Proc. of the 9th Intl. Conf. of Asian Society for Precision Engg. and Nanotechnology (ASPEN 2022)
15–18 November 2022, Singapore. Edited by Nai Mui Ling Sharon and A. Senthil Kumar
doi://10.3850/978-981-18-6021-8_OR-18-0206

A Study on the Stability of the Vibration Durability Test Jig for a Hydraulic Hose using the ANSYS Code module

Chae Sil Kim[1#], Hun Lee[2], Pil Geo Park[2]

1 Department of Mechanical Engineering, ChangwonNationalUniversity,20-2,Changwondaehak-ro,Uichang-gu,Changwon-si,Gyeongsangnam-do, RepublicofKorea
2 Department of Mechanical Engineering, ChangwonNationalUniversity,20-2,Changwondaehak-ro,Uichang-gu,Changwon-si,Gyeongsangnam-do, RepublicofKorea
Corresponding Author / Email: kimcs@changwon.ac.kr, TEL: +82-55-213-3602, FAX: +82-55-213-3611

KEYWORDS: Hydrualic Hose, Jig, Vibration Durability Test, ANSYS nCode module

A jig for fixing the hydraulic hose was damaged during the vibration durability test of the hydraulic hose. Kim and Jeon identified the cause through ANSYS mode analysis and fatigue analysis, designed a new jig, and successfully conducted the vibration durability test of the hydraulic hose with this new jig. It was argued that the stability of the jig must be ensured before the vibration durability test, and ANSYS mode analysis and fatigue analysis were presented as technique to ensure the stability. This article proposes another method to ensure the stability of the jig for vibration durability test of the hydraulic hose by performing cause analysis and simulation on the newly proposed jig using the ANSYS nCode module instead of ANSYS mode analysis and fatigue analysis.

1. Introduction

Vibration durability tests are required to ensure the ability to perform hydraulic hoses and stability against repeated loads and sudden impacts. To this end, a jig that secures the stability necessary for the test is essential. For this reason, vibration durability test and fatigue analysis of the jig of the hydraulic hose must be performed to confirm the stability and durability of the product. C. Kim et al. [(1)] designed the improved jig after ANSYS mode analysis and fatigue analysis for the damaged jig that was damaged during vibration durability test. The ANSYS mode analysis and fatigue analysis were performed on revised jigs and the stability and justification of improved jigs were presented. [(1)] This paper proposes a new method to ensure the stability of jigs for vibration durability test of hydraulic hoses

2. Cause analysis of broken jig

2.1. Vibration durability test of broken jig

Figure 1 shows the jig and sample of the damaged hydraulic hose. The vibration tolerance test used the Long Stroke vibration test equipment, and the specifications of the equipment are as specified in Table 1. The test environment of the jig was set as shown in Table 2. The breakage of the jig occurred 13 hours after the test. Cracks in the jig occurred at the bottom of the pipe and propagated in the vertical direction as shown in Figure 2.

Table 1 Long Stroke Vibration Test Equipment Spec.

Test Equipmet	Specification
Table Size	- Vertical: 800mm x 800mm - Horizontal: 800mm x 800mm
Max Payload	- 1000kg
Frequency Range	- 5~2600Hz
Max Displacement	- 100mm

Table 2 Test condition.

Type	Test Frequency	Acceleration	Test time
Dwell test	25Hz	7g (peak to peak)	100 hr

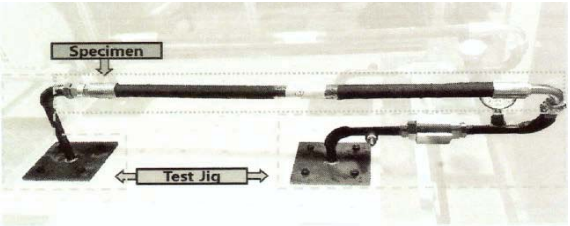

Fig. 1 Hydraulic Hose and Hydraulic Hose Jig

Fig. 2 Crack of the jig

2.2. Analysis using ANSYS nCode module

Use the ANSYS nCode module to analyze the cause of the jig breakage. ANSYS Modal analysis was conducted to determine whether the damaged jig was resonant. As a result of Modal analysis, the results are derived as shown in Table 3, and since the jig is not damaged by resonance, the S-N curve theory of ANSYSnCode module is applied based on the results analyzed by ANSYS Static Structure. The analysis conditions of the ANSYS Static Structure were set at an acceleration of 7g for the entire modeling in the same way as the test conditions. As a result of ANSYS Static Structure analysis, as shown in Figure 3, about 320 MPa, which exceeds the yield strength of SS400, which is a jig material, is applied, and it is confirmed that fatigue caused by repeated loads is the cause of the jig breakage. After that, as can be seen in Figure 4, fatigue analysis was performed by linking the ANSYS nCode module to the ANSYS Static Structure. The analysis results of ANSYSnCode module of damaged jig show that it is damaged when repeated load is applied 1.104 million times, as shown in Figure 5, and when converted to time life in consideration of the test frequency of 25 Hz, it is damaged for about 12.26 hours.

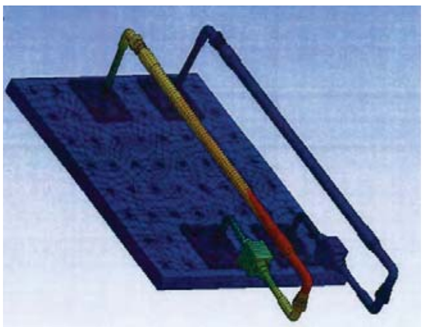

Fig. 3 Structural analysis results of broken jig

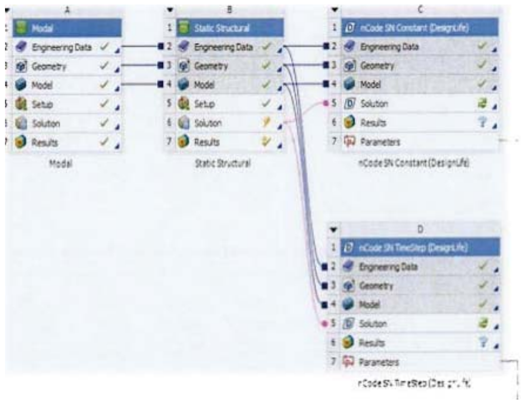

Fig. 4 ANSYS nCode Module solution

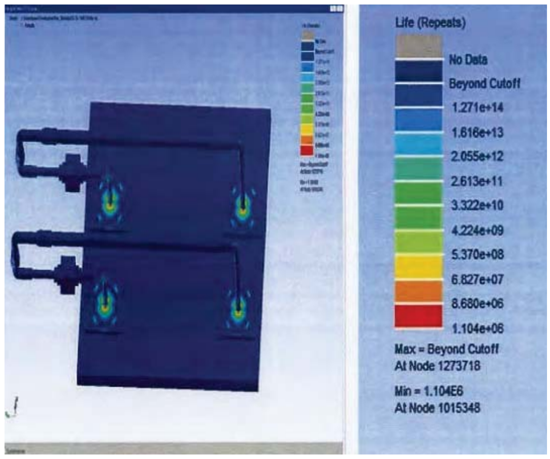

Fig. 5 Fatigue life result of broken jig

Table 3 Natural Frequency

Mode	Frequency(Hz)
1st	14.85
2nd	18.17
3rd	37.09

3. Analysis of revised jig

Figure 6 shows the CATIA 3D modeling of the improved jig. ANSYS Modal analysis was conducted to determine whether the revised jig resonates. As shown in Table 4, it was confirmed that there was no risk of resonance. The stress value acting on the jig is derived using ANSYS Static Structure, and the analysis is performed by applying the S-N curve theory of ANSYSnCode module based on the result value. The set value of structural analysis was the same as before, and an acceleration of 7g was applied to the entire modeling. As shown in Figure 7, it was found that about 180 MPavon-Mises stress acts. This acts less than 250 MPa, which is the yield strength of SS400, and it can be predicted that the stability of the jig has been improved. The fatigue life of the revised jig is then analyzed using the ANSYS nCode module. Figure 8 shows the results of fatigue life analysis of the ANSYSnCode module of the revised jig. The minimum life was calculated as 2.263x107 times, which is about 250hr when converted to time life.

When looking at these result values, the durability limit time of 100hr is satisfied.

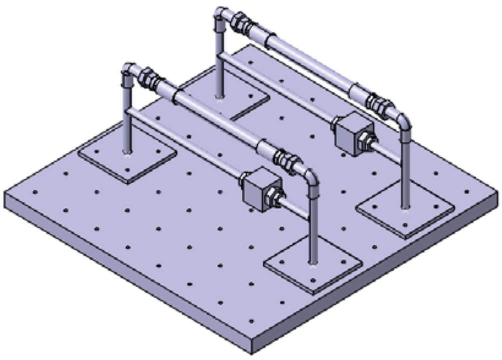

Fig. 6 CATIA 3D Modeling of the revised jig

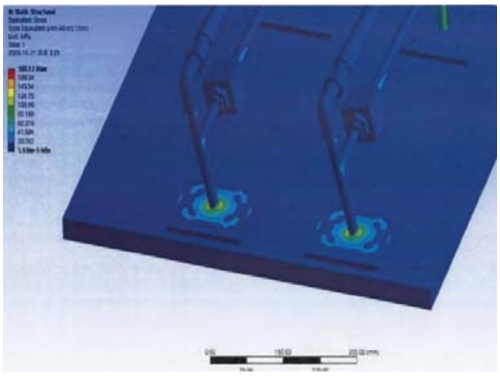

Fig. 7 Von-Mises stress of revised jig

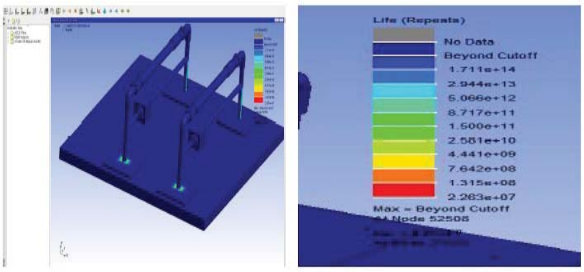

Fig. 8 ANSYS nCode module fatigue life results of revised jig

Table 4 Natural Frequency

Mode	Frequency (Hz)
1st	33.07
2nd	47.55
3rd	58.49

4. Conclusions

This paper reproduced a broken case of vibration durability test of broken jig through vibration analysis, and as a result, it was verified that it was broken by repeated load, not by resonance, presented an improved jig, and used ANSYSnCode module to predict fatigue life of two jigs. It was confirmed that the broken jig showed a life similar to the actual broken time of 13hr, and the revised jig showed a result similar to the result value of ANSYS fatigue analysis [1] at about 250hr. Through this, it is determined that the life of the jig may be predicted using the ANSYSnCode.

ACKNOWLEDGEMENT

This research was supported by the Vibration Durability Research Center, Changwon National University.

REFERENCES

1. C. Kim, “A Study on the Structural Stability of a Jig for Evaluating the Vibration Durability of a Hydraulic Hose.”, Journal of the Korean Society of Manufacturing Process Engineers, Vol.20, No1, pp. 125-131, 2021.
2. ReSEAT Analysis Report, “Damage Occurred during the Durability Test of Rubber Hose for Automobile Hydraulic Brake,: 2007.
3. Zhou, Y., Al-Bassyiouni, M. and Dasgupta, A., “Vibration Durability Assessment of Sn3.0Ag0.5Cu and Sn37Pb Solders Under Harmonic Excitation:, Journal of Electronic Packaging, Vol. 131, No. 1, 2009.
4. TAE SUNG Software & Engineering.INC, “Ansys nCodeDesignLife,” 2020.