

## REDUCTION OF DERIVATIVE PIPELINE EFFORT IN WATER TURBINE

LESZEK FLIS, STANISŁAW DOBROCIŃSKI

Polish Naval Academy, Faculty of Mechanical Engineering,  
ul. Śmidowicza 69, 81-103 Gdynia, Poland. E-mail: l.flis@amw.gdynia.pl

WOJCIECH KIELCZYŃSKI

Gdańsk University of Technology, Faculty of Mechanical Engineering,  
ul. G. Narutowicza 11/12, 80-233 Gdańsk, Poland. E-mail: wkilecz@pg.gda.pl

**Abstract:** The article presents an approach to evaluate and reduce the effort of the derivative pipeline in a water turbine of a hydroelectric power station in its vulnerable spots. Potential places of the biggest effort of the structure, as well as the coefficient of displacement and safety factor related to them were identified using FEM numerical method. Pipeline reinforcement was designed and made based on FEM analysis and non-destructive tests.

### 1. INTRODUCTION

Hydroelectric power station in Łapino on the river Radunia was put into service in 1927<sup>1</sup>. It is equipped with two identical water turbine generators no. 1 and no. 2. Each of them consists of two horizontal Francis turbines with a joint shaft (Manufactured by: Schichau Elbląg, 1925) which propel a synchronous generator directly coupled with them (Manufactured by: AEG, 1925). Basic operation and geometric parameters of the hydro turbine generator:

- nominal flow            11 m<sup>3</sup>/s;
- nominal head            13.8 m;
- nominal power          1147 kW;
- rotational speed        250 rev/min;
- rotor diameter          1.3 m.

The aim of the research featured in this article was to establish possible threats to work safety in the power plant in Łapino, which has been operating for over eighty years, and to protect it against failure with regard to material strength by eliminating the weakest links.

For this reason it was vital to evaluate structural components such as its branches, which – due to the lack of reinforcement and progressive corrosion processes – are highly susceptible to fractures caused by pressure generated by the water hammer effect.

The best example is the pipe break accident in 1997 [1]. The joint between the turbine connection pipe and the main penstock line was the point of rupture.

---

<sup>1</sup> <http://energa-hydro.pl/2,26,0,Galeria-Lapino.html>

Excessive water hammer effect caused by too rapid a shut-off of water flow during load discharge test of the turbo generator no. 2 [1] was deemed to be the direct cause of the incident. Low strength of the pipeline structure, caused mainly by low quality of welded joints (old welding technology) and the lack of reinforcement in the areas of high concentration of stresses, facilitated the break.

After the pipeline had been repaired it was critical to indicate the components with the highest effort and propose possible reinforcement of the pipeline wall in order to enhance its strength. With this aim a computer simulation of the pipeline load was conducted and on the basis of the simulation the structure effort was assessed. The next step was the design and construction of the pipeline reinforcement.

## 2. NUMERICAL SIMULATION OF PIPELINE EFFORT

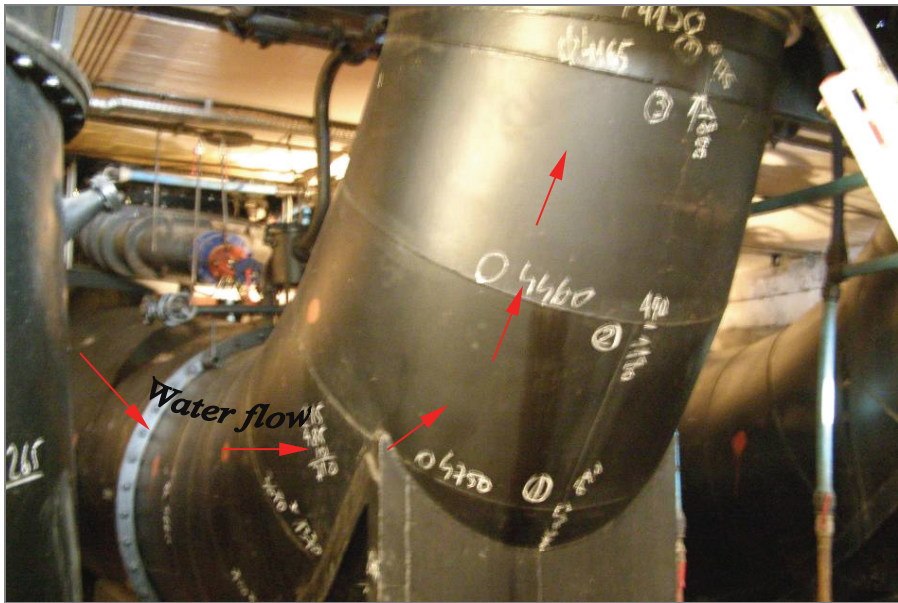
A general view of the water turbine generators of the hydroelectric station discussed is presented in Fig. 1. Figure 2 shows the view of water inlet pipelines between the turbo generators. A draft with selected dimensions is shown in Fig. 3.

The samples of pipeline material obtained from tensile test had the following yield strength ( $Re$ ) and ultimate strength ( $Rm$ ) values (mean values for three tests) [1]:

- tensile test in the rolling direction  $Re = 280$  MPa,  $Rm = 401$  MPa;
- tensile test transverse to the rolling direction  $Re = 294$  MPa,  $Rm = 431$  MPa.



Fig. 1. View of water turbine generators in Łapino.  
Source: <http://energia-hydro.pl/2,26,0,Galeria-Lapino.html>



Rys. 2. View of water inlet pipelines.

Source: prepared by the authors

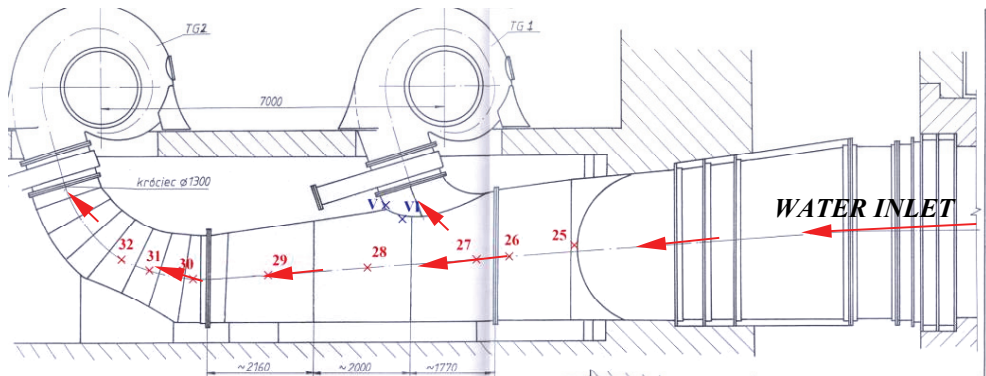


Fig. 3. Longitudinal section through the inlet pipe.

Source: Adamkowski A. et al., *Pomiar naprężenia najbardziej obciążonych elementów rurociągu derywacyjnego w ew. Łąpino w różnych warunkach obciążeń ciśnieniem wewnętrznym*, IMP, Gdańsk 2009

The analysis of stress in the pipeline was conducted in accordance with finite element method (FEM). Pipeline junction shape and geometry dimensions as well as tested strength properties of material were used in calculations. The calculations were made for a range of values of internal pressure. The components of calculated stress field were substituted with reduced stress following Huber's hypothesis.



The calculations did not require conducting an advanced nonlinear analysis, because their aim was to assess the values of dangerous pressure and the areas where the pressure induces stresses of 55% value of yield strength. As stipulated in specification for this type of structures the values should not be exceeded [1], [2].

### 3. DEVELOPMENT OF CALCULATION MODEL

One of the fundamentals for credibility of calculations is the appropriate representation of the actual structure in the form of digital representation, which directly translates into the quality of finite element mesh. The computer representation of a structure can be neither too simple, nor can it include all details such as pads, caps, etc. The geometric model should be a conscious simplification of the real object preserving its main design features. The above mentioned task entails experience and it is the most time consuming stage of the development of a computer simulation.

A CAD computer model of the object examined was created based on available data (Fig. 4). The model was used for a numerical analysis to assess the structure's level of effort.

The thickness of the pipeline wall is 8 mm. The geometric model and calculations were made for the shells.

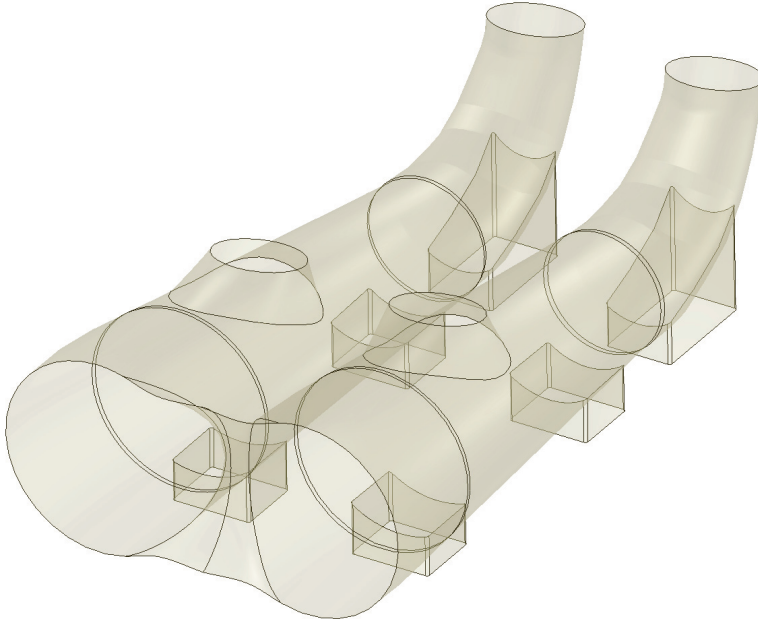


Fig. 4. Surface model of the pipeline.  
Source: prepared by the authors

#### 4. LOAD OF THE MODEL

On the basis of the operating data the nominal hydrostatic load was set at 0.12 MPa, equivalent to water gauge of 10.5 m which occurs as a difference of water levels between the influent water and the bottom of the pipeline. For dynamic load states, as recommended, an overload factor of 1.4 was taken (start) – then the value of pressure loading the structure is 0.16 MPa. The value of 0.16 MPa matches the pressure that arises in reality when the turbo generator is launched [1]. The stresses have to exceed the ultimate strength of the material in failure state. A 0.21 MPa pressure meets this requirement.

Hydrostatic pressure distribution shown in Fig. 5 remains the same. Appropriate colour ranges correspond to calculated variants, which are provided with comments (nominal load, overload, failure state).

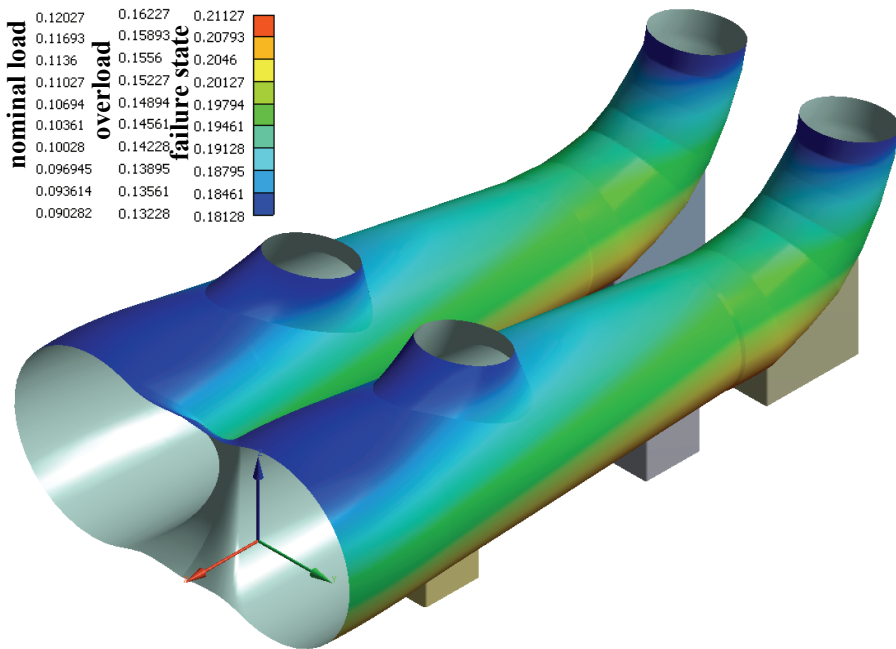


Fig. 5. Distribution of hydrostatic pressure in pipeline.

Source: prepared by the authors

#### 5. FINITE ELEMENT MESH AND ESTABLISHED BOUNDARY CONDITIONS

Finite element mesh is made out of shell finite elements with six degrees of freedom for each of the eight nodes per element (Fig. 6). The number of nodes in this range was 221231, with 73388 elements.

Red (Fig. 7) is used to mark the edges (nodes) whose degrees of freedom were eliminated. This way the places where the structure cannot displace were simulated. In general, these are the places where the structure is fixed to foundations, intake walls or where inlet flanges are fixed to the base of the turbine.

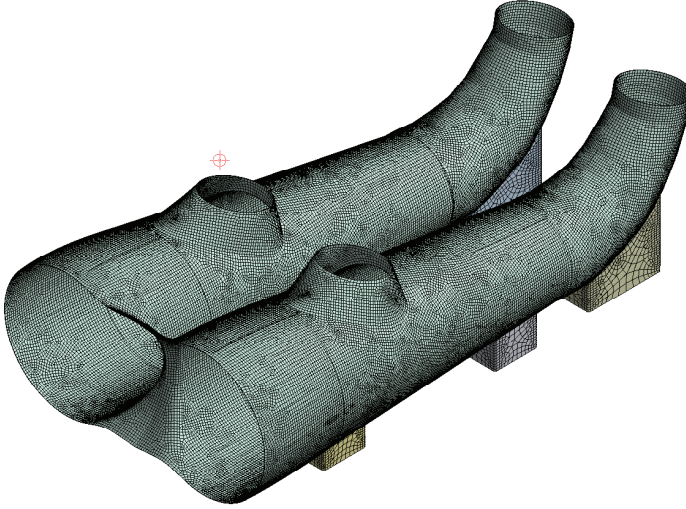


Fig. 6. Selected finite element mesh.  
Source: prepared by the authors

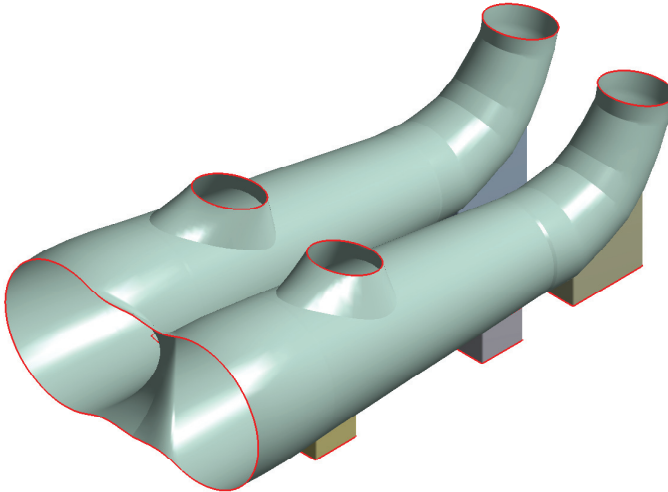


Fig. 7. Support of the model studied.  
Source: prepared by the authors

## 6. RESULTS OF CALCULATIONS

Figures 8 to 10 present the results of calculations as mapping of functions. Stresses reduced in accordance with Huber's hypothesis are shown in Fig. 8. The colour ranges represent, as before, nominal load, overload and failure state load, respectively. The value of reduced stress at its concentration points amounts to 80–150 MPa with nominal load. The maximum value of stress of 208 MPa (red colour in the range) may be ignored because this is the value for the point on the edge and its degrees of freedom were eliminated, thus it is the result of an approximation error of a finite element shape function. Stresses in notches under pressure of 0.16 MPa assume the value of 220 MPa and are lower than yield strength.

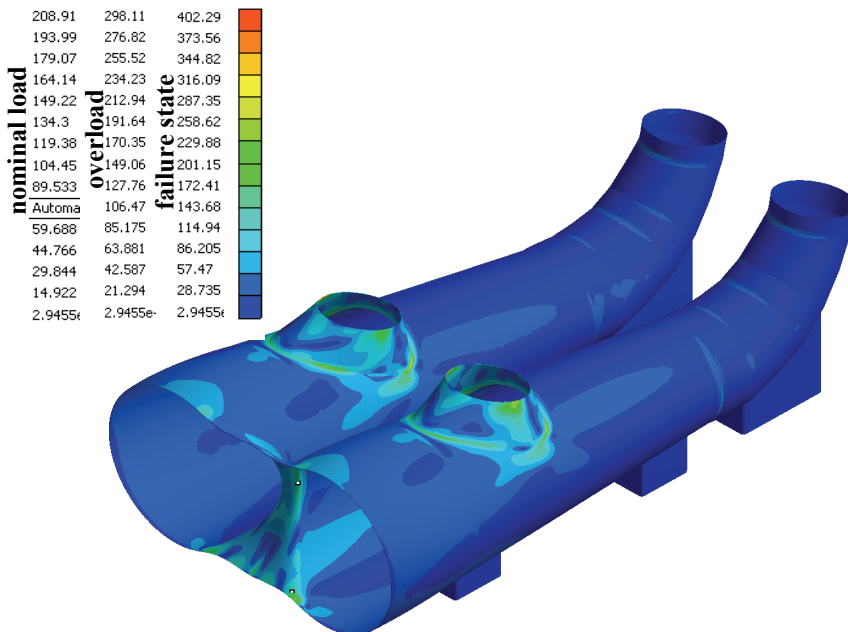


Fig. 8. Huber's reduced stress – colour range for 0.12, 0.16 and 0.21 MPa, respectively.

Source: prepared by the authors

Figure 9 illustrates resultant displacement. In nominal load state displacement value was equal to 3.7 mm; however, in overload state it was equal to 5.4 mm. The value of displacement in failure state was 7.4 mm. The last one does not make physical sense since the material exceeded its ultimate strength.

In the case of linear elastic analysis the distribution of the function of stress and displacement does not change as well, because only the load level is different for all the cases examined and not the pattern of the load. This is the advantage of the linear

elastic analysis, but only the results below the linear limit of elasticity of the material make sense and this is its disadvantage. In that case, the areas where the material is plasticised are impossible to determine unequivocally based on calculations. It is only possible to state that the yield strength or the ultimate strength of the material have been exceeded. Calculations and analysis of safety factor as the ratio of reference stress (usually yield strength) to calculated stress are useful in that case.

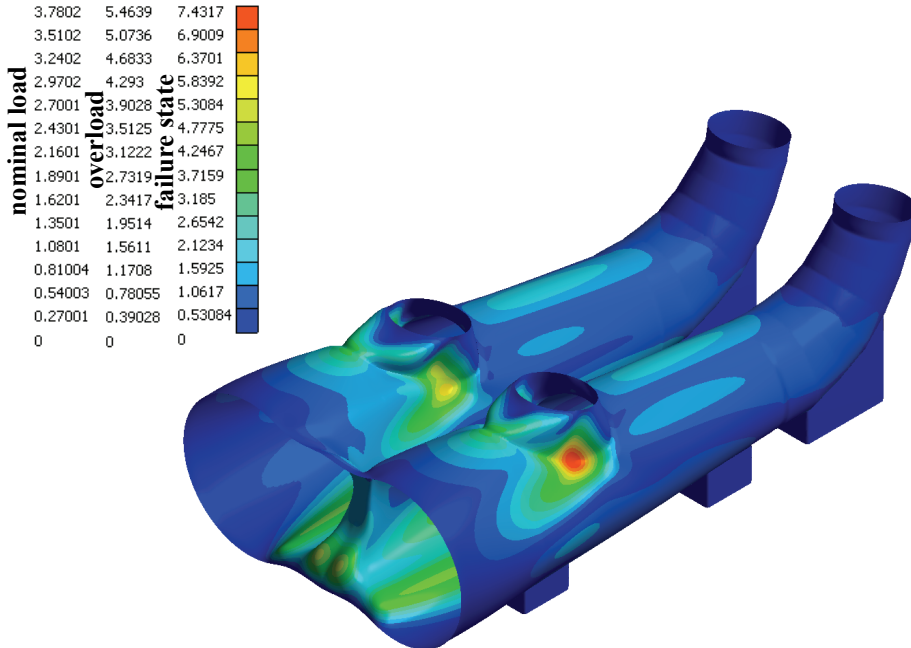


Fig. 9. Failure displacement (values in mm) – colour range for 0.12, 0.16 and 0.21 MPa, respectively.  
Source: prepared by the authors

Figure 10 shows the distribution of safety factors with loads of 0.12, 0.16 and 0.21, respectively. The distribution of safety factors is different, therefore a separate analysis of each coefficient is necessary. The minimum value for nominal load is: 1.4, for overload – 0.93 and the minimum value for failure state is 0.69. When the value of the safety factor is below 1.0, the yield strength of the material has been exceeded.

The obtained results of calculations allowed us to assess the distribution of stress in the pipeline examined. From the point of view of the ultimate strength of the structure the effort of the pipeline is a disadvantage. There is a high concentration of stress in the branch of the main line of the pipeline. A high level of stress concentration may also be found in the area where the coatings of the pipeline and the inlet connection pipe of the turbine are joined.



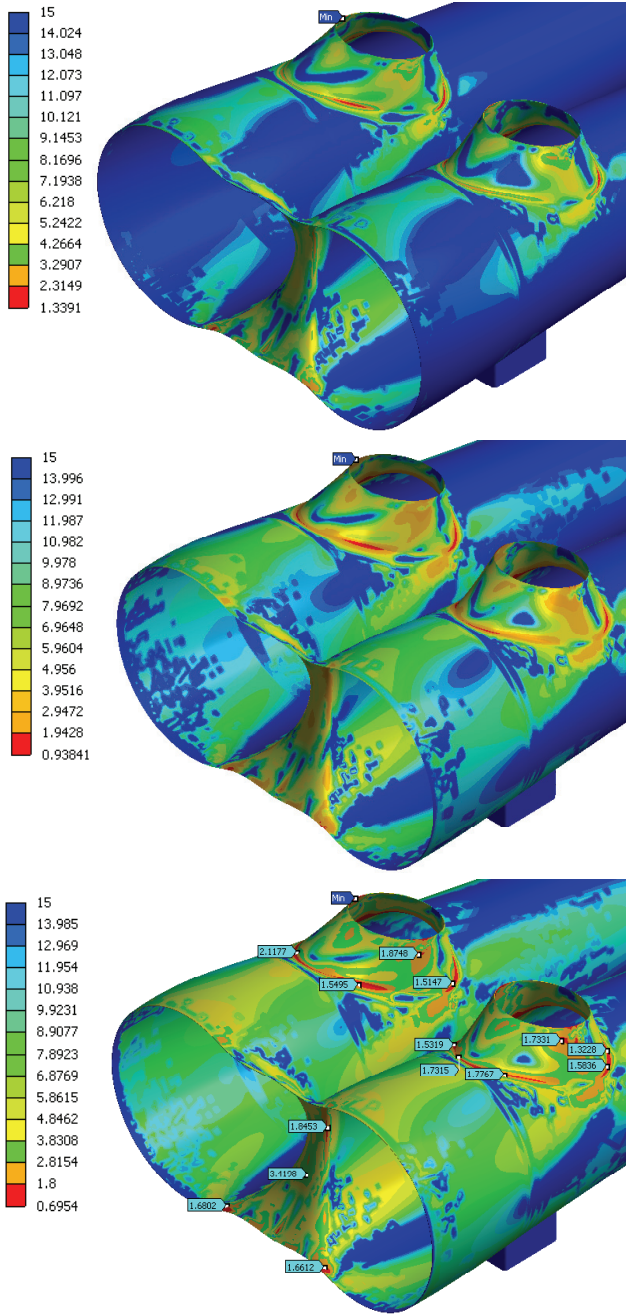


Fig. 10. Safety factors for 0.12, 0.16 and 0.21 MPa, respectively.  
Source: prepared by the authors

The tensile strength of the material was not exceeded under nominal pressure of 0.12 MPa, yet it has been exceeded in the other cases. The level of stress where it piles up has to be reduced, for example, with locally placed reinforcement.

## 7. THE IMPACT OF REINFORCEMENT ON THE DISTRIBUTION OF STRESS IN THE PIPELINE

The previous results have revealed the necessity for the reinforcement of the branch and the inlet connection pipe. Initial simulation of the reinforcement (Fig. 11) has shown that an increase on the wall thickness of the branch and the connection pipes reduces the stress by half and therefore lowers the stress much under the tensile strength even in failure state under a 0.21 MPa pressure.

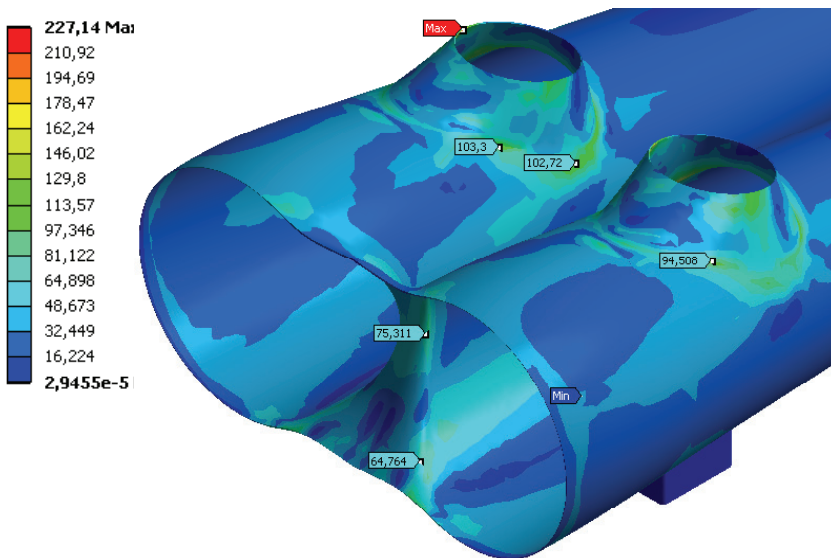


Fig. 11. Reduced stress for 0.21 MPa and an increase in local thickness of the branch and inlet pipes.

Source: prepared by the authors

The next step of the study was to suggest the best way to stiffen the components in question in relation to material strength, and first of all, how to construct the best support from the technological and economic point of view.

Using the knowledge concerning existing solutions (Fig. 12) designed by the authors, the computer simulation was used again and multiple alternatives were examined to match an optimal way to reinforce the pipeline. The calculations were carried out for the pressure of 0.21 MPa at failure state until the achieved value of the reduced stress was below the yield strength.



Fig. 12. Similar solutions.  
Source: prepared by the authors

Finally, bracing with 6 webs and a ring was selected using 6 mm thick plates (Fig. 13). The location of the webs symmetrically to longitudinal section plane where the biggest distortion is observed was considered the most effective. Cladding with a 3 mm plate was chosen for branch reinforcement.

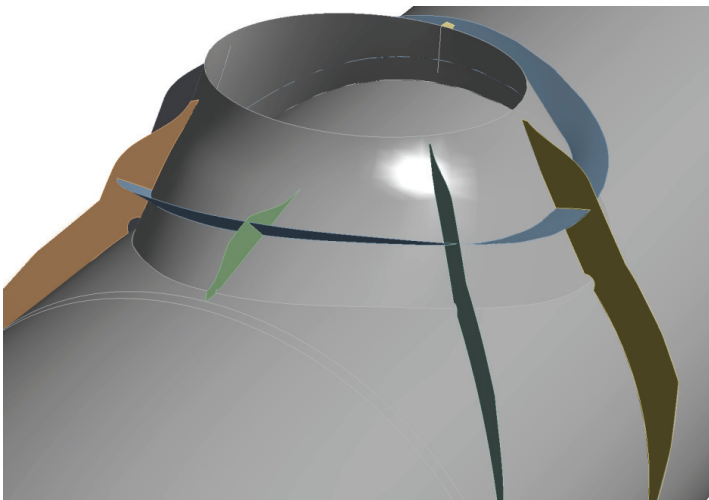


Fig. 13. Optimum location and shape of webs and bracing ring.  
Source: prepared by the authors

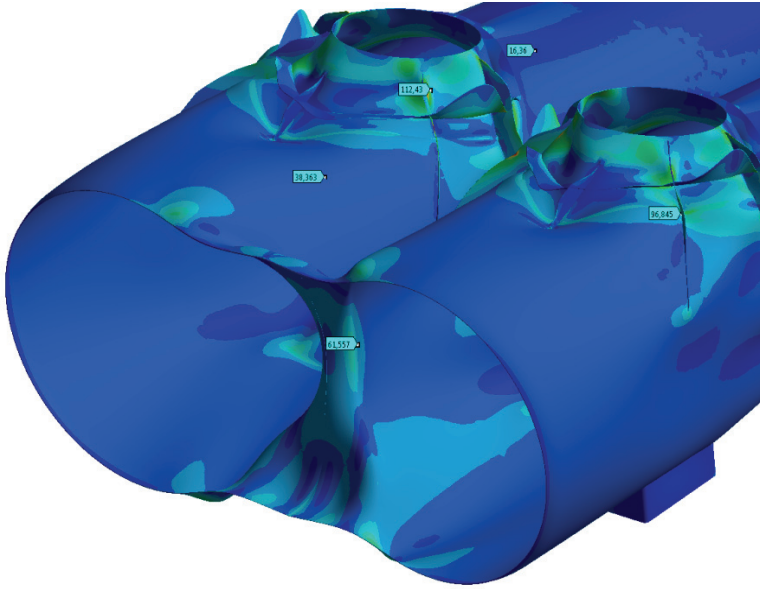


Fig. 14. Reduced stress in the pipeline with reinforcement under 0.21 MPa pressure below yield strength of the material  $Re = 280$  MPa (rescaled view).

Source: prepared by the authors

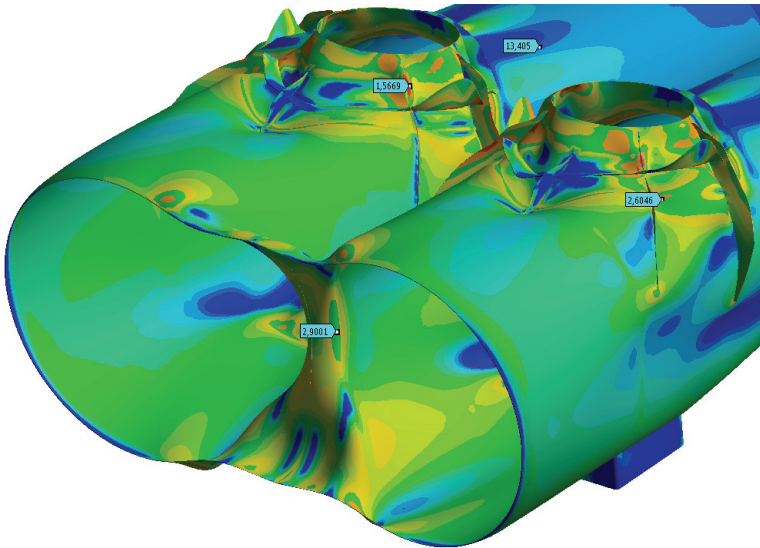


Fig. 15. Safety factor in the pipeline with reinforcement under 0.21 MPa pressure in the range above 1.5 (rescaled view).

Source: prepared by the authors

Attention should be paid to the shape of the web edge. The webs are not supposed to be stripes of plate attached at a right angle, but there should be an arc which gradually closes the width of the web to zero, which eliminates the concentration of stress [4] at the edge, i.e., at the point of contact between the web and the pipeline wall. It is essential to make a relief notch in webs located in the areas where the webs contact welded joint of the connection pipe and the pipeline.

## 8. REINFORCEMENT SET-UP AND ASSEMBLY

### 8.1. PLATING OF PIPELINE BRANCH

The assembly of the reinforcement components began from the central part (Figs. 16, 17). The reinforcement of the branch structure was divided into stages. First, the plating of the central part of the branch was laid, next the reinforcement plates of the bottom surface were laid symmetrically to the division plane. The last stage was creating the reinforced areas on the top surface of the branch. After careful fitting into the pipeline surface, each component was joined with tack welds along its perimeter, the plug welds were also made.



Fig. 16. Quadrangular components of the reinforcement in the process of tacking along centerline of the branch.

Source: prepared by the authors



Fig. 17. Structure welding of the reinforcement components to the coating of the pipeline branch and the completed painted plating of the pipeline branch.

Source: prepared by the authors

The next stage comprised of depositing circumferential fillet welds which joined three components simultaneously: adjoining edges of reinforcement components with pipeline coating, MAG welding process and 1.0 mm diameter solid wire with Ar + CO<sub>2</sub> gas shield were used.

## 8.2. BRACING OF INLET PIPE

The bracing prepared was tacked to inlet pipe plates and pipeline coating as specified in the project documentation (Fig. 18). Once all webs and reinforcement rings were tacked to both inlet pipes, welding work began. The developed welding procedure stipulated weld deposition symmetrically to longitudinal centerline of the web in addition to welding of individual webs symmetrically to inlet pipe centerline. The same welding process was used as for joining the reinforcement in the pipeline branch, i.e., MAG welding process.



Fig. 18. Assembly of webs to the inlet pipe and the pipeline coating.  
Source: prepared by the authors

In accordance with the experimental procedure visual tests (VT) and magnaflux tests (MT) were conducted to examine the fillet welds (Fig. 19). Quality level C, as defined in PN ISO 5817 standard, was presumed in the study. Test results have shown that the quality of the welds fits level B, which is higher than the presumed level.



Fig. 19. Bracing webs with reinforcement ring on the inlet pipe.  
Source: prepared by the authors

## 9. CONCLUSIONS

The numerical simulations have shown that in the case of hydrostatic load up to 0.21 MPa for failure state the effort of the material of the connection pipe is below yield strength  $Re = 280$  MPa, with extreme points reaching 375 MPa, when the lowest ultimate strength of the pipeline material is  $Rm = 401$  MPa. The extreme points might, basically, be omitted due to the fact that they usually are typical discretization errors caused by discontinuity of tangent deformation at finite element points of contact (especially when they occur at an angle) [3]. The material works below the yield strength with the load assumed as dynamic and overload factor of 1.4 to 1.16 MPa. In the case of nominal load of 1.12 MPa the effort of the material does not exceed 184 MPa with 1.5 overload factor, which is below permanent distortion level.

After the work had been completed, the pipeline was started and the hydroelectric plant is currently operating as scheduled. One way to verify the project is to measure the components of stress state on the walls of the modernized pipeline, though this has not been done yet .

*Calculations were carried out at the Academic Computer Center in Gdańsk.*

## REFERENCES

- [1] ADAMKOWSKI A. et al., *Pomiar naprężenia najbardziej obciążonych elementów rurociągu derywacyjnego w ew. Lapino w różnych warunkach obciążeń ciśnieniem wewnętrznym*, IMP , Gdańsk 2009.
- [2] BATHE K.J., *Finite Element Procedures*, Prentice Hall, New Jersey 1996.
- [3] DOBROCIŃSKI S., *Stabilność rozwiązań zagadnień odporności udarowej konstrukcji*, ITE, Akademia Marynarki Wojennej, Gdynia 2000.
- [4] MEYERS M.A., *Dynamic behaviour of materials*, A Wiley-Interscience Publication, 1994.

