Application of pipestress and ansys in stress analysis of nuclear pipe support – case study

S Dizdar[1], A Vucina[2], N Rasovic[2] and R Tomovic[3]

[1]BerDiz Consulting AB, Ranängsgatan 8-10, 41664 Göteborg, Sverige
[2]University of Mostar, Faculty of Mechanical Engineering, Computing and Electrical Engineering, Matice hrvatske bb, 88000 Mostar, Bosnia and Herzegovina
[3]University of Montenegro, Mechanical Engineering Faculty, Bul. Dzorda Vasingtona bb, 81000 Podgorica, Montenegro

E-mail: adisa@sve-mo.ba

Abstract: A power uprate of the nuclear power plant will affect some systems, which will be exposed to new loads, transients and operating parameters. After defining of new loads, transients and operating conditions, work to qualify a system begins with modelling of this pipe system as well as modelling of supports which are active in the system. Pipe supports in pipe model are defined depending on the supports function and also their correct stiffness. After analysis is done, reaction forces are obtained in points where pipe supports are defined in the pipe model. Reaction forces from pipestress analysis, in the points where pipe supports are defined, becomes attacking forces in pipe supports analysis. A complete calculation of support is explained as well as the way to use required standard. The calculation includes stiffness calculation, calculation of membrane stress and membrane plus bending stress. In order to qualify the support a limit load analysis is performed. Finally, it is showed that pipe support could be qualified according to the standard ASME NF-3200. This paper describes use of software Pipestress and Ansys in stress analysis of piping systems and pipe supports.

1. Introduction

In recent years, new designs for nuclear-related structures and components are necessary and optimization technique is considered as one of the tools to achieve better design [1]. Design and development of nuclear power plants have very strict demands for safety. Nuclear safety objective is to prevent any release of radioactive substances to the environment and mitigate consequences of such a release should it occur. Principles of nuclear safety imply multiple barriers (physical confinement of radioactive substances) and defence in depth or three levels of safety (prevent, protect, mitigate). Prevention implies design for maximum safety in normal operation and maximum tolerance for system malfunction i.e. use of design features inherently favourable to safe operation, emphasise quality, redundancy, inspectability and testability prior to acceptance for sustained commercial operation and over the plant lifetime. Protection assumes that incidents will occur in spite of care in design, construction and operation. Therefore it is important to provide safety systems to protect operators and the public and to prevent or minimise damage when such incident occur. Finally, mitigation means to provide additional safety systems as appropriate, based on the evaluation of effects of hypothetical accidents, where some protective systems are assumed to fail simultaneously with the accident they are intended to control. Therefore the nuclear power plants have to be designed

in accordance with the valid regulations. Nuclear piping is very important for operation safety in nuclear structures and stress analysis of nuclear piping is conducted in order to provide that the maximum stress does not exceed the limits according to valid regulations. Today, it is possible to provide stress analysis of piping system through different software as: Pipestress, Caepipe, Caesar II, Ansys etc. Pipestress [2] is used for calculation of a pipe systems for static, dynamic and thermal loads (e.g. weight, pressure, thermal expansion, earthquake, valve opening etc.). Based on results obtained in Pipestress analysis, Ansys [3] is used for calculations and evaluation of pipe supports in accordance with ASME standard (Section III, Division 1, Section NF-3200) [4].

2. Objectives

The main objective of this paper is to qualify pipe support according to the standard ASME NF-3200. Support is mounted in a pipe system that needs to be qualified for calculated loads according to the design specifications. This system belongs Safety Class 1 (in the nuclear industry all systems are divided into Class 1, Class 2, Class 3 and NNS). In a pipe model where support is placed, constraints are defined. How constraints appear on the system depends on the support function. In this case, as shown in Figure 1, the support holds a pipe in one direction, for this case in horizontal.
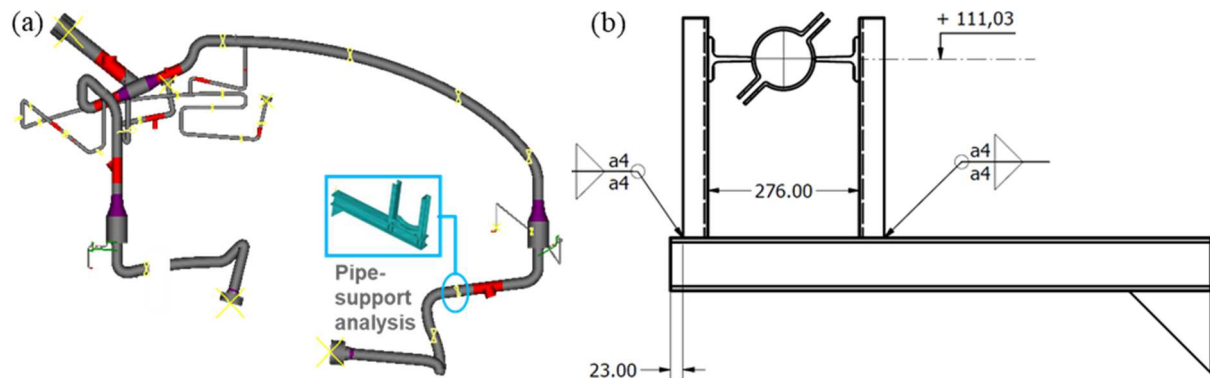

Figure 1. Pipe system (a) and layout of pipe-support (b).

Exact support stiffness is obtained from Ansys calculations and will be put into the Pipestress model. Pipe support has been made by carbon steel EN 1.0038. Allowed stress value for temperatures from 20°C to 350°C is obtained from [5]. Reaction forces from Pipestress analysis, in the points where pipe supports are defined, becomes active forces in pipe supports analysis in Ansys.

3. Stress analysis and evaluation

Stress analysis and evaluation of piping systems have been conducted in order to prove that the piping will not fail in various working conditions [6]. The use of Pipestress for stress analysis of nuclear piping systems [7] follows these general steps (Figure 2):

1. Establish the geometric model and finite element model in order to simulate various parameters of piping system (size, material, welding, valves, flanges etc.).
2. Apply boundary conditions i.e. add pipe supports and define point constraints. The stress of piping system and supports are closely related.
3. Apply loads in accordance with the design requirements. Loadings (the design, service and test loadings) shall be identified considering all plant or system operating and test conditions anticipated or postulated to occur during the intended service life of the component or support. Design loadings include design pressure, design temperature and design mechanical loads. Service loads are different combinations of loads that occur during operation of the reactor (Level A and B – the component or support must withstand these loadings without damage requiring repair. Primary and secondary stresses shall be accounted for all occurring loadings, no plastic deformations are allowed, fatigue analysis is required. Level C – the sets of limits

permit large deformations in areas of structural discontinuity which may necessitate the removal of the component from service for inspection or repair of damage to the component or support. Level D – this set of limits permits gross general deformations and damage requiring repair, which may require removal of the component from service).

4. Stress analysis. Combine loads for each condition in accordance with design requirements.
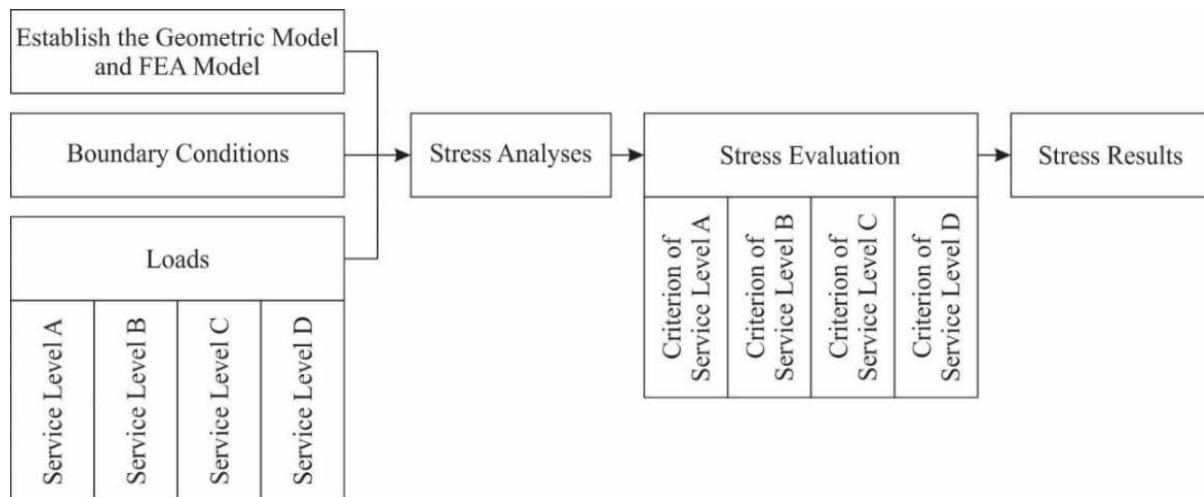5. Stress evaluation.

Figure 2. Stress analysis and evaluation for nuclear piping of Class 1,2,3.

At the same time with the modelling of piping system in Pipestress, modelling of pipe support in Ansys has been implemented. In the pipe model where the support is, constraints are defined. How constraints appear on the system depends on the support function. From Ansys calculations we get precise stiffness to be put into the pipestress model.

4. Case study

4.1 Stress Analysis of Nuclear Pipe Support
From pipe stress analysis, it is obtained reaction forces in points where support was defined (at the same time gets many other results like stresses, deformations etc.). The stiffness is calculated with a unit of 1 kN in the active directions of the support. A mean value of the displacement in the active direction is calculated for all nodes where the force is applied. Figure 3 shows the results as a contour plot.
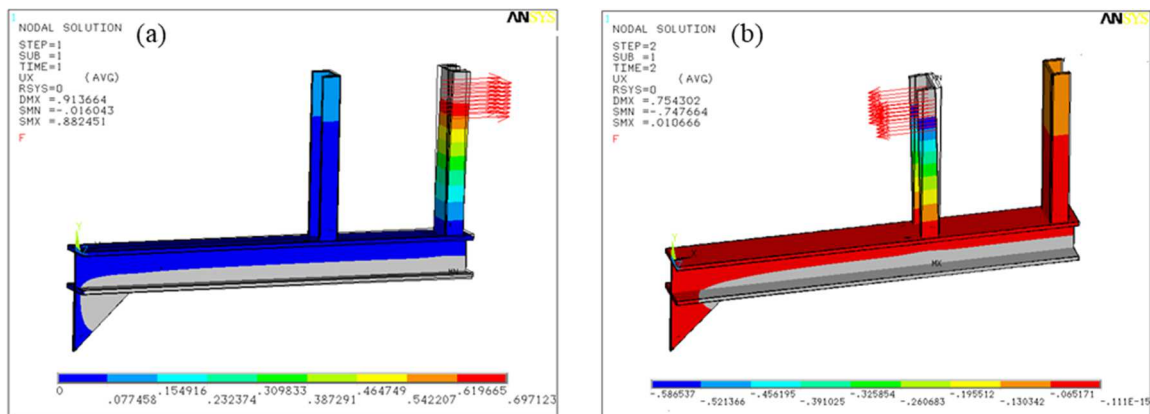

Figure 3. Mean displ. in positive x direction (a) and mean displ. in negative x direction (b).

The stiffness in the active directions is obtained from equation (1) and equation (2):

$$k_x = \frac{1}{0.69} = 1.4 \text{ kN/mm} \tag{1}$$

$$k_{-x} = \frac{1}{0.58} = 1.7 \text{ kN/mm} \tag{2}$$

Loads on the support are calculated in the piping analysis. Maximum and minimum forces (coordinates related to the ANSYS coordinate system) for each service level, respectively, are enveloped and presented in Table 1. A friction force, corresponding to 30% of the maximum level A load, is applied in the direction causing the highest stresses in the structure.

In order to establish the governing loading while evaluating the steel parts of the support, the loads given in Table 1 are scaled for each service level. With the acceptance limit for level A as the denominator the scale factors, K, become: 1.33 for level B and 1.5 for level C (NF-3221.2, F-1332.1 and F-1332.2) [8].

$$1.5 S_m = 1.5 \min\left\{ \left[\frac{2}{3} S_y(T) \right]; \left[\frac{S_u(20°)}{3} \right] \right\} = \min\left\{ [S_y(T)]; \left[\frac{S_u(20°)}{2} \right] \right\} \tag{3}$$

The scale factor for level D is calculated according to the equation (4):

$$K = \frac{\min\{\max[1.2 \cdot S_y; 1.5 \cdot S_m]; 0.7 \cdot S_u\}}{S_m} = \frac{\min\{\max[1.2 \cdot 211; 1.5 \cdot 120]; 0.7 \cdot 360\}}{120} = 2.1 \tag{4}$$

The loads are obtained from pipestress analysis and their values are shown in Table 1.

Table 1. Loads on support.

Service Level	Force F_y (N) (Ansys Coordinates)		The maximum forces F_y (N) scaled to level A
Level A	Max	606	948
	Min	-948	
Level B	Max	-784	753
	Min	-1002	
Level C	Signed	-1877	1251
Level D	Max	6223	5199
	Min	-10917	

After all loads are scaled down to level A, it can be seen in Table 1 that load Level D is a dimensioning load case for the support.

The support is evaluated by use of the same ANSYS-model as used for the calculation of the support stiffness. The maximum force of the applicable service level (from Table 1 – max. Level D force – 10917 N) and a friction force $F_{friction}$, are applied to the model. Both the membrane stress and membrane plus bending stress are calculated and compared to the allowable stress in Class 1 for the pertinent service level (material properties taken from ASME, Section II, Part D, [5]).

The allowable membrane stress in level D and class 1 is $K \cdot S_m = 2.1 \cdot 120 = 252$ MPa, and membrane plus bending stress is $1.5 \cdot K \cdot S_m = 1.5 \cdot 2.1 \cdot 120 = 378$ MPa (NF-3221.2, F-1332.1 and F-1332.2). The maximum force in level D is $F_y = 10,9$ kN and the used friction force $F_{friction} = 0.3 \cdot 0.948 = 0.28$ kN.

The results of the stress analysis are given in Figure 4 and Figure 5 and the results show that the membrane and membrane plus bending Tresca effective stresses are too high. Hence, further analyses are necessary in order to qualify support.
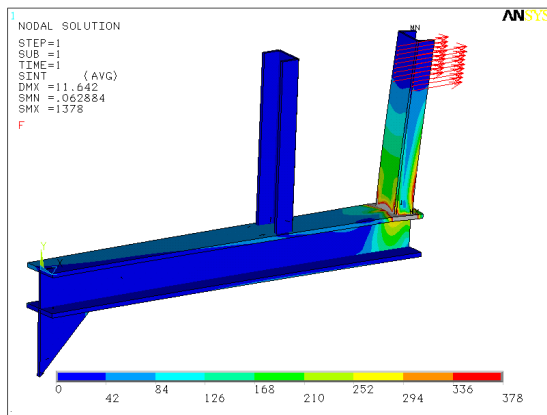
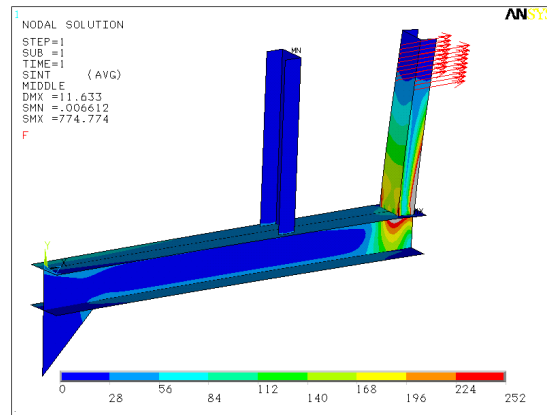**Figure 4.** Membrane + bending stresses.

Figure 5. Membrane stresses.

In order to qualify the support, a limit load analysis has been performed. The limit load is here defined as: the maximum load the structure carries when the accumulated plastic strains plus the elastic strains reach 5% somewhere in the structure. According to [5], the limit load, for steel EN 1.0038, in level A, B and C is related to the yield strength Sy=211 MPa, in level D Sy=252 MPa. The allowable force in Level A and B is 2/3 of the limit load, 0.8 x the limit load in level C and 0.9 x the limit load in Level D (NF-3221.4 and NB-3228.1 [9]).

Elastic ideal-plastic material constitutive behavior is assumed. The analysis does not include cyclic loading, whence isotropic hardening is used. Standard J2 plasticity (bilinear response ANSYS: BISO) is assumed and in order to secure convergence a (low) hardening modulus of 10 MPa is used. Linear shell elements (ANSYS SHELL181) are used to modeling support.

Figure 6 shows the von Mises total mechanical strain equal 5% when the support carries a vertically force of 8.1 kN (interpolated) combined with a friction force 0.3·0.948=0.28 kN and using a yield strength Sy=252 MPa. Hence, the limit load of the support is set to 8.1 kN and the allowable force in level D is equal to 8.1·0.9=7.3 kN (NF-3221.4 and NB-3228.1), which is lower than the maximum force, Fy (10,9 kN). The result of the limit analysis thereby shows that support cannot be qualified.
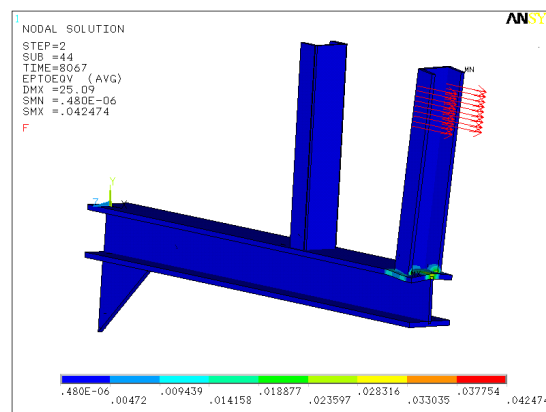

Figure 6. Total mechanical strains when support carries load of 8,1 kN and $F_{friction}$.

4.2. Qualification of nuclear pipe support

Since the pipe support could not be qualified for current loads according to ASME, Section III, NF-3200, there are two ways to go. The first way is to design much stronger support and the other way is to make changes in existing ones. In this case study, it is decided to modify existing support structure. Redesign was made to reinforce the support in order to carry current loads (Figure 7). The redesigned support must be verified using the same procedure as the actual design.
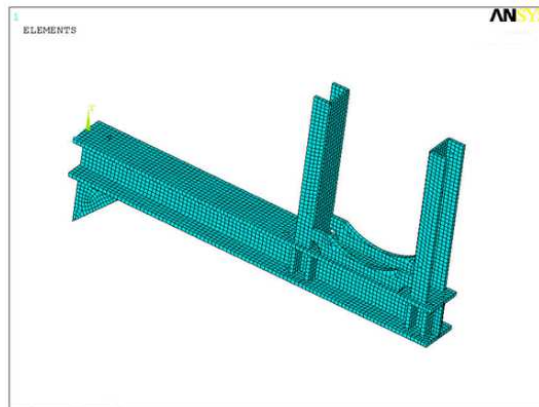
Figure 7. Model of redesigned support.

For redesigned support the stiffness is calculated with a unit of 1 kN in the active directions of the support. A mean value of the displacement in the active direction is calculated for all nodes where the force is applied. Figure 8 shows the results as a contour plot.
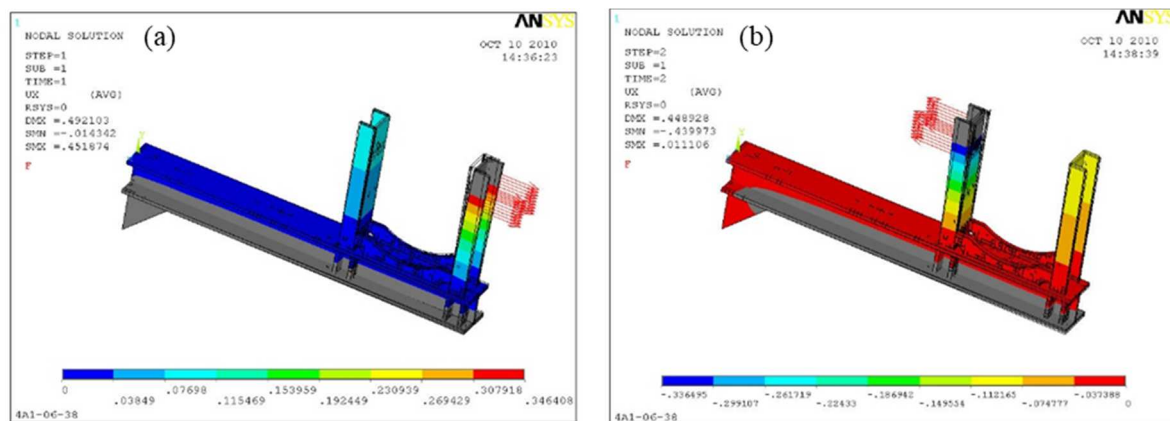

Figure 8. Mean displ. in positive x direction (a) and mean displ. in negative x direction (b).

The stiffness in the active directions is obtained from equation (5) and equation (6):

$$k_x = \frac{1}{0.346} = 2.89 \text{ kN/mm} \tag{5}$$

$$k_{-x} = \frac{1}{0.336} = 2.97 \text{ kN/mm} \tag{6}$$

The same loads are applied on redesigned support. The results of the stress analysis are given in Figure 9 and Figure 10 and the results show that the membrane and membrane plus bending. Tresca effective stresses are still too high. Hence, further analyses are necessary in order to qualify the support.

In order to qualify the support a limit load analysis has been performed. Figure 11 shows the von Mises total mechanical strain equals 5% when the support carries a vertically force of 13.4 kN (interpolated) combined with a friction force 0.3·0.948=0.28 kN and using a yield strength Sy=252 MPa.

Hence, the limit load of the support is set to 13.4 kN and the allowable force in level D is equal to 13.4·0.9=12.1 kN (NF-3221.4 and NB-3228.1), which is higher than the maximum force, Fy (10.9 kN). The result of the limit analysis thereby shows that the support can be qualified.
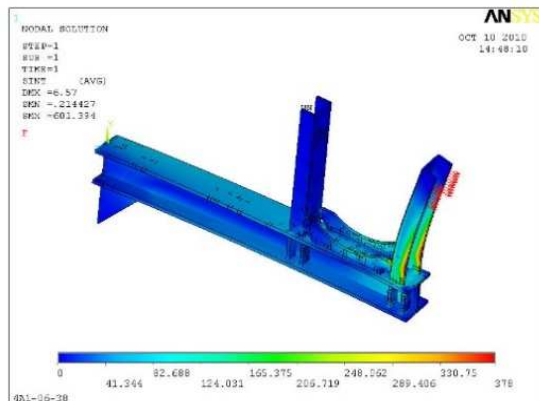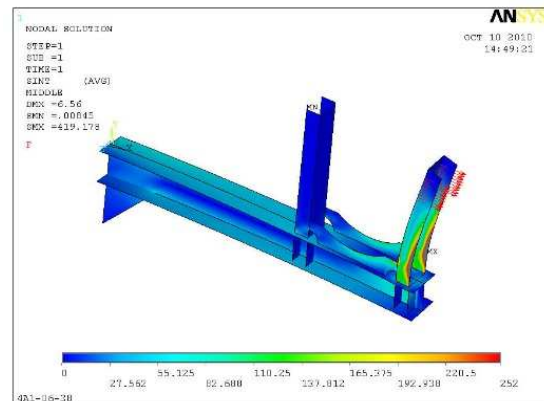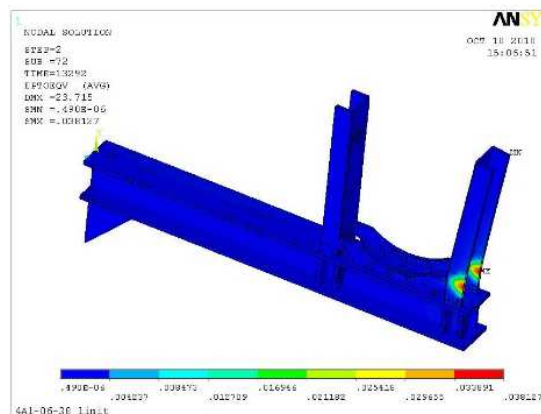
Figure 9. Membrane+bending stresses.

Figure 10. Membrane stresses.

Figure 11. Total mechanical strains when support carries load of 13.4 kN and $F_{friction}$.

5. Conclusion

In this case study it is showed that existing pipe support could not be qualified for defined loads according to ASME, Section III, NF-3200. In order to resolve the problem it was possible to design new stronger support or to make modifications on existing one. Very often it is difficult to make a whole new support for several reasons. The support is mounted several meters high and requires a lot of preparatory work to do this safely. It would be difficult to get new support because of many pipes around existing support. And finally, all this means that the reactor has to be stopped for several hours to replace the pipe support and it becomes too expensive. Therefore, existing support is modified and verification using the same procedure as the actual design is obtained. Limit load analysis is performed. The results of the limit load analysis show that redesigned support can be verified for current loads according to ASME, Section III, NF-3200.

References

[1] Choi W-S and Seo K-S 2011 Shape Optimization of a Torus Seal under Multiple Loading Conditions Based on the Stress Categories in the ASME Code Section III, *Nuclear Engineering and Design* **241**(8) 2653-2659

[2] Peps 6.0, DST Computer Services S.A, *Analysis Program for Nuclear & Industrial Piping*, https://www.dst.ch/ (accessed on May 2018)

[3] ANSYS ver.18.2, Mechanical Enterprise, *The Flagship Tool for Structural Simulation*, https://www.ansys.com/products/structures/ansys-mechanical-enterprise/ (accessed on May 2018)

[4] ASME Boiler and Pressure Vessel Code 2015, Section III, Division 1 – Rules for Construction of Nuclear Facility Components, Subsection NF-3200

[5]   ASME Boiler and Pressure Vessel Code 2015, Section II, Part D, Properties (metric)

[6]   Liu R, Fu Z and Li T 2013 Application of Peps in Stress Analysis of Nuclear Piping, *Journal of Applied Mathematics and Physics* **1** 57-61

[7]   Mao Q, Wang W and Zhang Y 2000 The Stress Analysis Evaluation and Pipe Support Layout for Pressurizer Discharge System, *Nuclear Power Engineering* **21**(2) 117-120

[8]   ASME Boiler and Pressure Vessel Code 2015, Section III, Rules for Construction of Nuclear Facility Components, Appendices, Appendix F

[9]   ASME Boiler and Pressure Vessel Code 2015, Section III, Division 1 – Rules for Construction of Nuclear Facility Components, Subsection NB-3200