Sensors and Materials, Vol. 36, No. 3 (2024) 1163–1177
MYU Tokyo

S & M 3589

Use of Finite Element Method Software to Assess the Safety of a Newly Designed Electric Motorcycle Frame

Hong-Mei Dai,[1] Bo-Hao Chen,[2] Chao-Ming Hsu,[2*]
Chiang-Lung Lin,[3] Hsien-Wei Tseng,[4] and Cheng-Fu Yang[5,6**]

[1]College of Art, Zhongqiao Vocational and Technical University, Shanghai 201514, China
[2]Department of Mechanical Engineering, National Kaohsiung University of Science and Technology, Kaohsiung 807, Taiwan
[3]Dongguan University of Technology, Guangdong-Taiwan College Industrial Science & Technology, Guangdong 523419, P.R. China
[4]College of Artificial Intelligence, Yango University, Fuzhou, Fujian 350015, China
[5]Department of Chemical and Materials Engineering, National University of Kaohsiung, Kaohsiung 811, Taiwan
[6]Department of Aeronautical Engineering, Chaoyang University of Technology, Taichung 413, Taiwan

(Received October 20, 2023; accepted February 26, 2024)

Keywords: electric motorcycle, frame, finite element analysis software, ANSYS

Two distinct approaches are available for scrutinizing the impact of diverse loads and the full-load capacity, modal characteristics, and fatigue life of an electric motorcycle frame. The initial method entails affixing stress sensors directly to the frame, whereas the second method leverages finite element method (FEM) software to conduct simulations. In this particular study, the simulation approach was adopted, and in this paper, the feasibility of establishing an analytical system for assessing an electric motorcycle frame was determined through the utilization of FEM software tools such as Visual Basic (VB) and ANSYS Parametric Design Language (APDL). Through the use of simulation analysis, we aimed to explore potential structural defects and provide directions for potential modifications, ultimately obtaining an electric motorcycle frame design that complies with regulations and exhibits improved performance. We first utilized computer-aided design software to create the motorcycle frame and generate a geometric model. Following this, FEM analyses were conducted to evaluate a prototype of the innovative electric motorcycle frame design. These analyses encompassed modes of force transmission in left and right bending moments, front and rear torques, and the results of postcollision deformation of the electric motorcycle frame. These analyses were carried out under specific conditions stipulated by the Vehicle Testing Center, covering a range of load conditions with various degrees of deformation and strength. This allowed us to assess whether the frame structure complies with regulatory standards.

1. Introduction

In recent years, global carbon emissions have been steadily increasing, exacerbating the global greenhouse effect and leading to more frequent extreme weather events. The rise in

---

[*]Corresponding author: e-mail: jammy@nkust.edu.tw
[**]Corresponding author: e-mail: cfyang@nuk.edu.tw

https://doi.org/10.18494/SAM4728

ISSN 0914-4935 © MYU K.K.
https://myukk.org/

carbon emissions not only affects the global climate but also contributes significantly to severe air pollution issues. According to research, promoting green energy electric motorcycles(1,2) and vehicles,(3,4) and replacing outdated two-stroke engines and old gasoline-powered motorcycles with zero carbon dioxide (CO_2) emission electric motorcycles are crucial for improving air quality and reducing the overall amount of domestic carbon emissions. Through this study, we aim to develop cost-effective and reliable electric motorcycle models that are more affordable than the existing market offerings. This approach seeks to increase the willingness of the general public to transition from traditional gasoline-powered motorcycles to electric ones, ultimately contributing to the government's efforts to foster the growth of the green energy industry and achieve energy savings and carbon reduction goals. The primary objective of this research is to leverage software simulations to design electric motorcycle frame structures efficiently. This methodology allows for the rapid development of electric motorcycle frames with enhanced safety features, reducing both development time and costs. By combining technological innovation with sustainability goals, this research aligns with the broader initiative to combat climate change and promote cleaner transportation options.

The primary purpose of an electric motorcycle frame is to bear loads. When the electric motorcycle frame structure is poorly designed or when the materials used in it experience a decrease in rigidity due to daily usage, its ability to withstand loads significantly decreases. This results in a decrease in performance. This study is focused on the analysis and testing of the electric motorcycle frame, including its rigidity, modal characteristics, and durability. During the operation of a vehicle, the electric motorcycle frame structure not only supports the weight of various components and the driver but also must withstand loads coming from the road surface (such as rough road conditions). These loads are transmitted to the electric motorcycle frame through the front and rear axles and the suspension system, and they are the primary cause of deformation of the electric motorcycle frame structure. Therefore, the rigidity of the electric motorcycle frame structure must be sufficient to withstand these external loads, preventing permanent deformation that could affect the alignment and compatibility with other components. This is crucial to preventing significant reduction in vehicle handling performance and, in severe cases, ensuring the safety of the driver.

In the past research, Rodriguez *et al.* harnessed the power of structural finite element analysis software to optimize the design of chopper-style motorcycle frames.(5) By leveraging insights gained through finite element analysis, their work represented a significant leap forward in enhancing both the strength and weight characteristics of these frames. This innovative approach was an integration of prior conventional design methods with the newfound capabilities of modern software tools. Vivek *et al.* employed Solidworks drawing software to meticulously craft a three-dimensional model of a two-wheeled motorcycle.(6) This intricately detailed model was then imported into finite element analysis software, where it was transformed into an analytical representation. Beyond this, their research encompassed the refinement of the motorcycle's drive system and suspension mechanisms. The outcomes of this comprehensive analysis not only offer valuable insights for powertrain design but also lay the groundwork for dynamic response assessments and computational analyses of related vehicle systems. In a distinct line of investigation, Choua and Hsiao conducted a comprehensive analysis using the mathematical dynamic model (MADYMO) software, adhering to the ISO13232 specification for evaluating collision protection devices for motorcycle riders.(7) Choua and Hsiao(7) meticulously followed established testing and analysis protocols, utilizing a prototype test motorcycle as the basis for their model. To ensure rider safety, a prototype airbag system was incorporated as the protective device. This research contributed significantly to improving the safety of motorcycle riders in collision scenarios.

Historically, two distinct approaches were employed to assess the impact of various loads on the modal behavior, full-load performance, and fatigue life of an electric motorcycle frame. One method involved the attachment of numerous sensors at different points across the designed electric motorcycle frame to measure stresses at various positions.(8–10) However, in other studies, the emphasis was shifted towards simulation analysis as a means to delve into the characteristics of the electric motorcycle frame, anticipate potential structural flaws, and offer guidance for modifications aimed at enhancing its mechanical properties.(11–13) In the analytical process, ANSYS, a finite element method (FEM) software program, which was applied to a scaled-down and lighter prototype frame structure of an electric motorcycle, was employed. The primary focus of the simulation was to assess frame deformation under various load conditions and identify vulnerable areas within the newly designed electric motorcycle frame structure that required reinforcement. Our research represents a shift towards a more comprehensive and predictive approach to frame design with the aim of creating electric motorcycles with improved structural integrity and performance.

To streamline our mechanical models, the specific coordinate systems for geometric modes were established first. Our primary goal was to employ a simulation analysis technology to assess the structural integrity of the electric motorcycle frame, enabling us to anticipate potential defects and provide guidance for enhancing its mechanical characteristics. This process of examining the force model of the prototype frame structure, which was lighter and smaller, was initiated using ANSYS.(12,14) Subsequently, components with various thicknesses and shapes were organized, dividing the frame elements into separate segments. This automation facilitated the creation of FEM models for the electric motorcycle frame, significantly reducing the workload and technical complexities associated with FEM pre-processing. The investigated system has the capability to automatically identify the model type on the basis of the analysis type and verify the model's geometry for errors. It also allows users to select suitable elements for analysis and even modify the element type through programming. These analyses encompass various modes of force transmission, including left and right bending moments and front and rear torques, as well as the postcollision deformation results of the electric motorcycle frame. Our findings confirm the feasibility and accuracy of the ANSYS simulation process, as validated by comparison with traditional manual analysis results.

2. Simulation Process and Parameters Used

Finite element analysis (FEA) technology has evolved into a core component of computer aided engineering (CAE). By employing CAE techniques, it becomes possible to reduce or even eliminate the time and cost associated with physical testing processes. Simulating the working conditions of components, such as loads and structures, using computer models allows for the calculation of stress, strain, and load distribution, enabling the evaluation of various performance aspects of a product during the design phase. Moreover, it facilitates the early detection and improvement of design issues, thus shortening the design and development cycle. Various methods fall under the umbrella of FEA, including the boundary element method, finite difference method, and FEM. In this article, the structural analysis of an electric vehicle is conducted by FEM with ANSYS software. The results presented in this paper shed light on the capabilities of FEM in analyzing stress aspects of the structures of the investigated electric motorcycle and provide a comparative advantage over other methodologies.

The frame of an electric motorcycle serves as the fundamental structure that bears external forces and loads. During its operation, this frame is subjected to repetitive and irregular loads, which can lead to fatigue damage over time. As a result, the design of the frame structure is a pivotal step in the development of an electric motorcycle with a primary focus on ensuring its rigidity and strength.(15) Our design process commenced with the utilization of SolidWorks, a computer-aided design (CAD) software program, to create a detailed 3D model of the frame at a one-to-one scale. Once we determined the appropriate mesh and node configurations, we conducted an extensive stress analysis to ensure that the design was free from any interference between different parameters.

Subsequently, we imported the designed frame into ANSYS to validate the distribution of stress within the structure. We executed a comprehensive and systematic analysis to ascertain whether the electric motorcycle frame met our strength expectations. To expedite this analysis process, we judiciously simplified the model without compromising its accuracy. Our chosen simplification principles were meticulously selected to ensure that the analysis results remained unaffected. These principles include the following:

(a) exclusive consideration of the frame structure of the motorcycle, omitting any structural elements that do not impact the analysis results, and
(b) disregarding the welding effects of the frame during the simulation process.

The simplified geometric representation of the frame for an electric motorcycle is visually depicted in Fig. 1.

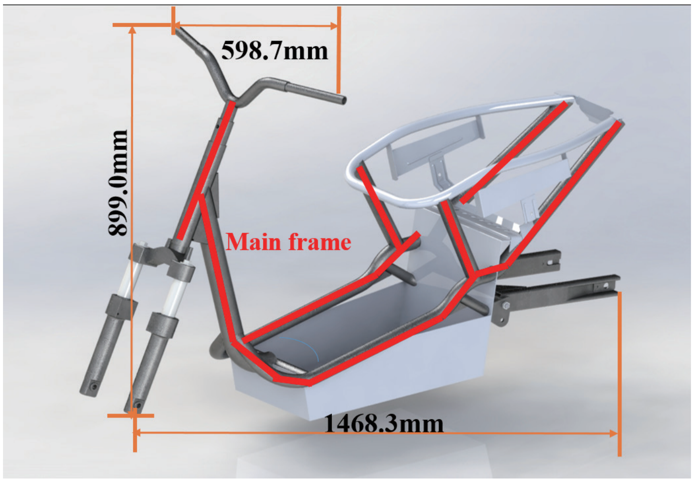

Fig. 1. (Color online) Schematic diagram for the frame of the investigated electric motorcycle.

We conducted a convergence analysis of the test grid, using the maximum stress location, as shown in Fig. 2(a), obtained from the preliminary analysis results as a reference point, as illustrated in Fig. 2(b). The analysis results revealed that convergence began when the number of grid elements approached approximately one million, with a calculation error of less than 3%. Consequently, we selected 1.1 million as the optimal grid size for this research analysis. This choice of grid size is noteworthy as it strikes a balance between computational efficiency and accuracy. The decision to use 1.1 million grid elements ensures that our calculations maintain a high level of precision while also being feasible in terms of computational resources. This optimization allows us to proceed with confidence in the reliability of our analysis results.

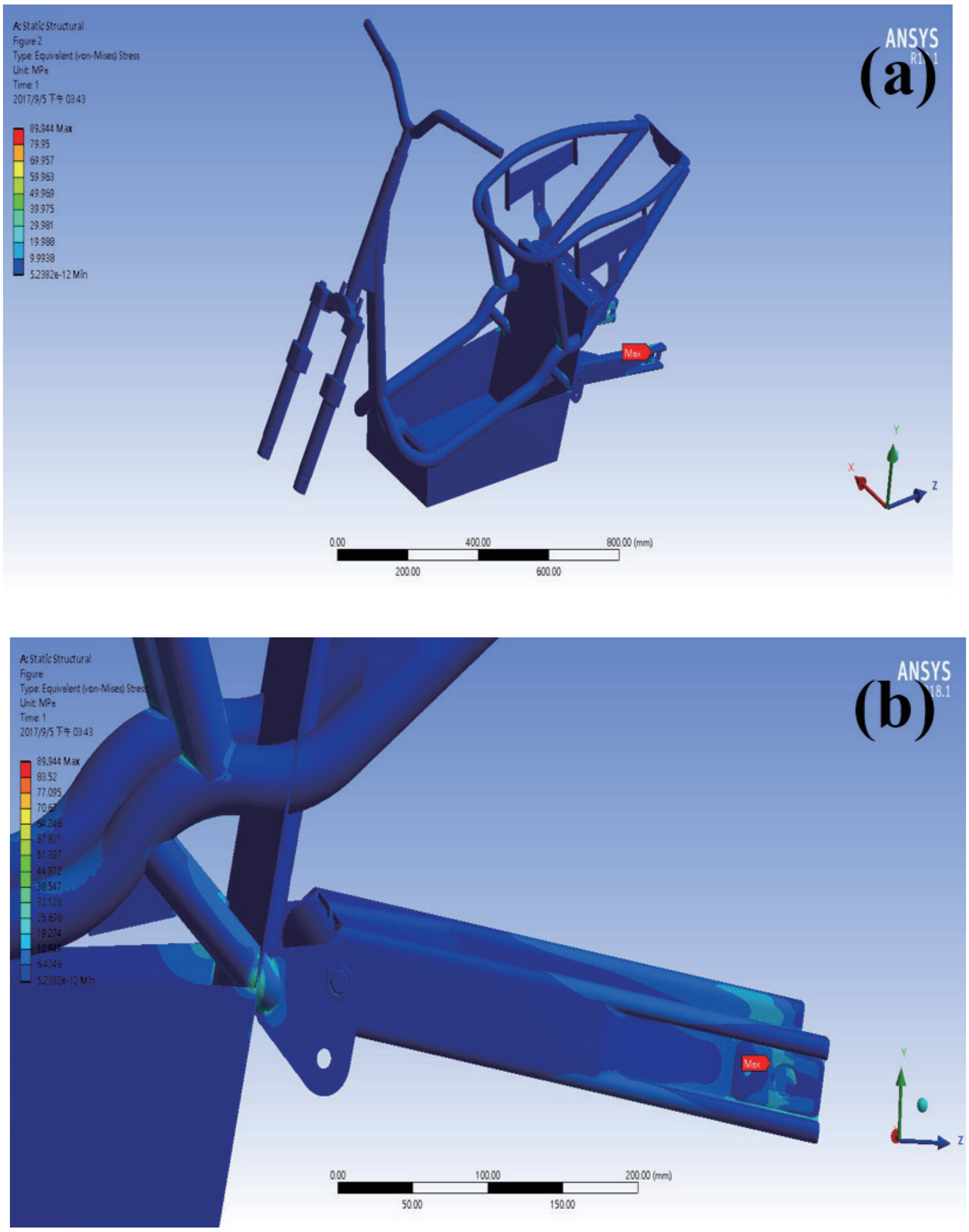

Fig. 2. (Color online) (a) Reference point for the location of the maximum stress and (b) reference point location.

During the optimization process, a critical step involves converting the original physical problem into a standardized mathematical model. This process can be broadly divided into the following three steps:

1. selection of design variables,
2. definition of constraints, and
3. specification of the objective function.

In this particular study, A36 structural steel was chosen as the material for the electric motorcycle frame. The parameters of this steel are detailed in Table 1, while an overview of the boundary conditions used in the initial frame analysis show a seat tube with a load of 500 N, and restrained rear and front axles. The von Mises yield criterion was employed to calculate the frame's stress, and the corresponding calculation equation is commonly known as the von Mises stress equation.(16) When assessing the stress components in terms of von Mises pressure, the equation takes the form

$$\sigma^2_{von\text{-}Mises} = \left(\sigma_x + \sigma_y\right)^2 - 3\left(\sigma_x\sigma_y - \sigma^2_{xy}\right)^2, \tag{1}$$

where σ_x, σ_y, and σ_z represent the principal stresses in three directions at the specific points within the structure. If we focus solely on the performance of principal stresses, the equation simplifies to

$$\sigma^2_{von\text{-}Mises} = 1/2\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_z - \sigma_x\right)^2\right]. \tag{2}$$

In this research, equivalent stress served as the metric for comparing the analysis results. The ANSYS Workbench FEM software utilizes a color-coded representation of stress values, facilitating the subsequent observation and discussion of the results.

In our analysis model, a mesh consisting of a combination of triangular and tetrahedral elements was employed, each having a size of 3 mm. This intricate mesh structure comprised a total of 2161306 nodes and 1112357 elements. To ensure accuracy and precision, we applied mesh refinements to specific regions, particularly at the junctions between the seat tube and the main frame, and between the main frame and the reinforcement tubes. These refinements were implemented to capture and account for potential stress concentration points accurately. For a

Table 1
Parameters of A36 steel.

	unit	value
Density	kg/m^{-3}	7850
Yield strength	MPa	250
Tensile strength	MPa	400
Bulk modulus	GPa	166.67
Shear modulus	GPa	76.92
Young's coefficient	GPa	200
Poisson's ratio		0.3

comprehensive overview of the mesh details used in this study, Table 2 provides a summary of the numbers of mesh nodes and grid elements that were meticulously investigated and employed. The welding junctions were set to have bonded contacts and employed an encrypted grid configuration, as depicted in Fig. 3, for the rear swingarm connection point. This step is primarily undertaken to align with real-world conditions and to accurately indicate the location when maximum stress or deformation occurs at the junction points. This ensures a more precise representation of the actual conditions in the analysis.

3. Simulation Results and Discussion

When an electric motorcycle is in motion on the road, its frame encounters a various of road conditions, including uneven surfaces and potholes. Moreover, when the electric motorcycle negotiates turns, the frame undergoes bending and torsional vibration. These operational loads are transmitted from the front suspension to the head tube and subsequently distributed

Table 2
Description of grid.

Number used for frame stiffness analysis	Complete frame	
525445	Mech nodes	2161306
360695	Grid elements	1112357
Grid type	Tet10	

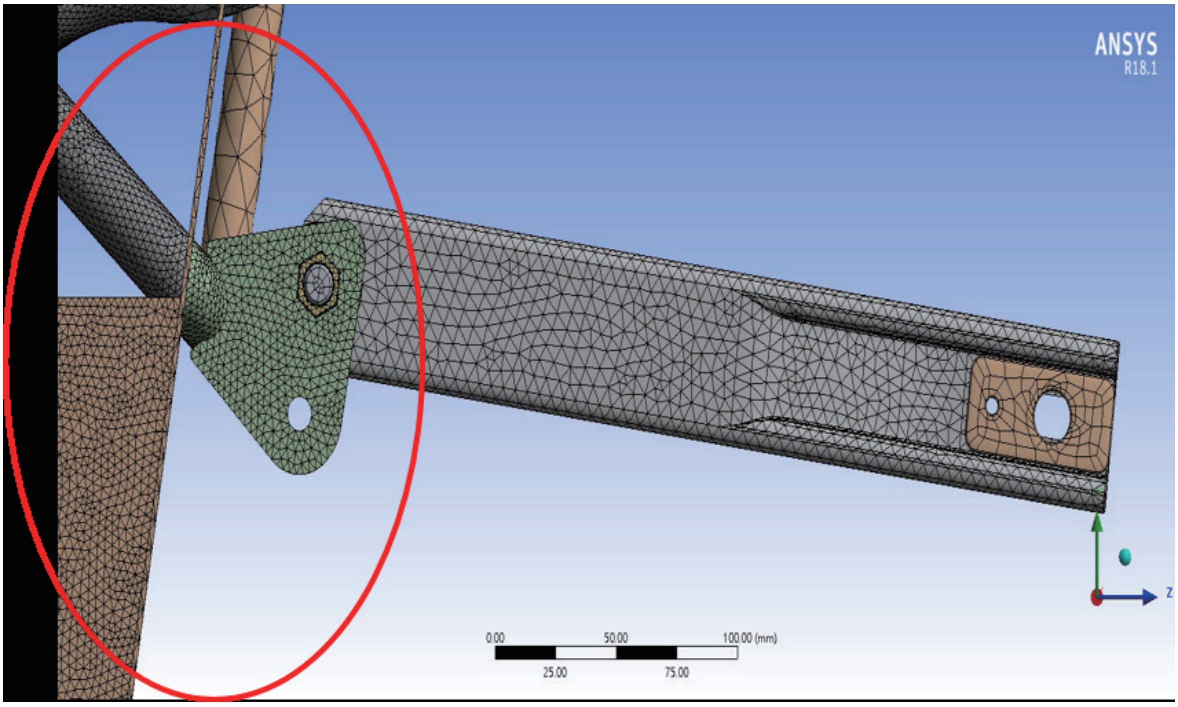

Fig. 3. (Color online) Densified area of the rear swingarm connection point for mesh refinement.

throughout the frame structure. Therefore, as discussed in the preceding sections, it is imperative to simulate the mechanism of force transmission to ensure that the design effectively disperses these loads rather than concentrates stress points. Insufficient frame rigidity can result in excessive deformation, making riding uncomfortable. Furthermore, it can have a detrimental impact on the electric motorcycle's stability and safety during operation. Consequently, the rigidity and strength of the frame structure are critical elements in the design process, warranting careful consideration and optimization. In the process of modeling development, we will conduct analyses to assess the torque and bending rigidity of the frame. When it comes to establishing boundary conditions, the frame model will be anchored in a fixed position relative to the front fork location. This anchoring will impose constraints on various degrees of freedom, encompassing translations along the x-, y-, and z-axes, as well as rotations around them.

Similarly, the model of the rear swingarm will be configured with contact bodies affixed to the rear fork, effectively limiting its degrees of freedom. The design of the frame in this study incorporates a Twin Shock suspension system, leading to a dual-suspension setup for the rear swingarm. This dual-suspension arrangement serves to mitigate any potential imbalances in forces experienced by the rear swingarm, a concern that can arise in single-sided suspension designs. The frame's different components will be interconnected using a combination of bolted and welded joints. Nevertheless, for the purposes of this study, which primarily focuses on analyzing stress distribution within the frame, specific welding or bolting details will be disregarded, and all frame connections will be treated as if they were perfectly joined. To ensure the accuracy of calculations at the connection points, mesh refinement settings will be employed. The analysis model will be simplified to represent the foundational considerations of frame strength from the initial design phase.

The concept of bending moment rigidity in the context of a frame refers to its ability to resist deformation when subjected to a force applied parallel to the frame's head tube. This force induces a bending angle in the frame. The calculation of bending moment rigidity involves multiplying the applied bending moment load by the radius of curvature of the resulting bending angle, with measurements in units of N-m. Conversely, defining the torsional rigidity of the frame's head tube entails the generation of a torsional angle when a force is applied perpendicular to the axis of the steering column. Owing to the frame's symmetrical design, it necessitates only one-directional bending moment analysis. To calculate torsional rigidity, the applied torque is divided by the twist angle, measured in units of N-m per radian. In this study, we applied specific boundary conditions to analyze torsional rigidity. A 100 N-m torque was exerted at the center of the frame head tube. Constraints were then imposed on the seat tube, and the impact of this 100 N-m torque on the bending moment of both the left and right sides was examined. Figure 4(a) depicts these boundary conditions, and the results of our analyses regarding left and right bending moment stress and deformation can be seen in Figs. 4(b)–4(d).

As shown in Figs. 4(b) and 4(c), the location experiencing the highest vertical bending moment (von Mises stress) was identified at the connection points between the main support shaft and the pedal base, and the maximum stress recorded at these junctions was 174.52 MPa. Therefore, the position exhibiting the maximum deformation was located at the two seat cushion support rods, as shown in Fig. 4(d), and the maximum deformation amounted to 13.689 mm.

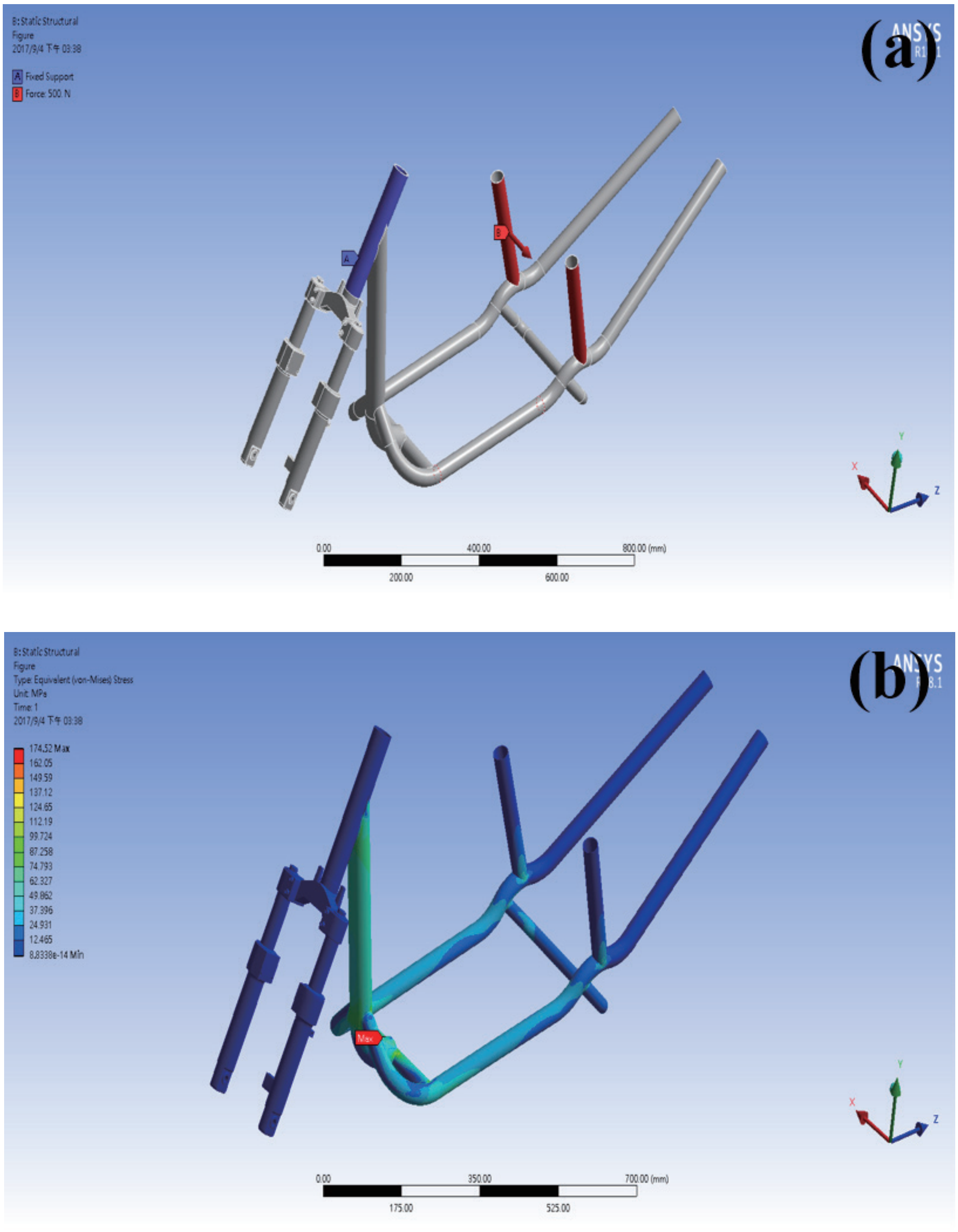

Fig. 4. (Color online) Geometry for the analyses of the left and right bending moments: (a) boundary condition, (b) stress analysis diagram, (c) position of maximum stress, and (d) position of maximum deformation.

This information is crucial for understanding how the frame responds to external forces and can be used to make design improvements or ensure the frame's structural integrity, particularly in cycling applications where frame rigidity is essential in performance and safety. Further investigations into optimizing the frame design based on these findings could lead to enhanced frame durability and overall performance.

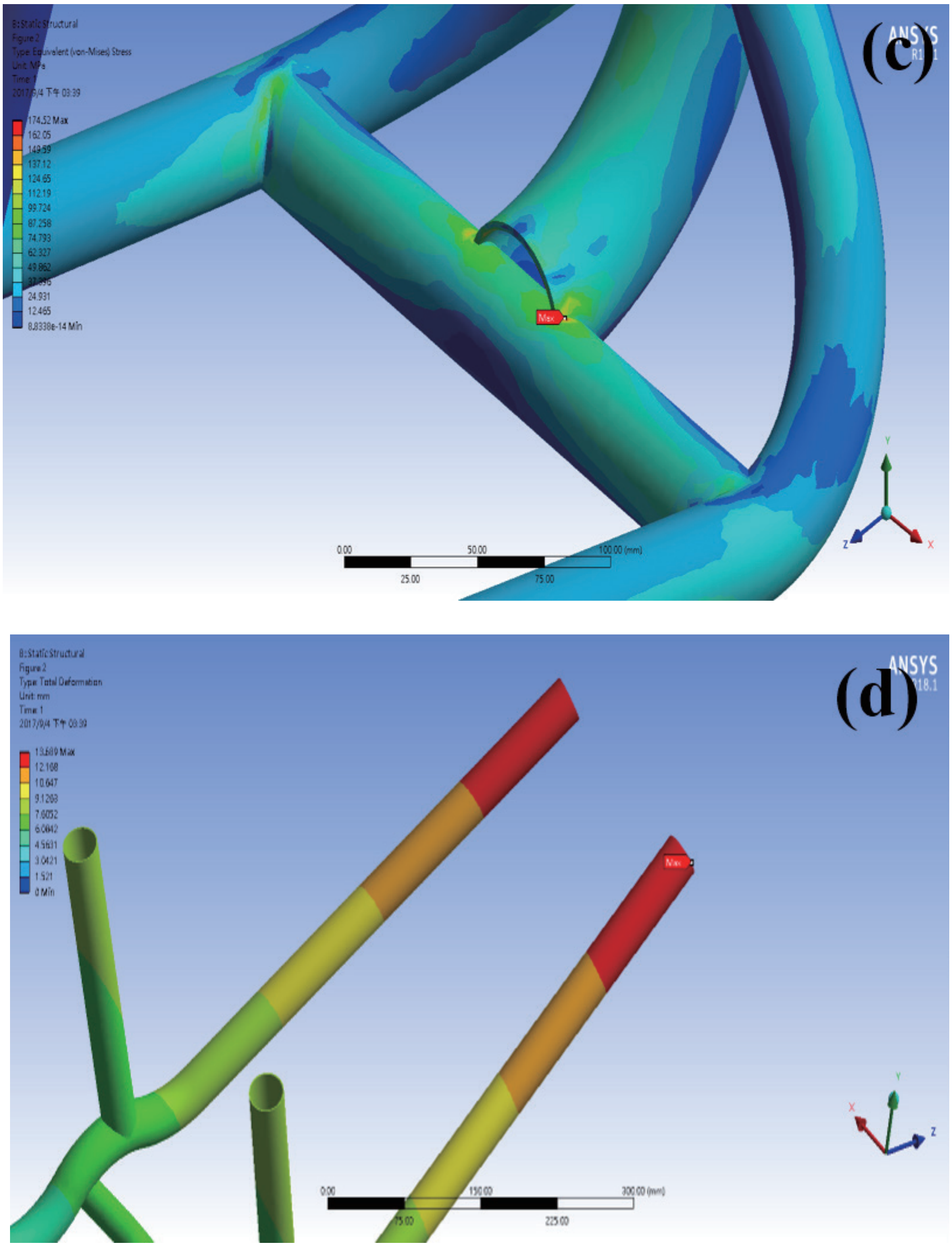

Fig. 4. (Color online) (Continued) Geometry for the analyses of the left and right bending moments: (a) boundary condition, (b) stress analysis diagram, (c) position of maximum stress, and (d) position of maximum deformation.

Also, the 100 N-m torque was exerted at the center of the frame head tube, and the impact of this 100 N-m torque on the front and rear torques was examined. Figure 5(a) depicts these boundary conditions, and the results of our analyses regarding left and right bending moment

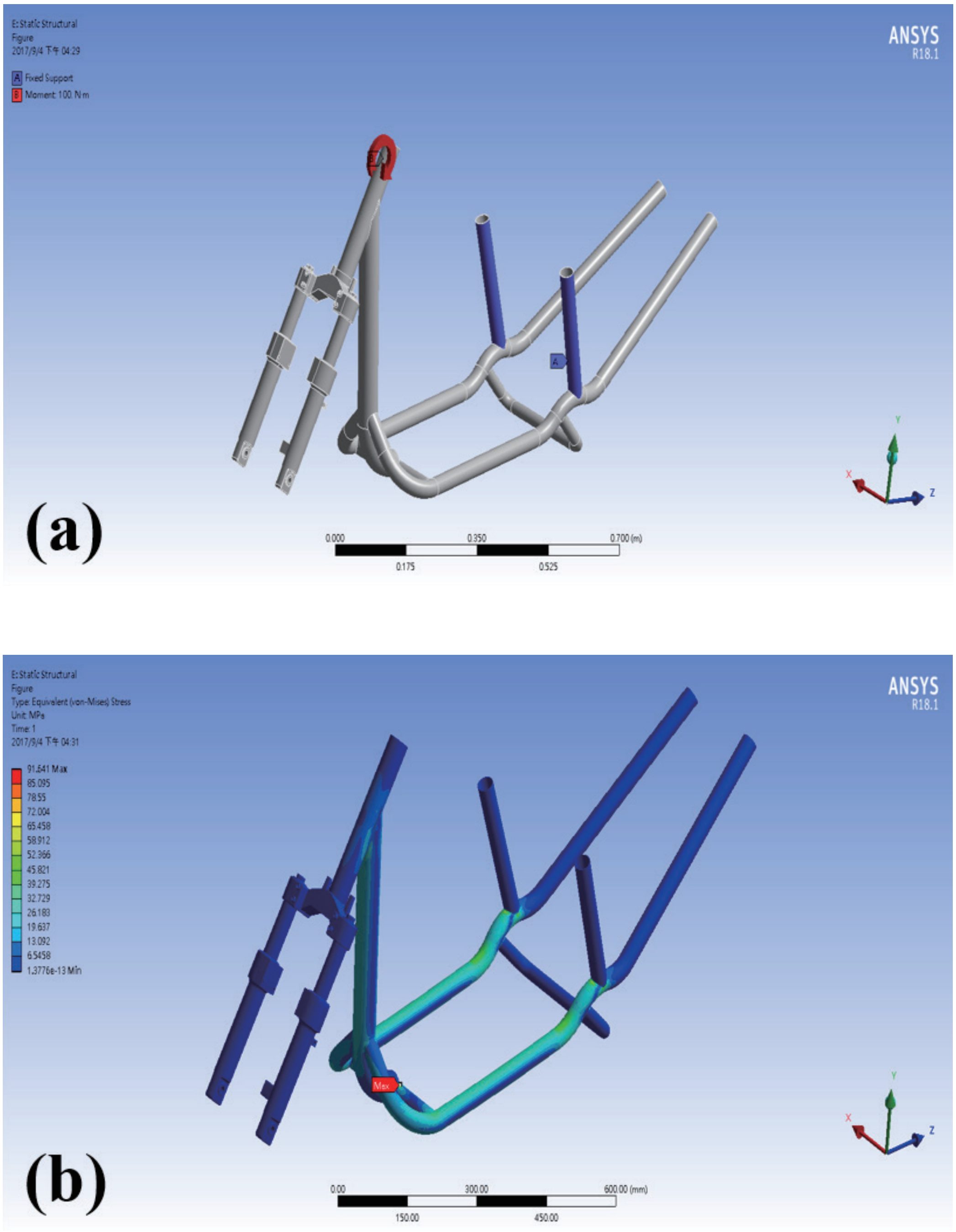

Fig. 5. (Color online) Geometry of analyses of front and rear torques: (a) boundary conditions, (b) stress analysis diagram, (c) maximum stress analysis diagram, and (d) deformation analysis diagram.

stress and deformation can be observed in Figs. 5(b)–5(d). As shown in Figs. 5(b) and 5(c), the maximum stress recorded at these junctions was 91.641 MPa and located at the connection point between the main support shaft and the pedal base. Therefore, the position exhibiting the

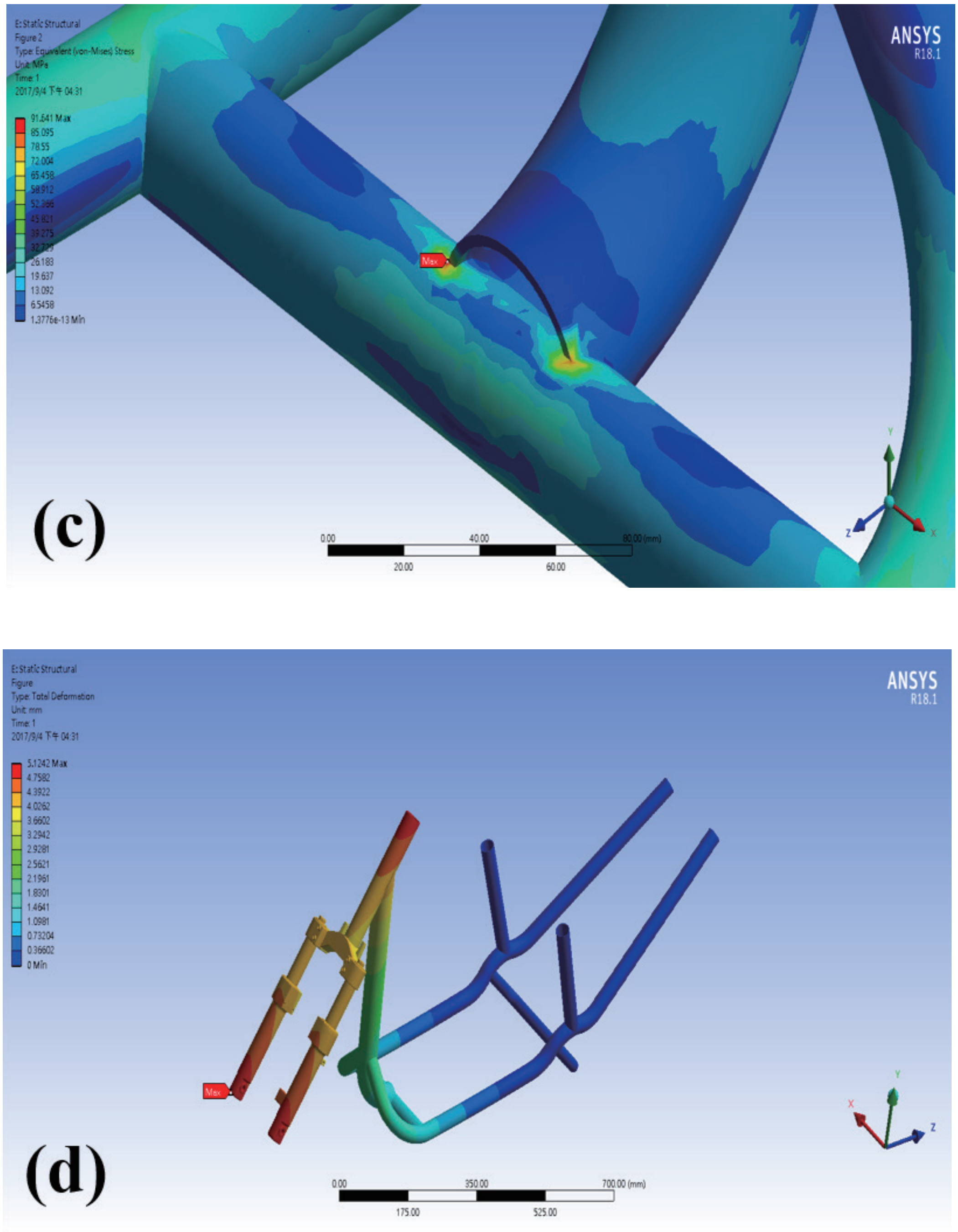

Fig. 5. (Color online) (Continued) Geometry of analyses of front and rear torques: (a) boundary conditions, (b) stress analysis diagram, (c) maximum stress analysis diagram, and (d) deformation analysis diagram.

maximum deformation was located at the faucet handle and front wheel support frame of the electric motorcycle frame, as shown in Fig. 5(d), and the maximum deformation amounted to 5.124 mm.

A transient dynamics model is used to simulate the results of a collision in the frame with boundary conditions as depicted in Fig. 6(a). The collision material is concrete and the impact velocity is 25.2 kilometers per hour, complying with the maximum speed specified in electric

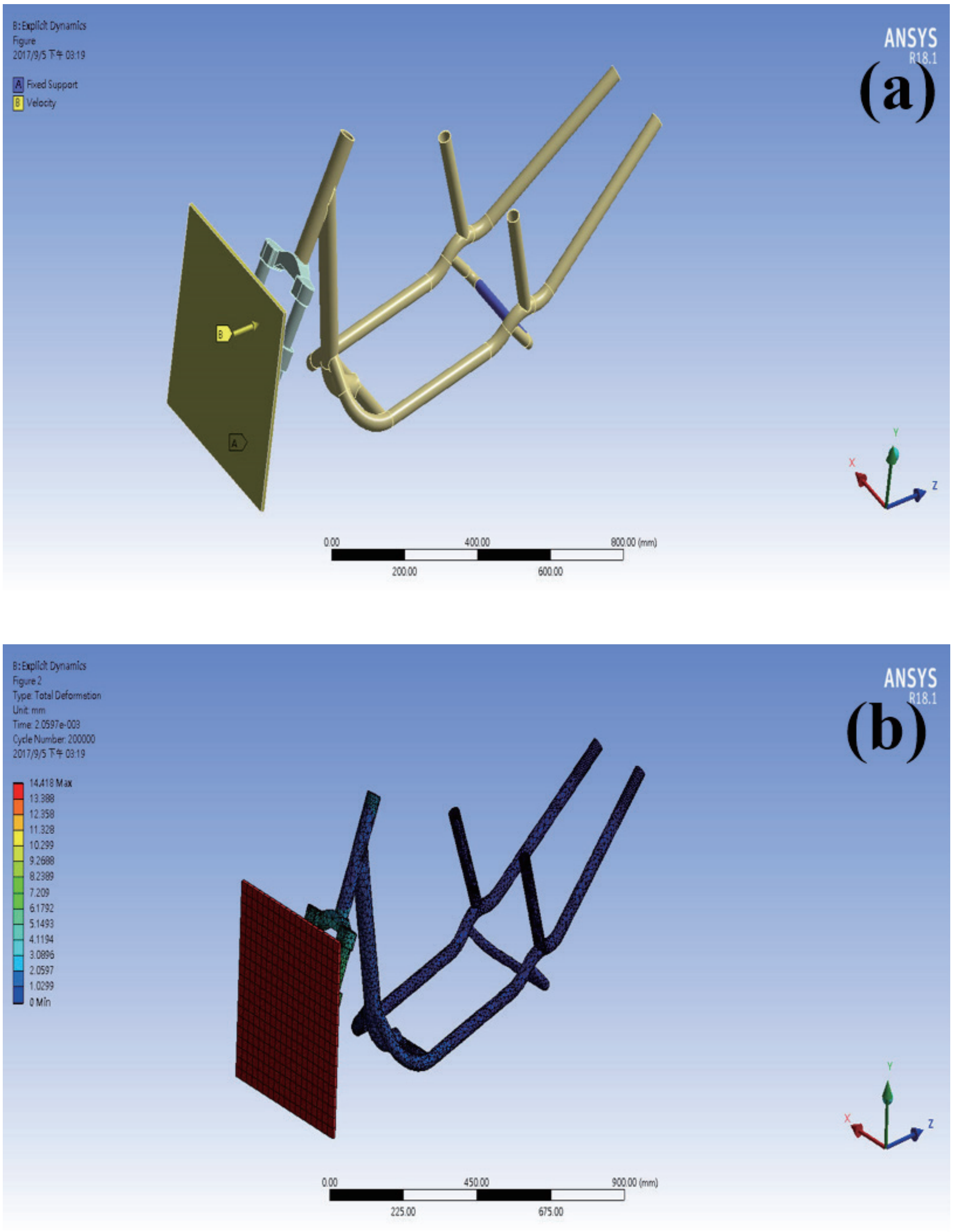

Fig. 6. (Color online) (a) Transient analysis boundary conditions, (b) postcollision results, and (c) maximum deformation position after collision.

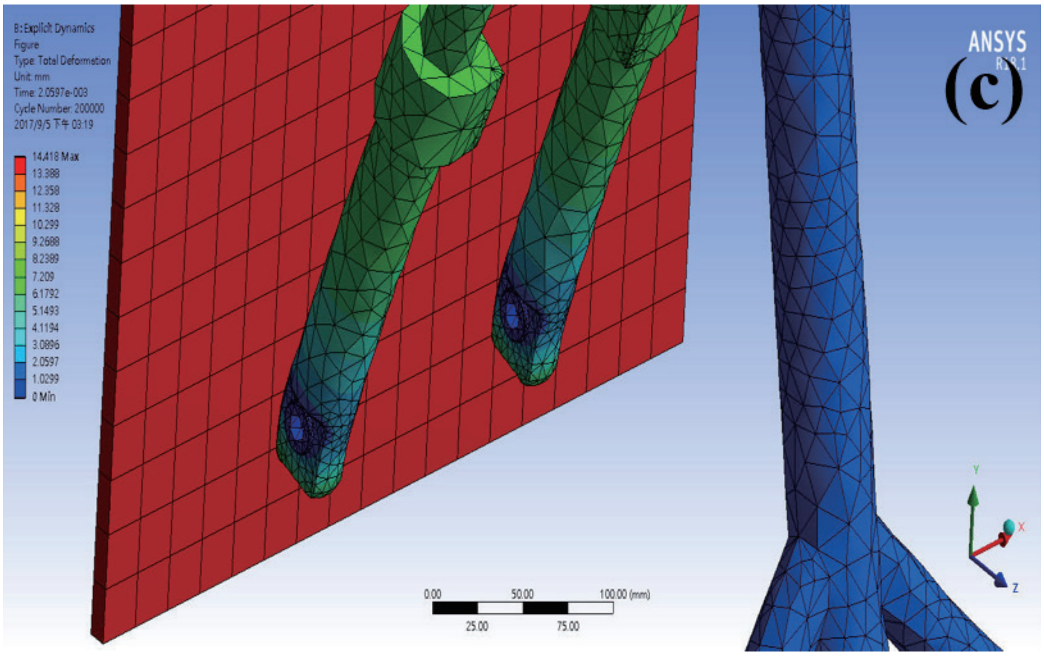

Fig. 6. (Color online) (Continued) (a) Transient analysis boundary conditions, (b) postcollision results, and (c) maximum deformation position after collision.

motorcycle regulations. The analysis results are shown in Figs. 6(b) and 6(c), with the maximum deformation occurring at the front shock absorber position. The use of transient dynamics modeling in this context implies that the simulation accounts for the dynamic response of the frame to the impact, taking into consideration factors such as inertia, forces, and the time-dependent behavior of the materials involved. The choice of steel as the collision material suggests a conservative approach, assuming a relatively rigid and unyielding impact. This choice might be appropriate for safety assessments and structural integrity evaluations. The impact velocity of 25.2 kilometers per hour aligns with the maximum speed specified in electric bicycle regulations, indicating that this simulation is designed to evaluate the frame's performance under realistic operating conditions. The identification of the maximum deformation occurring at the front shock absorber position is a critical finding. It implies that this part of the frame is the most vulnerable or sensitive to impact. Further analysis may be necessary to understand the implications of this deformation on the overall safety and functionality of the electric bicycle. Overall, this simulation provides valuable insights into how the frame behaves during a collision, aiding in design improvements or safety assessments of electric bicycles.

4. Conclusions

Analysis results indicated that convergence began when the number of grid elements approached approximately one million, with a calculation error of less than 3%. Therefore, 1.1 million was selected as the optimal grid size for this research analysis. The von Mises stress equation was employed and yielded the criterion to calculate the frame's stress. Simulation results revealed that the highest vertical bending moment occurred at the connection points between the main support shaft and the pedal base, with a maximum stress of 174.52 MPa recorded at these junctions. The position with the maximum deformation was observed at the two seat cushion support rods, where the deformation reached 13.689 mm. The maximum stress at these junctions was 91.641 MPa and was located at the connection point between the main support shaft and the pedal base. Additionally, the position with the maximum deformation was found at the faucet handle and front wheel support frame of the electric motorcycle frame, where the deformation amounted to 5.124 mm. When simulating an impact velocity of 25.2 kilometers per hour, the results indicated that this simulation was suitably designed to evaluate the frame's performance under realistic operating conditions. A critical finding was the identification of the maximum deformation occurring at the front shock absorber position.

Acknowledgments

The Works are Supported by the Great Project of Production, Teaching, Research of Fujian Provincial Science and Technology Department (2023H6025).

References

1 S. Rege, C. Khatri, M. Nandedkar, and N. Wagh: Int. J. Innov. Res. Sci. Eng. Technol. **6** (2017) 19500.
2 P. Jeyapandiarajan, G. Kalaiarassan, J. Joel, S. Rutwesh, F. T. Fastin, and B. Aditya: Proc. Mater. Today **5** (2018) 13563–13573.
3 P. Nigam, D. Sahu, and A. M. Bisen: Int. J. Eng. Res. Technol. **9** (2020) 527.
4 W. Sumbodo, R. Wahyudi, Setiadi, F. A. Kriswanto, and Budiman: J. Appl. Sci. Eng. **19** (2021) 948.
5 J. E. Rodriguez, A. L. Medaglia, and J. P. Casas: 2005 IEEE Design Symp., Systems and Information Engineering (Charlottesville, VA, USA, 2005) 277–285.
6 Vivek, L. Singh, and A. S. Johal: Int. J. Sci. Technol. Manag. **10** (2021) 7.
7 J. R. Choua and S. W. Hsiao: Int. J. Ind. Ergon. **35** (2005) 1047.
8 https://www.gazellebikes.com/en-gb/advice/electric-bike/sensors (accessed 10, 2023).
9 Y. J. Kim, N. Jeon, and H. Lee: Sens. Transducers **199** (2016) 67.
10 https://www.ezb2b.com/eng/v25-motorcycle-frame-body (accessed 10, 2021).
11 R. Kharul, S. Balakrishnan, D. Karedla, and S. S. You: SAE Int. J. Mater. Manuf. **3** (2010) 541.
12 H. M. Dai, B. H. Chen, C. M. Hsu, and C. F. Yang: Sens. Mater. **34** (2022) 1732.
13 D. Chindamo, M. Gadola, D. Armellin, and F. Marchesin: Appl. Sci. **7** (2017) 1220.
14 S. I. Cahyono, M. Anwar, K. Diharjo, T. Triyono, A. Hapid, and S. Kaleg: AIP Conf. Proc. **1788** (2017) 030084.
15 G. E. Roe and T. E. Thorpe: SAE Int. J. Passeng. Cars **98** (1989) 1319.
16 W. A. Siswanto and T. W. Besar Riyadi: Mater. Sci. Forum **961** (2019) 137.