Scientific Bulletin of the Politehnica University of Timisoara
Transactions on Mechanics, Special Issue
ISSN 1244-6077; Tom 53 (67)

The International Conference on Hydraulic Machinery and Equipments
Timisoara, October 16-17, 2008
Romania

STRESS AND DEFORMATIONS ON PELTON BLADE TURBINE CALCULATED BY FINITE ELEMENTS METHOD

Dorian NEDELCU
"Eftimie Murgu" University of Reşiţa
P-ta Traian Vuia no. 1-4, 0723 299 050, 0256 219134, d.nedelcu@uem.ro, http://www.uem.ro/

Constantin Viorel CÂMPIAN
"Eftimie Murgu" University of Reşiţa
P-ta Traian Vuia no. 1-4, 0722 224 801, 0256 219134, v.campian@uem.ro, http://www.uem.ro/

Ioan PĂDUREAN
"Politehnica" University of Timişoara
P-ta Victoriei no. 2, 0729 351201, padurean58@yahoo.com, http://www.upt.ro/

**Corresponding author*: Bd. A. I. Cuza, bl. 35, sc. 2. et. 1, ap.3, Reşiţa, Romania
Tel.: (+40) 0723 299050, Email: d.nedelcu@uem.ro

ABSTRACT

The Pelton blade turbine is a very complex geometry due to the blade shape and variable thickness. Analytical methods offer an appreciative calculus of the stress. Experimental methods can be used, but with expensive costs for blade model and technical measurements devices. As an alternative, numerical finite element method can be used. The paper describe step by step the procedure to generate the 3D solid geometry and the finite element method (FEM) used to calculate deformation and stress values for Pelton blade turbine, using mixed Autodesk Inventor and Cosmos Design Star software.

KEYWORDS

Pelton, turbine, blade, stress, deformations, finite element method.

INTRODUCTION

The shape and terminology of a Pelton blade is presented in figure 1 and figure 2. The purpose of Computer Aided Design (CAD) modelling is to obtain the blade as a solid object and not as a surface, because solid model of the blade can be used later for finite elements resistance calculus in a FEM software. Autodesk Inventor was chosen as CAD software.

THE BLADE CUP

Usually, there are known or can be obtained, from hydrodynamic calculus, the 3D coordinates of blade profiles, figure 3.

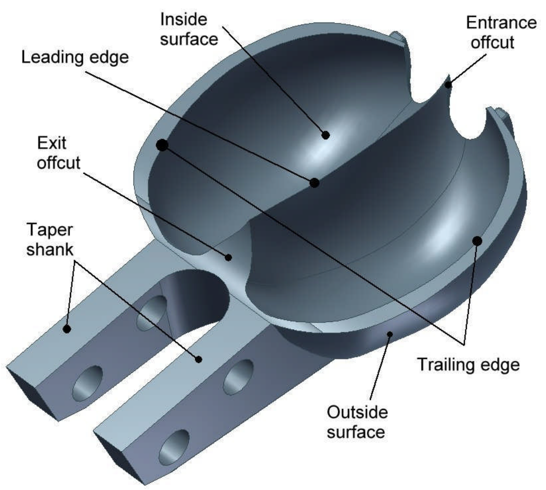

Figure 1 The Pelton blade terminology

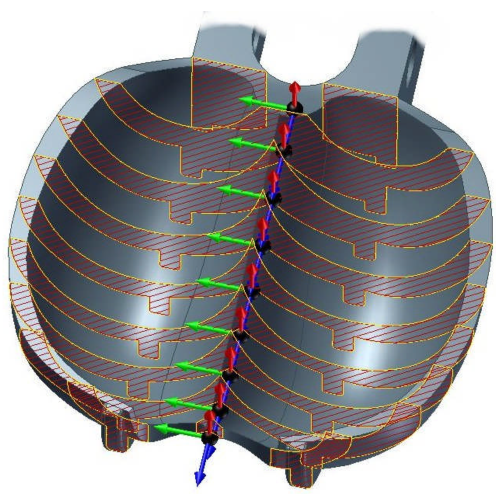

Figure 2 The Pelton blade shape

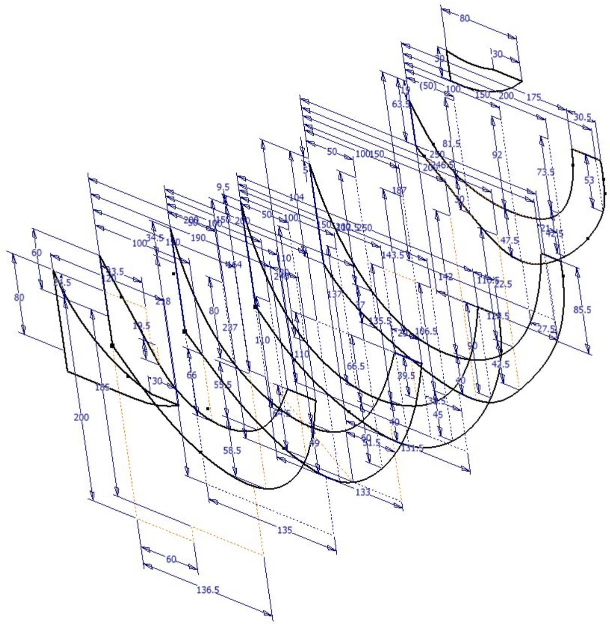

Figure 3 The 3D profiles of the blade

In Autodesk Inventor solid object can be created with **Loft** command, based on any number of closed loops, which may be curves in 2D sketches or 3D sketches. The blade cup will be generated by **Loft** command, applied to 3D profiles of blade, figure 4.

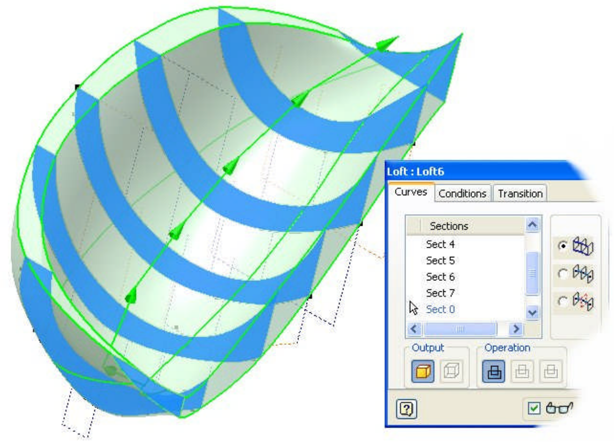

Figure 4 The blade cup generated by **Loft** command

THE TAPER SHANK

The **Extrude** command will be used to generate the taper shank. The command requires selection of the closed contour which will be extruded and of the **Join** & **To Next** command option, figure 5.

THE EXIT OFFCUT

The **Extrude** command will be used to generate the taper shank. The command requires selection of the closed contour which will be extruded and of the **Cut** command option, figure 6.

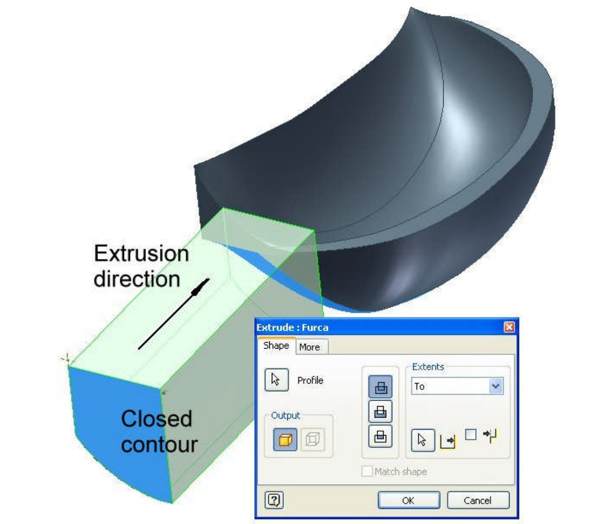

Figure 5 The taper shank generated by **Extrude** command

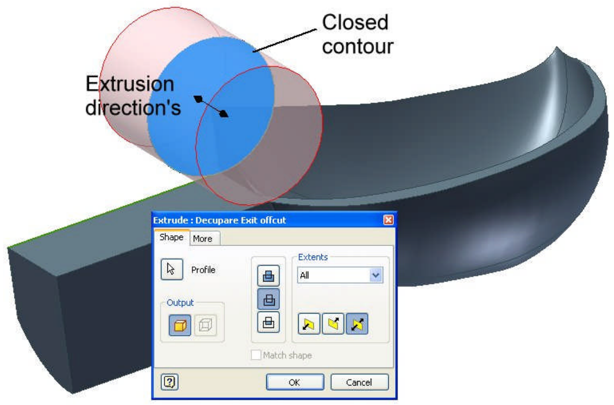

Figure 6 The Exit offcut generated by **Extrude** command

The **Mirror** command will be used to generate the mirror of the cup, taper shank and exit offcut, figure 7.

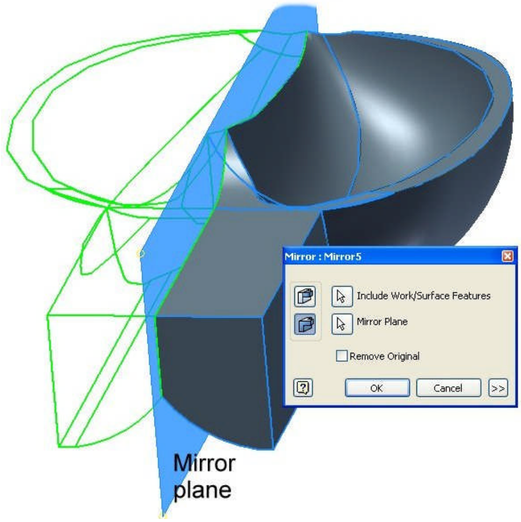

Figure 7 The **Mirror** command

THE TAPER SHANK OFFCUT

The **Extrude** command will be used to generate the taper shank offcut. The command requires selection of the closed contour which will be extruded and the **Cut** command option, figure 8.

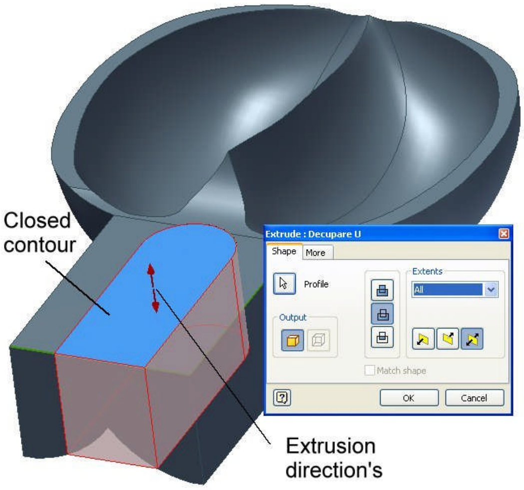

Figure 8 The taper shank offcut generated by **Extrude** command

THE ENTRANCE OFFCUT

The **Extrude** command will be used to generate the entrance offcut. The command requires selection of the closed contour which will be extruded and of the **Cut** command option, figure 9.

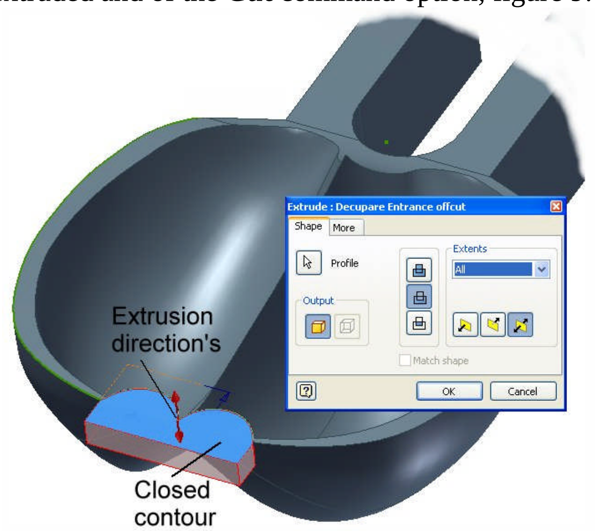

Figure 9 The entrance offcut generated by **Extrude** command

THE RIB

The **Extrude** command will be used to generate the rib. The command requires selection of the rib profile which will be extruded and of the **Join** command option, figure 10. The symmetrical rib will be generate by **Mirror** command, figure 11.

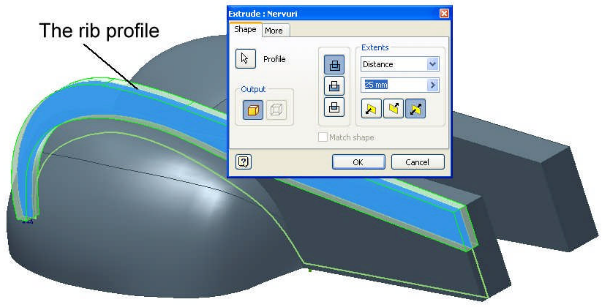

Figure 10 The rib generated by **Extrude** command

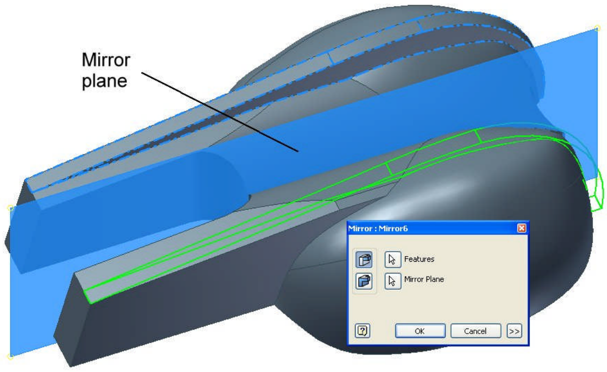

Figure 11 The symmetrical rib generated by **Mirror** command

THE TAPER SHANK HOLES

The **Hole** command will be used to generate the taper shank holes, figure 12. The command require to define the hole's position, diameter and depth.

Figure 12 The taper shank holes and fillets generated by **Hole / Fillet** commands

The final operation is to round the edges of the rib's blade, with command **Fillet**.

THE BLADE GEOMETRY

The blade geometry, figure 3, is taken from K 461 runner [1], [2] and scaled for do=165 mm. The main dimensions are presented in figure 13 and table 1 [3].

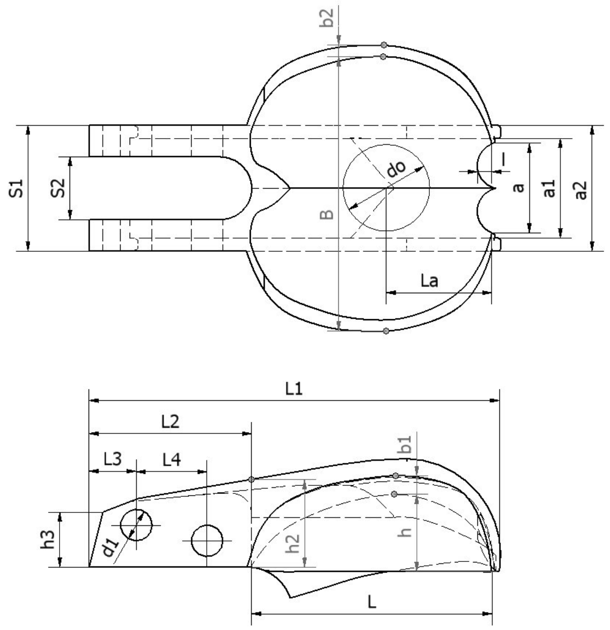

Figure 13 The blade main dimensions

Table 1

Parameter	Value	Parameter	Value
Do	1650	h2	225
do	165	h3	140
B	500	a1	225
L	472.5	a2	260
h	190	l	52.5
a	185	b1	30
La	260	b2	20
L1	800	S1	240
L2	300	S2	120
L3	90	d1	70
L4	135		

THE PELTON RUNNER

The Pelton runner will be assembled on a disc with bolts, figure 14. The main characteristics of the runner are presented in table 2:

Table 2

Number of the blades	Z=20
The characteristic diameter	Do=1650 mm
The jet diameter	do=165 mm
The head	H=543 m
The rotational speed	n=500 rpm

STRESS AND DEFORMATION CALCULATION

Linear static analysis was performed on a single blade with Cosmos Design Star software [4]. Linear static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.

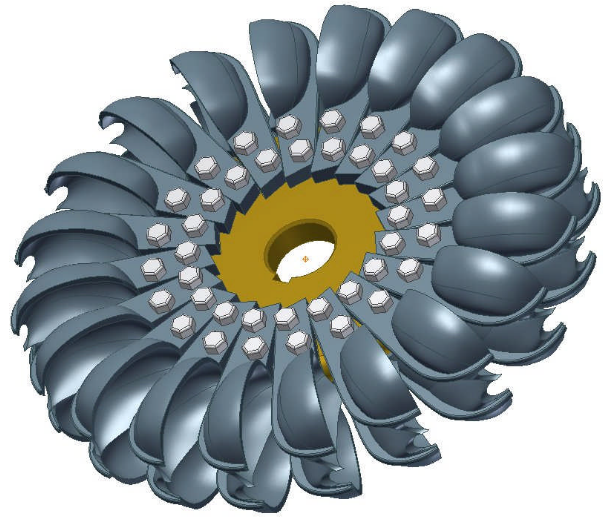

Figure 14 The Pelton runner

The required steps to perform static analysis are: import the model geometry, create a static analysis study, define a material, define adequate restraints, define some type of loading, mesh the model, start the analysis calculus and visualize the results.

The geometry of the blade is imported from Autodesk Inventor software as SAT file.

A study is completely defined by the type of analysis and related options, material assignments, loads and boundary conditions and mesh. So, it is possible to create a number of studies with different materials, loads, boundary conditions and meshes. For this analyze will be selected **Static** option for **Analysis Type**, **Solid** option for **Mesh Type** and **FFE solver**.

For **Static** type analyze it is necessary to define the material characteristics: the Modulus of Elasticity **E** and Poisson's ratio **ν**; also, the density must be defined when considering the effect of gravity and/or centrifugal loading, like in our case; selecting a material from the COSMOS library, will assign automatically these properties, table 3.

Table 3

Property Name	Value	Units
Material name	Alloy Steel (SS)	
Material Model Type	Linear Elastic Isotropic	
Elastic modulus	2.1e+011	N/m^2
Poisson's ratio	0.28	NA
Shear modulus	7.9e+010	N/m^2
Mass density	7700	kg/m^3
Tensile strength	7.2383e+008	N/m^2
Yield strength	6.2042e+008	N/m^2

Cosmos Design Star applies loads and boundary conditions to geometric entities as features that are fully associated with the current geometry.

Restraints must not allow any rigid body motion. The blade will be fixed in the taper shank holes, figure 15. The **Fixed** restraints impose 0 value for the translations and rotations of the selected entities.

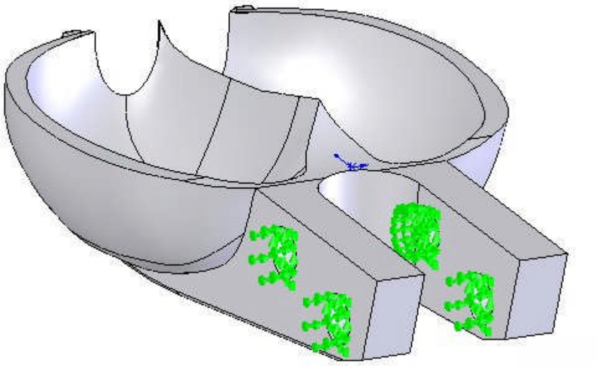

Figure 15 The **Fixed** restraints applied to in the taper shank holes

The loads applied to the blade are:

- the tangential force T, with value calculated by formula [3]:

$$T = 838 \cdot d_o^2 \cdot H = 12390\ N \qquad [1]$$

- the centrifugal load, generated for n=500 rpm.

If the runner is stuck, the centrifugal force is zero, but the tangential force reaches the maximum value and can be calculated with formula [3]:

$$T_{\max} = 2938 \cdot d_o^2 \cdot H = 44000\ N \qquad [2]$$

The tangential force, figure 17, is the result of the hydrodynamic jet action applied to the blade on **do**=165 mm circle. To apply the tangential force only in this area, the circle **do** must be projected on the inside surface of the blade. The **Split Line** tool projects a sketch to curved or planar faces. It divides a selected face into multiple separate faces, to enabling select operation of each face. The results are visible in figure 16.

The centrifugal load is defined by the value of angular velocity (radians or rpm) and rotation axis. In our case, the centrifugal load is generated around the rotation axis, figure 17, which correspond to Do=1650 mm diameter.

Because the model is a solid entity, tetrahedral solid elements will be used for meshing, figure 18; a parabolic tetrahedral element is defined by 4 corner nodes, 6 mid-side nodes, and 6 edges.

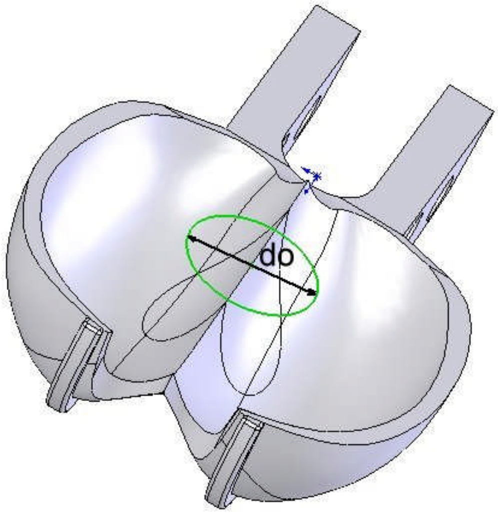

Figure 16 The projection of **do**=165 mm circle on the inside surface of the blade with **Split Line** tool

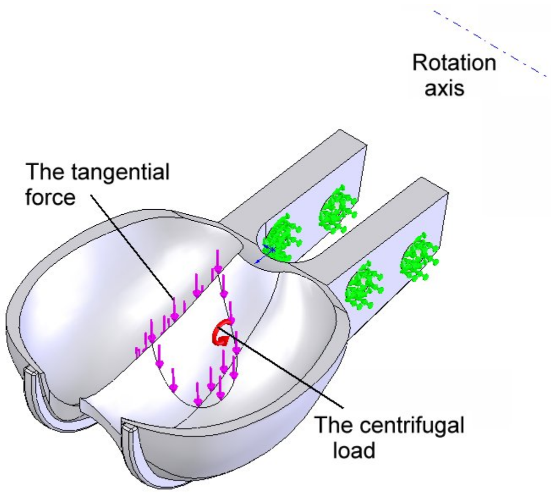

Figure 17 The loads applied to the blade

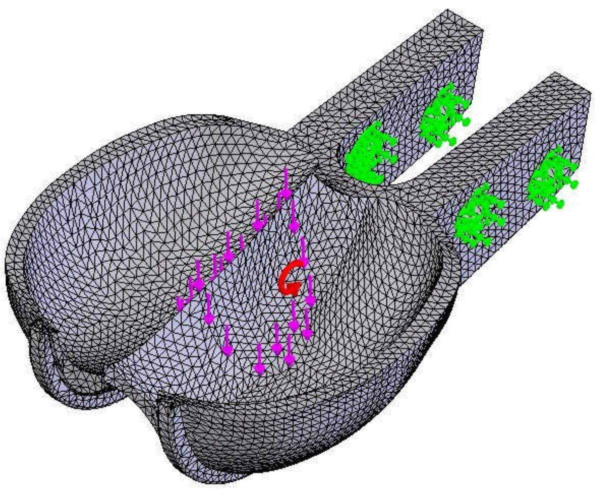

Figure 18 The mesh of the blade

After completely defining a study, the results can be obtained by running of the analyse study. Running a study calculates the results based on the geometry, material, loads and boundary conditions, and mesh. After the program finishes analyzing the study, there is possible to visualize the results. The final results are presented in table 4 and figures 19…20.

Table 4

Parameter	Value	U/M
Case 1		
The tangential force	12.390	N
The centrifugal load	Applied	-
The VonMises stress value	88.24	MPa
The displacement value	0.3138	mm
Case 2		
The maximal tangential force	44.000	N
The centrifugal load	No applied	-
The VonMises stress value	191.3	MPa
The displacement value	0.6167	mm

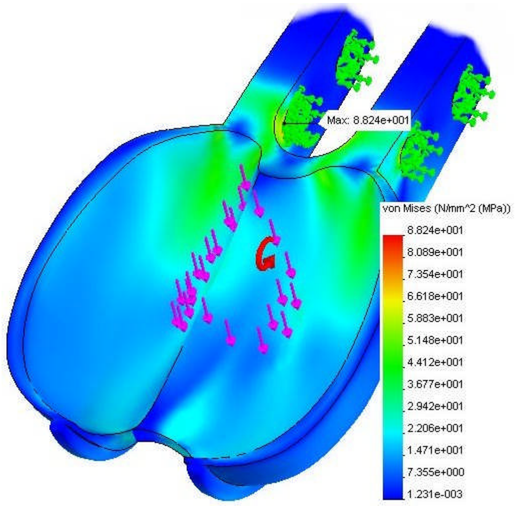

Figure 19 The VonMises plot – Case 1

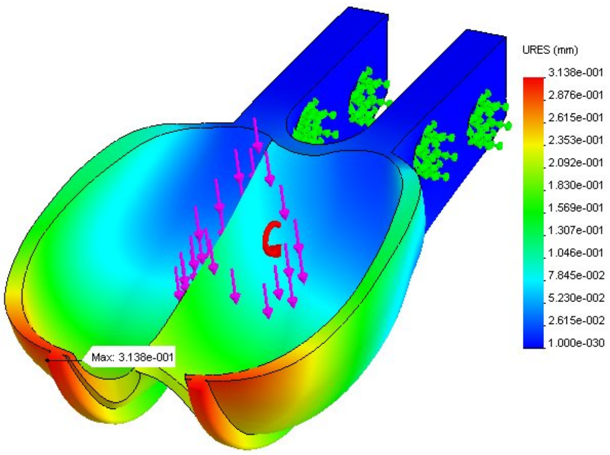

Figure 20 The displacement plot – Case 1

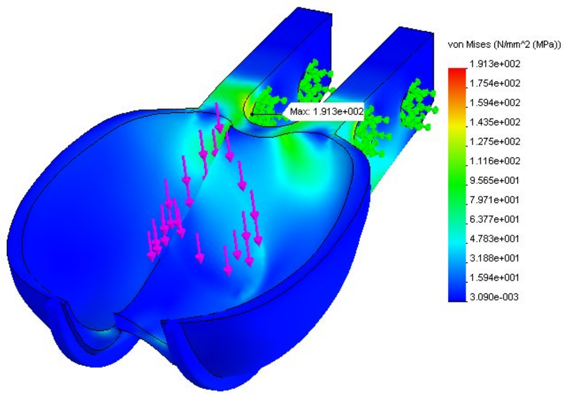

Figure 21 The VonMises plot – Case 2

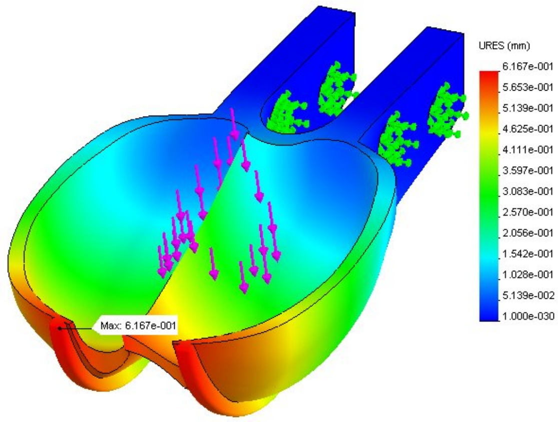

Figure 22 The displacement plot – Case 2

CONCLUSIONS

The maximum VonMises stress value is reached in the area of the taper shank hole and the maximum displacement value in the area of blade entrance.

The mixed software Autodesk Inventor and Cosmos Design Star offer to the designer engineer a numerical solution to verify the resistance of the Pelton turbine blade. The complex geometry of the blade must be modelled with high precision, to assure the curves continuity and correct results. The mesh in the sensible zones must be created much smaller for a good accuracy of the final results.

REFERENCES

1. Anton, I., - *Turbine hidraulice*, Editura Facla, Timişoara, 1979.
2. Edel, JU.J. – *Kovsovîe ghidroturbini*, Maşghiz, Moskova, 1963.
3. Pavel, D. - *Turbine hidraulice*, Editura Energetică de Stat, Bucureşti, 1954.
4. Mănescu, T., Nedelcu, D. - *Analiză structurală prin metoda elementului finit*, Editura Orizonturi Universitare, Timişoara, 2005.