Fatigue assessment of welded joints using the sub-modelling technique

João Milheiro
joaopmpmilheiro@gmail.com

Instituto Superior Técnico, Universidade de Lisboa, Portugal

November 2018

Abstract

Fatigue design of mechanical and structural elements is nowadays an essential field in engineering, that needs constant innovation to allow for ambitious projects to be carried out ensuring that the structures have large period of useful life. In the present work, the numerical results are combined with a standard based on experimental results and also with analytical extrapolation techniques. The extrapolation methods use different kinds of stress values as input and are intended to calculate the stress in certain weld points, where the finite elements method's results are not reliable. A fatigue assessment is performed in 3 different geometries where 3 attachments are welded with different arrangements. In a first step, the most demanding load is identified from among the 7 available and is then used to make the comparison of the geometries in terms of stress range on the weld details. As expected, there are no experimental values to compare with all the numerical results. That is the role of the standard for welded joints used, which has correction factors to be adapted to different geometries. The third geometry proposed, showed a decrease in the stress range of about 30%, depending on the method used.

Keywords: Fatigue assessment, stress range, hot-spot stress, weld geometry, FEA.

1. Introduction

The damages resulting from the fatigue process are progressive and permanent and can lead to the nucleation and propagation of cracks in the components.

The loads involved in the fatigue damage of components are most of the times below the yield stress of the material, so there is no plastic deformation and consequently there is no way of noticing visually a crack appearing and propagating

Some years ago, it was common in engineering to design components only considering the material's yield stress criterion, with the purpose of avoiding any permanent deformation in the structure. For conservative reasons, the allowable stress to consider in the project is then the yield stress divided by a safety factor that can be related to the material type, the loads type, the severity of the failure, etc.

The normal aspect of a surface broken due to fatigue is represented in figure 1. In the bottom there is an area with a smooth surface, where the crack propagated slowly and in a ductile way (depending on the material). The area on the top shows the result of a fast fragile, that as a brittle behaviour, after the instabilization is achieved.

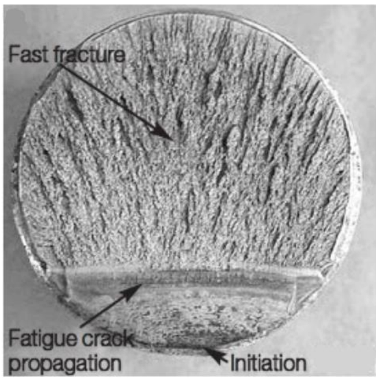


Figure 1 - Fatigue failure surface [1]

When designing components for fatigue, the most important stress parameter to consider is the stress range between the maximum and minimum value applied in the components. The yield stress value should also be considered to quantify the value of the cyclic load mean stress (the medium value of the fatigue load) that has negative effects on the fatigue life if it has a positive value. A positive mean stress indicates that the load has a tensile nature, pushing the S-N curve downwards and decreasing the endurance limit [2].

In order to adapt everything introduced before to the complexity of the geometry in this study and the lack of experimental results, the standard EN 13445-3 was taken in consideration. This standard was developed for a specific and controlled case and has correction factors and parameters to adapt for other cases, when the concern is the fatigue assessment of weld details.

2. Sub-modelling technique

The sub-modelling method was applied to the Vacuum Vessel (VV) structure to reduce its size, the number of elements of the mesh and consequently, the time required to perform each analysis. If the analysis is carried out with the full geometry, in addition to the heavy resulting mesh, the stresses evaluated would show value from all the geometry parts, including ranges that are way different from the values in the interest area.

To create a sub-model, a boundary surface is defined in some point of the geometry that excludes the unnecessary areas, leaving enough distance between the interest areas and the boundaries for the stresses and deformations to stabilize and create a field with its real values, that will be evaluated later to extrapolate the hot-spot stress range.

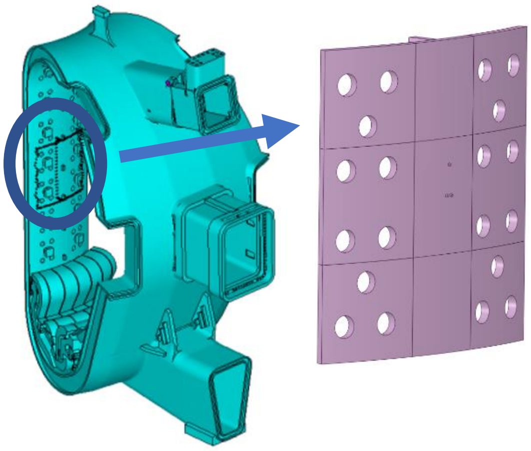


Figure 2 - Vessel geometry (a) before (b) and after applying the sub-modelling [3]

Figure 2 represents what was described. Figure 2 (b) is the geometry used in this study with the three bosses in the center. It is still a relatively big structure, where the element size of the mesh should not be set as too small (if not necessary), but shows already a large difference for the image in figure 2 (a), that is itself a section cut of the full model (for schematic purposes).

3. Extrapolation of the hot-spot stresses ranges

The hot-spot of a welded joint is the area around the weld toe and the weld root, where the stress value has a peak because of the discontinuities of the geometry and the notch effect, that comes from the weld shape (figure 3). It is considered a structural stress and differs from the notch stress (evaluated directly by the finite element method) for not including the non-linear stress peaks [4].

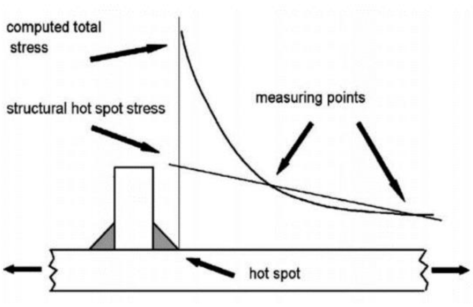


Figure 3 - Hot-spot stress [5]

The weld geometry type assessed in this study is represented in figure 3. It is a full penetration weld, so it occupies all the contact between the VV and the boss.

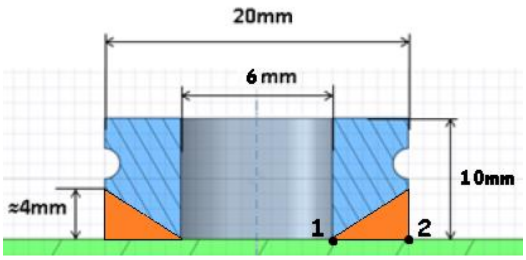


Figure 4 - Boss and weld dimensions [3]

The geometry of the weld material is represented in orange and the two critical points are represented with the numbers 1 and 2. The first point corresponds to the weld root and the hot-spot calculation requires a linearization of the sum of membrane and bending stress (equation 1) along the thickness of the base material (represented in green) and posterior extrapolation of the last value that will be the hot-spot stress range.

$$\sigma_{hs} = \sigma_{mem} + \sigma_{ben} \tag{1}$$

The stress range in the weld toe represented with the number 2 is calculated using a different method. The maximum and minimum principal stress are evaluated at the surface of the base material at certain distances and extrapolated directly to obtain the stress range at the hot-spot and there are four extrapolation method available. Depending if the assessment concerns the weld toe in contact with the boss or the weld toe in contact with the VV, there are two options: type “A” and type “B” (figure 5) extrapolation methods and each one has an expression to apply on a fine mesh and other for coarser meshes. All the four formulas are described in table 1 [6].

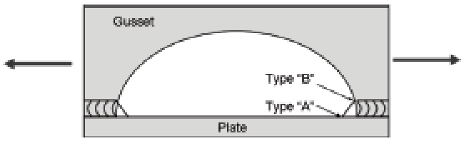


Figure 5 - Difference between weld toe types [7]

Table 1 - Surface extrapolation methods

Method	***Mesh type***	***Formula for σhs***
Type “A”	Coarse	$1.5\sigma_{0.5t} - 0.5\sigma_{1.5t}$
Type “A”	Fine	$1.67\sigma_{0.4t} - 0.67\sigma_{1.0t}$
Type “B”	Coarse	$1.5\sigma_{5mm} - 0.5\sigma_{15mm}$
Type “B”	Fine	$3\sigma_{4mm} - 3\sigma_{8mm} + \sigma_{12mm}$

Being the values underlined with the σ stress symbol, the distance at which the principal stresses are measured.

4. Case study

In addition to the first geometry provided with the exact location where the antenna needs to be positioned, two other geometries were created, each one representing a possible improved in the fatigue design, by changing the arrangement of the 3 bosses. The bosses can be moved if it does move the central point of the support, that marks the position of the antenna in the VV wall.

The material was chosen by ITER and is the austenitic stainless-steel type 316L (N)-IG.

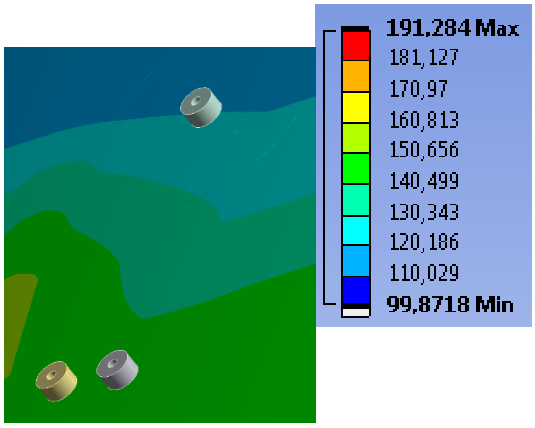
All the load cases provided consisted in an input file with displacement values to apply in the boundary surfaces of the sub-model, together in other load factor. Depending on the load case, the second load was a uniform temperature all over the geometry, a temperature with different values for each node, a pressure in the exterior of the VV wall, simulating a vacuum inside, a positive pressure inside the VV or an acceleration.

The main objective in this study is to combine the different analytical methods and the three geometries to find a set-up that meets the requirement of a safety coefficient equal to 20 for each load case individually and also obtain a safety coefficient under 10 with all the loads combined and applied cyclically.

5. Numerical analysis methodology

The files provided for input had all the necessary information but it was not sorted in a way that Ansys Workbench can import it. Matlab was used to get the input information organized and ready to apply in the model. All the data was originally divided in two complementary files and out of numerical order. With the help of this software, it was possible to gather everything in the desired format and file type required by Workbench.

The next step was to linearize the plasma temperatures provided and organized before. The local where the temperature is applied is defined by its coordinates and since the bosses' location is variable in this design phase, they do not have an associated temperature value, being at the reference 293.15 K, like is represented in figure 6.



(a) (b)

Figure 6 - Temperature (a) distribution, (b) scale [ºC], before linearization

This discrepancy creates very large peaks of stress, registering a maximum value of 523 MPa instead of 242 MPa at the weld toe if the temperatures are not linearized.

Most of the material properties varies with the body temperature, except for the Poisson's ratio for example, so all this information was entered as a setup in the analyses. Table 2 shows the variations with temperature for the first values.

Table 2 - Material properties

T [K]	ρ [kg/m³]	α [K⁻¹]	E [GPa]	K [W/m.K]	C_p [J/Kg.K]
293.15	7966	1.59E-05	200	13.9	470
323.15	7949	---	---	14.4	476
373.15	7932	1.64E-05	193	15.1	486
423.15	7910	---	---	15.8	497
473.15	7889	1.7E-05	185	16.5	508
523.15	7867	---	---	17.2	518
573.15	7846	1.75E-05	176	17.9	529

In the first analyses a relatively fine mesh is used to check which load is the most demanding for the components, because the stresses are evaluated all over the structure and it is more like a quantitative analysis that is only carried out once. When the extrapolation method are applied, a mesh convergence is performed exactly in the zone where the stresses will be evaluated. When a different load is applied to a previously converged model, it is not required to converge the results again, but when the geometry is changed, the process is performed again, resulting in 7 mesh convergences performed in total. Figure 7 shows an example of the local for which the convergence was made.

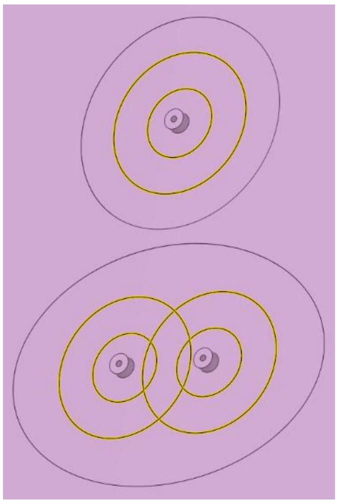


Figure 7 - Extrapolation circles for Type "A" method

Figure 8 is a graphic with the stress results converging to a point.

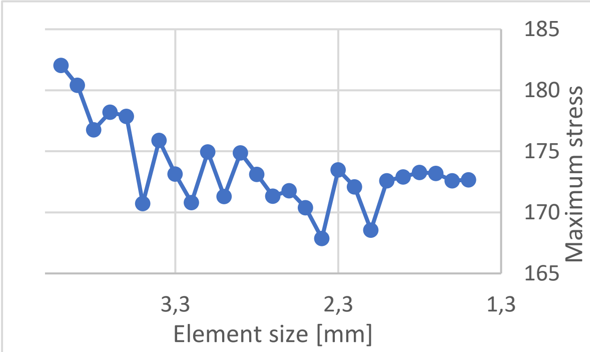


Figure 8 - Maximum von-Mises stress converging

Type "A" extrapolation method is not possible to apply in geometry 1 because there is not enough distance between the two bottom bosses to evaluate the principal stresses at the distance required. Type "B" and through the thickness extrapolations were carried out in the first geometry and σ_{hs} for both. The stress range value must be corrected for the difference in temperature and base material thickness, comparing to the material used in the standard's experimental test, so that the S-N curve can be used to obtain the number of allowed cycles, N_{max}. The mean temperature of the model is given by:

$$T^* = 0.75T_{max} + 0.25T_{min} \quad (2)$$

And the correction factor for temperature is:

$$f_{T^*} = 1.043 - 0.00043\,T^* \quad (3)$$

It is only needed to apply when the structure temperature is above 100 ºC.

The correction for the thickness is:

$$f_{ew} = \left(\frac{25}{e_n}\right)^{0.25} \quad (4)$$

Where e_n is the thickness of the VV.

The hot-spot stress value is corrected with the formula in equation (5).

$$\Delta\sigma_{corr} = \frac{\Delta\sigma_{hs}}{fT^{*}.f_{ew}} \qquad (5)$$

N_{max} is obtained with:

$$N = \frac{C}{\Delta\sigma_{corr}{}^{3}} \qquad (6)$$

C is defined as a property of the material and the fatigue class where it is inserted, depending on the load types, weld geometry, etc, and is represented in figure

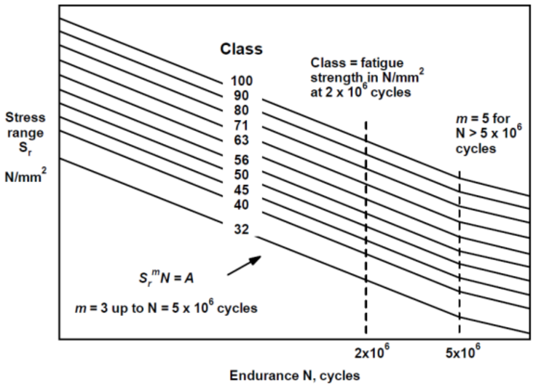


Figure 8 - S-N curve fatigue class

The last value to calculate is the usage factor, that is the inverse of the safety factor, n, and gives the same information but based on the values of N. It relates N_{max} evaluated before from the S-N curve with the N_E required in the project with equation (7):

$$UF = \frac{N_E}{N_{max}} \qquad (7)$$

The same procedures were repeated for geometry 2 and 3, performing new mesh convergences like explained before the usage factor was obtained for the plasma load in all these distinct situations.

The first geometrical change is only increasing the initial distance (L_0) between the top boss center and the 2 bottom bosses center by 40 mm (ΔL), just by moving the bottom bosses down. This distance is now 241 mm and figure 9 shows the new bosses' arrangement.

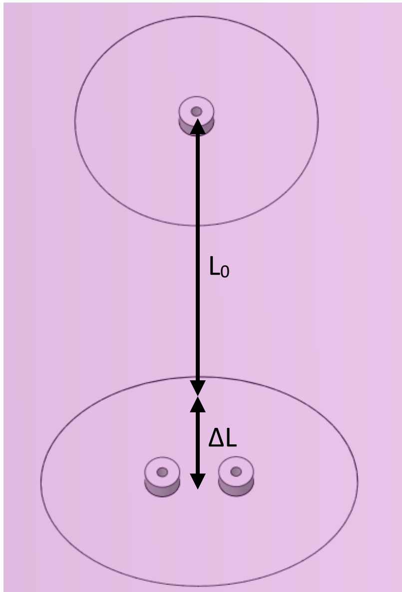


Figure 9 - Geometry 2 differences

The second geometrical change had the geometry 2 as base and consists in increasing the initial distance (L_0) between the 2 bottom bosses center by 47.5 mm (ΔL), just by moving the bottom right boss to the right, to check how the weld toe and root hot-spot stress get affected. This distance is now 90 mm and figure 10 shows the new bosses' arrangement.

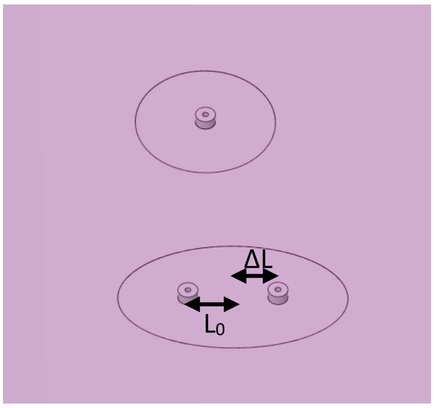


Figure 10 - Geometry 3 differences

The final task of the fatigue assessment was to repeat the process explained for all the other load cases, using the geometry that shows better results and applying the extrapolation method that shows the worst results in terms of fatigue. This step will be carried out with the geometry 3 and using the type "B" method that studies the failure in the weld toe in contact with the boss, as explained in figure 5.

6. Results and discussion

The results will be presented for all the tasks described in last section.

Table 3 - Worst load case scenario

Load Case	Maximum stress in whole geometry [MPa]	Maximum stress around the bosses [MPa]
Plasma formation	353.3	249.7
Normal operation pressure	50.3	17.4
Normal operation temperature	7.3	0.1
Baking pressure	116.3	40.2
Baking temperature	17.1	0.1
Plasma disruption	122.8	85.2
Seismic event	6.8	1.4

Table 3 resumes the values obtained statically for all the loads. Comparing the results, the plasma formation is by far the most demanding load to the structure in all the areas evaluated. It has to be taken in account that none of the values of table 5.1 is real (or precise) because they are obtained in local geometrical discontinuities. The highest value evaluated in the whole geometry always corresponds to a 90-degree shape (exactly or approximately) like, for example, the edge of the wholes of the VV far from the bosses. Those are boundary surfaces where the displacements are applied and are part of the sub-modelling technique surface, so they do not correspond to the real shape of the VV wall. The highest value evaluated around the bosses and in the bosses correspond to hot-spot. Its value is not accurate and is corrected in this study with the surface extrapolations. Therefore, the peak stress values obtained do not allow a quantitative comparison, but qualitatively they prove that the plasma formation is the most harmful load for the structure.

Some of the stress values obtained are above the yield stress for the material but nothing can be concluded regarding plastic deformations, because of the same reasons stated before.

Geometry 3 allowed to apply an extrapolation method that was not possible with the other two geometries because of the lack of space between the bosses. The results obtained cannot be compared with any other but the usage factor is only 68% of the limit imposed in the design rules, so it is acceptable. Maybe this extrapolation scheme would fail with the two other geometries, but the fact that a good margin was obtained with the most efficient geometry is enough for the design.

All the usage factor results at this point are listed below in table 4.

Table 4 - Usage factor values

	Type "B"	Through thickness	Type "A"
Geometry 1	0.077	0.0464	
Geometry 2	0.0607	0.0381	
Geometry 3	0.0468	0.0356	0.034

The third geometry studied has a usage factor below the limit of 0.05 in all the weld locations, so the structure can tolerate the plasma formation loads within the safety coefficient required. Since it is the most demanding load by far it means that it can support all the loads, individually. In the next sub-chapter, the usage factor is calculated for all the other loads and all the values are summed to check if the total is below the limit of 0.1. This will be assessed using a type "B" extrapolation for the weld toe in contact with the boss, since it is the weld detail with smaller safety margin or the most critical weld point.

Figure 11 shows a comparison between the variation of the peak stress evaluated directly by Ansys in the weld toe for the 3 geometries studied and the variation of the hot-spot stress extrapolated for the boss weld toe. This extrapolation method was chosen for the comparison because it has the biggest fluctuation among the geometries. The hot-spot is decreases almost by the same value in the geometrical optimizations. The peak stress evaluated numerically has a larger reduction in the first change but is almost constant in the second between the second and third. The reason behind this must be the definition of structural stress in chapter (2.3): this kind of evaluation considers all stress raising effects of a structural detail except those due to the local weld profile itself. By ignoring the local notch effect, the structural stress reflects better the other factors and changes.

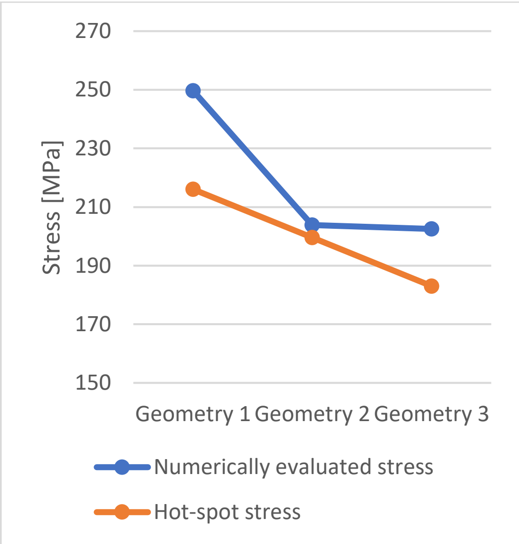


Figure 11 - Comparison between notch and hot-spot stress

Load Scenario	Usage Factor
Plasma formation	0.0468
Pressure envelope	0.0001
EM envelope	0.0032
Seismic envelope	0.0003
TOTAL	0.0504

These three usage factor values are added up with the plasma formation usage factor to obtain the project total usage factor of 0.0504. This result is around half of the limit imposed so the structural integrity of the welds are ensured for this load and number of cycles proposed. It is also verified that the other loads almost do not add damage to the structure, when compared with the plasma

7. Conclusions

In this section, some of the conclusions taken from this study are listed.

- The equivalent von-Mises stress only has positive values because the results are represented in modulus, but if the principal stresses are evaluated in any part of the geometry, the peak values are mostly negative. This makes sense because the structure has a high value of temperature in most of the load scenarios and tends to expand, resulting in compressive loads because it is constrained by itself;

- The geometry deformation is not a good tool to measure and quantify the improvements because they are mostly imposed by the displacements input files. The deformation values are not used normally in the mesh convergence because these values are easier to converge than the stress values. In this case it is even less advisable;

- A small load applied on any part of the geometry does not create a noticeable stress peak in the geometrical discontinuity of the weld geometry. The higher the loads on the model, the greater the stress variance between the weld and its surroundings. A plausible conclusion is that stress values result from the displacements, temperatures, pressure and acceleration applied in the model (and not from the weld notch effect) and are amplified by the geometrical discontinuities, leading to the peak values;

- As expected, the pre-existing stress in the VV wall before the bosses are added to the model have the greatest contribution to the peak stress in the weld. The stress range decreases when the critical bosses were moved to a less demanded zone of the VV, and this should be the procedure in future improvements of the geometry;

- The proximity between 2 bosses affects the stress range in its welds, as was noticed with geometry 3, since the stress values got lower for the same inputs. The stress peak from the notch effect of the surrounding bosses is one of the major contributions for the hot-spot stress;

- The plasma formation loads proved to be the most harmful by a considerable margin, which is supported by the previous conclusion since it is one of the loads with higher temperature values;

- The usage factor obtained with the third geometry and considering only the plasma formation is almost the same of the total usage factor that sums all the load envelopes. This is verified because in addition to the other load scenarios being less demanding, some of them are also applied less times to the structure, resulting in a greater margin to the maximum number of cycles allowed;

- When the fatigue assessment is carried out on the third geometry proposed, the welded joints respect the overall design requirements by a large margin in the weld root and in the weld toe in contact with the VV wall, but only a small margin is verified for the weld toe in contact with the boss lateral surface. It must be taken in consideration that the values imposed have a safety coefficient of n = 20 (the safety coefficient is the inverse of the usage factor) before the mechanical failure occurs, so the result is conservative;
- The methodology used is very specific for this case but it can be applied in other examples, by changing the correction factors, the extrapolation methods or the norm(s) followed.

References

[1] "Fatigue failure (LEFM): Part Two :: Total Materia Article." [Online]. Available: https://www.totalmateria.com/page.aspx?ID=CheckArticle&site=kts&LN=PT&NM=299. [Accessed: 03-Oct-2018].

[2] R. Budynas and K. Nisbett, *Shigley's Mechanical Engineering Design*. 2011.

[3] I. O. Dg and C. O. O. Ted, "Annex6 Fatigue assessment of OVC boss welded attachments to the VV Appendix justifying the protection against fatigue damage of OVC attachment welds to the," 2016.

[4] A. Lee, A. Martin, J. M. Martinez, V. Udintsev, and M. Walsh, "Fatigue assessment of standarized bosses to the VV INNER shell walls," 2017.

[5] A. F. Hobbacher, "The new IIW recommendations for fatigue assessment of welded joints and components - A comprehensive code recently updated," *Int. J. Fatigue*, vol. 31, no. 1, pp. 50–58, 2009.

[6] B. Giraud, J. M. Martinez, and M. Walsh, "Guideline for the fatigue assessment of Diagnostic attachments to VV welds," 2015.

[7] J.-M. Lee *et al.*, "Comparison of hot spot stress evaluation methods for welded structures," *Int. J. Nav. Archit. Ocean Eng.*, vol. 2, no. 4, pp. 200–210, 2010.