International Journal of
Automotive Manufacturing and Materials (IJAMM)

Article

Analysis and Countermeasure of Cracking of the Supercharger Inlet Pipe of a Diesel Engine

Chuanlong Yin *, Chaoqun Dai, Wenjuan Li, Guoqiang Yu, and Jifeng Deng

Dongfeng Automobile Co., Ltd. (DFAC), Wuhan 430000, China
* Correspondence: yinchuanlong@dfac.com
Received: 7 April 2025; Revised: 5 May 2025; Accepted: 6 May 2025; Published: 21 May 2025

Abstract: A crack occurred at the inlet pipe of a diesel engine supercharger during its operation, accompanied by an oil leakage phenomenon. This paper conducts an in-depth study on the cracking characteristics of the supercharger inlet pipe and its root causes from multiple aspects, including fracture analysis, simulation analysis, parts production process investigation, on-site assembly confirmation, and whole-machine modal measurement. Judging from the fracture morphology, the fundamental cause of the fracture is that the assembly displacement is excessive, and the installation stress generated thereby exceeds the design limit. As a result, stress concentration occurs at the fracture, leading to fatigue fracture during long-term repeated vibrations.

Keywords: analysis of fractures; analytical simulation; mode; displacement in assembly processes; stress concentration phenomena; fatigue breakage

1. Introduction

In order to meet the requirements for safe and reliable operation of the supercharger, the intermediate part of a general diesel engine supercharger is lubricated by oil cooling [1]. An inlet and return oil channel are provided in the middle of the supercharger. The function of the supercharger's inlet oil pipe is to introduce the high-pressure oil from the main oil channel into the supercharger intermediate. During the working process, affected by both the vibration of the supercharger and the vibration of the engine itself, as well as the accumulated tolerances [2] of the parts themselves and during assembly, the supercharger's inlet pipe must have a certain degree of flexibility to compensate for the vibrations. Generally, a flexible connection structure is adopted. In this paper, the focus is on the preparation of the hose structure [3].

In order to ensure the reliability and durability of the supercharger, the supercharger inlet pipe must possess excellent temperature resistance, pressure resistance, and corrosion resistance. Additionally, it requires sufficient strength and flexibility to endure the vibrations generated by the engine during its operation. During maintenance, it is essential to regularly inspect the supercharger oil-inlet pipeline for cracks or leaks. Any damaged components should be promptly replaced to guarantee the normal operation of the supercharger system.

2. Subject Background

Diesel engines in the market have continually experienced failures of cracking and oil leakage in the supercharger inlet pipes. The mileage at which these faults occur is more than 15,000 km. The positions of cracking and oil leakage are at the installation side of the cylinder block, as well as at the connection between the lute joint and the steel pipe, as depicted in Figures 1 and 2.

Figure 1. Cylinder mounting side.

Figure 2. Pa-type joint and steel pipe connection.

3. Cause Investigation and Analysis

3.1. Fracture Analysis

Firstly, a fracture analysis [4] of the faulty sample is conducted to determine its cracking characteristics. The cracking position of the tubing is at the connection between the lute joint and the steel pipe. Figure 3 presents the overall macroscopic morphology of the section; obviously, there are distinct fracture traces, and cracks spread from the outside to the inside. A scanning electron microscope was employed to observe the microscopic morphology of the fracture, and an obvious fatigue band morphology was revealed, as illustrated in Figure 4. Based on the fracture analysis, it can be determined that the cracking characteristic is fatigue fracture [5–7].

3.2. Simulation Analysis Check

3.2.1. Modal Analysis Results

Target value: The first-order mode frequency should be greater than 3600/60 × 2 × 1.1 = 132 Hz.

Simulation result: The first-order mode frequency is 138 Hz, as shown in Figure 5. This value meets the design requirements. However, it is close to the resonance frequency [8]. It is recommended to conduct physical testing. According to Figure 6, at around 110 Hz, there exists a resonance risk.

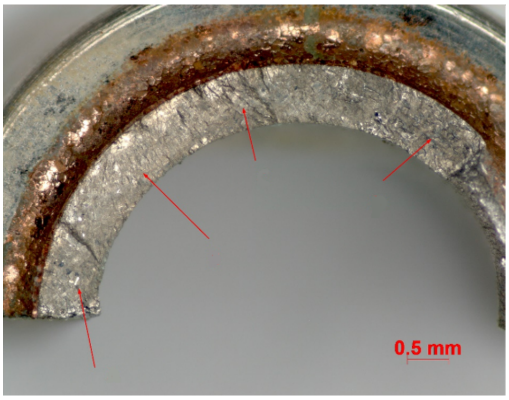

Figure 3. Macroscopic morphology of the section.

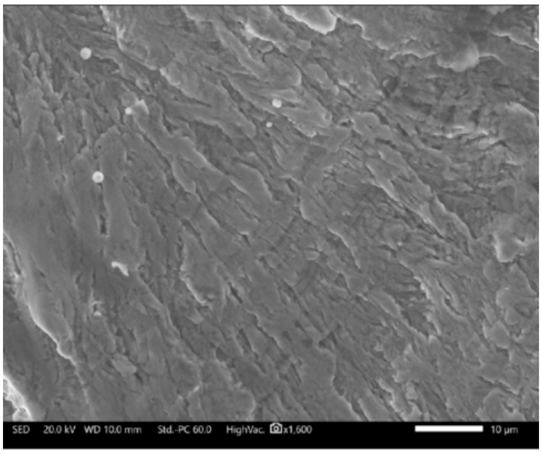

Figure 4. Micromorphology of the section.

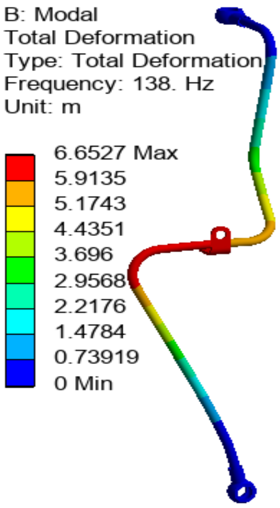

Figure 5. First-order mode (simulation).

3.2.2. Stress Analysis Result

Target value: The maximum equivalent stress should be less than 205 MPa. The maximum stress in the X-direction is 27.9 MPa, as depicted in Figure 7. The maximum stress in the Y-direction is 15.1 MPa, as shown in Figure 8. The maximum stress in the Z-direction is 32 MPa, as illustrated in Figure 9, and these values meet the design requirements.

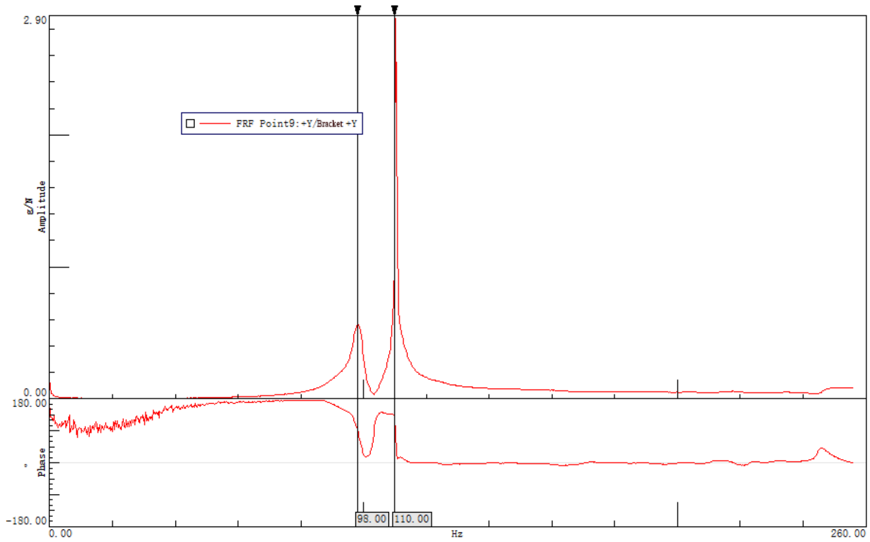

Figure 6. First-order mode (measured).

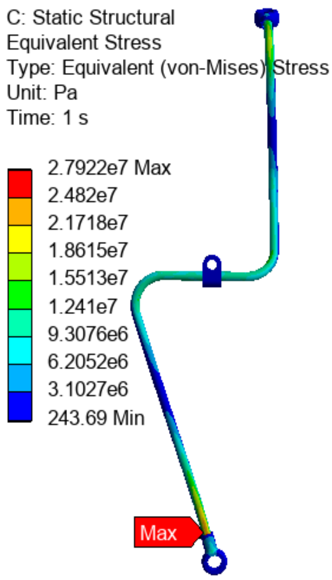

Figure 7. X-stress.

3.2.3. Simulation Analysis Result

The simulation results are presented in Table 1. The first-order mode frequency is 138 Hz, which is close to the resonant frequency of 132 Hz. A physical test was conducted, and the frequency related to the resonance risk is approximately 110 Hz. This frequency factor is a secondary contributor to the fracture. The maximum stress in the Z direction is 32 MPa, which is significantly lower than the yield strength of the steel pipe, which is 205 MPa. Therefore, it meets the design requirements.

3.3. Supplier Process Confirmation

3.3.1. Material Confirmation

Confirm the materials of the supplier's ball head, steel pipe, and copper ring, as well as the incoming inspection report. There is no abnormality found.

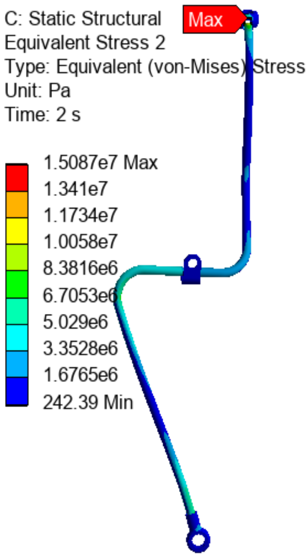

Figure 8. Y-stress.

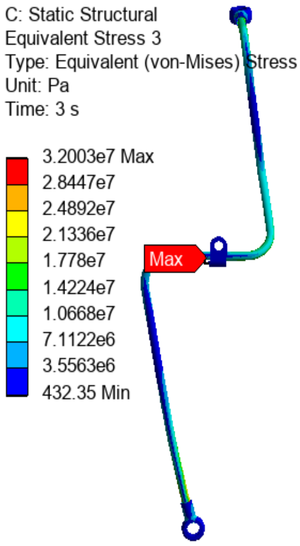

Figure 9. Z-stress.

Table 1. Results of simulation analysis.

	Calculation Content		**Design Requirement**	**Calculation Result**	**Conclusion**
Design Scheme	First order mode		>132 Hz	138 Hz	NG
	Maximum equivalent stress under acceleration load	X: 20 g	<205 MPa	27.9 MPa	√
		Y: 20 g		15.1 MPa	√
		Z: 20 g		32.0 MPa	√

3.3.2. Supplier Manufacturing Process Confirmation

Check the entire process of the supplier's pipe bending, point-pressure pre-positioning, brazing [9], correction, and watertight testing. Confirm that there are no 4M change risk points that could affect the fracture of the pipe fittings.

3.4. Factory Assembly Process Validation

3.4.1. Part Size Conformance Inspection

Five pieces of supercharger inlet tubing were randomly sampled for a size test, and the relative positional dimensions of the two mounting holes were found to be acceptable. Figure 10 shows the required dimensions as specified in the drawing, while Table 2 presents the measured dimensions.

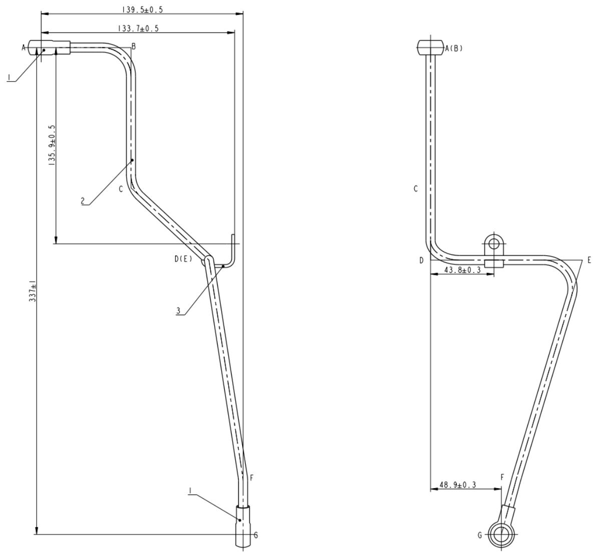

Figure 10. Drawings required dimensions.

Table 2. Measured dimensions.

Test Item	Technical Requirement	Part Quality Check Record				
		1	2	3	4	5
Dimension	337 ± 1	337.61	337.48	337.64	337.54	337.71
Dimension	139.5 ± 0.5	139.50	139.37	139.44	139.65	139.68
Dimension	48.9 ± 0.3	48.86	48.97	48.74	48.81	48.91

3.4.2. Assembly Process Validation

Online assembly process:

S1—Pre-tighten the bolts at both ends, as illustrated in Figure 11.

S2—Assist (to prevent the lute joint from rotating along with the bolt) in tightening the bolts on the supercharger side, as depicted in Figure 12.

S3—Tighten the bolts on the cylinder block side.

Offset Check: Once the above assembly is finished, the bolts on the side of the cylinder block are loosened, and it is discovered that the assembly offset is significant, as shown in Figure 13. Five pieces of supercharger inlet tubing are randomly chosen for the trial production. All of them exhibit offsets to varying degrees, among which the offsets of 1# and 5# are relatively larger, as presented in Table 3.

3.4.3. Assembly Stress Simulation

The offsets of 1# and 5#, which have relatively large offsets as shown in Table 3, were incorporated into the simulation analysis. The maximum stress of the 1# sample was found to be 259.7 MPa, as depicted in Figure 14. The maximum stress of the 5# sample is 246 MPa, as illustrated in Figure 15.

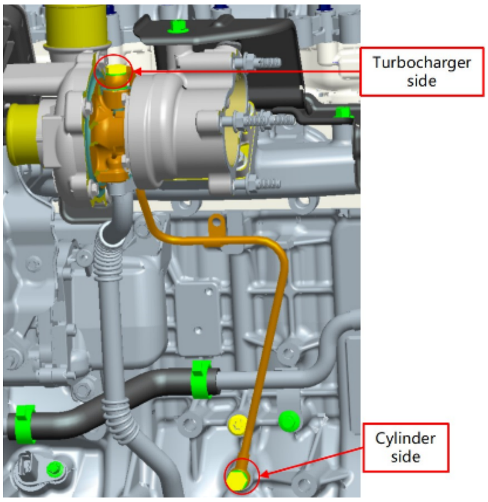

Figure 11. Pre-tighten bolts at both ends.

Figure 12. Accessories.

Figure 13. Offset after release.

According to the simulation results, once the offset is factored in, the stress increases significantly, surpassing the design limit value. Evidently, the excessive installation stress is the root cause of the fracture. The specific results are presented in Table 4.

Table 3. Measured offset.

Test Sample	Offset Direction (mm)		
	Up and Down	Left and Right	Front and Back
1#	6.1	6.7	0.3
2#	2.7	3.6	0.1
3#	1.9	2.7	0.1
4#	1	5.5	1
5#	1	7.5	1

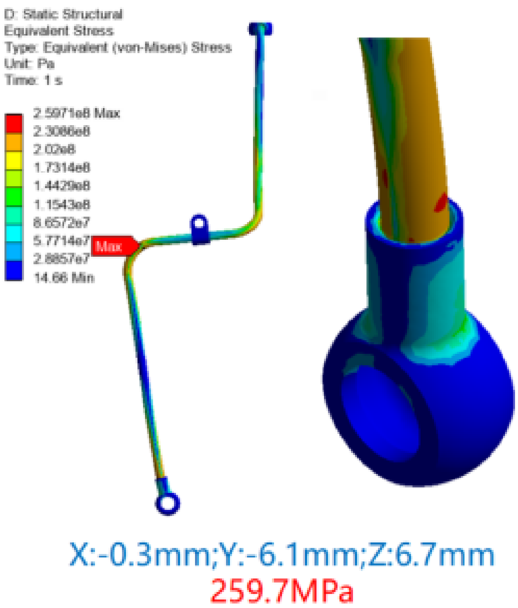

Figure 14. Simulation stress of sample 1#.

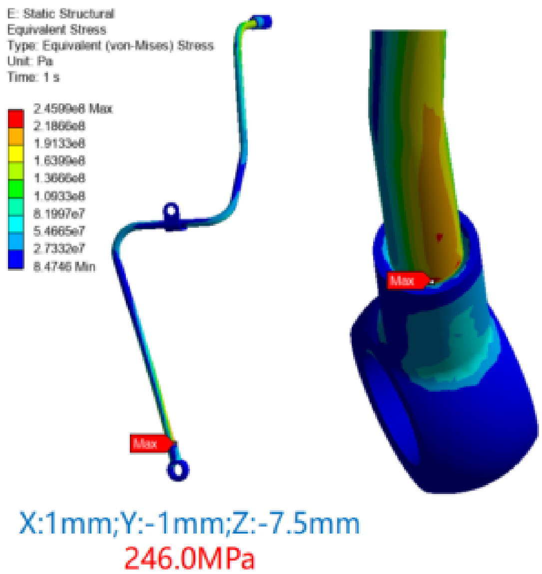

Figure 15. Simulation stress of sample 5#.

Table 4. Simulation results after adding an offset.

Scheme	Calculation Content		Design Requirement	Calculation Result	Conclusion
No displacement load	Maximum equivalent stress under acceleration load	X: 20 g	<205 MPa	27.9 MPa	√
		Y: 20 g		15.1 MPa	√
		Z: 20 g		32 MPa	√
Displacement load of the cylinder side pull joint	1# Sample (mm)	X: −0.3; Y: −6.1; Z:6.7.		259.7 MPa	X
	5# Sample (mm)	X: 1; Y: −1; Z: −7.5.		246 MPa	X

4. Improvement Measures

4.1. Confirmation of Improvement Direction

Based on the above analysis results, there are two main improvement directions: reducing the assembly stress [10] and increasing the modal.

4.2. Improvement Plan

By comparing and referring to the mature products of the same type, it has been determined to adopt a flexible hose structure, as illustrated in Figure 16. The objective of developing the hose structure is to reduce both the assembly stress and the vibration amplitude.

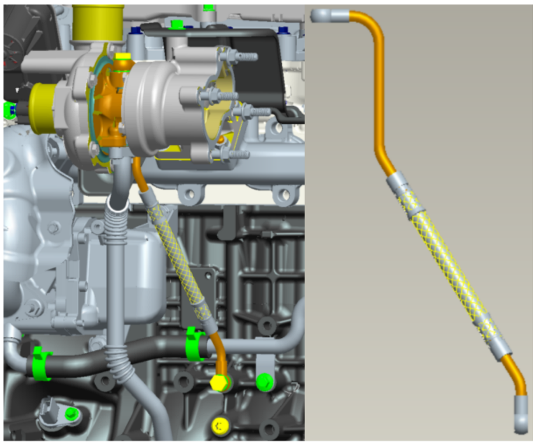

Figure 16. Preparation of the hose structure booster inlet pipe**.**

4.3. Simulation Analysis Confirmation

4.3.1. Modal Confirmation

Through simulation analysis, the first-order mode frequency is found to be 202 Hz, which is far higher than the resonance frequency of 132 Hz. This effectively and perfectly avoids resonance and meets the design requirements, as depicted in Figure 17.

4.3.2. Stress Confirmation

Through simulation analysis, the maximum stress in the X-direction is determined to be 99.94 MPa, and the maximum stress at the location of the crack is 56.85 MPa. These values are significantly lower than the yield strength [11] of the pipe material, which is 205 MPa. Thus, they meet the design requirements, as illustrated in Figure 18.

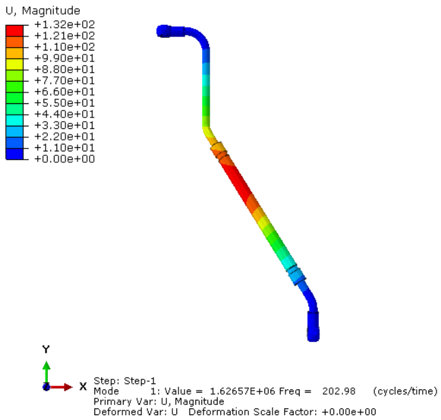

Figure 17. First order mode.

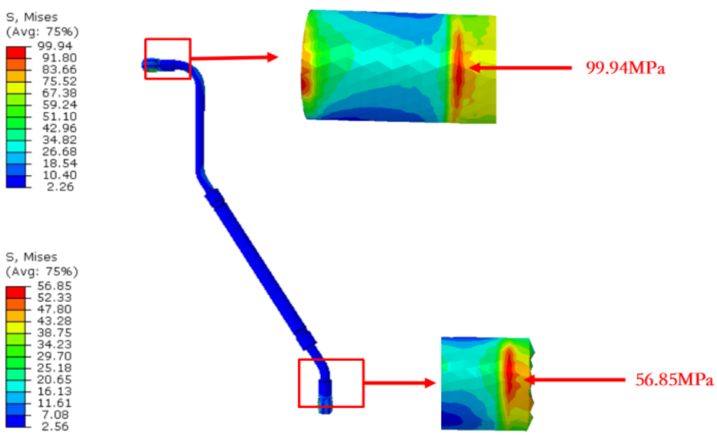

Figure 18. X-direction stress.

The maximum stress in the Y-direction is 98.2 MPa, and the maximum stress at the crack site is 60.5 MPa. These values are substantially lower than the yield strength [11] of the pipe material, which is 205 MPa. As a result, they satisfy the design requirements, as depicted in Figure 19.

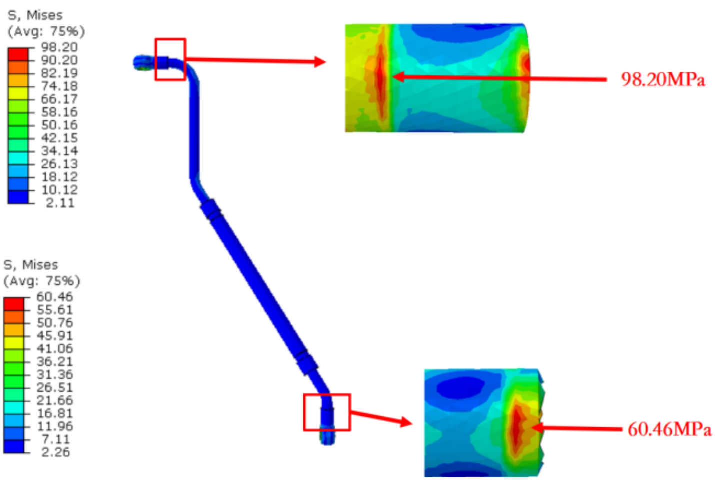

Figure 19. Y-direction stress.

The maximum stress in the Z-direction is 89.6 MPa, and the maximum stress at the location of the crack is 59.5 MPa. These values are far lower than the yield strength [11] of the pipe material, which is 205 MPa.

Thus, they meet the design requirements, as shown in Figure 20.

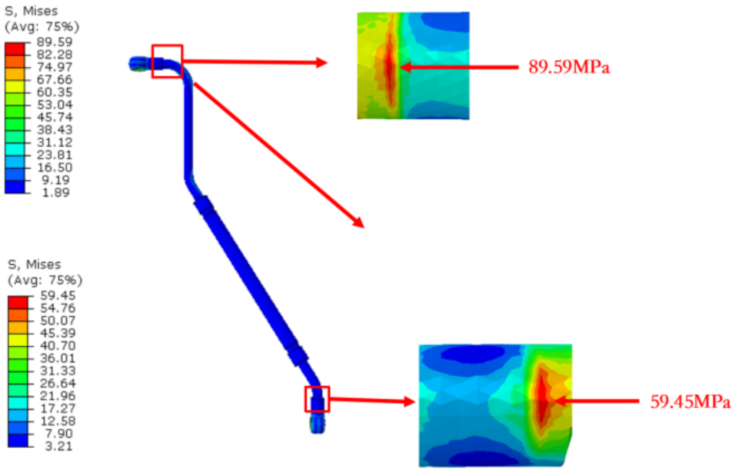

Figure 20. Z-direction stress.

4.3.3. Summary

From the simulation results, it can be seen that compared with the original plan, the flexible hose plan significantly reduces the stress and increases the modal frequency. This outcome is more in line with the envisioned improvement direction.

4.4. Confirmation of Durability for Individual, Complete, and Whole Vehicles

The development of the hose scheme has successfully completed the individual vibration durability test [12], the whole-machine vibration ultra-high speed durability test, and the whole-vehicle road test durability. Throughout these tests, no issues were found. Subsequently, the product has been launched into the market, and up to now, no problems have been reported.

4.5. Conclusion

From the above analysis, it can be concluded that the turbocharger inlet pipe featuring a flexible hose structure is capable of effectively addressing the issues of market-reported cracking and oil leakage.

5. Summary of the Topic and Prevention of Recurrence

(1) When the modal value obtained from the simulation analysis is close to the engine excitation frequency [13], it is essential to conduct a test on the physical modal value and carry out optimization according to the results of the physical test.

(2) In the case where the turbocharger oil inlet pipe has a pure steel pipe structure and lacks a fixed point in the middle, during the simulation analysis, it is necessary to take into account and increase the displacement resulting from the assembly process. After the turbocharger end is fixed, there will be a certain degree of displacement in the free state of the flange joint on the cylinder body side. At this moment, when fixing the cylinder body side, installation stress will be generated, and this displacement should be incorporated into the simulation.

(3) When there is no fixed point in the turbocharger oil inlet pipe, based on practical experience:

(a) For a total length within 300 mm, a pure steel pipe structure can be adopted.

(b) When the total length ranges from 300 mm to 400 mm, depending on the layout requirements and cost considerations, appropriate structures can be selected, such as pure steel pipes or braided hoses.

(c) If the total length exceeds 400 mm, a pure steel pipe structure is not suitable and a braided hose must be utilized instead.

Author Contributions: C. Y.: conceptualization, methodology, software, data curation, writing—original draft preparation, visualization, investigation, supervision, software, validation, writing—reviewing and editing; C. D.: reviewing; W.L.: reviewing; G.Y.: reviewing; J.D.: reviewing. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability: Statement: Not applicable.

Conflicts of Interest: The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

1. *Design Manual of Hydraulic Oil Cooling System for Construction Machinery*; China Petrochemical Publishing House: Beijing, China, 2022.
2. Avallone, E.A. *Mechanical Design Manual*; McGraw-Hill Education: New York, NY, USA, 2023
3. *Dynamic Fatigue Analysis of Braided Hydraulic Hoses in Construction Machinery*; SAE: Warrendale, PA, USA, 2024.
4. Tu, M.-J. *Fracture Diagnosis of Mechanical Failure*; Mechanical Industry Press: Beijing, China, 2023.
5. Shang, D. *Fatigue Strength Theory*; Science Press: Beijing, China, 2017.
6. Li, D.; Rong, L.; Mao, N.; Song, Y.; Liang, R.; You, J. Simulation of Vibration Characteristics of the Front MacPherson Suspension of a Sightseeing Vehicle. *Int. J. Automot. Manuf. Mater.* **2024**, *3*, 1.
7. Zhang, D.; Li, M.; Li, L.; Deng, J.; Li, Y.; Zhou, R.; Ma, L. Failure Analysis and Reliability Optimization Approaches for Particulate Filter of Diesel Engine after-Treatment System. *Int. J. Automot. Manuf. Mater.* **2025**, *4*, 2.
8. *Resonance and Suppression in Structural Dynamics*; Science Press: Beijing, China, 2023.
9. Zhang, Q. *Modern Brazing Technology and Application*; Machinery Industry Press: Beijing, China, 2022.
10 Wang, J. *Analysis and Control of Stress in Mechanical Assembly*; Machinery Industry Press: Beijing, China, 2023.
11. Y*ield Strength Design Manual for Engineering Materials*; Machinery Industry Press: Beijing, China, 2022.
12. *Engineering Machinery Vibration Durability Test Technical Manual*; Machinery Industry Press: Beijing, China, 2023.
13. He, L. *Engine Vibration Analysis and Control*; National Defense Industry Press: Beijing, China, 2023.