Civil Engineering Beyond Limits 4 (2023) 1891
Civil Engineering Beyond Limits
journal home: www.acapublishing.com/journals/1/cebel
Research Article

Behavior of Steel Sections with Different Geometries Under Impact Load

Elif Ağcakoca*,[1]
[1]Sakarya University, Civil Engineering Department

Keywords

Impact Load,
Steel section,
Finite element analysis,
Plastic deformation,
Weak axis.

Abstract

In this article, experimental and finite element analysis studies were carried out at different impact heights of steel beams with different geometric cross-sections. The behavior of elements with varying cross-sectional properties was examined when subjected to hammer loads dropped from different heights. In the experimental study, boundary conditions were established using a 2000 mm long rectangular cross-section beam element with pin and roller supports. The behavior of the steel beam under impact load was analyzed using Abaqus Explicit three-dimensional finite element models. Subsequently, a validation process ensuring convergence was performed before conducting a parametric study. 6 finite element models were constructed for the parametric study, comprising circular and ellipse-section steel elements with approximately equivalent unit weights. The cross-sectional geometry and hammer height were varied while keeping the boundary conditions consistent with the samples examined. As the height of the hammer drop increased, the displacement value, Von-Mises stress, and PEEQ (plastic equivalent strain) values of the elements also increased. When transitioning the cross-sectional shape of the beam element from a circle to an ellipse at a constant hammer height, it resulted in a decrease in the displacement value, Von-Mises stress, and PEEQ values. The results indicate that the maximum displacement, highest stress, and PEEQ value are observed in the E#2000 scenario. Similarly, the smallest displacement, lowest stress, and PEEQ value are exhibited in the C#1400 case.

1. Introduction

Steel structures serve as a vital solution for various openings in industrial facilities and remain a preferred choice in constructing high-rise buildings. When comparing steel structures to reinforced concrete ones, their principal advantages lie in being lighter, less susceptible to weather conditions during construction, and offering faster and more cost-effective construction processes.

Throughout their lifespan, buildings in our country can face static loads and particularly dynamic forces such as earthquake impacts. Designing and assembling structures under the influence of static and dynamic loads must adhere to current regulations. Additionally, due to various purposes of building usage or contemporary concerns such as increased terrorist threats, structures may encounter other dynamic effects.

Among these dynamic effects, the impact load's significance within a structure's economic lifespan becomes crucial for ensuring structural safety. Impact loads exerted on structures have the potential to cause severe damage, leading to disproportionate partial or complete collapses. Consequently, studying how a structure or its load-bearing elements respond to such impact loads is paramount.

These assessments, especially when calculating strategically important structures or during the initial design stages of bridge column bodies susceptible to impact effects, are crucial. Therefore, understanding the behavior of structural elements under impact loads holds significant importance in civil engineering. The behavior of an element under impact load depends on factors such as the element's hardness, the speed and mass of the impacting device, the height of the falling object, the total energy of the impacting object, and the boundary conditions of the elements. Hence, numerous factors play a crucial role in the design and analysis processes of structures. Numerical studies have gained significance in examining the influence of these diverse variables on the dynamic behavior of structures. In the literature, various test methods and design parameters are used to determine the behavior of elements under dynamic loads [1,2].

However, no specific regulations have been implemented in our country regarding impact loading. Impact loading can be categorized into two types based on the speed at which the hammer hits the sample: high-speed [3,4] and low-speed [5] impact loading. Studies have investigated the response of reinforced concrete and composite beams [6,7], steel and composite columns [8,9], and slabs [10,5] under low-speed impact effects, utilizing both experimental and numerical methods. Additionally, research has delved into the impact behavior of steel materials with different qualities or the behavior of reinforcement-free concrete elements [11,12].

While much of the research on steel sections has concentrated on circular [13,14,15,16,17] and rectangular [18] hollow sections, there remains a shortage of comprehensive studies on the impact loading of hollow elliptical geometry profiles on the weak axis. Furthermore, contemporary technological methods are employed to enhance structural performance under dynamic loads. Strengthening with fiber elements or creating composite elements has shown promise in reducing the impact effect, particularly when examining element behavior under impact loads.

One critical issue observed during impact loading is the formation of plastic deformation zones in hollow steel sections. This plastic deformation significantly affects the carrying capacity and bending strength of the section, potentially leading to collapse. Consequently, various studies have investigated methods to increase bearing capacity and reduce plastic notching under impact load. Experiments involving filling steel pipe sections with concrete (concrete-filled steel pipes) or mortar (double-walled steel pipe-shaped elements filled with mortar) have demonstrated a reduction in local plastic deformation in

*Corresponding Author: elifd@sakarya.edu.tr
Received 12 Dec 2023; Revised 30 Dec 2023; Accepted 30 Dec 2023
https://doi.org/10.36937/cebel.2023.1891

the impact region and overall deformation throughout the element [19,20,13].

The concept of moment of inertia reflects structural elements resistance to change when subjected to a load. Therefore, during the design process, considering both strong and weak axes is crucial to ensure optimal performance against these forces. Previous research holds significant importance, particularly in understanding structural behavior.

The aim of this study is to investigate the behavior of elements with identical weights but different cross-sectional features, such as circles and ellipses, under the influence of impact forces applied at various heights. Conducting more research in these areas can identify structural deficiencies by considering different risk scenarios during the design phase, even prior to assembly.

2. Test specimens and properties

The experiments were conducted at the Laboratory of the Civil Engineering Department of Sakarya University. Within the impact system, a hammer was precisely directed to the midpoint of a beam, strategically positioned on fixed and movable supports. Furthermore, to mitigate any out-of-plane movement, supports were strategically situated at the abutments. Preceding each test, specialized apparatus on sliding rails was systematically utilized to minimize potential friction encountered by the hammer. The experiments involved dropping a rectangular-shaped hammer from varying heights of 1400, 1600, and 2000 mm.

The data retrieved from the sample used for verification in the study remained consistent throughout the experimentation process and was securely mounted on the ground. The hammer geometry employed in the validation sample possessed a rectangular cross-section, as visually depicted in Figure 1. Crafted from S355 steel, the hammer boasted a fixed weight of 75 kg. For the finite element study, neither the geometry nor the weight of the hammer underwent alteration; solely the height was modified.

Detailed measurements and a schematic representation of the experimental setup are depicted in Figure 1. The reactive force generated during impact was meticulously gauged using two distinct load cells: one strategically positioned beneath the fixed support, and the other meticulously placed beneath the sliding support. The geometry of the hammer used for the verification specimen was rectangular, as exemplified in Figure 1, composed of S355 steel, and weighed 75 kg. In the context of the finite element analysis, the geometry and weight of the hammer remained unaltered; only the height of the hammer was subjected to alteration.

The details of the measurements and schematic representation of the experimental setup are presented in Figure 1. The reaction force during impact was measured using two separate load cells: one positioned under the fixed support and the other beneath the sliding support. The geometry of the hammer used for the verification specimen was rectangular, as shown in Figure 1, made of S355 steel, and weighed 75 kg. For the finite element analysis, the geometry and weight of the hammer remained unchanged; solely the height of the hammer was altered. In the finite element section of the study, parametric studies were carried out on 6 elements with two different geometries in circular and elliptical sections and the analysis results were evaluated. The diameter of the circular section element is 50 mm, the short axis of the element with ellipse geometry is 71 mm and the long axis is 162 mm wide. The element length is chosen as fixed and is 2000 mm. The rectangular-shaped hammer was dropped freely from different heights, such as 1400, 1600, and 2000 mm. The beam with circular and ellipse cross-sections examined in the finite element study was chosen to have the same weight. In the finite element study of the element with ellipse cross-section; In order to take into account, the most critical situation, an examination was made to apply the impact load on the weak axis. As shown in the illustration, the beam's right support acted as a pin, limiting all three-dimensional translations while permitting restricted out-of-plane rotations and unrestricted in-plane rotation. Conversely, the left support functioned as a roller, restricting vertical and out-of-plane translations but allowing free in-plane translation. Both the pin and roller support shared the same rotational degree of freedom.

3. Finite element modeling verification

Finite Element Method (FEM) is a widely employed mathematical approach for analyzing the behavior of complex structures. In this method, a structure or system is discretized into mathematically manageable finite elements with specific properties. The accuracy of this discretization process and the convergence property of finite element analyses are critically important in terms of accurately reflecting the actual behavior of the analyzed structure. As the number of finite elements increases, the obtained results converge closer to the real values. A three-dimensional finite element model was developed by using the results obtained in the experimental study in creating the finite element model. Steel profiles, hammer head geometry, supports are modeled as seen in Figure 2. The utilized software for performing 3D nonlinear finite element analysis was ABAQUS/Explicit [21]. During the modeling phase, the steel components were represented using the eight-node brick element with reduced integration (C3D8R) element type. The impactors were simulated as rigid bodies, also utilizing the C3D8R element.

For verification of the finite element model; A previously tested steel test sample with rectangular geometry was used. The hammer, which has a rectangular geometry, was allowed to fall freely with a weight of 0.75 kN onto the beam element with a fixed and sliding support from a height of 1400 mm [22]. In the verification study, the displacement-time graph obtained from the R#EXP sample was used in the verification of the ABAQUS finite element study. The previous finite element study was named R#EXP in Figure 3. For this article, the finite element model has been updated and named R#EXP Revise in Figure 4. While the maximum displacement value obtained from the previous study remained the same, a better validation of the residual displacement value was made in the Revised model. While the maximum displacement value obtained from the experimental study was measured as 30.1mm, the maximum displacement obtained as a result of the finite element was found to be 31.2mm. The difference between the experimental and finite element at the maximum displacement value was verified as 3.7%. In addition, while the residual displacement value obtained from the experimental study was measured as 13.5mm, the maximum displacement obtained as a result of the finite element was found to be 17.9. While in the previous study, there was an 88.1% difference in residual displacement between R#EXP and R#FEM experiment [22], the difference between R#EXP-Revise and R#FEM was verified as 4.4% for the residual displacement value. In the previous study, the ideal finite element mesh density could not be determined. Therefore, verification was achieved with a very large difference in residual displacement value. In addition, since the maximum displacement value was examined in the previous study, the residual displacement value is not important for that study. In this study, the finite element mesh density of the test sample and hammer head was selected as 10x10mm. The finite element mesh density is given in Figure 2. By finding the ideal value of the finite element mesh density, the data obtained with the finite element had sufficient convergence with the data obtained as a result of the experiment. When the height of the hammer dropped on the beam elements increases, the stress value in the element increases. It also causes different vibrations to occur in the element. Therefore, in many studies in the literature, the effect of changing hammer height on element behavior has been examined. Nasery in his study; He stated that the height of the hammer fall is proportional to the deformations occurring in the element [22]. Also, in a different work by Nasery; In case of using a hammer with different geometric heads, increasing the hammer height increased the deformation and displacement value of the element [23]. One of the important factors affecting the behavior of reinforced concrete and steel elements under impact load is the impact geometry used to apply the impact load in Table 1. The damage occurring in the element when rectangular hollow section steel beams are loaded from the weak axis under impact loads was examined. It has been stated that the formation of the hammer head in triangular, rectangular and circular shapes is effective on the displacement of the element. Additionally, in the study, it was observed that the most displacement and damage was caused by the triangular hammer head [23].

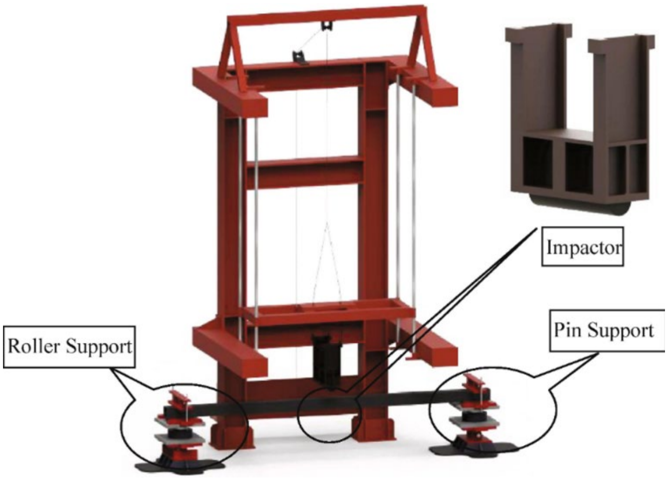


Figure 1. A three-dimensional view illustrating the experimental setup [22]

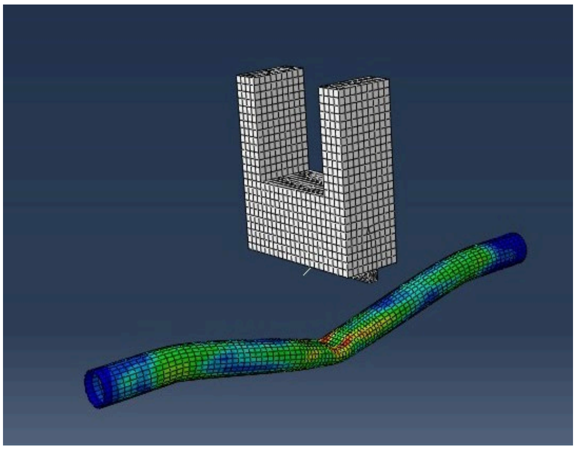


Figure 2. Finite element mesh density of hammer and test specimen

Changes in the impact geometry have significant effects on the deformation distribution in the elements and the damage that occurs in the structural elements as a result of the impact load [24]. It has been noteced that the least displacement and damage is caused by the semicircular hammer head [25].

Table 1. The characteristics of the specimen used in the validation study

Specimen code	Specimen Dimensions (mm)	Drop height (mm)	Impactor Weight (kN)	Head geometry shape
R#FEM	120×80×2000	2000	0.75	Rectangular
R#EXP	120×80×2000	2000	0.75	Rectangular

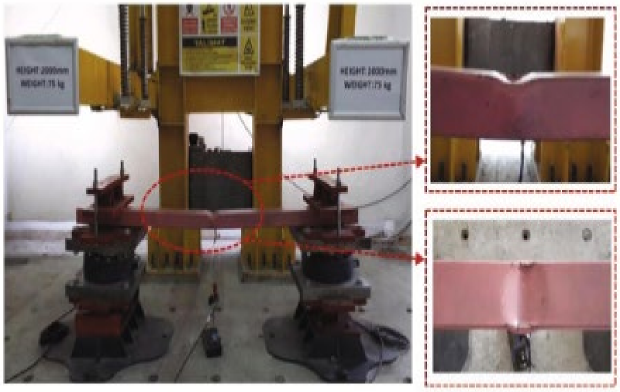


Figure 3. The post-experiment image of the R#EXP specimen [22]

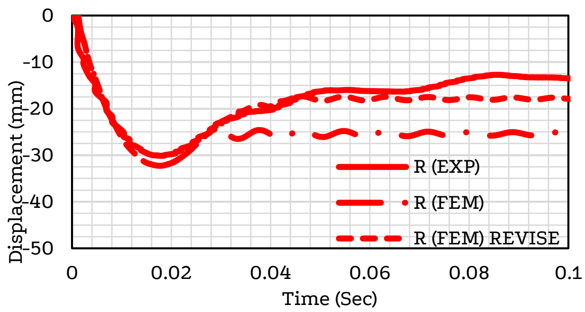


Figure 4. Validation graph of displacement-time for R#EXP, R#FEM and R#FEM REVISE specimens [22]

4. Parametric Study and Discussions

Parametric study was carried out after ensuring good agreement between the experimental and finite element (FE) models of the rectangular cross-section sample. The maximum displacement and residual displacement values in the Displacement-Time graph obtained from the experimental study had sufficient convergence, so parametric study was started. In the parametric study, a rectangular cap was left to fall freely on steel elements with circular and ellipse geometry. While selecting the cross-sectional dimensions of the steel profile with ellipse and circular cross-sections, care was taken to ensure that they had the same weight. In the parametric study, a hammer with rectangular geometry was allowed to fall freely onto a steel sample with a circular cross-section from a height of 1400, 1600, 2000 mm. A steel sample with an ellipse cross-section of the same weight was left to fall freely with a rectangular geometry hammer from a height of 1400, 1600, 2000 mm. In the parametric study, displacement-time graphs of the samples named C#1400, C#1600 and C#2000 were obtained under impact loading at 1400, 1600 and 2000 mm drop heights. When the height of the hammer dropped on the beam elements increases, the stress value in the element increases. It also causes different vibrations to occur in the element. Therefore, in many studies in the literature, the effect of changing hammer height on element behavior has been examined. Nasery in his study; He stated that the height of the hammer fall is proportional to the deformations occurring in the element [22]. Also, in a different work by Nasery; In case of using a hammer with different geometric heads, increasing the hammer height increased the deformation and displacement value of the element [23]. At the same time, the displacement-time graph of the samples named E#1400, E#1600 and E#2000, which have ellipse geometry determined to have the same weight, was obtained. In both series, deformation occurred in the -Z axis direction of the hollow ellipse and circular cross-section steel profiles. Plastic deflection started from the impact area and continued to the bottom surface of the steel beam. In addition, the damage in the area where the impact is applied is greater than in other parts of the beam.

Table 2. Sample characteristics in parametric study

Test series	Specimen code	Specimen Dimension (mm)	Drop height (mm)	Impactor Weight (kN)	Head geometry shape
	C#1400	120	1400	0.75	Rectangle
I	C#1600	120	1600	0.75	Rectangle
	C#2000	120	2000	0.75	Rectangle
	E#1400	162x72	1400	0.75	Rectangle
II	E#1600	162x72	1600	0.75	Rectangle
	E#2000	162x72	2000	0.75	Rectangle

Analysis and optimization of complex structures have become crucial in the fields of engineering and science. At this juncture, the role of parametric studies holds significant importance. Parametric analyses are employed to comprehend how systems or models behave under various variable parameters. This method allows for understanding the effects of different parameters on the performance of systems or models, thereby influencing the design process. Parametric studies are essential for determining the sensitivity of a system and assessing potential risks. Parametric analyses are used to foresee how a system might respond under different scenarios and to make comparisons among various alternatives. In the paremetric study, circle and ellipse elements with the same weight and different geometric properties were examined in Table 2. The impact load on the element with ellipse geometry is in the weak axis direction of the element. Inertia force is a force that resists a change in velocity or motion of an object. Samples used in parametric studies; For elements with circular and ellipse geometry, the wall thickness of the circle cross-section was chosen as 5 mm, and the wall thickness of the element with ellipse geometry was chosen as 4.78 mm. It has approximately the same unit weight. In determining the geometric properties of the sections to be used in the parametric study; The capacity of the computer on which the finite element model was solved and the solution time were effective. Impact effect occurs when an object is influenced by a force for a specific duration When the displacement-time graph of the C#1400, C#1600 and C#2000 samples consisting of a circular cross-section steel profile is examined; When the falling height of the hammer increased, the displacement value in the sample increased. Likewise, when the displacement-time graph of E#1400, E#1600 and E#2000 samples consisting of ellipse cross-section steel profiles is examined; When the falling height of the hammer increased, the displacement value in the sample increased. As a result of the hammer falling from the same height on samples with different cross-sections; The displacement value of C#1400 and E#1400 elements increased by 45%. Additionally, as a result of the hammer falling from the same height on samples with different cross-sections; The displacement value of C#1600 and E#1600 elements increased by 48%. Finally, as a result of the hammer falling from the same height on samples with different cross-sections; The displacement value of C#2000 and E#2000 elements increased by 34% in Figure 5.

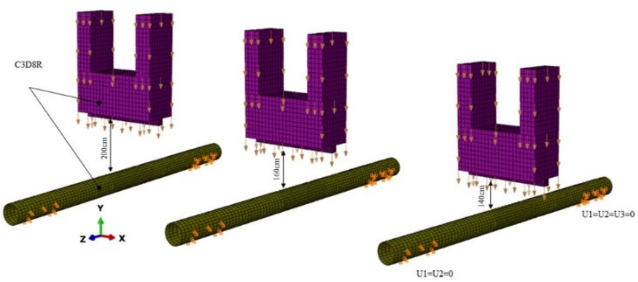
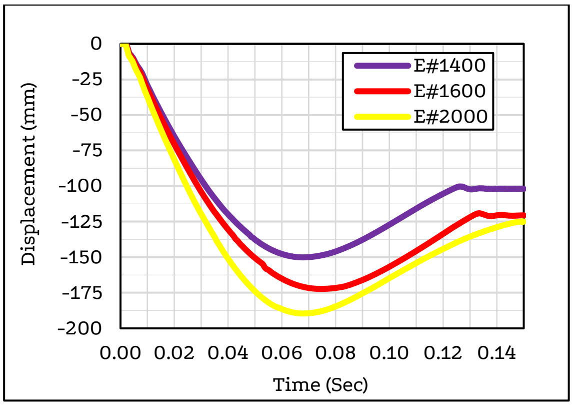
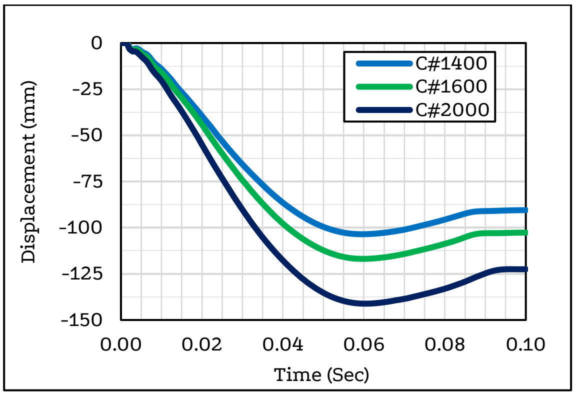
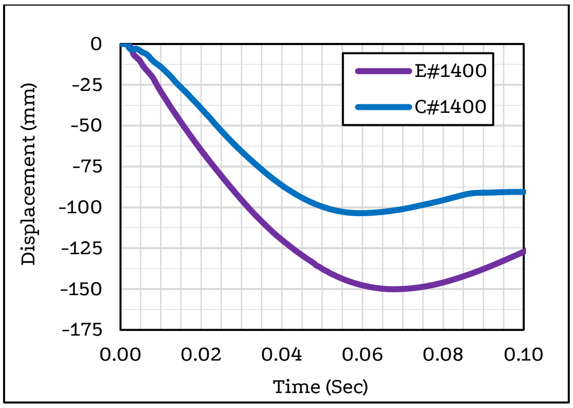
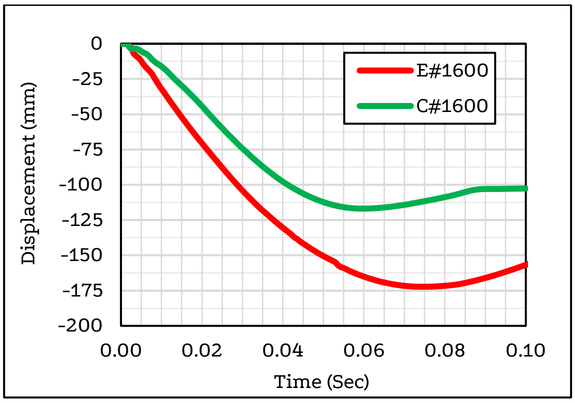


Figure 5. Parametric study finite element modules









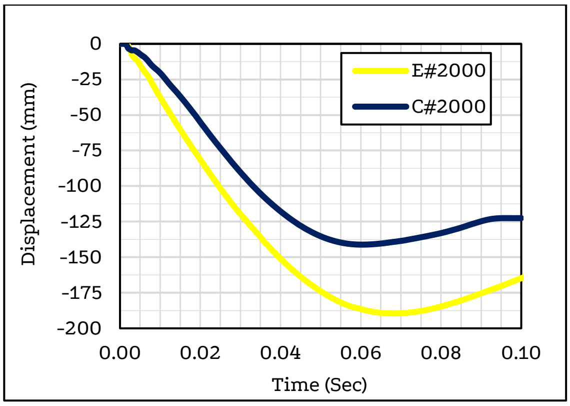


Figure 6. Numerical curves of displacement-time for Series I and Series II

Impact loading occurs when a sudden and short-term force is applied to a structure or system, resulting in immediate effects. In this context, the relationship between permanent displacement and maximum displacement determines the amount of movement or deformation that occurs under the applied impact force. The maximum displacement represents the highest deformation or movement that can occur due to the impact force applied to the structure. It signifies the pinnacle point of force and effect applied to the structure during impact loading. On the other hand, permanent displacement refers to the displacement that remains after the forces applied during impact have been removed and does not revert back. These permanent displacements arise when structural elements undergo plastic deformation or sustain damage during impact loading.

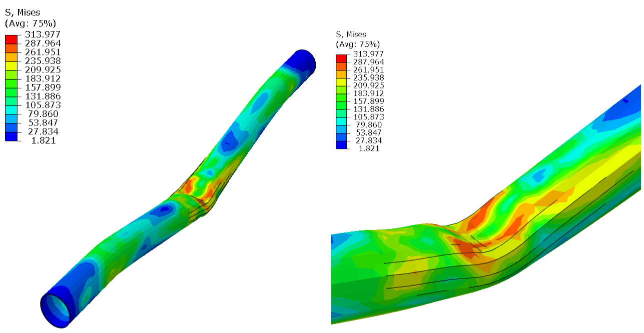
The relationship between these displacements during impact loading typically varies depending on factors such as structural material properties, geometry, and the duration of the impact effect. In some instances, maximum displacements can lead to permanent displacements, potentially causing structural damage or permanent deformations. Understanding this relationship is crucial for the design and resilience of structures against impact, and it is analyzed to ensure structural integrity and safety. During the simulation of impact loading, Von Mises stress criteria were considered and were calculated from a combination of normal and shear stresses. When the series 1 and series 2 Von Mises stress values were examined, the stress value increased as the height increased and it was seen in Figure 7. In the circular cross-section C#1400 to C#1600 samples formed by the Series 1 elements. A 20 cm height increase caused an 8.4% increase in the stress value. 20 cm height increase of C#1600 to C#2000 samples resulted in a 2% increase in the stress value. The same situation was observed in the 2nd Series elements with ellipse cross-section. 20 cm height increase in the E#1400 to E#1600 samples resulted in a 7% increase in the stress value. In addition, in the examination made using the same height and weight, a 16% increase in the stress value occurred in the C#1400 and E#1400 samples when the cross-section geometry changed from circle to ellipse. Similarly, in the analyzes performed using the same height and weight, a 19% increase in stress value occurred in C#1600 and E#1600 elements when the cross-section geometry changed from circle to ellipse. From the analysis of C#2000 and E#2000 sections, when the section geometry changed from circle to ellipse, there was a 13% increase in the Von Mises stress values obtained and Figure 5 can be examined for this situation.

During the simulation of impact loading, Equivalent Plastic Strain (PEEQ) criteria were taken into consideration. When the Series 1 and Series 2 PEEQ values were examined, the PEEQ value increased as the height increased and in Figure 8, a 20 cm height increase in the circular cross-section C#1400 to C#1600 samples formed by the Series 1 elements is % in the PEEQ value. It resulted in an increase of 50%. 20 cm height increase of C#1600 to C#2000 samples resulted in a 40% increase in PEEQ value. The same situation was observed in the 2nd Series elements with ellipse cross-section. 20 cm height increase in E#1400 to E#1600 samples resulted in a 31% increase in PEEQ value. 20 cm height increase of C#1600 to C#2000 samples resulted in a 25% increase in PEEQ value. In addition, in the examination using a hammer dropped from the same weight and height; It was seen that there was a 183% increase in PEEQ value in C#1400 and E#1400 samples with the change of cross-section geometry from circle to ellipse. Similarly, in the examination using a hammer dropped from the same weight and height; When the cross-section geometry changed from circular to elliptical, there was a 147% increase in PEEQ value in C#1600 and E#1600 elements. In addition, in the examination using a hammer dropped from the same weight and height; In the analysis of C#2000 and E#2000 sections, when the section geometry changed from circular to ellipse, there was a 122% increase in the PEEQ values obtained. Figure 9 can be examined for this situation.

Table 3. Results of the parametric study

Test series	Specimen code	Max Displ. mm	Residual Displ. mm	Von Mises MPa	PEEQ (Equivalent Plastic Strain)
	C#1400	103	90	295	0.155
I	C#1600	116	103	320	0.233
	C#2000	141	122	313	0.326
	E#1400	150	102	253	0.44
II	E#1600	172	120	270	0.577
	E#2000	189	125	276	0.726

The C#1400 specimen exhibits a 14% difference between the maximum displacement and the permanent displacement. Similarly, for the C#1600 and C#2000 elements, this difference stands at 12% and 15% respectively. In ellipsoid-shaped elements with heights of 1400mm, 1600mm, and 2000mm, the observed differences are 47%, 43%, and 51% respectively. Considering the cross-sectional properties of the elements, there is a greater degree of plastic deformation in the ellipsoid-shaped elements compared to those with circular geometry, specifically observed between C#1400 and E#1400. This trend persists in E#1600 and E#2000 elements as well. Consequently, the most significant factor is presumed to be the maximum displacement in Table 3. This conclusion is derived from the fact that in the hammer drops from heights of 1400mm, 1600mm, and 2000mm, the ellipsoid-shaped elements exhibit higher maximum displacement values compared to the circular-shaped elements. With regard to the element, some of the plastic deformation resulting from the maximum displacement reverts back, while the remainder persists as permanent deformation. Hence, elements with greater maximum displacement also exhibit more significant permanent deformation. Analyzing the specimen characteristics employed in the study reveals that elements with ellipsoid geometry undergo greater displacement when compared to circular elements of the same height. Consequently, a parallel increase is noted in the level of permanent deformation.



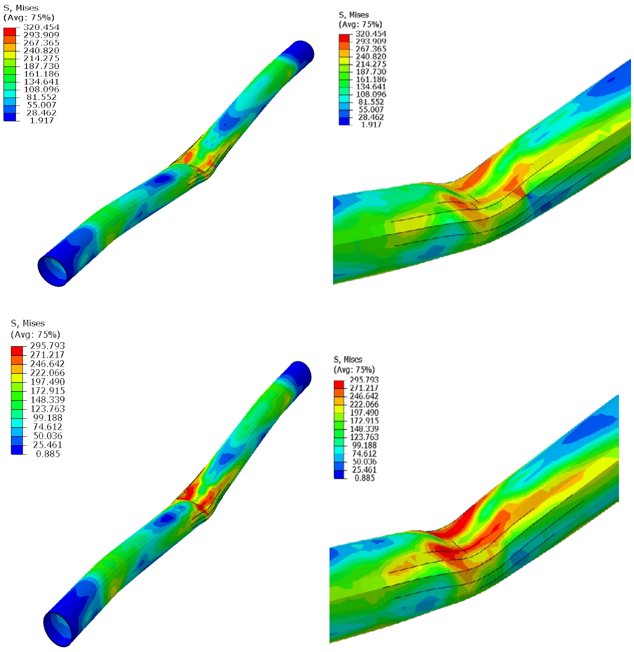


Figure 7. For Series I used in the parametric study; Von Mises stress distribution

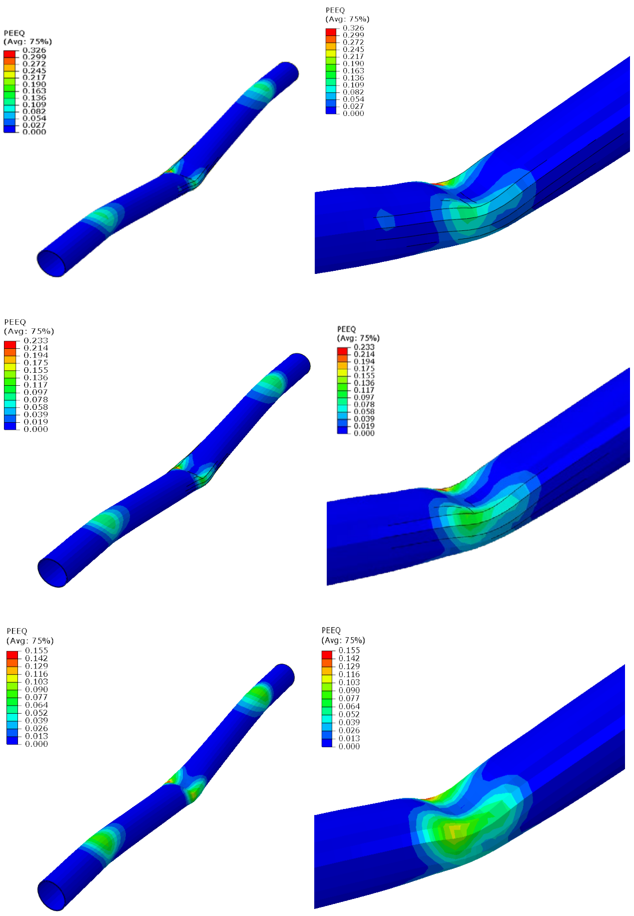


Figure 8. For Series I used in the parametric study; PEEQ ((Equivalent Plastic Strain) distribution

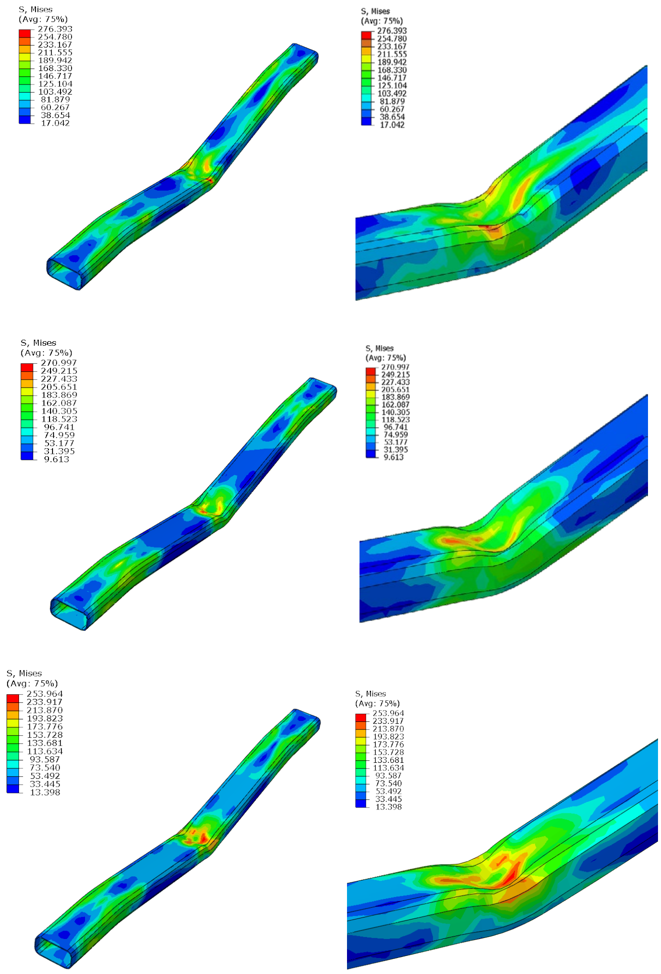
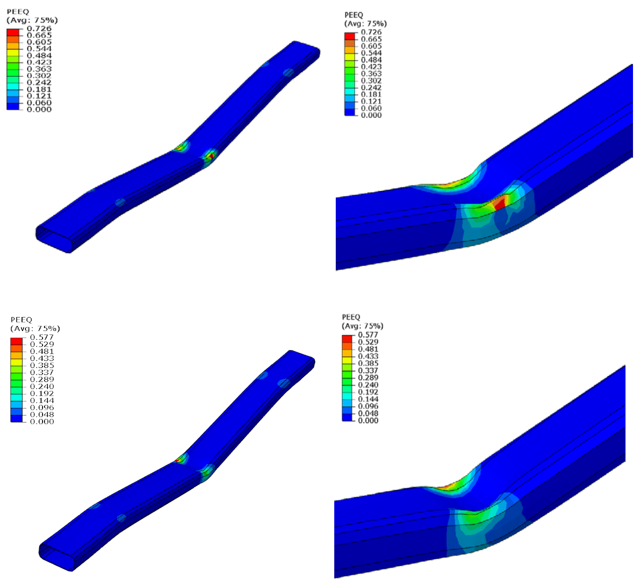


Figure 9. For Series II used in the parametric study; Von Mises stress distribution



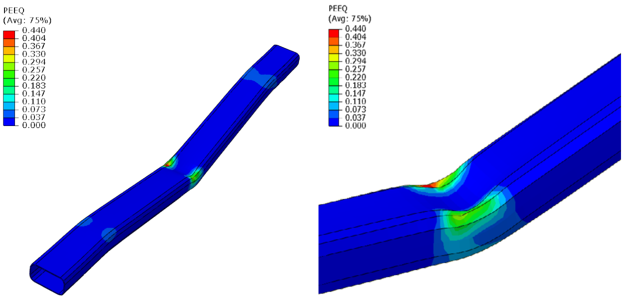


Figure 10. For Series II used in the parametric study; PEEQ ((Equivalent Plastic Strain) distribution

5. Results

The data acquired from the experimental study was used to validate the finite element study. A revision in the previous finite element analysis revealed an updated residual displacement value, decreasing from 88.1% to 4.4% in the revised model.

Upon reviewing the updated finite element study, a cross-check was performed, exposing a 3.7% error in the maximum displacement value and a 4.4% error in the residual displacement value between the experimental and finite element models.

Analyzing data from Series 1 and Series 2, it was evident that the C#1400 sample displayed the lowest Von Mises stress value, while the E#200 sample exhibited the highest. It was determined that the section geometry and hammer drop height significantly influenced stress occurrences within the element.

Within Series 1 and Series 2, the C#1400 sample demonstrated the lowest displacement value, whereas the highest was observed in the C#2000 sample. The section geometry and hammer drop height were identified as impacting the displacement values in the element. Notably, there was a 12% and 21% increase in displacement observed when hammers were dropped from heights of 1400 mm, 1600 mm, and 2000 mm, respectively, for circular cross-section elements. For ellipse cross-section elements, the displacement increased by 15% and 26% under the same height conditions.

Moreover, the lowest PEEQ value was obtained from the C#1400 sample within Series 1 and Series 2, while the E#2000 sample exhibited the highest PEEQ value. Again, section geometry and hammer drop height were indicated to impact stress values within the element.

Upon comparing elements with the same hammer height but different cross-sectional properties, it became evident that circular cross-section elements showcased lower stress values compared to ellipse counterparts. This disparity was noted as 14%, 18%, and 13% for hammers dropped from heights of 1400 mm, 1600 mm, and 2000 mm, respectively.

Similarly, analyzing data from elements with the same hammer height but different cross-sectional properties, it was observed that circular cross-section elements displayed lower PEEQ values compared to ellipse elements. This discrepancy amounted to 183%, 147%, and 122% for hammers dropped from heights of 1400 mm, 1600 mm, and 2000 mm, respectively. Considering the displacement, stress, and PEEQ values from the six samples in Series I and Series II, it was evident that stress values were consistent with displacement and acceleration values.

Declaration of Conflict of Interests

The author declares that there is no conflict of interest. They have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1.] Ellinas, C.P., and Alastair C.W., Damage on offshore tubular bracing members. IABSE colloquium, Copenhagen. 42 (1983).

[2.] Zhang, Shengming, P. Terndrup Pedersen, and Hasan Ocakli. "Collisions damage assessment of ships and jack-up rigs." Ships and Offshore Structures 10 (2015) 470-478.

[3.] Barr, B., and Baghli A., A repeated drop-weight impact testing apparatus for concrete. Magazine of Concrete Research 40(1988) 167-176.

[4.] Breen, C., Felicity G., and Martyn Pavier. Impact of thick CFRP laminates: the effect of impact velocity. Composites Part A: Applied Science and Manufacturing 36 (2005) 205-211.

[5.] Bambach, M.R., Validation of a general design procedure for the transverse impact capacity of steel columns. Journal of Constructional Steel Research 150 (2018) 153-161.

[6.] Farooq, U., and Peter M., Prediction of load threshold of fibre-reinforced laminated composite panels subjected to low velocity drop-weight impact using efficient data filtering techniques. Results in Physics 5 (2015) 206-221.

[7.] Yang, F. J., and Cantwell W.J., Impact damage initiation in composite materials. Composites Science and Technology 70 (2010) 336-342.

[8.] Kantar, E., R. Tuğrul E., and Özgür A., Nonlinear finite element analysis of impact behavior of concrete beam. Mathematical and Computational Applications 16 (2011) 183-193.

[9.] Nassirnia, M., et al. Hybrid corrugated members subjected to impact loading: Experimental and numerical investigation. International Journal of Impact Engineering 122 (2018) 395-406.

[10.] Saini, D., and Behrouz S., Investigation of concrete-filled steel tube beams strengthened with CFRP against impact loads. Composite Structures 208 (2019) 744-757.

[11.] Makarem, F.S., and Farid A., Nonlinear finite element modeling of dynamic localizations in high strength steel columns under impact. International Journal of Impact Engineering 52 (2013) 47-61.

[12.] Kantar, E., Addussamet A., and Anı Öz., Beton Dayanimindaki Değişimin Çarpma Davranişina Etkisi. Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi 26.1 (2011).

[13.] Li, W., et al. Behaviour of ultra-high strength steel hollow tubes subjected to low velocity lateral impact: experiment and finite element analysis. Thin-Walled Structures 134 (2019) 524-536.

[14.] Pichler, B., et al. Loading of a gravel-buried steel pipe subjected to rockfall. Journal of Geotechnical and Environmental Engineering 132 (2006) 1465-1473.

[15.] Yang, J. L., et al. Experimental study and numerical simulation of pipe-on-pipe impact. International Journal of Impact Engineering 36(2009) 1259-1268.

[16.] Anil, Ö., R. Tuğrul E., and Erkan K., Improving the impact behavior of pipes using geofoam layer for protection. International Journal of Pressure Vessels and Piping 132 (2015) 52-64.

[17.] Saleh, Z., Sheikh, M.N., Remennikov, A., Basu, A., Overload damage mechanisms of GFRP-RC beams subjected to high-intensity low-velocity impact loads. Composite Structures, 111578 (2020). 233-245.

[18.] Liu, B., and Guedes S,C., Plastic response and failure of rectangular cross-section tubes subjected to transverse quasistatic and low-velocity impact loads. International Journal of Mechanical Sciences 90 (2015) 213-227.

[19.] Li, W., et al. Behaviour of grout-filled double-skin steel tubular T-joint subjected to low-velocity impact. Thin-Walled Structures 144 (2019) 106270.

[20.] Zhang, R., Xu-dong Z., and Feng F., Plastic behavior of circular steel tubes subjected to low-velocity transverse impact. International Journal of Impact Engineering 114 (2018) 1-19.

[21.] ABAQUS/CAE v6.12 Programme, (2016) Dassault Systemes Simulia Corp. Providence, RI, USA.

[22.] Nasery, M.M, Ağcakoca E, and Yaman Z., Experimental and numerical analysis of impactor geometric shape effects on steel beams under impact loading. Structures 27 (202) 1118-1138.

[23.] Nasery, M. M. (2023). Investigation on Behaviours along Weak Axes of Steel Beam under Low Velocity Impact Loading: Experimental and Numerical. Buildings, 13(9), 2331.

[24.] Şengel, S., Erol, H., Yılmaz, T., & Anıl, Ö. (2022). Investigation of the effects of impactor geometry on impact behavior of reinforced concrete slabs. Engineering Structures, 263, 114429.

[25.] Liu, H., Liu, J., Ding, Y., Zhou, J., Kong, X., Blackman, B. R., ... & Dear, J. P. (2020). Effects of impactor geometry on the low-velocity impact behaviour of fibre-reinforced composites: an experimental and theoretical investigation. Applied Composite Materials, 27, 533-553.

How to Cite This Article

Ağcakoca, E., Behavior of Steel Sections with Different Geometries Under Impact Load, Civil Engineering Beyond Limits, 4(2023), 1891. https://doi.org/10.36937/cebel.2023.1891