AMM-055467

The 23rd Conference of the Mechanical Engineering Network of Thailand
November 4 – 7, 2009, Chiang Mai

Finite Element Simulation of a Tolerance-Ring Installation Process

Piriyakorn Jirawattanakasem [1] and Sujin Bureerat[1]

[1] Mechanical Engineering Department of Khon Kaen University, Khon Kaen, Thailand 40002

Abstract

Tolerance-ring (T-ring) installation is a process used to assemble a bearing to a hard disk drive (HDD) actuator arm. In this paper, the finite element (FE) model of a T-ring installation process is proposed. The model employs both solid and shell elements for predicting contact forces during the installation. The FE computational results are compared to that from experiment, and it is found to be acceptable. It is also found that the solid element provides better computational results than the shell.

***Keywords**:* Finite element analysis, Tolerance-ring installation, Hard disk drive, Contact analysis.

1. Introduction

The tolerance rings are used in the hard disk drive (HDD) manufacturing due to low cost, ease to rework, repeatability of assembly, high cycle time and fulfilling a resonance constraint.

This application is to mount a bearing cartridge, which consists of two miniature ball bearings, and a center shaft with or without a cartridge sleeve. A view of the assembly is shown in Fig. 1.

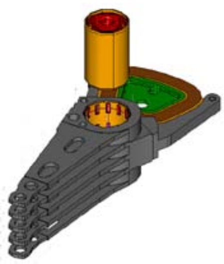

Fig. 1 Picture of tolerance ring assembly

A tolerance ring installation process starts by firstly placing an E-block on the fixture of a T-ring machine then putting a tolerance ring in the E-block. The next step is to place a bearing cartridge on the movable datum and seat on the outer sleeve. The top clamp of the T-ring machine presses on this outer sleeve until reaching the hard stop datum at the bottom. Once the bearing cartridge passed through the tolerance ring's bumps, plastic deformation occurred during the process remain while some of elastic deformations recover such that the contacting surfaces between the parts conform to each other. A view of the tolerance ring installation process is shown in Fig. 2.

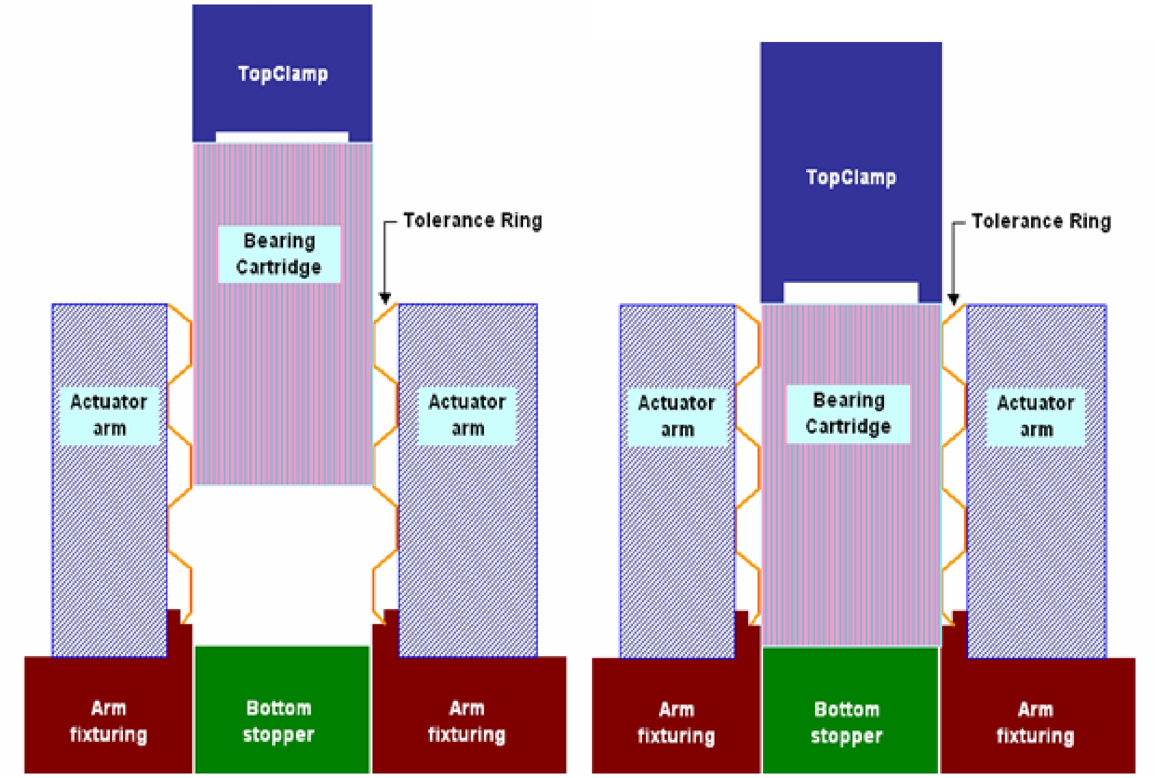

Fig. 2 Tolerance ring installation process

The aim of this work is to initiate the model to predict the contact force during installation and figure out the major contributor of this contact force. In the present work, the tolerance ring installation process is studied by performing a large-deformation explicit dynamic finite element analysis (FEA) using a commercial program. The E-block, tolerance ring and outer sleeve are opted as a model to be studied. The top clamp of Tolerance ring machine is moving with constant velocity 18 mm/s in the top-to-bottom.

2. FINITE ELEMENT ANALYSIS

The use of finite element analysis for material processing techniques, particularly for those involving contact mechanics, has been studied for many years [1-7]. Much research work towards the finite element procedures for contact-impact mechanics has been investigated [1-2, 5-6, 8]. A contact system of multiple deformable bodies can be thought of as the system having a static or dynamic equilibrium state such that contact constraints are imposed on the surfaces of the bodies contacting each other. The equation of motion of a body occupying a space domain Ω can be written as [1]

$$\frac{\partial \sigma_{ji}(\mathbf{x},t)}{\partial x_j} + b_i(\mathbf{x},t) = \rho a_i(\mathbf{x},t)$$

for i = 1, …, 3 and j = 1, …, 3 (1)

where **x** is a point on Ω, σ_{ji} is a Cauchy stress component, **b** is a body force vector, ρ is material density, and **a** is an acceleration vector. The boundary of the body Ω can be defined as Γ, and it is divided into three distinct parts as

$$\Gamma = \Gamma_d \cup \Gamma_f \cup \Gamma_c \quad (2)$$

where Γ_d is the boundary part having prescribed displacements, Γ_f is the boundary part that has prescribed forces, and Γ_c the boundary part contacting to other bodies. The prescribed boundary conditions can be written as

$$\begin{aligned} u_i(\mathbf{x},t) &= \bar{u}_i(\mathbf{x},t); \mathbf{x} \in \Gamma_d \\ \sigma_{ij}(\mathbf{x},t)N_j &= \bar{q}_i(\mathbf{x},t); \mathbf{x} \in \Gamma_f \end{aligned} \quad (3)$$

where $\bar{u}_i$ and $\bar{q}_i$ are prescribed displacement and boundary traction components respectively. N_j is the component of an outward normal vector at $\mathbf{x}$ on Γ_f. The initial conditions for displacement $(\mathbf{u})$ and velocity $(\mathbf{v})$ are also needed as:

$$\begin{aligned} \mathbf{u}(\mathbf{x},0) &= \bar{\mathbf{u}}(\mathbf{x}); \mathbf{x} \in \Omega \\ \mathbf{v}(\mathbf{x},0) &= \bar{\mathbf{v}}(\mathbf{x}); \mathbf{x} \in \Omega \end{aligned} \quad (4)$$

where $\bar{\mathbf{u}}$ and $\bar{\mathbf{v}}$ are prescribed displacement and velocity respectively. The contact constraints are assigned in such a way that the contact stress should be negative (or compressive) whereas there is no penetration between bodies. The contact constraints on Γ_f can be expressed as:

$$\begin{aligned} g(\mathbf{x},t) &= g(\mathbf{x},0) - \mathbf{u}(\mathbf{x},t)\cdot\mathbf{N} \geq 0; \mathbf{x} \in \Gamma_c \\ \bar{\mathbf{q}}_c(x,t)\cdot\mathbf{N} &\leq 0; \mathbf{x} \in \Gamma_c \end{aligned} \quad (5)$$

where g is the gap between contact surfaces, and $\mathbf{q}_c$ is the contact traction.

By using a finite element approach, the mathematical model (1) to (5) can be simplified to the matrix from of the system of second order differential equations. Using implicit and explicit time integration techniques can then solve the system of differential equations. Several numerical schemes dealing with contact mechanic finite element analysis have been proposed such as the Lagrange multiplier method, the augment Lagrangian method, and the penalty method. The methods have been implemented on numerous real world contact problems with success.

3. Tolerance Ring Installation Model

The four parts involving in the tolerance ring installation process, namely, an E-block housing hole, an outer sleeve of bearing cartridge, a tolerance ring, and a bottom datum that support the tolerance ring are modeled.

The E-block housing is meshed using 3-dimensional brick elements while the outer sleeve and the bottom datum are set as rigid bodies. Two types of elements, shells and solids, are used for meshing the T-ring so as to investigate the effect of using different element types.

During the tolerance ring installation, the arm is firstly placed on the fixture of the T-Ring machine. The ring is then put inside the arm bore whilst seating on the bottom datum. The bearing cartridge is placed on the movable supporting shaft and pressed by the top clamp with the constant velocity. This top clamp presses on the outer sleeve of the bearing cartridge until it reaches the stopper at the bottom.

The bottom datum is made of stainless steel and hardly deformed during the tolerance ring installation because it is used to support only the tolerance ring. Thus, it is simulated as a rigid body. The outer sleeve and tolerance ring are made of stainless steel while the arm is made of aluminum. The material models defined in the analysis are bilinear kinematics hardening. The friction coefficient between tolerance ring and outer sleeve is 0.70-0.95 [10]. The results of the reaction force at outer sleeve and tolerance ring are evaluated. The material properties used in the simulation are listed in Table 1.

Table 1 Material Properties used in the Finite Element Analysis

Material Properties	Type of Material	
	Stainless steel	***Aluminum***
Elastic modulus, *E* (MPa)	190,000	71,016
Yield stress, *Y* (MPa)	206	275
Poisson ratio	0.32	0.33
Mass density (kg/m^3)	7889	2700

All the elements used in the analysis are eight-node brick elements. To reduce the computational time, only outer sleeve, which is one of the components of a bearing cartridge, is used in the model. As for the arm, it is focused only on the bearing bore area while the arm tip and fantail area are excluded. Meshing of the structure is illustrated in Fig. 3.

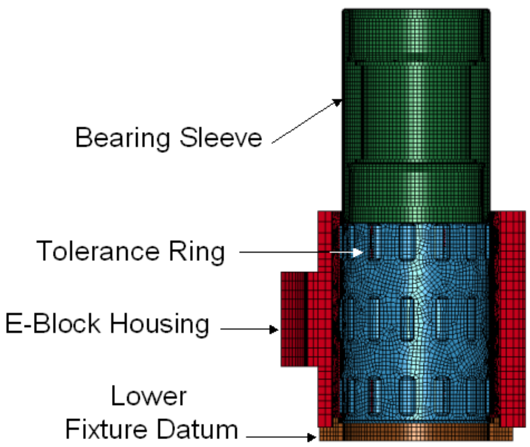

Fig. 3 Finite element grids of a tolerance ring installation

4. Simulation results

Fig. 4 displays the value of push-in force from FEA model of a tolerance ring. The lower graph is obtained form using solid elements whereas the upper graph is from using shells. The push-in force from experimentation is given in Fig. 5. From the figures, it can be seen that the graph of solid elements agrees well with that from the testing results. When using solid elements during the contact finite element process, the thickness of the ring is taken into consideration. Nevertheless, when using shells that have a higher-order polynomial function compared to solids, the ring is treated as having zero thickness although the effect of shell thickness is included in the mass and stiffness matrices. This means that the actual dimension of parts should be taken into account when performing finite element contact analysis.

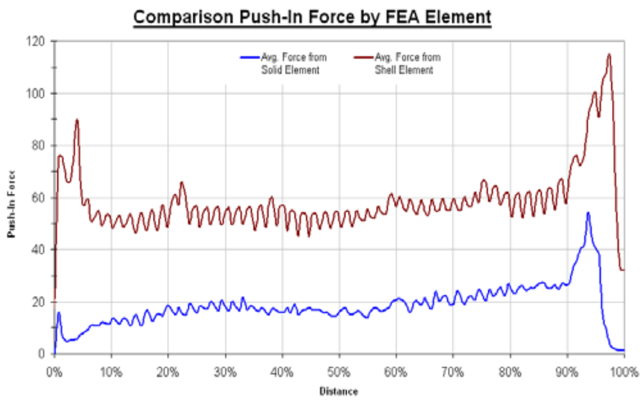

Fig. 4 Push-in force comparison between Shell and Solid element

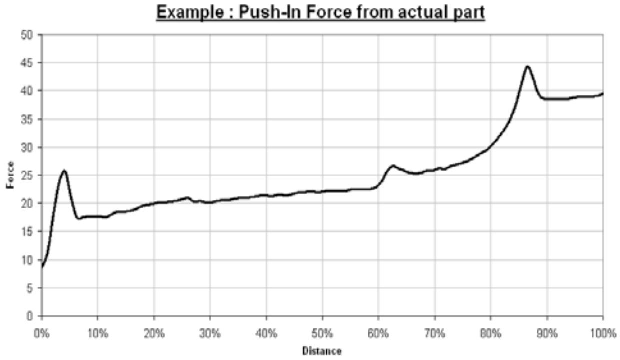

Fig. 5 Finite element grids of a tolerance ring installation [11]

5. CONCLUSIONS AND DISCUSSION

The finite element model of a T-ring installation process is proposed. Solid and shell elements are employed to model the T-ring. The computational results obtained from both finite element formulations are compared with those fro experimentation. It is found that solid element give the better and reasonable results. Base on the above-mentioned conclusion, the FEA model with solid T-ring will be continually used to figure out the major contributor to the push-in force. Moreover, it is also needed to investigate the impact of contact stresses to the parts.

6. Acknowledgement

This study is supported by Cooperate Project between National Electronics and Computer Technology Center (NECTEC) and Seagate Technology (Thailand) via Industry/University Cooperative Research Center (I/UCRC) in HDD Component, Khon Kaen University.

7. References

[1] Zhi-Hua Zhong, Finite element procedures for contact-impact problems, Oxford science publication, Oxford, 1993

[2] Wriggers P, Computational contact mechanics, Springer, Berlin, 2006

[3] Jaroslav Mackerle, Finite element methods and material processing technology, an addendum (1994-1996), Engineering Computations, Aug 1998 Volume: 15 Issue: 5 Page: 616 – 690

[4] Piela A.; Grosman F., Spatial modelling of swaging process using finite element method applied to axially-symmetrical problems, Journal of Materials Processing Technology, Volume 60, Number 1, 15 June 1996, pp. 517-522(6)

[5] F. H. Lin and A. A. Tseng, A finite element analysis of elasto-plastic contact problems in metal forming, Materials and Design, Volume 19, Issue 3, 1 June 1998, Pages 99-108

[6] Nagi Elabbasi, Jung-Wuk Hong and Klaus-Jürgen Bathe, On the Reliable Solution of Contact Problems in Engineering Design, International Journal of Mechanics and Materials in Design Volume 1, Number 1 / March, 2004

[7] Wang Xucheng, Chang Liangming and Cen Zhangzhi, Effective numerical methods for elasto-plastic contact problems with friction, Acta Mechanica Sinica, Volume 6, Number 4 / November, 1990

[8] Klaus-Jürgen Bathe, Finite Element Procedures, Prentice Hall, Englewood Cliffs, 1996

[9] Hariharan K, Balaji C. Material optimization: A case study using sheet metal-forming analysis. Journal of materials processing technology 2008; in-press

[10] Boonlong Kittipong, Experiment of friction coefficient between tolerance ring and outer sleeve, Seagate / December 2008

[11] Instron 3344, Instron Worldwide Headquarters, MA, USA. www.instron.com