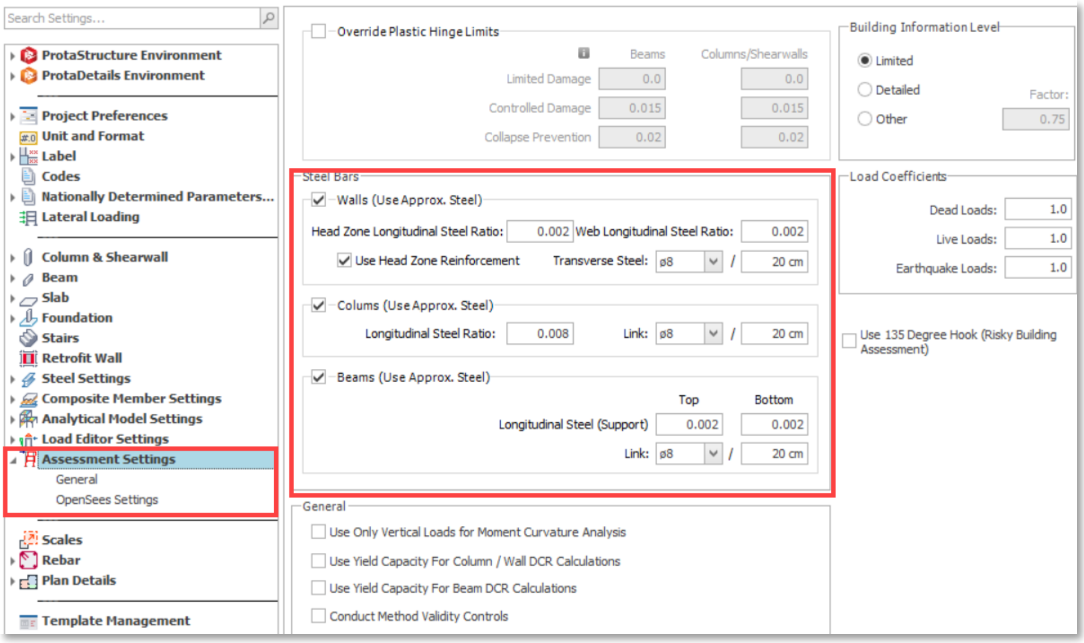
In case you don't have sufficient information on the detailed reinforcements in the members, you can always use 'Estimated Reinforcement' for member groups or individual members. Of course, this estimation must be backed by the site investigations and/or the design codes that were in effect when the building was designed.

Defining Approximate Reinforcement for All Members

To specify estimated reinforcement ratio for all members

1. Navigate to Assessment Settings on the Options menu

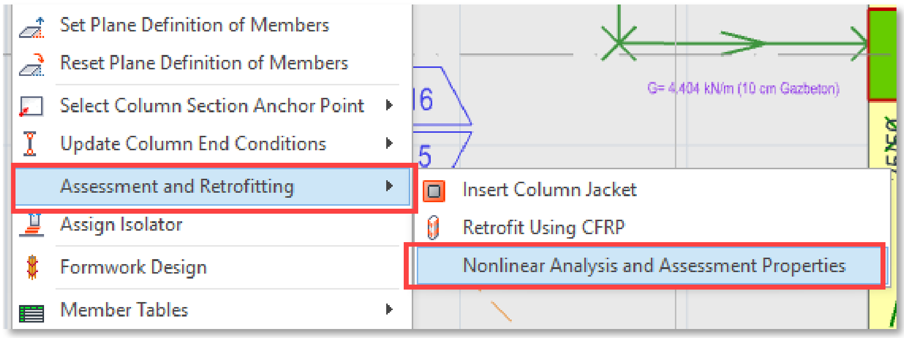
2. Enable the estimated bar ratios for Walls, Columns and/or beams.



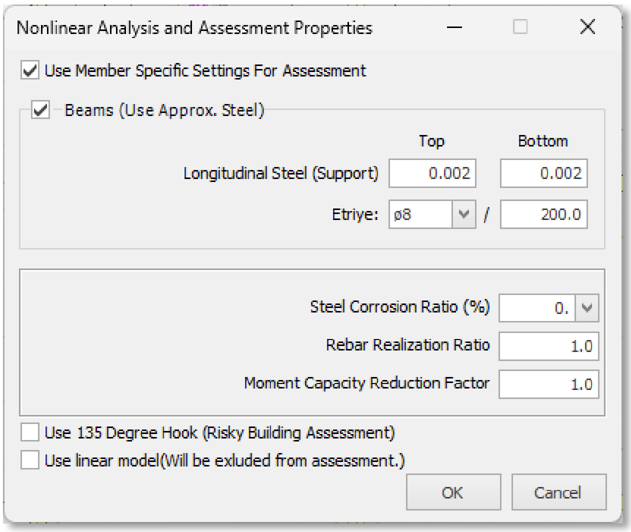
Defining Approximate Reinforcements for Individual Members

You can also assign **approximate reinforcement values to individual members**.

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



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



Important Note

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

<u>The approximate reinforcement assigned to individual members</u> have the priority in the assessment analysis. If no approximate reinforcement is assigned to individual members, then the globally defined values in the assessment settings <u>for the entire building</u> will be used. If global settings do not specify

any estimated reinforcement, then the actual detailed reinforcements on the members defined via design menus will be used.

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

Second Priority: Globally Approximate Reinforcement Ratios in the "**Settings > Assessment Settings**"

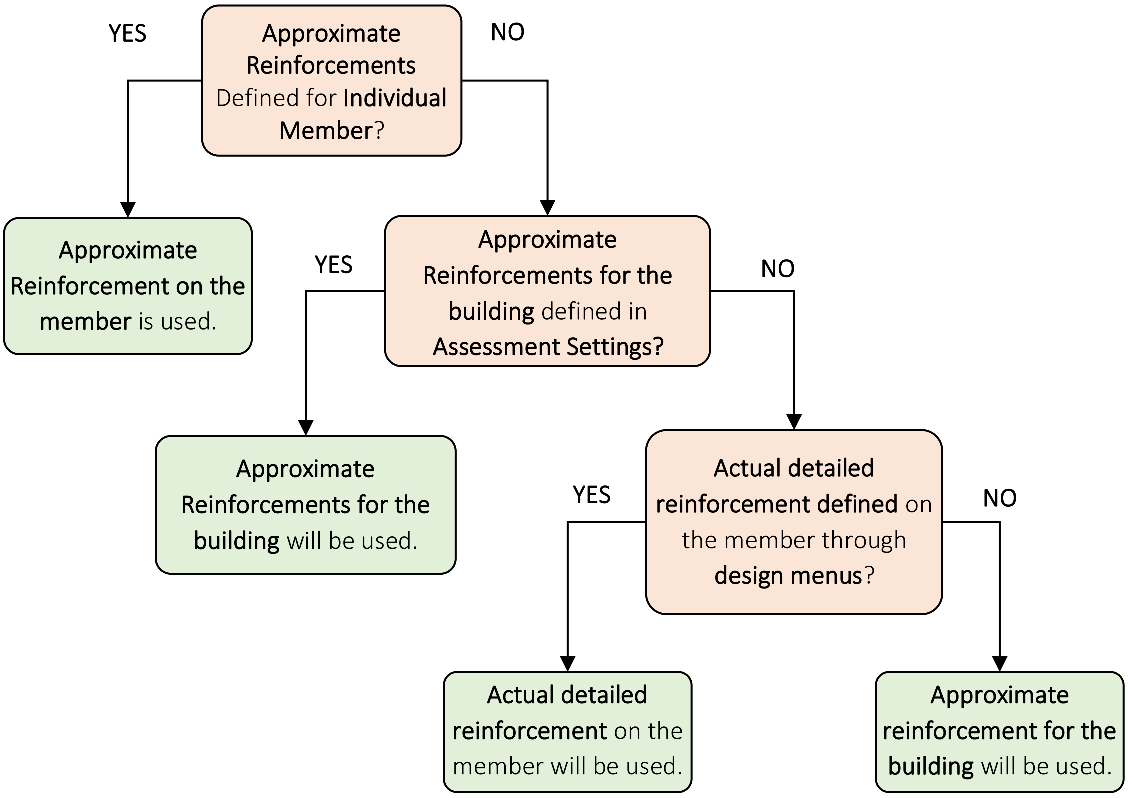
Third Priority: Actual detailed reinforcements defined through design menus directly on the member itself.

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

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

Flowchart for Estimated and Detailed Reinforcement Usage

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

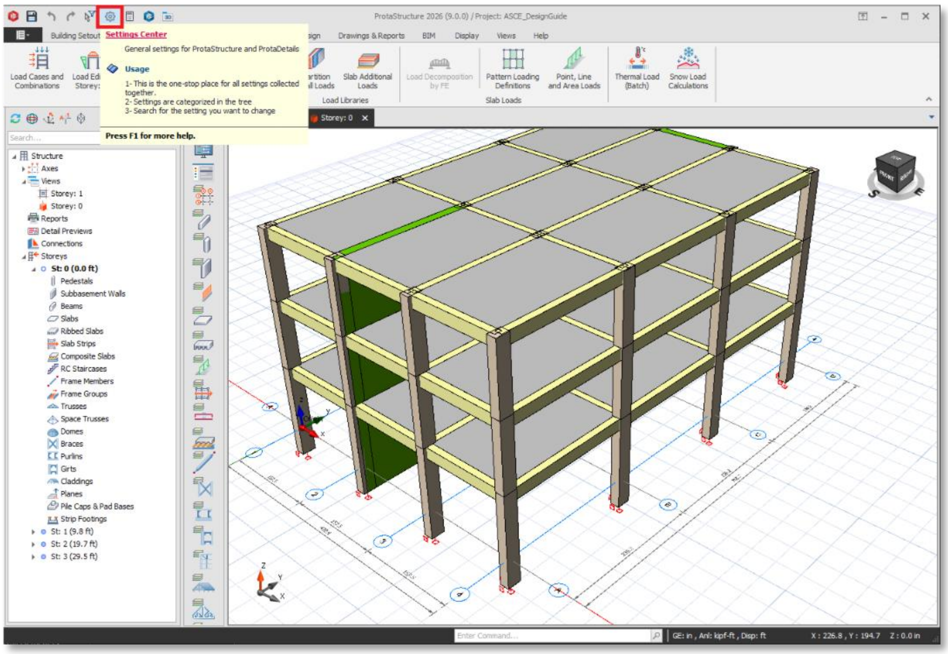


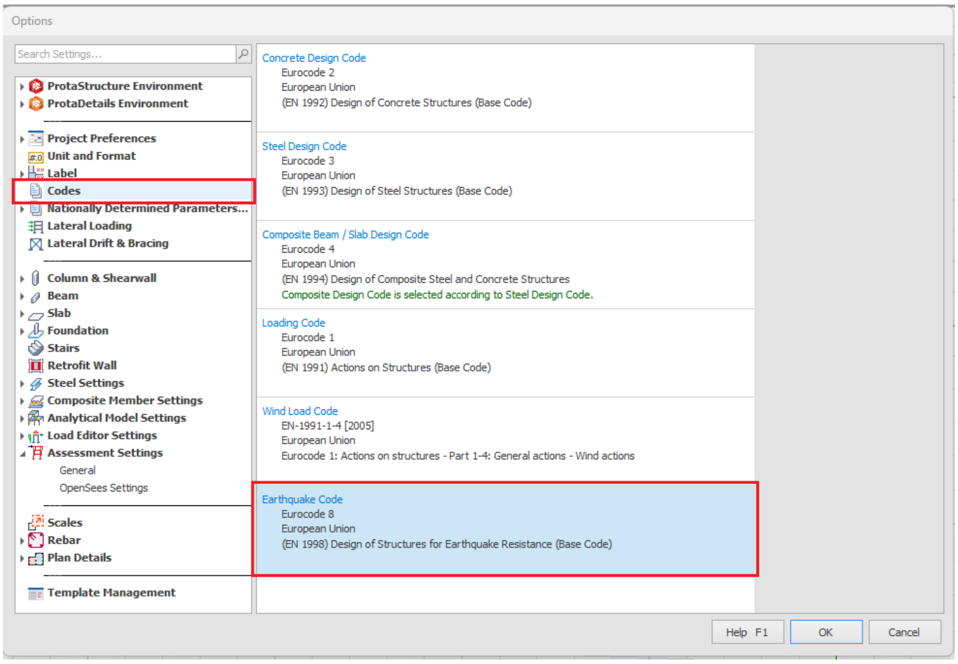
Earthquake Code

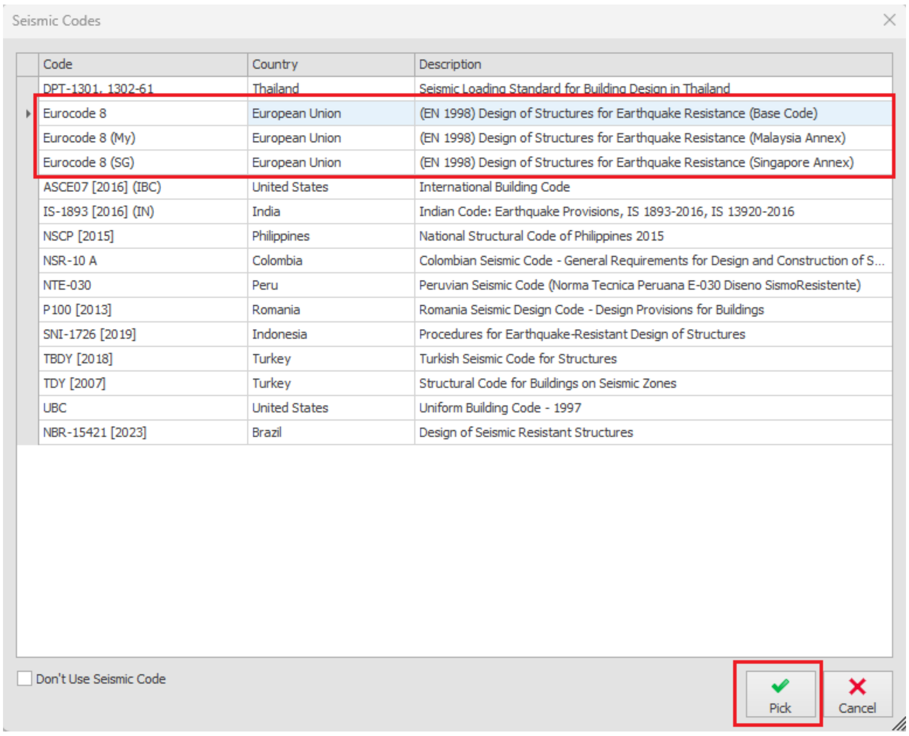
ProtaStructure supports a wide range of seismic design codes required for performance-based evaluations. To perform an analysis in accordance with the **Eurocode 8 Part 3** standard, follow the steps below:

1. Click on the **Settings** tab in the main menu.
2. In the dialog window, navigate to the **Codes** section.
3. From the **Earthquake Code** dropdown, select **Eurocode (8,My,SG)**

Applying these settings enables the necessary **seismic parameters** and **performance objectives** required for evaluating existing buildings under the **Eurocode 8 Part 3** framework.







Code	Country	Description
DPT-1301, 1302-61	Thailand	Seismic Loading Standard for Building Design in Thailand
Eurocode 8	European Union	(EN 1998) Design of Structures for Earthquake Resistance (Base Code)
Eurocode 8 (My)	European Union	(EN 1998) Design of Structures for Earthquake Resistance (Malaysia Annex)
Eurocode 8 (SG)	European Union	(EN 1998) Design of Structures for Earthquake Resistance (Singapore Annex)
ASCE07 [2016] (IBC)	United States	International Building Code
IS-1893 [2016] (IN)	India	Indian Code: Earthquake Provisions, IS 1893-2016, IS 13920-2016
NSCP [2015]	Philippines	National Structural Code of Philippines 2015
NSR-10 A	Colombia	Colombian Seismic Code - General Requirements for Design and Construction of S...
NTE-030	Peru	Peruvian Seismic Code (Norma Tecnica Peruana E-030 Diseno SismoResistente)
P100 [2013]	Romania	Romania Seismic Design Code - Design Provisions for Buildings
SNI-1726 [2019]	Indonesia	Procedures for Earthquake-Resistant Design of Structures
TBDY [2018]	Turkey	Turkish Seismic Code for Structures
TDY [2007]	Turkey	Structural Code for Buildings on Seismic Zones
UBC	United States	Uniform Building Code - 1997
NBR-15421 [2023]	Brazil	Design of Seismic Resistant Structures

By completing this step, **ProtaStructure** is configured to support both the definition of seismic parameters and the evaluation of existing buildings in accordance with the **Eurocode 8** standard. This ensures that all relevant settings align with performance-based design principles specified in the guideline.

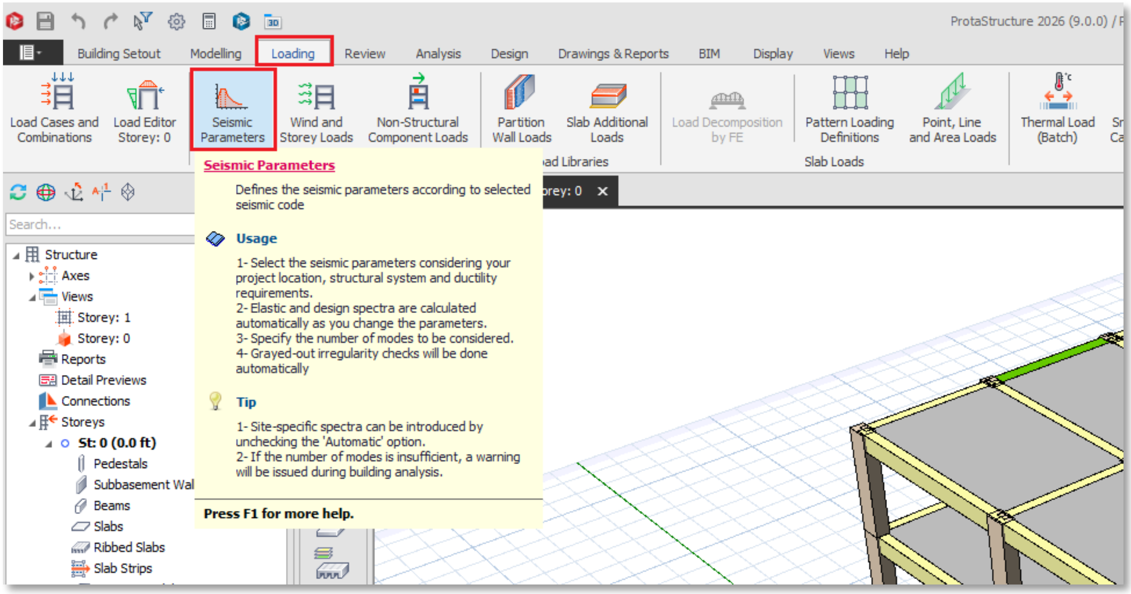
Important Note:

To carry out an assessment according to Eurocode 8, make sure that Eurocode 8 is selected under the Earthquake Code settings.

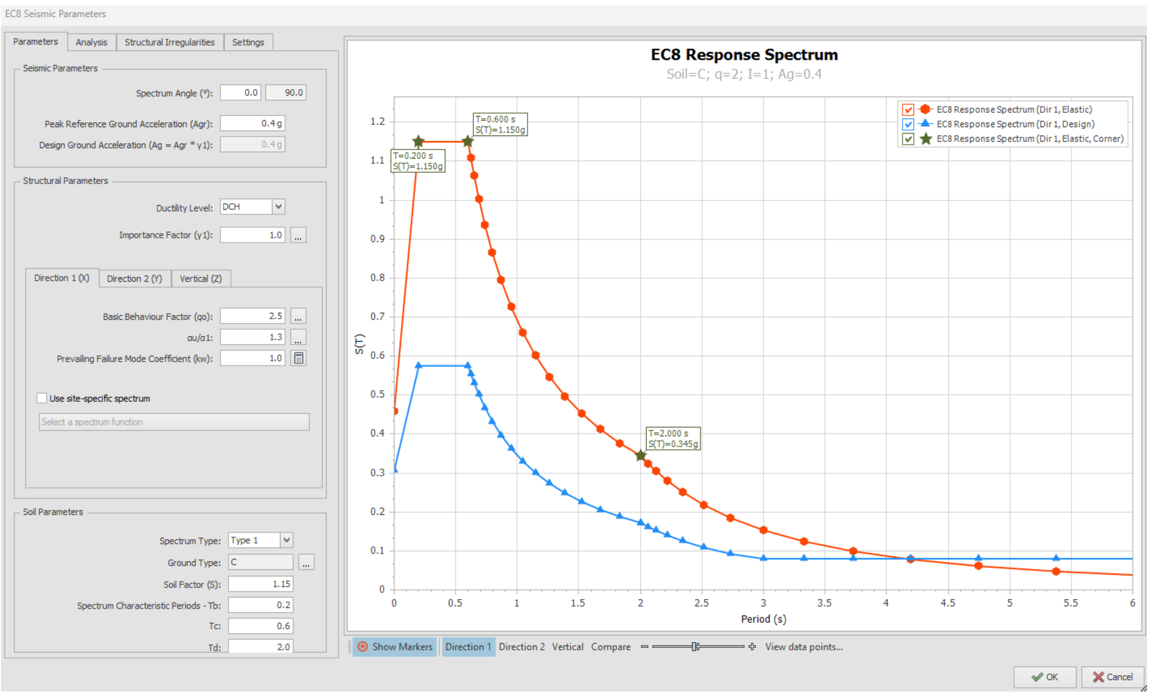
Seismic Parameters

In **ProtaStructure**, **seismic parameters** are defined under the **Loading** tab by selecting the **Seismic Parameters** section.

The figure below illustrates how to access the seismic parameters interface step by step.



After clicking the **Seismic Parameters** button, the interface shown below will appear.

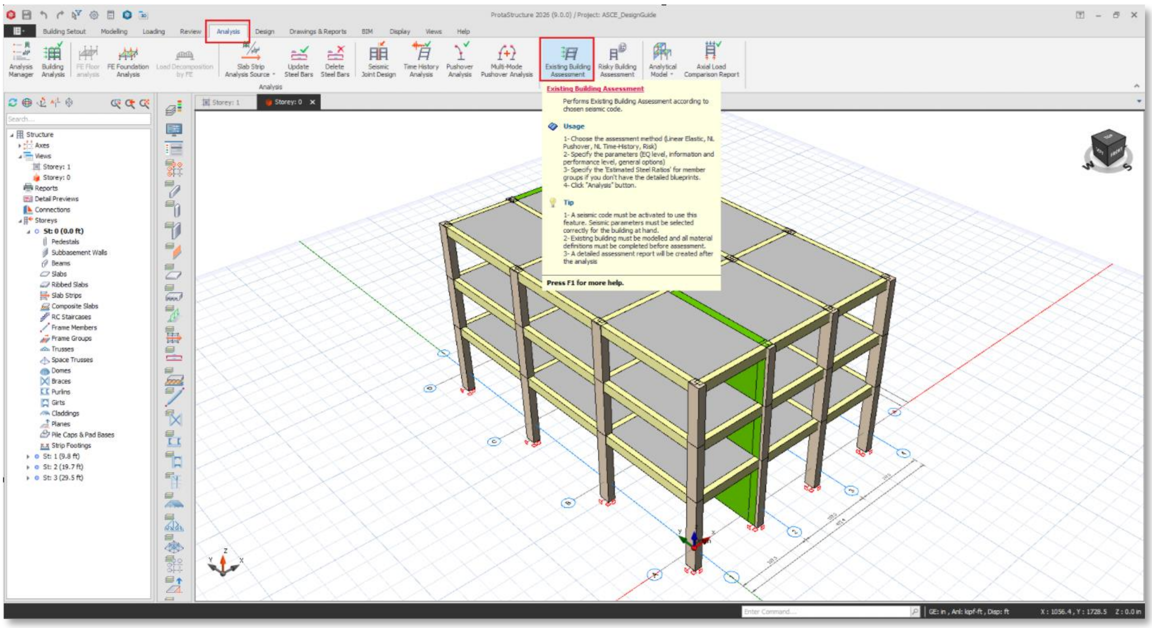


In this interface, seismic hazard levels and related earthquake parameters are defined as part of the existing building assessment process.

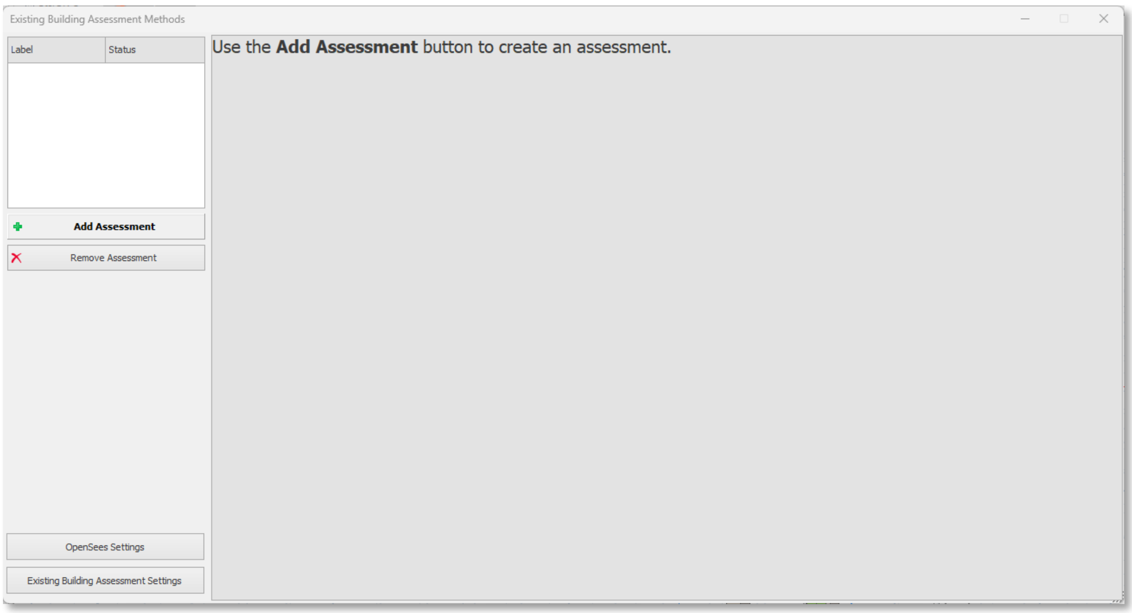
Additionally, this section is used to define key inputs such as response spectrum parameters, design spectral acceleration value (Agr), and the Site Class of the structure.

Existing Building Assessment

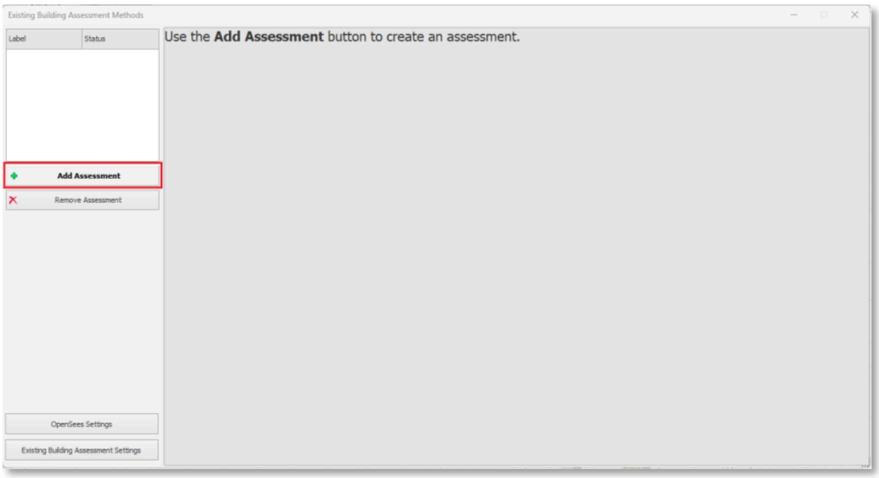
This section explains how to perform an **existing building assessment** step by step within **ProtaStructure**. To begin the process, go to the **Analysis** tab and click on the **Existing Building Assessment** button. The figure below illustrates where this option is located in the user interface.



After clicking the button, a new interface will appear where you can perform operations such as **creating or deleting assessment definitions**. This interface also includes access to **OpenSees settings** and **existing building assessment settings**. A detailed explanation of the **existing building assessment settings** will be provided in the following sections.

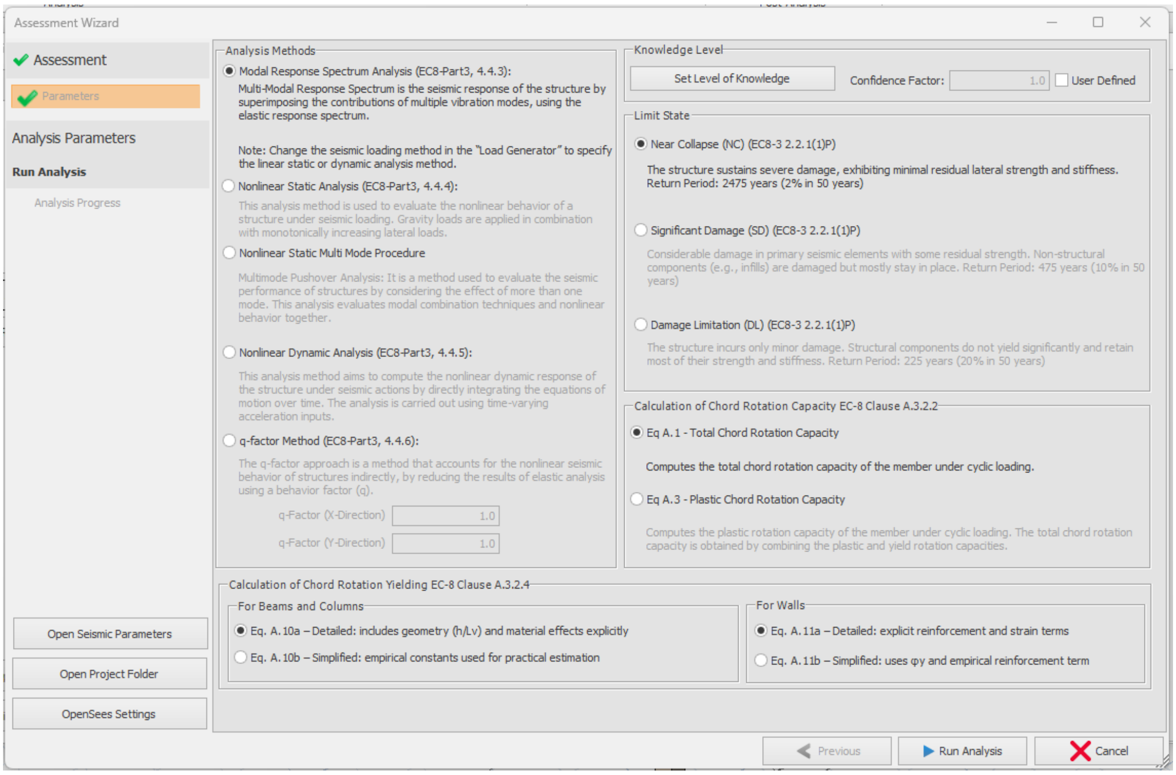


The **Label Status** section displays all previously created assessment scenarios. To create a new assessment, click the **Add Assessment** button.



After clicking the **Add Assessment** button, the **Assessment Wizard** interface will appear.

Assessment Wizard



After launching the **Assessment Wizard**, the first screen allows the user to define the **analysis method**, **knowledge factor**, and the target **limit state** for the assessment.

On the left side of the screen, the navigation panel lists the steps of the assessment process:

- Parameters
- Analysis Parameters
- Run Analysis

This panel allows users to progress through the assessment workflow in a structured and sequential manner.

Eurocode 8 Analysis Methods and Parameters

In this section, the user must select one of the **analysis methods defined in EN 1998-3:2005, Clause 4.4**, for evaluating the seismic performance of the structure:

- **Lateral Force Analysis (EC8-Part3, 4.4.2)**
  A linear elastic method that assumes the response is governed by the fundamental vibration mode. Seismic forces are applied as lateral loads distributed along the building height based on mass and mode shape.

- **Modal Response Spectrum Analysis (EC8-Part3, 4.4.3)**
  Multi-Modal Response Spectrum is the seismic response of the structure by superimposing the contributions of multiple vibration modes, using the elastic response spectrum.

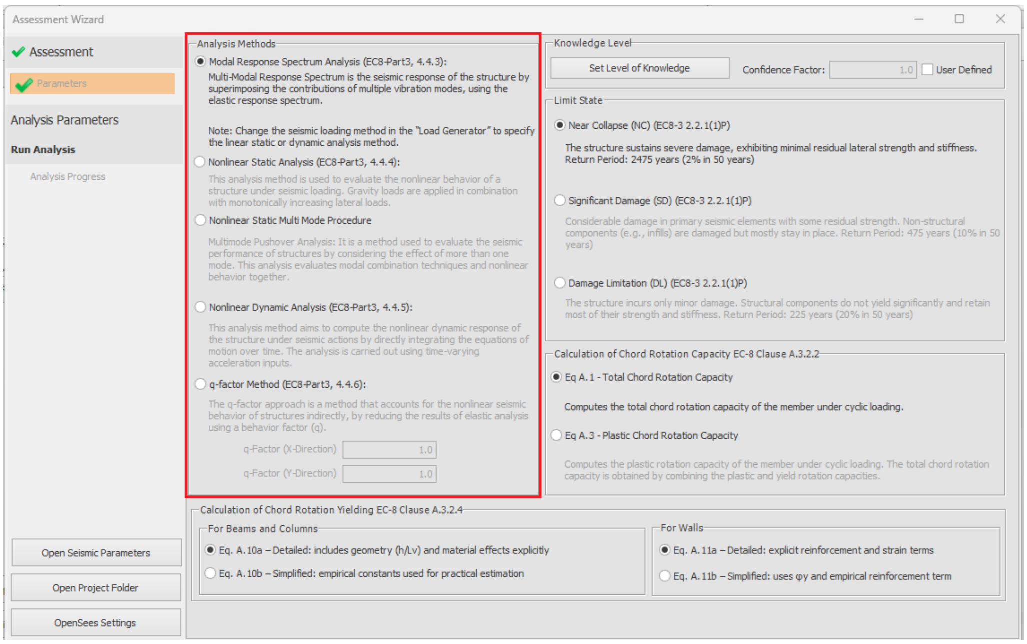
- **Nonlinear Static Analysis (EC8-Part3, 4.4.4)**
  Performs pushover analysis by applying monotonically increasing lateral forces, capturing nonlinear behavior and identifying plastic hinge formation and capacity curve.

- **Nonlinear Static Multi Mode Procedure**
  A ProtaStructure-specific approach that combines pushover analysis with multiple vibration modes to reflect modal contributions more accurately.
  Note: This method is not officially defined in EN 1998-3, but is implemented for enhanced nonlinear evaluation.

- **Nonlinear Dynamic Analysis (EC8-Part3, 4.4.5)**
  Time history analysis using recorded or synthetic accelerograms. Captures full nonlinear time-domain behavior of the structure under seismic loading.

- **q-factor Method (EC8-Part3, 4.4.6)**
  Allows reduced elastic analysis using a behavior factor (q) to account for energy dissipation through inelastic mechanisms. The user may assign different q-factors in X and Y directions.

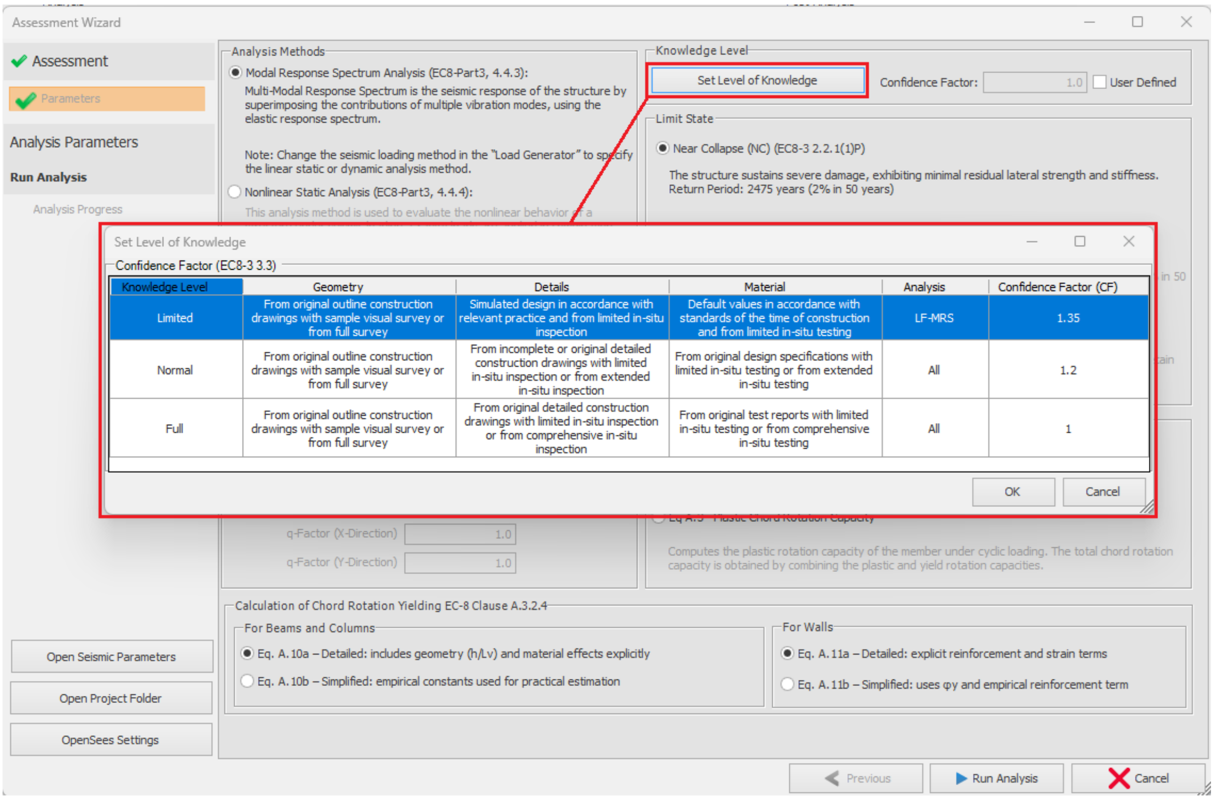
Important Note:
Seismic load method selected in the *Load Generator* must be consistent with the chosen analysis type (e.g., "Equivalent Static" for LFA or "Modal Response Spectrum" for Linear Methods).

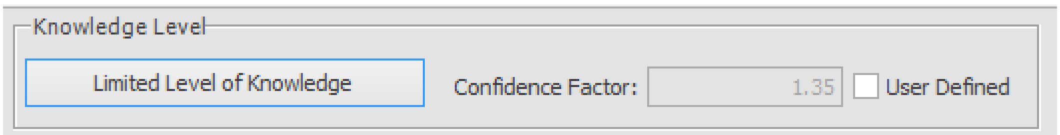


Knowledge Level and Confidence Factor

The **Knowledge Level** reflects the amount and quality of information available for the structure, including geometry, detailing, and material properties. It affects the **Confidence Factor (CF)** applied to capacity calculations.

- Use the **Set Level of Knowledge** button to assign one of the standard knowledge levels:
  - **KL1** – Limited knowledge → CF = 1.35 (Linear only)
  - **KL2** – Normal knowledge → CF = 1.20
  - **KL3** – Full knowledge → CF = 1.00
- Optionally, enable the **User Defined** checkbox to enter a custom Confidence Factor manually.





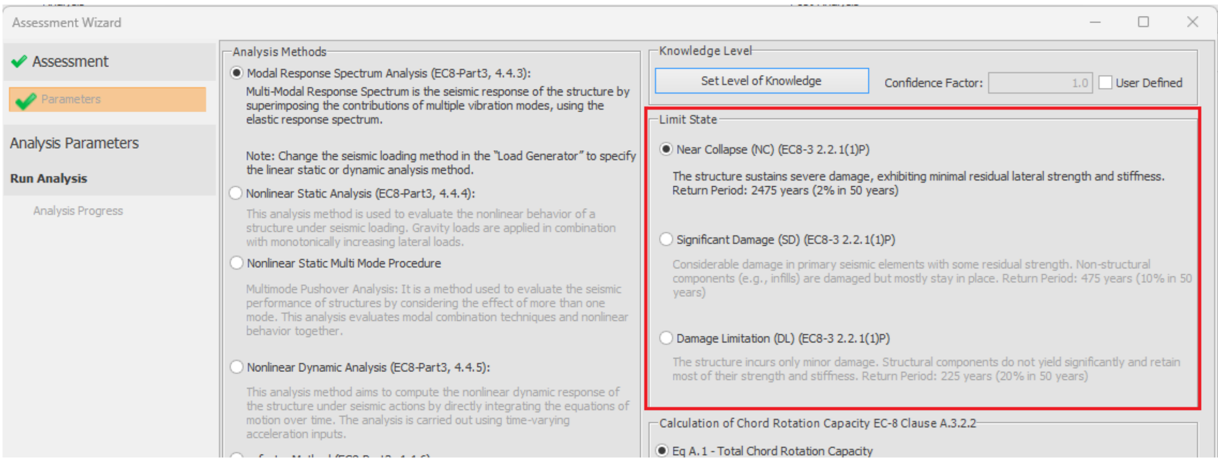
Limit State Selection

The user must select one target **Performance Level** for the assessment:

- **Near Collapse (NC)**
  Severe structural damage with minimal residual lateral capacity. Return period: **2475 years (2% in 50 years)**
- **Significant Damage (SD)**
  Considerable inelastic damage with limited strength retention. Return period: **475 years (10% in 50 years)**

- Damage Limitation (DL)
  Minor damage, structure remains largely elastic and usable. Return period: 225 years (20% in 50 years)

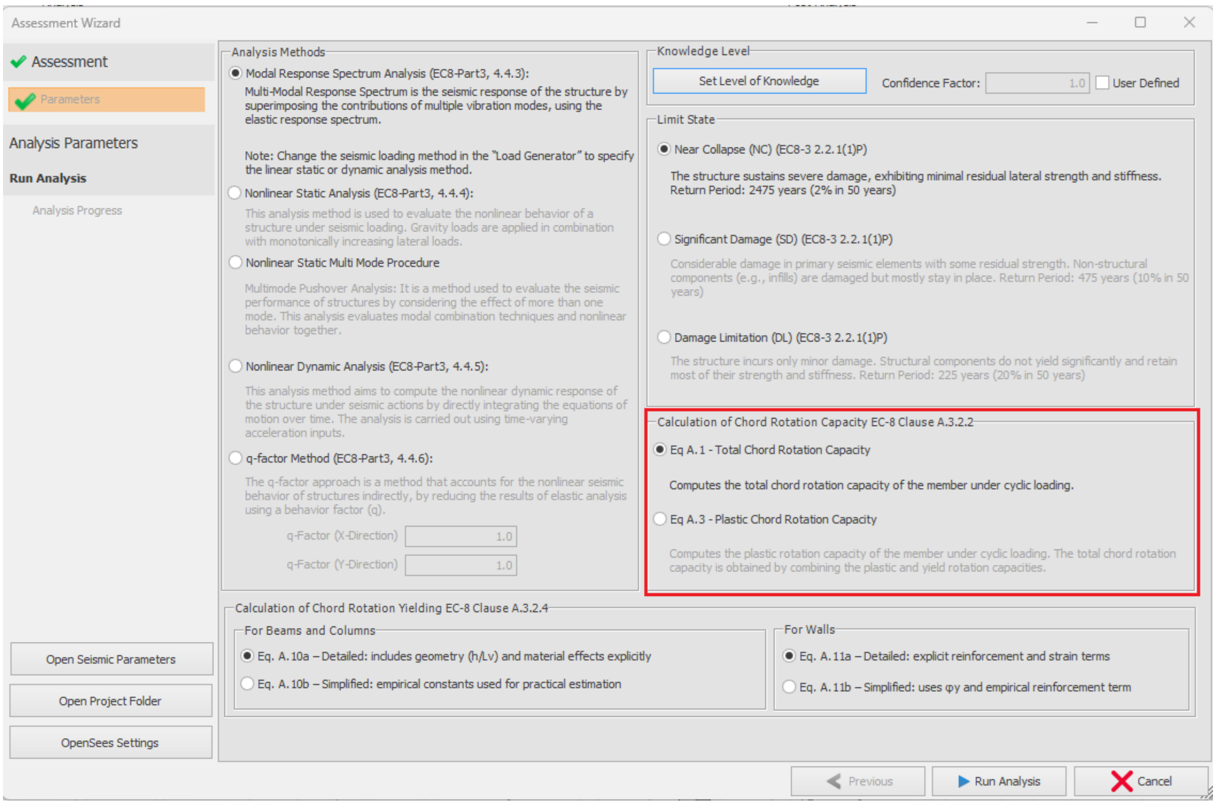
These limit states correspond to specific seismic hazard levels defined by the national annex or Eurocode default return periods.



Calculation of Chord Rotation Capacity (EN 1998-3, Clause A.3.2.2)

The user must choose how chord rotation capacity will be calculated:

- Eq. A.1 – Total Chord Rotation Capacity
  Includes both elastic and plastic deformation.
- Eq. A.3 – Plastic Chord Rotation Capacity
  Only the plastic portion of chord rotation is calculated separately.



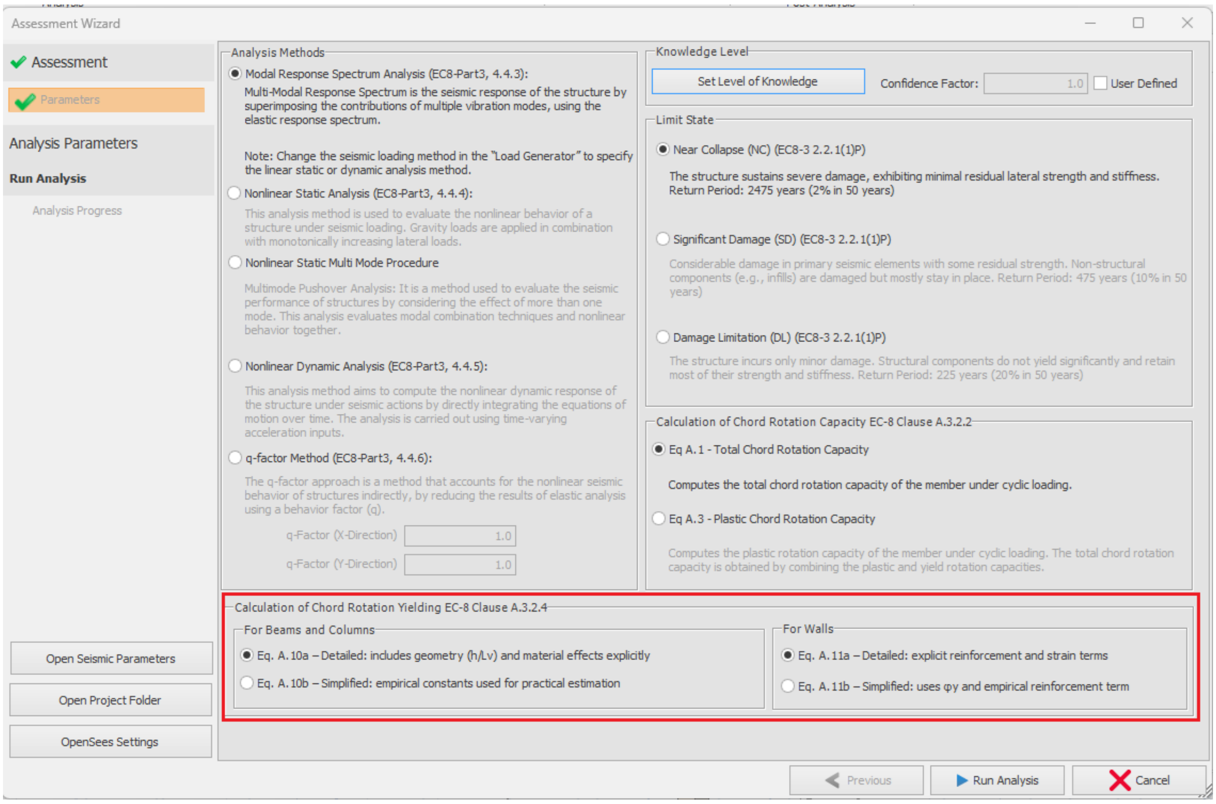
Yield Rotation Calculation (Clause A.3.2.4)

For **Beams and Columns**:

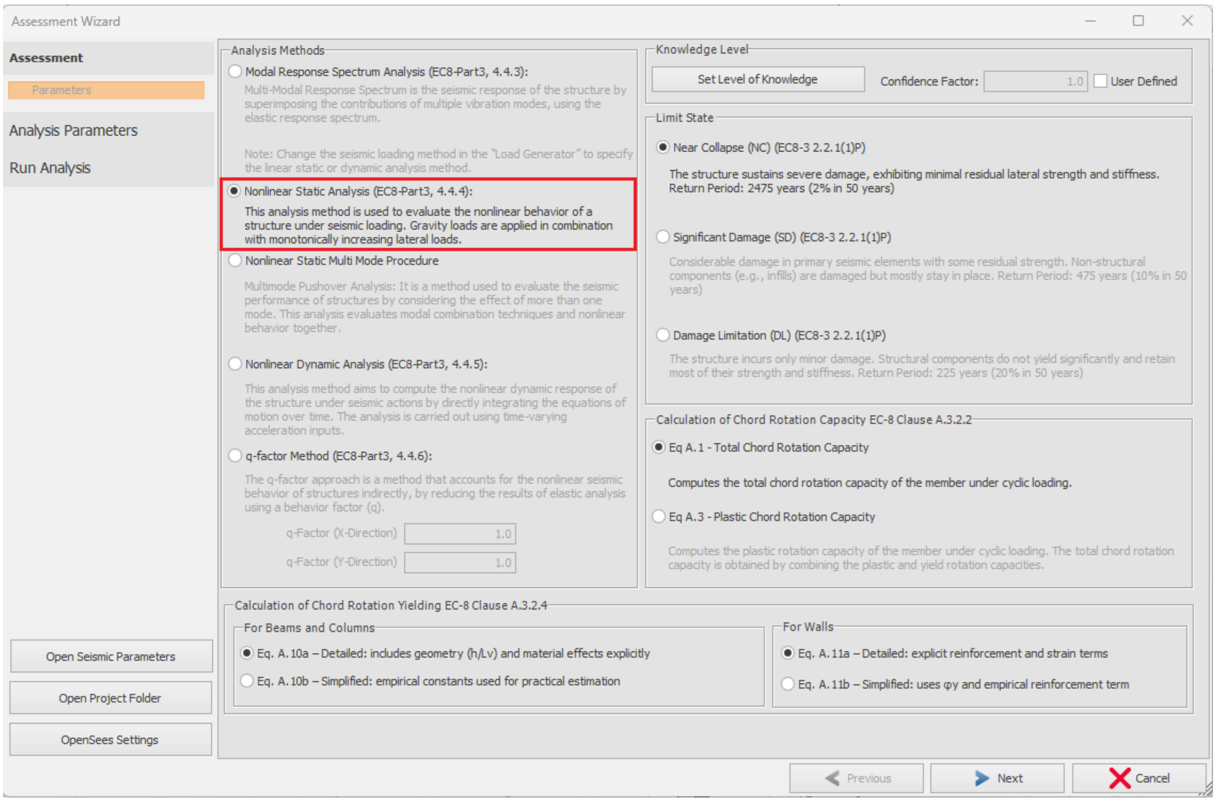
- **Eq. A.10a – Detailed method:** uses geometry (h/Lv) and explicit material parameters.
- **Eq. A.10b – Simplified method:** uses empirical constants for estimation.

For **Walls**:

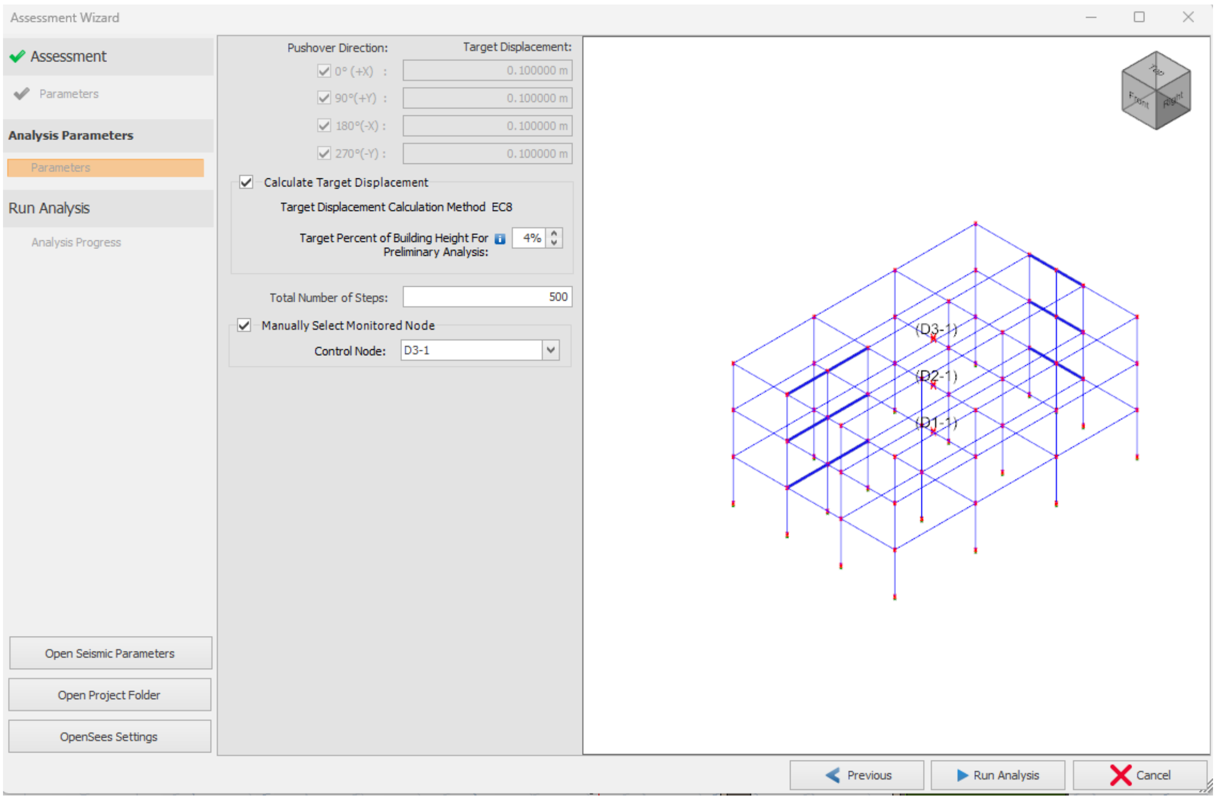
- **Eq. A.11a – Detailed method:** includes explicit reinforcement and strain parameters.
- **Eq. A.11b – Simplified method:** uses empirical reinforcement ratio and default values.



Nonlinear Static (Pushover) Analysis



After selecting **Nonlinear Static Procedure** in the **Analysis Methods** section and clicking the **Next** button, the following interface appears for defining analysis parameters.

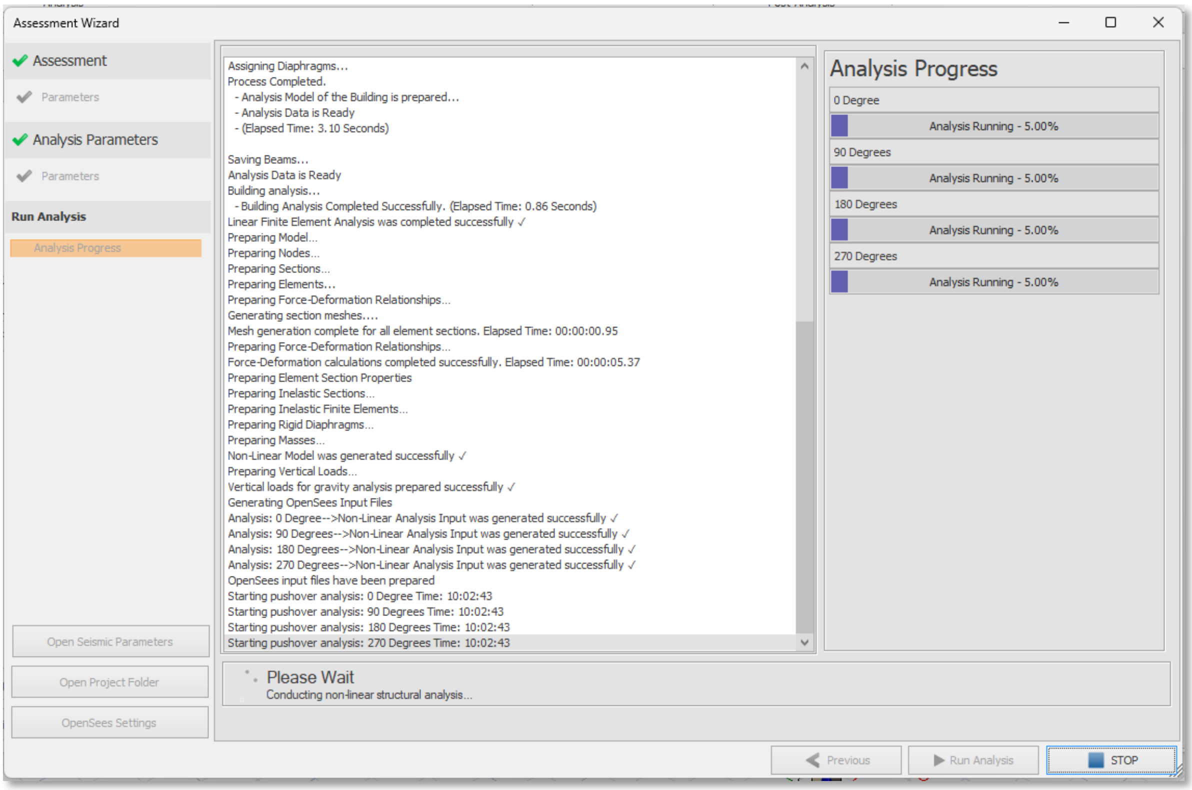


This screen is used to configure the nonlinear static (pushover) analysis based on **Eurocode 8.**

- Pushover Directions:
  Select the directions in which the pushover analysis will be performed (0° +X, 90° +Y, 180° -X, 270° -Y).
- Target Displacement Calculation:
  Target displacements are automatically calculated as per Eurocode 8.
  The *Target Percent of Building Height* defines the assumed displacement value for preliminary analysis (commonly 4%).
- Total Number of Steps:
  Defines the number of load increments used in the analysis.
- Control Node Selection:
  The node used to monitor lateral displacement. The pushover curve is generated based on this node's movement.

The 3D model view on the right displays the location of the selected control node.

After clicking the **Run Analysis** button, the interface transitions to the **Run Analysis** screen, where the nonlinear analysis process begins and its progress can be monitored. This screen executes the pushover analysis in the specified directions (0°, 90°, 180°, 270°).



Analyis log section displays a detailed log of the ongoing analysis process, including:

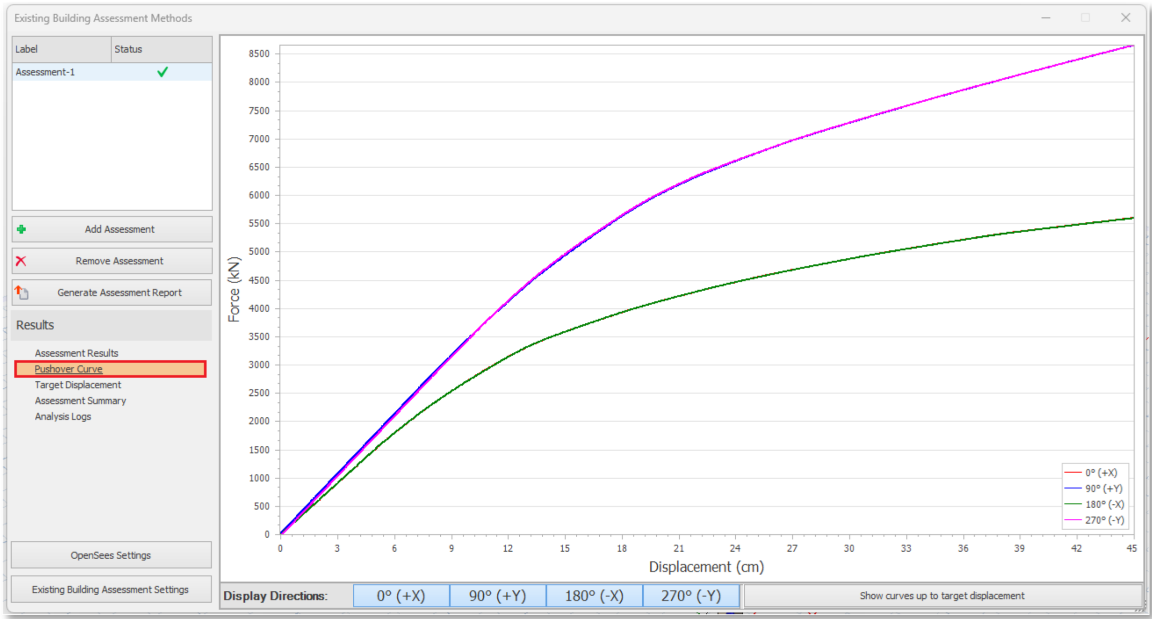
- Preparation of the structural analysis model
- Generation of section and force-deformation relationships
- Mesh creation
- OpenSees input file generation
- Launching pushover analysis for each selected direction.

Analysis progress shows real-time progress bars for each selected analysis direction (0°, 90°, 180°, 270°). It provides a quick overview of the current status and completion percentage of the pushover analysis in each direction.

At the bottom of the screen, the analysis status is displayed. Users can monitor the overall progress and, if needed, manually **stop the analysis** using the **STOP** button.

Nonlinear Static (Pushover) Analysis – Pushover Curve Screen

When the user selects **Pushover Curve** from the left-side navigation panel, the interface displays the **capacity curves** (pushover curves) for all selected loading directions.



These curves represent the **base shear–displacement relationship** and illustrate how the structure behaves under increasing lateral loads.

- The horizontal axis shows **lateral displacement** (cm)
- The vertical axis shows **base shear force** (kN)
- Each curve represents a specific **pushover direction**:
  - Red: 0° (+X)
  - Blue: 90° (+Y)
  - Green: 180° (–X)
  - Magenta: 270° (–Y)

Curves can be toggled individually using the **direction buttons** at the bottom of the screen. By default, curves are shown **up to the target displacement** calculated during the analysis.

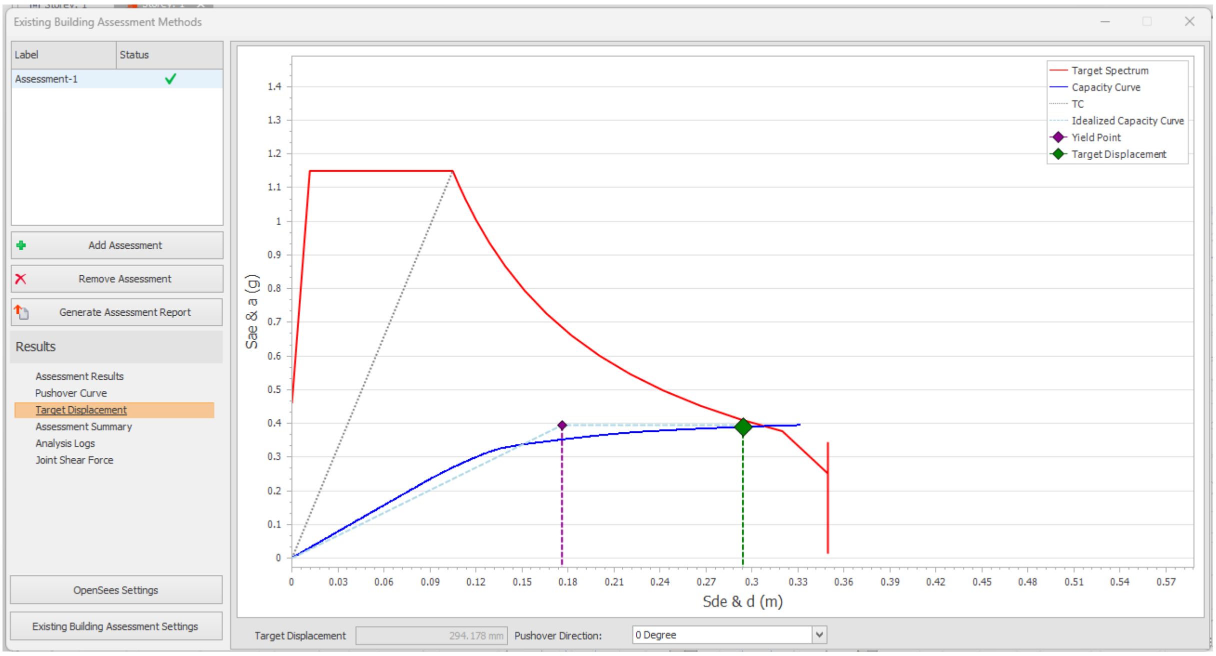
Tip:

These graphs allow users to evaluate the global capacity of the structure, identify yielding zones, and compare the performance of different loading directions.

Nonlinear Static (Pushover) Analysis – Target Displacement

When the user clicks on **Target Displacement** from the navigation panel on the left, a graph is displayed showing the **pushover curve**, the **idealized bilinear curve**, and the corresponding **target displacement point** for the selected direction.

This graph visually represents the location of the **target displacement**, which is used as the reference point for performance evaluation according to **Eurocode 8**.



Once the nonlinear static (pushover) analysis is completed and the user selects

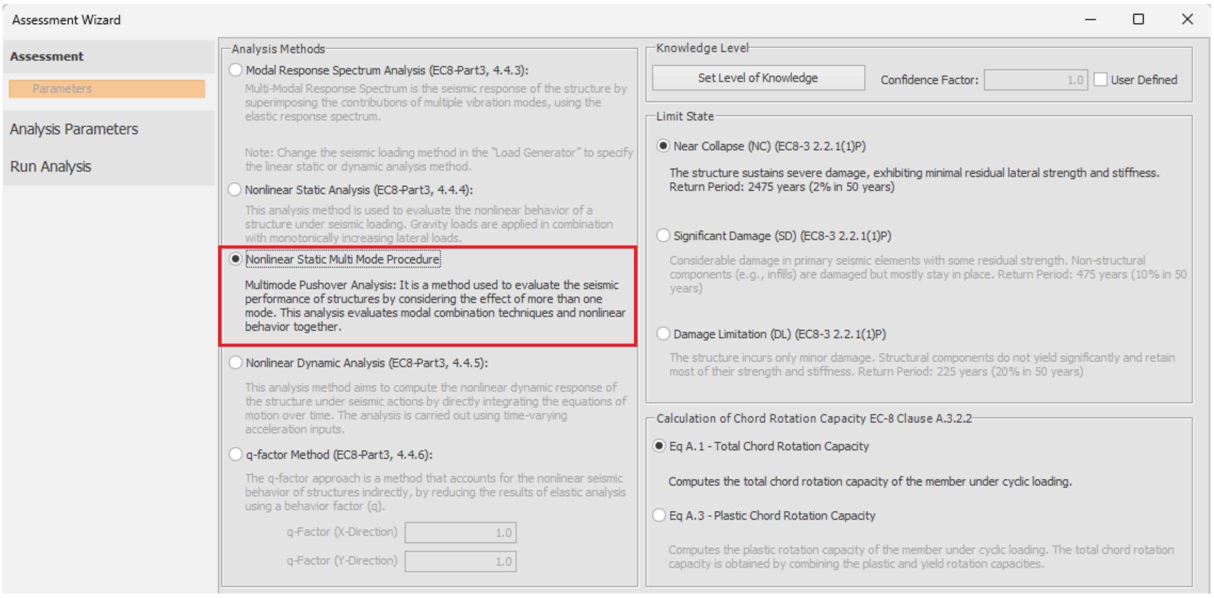
Results → Target Displacement,
the software displays the following elements in a single, interactive graph:

- **Target Spectrum (Red Curve)**: Elastic response spectrum derived based on user-defined seismic parameters.
- **Capacity Curve (Blue Curve)**: Base shear vs. displacement curve generated from the pushover analysis.
- **Idealized Capacity Curve (Dashed Line)**: Bilinear representation used to define yield and ultimate behavior.
- **Yield Point (Purple Marker)**: Identified from the idealized curve; used to determine ductility and performance levels.
- **Target Displacement Point (Green Marker)**: Computed as per **EC8-3 Annex B**. It is the control displacement for performance checks.
- **Corner Period (TC) Line (Gray Dashed)**: Used in the target displacement calculation to determine elastic vs. inelastic demand region.

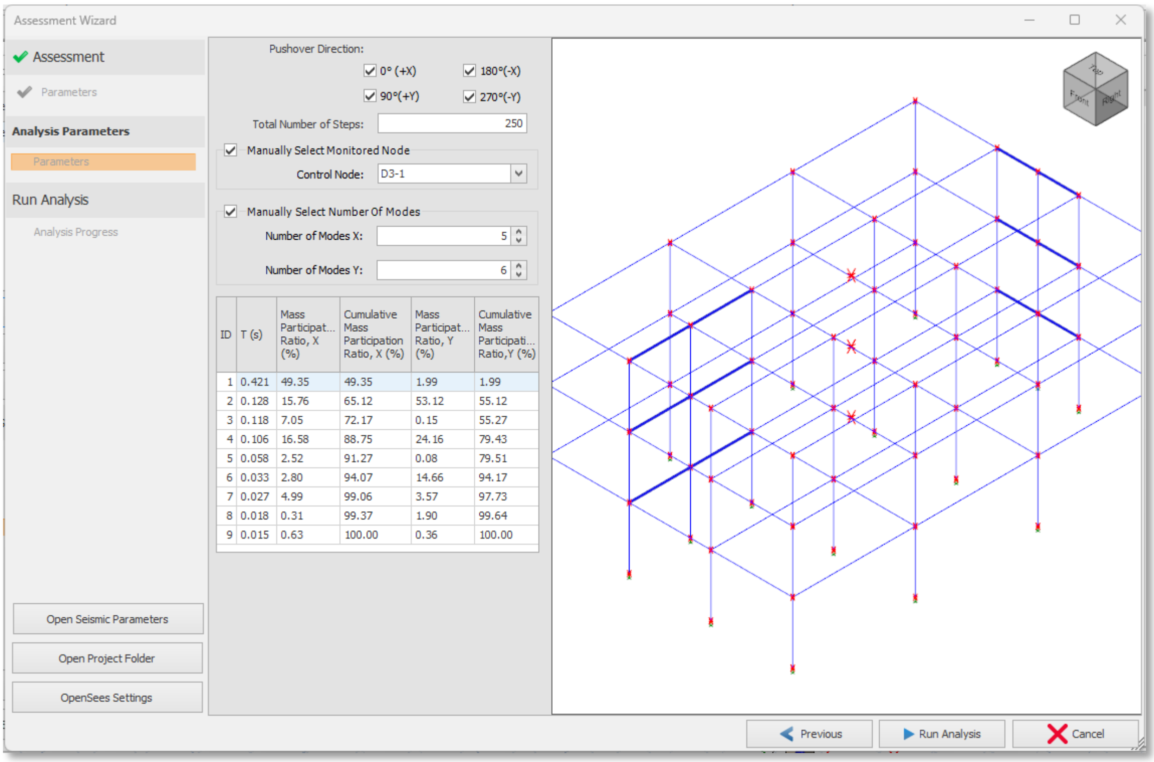
Nonlinear Static Multi Mode Procedure

In the **Assessment Wizard**, the **Analysis Methods** section allows the user to select the approach for seismic performance evaluation.

When **Nonlinear Static Multi Mode Procedure** is selected, it enables the use of **multimode pushover analysis**, which considers the combined effects of multiple vibration modes along with material nonlinearity.



After selecting **Nonlinear Static Multi Mode Procedure** in the **Assessment Wizard** and clicking **Next**, this screen appears. It is used to define all necessary parameters for multimodal nonlinear static (pushover) analysis.



Pushover Directions: Select the directions for which pushover analysis will be conducted:

- 0° (+X)
- 90° (+Y)
- 180° (–X)
- 270° (–Y)

Total Number of Steps: Defines how many load increments will be used during the analysis. A higher number yields a smoother and more detailed capacity curve.

Control Node Selection: The user can manually select a **control node** to monitor during the analysis. Typically, a node near the top of the structure is chosen.

Number of Modes Selection: Specify how many vibration modes to include in the analysis for each direction:

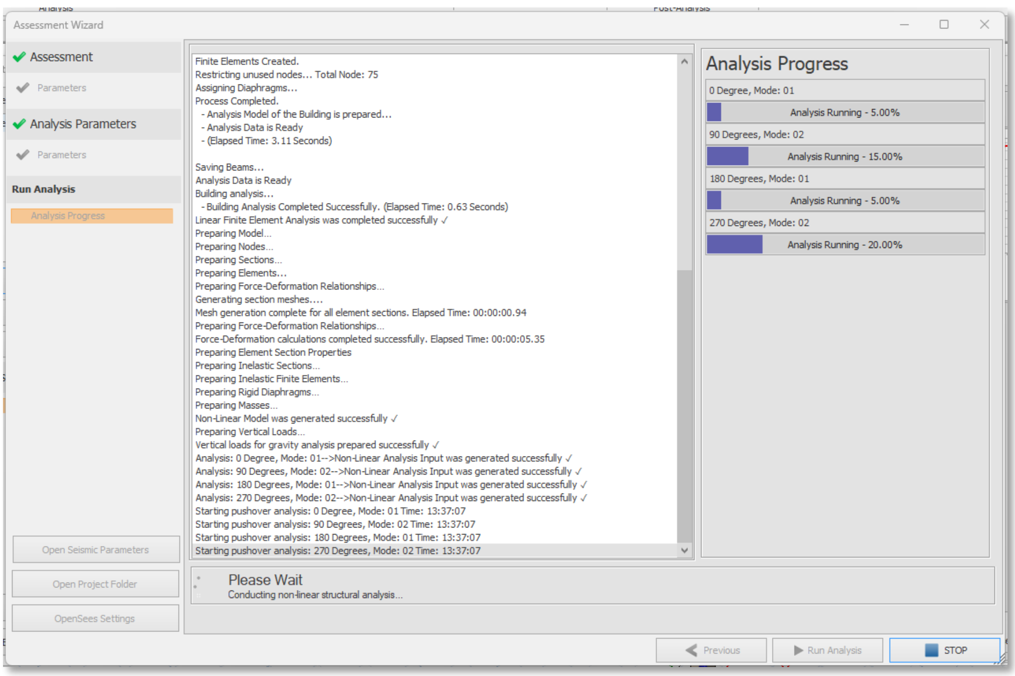
- Number of Modes X
- Number of Modes Y

Make sure to include enough modes to capture at least 90% of the total dynamic mass.

Mode Table: The table lists modal properties for each mode:

- **T (s):** Natural period of the mode
- **Mass Participation Ratio (%):** Contribution of the mode to total mass
- **Cumulative Mass Ratio (%):** Running total of mass contribution

After defining the necessary parameters for the **Nonlinear Static Multi Mode Procedure**, clicking the **Run Analysis** button initiates the process. The system then transitions to the **Analysis Progress** screen.



This screen allows the user to track each step of the analysis process in real-time. In multimodal analysis, it becomes especially useful as it displays progress per direction and per mode.

Analysis Log:

- Displays each operation being performed, such as model generation, element meshing, material assignments, and nonlinear file preparation.
- Each step is logged, including confirmation of success and timestamps for the start of each mode and direction.

Analysis Progress Bar:

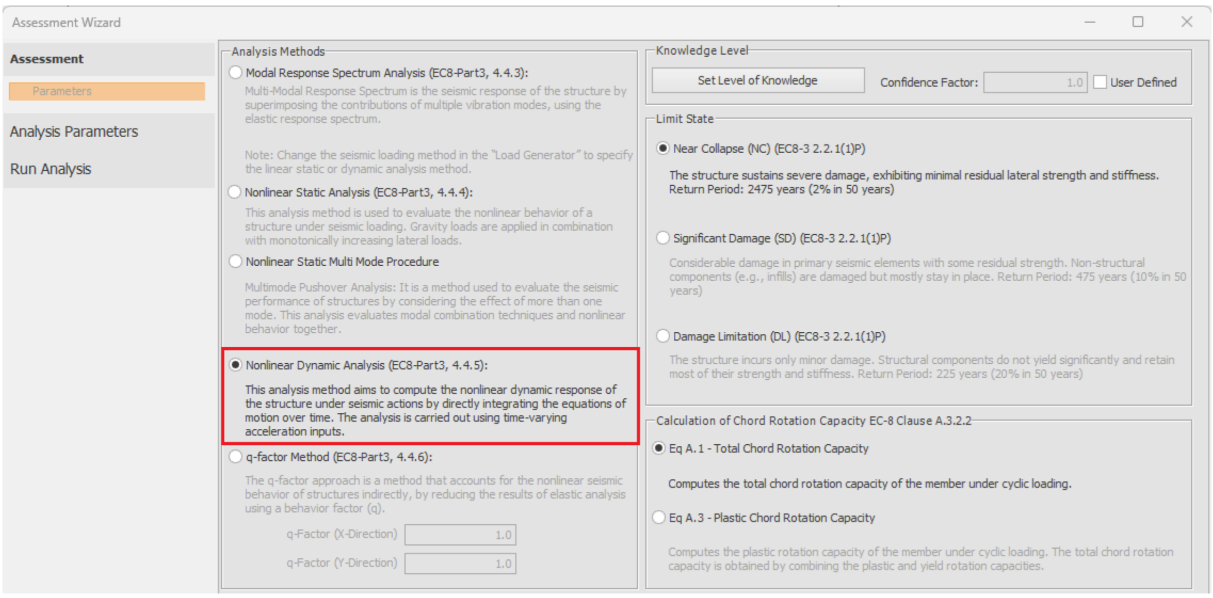
- Shows the real-time progress (%) for each selected direction (0°, 90°, 180°, 270°).
- Indicates which **mode** is currently being analyzed per direction (e.g., Mode 01, Mode 02).
- Helps quickly identify which direction or mode might be slowing down or failing.

Nonlinear Time History Dynamic Analysis

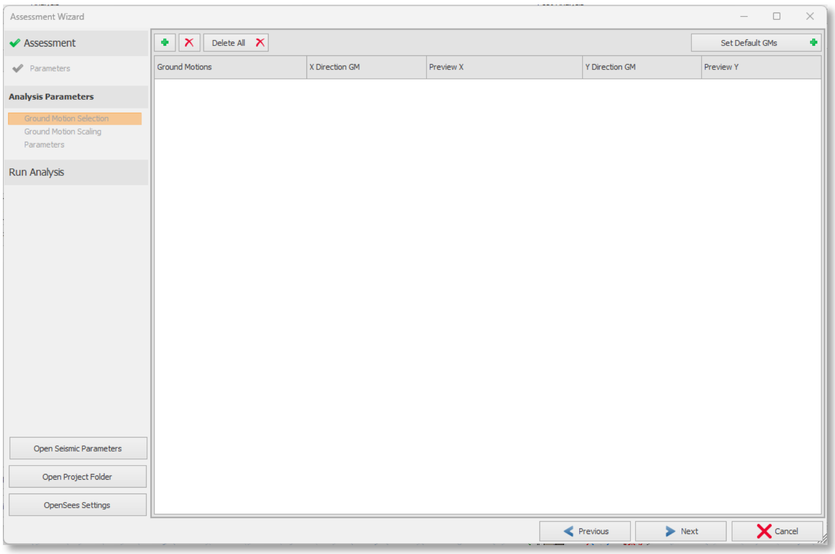
In the **Assessment Wizard**, the **Analysis Methods** section allows the user to select the approach for seismic performance evaluation.

When **Nonlinear Dynamic Procedure** is selected, it enables the use of a time history analysis method that directly incorporates the nonlinear behavior of structural components under real or simulated ground motion records.

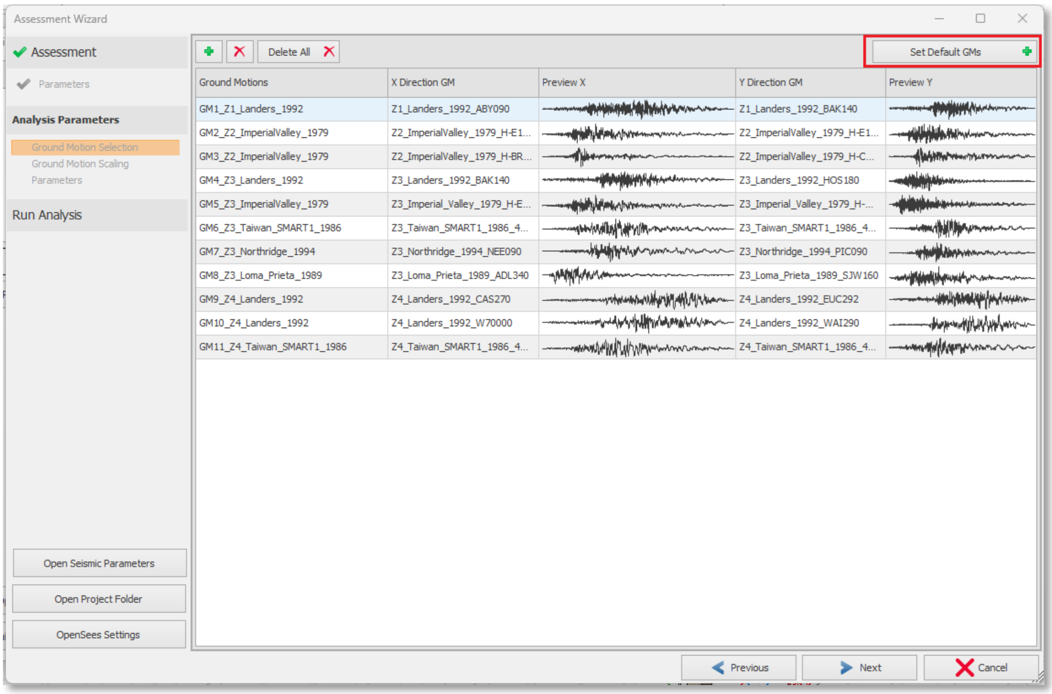
This method allows for a detailed evaluation of the seismic performance of structures by computing nonlinear time-dependent responses under ground motion records, in accordance with **EN 1998-3:2005 Clause 4.4.5**. The analysis captures accelerations, displacements, and internal forces through direct integration of the equations of motion using time-history data.



After clicking the Next button, the user is directed to the *Ground Motion Selection* interface, where earthquake records can be assigned for use in the nonlinear dynamic analysis.

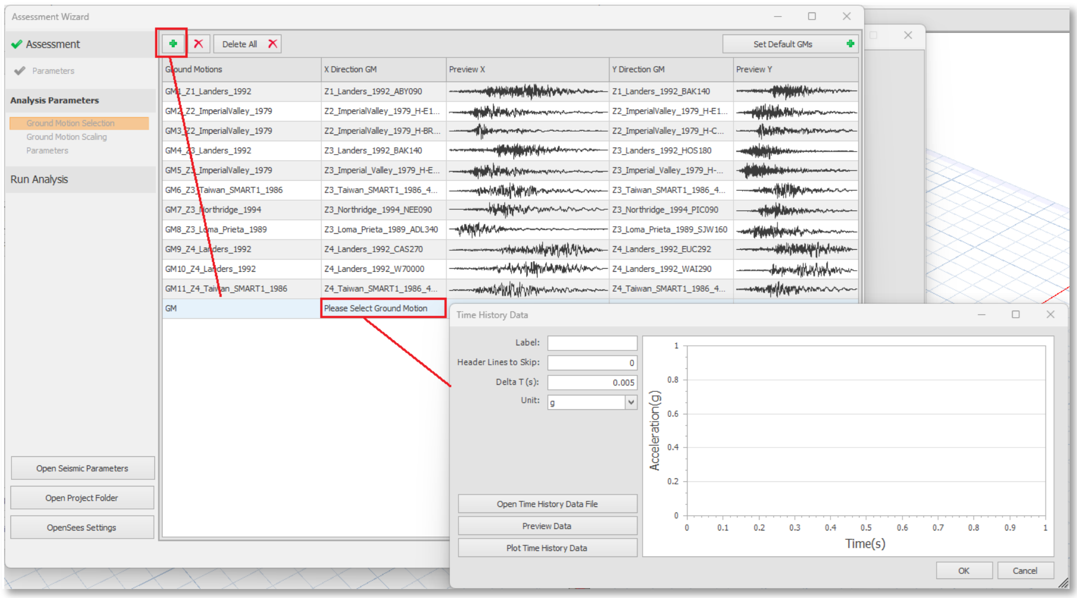


When the **Set Default GMs (Ground Motions)** button is clicked, the software automatically loads predefined earthquake records from the system library and assigns them to the X and Y directions for use in the dynamic analysis.



Each row in the table represents a pair of ground motions.The acceleration time history records are automatically mapped to **X Direction GM** and **Y Direction GM** fields. The **Preview X** and **Preview Y** columns display waveform previews of the assigned ground motions.

Within the **Assessment Wizard > Ground Motion Selection** screen, users can manually define and import custom ground motion records.

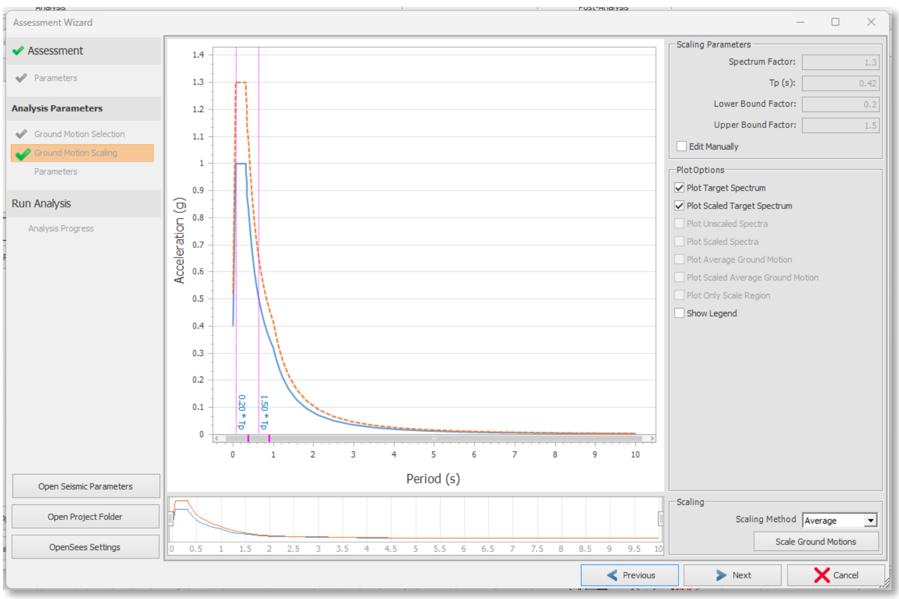


Click the ➕ **Add** button to insert a new row. Click on the **X Direction GM** or **Y Direction GM** cell of the new row. This will open the **Time History Data** input window.

Open Time History Data File, allows users to upload external acceleration time history files in .txt or similar formats. **Label,** define a custom label for the ground motion record. **Delta T (s)**, time step of the data (e.g., 0.005 s). **Unit,** select the unit of the data (g, m/s^2, etc.). **Preview**, a visual plot of the acceleration vs. time data will be shown on the right.

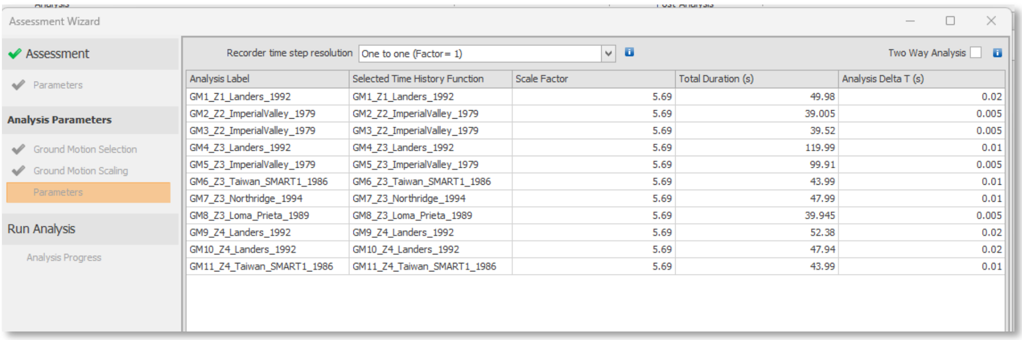
This feature provides full flexibility to use real recorded earthquakes or user-defined ground motions tailored to the specific needs of the analysis.

After completing the **Ground Motion Selection** step, clicking **Next** takes the user to the **Ground Motion Scaling** screen.



Here, the selected ground motions are **scaled to match the target spectrum**. Users may adjust the scaling parameters or proceed with the default values.

After completing the **Ground Motion Scaling** step and clicking **Next**, the user proceeds to the **Parameters** screen. This section displays and allows verification or adjustment of the analysis parameters for each loaded ground motion record.



Recorder time step resolution: Defines the time step resolution. *One to one (Factor = 1)* means the analysis time step is the same as the time history data. Higher factors reduce analysis time at the expense of accuracy.

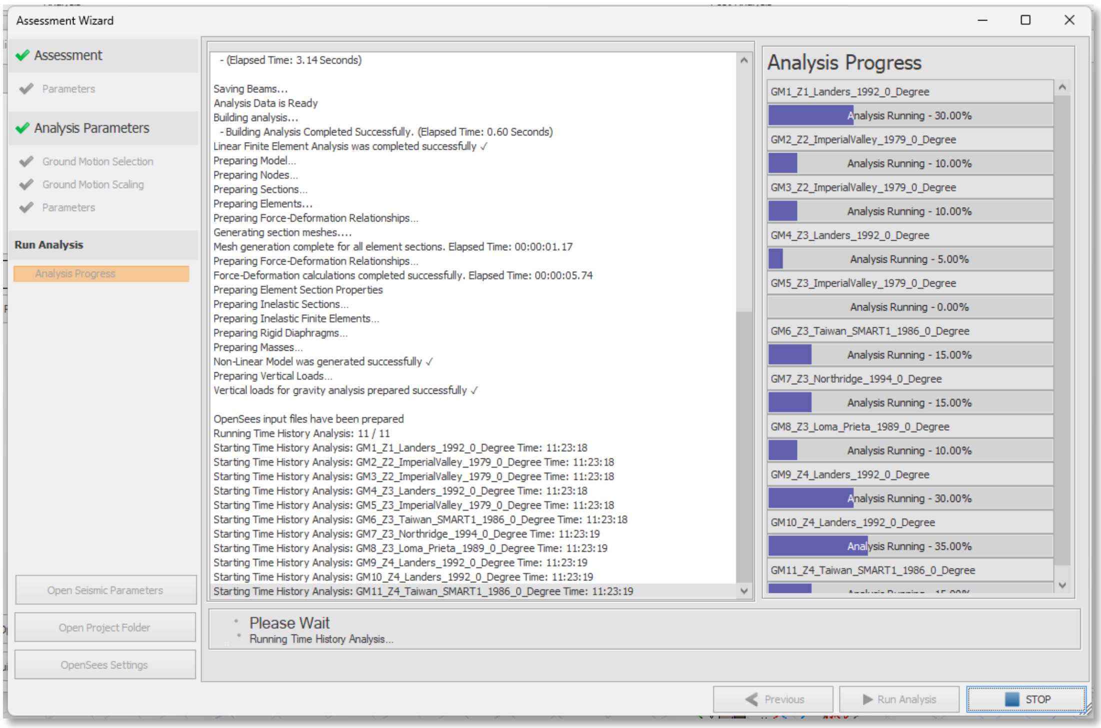
Two Way Analysis: When checked, each record is also analyzed in the orthogonal (90° rotated) direction by swapping X and Y components.

Scale Factor: Shows the scaling multiplier applied to match the target spectrum.

Analysis Δt: Indicates the time step used in the structural analysis for each ground motion.

This step is mostly automated, but the user can intervene if necessary before proceeding to the analysis.

After completing the **Parameters** step, click **Run Analysis** to initiate the analysis process. This screen provides real-time feedback during the **Nonlinear Time History Analysis** execution.



Analysis Log:

- Displays the progress log, including model setup, load applications, and analysis preparation steps.
- The starting time of each ground motion analysis is listed.

Analysis Progress:

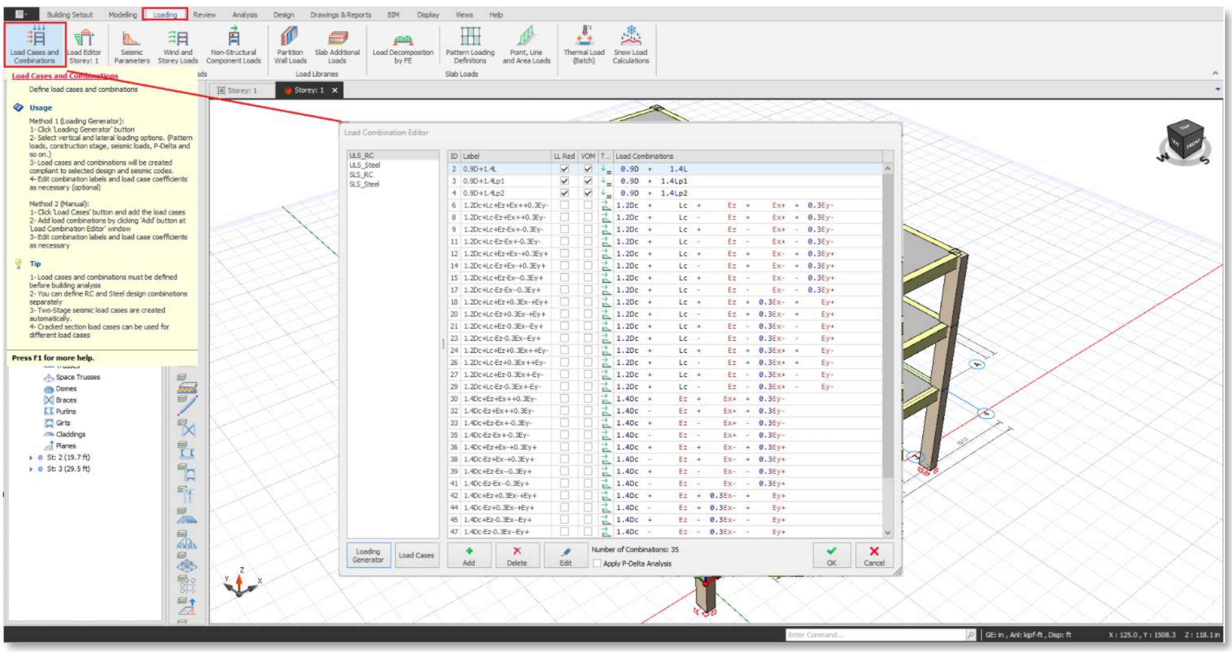
- Shows a percentage progress bar for each ground motion input.
- Once all records are completed, assessment results will be ready for review.

Linear Procedure Selection

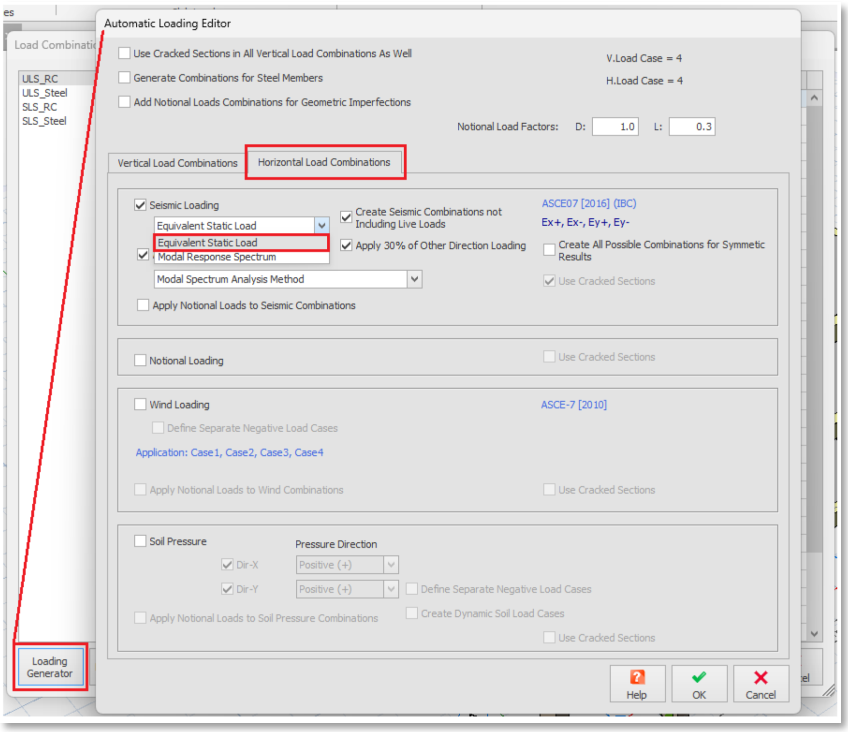
To define whether to use Lateral Force Analysis or Multi-Modal Response Spectrum Analysis for your assessment, follow the steps below in accordance with EN 1998-3:2005 Clause 4.4.2 and 4.4.3:

Steps:

1. Go to the **Loading** tab in the top ribbon.
2. Click on the **Load Cases and Combinations** icon.



3. In the pop-up window, press the **Load Generator** button.



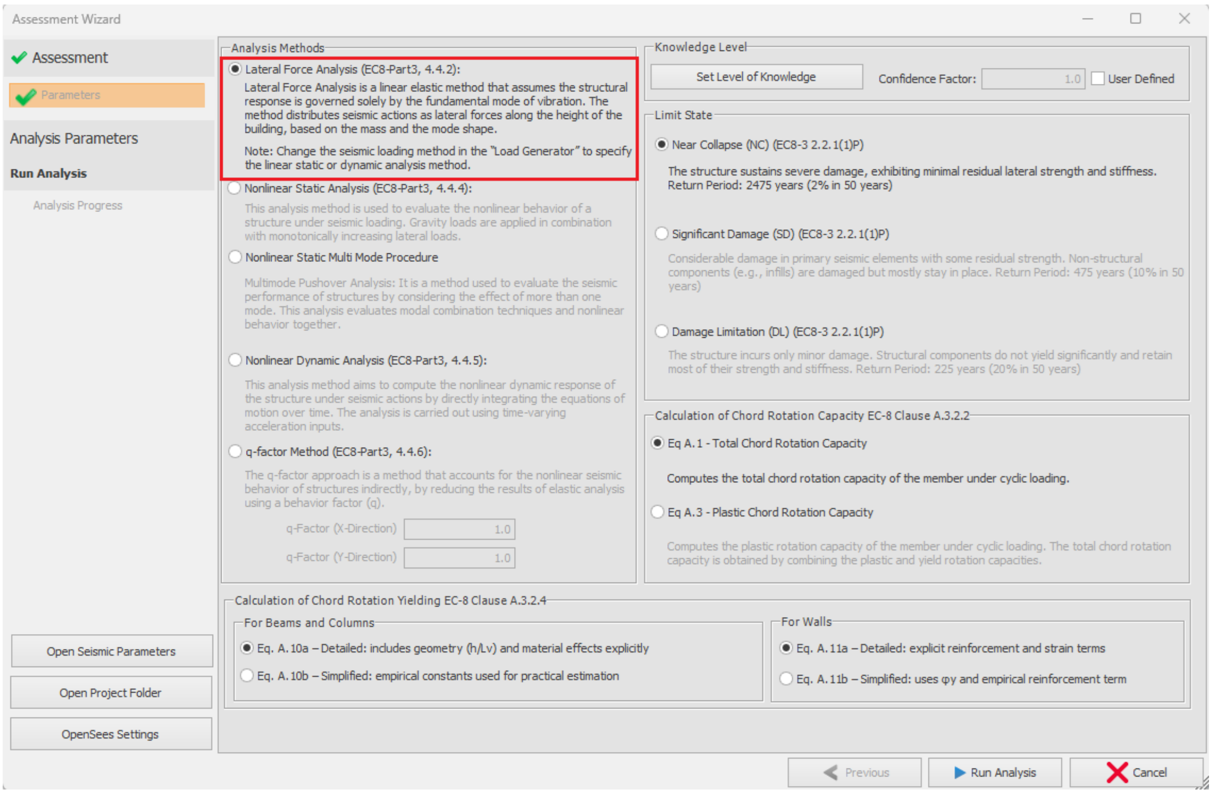
4. In the **Automatic Loading Editor** window:
   - Switch to the **Horizontal Load Combinations** tab.
   - Enable the **Seismic Loading** checkbox.
   - Choose the analysis method from the dropdown menu:
     - Equivalent Static Load → This activates Lateral Force Analysis
     - Modal Response Spectrum → This activates Multi-Modal Response Spectrum Analysis
5. Once the selection is made, click **OK** to apply the changes.

This configuration determines which linear analysis method (Lateral Force or Response Spectrum) will be used for the Eurocode 8 performance assessment in ProtaStructure.

Lateral Force Analysis

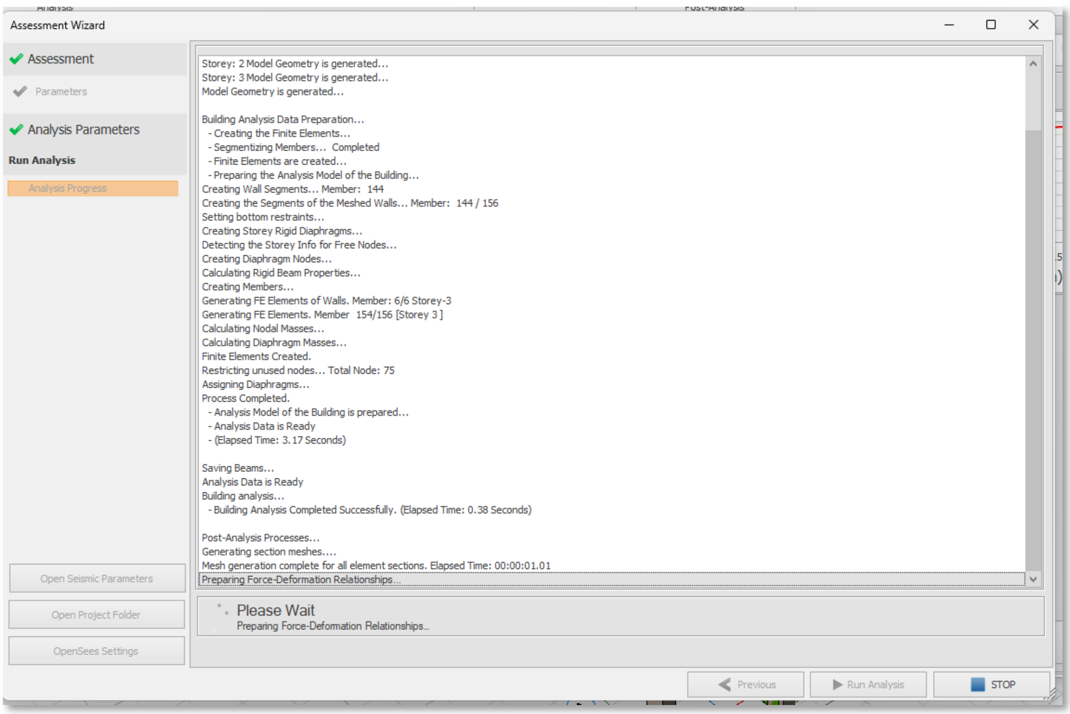
When the **Lateral Force Analysis** option is selected in the **Assessment Wizard**, the structure is analyzed using linear elastic assumptions.

This corresponds to the **Lateral Force Analysis** as described in **Eurocode 8 Part 3 Clause 4.4.2**.



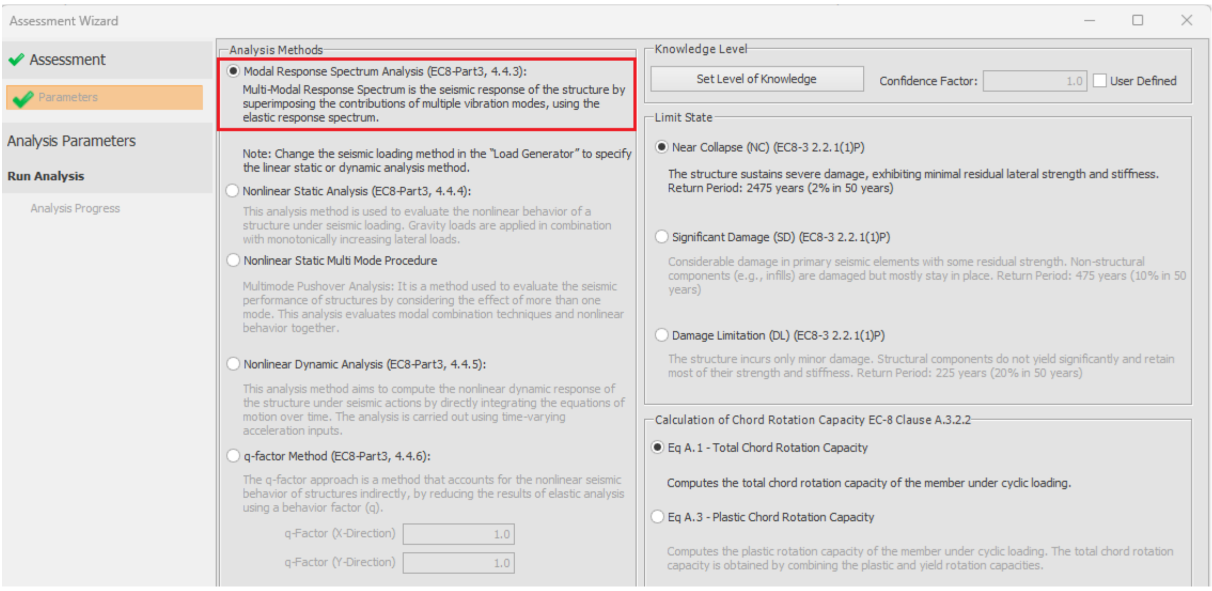
The **Lateral Force Analysis** is used to determine the distribution of seismic forces based on building height. Seismic loads and the corresponding internal forces and displacements are calculated using linear elastic static analysis. All necessary load combinations and analysis parameters are automatically handled by **ProtaStructure**. Therefore, no additional setup is required. Simply click **Run Analysis** to proceed with the assessment.

After clicking the Run Analysis button, a log window appears **showing** all the steps of the analysis process. This log **displays** each step from model generation to the completion of the analysis.



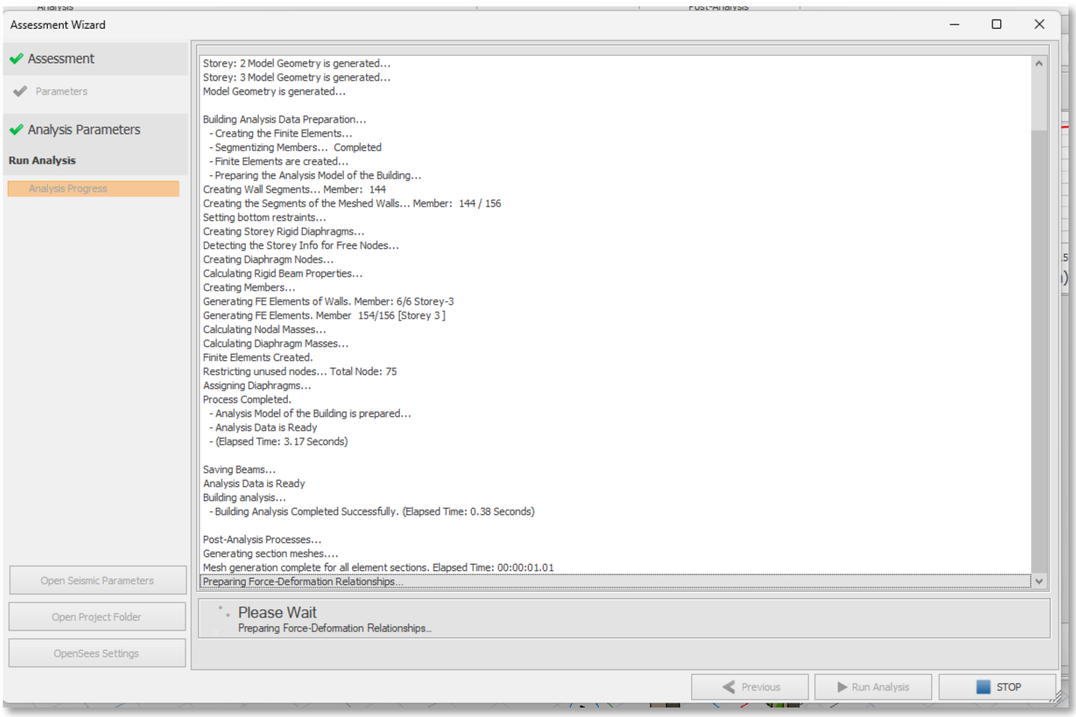
Modal Response Spectrum Analysis

In the Assessment Wizard, under the Analysis Methods section, select Linear Elastic Procedure, and ensure that the option titled **Modal Response Spectrum Analysis** (Eurocode 8 Part 3 Clause 4.4.3) is activated. To apply this method correctly, the "Modal Response Spectrum" option must be selected from the Load Generator interface.Once selected, the user can proceed by clicking Run Analysis.



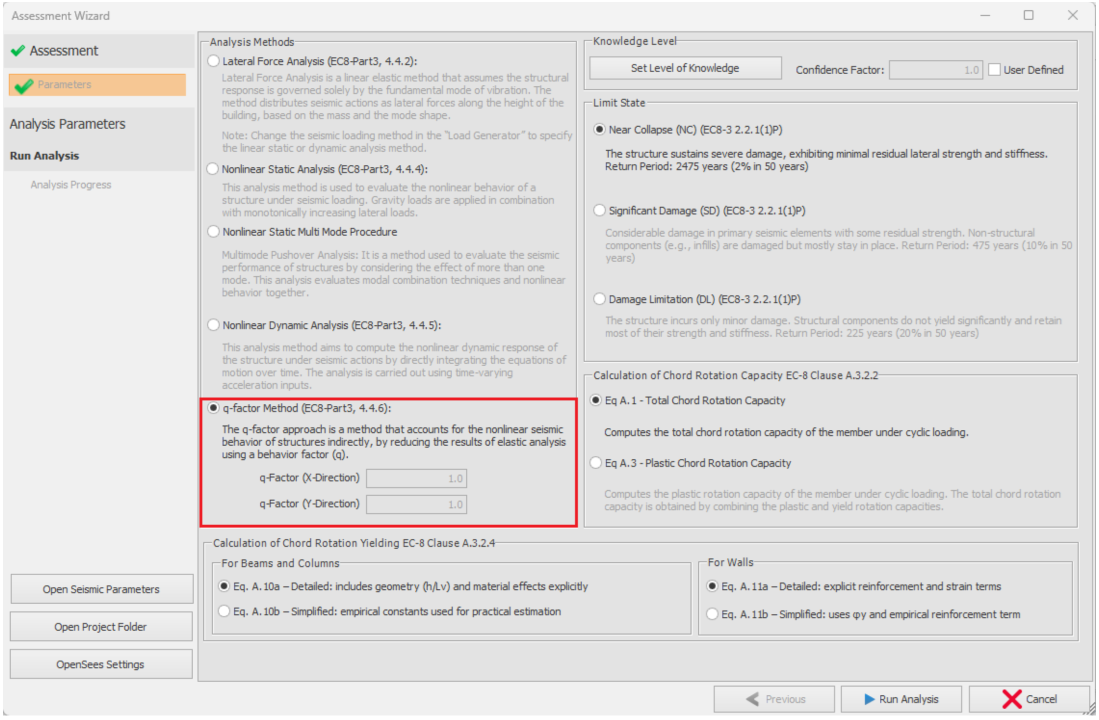
This method evaluates the seismic response of a building using modal or response spectrum analysis, offering a more realistic assessment of elastic behavior under seismic loads.

After clicking the Run Analysis button, a log window appears **showing** all the steps of the analysis process. This log **displays** each step from model generation to the completion of the analysis.



Q-Factor Analysis

The **q-factor approach** provides a simplified method to account for the nonlinear seismic behavior of structures by **reducing the elastic spectrum** using a **behavior factor (q)**. Instead of performing full nonlinear analyses (e.g., pushover or dynamic time-history), elastic forces are scaled down by a factor reflecting expected ductility and overstrength of the structure.

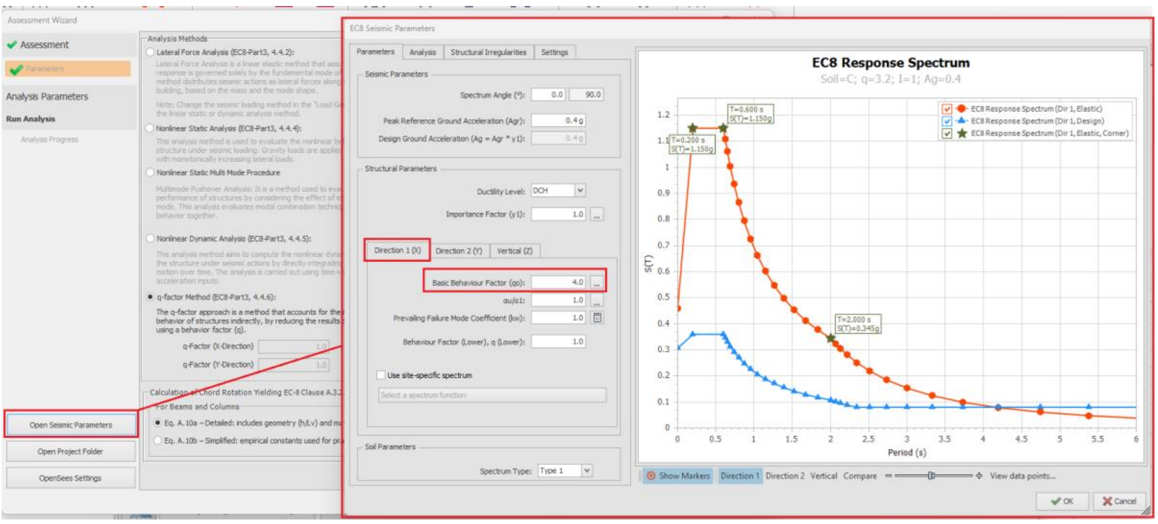


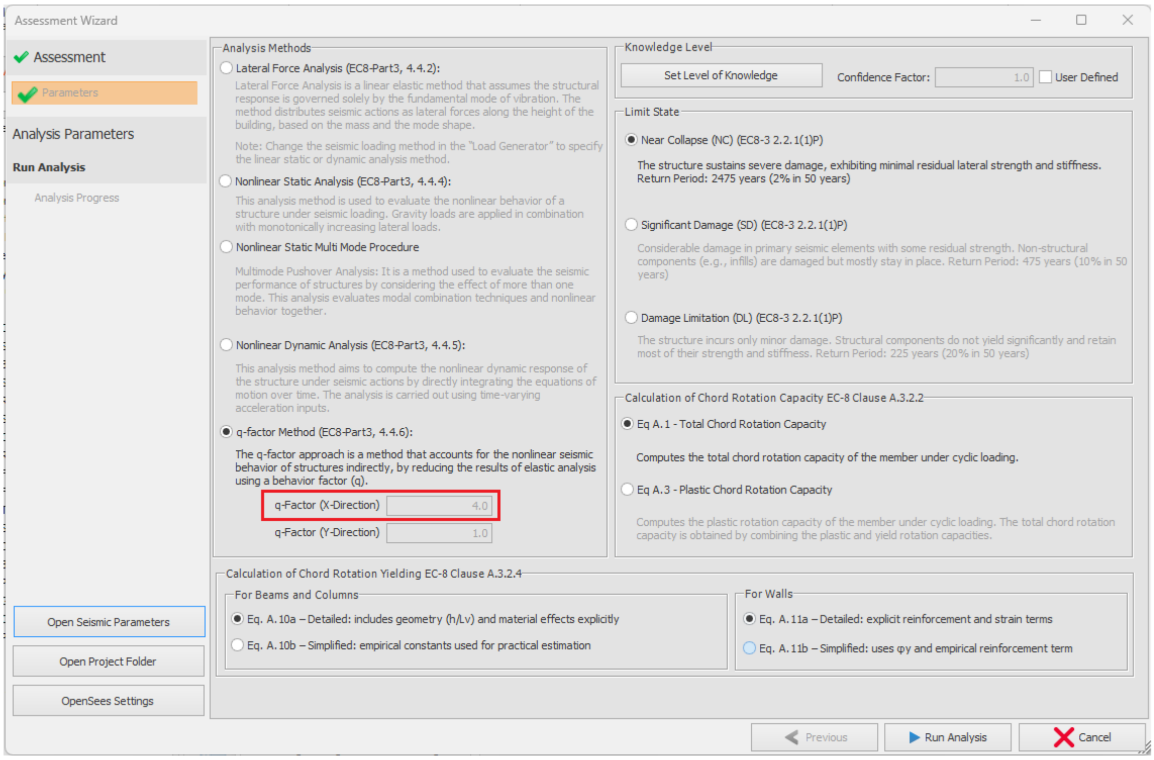
Once selected, **q-factor values** must be assigned for both horizontal directions:

- q-Factor (X-Direction)
- q-Factor (Y-Direction)

You can set these values via the button:

- Open Seismic Parameters → Direction 1 (X) / Direction 2 (Y) → Basic Behavior Factor (q)





When q-factor is activated, the elastic spectrum Se(T) is divided by q:

$$S_d(\mathrm{T}) = \frac{S_e(\mathrm{T})}{\mathrm{q}}$$

This results in lower design accelerations, and hence reduced base shear and member forces. The design still uses linear analysis techniques, but results reflect nonlinear energy dissipation via ductility.

Important Considerations:

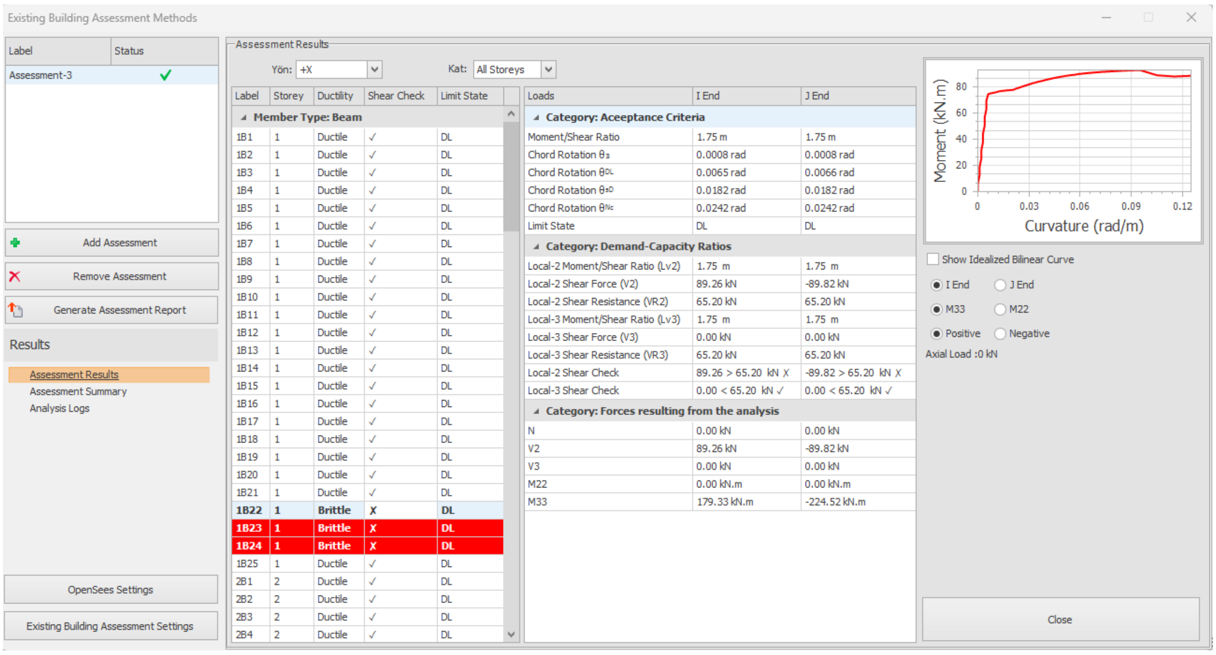
This method does not capture local failures, plastic hinge formation, or sequence of yielding. It's most suitable for perliminary assessments or when global performance checks are acceptable.

Plastic hinge behaviour and performance states are not evaluated directly.

Member-based assessments are performed using demand values from reduced elastic results. Thus, accuracy depends heavily on chosen q-factor values.

Assessment Results

After the seismic analysis is completed, the user is directed to the **Assessment Results** screen, where the performance of individual structural members is evaluated according to **Eurocode 8 – Part 3** limit states (DL, SD, NC) and local verification checks such as **chord rotation** and **shear capacity**.



Assessment Definition Panel

This section displays the current assessment scenario:

- **Label**: The name of the assessment (e.g., Assessment-3). Users can rename it.
- **Status**: Shows whether the assessment ran successfully (✓ icon).

It also includes quick-access options to:

- Add a new assessment
- Remove an existing assessment
- Generate a detailed performance report

Member List and Categories Table

All structural elements for the selected story are listed, with the following attributes:

- **Label** (e.g., 1B6)
- **Story Level**
- **Ductility Classification**: Ductile or Brittle (based on shear check)
- **Shear Check**: Pass (✓) or Fail (X)
- **Limit State**: DL, SD, or NC
- **Member Type**: Beam, Column, Wall

Note:

Members classified as *brittle* (due to failing shear check) or evaluated at the Near Collapse (NC) limit state are highlighted in red.These are considered inadequate and may require retrofitting.
Detailed Member Evaluation

When a member is selected, its detailed verification data is shown on the right side of the screen:

Category: Acceptance Criteria

- Moment/Shear Ratio (Lv)
- Chord Rotation values:
  - θ_3 (demand)
  - θ_{DL} ,θ_{SD}, θ_{NC} (limit values from EC8 A.3.2)
- Limit State classification: DL, SD, NC

Category: Demand–Capacity Ratios

- Local-2 Shear Check:
  Checks whether the shear demand in the Local-2 direction (V_2) does not exceed the shear capacity (V_{R2}) of the element.
- Local-3 Shear Check:
  Checks whether the shear demand in the Local-3 direction (V_3) does not exceed the shear capacity (V_{R3}) of the element.

Category: Forces from Analysis

- Axial force (N), shear forces (V2, V3), moments (M22, M33) at I-End and J-End

Moment–Curvature (M–ϕ) Plot

Displays the nonlinear moment–curvature behavior of the selected element, calculated from material and section properties.

Options:

- Toggle between I-End and J-End
- Switch between M22 and M33 moment axes
- View positive or negative loading behavior
- Enable idealized bilinear curve overlay (used in plastic hinge evaluations)

The Assessment Summary screen provides a comprehensive overview of the evaluation results for all directions in a clear, tabular format. It summarizes key assessment parameters, element distributions per direction.

Category: Assessment Parameters

Lists the general configuration used in the analysis:

- Seismic Hazard Level (e.g., Significant Damage)

- **Performance Objective** (e.g., Significant Damage)
- **Analysis Type** (e.g., Linear Elastic Procedure)
- **Knowledge Factor** used in capacity reduction

Category: Count

Shows the total number and percentage distribution of members by type:

- **Total Member Count**
- **Column Member Count**
- **Beam Member Count**
- **Wall Member Count**

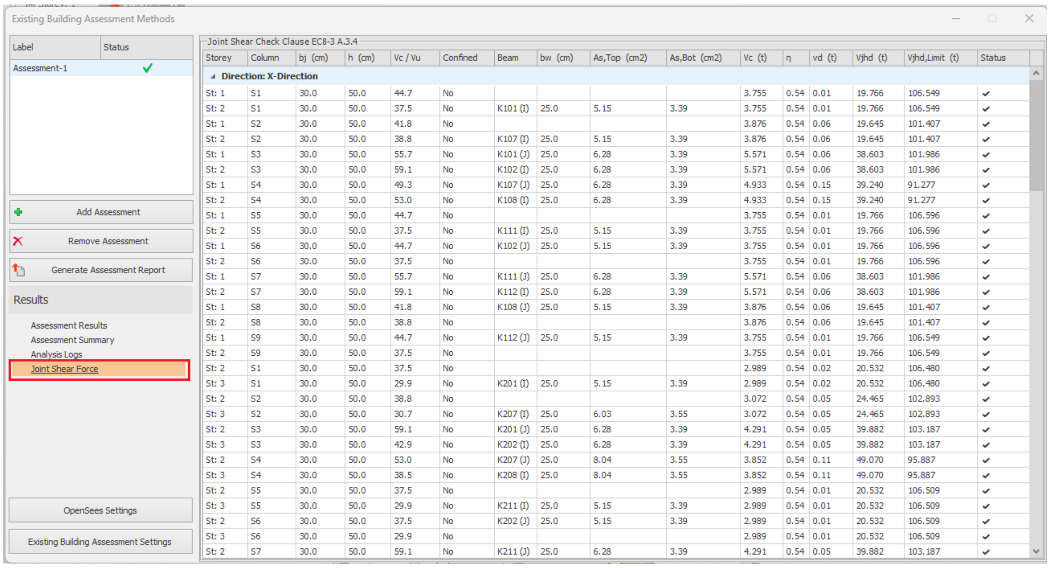
Category: Assessment

Performance classification for each direction is also provided:

- Number of members satisfying **NC,SD,DL** states
- Count of members classified as **Shear Check Passed** or **Shear Check Failed**

Joint Shear Check

This section of the report evaluates the **joint shear strength of beam-column connections**, as defined in **Eurocode 8 – Part 3, Annex A.3.3.4**. The check is performed for each joint in the selected direction (e.g., X or Y), and results are displayed **story by story**.



Storey	Column	bj (cm)	h (cm)	Vc / Vu	Confined	Beam	bw (cm)	As,Top (cm2)	As,Bot (cm2)	Vc (t)	η	vd (t)	Vjhd (t)	Vjhd,Limit (t)	Status
St: 1	S1	30.0	50.0	44.7	No					3.755	0.54	0.01	19.766	106.549	✔
St: 2	S1	30.0	50.0	37.5	No	K101 (I)	25.0	5.15	3.39	3.755	0.54	0.01	19.766	106.549	✔
St: 1	S2	30.0	50.0	41.8	No					3.876	0.54	0.06	19.645	101.407	✔
St: 2	S2	30.0	50.0	38.8	No	K107 (I)	25.0	5.15	3.39	3.876	0.54	0.06	19.645	101.407	✔
St: 1	S3	30.0	50.0	55.7	No	K101 (J)	25.0	6.28	3.39	5.571	0.54	0.06	38.603	101.986	✔
St: 2	S3	30.0	50.0	59.1	No	K102 (I)	25.0	6.28	3.39	5.571	0.54	0.06	38.603	101.986	✔
St: 1	S4	30.0	50.0	49.3	No	K107 (J)	25.0	6.28	3.39	4.933	0.54	0.15	39.240	91.277	✔
St: 2	S4	30.0	50.0	53.0	No	K108 (I)	25.0	6.28	3.39	4.933	0.54	0.15	39.240	91.277	✔
St: 1	S5	30.0	50.0	44.7	No					3.755	0.54	0.01	19.766	106.596	✔
St: 2	S5	30.0	50.0	37.5	No	K111 (I)	25.0	5.15	3.39	3.755	0.54	0.01	19.766	106.596	✔
St: 1	S6	30.0	50.0	44.7	No	K102 (J)	25.0	5.15	3.39	3.755	0.54	0.01	19.766	106.596	✔
St: 2	S6	30.0	50.0	37.5	No					3.755	0.54	0.01	19.766	106.596	✔
St: 1	S7	30.0	50.0	55.7	No	K111 (J)	25.0	6.28	3.39	5.571	0.54	0.06	38.603	101.986	✔
St: 2	S7	30.0	50.0	59.1	No	K112 (I)	25.0	6.28	3.39	5.571	0.54	0.06	38.603	101.986	✔
St: 1	S8	30.0	50.0	41.8	No	K108 (J)	25.0	5.15	3.39	3.876	0.54	0.06	19.645	101.407	✔
St: 2	S8	30.0	50.0	38.8	No					3.876	0.54	0.06	19.645	101.407	✔
St: 1	S9	30.0	50.0	44.7	No	K112 (J)	25.0	5.15	3.39	3.755	0.54	0.01	19.766	106.549	✔
St: 2	S9	30.0	50.0	37.5	No					3.755	0.54	0.01	19.766	106.549	✔
St: 2	S1	30.0	50.0	37.5	No					2.989	0.54	0.02	20.532	106.480	✔
St: 3	S1	30.0	50.0	29.9	No	K201 (I)	25.0	5.15	3.39	2.989	0.54	0.02	20.532	106.480	✔
St: 2	S2	30.0	50.0	38.8	No					3.072	0.54	0.05	24.465	102.893	✔
St: 3	S2	30.0	50.0	30.7	No	K207 (I)	25.0	6.03	3.55	3.072	0.54	0.05	24.465	102.893	✔
St: 2	S3	30.0	50.0	59.1	No	K201 (J)	25.0	6.28	3.39	4.291	0.54	0.05	39.882	103.187	✔
St: 3	S3	30.0	50.0	42.9	No	K202 (I)	25.0	6.28	3.39	4.291	0.54	0.05	39.882	103.187	✔
St: 2	S4	30.0	50.0	53.0	No	K207 (J)	25.0	8.04	3.55	3.852	0.54	0.11	49.070	95.887	✔
St: 3	S4	30.0	50.0	38.5	No	K208 (I)	25.0	8.04	3.55	3.852	0.54	0.11	49.070	95.887	✔
St: 2	S5	30.0	50.0	37.5	No					2.989	0.54	0.01	20.532	106.509	✔
St: 3	S5	30.0	50.0	29.9	No	K211 (I)	25.0	5.15	3.39	2.989	0.54	0.01	20.532	106.509	✔
St: 2	S6	30.0	50.0	37.5	No	K202 (J)	25.0	5.15	3.39	2.989	0.54	0.01	20.532	106.509	✔
St: 3	S6	30.0	50.0	29.9	No					2.989	0.54	0.01	20.532	106.509	✔
St: 2	S7	30.0	50.0	59.1	No	K211 (J)	25.0	6.28	3.39	4.291	0.54	0.05	39.882	103.187	✔

For each joint, the following parameters are listed:

- **Storey / Column / Beam**: Geometric identification of the joint
- b_j : Effective joint width

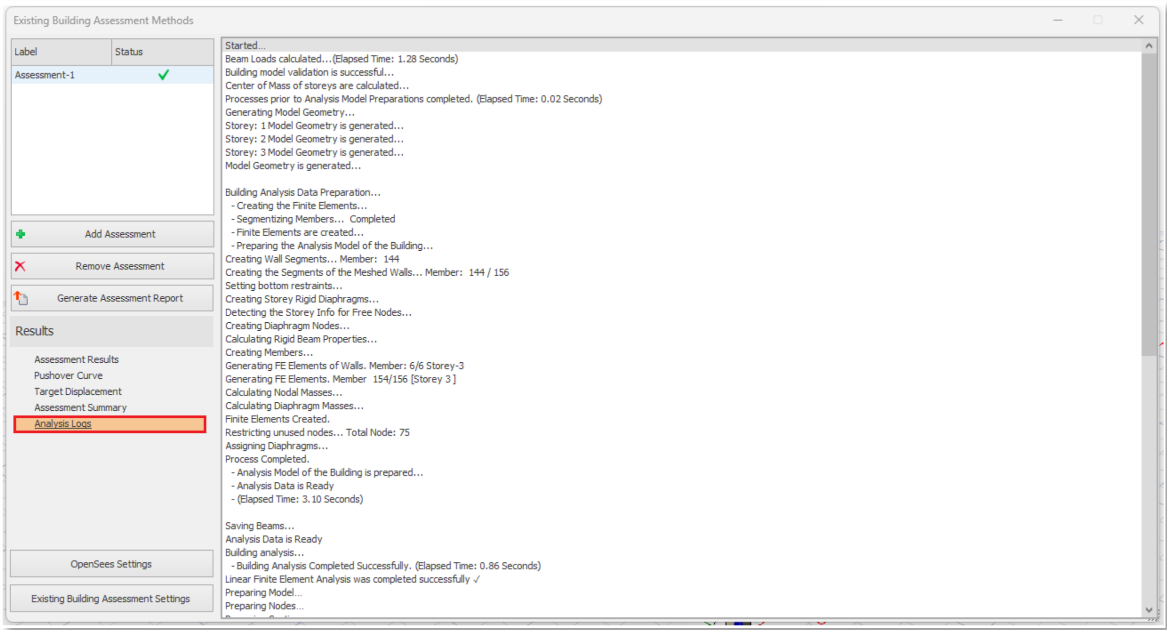
- c_j : Effective joint depth
- V_c : Concrete shear strength of the joint panel
- V_u : Shear demand on the joint
- $A_{s,Top}$ / $A_{s,Bot}$: Top and bottom longitudinal beam reinforcement
- b_w : Beam width
- **Confined**: Whether the joint is confined or not (affects Vc)
- v_d / v_h / V_{jhd}: Design shear stress terms calculated per EC8 formulation
- $V_{jhd,Limit}$: Joint shear capacity per design
- **Status**: ✓ indicates the joint satisfies the EC8 joint shear capacity check

Note:

The Joint Shear Check results are shown only when the Building Analysis has been performed.

Analysis Logs

The **Analysis Logs** section provides a detailed, real-time record of the entire analysis process.



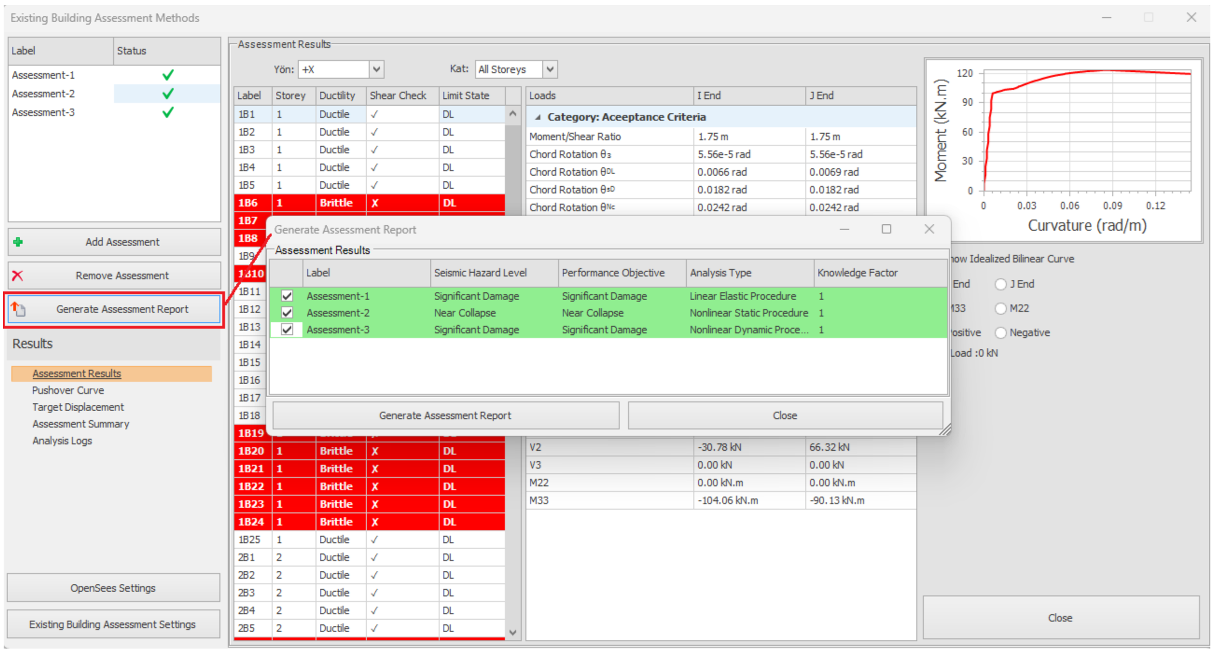
Assessment Report

Within the **Existing Building Assessment Methods** interface, all completed evaluation scenarios are listed under the **Label** section. The user can click on the **"Generate Assessment Report"** button located in the left panel to open the reporting screen.

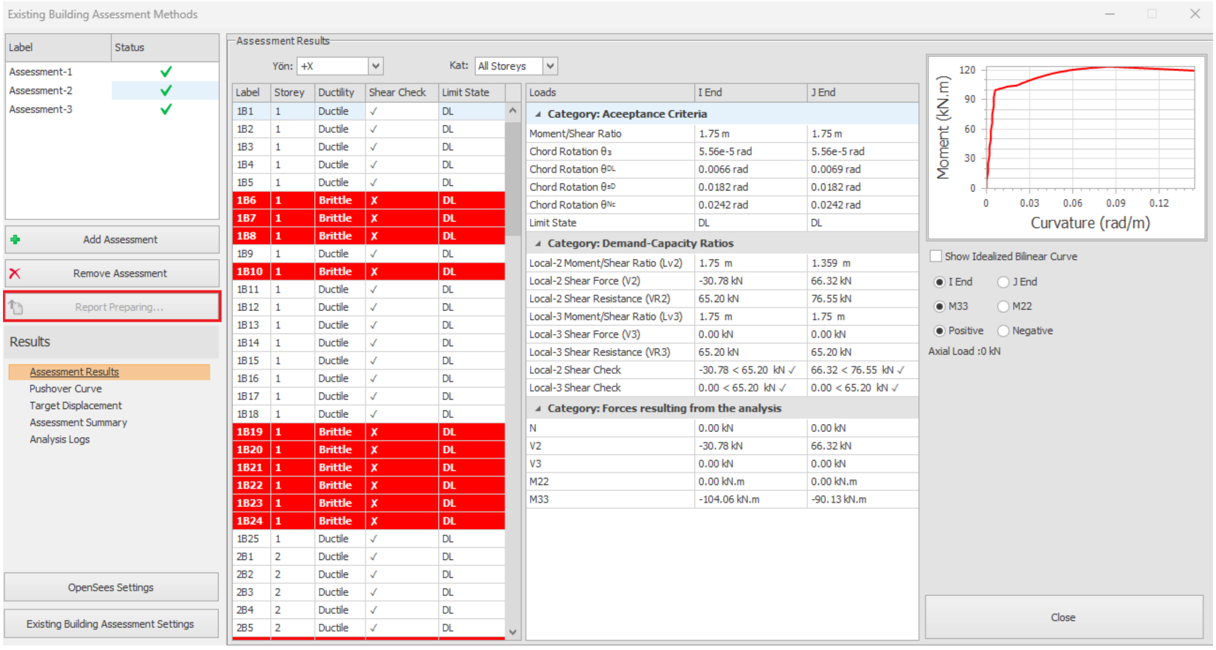
The pop-up window provides summary information for each assessment:

This window displays a summary of all assessment scenarios created by the user. Each row provides key parameters associated with the selected performance evaluation:

- Label: Name of the assessment (e.g., Assessment-1).
- Seismic Hazard Level: The return period selected for the assessment (e.g., 475 years for Significant Damage, 2475 years for Near Collapse), based on EN 1998-3 §2.1(3)P.
- Performance Objective: Target limit state selected by the user — Damage Limitation (DL), Significant Damage (SD), or Near Collapse (NC).
- Analysis Type: The structural analysis method applied —
    - Lateral Force Analysis (Linear)
    - Response Spectrum Analysis (Linear)
    - Nonlinear Static Analysis (Pushover)
    - Nonlinear Dynamic Analysis (Time History)
- Knowledge Factor: Confidence Factor (CF) applied, based on the selected Knowledge Level:
    - KL1 → CF = 1.35
    - KL2 → CF = 1.20
    - KL3 → CF = 1.00



When generate assesment report button from popup screen. Report will be preparing.



When the **"Generate Assessment Report"** button is clicked in the **Existing Building Assessment Methods** screen, a comprehensive report is generated by combining all selected assessment scenarios. This report provides a detailed overview of how the structure performs under different analysis methods.

Report Content:

- Assessment Parameters
  - Selected analysis method (e.g., *Lateral Force Analysis)*
  - Limit State (e.g., *Near Collapse*, Significant Damage, etc.)
  - Seismic hazard level (e.g., *Near Collapse*, Significant Damage, etc.)
  - Knowledge factor level
- Assessment Summary by Performance Limit State
  - Understand the structural performance per story and per element type
  - Identify the critical stories where elements begin to exceed SD or NC thresholds
  - Quantify how many members are within acceptable rotation demands at each limit state
- Ductility Classification Summary
  - Clear identification of brittle elements per story, which are more critical for retrofitting
  - Visibility into which structural component types (e.g., beams vs columns) are mote vulnerable to shear failure
- Joint Results (Per Member, Story-Wise)

  - Internal forces (axial, shear and moments) for each end (Top/Bottom or I/J) of every structural member
  - Reported per story
- Shear Capacity Check (Per Member, Story-Wise)
  - Shear demand vs capacity checks in Local-2 and Local-3 directions
  - Capacity (V_{R2}, V_{R3}) and demand (V_2, V_3) compared
  - Separate checks at top and bottom (I and J) ends
  - Pass/fail status (✓ or ✗) provided explicitly
- Chord Rotation Check (Per Member, Story-Wise)
  - Chord rotation demand (θ_2, θ_3) vs. EC8 limit values (θ_{DL}, θ_{SD}, θ_{NC})
  - Evaluated seperately at Top and Bottom ends
  - Final assigned Limit State per end (DL, SD, NC)
- Joint Shear Check (Per Member, Story-Wise)
  - Evaluates beam-column joint panel shear strength based on EC8 Clause A.3.3.4. Reported per story and per direction (X / Y)
  - Column and beam geometry
  - Reinforcement area
  - Concrete shear capacity
  - Shear demand
  - Calculated stress values
  - Joint confinement status
  - Final status (✓ or ✗)

Member Seismic Detailing

In the Eurocode 8–3 assessment module of ProtaStructure, users can define whether each member type (columns, beams, and walls) is detailed for seismic resistance, as described in Clause A.3.2.2 of EN 1998-3:2005. This setting has a direct impact on the calculation of chord rotation capacity.

Global Settings

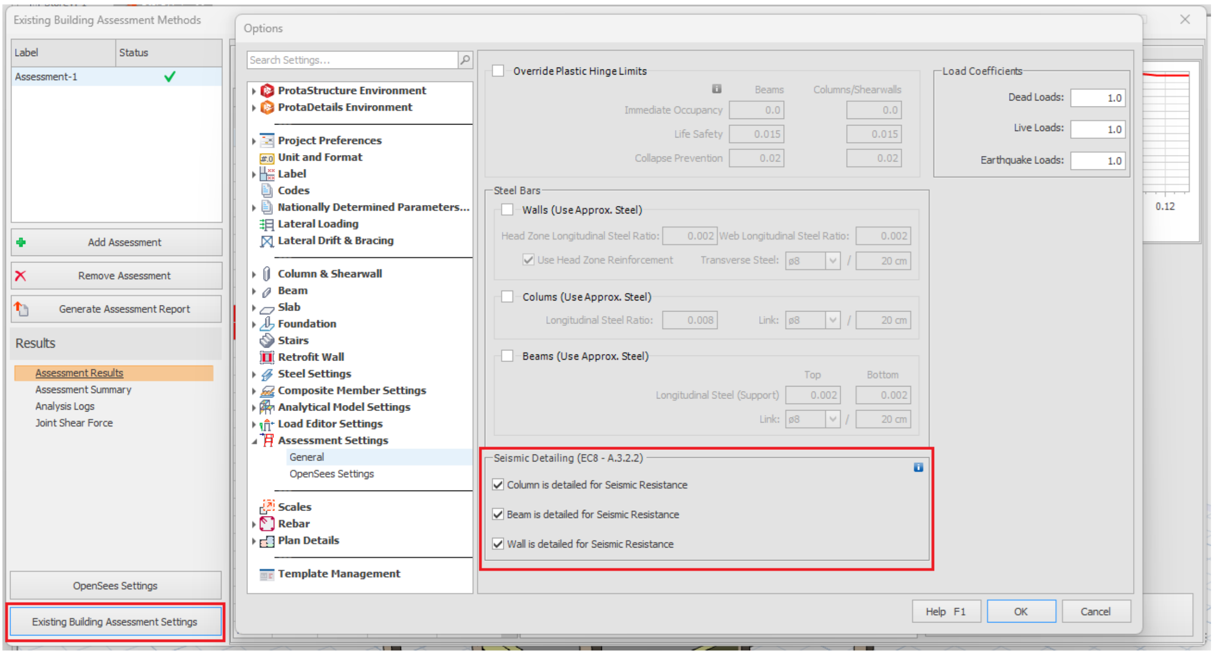
Existing Building Assessment Settings > Assessment Settings > General

This section includes a global checkbox setting for:

- Column is detailed for seismic resistance
- Beam is detailed for seismic resistance

- Wall is detailed for seismic resistance

These global definitions apply to all elements unless overridden by member-specific inputs.

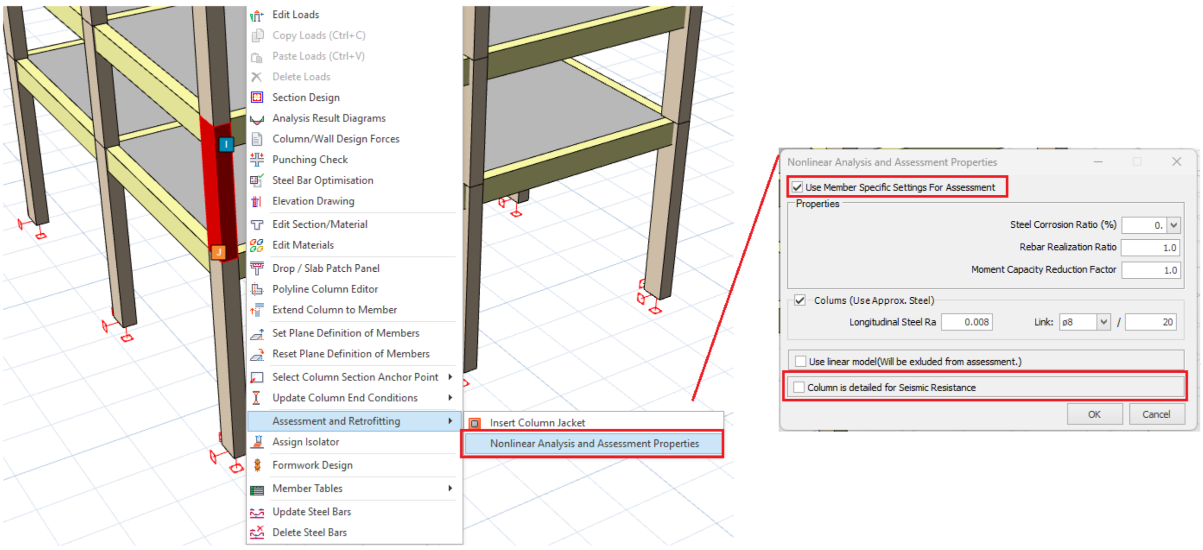


Member-Specific Settings

Right-clicking any member and selecting Assessment and Retrofitting > Nonlinear Analysis and Assessment Properties opens the custom settings window.

When **"Use Member Specific Settings for Assessment"** is checked, the user can override global assumptions and define **element-specific seismic detailing and degradation properties** for individual **columns, beams, or walls**.

This is useful for capturing local deficiencies, corrosion scenarios, or retrofitted sections.



Thank You…

Thank you for choosing the ProtaStructure Suite product family.

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

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

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

The Prota Team

PROTA Structure®

PROTA Steel®

PROTA Details®

PROTA BIM®