Brittle fracture analysis of primary nozzle of VVER 440 reactor pressure vessel

Keinanen H.,(1) Talja H.(2)
(1) Tectnical Research Center of Finland, Finland
(2) VTT Manufacturing Technology, Finland

ABSTRACT

The stage 2 of the IAEA Coordinated Research Programme on Management of Ageing of Reactor Pressure Vessel (RPV) Primary Nozzle is aimed at comparing methodologies for evaluating the current condition, assessing structural integrity and predicting remaining life of a selected VVER-440/213 RPV nozzle. The individual partners perform a detailed evaluation, considering brittle fracture, leak-before-break or fatigue failure phenomena and related assessment methods.

This paper presents the brittle fracture analysis of the selected primary nozzle from one unit of the Hungarian VVER-440 nuclear power plant PAKS performed by VTT. It is assumed in the analysis, that there is a circumferential inside surface crack in the fusion line of a bimetallic weld. The nozzle is simultaneously subjected to operational pressure and to a postulated emergency coolant flow transient.

The results show that for the two alternate transients the safety margin against brittle fracture is relatively large.

1 INTRODUCTION

The objectives of this IAEA Coordinated Research Programme (IAEA CRP) are to exchange information on the state-of-the art in assessing remaining life of the reactor pressure vessel (RPV) nozzles and mitigating effects of ageing, and to perform a collaborative case study. Organizations from Bulgaria, the Czech Republic, France, Germany, Hungary, Russia, the United Kingdom, Finland and USA as well as CEC/IAM Joint Research Centre are taking part in the programme.

The stage I results of the IAEA CRP have been reported in [2]. In the stage I, RPV nozzles of different designs were described, the operating experiences were reviewed and significant ageing mechanisms were identified. During stage II different approaches were used to assess the structural integrity of a selected individual RPV nozzle portion taking into consideration relevant ageing mechanisms. Within stage III of the programme, the results and findings of stage II are summarised.
The brittle fracture analysis of the VVER 440/213 primary nozzle of the Hungarian nuclear power plant PAKS is presented in the following. In a welded joint of a Russian RPV nozzle a real fusion line crack has been found with a depth of more than 11 mm and circumferential extension equal to half-perimeter [6].

The nozzle was thought to be subjected to operational pressure and to emergency coolant flow transients. The first transient was stratified flow, which caused strip cooling loading conditions. The second one was perfect mixing, which caused axisymmetric cooling loading conditions. A detailed presentation of the brittle fracture analysis can be found in [3].

2 DESCRIPTION OF THE NOZZLE

The VVER-440 RPV shell is made of a quenched and tempered CrMoV low alloyed ferritic fine grained steel of Russian type 15Kh2MFA (2.5-3.0Cr;0.6-0.8Mo;0.25-0.35V;≤0.4Ni). The inner surface of the RPV shell has a double layer submerged-arc strip welded cladding. The surfaces of nozzle inner radius and nozzle hole area have also a double layer cladding, which is made with manual arc-welding. The cladding is stress relieved after welding. The nozzle weld groove surfaces on shell have a double-layer buttering manually welded with electrodes and stress relieved after welding.

The primary nozzle is welded with manual-arc welding to the buttered grooves on shell. No post weld heat treatment is applied. The inner and outer weld surfaces are grinded smoothly after welding. The safe-end and the pipe are made of 08Kh18N10T (18Cr10Ni, Ti stabilised austenitic steel (AISI 321)).

Thermal properties for the ferritic steel 15Kh2MFA and all other (austenitic) materials are given in Table 1. The mechanical properties for the nozzle were obtained from [2].

Table 1. Thermal properties for the nozzle materials. Density (ρ) is 7800 kg/m³.

<table>
<thead>
<tr>
<th>Temperature (°C)</th>
<th>20</th>
<th>100</th>
<th>300</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conductivity (N/(sK))</td>
<td>15Kh2MFA</td>
<td>34.7</td>
<td>36.7</td>
</tr>
<tr>
<td></td>
<td>All other</td>
<td>14.5</td>
<td>15.7</td>
</tr>
<tr>
<td>Specific heat capacity per unit volume (N/(mm²K))</td>
<td>15Kh2MFA</td>
<td>3.43</td>
<td>3.87</td>
</tr>
<tr>
<td></td>
<td>All other</td>
<td>3.72</td>
<td>3.95</td>
</tr>
</tbody>
</table>

3 COMPUTATIONAL MODEL

3.1 Finite element computation

The numerical computations were made using Abaqus 5.4 finite element software. The analyses were thermo-elastic-plastic small strain and small displacement analyses. Materials were modelled using von Mises plasticity.

A 30 degree section of the reactor pressure vessel (RPV) wall was modelled. The total height of the model was 3000 mm. The modelled length of the straight portion of the inlet pipe was 876 mm. The finite element model is presented in Fig. 1. A circumferential crack with depth of 18.1 mm was modelled. The crack tip was modelled as a blunted notch with a radius of 0.05 mm. The finite element mesh in the vicinity of crack tip is too coarse to characterise near tip strain/stress fields.
The normal operating pressure 12.4 MPa and corresponding resultant forces were modelled on the free (vertical) surface of the pipe and on the horizontal lower surface of the pressure vessel wall. The initial temperature of the vessel was 265°C. In the computation model the width of the stratified area was 90° in the circumferential direction of the pipe. The width of the strip cooled area was limited at the lower part of the model to approximate real conditions. The inner surface temperature varied from 133 to 265°C and the heat transfer coefficient from 1500 to 3000 W/m²K.

Symmetry boundary conditions were modelled on the upper face and on the vertical faces of the model. The nodes on the upper end of the model were fixed in the axial direction. The lower face of the model as well as of the free face of the pipe are constrained to remain planar. These planes are allowed to translate but not to rotate.

In the "free" face of the pipe the nodes were fixed to the initial plane which was free to translate but not to rotate (‘free’ boundary conditions). An alternative modelling was made in
such a way that axial shrinkage of the pipe was hindered (‘fixed’ boundary conditions). A prescribed displacement was given to the “free end” of the pipe. The magnitude of the displacement corresponded to that caused by the pressure loading.

J-integral was calculated using the contour integral method of Abaqus 5.4. From J-integrals stress intensities were calculated using the well-known plane strain formula.

3.2 Simplified computation

For comparison, the both loading cases (strip cooling caused by stratified flow and perfect mixing) were analysed also by simplified fracture assessment methods. The engineering fracture assessment program system "MAST" [5] available at VTT Manufacturing Technology was applied. Because there were not direct solutions for the nozzle geometry in the program system, some essential simplifications were made.

The geometry was described as an infinitely long cylinder with constant radius and wall thickness. The wall thickness corresponded to that at the location of the crack. In the case of strip cooling the program does not consider the circumferential heat flow. This gave slightly too small (conservative) temperature values in the strip area.

In stress intensity factor computation there was no solution available for $R/t$ values smaller than 5. The temperature and stress distributions due to the thermal shock are mainly defined by the wall thickness, not by the radius. Thus the inner radius was taken as 475 mm and the internal pressure value was correspondingly scaled down to 6.48 MPa. In the stress intensity factor calculation an axisymmetric crack case with axisymmetric loading was utilised.

4 RESULTS

The computed crack tip temperatures at different crack front locations are shown in Fig. 2 for both perfect mixing and stratified flow loading condition. The temperature of the crack tip remains rather high during the both transients.

![Figure 2](image_url)

Fig. 2. The computed crack tip temperatures for both perfect mixing (p) and stratified flow (s) loading condition at selected crack front locations (angular distance from crack plane (°) is shown).
To assess brittle fracture the ‘emergency condition’ -curve from the former Soviet code [4] was used giving

\[ K_{13} = 26 + 36 e^{0.02(T-T_k)} \]  

(1)

The curve is valid for all Russian pressure vessel materials and corresponding welded joints. The curve represents the lower bound of the experimental data collected from an "eye ball" estimate [1]. In this case the given value for \(T_k\) was 50° C.

Figs. 3, 4 and 5 show the calculated stress intensity factors as a function of crack tip temperature. The results of the engineering calculation (MASI) are included in the Figures. Considering the remarkable simplifications in the engineering calculation, it can be said that the agreement between engineering assessment and finite element results is reasonable.

In the perfect mixing loading case the ‘fixed’ boundary condition gives higher stress intensity factor values than ‘free’ boundary conditions. In the stratified flow loading case the differences were small.

The material toughness curve is not presented, because it would be located on the left (safe) side of the figures outside the scaling range. The results confirm, that safety margin against brittle fracture is relatively large under the assumed conditions.

![Graph showing calculated stress intensity factors as a function of crack tip temperature](image.png)

Fig. 3. Calculated stress intensity factors as a function of crack tip temperature, perfect mixing loading conditions causing smooth cooling. Results are shown for both ‘free’ and ‘fixed’ boundary conditions at selected crack front locations (angular distance from crack plane (°) is shown).
Fig. 4. Calculated stress intensity factors as a function of crack tip temperature, stratified flow loading conditions causing strip cooling. Results are shown for ‘free’ boundary conditions at selected crack front locations (angular distance from crack plane (°) is shown).

Fig. 5. Calculated stress intensity factors as a function of crack tip temperature, stratified flow loading conditions causing strip cooling. Results are shown for ‘fixed’ boundary conditions at selected crack front locations (angular distance from crack plane (°) is shown).
5 SUMMARY AND CONCLUSIONS

This report describes the brittle fracture analysis of primary nozzle of VVER 440/213 reactor pressure vessel of nuclear power plant PAKS. The work belongs to stage II of IAEA Coordinated Research Programme on Management of Ageing of Reactor Pressure Vessel Primary Nozzle.

A hypothetical circumferential crack was located in the fusion line of the bi-metallic weld and the nozzle was subjected to operating pressure and thermal transient due to emergency coolant flow. Coolant flow was either perfectly mixed causing smooth cooling conditions or stratified causing strip cooling conditions.

A three dimensional finite element model with circumferential crack was created and J-integral values were computed for the specified transients. Computed stress intensities from J-integrals were compared to the given material toughness curve from the former Soviet code.

For the given transients and given initial conditions it can be concluded, that brittle fracture was not predicted. The margin of safety against brittle fracture was relatively large. The non-severe thermal boundary conditions caused quite mild thermal transient.

Welding residual stresses were not taken into account in the finite element computations. For realistic modelling of the welding residual stresses, a comprehensive simulation analysis should have been done. This was outside the scope and resources of the presented work.

The modelling of the boundary condition at the free edge of the inlet pipe had a significant effect on the results. The selection of certain pipe length with the boundary condition of nodes fixed to a plane, which was constrained to displace in the axial direction of the pipe an amount equal to that caused by pressure loading, was thought to be a conservative approximation of the real situation.

A simplified engineering assessment provided results with a reasonable agreement with the detailed finite element analysis.

ACKNOWLEDGEMENT

The work reported here belonged to the project "Structural Analyses of Nuclear Power Plant Components" being a part of Nuclear Power Plant Structural Safety Research Programme (RATU 2) funded by the Ministry of Trade and Industry (KTM), Finnish Centre for Radiation and Nuclear Safety (STUK), IVO International Ltd, Teollisuuden Voima Oy (IVO) and VTT Manufacturing Technology.
REFERENCES


