*Transactions of the Korean Nuclear Society Autumn Meeting*
*Gyeongju, Korea, October 25-26, 2012*

# Mechanical Integrity Evaluation of X2-Gen Bottom Nozzle

Jin-Sun Kim, Joon-Kyoo Park, Sang-Yoon Jeon, Jung-Min Suh
*KEPCO Nuclear Fuel, Daejeon, 305-353, Korea*
*jinsun@knfc.co.kr*

## 1. Introduction

The bottom nozzle is a component of fuel assembly for structural support, distribution of reactor coolant and positioning of other fuel components. It consists of a stainless steel flat plate with an array of flow holes, legs at each of the four corners and skirt plates for connecting each leg.

In general, the test and finite element analysis (FEA) are performed to verify mechanical integrity of bottom nozzle. The design load conditions considered in evaluation are Condition I, II and Condition III, IV. The bottom nozzle is tested in the real condition of the worst load and manufacturing case, and analyzed using the conservative boundary conditions to confirm the mechanical integrity.

In this study, FEA with test boundary conditions is presented. Also, the test results are compared with FEA results to verify the validity of the analysis.

## 2. Mechanical Test

The bottom nozzle was tested under 4g design shipping load and 4,082 lbs LOCA(Loss of Coolant Accident) design load. All test loads corrected for as-built material property, plate thickness and test temperature.

The test was performed at room temperature in air condition utilizing the Instron Universal Testing Machine. The test arrangement is shown in Figure 1.

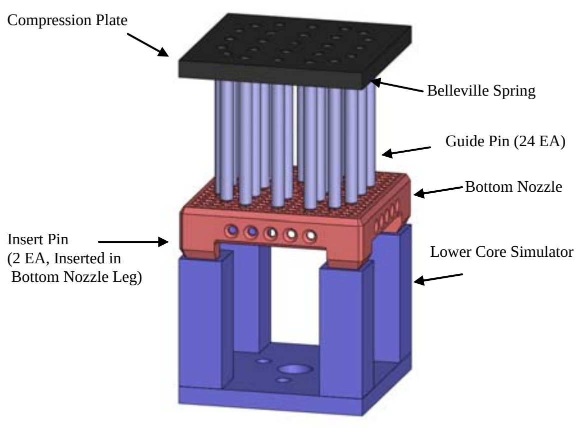



Figure 1. Bottom Nozzle Test Arrangement

For shipping loads, since this load is more conservative than Condition I, II loads. Thus no further test under these conditions is necessary. For Condition III & IV loads, the LOCA load was applied to LOCA analysis of LOCA analyses of reactor vessel and internals for the Kori Units 3 & 4 (KGA/KHB) and Yonggwang Units 1 & 2

The test load was applied axially through a set of Belleville springs which simulates the stiffness of the fuel assembly guide tubes. The bottom nozzle was placed on the lower core simulator with two insert pins to simulate the in-core boundary conditions.

The LVDTs and strain gages were mounted on the top and bottom surface of bottom nozzle to monitor the deflections and strains. The setup of test equipment is shown in Figure 2.

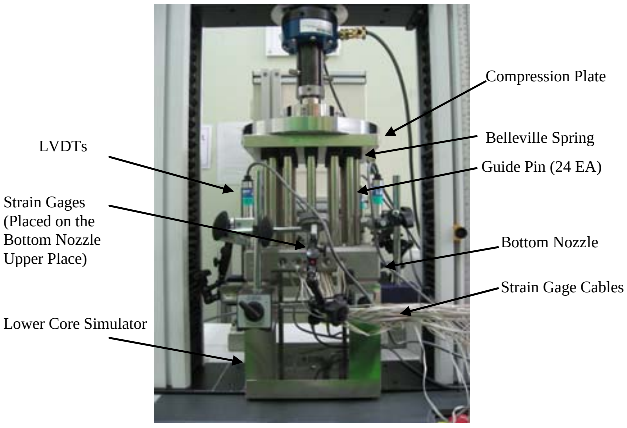



Figure 2. Bottom Nozzle Test Setup

Before the test was performed, the strain gages and LVDTs had been conditioned twice by loading and unloading the nozzle to 1,000 lbs. These were then balanced and zeroed.

## 3. FEA and Comparison

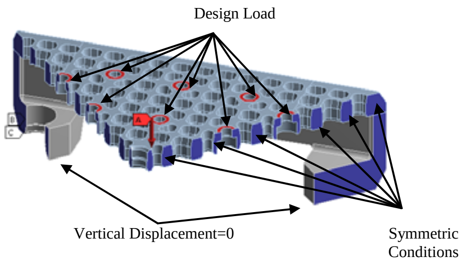
### *3.1. FEA*

The 3D solid model was generated using the SolidWorks 2011[1]. The model was then meshed and analyzed using the ANSYS V12.0[2]. Since geometry and applied load of the bottom nozzle are symmetric, only a 1/4 of bottom nozzle was modeled.

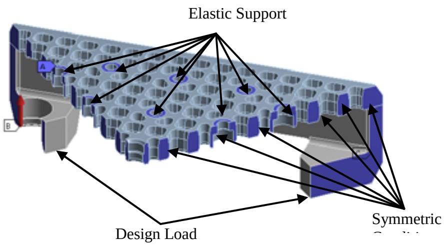
Figure 3 shows the 2 types of boundary conditions for analysis. The symmetric boundary conditions were applied along symmetric cut sections.

As shown in Figure 3(a), the design loads for shipping and LOCA conditions are uniformly applied to the contact area between the bottom nozzle and the guide tube. The bottom face of legs was vertically constrained.

Figure 3(b) shows that the axial restraint condition from the guide tube using the elastic support element given by the stiffness of Belleville springs and the design loads for shipping and LOCA conditions are uniformly applied to bottom place of legs.




(a) Type 1 Boundary Condition




(b) Type 2 Boundary Condition

Figure 3. Boundary Condition of FEA

### *3.2. Comparison with Test Results*

The results of 2 types FEA were compared with test results to verify the validity of the analysis. The mounted strain gage and LVDT positions are shown in Figure 4.

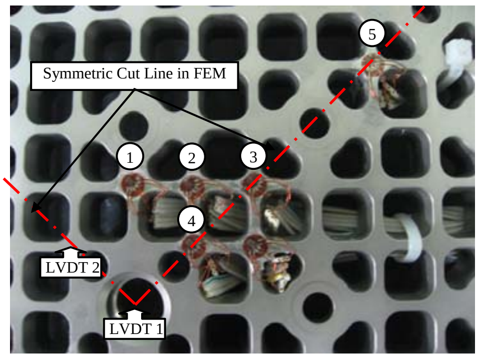



Figure 4. Positions of Strain Gage and LVDT

The stress intensity distribution of bottom nozzle is shown in Figure 5.

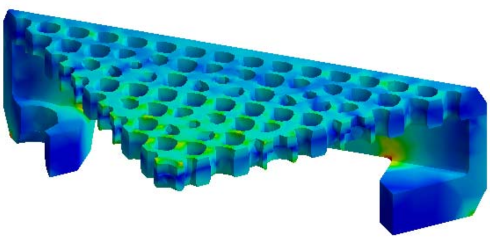


Figure 5. Stress Intensity Distribution

The comparison between the test and FEA can be assessed using the normalized values. Table 1 shows the normalized values of the maximum stress and deflection at each position. The normalized values of stress at the strain gage position are the ratio of the maximum principal stresses. The values at LVDT 1 and LVDT 2 are the maximum deflection that was measured at the center bottom region.

The type 2 FEM result is more similar with test result than type 1. Especially, The difference in the two analysis type's deformation result was evident, because type 1 FEM result did not simulate the gap between bottom nozzle and lower core support plate by whole bottom nozzle deformation.

The maximum deflection of the type 2 FEA is larger than the Type 1. It means the previous FEA is more conservative. Finally, the type 2 FEA is nore conservative and the results agree relatively well with the test results considering an error and uncertainty of measurement.

Table 1. Comparison of the Test and FEA

| | Positions | Normalization(=FEA/Test) | | | |
|---|---|---|---|---|---|
| | | Type 1 | | Type 2 | |
| | | Shipping | LOCA | Shipping | LOCA |
| Stress | 1 | 0.82 | 0.85 | 0.89 | 0.90 |
| | 2 | 0.88 | 0.88 | 1.06 | 1.06 |
| | 3 | 0.88 | 0.87 | 1.01 | 1.00 |
| | 4 | 0.97 | 0.96 | 0.99 | 0.99 |
| | 5 | 0.80 | 0.81 | 1.10 | 1.10 |
| | Avg. | 0.87 | 0.87 | 1.01 | 1.01 |
| Def. | LVDT 1 | 0.57 | 0.57 | 0.90 | 0.90 |
| | LVDT 2 | 0.64 | 0.64 | 0.92 | 0.92 |

## 4. Conclusion

In this study, the bottom nozzle was tested in the real conditions to evaluate the mechanical integrity of bottom nozzle and then the 2 kinds of FEA conditions were performed to calculate the more precise values of stress.

The type 2 FEA with elastic supported on the guide tube contact surface is more similar than type 1 FEA and estimated larger stresses than the stress from the test.

In conclusion, All FEM and test results sufficiently qualify mechanical requirements but the type 2 FEA boundary condition has a better agreement with test results. It is assure conservative.

## REFERENCES

[1] SolidWorks Corporation, "SolidWorks Office Professional 2011 ", 2011.
[2] ANSYS, Inc., "ANSYS Workbench V12.0", 2012.