ProtaStructure Design Guide

Assessment & PBD to ASCE/SEI 41

Version 1.0 March 2025

Please contact us for your training and technical support queries

asiasupport@protasoftware.com

globalsupport@protasoftware.com

Publisher

PROTA SOFTWARE

Limitation of Responsibilities

Prota shall not be held responsible for any losses caused by documentation, software, or usage errors.

In addition to Prota License Agreement Terms, it is the responsibility of the user.

- to check results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- make sure the software is correctly used per the reference manual and documentation,

Intellectual Property

ProtaStructure is a registered trademark of **Prota Software Inc,** and all intellectual property rights belong to **Prota Software Inc**. Documentation, training, and reference manuals. Any program component cannot be copied, distributed, or used in violation of the license agreement.

Trademarks

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

Table of Contents

Introduction

The Seismic Evaluation and Retrofit of Existing Buildings Standard ASCE/SEI 41-17 is a significant reference in the field of structural engineering. This standard delineates deficiency-based and systematic procedures for evaluating and retrofitting existing buildings for seismic resilience, utilizing performance-based principles. ASCE/SEI 41-17 provides a three-tier process that aligns building performance levels with seismic hazard levels, integrating targeted structural performance and the performance of non-structural components.

The standard encompasses analysis procedures, acceptance criteria, and requirements for various building components and systems, including foundations, steel, concrete, masonry, wood, cold-formed steel, architectural, mechanical, and electrical components, as well as seismic isolation and energy dissipation systems. It offers checklists for different building types and seismicity levels to support the Tier 1 screening process.

The latest edition of ASCE/SEI 41-17 updates and revises previous editions, incorporating significant revisions on fundamental performance objectives for existing buildings and the evaluation of force-controlled actions. Notably, it reconfigures the nonlinear dynamic procedure and updates provisions for steel and concrete columns and unreinforced masonry.

ASCE/SEI 41-17 is essential for structural engineers focusing on enhancing the seismic resilience of existing buildings and for building code officials reviewing such work. It also holds value for architects, construction managers, academic researchers, and building owners, providing a comprehensive framework to address seismic vulnerabilities in existing structures.

ASCE 41/17 Seismic Definitions

In this section, the steps and definitions for the evaluation of existing buildings within ASCE 41/17 will be discussed.

Performance Levels

ASCE 41/17 contains four performance levels. These performance levels are the targeted or defined performance objectives under specific seismic loads. The performance levels are outlined below:

Operational Performance Level (1-A)

According to ASCE 41-17's Operational Level (1-A), the structure does not require any repair measures. The strength and stiffness properties of the structural elements are maintained without significant yielding. All systems important for normal operation are functional. Non-structural components, such as partitions and infills, should not be damaged. (ASCE 2.3.3.1)

Immediate Occupancy Performance Level (1-B)

According to ASCE 41-17, the Immediate Occupancy Condition (1-B) post-earthquake is a condition where building operations are expected to continue uninterrupted during and after the design earthquake, with the exception of some minor functions. Structural elements retain their durability and stiffness properties. A few minor cracks may occur in the structure. (ASCE 2.3.3.2)

Life Safety Performance Level (3-C)

According to ASCE 41-17, Life Safety (3-C) is a condition where moderate damage to the structure is expected during the design earthquake, but repair may not be economically feasible. Structural elements retain some residual strength and stiffness. Although non-structural components may be damaged, partitions, and infill walls do not experience out-of-plane failure. Moderate permanent displacements are present. (ASCE 2.3.3.3)

Collapse Prevention Performance Level (5-D)

According to ASCE 41-17, Collapse Prevention (5-D) is a condition where significant (generally irreparable) damage to the structure is expected during the design earthquake, and the structure is unlikely to survive another earthquake. The structure exhibits a permanently low level of lateral stiffness and strength, but the vertical elements are still capable of carrying vertical loads. Most non-structural components have collapsed, and large permanent displacements are present. (ASCE 2.3.3.4)

Seismic Hazard Levels

The seismic hazard section of ASCE/SEI 41-17 provides comprehensive guidelines for characterizing and assessing seismic hazards for the seismic evaluation and retrofitting of existing buildings. The primary objective of seismic hazard assessment is to determine the expected ground motion parameters at a building site during an earthquake. This information is necessary to design seismic retrofits that enhance the building's resilience against seismic forces.

BSE-1E (2.4.1.4): Represents a seismic event with a 20% probability of exceedance within 50 years, corresponding to a return period of 225 years. Used for initial evaluation and retrofit design to achieve a basic safety level for existing buildings.

BSE-2E (2.4.1.3): Represents a seismic event with a 5% probability of exceedance within 50 years, corresponding to a return period of 975 years. Intended for more rigorous evaluation and retrofit design to ensure building safety during more severe earthquakes.

BSE-1N (2.4.1.2): Represents a seismic event with a 10% probability of exceedance within 50 years, corresponding to a return period of 475 years. Typically used as a criterion for designing new buildings to a standard safety level.

BSE-2N (2.4.1.1): Represents a seismic event with a 2% probability of exceedance within 50 years, corresponding to a return period of 2,475 years. Used to design critical infrastructure and facilities to remain operational during and after a major seismic event.

Significance of Seismic Hazard Levels

Risk Assessment: Helps understand and mitigate seismic risks specific to the location of a building.

Design Guidance: Provides clear criteria to ensure the necessary level of protection based on expected seismic activity.

Safety Standards: Ensures that buildings are evaluated and retrofitted to meet appropriate safety standards, reducing the risk of damage and loss of life during an earthquake.

Level of Seismicity

The level of seismicity is defined in four levels (ASCE 41/17 2.5).

Very Low Seismicity: Areas where the probability of significant ground motion is relatively very low.

Low Seismicity: Areas where the probability of significant ground motion is relatively low. Buildings in these regions require basic measures to ensure safety and prevent collapse.

Moderate Seismicity: Regions where the probability of experiencing significant ground motion is moderate. Retrofit designs must account for more frequent and stronger seismic events compared to regions of low seismicity.

High Seismicity: Areas where the probability of experiencing severe ground motion is high. Buildings in these regions require the strictest retrofit measures to ensure they withstand intense seismic forces and protect occupants.

$$S_{DS} = \frac{2}{3} * \mathrm{F}_a S_s \quad \text{(Eq. 2-4)}$$

$$S_{D1} = \frac{2}{3} * \mathrm{F}_v S_1 \quad \text{(Eq. 2-5)}$$

Table 2-4. Level of Seismicity Definitions

Level of Seismicity[a]	S_{DS}	S_{D1}
Very low	<0.167 g	<0.067 g
Low	≥0.167 g <0.33 g	≥0.067 g <0.133 g
Moderate	≥0.33 g <0.50 g	≥0.133 g <0.20 g
High	≥0.50 g	≥0.20 g

Soil Classification

Soil classification is an important component in the seismic evaluation and retrofitting of existing buildings. It helps understand how local soil conditions can affect seismic waves and ground shaking during an earthquake.

Soil classification categorizes the soil and rock conditions at a building site to adjust seismic design parameters. Different soil conditions can amplify or attenuate seismic waves, which influences the building's seismic response.

The standard classifies sites into different categories based on the average properties of the top 30 meters (100 feet) of the soil profile.

According to ASCE 41/17, soil classes consist of 6 categories; (ASCE 7 20.3-1)

Table 20.3-1 - Site Classification

Table 20.3-1: Site Classification

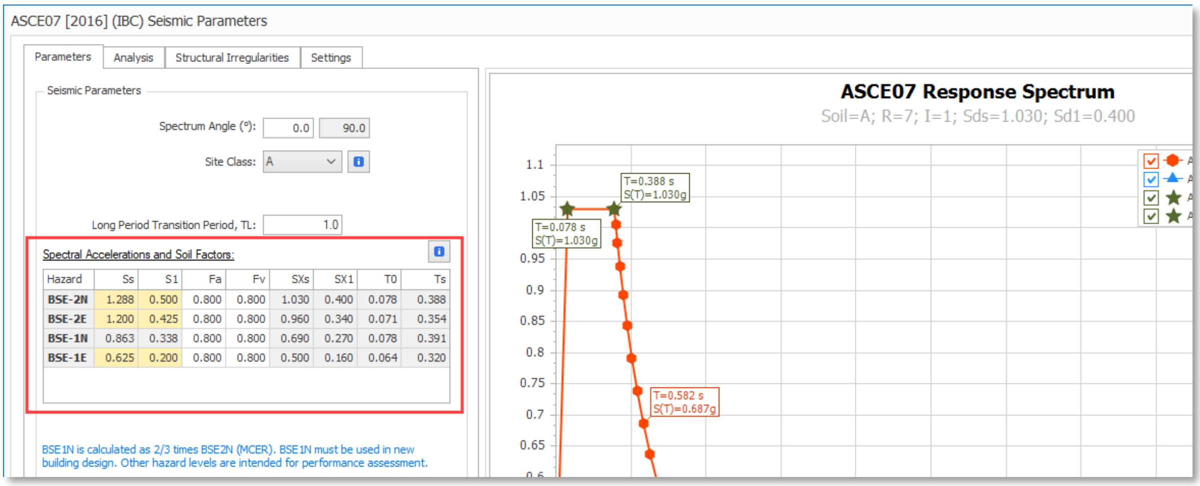
Site Class	$\bar{V}_s$ (ft/s)	$\bar{N}$ or $\bar{N}_{ch}$	$\bar{S}_u$ (lb/ft^2)
A. Hard Rock	> 5000	N/A	N/A
B. Rock	2500 to 5000	N/A	N/A
C. Very dense soil and soft rock	1200 to 2500	> 50	> 2000
D. Stiff Soil	600 to 1200	15 to 50	1000 to 2000
E. Soft Clay Soil	< 600	< 15	< 1000
	Any profile with more than 10 ft of soil that has the following characteristics: - Plasticity Index PI > 20 - Moisture Content w_{mc} ≥ 40% - Undrained Shear Strength $\bar{S}_u$ < 500 lb/ft^2		
F. Soils requiring site response analysis in accordance with Section 21.1	See Section 20.3.1		

Note: For SI: 1 ft = 0.3048 m; 1 ft/s = 0.3048 m/s; 1 lb/ft^2 = 0.0479 kN/m^2

Soil classification helper table in ProtaStructure

Definition of Seismic Parameters

Short Period Spectral Acceleration, S_S, and 1-sec Period Spectral Acceleration, S_1, should be obtained for hazard levels **BSE-2N, BSE-2E, BSE-1N, BSE-1E** and entered in ProtaStructure Seismic Parameters UI.



ASCE07 Seismic parameters UI in ProtaStructure for seismic hazard definition.

Important Note:

BSE-1N is calculated as 2/3 times **BSE-2N (MCER)** and editing of spectral acceleration values is not allowed. BSE-1N must be used in new building design. Other hazard levels in ProtaStructure are intended for performance assessment. *(ASCE41-17 2.4.1.2)*

Site response coefficients, F_a and F_v values are automatically calculated and can be customized if necessary.

Table 11.4.1 and 11.4.2 - Site Response Coefficients

Site Coefficient, Fa

Site class	Mapped Risk-Targeted Maximum Considered Earthquake (MCE_R) Spectral Response Acceleration Parameter at Short Period, S_s					
	$S_s \le 0.25$	$S_s = 0.5$	$S_s = 0.75$	$S_s = 1.0$	$S_s = 1.25$	$S_s \ge 1.50$
SA	0.8	0.8	0.8	0.8	0.8	0.8
SB	0.9	0.9	0.9	0.9	0.9	0.9
SC	1.3	1.3	1.2	1.2	1.2	1.2
SD	1.6	1.4	1.2	1.1	1.0	1.0
SE	2.4	1.7	1.3	*ASCE 07 11.4.8*		
SF	*ASCE 07 11.4.8* (Site-specific response analysis required)					

Site Coefficient, Fv

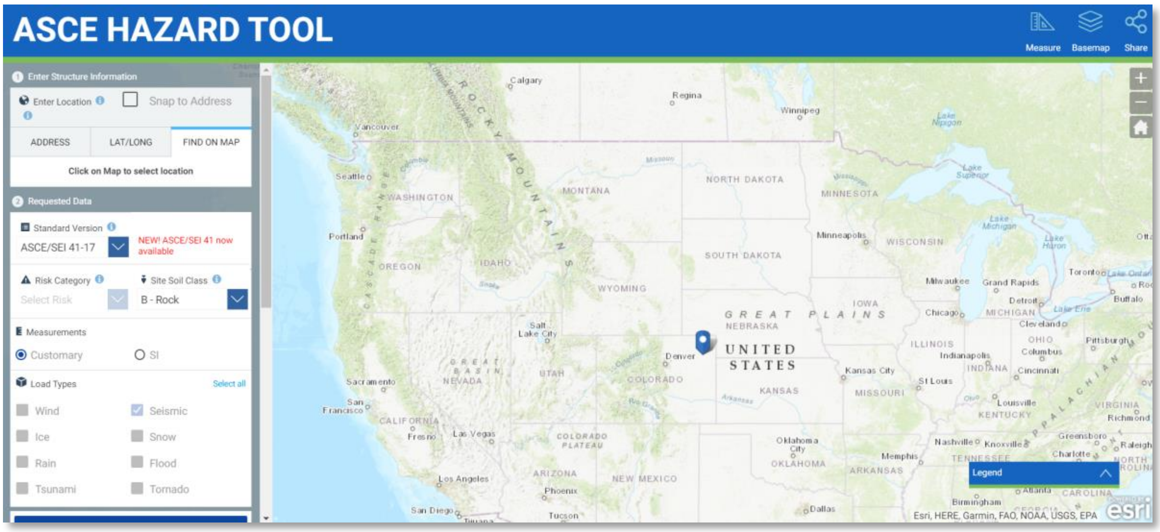
Site class	Mapped Risk-Targeted Maximum Considered Earthquake (MCE_R) Spectral Response Acceleration Parameter at 1-s Period, S_1					
	$S_1 \le 0.1$	$S_1 = 0.2$	$S_1 = 0.3$	$S_1 = 0.4$	$S_1 = 0.5$	$S_1 \ge 0.6$
SA	0.8	0.8	0.8	0.8	0.8	0.8
SB	0.8	0.8	0.8	0.8	0.8	0.8
SC	1.5	1.5	1.5	1.5	1.5	1.4
SD	2.4	2.2	2.0	1.9	1.8	1.7
SE	4.2	*ASCE 07 11.4.8*				
SF	*ASCE 07 11.4.8* Site-specific response analysis required					

Helper table in ProtaStructure for site response coefficients, F_a and F_v

Obtaining Spectral Acceleration Values, S_s and S_1

Values of S_s and S_1 are specific to the project site and must be obtained from an official seismic hazard map.

ASCE Hazard Tool (https://ascehazardtool.org) offers an interactive tool for picking S_s and S_1 values on a map. You can also use this online tool for generating lateral and vertical acceleration spectra for different seismic hazard levels and cross check with ProtaStructure.



The UI for interactive ASCE Hazard Tool web application.

Definitions of S_{XS} and S_{X1}

The short-period spectral acceleration S_{XS} and and the 1-second period spectral acceleration S_{X1} parameters are significant seismic parameters. The equations used for the calculations of these parameters are:

$$S_{XS} = F_a S_s \quad \text{(Eq 2-1)}$$

$$S_{X1} = F_v S_1 \quad \text{(Eq 2-2)}$$

General Horizontal Response Spectrum

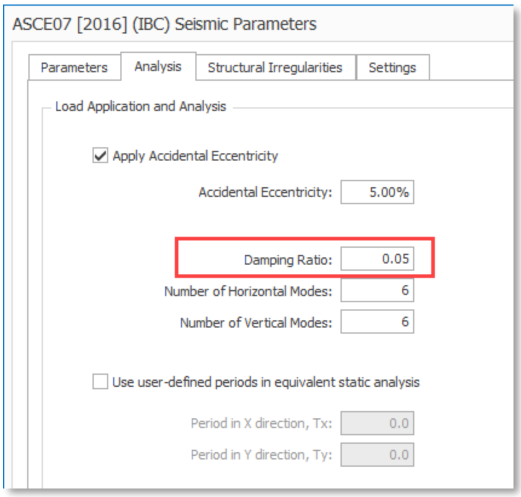
The General Horizontal Response Spectrum is an enhanced version of the horizontal spectrum definition used in ASCE 7 11.4.6.

The spectral response acceleration S_a is plotted against the structural period, T, horizontally, using S_{XS}/B_1 and S_{X1}/B_1 instead of S_{DS} and S_{D1}, respectively.

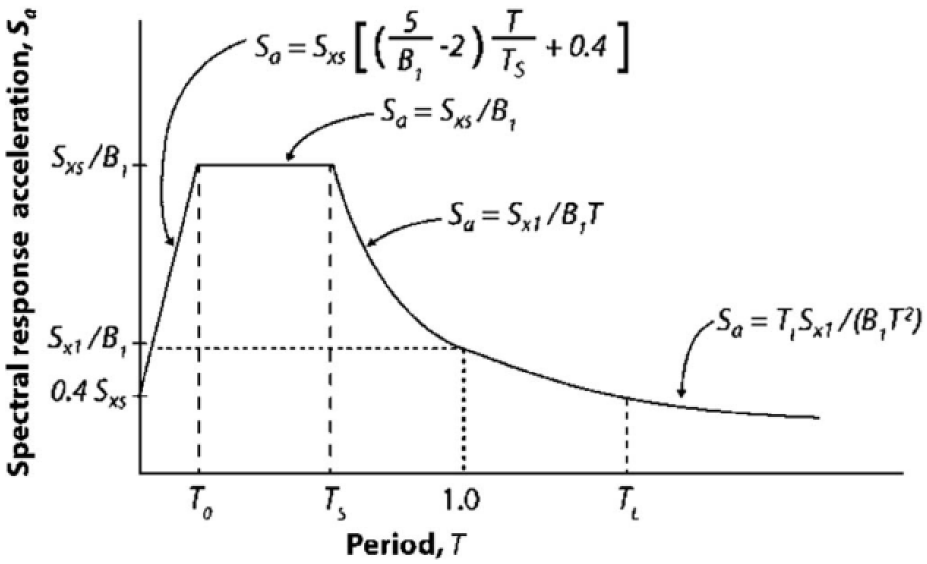
The equation for the value of B_1 is provided below, where β is the effective viscous damping ratio.

$$B_1 = 4/[5.6 - \ln(100\beta)] \quad \text{(Eq 2-3)}$$

Effective viscous damping ratio can be entered in ProtaStructure using **Seismic Parameters > Analysis > Damping Ratio** field.



The general horizontal response spectrum graph is provided below:



$T_O = 0.2\ S_{D1}/S_{DS}$ (ASCE07-22 11.4.5.2)

$T_S = S_{D1}/S_{DS}$ (ASCE07-22 11.4.5.2)

ASCE 41-17 Building Assessment

The ASCE 41-17 building assessment stages are as follows:

- **Tier 1 Screening** - ASCE 41/17 Section 4
- **Tier 2 Deficiency-based Evaluation and Retrofit** - ASCE 41/17 Section 5
- **Tier 3 Systematic Evaluation and Retrofit** - ASCE 41/17 Section 6

Important:

Only **Tier 3** Assessment is within the scope of ProtaStructure.

Tier 3 Systematic Assessment and Retrofit

The Tier 3 Systematic Assessment and Retrofit process provides a comprehensive and detailed analysis of a building's seismic performance. This involves the most rigorous evaluation and retrofit design, ensuring that buildings can meet high performance objectives under seismic loads.

Components of a Tier 3 evaluation are:

Detailed Construction Information: Gathering comprehensive construction information, including structural and architectural drawings, material properties, and detailed field investigations.

Comprehensive Condition Assessment: Conducting a wide-ranging assessment of the building's current condition, identifying any existing damages, deterioration, or other issues that may affect seismic performance.

Advanced Analytical Methods: Utilizing advanced linear and nonlinear analysis methods to accurately model and evaluate the building's response to seismic forces. Both static (pushover) and dynamic (time-history) analyses are frequently used.

Performance Objectives: Clearly defined performance objectives that guide the evaluation process, ranging from Immediate Occupancy to Collapse Prevention. These objectives determine the rigor of the evaluation and retrofit criteria.

Main Steps of Tier 3 Evaluation

Data Collection and Documentation: Collect detailed information about the building's configuration, materials, and current conditions. This includes comprehensive architectural and structural plans, material tests, and field surveys.

Detailed Condition Assessment: Conduct a through inspection to document any existing structural damage, material deterioration, and other relevant conditions.

Analytical Modeling: Develop a detailed analytical model that accurately represents the building's structural behavior under seismic loads. Advanced analysis techniques, such as finite element modeling, can be used to capture complex interactions within the structure.

Evaluation of Structural Components: Assess each structural component using both linear and nonlinear procedures to determine its capacity and performance under seismic loading. Identify and quantify deficiencies that could impair the building's performance.

Retrofit Design: Design retrofit measures to address identified deficiencies. This includes adding new structural elements, strengthening existing components, and improving connections. Retrofit designs must be detailed and precise to meet the specified performance objectives.

Verification and Validation: Conduct additional analyses to verify that the retrofitted building meets the desired performance levels. This may involve iterative cycles of modeling, analysis, and design adjustments. Validate the retrofit design through peer reviews, sensitivity analyses, and, if necessary, physical testing of components.

Analysis Procedures and Acceptance Criteria

Before presenting the analysis methods and acceptance criteria, the topics of Knowledge Level and Performance Objectives must be explained.

Knowledge Level

The Knowledge Factor is used to account for the confidence level in the data and information available about the existing building. It reflects the completeness and reliability of the information used in the seismic evaluation and retrofit design process. (ASCE 41/17 6.2.4)

Data Collection

The Knowledge Factor is influenced by the scope and quality of the data collected during the building evaluation. This includes construction drawings, material properties, and field surveys.

Knowledge Levels

The standard defines different knowledge levels based on the comprehensiveness of the data.

Comprehensive Knowledge Level (KL1): Complete and accurate information is available, including detailed construction drawings and comprehensive material tests. (10.2.2.4.2)

Usual Knowledge Level (KL2): Partial information is available, supported by field verification and testing. (10.2.2.5)

Minimum Knowledge Level (KL3): Limited or unreliable information is available, requiring extensive assumptions and generalizations. (10.2.2.5)

Determination of Knowledge Level Value

The knowledge level values are obtained from the Table 6-1 Data Collection Requirements in ASCE 41/17.

Table 6-1. Data Collection Requirements

	Level of Knowledge					
Data	**Minimum**		**Usual**		**Comprehensive**	
Performance Level	Life Safety (S-3) or lower		Damage Control (S-2) or lower		Immediate Occupancy (S-1) or lower	
Analysis Procedures	LSP, LDP		All		All	
Testing	No tests[a]		Usual testing		Comprehensive testing	
Drawings	Design drawings	Field survey drawings prepared in absence of design drawings	Design drawings	Field survey drawings prepared in absence of design drawings	Design drawings	Field survey drawings prepared in absence of design drawings
Condition Assessment[b]	Visual	Comprehensive	Visual	Comprehensive	Visual	Comprehensive
Material Properties	From design drawings (or documents)[c]	From default values	From design drawings (or documents) and tests	From usual tests	From design drawings (or documents) and tests	From comprehensive tests
Knowledge Factor (κ)[d]	0.9[e,f]	0.75	1.00	1.00	1.00	1.00

Note: LSP, linear static procedure; LDP, linear dynamic procedure.

a. Except for cases where default material properties not provided in this standard.

b. If sufficient component detailing information is not available from the design drawings, any missing information must be completed through a comprehensive assessment.

c. In cases where material properties are missing in the design drawings (or documents), default values with κ = 0.75 can be used.

d. For additional material special requirements and limitations, see Sections 9 to 12.

e. If the building meets the benchmarking requirements of Table 3-2, κ = 1.0.

f. If inspection or test records are available to validate the design drawings, κ = 1.0.

Determination of Performance Objectives

The performance objectives are provided within Table C2-2 Performance Objectives in ASCE 41/17. The target performance for the analyses is selected based on these objectives. The selected objectives are classified as advanced objectives and limited objectives.

Table C2-2. Performance Objectives

	Target Building Performance Levels			
Seismic Hazard Level	**Operational Performance Level (1-A)**	**Immediate Occupancy Performance Level (1-B)**	**Life Safety Performance Level (3-C)**	**Collapse Prevention Performance Level (5-D)**
50%/50 years	a	b	c	d
BSE-1E(20%/50 years)	e	f	g	h
BSE-2E(5%/50 years)	i	j	k	l
BSE-2 N(ASCE 7 MCE_R)	m	n	o	p

Notes: Each cell in the above matrix represents a discrete Performance Objective.
The Performance Objectives in the matrix above can be used to represent three types of Performance Objectives, as discussed below, that might be selected for a building that is assigned to Risk Category I or II, as follows:

Basic Performance Objective for Existing Buildings (BPOE)	**g and l**
Enhanced Objectives	g and either i, j, m, n, o, or p l and either e or f g and l plus either a or b k, m, n, or o alone
Limited Objectives	g alone l alone c, d, e, or f

Demand – Capacity Ratio (DCR)

The magnitude and distribution of inelastic demands for existing and added primary elements and components should be defined by demand-capacity ratios (DCRs). The DCR equation is provided in as 7-16.

$$DCR = \frac{Q_{UF}}{Q_{CE}} \quad \text{(Eq 7-16)}$$

- DCRs must be calculated for every action of each primary component (e.g., axial force, moment, or shear).
- The critical action for a component will be the action with the largest DCR.
- The DCR for this action will be referred to as the critical component DCR.
- The largest DCR for any element on a particular floor will be referred to as the critical element DCR for that floor.
- If an element on a particular floor contains multiple components, the component with the largest calculated DCR will be designated as the critical component for that element on that floor.

Linear Procedures

Linear procedures are analytical approaches used to evaluate the seismic performance of existing buildings under the assumption of linear elastic behavior. These procedures do not consider material nonlinearities and are suitable for preliminary assessments, regular buildings, or when nonlinear analysis is not feasible. ASCE/SEI 41-17 outlines two types of linear procedures: the Linear Static Procedure (LSP) and the Linear Dynamic Procedure (LDP), each with specific modeling, load combination, and acceptance criteria requirements.

Linear Static Procedure

If the Linear Static Procedure (LSP) is chosen for seismic analysis of the building, seismic forces, the distribution of these forces according to the building height, and the corresponding internal forces and system displacements should be determined using a linear elastic static analysis as specified in ASCE41/17 7.4.1.

Linear Dynamic Procedure

The Linear Dynamic Procedure (LDP) is used for seismic analyses that consider the dynamic response of buildings. This procedure involves the use of a detailed mathematical model to simulate the behavior of a building under seismic loads.

LDP includes calculating the dynamic response of a building to seismic ground motions, typically using modal analysis or response spectrum analysis. (ASCE 41/17 Section 7.4.2.1)

Force-Controlled Actions in Linear Procedures

Force-controlled actions refer to structural responses that are highly dependent on the ability of components to withstand applied forces without significant deformation. These components are typically characterized by brittle behavior, meaning they cannot undergo large inelastic deformations before failure.

Demands: For force-controlled actions, the demand is determined by forces applied to the structural component, such as axial loads, shear forces, and bending moments.

Capacity: The capacity is the maximum force the component can withstand without failure. This is based on the material's strength properties and the component's design.

Evaluation: The adequacy of force-controlled components is evaluated by ensuring that the applied forces do not exceed the component's capacity. The Demand-Capacity Ratio (DCR) equation is provided below. For details, please refer to the relevant section of ASCE41.

$$\mathrm{Q_{UF}} = \mathrm{Q_G} \pm \frac{\chi \mathrm{Q_E}}{C_1 C_2 J} \quad \text{(Eq. 7-35)}$$

Deformation-Controlled Actions in Linear Procedures

Deformation-controlled actions refer to structural responses that are highly dependent on the ability of components to undergo significant deformations without losing their load-carrying capacity. These components exhibit ductile behavior, meaning they can undergo large inelastic deformations before failure.

Demands: For deformation-controlled actions, the demand is determined by deformations or displacements applied to the structural component, such as displacements, rotations, and drifts.

Capacity: The capacity is the maximum deformation the component can undergo without losing its load-carrying capability. This is based on the ductility and inelastic deformation properties of the material and its design.

Evaluation: The adequacy of deformation-controlled components is evaluated by ensuring that the applied deformations do not exceed the component's deformation capacity. The Demand-Capacity Ratio (DCR) equation is provided below. For details, please refer to the relevant section of ASCE41.

$$\mathrm{Q_{UD}} = \mathrm{Q_G} + \mathrm{Q_E} \quad \text{(Eq. 7-34)}$$

Acceptance Criteria for Deformation-Controlled Actions in Linear Procedures

Acceptance criteria for deformation-controlled actions for LSP or LDP in primary and secondary components are provided by Equation 7-36.

$$mkQ_{CE} > Q_{UD} \quad \text{(Eq. 7-36)}$$

m = Component capacity modification factor to account for the expected ductility associated with this action at the selected Structural Performance Level given in **ASCE41 Tables 10-10a, 10-10b, 10-13, 10-21, 10-22**

Q_{CE} = Expected strength of a component's deformation-controlled action at the considered deformation level.

Q_{UD} = Expected strength of a component's deformation-controlled action at the considered deformation level.

K = Knowledge level coefficient

Acceptance Criteria for Force-Controlled Actions in Linear Procedures

Acceptance criteria for force-controlled actions for LSP or LDP in primary and secondary components are provided by Equation 7-37.

$$kQ_{CL} > Q_{UF} \quad \text{(Eq. 7-37)}$$

K = Knowledge level coefficient

Q_{CL} = Strength of a component's force-controlled action at the considered deformation level.

Q_{UF} = Force-controlled action resulting from the combination of gravity loads and earthquake forces.

Nonlinear Procedures

Nonlinear Procedures are analytical methods used to assess the seismic performance of structures by directly considering the nonlinear behavior of structural components. ASCE 41-17 outlines two primary nonlinear procedures: the Nonlinear Static Procedure (NSP) and the Nonlinear Dynamic Procedure (NDP).

Nonlinear Static Procedure

The Nonlinear Static Procedure (NSP) is a method used to evaluate the performance of structures during seismic events. The NSP aims to determine the maximum displacement that a structure may experience under a certain level of seismic hazard. It uses a mathematical model that directly accounts for the nonlinear response of materials and provides reasonable estimates of the internal forces expected during seismic events. (ASCE 41/17 7.4.3)

Modeling and General Requirements for NSP

Selection of Control Node: A control node should be selected to represent the overall displacement demand of the structure. This node is typically located at the roof or a significant point in the structural system.

Selection of Seismic Force Patterns: Seismic force patterns should be selected to reflect the distribution of inertial forces. Common patterns include uniform, triangular, and mode shapes derived from eigenvalue analysis.

Determination of Fundamental Period: The fundamental period of the building should be determined using appropriate methods. This period is critical for understanding the dynamic characteristics of the structure and is typically found using modal analysis.

Application of Analysis Method: The method involves the application of monotonically increasing lateral loads representing the inertial forces in an earthquake until the target displacement is reached. The relationship between the base shear force and the lateral displacement of the control node should be established for displacements ranging from 0 to 150% of the target displacement (δt).

Inclusion of Gravity Loads

Component Gravity Loads: The model should include gravity loads acting on components to accurately reflect the combined effects of seismic and gravity forces.

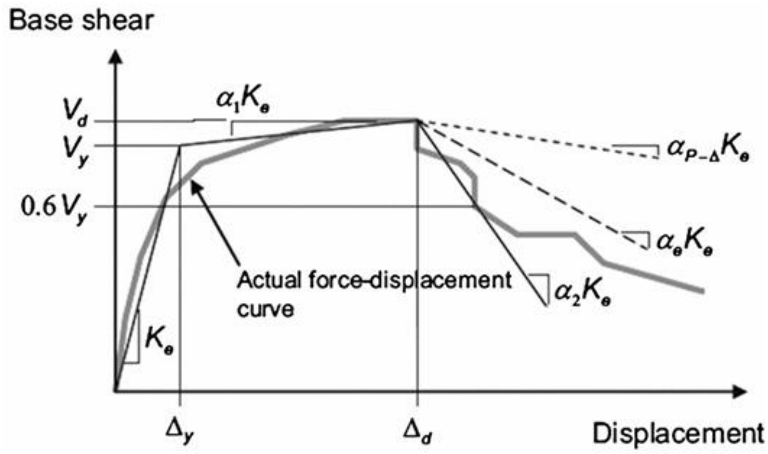
Seismic Force Directions: Seismic forces should be applied in both positive and negative directions. The maximum seismic effects should be used for analysis.

Force-Deformation Response: The model should be detailed to represent the force-deformation response along the length of each component. This helps identify locations of nonlinear action and ensures accurate analysis of the structure's behavior.

Force-Displacement Behavior: The force-displacement behavior of all components should be explicitly included in the model using full backbone curves. These curves should include strength degradation and any residual strength.

Idealized Force-Displacement Curve for NSP

This section explains how to develop an idealized force-displacement curve for the Nonlinear Static Procedure (NSP). The objective is to convert the complex nonlinear relationship between base shear and control node displacement into a more manageable form for analysis. (ASCE 41/17 7.4.3.2.4)



Target displacement is determined by:

$$S_t = C_0 C_1 C_2 S_a \frac{T_e^2}{4\pi^2} g \quad \text{(Eq. 7-28)}$$

C_0: Modification factor used to relate the spectral displacement of an equivalent single-degree-of-freedom (SDOF) system to the roof displacement of the building. This factor adjusts the spectral displacement to reflect the actual roof displacement of a multi-degree-of-freedom (MDOF) building system. (ASCE 41/17 Table 7-5)

C_1 = Modification factor used to relate the expected maximum nonlinear displacements to the displacements calculated for linear elastic response. This factor accounts for the increase in displacement due to the nonlinear behavior of building materials. (ASCE41-17 eq. 7-29)

C_2 = Modification factor representing the effect of pinched hysteretic shape, stiffness degradation, and strength deterioration on the maximum displacement response. This factor adjusts for hysteresis effects in material behavior, as well as reductions in stiffness or strength. (ASCE41-17 eq. 7-30)

T_e = Effective period. (ASCE41-17 7.4.3.2.5)

S_a: Response spectrum acceleration at the building's effective fundamental period and damping ratio.

g: Acceleration due to gravity.

$$T_e = T_i \sqrt{\frac{K_i}{K_e}} \quad \text{(Eq. 7-27)}$$

T_i: Elastic fundamental period in the relevant direction, determined through eigenvalue analysis.

K_i: Elastic lateral stiffness in the relevant direction, calculated from the idealized Force-Displacement curve for the Nonlinear Static Procedure (NSP).

K_e: Effective lateral stiffness in the relevant direction, derived from the idealized Force-Displacement curve for the Nonlinear Static Procedure (NSP).

Nonlinear Dynamic Procedure

The Nonlinear Dynamic Procedure (NDP) utilizes a mathematical model that explicitly incorporates the nonlinear load-deformation characteristics of individual building components. This model is subjected to ground motion acceleration time histories representing seismic shaking to obtain forces and displacements. The results are directly compared to the specified acceptance criteria. (ASCE 41/17 Section 7.4.4)

The general modeling and analysis requirements for NDP are similar to those of the Nonlinear Static Procedure (NSP), except that control nodes and target displacements are not considered. Instead, the analysis directly uses ground motion acceleration data to compute displacements and forces.

Unacceptable Conditions in NDP

Due to (ASCE 41/17 Section 7.5.3.2.1):

Analytical Convergence: The analysis must convergo to a solution. Non-convergence may indicate potential global instabilitiy or modeling errors.

Deformation Limits: Predicted deformations in deformation-controlled elements must remain valid modeling ranges.

Force-Controlled Actions: For critical force-controlled actions modeled elastically, the estimated demands must not exceed the expected capacities.

Gravity Load Capacity: Elements no explicitly modeled must not exceed deformation limits that would compromise their ability to carry gravity loads.

Acceptance Criteria for Force-Controlled Actions in Nonlinear Procedures

This section defines the acceptance criteria for **force-controlled components** evaluated using nonlinear static (NSP) or nonlinear dynamic (NDP) procedures.

Force-controlled components must satisfy the following condition:

$$\gamma\chi(Q_{UF} - Q_G) + Q_G \leq Q_{CL} \quad \text{(Eq. 7-38)}$$

Where:

- Q_{UF}: Force-controlled demand obtained from NSP or NDP
- Q_G: Gravity load demand
- Q_{CL}: Lower-bound component capacity
- γ: Load factor (Table 7-8)
- χ: Demand modifier (based on performance level)

Table 7-8. Load Factor for Force-Controlled Behaviors

Action Type	γ
Critical	1.3
Ordinary	1.0
Noncritical	1.0

The demand modifier reflects the performance objective of the structure. Higher performance levels require stricter control.

χ Demand modifier factor:

1.0 for Collapse Prevention (CP)

1.3 for Life Safety (LS) or Immediate Occupancy (IO)

Acceptance Criteria for Deformation-Controlled Actions in Nonlinear Procedures

In nonlinear procedures, acceptance criteria are determined by calculating the plastic rotation or deformation capacity of structural components based on design parameters and performance objectives. These criteria vary depending on whether the component is a column, beam, or structural wall, and whether its behavior is governed by flexure or shear. The acceptance values for different performance levels are provided in **Tables 10-7, 10-8, 10-9, 10-19, and 10-20 of ASCE 41-17**, depending on the component type and governing behavior.

The elastic limit used in these procedures is typically determined through moment-curvature analysis of the cross-section, which provides a basis for identifying yield points and evaluating nonlinear deformation capacity.

The general workflow is:

1. Determine design parameters such as reinforcement ratios, axial load ratio, shear capacity, etc.
2. Select the target performance level (Immediate Occupancy, Life Safety, or Collapse Prevention).
3. Use relevant acceptance tables to obtain plastic rotation limits (or other deformation parameters).
4. Compare calculated demands plastic rotation (e.g., drift, chord rotation) with the limits to assess compliance.

For reinforced concrete columns without spiral or seismic ties, acceptance criteria depend on:

1. Axial load ratio
2. Transverse & longitudinal reinforcement ratios
3. Shear demand at yielding

4. Plastic rotation values are obtained from standard acceptance tables. Limits are stricter when transverse reinforcement is low or poorly anchored.

For reinforced beams, acceptance depends on:

1. Reinforcement ratios (total and compression side)
2. Shear demand and concrete strength
3. Plastic rotation values are taken from corresponding tables based on performance level.

For flexure-controlled walls, acceptance is based on:

1. Plastic rotation capacity derived from reinforcement layout and axial load.
2. Proper confinement of boundary elements increases acceptance limits.

For shear-controlled walls and coupling beams:

1. Walls → acceptance is based on inter-story drift
2. Coupling beams → acceptance is based on chord rotation
3. If the axial load exceeds 15% of the gross section capacity, the wall must be treated as force-controlled.
4. Short coupling beams with continuous reinforcement and strong confinement are permitted to use higher acceptance limits.

Application of ASCE 41-17 in ProtaStructure

In this section, a structural model will be created in **ProtaStructure** and evaluated based on the performance criteria defined in **ASCE 41-17**. The initial part of the section will present general information about the model, including geometry and material properties. Following this, the evaluation parameters and acceptance criteria for structural components such as columns, beams, and walls will be discussed in accordance with ASCE 41-17.

The user interface of ProtaStructure for existing building assessments will be utilized to define analysis parameters and performance objectives. Evaluation results for individual components will be visualized within the interface. Finally, a sample ProtaStructure report generated from a performance assessment compliant with ASCE 41-17 will be presented.

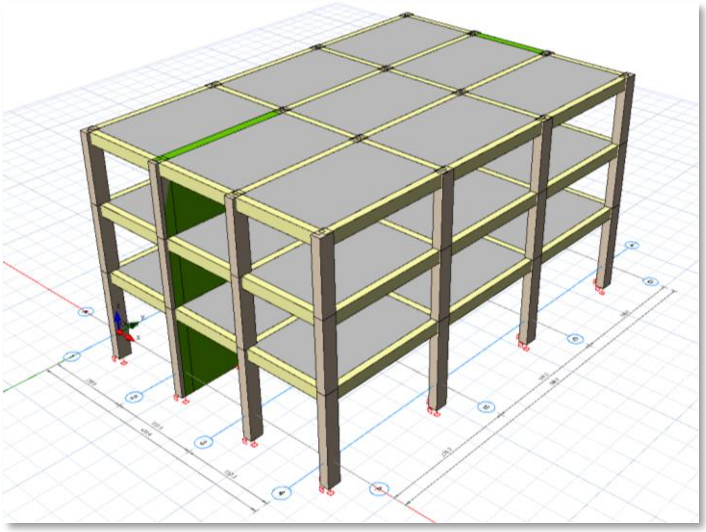
Structural Model Properties

In this section, the properties of the structural model created in ProtaStructure will be presented. The characteristics of the model are provided in the table below.

Structural Model Properties

Number of Stories		3
Storey Height		3 x 9.8 ft = 29.5 ft
Direction X Span Length		3 x 157.5 ft = 472.4 ft
Direction Y Span Length		3 x 236.2 ft = 708.7 ft
Column	Width	15.7 in
	Depth	23.6 in
	Concrete Material	C320
	Steel Material	SD60
Beam	Width	11.8 in
	Depth	19.7 in
	Concrete Material	C300
	Steel Material	SD60
Wall	Width	246.1 in
	Depth	15.7 in
	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:

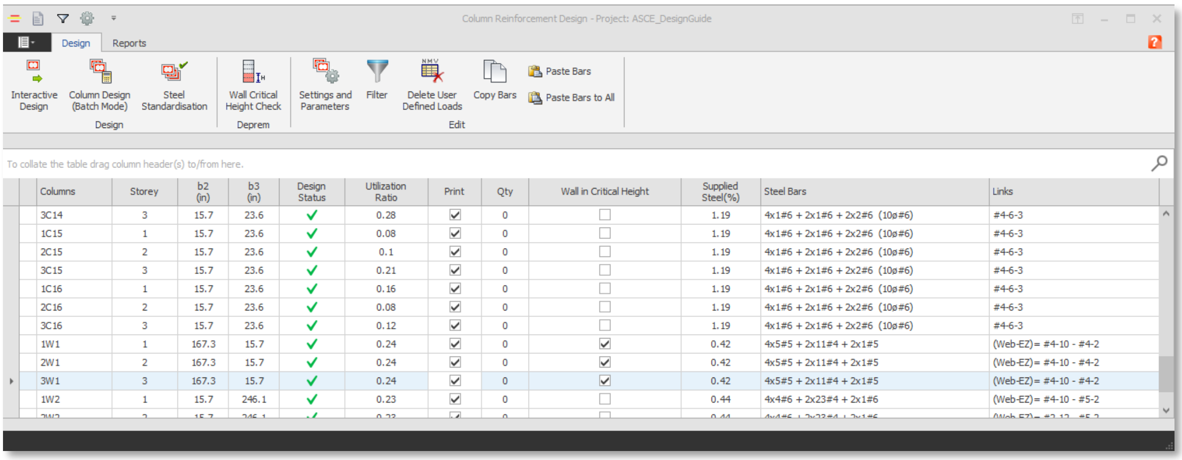
If you are specifying the detailed reinforcements for an existing building, the design status in the design modules may indicate a FAILURE with a RED CROSS sign. Do not worry about this and make sure that you have entered longitudinal and transverse reinforcement as you see in the bluprints. Click OK 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

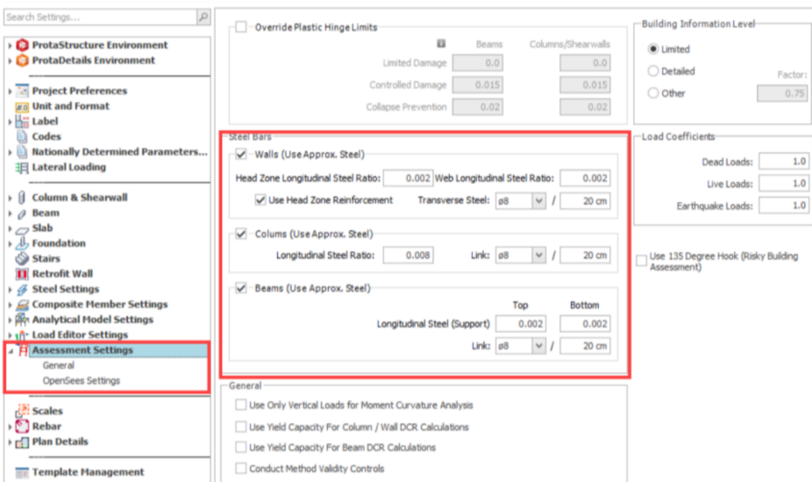
In case you don't have sufficient information on the detailed reinforcements in the members, you can always use 'Estimated Reinforcement' for member groups or individual members. Of course, this estimation must be backed by the site investigations and/or the design codes that were in effect when the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

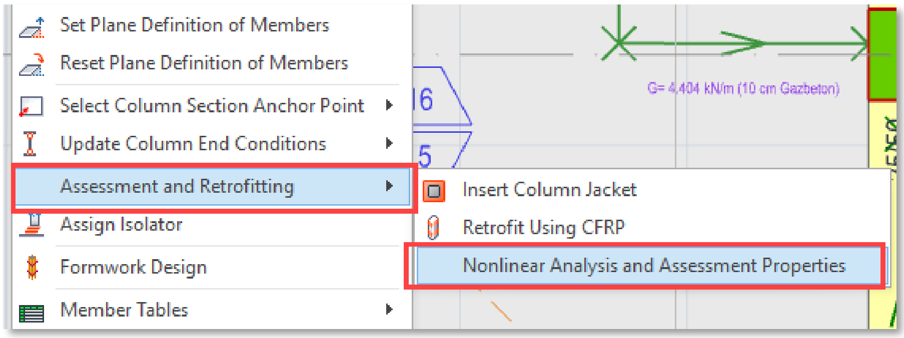
2. Enable the estimated bar ratios for Walls, Columns and/or beams.



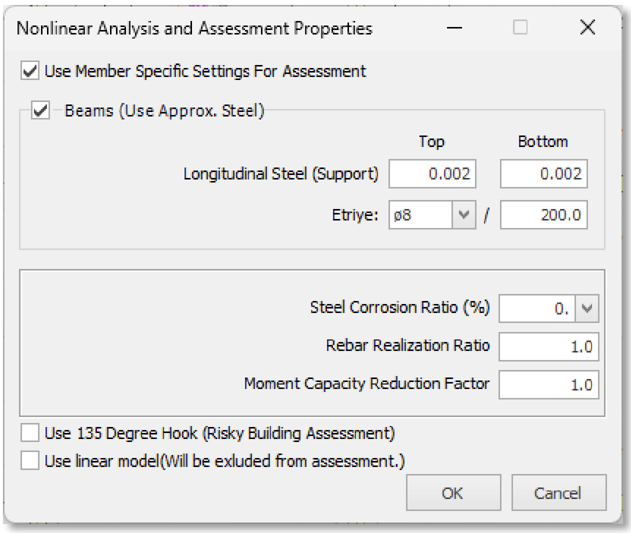
Defining Approximate Reinforcements for Individual Members

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**"

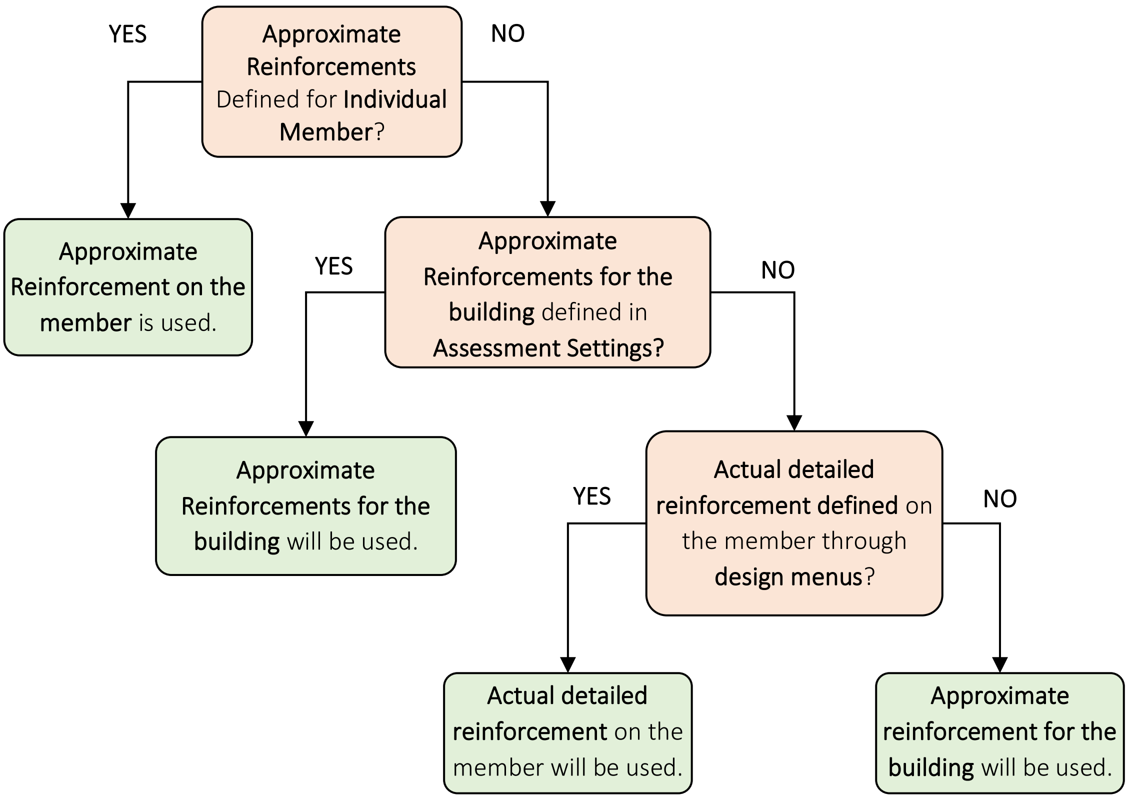
Third Priority: Actual detailed reinforcements defined through design menus directly on the member itself.

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.

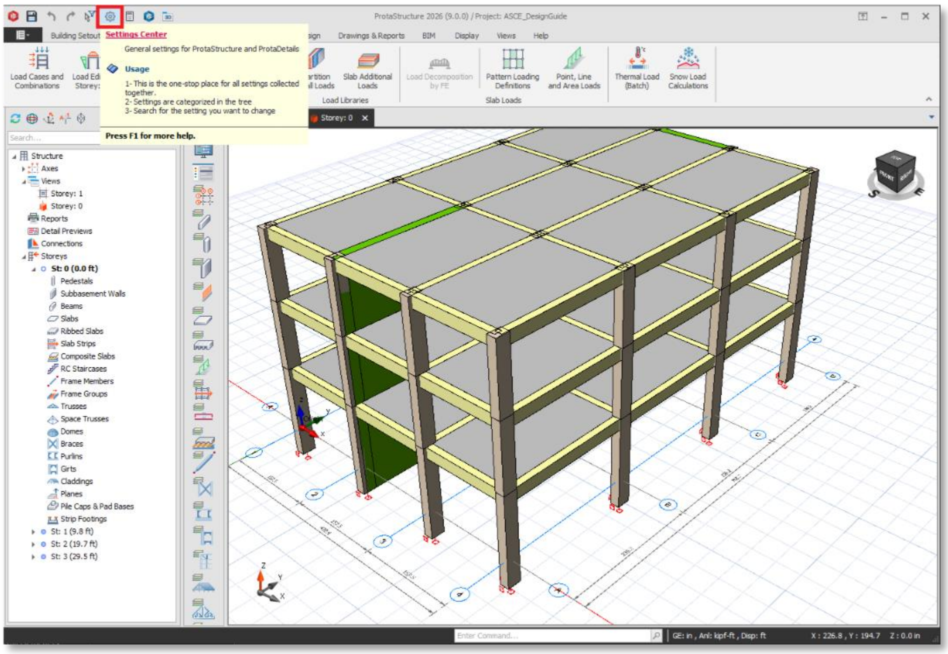


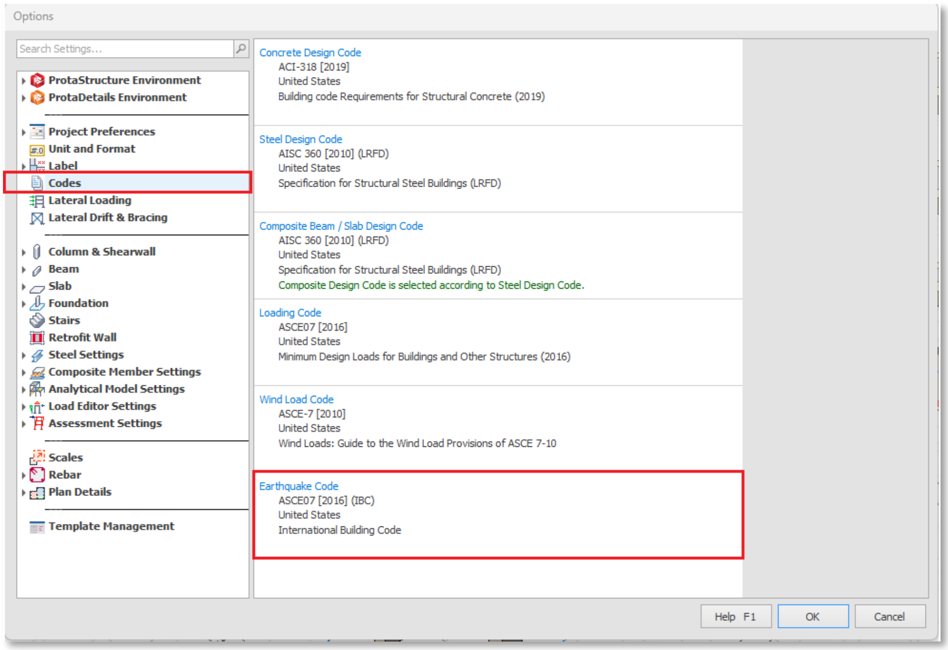
Earthquake Code

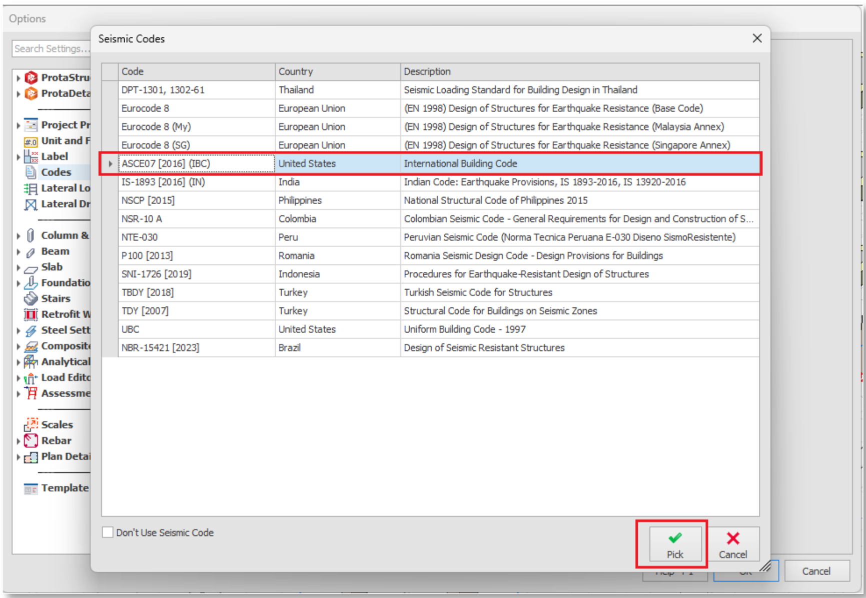
ProtaStructure supports a wide range of seismic design codes required for performance-based evaluations. To perform an analysis in accordance with the **ASCE 41-17** standard, follow the steps below:

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 **ASCE07 [2016] (IBC)**.

Applying these settings enables the necessary **seismic parameters** and **performance objectives** required for evaluating existing buildings under the **ASCE 41-17** framework.







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 **ASCE 41-17** standard. This ensures that all relevant settings align with performance-based design principles specified in the guideline.

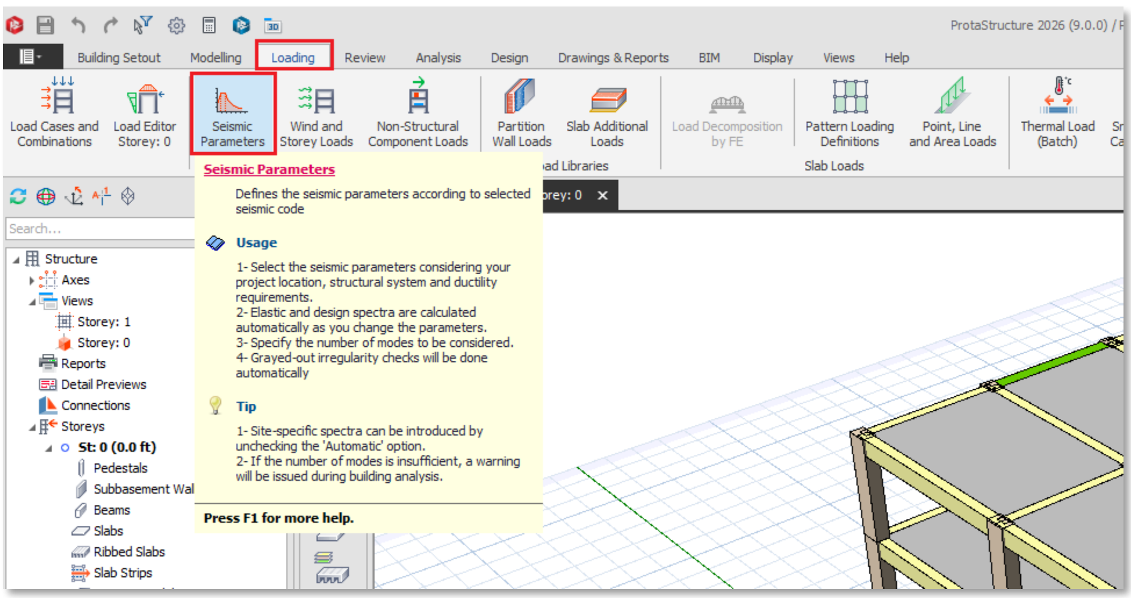
Important Note:

To carry out an assessment according to ASCE 41-17, make sure that ASCE07 [2016] (IBC) is selected under the Earthquake Code settings.

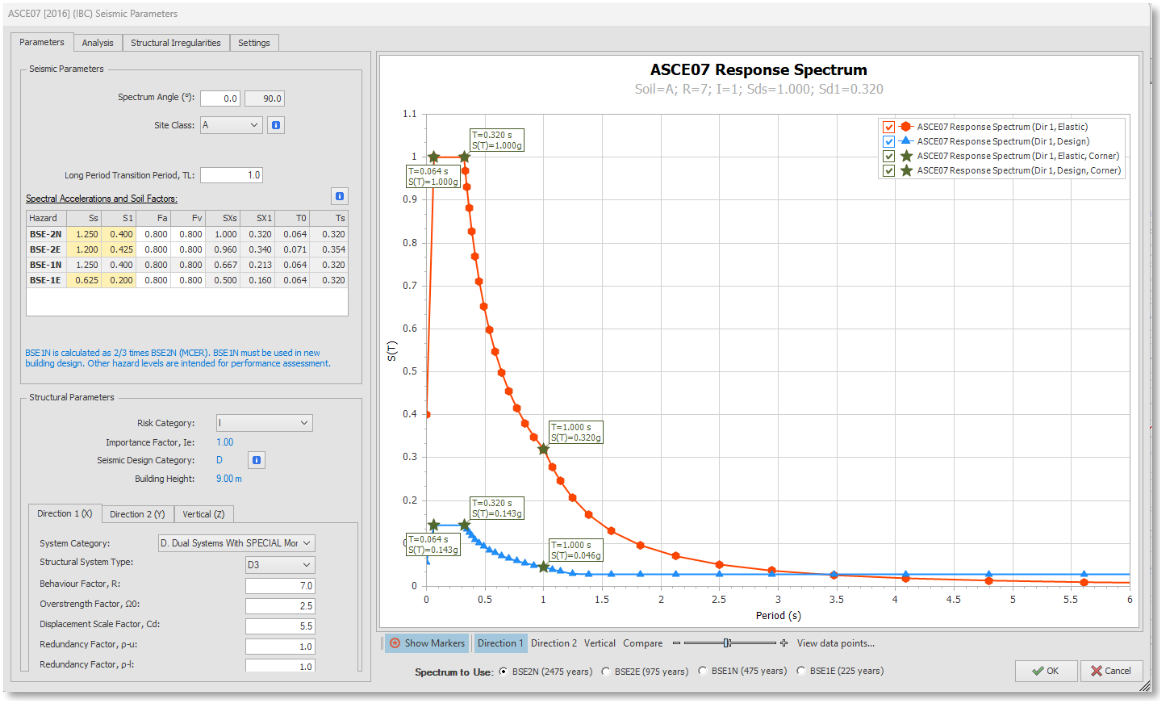
Seismic Parameters

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 values (SDS, SD1), and the Site Class of the structure.

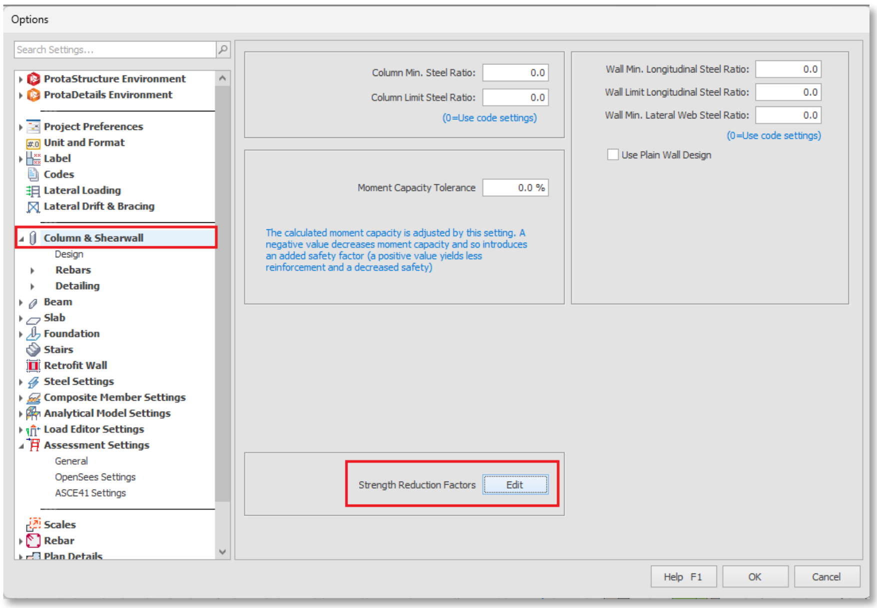
Strength Reduction Factors For Column and Shearwall

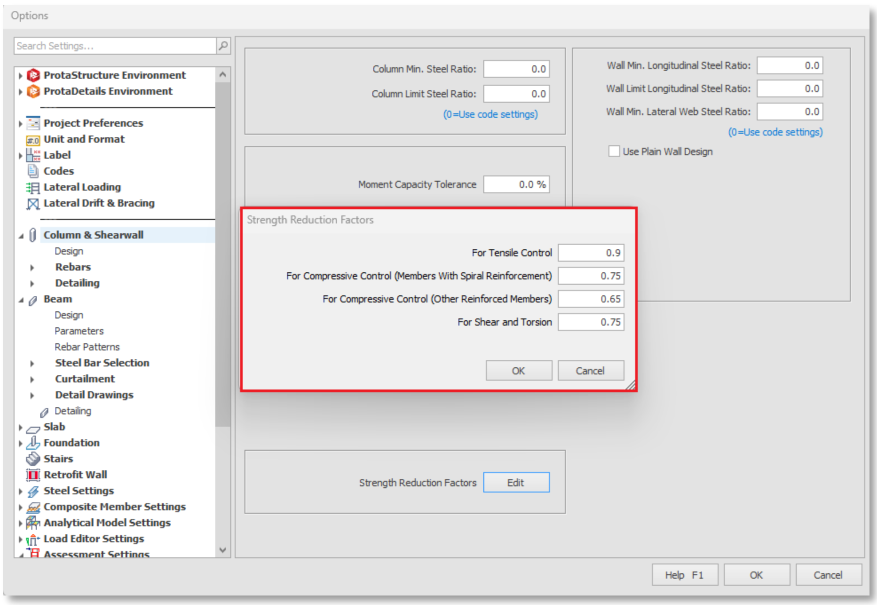
After calculating the capacity of **column and shear wall elements** based on their cross-sectional and material properties, you can apply **strength reduction factors** to account for required safety margins in accordance with design principles.

To define these factors in **ProtaStructure**, follow the steps below:

1. Open the **Settings** panel.
2. Navigate to the **Columns and Shear Walls** section.
3. Click on **Strength Reduction Factors**.

In this interface, you can assign **reduction factors** for different **force components** based on the expected behavior and performance objectives of each element.





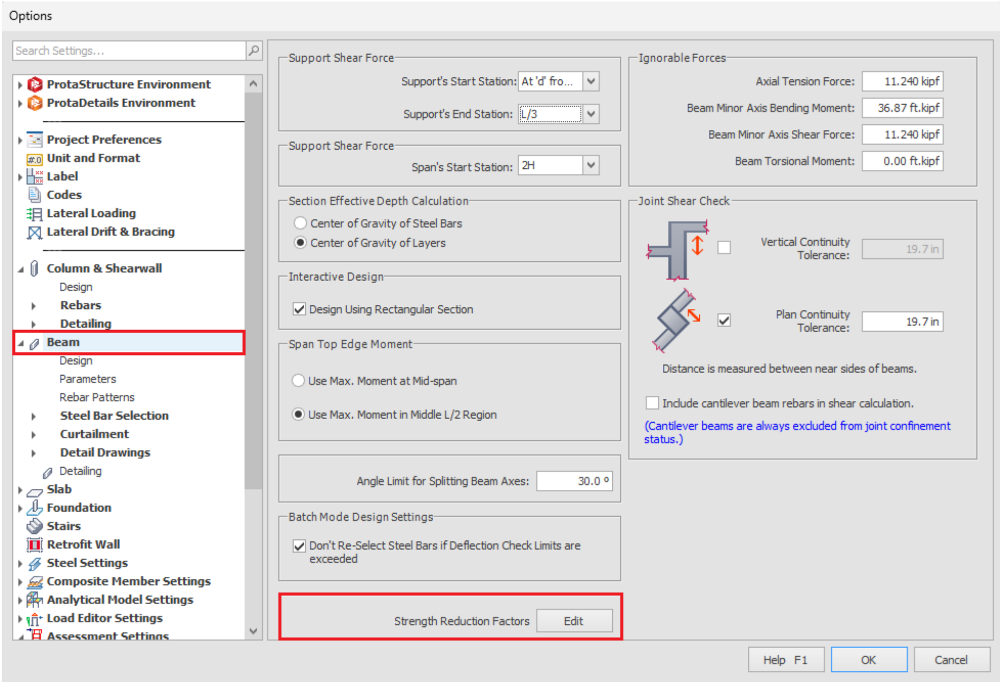
Strength Reduction Factors For Beams

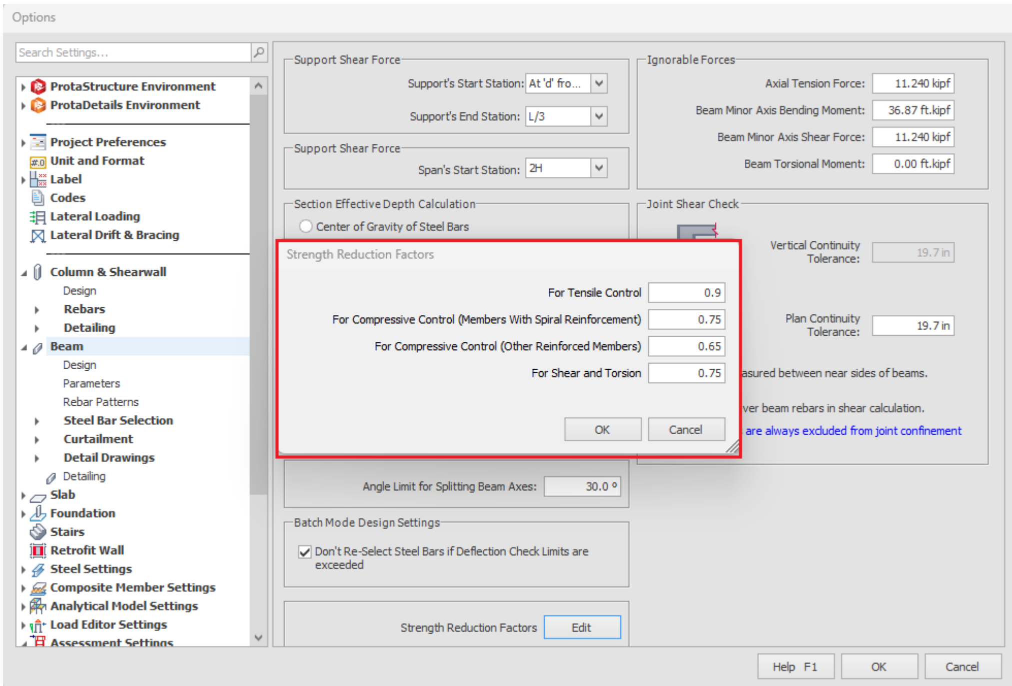
After determining the capacity of **beam elements** based on their cross-sectional geometry and material properties, you can apply **strength reduction factors** to adjust the capacities according to the force components involved.

To configure these factors in **ProtaStructure**, follow the steps below:

1. Open the **Settings** panel.
2. Navigate to the **Beams** section.
3. Click on **Strength Reduction Factors**.

In this interface, you can assign appropriate **strength reduction factors** for each **force component** (such as bending, shear, and axial force) based on design requirements and performance objectives.



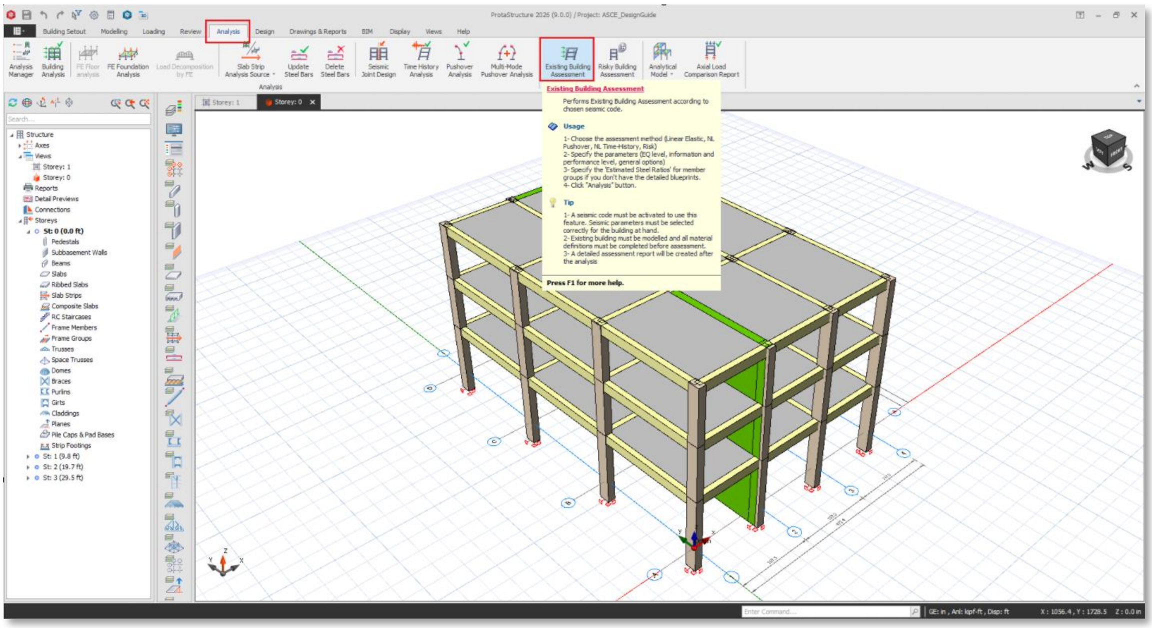


Important Note:

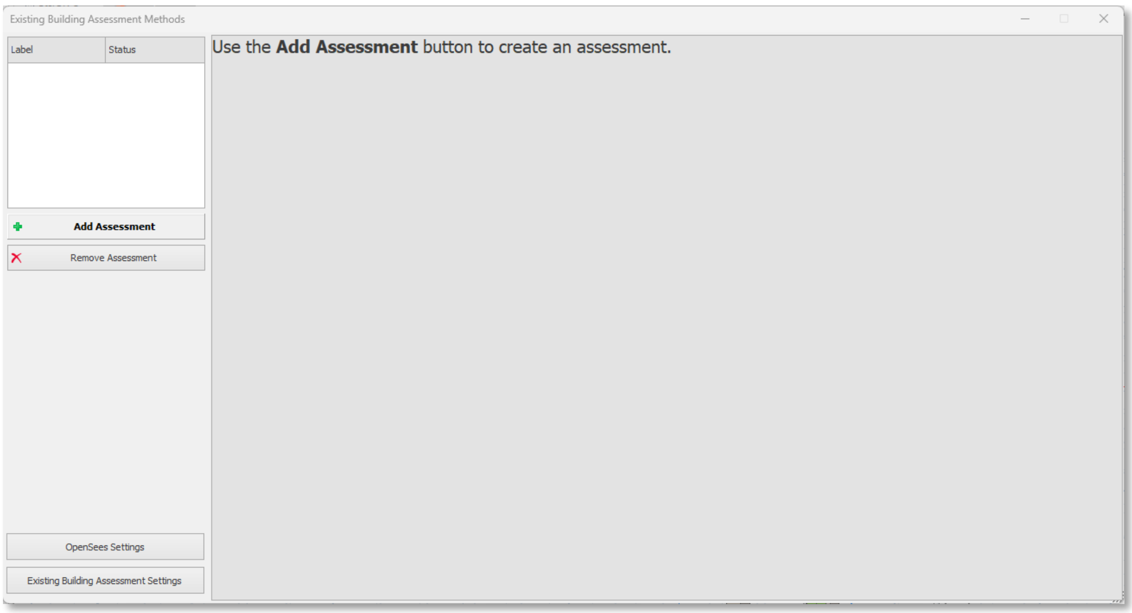
The strength reduction factors defined in this section play a critical role in determining the capacities of structural elements during existing building assessment.

Existing Building Assessment

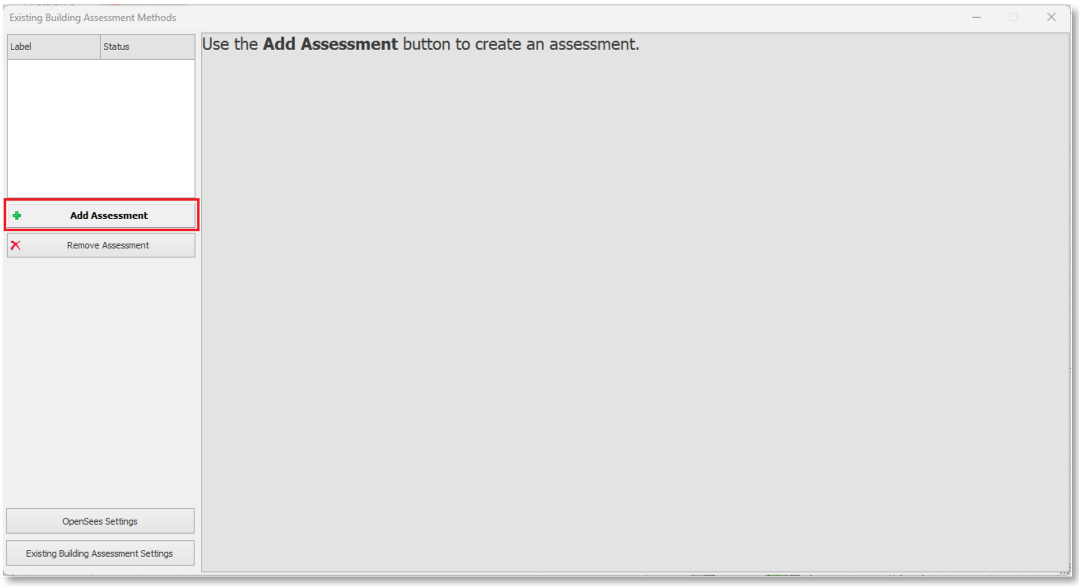
This section explains how to perform an **existing building assessment** step by step within **ProtaStructure**. To begin the process, go to the **Analysis** tab and click on the **Existing Building Assessment** button. The figure below illustrates where this option is located in the user interface.



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.

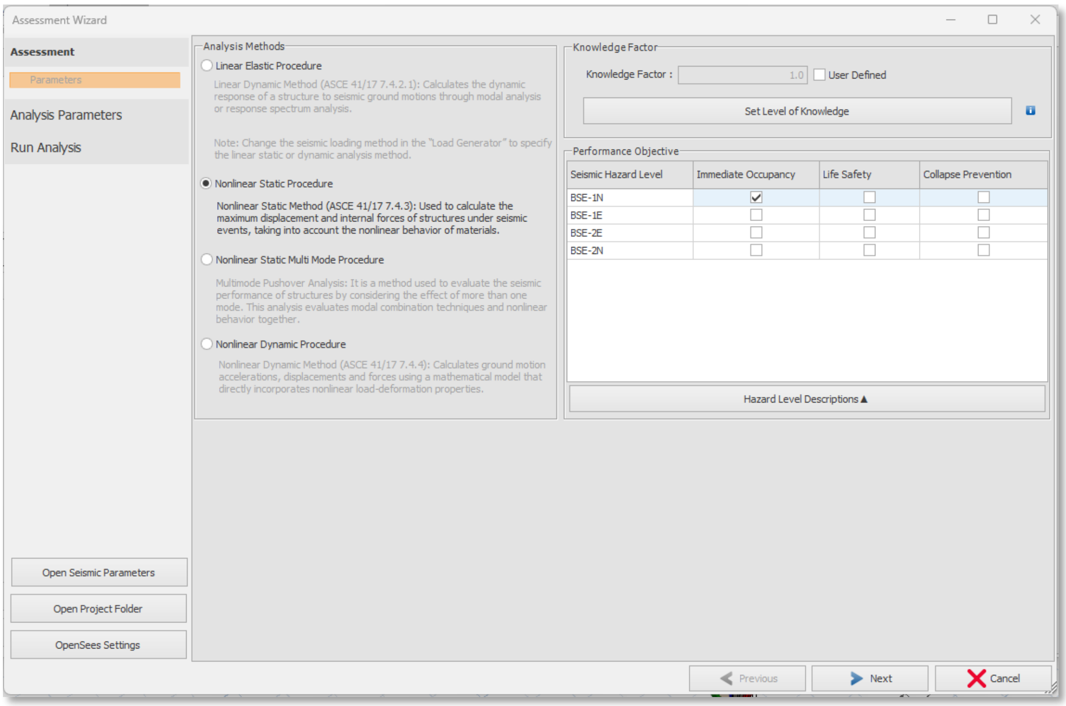


The **Label Status** section displays all previously created assessment scenarios. To create a new assessment, click the **Add Assessment** button.



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 **performance objective** 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.

The user must choose one of the analysis methods defined in **ASCE 41-17 Chapter 7**:

- **Linear Elastic Procedure**
  Evaluates structural response using either static or dynamic linear methods:
  - *Linear Static Procedure (LSP):* Calculates seismic forces and distribution using linear static analysis.
  - *Linear Dynamic Procedure (LDP):* Calculates dynamic response using modal or response spectrum analysis.
    (ASCE 41-17 §7.4.1 & §7.4.2.1)

Note: The selected method must match the seismic loading type set in the **Load Generator**.

- **Nonlinear Static Procedure**
  Calculates internal forces and displacements under seismic loads by considering material nonlinearity using a pushover analysis method.
  (ASCE 41-17 §7.4.3)
- **Nonlinear Static Multi-Mode Procedure**
  Evaluates structural performance by combining nonlinear pushover analysis with multiple vibration modes.
- **Nonlinear Dynamic Procedure**
  Uses time history analysis to calculate accelerations, displacements, and internal forces in a nonlinear model.
  (ASCE 41-17 §7.4.4)

Important Note:

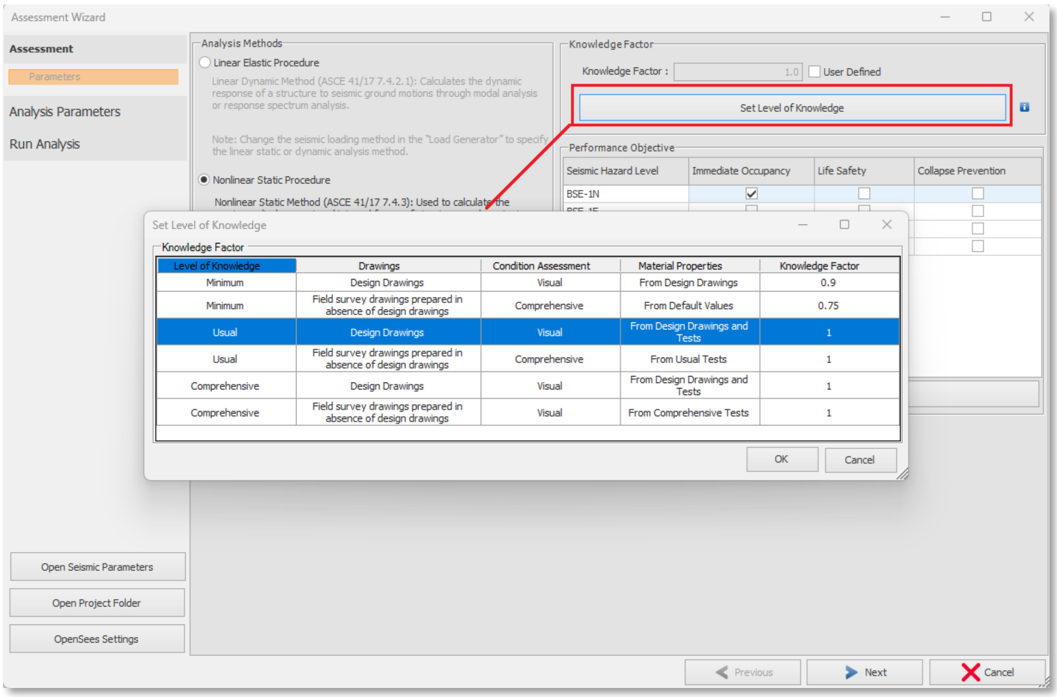
The Nonlinear Static Multi-Mode Procedure is not a standard analysis method defined in ASCE 41-17. However, it is available in ProtaStructure as one of the nonlinear analysis methods.

Assessments conducted using this method are still based on the principles of nonlinear evaluation and are compatible with performance-based assessment workflows.

The **Knowledge Factor** reflects the level of information available for the structure (e.g., as-built drawings, material tests, construction quality). This factor directly affects the strength reduction applied in capacity calculations.

- Default value: **1.0**
- To manually adjust it, check **User Defined**
- Use the **Set Level of Knowledge** button to assign a predefined value based on ASCE 41-17 criteria (Minimum, Usual, Comprehensive)

To define the knowledge factor in **ProtaStructure**, click the **Set Level of Knowledge** button located in the **Assessment Wizard** under the **Knowledge Factor** section.

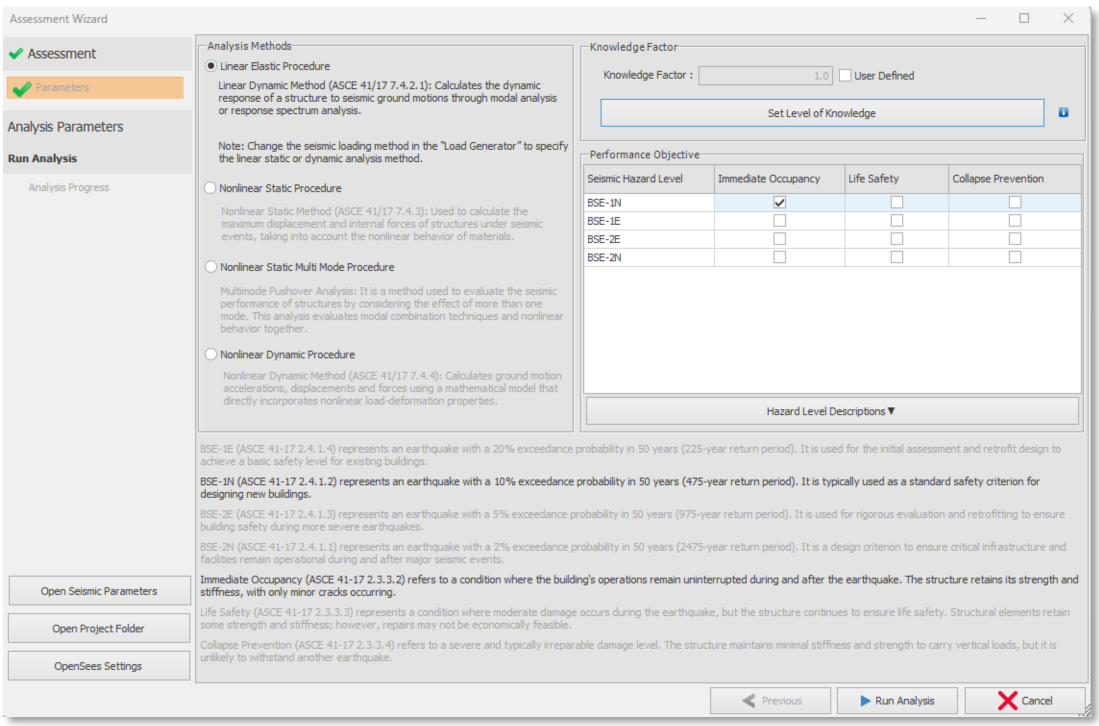


In this section, users must select:

- One **Seismic Hazard Level** (BSE-1N, BSE-1E, BSE-2E, or BSE-2N)
- One corresponding **Performance Objective** (Immediate Occupancy, Life Safety, or Collapse Prevention)

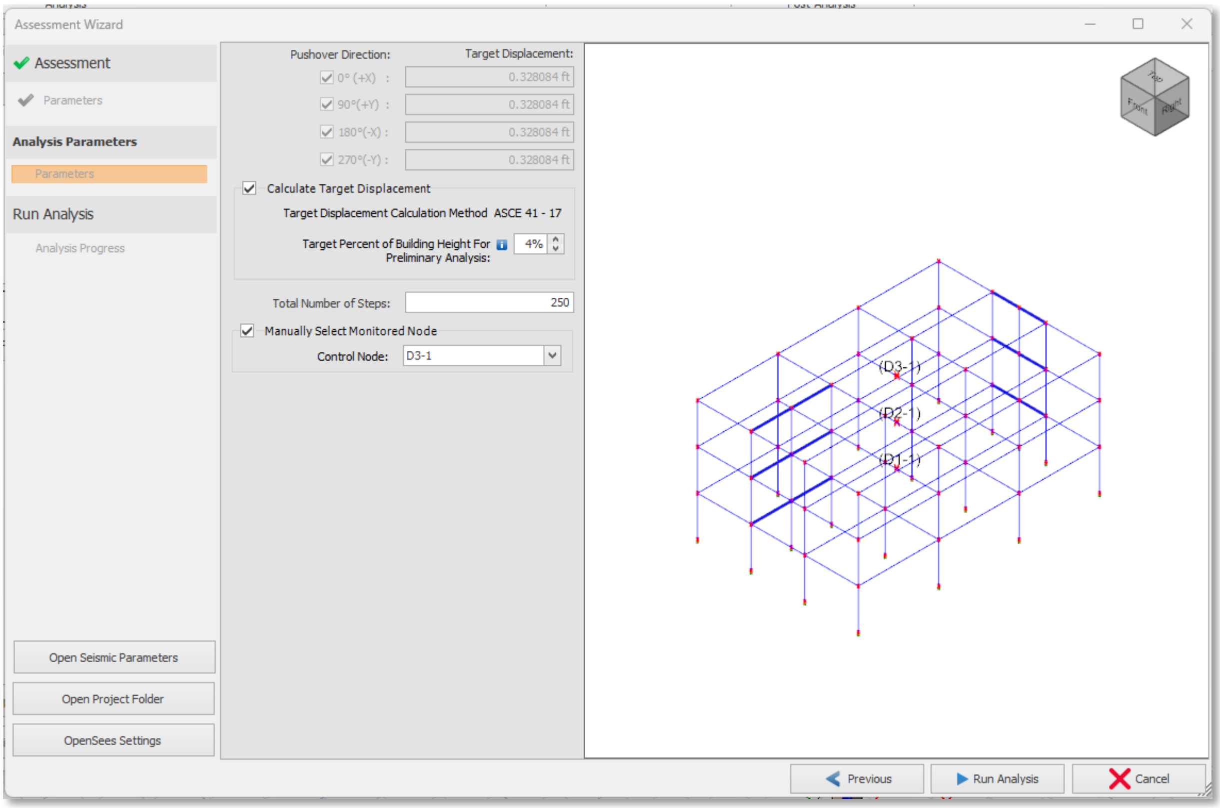
Tip:

Use the **Hazard Level Descriptions** button to view details for each hazard level, as described in ASCE 41-17 Table 2-1.



Nonlinear Static Procedure

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

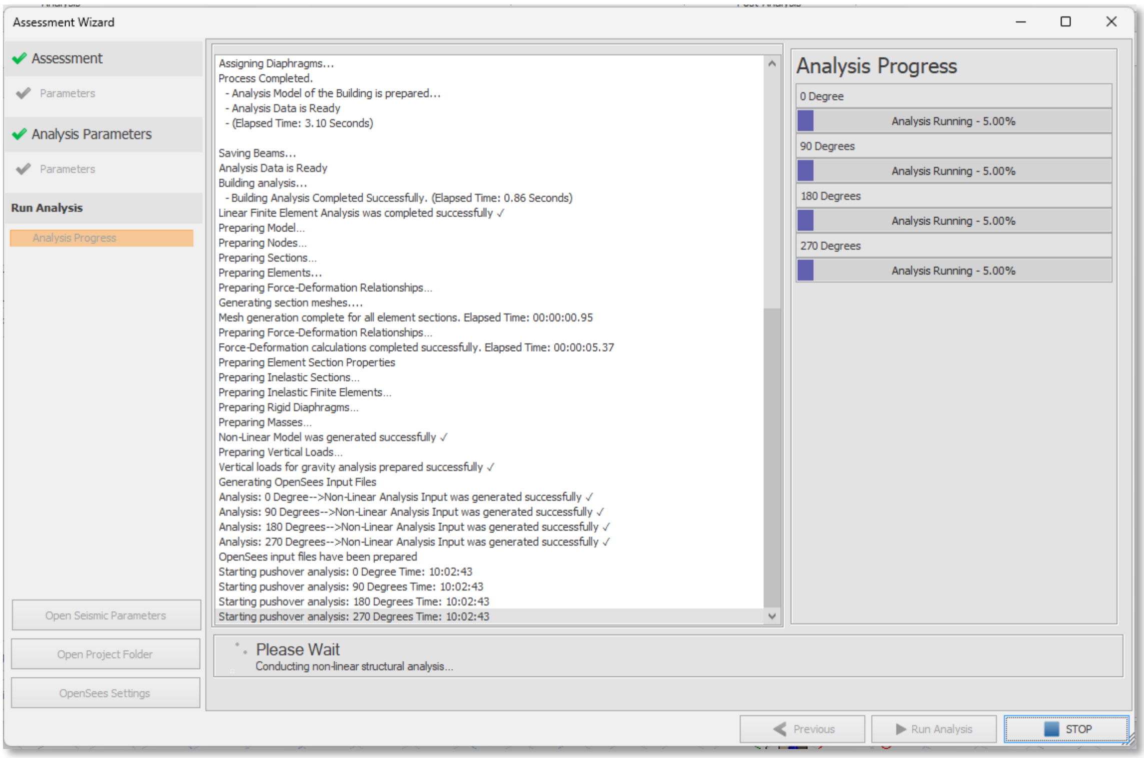


This screen is used to configure the nonlinear static (pushover) analysis based on **ASCE 41-17**.

- **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 ASCE 41-17.
  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:

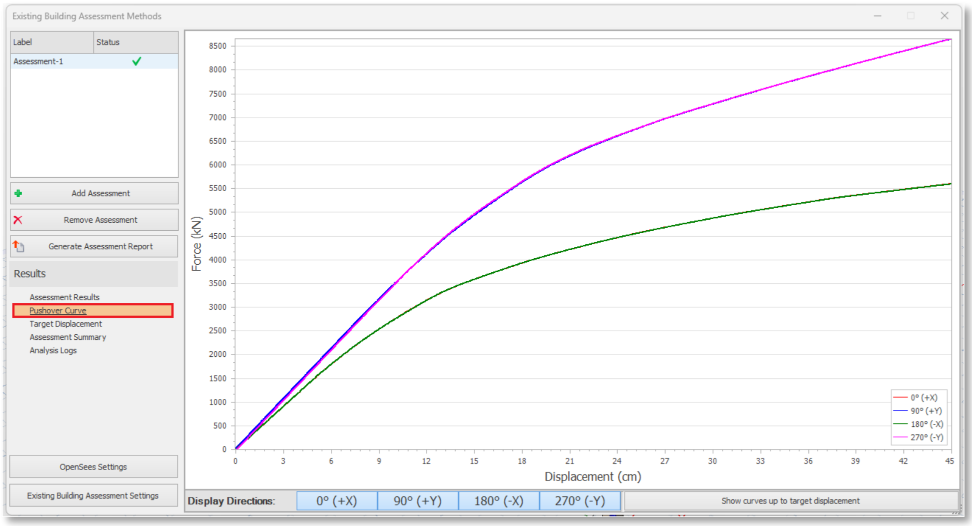
- Preparation of the structural analysis model
- Generation of section and force-deformation relationships
- Mesh creation
- OpenSees input file generation
- Launching pushover analysis for each selected direction.

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 Procedure – 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.

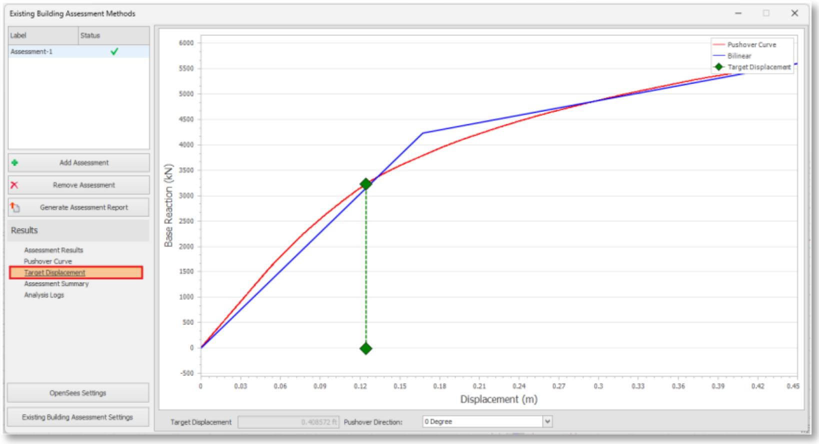
Tip:

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 Procedure – 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.

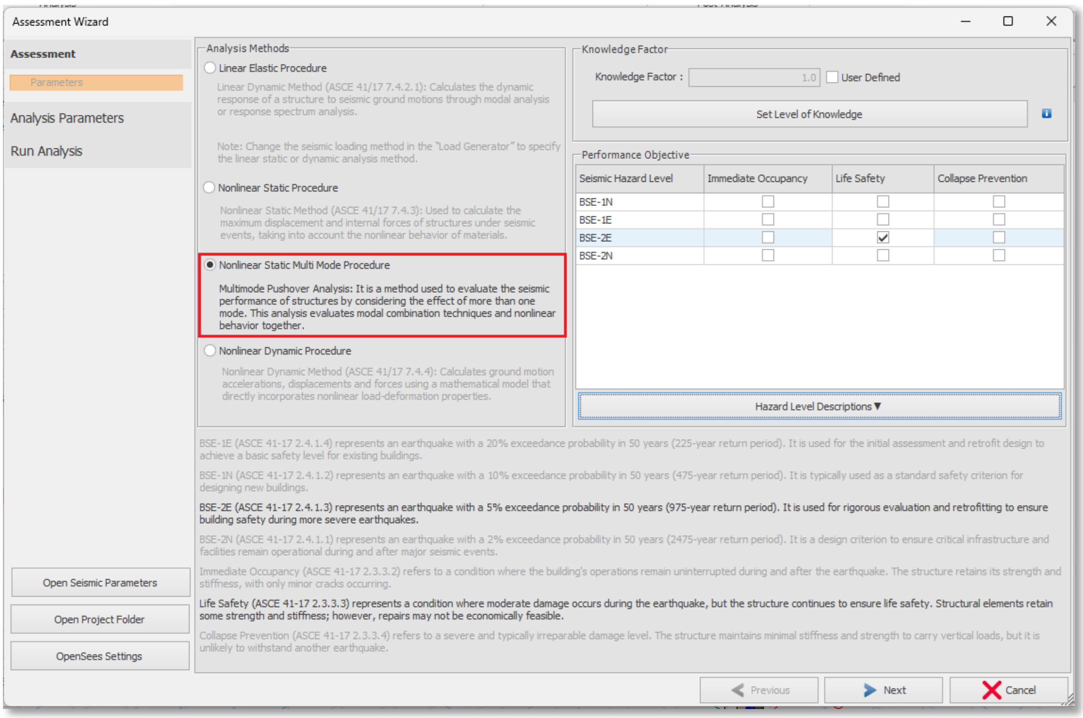
This graph visually represents the location of the **target displacement**, which is used as the reference point for performance evaluation according to **ASCE 41-17**.



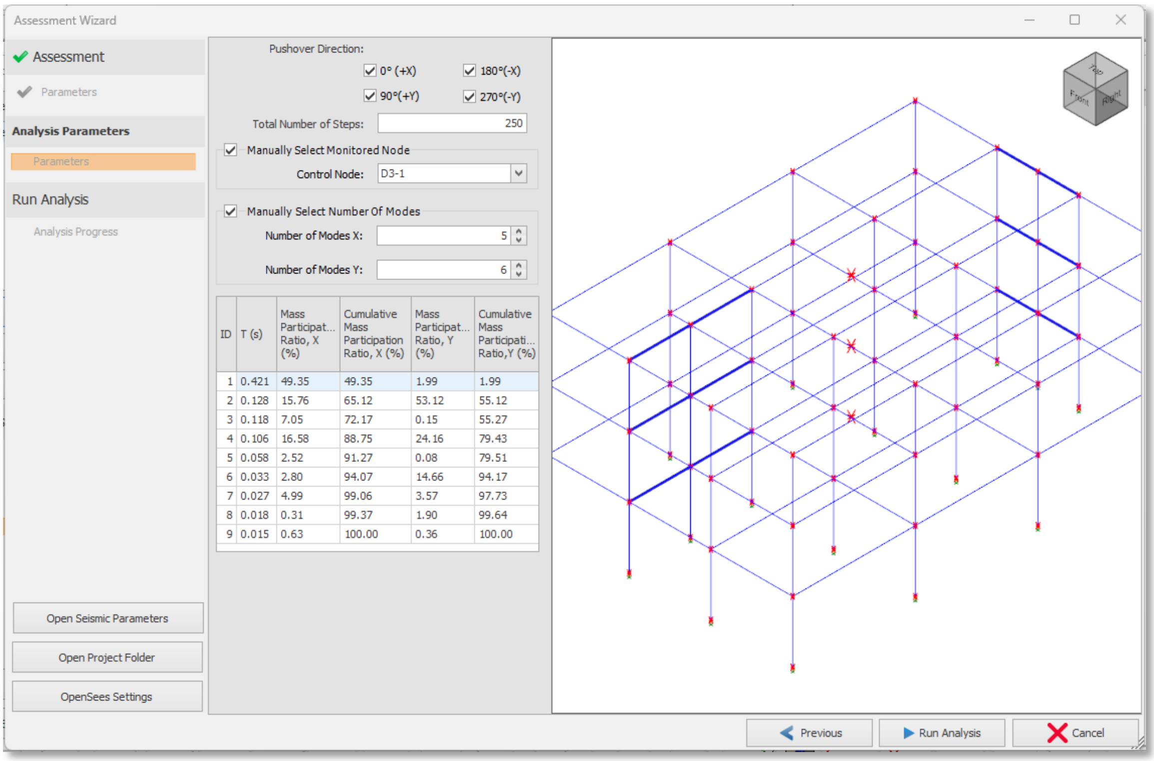
Nonlinear Static Multi Mode Procedure

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:

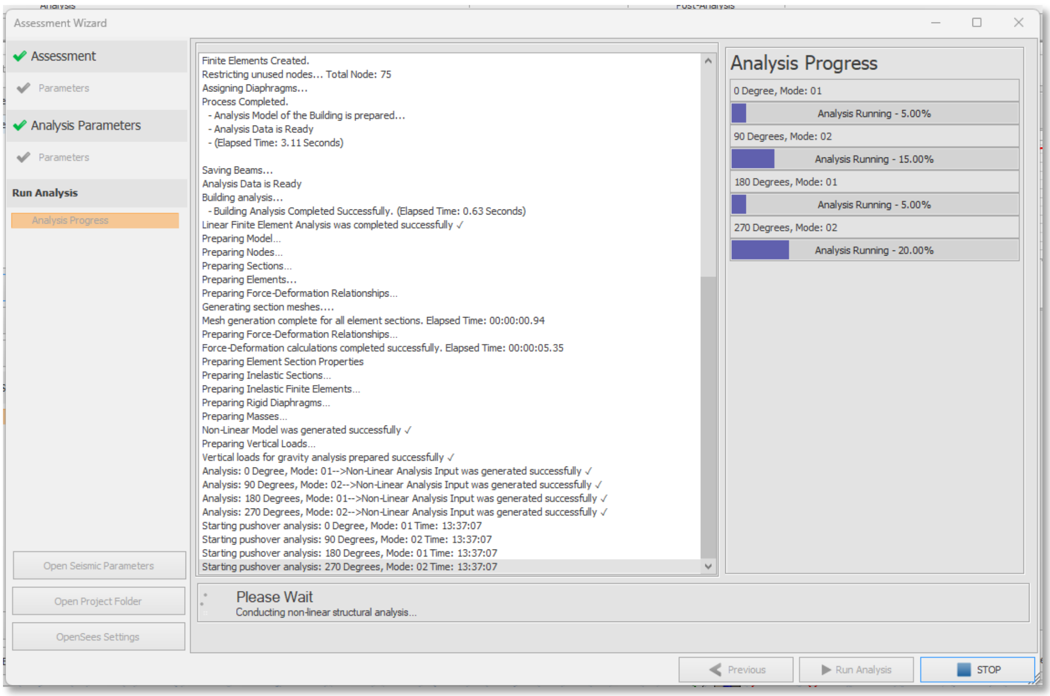
- **Number of Modes X**
- **Number of Modes Y**

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:

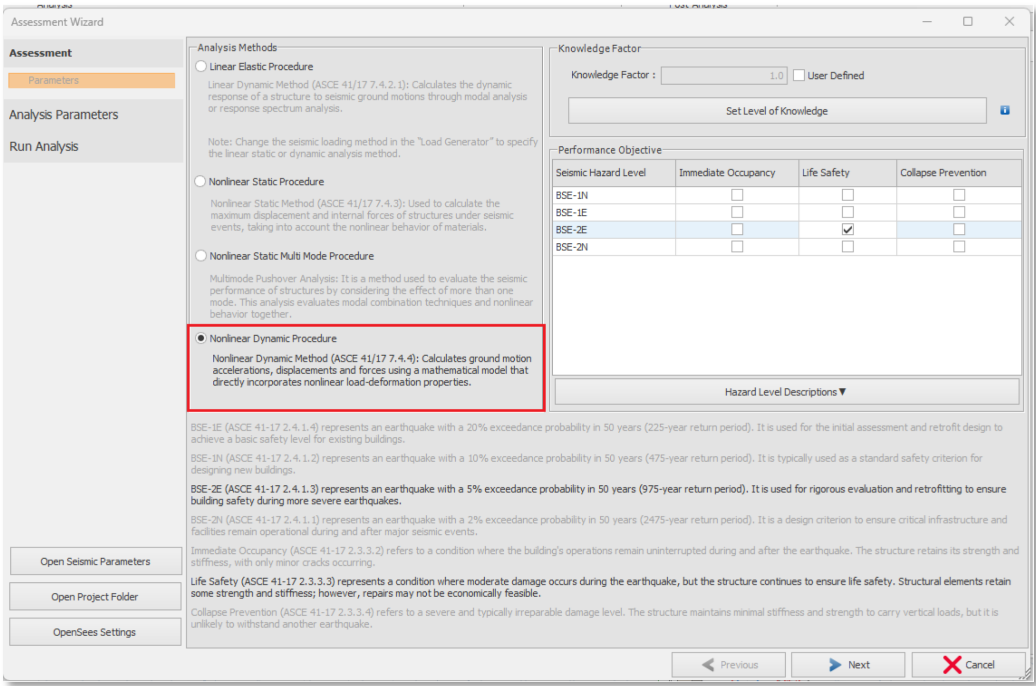
- Shows the real-time progress (%) for each selected direction (0°, 90°, 180°, 270°).
- Indicates which **mode** is currently being analyzed per direction (e.g., Mode 01, Mode 02).
- Helps quickly identify which direction or mode might be slowing down or failing.

Nonlinear Dynamic Procedure

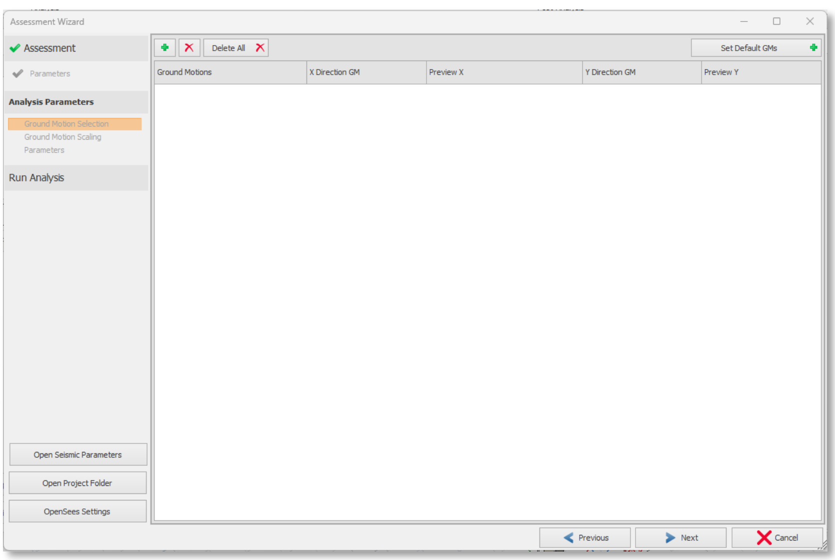
In the **Assessment Wizard**, the **Analysis Methods** section allows the user to select the approach for seismic performance evaluation.

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.

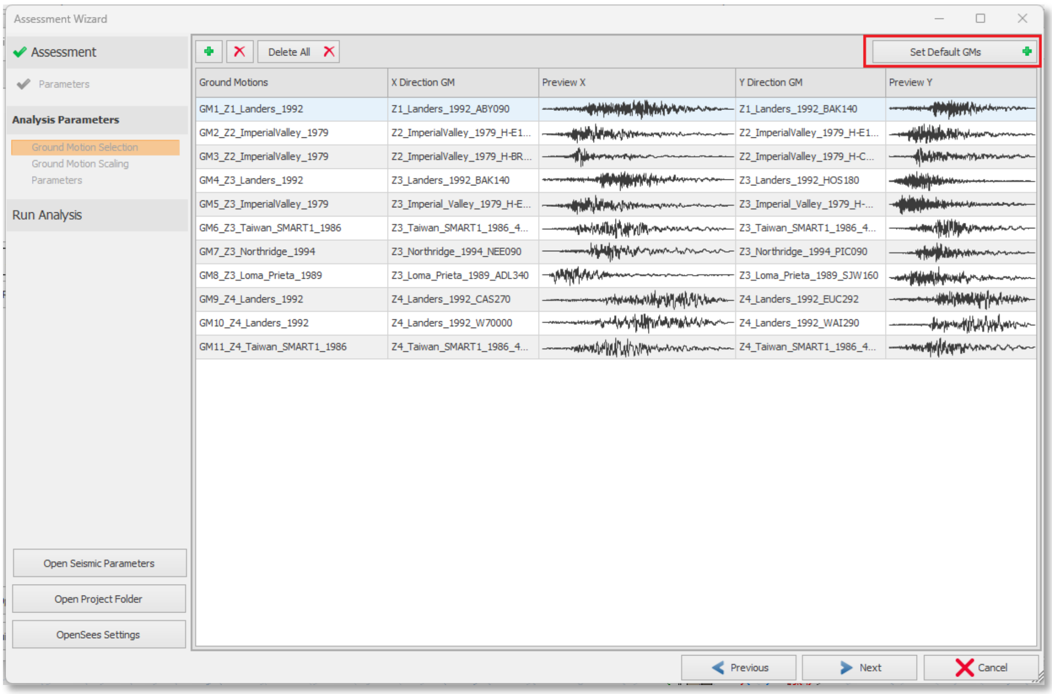
This method allows for highly detailed seismic response evaluation by calculating accelerations, displacements, and internal forces using a nonlinear time-domain simulation as described in **ASCE 41-17 Section 7.4.4**.



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.

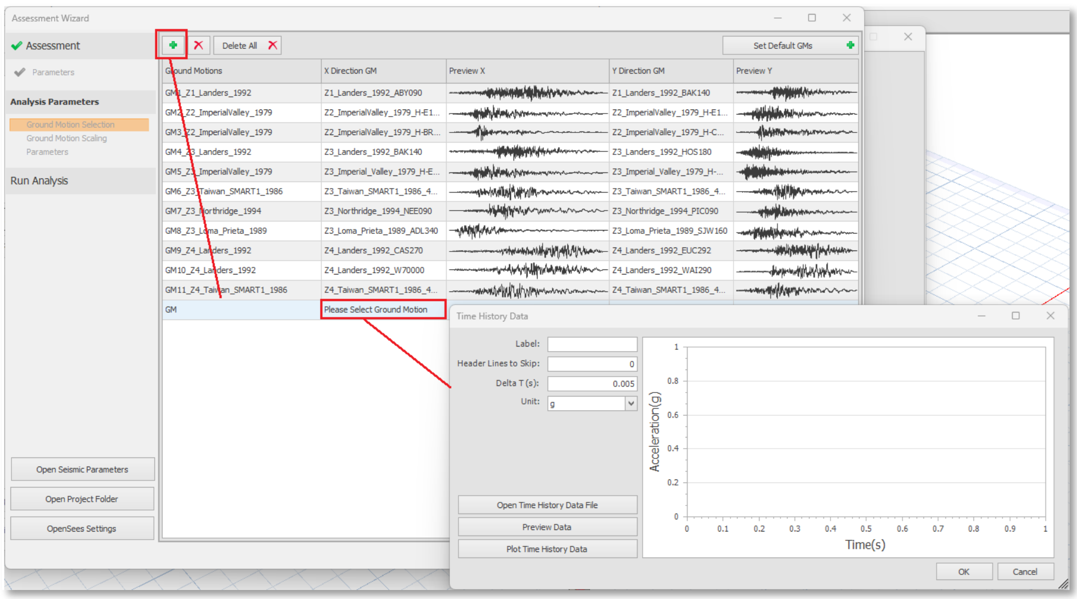


When the **Set Default GMs (Ground Motions)** button is clicked, the software automatically loads predefined earthquake records from the system library and assigns them to the X and Y directions for use in the dynamic analysis.



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.

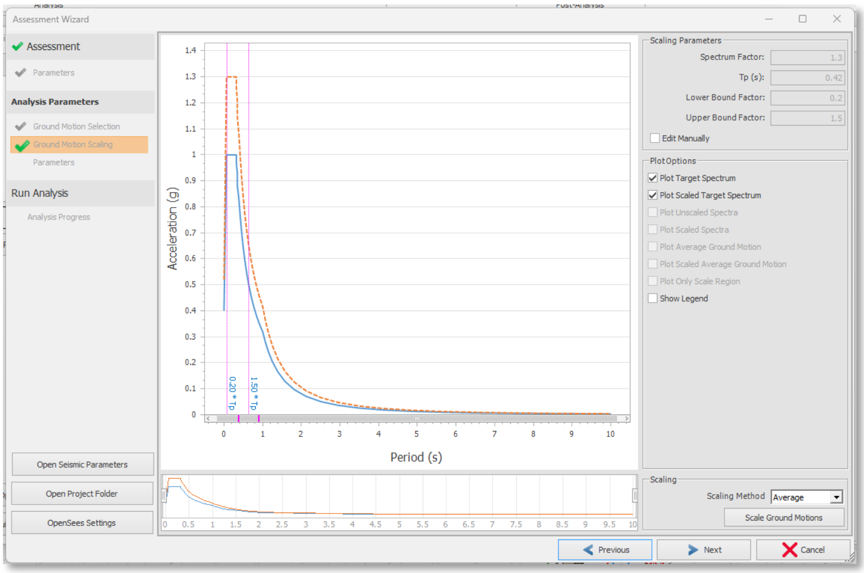


Click the ✚ **Add** button to insert a new row. Click on the **X Direction GM** or **Y Direction GM** cell of the new row. This will open the **Time History Data** input window.

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/s², etc.). **Preview**, a visual plot of the acceleration vs. time data will be shown on the right.

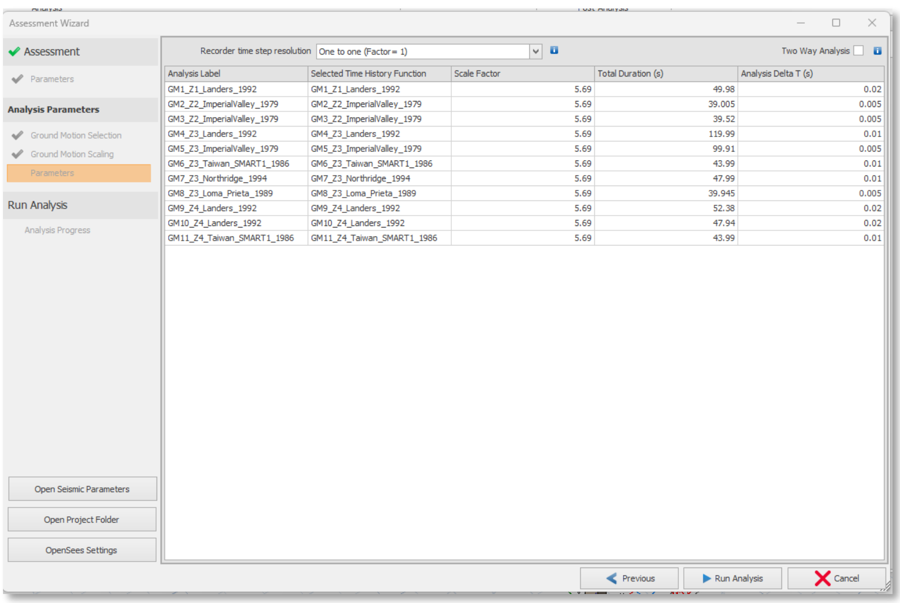
This feature provides full flexibility to use real recorded earthquakes or user-defined ground motions tailored to the specific needs of the analysis.

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.

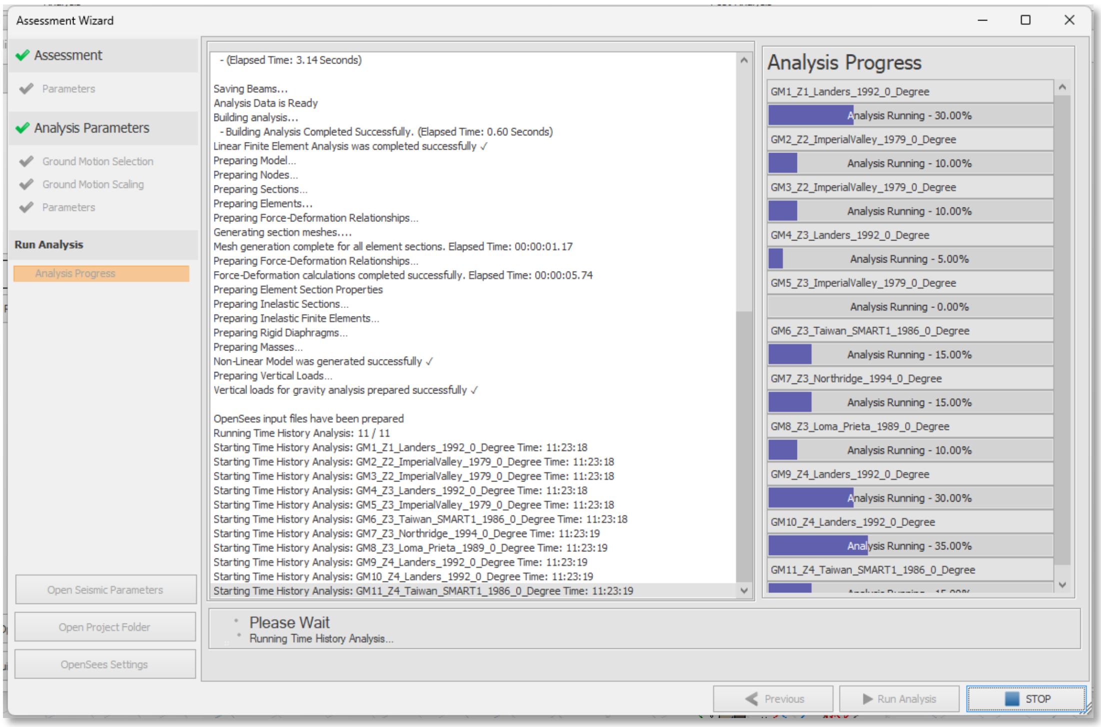
Two Way Analysis: When checked, each record is also analyzed in the orthogonal (90° rotated) direction by swapping X and Y components.

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.

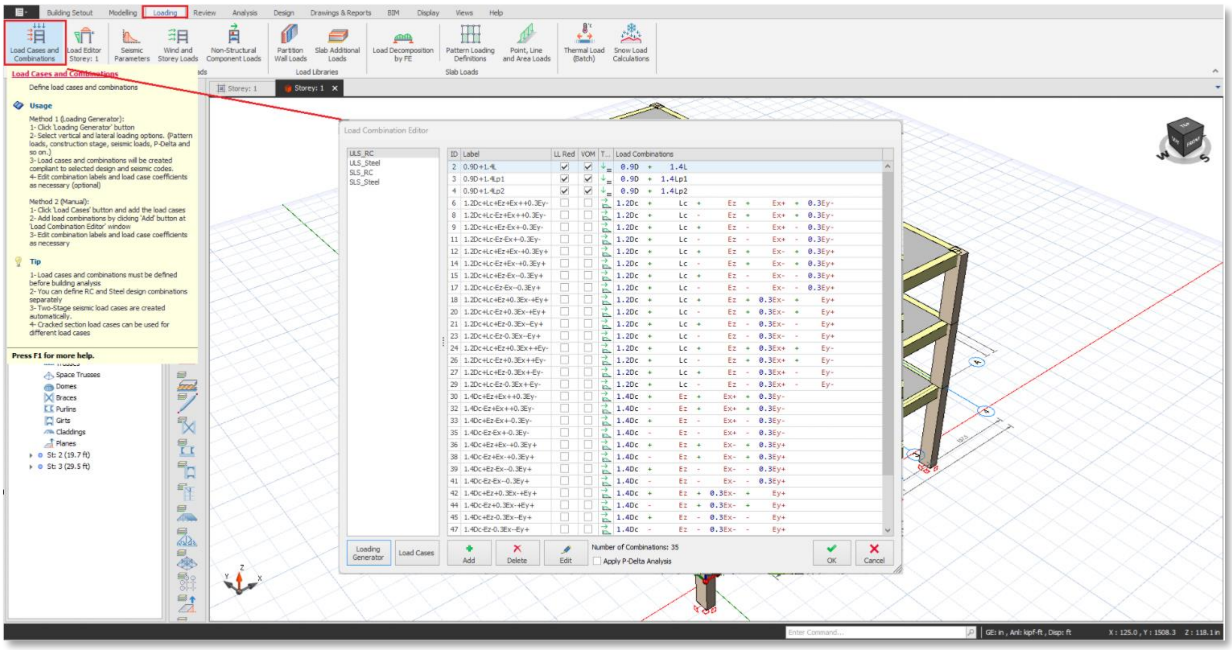
Linear Procedure Selection

To define whether to use **Linear Static Procedure (LSP)** or **Linear Dynamic Procedure (LDP)** for your analysis, follow the steps below:

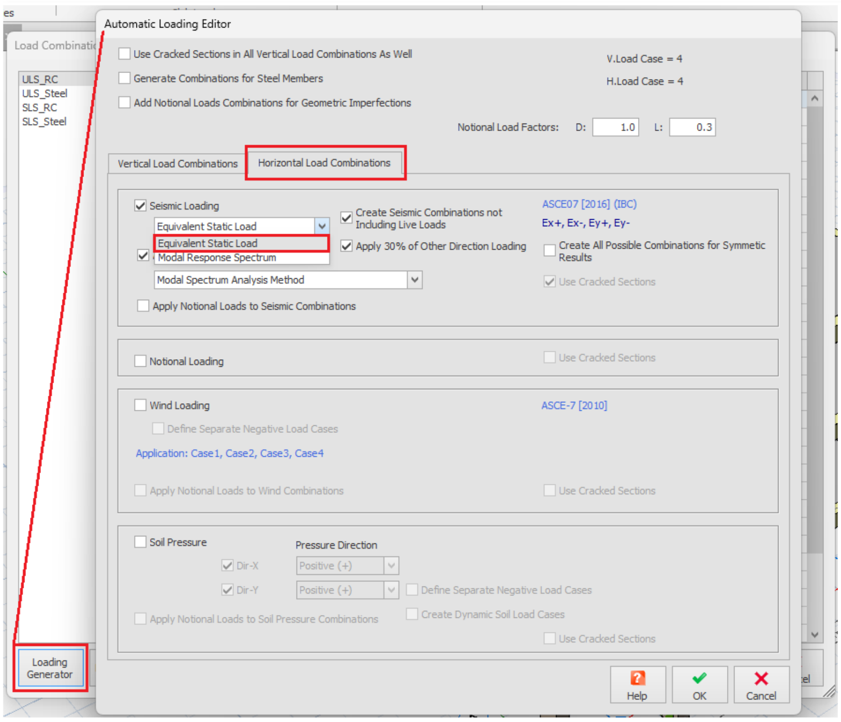
Steps:

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.



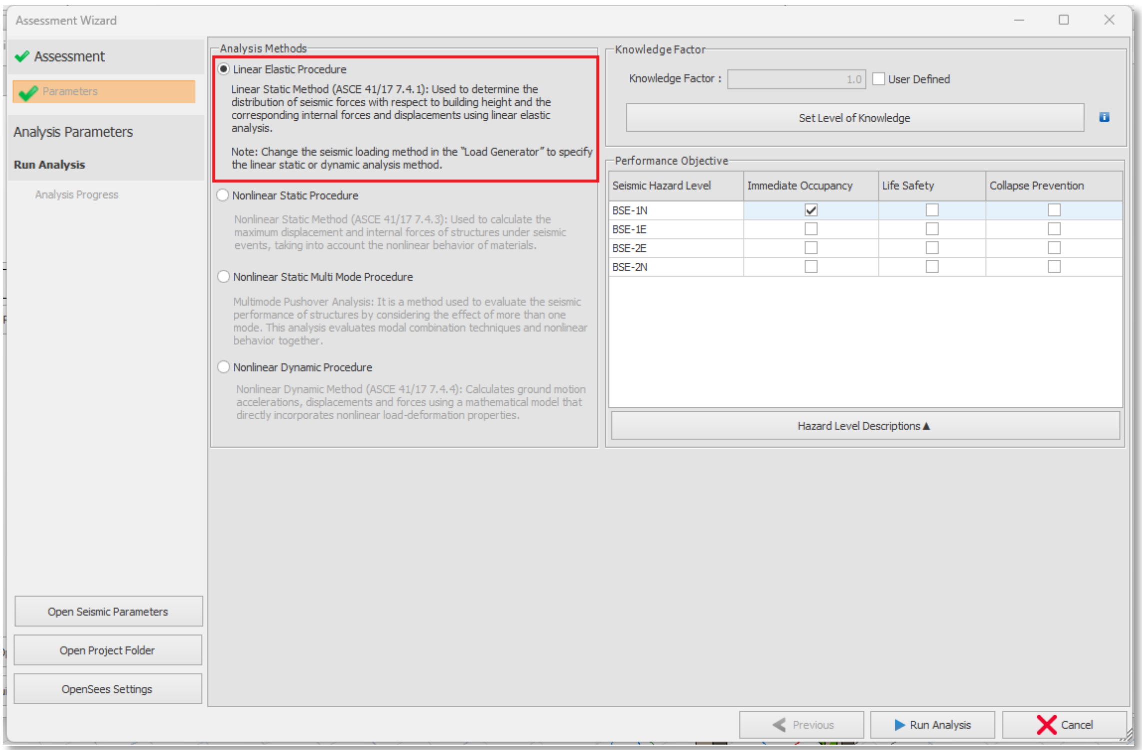
4. In the **Automatic Loading Editor** window:
    - Switch to the **Horizontal Load Combinations** tab.
    - Enable the **Seismic Loading** checkbox.
    - Choose the analysis method from the dropdown menu:
        - **Equivalent Static Load** → This activates **LSP**.
        - **Modal Response Spectrum**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

This configuration determines which linear analysis method (LSP or LDP) is used for building performance assessment under ASCE 41-17.

Linear Static Procedure

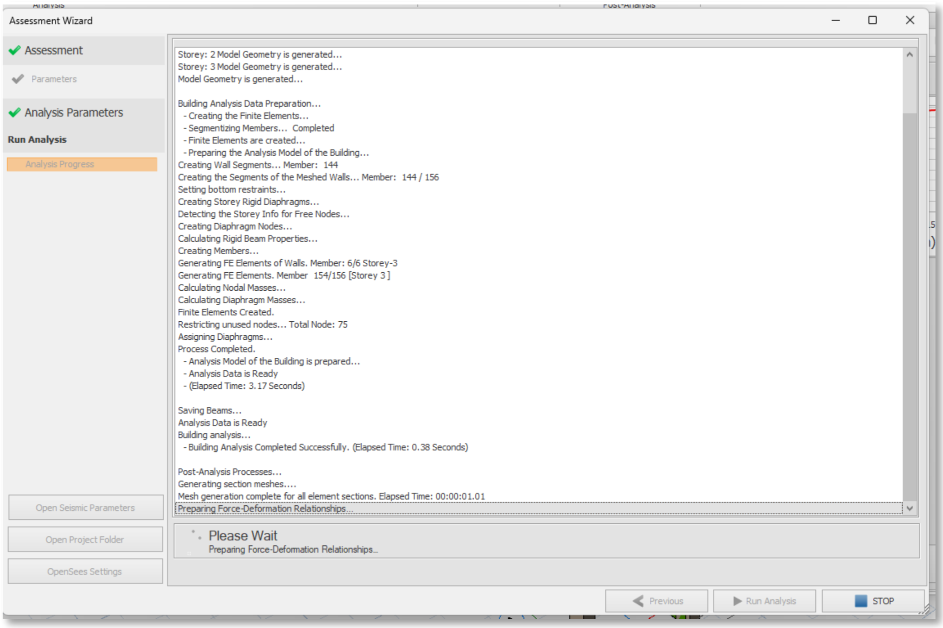
When the **Linear Elastic Procedure** option is selected in the **Assessment Wizard**, the structure is analyzed using linear elastic assumptions.

This corresponds to the **Linear Static Procedure** as described in **ASCE 41-17 Section 7.4.1**.



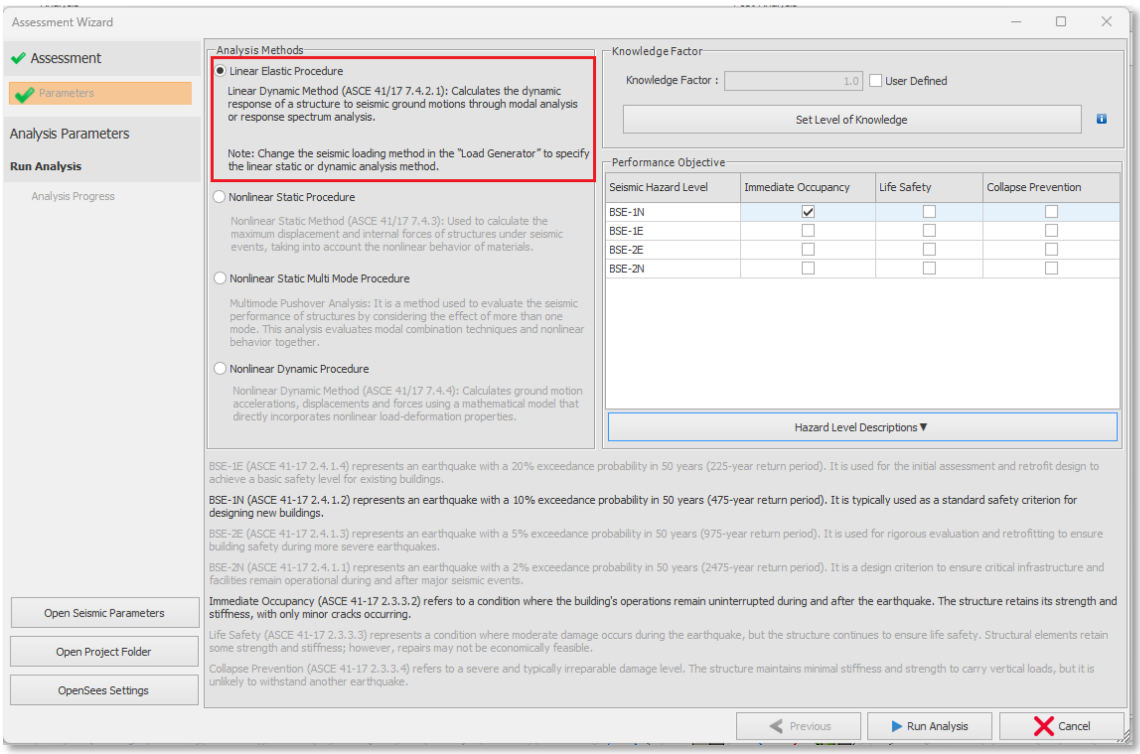
The **Linear Static Procedure** is used to determine the distribution of seismic forces based on building height. Seismic loads and the corresponding internal forces and displacements are calculated using linear elastic static analysis. All necessary load combinations and analysis parameters are automatically 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.



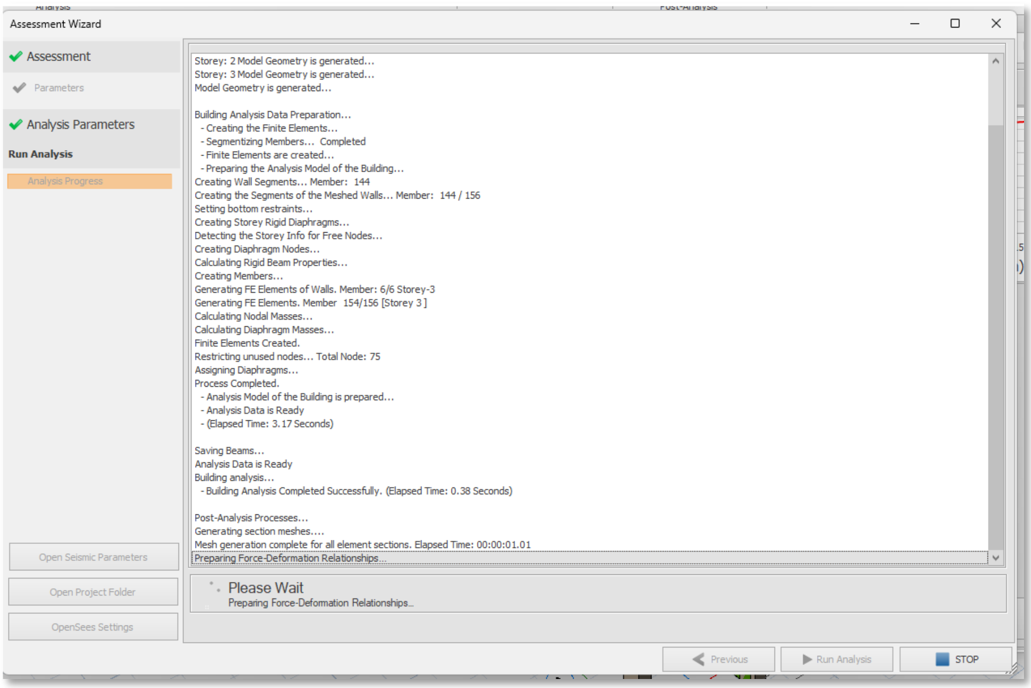
Linear Dynamic Procedure

In the Assessment Wizard, under the Analysis Methods section, select Linear Elastic Procedure, and ensure that the option titled Linear Dynamic Method (ASCE 41/17 7.4.2.1) 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.



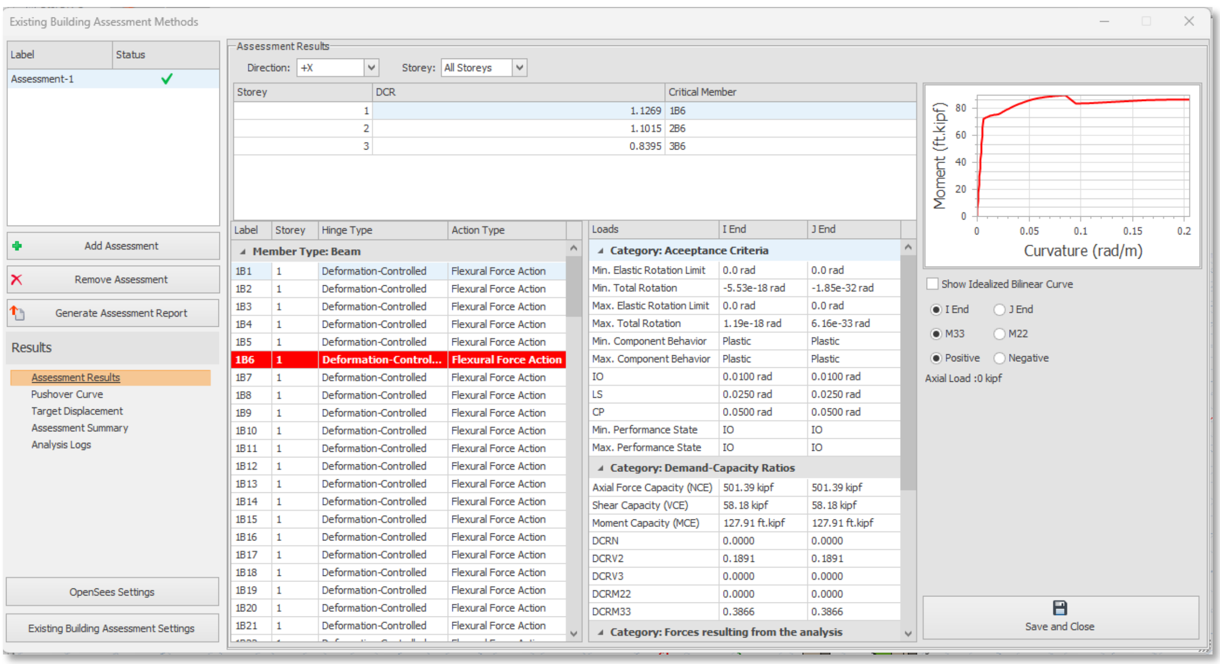
This method evaluates the seismic response of a building using modal or response spectrum analysis, offering a more realistic assessment of elastic behavior under seismic loads.

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.



Assessment Results

After the analysis is successfully completed, the user is directed to the **Assessment Results** screen, where the performance of individual structural members is displayed based on the selected evaluation direction and story level.



Assessment Definition panel shows the current assessment's **Label** and **Status**:

- **Label:** The name of the assessment (e.g., Assessment-1). Users can rename it as needed.
- **Status:** Indicates whether the assessment was completed successfully (✓ icon shown).

Additionally, this section includes shortcuts to:

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

Story-Level Critical Member Summary panel displays for each selected analysis direction:

- The **member with the highest DCR** in each story
- The corresponding **DCR value** and **critical member label**

This helps identify which story contains the governing element that may limit overall structural performance.

Member List and Categories table lists all members belonging to the selected story. Each row includes:

- **Label** (e.g., 1B6)
- **Story** level
- **Hinge Type** (Deformation-Controlled or Force-Controlled)
- **Action Type** (e.g., Flexural, Shear, Axial)
- **Member Type** (Beam, Column, Wall)

Note:

Members with DCR > 1.0 or those that fail to meet the target performance level are higlighted in red. These are considered inadequate and may require retrofitting.

Detailed Member Evaluation section provides complete assessment results for the selected member:

Category: Acceptance Criteria

- Elastic and total rotation limits (IO, LS, CP)
- Minimum and maximum performance states
- Component behavior (Elastic,Plastic)

Category: Demand–Capacity Ratios

- Capacities: Axial (NCE), Shear (VCE), Moment (MCE)
- Calculated demands and DCR values for each force component

Category: Forces from the Analysis

- Internal force results at I-End and J-End

Moment–Curvature Plot shows the **Moment–Curvature (M–ϕ)** relationship of the selected element.

The user can:

- Enable the **idealized bilinear curve**
- Switch between **I-End** and **J-End**
- Toggle between **M22** and **M33** moment components
- View either **positive** or **negative** direction behavior

Assessment Summary

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, and statistical DCR results per direction.

Category: Assessment Parameters

Lists the general configuration used in the analysis:

- **Seismic Hazard Level** (e.g., BSE-1N)
- **Performance Objective** (e.g., Immediate Occupancy)
- **Analysis Type** (e.g., Nonlinear Static 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: Directional Results

For each pushover direction (+X, +Y, –X, –Y), the following data is reported:

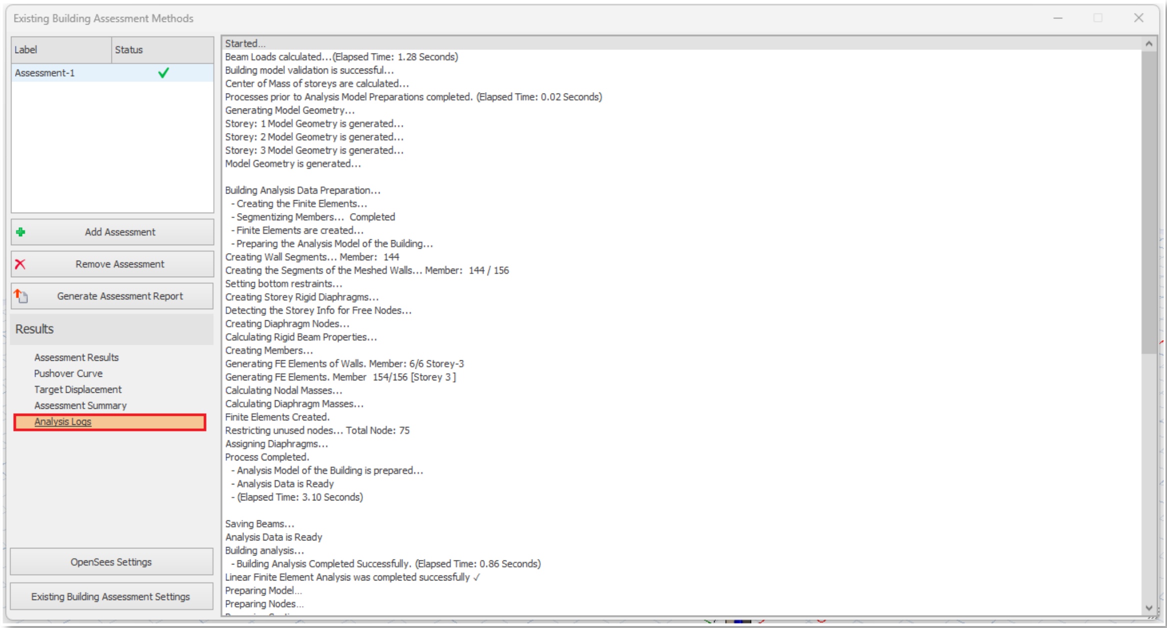
- **Minimum DCR**
- **Maximum DCR**
- **Mean DCR**
- **Standard Deviation of DCR**

Performance classification for each direction is also provided:

- Number of members satisfying **Elastic**, **IO**, **LS**, or **CP** states
- Count of members classified as **Acceptable** or **Not Acceptable**

Analysis Logs

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

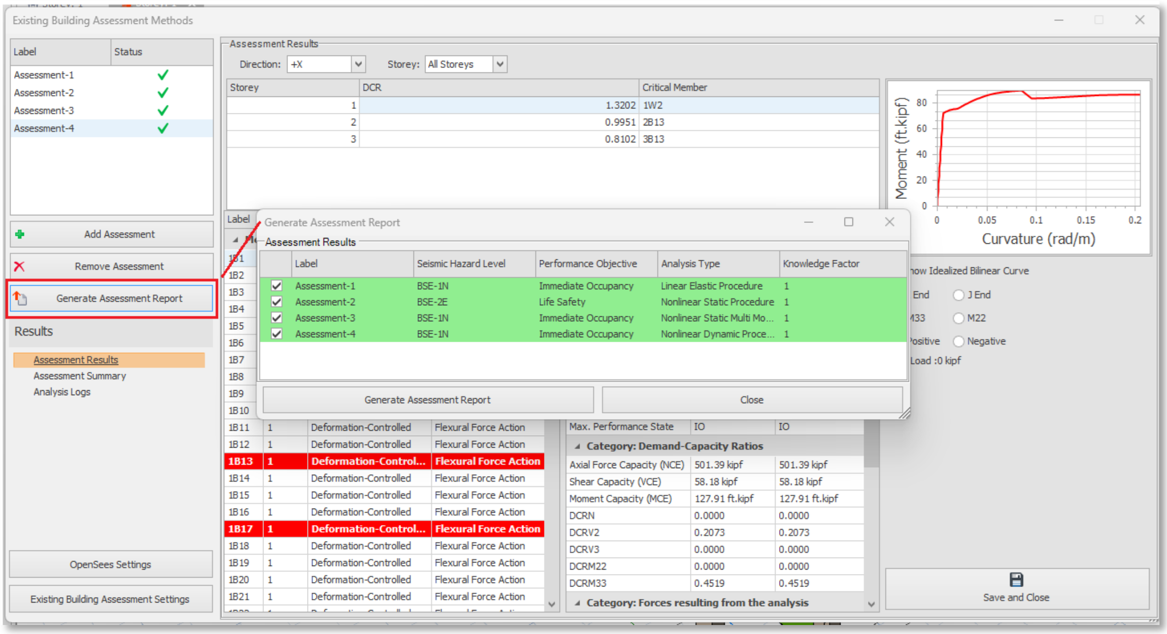


Assessment Report

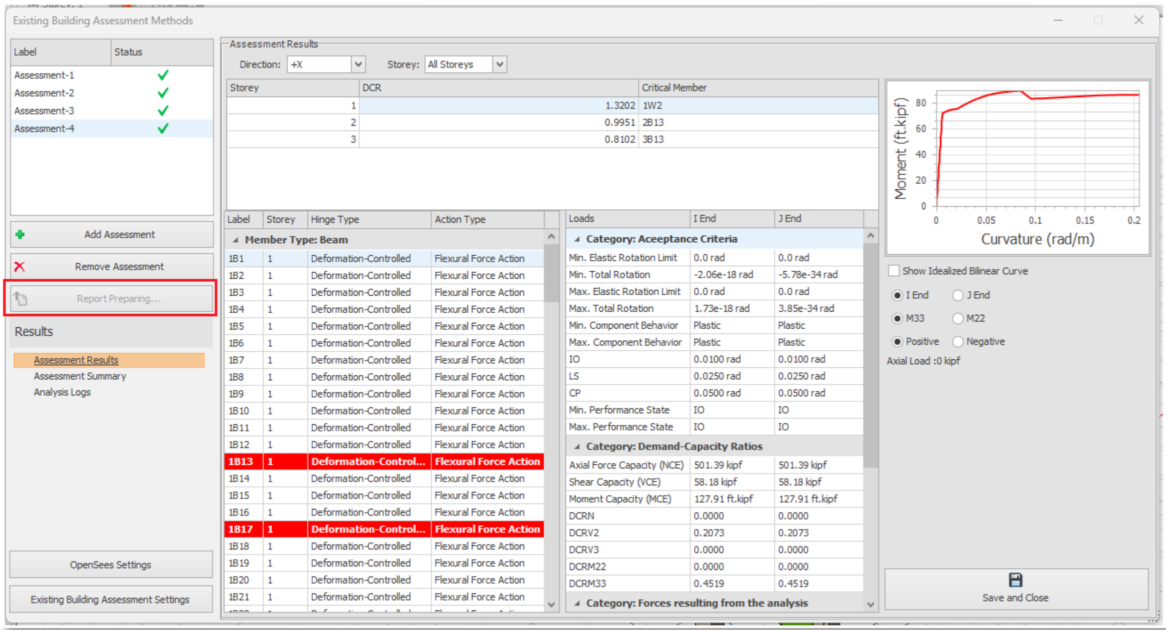
Within the **Existing Building Assessment Methods** interface, all completed evaluation scenarios are listed under the **Label** section. The user can click on the **"Generate Assessment Report"** button located in the left panel to open the reporting screen.

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

Label: Assessment name,**Seismic Hazard Level**: Selected seismic hazard level (e.g., BSE-1N, BSE-2E), **Performance Objective**: Target performance level (IO, LS, CP),**Analysis Type**: Applied analysis method (LSP, LDP, NSP, etc.),**Knowledge Factor**: Applied knowledge factor,The user can select multiple scenarios and generate a combined report by clicking the **Generate Assessment Report** button.



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.

Assessment-1 Analysis Method: Linear Elastic Procedure - Performance Objective: Immediate Occupancy - Seismic Hazard Level: BSE-1N Assessment Report

Assessment Parameters	Value
Seismic Hazard Level :	BSE-1N
Performance Objective :	Immediate Occupancy
AnalysisType :	Linear Elastic Procedure
Knowledge Factor :	1.0
Level of Seismicity :	Very Low

Assessment-1 - Direction +X

Assessment-1 Summary Table

Label	Value
Minimum - DCR:	0.017
Maximum - DCR:	0.603
Mean - DCR:	0.142
Standard Deviation - DCR:	0.094

Assessment-1 Deformation Controlled Members Summary Table

Story	Members	Performance State: Acceptable	Performance State: Not Acceptable
1	Columns : 16 Beams : 25 Walls : 2	43	0
2	Columns : 16 Beams : 25 Walls : 2	43	0
3	Columns : 16 Beams : 25 Walls : 2	43	0

Assessment-1 Force Controlled Members Summary Table

Story	Members	Performance State: Acceptable	Performance State: Not Acceptable
1	Columns : 0 Beams : 0 Walls : 0	0	0
2	Columns : 0 Beams : 0 Walls : 0	0	0
3	Columns : 0 Beams : 0 Walls : 0	0	0

Assessment-1 Storey Assessment

Story	Demand-Capacity Ratios	Critical Member
1	0.603	1W1

Report Content:

- Assessment Parameters
  - Selected analysis method (e.g., *Linear Elastic Procedure*)
  - Performance objective (e.g., *Immediate Occupancy*, *Life Safety*, etc.)
  - Seismic hazard level (e.g., *BSE-1N*, *BSE-2E*)
  - Knowledge factor level
- Summary Table
  - Provides statistical values of **DCR (Demand-Capacity Ratio)** such as minimum, maximum, mean, and standard deviation
  - Helps identify the overall strength performance of the structure
- Deformation-Controlled Members Summary
  - Displays deformation-controlled elements (e.g., beams and some walls) per story
  - Number of members **meeting or not meeting** the performance criteria is clearly listed
- Force-Controlled Members Summary
  - Shows the performance of force-controlled elements (typically columns)
  - Acceptable vs. unacceptable status per story is displayed
- Storey Assessment

- Presents **maximum DCR values and critical members** for each story
- Helps easily detect the weakest story in terms of performance

- DCR Details (Demand-Capacity Ratios)
  - **Individual DCR values for each element** are provided
  - Elements with **DCR > 1.0** are highlighted and considered to exceed performance limits
  - DCR components may also be shown to indicate which demand type is governing
- Structural Members & Element Behavior
  - Classification of members (columns, beams, walls) based on performance levels such as **IO (Immediate Occupancy), LS (Life Safety), CP (Collapse Prevention)**
  - Provides insights into whether members meet their expected behavior

Hinge Properties and Condition Selection

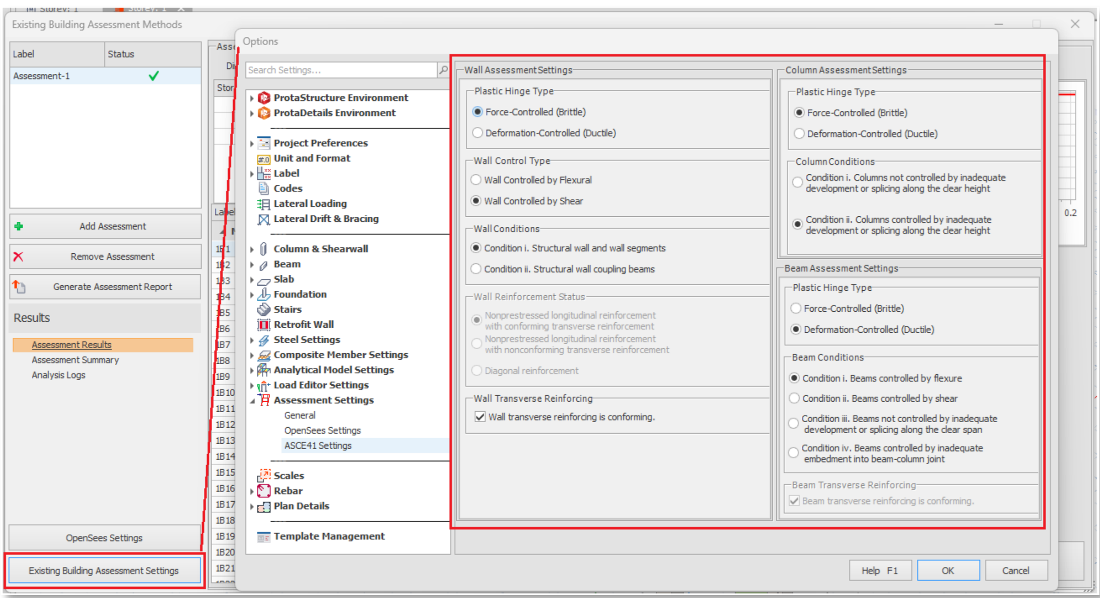
In the **ProtaStructure Existing Building Assessment Methods** module, users can configure assessment criteria both globally and on a per-member basis. These settings define how columns, beams, and shear walls are evaluated according to ASCE 41-17 guidelines.

Global Settings

Accessible via Existing Building Assessment Settings > ASCE41 Settings. These settings include:

- **Plastic Hinge Type** (Force-Controlled or Deformation-Controlled),
- **Column / Beam / Wall Conditions** (e.g., controlled by flexure, shear, or inadequate development),
- **Reinforcement Conditions** (e.g., compliance of transverse reinforcement).

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

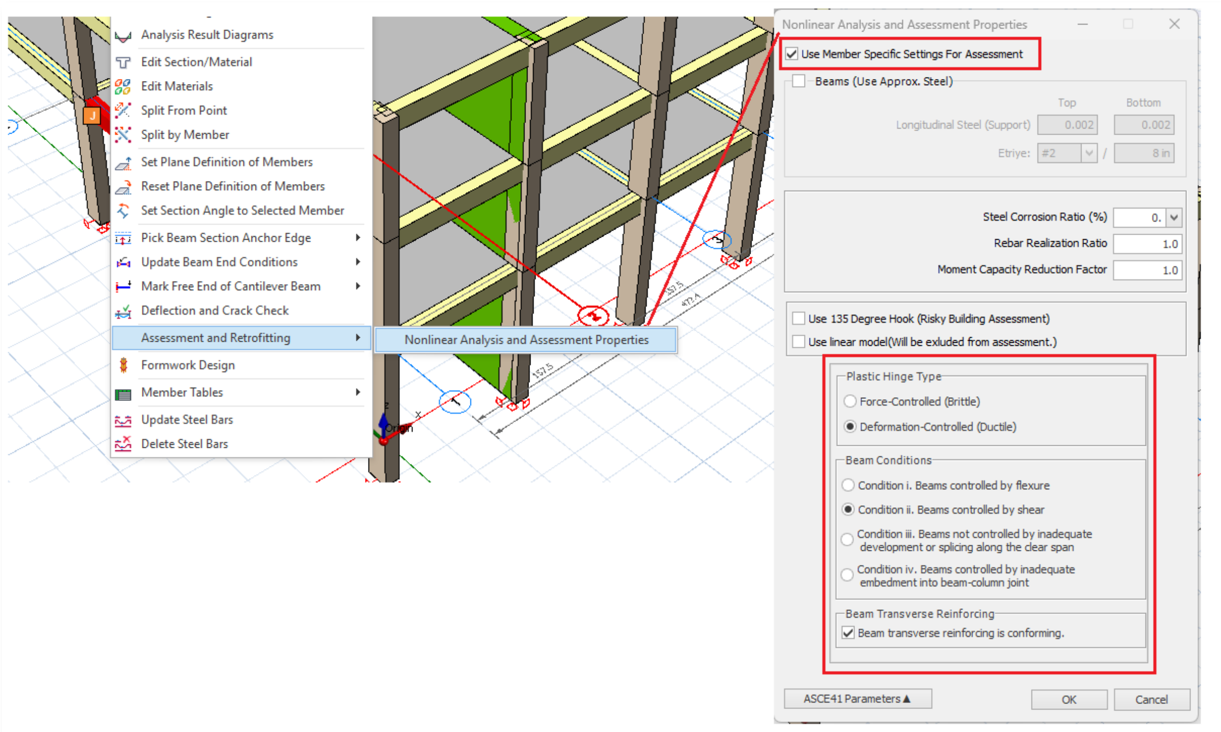


Member-Specific Settings

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 define hinge behavior and reinforcement compliance independently for that specific element.

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®