RESEARCH ARTICLE / НАУЧНАЯ СТАТЬЯ

Structural analysis of high-rise building using ETABS and RSA software

Sophia A. Pechorskaya[1], Vera V. Galishnikova[2], Shishai B. Gebreslassie[1]*, Habte Y. Damir[1]

[1]*Peoples' Friendship University of Russia (RUDN University), 6 Miklukho-Maklaya St, Moscow, 117198, Russian Federation*
[2]*Moscow State University of Civil Engineering (National Research University), 26 Yaroslavskoye Shosse, Moscow, 129337, Russian Federation*
*shifu558@gmail.com

Abstract. There are numerous software applications available to analyze and design massive projects in short time. Most of the software have 2-dimensional and 3-dimensional tools, while various structural engineers employ the use of the 3D analysis and design tools in their day-to-day tasks because of its ease to operate and graphical user interface. These tools help in modeling, analyzing, and design of structures much more efficient. Despite the availability of numerous software products, there are confusions on the software to be used in the analysis and design of specific building structures. There is a need of studying the strength and weakness of some of these software tools to help structural engineers in the selection of the best application in their daily tasks. The aim of this study is to investigate the structural analysis of high-rise building with ETABS and RSA software and compare the influences of the structural analysis results from the two software in design. The comparison between the axial forces and moment from the results of ETABS and RSA software are presented. Case studies are considered to analyze the structure with the gravitational loads and lateral loads due to wind load by the two software applications. The case studies include a thirty-stories reinforced concrete building frame. The results of the analysis of the frame are compared and their difference is presented. From the analysis, the results show that, the moments and forces presented from RSA are bigger than that of ETABS.

Keywords: ETABS software, robot structural analysis software, wind load, high-rise building

Расчеты несущего каркаса высотного здания в программных комплексах ETABS и RSA

С.А. Печорская[1], В.В. Галишникова[2], Ш.Б. Гебреслассие[1]*, Й.Д. Хабте[1]

[1]*Российский университет дружбы народов, Российская Федерация, 117198, Москва, ул. Миклухо-Маклая, д. 6*
[2]*Московский государственный строительный университет, Российская Федерация, 129337, Москва, Ярославское шоссе, д. 26*
*shifu558@gmail.com

Аннотация. Существует множество расчетных программных комплексов, позволяющих быстро и эффективно выполнять расчеты и проектировать масштабные проекты. Большинство расчетных программ оснащено 2D- и 3D-инструмен-

Sophia A. Pechorskaya, teacher of the Department of Civil Engineering, Academy of Engineering, Ph.D.; ORCID iD: https://orcid.org/0000-0001-5933-5161.
Vera V. Galishnikova, Director of the International Department, Doctor of Technical Sciences, Professor; ORCID iD: https://orcid.org/0000-0003-2493-7255, Scopus ID: 55303553900, eLIBRARY SPIN-code: 2765-7069.
Shishai B. Gebreslassie, master student of the Department of Civil Engineering; ORCID iD: https://orcid.org/0000-0001-5683-829X, eLIBRARY SPIN-code: 4316-4841.
Habte Y. Damir, PhD student of the Department of Civil Engineering; ORCID iD: https://orcid.org/0000-0002-7275-6750.

Печорская Софья Александровна, преподаватель департамента строительства, Инженерная академия, ORCID iD: https://orcid.org/0000-0001-5933-5161.
Галишникова Вера Владимировна, директор международного отдела, доктор технических наук, профессор; ORCID iD: https://orcid.org/0000-0003-2493-7255, Scopus ID: 55303553900, eLIBRARY SPIN-код: 2765-7069.
Гебреслассие Шишай Берхане, магистрант департамента строительства, Инженерная академия; ORCID iD: https://orcid.org/0000-0001-5683-829X, eLIBRARY SPIN-код: 4316-4841.
Хабте Йоханнес Дамир, аспирант департамента строительства, Инженерная академия; ORCID iD: https://orcid.org/0000-0002-7275-6750.

тарием, который благодаря простоте управления и пользовательского графического интерфейса дает возможность конструкторам-проектировщикам в процессе повседневной проектной работы разрабатывать пространственные расчетные модели зданий и сооружений. Применение расчетных программных комплексов способствует повышению эффективности моделирования, анализа и проектирования строительных конструкций. Значительное разнообразие расчетных программ вызывает сложности при выборе программного обеспечения, наиболее подходящего для решения конкретных инженерных задач, возникающих при проектировании. Для правильного выбора расчетного программного комплекса необходимо иметь четкое представление об основных особенностях, достоинствах и недостатках представленных на рынке программных продуктов. Целью работы является исследование структурного анализа высотного здания с помощью программ ETABS и RSA. Представлено сравнение между силами стержня и моментом по результатам использования программ ETABS и RSA. Рассмотрены тематические исследования для анализа конструкции с гравитационными нагрузками и боковыми нагрузками из-за ветровой нагрузки двумя программными приложениями. Тематические исследования включают тридцатиэтажный железобетонный каркас здания. Сравниваются результаты анализа кадра и представлено их различие. Установлено, что моменты и силы, представленные RSA больше, чем полученные в ETABS.

Ключевые слова: программный комплекс ETABS, программный комплекс RSA, ветровая нагрузка, высотное строительства

Introduction

Creating tall structures first began as an expression of wealth, power and structural engineering advancement. However, with the rapid population growth and expanding urbanization, tall structures remain highly relevant for both commercial and residential construction. This demonstrates that high-rise buildings are profitable in terms of economics crisis and recession [1]. As indicated by Willis (2016), structural engineers generally consider skyscrapers with a minimum 1:10 or 1:12 ratio the width of the building's base to its height to be slender. However, many of the newest skyscrapers far exceed this ratio. For instance, '111 West 57th Street' tower in New York has a slenderness ratio of 1:24; this is more than twice as slender as what is required to be considered 'slender' [2–5].

As the height increases the rigidity and stability requirements become more important and they are often the dominant factors in design. Different types of structural systems are to be used to resist the effect of lateral loads on the buildings. They are rigid frame structures, braced frame structures, shear wall frame structures, outrigger systems, and tubular structures. Lateral load resisting systems are structural elements which resist seismic, wind and eccentric gravity loads [5]. Any high-rise building should be constructed only after being analyzed under wind loads and also seismic loads if required. A building as a shelter represents a physical division of the human habitat. Seismic force is the major cause for collapse of many high-rise structures. Seismic zone plays an influential role in the earthquake resistant design of building structures [6]. Generally, horizontal loads, such as wind loads and seismic loads, vibrate super high-rise building strongly, threatening its structural serviceability and safety [7]. In high rise buildings beam and column dimensions work out to be very large and reinforcement at the beam column joints are quite heavy which results in clogging at joints and to remove these kinds of practical problems we use shear walls as a key element to provide adequate stiffness [8; 9].

The exponential growth in computational power in recent years is continuously narrowing the industry academia gap by providing the cutting-edge research and technology to practicing engineers at their doorstep. As a result, the structural designers nowadays are equipped with far more aids and tools compared to a couple of decades ago. Moreover, recent advancements in nonlinear modeling techniques have also opened a whole new research area dealing with constructing computer models with close-to-real behaviors. With such a range of options available, the choice of modeling scheme and the analysis procedure for design decision making often becomes a matter of "the more the sweat; the more the reward" for designer [10–12]. With the advent of different structural software, in the present situations of construction industry, the buildings that are being constructed are gaining significance, in general, those with the best possible outcomes. Those software mainly used for structures like high-rise buildings, steel and concrete structures [13; 14]. Most researches and specifications conducted on the high-rise buildings have focused on the effects of seismic and wind load separately as it respectively pertains to the strength and serviceability requirements. The meteorological phenomena indicated that the atmospheric pressure varies vastly during earthquakes occurrence. As reported in series of literature strong winds were often accompanied by the occurrence of earthquakes [15; 16].

ANALYSIS AND DESIGN OF BUILDING STRUCTURES

Today, numerous design developments and enhancements arise which include structural design software which seeks to promote the capacity of a structural engineer to produce safe designs especially when battling with complex building structures. Most of the software applications have diverse and integrated functions, like modelling building structures of various materials, structural element analysis, geometrical error detections, and report output and graphical files [17].

ETABS (Extended Three-Dimensional Analysis of Building Structure) is an integrated building design software developed by Computers and Structures Inc., also known as CSI. It is one of the most powerful software in structural engineering in the design of high-rise buildings. It is used worldwide due to its features in rapid modeling of framing systems and in analyzing large and complicated building structures. A big advantage ETABS offers is the shear wall design and offers design of composite beam floors and auto-meshing of floors [18]. Autodesk Robot Structural Analysis (RSA) Professional software provides structural engineers with advanced building simulation and analysis capabilities for large, complex structures. It has BIM-integrated workflows by having the ability to exchange data with Revit and other design tools. Robot can be used for wind load simulation, static, modal and non-linear analysis of structures, finite element auto-meshing, structural design, among others [19]. The aim of this paper is to investigate the structural analysis of high-rise building with ETABS and RSA software and to study the influence of the structural analysis results from the two software in design. Based on the results of structural analysis on the column elements and the link-beam elements to give a clear information that when we can use those software from the practical point of view in design industries.

Materials and methods

Structural BIM, being a subset of BIM, can be the fundamental information for structural engineers, such as geometry, material properties, sectional properties, loads, load combinations, boundary conditions, and so on. Accordingly structural BIM can be utilized for structural analysis for the generation of drawings and reports. Structural BIM provides for any engineering project a flexible environment of interoperability and collaboration for relevant areas. Many companies invested in BIM technology. In this field Autodesk is consider as the market leader. Autodesk Revit gives direct link with numerous of the common structural analysis software such, ETABS, SAP2000, SAFE, PROKON and Autodesk Robot structural analysis (RSA) which is fully compatible with Autodesk Revit, and seamless interaction is permitted between the model of BIM and structural model [20; 21].

There are some specific points that demonstrate in which the two software differ from each other. For example: in ETABS, when we choose a membrane element, applying the area loads to the slab, all loading is transferred as tributary to the frame object. ETABS will then show the tributary uniform line load to each beam (must dump 100% of the loads to the support elements) and it does not consider the relative stiffness of the structural elements. But in RSA the concrete slab which is modeled as shell element has out-of-plane stiffness and will share some of the load carrying capacity through flexure of the slab that means it considers the relative stiffness of the structural elements. In ETABS software shell objects are meshed automatically into elements needed for analysis. But in RSA software, different meshing options are available for the shell objects. ETABS software automatically generate and apply wind loads to diaphragms or to walls and frames based on the various design codes. While in RSA the wind loads simulation option allows to simulate a wind flow around the structure and generate wind loads automatically. The program uses the flow simulation results to trigger the automatic load generation at the best moment. ETABS has many different link elements available for users to accurately represent the behavior of a structure. Link elements include linear, multi-linear elastic, multi-linear plastic, gaps, hooks, dampers, friction isolators and rubber isolators. RSA, in contrast, doesn't have those link elements. In ETABS the link-beam is considered as a bar elements with two nodes, but in RSA it is considered as shell and meshed that has more than two nodes.

A three-by-three bays of thirty stories high-rise building was considered in this study. A 3D model was prepared in Autodesk Revit structural, and then the model was exported to Robot structural analysis for the structural analysis. To understand the integration process between Revit and structural analysis software and on how to obtain the best benefits from this process in terms of time and effort, it is necessary before modeling to understand the concept of physical and analytical representations of structure. The physical model (Figure 1) is the true model that contains all information, and this is what was modeled inside Revit and its used for coordination as well as documentation, while analytical model is transfers just the information needed by structural engineer, and then it was modeled automatically in the background of physical model inside Revit. The analytical model is what the structural analysis software deals with it. It is used for structural analysis and design where structural loads, load combinations and boundary conditions that can be easily added with in Revit, that model is only exported for structural analysis to RSA.
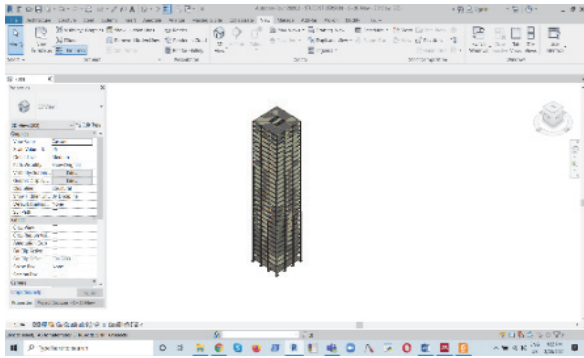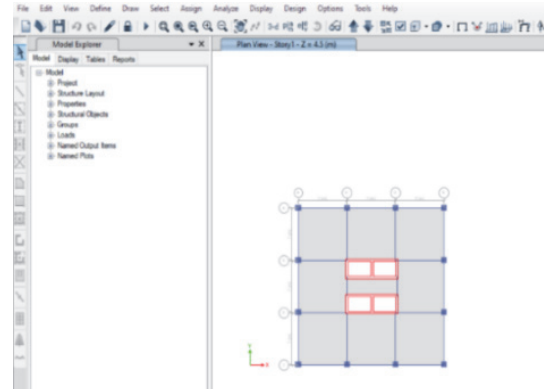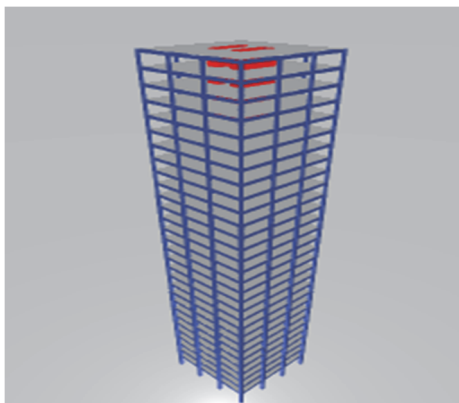
Figure 1. 3D model in Revit
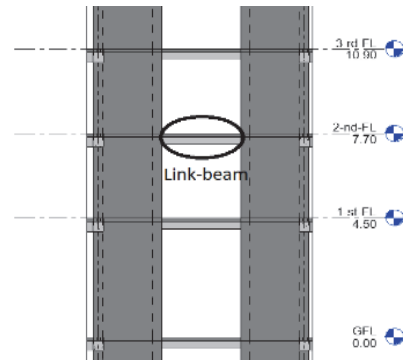
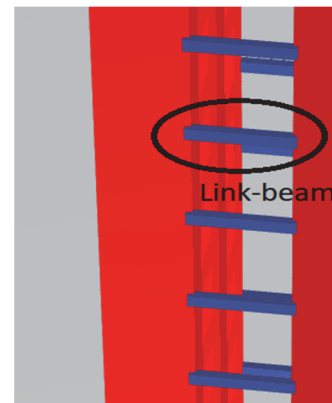**Figure 2.** Link-beam in Revit
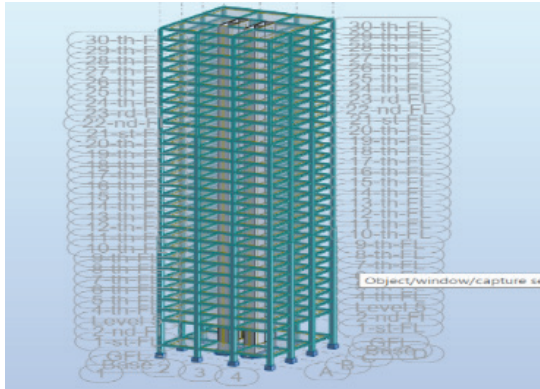
Figure 3. 3D model in Robot

Figure 4. Floor plan in ETABS

Figure 5. 3D model in ETABS

Figure 6. Link-beam in ETABS

Exporting the model to Robot Structural Analysis was performed using Revit Extension. Then, the model was directly opened at RSA as in Figure 3. Another with the same geometrical shape of 3D model was also prepared in ETABS software and structurally analyzed. The floor plans in ETABS (Figure 4) and the 3D model (Figure 5) has the same geometrical shape and characteristics, the only difference is that they are modeled in two different software and structural analyzed in two different software.

These building models from the two software are characterized by reinforced concrete structure moment resisting frame with provision of shear wall at the central lift core connected each other with a link-beam. A square plan (Figure 4) with 21×21 m was considered with 3 bays in longitudinal direction and 3 bays in transverse direction. The total heights is 97.30 m, with equal story heights of 3.20 m except the first storey, which is 4.5 m high. A link-beam (Figures 2 and 6) with (450×700) mm throughout the building height and that of column with (1500×1500) mm for the storeys (1–10), (900×1000) mm for the storeys (11–20) and (600×600) mm for the storeys (21–30) were provided. Slab with thickness of 250 mm and shear wall with thickness of 300 mm were taken.

The frame was loaded with gravity load and lateral (wind) loads while the end supports of columns are considered as pinned. The un-factored dead loads including self-weight and live loads are taken as 10.30 and 5.0 kN/m^2 respectively. The building has been designed in accordance with ACI-381-14. The cross-section capacities have been computed by considering a characteristic strength of 24 N/mm^2 for concrete and a characteristic yield strength of 420 N/mm^2 for both longitudinal and transverse steel. All the models were analyzed for gravity loads and lateral (wind) loads as per ACI-381-14 standards and to have consistency, the results were taken from the same load combination. Design data taken for the wind load pattern according to ASCE-7-16, are: wind speed = 90 mph, exposure type – gound elevation factor – 1.0, topographical factor – 1.0, gust factor – 0.85, directionality factor – 0.85, windward coefficient C_{pw} – 0.80 and leeward coefficient .C_{pl} – 0.50.

Results and discussion

Comparison of maximum bending moment and axial force in the columns. In complex and high-rise building design, usually the design process is done by structural analysis and design software. Especially in construction industrial company using those software is a common way of design. The design process is started after completing the analysis. From that point of view we can understand that the structural analysis results govern the amount of steel area required (bar diameter, number and spacing between bars) with the sufficient cross-sectional size of the structural elements (column, beams, walls and slabs). So the accuracy of the analysis results are very important for our structural design.

The case study considered the maximum force values, from the output results of same load combination used in both software, as per ACI381-14. The magnitude of maximum axial force calculated using ETABS and RSA software were compared in the edge columns of the building. A comparison was made by taken magnitude of the columns force for the base edge columns, middle story edge columns and top story edge columns. As shown in Table 1 the results of RSA in the vertical forces (axial forces) are larger, at the base, at the middle and at the top of the story, than the results from ETABS software.

Table 1

Comparative result of axial forces from ETABS and RSA software for the edge columns

Height, m	Axial force, kN		Difference in results, %
	ETABS results	*RSA results*	
At base	15379.31	16724.72	8.75
At the middle (49.3 m)	6917.86	7381.78	6.71
At the top (97.3 m)	368.36	390.90	6.12

Similarly for the corner columns, as displayed in Table 2, the axial forces of RSA are larger, at the base, at the middle and at the top of the storey, than the results of ETABS software. In all the axial forces of the structural system, RSA results are bigger than ETABS results for the different heights of the building columns. However, in both tables the relative differences in the results decrease with increasing level height of the storeys.

Table 2

Comparative result of moments from ETABS and RSA software for the corner columns

Height, m	Axial force, kN		Difference in results, %
	ETABS results	*RSA results*	
At base	11385.14	12427.75	9.24
At the middle (49.3 m)	4793.44	5087.07	6.13
At the top (97.3 m)	221.81	235.03	5.96

Comparison of maximum bending moment in the link-beams. In high-rise building design, the main purpose of the link-beam is a lateral force resistant component of a structure. Link-beam is mainly used effectively when two shear wall or any other elements that are used to withstand combined lateral loads. This consequently decrease the overturning effects and improve overall stiffness of a system. From that points we can understand that the link-beam has a great role in the stiffness of the structure, so the analysis results are necessary for designing the link-beams with its capacity that can resist the proposed loads. The study again, considered the maximum

moment values, from the output results of same load combinations used in both software, as per ACI381-14. A comparison was made among the magnitudes of the link-beams moment and shear force for the base storey, middle storey and top storey. As summarized in Tables 3 and 4 the results of RSA in the link-beams moments and shear forces are bigger than the results from ETABS software. In those structural elements, the differences in percentage are a bit bigger than the differences observed earlier in the column elements. The reason for that differences in the results are, in ETABS the link-beam is consider as a bar elements with tow nodes, but in RSA it is consider as a mesh shell elements with more than two nodes.

Table 3

Comparative result of shear force from ETABS and RSA software for the link-beams

Height, m	Shear force, kN		Difference in results, %
	ETABS results	*RSA results*	
At base	42.72	48.86	14.37
At the middle (49.3 m)	45.64	50.98	11.70
At the top (97.30 m)	38.53	42.61	10.59

Table 4

Comparative result of moment from ETABS and RSA software for the link-beams

Height, m	Moment, kN·m		Difference in results, %
	ETABS results	*RSA results*	
At base	32.36	41.68	28.80
At the middle (49.3 m)	37.25	44.79	20.24
At the top (97.30 m)	31.53	37.81	19.96

Conclusion

This paper presents the results of the structural analysis of high-rise reinforced concrete building with the two software programs. Considering the structural 3D modelling from Revit exporting to Robot structural analysis for the analysis, and the same structure modeled in ETABS and structurally analyzed in ETABS. From those results, it is concluded that, the results from RSA is bigger than the results from ETABS software. Although all the analyses are generated based on the same inputs and with the same design codes, the results of the moments and forces are found to be different. It is observed that from the difference between the two software's results, nonetheless how small it may seem, can still give difference in the design for the amount of reinforcement and the cross-sectional size of the structural elements. From this we can concluded that those differrences are most likely from the way the two software's considered different way of generating the results. RSA is a finite element, its solution is based on generating a finite element mesh and distributing loads on nodes of this mesh in which relative stiffness of the structural elements are considered, in contrast, in ETABS the floor is treated as diaphragm (rigid connection of nodes for the slabs) in which the relative stiffness of the structural elements are not considered. In ETABS the link-beam is considered as a bar elements with two nodes, but in RSA it is considered as a meshed shell elements with more than two nodes. In addition to that also the way of meshing, that is automatically meshing in ETABS, but different options of meshing in RSA, can be affected the final results.

Finally, this paper recommends further comparative study of those software to investigate overall design output differences and material cost impression.

References

1. Shuvalova E. High-rise construction in Russia: Asian way vs. Middle-Eastern way. CTBUH 2015 New York Conference. 2015:168–175.

2. Walsh P., Saleh A., Far H. Evaluation of structural systems in slender high-rise buildings. *Aust. J. Struct. Eng.* 2018;19(2):105–117. http://dx.doi.org/10.1080/13287982.2018.1449597

3. Lande P.S., Takale P. Analysis of high rise building with transfer floor. *Int. Res. J. Eng. Technol.* 2018;5:2483–2488.

         ANALYSIS AND DESIGN OF BUILDING STRUCTURES

4. Saleem H., Shrivastava L.P. A Comparative Study on high rise building for various geometrical shapes subjected to wind load of RCC & composite structure using ETABS. *IRJET*. 2019:6(12):1553–1558.

5. Kawade M.P., Bangde V.S., Sawai G.H. Seismic analysis of tall building with central core as tube structure. *IJAEM*. 2020;2(6):300–310. http://dx.doi.org/10.35629/5252-0206300310

6. Lokesh Nishanth C.H., Sai Swaroop Y., Jagarapu D.C.K., Jogi P.K. Analysis and design of commercial building with different slab arrangements using ETABS. *Mater. Today Proc.* 2020;33:700–704. http://dx.doi.org/10.1016/j.matpr.2020.05.823

7. Lu Z., He X., Zhou V. Performance-based seismic analysis on a super high-rise building with improved viscously damped outrigger system. *Struct. Control Heal. Monit.* 2018;25(8):1–21. http://dx.doi.org/10.1002/stc.2190

8. Akhil Ahamad S., Pratap K.V. Dynamic analysis of G + 20 multi storied building by using shear walls in various locations for different seismic zones by using ETABS. *Mater. Today Proc.* 2021;43(2):1043–1048, http://dx.doi.org/10.1016/j.matpr.2020.08.014

9. Secer M., Zamani A., Isler Y. A practical compensation method for differential column shortenings in high-rise reinforced concrete buildings. *Period. Polytech. Civ. Eng.* 2020;65(1):242–254. http://dx.doi.org/10.3311/PPci.16028

10. Najam F.A., Warnitchai P. A modified response spectrum analysis procedure to determine nonlinear seismic demands of high-rise buildings with shear walls. *Struct. Des. Tall Spec. Build.* 2018;27(1):1–19. http://dx.doi.org/10.1002/tal.1409

11. Karrar W.S., Shyama A.M., Jassim M. High-rise building wind analysis using computational fluid dynamics and dynamic analysis using etabs program. *Int. J. Emerg. Trends Eng. Res.* 2020;8(7):3994–4012. http://dx.doi.org/10.30534/ijeter/2020/172872020

12. Ren X., Fan W., Li J., Chen J. Building information model-based finite element analysis of high-rise building community subjected to extreme earthquakes. *Adv. Struct. Eng.* 2019;22(4):971–981. http://dx.doi.org/10.1177/1369433218780484

13. Chandrasekhar Reddy K., Lalith Kumar G. Seismic analysis of high-rise buildings (G+30) by using ETABS. *Int. J. Tech. Innov. Mod. Eng. Sci.* 2019;5(3):174–181.

14. Kumar A., Kushwaha N. A review paper on progressive collapse assessment of asymmetric high rise building and its modelling using ETABS software. *Int. J. Res. Appl. Sci. Eng. Technol.* 2020;8(IX):68–73. http://dx.doi.org/10.22214/ijraset.2020.31310

15. Zheng X.W., Li H.N., Bin Yang H.N., Li G., Huo L.S., Liu Y. Damage risk assessment of a high-rise building against multihazard of earthquake and strong wind with recorded data. *Eng. Struct.* 2019;200:109697. http://dx.doi.org/10.1016/j.engstruct.2019.109697

16. Tan S., Moinuddin K. Systematic review of human and organizational risks for probabilistic risk analysis in high-rise buildings. *Reliab. Eng. Syst. Saf.* 2019;188:233–250. http://dx.doi.org/10.1016/j.ress.2019.03.012

17. Odeyemi S.O., Akinpelu M.A., Abdulwahab R., Ibitoye B.A., Amoo A.I. Evaluation of selected software packages for structural engineering works. *ABUAD J. Eng. Res. Dev.* 202;3(2):133–141.

18. *ETABS 2016: software verification examples*. Computer and Structure, Inc., 2017.

19. Marsh K. *Autodesk robot structural analysis professional 2015: essentials.* Marsh API LLC; 2014.

20. Hadi A.S., Abd A.M., Mohammed Mahmood. Integrity of Revit with structural analysis softwares. *IOP Conf. Series: Materials Science and Engineering*. 2012;1076:012119. http://dx.doi.org/10.1088/1757-899X/1076/1/012119

21. Kamble T.R., Awchat G.D. Seismic analysis and design of multi-storied RC building using STAAD Pro and ETABS. 2018;3(8)4–7.