Scholar Publishing Group
Kinetic Mechanical Engineering
https://doi.org/10.38007/KME.2020.010206
ISSN 2790-2676 Vol. 1, Issue 2: 45-53

New Energy Vehicle Development Strategy Integrated with Finite Volume Method

Nasser Liser*

Vytautas Magnus University, Lithuania

**corresponding author*

Keywords: Finite Volume Method, New Energy Vehicle, Genetic Algorithm, Frame Optimization

Abstract: As one of the important parts of the automobile, the frame bears the mass load of the cab, powertrain and all other parts, and bears the vibration, impact and distortion from the road under different working conditions. If it can meet the performance requirements of the frame, improve the utilization rate of materials and realize the lightweight of the frame, it will be very valuable for engineering use. This paper mainly studies the development strategy design of new energy vehicles integrated with finite volume method. Firstly, this paper analyzes the principle and application of finite volume method, establishes the finite element model of local topology optimization of electric vehicle frame in Hypermesh, and uses genetic algorithm to complete the multi-condition topology optimization of electric vehicle frame in Hypermesh. A three-dimensional model of small electric vehicle frame based on topology optimization results was constructed.

1. Introduction

Now people compared with traditional fuel vehicles to the requirement of pure electric vehicles, not only to meet the requirement of safety and handling stability, and also for the car comfort, lightweight, pollution index put forward higher request, in the past to the traditional design method of fuel car already cannot satisfy the requirement of the pure electric vehicles now use [1-2]. At present, the frame of electric vehicles only uses the old models and only changes the power system. However, due to the heavy battery weight of electric vehicles, the frame structure of traditional cars may not meet the new requirements of strength, stiffness and lightweight at the same time, so the frame structure has become the key to the research of electric vehicles [3]. Therefore, it is an important aspect of electric vehicle design and development to complete the frame structure matching with electric vehicle power system through advanced structural design methods. Different from the traditional frame, the new concept design can improve the handling stability, crash safety and ride comfort of electric vehicles, and maximize the lightweight of the frame structure to improve the endurance of electric vehicles [4]. Especially at the present stage when the development of battery technology still restricts the development of electric vehicles, improving the endurance capacity means that the whole vehicle cost can be significantly reduced, which has engineering application value and is conducive to the promotion of electric vehicles [5].

In the research and development process of new models, although the traditional conservative design method has a high safety factor, the frame is bulky, wastes energy and is not conducive to environmental protection. Meanwhile, the analysis of vibration and noise is limited to empirical calculation and experimental analysis [6]. However, with the development of finite element simulation analysis technology, finite element simulation analysis technology can be combined with vibration, noise, strength and other vehicle problems, which can greatly improve vehicle driving safety, riding comfort, environmental protection and other characteristics while reducing production costs [7]. The end of the 20th century, the researchers have begun to study of finite element technology, some scholars have vertical torsion static load, horizontal torsion for static load and dynamic load of the frame, put forward a hybrid analysis method, the finite element method (fem) ideal into a beam element and side beam section of beam element, simplifies the calculation process, at the same time, The scholar compared the analysis results with the experimental results and found that the calculated results were in good agreement with the experimental data [8]. In the 21st century, the development of finite element technology is becoming more and more mature, and the research on the application of finite element technology in the field of automobiles is becoming more and more mature [9]. Performance of some scholars for cars or light trucks and chassis to make use of the finite element analysis software ls-dyna based on nonlinear finite element analysis of dynamic stress, they use a virtual proving ground (VPG) method to obtain the frame displacement history and dynamic stress distribution, and combining the impact test, field test, the calculated results with the experimental results, The feasibility of the integrated life prediction method is verified [10].

Electric vehicles instead of ordinary fuel vehicles can make a great contribution to resource conservation and environmental protection, and the lightweight design of the frame can improve the performance of the whole vehicle, which has very good significance.

2. Optimization of New Energy Vehicle Frame Based on Finite Volume Element

2.1. Finite Volume Element

Finite element analysis method is an efficient and superior numerical calculation method. It develops with the development of computer technology and is gradually applied to various engineering fields, and the related finite element software is also constantly developed and promoted [11]. Ansys, Abaqus, Altair HyperWorks and other software are several popular finite element analysis software at present, and many companies as well as experts and scholars regard them as common tools to solve structural analysis problems [12]. HyperWorks is a more comprehensive CAE analysis software, excellent pre - and post-processing and design optimization function is its biggest characteristic, often used to deal with various structural performance analysis problems.

In essence, finite element analysis is a numerical analysis and calculation method, which is mainly used to solve continuous problems. With the rapid development of computer technology, finite element analysis technology is gradually developed, and used to deal with the analysis of various complex structures. Due to its high applicability, finite element analysis method has been widely used in various engineering practical fields, successfully solving a large number of relatively complex continuum problems and field problems, and promoting the rapid development of the engineering field [13]. It is difficult to numerically solve the continuum of mechanical parts with infinite degrees of freedom by traditional analytical methods. Finite element analysis is different. It divides the structure of the object under study into finite non-overlapping elements, and then connects these scattered finite elements in a certain way to re-simulate the structure of the object under study. Through this method, the original solution of the continuous problem with infinite degrees of freedom is transformed into the solution of the discrete problem with finite degrees of freedom [14]. Therefore, engineering practical problems that cannot be solved by numerical methods are reasonably simplified to structural problems that can be solved by numerical methods. Generally speaking, finite element analysis has its specific analysis process: Firstly, the research object is divided into finite non-overlapping units, which are connected with each other at their nodes. A structure is discretized into finite non-overlapping units, which are connected with each other at the nodes of the units. The displacement function at each unit node is represented by the displacement of the unit [15]. Then, based on the relationship among stress, strain and displacement and the principle of virtual work, the mechanical properties of the elements are analyzed one by one. Finally, the whole structural equations of the research object are established, and the final solution result is obtained by solving the equations [16].

2.2. Establishment of Finite Volume Metamodel for New Energy Vehicles

The frame parts of the new energy bus are basically rectangular steel, and the frame adopts a three-stage structure including the front, middle and rear frame. To late can accurately of the new energy bus frame finite element simulation analysis and structure optimization, the need to establish a mathematical model of the frame structure, the model must be complete and reflect the design parameters of bus frame, and so before finite element simulation analysis, must establish correct 3 d model for the passenger car chassis [17]. In the frame modeling stage, the model should be as close as possible to the actual prototype of the frame, and the parts that have less influence on the overall force of the frame can be simplified, so that the analysis results can not only maintain a high consistency with the real results, but also improve the computational efficiency. The geometric model of the bus frame structure can be directly established in the HyperMesh module of the finite element analysis software HyperWorks, but due to the complexity of the frame structure, this modeling method is inefficient, and the use of professional three-dimensional modeling software such as SolidWorks for modeling can save a lot of time [18].

Before establishing the finite element model of the new energy bus frame structure, it is necessary to simplify the bus frame.

The non-bearing components and welds of the frame are ignored, and these parts have little influence on the whole frame. Simplifying them can reduce the time needed to establish the finite element model of the frame to a certain extent, and make the model more concise. Ignoring the non-bearing components of the frame will lead to the deterioration of the mechanical properties of the frame and make the finite element analysis result of the frame larger, but the final analysis result is safer.

Some rounded corners and small holes of the frame are ignored, which have little influence on the actual force of the frame of the new energy bus, but will increase the number of grid of the bus frame, reduce the quality of the grid of the frame, resulting in distortion of the finite element analysis results.

In the area where the frame stress does not change much, the mesh size can be larger. In the area where the frame stress changes greatly and stress concentration is easy to occur, the grid size should be smaller as far as possible, and the grid should be smooth in the concentrated part. The STP format model was imported into SCDM to simplify the frame and extract the intermediate surface.

In finite element analysis, the contact setup can conduct normal and tangential forces between the contact surfaces of the two parts, so that the finite element model has the correct force conduction path. The contact between the rectangular steels of the frame of the new energy bus is set as linear contact, so the parts are bonded together by direct bias method.

Because the hard points will disturb the grid division, remove unnecessary hard points on the middle surface; The bolt hole is round, and when it is meshed, it is easy to make the inner Angle of the surrounding grid too large or too small. Therefore, Washer is set up on the outside of the bolt hole, which is convenient to divide the mesh of high quality and can avoid stress concentration. Use free edges to divide the irregular geometry properly, so as to form a good mesh later.

The quality of finite element mesh of bus frame has an important influence on the accuracy of finite element analysis results. The Quality Index of HyperMesh quality inspection panel can be used to detect the mesh quality. edges of HyperMesh, duplicates icates, qualityindex and other commands are used to suture discontinuous mesh, delete repeated mesh and optimize mesh. Finally, the number of failed grids was 0, the Comp.QI value was 0.01, and the proportion of cells with good quality was 99.9%.

3. Optimization Design of Vehicle Frame Reliability Based on Genetic Algorithm

The objective function of the frame is the minimum mass (or volume) of the frame and always greater than 0. In this case, the fitness function is:

$$Fit(W(d)) = W(d) \tag{1}$$

In the early stage of evolution, the reliability of some individuals may be difficult to meet the target reliability. Therefore, this paper uses the penalty function method to add a penalty term to those individuals who cannot meet the reliability requirements. In this case, the fitness function can be written as follows:

$$Fit^{new}(W(d)) = (1 + v \cdot F) Fit(W(d)) \tag{2}$$

Where, v is the penalty function index. If an individual does not meet the reliability requirement, then v=1, otherwise v=0; F is a big enough integer.

The fitness values of all individuals are sorted in contemporary times, and the individual with the smallest fitness value is the optimal individual. The fitness values corresponding to the optimal individual and the position of the group they belong to are recorded and retained to contemporary times.

In this paper, the frame structure is based on roulette to select individuals in the group. Specifically, the proportion of the fitness value of each individual in the current population and the sum of the fitness value of the whole population is first solved. Secondly, the probability that the corresponding individual can be selected is determined based on this proportion. In general, the smaller the proportion of individuals, the more likely they are to be selected. In the roulette method, the specific operation is as follows: Assuming that there are n individuals in the whole group, and the fitness value of the ith individual is Fitnew, the probability that this individual can be selected is:

$$P_i = 1 - \frac{Fit^{new}(W(d_i))}{\sum_{i=1}^{n} Fit^{new}(W(d_i))}, (i = 1,2,\ldots,n) \tag{3}$$

4. Analysis of Experimental Results

4.1. Frame Reliability Design Results

The reliability optimization design of the frame structure is carried out, and the frame parts with large mass are selected as the design variables. The final optimization plate thickness of the frame reliability optimization design is shown in Table 1.

Table 1. Reliability optimization frame final optimization plate thickness

	1	2	3	4	5
Initial thickness	7	5	5.2	6	4.6
Optimization of thickness	7.03	5.6	4.8	5.31	4.94
Final value	7	5.6	4.8	5.3	4

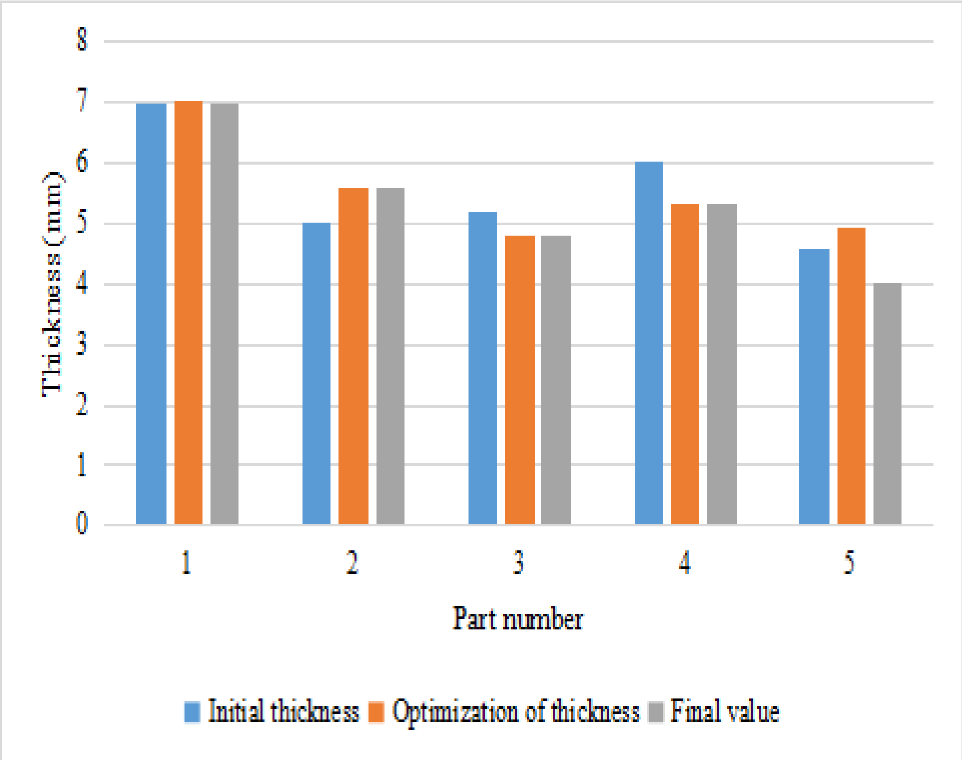

Figure 1. Finally optimized plate thickness results

As shown in FIG. 1, it can be seen that the optimization value of component 1 has little change and the initial plate thickness is adopted, while the other components have been optimized to different degrees.

4.2. Deterministic Optimization of Frame

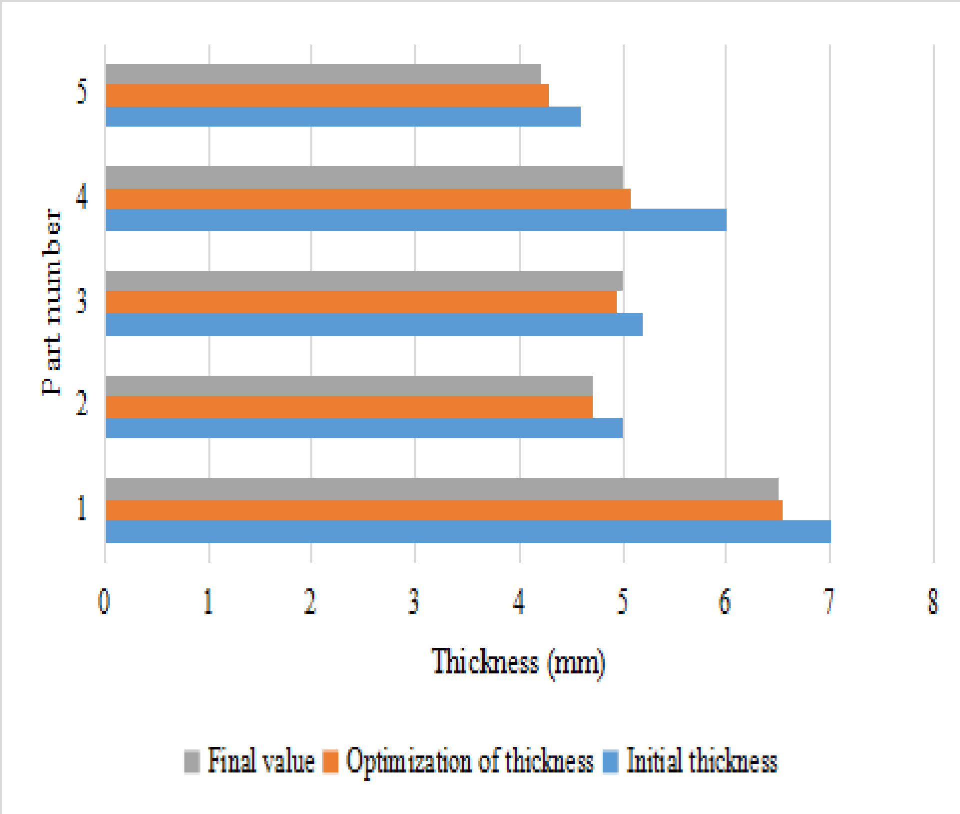

Figure 2. Deterministic optimization of frame finally optimized plate thickness

Firstly, the plate thickness of five frame parts is taken as the design variable. Secondly, the allowable stress of four typical working conditions is 192MPa as the constraint condition. Thirdly, the objective function is to minimize the total mass of the frame, and the constraint condition and objective function are the response of the optimization problem. Finally, the deterministic optimization of the frame is completed. The final frame design variable plate thickness obtained through deterministic optimization design is shown in FIG. 2.

4.3. Comparison between Reliability Optimization and Deterministic Optimization

Table 2. Comparison of deterministic optimization and reliability optimization results

	Weight loss/kg	Percentage of weight loss /%	Reliability of bending	Reliability of braking
Deterministic optimization	4.87	2.97	0.986	0.981
Optimization of reliability	4.32	2.64	0.952	0.960

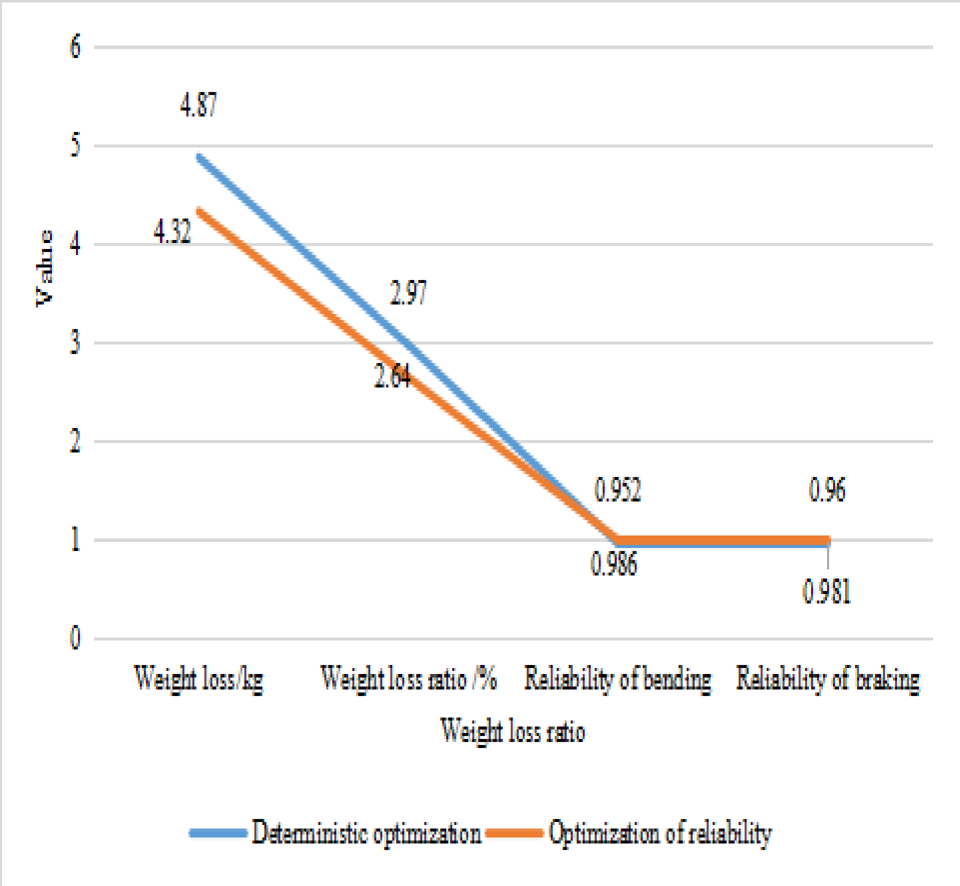

Figure 3. Comparison of results of different optimization modes

It can be seen from Table 2 and Figure 3 that the frame weight is reduced by 4.532kg with a weight loss ratio of 2.52% after reliability optimization, and 4.87kg with a weight loss ratio of 2.97% after deterministic optimization. In addition, the reliability of the two working conditions is greater than 0.98, so both meet the reliability requirements. In general, although the weight loss of reliability optimization is not as good as that of deterministic optimization, the reliability of the frame in torsion condition using deterministic optimization method does not meet the requirements. Therefore, after the reliability optimization design, the frame not only improves the reliability to meet the safety requirements, but also reduces the quality to meet the lightweight requirements.

5. Conclusion

In this paper, the finite element analysis and structural optimization of the new energy bus frame are carried out for the strength, stiffness, vibration resistance and fatigue durability requirements of the new energy bus frame. Firstly, the application of finite element analysis technology in frame performance analysis is summarized. According to the results of finite element analysis, the multi-stiffness topology optimization of bus frame is realized. According to the results of topology optimization, the frame is improved twice to improve the bearing capacity of the frame on the basis of satisfying the strength, stiffness, vibration resistance and fatigue durability. Due to the limitations of my knowledge, time and research conditions, the content of the study is still imperfect, and the work to be further studied mainly includes: further improvement of the frame is needed based on the results of random vibration analysis and frame life fatigue analysis.

Funding

This article is not supported by any foundation.

Data Availability

Data sharing is not applicable to this article as no new data were created or analysed in this study.

Conflict of Interest

The author states that this article has no conflict of interest.

References

[1] Chen J, Kwak Y, Xu M, et al. Topology and modular size optimization of small electric vehicle frame based on cross-section contribution analysis. Structural and Multidisciplinary Optimization, 2020, 64(6): 4287-4304. https://doi.org/10.1007/s00158-021-03075-y

[2] Navale A B, Chippa S P, Chougule D A, et al. Crashworthiness aspects of electric vehicle design. International journal of crashworthiness, 2020, 26(4): 368-387. https://doi.org/10.1080/13588265.2020.1718462

[3] Blatnický M, Sága M, Dižo J, et al. Application of light metal alloy EN AW 6063 to vehicle frame construction with an innovated steering mechanism. Materials, 2020, 13(4): 817. https://doi.org/10.3390/ma13040817

[4] Ali S M N, Hossain M J, Wang D, et al. Robust sensorless control against thermally degraded speed performance in an im drive based electric vehicle. IEEE Transactions on Energy Conversion, 2020, 35(2): 896-907. https://doi.org/10.1109/TEC.2020.2968547

[5] Teli M D, Chavan U S, Phakatkar H G. Design, analysis and experimental testing of composite leaf spring for application in electric vehicle. Int. J. of Innovative Technology and Exploring Engineering (IJITEE), 2019, 8(9): 2882-2891. https://doi.org/10.35940/ijitee.I8744.078919

[6] ALVALI G T, Balbay A, Güneş S, et al. Design and implementation of a three-wheel multi-purpose electric vehicle with finite elements analysis. International Journal of Automotive Science and Technology, 2020, 5(2): 116-125. https://doi.org/10.30939/ijastech..871684

[7] Ebrahimi J, Salari O, Eren S, et al. Efficiency improved multi-source inverter for hybrid energy storage systems in electric vehicle application. IEEE Transactions on Power Electronics, 2020, 37(2): 1982-1997. https://doi.org/10.1109/TPEL.2020.3104759

[8] Sharma S, Obaid A J. Optimal Design, Simulation and implementation of Solar Photo-voltaic Panels in Hybrid electric vehicles using CATIA V5R19 software integrated with ANSYS 13.0 versions//Journal of Physics: Conference Series. IOP Publishing, 2020, 1530(1): 012124. https://doi.org/10.1088/1742-6596/1530/1/012124

[9] Rezvani Z, Jansson J, Bengtsson M. Consumer motivations for sustainable consumption: The interaction of gain, normative and hedonic motivations on electric vehicle adoption. Business Strategy and the Environment, 2018, 27(8): 1272-1283. https://doi.org/10.1002/bse.2074

[10] Haruna J O, Sanni S O. Analysis and control of offboard bidirectional plug-in electric vehicle (PEV) charger for vehicle to grid operation. Nigerian Journal of Technology, 2018, 37(4):

1083-1086. https://doi.org/10.4314/njt.v37i4.30

[11] Carstensen C, Dond A K, Nataraj N, et al. Three first-order finite volume element methods for Stokes equations under minimal regularity assumptions. SIAM Journal on Numerical Analysis, 2018, 56(4): 2648-2671. https://doi.org/10.1137/17M1134135

[12] Rachid A, Bahaj M, Fakhar R. Finite volume element approximation for time dependent convection diffusion reaction equations with memory. Computational Methods for Differential Equations, 2020, 9(4): 977-1000.

[13] Camargo J T, White J A, Borja R I. A macroelement stabilization for mixed finite element/finite volume discretizations of multiphase poromechanics. Computational Geosciences, 2020, 25(2): 775-792. https://doi.org/10.1007/s10596-020-09964-3

[14] Melvin T, Benacchio T, Shipway B, et al. A mixed finite‐element, finite‐volume, semi‐implicit discretization for atmospheric dynamics: Cartesian geometry. Quarterly Journal of the Royal Meteorological Society, 2019, 145(724): 2835-2853. https://doi.org/10.1002/qj.3501

[15] Cardiff P, Demirdžić I. Thirty years of the finite volume method for solid mechanics. Archives of Computational Methods in Engineering, 2020, 28(5): 3721-3780. https://doi.org/10.1002/qj.3501

[16] Behrens B A, Dröder K, Hürkamp A, et al. Finite Element and Finite Volume Modelling of Friction Drilling HSLA Steel under Experimental Comparison. Materials, 2020, 14(20): 5997. https://doi.org/10.3390/ma14205997

[17] Hodges B R. Conservative finite-volume forms of the Saint-Venant equations for hydrology and urban drainage. Hydrology and Earth System Sciences, 2019, 23(3): 1281-1304. https://doi.org/10.5194/hess-23-1281-2019

[18] Coulet J, Faille I, Girault V, et al. A fully coupled scheme using virtual element method and finite volume for poroelasticity. Computational Geosciences, 2020, 24(2): 381-403. https://doi.org/10.1007/s10596-019-09831-w