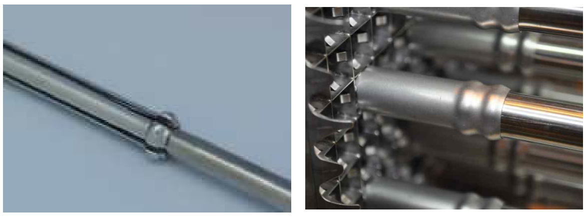
Shape Optimization Analysis of the Bulge Tool

Joo Young Yoo *, Nam Gyu Park, Jung Min Suh, Kyeong Lak Jeon
KEPCO NF Co., 1047, Nuclear Fuel Technology Office, Daejeon, 305-353, Korea
**Corresponding author: jyyoo@knfc.co.kr*

1. Introduction

The grid sleeves are mechanically fastened to guide tubes and to instrumentation tubes by means of an expansion joint (or bulgy joint) as shown in Fig. 1. Two layers bulge between dashpot tube and guide tube is adopted in ACE7, and three layers bulge in dashpot tube, guide tube, and sleeve is reviewed in the developing fuel.



(a) a bulge tool set (b) bulge joints
Fig. 1. Bulge tools and a product

Based on the field experience, bulge tool fractures are rarely occurred in the two layers bulge process. When the fracture occurred, it has been observed that the crack is in the neck such as shown in Fig. 2.

If three layers bulge process is applied, fractures could be occurred frequently. As a result, it is necessary to redesign the tool to enhance fatigue resistance.

(a) standard state (b) fracture state
Fig. 2. Fracture of the bulge tool

To robustly redesign the neck of the bulge tool, in this study, a finite element method is developed and analyzed.

Investigating acquired data from simulations, we can find out optimized design considering minimization of stress intensity and fatigue fracture risks throughout shape optimization analysis.

2. Simulation Model

In advance of shape optimization analysis, some important data and manufacturing information were investigated to simulate bulge process and to acquire more accurate results.

2.1 Material Properties

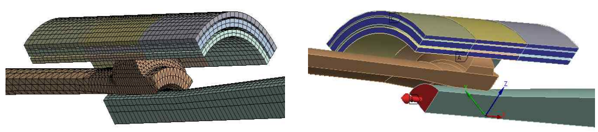
The bulge tool and bulge taper pin are cold steel alloy tools and the strengths are listed in Table 1. The guide tube and the dashpot tube are zirconium alloy materials and sleeve is made of steel.

Table 1 Strength of the bulge tool set

	Bulge Tool	Bulge Taper Pin
Yield Strength [MPa]	1.91E+03	1.9E+03
Tensile Strength [MPa]	1.95E+03	2.2E+03

2.2 Range of Model

To reduce analysis time, one-quarter model was used as shown in Fig. 3. This model has 4 directional symmetric structures.



(a) one-quarter model (b) boundary condition
Fig. 3. Simulation model

For completion of bulge joint simulation, a boundary condition was set to cross section of the bulge taper pin. Displacement condition is applied on the cross section, and it is back and forth motion with 80 mm distances along the axial direction.

3. Simulation Results and Discussions

3.1 Coefficient of Friction

It was needed to consider how friction coefficient affects stress intensity on the neck of the bulge tool. To do this, the ranges of coefficient factors were selected from 0 to 0.2. Fig. 4. shows the results of stress intensity on the neck of the bulge tool in three layers bulge simulation. When the bulge process is performed in KEPCO NF factory, grease is coated over bulge tools to reduce overloads of the machines. This grease coefficient factor is close to 0.05 [1].

The stress level at 0.05 is similar to stress level at 0 in Fig. 4. It means that the coefficient is not critical to the simulation. For this reason, the friction coefficient was ignored in this paper.

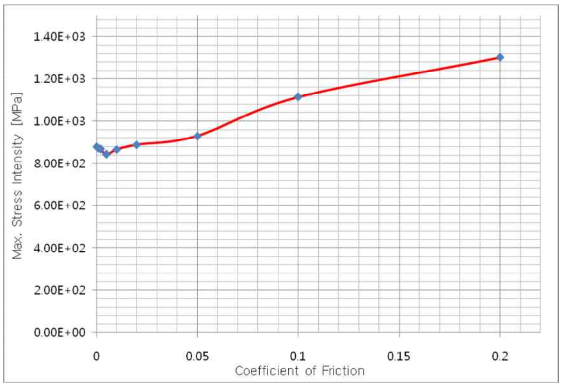


Fig. 4. Developed stress as a function of friction coefficient

3.2 Verification

One cycle of bulge process is that the bulge taper pin moves to back and forth in the bulge tool. And then, the head of the bulge tool swells out so that bulge shape can be formed on the tubes. Fig. 5. shows the simulated results of the deformation profiles of outer tube in the radial direction. As shown in Fig. 5, it is noticed that a little restoring range can be seen during plastic deformation. And final outer bugle diameter is 14.80 mm after bulging. The deformation is satisfied with the designated specifications.

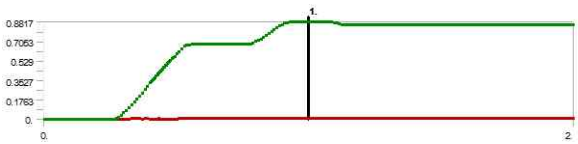


Fig. 5. Profiles of displacement during bulge process

3.3 Optimization

To improve more robust shape of the bulge tool in three layers tubes, we have to know how large stress developed on the neck according to many radius types as below objectives :

Minimize : Stress intensity
Subject to : 0.4 mm $\leq$ R $\leq$ 2.0 mm

Fig. 6. shows various results that range from radius of 0.4 mm to radius of 1.8 mm. Among these results, radius of 1.4 mm has the lowest level of stress.

In addition, the stress around the radius of 1.4 mm is lower than 650MPa which is a fatigue limit [2, 3]. That means the neck of the bulge tool at radius of 1.4 mm can be safe from fatigue fracture risks.

However, the radius of 1.8 mm shows drastically increasing stress. The reason is that rounding face contacts to dashpot tube during deformation. Also, it has one more problem. If the radius 1.8 mm is applied to bulge tool neck, it may deform different bulge shape in terms of the designated bulge shape. Therefore, it can be concluded that the radius of 1.4 mm is the best optimization among the results considering stress concentration and fatigue fracture.

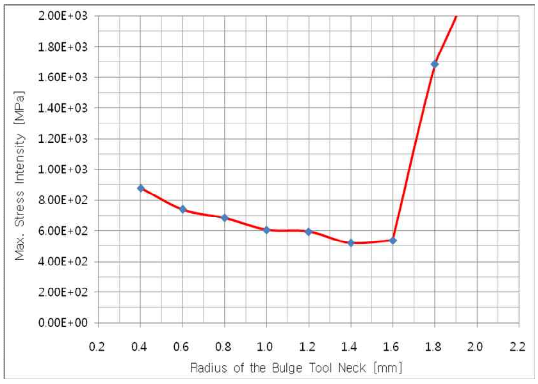
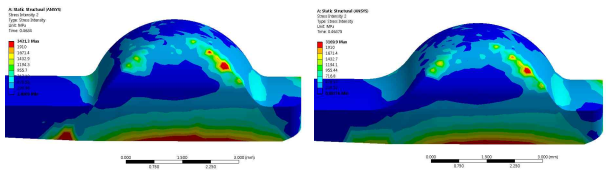


Fig. 6. Developed stress as a function of radius of neck

3.4 Improvement

Fig. 7. shows stress intensity distributions at radius of 0.4 mm and at radius of 1.4 mm. The neck with radius of 1.4 mm shows lower stress intensity levels than that of the design with radius of 0.4 mm. Therefore, it can be assured that stress concentration was reduced.



(a) previous (R 0.4 mm)   (b) optimized (R 1.4 mm)
Fig. 7. Stress distributions of the bulge tool

4. Conclusions

For improvement of the current bulge tool, in this study, ANSYS program is used. And various redesign ranges were carried out to estimate the levels of stress distributions on the neck of the bulge tool. Finally, improved design can be obtained from shape optimization analysis.

REFERENCES

[1] Chung Kyun Kim, “Tribology”, 2006.
[2] Norman E. Dowling, “Mechanical Behavior of Materials”, 1993.
[3] Julie A. Bannantine, Jess J. Comer, James L. Handrock, “ Fundamentals of Metal Fatigue Anaysis”, 1999.