한국기계가공학회지, 제20권, 제1호, pp. 125～131(2021.01) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 20, No. 1, pp. 125~131(2021.01) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2021.20.01.125

A Study on the Structural Stability of a Jig for Evaluating the Vibration Durability of a Hydraulic Hose

Chae-Sil Kim[*,#], Min-Seong Jun[**]
[*]School of Mechanical Engineering, Changwon National Univ.
[**]Vibration Duarability Research Center, Changwon National Univ.

유압 호스의 진동 내구성 평가를 위한 지그의 구조적 안정성에 관한 연구

김재실[*,#], 전민승[**]
[*]창원대학교 기계공학부, [**]창원대학교 진동내구성연구센터

(Received 07 December 2020; received in revised form 07 January 2021; accepted 11 January 2021)

ABSTRACT

For the vibration reliability tests, the jigs for mounting the test specimen on a vibration reliability tester are required. The dynamic stabilities of the jigs should be verified before the tests for obtaining the accurate reliability of the test specimen. This paper proposes an analytical technology for ensuring the dynamic structural stability for the test setup including the jig. The technology includes the mode analyses for checking resonance, the harmonic analyses for evaluating the dynamic structural stability of test setup including the jig, and the fatigue analyses for obtaining the durable reliability time with calculating the life cycles at the area of weakness. The cause investigation of the damaged jig during vibration reliability test of a rubber hose and the design of new revised jig are performed by using the technology. The vibration reliability test for the rubber hose with the new revised jig by analysis results is successfully conducted without any problem. Therefore the jig's design technology proposed in this paper may be useful for other items as well.

Keywords : Hydraulic Hose(유압호스), Vibration Durability Test(진동내구성시험), Finite Element Analysis(유한요소해석)

1. Introduction

Recently, the vibration reliability tests for mechanical systems have been increased to improve their performances and to ensure their stabilities under the repeatable loads and sudden shocks. For these tests, the fixing jigs for mounting the systems on a vibration reliability tester are required. Before the tests, the dynamic stabilities of the fixing jigs should be verified. This paper proposes an analytical technology for ensuring the dynamic structural stability for the test setup including the jig.

Corresponding Author : kimcs@changwon.ac.kr
Tel: +82-10-4180-3607, Fax: +82-55-213-3611

A jig was damaged during evaluating the vibration reliability of a rubber hose at the vibration durability research center in Changwon National University. The finite element (FE) analysis is performed for investigating the cause of the jig's damage.

Another jig is newly devised on the basis of the investigated results for the damaged jig. And the FE model for the new jig is simulated to check its stability. Finally, the vibration reliability test for the rubber hose with the new revised jig is successfully conducted without any problem.

2. A broken jig during reliability test for a rubber hose

2.1 The description of reliability test

Figure 1 shows the test setup conducting the vibration reliability test for a hydraulic rubber hose with maintaining the inner pressure of 42MPa. The test fails if the leakage or pressure reduction is found during the test period. The jig for mounting the test specimen shall, of course, remain structurally stable during the test period. This is because the reliability of the durability assessment of the test specimen could be ensured if the stability of the jig is ensured

The jig used for the reliability test of the rubber hose is a series of gas hole, valve, and pressure gauge on only one part, considering the convenience of refueling, pressurization, and gauge reading during test, and is empirically produced by the company without the jig's dynamic stability verification.

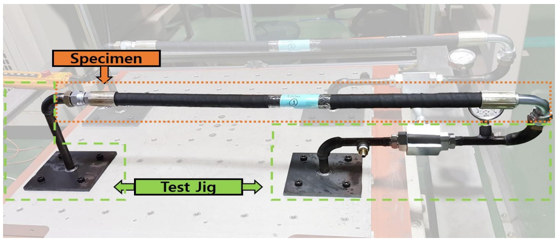

Fig. 1 reliability test for a hydraulic rubber hose

Fig. 2 Long stroke vibration test equipment

Figure 2 shows the long stroke vibration test equipment (J260/IMV, Japan) used for this reliability test for the rubber hose at the vibration durability research center in Changwon National University. Many vibration reliability tests and shock tests for parts and sub-assemblies of electric machines, transportation machineries, machine tools, and so on can be conducted with the test equipment under the specification as shown in Table 1.

The failure was found in 13 hours after the test which began under the test conditions as shown in Table 2.

Table 1 Test Equipment Specification

Test Equipment	Specification
- Table size	- Vertical: 800mm×800mm Horizontal:800mm×800mm
- Max Payload	- 1000kg
- Frequency Range	- 5～2600Hz
- Max Displacement	- 100mm

Table 2 Vibration durability test condition

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

Fig. 3 The damage of jig

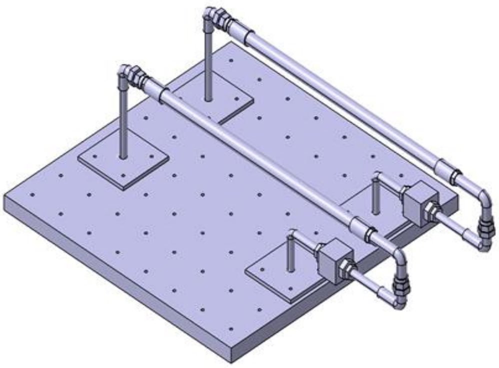

Fig. 4 CATIA 3D model of test jig

A part of the jig as shown in Fig. 3 was broken during the vibration durability test. The crack was occurred in the lower pipe of the jig and propagated vertically.

2.2 Finite element analyses for the model of damaged jig

A three dimensional(3D) model using CATIA with the drawings provided from the company is constructed shown in Fig. 4. And then a FE analysis model with importing the CATIA 3D model using ANSYS is completed for investigating the cause of the jig's damage.

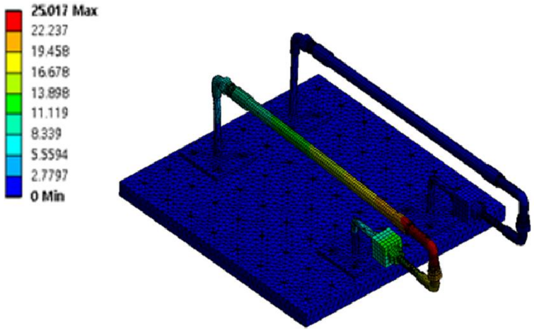

Fig. 5 1^{st} Mode Shape

Table 3 The analyzed natural frequencies

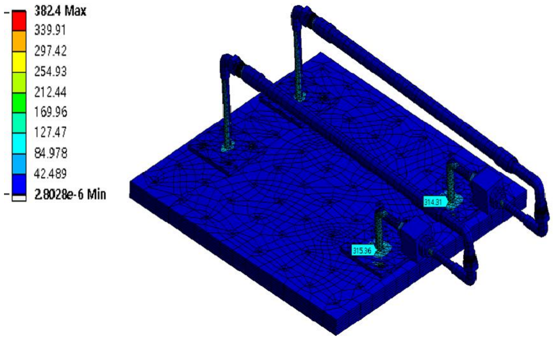

Fig. 6 von-Mises stress distribution

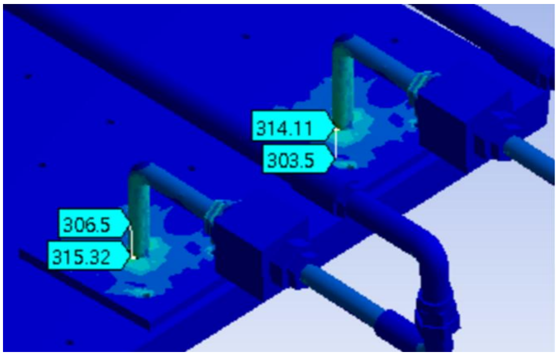

Fig. 7 von-Mises stress at the damage spot

First, mode analysis for this model is performed to check the resonance. The degrees of freedom of the nodes at the bottom of fixture are restricted as the boundary conditions for the mode analysis. Figure 5 shows the 1st mode shape, one of the mode analysis results. Since the calculated natural frequencies are listed in Table 3 are higher than the exciting frequency of 25Hz as shown in Table 2, no dangerousness due to the resonance exists. This leads that the cause of the jig damage may not be the resonance and that the other analyses for the damage are necessary.

Next, ANSYS harmonic analysis is continued to investigate the cause of jig damage. For the harmonic analysis, the 7g peak to peak acceleration with the frequency of 25 Hz was applied to the entire test model in the same way as the actual test conditions.[5]

Figure 6 shows von-Mises stress distribution, one of the harmonic analysis results. It can be seen that the equivalent stress of about 310 MPa was calculated at the actual damage spot as shown in Fig. 7. Since the stress at the damage spot exceeds the yield strength of 250 MPa for SS400, which is the material of the jig, fatigue due to repeated loads may be estimated as the cause of damage.

So a fatigue analysis is finally performed under applying Goodman failure line and fully reversed load with average stress of 0 and no average stress modification.

Figure 8 shows the calculated life distribution which is one of the fatigue analysis results, and the minimum repeatable number of the jig of 1.1507 million. The minimum life is about 13 hours at 25Hz which is much lower than the durability test time of 100 hours. This is the explicit evidence of the damage of jig. Therefore a new jig should be revised for the safe test.

3. Analyses and test for a new revised jig

3.1 The device of new jig

As the results of the harmonics and fatigue analysis show that the damaged area is weaker than the shape of the other straight support, the damaged area is modified to a straight line form, such as the shape of the other part. Figure 9 shows the CATIA 3D model for the new revised jig.

3.2 Finite element analyses for the model of revised jig

A FE analysis model with importing the CATIA 3D model for the new revised jig using ANSYS is completed for investigating the safety for the resonance and the structural stability.

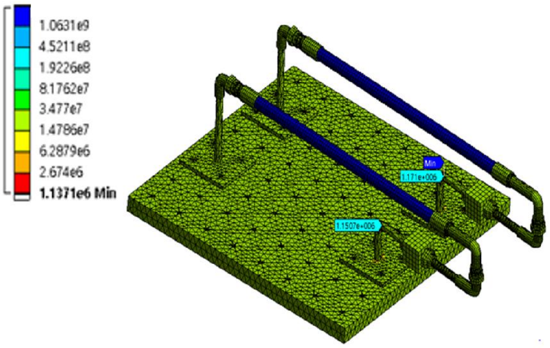

Fig. 8 Life distribution by the fatigue analysis

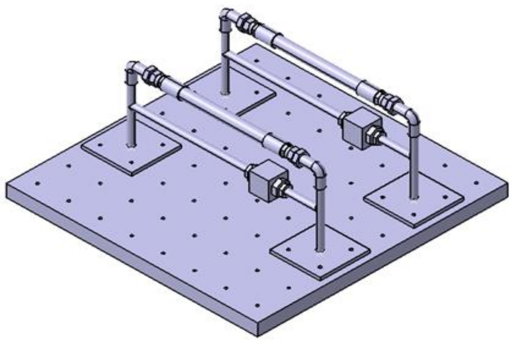

Fig. 9 CATIA 3D model for the new jig

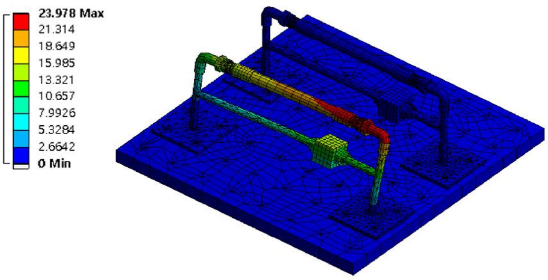

Fig. 10 1st Mode Shape for the new revised jig

Table 4 The natural frequencies for the new jig

Mode	Frequency[Hz]	Mode	Frequency[Hz]
1st	43.238	4th	117.09
2nd	45.258	5th	119.29
3rd	117.08	6th	119.3

First, mode analysis for this model is conducted to check the resonance. The degrees of freedom of the nodes at the bottom of fixture are restricted as the boundary conditions for the mode analysis. Figure 10 shows the 1st mode shape for the new revised jig.

Since the calculated natural frequencies are listed in Table 4 are much higher than the exciting frequency of 25Hz as shown in Table 1, the revised jig may be safe for resonance.

Next, ANSYS harmonic analysis for the revised jig is continued to investigate the structural stability. For the harmonic analysis, the 7g peak to peak acceleration with the frequency of 25 Hz is applied to the entire test model in the same way as the actual test conditions.[3] Figure 11 shows von-Mises stress distribution, one of the harmonic analysis results. Although it can be seen that the maximum von-Mises equivalent stress still exists at the damaged spot, the calculated equivalent stress of about 115MPa as shown in Fig. 12 is less than the yield strength of 250MPa for SS400, which is the material of the revised jig. This means that the structural stability for the revised jig may be ensured.

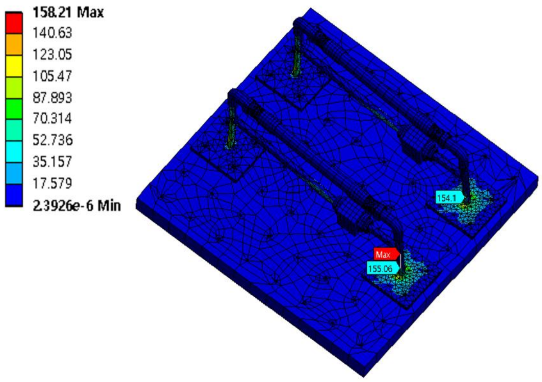

Fig. 11 von-Mises stress for the revised jig

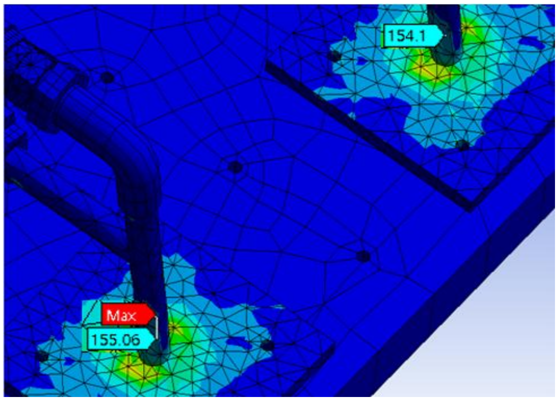

Fig. 12 Maximum stress for the revised jig

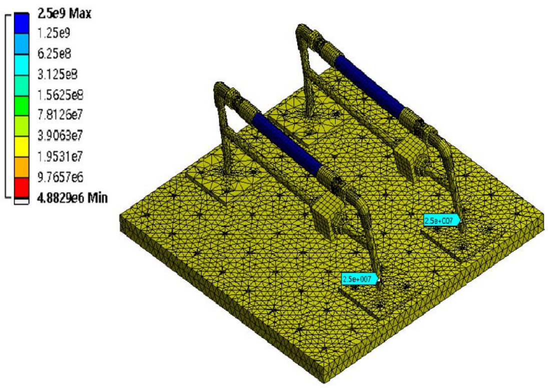

Fig. 13 Life distribution by the fatigue analysis for the revised jig

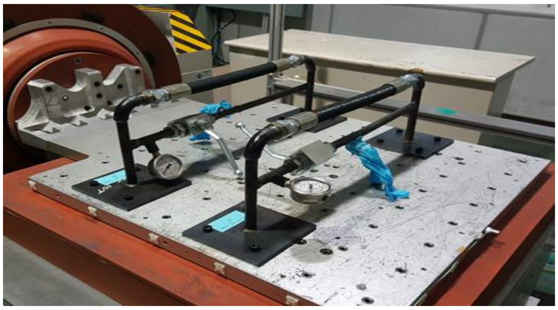

Fig. 14 Reliability test of the rubber hose with the new revised jig

Finally a fatigue analysis for the revised jig is performed under applying Goodman failure line and fully reversed load with average stress of 0 and no average stress modification, as the same condition of damaged jig.

Figure 13 shows the calculated life distribution for the revised jig and the minimum repeatable number of the jig of 25 million. The minimum life is about 277 hours at 25Hz which is much higher than the durability test time of 100 hours. This leads that the jig can be reliability test of rubber hose with the revised jig.

3.3 Reliability test of the rubber hose with new revised jig

An improved jig is manufactured based on the analysis results. Figure 14 shows the actual reliability test setup under the test condition in Table 2. The test is successfully performed without any failure during test durable time of 100hrs. Therefore the rationality of the analytical method for securing the stability of the jig proposed in this paper has been confirmed.

4. Conclusion

A jig was damaged during the test for evaluating the vibration reliability of a rubber hose. In this paper, the FE analyses, including mode analysis, harmonic analysis and fatigue analysis, are performed for investigating the cause of the jig's damage. It could be estimated that the damage was caused by the fatigue due to the repeated load on the jig by the FE analyses results.

A new jig was designed with the damaged part which is designed as the shape of the other side. It is also designed to be structurally robust by placing the test hydraulic hoses and the filling hole -valve -pressure gauge in parallel and connecting the two poles each other. The FE analyses for the revised jig model were conducted with the same manner as the broken jig model. The results showed that the jig model could be safe for resonance and be enough to its fatigue life. Finally, the reliability test of rubber hose with the revised jig was successfully performed without any failure during test durable time of 100hrs.

Therefore, this concludes that the dynamic stabilities of test setup including the jigs should be ensured before the vibration reliability tests and that the jig's design technology proposed in this paper may be useful for other items as well.

Acknowledgement

This research was supported by Changwon National University in 2019~2020.

.

References

1. ReSEAT Analysis Report, “Damage Occurred during the Durability Test of Rubber Hose for Automobile Hydraulic Brake,” 2007.
2. Spater g., “How to do modal analysis in Ansys workbench”, 2012. https://www.youtube.com/watch?v=uwim3BdINcs
3. Peter To, Ansys 18.2 “Natural frequency and harmonic response of an I beam”, 2018.

https://www.youtube.com/watch?v=Mofu3Ux21kE
4. TAE SUNG Software & Engineering.INC, "How to Use ANSYS Workbench, Sigmapress," 2007.
5 . TAE SUNG Software & Engineering.INC, "Work bench Fatigue Module Analysis," 2017. https://www.youtube.com/watch?v=l—WAlOants&t=1s
6. Stephen K., CATIA V5 - "Basic Part design tutorial with audio," 2014. https://www.youtube.com/watch?v=3fCkU6EqwuU
7. 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, Vo. 131, No. 1, 2009.