Funded by the European Union

Hydraulic Turbines

Author: doc.dr. Adis Bubalo
Džemal Bijedić University of Mostar
Mašinski fakultet Mostar

Turbomachinery 15.04.2025

"Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union. Neither the European Union nor the granting authority can be."

Partnership for Promotion and Popularization of Electrical Mobility through Transformation and Modernization of WB HEIs Study Programs/PELMOB

Call: ERASMUS-EDU-2022-CBHE-STRAND-2
Project Number: 101082860

Hidraulične turbine

Hydro-electric power accounts for up to 20% of the world's electrical generation.

Hydraulic turbines come in a variety of shapes determined by the available head and a number of sizes determined by the flow rate through the device

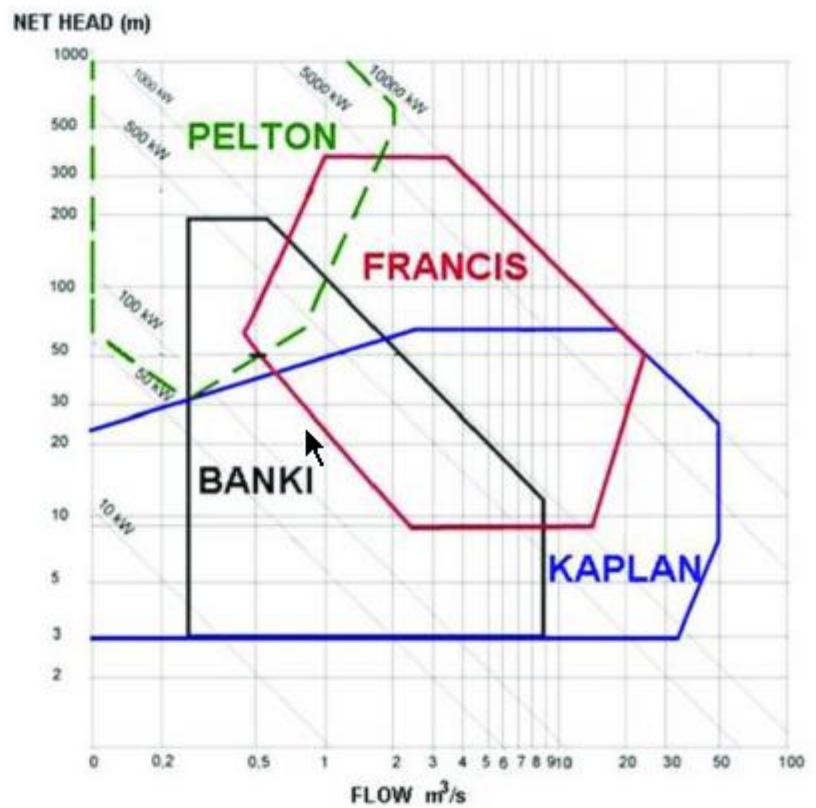
Three main type:

Pelton

Francis

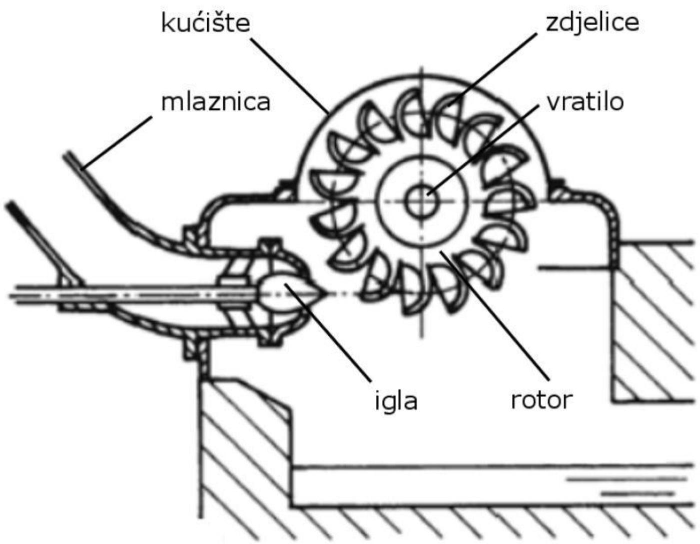
Kaplan

Other types are variation of main types

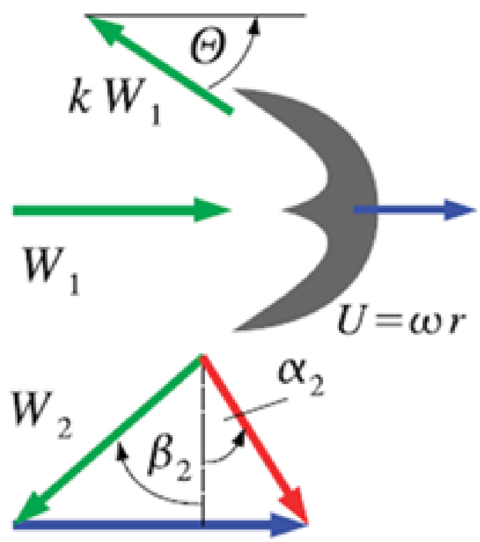


Pelton turbine

- Operation range for heads 300 do 4000 m.
- Efficiency over 90 %
- Wide operation range, even on very low discharge



Presjek Peltonove turbine s osnovnim konstruktivnim dijelovima.



<u>Flow analyse of Pelton runner</u> is straightforward and can be found in any basic fluid mechanics as a simple control volume can be drawn around the rotor blade and the force determined from a simple analysis.
Alternatively the results can be obtained from the Euler equation as follows.

$$P = \dot{m}\omega(V_{2\theta}r_2 - V_{1\theta}r_1)$$

so the key parameters to work out are the inlet tangential velocity and the exit tangential velocity..

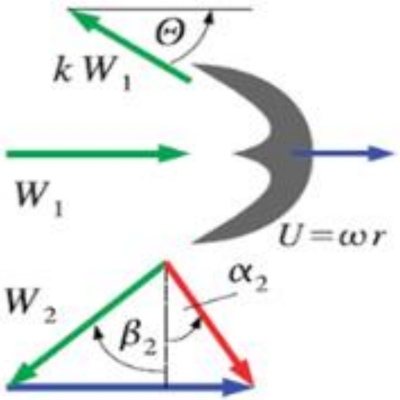
At inlet the situation is straightforward $V_{1\theta} = V_j$ the jet velocity, as the nozzle directs the flow only in the tangential direction.

At exit the velocity triangle shown in Figure must be used.

Note that: $W_1 = W_j - U$ so $W_2 = kW_1 = k(V_j - U)$ where ***k*** is an empirical coefficient for frictional losses in the bucket.

From the velocity triangle: $V_{2\theta} = U + W_{2\theta}$

$$W_{2\theta} = -W_2 \cos\Theta$$



$$W_{2\theta} = -W_2 \cos\Theta$$

where Θ is the bucket angle shown in Figure angle made by the bucket

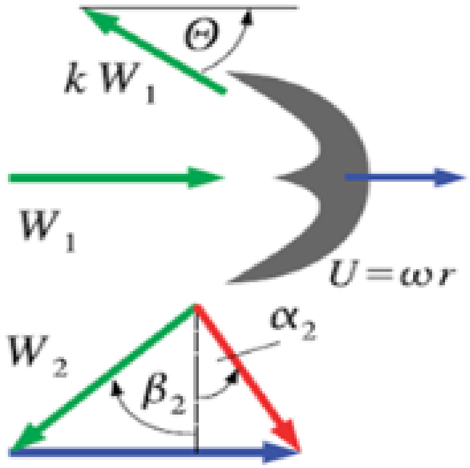
The negative sign arises as our sign convention is "positive in the direction of rotation" Substituting yields:

$$V_{2\theta} = U - W_2 \cos\Theta = U - k(V_j - U)\cos\Theta = -kv_j \cos\Theta + U(1 + k\cos\Theta)$$

Substitute this into the expression for power:

$$P = \dot{m}U(V_j - U)(1 + k\cos\Theta)$$

In the Pelton wheel all the pressure drop occurs in the stator or nozzle so the machine can be classified as an **impulse machine**.



Funded by the European Union

The analysis of the Pelton wheel is extremely straightforward, other hydraulic machines required a more nuanced approach. This is a three-step process:

1. Given the flow rate apply the continuity equation to get the radial or axial velocity.
2. The geometry (blade angles and radii) will yield absolute and relative velocities by means of velocity triangles.
3. Power is obtained from the Euler Equation.

Empirically obtained values of efficiency (or estimated values based on prior experience) link the power produced and the head drop across the machine. There are two methods of doing this depending of what information is available

Analysis Approach

here are two methods of doing this depending of available information:

If the total head is given this yields the ideal power:

$$P_{ideal} = \rho \dot{Q} g H = \dot{m} g H$$

iven the ideal power, we have the actual power (from the Euler equation above) this yields the efficiency.

If loss information is given, the actual head is obtained from:

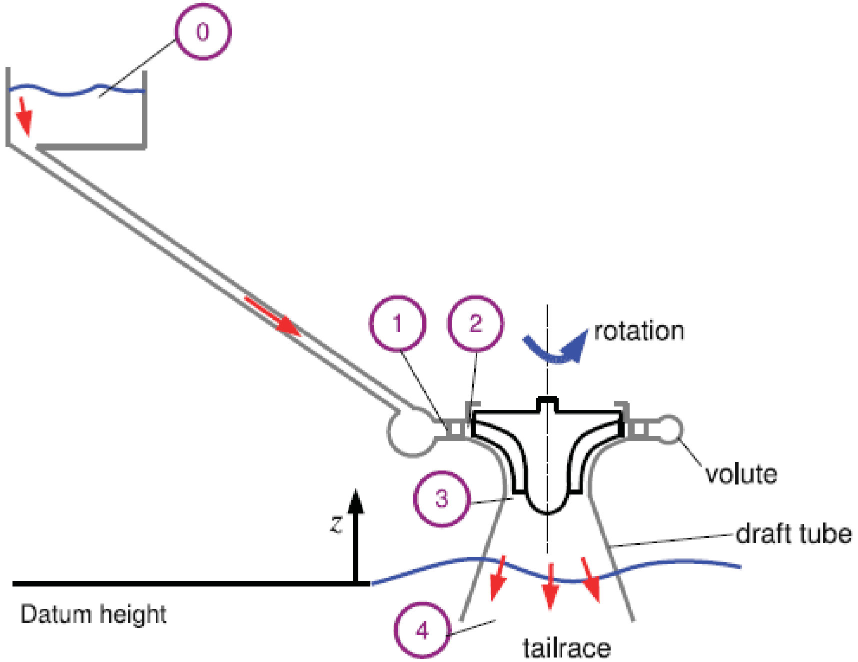
$$H_{actual} = \frac{P_{actual}}{\rho \dot{Q} g} + H_{loss}$$

The ideal head is given by : Hactual – Hloss so given the actual power and the
losses efficiency can be obtained

Flow analyse through Francis turbine

1. Inlet to Guide Vanes. There may be a small amount of swirl at inlet to the guide vanes.
2. Exit from Guide Vanes. Flow is deflected by the guide vanes (stator) to give high swirl velocity, and thus high angular momentum

3. Exit from Runner. The aim here is to design at exit for zero absolute swirl as otherwise kinetic energy is wasted.
4. Draft Tube. (Station 4) This diffuser allows recovery of some of the exit kinetic energy of the axial flow.

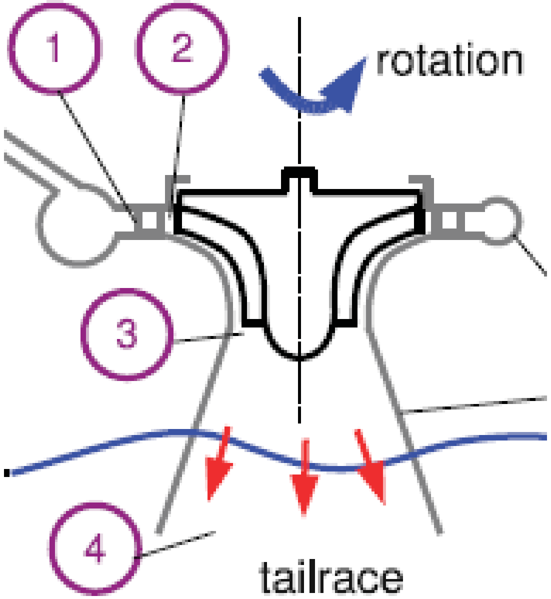


Francis turbina

Flow from station to 1 station 2 through guide vanes is radially inwards, while being deflected and gaining angular momentum.

First we apply continuity: Flow rate: Q1 = Q2

$$Q_2 = V_{2r} 2\pi r_2 b_2 \text{ and } Q_1 = V_{1r} 2\pi r_1 b_1$$



If b1= b2 (constant widthe vanes) then

$V_{2r} r2 = V_{1r} r1$

Now the guide vanes have a non-zero thickness, and this introduces a blockage t at station 2 so to allow for this:

$$Q_2 = 2\pi r_2 b_2 (1 - t) V_{2r}$$

Usually t = 0,08 or about 8% of the area is blocked by the inlet guide vanes.

Francis turbine

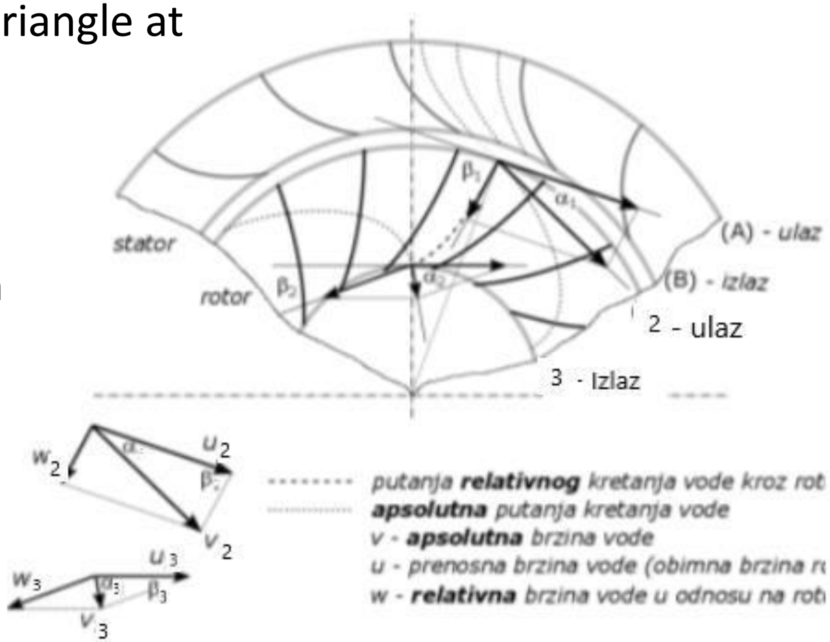
The detailed analysis process is as follows:

From the values Vr2, the flow angle at exit from the vanes , and the blade speed construct the velocity triangle at vane exit, 2

Usually, the design is for no exit swirl to reduce the exit kinetic energy that is lost, V3Ɵ = 0, thus from the blade speed ωR3, the velocity triangle at runner exit, 3 can be constructed.

The runner inlet and exit relative flow angles (to give the geometry of the runner blades) are obtained and also the power from the change in angular momentum across the runner

Finally, to get the efficiency, the height of the reservoir and information on losses is required.



Francis turbin runner‘s design

Example of calculation

Consider a machine with the following specification:

Outlet runner diameter, $2_{r2} = 2$ m

No of rotation, N = 200 rev/min

Guide vane height, $b_2 = 0.3$ m

Vane blockaget=0.08

Vane exit angle $\alpha_2 = 75^\circ$

Impeller designed for axial flow at exit, $\alpha_3 = 0$, and $r_{3m} = 0.5\ m;\ b_3 = 0.4\ m$

Head, H_o=63m, Flow Q = 12 m3 / s

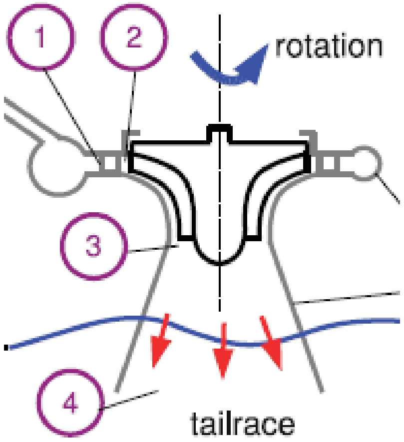
Flow losses : 2 m head loss intake pipe, 0.5 m draft tube loss

Draft tube velocity 4 m/s

Determine the relative angles at inlet and exit of runner to give a preliminary design for the runner geometry. Also find the power output.

Solution

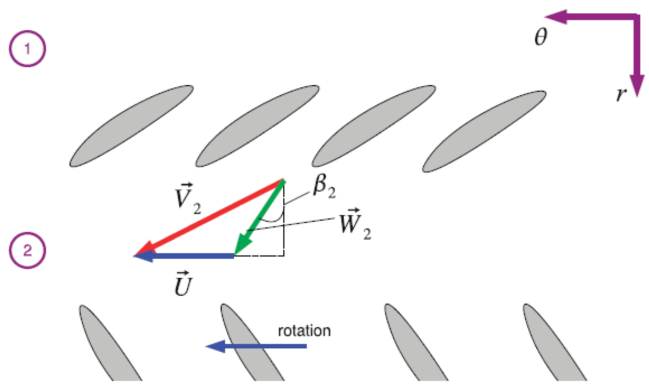
The strategy is to apply the analysis steps outlined earlier. The most part is drawing the velocity triangles at station 2 and station 3.



At exit from guide vanes , station 2:

$$Q = 2\pi r_2 b_2 (1 - t) V_{2r} \implies 12 = 2\pi \times 1.0 \times 0.3(1 - 0.08) V_{r2}$$

Working the numbers though yields: V2r = 6,92 m/s



Velocity triangles at point 2

To determine power the Euler equation only requires values at inlet and exit of the turbine so a sketch at inlet and exit will suffice.

In reality the turbine blade curves around into the axial direction which can be represented using CAD for example but is not easy in a sketch. The velocity triangle is in the tangential radial plane

$$V_{2\theta} = V_{2r} \tan \alpha_2 = 6.92 \tan 75^\circ \implies V_{2\theta} = 25.83\, m/s$$

$$V_2 = \sqrt{V_{2r}^2 + V_{2\theta}^2} = \sqrt{25.83^2 + 6.92^2} = 26.74\, m/s$$

From velocity triangles:

$$V_{2\theta} = W_{2\theta} + \omega r_2$$

$$\omega r_2 = 2\pi \times \frac{200}{60} \times 1.0 = 20.94\, m/s$$

$$\Rightarrow W_{2\theta} = V_{2\theta} - \omega r_2 = 25.83 - 20.94 = 4.89\, m/s$$

So we can now work out the relative flow angle at 2::

$$\tan \beta_2 = \frac{W_{2\theta}}{V_{2r}} \implies \beta_2 = \tan^{-1}\left(\frac{4.89}{6.92}\right) = 35.2^\circ$$

The velocity of a jet expanding is $\sqrt{2gH}$ For this machine is $V_2/\sqrt{2gH} = 0.76$ so not all the acceleration of the fluid occurs over the stator so the reaction R > 0 and the machine is **not an impulse turbine.**

CONTINUITY AT POINT 3

The flow area is given by: $2\pi r_{3m} b_3$ so: ⟹ $Q = 2\pi r_{3m} b_3 V_{3x}$

Thus: $V_{3x} = 12/\left(2\pi \times 0.5 \times 0.4\right) = 9.55\, m/s$

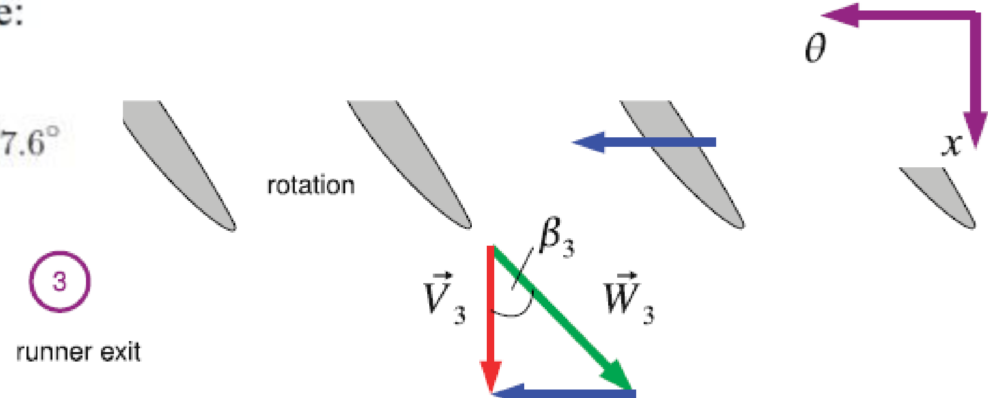
VELOCITY TRIANGLE AT POINT 3

No exit swirl $V_{3\theta} = 0$

$$\therefore W_{3\theta} = -\omega r_{3m} = -2\pi \times \frac{200}{60} \times 0.5 = -10.47\, m/s$$

So we can now work out the relative flow angle:

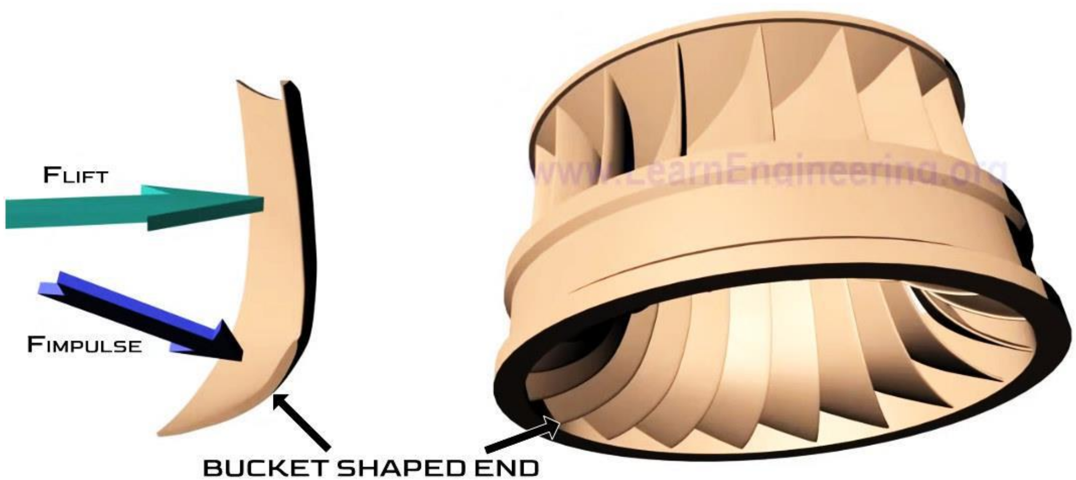
$$\tan\beta_3 = \frac{W_{3\theta}}{V_{3x}} \implies \beta_3 = \tan^{-1}\left(\frac{-10.47}{9.549}\right) = -47.6^\circ$$

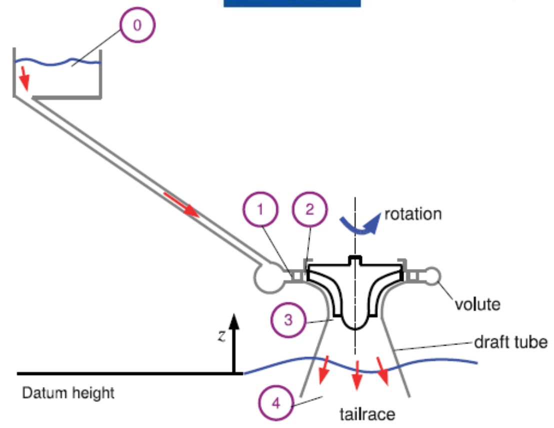


Power output: Recall that:

$$P = \dot{m}\omega\left(r_2 V_{2\theta} - r_{3m} V_{\theta 3}\right) \quad \text{kako je} \quad V_{3\theta} = 0$$

$$P = \dot{m}\omega r_2 V_{2\theta} = 1000 \times 12 \times 2\pi \times \frac{200}{60} \times 25.82 = 6.49\ MW$$

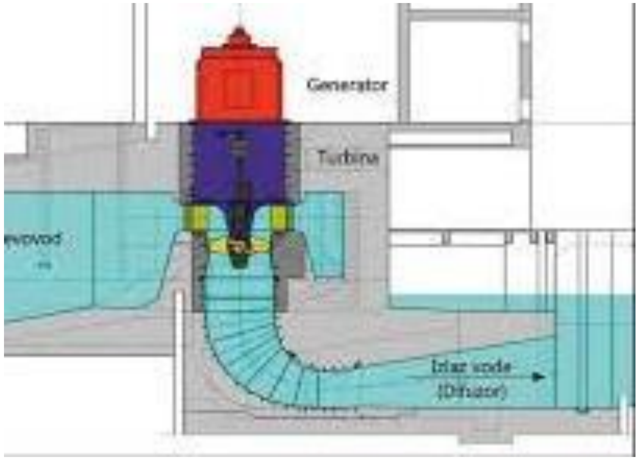




Draft tube analyse

The draft tube is a conical diffuser with around 7◦ divergence which reduces the exit kinetic energy in the departing fluid and therefore increases the efficiency of the machine as a whole.

In this example the aim is to reduce exit kinetic energy at station 4, note that h4 is normally below
the height of the river that the flow exits into.



Total head at station „0“ is:

$$\frac{p_o}{\rho g} + \frac{V_o^2}{2g} + z_o$$

But po = 0 as in hydraulics we always use gauge pressure where zero is the ambient atmospheric pressure, in the reservoir V0 = 0 so the total head at station zero is simply given by H0 = z0

We need to find the head across turbine: $\Delta H = H_1 - H_3$

$H_1 = H_o - h_{fp}$ where h_{fp} is the pipe head loss due to friction

Now examine the total head from the other end of the machine starting **at station 4.**

If loss in draft tube is given by h_{fDT} $H_4 = H_3 - h_{fDT} \implies H_3 = H_4 + h_{fDT}$

The total head at station 4 is given by

$$H_4 = \frac{p_4}{\rho g} + \frac{V_4^2}{2g} + z_4$$

The pressure at station 4 must be equal to the pressure of the fluid in the river as a whole so is given by the pressure due to a height of fluid: $p_4 = \rho g h_4.$ $z_4 = -h_4$
substituting gives:

$$H_4 = \frac{\rho g h_4}{\rho g} + \frac{V_4^2}{2g} - h_4 = \frac{V_4^2}{2g}$$

Thus

$$H_3 = h_{fDT} + \frac{V_4^2}{2g}$$

So head across turbine: $H_1 - H_3 = H_o - \frac{V_4^2}{2g} - h_{fp} - h_{fDT}$

simplifying slightly: $H_1 - H_3 = H_o - h_{fp} - \left(h_{fDT} + \frac{V_4^2}{2g}\right)$

So for maximum head change across turbine and therefore the maximum power the design aim is to keep V_4 , h_{fp}, h_{fDT} **as low as posible.**

To apply this information on pipe flow losses and draft tube losses is required and the velocity at station 4. In this example the information is provided: :

$$h_{fp} = 2\,m, h_{fDT} = 0.5\,m, V_4 = 4\,m/s.$$

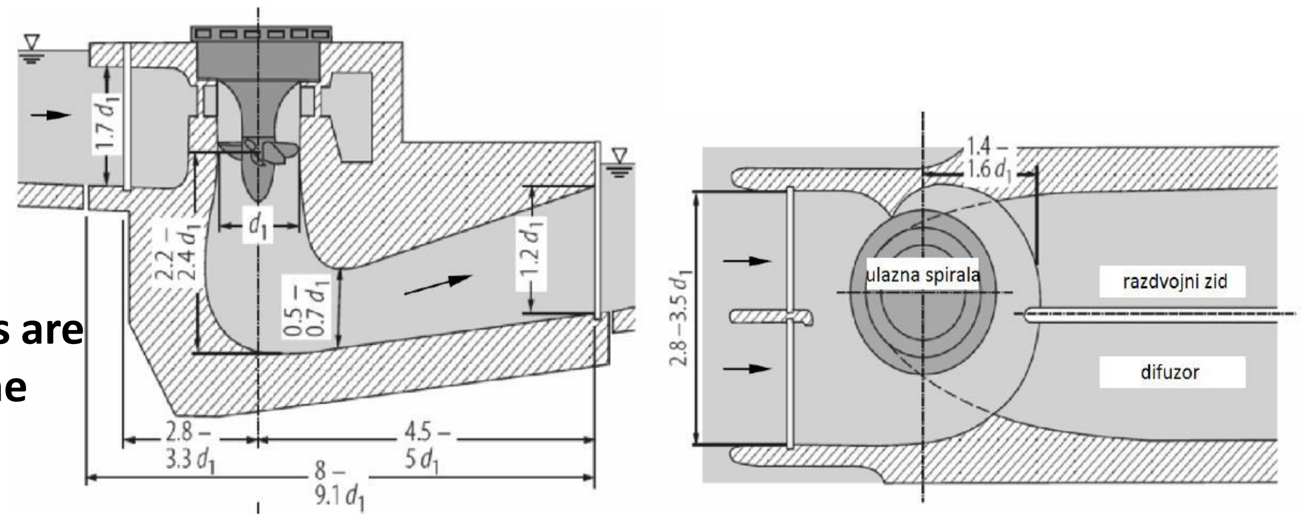
In this example $V_3 = V_{3x} = 9.55$ m/s, so the exit velocity is considerably reduced by the draft tube

$$H_1 - H_3 = 63 - \frac{4^2}{2g} - 2 - 0.5 = 59.68\,m$$

The turbine efficiency may then be worked out:

$$\eta_T = \frac{P_{actual}}{P_{ideal}} = \frac{6.49 \times 10^6}{1000 \times 12 \times 59.68 \times 9.81} = 0.92$$

DIFUZORI



Draft tubes or suction pipes are pipe structures placed at the outlet of reaction turbines that have the role of:

- they lead water from the turbine towards the outlet structure, i.e. the lower reservoir,
- insulate the flow of water upon exiting the runner from atmospheric pressure,
- ensure the utilization of the full water head (height difference between the level of the free surface of the upper and lower reservoirs) regardless of the position of the turbine,
- reduce the velocity of the water at the otlet of turrbine to reduce energy losses,
- reduce the pressure at the very exit from the runner, so that the power delivered to the turbine is as high as possible.

DIFUZORI

- The main constructive feature of the draft tube is the increase in cross-sectional area in the direction of water movement.

- Upon exiting the rotor, the water passes through a diffuser so that it is not exposed to atmospheric pressure. Otherwise, it would possess relatively high kinetic energy, which would remain unused. The usable head of water would be reduced, determined by the water level in the upper reservoir and the level of the turbine outlet.

- Thanks to the diffuser, the turbine can be placed above the bottom water level, while the water is led further towards the lower reservoir in a closed path. Thus, the entire water head in between is used for energy conversion on the turbine upper and lower tank and the efficiency of the plant is not reduced.

- Another advantage is achieved by using a diffuser: higher water flow speeds are achieved at the rotor outlet. This will not lead to large energy losses, because the water in the diffuser slows down, and by choosing a higher speed at the rotor outlet with the same given flow rate, it is possible to use a smaller turbine diameter.

CFD simulation – turbine parts preprocessing