Design and performance analysis of a retractable electric vehicle based on modern numerical simulation theory

Yuanyuan Zhang*, Jianghui Xin, Zhendong Zhao
Nanjing Institute of Technology, 1 Hongjing Ave., Jiangning District, Nanjing, Jiangsu, China 211167
Corresponding author: tangyuanyuan1214@163.com

Abstract. In view of environmental pollution, energy shortage and urban traffic congestion and other issues, a drawer type retractable electric vehicle (REV) is designed. Through the establishment of the frame and the vehicle's three-dimensional entity model, the frame of the static characteristics and modal characteristics are analyzed by using the finite element numerical simulation theory. The results show that the frame vertical stiffness is not reasonable, thereby improving the frame structure. Based on multi-body dynamics theory, the vehicle dynamics model is established, and the ride comfort of the vehicle is checked. The results show that the comfort of the vehicle is in line with the general requirements of the vehicle. The design and analysis method of this paper has enriched the design method of modern electric vehicle, which can embody the characteristics of light weight, clean energy, effective space utilization and so on, as well as has certain engineering application value.

1. Introduction
With people’s attention to the environmental and energy problems, electrification has become a mainstream trend to solve these problems. On the other hand, with the accelerated process of urbanization, the lack of parking spaces has become more and more prominent and become the factor that hinder the development of the city[1]. The folding car can not only solve these problems, but also improve the flexibility of the car. At present, the folding car can be divided into two types, one is chassis folding and another one is body folding[2].

As the assembling base of the car, frame not only used to support and connect automobile parts[3], but also bear various loads inside and outside the car as the main bearing component, and its performance directly decides the vehicle’s performance. Similarly, the frame of the retractable electric vehicle (REV) is not only the assembling base of the whole vehicle, but also the foundation to realize the telescopic function of the vehicle. So its performance is very important. CAE technologies like Finite Element Analysis (FEA) method and Multibody Dynamics theory are important tools for modern automobile design, and they are significant to improve the quality of automotive products, reduce developing & manufacturing cost and shorten the development cycle. During the research and development of the telescopic electric vehicle, the advanced 3D design software, Finite Element Analysis (FEA) method and Multi-rigid-body Dynamics theory are used to analyze the strength and stiffness of the frame, modal analysis and the vehicle’s riding comfort. They can also effectively improve the efficiency of structure improvement, and ensure the reasonability of the design.
2. Design of REV

While the body is folded, the change is the useful space of the body but not the attitude, and it will not affect the driver ride space and posture. Thus, before and after folding it can be run normally. So its could be applied in any length vehicle of four or more in theory. The telescopic vehicle has a wider application range and more development prospects.

2.1 Performance parameter design of REV

The size of the human body determines the space and range of activities, which is the basis of human computer systems or product design. According to the basic data of the static body size of the Chinese adult human dimension from GB10000-1988 (male aged 18-60 years, women aged 18-55), the length and width of the frame can be roughly determined. Reference to the overall size of electric vehicle parameters and the layout of battery, hydraulic systems and other components of the actual needs, the main performance parameters of REV are determined as:

1. Outline dimensions of vehicle: 2.8m×1.2m (elongated four seats); 2.2m×1.2m (shortened two-seats)
2. Vehicle elongation 2100mm; vehicle shorten the wheelbase 1500mm
3. Wheelbase: 1100mm

2.2 Structure design of REV frame

The frame adopts the flexible type like drawer, and the simplified schematic shows as figure 1, flexible principle is that the front wheels are fixed and the behind frame can slide in the front frame. While retractable push the behind frame to slide forward or backward by the hydraulic cylinder, the telescopic frame is realized, and the converging section will overlap together like a drawer.

Fig. 1 Schematic diagram of expansion and contraction

The frame of the telescopic electric vehicle adopts a steel tube frame structure, the main material is a square steel tube, and the three-dimensional solid model of the telescopic electric vehicle frame is shown in Figure 2.

Frame body can be divided into two parts before, after, which are connected by the hydraulic cylinder and provide power to realize the function of the telescopic frame. The straight roller bearings or POM plates are installed at between front and rear frame to reduce the sliding friction. At the same time, in order to enhance the elongate frame longitudinal stiffness, a clamping tenon is designed after elongation. The locking function of the clamping tenon and the hydraulic system could ensure the longitudinal strength of the vehicle frame. The specific structure of the middle part of the frames are shown in Figure 3 and 4.

Fig 2. The model of the folding state of the folding frame
2.3 **Design of Hydraulic system**

In order to realize the automatic extension of the front and rear of the frame, two hydraulic cylinders which symmetrical arrangement is added to provide power for the vehicle frame, the hydraulic cylinder and the back frame is connected, a piston rod is connected with the front frame, through the telescopic movement of the hydraulic cylinder is driven by the front and rear frame action, so as to realize the frame of the telescopic function.

In the actual process, due to the influence of many factors such as manufacturing errors by hydraulic cylinder bore, leakage, friction resistance is not equal, load distribution is not uniform, air mixed, it is difficult to get synchronous motion [4].

The hydraulic system with throttling synchronous circuit is adopted to avoid stuck when the front and rear frame suffer different resistance by the obstacles. And different flows of hydraulic cylinders are compensated. At the same time, the two-way telescopic synchronization of the two hydraulic cylinders are achieved by flow divider value, and the ride comfort is obtained.

3. **Finite Element Analysis of Frame**

3.1 **Statics performance analysis**

The static strength analysis of frame structure under the bending working condition and twisting working condition based on the finite element analysis method, which mainly includes two aspects, stiffness and strength analysis. It can provide a certain technical analysis basis for follow-up work [5].

As the scalable electric vehicle has special form of the body frame structure, which includes long and short two kinds of frame structure. But the two frame structures are both belong to the frame type. The frame structure type is changeless before and after scaling, and the integral stiffness is increased after the frame overlap. So the extension structure will be selected to the static performance of structure.

The main purpose of the simulation analysis is to calculated the size of the frame structure and the size of parameters such as displacement under fixed load and to analysis the deformation and the stress of the structure by external, so that to check the strength of the structure and optimize the structure.

3.1.1 **The establishment of the frame finite element model**

In order to reduce the complexity of the model, it is necessary to clear geometry including remove the installation hole, small rounded the geometrical characteristics of little impact on overall statics performance, when the model will be imported into of finite element software. The model need to be repaired, such as free edges’ suture, lack of the surface’s reconstruction.

The division of finite element mesh need to determine appropriate grid size, the frame adopts rules of square steel, and is the basic right Angle connection between square steel, according to the actual size of the square steel, 10 mm mesh size is determined to before and after frame respectively mesh, then the before and after frame is defined in accordance with the actual situation of connection, thus the simulation of real situation could be completed, the simulation accuracy could be improved.

Considering that this body is illustrated the type body, materials belong to thin-walled structure, its thickness is far less than its length, so the meshing with two-dimensional plate and shell element, it can directly reflect the vertical and horizontal beam connection, local reinforcement plate, various ancillary support, etc., and can be very image, a high precision reflect their true deformation and stress conditions.

Frame production materials belong to steel.
3.1.2 Treatment on Boundary condition

Weather statics analysis of boundary condition processing is reasonable, it is directly related to the correctness of the simulation results and accuracy. Bending condition is simulated in a horizontal well pavement, four-wheel landing uniform motion in a straight line at the condition of the frame for the load response. When bending condition analysis was carried out on the frame, it shall in its on the load and the weight of its own superior to a dynamic load coefficient. Experience, according to research on the frame to analyze bending condition, the maximum dynamic load coefficient is less than 2.5 best[6]. In this article, as a result of the chassis frame special fission structure, overall frame strictly checking conditions than normal, when bending condition analysis to select the dynamic load coefficient is 2.5.

Torsional condition is corresponding to the car across the steps or after potholes zone of the road, in the torsional conditions of dynamic load changing with time is very small, so the maximum dynamic load coefficient also need not too big, generally choose 1.3[6]. The simulation model is established, which is bound to the three directions of the left front suspension, the left rear suspension and the right rear suspension at the same time, and the Z direction displacement constraint is added -20mm at the position of the right front suspension at the same time.

The boundary condition of the frame is processed as follows (shown in Table 1):

(1) The gravity of the frame itself. By giving a density value to the vehicle frame, a gravity acceleration is applied in the direction of the vertical frame.
(2) The power battery pack and the weight of the hydraulic system are added to the frame in a concentrated mass.
(3) The weight of the occupant is uniformly loaded on the frame by means of a concentrated load.

In the simulation, in order to avoid the influence of the rigid body displacement of the frame, the ideal constraint is used to the frame, that is to skip the suspension and will be bound directly to the suspension and the frame of the connection. As shown in Table 2, the X is the frame horizontal, Y is the frame longitudinal and Z is for the vertical.

| Name of load     | Loading position                                      | Direction | Size  |
|------------------|-------------------------------------------------------|-----------|-------|
| weight of passengers | Pliotributed in four seat positions                    | -z        | 4x750N|
| weight of batteries | Installed in the rear frame batteey installation location | -z        | 500N  |
| weight of frame   | loaded with gravity acceleration                       | -z        | 2600N |

| Condition | Left front suspension | Right front suspension | Left rear suspension | Right rear suspension | Load coefficient |
|-----------|-----------------------|------------------------|----------------------|-----------------------|------------------|
| Bending   | X,Y,Z                 | X,Y,Z                  | X,Y,Z                | X,Y,Z                 | 2.5              |
| Torsion   | X,Y,Z                 | Z(-20mm)               | X,Y,Z                | X,Y,Z                 | 1.3              |

3.1.3 Results of static performance analysis

(1) Bending condition

The frame displacement contours are shown in Figure 5, Figure 6 and the maximum displacement of the frame in the frame of the central (i.e., front and rear frame intersection), and the maximum displacement volume 0.9306mm. The main reason is that the load (passenger weight) of the load is evenly loaded in the middle of the frame and the frame is not the whole (divided into two parts), therefore, there will be a large deformation of the middle of the frame phenomenon.
The stress of the frame image is shown in Figure 7 and the frame of the maximum stress for 75.33MPa, mainly in the right angle connection of front and rear suspension mounting frame structure and the middle of the frame and front and rear frame intersection junction. Material to make the frame is a hollow square steel, the material is 45 steel and the yield limit is 355MPa. Comparison shows that the maximum stress of the frame is less than the allowed material yield limit.

(2) Torsion condition

Torsion condition analysis results show that the maximal deformation displacement of the frame is 1.069mm and the maximum displacement is in the central of the frame (i.e., front and rear frame intersection). The maximum stress is 96.79MPa, mainly in the right angle connection of front and rear suspension mounting frame structure and the middle of the frame and front and rear frame intersection junction. Torsion condition of the maximum displacement and the maximum stress generation are the same of the bending condition under the torsional condition, although values are a litter bigger than bending condition, but in general meet the requirements, all in the allowable range.

From the comprehensive analysis of the frame under bending and torsion condition simulation analysis results we can see, the frame's stiffness and strength meet the requirements, and the car's stiffness and strength are much larger than the average car, but taking into account the telescopic car's special vehicle design, the frame is designed for the form of the front and rear of the frame, in order to ensure the telescopic function, the allowed deformation of the frame is smaller, otherwise it will be stuck when it works. So compared to the unitary frame, telescopic cars need larger stiffness and strength in order to ensure the realization of the function and cars' safety, so the car's stiffness and strength is qualified and reasonable.

3.2 Modal analysis

Modal analysis is the basis of dynamic analysis. It can grasp the influence of the frame to the exciting force through the modal analysis of the frame so as to find the weak link of the structure and provide the basis for the development of the vehicle.

Modal analysis is from the overall structure of the problem, it considers the larger factors of vibration in the analysis. The road and the power of the motor to the frame of the excitation will cause vibration. If the frame structure design is not reasonable, the frame will produce bending, torsion and other deformation. It directly affects the vehicle ride comfort and even its service life [6].

Because the frame structure of the front and rear of the telescopic vehicle is similar to the frame structure, the structure type has not changed, but the whole vehicle size has changed a little. It can be known that the overall stiffness of the frame will be increased and the natural frequency will become larger, so the finite element model of the vehicle frame is selected to establish the modal analysis, and the simulation analysis is carried out. Because the modal analysis does not need to load frame and this is the free modal analysis of the frame, we can establish the finite element analysis model of the free modal after removing via static analysis when applied loads and constraints. The calculation results omit frame rigid modal extraction, frame six mode, as shown in Table 3 and figure 7.
Table 3. Modal analysis results of frame

| Order | Modal frequency/Hz | Modal characteristics |
|-------|--------------------|-----------------------|
| 1     | 12.26              | Lateral bending       |
| 2     | 20.77              | Around the longitudinal twist |
| 3     | 26.08              | Transverse partial bending |
| 4     | 41.51              | Around the longitudinal twist |
| 5     | 51.59              | Back end winding      |
| 6     | 63.11              | The front end is twisted around the longitudinal axis and the back end of the winding shaft is curved |

In the actual process of vehicle driving, the external environment and the whole vehicle system will be excited to the frame, which will have a great impact on the performance of the frame. In this paper, the main analysis of the road, power motor and suspension of the frame of the incentive:

1. The excitation of road surface is mainly vertical vibration of 10Hz;
2. The natural frequency of the non-sprung mass is generally 6-15Hz[6];
3. The rated speed of the motor is 1500r/min, and the vibration frequency is 25Hz.

The modal simulation results show that the first order natural vibration frequency of the frame is in the natural frequency of the non sprung mass, the first two-order natural vibration frequency of the frame is in the vibration frequency of the motor. In order to avoid the resonance of the frame, the first mode frequency should be avoided in the external environment and the vehicle system may have the excitation frequency of the frame. Through the first and second order modal analysis, we can know that the first order vibration mode is along the frame lateral bending deformation; the second order vibration mode is along the longitudinal torsion deformation, in which the rear part of the frame is larger than the front part. Therefore, the longitudinal stiffness and lateral stiffness of the frame will be improved, the need for the rear of the frame to improve the design to optimize its performance.

![Frame first order mode shape](image)

4. Optimization and improvement to the frame

By analyzing the results of the simulation and the actual production process, it can be found that there are still some problems in the frame:

1. The frame of the first order natural frequency of vibration in the unsprung mass frequency band, first order and second order natural frequency are within the range of the vibration frequency of the motor and modal shape is mainly along the frame lateral bending deformation and deformation along the longitudinal torsion. It is known that the longitudinal stiffness of the frame and the lateral stiffness of the middle and rear of the vehicle need to be improved, and the rear part of the frame is improved;
2. The clearance between the cart telescopic special sub frame design, before and after the frame is not directly connected together, actual production process in the presence of errors will cause front and rear frame becomes large, will lead to the middle frame deformation quantity.

In view of the existing problems of the frame to make the improved design as shown in figure 8:

1. by adding 2 and 1 original longitudinal stringers, constitute a frame through the longitudinal axis of the main beam to strengthen frame longitudinal stiffness;
(2) the displacement and deformation of the beam 1 is further constrained by the extension of the rod 1 (that is, 2).

Fig. 8 Improved frame structure

Fig. 9 First order modal vibration mode of the frame

Modal analysis Results of the optimized model are shown in Table 4 and Figure 9, the first-order modal 27.61Hz avoid the outside and retractable car itself vibration frequency and reduce the possibility of resonance in the frame.

Table 4. Modal analysis results of the frame

| Modal order | Modal frequency/Hz | Modal characteristics          |
|-------------|--------------------|--------------------------------|
| 1           | 27.61              | Around the longitudinal twist   |
| 2           | 30.45              | Lateral bending                 |
| 3           | 41.97              | Front winding                   |
| 4           | 50.52              | Around the longitudinal twist   |

5. Simulation Analysis of vehicle ride comfortable

5.1 Flexible Vehicle Multi-body Dynamic Model

The process of multi body dynamics modeling is an appropriate abstraction and simplification of the vehicle. In order to make the simulation result consistent with the actual situation, we must study the restraint and movement of each component. The main technical data of REV is shown in Table 5.

Table 5. Main technical data of REV

| Parameter name     | data     | Parameter name     | data     |
|--------------------|----------|--------------------|----------|
| People in car      | 4        | curb weight (Kg)   | 400      |
| Front track (mm)   | 1100     | Wheelbase (mm)     | 2100 (1500) |
| Front tire radius (mm) | 250   | Rear wheel (mm)    | 1100     |
| Vehicle width (mm) | 1200     | Rear tyre radius (mm) | 250     |
| Vehicle length (mm) | 2800 (2200) | Maximum load capacity (Kg) | 300     |

Using constraints, deputy to conform to the actual situation of vehicle parts of the constraint, vehicle dynamics model is set up as shown in Figure 10.

Fig. 10 Multi-body dynamics model of flexible electric vehicle

The multi-body dynamics model of the flexible electric vehicle includes 36 dynamic parts. The crew, the battery and the hydraulic system weight, simplified to 7 point ball quality side is fixed on the frame. The constraint types and quantity are shown in Table 6.
Table 6. Constraint types and quantity

| Constraint type | quantity | Constraint type | quantity |
|-----------------|----------|-----------------|----------|
| Rotating        | 17       | Ball            | 19       |
| Fixed           | 6        | translational joint | 3       |
| Hooke           | 5        | Point line      | 2        |
| Vertical        | 1        | coupling        | 3        |

The multi-body dynamics model of the vehicle has 56 constraints, plus two drivers, a drive steering wheel, and another driving motor.

5.2 Methods for the evaluation of ride comfort

According to the relevant standards, the simulation test evaluation method of ride comfort is divided into basic evaluation method and auxiliary evaluation method:

(1) The basic evaluation method (the peak coefficient of vibration waveform is less than 9)

- Curve of acceleration time domain;
- The acceleration power spectral density is obtained by the fast Fourier transform;
- The mean square value of weighted acceleration is obtained;
- The calculated weighted vibration level;
- Based on the weighted RMS acceleration relationship between the subjective evaluation and evaluation, as in equation (1),

\[ a_w = \left( \int_{0.5}^{80} \omega^2(f)Ga(f)df \right)^{\frac{1}{2}} \]  

Vertical direction, \( \omega_k \) in equation (2),

\[ \omega_k = \begin{cases} 
0.5 & (0.5 < f < 2) \\
 f/4 & (2 < f < 4) \\
 1 & (4 < f < 12.5) \\
 12.5/f & (12.5 < f < 80) 
\end{cases} \]  

Horizontal direction, \( \omega_k \) in equation (3),

\[ \omega_k = \begin{cases} 
1 & (0.5 < f < 2) \\
2/f & (2 < f < 80) 
\end{cases} \]  

X, Y, Z total weighted acceleration root mean square value in equation (4),

\[ a_r = \left[ (1.4a_{xw})^2 + (1.4a_{yw})^2 + a_{zw} \right]^{\frac{1}{2}} \]  

Conversion formula of weighted vibration level \( L_{aw} = 20\log(a_w/a_0) \), \( a_0 = 10^4 \text{m/s}^2 \)

(2) Auxiliary evaluation method (peak coefficient is greater than 9)

The evaluation index is the root mean square value of 4, which can better evaluate the influence of the vibration caused by the bump on the road surface, as in equation (5),

\[ VDV = \left[ \int_0^T a^4_w(t)dt \right]^{0.25} / \text{ms}^{-1.75} \]  

5.3 Simulation of vehicle ride comfort

Combined with the flexible use of the environment for the road traffic in good condition of non-public transport road, focusing on the expansion of the former, after the B-grade Random Road on the ride comfort of the analysis.

The simulation time of ride comfort simulation is set to 30s, and the sampling frequency is 100Hz. The telescopic vehicle maximum speed is 35km / h, and Simulation of the vehicle is divided into
15km/h, 25km/h, 35km/h, three speed are analyzed. By using step function to drive the vehicle in the proportion 10~15s uniform accelerated motion.

The simulation show that the vertical acceleration of the 20~30s driver's seat is in the time domain curve (Figure 11).

![Fig. 11 Time curve of vertical acceleration on 35km/h](image)

![Fig.12 Power spectrum density curve of vertical acceleration on 35km/h](image)

Using fast Fourier transform of vertical vertical acceleration time history curve of spectrum analysis to the acceleration power spectral density curves (Figure 12), and then use the formula (1) to calculate the root mean square value of weighted acceleration, calculation results and the results of the evaluation are shown in Table 7.

It can be seen from the simulation results, the driver's seat, the weighted acceleration RMS value is less than 0.315 in the shortened state, and the other conditions are more than 0.315. The reason may be that the seat is not equipped with damping device. In addition, it is better to evaluate the comfort of the shorter time than the elongation. In order to verify the simulation results, carried out a real car test, from the test experience can be seen, up and down significantly, ride comfort is not very ideal, ride comfort is to be further improved.

| Speed (km/h) | Extension state | Shortened state |
|-------------|-----------------|-----------------|
| 15          | 0.58            | 0.29            |
| 25          | 0.61            | 0.43            |
| 35          | 0.9             | 0.59            |

### Table 7. Simulation results of ride comfort under random input

6. Conclusion

The characteristics of the existing folding vehicle structure are analyzed in this paper. Combined with the actual design requirements, a REV is designed, the vehicle is realized by the hydraulic telescopic, to achieve automation of vehicle expansion. The advanced three-dimensional design software and the finite element analysis method are make full use in the design of telescopic electric vehicle, the 3D entity model of vehicle is established, static characteristics (stiffness and strength) and modal characteristics are analysed on the frame, the weak part of the frame are found out and improved. Finally, multi-body dynamics model of the vehicle is established, simulation analysis on the vehicle's ride comfort, suggestions and improvement of telescopic electric vehicle comfort are evaluated.

7. References

[1] Francesco B and Manuel S, 2016, Driver's braking behavior approaching pedestrian crossings: a parametric duration model of the speed reduction times, *Journal of Advanced Transportation*. Papers, vol 50, pp: 630-46.

[2] Xinxing Yang, Lihua Feng, and Wei Peng, 2012, Automobile exhaust pollution and its harm, *Journal of Automobile Engineering*, vol 6, pp: 10-22.

[3] Dong Jianhua, Zhang Sanchuan, Chu Zheng and Guo Zhengyang, 2013, Mechanism design and analysis of super-micro foldable electric vehicle frame, *Manufacturing Automation*, vol 35, pp: 103-05.
[4] Martin Ranlöf, Johan Bladh and Urban Lundin, 2013, Use of a finite element model for the determination of damping and synchronizing torques of hydroelectric generators, *International Journal of Electrical Power & Energy Systems*, vol 44, pp: 844-851.

[5] Ermias Koricho and Giovanni Belingardi, 2015, An experimental and finite element study of the transverse bending behaviour of CFRP composite T-joints in vehicle structures, *Composites Part B: Engineering*, Vol 79, pp: 430-443.

[6] Serkan Sezen, Ercument Karakas, Kadir Yilmaz and Murat Ayaz, 2016, Finite element modeling and control of a high-power SRM for hybrid electric vehicle, *Simulation Modelling Practice and Theory*, vol 62, pp: 49-67.

**Acknowledgments**

This work is supported by the National Nature Science Foundation of China (51405221) , sponsored by Qing Lan Project. The authors would like to express appreciations for their assistance and attention.