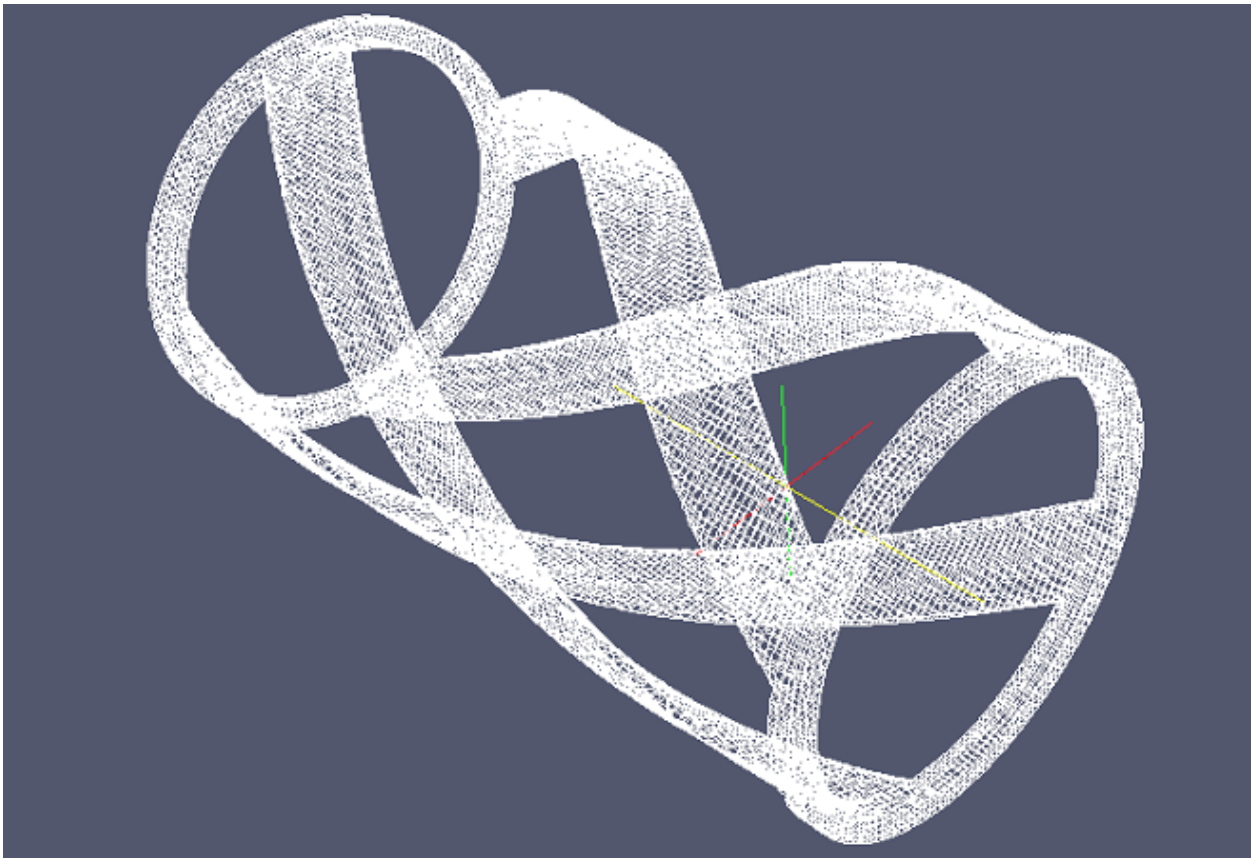
Society of Naval Architects and Marine Engineers **Propeller & Shafting Conference 2023**
sname.org **Norfolk VA USA**

Research of the Vortex Propulsor Concept

Richard Hayman (M), Ibrahim Mesinovic (V)*, Dijana Basta (V)**
* Swedish Society of Marine Engineers
** Aeronautical Institute of Croatia



Abstract:

Recent research in the possible applications of the vortex propulsor – Vor-Prop - have included open cylinder model, a shafted coaming contained compact model and a double counter rotational internal turbine model.

CFD analysis indicates efficiencies in applications including ship propulsion, UAV propulsion, industrial and agricultural pumping, and medical devices. Prototypes and testing have been pursued at MARIN, Netherlands, and Swedish laboratories.

RESEARCH MODELS:

The principle of the vortex propulsor is to create pressure differentials by spinning helical foils arrays, with a low center vortex drawing water into the cylinder and providing flow thrust. This infers a large volume of water being drawn through the center passage. The effectiveness of this concept and its applications are still to be determined in various designs.

The original hydraulic principle was developed by Alexander Gorlov for electrical generation captured by helix foil array in tidal and river flow sites.

WATER TURBINE GENERATION:

Helix hydrofoil blade array turbines are a proven technology as currently deployed in tidal & river flows. The advantage is in low flow generation versus comparable screw propeller water turbines.



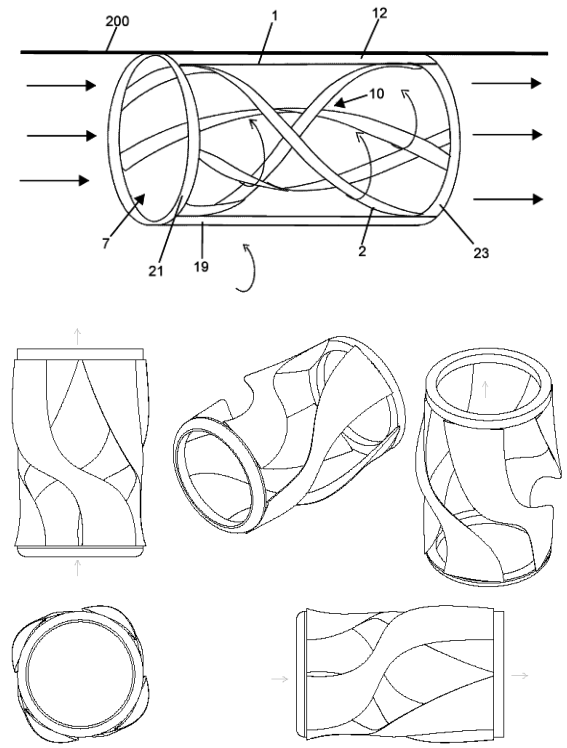
This installation, funded by the US Dept. of Energy is in the Coobscook Bay, Maine.

Our subsequent research to turn this energy capture design into propulsion has indicated efficiencies leading to various models and testing. Whether the 'tangential' velocity can be used as 'axial' velocity is the crux of the investigation.

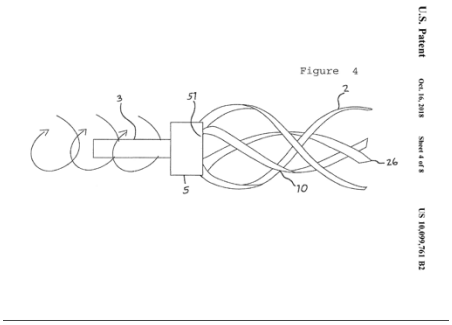
The possible use of Vor-Prop for marine propulsion leads to shaft mounted versions for comparison to standard screw prop, Sharrow style and other evolved propeller designs. It may be unlikely that any Vor-Prop would be useful in high rpm high speed propulsion. There may be use in low rpm 'service' speed for vessels of various sizes.

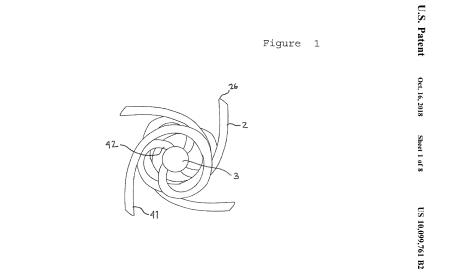
The patent document describes possible use of Vor-Props integrated in hulls as large cylindrical pumps to draw and expel large water volumes to supplement any aft propellers.

A U.S. Patent #10,099,761 was issued in 2018 under the title "Water Turbine Propeller".



Patent drawings: Cylindrical:



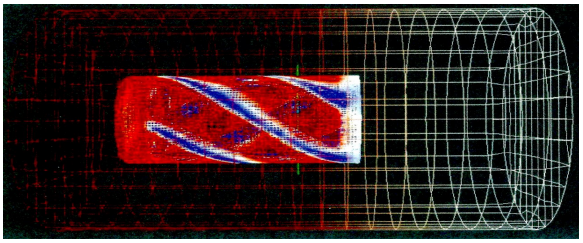


Patent drawings: Shaft Mount

The name, Vor-Prop, has been registered as trademark and research sponsored by Vor-Prop LLC.

COMPUTER MODELS:

Simulations were realized by Joe Baroutey, Blake Koop and Matt Sinott of Computing Future Solutions using pressure isocontours and vorticity profiles on the open center cylindrical Vor-Prop.



Computer hydro flow simulation

Computer hydro flow simulation and analysis:

Simulated Volume	600mm x 1020mm x 600mm
Blade Thickness	50.0mm
Blade Height	12.5mm

Initial pressure differentials:

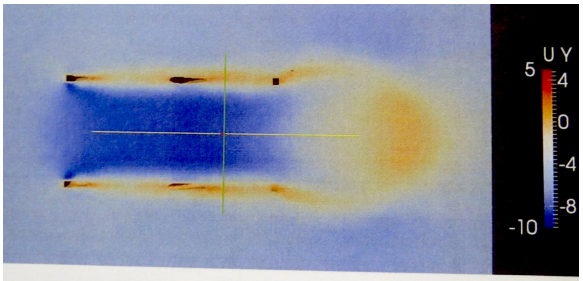


Figure 13: Vor-Prop (wide blade) Axial Velocity Profile

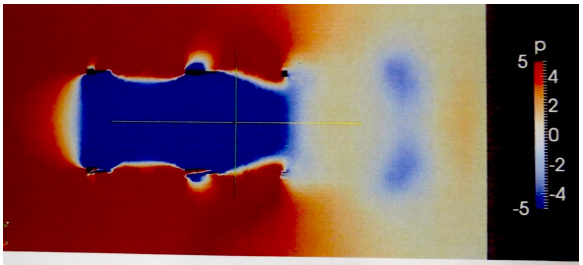


Figure 14: Vor-Prop (wide blade) Pressure Gradient

This indicates continuous flow pressure effect.

SIMULATION SUMMARY:

Recommendations:

1 Increasing the blade width added structure to the flow pattern and focused the velocity profile. It may be beneficial to simulate the wide blade model further, as the pressure gradient looks as though it is still developing. Widening the blade further and adding those results to the comparison would be helpful as well.

2. The velocity profile is a strong indicator of the effectiveness of the Vor-Prop. Conservation of momentum in this system requires that the forward momentum of any marine vessel be equal and opposite to the momentum of the water displaced by the Vor-Prop. Focusing this velocity profile (and therefore momentum) should be the goal when altering design parameters.

3. Another strong indicator is the pressure difference on the blades themselves. Notice in the high pressure and low pressure regions on either side of the Vor-Prop blades. This is a very direct visualization of the net force which creates motion.

APPLICATIONS:

If this hydraulic principle is proven and utilized, the Vor-Prop may have applications in marine propulsion in various sizes from small craft to large ships. An installation integrated into hull construction may be most effective.

The goal is a propulsor with low rpm moving large water volumes with high torque, no cavitization and a more efficient flow in the aft wash. The result could be comparable to marine animal propulsion such as squid, taking in large volumes of water to expel and move forward. The greater the flow through, the more efficient the propulsor would be. There would be an optimum foil design, cylinder diameter and length calculated to the application.

Manufacturing of the Vor-Prop may prove more efficient as non-metal foils could be extruded and assembled and installed as per the vessel requirements.

The size and length of cylinder and the number and position of foil blades will be the determinate of efficacy as prototypes will determine.

Small vessel engines could have fewer foil blades. Larger versions for ships may realize more blades and different power delivery attachment.

Mechanical linkage or electrical induction would drive the Vor-Prop as per vessel design.

An air cushion or sealed oil bearing system for a larger Vor-Prop rotating cylinder could mitigate friction in the system.

INSTALLATION:

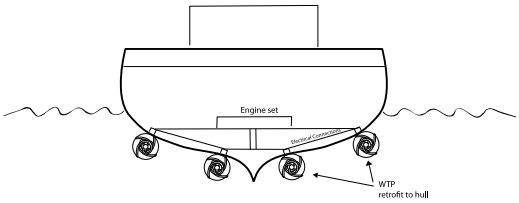
A larger Vor-Prop configuration could be built integrated or attached to a vessel hull.

The Vor-Prop may be most effective at slow speed and for large vessel propulsion with low rpm diesel and electric engine applications. A moderate engine speed will be matched by greater torque at a lower relative rpm than existing screw propellers. High rpm parameters will also be tested for small craft installation.

If Vor-Prop rotation of 3 or 4 times hull speed may match current speed levels and yield a significant reduction in fuel costs, the design may develop into an alternative marine propulsion system.

Questions of rudder placement or direction reversibility are yet to be studied.

The general description of the Vor-Prop for patent filings includes both hull integrated and fitted onto standard propeller shafts. As ongoing research follows the demonstrable effects of the design - variables of size, number of foils blades, rotational speed and flow dynamics - the optimization and variations of the design will indicate its applications.



Monohull external Installation

Application example: A swath hull design (small water area twin hull) could install the Vor-Prop into the subsurface hulls with intake openings and smooth propulsive flow from the ends.



SWATH vessel with screw propellers

Additional applications of the Vor-Prop may be found in industrial pumps, irrigation systems and medical devices. Comparative tests with various hydraulic pump units are yet to be made.

BENEFITS:

The advantages of Vor-Prop for ocean sustainability include greater fuel efficiency to reduce pollution, reduction of noise and vibrations.

ENVIRONMENTAL:

Vor-Prop would be less injurious to marine life than faster churning screw propellers. It would also reduce ‘prop chop’ of marine life as the helix would create its own exterior buffer pressure, as has been demonstrated in Gorlov generators. Any life drawn into the flow would be turned about and sent through.

SPECULATIONS:

Discussions with other design engineers include:

Agricultural Vor-Prop pumps with low pressure and high volumes.

Medical devices utilizing implanted micro Vor-Prop pumps in cardiac and gastrointestinal implants to lessen damage to cell structures.

Large scale Vor-props installed to propel and position large barges in offshore sites in place of ocean tug boats.

Larger scale Vor-Props to divert river flows for collecting plastic debris.

PROTOTYPE TEST MODELS

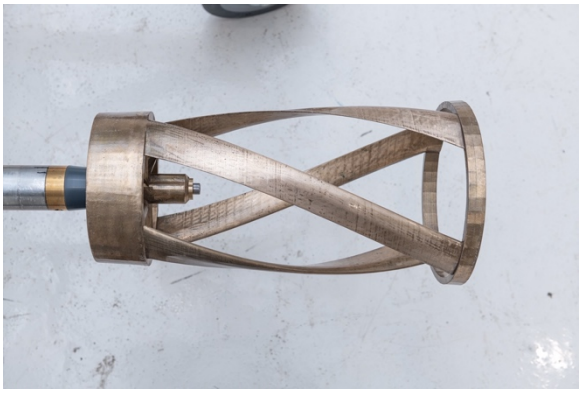
These are the recent prototype test models:

MARIN, Netherlands



MARIN Model

This model was cast in bronze per the specifications of the test equipment at MARIN. The aim was to measure any effect of the helical foils. For shaft mounting, a ring with blade struts was added to the cylinder array.

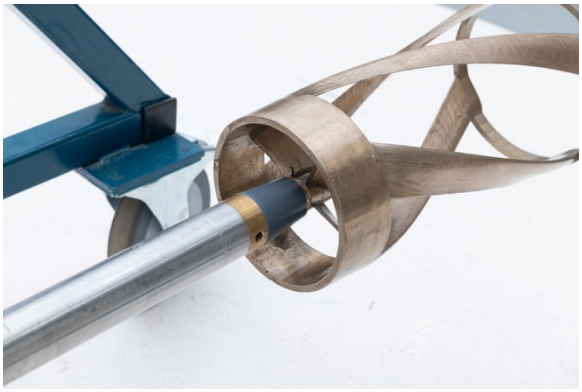


MARIN Model



MARIN Model

This model would have had a fraction of the weight had it been made from aviation ABS or aluminum.. The MARIN laboratory did not accept these materials as unstable for testing.



MARIN Model

Due to the extended weight of the helical array, the model produced imbalanced motion vibrations and stressed the sensors imbedded in the mounting. Proposed rpm of 1500 was stopped at 700 rpm due to this vibration.



MARIN Model

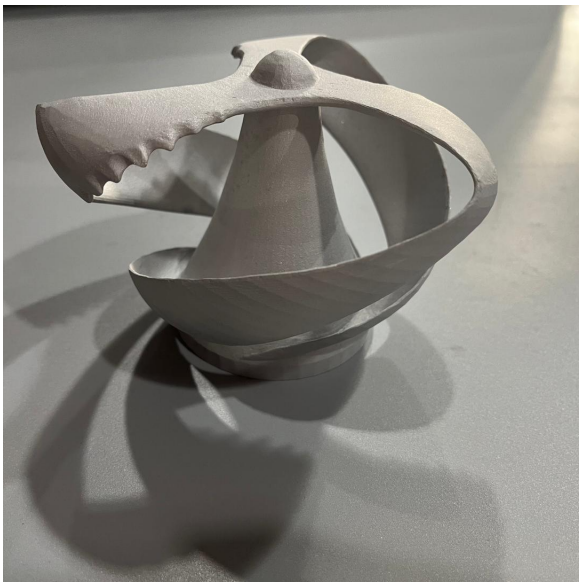
No data from the foils array was gathered. Propulsion was measured in the whole unit, but undifferentiated from the impeller mounting designed to hold the foil array. MARIN was unable to measure the cylindrical array as an open unit.

The thrust data generated showed no advantage versus comparable screw propeller. This effort was inconclusive, as this design model is a hybrid of shafted designs and the open cylinder design.

We have yet to identify a test lab that has the facilities to tank test the cylindrical model.

DIJANA BASTA Model

This is small size shaft mounted prototype designed by Dijana Basta:



Dijana Basta Model

This Vor-Prop model has been printed in aluminum with 6 cm radial width, 7.5 cm depth on shaft fitting is being tested.



Dijana Basta Model

This model may prove effective for alternative to small propellers. Tank tests are yet to be performed.

Ibrahim Mesinovic Models

Vor-Prop virtual models as designed by Ibrahim Mesinovic:

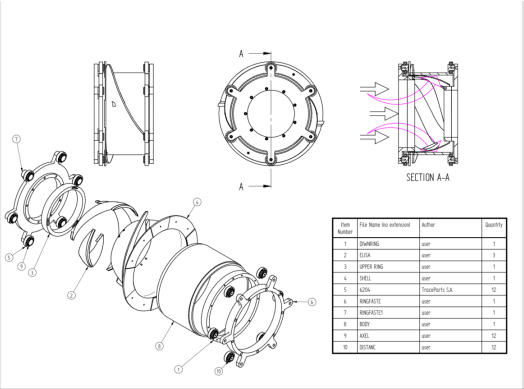
COAMING MODEL:



This prototype is contained in a coaming structure which directs and boosts the pressure differentials of the Vor-Prop, is yet to be built and tested.

THRUSTER MODEL

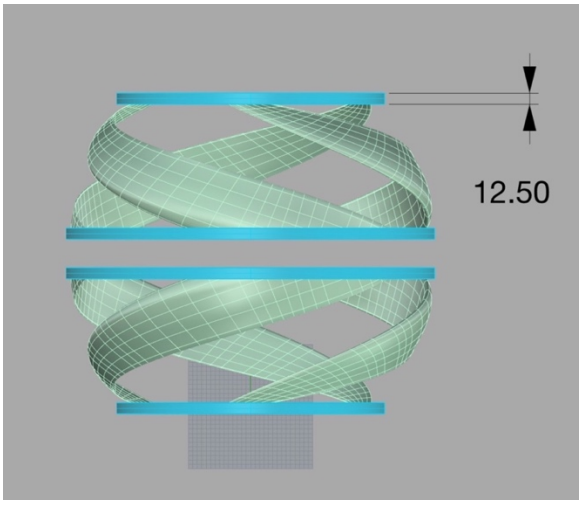
Another proposed use of the Vor-Prop is as thruster driver, installed in ships or other pumping applications.



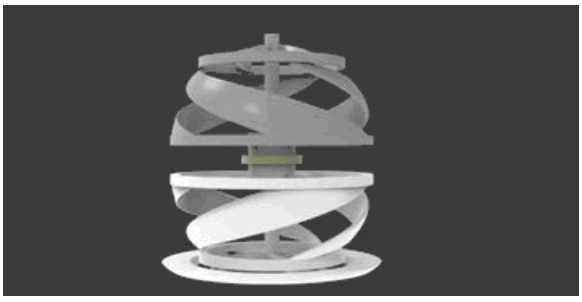
Thruster Model

DOUBLE VOR-PROP MODEL

A double Vor-Prop combines two semi-spherical helical foil blade arrays rotated in opposite directions.



Double Vor-Prop



Double Vor-Prop

This model has been tested as propulsor for an autonomous underwater vessel with pressure generated to supply outlet water jets with high pressure water volume for maneuvering.

Each array is driven by power application from top and bottom connectors creating pressure differential with a low pressure vortex drawing water in from above and below, expelling at the horizontal center into hydraulic gates controlled for variable propulsion.

Variable speeds applied to each Vor-Prop array may alter the internal pressures for changing control and directionality, directing propulsion in sub-sea applications. Greater efficiency may be realized compared to external screw propellers or water jet pump outlets.

The resultant maneuverability of the UAV with this double Vor-Prop sphere is under prototype modeling and testing in Sweden. A functioning model will be available for production in the next year.

These various efforts have been more indicative than conclusive. More designs and tests are required to realize possible applications of the Vor-Prop concept.

PROJECT GOALS:

- Research hydraulic principle of vortex driven thrust
- Provide comparables to screw propeller and water jet technology
- Enable new vessel hull integrated propulsion
- Reduce fuel consumption and pollution
- Reduce aquatic noise and vessel vibration
- Test other applications of vortex propulsion for use in other industries that move liquids via pumps and water diversion systems.

QUESTIONS FOR VOR-PROP:

1: Is this "better" than traditional means, i.e. efficiency, or is it giving designers an alternate set of constraints to balance against in a given design?

- *Vor-Prop will induce a flow through effect that will be less turbulent than a screw propeller, the greater the flow, the more thrust. This may be more efficient by a factor that would make significant saving in fuel costs.*

2: Is there some type of economy of scale by lining these up end to end, perhaps adjusting the pitch of the latter sections?

- *We propose the Vor-Prop be integrated in hull construction to utilize the longest length possible for better performance. The size and pitch of the hydro-blades is key to the impelling of flow and thrust. The blades may also be flexible and adjustable as are some screw propellers and turbines.*

3: Is this meant to be enclosed or exposed? If enclosed, would it help to restrict the end a little bit to optimize thrust, as is a rocket nozzle?

- *Cylindrical casing will be tested once the hydraulic principles are better defined. The cowling around screw propellers (Kort nozzles, etc) focuses flow for use in steerable props at the cost of more friction of water flow. Vor-Prop may be produced in different models, some for major ships, others small for bass fisher size electric outboard motors, and various other applications.*

5: What is the difference in performance between compressible atmosphere - air - versus fluids?

- *Water is 800 times the density of air and not compressible after dissolved gases are expelled. This is the cause of cavitization as high pressure gas eats away at screw propeller blades as they bite the water flow at tip pressures. Vor-Prop will distribute this pressure along the helix blades. We do not know the effect of Vor-Prop at higher rpm. Our simulations have been at 1500 rpm, in the standard range for small craft propellers. The very large props on large ships have lower rpm, which we believe may be matched with very long Vor-Props for more efficiency. Very high speeds for racing boats may not be feasible. Advanced tank tests will indicate limitations or adaptions for specific applications.*

6: Rather than constraining your design by mechanical linkages, have you considered electric drive? You would have so fewer moving parts, and direct control.

- *We have renderings for shaft mountings to retrofit existing vessels, but better performance may occur with electrical induction drive instead of mechanical linkages. There may be different means to drive the Vor-Prop depending on the design of vessel as per purpose.*

7: Have you considered smaller sizes, perhaps things that could be 3D printed? If you made one an inch in diameter, how well would it work?

- *We have made the 60 cm long metal model as seen on the video for visualization and initial water tests. Most propeller research testing is done at a smaller scale for initial data measurement, then scaled up to application. Our 3D printer specialist is planning a carbon fiber print with titanium coating to provide more rigidity for water testing. Micro or macro scale Vor-Props may have different configurations – number of blades, length, diameter, etc. There may be applications in medical, agricultural and other industries as the research continues.*

ANSWER TO REVIEWER CHAD OLDFIELD:

Thank You for your consideration and analysing of our Vor-Prop concept.

As we develop this project, it is likely to have balance and vibrations issues.

Also, the CFD analyses that we have made are made just for blades, without calculating whole turbine as a one working part. We do not have final design and calculations. We have initial numbers for our unspecific type of turbine blades.

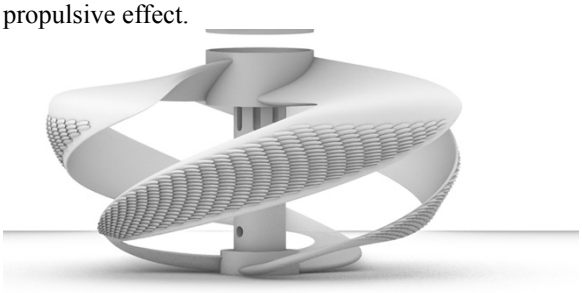
In our Vor-Prop design we follow natural shapes combined with research of Victor Schauberger. We believe that our improvement of some specific shapes will result with better performances than we have today in other types of blades.

To overcome recent issues with prototypes, we will use egg shape cone tip for the blade mount structure. The pressure differences between the lower-pressure region behind the cone and the higher-pressure region surrounding it should be sufficient to create a net force for propulsion. Careful design and optimization of the system will be necessary to achieve this effect.

Achieving a stable and controllable vortex behind the cone is crucial for harnessing its benefits for propulsion. The shape and size of the cone, as well as the turbine's characteristics, will influence the vortex formation and fluid dynamics.

The turbine blades' design will be optimized to capture the impulsion effect caused by pressure mediation in front of the cone. The blade shape, angle, and orientation will impact the efficiency of

the propulsion system. We have already proven in CFD analyzes that blades have much better characteristics if they are covered with a relief like fish scales which induce micro eddies and add to the propulsive effect.



Fishscale Reliefs

Comparison Between Vorprop and Fishscale Turbine

	Vorprop turbine	Fishscale Turbine
Max Velocity	15 m/s	17m/s
Max total enthalpy	1238542 J/kg	1240314.55 J/kg
Dynamic pressure	3885.5Pa	4113.23 Pa
Equivalent Torque	1033.11N.m	422.18N.m
Equivalent thrust	14428.4N	29832N

By combining a single cone design with fish scales on the blades and leveraging the insights gained from the CFD analysis and physical testing, we have the potential to create an efficient and eco-friendly propulsion solution that could benefit the marine industry.

Dijana Basta

ANSWER TO REVIEWER WILLIAM WOOD:

You rightly identified the crux of our research, to create axial velocity from a structure that has been developed to capture tangential velocity in the helix water turbine. Vortex forces and hydro foil hydraulics have many vectors, some as interference, others as useful dynamic effects. We have identified pressure differentials that indicate potential propulsive effects in simulation of the cylindrical model, and when confined in the coaming contained model and double Vor-Prop turbine models.

The shafted models are inherently problematic due to flow interference of the mounting impeding the development of vortex induced pressures. These models have not yet been fully measured in tank tests. The prototype cast for MARIN proved unfeasible due to its extended form, weight and vibrations. What does not work may lead to designs that do work in specified applications. We will continue our research to optimize the forms that realize our hydraulic concepts.

Richard Hayman

DEVELOPMENT TEAM:

Richard Hayman, Capt. USMM,
Member, SNAME

Ibrahim Mesinovic, marine engineer and inventor, Sweden

Dijana Basta, aeronautical engineer and materials specialist, Croatia

Dan Freedman, SUNY Advanced Manufacturing Center, New Paltz NY

Bil Becker, Aerotecture International, Inc., Chicago
Inventor and designer of helix wind turbines

John Sellers, inventor and engineer, designer of turbine generator control systems, magnetic propulsion for Royal Navy submarines

Joe Beyroutey, Blake Koop, Matt Sinnott,
Computing Future Solutions, Bethlehem, PA

Prof. Jacques Hadler, Webb Institute

Jack Langford, sculptor and foundry director

Allen Rubenstein, Jason Wachler, patent attorney, Gottlieb, Rackman, Reisman, P.C. New York

Keith Weltsch, trademark & licensing attorney, Scully, Scott, Murphy & Presser PC

REFERENCES

BOSE, NEIL, *Marine Powering and Propulsors,* SNAME, 2008

KUNDU, PIJUSH, COHEN, IRA, *Fluid Mechanics*, 4th ed.

GERR, DAVID, *Propeller Handbook*, International Marine Pub. Camden ME, 1989

GORLOV, A.M. *Helical Turbines,* Marine Technology, 35, No 3, pp. 175-182, 1998

SCHAUBERGER, VICTOR, *The Energy Evolution*, Gateway / Gill & Macmillan, Dublin, 2000

ADDENDUM:

CFD analysis of Vor prop project

By Ibrahim Mesinovic

Contents

Objective
CFD SIMULATION APPROACH
Meshing
Calculation
Post processing
Conclusion

Objective

This study presents computational investigation of the flow in axial water turbines using Finite Volume based commercial CFD package namely Fluent. Based on three dimensional numerical flow analysis and fluid-structure interaction, the flow characteristics of water turbines governing equations.

Mainly the existing generated forces and moments on blades because of the interaction between water and the solid of blades.

CFD SIMULATION APPROACH

CFD is fundamentally based on the governing equations of fluid dynamics. They represent mathematical statements of the conservation laws of physics, where the following physical laws are adopted:

1. Mass is conserved for the incompressible fluid at steady state as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

where, the fluid velocity at any point in the flow field is described by the local velocity components and which are functions of location (x,y,z) and time (t).This equation is precisely the partial differential form of the continuity equation.

2. Newton's second law: the rate of change of momentum equals the sum of forces acting on the fluid

Where pressure gradient, and other corresponding terms in these three equations and the normal and tangential stresses acting the control volume body forces that may gravity, centrifugal, Coriolis, and electromagnetic forces.

3. Standard κ-ε mode

Assumptions

In this study, the preprocess including model and non-structure mesh generation was completed.

Since the working fluid is ocean water, the CFD simulations were calculated based on assumptions as follows:

1. Incompressible flow.
2. Steady-state conditions.
3. Thermal effects are negligible.
4. Smooth walls with no-slip boundary condition.

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+F_x$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+F_y$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+F_z$$

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial t}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon c)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_1\frac{\varepsilon}{k}G_k-C_2\rho\frac{\varepsilon^2}{k}$$

Calculation

Calculation Solver:

3D Segregated Steady Implicit Solver Turbulence Model: Standard k–ε model, standard wall functions. Multiple Reference Frames (MRF) approach was used to modeling the rotating flow. This approach adopts two reference frames; one stationary frame related to the body walls around impeller; the other related to the rotating impeller. The fluid zone is divided into two separate regions, while the other, close to the rotating impeller, is related to the moving zone. The momentum equations and closure models are resolved in the separate zones and a steady state approximation is made at the zone interface. Reliable results can be given only by a well converged, well posed and grid independent simulation. Convergence is determined by the order of magnitude of the residues. Two different convergence tolerances are calculated, one is 10^{-4}, and the other is 10^{-6} for continuity of flow in this study. Running the solver such that residuals fall one

more order of magnitude means that more iteration is done to improve the solution quality. It should be noted that convergence criteria must assure that the results do not change as the iterations proceed. In this study the mass flow rate change was monitored

When the mass flow rate stays at a certain number and does not change as the iterations continue, then it can be stated that solution has converged. It was seen that this criterion was satisfied when the continuity and momentum residues fell below 10^{-6}.

Meshing

The mesh, or grid, is the discretized domain in which the flow problem is to be solved. The flow properties are calculated in each cell, and hence, as briefly described, the density of the cells (i.e. the mesh resolution) is of great importance. Basically, the grid could be either structured or unstructured. In an unstructured grid, the cells are arranged in an apparently random fashion in contrast to a structured grid.

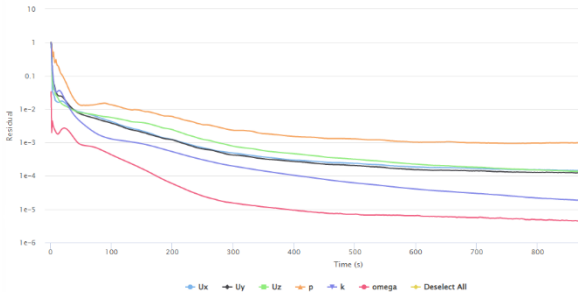
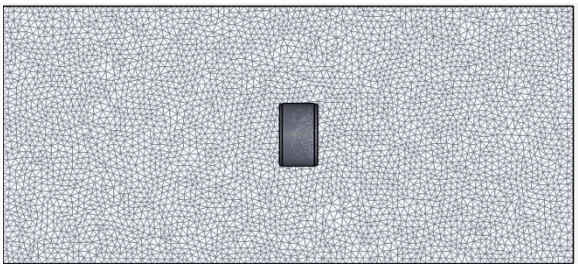


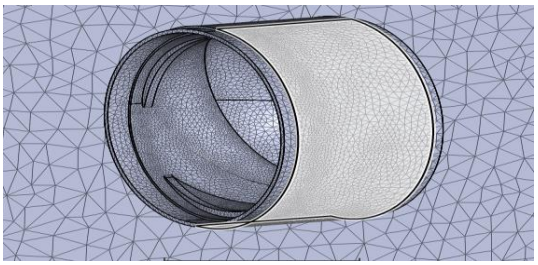
Figure Convergence tolerance 10-4

Further, the mesh could be either single- block or multi-block. For instance, in a structured single-block mesh, the gridlines must pass through the entire domain. For complex geometries, multi-block grids are necessity as it is in our case.

The final judgment on the quality of the division is made through several points, the most important of which is the overall mesh quality which ranges from 0.035 to 1.0
As it is shown in the next figure.

Overall mesh quality:	0.107 (Acceptable range: 0.035 to 1.0)
This mesh meets our quality criteria.	





The way of meshing in our case it divide the domain to two main sources ,the first one is computational domain and the second which is needed to be specific in number of elements ,shape and quality in the region which the flow will be influenced by the structure as the casing and the blades of the turbine .So, the sizing of elements has become more smaller and more numerous in the region of blades with more inflation considerations around the layer where the fluid structure interaction happens .

Mesh densities in the regions: single turbine unit and free flow. Mesh inflation around the blade and the casing. In our case, different types of elements have been used: prims, tetrahadedra, quadrangles,.....,etc

As it is shown in the next table the used type of element and its number:

Number of nodes:	331962
Number of edges:	4153
Number of faces:	2487322
Number of volumes:	1134452
Number of triangles:	2058503
Number of quadrangles:	428819
Number of tetrahedra:	877050
Number of hexahedra:	60692
Number of prisms:	148388
Number of pyramids:	48322

Post processing

Path line plots:

Path lines are parallel to the mean velocity vector, where they trace the flow pattern using massless particles.

Vector plots: A vector plot provides the means whereby a vector quantity is displayed at discrete points (usually velocity, with arrows) whose orientation indicates direction and whose size indicates magnitude.

To calculate the extracted power, the first law of thermodynamics was used, which indicates that the rate of change of energy equals the sum of rate of heat addition on and the rate of work done on the fluid.

Where, **Q** is the heat transfer, **W** is the work being done, h is the specific enthalpy, **0.5 C^2** is the kinetic energy per unit mass and **gz** is the potential energy per unit mass. For an incompressible fluid (liquid water), in the absence of friction, the maximum work output from the turbine is

$$Q - W = m\left[(h_2 - h_1) + \frac{1}{2}(c_2^2 - c_1^2) + g(z_2 - z_1)\right]$$ $\frac{1}{2}c^2$

$$\dot{W} = \dot{m} g [H_1 - H_2]$$

$$\text{where, } gH = \frac{p}{\rho} + \frac{1}{2}c^2 + gz$$

For a control volume and steady state, the law of moment of momentum can be obtained.

which states that, the sum of the moments of the external forces acting on fluid temporarily occupying the control volume is equal to the net time rate of efflux of angular momentum from the control volume

$$\tau = \dot{m}(r_1 C_{u1} - r_2 C_{u2})$$

From Euler's equation the turbine extract power is given as follows:

$$\dot{W} = \tau\omega = \omega\dot{m}(r_1 C_{u1} - r_2 C_{u2})$$

In axial turbine $r_1 = r_2$.

The extract power of 1.02 kw at a nozzle inlet speed of 1.8m/s for a first model with a 0.2m diameter wound turbine is very encouraging.

The following figure shows the forces existing on the blades.

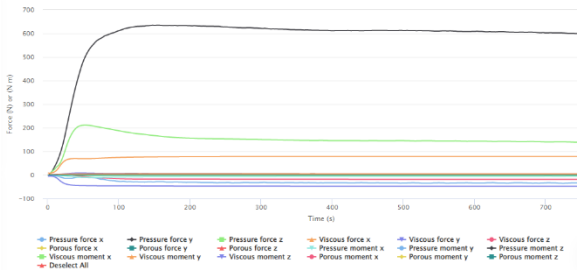


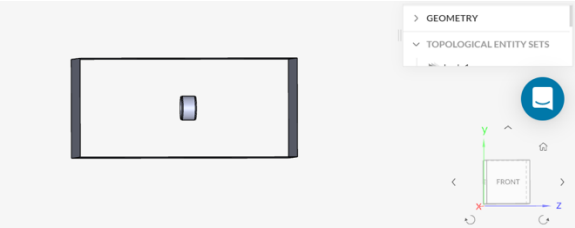
Figure 1 forces and moments of blade

Type value	unit
Pressure force X 32.5	N
Pressure force y 587.6	N
Pressure force Z 133.8	N
Viscous force X 78.6	N
Viscous force Y 5.6	N
Viscous force Z 18.3	N
Viscous moment X 1.2	N.m
Viscous moment Y 4.7	N.m
Viscous moment Z 47.4	N.m
pressure moment X 5.3	N.m
pressure moment Y 0.025	N.m
pressure moment Z 5.1	N.m

Turbine axis

The orientation of the blade as it exists in the computational domain the Z direction is the axial direction of the blade and both of x,y directions are the radial direction of flow.

As it is shown in the next figure



Turbine characteristics

Rotating speed (rpm).	1500
Mass flow rate (kg/s)	18.5
Tangential velocity:(cu)m/s	1.8
Power (kw)out.	1.0205
Torque in x direction N.m	6.66

Results

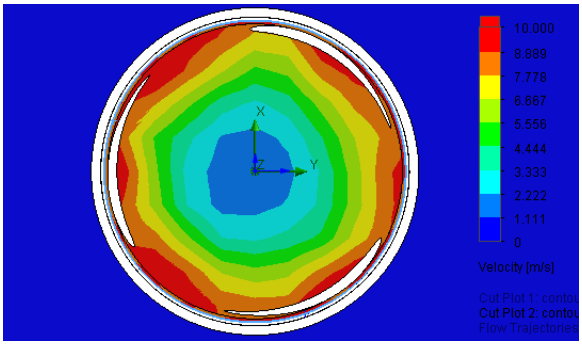


Figure Velocity contour diagram of single turbine unit placed in free flow of 8 m/s

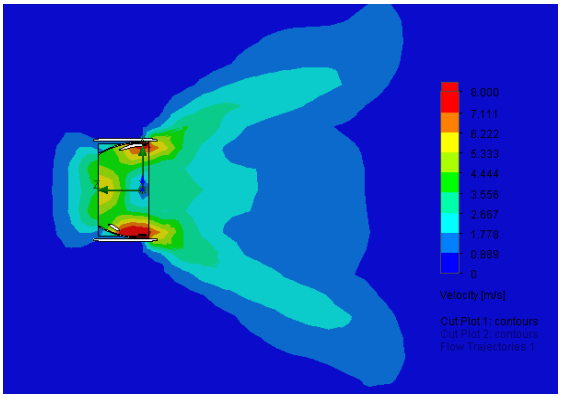


Figure Velocity contour diagram of single turbine unit normal view

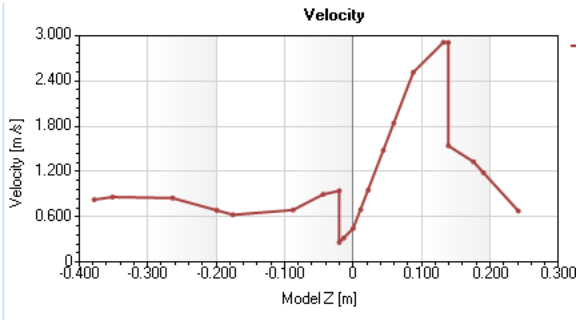


Figure axial velocity across the turbine

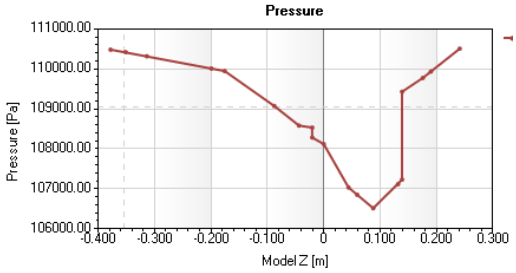


Figure axial pressure difference across the turbine

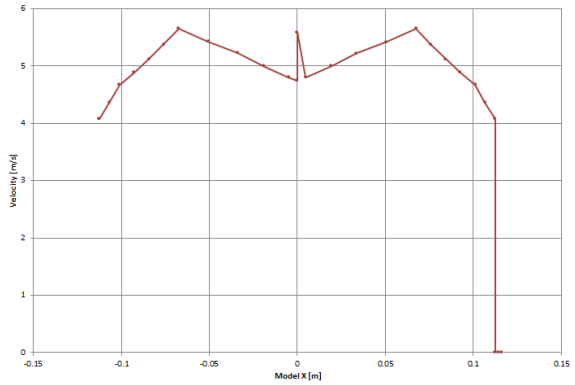


Figure Radial velocity distribution at the inlet of turbine

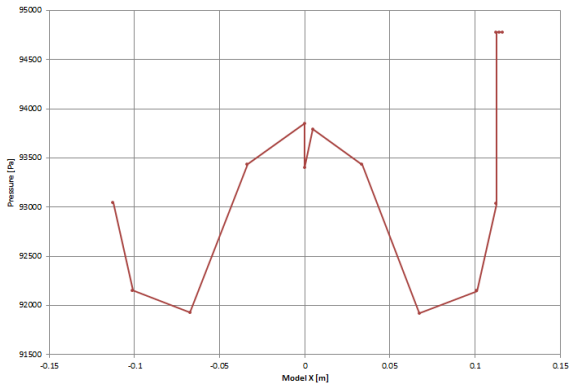


Figure Radial pressure difference at the inlet of turbine

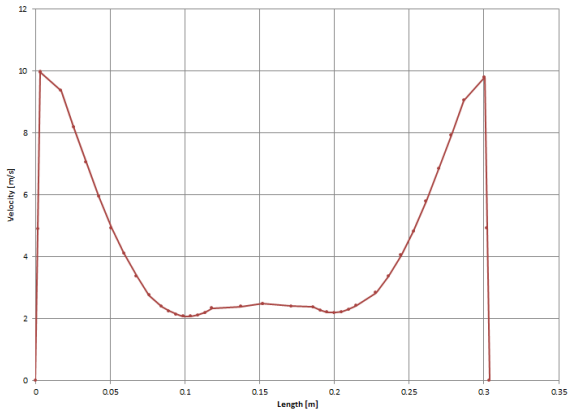


Figure Radial velocity distribution at the outlet of turbine

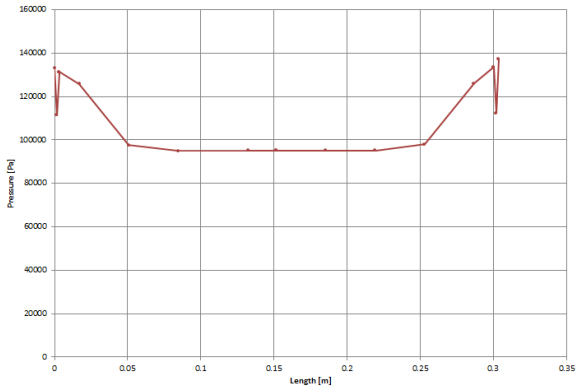


Figure Radial pressure difference at the outlet of turbine

From the previous results, it is possible to determine what is the possible improvement of the distribution of speeds at the inlet and outlet of the turbine, as well as the pressure difference at the inlet and outlet of the turbine.

It is also important to anticipate water movement after getting out from turbine there are some areas where water circulation occurs.

It is important to reduce those areas as these areas reduce the tangent velocity of water after turbine, which affect badly the extract torque and power of the turbine.

So, the next action is: How can you reduce those eddies that occurred in the water.

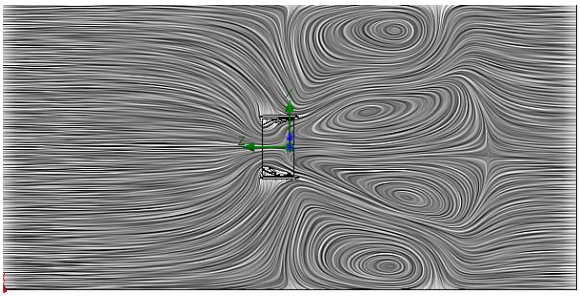


Figure streamlines of water show the areas of circulations of water

Conclusion

The mesh which was used, multi block grid unstructured types like were mentioned ***prims, tetrahadedra, quadrangles*** and the number of meshing is about 4.5 million as they were mentioned. The overall quality of meshing is acceptable as previous mentioned.

The analysis was done with some assumptions as they were previously described. The residuals of

solution were about 10^-4 which is considered acceptable.

The results as were discussed, the difference in the tangential velocity between exit and inlet is about 1.8 m/s, generated water with flow rate about 18.5 kg/s and the efficiency of the turbine is about 0.68 as there some obvious losses in circulation of water after water passing away the turbine which decreases the tangential velocity of the turbine.

video introduction:
https://youtu.be/1nDOpNKP1-E

Conference
presentation:
https://youtu.be/XKHHzhJhkcI?si=G_mrnBcxLb1lyLQDhttps://youtu.be/XKHHzhJhkcI?si=G_mrnBcxLb1lyLQD

Thank you for your interest and recommendations.

Richard P. Hayman, Director

Vor-Prop LLC
290 Quaker Rd.
Pomona NY 10970 USA

riphear@aol.com 1-914-263-6716