A Study of Tolerances Effect of the Interference Fit in a Retaining Ring Generator 60 MW at Dieng Geothermal Component Using FEM

Aditya Sukma Nugraha, Imam Djunaedi, and Hilman Syaeful Alam

***Abstract*—In fact shrink fit an installation technique that has been chosen to assembly two components. In this paper will be learned as a result of differences in the size of the interference in two pairs of components in the generator. Variations in diameter will be selected to determine the effects of the stresses and strains that occur. Methods of solution were performed using Finite Element Method (FEM). This method was chosen because of the kind of research that is found, the use of this model will get the results that will be close to the actual results. The solution obtained shows that a small variation in size (each variation of 0.03 mm) will have a major impact on the stresses and strains obtained.**

***Index Terms*—Shrink fit, interference, FEM, stress, strain.**

I. Introduction

In the manufacturing industry, connecting the two components in the assembly widely used shrink fitt method. This method is usually accomplished by a change in the relative size at the time of assembly. The thermal shrink uses the phenomenon of expansion during the heating of materials. In Fig. 1 can be seen that the mechanical interference with little or no pressure needed for assembly. For examples the heating process at this stage using electromagnetic induction process by placing the inductive material, there will be a large magnetic field, so that the electric current can flow in the metal and the heat arising as a result of the process [1]. Heat generated by this process is determined by the frequency of the magnetic field and the material permeability [2].

Analysis of shrink fit is important because the geometrical complex, material permeability and loaded make different effect. Many research calculated of the shrink fit loaded by conventional method [3]. In this paper calculated of the retaining ring inteference fit part analysis with Finite Element Method (FEM). Robust analysis performed with ANSYS Workbench, in the process in shrink fit will be modeled by 3D FEM in solidwork then import by ANSYS Design Modeller. On the other hand, FEM analysis using for calculated pressure, stress, and deformation in the shrink fit component.

Actually, based by the result of the installation process make components increased internal pressure, and then allowing of the compound cylinder [4]. The assumption of the analysis can be solved by many method, one of them is analysis of interference fit using simplification with strain hardening on the hub and elastic approach at the shaft solid [5]. Another approach calculation done by Jahed, *et al.*, they conduct research with the axisymetric method with elastic-plastic analysis to predicts residual stress at the component [6].

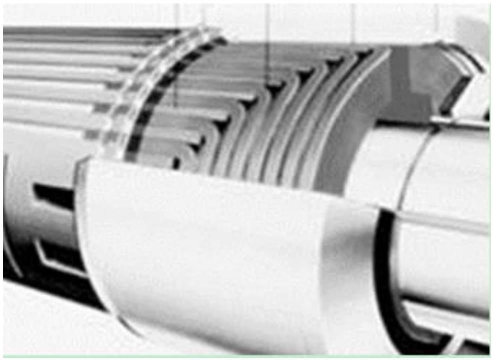


Fig. 1. Interference fit in generator system.

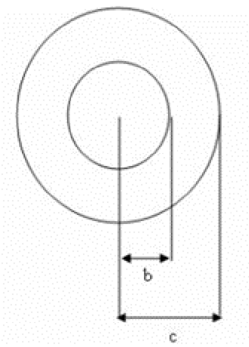


Fig. 2. Geometrical of component.

II. Calculation Procedure

On the theory, interference fit procedure is used heat to produce interference between two part together mecahnically, on the other hand to make one of which is inserted in to the other part. After assembly process, the part return to the original dimension, so this process produce the stress and deformation on the some areas.

As on handbook [7], [8], value of the area contact pressure can be solved by an equation:

Manuscript received February 27, 2016; revised July 25, 2017.

Aditya Sukma Nugraha is with 1Research Center for Power and Mechatronics, Indonesian Institute of Sciences, Jl. Sangkuriang Komp. LIPI Gd. 20, Bandung, 40135, Indonesia (e-mail: adit003@lipi.go.id).

Imam Djunaedi was with Research Center of Physics, Indonesian Institute of Sciences, Jl. Sangkuriang Komp. LIPI Gd. 80, Bandung, 40135, Indonesia (e-mail: imam_djunaedi@yahoo.com).

Hilman Syaeful Alam is with Technical Implementation Unit for Instrumentation Development, Indonesian Institute of Sciences, Jl. Sangkuriang Komp. LIPI Gd. 30, Bandung, 40135, Indonesia (e-mail: hilm003@lipi.go.id).

doi: 10.18178/ijmmm.2017.5.4.334

$$P = \frac{E.\delta}{b}\left(\frac{b^2\left(c^2 - b^2\right)}{2b^2c^2}\right) \tag{1}$$

where is P: Pressure in contact area, δ: radial interference at the assembly system, E : Young's modulus and the b and c is inner diameter and outer diameter like Fig. 2.

After we calculated the pressure at the contact area, we can determining hoop stress and tangential stress at the shrink fit component. And then, radial and tangential strees at the inlet cylinder can be solved by equation [8], [9]: σ

$$\sigma_{r(inner)} = \frac{c^2P}{\left(c^2 - b^2\right)}\left(1 - \frac{b^2}{r^2}\right) \tag{2}$$

$$\sigma_{\theta(inner)} = \frac{c^2P}{\left(c^2 - b^2\right)}\left(1 + \frac{b^2}{r^2}\right) \tag{3}$$

Finally, radial and tangential stress at the outlet cylinder can be solved by equation:

$$\sigma_{r(Outer)} = \frac{b^2P}{\left(c^2 - b^2\right)}\left(1 - \frac{b^2}{r^2}\right) \tag{4}$$

$$\sigma_{\theta(Out4r)} = \frac{b^2P}{\left(c^2 - b^2\right)}\left(1 + \frac{b^2}{r^2}\right) \tag{5}$$

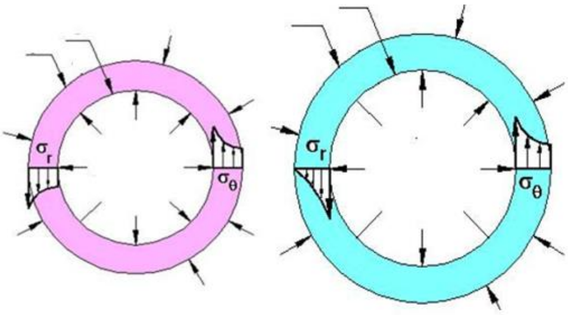


Fig. 3. Radial and hoop stress distribution.

To illustrated the above equation can be seen in Fig. 3, that the hoop stress distribution the hoop stress is the force worked at circumferentially (perpendicular both to the axis and to the radius of the object) in both directions in the cylinderical component and the radial stress is a component of the stress tensor in cylindrical coordinates.

III. Data and Numerical Model

In this paper, geometry of the interference fit make standard code in mechanical role. Dimension constraint use ISO 286-1(2010), ISO 286-2(2010) and ANSI B4.2 (1978), with the nominal size 838 mm and use the above standard resulting size tolerance can be seen in Table I.

TABLE I: The Tolerance size of Shrink Fit Component

	Hole	
Parameter	Value	Unit
Designation	838 H7	mm
Hole Upper Deviation	90	μm
Hole Lower Deviation	0	μm
Maximum Hole Size	837.944	mm
Minimum Hole Size	838	mm
	Shaft	
Parameter	Value	Unit
Designation	838 h6	mm
Shaft Upper Deviation	0	μm
Shaft Lower Deviation	-56	μm
Minimum Shaft Size	838	mm
Maximum Shaft Size	838.09	mm
	Fit	
Designation	838 H7/h6	
Fit Type	Transation Fit	

TABLE II: Parameter for Dynamic Calculation

Density	7850 kg.m^3
Young Modulus	20×10^7 Pa
Poisson Ratio	0.3

IV. Calculation and Result

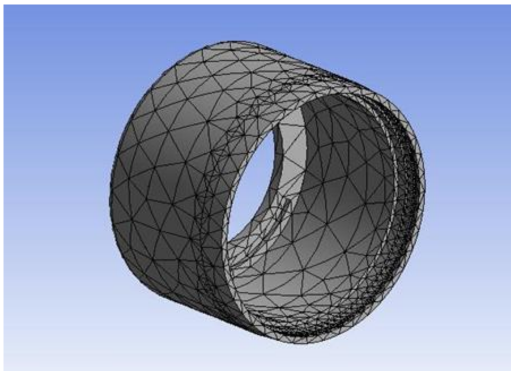
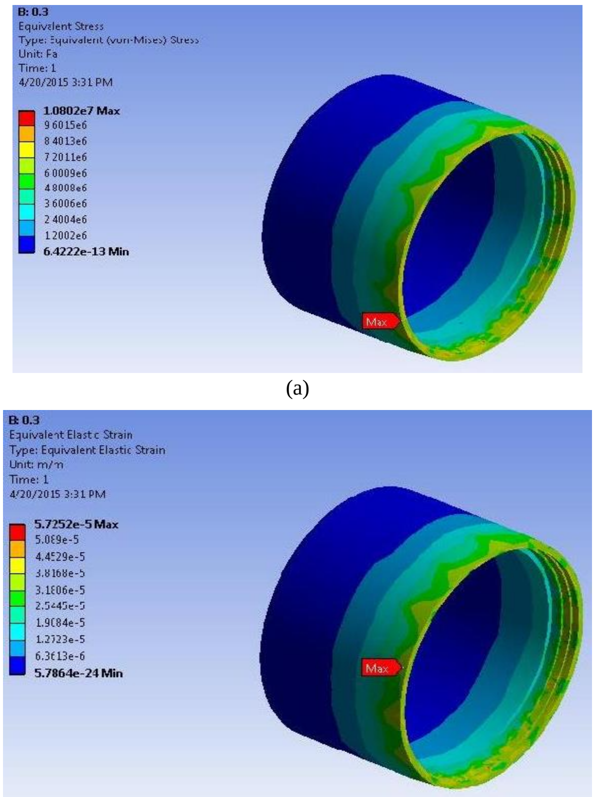


Fig. 4. Finite Eelement mesh.



(a)

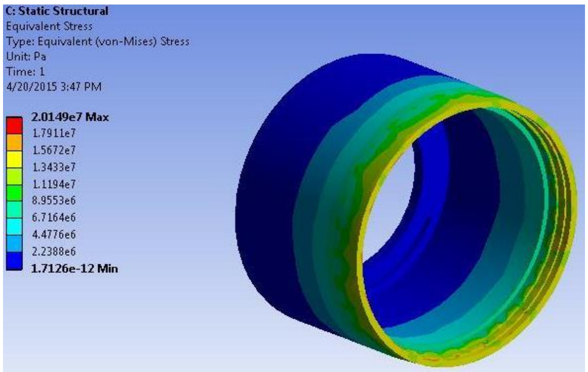
(b)

Fig. 5. (a)stress and (b) strain rresults by ANSYS Workbench 1(Hole inner diameter 838 mm, Shaft diameter 838.03 mm).

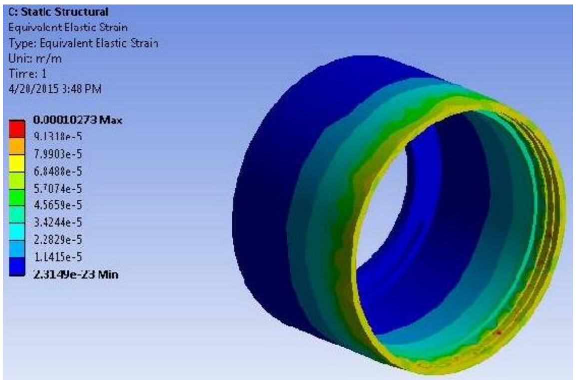
In this study, the instalaition process of retaining ring to generator system, by adding an initial pressure of 32 mpa. At the time of calculation, fem analysis with ANSYS applied 17125 elements and 28855 nodes are shown in Fig. 4. The meshing using tetrahedral mesh pattern. In 3D finite element modeling, this meshing prefer strongly and powerfull to analysis some problem. Steel is selected as the material of the retaining ring. The parameter of the material properties are shown in Table II, the content is density, young modulus and pisson ratio, respectively, and the the paramer can be used in simulation in retaing ring.

Results by ANSYS Workbench Analysis are shown in the Figs. 5 to 7 and Table III. Fig. 5 shows the countour of von misses stress and elastic strain for the loads and boundary condition with hole inner diameter 838 mm, Shaft diameter 838.03 mm. From Figs. 5 and 7, the highest value of equivalent stress result of assembly of retaining ring into generator winding is 1,08 $\times$ 10^7 Pa, the located in end thin walls, as seen in the picture inte and the maximum value of deformation at the surface of thin wall, with rate 5,57 $\times$ 10^5 of elastic strain.

In the other hand, Fig. 6 shows ilustrates the countour of von misses stress and elastic strain for the loads and boundary condition with hole inner diameter 838 mm. The trend alamost same with happening inte Fig. 5, but the velue much greater compairising before. It can be seen from picture 6, he ratio is almost doble result than before. The maximum value of the equivalent stress is 2,01 $\times$ 10^7 Pa located in end thin walls of retaing ring and the maximum value of elastic strain is 1,04 $\times$ 10^4.



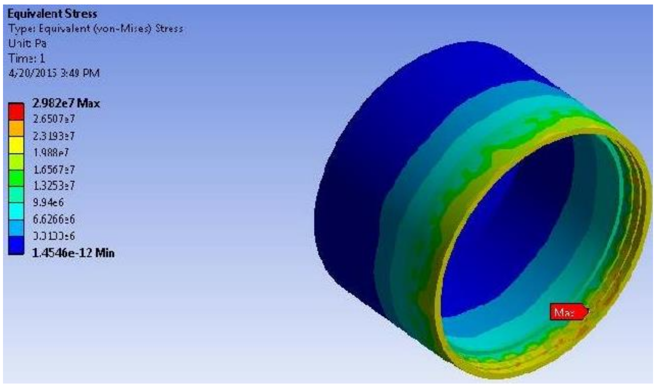
(a)



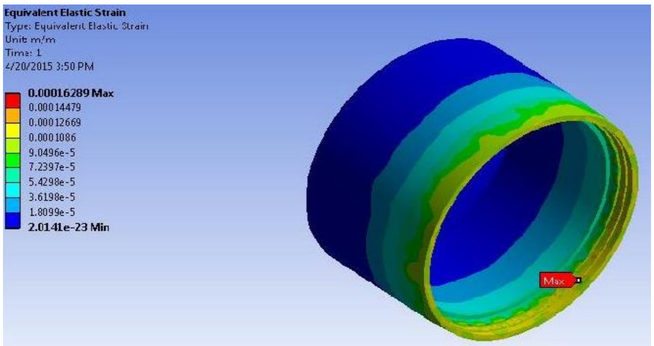
(b)

Fig. 6. (a) stress and (b) strain results by ANSYS Workbench 2(Hole inner diameter 838 mm, Shaft diameter 838.06 mm).

Lastly, in Fig. 7 discribes the countour of von misses stress and elastic strain for the loads and boundary condition with hole inner diameter 838 mm, Shaft diameter 838.06 mm. It can be seen the result of the simulation shows that in this simulation give a highest equavalent stress with 2,98 $\times$ 10^7 Pa and the maximum value of elastic strain is aproximate 1,06 $\times$ 10^4.



(a)



(b)

Fig. 7. (a) stress and (b) strain results by ANSYS Workbench 1(Hole inner diameter 838 mm, Shaft diameter 838.09 mm).

As described in Figs. 5-7, three cases with different combinations of initial interferences were performed. Detailed values of initial interferences are summarized in Table III. The highest stress is happened in last model with 2,95 $\times$ 10^7 of the stress.

TABLE III: THE RESULT OF SIMULATION

Density	Intereference (mm)	Max. Stress (Pa)	Elastic Strain
1st shrink fitt	0.03	1.08×10^7	5.7×10^{-5}
2st shrink fitt	0.06	2.01×10^7	1.02×10^{-4}
3st shrink fitt	0.09	2.95×10^7	1.62×10^{-4}

V. CONCLUSION

From the Table III it is clear that geometry tolerances between the shaft and shrink fit component will affect the amount of stress at the part of interference fit. In In this paper can be seen that the variation of the diameter of the small differences will cause a large effect on the stress and elastic strain that occurs. For further studies should be included the dynamic aspects in the calculation.

ACKNOWLEDGMENT

The author would like to tahnk to Research Centre for Electrical Power and Mechatronics- Indonesian Institute of Science, for facilitating the Finite Element Analysis Simulation using ANSYS.

REFERENCES

[1] S. Bin, H. Foroogh, T. Chris, and S. Davis, "Residual stresses in machined and shrink-fitted assemblies," *Journal JCPDS-International Centre for Diffraction Data*, pp. 675-682, 2009.

[2] R. Valery, L. Don, L. C. Raymond, and B. Micah, *Handbook of Induction Heating*, CRC Press.

[3] K. Niranjan, "Optimization of autofrettage-reautofgrettage percent and shrink-fit combination for optimum fatigue life in multilayer pressure vessels," *Journal of Applied Mechanical Engineering*, vol. 1, issue 3, pp. 1-7, 2012.

[4] A. P. Boresi, R. J. Schmidt, and O. M. Sidebottom, *Advanced Mechanics of Materials*, John Wiley & Sons Ltd, 1993.

[5] X.-L. Gao and S. N. Atluri, "An elasto-plastic analytical solution for the shrink-fit problem with a thin strainhardening hub and an elastic solid shaft," *Mathematics and Mechanics of Solids*, pp. 335-349, 1997.

[6] J. Hamid and B. Jalal, "An axisymmetric method of creep analysis for primary and secondary creep," *International Journal of Pressure Vessels and Piping*, pp. 597–606, 2003.

[7] O. K. Khoee, M. H. Abolbashari, and R. Tabasian, "Thickness optimization of cylindrical shell in shrink fit," *International Journal of Advanced Engineering Applications*, pp. 29-31, 2008.

[8] J. E. Shigley *et al*., *Mechanical Engineering Design*, McGraw-Hill Higher Education, 2011.

[9] ASME pressure vessel and boiler code section 8, Div. 3, *American Society of Mechanical Engineering*, New York, 2007.

Aditya Sukma Nugraha received his B.S. degree in mechanical engineering from Diponegoro University, Indonesia in 2006 and M.S. degree degree in mechanical engineering from InstitutTeknologi Bandung, Indonesia in 2014, He worked as a researcher at Indonesian Institute of Sciences (LIPI) from 2008 until now. His research areas are mechatronics, vibration, finite element method analysis, energy conversion, engineering design.

Imam Djunaedi received his B.S. degree in mechanical engineering from InstitutTeknologi Bandung, Indonesia in 1984. He had been research as a Ph.D research student at School of Computer & Electrical Engineering Curtin University of Technology, Australia from 1993 to 1996. He worked as a researcher at Indonesian Institute of Sciences (LIPI) from 1986 until now. His research areas are renewable energy, optimization, mechatronics, thermal engineering and power plant.

Hilman Syaeful Alam received his B.S. degree in mechanical engineering from JenderalAchmadYani University, Indonesia in 2003 and M.S. degree in mechanical engineering from InstitutTeknologi Bandung, Indonesia in 2010, He worked as a researcher at Indonesian Institute of Sciences (LIPI) from 2006 until now. His research areas are power generation, renewable energy, energy conversion, engineering design, intelligent material & structures, instrumentation& control, finite element and computational mechanics.