Finite Element Analysis of Joint in Prefabricated Carbon Fiber Composite Truss

Yuxuan Gao *

Department of Chang'an Dublin International College of Transportation, Chang'an University, xi'an, China

* Corresponding Author Email: 2022902940@chd.edu.cn

Abstract. Carbon fiber composite truss, as an innovative structural form, has been widely used in the construction industry. However, the joint is serving as a crucial component of the truss structure construction. In order to improve the design level of the truss, this paper mainly studies the prefabricated carbon fiber composite truss joints. Firstly, the finite element analysis method is used to simulate the truss joint, and the stress nephogram under different geometric conditions is obtained. Then, the stress deformation mode and stress distribution mode are analyzed in depth. The calculation results are compared with similar steel pipe truss joints, and the accuracy of stress distribution is verified. Finally, the relationship between stress concentration area, stress concentration degree, maximum stress size and geometrical parameters of joint is revealed. Results in this paper can guide the design of this kind of truss structures, which can help develop more efficient and durable construction projects

Keywords: CFRP; truss, joint; ABAQUS; Finite Element Analysis.

1. Introduction

Truss structures exhibit high material utilization rates, rational load distribution, lightweight construction, and easy assembly, making them extensively employed in aerospace, construction, machinery, and other fields. Traditional trusses predominantly adopt steel structures; however, carbon fiber composite materials (CFRP) offer numerous advantages over traditional steel due to their lightweight nature, high strength properties, corrosion resistance capabilities, heat resistance characteristics, and small thermal expansion coefficients. Consequently, carbon fiber truss structures have emerged as a prominent area of research owing to the aforementioned benefits. In specific domains such as deep-sea exploration and aerospace applications where unique requirements exist, carbon fiber truss structures are gradually replacing conventional steel trusses.

The joint connection within a truss is considered the most critical component since it involves various processing methods that can be relatively challenging with unstable performance outcomes significantly impacting the overall integrity of the structure. Composite truss joints are primarily categorized into metal joints and composite joints while exhibiting distinct geometric shapes based on different types of trusses.

According to the current research status, Fox and Swaim conducted experimental analysis on the static strength characteristics of bolts [1]. McCarthy et al. [2] extensively investigated the clearance of bolt holes and analyzed its impact on various factors such as multi-nail load distribution and extrusion strength. AlaatinAkt examined the influence of layup sequence on bolt hole strength [3], while Gu conducted comprehensive research on factors affecting mechanical strength in composite joints [4]. Yan conducted research on load-free division algorithms for complex joints [5]. However, there is limited research available in related fields concerning prefabricated carbon fiber composite truss joints.

Therefore, this paper focuses on analyzing the stress of joints in carbon fiber composite trusses. Firstly, a joint model for CFRP trusses was established using shell elements in the finite element analysis software ABAQUS. The influence of different geometric parameters on the mechanical properties of the joint was analyzed through single parameter analysis. Then stress cloud maps and

load displacement curves were drawn and analyzed to determine how these geometric parameters affect the performance of CFRP truss joints.

2. Methodology

2.1. Purpose of Research

The joint is the most critical component in the composite truss, characterized by significant processing difficulty and unstable performance. This paper focuses on studying K-shaped joints in prefabricated carbon fiber composite plane trusses. Finite element analysis models are established using shell elements to investigate the influence of different geometric parameters on structural stability bearing capacity:

(1) The data and finite element simulation results are analyzed to understand the stress distribution law of nodes at each stage of test loading.

(2) The data and finite element simulation results are analyzed also to conduct a detailed study of the test scheme, device, process, test phenomenon, and results while combining previous tests, and find some method to improve the design CFRP truss.

(3) The data and finite element simulation results are analyzed and compared to verify the applicability of this finite element method to such joints.

2.2. Method of 3D Modeling

The units commonly used in ABAQUS 3D modeling are categorized into solid, beam, shell, and point units. The selection of different units for modeling should be based on the specific characteristics of the objects being numerically simulated in order to optimize computational efficiency and achieve more accurate results. In this study, shell element modeling is predominantly employed. To analyze force conditions under various geometric parameters, multiple finite element analysis simulations need to be conducted. Adopting solid unit modeling would result in prolonged analysis time, increased complexity, and a significant investment of manpower and time resources [6]. When the thin wall thickness of a structure is much smaller than its overall size and there is negligible stress perpendicular to the thickness direction or it can be disregarded entirely, shell elements can effectively simulate such structures. In ABAQUS, assigning cross-section properties to a component enables easy adjustment of shell element thicknesses. This greatly facilitates analyzing wall thickness ratios and diameter-to-thickness ratios for different branches and main pipes.

Shell elements have been widely utilized in numerical simulation experiments involving steel truss joints; moreover, carbon fiber composite materials possess superior strength characteristics with trusses made from these materials having thinner tube walls that are better suited for shell element modeling purposes. Overall, utilizing shell element modeling not only simplifies the simulation process but also has minimal impact on experimental accuracy. This approach forms the primary focus of this paper. The meshing is shown in Fig. 1.

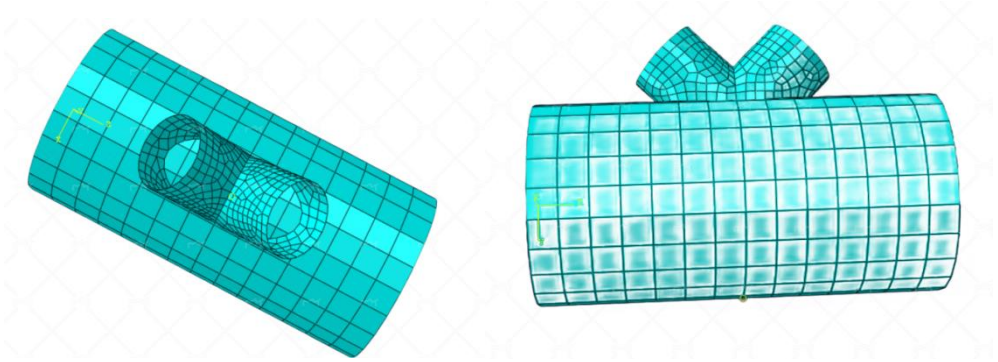


Fig 1. The Meshing.

In terms of load application, this paper mainly applies the boundary load of the shell on the two branches of the pipe. And in terms of boundary conditions, this paper adopts the boundary condition of completely fixing both sides of the bamboo pipe. The load and constraint are shown in Fig. 2.

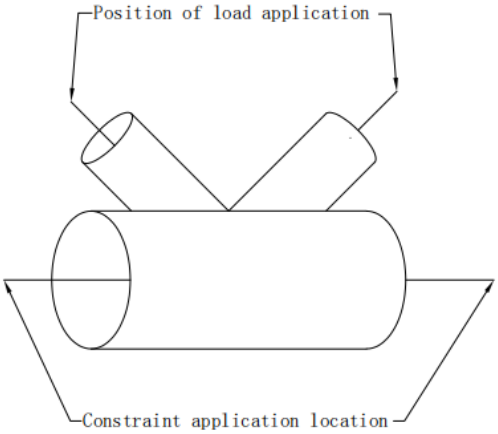


Fig 2. Load and Constraint.

2.3. Method of Research

In terms of the analysis method, this paper primarily employs a single-parameter analysis approach. This involves systematically varying one parameter at a time while keeping all other parameters constant, thereby examining its influence on the research object. The rationale behind adopting this methodology lies in the existence of numerous geometric parameters that potentially impact the mechanical properties of joints. By studying these parameters individually, any confounding effects from other variables can be avoided.

Specifically, this study focuses on analyzing how variations in the main outer diameter (D) and wall thickness (T), as well as branch outer diameter (d) and wall thickness (t), affect joint performance. To facilitate comparison and investigation, these influencing factors are transformed into dimensionless geometric parameters denoted by D and T for the main pipe's outer diameter and wall thickness respectively, while d and t represent those for the side pipe. Variable that needed is shown in Table 1 [6].

Table 1. The calculation formula of each variable.

Outer diameter ratio of side pipe to main pipe	Outer diameter to thickness ratio of side pipe	Outer diameter to thickness ratio of main pipe	Angle between main and side pipe
A=d / D	B=t / T	C=D / T	θ

2.4. Material Property

The density, elasticity-related properties, and failure mode of the carbon fiber material used in the ABAQUS have been defined. The elastic type is Engineering Constants. The type of damage is Hashin damage. The specific data for density equals to 50 kg/m^3, while the remaining details can be found in Tables 2-4 [7-9].

Table 2. Parameter of Elastic

Properties	E_1 (Gpa)	E_2 (Gpa)	E_3 (Gpa)	Nu_{12}	Nu_{13}	Nu_{23}	G_{12} (Gpa)	G_{13} (Gpa)	G_{23} (Gpa)
Value	127	10.3	10.3	0.3	0.3	0.3	5.7	5.7	3.2

Table 3. Parameter of Hashin Damage.

Properties	X_T (Mpa)	X_C (Mpa)	Y_T (Mpa)	Y_C (Mpa)	S_L (Mpa)	S_T (Mpa)
Value	2070	1360	102	276	186	186

Table 4. Damage Revolution.

Properties	G_{XT} (kJ/m^2)	G_{XC} (kJ/m^2)	G_{YT} (kJ/m^2)	G_{YC} (kJ/m^2)
Value	218	104	1.7	2.0

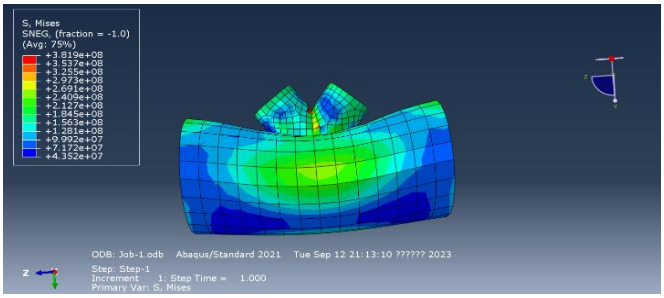
3. Result and Discussion

3.1. Outer Diameter Ratio of Side Pipe to Main Pipe (A)

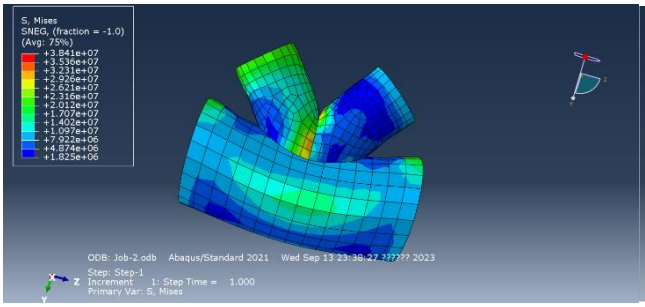
The first step is to analyze the nodes with different outer diameter ratios of main side pipes. A series of experimental groups are set up in this paper, and the parameters are shown in Table 5, and all of the outer diameter equals to 0.1m. The stress cloud diagram of joint with different ***A*** is shown in Fig. 3.

Table 5. Parameters of Different ***A***.

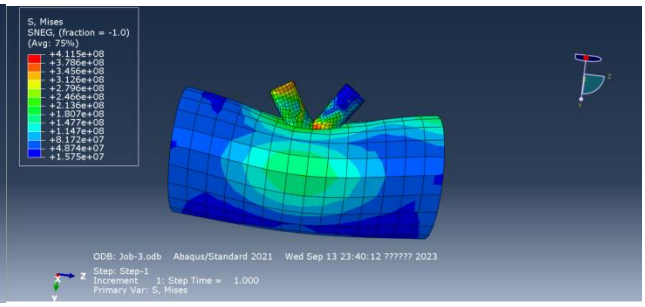
A	Thickness of side pipe (m)	Thickness of main pipe (m)	The magnitude of Total force act on the side pipe (N)	The magnitude of the uniform load (N*m^-1)	Outer diameter of side pipe
5:2	0.01	0.01	1e7(0.004*2pi)	1e7N/m	0.04
5:3	0.01	0.01	6.67e6/(0.006*2pi)	6.67e6N/m	0.06
5:1	0.0	0.01	2e7/(0.008*2pi)	2e7N/m	0.02



(a) ***A*** =5:2



(b) ***A*** =5:3



(c) ***A*** = 5:1

Fig 3. The Stress Cloud Diagram of Joint with Different ***A***

Focusing on stress, it can be seen from the stress cloud diagram analysis of the different outer diameter ratio of the main side pipe that the outer diameter ratio of the main side pipe is for. It can be seen from the stress cloud diagram analysis of the different outer diameter ratio of the main side pipe that the external diameter ratio of the main side pipe has not a great influence on the stress concentration position in the truss structure, but to a certain extent, it affects the stress concentration degree of the main branch. And it can be seen that the diameter of the side pipe is larger relative to the main pipe. The stress distribution is more concentrated but the maximum stress is smaller.

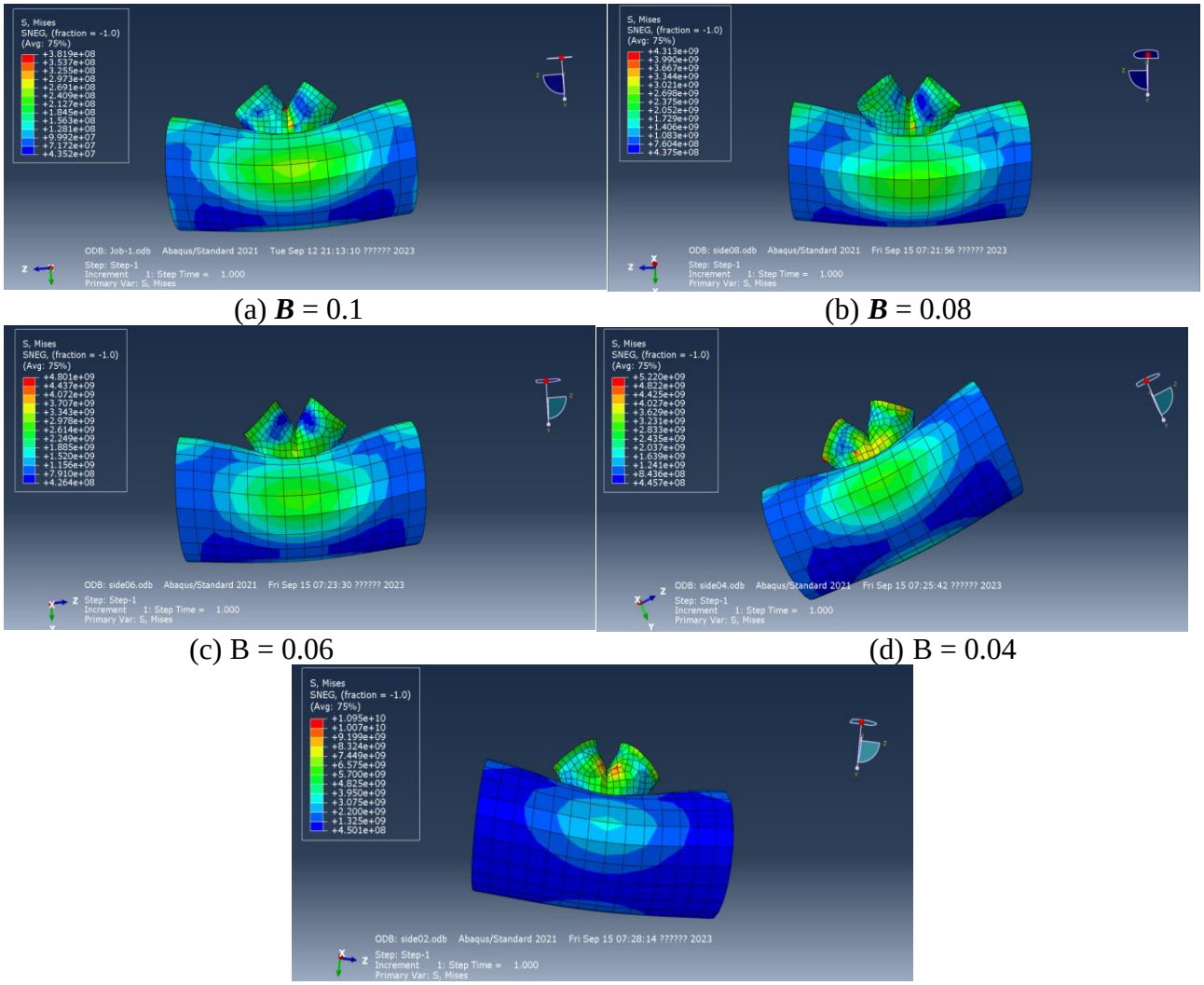
In terms of deformation, the node with a larger outer diameter ratio of the main pipe and side pipe has a smaller deformation degree of the main pipe. The reason for this phenomenon may be that the node with a larger outer diameter of the side pipe has a larger area of the main pressure area. The branch deformation is not affected by this parameter

3.2. Outer Diameter to Thickness Ratio of Side Pipe (B)

The second step is to analyze the nodes with different Outer diameter to thickness ratio of side pipe. A series of experimental groups are set up in this paper, all the outer diameter of main pipe equals to 0.1 m and all outer diameter of side pipe equals to 0.04m. And the other parameters are shown in the Table 6. The stress cloud diagram of joint with different B is shown in Fig. 4.

Table 6. Parameters of Different ***B***

A	Thickness of side pipe (m)	Thickness of main pipe (m)	The magnitude of Total force act on the side pipe (N)	The magnitude of the uniform load (N*m^-1)	***C***
5:2	0.01	0.01	1e7/(0.004*2pi)	1e6	0.1
5:2	0.01	0.008	1e7/(0.004*2pi)	1e6	0.08
5:2	0.01	0.006	1e7/(0.004*2pi)	1e6	0.06
5:2	0.01	0.004	1e7/(0.004*2pi)	1e6	0.04
5:2	0.01	0.002	1e7/(0.004*2pi)	1e6	0.02



(a) ***B*** = 0.1 (b) ***B*** = 0.08

(c) B = 0.06 (d) B = 0.04

(e) B = 0.02

Fig 4. The Stress Cloud Diagram of Joint with Different ***B.***

According to the analysis of the output stress cloud image, the larger the diameter to thickness ratio of the side pipe is, the larger the stress distributed on the main pipe is, but the stress is more dispersed, but the deformation degree of the main pipe is not affected by this value. The stress on the side pipe is smaller, and its deformation degree is also smaller, but the stress distribution concentration area is unchanged.

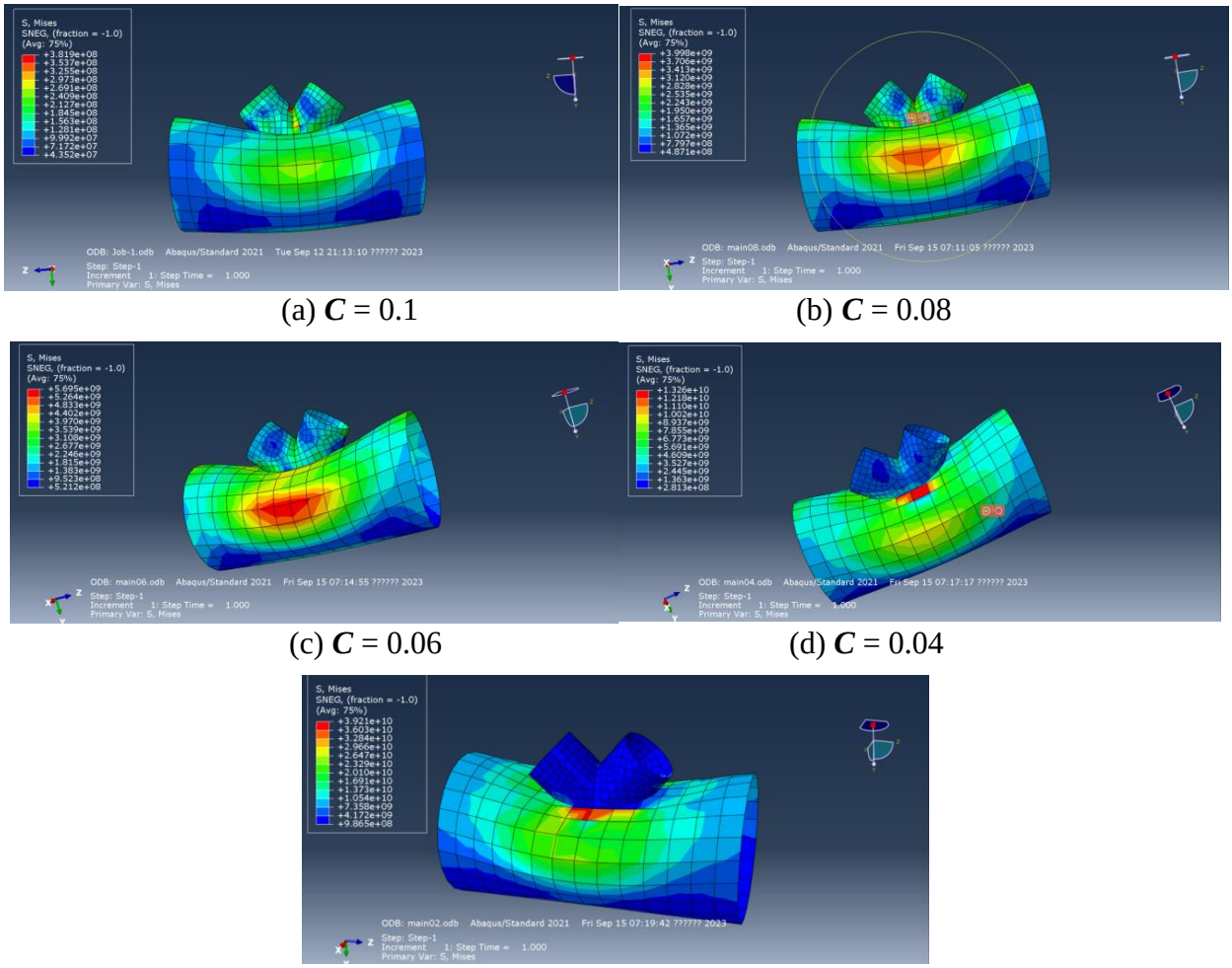
In terms of deformation, the size of the diameter to thickness ratio of the side pipe has no obvious effect on the deformation degree of the main pipe, while for the side pipe, a smaller diameter to thickness ratio will significantly aggravate the deformation, which also verifies the previous guess of the reason for the small deformation degree of the side pipe.

3.3. Outer Diameter to Thickness Ratio of Main Pipe (C)

The third step is to analyze the nodes with different Outer diameter to thickness ratio of side pipe. A series of experimental groups are set up in this paper, and all of the outer diameter of main pipe equals to 0.1 m and all of outer diameter of side pipe equals to 0.04m. And the other parameters are shown in Table 7. The stress cloud diagram of joint with different C is shown in Fig. 5.

Table. 7 Parameters of Different ***C***

A	Thickness of side pipe (m)	Thickness of main pipe (m)	The magnitude of Total force act on the side pipe (N)	The magnitude of the uniform load (N*m^-1)	***C***
5:2	0.01	0.01	1e7/(0.004*2pi)	1e6	0.1
5:2	0.008	0.01	1e7/(0.004*2pi)	1e6	0.08
5:2	0.006	0.01	1e7/(0.004*2pi)	1e6	0.06
5:2	0.004	0.01	1e7/(0.004*2pi)	1e6	0.04
5:2	0.002	0.01	1e7/(0.004*2pi)	1e6	0.02



(a) ***C*** = 0.1 (b) ***C*** = 0.08

(c) ***C*** = 0.06 (d) ***C*** = 0.04

(e) ***C*** = 0.02

Fig 5. The stress cloud diagram of joint with different *C*

Focusing on stress, the stress cloud diagram analysis reveals that as the ratio of diameter to thickness of the main pipe increases, the stress concentration area shifts from the side pipe to the abdomen of the main pipe wall. Consequently, stress becomes more dispersed maximum stress values

and reduced stress distribution on the side pipe. This parameter significantly influences stress distribution but has minimal impact on strain in both the main pipe and its branches.

In terms of deformation, the node with a larger diameter and thickness of the main pipe has a smaller deformation degree, and the side pipe does not show obvious deformation in this part of the test, the reason is that the side pipe here has a larger diameter and thickness ratio, which is not easy to deform. In addition, it can also be judged that the diameter thickness ratio of the main pipe has little effect on the deformation of the side pipe.

3.4. Stress Distribution

In terms of stress distribution, no matter what parameter, the distribution pattern is similar to some extent. The main stress is mainly concentrated in the side wall and the bottom, and the stress distributed in the side wall is greater than the bottom. On the side pipe, the stress distributed near the node and the applied load is larger, and the middle stress appears a valley value.

According to the loading experiment of the steel pipe truss by Gong Song et al., the failure position of the steel pipe truss during loading is near the joint of the side pipe. In this respect, the performance of the CFRP pipe is the same as that of the steel pipe truss. However, in the experiment of Gong Song et al., there was no obvious deformation of the main pipe, which was different from the experiment in this paper. According to 3.1, the deformation of the main pipe of the node with a larger branch wall thickness was less obvious [10].

4. Conclusion

In this study, a series of truss joints with different geometric characteristics were established by using ABAQUS finite element simulation and analysis software, and a single parameter analysis method was used to calculate the static forces of all the models, and the conclusions were compared with those obtained by predecessors. The following conclusions are obtained:

(1) Due to the stress zone in the side pipe near the main pipe, the main pipe belly and the bottom of the main pipe, reinforcement should be considered at these places during truss timing.

(2) Revised sentence: Increasing both the main pipe size and the diameter-to-thickness ratio can effectively reduce node deformation. Additionally, these parameters contribute to reducing excessive local stress and promoting a more uniform stress distribution at joints outer diameter ratio of the main side pipe not only affects deformation but also influences stress distribution. A smaller outer diameter difference between pipes can minimize maximum stress and deformation.

(3) Only K-type joints were considered in this experiment, while other types of joints were not analyzed. This study draws conclusions solely based on numerical simulation analysis and data obtained from similar experiments conducted by predecessors, without experimental verification. Further research is needed.

References

[1] Fox D E, Swam K W. Static Strength Characteristics of Mechanically Fastened Composite Joint NASA/TM-1999-209735

[2] McCarthy C T, McCarthy M A. Three-dimensional finite element analysis of single-bolt, single-lap composite bolted joints, 2005, 71(2), 159-175.

[3] Alaattin Aktas. Bearing strength of carbon epoxy laminates under static and dynamic loading. Composite Structures, 2005, 67(4): 495-489.

[4] Gu Yilei. Experimental Study on Extrusion Strength of Composite Connectors containing Delamination. Mechanical Engineering Materials, 2000, 24(4): 4-7, 17.

[5] Yan Guoliang. Research on Load Distribution Engineering Algorithm for Mechanical Connection of Complex Joints. Northwestern Polytechnical University, 2005.

[6] Fan Shunyuan. Research on Finite Element Analysis Model of Intersecting Joint Pipe Truss. Shenzhen University, 2015.

[7] Rahimian Koloor S S, Karimzadeh A, Yidris N, Petrů M, Ayatollahi M R and Tamin M N. An Energy-Based Concept for Yielding of Multidirectional FRP Composite Structures Using a Mesoscale Lamina Damage Model. Polymers, 2020, 12(1): 157.

[8] Wang, Fuji, Jin, Xinghai, Wang, Xiaonan et al. Numerical Investigation on the Influences of Processing Conditions on Damage in the CFRTP Cutting Using a Novel Elastic–plastic Damage Model, 2022, 29(5): 2063-2094.

[9] Wang Zengsheng, Liu Jianjun, Han Xiao, et al. Design and development of a Large Carbon fiber Composite Truss Structure. New Chemical Materials, 2018, 46(05): 71-74.

[10] Gong Song. Experimental Study and Finite Element Analysis on Static Performance of Pipe Truss Joints in Coal Transport Trestle. Hebei University of Engineering, 2018.