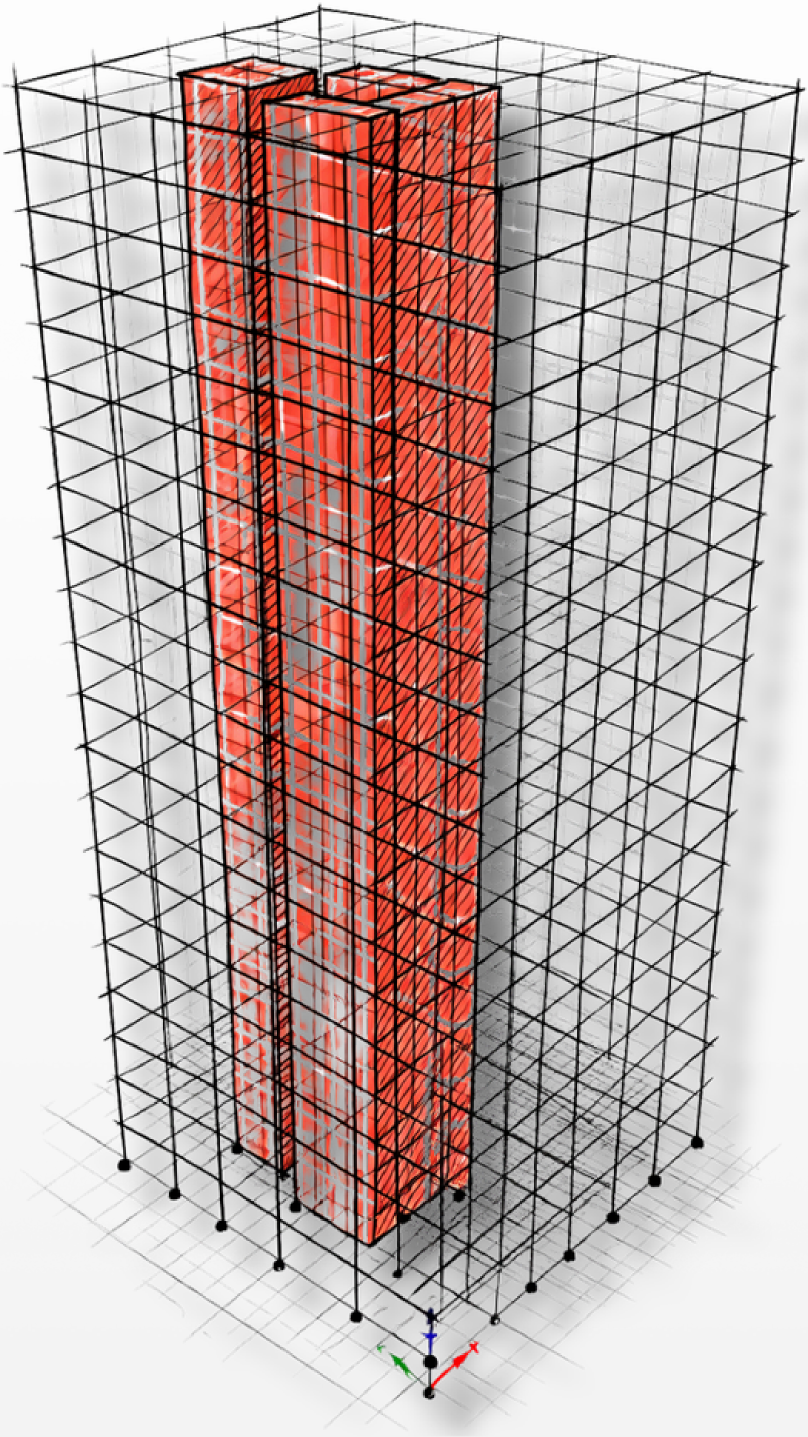
SQVe
Engineering



Your Assistant for Structural Model Verification

SQVE-MV-ET-01 | VER 2.0

SQVE MODEL VERIFICATION TOOL

1. Why This Tool?

Structural model verification is often time-consuming and prone to oversight when performed manually. This tool is designed to **save time on every model iteration** by eliminating repetitive manual checks and delivering verification results within a few seconds to a few minutes, depending on model complexity**.** It helps engineers **avoid missing critical errors** by automatically identifying key issues that are often overlooked during manual review. By providing **instant structural insights**, the tool enables quick understanding of model behavior without the need to navigate multiple ETABS tables and outputs. It also supports a **standardized QA/QC process**, ensuring consistent and reliable verification across projects, teams, and iterations. Overall, the tool enhances efficiency while supporting a structured and reliable QA/QC process, allowing engineers to **focus more on critical engineering decisions rather than routine checking tasks**.

2. Current Scenario in Structural Engineering

Structural engineering practice is undergoing a significant transformation. With the continuous revision of design standards and increasing project complexities, engineers are required to handle more advanced analysis and design requirements. At the same time, project delivery timelines are becoming tighter, creating additional pressure on design teams. Compounding this challenge is the evolving workforce landscape. Structural consultancies are often in a continuous hiring mode, and new engineers—whether experienced or fresh graduates—require time to align with company-specific processes and expectations. Additionally, a noticeable gap exists between academic learning and practical industry requirements, making it essential for organizations to establish robust systems that ensure consistency and reliability in design outputs.

3. Importance of Model Verification in Quality Assurance

Quality assurance is a critical component of any structural engineering project, directly influencing both the safety and cost-effectiveness of the structure. Within this framework, model verification plays a vital role. A structural analysis model must be thoroughly checked through multiple parameters, which involves reviewing numerous tables, graphical outputs, and analytical results. This process is often time-consuming and prone to oversight if not handled systematically. Inadequate or delayed verification can lead to significant rework during later stages of the project. In more critical cases, it may impact the structural safety or result in inefficient designs. Therefore, a structured and reliable approach to model verification is essential to ensure accuracy, reduce rework, and maintain high standards of engineering quality.

4. Introduction to SQVE-MV-ET-01

To address the challenges highlighted in modern structural engineering practice, particularly increasing complexity, reduced timelines, and the critical need for robust quality assurance, SQVE has initiated the development of specialized engineering tools.

As a first step in this direction, we present: SQVE-MV-ET-01 – Model Verification Tool for ETABS

This tool is specifically designed to assist structural engineers in performing systematic, fast, and reliable model verification as a key component of Quality Assurance (QA/QC).

5. Phased Development Strategy

Recognizing the wide scope of structural model verification, the tool is being developed through a **phased development strategy**, with each phase progressively expanding the range and depth of automated verification checks.

Phase 1 – Version 1.0

The first release focused on **critical seismic and dynamic model verification checks**, particularly those having a significant influence on structural analysis and seismic design. The checks primarily covered mass definition, seismic loading, response spectrum analysis and dynamic behaviour.

Phase 2 – Current Release – Version 2.0

The current release significantly expands the verification capability beyond seismic and dynamic checks. Phase 2 introduces an expanded set of automated checks covering **model geometry, connectivity, supports, material and section properties, stiffness modifiers, diaphragms, member assignments, meshing, load assignments, seismic input parameters, storey shear, eccentricity ratio and drift summary.**

6. Scope of Checks – Phase 1 & Phase 2

The current release of SQVE-MV-ET-01 incorporates automated verification checks covering the following aspects of an ETABS structural model:

A. Model Definition & Geometry

- Model Units
- Story Geometry Database
- Overall Building Dimensions
- Mass Source Verification
- Mass Source Direction Check
- Story Mass Validation
- Adjacent Story Mass Difference Check

B. Model Connectivity & Structural Elements

- Duplicate Point Object Verification
- Duplicate Beam Verification
- Collinear Overlapping Beam Verification
- Duplicate Column Verification
- Overlapping Column Verification

- Floating Column Verification

C. Supports & Boundary Conditions

- Support Restraint Verification
- Support Restraint Classification and Consistency Verification
- Unexpected Joint Restraint Assignments

D. Material & Section Properties

- Material Properties
- Beam Material Summary
- Column Material Summary
- Slab Material Property Summary
- Wall Material Property Summary
- Beam and Column Property Assignment Verification
- Slab Modeling Type Summary
- Wall Modeling Type Summary

E. Stiffness Modifiers

- Beam Stiffness Modifier Summary
- Column Stiffness Modifier Summary
- Slab Stiffness Modifier Summary
- Wall Stiffness Modifier Summary

F. Diaphragm & Member Assignment

- Diaphragm Assignment Verification
- Frame Member End Length Offset Summary
- Insertion Point / Cardinal Point Summary
- Member Release Summary
- Slab and Wall Property Assignment Verification

G. Floor & Wall Meshing

- Floor Meshing Strategy Summary
- Floor Meshing Auto Cookie Cut Configuration
- Floor Meshing Add Restraints Review

- Wall Meshing Strategy Summary
- Wall Meshing Add Restraints Review
- Auto Edge Constraint Summary
- Wall Pier Label Assignment Verification
- Spandrel Label Summary

H. Gravity Load & Load Assignment

- Gravity UDL Sign Convention Verification
- Gravity Point Load Sign Convention Verification
- Shell Uniform Load Set Sign Convention Verification
- Area Uniform Load Sign Convention Verification
- Gravity Load Joint Displacement Summary

I. Seismic Load & Static Earthquake Parameters

- Seismic Load Pattern Verification
- Maximum Static Earthquake Base Shear Check
- Base Shear Comparison with Gravity Loads
- Vertical Seismic Force Check
- Static Earthquake Load Direction Verification
- Static Earthquake Load Story Range Verification
- Static Earthquake Force Time Period Definition Verification
- Static Earthquake Force Input Parameter Verification

J. Response Spectrum & Dynamic Behaviour

- Presence of Response Spectrum Load Cases
- Response Spectrum Base Shear – X Direction
- Response Spectrum Base Shear – Y Direction
- Response Spectrum Base Shear – Vertical Direction
- Response Spectrum Scaling Verification
- 65% Mass Participation Check
- 90% Mass Participation Check
- First Natural Time Period

- Torsional Mode Identification
- Closely Spaced Modes Check
- Torsional Eccentricity Override Check
- Eccentricity Ratio Summary
- Response Spectrum Storey Shear Summary
- Seismic Drift Summary

7. Key Features & Benefits

Once the structural model analysis is completed, the tool performs automated verification of all identified checks and generates **reports** (detailed & summary) **within seconds to a few minutes**, depending on the size and complexity of the model. The report provides **detailed explanations for each verification check**, highlights **warnings and critical observations**, and offers **clear recommendations for further engineering action**, enabling engineers to quickly understand and act on key aspects of the model.

The tool is designed to support engineers across **multiple stages of the project lifecycle**, including model development, iterative analysis, self-checking, senior-level review, and proof-checking by third-party consultants. It delivers **instant insights into structural behavior** without requiring **navigation through multiple ETABS tables**, **manual extraction of results**, or **time-consuming graphical checks**, thereby simplifying the overall verification process.

Built as a **lightweight and high-performance solution**, the tool ensures **fast execution**. It operates in a **read-only mode** and does **not modify the ETABS model**, eliminating any risk of unintended changes. At the same time, it enforces a **consistent QA/QC methodology** and ensures that **critical checks are not missed** during the verification process.

By automating repetitive verification tasks, the tool **substantially reduces manual effort** and **accelerates the model verification workflow**, enabling engineers to focus more on engineering judgment and decision-making rather than routine checking activities, thereby enhancing **efficiency**, **consistency** and **productivity**.

The tool is particularly useful for **quick validation during model development**, **iterative checking during trial-and-error modeling**, **pre-review before submission**, and **rapid screening during design reviews**. It also supports **internal QA/QC workflows** and **independent model reviews**, making it a valuable aid across different engineering roles.

Overall, the tool helps **identify critical issues at early stages**, **improves model quality and reliability**, and enables **faster, more confident engineering decisions**. It transforms model verification into a **structured, efficient, and dependable process** aligned with modern engineering practices.

"SQVE-MV-ET-01 transforms model verification into a fast, structured, and reliable process, enabling engineers to focus on critical design decisions."

*The summary report and detailed output generated by the software are provided in **Annexure A & Annexure B** respectively**.***

8. Correlation and Use of the Detailed and Summary Reports

The **Summary Report** provides a **concis**e, **high-level overview** of the model verification results and is intended primarily for quick review and identification of areas requiring attention. It consolidates the outcome of the individual verification sections by presenting the section title, verification status such as PASS, WARNING, SUMMARY/INFORMATION, and the corresponding key observations, recommendations or verification messages. The Summary Report enables the user to **rapidly** understand the overall condition of the analytical model, identify warnings or exceptions, and recognize items that require further engineering review without initially going through the complete underlying data.

The **Detailed Report** provides the **supporting information** and model data associated with the individual verification checks. Depending on the nature of the check, it may contain the purpose and importance of the verification, methodology or basis of assessment, extracted model information, detailed tables, exception tables, observations, verification results, recommendations and other relevant supporting information. It is intended for **in-depth review**, **investigation** and **engineering verification**, particularly where the Summary Report identifies a warning, exception, unusual condition or an item requiring user verification. The Detailed Report therefore provides the traceability and supporting evidence necessary for the engineer to understand how the reported result relates to the underlying analytical model.

The Summary Report and Detailed Report are intended to be used together as complementary documents. The **Summary Report** may be used as the **first level of review** to quickly navigate through the overall verification results and identify the sections requiring attention. The corresponding section in the **Detailed Report** can then be referred to for the **underlying model data**, **detailed observations**, **exception information** and **recommendations**. Conversely, while reviewing a particular item in the Detailed Report, the corresponding Summary Report entry provides a convenient consolidated indication of its overall verification status. Together, the two reports provide a structured overview-to-detail review workflow, enabling efficient screening of the model while retaining access to the detailed information required for engineering assessment and decision-making.

9. Important Notes & Limitations

This tool currently performs a defined set of automated verification checks within the Phase 2 scope and is not intended to constitute a complete model verification system. The results generated are indicative in nature and intended for preliminary assessment only.

- Accuracy of results depends on Model inputs, ETABS settings & Analysis conditions
- The verification logic is based on generalized modeling practices
- Certain project-specific conditions and complexities may not be fully captured
- This tool is designed to assist engineers, but does NOT replace engineering judgment
- All outputs must be reviewed and validated by a qualified structural engineer

The objective of the tool is to:

- Highlight critical modeling aspects
- Identify potential areas requiring attention
- Enable focused and efficient model review

The tool will be continuously refined and enhanced based on feedback from practicing engineers.

10. Software Dependency

- Developed using ETABS API integration
- Built on .NET Standard 2.0 framework (DLL-based architecture)
- Tested and validated with:
    - ETABS Version 22
    - ETABS Version 23

⚠️ The tool may not function correctly for ETABS versions below v22

11. Free Trial & Flexible Upgrade Options

🎯 **30-Day Free Trial Period:**

✓ Install and use the software free of cost for 30 days.
✓ Evaluate the tool using your own ETABS models and workflows.
✓ Helps consultants understand the practical benefits before subscription.

🔄 **Flexible Upgrade Options:**

✓ After completion of the free trial period, users can upgrade to:
  • 1-Month Plan
  • 6-Month Plan
  • 12-Month Plan
✓ Flexible subscription options based on project requirements and usage needs.

12. Access Link for 30-Day Free Trial

Register using the link below to get access to the 30-day free trial version of the software.

https://sqve-academy.thinkific.com/courses/sqve-mv-et-01

Once registered, you will receive access to:

- ✓ Software download link
- ✓ Recorded demonstration videos
- ✓ Software brochure and features
- ✓ Installation procedure
- ✓ License key generation process
- ✓ User guidance and support information

All relevant information and resources required to start using the software are available through this registration link.

13. Pricing & Licensing

License Cost (Exclusive of GST)

Duration	Price
1 Month	₹2,000 + GST
6 Months	₹10,000 + GST
12 Months	₹18,000 + GST

GST (18%) will be applicable as per government regulations. GST credit can be availed by registered businesses.

Team / Multi-License Discounts

Attractive discounts are available for organizations and teams:

- 2 to 5 Licenses → 20% Discount
- 6 or More Licenses → 30% Discount

Discount is applied on the base license cost before GST.

14. How to Purchase a License

Step 1: Select Your Plan

Choose: Duration → 1 Month / 6 Months / 12 Months

Number of Licenses

Step 2: Make Payment

Payment link for different options:

1. 1 License | 1 month
2. 1 License | 6 months
3. 1 License | 12 months
4. Customised option

- To confirm the total cost for multiple licenses, please contact us at: apps@sqveconsultants.com.

Step 3: Installation & Activation

After payment, you will receive:

- Software installation link
- License activation instructions
- Further onboarding guidance

15. Support & Assistance

For any queries related to pricing, licensing, or payment:

📧 Email: apps@sqveconsultants.com

ANNEXURE A

Sample Summary Report

Structural Model Verification Report - Summary

Automated Structural Model Verification Tool for ETABS - SQVE-MV-ET-01 - Version 2.0

Model File: 3-Tall building-drift_check

Report Generated On: 27-Aug-2026 | 14:59:44

Introduction

This document provides a consolidated summary of the key results, observations and information extracted from the detailed Structural Model Verification Report generated by the Automated Structural Model Verification Tool for ETABS. It is intended to provide a quick overview of the model verification results and to help the user identify sections requiring further attention or engineering review. This summary report shall be read in conjunction with the detailed report. The detailed report contains the applicable assumptions, methodology, extracted data, detailed results, recommendations and engineering review notes associated with each individual verification section.

Interpretation of Symbols

✓ **PASS** – The implemented check is satisfied based on the available model information.

⚠ **WARNING** – The implemented check identifies a condition requiring review.

i **SUMMARY** – Information is extracted and presented for user verification; no independent pass/fail determination is made by the tool for that item.

Verification Summary

Section	Summary / Verification Result	Status
1. Introduction	This report presents automated Phase-2 structural model verification checks to assist engineers in identifying potential modelling issues and reviewing model consistency. Future versions will incorporate additional verification and engineering checks.	i
2. Model Units	Units considered in the model: kN, m	i
3. Story Geometry Database	30 stories with Point Objects (Nodes) are present in the model. Story heights observed: 3.5 m (30 stories).	i
4. Overall Building Dimensions	Overall height of the analytical model is 105 m. Maximum plan dimension in the global X direction is 32 m at Story30. Maximum plan dimension in the global Y direction is 32 m at Story30.	i
5. Duplicate Point Object Verification	No duplicate Point Objects are detected within the specified duplicate node tolerance.	✓
6. Duplicate Beam Verification	No duplicate beams are detected in the analytical model.	✓
7. Collinear Overlapping Beam Verification	No collinear overlapping beams are detected in the analytical model.	✓
8. Duplicate Column Verification	No duplicate Column Objects are detected in the analytical model.	✓

Section	Summary / Verification Result	Status
9. Overlapping Column Verification	No collinear overlapping Column Objects are detected in the analytical model.	✓
10. Floating Column Verification	No floating columns are detected in the analytical model.	✓
11. Support Restraint Verification	All identified bottom joints have valid restraint assignments (61 joints are supported).	✓
12. Support Restraint Classification And Consistency Verification	All supported bottom joints have consistent restraint assignments. All supports are Pinned.	✓
13. Unexpected Joint Restraint Assignments	No unexpected joint restraint assignments were identified. All restraint assignments are located at the bottom of the structure (foundation level).	✓
14. Material Properties	The analytical model contains 1 concrete grade (M30), 2 reinforcement grades (HYSD415, HYSD500), and 1 structural steel grade (Fe345), and 1 other material (Tendon).	i
15. Beam Cover Summary	TC represents Top Cover and BC represents Bottom Cover for beams (Cover to Longitudinal Rabar Group Centroid). All concrete beam properties have the same cover combination- 3 beam properties (TC : 60 mm, BC : 60 mm).	i
16. Column Cover Summary	Clear cover for confinement bars. All concrete column properties have the same cover value - 2 column properties (Cover : 40 mm).	i
17. Beam Material Summary	All concrete beam section properties are assigned the same material grade (M30) - 3 beam properties (M30).	i
18. Beam Stiffness Modifier Summary	The analytical model contains 2 different beam stiffness modifier sets: 2 beam properties (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 0.35, I22 = 0.35, Mass = 1, Weight = 1); 1 beam property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 1, I22 = 1, Mass = 1, Weight = 1).	i
19. Column Material Summary	All concrete column section properties are assigned the same material grade (M30): 2 column properties.	i
20. Column Stiffness Modifier Summary	The analytical model contains 2 different column stiffness modifier sets: 1 column property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 0.7, I22 = 0.7, Mass = 1, Weight = 1); 1 column property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 1, I22 = 1, Mass = 1, Weight = 1).	i
21. Beam And Column Property Assignment Verification	All identified beam members (3690) have been assigned valid beam section properties and all identified column members (570) have been assigned valid column section properties. No incorrect property assignments were detected in the analytical model.	✓
22. Slab Material Property Summary	All slab section properties are assigned the same material grade (M30): 3 slab properties.	i
23. Slab Modeling Type Summary	All slab section properties are assigned the same modeling type (Shell-Thin): 3 slab properties.	i

Section	Summary / Verification Result	Status
24. Slab Stiffness Modifier Summary	The analytical model contains 2 different slab stiffness modifier sets: 2 slab properties (f11 = 1, f22 = 1, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1 slab property (f11 = 0.25, f22 = 0.25, f12 = 0.25, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1).	ℹ
25. Wall Material Property Summary	All wall section properties are assigned the same material grade (M30): 3 wall properties.	ℹ
26. Wall Modeling Type Summary	All wall section properties are assigned the same modeling type (Shell-Thin): 3 wall properties.	ℹ
27. Wall Stiffness Modifier Summary	The analytical model contains 3 different wall stiffness modifier sets: 1 wall property (f11 = 0.7, f22 = 0.7, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1 wall property (f11 = 0.35, f22 = 0.35, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1 wall property (f11 = 1, f22 = 1, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1).	ℹ
28. Diaphragm Assignment Verification	The analytical model contains 30 area objects: 30 area objects (Rigid diaphragm).	ℹ
29. Frame Member End Length Offset Summary	All 4260 frame members are assigned Auto End Length Offsets.	ℹ
30. Insertion Point / Cardinal Point Summary	All 3690 beam members are assigned the same cardinal point (8 (Top Center)). All 570 column members are assigned the same cardinal point (5 (Middle Center)). The "Do not transform frame stiffness" option is enabled for all 3690 beam members, and the option is enabled for all 570 column members.	ℹ
31. Frame Auto Mesh Options Summary	Auto Mesh, At Intermediate Joints and At Intersections are enabled for all 4260 frame members.	✓
32. Member Release Summary	No member end releases are assigned to any frame member.	ℹ
33. Slab And Wall Property Assignment Verification	No inadvertent RCC slab-to-wall or RCC wall-to-slab section property assignments were detected in the analytical model.	✓
34. Floor Meshing Strategy Summary	All 30 floor areas are assigned the same floor meshing strategy (Default).	ℹ
35. Floor Meshing Auto Cookie Cut Configuration	No floor area object has been assigned the Auto Cookie Cut meshing strategy in the analytical model.	ℹ
36. Floor Meshing Add Restraints Review	The Add Restraints option is not enabled for any floor area object. All 30 floor area objects are assigned with the Add Restraints option disabled.	✓
37. Wall Meshing Strategy Summary	Engineering review is recommended to verify that the 1170 wall areas are intentionally and adequately meshed in accordance with the intended structural modelling approach.	⚠
38. Wall Meshing Add Restraints Review	The Add Restraints option is enabled for 1170 wall area objects and disabled for 0 wall area objects.	⚠
39. Auto Edge Constraint Summary	Auto Edge Constraint has been assigned to all 1200 floor and wall area objects identified in the analytical model.	ℹ

Section	Summary / Verification Result	Status
40. Wall Pier Label Assignment Verification	No Pier Labels were identified in the analytical model.	⚠
41. Spandrel Label Summary	No Spandrel Labels were identified in the analytical model.	⚠
42. Gravity Udl Sign Convention Verification	All 3690 distributed force load assignments in the Gravity direction pass the sign convention check and have positive Force A and Force B values.	✓
43. Gravity Point Load Sign Convention Verification	No concentrated point loads assigned in the Gravity direction are found in the analytical model. Hence, no Gravity Point Load sign verification is required.	i
44. Shell Uniform Load Set Sign Convention Verification	No Shell Uniform Load Sets are found in the analytical model. Hence, no Shell Uniform Load Set sign convention verification is required.	i
45. Area Uniform Load Sign Convention Verification	All 30 Area Uniform Load assignments with Direction specified as Gravity pass the sign convention check and have positive Load Values.	✓
46. Gravity Load Joint Displacement Summary	9542 joint displacement records are reviewed for gravity load deflection, including internal mesh joints. The maximum joint displacements identified for the individual gravity load cases (without load combinations) are UX = 10.0 mm, UY = -36.0 mm, and UZ = -33.6 mm.	i
47. First Natural Time Period	The first natural time period of the analytical model is 3.73 seconds.	i
48. Static Earthquake Load Direction Verification	Auto Seismic Load Patterns are identified in both the global X and global Y directions.	✓
49. Static Earthquake Load Story Range Verification	For the static earthquake load patterns EQX & EQy : Top Story = Story30, Bottom Story = Base.	i
50. Static Earthquake Force Time Period Definition Verification	All identified Auto Seismic Load Patterns are defined using User Defined Period Type. For the static earthquake load patterns (EQX), the User Defined Time Period is 2.2 sec in the global X direction. For the static earthquake load patterns (EQy), the User Defined Time Period is 2.2 sec in the global Y direction.	i
51. Static Earthquake Force Input Parameter Verification	For the static earthquake load patterns (EQX) in the global X direction, the seismic input parameters are (Z = 0.36, Site Type = II, I = 1.2, R = 5); For the static earthquake load patterns (EQy) in the global Y direction, the seismic input parameters are (Z = 0.36, Site Type = II, I = 1, R = 5).	i
52. Seismic Mass Source Verification	The seismic mass obtained from the Static Earthquake Load Pattern and the ETABS Joint Mass Table are matching within the specified 2% tolerance in the program.	✓
53. Mass Source Direction Check	Vertical mass is absent.	⚠
54. Story Mass And Adjacent Story Mass Difference Check	No intermediate adjacent story mass difference is identified after excluding the first and last adjacent story pairs.	i
55. Static Earthquake Base Shear	In the X direction, the governing static earthquake Base Shear is 9432.5 kN, which is 2.7% of the gravity load. In the Y direction, the governing static earthquake Base Shear is 8476.9 kN, which is 2.4% of the gravity load.	i

Section	Summary / Verification Result	Status
56. Response Spectrum Load Cases Detected	4 Response Spectrum Load Cases are detected in the analytical model (RSX, RSY, RSX-DR, RSY-DR).	i
57. Response Spectrum Base Shear - X Direction	All 2 Response Spectrum Load Cases are unscaled in the X direction (RSX, RSX-DR).	⚠
58. Response Spectrum Base Shear - Y Direction	Out of 2 Response Spectrum Load Cases having predominant Y-direction base reaction, 1 is scaled to meet or exceed the governing static earthquake Base Shear (RSY), while 1 is below the governing static earthquake Base Shear (RSY-DR).	⚠
59. Response Spectrum Base Shear - Vertical Direction	No Response Spectrum Load Case having predominant vertical (Z-direction) base reaction was identified.	⚠
60. 65% Mass Participation Check | Torsion Mode Check	65% cumulative translational mass participation is achieved in both the global UX and UY directions, mixed mode behaviour is detected, and the torsional mode does not occur before the required translational modes.	⚠
61. Closely Spaced Modes Check	The modal separation is greater than the minimum separation criterion of 10%.	✓
62. 90% Mass Participation Check	Horizontal mass participation is adequate in both the UX and UY directions, while vertical mass participation is absent.	⚠
63. Torsional Eccentricity Override Check	Diaphragm-specific torsional eccentricity override values are not available in both the X and Y directions.	⚠
64. Eccentricity Ratio Summary	The maximum absolute Ex/Lx ratio is 0.11 at Storey: Story30, while the maximum absolute Ey/Ly ratio is 0.23 at Storey: Story8.	i
65. Response Spectrum Storey Shear Summary	The maximum absolute cumulative storey shear in the X direction is 9417.2 kN at Storey: Story1, while the maximum absolute cumulative storey shear in the Y direction is 9227.1 kN at Storey: Story1.	i
66. Seismic Drift Summary	The maximum seismic drift in the X direction is 0.0009 (Δ/H) at Story12 (RSX-DR), while the maximum seismic drift in the Y direction is 0.0008 (Δ/H) at Story13 (RSY-DR).	i
67. Scope And Limitations	This summary report provides a quick automated review of selected aspects of the ETABS structural model and highlights items requiring further attention. It is intended as preliminary guidance only and does not replace detailed engineering review and engineering judgement.	i

ANNEXURE B

Sample Detailed Report

STRUCTURAL MODEL VERIFICATION REPORT

Automated Structural Model Verification Tool for ETABS | SQVE-MV-ET-01 | Ver 2.0

Model File: D:\1-Tech\Software demo\1-Tall building\3-Tall building-drift.EDB

Report Generated: 27-Aug-2026 | 14:59:44

Viewing Recommendation: This report is optimised for A4 viewing and printing. The browser zoom function may be used to adjust the display scale for comfortable on-screen viewing.

1. INTRODUCTION

Model verification is an essential step in structural quality assurance. The Structural Model Verification Tool performs automated checks on selected parameters of the ETABS analytical model to assist engineers in reviewing model consistency, identifying potential modelling issues, and improving the overall quality of the analytical model at an early stage of the design process.

This report presents the results of the Phase-2 verification checks. Additional automated verification modules and engineering checks will be incorporated in future versions of the tool to further enhance the reliability, completeness, and efficiency of structural model review.

Summary: This report presents automated Phase-2 structural model verification checks to assist engineers in identifying potential modelling issues and reviewing model consistency. Future versions will incorporate additional verification and engineering checks.

2. MODEL UNITS

The units currently defined in the ETABS analytical model are presented below for user reference and verification. The user should confirm that the force, length, and temperature units are consistent with the intended modelling, analysis, and design units before reviewing the verification results.

Model Units

Force	Length	Temperature
kN	m	C

Summary: Units considered in the model: kN, m

3. STORY GEOMETRY DATABASE

The story-wise geometry extracted from the ETABS analytical model is presented below for user reference and verification. The table includes the story height, elevation, and maximum plan dimensions of each story.

The user should review these values to confirm that the reported story heights, elevations, and overall plan dimensions are consistent with the intended structural geometry. This information also helps identify modelling elements or joints that may have been placed outside the intended building boundary.

Story Geometry Summary

Sr.	Story	Height (m)	Elevation (m)	Max. X Dimension (m)	Max. Y Dimension (m)
1	Story30	3.5	105.000	32.000	32.000

Sr.	Story	Height (m)	Elevation (m)	Max. X Dimension (m)	Max. Y Dimension (m)
2	Story29	3.5	101.500	32.000	32.000
3	Story28	3.5	98.000	32.000	32.000
4	Story27	3.5	94.500	32.000	32.000
5	Story26	3.5	91.000	32.000	32.000
6	Story25	3.5	87.500	32.000	32.000
7	Story24	3.5	84.000	32.000	32.000
8	Story23	3.5	80.500	32.000	32.000
9	Story22	3.5	77.000	32.000	32.000
10	Story21	3.5	73.500	32.000	32.000
11	Story20	3.5	70.000	32.000	32.000
12	Story19	3.5	66.500	32.000	32.000
13	Story18	3.5	63.000	32.000	32.000
14	Story17	3.5	59.500	32.000	32.000
15	Story16	3.5	56.000	32.000	32.000
16	Story15	3.5	52.500	32.000	32.000
17	Story14	3.5	49.000	32.000	32.000
18	Story13	3.5	45.500	32.000	32.000
19	Story12	3.5	42.000	32.000	32.000
20	Story11	3.5	38.500	32.000	32.000
21	Story10	3.5	35.000	32.000	32.000
22	Story9	3.5	31.500	32.000	32.000

Sr.	Story	Height (m)	Elevation (m)	Max. X Dimension (m)	Max. Y Dimension (m)
23	Story8	3.5	28.000	32.000	32.000
24	Story7	3.5	24.500	32.000	32.000
25	Story6	3.5	21.000	32.000	32.000
26	Story5	3.5	17.500	32.000	32.000
27	Story4	3.5	14.000	32.000	32.000
28	Story3	3.5	10.500	32.000	32.000
29	Story2	3.5	7.000	32.000	32.000
30	Story1	3.5	3.500	32.000	32.000

N/A indicates that no Point Objects (Nodes) are present at the corresponding story level.

Summary: 30 stories with Point Objects (Nodes) are present in the model. Story heights observed: 3.5 m (30 stories).

---

4. OVERALL BUILDING DIMENSIONS

The overall dimensions of the analytical model are summarized below for user reference and verification. The overall building height is determined from the difference between the maximum and minimum Z coordinates of all Point Objects in the analytical model.

The maximum plan dimensions in the global X and Y directions are identified from the Story Geometry Database. The user should review these values to confirm that the reported overall dimensions are consistent with the intended structural geometry.

Overall Building Dimensions

Parameter	Value	Governing Story
Overall Building Height (m)	105	N/A
Maximum X Dimension (m)	32	Story30
Maximum Y Dimension (m)	32	Story30

Summary: Overall height of the analytical model is 105 m. Maximum plan dimension in the global X direction is 32 m at Story30. Maximum plan dimension in the global Y direction is 32 m at Story30.

The maximum plan dimensions in the global X and Y directions represent the coordinate - to - coordinate extent between the extreme Point Objects of the analytical model in the respective global direction.

5. DUPLICATE POINT OBJECT VERIFICATION

This verification checks the analytical model for duplicate Point Objects (Nodes) located within the specified duplicate node tolerance. Duplicate Point Objects may be created unintentionally during modelling, importing models from external software, or incomplete model editing.

Such duplicate Point Objects should be reviewed, as unintended duplicate nodes may result in incorrect structural connectivity or other modelling inconsistencies. The verification is performed using a duplicate node tolerance of 10 mm.

✓ PASS: No duplicate Point Objects are detected within the specified duplicate node tolerance.

6. DUPLICATE BEAM VERIFICATION

This verification checks the analytical model for duplicate beams. Two beams are considered duplicate when they have identical end joints, irrespective of the member connectivity direction. The verification identifies beams having identical Joint-I and Joint-J locations, including cases where the Joint-I and Joint-J sequence is reversed.

Duplicate beams may be created unintentionally during modelling, importing models from external software, or repeated member creation. The presence of duplicate beams does not necessarily indicate a modelling error. However, unintended duplicate Frame Objects should be reviewed and corrected where appropriate.

✓ PASS: No duplicate beams are detected in the analytical model.

---

7. COLLINEAR OVERLAPPING BEAM VERIFICATION

ℹ Note:

This verification checks the analytical model for collinear overlapping beams. Unlike duplicate beams, the members identified in this verification do not have identical end joints. Instead, the verification identifies beams located along the same geometric line that share a common overlapping length. The verification identifies complete overlap, partial overlap, and cases where one beam is located entirely within another.

The verification is applicable to horizontal beams aligned along the global X-axis, global Y-axis, and beams inclined in plan. Beams inclined in elevation (sloping beams) are presently not included in this verification and should be reviewed separately. Collinear overlapping beams may be created unintentionally during modelling, copying, importing models from external software, or model editing. The presence of collinear overlapping beams does not necessarily indicate a modelling error. However, unintended overlapping members should be reviewed and corrected where appropriate.

✓ PASS: No collinear overlapping beams are detected in the analytical model.

---

8. DUPLICATE COLUMN VERIFICATION

ℹ Note:

This verification checks the analytical model for duplicate columns. Two columns are considered duplicate when they have identical end joints, irrespective of the member connectivity direction. The verification identifies columns having identical end joints, including cases where the connectivity direction is reversed.

Duplicate columns may be created unintentionally during modelling, importing models from external software, or repeated member creation. The presence of duplicate columns does not necessarily indicate a modelling error. However, unintended duplicate columns should be reviewed and corrected where appropriate.

✓ PASS: No duplicate Column Objects are detected in the analytical model.

---

9. OVERLAPPING COLUMN VERIFICATION

This verification checks the analytical model for collinear overlapping columns. Unlike duplicate columns, the members identified in this verification do not have identical end joints. Instead, the verification identifies columns located along the same geometric line that share a common overlapping length, including cases of complete overlap, partial overlap, and where one column is located entirely within another.

The verification is applicable to vertical columns aligned predominantly along the global Z-axis. Minor coordinate differences within the specified geometric tolerance are treated as numerical variations, and such columns are considered vertical for the purpose of this verification. Columns inclined in plan or elevation (sloping columns) are presently not included in this verification and should be reviewed separately.

Collinear overlapping columns may be created unintentionally during modelling, copying, importing models from external software, or model editing. The presence of overlapping columns does not necessarily indicate a modelling error. However, unintended overlapping columns should be reviewed and corrected where appropriate.

✓ PASS: No collinear overlapping Column Objects are detected in the analytical model.

10. FLOATING COLUMN VERIFICATION

This verification checks the analytical model for floating columns. This verification also assists in validating the intended structural system by confirming that columns designed to function as floating columns have been modelled correctly. It can help identify situations where an unintended supporting column has been introduced below an intended floating column, thereby altering the intended load path.A floating column is defined as a vertical load-carrying column that terminates at an intermediate level without continuing vertically to another supporting column below. Such columns may be intentionally provided as part of the structural system, for example above transfer beams or transfer girders, or they may represent unintended modelling inconsistencies. The verification identifies columns that do not have vertical continuity below their terminating joint and reports them for engineering review.

Columns whose bottom elevation coincides with the lowest column base in the analytical model are treated as foundation columns and are excluded from this verification. The present verification assumes a common foundation level based on the minimum column base elevation in the analytical model. Models having multiple foundation levels, stepped foundations, or sloping foundation systems should therefore be reviewed carefully, as additional engineering judgement may be required when interpreting the reported results. The presence of floating columns does not necessarily indicate a modelling error. However, all reported floating columns should be reviewed to confirm that they are intentional and consistent with the intended structural load path. The present verification is applicable only to Column Objects modelled using line elements in the analytical model. Columns represented using shell or area elements are presently excluded from this verification and should be reviewed separately where applicable. Columns supported by walls, shell elements, plate elements, or other analytical elements instead of another Column Object will also be reported as floating columns in the present verification. This is because the verification checks only the continuity between Column Objects and does not evaluate load transfer through shell or area elements. Such cases should therefore be reviewed using engineering judgement.

✓ PASS: No floating columns are detected in the analytical model.

11. SUPPORT RESTRAINT VERIFICATION

This verification checks whether restraint assignments have been provided to the bottom-most joints of the analytical model. The verification identifies the lowest joints of the structural model and confirms whether the required support restraints have been assigned. Joints without restraint assignments are reported for engineering review, as missing supports may result in unstable structural behaviour, incorrect load transfer, or analysis errors.

The present verification assumes that the structural supports are located at the lowest level of the analytical model. Models having multiple foundation levels, stepped foundations, split-level basements, sloping sites, or supports provided at different elevations should therefore be reviewed carefully, as additional engineering judgement may be required while interpreting the reported results.

The current version of the software considers only one type of foundation/support system in a model- Joint Restraints, Spring Supports, or Area Springs. Models with a combination of these support types are not supported in the current version. For such models, the report may not produce accurate results, and the findings should be verified through engineering review.

✓ PASS: All identified bottom joints have valid restraint assignments (61 joints are supported).

---

12. SUPPORT RESTRAINT CLASSIFICATION AND CONSISTENCY VERIFICATION

This verification identifies the restraint condition assigned to each supported bottom joint in the analytical model and classifies the support as Fully Fixed, Fully Pinned, or Partially Restrained based on the translational (UX, UY, UZ) and rotational (RX, RY, RZ) restraint assignments. The predominant support type adopted in the model is then identified and compared with all other supports to determine the overall consistency of the support system.

The presence of different support restraint conditions does not necessarily indicate a modelling error. However, support conditions that differ from the predominant support philosophy should be reviewed to confirm that they are intentional and consistent with the structural analysis assumptions, foundation detailing and structural drawings.

Support Restraint Classification Summary

Support Type	Number of Supports
Fixed	0
Pinned	61
Partially Restrained	0
Without Restraint Assignment	0

✓ PASS: All supported bottom joints have consistent restraint assignments. All supports are Pinned.

---

13. UNEXPECTED JOINT RESTRAINT ASSIGNMENTS

This verification identifies joint restraint assignments that are located at elevations other than the identified foundation support level in the analytical model. For the purpose of this verification, the foundation support level is considered as the bottom-most restrained joint level identified in the structural model. Any joint restraint assignment detected above this level is reported for engineering review.

This verification is intended to identify potentially unintended support conditions that may alter the structural behaviour of the analytical model. Models containing stepped foundations, split-level foundations, existing building connections or intentionally restrained joints above the foundation level may also be reported by this verification. Such cases do not necessarily indicate a modelling error and should therefore be reviewed using engineering judgement.

✓ PASS: No unexpected joint restraint assignments were identified. All restraint assignments are located at the bottom of the structure (foundation level).

14. MATERIAL PROPERTIES

This section provides a summary of all material properties defined in the analytical model. The information is intended to provide a quick reference for verifying that the required concrete, structural steel, reinforcing steel and other material grades have been defined before interpreting the analysis and design results.

The presence of a material property in this summary does not confirm its assignment to any structural element. Similarly, the absence of an expected material property may indicate that the corresponding material has not been defined in the analytical model. The listed material properties should therefore be reviewed to confirm that they are consistent with the intended design assumptions, project specifications and structural drawings.

Material Properties

Sr.	Material Property
1	Fe345
2	HYSD415
3	HYSD500
4	M30
5	Tendon

Summary: The analytical model contains 1 concrete grade (M30), 2 reinforcement grades (HYSD415, HYSD500), and 1 structural steel grade (Fe345), and 1 other material (Tendon).

15. BEAM COVER SUMMARY

This section summarizes the top and bottom concrete cover (cover to Longitudinal Rabar Group Centroid) specified for the reinforced concrete beam section properties defined in the analytical model. The summary is presented to facilitate engineering review of the concrete cover adopted for different beam section properties and to enable quick identification of beam section properties having cover values different from the predominant beam cover philosophy adopted in the analytical model.

The reported cover values are extracted directly from the beam section property definitions in the analytical model. The presence of different cover values does not necessarily indicate a modelling error. Different cover values may be intentionally assigned for durability requirements, fire resistance, exposure conditions or construction considerations. Such cases should therefore be reviewed using appropriate engineering judgement.

Beam Cover Summary

Top Cover (mm)	Bottom Cover (mm)	Beam Properties
60	60	3

Summary: TC represents Top Cover and BC represents Bottom Cover for beams (Cover to Longitudinal Rabar Group Centroid). All concrete beam properties have the same cover combination- 3 beam properties (TC : 60 mm, BC : 60 mm).

---

16. COLUMN COVER SUMMARY

This section summarizes the clear concrete cover (Clear Cover for Confinement Bars) specified for the reinforced concrete column section properties defined in the analytical model. The summary is presented to facilitate engineering review of the concrete cover adopted for different column section properties and to enable quick identification of column section properties having cover values different from the predominant column cover philosophy adopted in the analytical model.

The reported cover values are extracted directly from the column section property definitions in the analytical model. The presence of different cover values does not necessarily indicate a modelling error. Different cover values may be intentionally assigned to satisfy durability requirements, fire resistance, exposure conditions or construction considerations. Such cases should therefore be reviewed using appropriate engineering judgement.

Column Cover Summary

Clear Cover (mm)	No. of Column Properties
40	2

Summary: Clear cover for confinement bars. All concrete column properties have the same cover value - 2 column properties (Cover : 40 mm).

17. BEAM MATERIAL SUMMARY

This section summarizes the distribution of material grades assigned to reinforced concrete beam section properties defined in the analytical model. The predominant material grade is identified to facilitate quick engineering review of modelling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of section properties assigned to each material grade. The calculation does not consider the actual quantity of concrete (such as volume in cubic metres) associated with these properties.

Accordingly, the reported percentages are intended only for indicative and comparative purposes, providing a quick reference to the user, and should not be interpreted as a representation of material quantity distribution within the structure.

Beam Material Summary

Material	Number of Beam Properties	Percentage (%)
M30	3	100.0

Predominant Material:

M30 (3 Properties - 100.0%)

Summary: All concrete beam section properties are assigned the same material grade (M30) - 3 beam properties (M30).

---

18. BEAM STIFFNESS MODIFIER SUMMARY

This section summarizes the stiffness modifiers assigned to the reinforced concrete beam section properties defined in the analytical model. The predominant stiffness modifier set adopted for the majority of beam section properties is identified and presented to facilitate engineering review of the modelling assumptions adopted for concrete beam elements.

The reported stiffness modifiers are extracted directly from the beam section property definitions in the analytical model. This section does not verify the correctness or appropriateness of the assigned stiffness modifier values. Different stiffness modifier values may be intentionally assigned to represent cracked section properties, transfer members, coupling beams, deep beams or other project-specific modelling assumptions. The reported information should therefore be reviewed using appropriate engineering judgement.

Beam Stiffness Modifier Summary

Beam Property	Area Modifier	Shear 2 Modifier	Shear 3 Modifier	Torsion Modifier	Moment I33 Modifier	Moment I22 Modifier	Mass Modifier	Weight Modifier
Majority of Beam Properties (2 Properties - 66.7%)	1	1	1	1	0.35	0.35	1	1

⚠ WARNING: Multiple beam stiffness modifier sets have been identified in the analytical model. Beam section properties having stiffness modifiers different from the predominant stiffness modifier set are listed below for engineering review.

→ Recommendation :

Review the beam section properties listed below and confirm that the reported stiffness modifiers are intentional and consistent with the structural design philosophy, analysis assumptions and project-specific modelling requirements.

Beam Properties Having Stiffness Modifiers Different from the Predominant Modifier Set

Beam Property	Area Modifier	Shear 2 Modifier	Shear 3 Modifier	Torsion Modifier	Moment I33 Modifier	Moment I22 Modifier	Mass Modifier	Weight Modifier
ConcBm	1	1	1	1	1	1	1	1

Summary: The analytical model contains 2 different beam stiffness modifier sets: 2 beam properties (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 0.35, I22 = 0.35, Mass = 1, Weight = 1); 1 beam property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 1, I22 = 1, Mass = 1, Weight = 1).

19. COLUMN MATERIAL SUMMARY

This section summarizes the distribution of material grades assigned to reinforced concrete column section properties defined in the analytical model. The predominant material grade is identified to facilitate quick engineering review of modelling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of section properties assigned to each material grade. The calculation does not consider the actual quantity of concrete (such as volume in cubic metres) associated with these properties.

Accordingly, the reported percentages are intended only for indicative and comparative purposes, providing a quick reference to the user, and should not be interpreted as a representation of material quantity distribution within the structure.

Column Material Summary

Material	Number of Column Properties	Percentage (%)
M30	2	100.0

Predominant Material:

M30 (2 Properties - 100.0%)

Summary: All concrete column section properties are assigned the same material grade (M30): 2 column properties.

---

20. COLUMN STIFFNESS MODIFIER SUMMARY

This section summarizes the stiffness modifiers assigned to the reinforced concrete column section properties defined in the analytical model. The summary is presented to facilitate engineering review of the stiffness modifier values adopted for different column section properties and to enable quick identification of column section properties having stiffness modifiers different from the predominant stiffness modifier philosophy adopted in the analytical model.

The reported stiffness modifiers are extracted directly from the frame section property definitions in the analytical model. The presence of different stiffness modifier values does not necessarily indicate a modelling error. Different stiffness modifiers may be intentionally assigned to represent cracked section properties, different stages of construction, retrofit measures, special design requirements or other modelling assumptions. Such cases should therefore be reviewed using appropriate engineering judgement.

Concrete Column Stiffness Modifier Summary

Column Properties	Area Modifier	Shear 2 Modifier	Shear 3 Modifier	Torsion Modifier	Moment I33 Modifier	Moment I22 Modifier	Mass Modifier	Weight Modifier
Majority of Column Properties (1 Properties - 50.0%)	1	1	1	1	0.7	0.7	1	1
1 Properties (50.0%)	1	1	1	1	1	1	1	1

⚠ WARNING: Multiple stiffness modifier sets have been identified for reinforced concrete column section properties. Column section properties having stiffness modifiers different from the predominant stiffness modifier set are listed below for engineering review.

→ Recommendation :

Review the column section properties listed below and verify that the reported stiffness modifier values are intentional and consistent with the structural analysis assumptions, applicable design standards and project requirements.

Column Properties Having Stiffness Modifiers Different from Predominant Modifier Set

Column Property	Area Modifier	Shear 2 Modifier	Shear 3 Modifier	Torsion Modifier	Moment I33 Modifier	Moment I22 Modifier	Mass Modifier	Weight Modifier
ConcCol	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00

Total Concrete Column Properties: 2

Unique Stiffness Modifier Sets: 2

Summary: The analytical model contains 2 different column stiffness modifier sets: 1 column property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 0.7, I22 = 0.7, Mass = 1, Weight = 1); 1 column property (AMod = 1, A2 = 1, A3 = 1, J = 1, I33 = 1, I22 = 1, Mass = 1, Weight = 1).

21. BEAM AND COLUMN PROPERTY ASSIGNMENT VERIFICATION

This section verifies the assignment of reinforced concrete (RCC) beam and column section properties in the analytical model. The check specifically identifies cases where an RCC column section property is assigned to a beam member or an RCC beam section property is assigned to a column member. Section properties that are not identified as RCC beam or RCC column properties, such as steel or other structural member properties, are not flagged by this check.

The purpose of this verification is to identify potential inadvertent interchange of RCC beam and column section properties that may affect the intended modelling and subsequent structural design. The check is based on the RCC beam and RCC column section property databases available in the model and is intended to assist the engineer in reviewing potential assignment inconsistencies.

✓ PASS: All identified beam members (3690) have been assigned valid beam section properties and all identified column members (570) have been assigned valid column section properties. No incorrect property assignments were detected in the analytical model.

22. SLAB MATERIAL PROPERTY SUMMARY

This section summarizes the distribution of material grades assigned to slab section properties defined in the analytical model. The predominant material grade is identified to facilitate quick engineering review of modelling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of slab section properties assigned to each material grade. The calculation does not consider the actual quantity of concrete (such as volume in cubic metres) associated with these properties.

Accordingly, the reported percentages are intended only for indicative and comparative purposes, providing a quick reference to the user, and should not be interpreted as a representation of material quantity distribution within the structure.

Slab Material Summary

Material	Number of Slab Properties	Percentage (%)
M30	3	100.0

Summary: All slab section properties are assigned the same material grade (M30): 3 slab properties.

23. SLAB MODELING TYPE SUMMARY

This section summarizes the distribution of modeling types assigned to slab section properties defined in the analytical model. The predominant modeling type is identified to facilitate quick engineering review of modeling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of slab section properties assigned to each modeling type. The calculation does not consider actual slab area, stiffness contribution, or structural behavior.

Accordingly, the reported percentages are intended only for indicative and comparative purposes. Engineering judgement is required to assess the suitability of the modeling approach.

Slab Modeling Type Summary

Modeling Type	Number of Slab Properties	Percentage (%)
Shell-Thin	3	100.0

Summary: All slab section properties are assigned the same modeling type (Shell-Thin): 3 slab properties.

---

24. SLAB STIFFNESS MODIFIER SUMMARY

This section summarizes the stiffness modifiers assigned to slab section properties defined in the analytical model. The summary facilitates engineering review of stiffness assumptions and helps identify slab properties having stiffness modifiers different from the predominant modelling philosophy adopted in the model.

The reported stiffness modifiers are extracted directly from slab property definitions. The presence of different modifier values does not necessarily indicate an error, as they may represent cracked sections, different analysis assumptions or design requirements. Engineering judgement is therefore required while reviewing the results.

Slab Stiffness Modifier Summary

Description	f11	f22	f12	m11	m22	m12	v13	v23	Mass	Weight
Majority of Slab Properties (2 Properties - 66.7%)	1	1	1	1	1	1	1	1	1	1
1 Properties (33.3%)	0.25	0.25	0.25	1	1	1	1	1	1	1

⚠ WARNING: Multiple stiffness modifier sets have been identified for slab properties.

➜ Recommendation :

Review slab properties having modifiers different from the predominant set and verify modelling consistency.

Slab Properties with Modifiers Different from Predominant Set

Slab Property	f11	f22	f12	m11	m22	m12	v13	v23	Mass	Weight
Slab-200	0.25	0.25	0.25	1.00	1.00	1.00	1.00	1.00	1.00	1.00

Total Slab Properties: 3

Unique Stiffness Modifier Sets: 2

Summary: The analytical model contains 2 different slab stiffness modifier sets: 2 slab properties (f11 = 1, f22 = 1, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1

slab property (f11 = 0.25, f22 = 0.25, f12 = 0.25, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1).

25. WALL MATERIAL PROPERTY SUMMARY

This section summarizes the distribution of material grades assigned to wall section properties defined in the analytical model. The predominant material grade is identified to facilitate quick engineering review of modelling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of wall section properties assigned to each material grade. The calculation does not consider the actual quantity of concrete (such as volume in cubic metres) associated with these properties.

Accordingly, the reported percentages are intended only for indicative and comparative purposes and should not be interpreted as a representation of material quantity distribution within the structure.

Wall Material Summary

Material	Number of Wall Properties	Percentage (%)
M30	3	100.0

Predominant Material:

M30 (3 Properties - 100.0%)

Summary: All wall section properties are assigned the same material grade (M30): 3 wall properties.

26. WALL MODELING TYPE SUMMARY

This section summarizes the distribution of modeling types assigned to wall section properties defined in the analytical model. The predominant modeling type is identified to facilitate quick engineering review of modeling consistency.

It is important to note that the predominance and corresponding percentage distribution are determined solely based on the number of wall section properties assigned to each modeling type. The calculation does not consider actual wall area, stiffness contribution, or structural behaviour.

Accordingly, the reported percentages are intended only for indicative and comparative purposes. Engineering judgement is required to assess the suitability of the modeling approach.

Wall Modeling Type Summary

Modeling Type	Number of Wall Properties	Percentage (%)
Shell-Thin	3	100.0

Summary: All wall section properties are assigned the same modeling type (Shell-Thin): 3 wall properties.

---

27. WALL STIFFNESS MODIFIER SUMMARY

This section summarizes the stiffness modifiers assigned to wall section properties defined in the analytical model. The summary facilitates engineering review of stiffness assumptions and helps identify wall properties having stiffness modifiers different from the predominant modelling philosophy adopted in the model.

The reported stiffness modifiers are extracted directly from wall property definitions. The presence of different modifier values does not necessarily indicate an error, as they may represent cracked sections, different analysis assumptions or design requirements. Engineering judgement is therefore required while reviewing the results.

Wall Stiffness Modifier Summary

Description	f11	f22	f12	m11	m22	m12	v13	v23	Mass	Weight
Majority of Wall Properties (1 Properties - 33.3%)	0.70	0.70	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
1 Properties (33.3%)	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
1 Properties (33.3%)	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00

⚠ WARNING: Multiple stiffness modifier sets have been identified for wall properties.

→ Recommendation :

Review wall properties having modifiers different from the predominant set and verify modelling consistency.

Wall Properties with Modifiers Different from Predominant Set

Wall Property	f11	f22	f12	m11	m22	m12	v13	v23	Mass	Weight
Wall-350	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
Wall1	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00

Summary: The analytical model contains 3 different wall stiffness modifier sets: 1 wall property (f11 = 0.7, f22 = 0.7, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1 wall property (f11 = 0.35, f22 = 0.35, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1); 1 wall property (f11 = 1, f22 = 1, f12 = 1, m11 = 1, m22 = 1, m12 = 1, v13 = 1, v23 = 1, Mass = 1, Weight = 1).

28. DIAPHRAGM ASSIGNMENT VERIFICATION

Diaphragms play a critical role in distributing lateral forces to vertical load-resisting elements. In typical structural analysis, rigid diaphragms are assumed to transfer lateral loads efficiently to the vertical structural system. The assignment of appropriate diaphragm types is therefore essential to ensure that the intended load transfer mechanism is correctly represented in the analytical model.

The summary below provides an overview of diaphragm assignments in the model, including unassigned cases (None). The predominant diaphragm type is identified based on the number of area objects assigned to each diaphragm. This assessment is based purely on the count of elements and does not consider the actual geometric area of the elements; hence, the percentages reported are indicative and intended only for quick review.

The identified predominant diaphragm type, along with area objects having different or missing diaphragm assignments, should be reviewed by the user. The suitability of diaphragm assignments must be verified considering the actual structural system, modelling intent, and project-specific requirements.

Diaphragm Assignment Summary

Diaphragm	Rigidity	Number of Area Objects	Percentage (%)
D1	Rigid	30	100.0

Summary: The analytical model contains 30 area objects: 30 area objects (Rigid diaphragm).

29. FRAME MEMBER END LENGTH OFFSET SUMMARY

End Length Offsets may be assigned to frame members to account for the physical dimensions and connectivity of beams and columns at their ends. The applicability and magnitude of such offsets depend on the structural configuration, member geometry, connection conditions, and project-specific modelling requirements.

This section summarizes the End Length Offset assignments identified for frame members in the analytical model. The offset distribution is presented based on the number of frame members having identical Offset I and Offset J values and is intended to provide a quick indication of the modelling approach adopted.

The reported percentages are based solely on the number of frame members and do not represent actual member length, volume, or any other physical quantity. Accordingly, the summary is indicative in nature, and the assigned offsets should be reviewed by the user based on the actual structural system and modelling requirements.

Frame Member End Length Offset Summary

Category	Number of Frame Members	Percentage (%)
Total Frame Members	4260	100.0
Auto End Length Offset	4260	100.0
Manual End Length Offset	0	0.0

Summary: All 4260 frame members are assigned Auto End Length Offsets.

30. INSERTION POINT / CARDINAL POINT SUMMARY

Insertion points define the relative position of a frame member with respect to its connecting joints and include the assigned cardinal point and joint offsets. They are useful where the analytical member axis does not coincide with the physical location of the structural member, such as beams framing into the face of columns or members requiring an intentional eccentricity to represent the actual structural arrangement.

When an insertion point or joint offset is used to represent an actual structural eccentricity, the option "Do not transform frame stiffness for offsets from centroid for non-P/T floors" should generally remain unchecked where the eccentricity is intended to be considered in the analytical stiffness representation. If this option is checked, ETABS does not transform the frame stiffness to account for the cardinal point or joint offsets, and the insertion point and offsets are used for graphical display only. The applicability of the stiffness transformation should therefore be reviewed based on the intended structural modelling approach and project requirements.

The predominant cardinal point for beams and columns is identified based on the number of frame members assigned to each cardinal point. The predominant value is therefore based on the frequency of assignment and does not represent an assessment of the actual structural requirement or physical geometry. The predominant cardinal point, as well as members having cardinal points different from the predominant value, shall be reviewed by the user based on the actual structural system, member geometry, framing arrangement and project-specific modelling requirements.

Insertion Point / Cardinal Point Summary

Member Type	Total Members	Predominant Cardinal Point	Members Assigned
Beam	3690	8 (Top Center)	3690
Column	570	5 (Middle Center)	570

Frame Stiffness Transformation Summary

Member Type	Total Members	Stiffness Transformation Enabled	Do Not Transform Stiffness
Beam	3690	0	3690
Column	570	0	570

⚠ WARNING:3690 beam member(s) have the "Do not transform frame stiffness" option enabled.

⚠ WARNING:570 column member(s) have the "Do not transform frame stiffness" option enabled.

→ Recommendation :

Where insertion points or joint offsets are intended to represent actual structural eccentricities, review the stiffness transformation setting and verify that the selected option is consistent with the intended analytical behaviour.

Summary: All 3690 beam members are assigned the same cardinal point (8 (Top Center)). All 570 column members are assigned the same cardinal point (5 (Middle Center)). The "Do not transform frame stiffness" option is enabled for all 3690 beam members, and the option is enabled for all 570 column members.

31. FRAME AUTO MESH OPTIONS SUMMARY

Automatic meshing options for frame members are important for ensuring that the analytical model contains appropriate joints at intermediate locations and at intersections between connected frame elements. These joints allow the finite element model to properly represent connectivity and facilitate appropriate transfer of forces between connected structural members.

This verification checks whether Auto Mesh, At Intermediate Joints, and At Intersections options are enabled for the frame members. The check is performed for each frame member individually, and a member is identified for review if any one of these three options is not enabled.

The assignment of these meshing options should be reviewed with respect to the structural system, member connectivity, and modelling requirements of the project. The verification provided by this section is intended to identify potential modelling conditions requiring engineering review and does not by itself establish the adequacy of the selected meshing parameters.

Frame Auto Mesh Options Summary

Parameter	Number of Frame Members	Percentage (%)
All Required Options Enabled	4260	100.0
Requiring Review	0	0.0

✓ PASS: Auto Mesh, At Intermediate Joints and At Intersections are enabled for all 4260 frame members.

No frame members were identified requiring review based on the selected auto mesh options.

---

32. MEMBER RELEASE SUMMARY

Member releases define the degrees of freedom released at the ends of frame members and therefore influence the force transfer, connectivity, and structural behaviour represented in the analytical model. Appropriate member release assignments are required to ensure that the analytical model represents the intended structural system and connection conditions.

The present check identifies the release configuration represented in the member release data by considering the release conditions separately at End I and End J. Members having identical End I and End J release configurations are grouped together, and the configuration assigned to the maximum number of members is identified as the predominant member release configuration.

The predominant release configuration is determined only on the basis of the number of frame members having the same release configuration. No consideration is given to member length, section properties, stiffness, forces, or other structural parameters. The predominant configuration as well as the exceptional configurations identified in the model shall be reviewed by the user based on the intended structural system, connection behaviour, and specific project requirements. The total number of frame members represents the complete frame-member population of the analytical model, whereas the members with releases represent frame members identified with release assignments in the member release data. Therefore, the number of members with releases may be lower than the total number of frame members.

Summary: No member end releases are assigned to any frame member.

---

33. SLAB AND WALL PROPERTY ASSIGNMENT VERIFICATION

This verification checks for inadvertent interchange of reinforced concrete (RCC) slab and wall section properties in the analytical model. The check identifies cases where an RCC wall property is assigned to a floor object or an RCC slab property is assigned to a wall object.

Section properties that are not identified as RCC slab or RCC wall properties, such as deck, steel or other structural area properties, are not flagged by this check. The verification is therefore intended specifically to identify potential RCC slab-to-wall or wall-to-slab property assignment errors and does not constitute a general validation of all area section property assignments.

The results are intended to assist the engineer in reviewing potential RCC area-property assignment inconsistencies with reference to the intended structural system, element function, geometry and project-specific modelling requirements.

RCC Slab / Wall Property Cross-Assignment Summary

Parameter	Number of Area Objects	Percentage (%)
Floor Objects	30	2.5
Wall Objects	1170	97.5
No RCC Cross-Assignments	1200	100.0
RCC Cross-Assignments	0	0.0

✓ PASS: No inadvertent RCC slab-to-wall or RCC wall-to-slab section property assignments were detected in the analytical model.

34. FLOOR MESHING STRATEGY SUMMARY

Appropriate meshing of floor elements is important for finite element analysis because the floor area must be discretized into suitable finite elements while maintaining appropriate connectivity with the surrounding structural elements. An appropriate meshing strategy helps the analytical model represent the intended geometry, load transfer and structural behaviour.

The present check summarizes the floor meshing strategy assigned to the floor area objects in the analytical model. Default Meshing and Auto Cookie Cut are considered acceptable meshing approaches, with Auto Cookie Cut generally providing a preferred approach where appropriate compatibility with beams and walls is required. The suitability of the selected meshing strategy shall be reviewed with respect to the structural system and project-specific modelling requirements.

The predominant meshing strategy and the corresponding percentage are determined solely on the basis of the number of floor area objects assigned to each meshing strategy. The calculation does not consider the actual physical area of the floor objects. Accordingly, the reported percentages are indicative values intended to provide a quick review of modelling consistency and shall not be interpreted as a representation of the physical floor area distribution.

It is recommended that the user review the analysis mesh generated by the software and confirm that the resulting mesh is appropriate for the intended structural system, member connectivity, geometry and project-specific analysis requirements.

Floor Meshing Strategy Summary

Meshing Strategy	Number of Floor Areas	Percentage (%)
Default	30	100.0

Predominant Floor Meshing Strategy:

Default (30 Floor Areas - 100.0%)

The Default floor meshing strategy has been assigned throughout the model. The Default option is considered an acceptable meshing approach; however, the generated analysis mesh should be reviewed by the user to confirm its suitability for the intended structural system.

Summary: All 30 floor areas are assigned the same floor meshing strategy (Default).

---

35. FLOOR MESHING AUTO COOKIE CUT CONFIGURATION

Auto Cookie Cut is a floor meshing strategy used to generate compatible finite element mesh around intersecting structural elements. Appropriate mesh generation at beams and walls is important to maintain connectivity between floor elements and the surrounding structural system.

For floor areas assigned the Auto Cookie Cut meshing strategy, the 'At Beams' and 'At Walls' options are considered mandatory checks in the present verification. These options should be enabled where required to ensure that the generated floor mesh is appropriately compatible with the intersecting beam and wall elements. The 'At Grids' option is considered optional in this verification.

The following check identifies the Auto Cookie Cut configuration assigned to the floor area objects and highlights any floor areas where the mandatory 'At Beams' or 'At Walls' option is not enabled. The final suitability of the selected settings shall be reviewed by the user with respect to the structural system, geometry, connectivity and project-specific modelling requirements.

Summary: No floor area object has been assigned the Auto Cookie Cut meshing strategy in the analytical model.

Therefore, the Auto Cookie Cut-specific configuration check for 'At Beams', 'At Walls' and 'At Grids' is not applicable.

The floor meshing strategy identified in Section 34 should nevertheless be reviewed by the user with respect to the intended structural system and project-specific modelling requirements.

---

36. FLOOR MESHING ADD RESTRAINTS REVIEW

The Add Restraints option in floor meshing controls whether restraints are considered at the edges or corners of floor area objects during mesh generation. Unintended restraint assignments may introduce additional constraints into the analytical model and may therefore influence the structural response.

For typical floor-area modelling, the Add Restraints option is generally expected to remain disabled unless restraints at the corresponding floor edges or corners are intentionally required to represent the actual structural behaviour. Accordingly, the present check identifies the floor areas for which this option has been enabled so that the assignment can be reviewed against the intended structural system and project-specific modelling requirements.

The percentage distribution is determined solely on the basis of the number of floor area objects for which the Add Restraints option is assigned. The calculation does not consider the actual physical area of the floor objects. Accordingly, the reported percentages are indicative values intended only to provide a quick review of modelling consistency.

Floor Meshing Add Restraints Summary

Add Restraints	Number of Floor Areas	Percentage (%)
Yes	0	0.0
No	30	100.0

✓ PASS: The Add Restraints option is not enabled for any floor area object. All 30 floor area objects are assigned with the Add Restraints option disabled.

---

37. WALL MESHING STRATEGY SUMMARY

Appropriate meshing of wall elements is important for finite element analysis because wall area objects must be discretized into suitable finite elements while maintaining appropriate connectivity with adjoining structural elements. An appropriate wall meshing strategy helps the analytical model represent the intended wall geometry, load transfer and structural behaviour.

For wall elements, Auto Rectangular Mesh is considered the preferred meshing approach where automatic subdivision of wall objects into rectangular finite elements is appropriate. Other meshing approaches, including Default Mesh or user-defined Mesh Nv x Nh, may also be intentionally used depending on the wall geometry, structural system and project-specific modelling requirements. Therefore, the selected wall meshing strategy should be reviewed in the context of the actual analytical model.

The predominant wall meshing strategy is identified solely on the basis of the number of wall area objects assigned to each meshing strategy. The calculation does not consider the actual physical area of the wall objects. Accordingly, the reported percentages are indicative values intended only to provide a quick review of modelling consistency and shall not be interpreted as a representation of the physical wall area distribution.

The predominant wall meshing strategy, as well as wall areas assigned with a meshing strategy different from the predominant strategy, shall be reviewed by the user with respect to the intended structural system, wall geometry, connectivity and project-specific analysis requirements.

Wall Meshing Strategy Summary

Meshing Strategy	Number of Wall Areas	Percentage (%)
Default	1170	100.0

⚠ WARNING: Engineering review is recommended to verify that the 1170 wall areas are intentionally and adequately meshed in accordance with the intended structural modelling approach.

---

38. WALL MESHING ADD RESTRAINTS REVIEW

The Add Restraints option in wall meshing controls whether additional restraints are considered at the edges or corners of wall area objects during mesh generation. Unintended restraint assignments may introduce additional constraints into the analytical model and may therefore influence the structural response.

For typical wall-area modelling, the Add Restraints option is generally expected to remain disabled unless additional restraints at the corresponding wall edges or corners are intentionally required to represent the actual structural behaviour. Accordingly, the present check identifies the wall areas for which this option has been enabled so that the assignment can be reviewed against the intended structural system, support conditions and project-specific modelling requirements.

The percentage distribution is determined solely on the basis of the number of wall area objects for which the Add Restraints option is assigned. The calculation does not consider the actual physical area of the wall objects. Accordingly, the reported percentages are indicative values intended only to provide a quick review of modelling consistency.

Wall Meshing Add Restraints Summary

Add Restraints	Number of Wall Areas	Percentage (%)
Yes	1170	100.0
No	0	0.0

⚠ WARNING: The Add Restraints option is enabled for 1170 wall area objects and disabled for 0 wall area objects.

Engineering review is required to verify whether the additional restraints at the corresponding wall edges or corners are intentionally required based on the structural system, support conditions and project-specific modelling requirements.

39. AUTO EDGE CONSTRAINT SUMMARY

Auto Edge Constraints are used in the analytical model to establish displacement compatibility between adjoining shell elements and between shell elements and other structural elements where the connected mesh does not provide direct joint-to-joint connectivity. They can be particularly useful at locations where shell meshes are incompatible or where structural elements connect to the edge of a shell element rather than directly to a corner joint.

The present check identifies the number and distribution of floor and wall area objects for which Auto Edge Constraint assignments are present in the analytical model. The check is intended to provide a quick indication of the assignment made in the model and does not determine whether an Auto Edge Constraint is structurally required or whether an existing assignment is appropriate.

The requirement and suitability of Auto Edge Constraints shall be reviewed by the user with respect to the actual shell mesh, adjoining structural elements, connectivity, structural system and project-specific modelling requirements. Particular attention should be given to locations where the use of an edge constraint may unintentionally influence the intended load-transfer mechanism.

The percentages reported in this section are based on the number of floor and wall area objects and do not represent actual physical area proportions. The results are therefore intended only as an indicative and quick reference for engineering review.

Auto Edge Constraint Summary

Auto Edge Constraint	Number of Area Objects	Percentage (%)
Assigned	1200	100.0
Not Assigned	0	0.0

Summary: Auto Edge Constraint has been assigned to all 1200 floor and wall area objects identified in the analytical model.

The assignment should nevertheless be reviewed by the user to confirm that Auto Edge Constraints are appropriate for the actual shell mesh, adjoining structural elements, connectivity and intended load - transfer mechanism.

40. WALL PIER LABEL ASSIGNMENT VERIFICATION

Pier Labels are used in ETABS to identify and group wall elements for pier-based wall design and design-result reporting. Correct assignment of Pier Labels is therefore important for the subsequent design and interpretation of wall elements.

The present check verifies whether a Pier Label assignment has been identified for each wall area object in the analytical model. The verification is performed by comparing the UniqueName of each wall object with the UniqueName available in the Pier Label assignment database.

The present check verifies only the presence of a Pier Label assignment. It does not verify whether the assigned Pier Label is appropriate for the actual structural system, wall geometry, design intent or project-specific requirements. Engineering review of the assigned Pier Labels is therefore recommended.

⚠ WARNING: No Pier Labels were identified in the analytical model.

Pier Labels are required for performing the design of wall elements.

Engineering review is required to verify and assign the appropriate Pier Labels to the wall elements.

41. SPANDREL LABEL SUMMARY

Spandrel Labels are used in ETABS to identify and group wall portions that form a spandrel for analysis, design and reporting of spandrel forces. Spandrel definitions may be applicable to horizontal wall segments, coupling beams and wall portions located above or below openings.

The present check identifies whether Spandrel Label assignments are present in the analytical model and reports the number of shell elements for which Spandrel Labels have been identified. This check does not verify whether the assigned Spandrel Labels are appropriate for the actual structural system or project requirements.

Engineering review is therefore required to determine whether Spandrel Labels are required and, where assigned, whether the assignments are appropriate for the intended wall configuration.

⚠ WARNING: No Spandrel Labels were identified in the analytical model.

Review whether Spandrel Labels are required for the modelled wall configuration, particularly where horizontal wall segments, coupling beams or wall portions above or below openings are present.

Engineering review is required to verify the applicability of Spandrel Label assignments based on the actual structural system and project requirements.

---

42. GRAVITY UDL SIGN CONVENTION VERIFICATION

This section verifies the sign convention of uniformly distributed force loads assigned to frame elements in the analytical model when the load direction is specified as Gravity. For a distributed force load assigned in the Gravity direction, the load intensity is expected to be positive so that the load acts in the intended downward Gravity direction.

A negative Force A and/or Force B value for a Gravity-directed distributed load indicates that the corresponding load component acts opposite to the intended Gravity direction and may therefore represent an upward-applied load. Such assignments are identified for engineering review.

This verification is limited to distributed loads having Load Type as Force and Direction as Gravity. Distributed loads assigned in Global, Projected, Local or other directions are not verified in this section because their sign convention depends upon the selected loading direction and coordinate system.

The check verifies only the sign convention of the distributed load intensities. The adequacy or correctness of the load magnitude is not evaluated by this section.

✓ PASS: All 3690 distributed force load assignments in the Gravity direction pass the sign convention check and have positive Force A and Force B values.

---

43. GRAVITY POINT LOAD SIGN CONVENTION VERIFICATION

This section verifies the sign convention of concentrated point force loads assigned to frame elements in the analytical model when the load direction is specified as Gravity. For a point force load assigned in the Gravity direction, the load intensity is expected to be positive so that the load acts in the intended downward Gravity direction.

A negative or zero Force value for a Gravity-directed point load indicates that the corresponding load does not act as a positive Gravity-directed load and should therefore be reviewed to confirm the intended loading direction and loading convention.

This verification is limited to point loads having Load Type as Force and Direction as Gravity. Point loads assigned in Global, Projected, Local or other directions are not verified in this section because their sign convention depends upon the selected loading direction and coordinate system.

The check verifies only the sign convention of the point load intensity. The adequacy or correctness of the load magnitude is not evaluated by this section.

Summary: No concentrated point loads assigned in the Gravity direction are found in the analytical model. Hence, no Gravity Point Load sign verification is required.

---

44. SHELL UNIFORM LOAD SET SIGN CONVENTION VERIFICATION

This section verifies the sign convention of Shell Uniform Load Sets defined in the analytical model. Shell Uniform Load Sets are considered as uniformly distributed gravity loads acting on shell elements and, accordingly, the assigned Load Value is expected to be positive.

The verification checks the Load Value defined for each Shell Uniform Load Set. A positive Load Value is considered consistent with the adopted Gravity load convention, whereas a zero or negative Load Value is identified for engineering review.

This verification is limited to the sign convention of the defined Shell Uniform Load Sets. It does not verify whether the load set has been assigned to shell elements, whether the assigned load magnitude is adequate, or whether the selected Load Pattern is appropriate. These aspects require separate engineering review.

Summary: No Shell Uniform Load Sets are found in the analytical model. Hence, no Shell Uniform Load Set sign convention verification is required.

---

45. AREA UNIFORM LOAD SIGN CONVENTION VERIFICATION

This section verifies the sign convention of uniformly distributed loads assigned to shell elements in the analytical model.

The verification is applicable only to Area Uniform Load assignments for which the Direction is specified as Gravity. For such assignments, a positive Load Value is expected so that the load acts in the intended downward Gravity direction.

This verification checks only the sign of the Load Value and does not verify the adequacy of the assigned load magnitude, the suitability of the selected Load Pattern, or whether the load has been appropriately assigned to the intended shell elements. These aspects require separate engineering review.

Area Uniform Load Verification Summary

Total Area Uniform Load Assignments: 30

Gravity Direction Load Assignments Checked: 30

✓ PASS: All 30 Area Uniform Load assignments with Direction specified as Gravity pass the sign convention check and have positive Load Values.

---

46. GRAVITY LOAD JOINT DISPLACEMENT SUMMARY

This section summarizes the maximum joint displacements in the global X, Y and Z directions identified for the relevant gravity load cases extracted from the analytical model.

Dead load and live load related load patterns are identified, and the corresponding load cases are considered for this review. For each identified gravity load case, the maximum absolute joint displacement in the global X, Y and Z directions is identified, while the original sign of the governing displacement is retained for reporting.

The reported displacement values are presented in millimetres and are intended to facilitate a quick engineering review of the overall equilibrium and stability of the analytical model under gravity loading.

Under gravity loading, the lateral displacement in the global X and Y directions would generally be expected to be relatively small. The vertical displacement in the global Z direction should also be reviewed for reasonableness considering the structural system, member sizes, loading and support conditions.

Significantly large lateral or vertical displacement values may indicate modelling inconsistencies, inadequate restraint, connectivity issues, incorrect member properties, support conditions or other abnormalities in the analytical model. Such observations should be reviewed by the engineer before proceeding with further structural verification.

This section does not verify the reported displacement values against any permissible serviceability or deflection limit. The purpose of this check is to identify potentially abnormal displacement behaviour under gravity loading and facilitate engineering review.

Gravity Load Joint Displacement Summary

Load Case	Maximum UX (mm)	Maximum UY (mm)	Maximum UZ (mm)
Dead	10.0	-36.0	-33.6
Live	7.2	-16.9	-16.0

Summary: 9542 joint displacement records are reviewed for gravity load deflection, including internal mesh joints. The maximum joint displacements identified for the individual gravity load cases (without load combinations) are UX = 10.0 mm, UY = -36.0 mm, and UZ = -33.6 mm.

47. FIRST NATURAL TIME PERIOD

This section presents the first natural time period obtained from the modal analysis of the analytical model.

The first natural time period provides an indicative measure of the overall dynamic characteristics of the structural model and may provide an early indication of whether the model is behaving as expected.

An unusually low or high first natural time period may indicate potential modelling issues related to the mass, stiffness, structural connectivity, support conditions or other characteristics of the analytical model. Where the reported value appears abnormal, a detailed engineering review of the model is recommended before proceeding with further verification.

It is important to note that this section does not verify the first natural time period against any codal or project-specific permissible limit. The value is presented only for quick engineering review and to facilitate identification of potentially abnormal dynamic behaviour of the analytical model.

First Natural Time Period

Parameter	Value	Unit
First Natural Time Period	3.73	sec

Engineering Review:

The reported first natural time period should be reviewed by the engineer considering the structural system, overall building configuration, mass distribution, stiffness characteristics and support conditions.

Where the value is found to be significantly different from the expected behaviour of the structure, the analytical model should be reviewed before proceeding with further structural verification.

Summary: The first natural time period of the analytical model is 3.73 seconds.

---

48. STATIC EARTHQUAKE LOAD DIRECTION VERIFICATION

This section verifies whether static earthquake force patterns are defined in both principal horizontal directions of the analytical model, namely the global X and global Y directions.

The check is based on the Auto Seismic Load Pattern Definition table extracted from ETABS and identifies the static earthquake load patterns assigned in the X and Y directions, including the corresponding eccentricity variants where applicable.

The availability of static earthquake force patterns in both principal directions is important because these load patterns provide the reference static seismic forces required for subsequent verification and scaling of the response spectrum seismic forces in the corresponding directions.

Accordingly, absence of a static earthquake force pattern in either the global X or global Y direction may prevent the intended direction-specific scaling and comparison of response spectrum seismic forces. Such a condition shall therefore be reviewed before proceeding with the subsequent seismic force scaling verification.

This section verifies only the availability and direction of the static earthquake load patterns. The adequacy of the seismic load magnitude and the scaling of response spectrum forces are not evaluated in this section and will be addressed in subsequent verification steps.

Seismic Load Direction Summary

Load Case	Direction	X Direction?	X + Ecc?	X - Ecc?	Y Direction?	Y + Ecc?	Y - Ecc?
EQX	X	Yes	No	No	No	No	No
EQy	Y	No	No	No	Yes	No	No

✓ PASS: Auto Seismic Load Patterns are identified in both the global X and global Y directions.

---

49. STATIC EARTHQUAKE LOAD STORY RANGE VERIFICATION

This section verifies and reports the Top Story and Bottom Story defined for each static earthquake force pattern in the analytical model.

The selected story range defines the portion of the analytical model over which the corresponding static earthquake force pattern is applied. Verification of the Top Story and Bottom Story is therefore important to confirm that the intended portion of the structure is considered for the static earthquake force.

The reported story range is extracted directly from the ETABS Auto Seismic Load Pattern Definition table for the identified static earthquake force patterns in the global X and global Y directions.

Correct identification of the Top Story and Bottom Story is particularly important because the static earthquake force patterns are subsequently used as reference cases for seismic force verification and response spectrum force scaling.

The software does not independently determine whether the selected story range is appropriate for the structural system. The reported Top Story and Bottom Story are provided for engineering review, and any inconsistency with the intended seismic load application shall be reviewed before proceeding with subsequent seismic verification.

Seismic Load Story Range Summary

Load Case	Direction	Top Story	Bottom Story
EQX	X	Story30	Base
EQy	Y	Story30	Base

Summary: For the static earthquake load patterns EQX & EQy : Top Story = Story30, Bottom Story = Base.

---

50. STATIC EARTHQUAKE FORCE TIME PERIOD DEFINITION VERIFICATION

This section verifies the time period definition used for the static earthquake force calculation for each identified Auto Seismic Load Pattern.

The time period is a critical input in the calculation of static earthquake force because it directly influences the seismic design coefficient and, consequently, the calculated design base shear.

The time period used for the static earthquake force calculation is required to be determined in accordance with the applicable provisions of IS 1893, IS 16700, or other governing seismic design provisions applicable to the project and structural system.

The manually determined time period shall be entered in ETABS as a User Defined Time Period. Accordingly, a User Defined Period Type is required for the static earthquake force definition so that the time period adopted for the seismic force calculation is explicitly established by the engineer.

If the Period Type is Program Defined, a Warning is issued indicating that the static earthquake force definition is not based on a User Defined Time Period. The engineer shall review the model and confirm that the time period has been determined and entered in accordance with the applicable seismic design provisions.

Where the Period Type is User Defined, the User Defined Time Period entered in ETABS is reported for engineering review.

In the current version of the software, the numerical value of the User Defined Time Period is not independently calculated or cross-checked against IS 1893, IS 16700, or other applicable provisions. The software only verifies the period definition and reports the time period value available in the ETABS Auto Seismic Load Pattern Definition table.

The reported User Defined Time Period shall therefore be reviewed by the engineer against the applicable seismic design provisions before proceeding with further verification of the static earthquake force and related seismic parameters.

Seismic Period Definition Summary

Load Pattern	Direction	Period Type	User Defined Period (sec)	Period Used (sec)
EQX	X	User Defined	2.20	2.20
EQy	Y	User Defined	2.20	2.20

Summary: All identified Auto Seismic Load Patterns are defined using User Defined Period Type. For the static earthquake load patterns (EQX), the User Defined Time Period is 2.2 sec in the global X direction. For the static earthquake load patterns (EQy), the User Defined Time Period is 2.2 sec in the global Y direction.

51. STATIC EARTHQUAKE FORCE INPUT PARAMETER VERIFICATION

This section reports the principal input parameters defined for the static earthquake force calculation for the identified Auto Seismic Load Patterns, including Zone Factor (Z), Site Type, Importance Factor (I) and Response Reduction Factor (R).

These parameters are important inputs to the calculation of the static earthquake force and directly influence the seismic design coefficient and the resulting design Base Shear.

The static earthquake force parameters are particularly important because the resulting Base Shear will subsequently be used as the reference value for the scaling and verification of the response spectrum seismic forces in the corresponding principal directions.

The values reported in this section are extracted directly from the ETABS Auto Seismic Load Pattern Definition table and are presented for quick reference and engineering review. The engineer shall verify these parameters against the intended seismic design basis and the applicable provisions of the governing seismic design standard.

In the current version of the software, these input parameters are not independently calculated or validated against the applicable code provisions. The software only extracts and reports the values defined in the analytical model.

Any discrepancy or inappropriate value identified during engineering review shall be resolved before proceeding with subsequent verification of the static earthquake Base Shear and response spectrum force scaling.

Seismic Design Input Parameter Summary

Load Pattern	Direction	Zone Factor (Z)	Site Type	Importance Factor (I)	Response Reduction Factor (R)
EQX	X	0.36	II	1.2	5
EQy	Y	0.36	II	1	5

Summary: For the static earthquake load patterns (EQX) in the global X direction, the seismic input parameters are (Z = 0.36, Site Type = II, I = 1.2, R = 5); For the static earthquake load patterns (EQy) in the global Y direction, the seismic input parameters are (Z = 0.36, Site Type = II, I = 1, R = 5).

52. SEISMIC MASS SOURCE VERIFICATION

This section compares the seismic mass obtained from two independent sources within the analytical model - the seismic mass derived from the Static Earthquake Load Pattern and the seismic mass obtained from the ETABS Joint Mass Table.

The comparison is important because the seismic mass directly influences the earthquake force calculation. Any significant inconsistency between these two sources may indicate an issue in the seismic load definition, mass source definition, story range selection or other aspects of the analytical model.

In particular, an incorrect Top Story or Bottom Story selection in the Static Earthquake Load Pattern may cause the seismic weight considered for the static earthquake force to differ from the total seismic mass represented by the ETABS Joint Mass Table. Such an inconsistency should therefore be reviewed before proceeding with further seismic verification.

The seismic weight reported by the Static Earthquake Load Pattern is converted from kN to kg using the acceleration due to gravity of 9.81 m/s^2.

This section performs only a comparison of the two extracted seismic mass values. It does not independently calculate the seismic mass or verify the adequacy of the mass source against code requirements.

Mass Source Multipliers for Different Load Patterns are also presented below for user verification. These values are extracted from the analytical model and are provided for ready reference. No independent verification or review of the individual mass source multipliers is performed in this section.

Mass Source Multipliers for Different Load Patterns:

Dead: 1.00

Live: 0.50

Seismic Mass Comparison

Mass Source	Mass (kg)
Static Earthquake Load Pattern	36004437.7
ETABS Joint Mass Table	36177563.0

Percentage Difference in Seismic Mass: 0.48 %

✓ PASS: The seismic mass obtained from the Static Earthquake Load Pattern and the ETABS Joint Mass Table are matching within the specified 2% tolerance in the program.

No significant discrepancy was identified between the two seismic mass sources based on this verification.

---

53. MASS SOURCE DIRECTION CHECK

This section verifies the presence of structural mass in the three translational directions, namely UX, UY and UZ, based on the mass values extracted from the ETABS Joint Mass Table of the analytical model.

The presence of mass in the relevant translational directions is important for the correct representation of the dynamic behaviour of the structure and for generation of the corresponding seismic inertial forces. The total mass available in each direction is therefore reported for quick reference and user verification.

A warning is reported if the mass is absent or effectively zero in any of the three translational directions. Absence of mass in a particular direction may prevent the corresponding seismic inertial force from being generated as intended. In such cases, the user should review the mass source definition and confirm that the intended structural mass is appropriately represented before proceeding with further seismic analysis and verification.

Mass Source Direction Summary

Direction	Total Mass (kg)
UX	36177563.0
UY	36177563.0
UZ	0.0

⚠ WARNING: Vertical mass is absent.

→ Recommendation : If vertical earthquake effects

are required to be considered as per the IS code provisions,

please enable vertical mass in the ETABS Mass Source definition.

---

54. STORY MASS AND ADJACENT STORY MASS DIFFERENCE CHECK

This section presents the seismic mass at each story level based on the mass values extracted from the ETABS Joint Mass Table and reports the percentage difference in mass between adjacent stories.

The story-wise mass distribution and adjacent story mass difference are presented together to facilitate identification of sudden increases or decreases in mass along the height of the structure and to assist in identifying potential mass irregularities.

Significant variation in mass between adjacent stories, particularly where the adjacent stories have similar structural configuration, may indicate missing loads, incorrect gravity load assignments or other modelling inconsistencies. Such locations should be reviewed by the engineer.

The reported values are provided for engineering review and do not by themselves establish whether a mass irregularity exists in accordance with the applicable code provisions.

Story Mass and Adjacent Story Mass Difference Summary

Story Name	Story Mass (kg)	Adjacent Story Pair	Mass Difference (%)
Base	160827.40	N/A	N/A
Story1	1205918.7	Base → Story1	86.7
Story2	1205918.7	Story1 → Story2	0.0
Story3	1205918.7	Story2 → Story3	0.0
Story4	1205918.7	Story3 → Story4	0.0
Story5	1205918.7	Story4 → Story5	0.0
Story6	1205918.7	Story5 → Story6	0.0
Story7	1205918.7	Story6 → Story7	0.0
Story8	1205918.7	Story7 → Story8	0.0
Story9	1205918.7	Story8 → Story9	0.0
Story10	1205918.7	Story9 → Story10	0.0
Story11	1205918.7	Story10 → Story11	0.0

Story Name	Story Mass (kg)	Adjacent Story Pair	Mass Difference (%)
Story12	1205918.7	Story11 → Story12	0.0
Story13	1205918.7	Story12 → Story13	0.0
Story14	1205918.7	Story13 → Story14	0.0
Story15	1205918.7	Story14 → Story15	0.0
Story16	1205918.7	Story15 → Story16	0.0
Story17	1205918.7	Story16 → Story17	0.0
Story18	1205918.7	Story17 → Story18	0.0
Story19	1205918.7	Story18 → Story19	0.0
Story20	1205918.7	Story19 → Story20	0.0
Story21	1205918.7	Story20 → Story21	0.0
Story22	1205918.7	Story21 → Story22	0.0
Story23	1205918.7	Story22 → Story23	0.0
Story24	1205918.7	Story23 → Story24	0.0
Story25	1205918.7	Story24 → Story25	0.0
Story26	1205918.7	Story25 → Story26	0.0
Story27	1205918.7	Story26 → Story27	0.0
Story28	1205918.7	Story27 → Story28	0.0
Story29	1205918.7	Story28 → Story29	0.0
Story30	1045093.3	Story29 → Story30	13.3

Summary: No intermediate adjacent story mass difference is identified after excluding the first and last adjacent story pairs.

55. STATIC EARTHQUAKE BASE SHEAR

This section summarizes the governing static earthquake Base Shear in the principal horizontal directions and compares it with the gravity-equivalent load corresponding to the seismic mass of the analytical model.

The governing static earthquake Base Shear in each direction is obtained from the maximum Base Shear identified from the ETABS Auto Seismic Load Pattern results. Where multiple static earthquake load patterns are available in the same direction, the maximum Base Shear is considered.

The seismic mass used for this comparison is obtained from the ETABS Joint Mass Table. The mass is converted from kg to its corresponding gravity-equivalent load in kN using an acceleration due to gravity of 9.81 m/s^2.

The ratio of the governing static earthquake Base Shear to the corresponding gravity-equivalent seismic mass load is calculated and expressed as a percentage. This provides an immediate indication of the magnitude of the static earthquake force relative to the seismic mass of the structure.

Subject to the use of an appropriate time period and seismic design parameters in accordance with the applicable Indian Standard provisions, the resulting value represents the reference seismic force level in the respective direction and is subsequently used for response spectrum force scaling.

Static Earthquake Base Shear and Seismic Force Ratio

Direction	Seismic Mass (kg)	Gravity-Equivalent Load (kN)	Governing Base Shear (kN)	Base Shear / Gravity-Equivalent Load	Seismic Force as (%) of Gravity Load
X	36177563.0	354901.9	9432.5	0.0266	2.66
Y	36177563.0	354901.9	8476.9	0.0239	2.39

Summary: In the X direction, the governing static earthquake Base Shear is 9432.5 kN, which is 2.7% of the gravity load. In the Y direction, the governing static earthquake Base Shear is 8476.9 kN, which is 2.4% of the gravity load.

56. RESPONSE SPECTRUM LOAD CASES DETECTED

This section identifies the Response Spectrum Load Cases defined in the analytical model and presents their names for user verification.

The presence of an appropriate number of Response Spectrum Load Cases is important for representing the intended seismic excitation in the principal horizontal directions and, where applicable, for considering torsional eccentricity and vertical seismic effects.

For example, where torsional eccentricity is required to be considered in both principal horizontal directions, two Response Spectrum Load Cases may be required in the X direction and two in the Y direction, together with a vertical Response Spectrum Load Case where vertical seismic effects are required to be considered.

The Response Spectrum Load Cases identified below are therefore presented for ready reference and engineering review. No independent check of the adequacy or completeness of the Response Spectrum Load Case definition is performed in this section.

Response Spectrum Load Cases Detected

Sr. No.	Response Spectrum Load Case
1	RSX
2	RSY
3	RSX-DR

Sr. No.	Response Spectrum Load Case
4	RSY-DR

Summary: 4 Response Spectrum Load Cases are detected in the analytical model (RSX, RSY, RSX-DR, RSY-DR).

57. RESPONSE SPECTRUM BASE SHEAR - X DIRECTION

This section identifies the Response Spectrum Load Cases for which the absolute X-direction base reaction is predominant and presents the corresponding X-direction Response Spectrum base shear for engineering review.

The predominant direction of each Response Spectrum Load Case is identified from the absolute magnitude of the resulting base reactions in the global X, Y and Z directions. A Response Spectrum Load Case is included in this section when the absolute X-direction base reaction is greater than or equal to the absolute Y-direction and Z-direction base reactions.

The X-direction Response Spectrum base shear is compared with the governing static earthquake base shear in the X direction. This comparison provides a quick review of whether the Response Spectrum base shear satisfies the corresponding static earthquake base shear used as the minimum reference seismic force.

Where the Response Spectrum base shear is equal to or greater than the governing static earthquake base shear, the result is reported as Pass.

Where the Response Spectrum base shear is lower than the governing static earthquake base shear, the result is reported as UNSCALED. Such a condition may indicate that the Response Spectrum Load Case has been intentionally defined with unscaled forces for seismic drift or serviceability evaluation. The user should verify the intended purpose of the load case.

If a Response Spectrum Load Case having a lower base shear is intended to be used for strength design, appropriate scaling of the Response Spectrum forces should be applied so that the required minimum seismic force is achieved in line with applicable IS codes (para no. 7.7.3 of IS 1893 (Part 1)-2016).

No independent calculation of the Response Spectrum base shear is performed in this section.

Response Spectrum Base Shear - X Direction

Response Spectrum Load Case	Response Spectrum Base Shear Vx (kN)	Static Earthquake Base Shear Vx (kN)	Result
RSX	9417.2	9432.5	UNSCALED
RSX-DR	4956.4	9432.5	UNSCALED

UNSCALED: The Response Spectrum Load Case(s) have predominant X-direction base shear lower than the governing static earthquake base shear. The Response Spectrum Load Case(s) appear to be unscaled and may have been defined for seismic drift or serviceability evaluation. The user should verify the intended purpose of these Response Spectrum Load Cases.

If these Response Spectrum Load Cases are intended to be used for strength design, appropriate scaling should be applied so that the required minimum seismic force is achieved in line with para no. 7.7.3 of IS 1893 (Part 1)-2016.

⚠ WARNING: All 2 Response Spectrum Load Cases are unscaled in the X direction (RSX, RSX-DR).

Engineering review is required to confirm whether the Response Spectrum Load Cases identified above with lower base shear are intentionally defined for drift or serviceability evaluation or require scaling for strength design.

58. RESPONSE SPECTRUM BASE SHEAR - Y DIRECTION

This section identifies the Response Spectrum Load Cases for which the absolute Y-direction base reaction is predominant and presents the corresponding Y-direction Response Spectrum base shear for engineering review.

The predominant direction of each Response Spectrum Load Case is identified from the absolute magnitude of the resulting base reactions in the global X, Y and Z directions. A Response Spectrum Load Case is included in this section when the absolute Y-direction base reaction is greater than or equal to the absolute X-direction and Z-direction base reactions.

The Y-direction Response Spectrum base shear is compared with the governing static earthquake base shear in the Y direction. This comparison provides a quick review of whether the Response Spectrum base shear satisfies the corresponding static earthquake base shear used as the minimum reference seismic force.

Where the Response Spectrum base shear is equal to or greater than the governing static earthquake base shear, the result is reported as Pass.

Where the Response Spectrum base shear is lower than the governing static earthquake base shear, the result is reported as UNSCALED. Such a condition may indicate that the Response Spectrum Load Case has been intentionally defined with unscaled forces for seismic drift or serviceability evaluation. The user should verify the intended purpose of the load case.

If a Response Spectrum Load Case having a lower base shear is intended to be used for strength design, appropriate scaling of the Response Spectrum forces should be applied so that the required minimum seismic force is achieved in line with applicable IS codes (para no. 7.7.3 of IS 1893 (Part 1)-2016.).

No independent calculation of the Response Spectrum base shear is performed in this section.

Response Spectrum Base Shear - Y Direction

Response Spectrum Load Case	Response Spectrum Base Shear Vy (kN)	Static Earthquake Base Shear Vy (kN)	Result
RSY	9227.1	8476.9	PASS
RSY-DR	4856.4	8476.9	UNSCALED

UNSCALED:

The Response Spectrum Load Case(s) have predominant Y-direction base shear lower than the governing static earthquake base shear. The Response Spectrum Load Case(s) appear to be unscaled and may have been defined for seismic drift or serviceability evaluation. The user should verify the intended purpose of these Response Spectrum Load Cases.

If these Response Spectrum Load Cases are intended to be used for strength design, appropriate scaling should be applied so that the required minimum seismic force is achieved in line with para no. 7.7.3 of IS 1893 (Part 1)-2016..

⚠ WARNING: Out of 2 Response Spectrum Load Cases having predominant Y-direction base reaction, 1 is scaled to meet or exceed the governing static earthquake Base Shear (RSY), while 1 is below the governing static earthquake Base Shear (RSY-DR).

Engineering review is required to confirm whether the Response Spectrum Load Case identified above with lower base shear is intentionally defined for drift or serviceability evaluation or requires scaling for strength design.

59. RESPONSE SPECTRUM BASE SHEAR - VERTICAL DIRECTION

This section identifies the Response Spectrum Load Cases for which the absolute vertical (Z-direction) base reaction is predominant and presents the corresponding vertical Response Spectrum base shear for engineering review.

The predominant direction of each Response Spectrum Load Case is identified from the absolute magnitude of the resulting base reactions in the global X, Y and Z directions. A

Response Spectrum Load Case is included in this section when the absolute Z-direction base reaction is greater than or equal to the absolute X-direction and Y-direction base reactions.

The vertical Response Spectrum base shear is presented for user verification. Unlike the horizontal Response Spectrum checks, no comparison with the static earthquake base shear is performed in this section.

The purpose of this section is therefore limited to identifying and reporting Response Spectrum Load Cases having predominant vertical response and the corresponding vertical base reaction obtained from the analytical model.

The reported vertical Response Spectrum force should be reviewed by the engineer to confirm that vertical seismic effects have been considered where required by the applicable code provisions and the structural system.

Response Spectrum Base Shear - Vertical Direction

Response Spectrum Load Case	Response Spectrum Base Shear Vz (kN)

⚠ WARNING: No Response Spectrum Load Case having predominant vertical (Z-direction) base reaction was identified.

The presence and magnitude of the vertical Response Spectrum force should be reviewed by the engineer with reference to the applicable code provisions and the requirements of the structural system (para no. 6.3.3.1 of IS 1893 (Part 1)-2016 | Ammendment no. 2, para no. 6.4.6 of IS 1893 (part 1)-2016, para no. 7.7.3 of IS 1893 (Part 1)-2016, para no. 6.3.1 of IS 16700-2023.

---

60. 65% MASS PARTICIPATION CHECK | TORSION MODE CHECK

This section reviews the modal mass participation in the global UX and UY translational directions and identifies the translational and torsional characteristics of the initial vibration modes of the analytical model.

The check provides a quick verification of whether sufficient translational modal mass participation is achieved in the principal horizontal directions and whether a torsional mode occurs at an early stage of the modal sequence (Table 5, Sl. No. (i) of IS 1893 (Part 1)-2016 , para no.5.5.1 of IS 16700-2023). The 65% mass participation criterion is reviewed with reference to Table 6, Sl. No. (vii) of IS 1893 (Part 1)-2016 | Amendment no. 2.

Mode classification is based on the relative mass participation in the translational UX and UY directions and the rotational RZ direction. The classification is used to identify translational, mixed and torsional modes and to review the sequence in which these modes occur.

For the purpose of this review, a mode is classified as a Pure Translational Mode when mass participation is dominant in a single translational direction (UX or UY), while participation in the rotational direction (RZ) and the other translational direction remains negligible. A Mixed Mode is identified when significant mass participation occurs in one or more translational directions along with noticeable rotational participation (RZ), or when the participation is distributed across multiple directions. A Torsional Mode is identified when mass participation is dominant in the rotational direction (RZ), while participation in the translational directions (UX and UY) remains negligible. Modes having negligible mass participation in both translational and rotational directions are classified as Other Modes for the purpose of this report.

Please refer to Table 6, Sl. No. (vii) of IS 1893 (Part 1)-2016 | Amendment No. 2 for the applicable requirement regarding 65% mass participation. Where a large podium is present in the lower storeys of the building, the 65% mass participation requirement is not applicable. Where a large podium is not present in the lower storeys of the building, the required 65% mass participation shall be achieved, and the analytical model should therefore be reviewed and revised, where required, to achieve the required mass participation.

The results presented in this section are intended for quick verification of the analytical model. It is recommended that the engineer additionally review the corresponding graphical mode shapes in ETABS along with mass participation to confirm the mode classification and to verify the presence or absence of significant torsional behaviour.

Modal Classification and Translational Modes Used

Mode	Mode Classification	Natural Period (sec)	Used in First Three Translational Modes
Mode 1	Mixed (Translation + Torsion)	3.73	Yes
Mode 2	Mixed (Translation X & Y)	3.19	Yes
Mode 3	Mixed (Translation + Torsion)	2.26	Yes

The translational modes identified above and marked as 'Yes' are the modes considered for the 65% translational mass participation review.

Cumulative Translational Mass Participation

Direction	Cumulative Mass Participation (%)
UX	68.3
UY	70.7

⚠ WARNING: 65% cumulative translational mass participation is achieved in both the global UX and UY directions, mixed mode behaviour is detected, and the torsional mode does not occur before the required translational modes.

---

61. CLOSELY SPACED MODES CHECK

This section reviews the separation between the first three predominant mode shapes to identify potential closely spaced modal behaviour.

The check is performed using the existing modal periods identified from the analytical model and the established minimum modal separation criterion of 10%. Closely spaced modes may result in changes in the predominant modal characteristics and may lead to interaction between translational and torsional modal behaviour.

The present verification considers the first three mode shapes as the primary predominant modes for this screening check. Where higher modes exhibit significant mass participation or are otherwise identified as predominant modes, the engineer should review those modes separately for potential closely spaced behaviour.

The results presented in this section are intended to facilitate verification of the applicable requirements of Table 6, Serial No. (vii) of IS 1893 (Part 1)-2016, Amendment No. 2 and/or Para 5.5.1 of IS 16700-2023. The engineer should review the modal characteristics and applicable code provisions before final acceptance.

Closely Spaced Modes Check

Mode	Period (sec)	Minimum Required Separation (%)
Mode 1	3.73	10
Mode 2	3.19	10
Mode 3	2.26	10

✓ PASS: The modal separation is greater than the minimum separation criterion of 10%.

62. 90% MASS PARTICIPATION CHECK

This section verifies the cumulative modal mass participation in the UX, UY and UZ directions based on the modal analysis results extracted from the analytical model.

As per Clause 7.7.5.2 of IS 1893 (Part 1)-2016, the number of modes considered in the earthquake analysis should be such that the sum of modal masses of the modes considered is at least 90% of the total seismic mass in the considered direction.

The frequency range of the modes considered is also reviewed with reference to the 33 Hz frequency limit stated in the same provision. Where modes beyond 33 Hz are not explicitly included in the modal combination, their inertia contribution is required to be addressed through an appropriate missing-mass correction procedure in accordance with the applicable structural dynamics principles.

This section therefore collates the total number of modes considered, the maximum frequency achieved, the modal analysis type and the cumulative mass participation achieved in the UX, UY and UZ directions. The information is presented for ready reference and engineering verification of the adequacy of the modal analysis.

Modal Analysis Summary

Parameter	Value
Total Modes Considered	30
Maximum Frequency Achieved (Hz)	17.4
Modal Analysis Type	Eigenvalue Analysis

Cumulative Mass Participation Summary

Direction	Cumulative Mass Participation (%)	Minimum Required (%)
UX	98.4	90
UY	98.2	90
UZ	0.0	90

Modes Required to Reach 90% Mass Participation

Direction	Mode Required to Reach 90%
UX	Mode 10
UY	Mode 9
UZ	Not Reached

⚠ WARNING: Horizontal mass participation is adequate in both the UX and UY directions, while vertical mass participation is absent.

→ Recommendation :

Verify whether vertical seismic force is required as per IS 1893 (Part 1)-2016 (para no. 6.3.3.1 of Amendment no. 2) and/or IS 16700-2023 (para no. 6.3.1).

If applicable, ensure that vertical mass is included in the Mass Source definition by enabling the 'Include Vertical Mass' option in the model.

If vertical mass is already enabled, consider increasing the number of modes to ensure adequate mass participation in the vertical direction.

---

63. TORSIONAL ECCENTRICITY OVERRIDE CHECK

This section verifies the presence of diaphragm-specific override eccentricity values defined in the analytical model for Response Spectrum Load Cases.

As per Clause 7.8.2 of IS 1893 (Part 1)-2016, design eccentricity is required to account for the inherent static eccentricity between the centre of mass and centre of resistance together with the prescribed accidental eccentricity. The applicable design eccentricity is determined using the two expressions specified in the code, with the more severe effect governing.

For consideration of the two design eccentricity conditions in each principal horizontal direction, separate earthquake force cases are required in the X and Y directions. The inherent eccentricity should be determined at each floor level from the difference between the centre of mass and centre of resistance, and the corresponding design eccentricities should be calculated and entered as diaphragm-specific override eccentricity values in the analytical model.

The present verification checks only whether diaphragm-specific override eccentricity values are present in the model. The numerical values of eccentricity are not independently calculated or verified in this section.

Where diaphragm-specific eccentricity override values are available, only the maximum eccentricity identified in the X and Y directions is reported for quick reference and user verification. The engineer should independently verify the eccentricity values against the applicable code provisions and the centre of mass and centre of resistance at each floor level.

Where the model uses an eccentricity ratio option applicable to all diaphragms instead of diaphragm-specific override eccentricity values, the present version of the software will report a warning because this verification specifically checks for diaphragm-specific override values.

Response Spectrum Load Cases Considered for Eccentricity Verification

Sr. No.	Response Spectrum Load Case
1	RSX
2	RSY
3	RSX-DR
4	RSY-DR

⚠ WARNING: Diaphragm-specific torsional eccentricity override values are not available in both the X and Y directions.

64. ECCENTRICITY RATIO SUMMARY

This section presents the eccentricity and eccentricity ratio obtained from the Centre of Mass and Centre of Rigidity data extracted from the analytical model. The eccentricity is evaluated separately in the X and Y directions at each story, and the corresponding eccentricity ratios are obtained by comparing Ex with the plan dimension Lx and Ey with the plan dimension Ly, respectively.

The eccentricity ratio provides a quick indication of the relative eccentricity between the Centre of Mass and Centre of Rigidity with respect to the corresponding plan dimension of the structure. The calculated values are presented for each story, together with a summary identifying the maximum positive and maximum negative eccentricity ratios in the X and Y directions.

Interpretation of eccentricity sign: The Centre of Rigidity is considered as the reference point (zero position) for interpretation of the eccentricity. A positive eccentricity indicates that the Centre of Mass is located in the positive global coordinate direction relative to the Centre of Rigidity, whereas a negative eccentricity indicates that the Centre of Mass is located in the negative global coordinate direction relative to the Centre of Rigidity. The positive and negative signs therefore indicate the direction of eccentricity and do not represent positive or negative magnitude of eccentricity.

Interpretation of eccentricity sign: The Centre of Rigidity is considered as the reference point (zero position) for interpretation of the eccentricity. A positive eccentricity indicates that the Centre of Mass is located in the positive global coordinate direction relative to the Centre of Rigidity, whereas a negative eccentricity indicates that the Centre of Mass is located in the negative global coordinate direction relative to the Centre of Rigidity. The positive and negative signs therefore indicate the direction of eccentricity and do not represent positive or negative magnitude of eccentricity.

For this calculation, diaphragm definition is required in the analytical model. The Centre of Mass and Centre of Rigidity calculation option must also be enabled in the ETABS analysis options, and the analysis must be performed after enabling this option. The eccentricity and eccentricity ratio reported in this section are therefore dependent on the availability of the corresponding Centre of Mass and Centre of Rigidity data in the ETABS analytical database.

The results presented in this section are intended for quick engineering verification of the variation of eccentricity along the height of the structure. Significant or unusual eccentricity may warrant further review of the distribution of structural stiffness and the resulting Centre of Rigidity at the respective story levels.

Maximum and Minimum X-Direction Eccentricity Ratio

Condition	Story	Eccentricity Ex (m)	Plan Dimension Lx (m)	Ex/Lx
Maximum Positive	Story30	3.58	32.00	0.112

Maximum and Minimum Y-Direction Eccentricity Ratio

Condition	Story	Eccentricity Ey (m)	Plan Dimension Ly (m)	Ey/Ly
Maximum Negative	Story8	-7.52	32.00	-0.235

Story-wise Eccentricity Ratio Summary

Story	Eccentricity Ex (m)	Eccentricity Ey (m)	Lx (m)	Ly (m)	Ex/Lx	Ey/Ly
Story30	3.58	-5.83	32.00	32.00	0.112	-0.182
Story29	3.40	-5.62	32.00	32.00	0.106	-0.176
Story28	3.39	-5.72	32.00	32.00	0.106	-0.179
Story27	3.39	-5.82	32.00	32.00	0.106	-0.182
Story26	3.39	-5.93	32.00	32.00	0.106	-0.185
Story25	3.38	-6.05	32.00	32.00	0.106	-0.189

Story	Eccentricity Ex (m)	Eccentricity Ey (m)	Lx (m)	Ly (m)	Ex/Lx	Ey/Ly
Story24	3.38	-6.16	32.00	32.00	0.106	-0.193
Story23	3.37	-6.27	32.00	32.00	0.105	-0.196
Story22	3.37	-6.38	32.00	32.00	0.105	-0.199
Story21	3.37	-6.49	32.00	32.00	0.105	-0.203
Story20	3.36	-6.59	32.00	32.00	0.105	-0.206
Story19	3.36	-6.70	32.00	32.00	0.105	-0.209
Story18	3.35	-6.80	32.00	32.00	0.105	-0.213
Story17	3.34	-6.90	32.00	32.00	0.104	-0.216
Story16	3.34	-7.00	32.00	32.00	0.104	-0.219
Story15	3.33	-7.10	32.00	32.00	0.104	-0.222
Story14	3.32	-7.19	32.00	32.00	0.104	-0.225
Story13	3.31	-7.27	32.00	32.00	0.103	-0.227
Story12	3.29	-7.35	32.00	32.00	0.103	-0.230
Story11	3.28	-7.42	32.00	32.00	0.102	-0.232
Story10	3.26	-7.48	32.00	32.00	0.102	-0.234
Story9	3.23	-7.51	32.00	32.00	0.101	-0.235
Story8	3.20	-7.52	32.00	32.00	0.100	-0.235
Story7	3.16	-7.48	32.00	32.00	0.099	-0.234
Story6	3.10	-7.37	32.00	32.00	0.097	-0.230
Story5	3.01	-7.18	32.00	32.00	0.094	-0.224
Story4	2.90	-6.83	32.00	32.00	0.091	-0.214

Story	Eccentricity Ex (m)	Eccentricity Ey (m)	Lx (m)	Ly (m)	Ex/Lx	Ey/Ly
Story3	2.73	-6.29	32.00	32.00	0.085	-0.196
Story2	2.52	-5.48	32.00	32.00	0.079	-0.171
Story1	2.21	-4.28	32.00	32.00	0.069	-0.134

Information: The eccentricity and eccentricity ratio values presented above are extracted and calculated from the analytical model data and are provided for quick engineering verification of the variation of eccentricity along the height of the structure.

The maximum positive and maximum negative eccentricity ratios identify the critical eccentricity conditions obtained from the story-wise results. The reported values should be reviewed together with the Centre of Mass, Centre of Rigidity and structural stiffness distribution of the corresponding story levels.

Engineering review is recommended where significant or unusual eccentricity is observed, particularly where the eccentricity ratio varies substantially between adjacent stories or indicates potential stiffness irregularity.

Summary: The maximum absolute Ex/Lx ratio is 0.11 at Storey: Story30, while the maximum absolute Ey/Ly ratio is 0.23 at Storey: Story8.

---

65. RESPONSE SPECTRUM STOREY SHEAR SUMMARY

This section presents the Response Spectrum storey shear values obtained from the Story Forces data extracted from the analytical model. The results are reported for the governing Response Spectrum load cases identified separately in the X and Y directions.

The program identifies all Response Spectrum load cases available in the analytical model. The governing Response Spectrum load case in the X direction is identified based on the maximum base shear VX, while the governing Response Spectrum load case in the Y direction is identified based on the maximum base shear VY.

The cumulative storey shear values reported in this section are extracted directly from the ETABS Story Forces table. The absolute storey shear is derived from the cumulative storey shear values and represents the shear contribution associated with the respective storey level.

The absolute storey shear provides an additional representation of the lateral shear demand along the height of the structure and may assist the engineer in reviewing the distribution of lateral force between adjacent storeys.

Important Note: The cumulative storey shear values presented in this section are ETABS-derived results obtained from the Story Forces table. In certain analytical models, the cumulative storey shear values may not exactly match the corresponding support or base reaction values reported by ETABS.

Such differences may occur depending on the analytical modelling of foundations, raft or pile-cap levels, retaining walls, support conditions and the manner in which Response Spectrum results are reported by ETABS.

The results presented in this section are intended for user verification and reporting purposes only. No independent engineering verification of the Response Spectrum storey shear values is performed by this program.

Governing X-Direction Response Spectrum Load Case: RSX

Governing Y-Direction Response Spectrum Load Case: RSY

Response Spectrum Storey Shear Summary

Storey	X Cumulative Shear VX (kN)	X Absolute Shear VX (kN)	Y Cumulative Shear VY (kN)	Y Absolute Shear VY (kN)
Story30	753.3	753.3	1056.8	1056.8

Storey	X Cumulative Shear VX (kN)	X Absolute Shear VX (kN)	Y Cumulative Shear VY (kN)	Y Absolute Shear VY (kN)
Story29	1474.9	721.6	2015.9	959.1
Story28	2051.4	576.5	2746.5	730.6
Story27	2510.2	458.8	3299.9	553.4
Story26	2883.6	373.4	3721.0	421.1
Story25	3200.2	316.6	4042.5	321.5
Story24	3482.2	282.0	4290.7	248.2
Story23	3741.1	258.9	4488.8	198.1
Story22	3973.2	232.0	4657.7	168.9
Story21	4167.7	194.5	4814.9	157.2
Story20	4320.9	153.2	4971.0	156.1
Story19	4438.9	118.0	5128.5	157.5
Story18	4532.6	93.8	5287.8	159.3
Story17	4614.1	81.5	5449.8	162.0
Story16	4696.1	82.0	5612.5	162.8
Story15	4794.1	98.0	5774.5	162.0
Story14	4923.5	129.4	5939.5	164.9
Story13	5096.0	172.4	6111.5	172.1
Story12	5322.0	226.1	6293.2	181.7
Story11	5607.6	285.6	6491.1	197.9
Story10	5948.7	341.1	6713.8	222.7
Story9	6338.0	389.3	6964.3	250.5

Storey	X Cumulative Shear VX (kN)	X Absolute Shear VX (kN)	Y Cumulative Shear VY (kN)	Y Absolute Shear VY (kN)
Story8	6767.5	429.5	7243.3	279.0
Story7	7224.5	457.0	7552.2	308.9
Story6	7691.7	467.2	7884.1	331.9
Story5	8149.9	458.2	8222.5	338.4
Story4	8581.8	431.9	8555.2	332.7
Story3	8968.2	386.4	8867.0	311.9
Story2	9268.4	300.2	9111.3	244.3
Story1	9417.2	148.8	9227.1	115.8

Information: The cumulative storey shear values are extracted from the ETABS Story Forces table for the governing Response Spectrum load case in each principal direction.

The absolute storey shear values are derived from the corresponding cumulative storey shear values and are provided to assist in reviewing the distribution of lateral shear demand along the height of the structure.

The reported values are provided for user verification only. Engineering review is recommended where significant differences are observed between Story Forces results and support or base reaction results.

Summary: The maximum absolute cumulative storey shear in the X direction is 9417.2 kN at Storey: Story1, while the maximum absolute cumulative storey shear in the Y direction is 9227.1 kN at Storey: Story1.

66. SEISMIC DRIFT SUMMARY

The seismic drift values presented below are extracted from the ETABS analytical model for the individual seismic drift load cases defined in the model.

The program reports drift only for individual seismic load cases (not load combinations) where the corresponding load pattern is defined as Seismic (Drift) and the load pattern and load case have the same name; otherwise, the seismic drift summary will not be reported in this section.

The maximum story drift in the principal X and Y directions is reported along with the corresponding storey height and inter-storey displacement for quick user review.

This section is intended only for reporting and user verification. No independent verification or codal pass/fail check is performed by the program.

The reported drift values shall be reviewed by the user with reference to the applicable provisions of IS 1893 (Part 1)-2016 and IS 16700-2023.

Seismic Drift Summary

Direction	Storey	Load Case	Storey Height (m)	Inter-storey Displacement (mm)	Drift (Δ/H)
X	Story12	RSX-DR	3.50	3.02	0.000862
Y	Story13	RSY-DR	3.500	2.660	0.0008

Inter-storey displacement is derived from the reported drift and corresponding storey height.

The reported values are provided for quick user review only and do not constitute an independent codal verification.

The user shall verify the reported drift and inter-storey displacement against the applicable requirements of IS 1893 (Part 1)-2016 and IS 16700-2023.

Summary: The maximum seismic drift in the X direction is 0.0009 (Δ/H) at Story12 (RSX-DR), while the maximum seismic drift in the Y direction is 0.0008 (Δ/H) at Story13 (RSY-DR).

67. SCOPE AND LIMITATIONS

This report provides a quick automated review of selected aspects of the ETABS structural model. The information presented is intended only as preliminary guidance to help identify potential areas that may require further attention during model review. This tool does NOT replace a detailed engineering review of the structural model, and engineering judgement should always be exercised while interpreting the results.

The purpose of this tool is to assist engineers in identifying critical modelling aspects and to help focus attention on specific areas of the model. Only a limited set of checks are currently implemented, and the logic implemented in this tool will continue to be refined and expanded based on feedback received from practicing engineers.

Summary: This summary report provides a quick automated review of selected aspects of the ETABS structural model and highlights items requiring further attention. It is intended as preliminary guidance only and does not replace detailed engineering review and engineering judgement.

For suggestions, queries, or feedback, please contact: apps@sqveconsultants.com

Total processing time of modal verification tool : 4.4 seconds