



The 4th International Conference  
"Computational Mechanics  
and Virtual Engineering"  
COMEC 2011  
20-22 OCTOBER 2011, Brasov, Romania

## FINITE ELEMENT ANALYSIS AND DYNAMIC ANALYSIS OF THE OUTER BEARING BUSH FROM THE BLADE ADJUSTMENT MECHANISM OF KAPLAN TURBINES

**Camelia Jianu, Constantin.V. Câmpian, Vasile Cojocaru, Dorian Nedelcu**

"Eftimie Murgu" University of Resita, Center for Research in Hydraulics, Automation and Thermal Processes  
CCHAPT, Resita, ROMANIA, e-mail: jianu\_camelia@yahoo.com, v.campian@uem.ro, v.cojocaru@uem.ro,  
d.nedelcu@uem.ro

**Abstract:** The objective of this paper is to analyze through the finite element method (FEM) and the motion analysis the dynamical behavior of the bushing adjustment mechanism of Kaplan turbines. The motion generated by the servomotor is transmitted from the fork head through the connecting rod to the pin lever – trunnion – blade subassembly. This subassembly of the Kaplan turbine is supported on the hub by two bushings: an outer bush and an inner bush. Analysis of the Kaplan turbine's bushing was made in order to determine the stresses distribution, the displacements, the deformations and the factor of safety distribution. A three-dimensional model of the Kaplan turbine runner was generated based on the designed data. Finite element analysis and motion analysis was performed using SolidWorks package. Performing numerical calculations, the entire loading process history, the deformed geometry, the nodal velocity and the distribution of stresses, displacements and deformations are obtained. Results predicted by the finite element method show that this method is efficient and accurate and in good agreement with the theoretical and experimental values. Results from the current analysis can be used for further studies in design of the Kaplan turbine runners as part of a continuous product development process.

**Keywords:** finite element method, outer bearing bush, motion analysis, Kaplan turbine runner.

### 1. INTRODUCTION

Finite elements analysis (FEA) is a widely accepted computer simulation methodology for modeling, evaluation and optimization of product's mechanical and structural design from a vast array of engineering fields [1, 2, 3]. FEA is capable of reducing the design time as well as the expenses of extensive physical prototyping. Recent developments in computer hardware, numerical solutions and design optimization software are providing faster and cheaper results in an optimal design process [4].

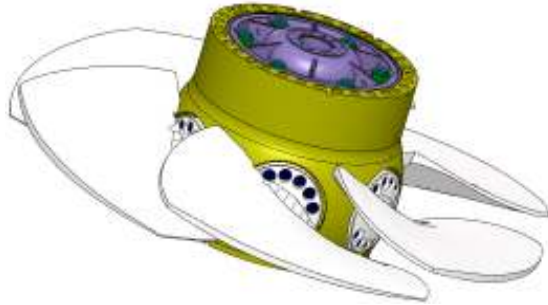
Motion simulation software gives the reaction forces/moments acting on each component. The reactions and the body forces acting on each component can be exported to finite element analysis studies in order to perform stress analysis [5].

A Kaplan turbine runner must be designed according to an optimal cost-efficiency and long-term productive life for coping with the evolution of industrial technology, materials, energy sources and environmental regulations [6].

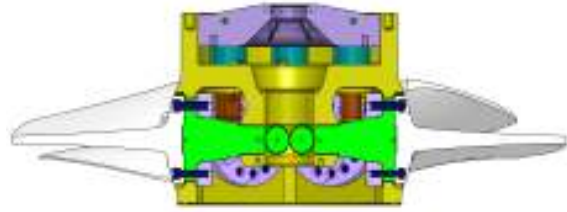
The objective of this paper is to analyze through the finite element method (FEM) and the motion analysis the dynamical behavior of the bushing adjustment mechanism of Kaplan turbines. The motion generated by the servomotor is transmitted from the fork head through the connecting rod to the pin lever – trunnion – blade subassembly. This subassembly of the Kaplan turbine is supported on the hub by two bushings: an outer bush and an inner bush. Analysis of the Kaplan turbine's bushing was made to determine the stresses distribution, displacements, deformations and the factor of safety distribution.

## 2. 3D GEOMETRY

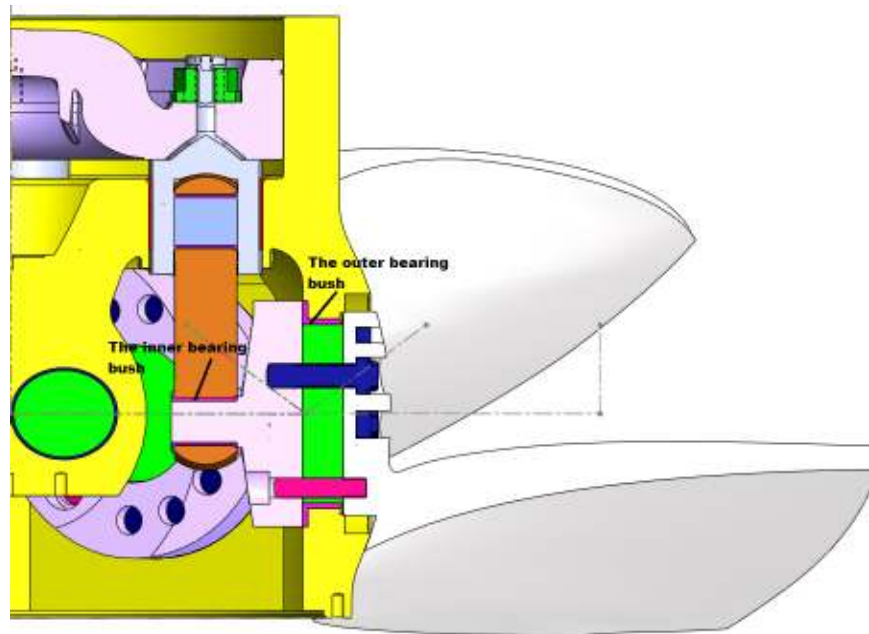
The assembly of the Kaplan turbine runner has a complex 3D geometry (figures 1-2). For this research the geometry was generated in SolidWorks [7], [8]. The outer bearing bush (1398 mm internal diameter) and the inner bearing bush (478 mm internal diameter) are presented in figure 3.



**Figure 1:** The 3D geometry of the Kaplan turbine runner assembly



**Figure 2:** A section through Kaplan turbine runner assembly



**Figure 3:** The outer bearing bush and the inner bearing bush

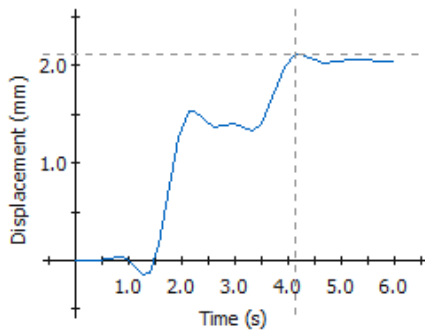
## 3. MOTION ANALYSIS

The required steps in order to perform static analysis are: import of the model geometry, select Motion Analysis from the simulation options pull-down menu, fix and move the components, specify the Input Motion, specify simulation time, run the simulation, visualize the results and transfer of the motion loads in the Solidworks Simulation stress analysis software [9].

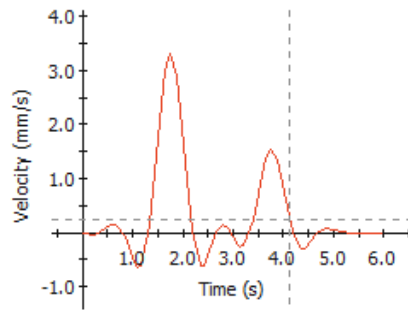
The input data for analysis with SolidWorks are:

- Total time was 6 seconds;
- Linear motor with the law of motion described in Figure 4-7. The motor will be located at the face of the fork head, Figure 8, [10].
- The gravitational force applied on the direction OY;
- The contact between the bushings and joint parts (the coefficient of friction is  $\mu_k = 0.12$ );
- The loads applied on the blade are:
  - The tangential force,  $F_T = 1238.2$  kN. The force is applied perpendicularly on the axis plane of the machine in the center of the pressure of the assembly. The radius of the center of gravity is  $R_{cg} = 2187$  mm;

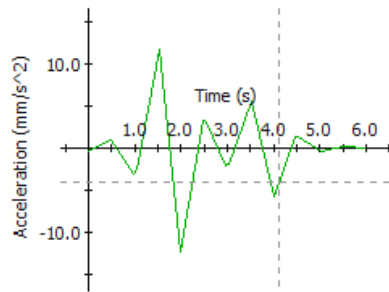
- The centrifugal force,  $F_C=3992.315$  kN. The force applied by the blade to the outside axis;
- The axial force,  $F_A=1845.656$  kN. The force applied by the machine axis direction, in the center of the pressure;



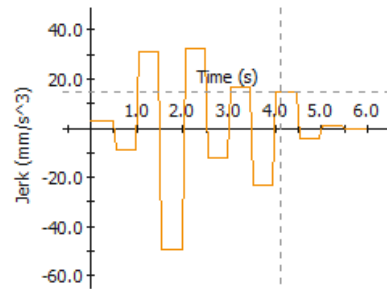
**Figure 4:** Displacement (Interpolation type-cubic spline)



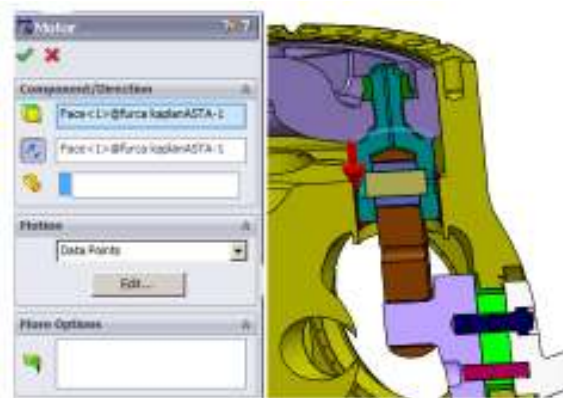
**Figure 5:** Velocity (Interpolation type-cubic spline)



**Figure 6:** Acceleration (Interpolation type-cubic spline)



**Figure 7:** Jerk (Interpolation type-cubic spline)



**Figure 8:** The motor located on the face of the fork head

To determine the radius of the center pressure point of application of forces the next equation was used [10]:

$$R_{cp} = L_s = \frac{D}{3} \cdot \frac{1 - \left(\frac{d}{D}\right)^3}{1 - \left(\frac{d}{D}\right)^2} \cdot \frac{\sin \beta}{\beta} \quad (1)$$

where:  $D$ = outside diameter of rotor;  $D=9500$  mm  
 $d$ = diameter of the rotor hub;  $d=4245$  mm  
 $2\beta$ = winding angle of the hub by the inner blade profile;  $2\beta=84.870^\circ$

The results was  $R_{cp}=3283$  mm.

The load scheme is given in figure 9.

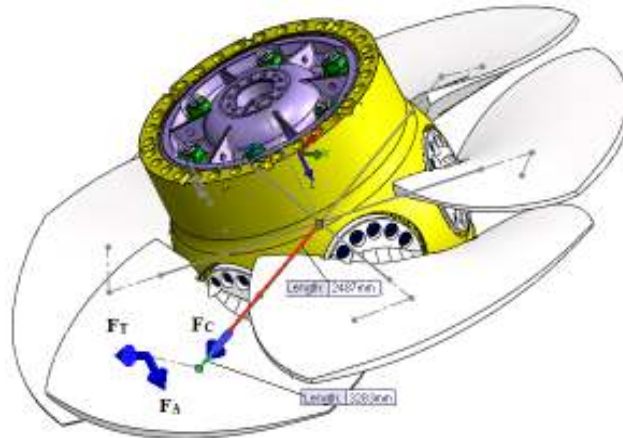


Figure 9: The load scheme

#### 4. TRANSFER AND STRUCTURAL ANALYSIS

Simulation is made for the outer bearing bush and then the requirements set are transferred from SolidWorks Motion to SolidWorks Simulation for structural analysis, figure 10.

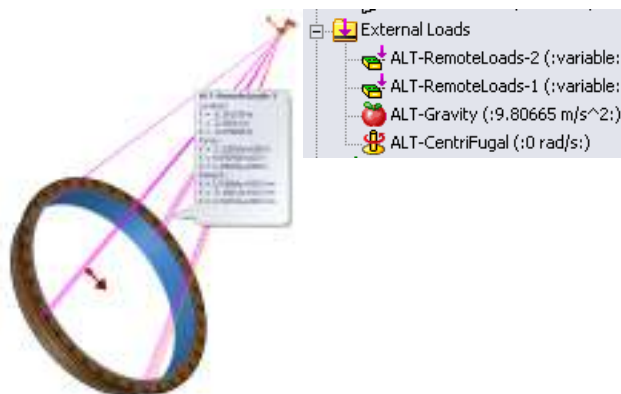


Figure 10: The loads applied on the outer bearing bush

#### 5. STRESS AND DEFORMATION CALCULATION

A linear static analysis calculates the displacements, the stresses and the reaction forces under the effect of applied loads [11], [12]. The required steps to perform static analysis are: import the model geometry, define a material, define adequate restraints, mesh the model, start the analysis calculus and then visualize the results [13].

##### 5.1. The selection of the material

For Static type analysis it is necessary to define the material characteristics: the modulus of elasticity  $E$  and Poisson's ratio  $\nu$ ; also, the density must be defined in order to have the effect of gravity and/or centrifugal loadings; selecting a material from the SolidWorks library will automatically assign these properties. For this study the selected material was *Tin bearing bronze*, with Elastic modulus  $1.1 \times 10^{11}$  N/m<sup>2</sup> and Poisson's ratio  $\nu=0.33$ .

##### 5.2. The restraints

The following constraints are being introduced for rebuilding the joints of the whole piece:

- On top of the cylindrical surface, contact with turbine hub, suppressing the degrees of freedom perpendicular to the surface;
- On the front surface of the bushing, suppressing all the degrees of freedom.

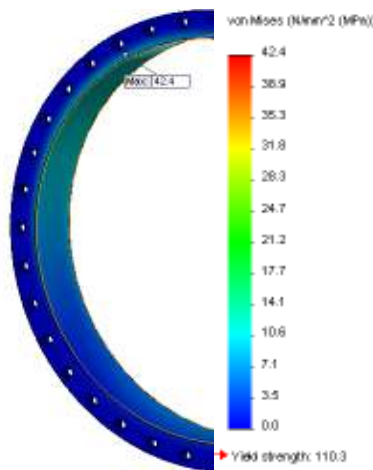
### 5.3. The mesh

The model is a solid entity. Tetrahedral solid elements will be used for meshing. A parabolic tetrahedral element is defined by 4 corner nodes, 6 mid-side nodes, and 6 edges. The Kaplan turbine lever is meshed with High mesh quality, resulting 333541 nodes and 207912 finite elements.

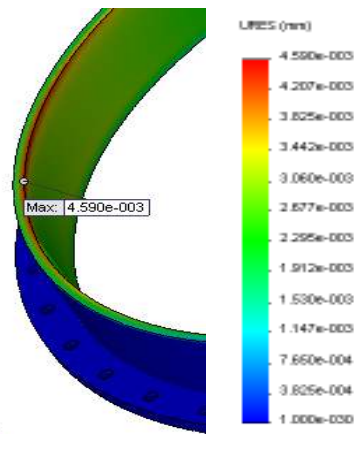
## 6. RESULTS

After the setup of the analysis conditions (the geometry, the material, the restraints, the loads and the mesh) it is possible to run the analysis. The software offers different solvers to handle different types and sizes of problems more efficiently [14].

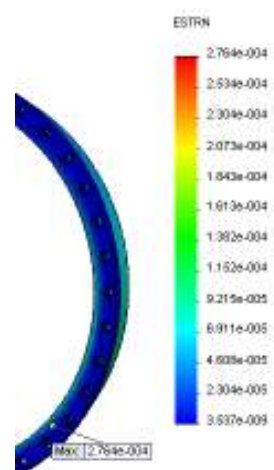
Graphical distribution of stresses is shown in figure 11. Graphical distribution of resultant displacements is presented in figure 12. Graphical distribution of resultant deformation distribution is shown in figure 13.



**Figure 11:** The VonMises distribution



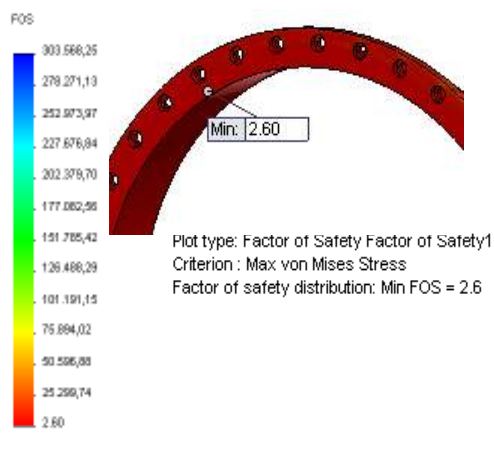
**Figure 12:** The resultant displacements



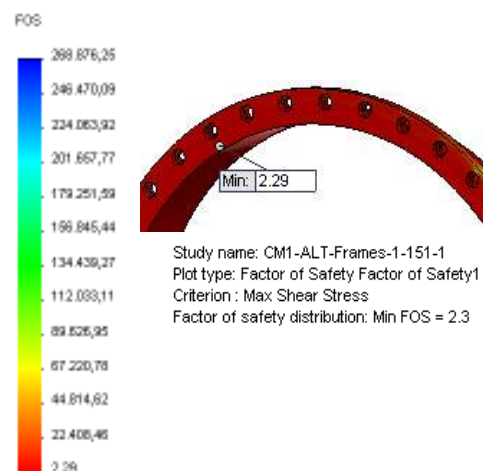
**Figure 13:** The ε resultant deformations distribution

Graphical representation for factor of safety distribution is shown according to:

- criterion: Max von Mises Stress; factor of safety distribution: Min FOS = 2.60 (figure 14);
- criterion: Max Shear Stress; factor of safety distribution: Min FOS = 2.29 (figure 15);



**Figure 14:** Factor of safety distribution Min FOS = 2.6. Criterion: Max von Mises Stress



**Figure 15:** Factor of safety distribution: Min FOS = 2.29 Criterion: Max Shear Stress

### 3. CONCLUSION

The analysis of the outer bearing bush was made to determine the stress distribution, displacement, deformation and the factor of safety distribution.

A three-dimensional model of the outer bearing bush with a complex geometry was generated based on the designed data. Finite elements analysis was performed using SolidWorks Simulation software and motion analysis was performed using SolidWorks Motion.

Results predicted by the finite element method show that the presented process is efficient and accurate and in good agreement with the theoretical and experimental values. The maximum value of the Von Mises stress is 42.4 [MPa], lower than yield strength (110,3 MPa).

As further research, a fatigue analysis of outer bearing bush is to be accomplished. Results from the current analysis can be used for further studies in designing of the outer bearing bush as part of a continuous product development process.

### REFERENCES

- [1] Kurowski P.M: Finite Element Analysis For Design Engineers, Society of Automotive Engineers, 2004, ISBN 978-0-7680-1140-1.
- [2] Reddy J.N.: An Introduction To The Finite Element Method – Third Edition, McGraw-Hill College, 2004.
- [3] Singiresu S.R.: The Finite Element Method in Engineering – Fourth Edition, Elsevier Inc., USA, 2005.
- [4] Logan D.L.: First Course in the Finite Element Method – Fourth Edition, Thomson Learning, 2006.
- [5] An Introduction to Motion Analysis Applications with SolidWorks Motion, Instructor Guide, Dassault Systèmes SolidWorks Corporation.
- [6] Braess D.: Finite Elements: Theory, Fast Solvers, and Applications in Solid Mechanics – Third Edition, Cambridge University Press, UK, 2007, ISSN 1431-1550.
- [7] Jianu C.: “Computer Aided Design of Kaplan Turbine piston with SolidWorks” *Analele Universității „Eftimie Murgu”*, Reșița, Anul XVII, Nr.2, 2010, ISSN 1453-7397.
- [8] Jianu C., Budai A. M., Câmpian C. V.: “Computer Aided Design of the link-fork head-piston assembly of the Kaplan turbine with Solidworks” *Analele Universității „Eftimie Murgu”*, Reșița, Anul XVII, Nr.2, 2010, ISSN 1453-7397.
- [9] Nedelcu D., Campian C.V., Padurean I.: Stress and deformations on Pelton blade turbine calculated by finite elements method, The International Conference on Hydraulic Machinery and Equipments Timisoara, October 16-17, 2008 Romania, Scientific Bulletin of the Politehnica University of Timisoara Transactions on Mechanics, Special Issue ISSN 1244-6077; Tom 53 (67)
- [10] Strength and lifetime duration calculus for runner blade lever, CCHAPT, Technical Report No.U-09-400-289, November, 2009.
- [11] Nedelcu D., Mănescu T. Ș., Câmpian C. V: Finite Element Through COSMOS M/Design STAR, FME Transactions, Volume 32, Number 1, 2004, University of Belgrade, Faculty of Mechanical Engineering, YU ISSN 1451-2092, 2004.
- [12] Mănescu T., Nedelcu D.: Finite element method structural analysis, *Analiză structurală prin metoda elementului finit*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-217-6, Octombrie, 2005.