International Journal of Modern Manufacturing Technologies
ISSN 2067–3604, Vol. XIV, No. 2 / 2022
https://doi.org/ 10.54684/ijmmt.2022.14.2.69

# SPIRAL TURBINE CASING - IMPELLER ASSEMBLY DESIGN AND FLUID FLOW ANALYSIS

**Constantin Luois Dumitrache, Dumitru Deleanu**

Maritime University of Constanta, Department of General Engineering Sciences, Mircea cel Batran street, No. 104, Constanta, 900663, Romania

Corresponding author: Constantin Luois Dumitrache, ldumitr@yahoo.com

***Abstract:*** Turbine is a hydraulic engine where energy transfer occurs both based on the kinetic energy and potential (pressure) of the water. In this article we have designed the Spiral Turbine Casing – Impeller assembly with dimensions, geometry from technical documentation forms and the articles of other authors. CAD design of this assembly is a challenge and was done using .NX SIEMENS. The study of the fluid flow was done with ANSYS CFX by importing this assembly, some conclusions were drawn regarding the fluid velocities and the pressure field. The cavitation phenomenon that may occur in the operation of the turbine was also taken into account and some conclusions were drawn regarding the effects of this phenomenon. If the presence of vapours in the hydraulic circuit is found, then the cavitation can develop and this fact can be highlighted with Vapour Volume Fraction.

***Key words:*** spiral turbine, fluid flow analysis, cavitation, NX Siemens, ANSYS CFX.

## 1. INTRODUCTION

Hydraulic turbines, also called "hydraulic engines", are transforming hydraulic energy in mechanical energy with high efficiencies and which constitutes a basic component in economic development.

Hydraulic turbines use cheap and inexhaustible energy, are competitive, these include Francis turbines (figure 1) that have a specific rotational speed $n_s$ within the range 60÷500 rpm and better works when it is used with waterfalls H=70÷500 m.

The operating fluid flow is brought by the forced pipe to spiral casing, which ensures the uniform distribution flow of water on the periphery of the stator, the fluid flow guide device and then the impeller. Inside the spiral casing takes place the partial transformation of the potential energy of water into kinetic energy. The stator is composed of a number of stator columns, usually embedded in the spiral casing and profiled, which have the role of supporting the construction and directing the water to the fluid flow guide device and impeller [2].

The fluid flow guide device, consisting of 12 to 36 blade guides, with the axis placed on a cylinder, whose spatial position is simultaneously adjustable by means of fluid flow adjustment mechanism, its main role is to adjust the flow in order to balance the turbine engine torque with the resistant torque of the electric generator and to create a large circulation at the entrance to the impeller [3].

The Francis turbine impeller consists of a number of 11÷19 spatially twisted fixed blades. Inside of the impeller takes place the conversion of the hydraulic energy of the water into mechanical energy, transmitted to the turbine shaft and then the electric generator.

The suction tube (diffuser) is the constructive element of the turbine that ensures the evacuation of the flow passed through the impeller to the escape channel. Inside, is recovering the remaining kinetic energy from impeller outlet and suction height if this is positive (when the impeller is above downstream water level).

## 2. FUNCTIONAL PARAMETERS OF TURBINES

A turbine is characterized by the following parameters:
- flow, Q [$m^3$/s];
- waterfall, H [mcol $H_2O$];
- power, P [KW, HP];
- rotational speed, n [rpm];
- efficiency, $\eta_T$;
- geometric suction height, $H_S$ [mcol $H_2O$];
- the cavitation coefficient, σ.

### 2.1. Turbine flow

This is defined as the amount of water entering the turbine in the unit of time. Flow is expressed in units of volume (Q), weight (G) or mass (m), related to the unit of time.

### 2.2. Turbine waterfall

It is defined as the specific energy of fluid, which is expressed as the total energy (E) in relation to the weight of the fluid (G) or at the mass (m):

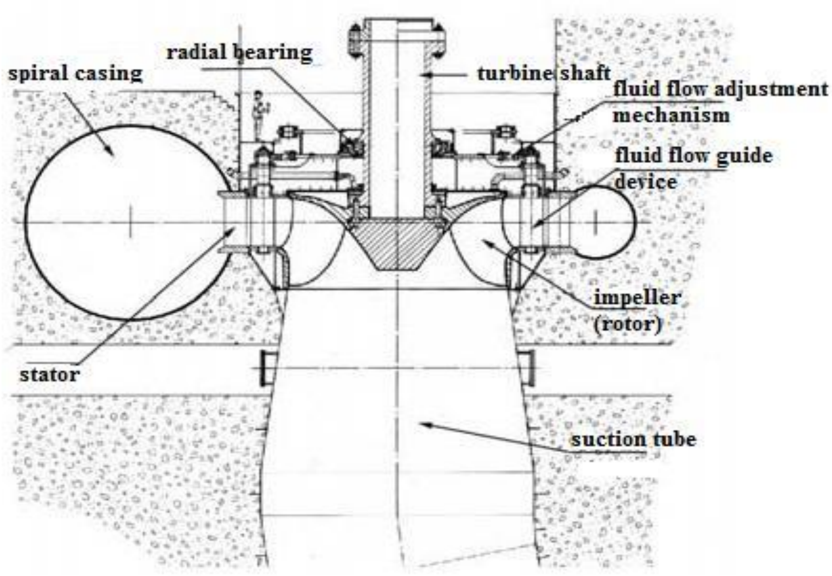


Fig. 1. Francis turbine component parts – axial section, [1]

$$e=\frac{E}{G}=\frac{p}{\gamma}+z+\frac{v^2}{2g} \quad [mcolH_2O] \qquad (1)$$

$$Y=\frac{E}{M}=\frac{p}{\rho}+g\,z+\frac{v^2}{2} \quad [J/N] \qquad (2)$$

Waterfall, because it expresses the energy consumed by the system to perform mechanical work, is defined as the difference between the specific energies at the entrance and exit of the system (system that can be the whole hydroelectric power plant or turbine only).

***2.2.1. Hydroelectric power plant gross waterfall***

The gross waterfall $H_b$ is defined as the difference between the energy specific flow of fluid from the upstream water storage and the downstream water drainage channel (figure 2):

$$H_b=e_{Am}-e_{Av}=\frac{p_{Am}-p_{Av}}{\gamma}+z_{Am}-z_{Av}+\frac{v_{Am}^2-v_{Av}^2}{2g} \qquad (3)$$

or

$$Y_b=Y_{Am}-Y_{Av}=\frac{p_{Am}-p_{Av}}{\rho}+g\left(z_{Am}-z_{Av}\right)+\frac{v_{Am}^2-v_{Av}^2}{2} \qquad (4)$$

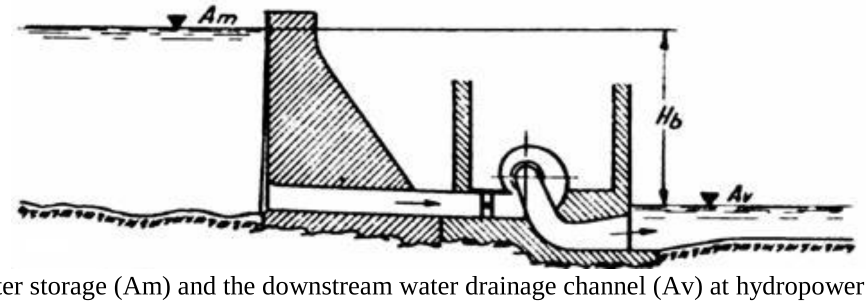


Fig. 2. Upstream water storage (Am) and the downstream water drainage channel (Av) at hydropower plant with turbine [1]

If we consider the pressures $p_{Am}=p_{Av}=p_{atm}$ they are equal each other and with atmospheric pressure the above relations become:

$$H_b=z_{Am}-z_{Av}\ ;\ Y_b=g\left(z_{Am}-z_{Av}\right) \qquad (5)$$

So, the gross waterfall is the vertical difference between the water levels in the dam lake and the escape channel of the hydroelectric power plant.

If we consider the section at the turbine inlet marked with i and at the output with e, the turbine waterfall is expressed as:

$$H=\frac{p_i-p_e}{\gamma}+z_i-z_e+\frac{v_i^2-v_e^2}{2g} \qquad (6)$$

or

$$Y=\frac{p_i-p_e}{\rho}+g\left(z_i-z_e\right)+\frac{v_i^2-v_e^2}{2} \qquad (7)$$

Because when the current passes through the dam supply system there are hydraulic leaks at the turbine

($\Sigma h_{pad}$) and at the outlet of the turbine the water has the energy ($e_e$=$\Sigma h_{pe}$) turbine waterfall is less than the gross waterfall:

$$H = H_b - \Sigma h_{pad} - \Sigma h_{pe} \qquad (8)$$

Based on the notations in figure 3, Francis turbine waterfall is given by the relation:

$$H = \left(z_i + a_i - z_e^*\right) + \frac{p_{mi}}{\gamma} + \frac{v_i^2 - v_e^2}{2g} \qquad (9)$$

where $p_{mi}$ represents the pressure indicated by the manometer in point i.

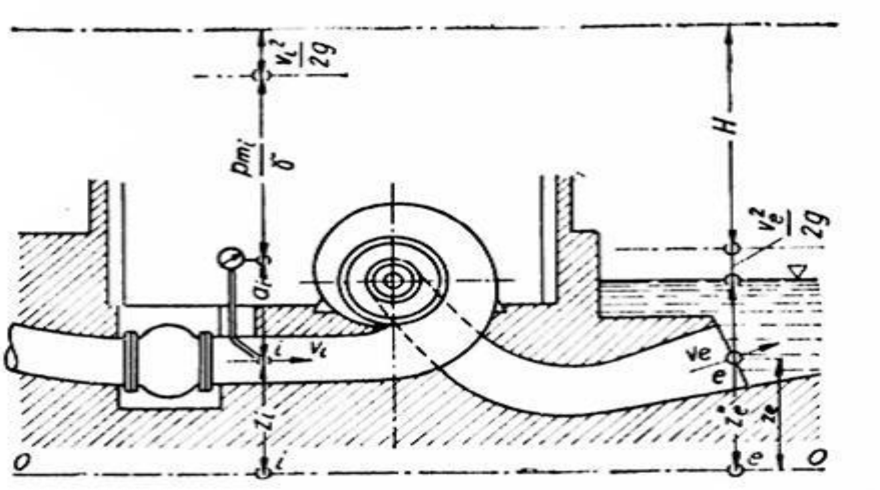


Fig. 3. Input and output parameters for the Francis turbine [1]

### *2.2.2. Effective turbine waterfall*

Effective turbine waterfall is the specific useful energy provided by turbine to shaft, after part of the turbine waterfall was spent for covering energy losses:

$$H_{ef} = H - \Sigma h_{pT} \qquad (10)$$

Often instead of hydraulic losses the term of hydraulic efficiency is defined such as:

$$\eta = \frac{H - \Sigma H_{pT}}{H} = \frac{H_u}{H} \qquad (11)$$

### 2.3. Turbine power and efficiency

The power of a turbine means the knowledge of several terms:

- $P_{am}$ - the power of the hydroelectric power plant;
- $P_a$ - hydraulic power or power absorbed by the turbine;
- $P_u$ - effective power obtained from the hydroelectric power plant.

$$P_{am} = \gamma\, Q\, H_b \quad ; \quad P_a = P_h = \gamma\, Q\, H \quad ; \quad P_u = \gamma\, Q\, H_u \qquad (12)$$

### *2.3.1. Turbine efficiency*

Performance or efficiency is generally understood to be the ratio between effective energy supplied by the hydroelectric power plant and energy absorbed it [4]:

$$\eta = \frac{P}{P_a} = \frac{P}{\gamma Q H} = \frac{M\,\omega}{\gamma Q H} \qquad (13)$$

where M represent the turbine shaft torque and ω the value of angular velocity.

### 2.4. Geometric suction height

This parameter, directly related to the cavitation behaviour of the turbine, has a great influence on the turbine' operation, both in terms of efficiency as well as safety during operation.

Geometric suction height is defined as the vertical difference between a reference plane (the middle plane of the fluid flow guide device at the Francis turbine), and the plane of downstream water level (figure 4).

### 2.5. Coefficient cavitation

If the geometric suction height of the turbines exceeds a certain value (theoretically 10.33m) a severe depression is created at the exit of the turbine impeller, which facilitates the appearance of some areas filled with vapours. Thus, the turbine operates in cavitation. This is a complex physical process that occurs in a vapours of liquid in the area of high speeds and low pressures. The primary effects of cavitation are [5]:

- loud noises and vibrations;
- intense material damage of turbine flow parts;
- decreased turbine efficiency.

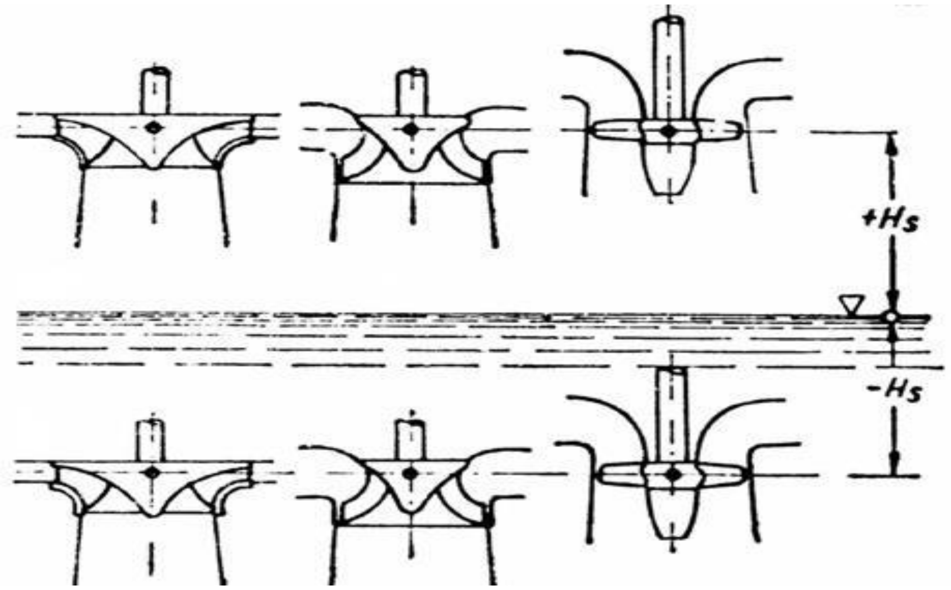

Fig. 4. The middle plane of the fluid flow guide device at the Francis turbine [1]

These manifestations intensify when a certain regime is exceeded called the limit. The cavitation coefficient is expressed as:

$$\sigma = \frac{A - A_t \mp H_s}{H} \qquad (14)$$

where: A - represents the atmospheric pressure (mcol $H_2O$), $A_t$ - the pressure of water vaporization (mcol $H_2O$). Knowing the value of the cavitation coefficient allows the calculation of the maximum suction height ($H_s$), respectively the mounting conditions of the turbine:

$$H_{s\,adm} \leq A - A_t - \frac{h}{900} - \sigma H \qquad (15)$$

where h represents the altitude of the hydroelectric power plant.

## 3. THE CAD MODELS OF SPIRAL TURBINE CASING AND IMPELLER

The NX CAD models for the hydraulic components of the turbine were made based on the dimensions and geometry presented by some authors in their works [6]. The construction of the spiral turbine casing CAD model is focused on "Extrude" operations to make the flange inlet part and blade. The challenge part of the model is that one containing the spiral turbine casing part made by creating a "Spline" guide (figure 5) along which the spiral contour of the casing through which the fluid will pass was drawn using the "Swept" option.

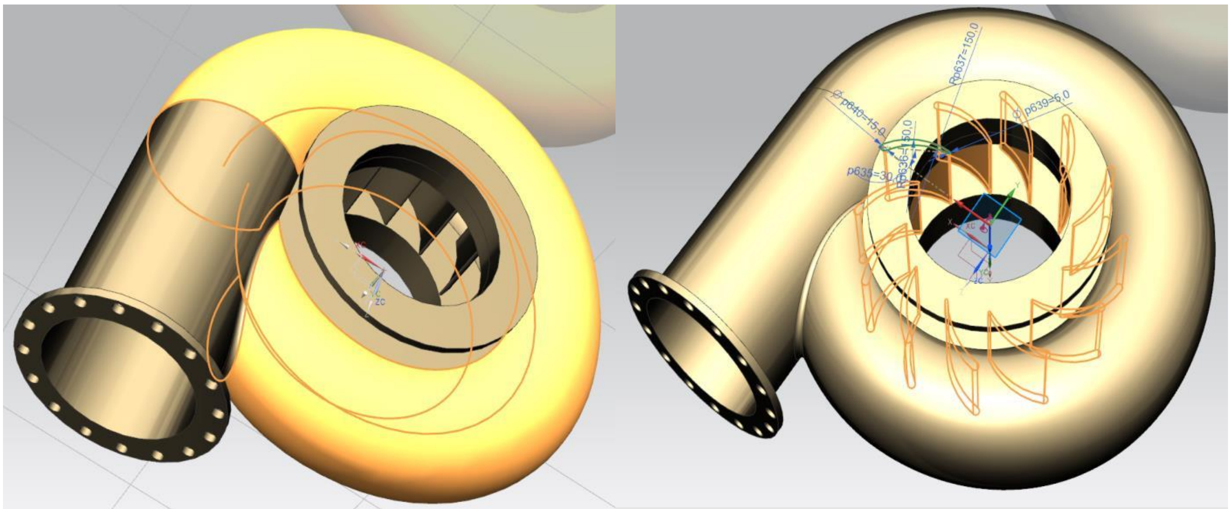

Fig. 5. Spiral turbine casing with spline guide curve, swept operation (orange) and stator blades

The construction of the stator blade started from a sketch that was later extruded (green in figure 5), next by multiplying the blade with "Pattern Circular" operation, a total of 12 blades were obtained.

The sketch of the figure 6 was used in the construction of the impeller, which was later subjected to a “Revolve” operation.
The impeller blade was made by drawing a “Curve surface” on the circular surface of the impeller (figure 7). Subsequently, it was subjected to a “Law extension” operation and to achieve the layer “Thicken” of the blade.
After making the blade (figure 7), the operation of cutting and adjusting the shape was applied using “Trimmed sheet”. By multiplying using the “Pattern circular” operation, a total of 10 blades were obtained [7].
The two 3D models were assembled using "Concentric" and "Center" constraints (figure 8).
At these two 3D models were joined by a diffuser (figure 8) that has a flange that is subjected to a “Concentric” type constraint operation in relation to the outlet flange of the spiral turbine casing.

## 4. COMPUTATIONAL FLUID DYNAMIC (CFD) ANALYSIS

The NX CAD assembly is imported into the “Design Modeler” from ANSYS CFX, the fluid domain is determined with the “Fill” option.
This fluid’ domain has been divided into five subdomains. The transition from one subdomain to another is done through interfaces. This was done because a subdomain of fluid will be rotated by the impeller blades and it is framed by other static subdomains thus creating a stratification of subdomains. This stratification helps a lot in applying the boundary conditions (figure 9).

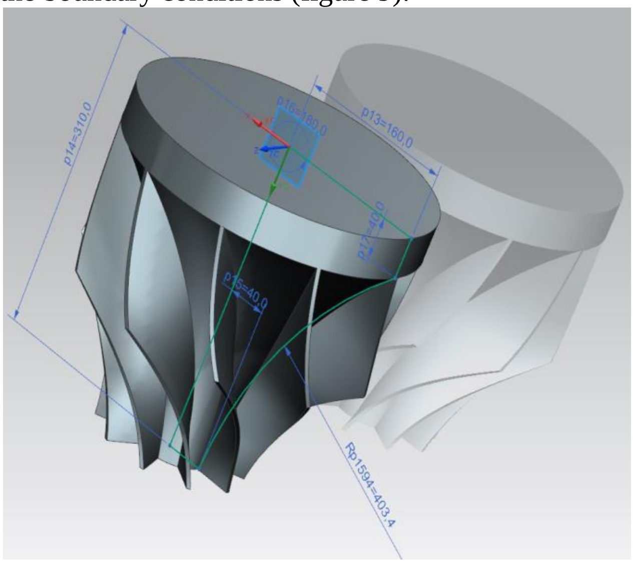


Fig. 6. “Revolve” of sketch (green) and dimensions

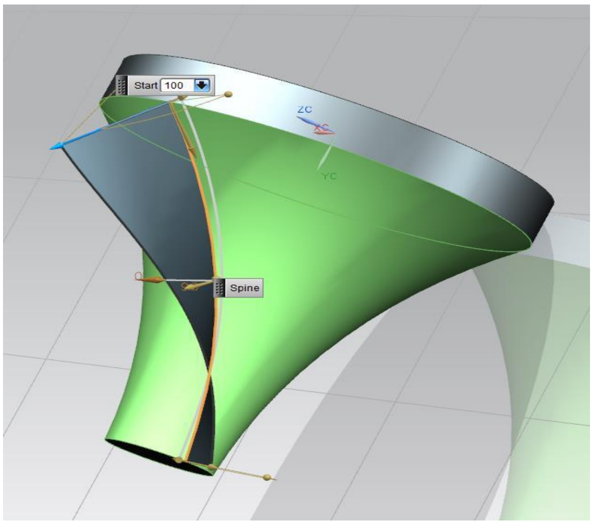


Fig. 7. “Curve surface” (orange) and “Law extension” operations

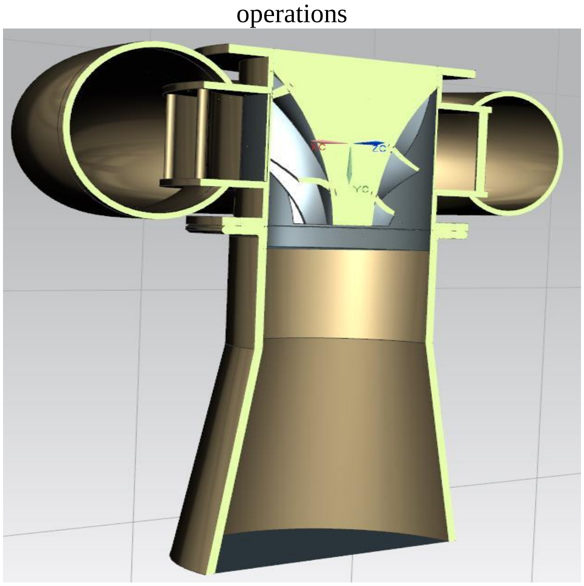

Fig. 8. Assembled NX CAD models (section viewing), spiral casing, impeller and diffuser

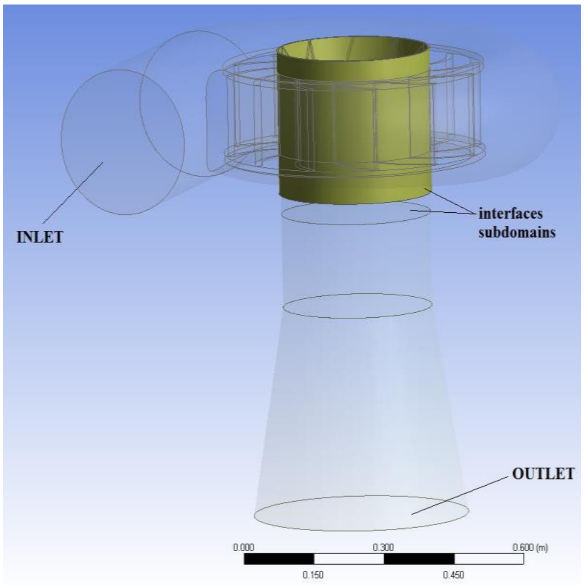


Fig. 9. Impeller subdomain (the rotating one) surrounded by two other static fluid subdomains, (one is coloured in green the other is transparent)

Through the discretization operation with finite volumes, 439938 nodes and 2251536 microvolumes were obtained (figure 10).

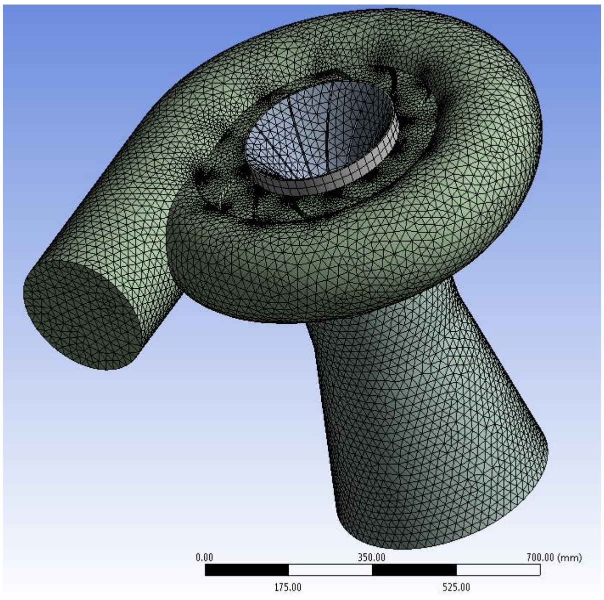


Fig. 10. Fluid subdomains volumes mesh

### 4.1. Boundary conditions

Water with 25°C its well-known properties and a vaporization pressure of 3170 Pa were used to model the flow. The “Bulk Flow Mass Rate” of the inlet fluid was set at 100 kg/s. The boundary type condition for outlet was set “Opening”, option “Opening pressure and dimension” with relative pressure equal with 30000 Pa. The cavitation model used is that of Reyleigh-Plesset one with the saturation pressure of 3170 Pa. The relative pressure of the model is set to zero.

When switching from one subdomain to another, boundary conditions of the interface type were used. When switching from a static subdomain to a rotating subdomain, the “Frozen Rotor” procedure was applied. The same procedure was then used for the transition interface from the rotating subdomain to the static subdomain [8].

It was considered that the impeller rotational speed is 100 rpm considered in relation to an Oy axis, the value being taken with the plus sign because the direction of rotation is clockwise direction.

Wall boundary condition, option “No slip wall” were also used on the liquid surfaces that come in contact with the impeller blades as well as with the walls at the fluid inlet and the spiral part of turbine casing.

## 5. RESULTS AND DISCUSSION

In figure 11 we notice that the water velocity (vertical section field) is high at the entrance on the blades from the fluid flow guide device (stator), this fact helping to set in motion the rotor blades. In the figure 12, the values of absolute pressure (vertical section field) are high on the spiral turbine casing walls but also on the fluid flow guide device blades (stator).

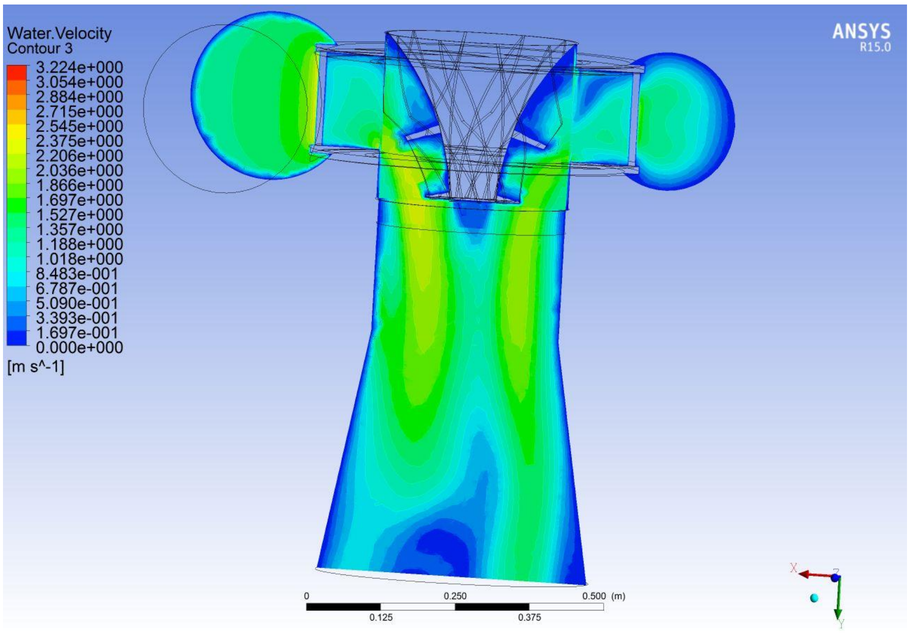


Fig. 11. Water velocity – vertical section field

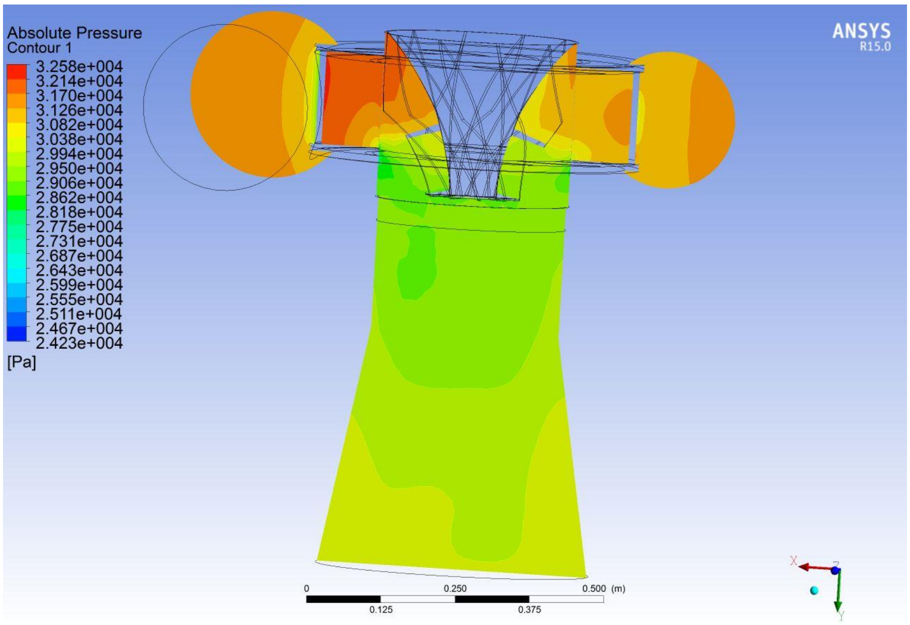

Fig. 12. Absolute pressure – vertical section field

The absolute pressure field (figures 13) show that the lowest pressure is (2.7e04÷2.9e04) Pa which is higher of the vaporization pressure (3170 Pa) that was taken into account to model the cavitation phenomenon. These low absolute pressures are located in the vicinity of the stator blades at the inlet of the liquid in the fluid flow guide device.

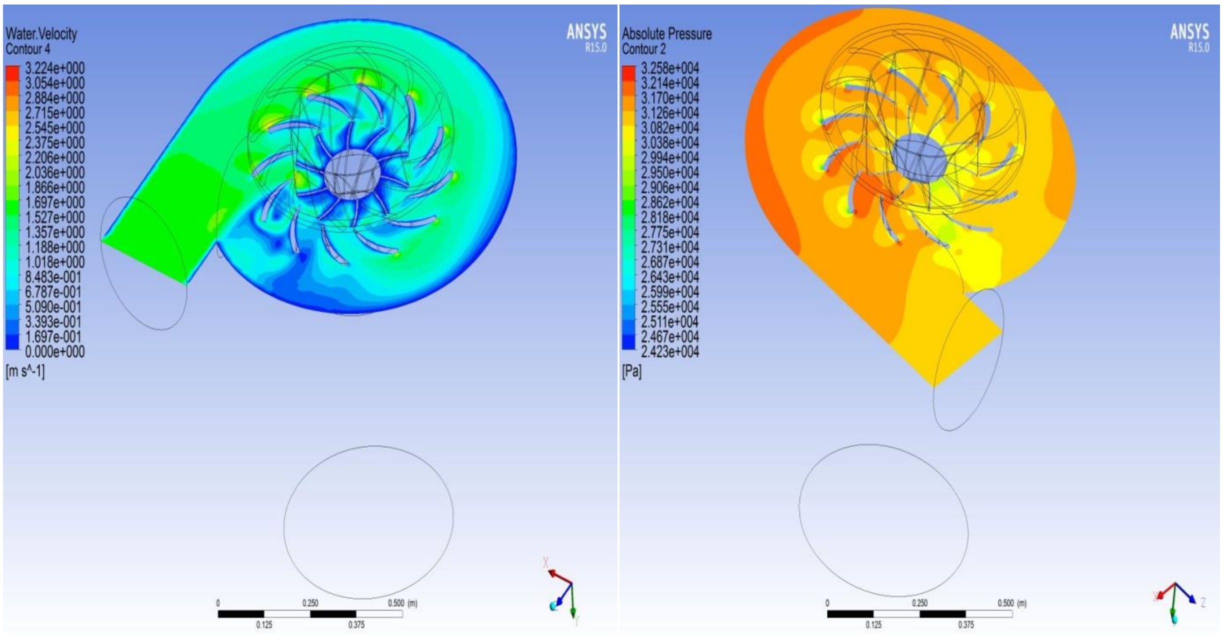

Fig. 13. Velocity and absolute pressure – orizontal section fields

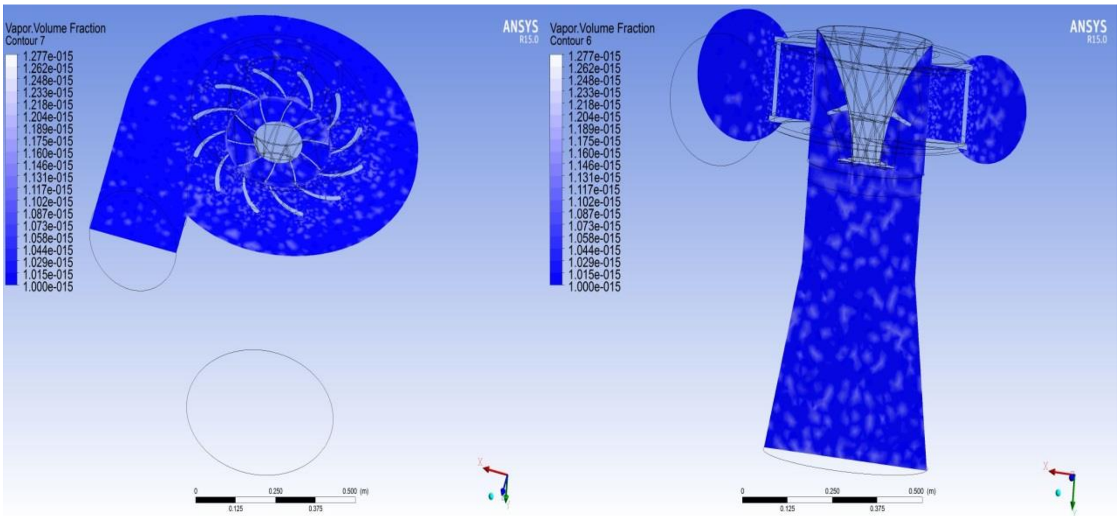

Fig. 14. Vapour Volume Fraction – horizontal and vertical section fields

## 6. CONCLUSIONS

Cavitation is a complex process that occurs in a stream of liquid in the area of high velocity and low pressures. The primary effects of cavitation are:

- loud noises and vibrations;
- intense damage to the material (spiral casing, fluid flow guide device, rotor blades)
- decreases turbine efficiency.

These manifestations intensify when a certain regime (limit) is exceeded. Figures 11, 12, and 13 show which are the areas with high liquid velocity but also the areas with lowest pressures, so areas susceptible to cavitation formation. "Vapour Volume Fraction" was considered to study this phenomenon.

In order to highlight the possibility of the formation of vapours, "Vapour Volume Fraction" can be drawn, light-coloured areas could favour the appearance of vapours (figure 14). These small areas ($1.27 \cdot 10^{-15}$) can be considered germs from which vapours can develop. Finally, given that the value of the minimum absolute pressure (2700-2900) Pa is higher than the value of the vapour pressure 3170Pa, the cavitation phenomenon does not occur, the turbine works in good condition. Vapour Volume Fraction may indicate areas with germs from which vapours may be produced.

## 7. REFERENCES


1. Padurean, I., Anton, I., (2001). *Turbine hidraulice Francis. Parametri fundamentali. Calcule de proiectare*. Eurostampa Publishing House, Timișoara.

2. Andritz (2022). *Hydro turbine animation*, Francis, Andritz Group, https://www.youtube.com/watch?v=qp3J1eRtKn0, accessed 10.02.2022.

3. Mauro, S., Lanzafame, R., S. Brusca, S., Messina, M., (2019). *Unsteady computational fluid dynamics analysis of the hydrodynamic instabilities in a reversible Francis turbine used in a storage plant*, Heliyon, 5(9), e02441.

4. Axinte, T., Nutu, C., et al., (2016). *Advanced analysis of the transverse bulkhead of the general cargo ship*, IOP Conference Series: Materials Science and Engineering, 145, 082005.

5. Eidenberg, P., (1961). *Cavitation*, Section 12, Handbook of Fluid Mechanics, pp. 121-128, McGraw Publishing House.

6. *#NX CAD #Francis spiral turbine casing #tutorial 13.0* – CAD CAM Solution, Meerut, https://www.youtube.com/watch?v=ZvyKOalpJKM&t=14s, accessed date: February 10, 2022

7. *#NX CAD #Turbine #Rotor Design #Impeller* – CAD CAM Solution, Meerut https://www.youtube.com/watch?v=jML6oioL2M0&t=19s, accessed date: February 15, 2022

8. Calimanescu I., Stan L.C., *CFD Optimization Study of a Centrifugal Pump in Cavitation*, Proceedings Volume 11718, Advanced Topics in Optoelectronics, Microelectronics and Nanotechnologies, X 117181W, 2020, DOI: 10.1117/12.2571239



Received: June 17, 2022 / Accepted: December 15, 2022 / Paper available online: December 20, 2022 © International Journal of Modern Manufacturing Technologies