20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20)
Espoo, Finland, August 9-14, 2009
SMiRT 20-Division V, Paper 1670

The Effect of Cover Unit Drop on Loviisa NPP Primary Circuit During the Maintenance Outage

J. Kähkönen and P. Varpasuo
Fortum Nuclear Services Ltd, Espoo, Finland
email: jukka.kahkonen@fortum.com; pentti.varpasuo@fortum.com

Keywords: Structural Impact, PWR, Primary Circuit, Finite Element Method

1 ABSTRACT

Information about the damage caused by the cover unit drop to the primary circuit during maintenance outage was needed for the probabilistic risk assessment update. To assess the damage, a large finite element method model of the primary circuit was prepared. The cover unit drop was modeled as a 75 ms event where 180 ton rigid object impacts on the reactor pressure vessel main flange.

Based on the analysis, the reactor pressure vessel suffered local damages near the supporting area and the supporting foundation damaged in some extent. Slight plastic deformations were observed at the primary piping near the pressure vessel. However, the integrity of the primary circuit was preserved.

2 INTRODUCTION

The Loviisa Nuclear Power Plant (NPP) in Finland obtained an extended operation license for the period of 2007-2027 in July 2007. One of the conditions set in the safety assessment document that accompanied the license extension approval was the requirement to update the probabilistic risk assessment (PRA) of the plant. For the PRA update, knowledge of damage caused by a very unlike accident of cover unit dropping during the maintenance outage was needed.

For this purpose a large finite element model (FEM) of the primary circuit was prepared. The model included all the major primary circuit components and the supporting foundation for the reactor pressure vessel (RPV).

3 FEM -MODEL OF THE PRIMARY CIRCUIT

The FEM -modelling and analysis were carried out using Abaqus/CAE and Abaqus/Explicit-6.7-EF softwares [1] with nonlinear material properties and contact formulations. The following parts of the primary circuit were modelled:

- The RPV and simplified geometry of the RPV internals (Figures 1 and 2).
- Primary piping accurately (Figure 1). Also the roller supports of the main valves were modeled.
- The steam generators (Figure 1) were modeled as stiff shell structures with realistic mass properties.
- The foundation was modeled as solid structure (Figures 3 and 4) with brittle cracking concrete material model available in Abaqus/Excplicit. The steel supporting ribs are embedded in side the foundation.
- The steel supporting ring on top of the foundation (Figures 3 and 4) was modeled accurately. Also the wedge structures underneath of the supporting ring were modeled.
- The pressurizer, the pressurzer line and the emergency feed lines didn't include in the model.

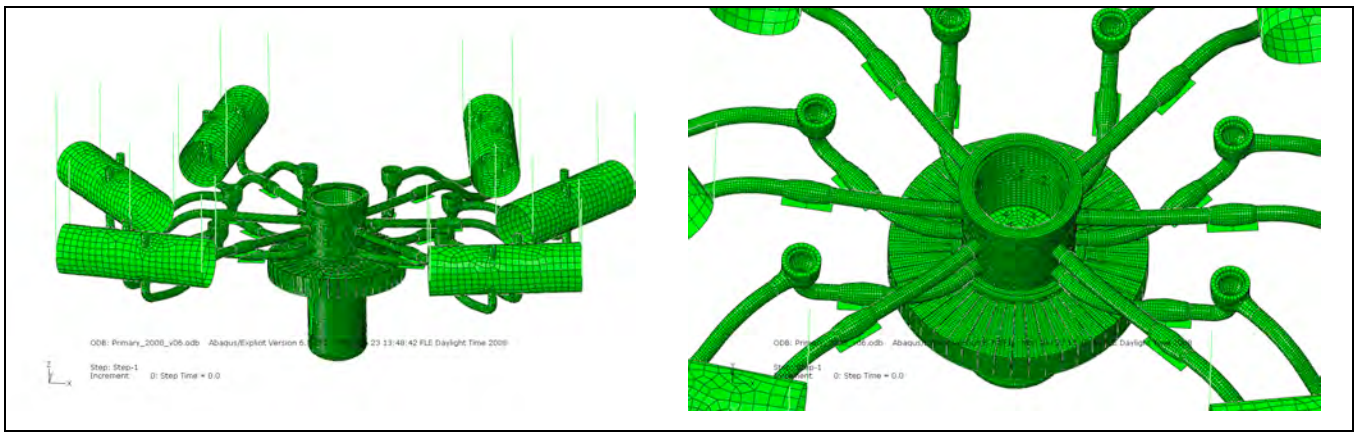

Figure 1. The FEM -model of the Loviisa NPP primary circuit.

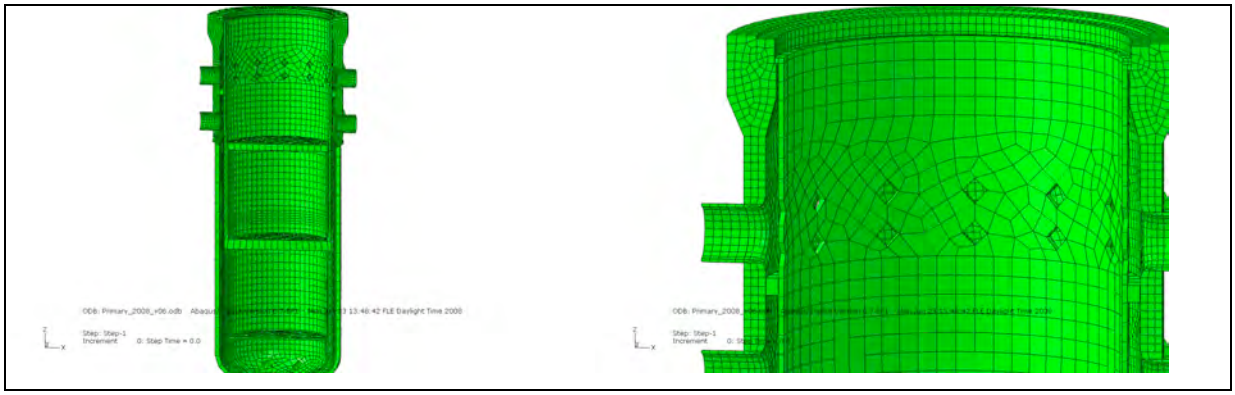

Figure 2. The FEM -model of the RPV and the internals.

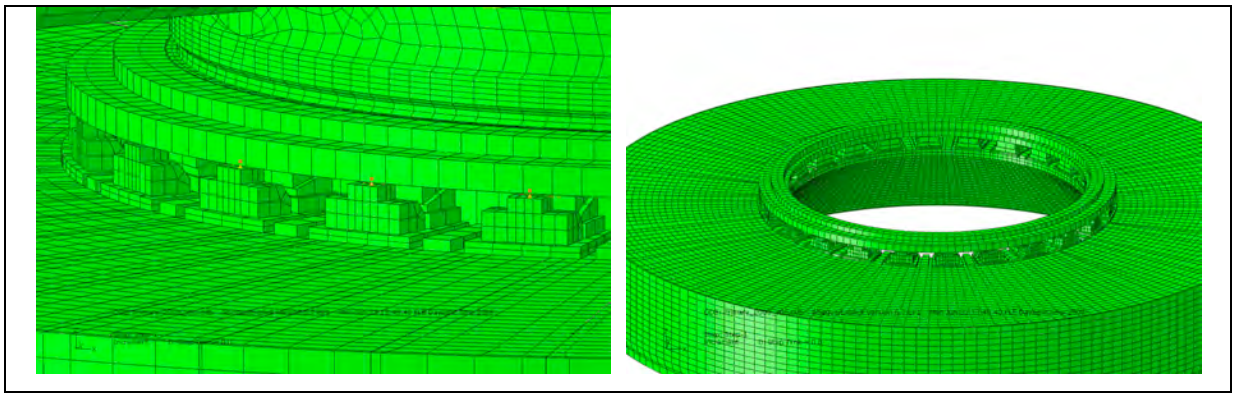

Figure 3. The supporting ring on top of the foundation.

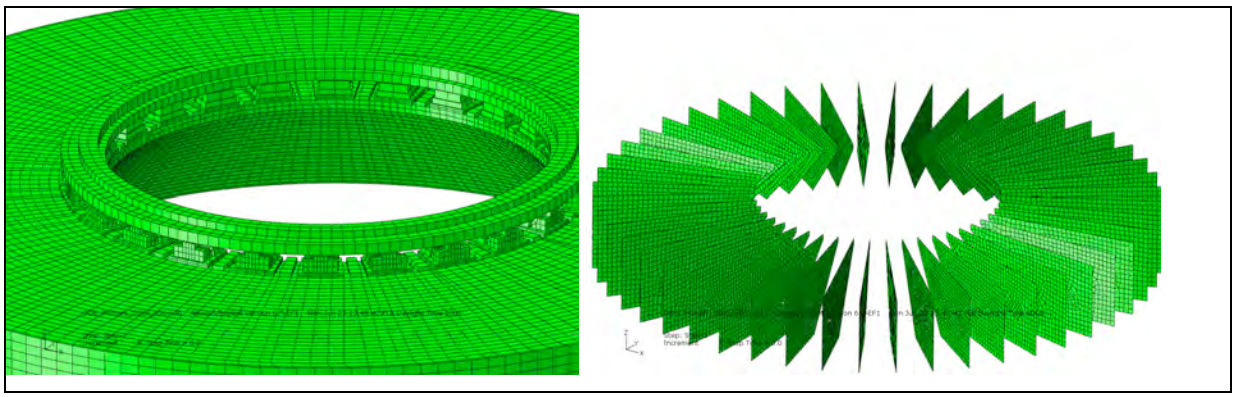

Figure 4. The supporting steel ribs inside the foundation.

The model has ca. 200 000 elements and about 900 000 degrees of freedom. The cover unit dropping accident was considered in FEM analysis as an event where 180 ton rigid object impacts directly on the RPV

main flange with velocity of 17 m/s. This impact velocity corresponds roughly situation where object is dropped from the height of 15 meters. The situation is illustrated in figure 5.

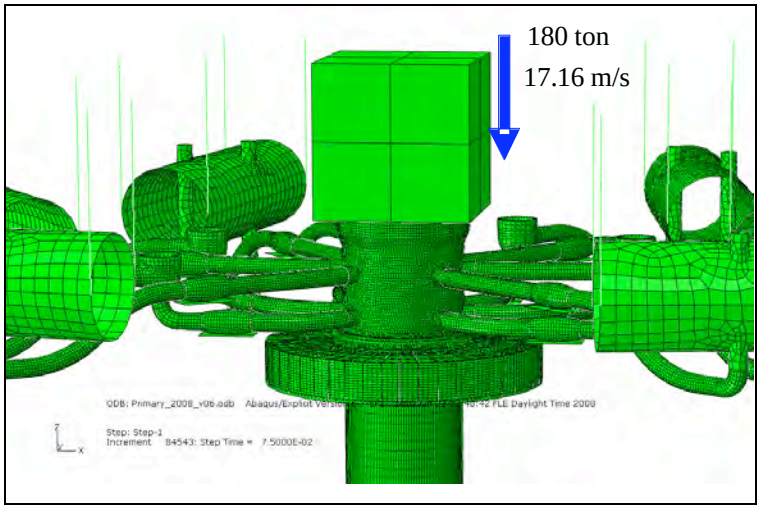

Figure 5. The dropping accident modelling. 180ton rigid block hits the RPV main flange with the velocity of 17.16 m/s.

4 RESULTS

In the Figure 6 the RPV main flange displacement during 75 ms event is illustrated by plotting the vertical displacements of three main flange nodes. The positive direction of the displacement is opposite to the velocity vector shown in Figure 5.

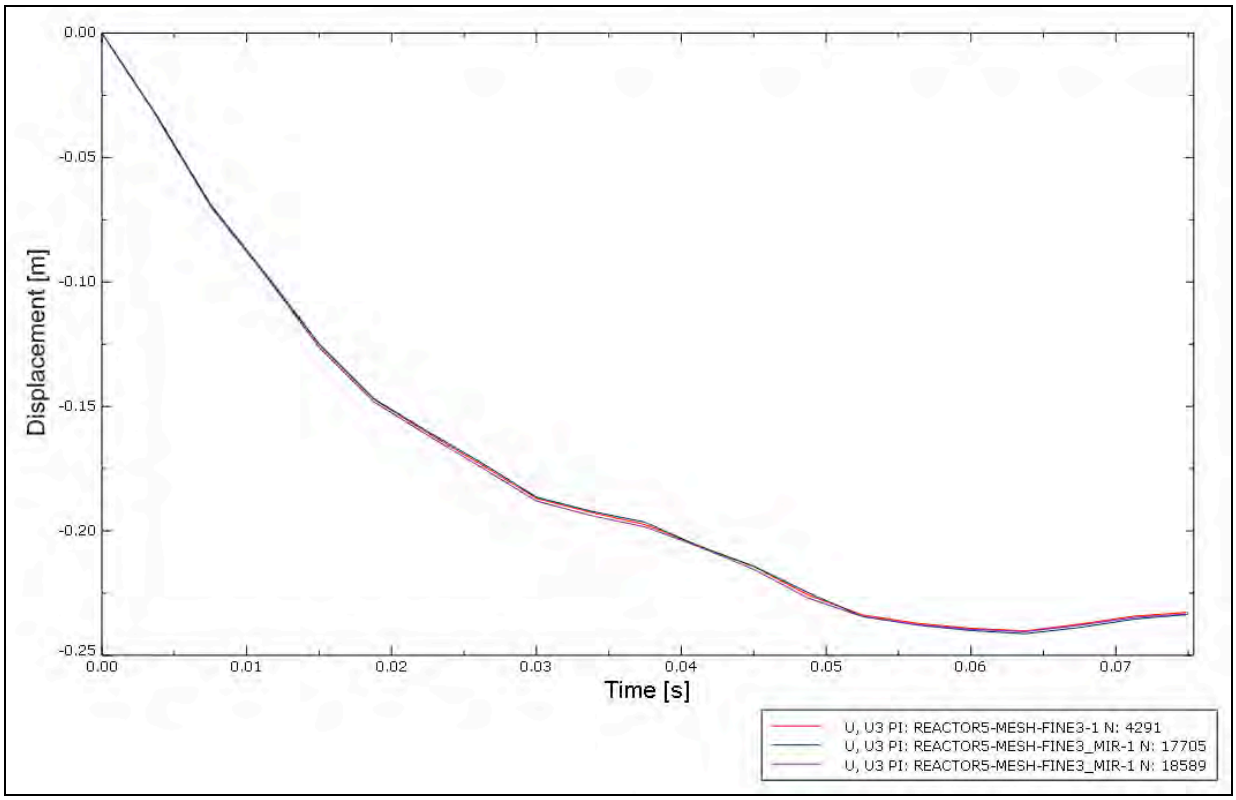

Figure 6. The deformation of the RPV main flange during 75 ms. Based on the figure the main flange of the RPV presses down about 25 cm.

The local nature of the primary circuit plastic deformations after 75 ms of the impact can be seen in Figure 7 where the contours of the so called work equivalent plastic strain shown.

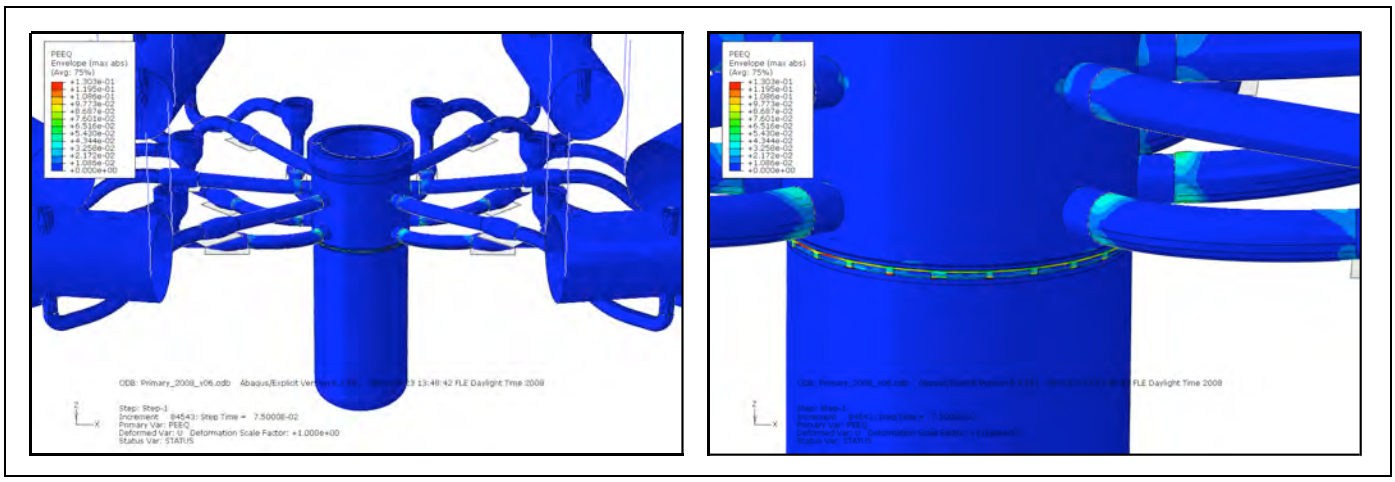

Figure 7. The work equivalent plastic strain distribution of the primary circuit after 75 ms of the impact. The maximum value is 13 %. The RPV damages locally near the supporting area.

5 CONCLUSION

Based on the FEM analysis the following conclusions could be made: 1) the primary piping doesn't exhibit much plastic deformations, 2) the RPV damages locally at the supporting area, 3) the supporting foundation damages severely but 4) the integrity of the primary circuit is not jeopardized.

REFERENCE

[1] ABAQUS version 6.7-EF1, 2007. Users documentation, Simulia Inc., Dassault Systèmes.