Stress analysis of pipeline as hydropower plants structural element

Stefan Ćulafić[1*], Taško Maneski[2], Darko Bajić[1]
[1] University of Montenegro, Faculty of Mechanical Engineering, Montenegro
[2] University of Belgrade, Faculty of Mechanical Engineering, Serbia

This paper describes pipeline stress analysis, dominantly branch junctions, as structural element in hydro-power plant. Pipeline is exposed to internal pressure which is present under working conditions. Analysis of stresses in the pipeline of the hydropower plant is based on analytical, numerical and experimental methods. In this paper we will define critical elements of the pipeline. After that, we will determine critical areas in the branch junction, under experimental conditions, where strain gauges should be installed. Results that are obtained show that a boiler formula can be efficiently applied in the stress analysis. Also, correlation between the internal pressure and the maximum circumferential stresses in the elastic zone is given. In the last sections of the paper, limit value of the internal pressure as load for which stress in the zone of plasticity appears and the safety factor of the branch junction in the exploitation conditions are determined. The contribution of this work is the unification and deepening of the topic related to the problem of the hydro-power structural elements testing.

Keywords: pipeline, branch junction, hydropower plant, strength analysis, shell intersection

Highlights*:*

- A boiler formula can be efficiently applied in the stress analysis.
- Correlation between the internal pressure and the maximum circumferential stresses in the elastic zone is given
- Limit value of the internal pressure as load for which stress in the zone of plasticity appears.
- Safety factor of the branch junction in the exploitation conditions are determined.

0 INTRODUCTION

When studying hydropower plants, special attention should be paid to the analysis of the strength of certain parts of the hydropower plant. With a high-quality analysis of the stress of individual parts of the plant, it is possible, with sufficient accuracy in real conditions, to anticipate the critical areas for remediation, evaluation and reduction of the maintenance costs, which should ultimately extend the lifetime of the power plant.

The interesting shell problem has not yet been investigated sufficiently. The practical importance of this problem requires further investigation, including that of the elastic stress analysis of the intersecting shells of the various shapes subjected to different loadings and the elastic plastic analysis [1].

Geometry of hydropower plants structural elements is very complex with large number of discontinuities, so problem of stress and strain analysis on these elements is also complex. Analytical analysis is possible only in small number of very simple cases. Numerical analysis based on finite elements method is used when analysing stresses of structural elements in pipelines, primarily pipeline branch junctions. It is a very real problem of defining locations, areas, in which yield stresses are observed, defining stress concentration factor, limit load, and burst pressure [2-6]. The use of experimental methods is very difficult under real, working conditions, because it is not possible to vary internal pressure in real conditions in some structural elements such as branch junction, until plastic strains are observed under working conditions. Because of this, it is more convenient to perform experiments on the model of the structural element under laboratory conditions and, on the basis of these results, make conclusions about what is happening in the real elements.

By combining numerical and experimental methods, it has been shown that the most accurate results are obtained when determining critical stresses which in some places, can lead to problems in the exploitation and functioning. Subject of this paper is the analysis of stress distribution in structural elements of a pipeline, moreover:

- applying analytical procedures,
- FEM application on the real element with real dimensions,
- FEM application on the element model,
- applying experimental analysis on the model under laboratory conditions.

Corr. Author's Address: University of Montenegro, Faculty of Mechanical Engineering , Blvd. G. Wash., 81000, Podgorica, Montenegro, sculafic@ucg.ac.me.

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p
Received for review: yyyy-mm-dd

Received revised form: yyyy-mm-dd
DOI code
Original / Review / Short Scientific Paper
Accepted for publication: yyyy-mm-dd

1 METHODS

Analytical, numerical and experimental procedures were applied in the analysis of the stress of structural elements. Specificity is reflected in the fact that the experiment was realized not on a real structural element but on its model. Numerical analysis was also carried out on a real structural element and on a structural element model.

2 ANALYTICAL AND NUMERICAL STRESS ANALYSIS OF PIPELINE

Main characteristic of pipes in the pipeline is that their radius is much larger than the thickness (*R*>>*t*), so it can be adopted that these pipes are actually shell pipeline.

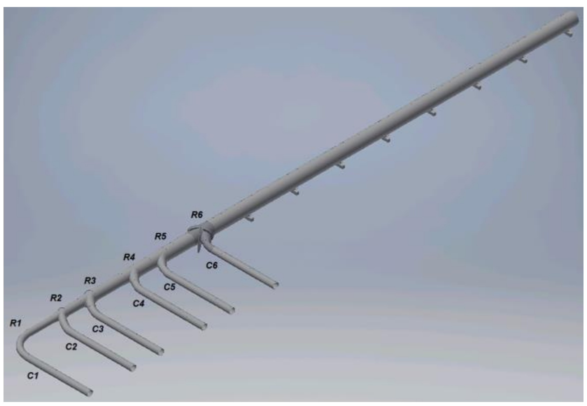


Fig. 1. *FEM pipeline model*

Static linear finite element analysis in the elasticity area has been performed for the pipeline, i.e. its straight main tube parts and the knee part using Autodesk INVENTOR 2016 software. Due to a very complex geometry that has been analyzed, FE analysis was used with caution, and also was confirmed with stand-alone software package KOMIPS. FEM mesh was generated using 3D iso-parametric solid elements. 3D model of the pipeline is shown on the Fig. 1. Pipeline length is approximately 200m, and it is exposed to 51 bars of internal pressure under exploitation conditions. It can be remarked that basic elements of the pipeline are branch junctions: R1, R2, R3, R4, R5, R6 (where branch junctions are actually knee pipes which direct the water) and pipes: C1, C2, C3, C4, C5, C6.

Analytical solution for the parts of the pipeline which is made of straight tube with no junctions or nozzles is given. Also, analytical solution is given for the pipeline knee (Fig. 2).

Analytical equations for the determination of stresses in the torus shell pipeline (Fig. 2) are known as follows:

$$\boldsymbol{\sigma^{O}} = \frac{\boldsymbol{p \cdot R(2a \pm R)}}{\boldsymbol{2t(a-R)}} \tag{1}$$

$$\boldsymbol{\sigma^{p}} = \frac{\boldsymbol{p \cdot R}}{\boldsymbol{2t}} \tag{2}$$

where:

p - internal pressure,
R - radius of a circle cross section,
a - radius of a torus,
σ^{O} - circumferential stress,
σ^{p} - longitudinal stress.

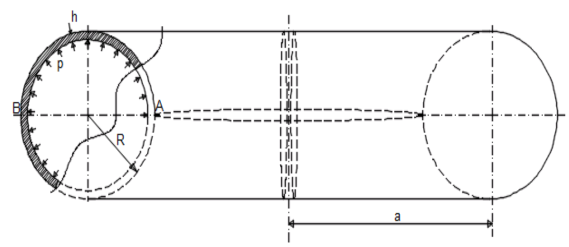


Fig. 2. *Torus shell part of the pipeline*

Especially, if $a = \infty$ it is a cylindrical shell, and if $a = 0$ it is a spherical shell. On the basis of Eq. (1), the stresses can be calculated on all parts of the pipeline, except for branch junctions. It is thus possible to calculate the stresses on the knee part R1 and the straight pipes of the pipeline (C1,…, C6).

So, for points A and B of the knee R1 pipe (*a*=4650 mm, *R*= 600 mm, *t*= 18 mm, *p*= 51 bar):

$\sigma_A^O = 182.6$ MPa,
$\sigma_B^O = 160.3$ MPa,
$\sigma^P = 85$ MPa,

and for the pipes C1 and C6:

C1: $\sigma^O = 170$ MPa,
$\sigma^P = 85$ MPa,
C6: $\sigma^O = 212.5$ MPa,
$\sigma^P = 106.25$ MPa,

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p

DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

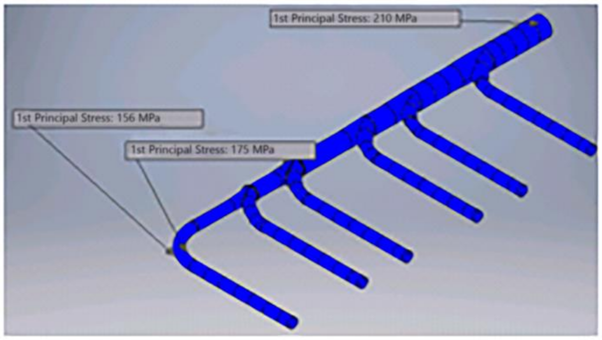


***Fig. 3.** FEM pipeline model*

Since there was one plane of symmetry, boundary conditions were the following: all translations normal to the plane of symmetry and all rotations in the plane of symmetry are constrained.

Material used in the FEM analysis was similar to the material properties of the pipeline under real working conditions. This is also important because of the correlation with the analytical results for which we have used Young's elasticity modulus for steel. Constant internal pressure of 51 bars was the implied load used in the finite element analysis.

Based on the numerical and analytical analysis and stress values given in Fig. 3, the conclusion is that pipeline branch junctions, especially branch junction number one, are most affected elements of whole pipeline, so just branch junction number one will be the subject of further analysis.

2.1. FEM analysis of the pipeline branch junction (real dimensions)

Pipeline branch junctions (cylinder-to-cylinder intersections) are very often used in industrial engineering. Reduction of the base material due to penetration of the intersecting cylinder is the cause for stress concentration.

As authors indicate in [6] cylinder-to-cylinder intersections are a very common occurrence in many industrial applications. Difficulties in obtaining analytical evaluations of the stress distributions in the disturbed regions near the intersection of comparable size shells originally stemmed from the complicated geometrical shape of the intersection line. The intersection curve of the middle surfaces of the cylinders is neither rotationally symmetric, nor on a plane curve, but rather is a spatial curve. Besides, the sharp discontinuities of curvatures across the intersection curve increase the stress.

Therefore, the presence of the stress concentration is inevitable and, as a consequence, constitutes a major consideration in the design.

In the papers [7-9] analysis of the pipeline branch junction in real dimensions is performed using FEM. Main pipe diameter on the branch junction entrance is 2.5m, while pipe diameter at the exit is 2.35m. FEM was performed using AUTODESK Inventor 2016 software, where we created the geometric model and performed the stress-strain analysis.

In Fig. 4, Fig. 5, Fig. 6 and Fig. 7 yield stresses are given when pipeline branch junction in real dimensions is exposed to 20 bar, 50 bar and 84 bar pressure.

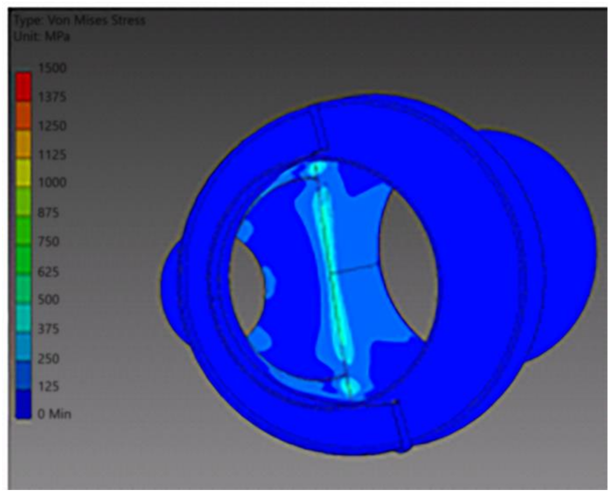


***Fig. 4.** FEM branch junction under 20 bar pressure*

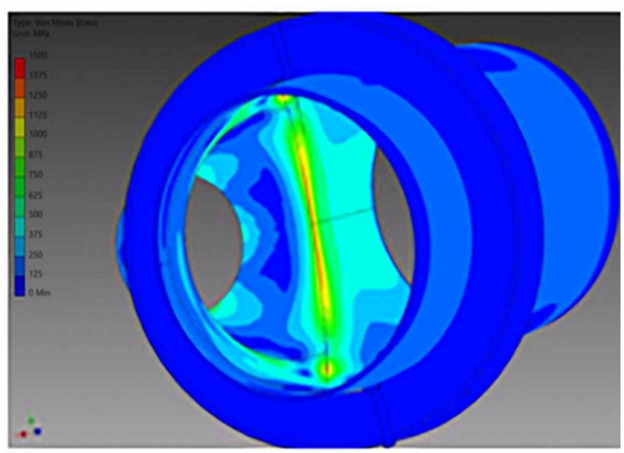


***Fig. 5.** FEM branch junction under 50 bar pressure*

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p

DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

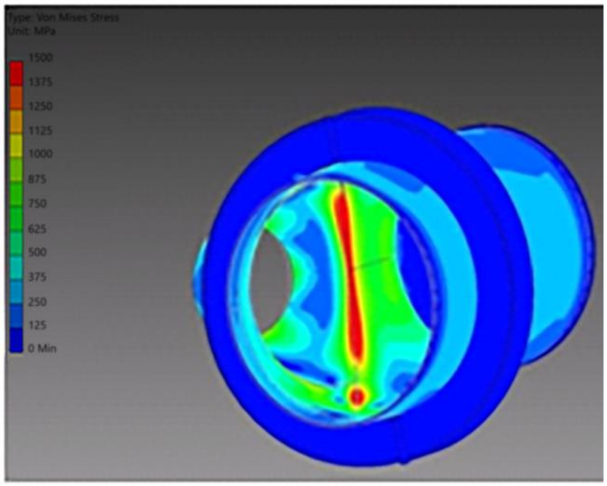


Fig. 6. *FEM branch junction under 84 bar pressure*

Table 1.*Stress values for MP1.*

Pressure values [bar]	Stress values for MP1 [MPa]
20	111
50	284
84	458

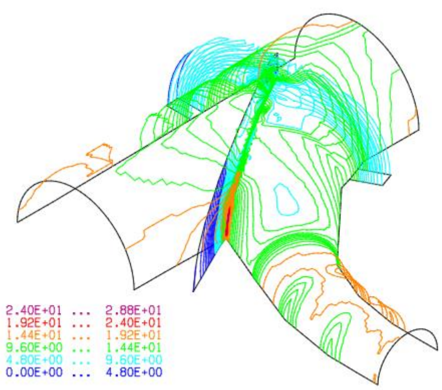


Fig. 7. *Equivalent stress under 51 bar pressure*

Results of the FEM analysis shows that yield stress appears in the area next to anchor of the pipeline branch junction. This particular area was specified (and referred to in Table 1 as MP1) as area of highest stress values. In Table 1 values of maximal circumferential stresses are given as a function of internal pressure values. Generally speaking, it can be said that the dependence of the value of stresses on pressure is very satisfactory linear in the field of elastic strains, which further means that the branch junction, which is essentially a shell, is not loaded on bending.

3 EXPERIMENTAL

3.1. Branch junction model manufacture

Material used for the construction of pipe elements of the real object is NIOVAL 47, manufacturer SIJ – Slovenian Steel Group (ex Železara Jesenice, Slovenia). The mechanical properties of this material are given in Table 2. In the absence of NIOVAL 47, which, due to exploitation problems, ceased to be produced during the 1970s, we analysed the steel of the same class with the most similar mechanical properties. That is steel S355J2 + AR. It was used to produce the branch junction model. The mechanical properties of this material are given in Table 3.

Table 2. *Mechanical properties of the material NIOVAL 47*

Tensile strength *Rm*, MPa	Yield strength *Re*, MPa	*Elongation A*, %
650	470	24

Table 3. *Mechanical properties of the material S355J2+AR*

Tensile strength *Rm*, MPa	Yield strength *Re*, MPa	*Elongation A*, %
554	360	28.2

Branch junction model (partitions Ø2500/Ø2350/Ø1200 mm) was made [10] of steel S355J2+AR, based on boiler formula. Branch model was created with following characteristics:

- model dimensions are five times smaller than the real object,
- thickness is 10 times smaller.

Calculus pertaining to stresses in the branch junction construction is analytically possible only in cylindrical parts of the junction, except for stiffeners, ribs and holes. This calculus is defined also with the standards for pressure vessels. Formula in which the stresses calculus is obtained is called boiler formula.

Boiler formula for stresses calculus on the cylinder (pipe, vessel) exposed to internal pressure without ribs and holes is Eq. (2) for longitudinal direction of a cylinder and for:

- circumferential direction of a cylinder:

$$\boldsymbol{\sigma^o} = \frac{\boldsymbol{p \cdot R}}{\boldsymbol{t}} \qquad (3)$$

where:

p [bar]-fluid internal pressure,

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p
DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

R [mm]-cylinder radius,
t [mm]- cylinder thickness,

From Eq. (2) and Eq. (3) observation can be made that values of the stresses in circumferential direction are twice the value along the longitudinal direction.

Branch junction model should give the same stress like the real object. This is secured by the application of similarity method in the following manner:

$$\boldsymbol{\sigma^o = \frac{p \cdot R}{t}\left(= \frac{\frac{p}{2}\frac{R}{5}}{\frac{t}{10}}\right) = \frac{p_{model} \cdot R_{model}}{t_{model}}} \qquad (4)$$

Derivation of the same stress value is as follows:

$$\boldsymbol{p_{model} = \frac{p}{2},\ R_{model} = \frac{R}{5}, t_{model} = \frac{t}{10}.}$$

Real branch junction: p=50 bar, R=1250 mm, t=36 mm, σ^o=174 MPa, (R=34.77t) .

Branch junction model: p=25 bar, R=250 mm, t=4 mm, σ^o=156 MPa.

This means that the branch junction model for the same value of the pressure has twice the value of circumferential stresses. This means the branch junction model has to be subjected to two times lower pressure so it could be correlated to the real branch.

In terms of manufacturing we have adopted the previous relations. Since there was no 3.6 mm sheet, 4mm sheet was adopted. In this way around 10% smaller values of stresses are obtained. Anchor stresses are same, since thicknesses are 8 mm for the model and 80 mm for the real branch.

Similarity method is applied on this branch junction model, because in its construction there is negligible presence of bending stresses, as we concluded in the real branch analysis, which depends on a square (t^2) of the thickness of the material.

3.2. FEM analysis of branch junction model

Three-dimensional model of the branch junction of basic dimensions and thicknesses is shown on Fig.8. It was made for the requirements of the strength analysis using FEM. Branch geometry was modelled using surfaces.

In numerical modelling, the branch is subjected to 10 bars of internal pressure on the walls. Due to linear static nature of the analysis of the construction (obtained stress has linear character compared to the given pressure) it is possible to use scaling to obtain results for different values of internal pressure. Having in mind the symmetry of the branch in longitudinal direction, one half of the branch was analysed.

Mesh was more detailed in the areas of calotte penetration, where we expected to get higher values of stress. In order to control and confirm experimental analysis using strain gauges, FEM model was made as well as necessary calculus.

Fig. 10 shows the positions of strain gauges on the branch. These exact locations will be used for the results comparison between experimental tests and numerical results.

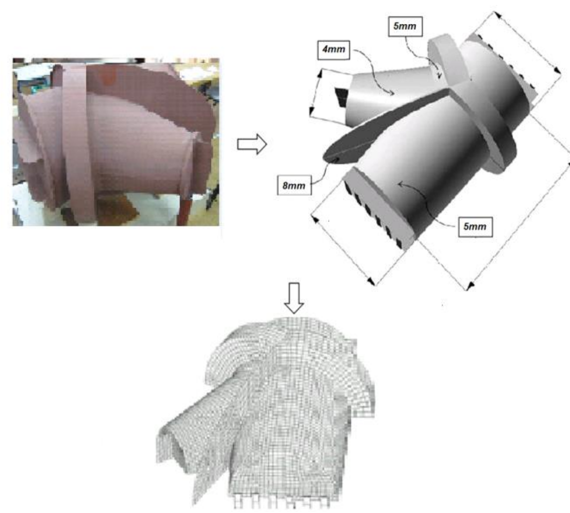
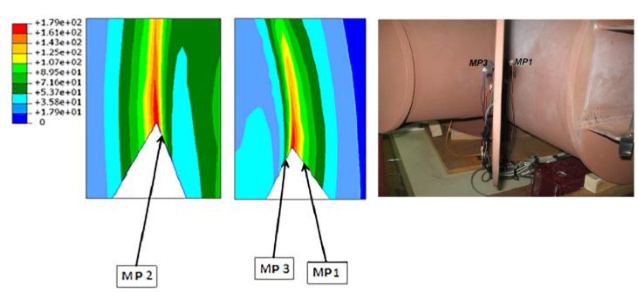


Fig. 8. *Branch junction model*

Boundary conditions are given as two constraints: translation and rotation, and since one plane of symmetry exists, boundary conditions that were used were that all the nodes on the symmetric section were constrained against deformation in the perpendicular direction.

Fig. 9 shows the results of the FEM analysis as Von Mises yield stress and also as stresses in circumferential (vertical) direction. All results are related to the pressure of 10 bars. Also, Table 4 shows exact values of circumferential stresses readings within the FEM model. These values are referred to as measuring positions MP1, MP2 and MP3.



Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p
DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

Fig. 9. *Measuring positions of strain gauges*

Table 4. *Values of stresses at measuring positions*

	MP1	MP2	MP3
Stress in circumferential direction [MPa]	115	89	58
Von Mises stress [MPa]	107	84	63

The following conclusions can be made based on the FEM analysis of the real branch junction and branch junction model:

1. Boiler formula gives, with very satisfactory accuracy, the correlation between the results on the real branch and branch model. This lies in the fact that the circumferential stresses on the MP1 are about the same, and amount to 111 MPa on the real branch junction and 115 MPa on the branch junction model. As the branch junction model is made on the basis of the boiler formula, we come to the previously stated conclusion. Furthermore, laboratory investigations have shown, based on the results obtained, what will happen with the branch junction under working conditions.

2. Yield stresses are visible in the area of the junctions of the main pipe (Ø2500 mm) and anchor and branch pipe (Ø1200 mm) and anchor, though it should be expected that measurements will show that critical gauge position is position one.

3.3. Experimental analysis of branch junction model

Based on the results obtained using the FEM it is possible to define areas at which strain gauges should be placed.

Experimental testing of the branch junction model should give answers to several questions:

- Are results of the stresses obtained using the FEM concurrent with the results obtained based on the experiment?

- What is the value of the internal pressure inside the branch junction model, in which first plastic strains appears?

Knowing this value and based on the similarity theory, it is possible to determine the internal pressure in the real branch junction precisely before plastic strain occurs.

Measuring places are designated as follows: MP1, MP2, MP3,..., MP8. Due to the complexity of the material, and observed values of stresses in some measuring spots, focus of our attention will be on the measuring positions MP1, MP2, MP3. (Fig. 10)

- MP1 - circumferential direction of bigger cylinder,
- MP2 - vertical direction of anchor,
- MP3 - circumferential direction of the conical part of the model.

Fig. 10. *Measuring positions: MP1, MP2, MP3*

Numerical analysis of the branch junction in real conditions and the branch junction model have shown that biggest stresses are in MP1 (Table 3 and Table 4). This should also be measured throughout the experiment.

Twenty-three experiments were conducted for each measuring position.

Values of the stress for any value of the pressure are obtained when scaling these values by a given factor.

In the following Fig. 11 and Fig. 12 graphical representation of measured stresses in all measuring positions and experiments conducted are given as the function of pressure and time.

The following colours are used in order to facilitate graphical representation (Fig. 11 and Fig. 12):

-black line -pressure,
- red line - MP1,
- blue line - MP2,
- purple line - MP3,
- light green line - MP5,
- light brawn line - MP6,
- dark green line - MP7,
- dark brawn line - MP8,

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p

DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

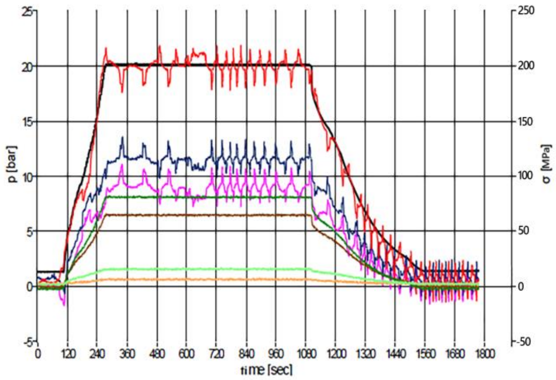


Fig. 11. *Internal pressure and stress as the function of time (elastic strain)*

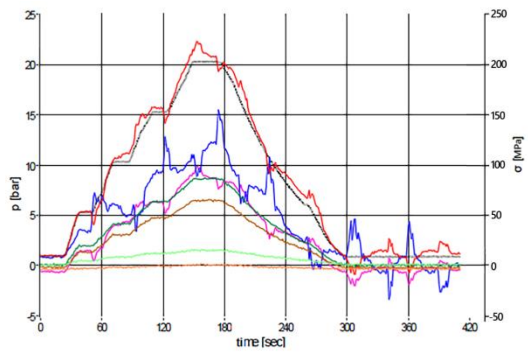


Fig. 12. *Internal pressure and stress as the function of time (elastic strain)*

It can be seen from Fig. 11 and Fig. 12 how the stresses change at all measuring positions with the change of internal pressure. At the pressure of 20 bar, the value of the stress on MP1 is slightly higher than 200 MPa. It is also noted that the MP1 stress is dominant in relation to the stresses at other measuring positions. After that, the highest stress value is on MP2. Also, it should be noted that after unloading, the internal pressure returns to the initial, i.e. zero, and that this release is accompanied by the stresses that also return to zero. This further means that, as far as the stress condition is concerned, the process is related to elasticity, i.e. stresses do not reach the value that belongs to plastic strain beyond the yield point.

.

Table 5. *Mean values of stress on measuring positions (MP)*

Stress - MP	Pressure value [MPa] - 10 bar
MP1	110
MP2	75
MP3	50

Table 5 shows mean values based on all experiments at MP1, MP2, MP3, when pressure is reduced by 10 bar.

This scaling only applies until yield point and enables the acquisition of any value of pressure stresses.

4 RESULTS

4.1. Comparison of the measurements and the FEM calculation

Table 6 gives values of stress intensities for the positions MP1, MP2, MP3 at the branch junction model obtained when the FEM calculation is applied and when measuring values (for 10 bar) were obtained [11]. General conclusion is that these values are very close, which puts these two methods in equal position and simultaneously confirms each other.

Table 6. *Mean values of stresses at MP1, MP2, MP3*

	Stress [MPa]	
	FEM	Measuring
MP1	115	110
MP2	89	75
MP3	58	50

4.2. Determination of limit internal pressure

What happens when the pressure is increased to 30 bar? The stress on MP1 reaches the value of about 370 MPa (Fig. 13). It is also important to note that after unloading, i.e. bringing the internal pressure to zero, the stress value on MP1 does not return to zero, but has the value of 50 MPa. This means that there is a permanent deformation corresponding to this stress. When, after the permanent deformation, the branch junction model is again submitted to the pressure of 30 bar, the stresses at MP1, after unloading return to the new starting level of 50 MPa.

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p

DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

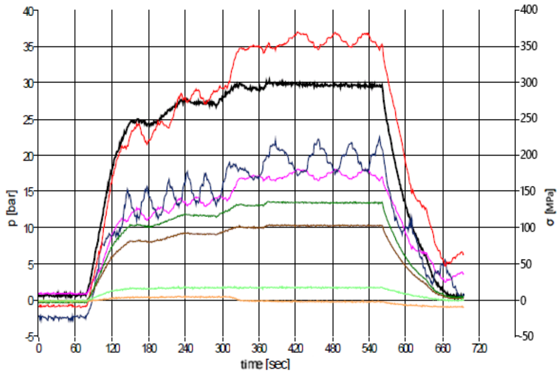


Fig. 13. *Internal pressure and stress as the function of time (plastic strain)*

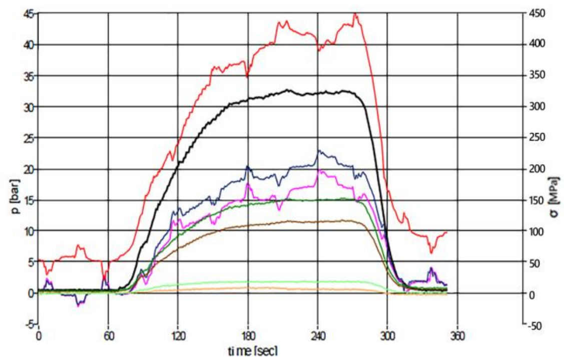


Fig. 14. *Internal pressure and stress as the function of time (plastic strain)*

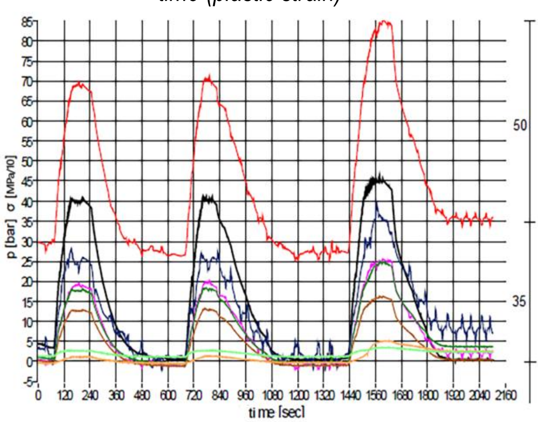


Fig. 15. *Internal pressure and stress as the function of time (plastic strain)*

From Fig. 14 it can be seen that when the pressure is increased to 32-33 bars, additional permanent deformation of the branch model at MP1 is made, for additional 50 MPa. When the branch model is unloaded, the stress at MP1 is 100 MPa. Fig. 15 shows the behaviour of the stress on the MP1 model of the branch when the internal pressure reaches the value of 40-45 bar. It can be seen that the stress value at MP1 reaches up to 850 MPa.

The pressure of 45 bars on the branch junction model corresponds to 90 bar on the real branch junction. From the diagram shown in Fig. 15, it can be seen that the stress value is about 850 MPa. If we deduct from this value the value of the stress due to the plastic strain of about 350 MPa, we obtain the value of about 500 MPa, which corresponds to the internal pressure of 90 bars on the real branch or 45 bars on the branch model. This further means that the result given in Table 1 is logical (pressure of 42 bars on the branch model or 84 bars on the real branch) and corresponds to the maximum stress value of 458 MPa at MP1. The explanation lies in the fact that the results given in Table 1 are related to the assumption that all analyses based on the finite element method are related to the field of elasticity.

4.3. Determination of safety factor

In the paper [12] a procedure for determination of the stress concentration factor was given. As the continuation of this paper, we will determine the safety factor.

Initial plastic strains of the branch junction model appear on MP1 under the pressure of 30 bars.

Calculation of necessary pressure for the occurrence of the initial plastic strains on the real branch junction, on the measuring position MP1 (position of maximum stress) is as follows:

$$p=30*0.9*(47/36)*2=70.5 \text{ bars.} \qquad (5)$$

Factor 0.9 represents the relation of thickness of real branch junction and branch junction model on the position MP1 - 36/(4*10).

Relation 470/360 represents the relation of yield point of the material of the branch (NIOVAL 47) and of the branch model (St355J/AR).

Factor 2 is model factor, which refers to the pressure.

Safety factor in the branch junction exploitation in relation to the plastic strains is: 70.5/51=1.38.

From Fig. 15 (blue line) it can be seen that the first plastic strains on MP2 appear when the pressure reaches 45 bars corresponding to the stress of 70-80 MPa. If the pressure of 45 bars is reduced by 5 bars, we conclude that at the pressure is about 40 bar, the first plastic strains appear.

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p
Received for review: yyyy-mm-dd

Received revised form: yyyy-mm-dd
DOI code
Original / Review / Short Scientific Paper
Accepted for publication: yyyy-mm-dd

Calculation of necessary pressure for the appearance of initial plastic strain on the real object of pipeline branch junction on MP2 is:

$$p=40*(47/36)*2=104.44 \text{ bars.} \qquad (6)$$

Stress values on other measuring positions are even lower.

5 CONCLUSION

This paper tries to use analytical, numerical and experimental methods to describe problem of the stress analysis in the pipeline. It is shown that the critical structural element of the pipeline is the pipeline branch junction. The following analyses have been carried out:

1. Analytical and numerical analysis for the pipes of the pipeline and the knee section of the pipeline,
2. Numerical analysis of the branch junction in real dimensions,
3. Numerical analysis of the branch junction model,
4. Experimental analysis of the branch junction model.

Based on Eq. (2) and Eq. (3), we defined the measuring places where the strain gauges should be placed. Also, it is shown that the boiler formula is correct for loads in which plastic strain zone is not reached. Our paper also shows the linear correlation between internal pressure and maximum circumferential stress, which means that bending stresses can be neglected, i.e. the branch junction can be treated and observed as a membrane shell. In case of appearance of the trapped air pockets, pressure will become unsteady [13], [14]. This paper analysis only steady pressure.

Based on the experimental analysis of the branch junction model it can be concluded how will real-dimension branch junction behave under internal pressure, under real working conditions. It was shown that the results pertaining to obtaining stresses with numerical method on the real branch junction, results pertaining to obtaining stresses with numerical method on the branch model junction and results pertaining to the experimental determination of stress values are such that satisfactory accuracy has been reached. Also, based on the experimental analysis, limit pressure value was defined based on the point when the first plastic strain is noted, which enabled us to define the maximum value of pressure under working conditions on the real branch junction necessary to achieve plastic strain. This enabled us to define safety factor in the branch junction exploitation and for the whole pipeline as well.

6 REFERENCES

[1] Skopinsky, V.N.(1993).Numerical stress analysis of intersecting cylindrical shells. *Journal of pressure vessel technology*, vol 115, no. 3, p. 275-282, DOI: 10.1115/1.2929528.

[2] Barry R.M., Venter G.(2019).Analysis of reinforcement designs for specials in steel pipelines. *International journal of pressure vessels and piping*, vol. 169, no. 1, p. 204-214, DOI: 10.1016/j.ijpvp.2018.12.005.

[3] Kim, S.H., Gim J.M., Wang, M., Kim Y.J.(2016), Simplified limit load estimation using ma-tangent method for branch pipe junctions under internal pressure and in-plane bending. *Procedia Structural Integrity*, vol. 2, p. 2583-2590, DOI: 10.1016/j.prostr.2016.06.323

[4] Plancq, D., Berton, M.N.(1998).Limit analysis based on elastic compensation method of branch pipe tee connection under internal pressure and out-of-plane moment loading. *International journal of pressure vessels and piping*, vol. 75, no. 11, p. 819-825, DOI: 10.1016/S0308-0161(98)00085-4.

[5] Kim, Y.J, Lee, K.H., Park, C.Y(2008).Limit load for piping branch junctions under internal pressure and in-plane bending- extended solutions. *International journal of pressure vessels and piping*, vol. 85, no. 6, p. 360-367, DOI: 10.1016/j.ijpvp.2007.11.007

[6] Sang, Z.F, Xue, L.P., Lin, Y.J., Widera, G.E.O.(2002). Limit and burst pressures for a cylindrical shell intersection with intermediate diameter ratio. *International journal of pressure vessels and piping*, vol. 79, no. 5, p. 341-349, DOI: 10.1016/S0308-0161(02)00028-5.

[7] Sedmak, A.(2011). *Program for an on-line monitoring of metal state in pipeline no. 3 at HPP Perucica.* Mechanical engineering faculty, Podgorica, Montenegro.

[8] Culafic, S. (2015).Numerical and experimental analysis of stress state in pipeline branch in HP Perućica. Master thesis.

[9] Ćulafić, S., Bajić, D., Maneski, T.(2015). Numerical model of pipeline branch in HP Perucica. Proceedings of the

Strojniški vestnik - Journal of Mechanical Engineering vol(yyyy)no, p-p

DOI code
Original / Review / Short Scientific Paper
Received for review: yyyy-mm-dd
Received revised form: yyyy-mm-dd
Accepted for publication: yyyy-mm-dd

2nd International conference "New Technologies NT-2015", p. 27-32 .

[10] Bajic, D., Maneski, T.(2011). *Investigation of branch junction A6 pipeline no. 3. at HPP Perucica*. Mechanical engineering faculty, Podgorica, Montenegro

[11] Bajić, D., Ćulafić, S.(2017).*Comparasion of numerical and experimental results of stress-deformation state in a pipeline branch*. Scientific Proceedings XIV International Congress "Machines. Technologies. Materials.", p. 29-32.

[12] Bajić, D., Momčilović, N., Maneski, T., Balać, M., Kozak, D., Ćulafić, S.(2017). Numerical and experimental determination of stress concentration factor for a pipe branch model. *Tehnički vjesnik*, vol. 24, no. 3, p. 687-692, DOI: 10.17559/TV-20151126222916.

[13] Bergant, A., Tijsseling, A., Kim, Y.I., Karadžić, U., Zhou, L., Lambert, M., Simpson, A.(2018). Unsteady Pressures Influenced by Trapped Air Pockets in Water-Filled Pipelines. *Strojniški vestnik - Journal of Mechanical Engineering,* vol. 64, no. 9, p. 501-512, DOI: 10.5545/sv-jme.2018.5238.

[14] Bergant, A., Karadžić, U., Tijsseling, A.S.(2016). Dynamic water behavior due to one trapped air pocket in a laboratory pipeline apparatus. *IOP Conference Series. Earth and Environmental Science*, vol. 49, p. 967-976, DOI:10.1088/1755-1315/49/5/052007.