CFD-based Parametric Modeling and Optimization at ABS:

Study of Hydrokinetic Turbine

Dae-Hyun Kim, Ph.D. | April 1, 2026

Who We Are

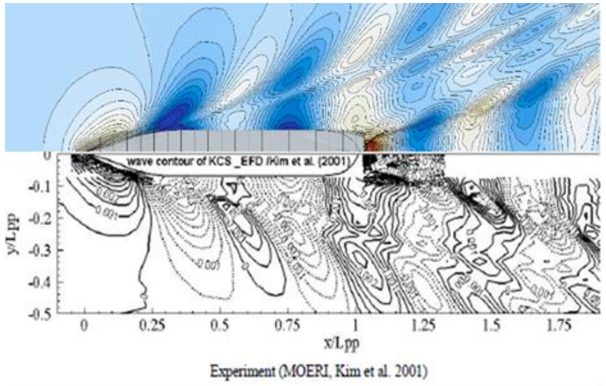
- American Bureau of Shipping (ABS) is a classification society founded 1862.
- ABS clients have asked for assistance in making their ships more energy efficient and environmentally friendly.
- ABS Technology, the internal research division of ABS, has been using CAESES software for many years.
- ABS Mission Statement:

> *To serve the public interest as well as the needs of our members and clients by promoting the security of life and property and preserving the natural environment.*

Evaluation Capabilities

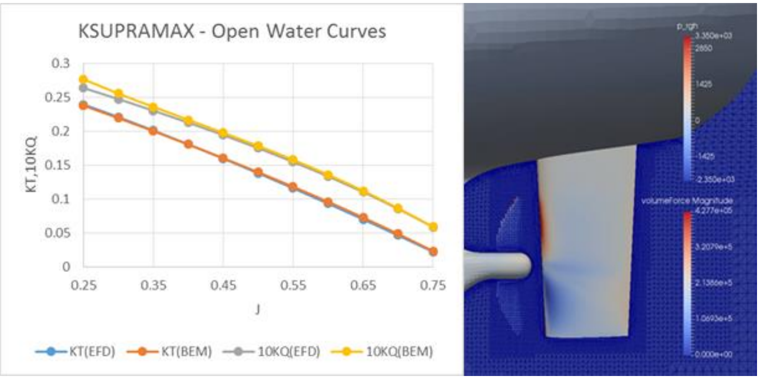
Hull Form Evaluation

Dedicated HPC (High Performance Computing) center and highly automated simulation preparation make it possible to evaluate hundreds of hull forms daily using RANS (Reynolds-Averaged Navier Stokes) CFD tools. The computation results have been extensively verified and validated against experimental data.



Propeller Evaluation

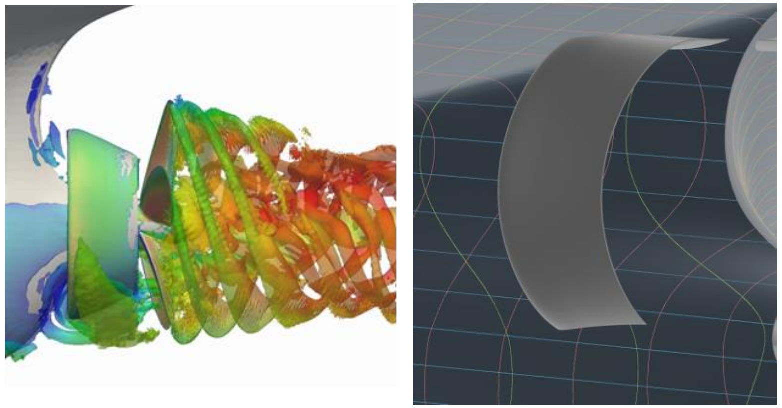
CFD tools with various levels of fidelity can be utilized to evaluate the performance of a propeller in open water or behind a hull. The available tools include RANS tools, Potential theory propeller tools, and the coupling of RANS and Potential theory propeller tools.



Experimental Results Courtesy of KTTC

ESD Evaluation

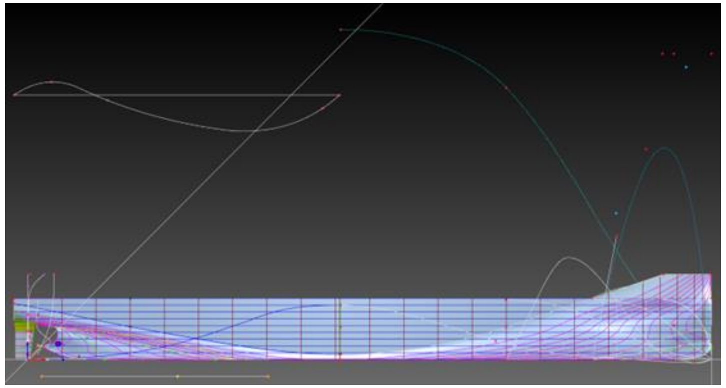
Evaluation of ESD (Energy Saving Devices) often requires the simulation of rotating propellers with the highest fidelity, but lower fidelity simulations are also used, such as body force propeller. The shape and alignment of different appendages such as rudder bulbs and bilge fins can also be evaluated.



Optimization Capabilities

Global Hull Form Optimization

It has the advantage of reducing operating expenses and helping with regulatory compliance without any additional capital expenditure for new builds. RANS CFD tools are routinely used in model scale and full scale. A complex set of design constraints can be considered during the optimization.



Local Hull Form Optimization

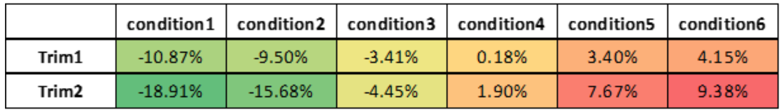
This type of optimization can significantly improve the energy efficiency of a vessel with a relatively small investment in modifications. One example is a bulbous bow retrofit optimized for a new operational profile. Aftship optimization for improving propulsion efficiency is also routinely carried out.

© Juozas Baltiejus/ Shutterstock

Trim Optimization

Operating a vessel in the optimized trim could result in a significant fuel cost without any hull modification. The significant portion of saving comes from reduced residual resistance, because hull forms were often optimized for a design draft and speed. The results are used in connection with cargo loading software.

Trim Optimization for power saving



	condition1	condition2	condition3	condition4	condition5	condition6
Trim1	-10.87%	-9.50%	-3.41%	0.18%	3.40%	4.15%
Trim2	-18.91%	-15.68%	-4.45%	1.90%	7.67%	9.38%

Typical Optimization Process at ABS

1. Initial Review and Evaluation

- Problem Definition
  - Objective function
  - Constraints
  - Operation Profile
- Evaluation of Baseline
  - Proper evaluation methodology
  - Benchmark test, if available

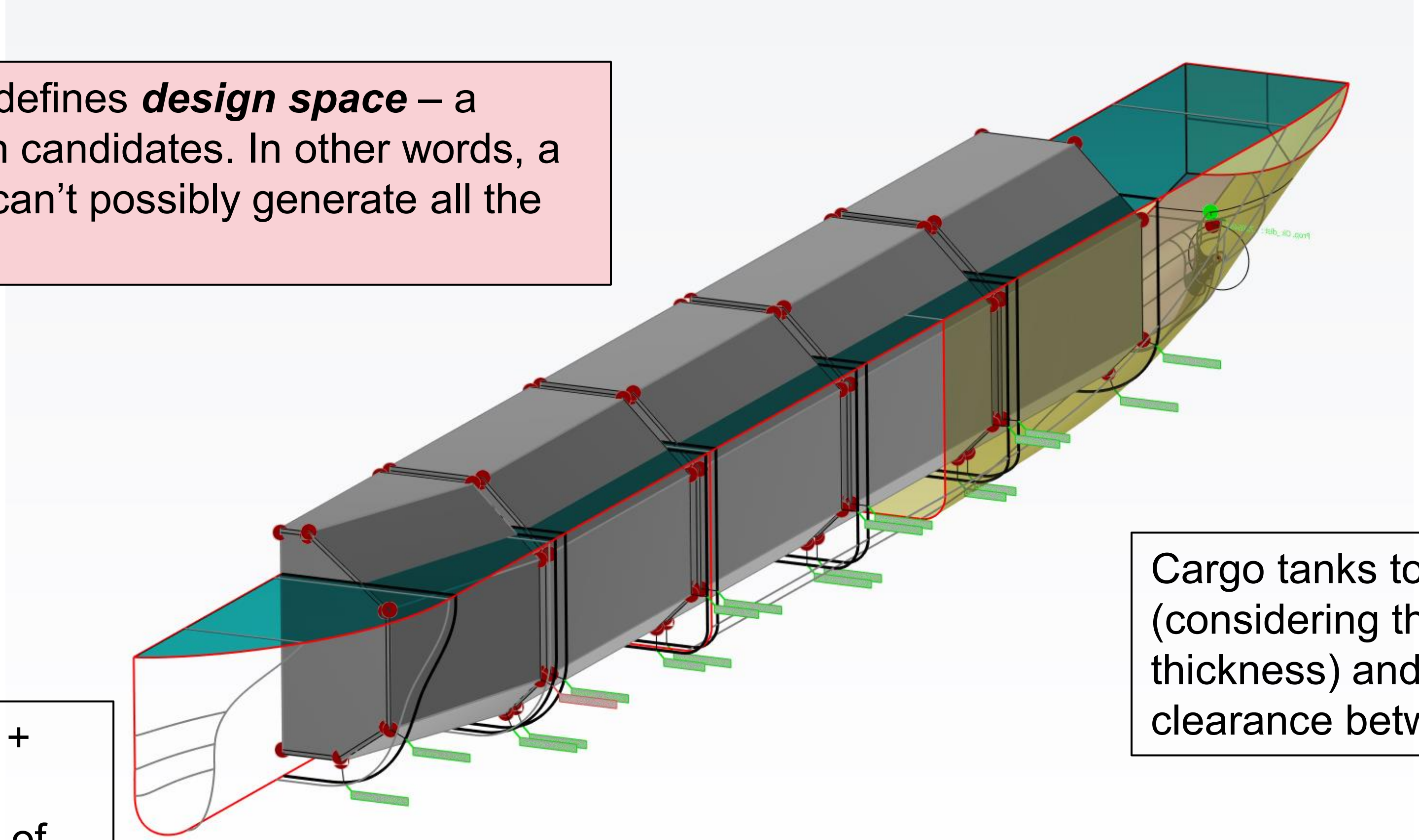
2. Design Space Exploration

- Parametric Modeling of Baseline
  - Fully parametric
  - Partially parametric
- Design Variables
  - What to change? In what order?
  - How (and how much) to change?
- Accuracy of Evaluation vs Speed of Evaluation
- Optimization Algorithm and Strategy

3. Selection of Final Design

- Choose one final design among multiple design candidates
  - Consider other characteristics not directly captured in the objective function, if suitable.
- Additional Fairing/Adjustment
- Evaluation of Final Design
  - Amount of Improvement

Fully Parametric Approach (CAESES User Conference 2024)



Parametric model defines ***design space*** – a collection of design candidates. In other words, a parametric model can’t possibly generate all the feasible designs.

Realistic way of modeling twin-skeg stern (asymmetry between inner and outer skeg surfaces)

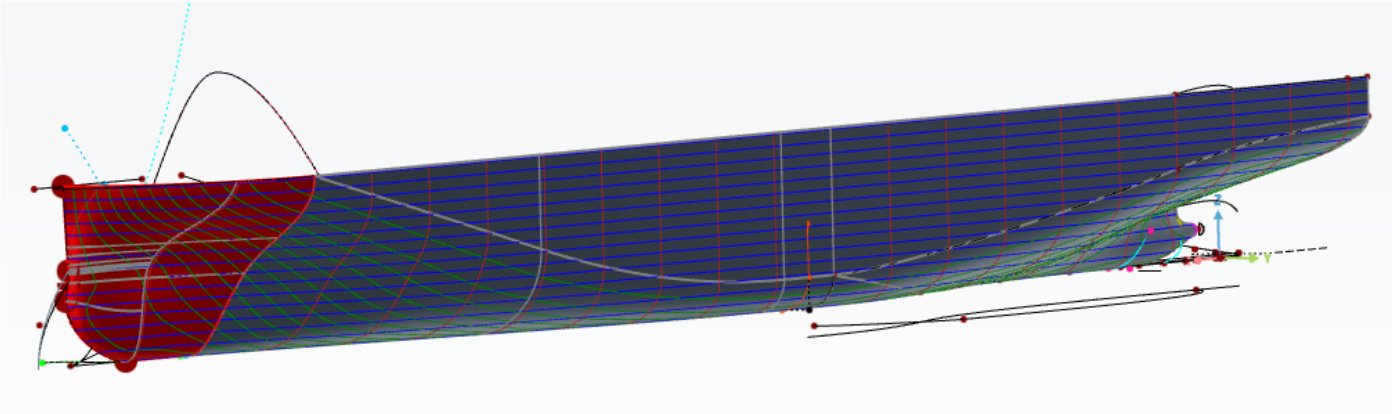
Cargo tanks to calculate volume (considering the insulation thickness) and check the required clearance between hull and tank

Parametric bulbous bow + shoulder with curves for sectional area curve, flat of side, flat of bottom, design waterline, and so on

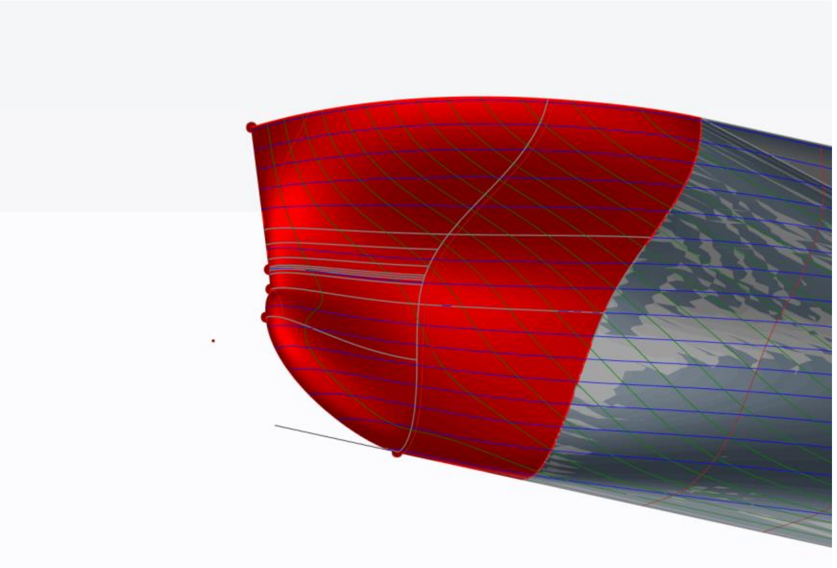
For containerships, container boxes could be modeled. Or, for navy ships, weapon systems in the forebody could be modeled.

Fully Parametric + Partially Parametric Approach

Reproducing an “organic” design candidate with existing parametric model can be complicated.



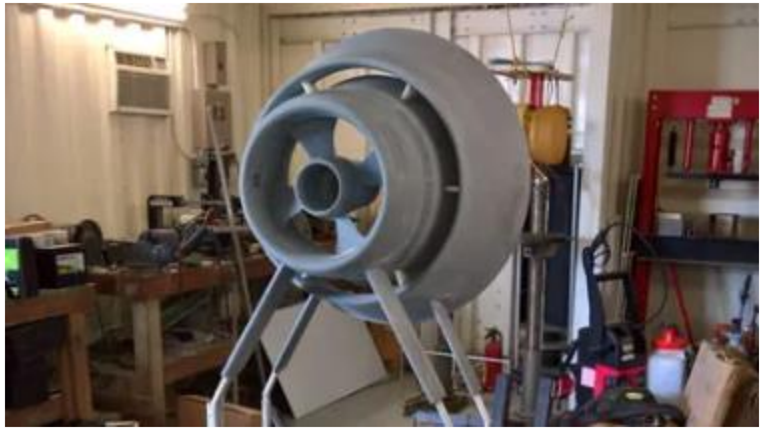
Fully Parametric Approach

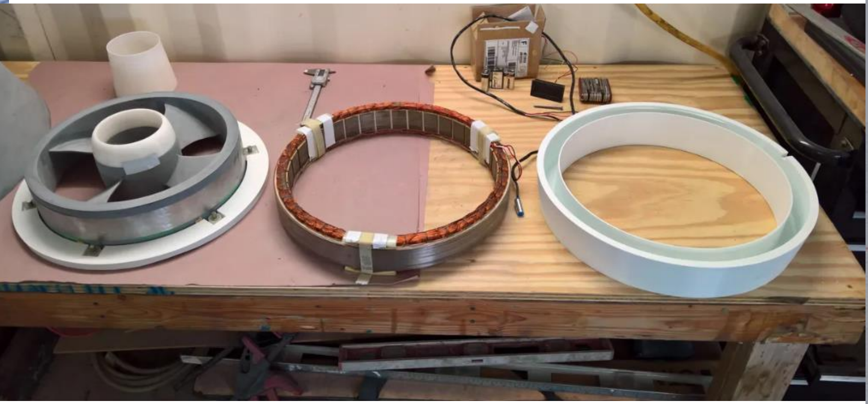
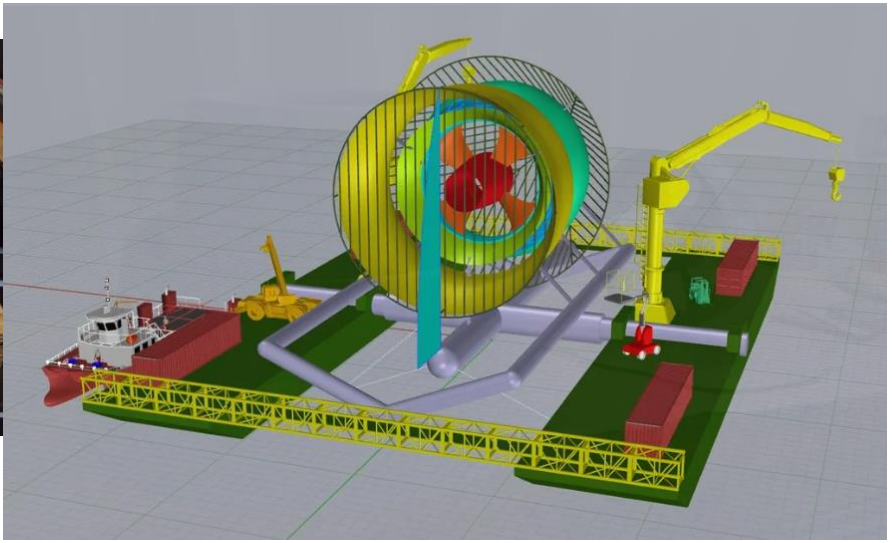


Partially Parametric Approach

Both fully parametric and partially parametric approaches may be necessary for a typical ABS hull optimization project.

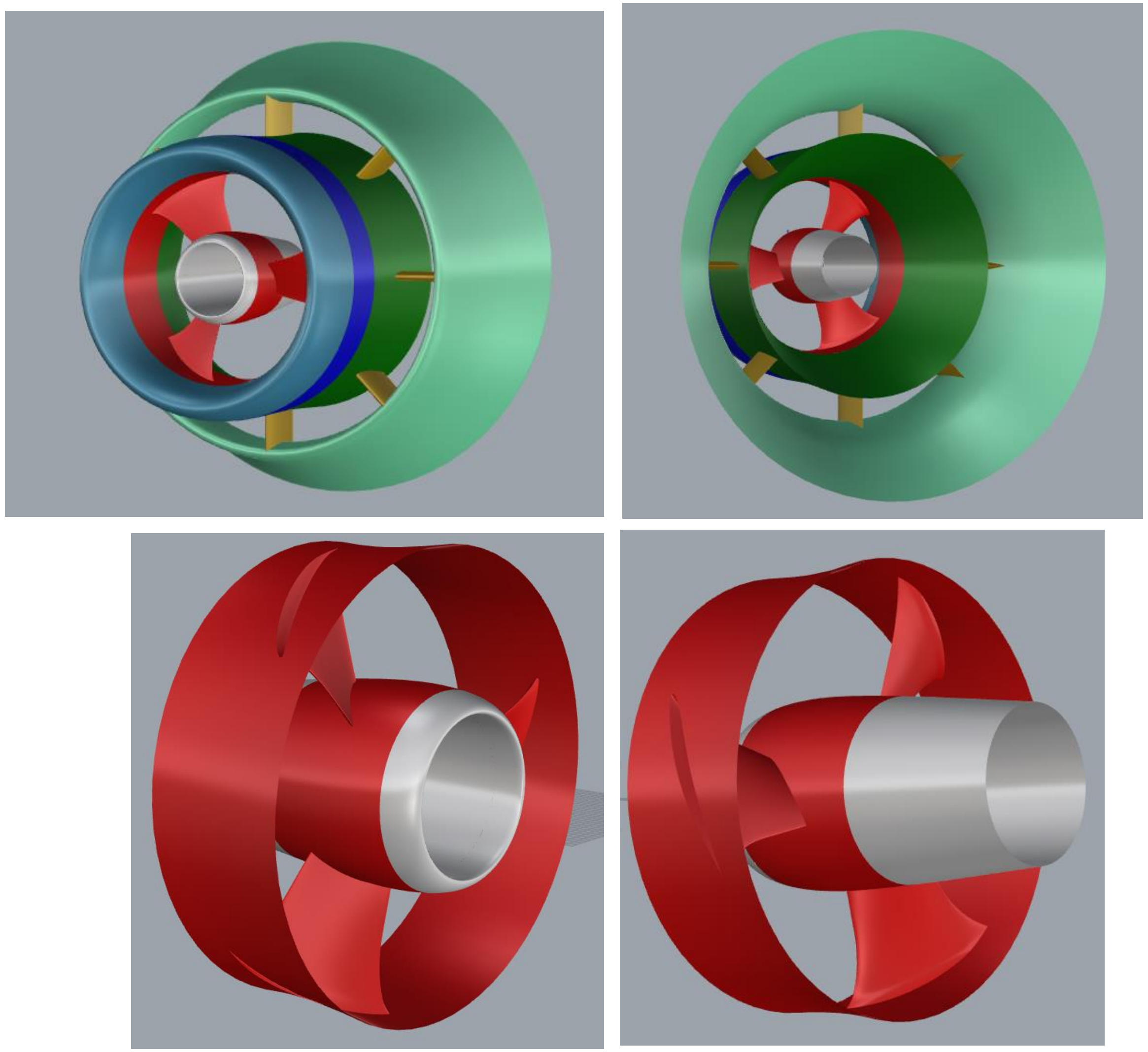
Case Study: Tidal/Current Turbine Design by HEC

Source: https://hydrokinetic-energy.com
HEC: Hydrokinetic Energy Corp

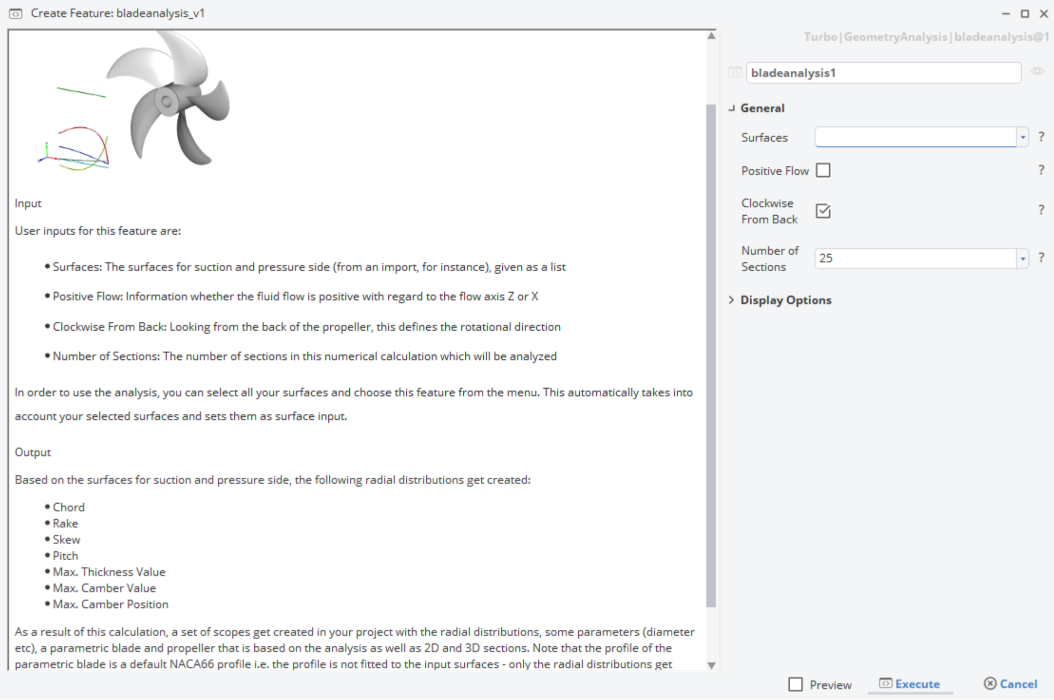
CFD Modeling of the Current Turbine



Turbine Anatomy: Stator vs. Rotor

Blade Analysis in Turbo Module

- Compared to ship examples, it is relatively easy, using Blade Analysis Feature in CAESES, to parametrically model different components of the turbine.
- Blade Analysis Feature is extremely useful in “cleaning” existing 3D propeller model.



Input

User inputs for this feature are:

- Surfaces: The surfaces for suction and pressure side (from an import, for instance), given as a list
- Positive Flow: Information whether the fluid flow is positive with regard to the flow axis Z or X
- Clockwise From Back: Looking from the back of the propeller, this defines the rotational direction
- Number of Sections: The number of sections in this numerical calculation which will be analyzed

In order to use the analysis, you can select all your surfaces and choose this feature from the menu. This automatically takes into account your selected surfaces and sets them as surface input.

Output

Based on the surfaces for suction and pressure side, the following radial distributions get created:

- Chord
- Rake
- Skew
- Pitch
- Max. Thickness Value
- Max. Camber Value
- Max. Camber Position

As a result of this calculation, a set of scopes get created in your project with the radial distributions, some parameters (diameter etc), a parametric blade and propeller that is based on the analysis as well as 2D and 3D sections. Note that the profile of the parametric blade is a default NACA66 profile i.e. the profile is not fitted to the input surfaces - only the radial distributions get

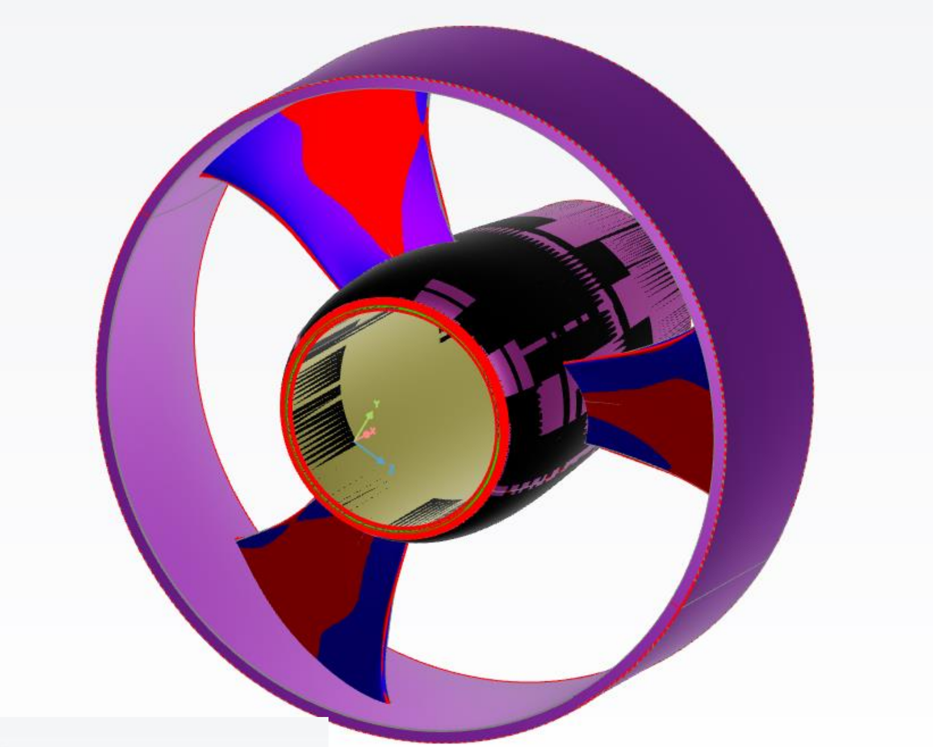
bladeanalysis1

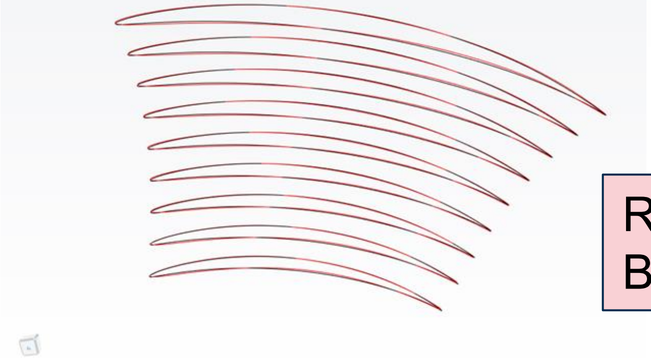
Surfaces:

Positive Flow ☐

Clockwise From Back ☑

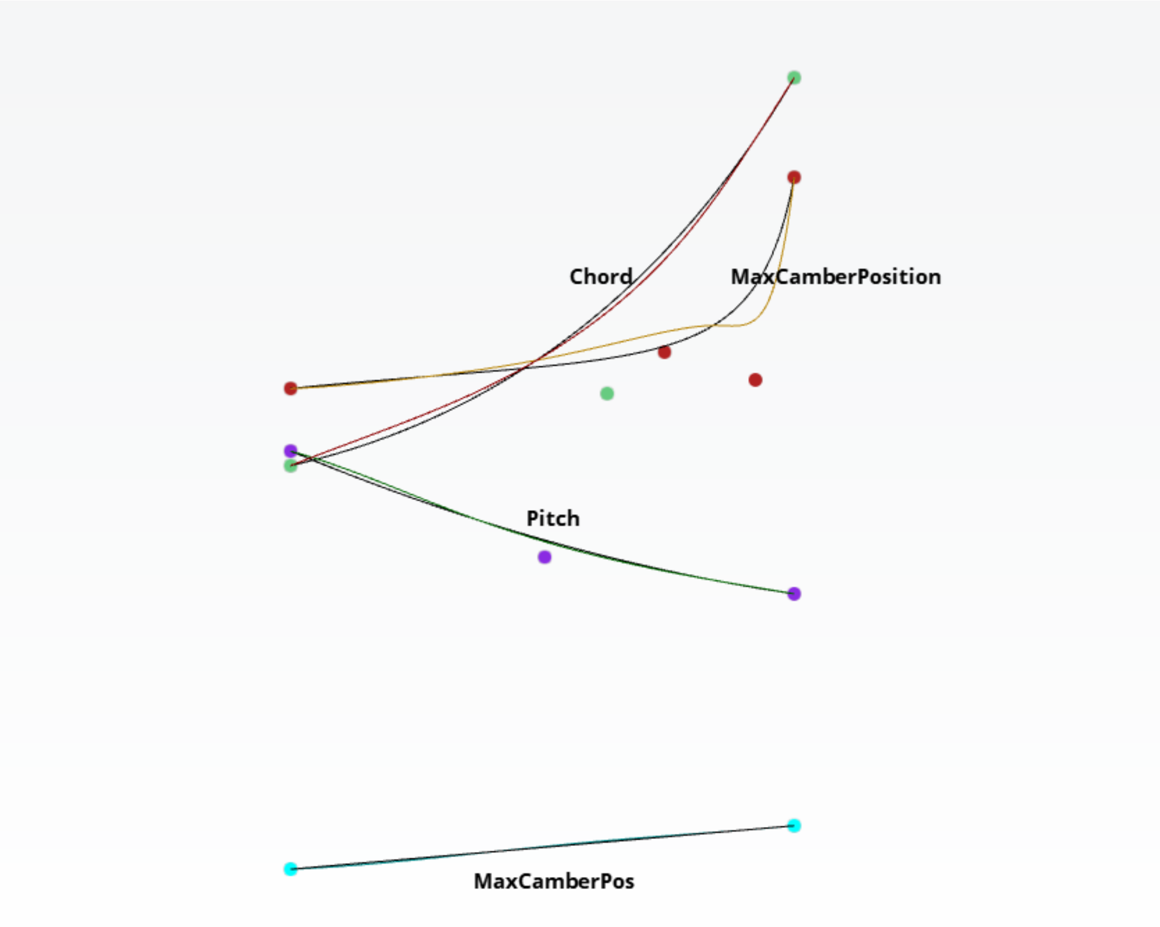
Number of Sections 25

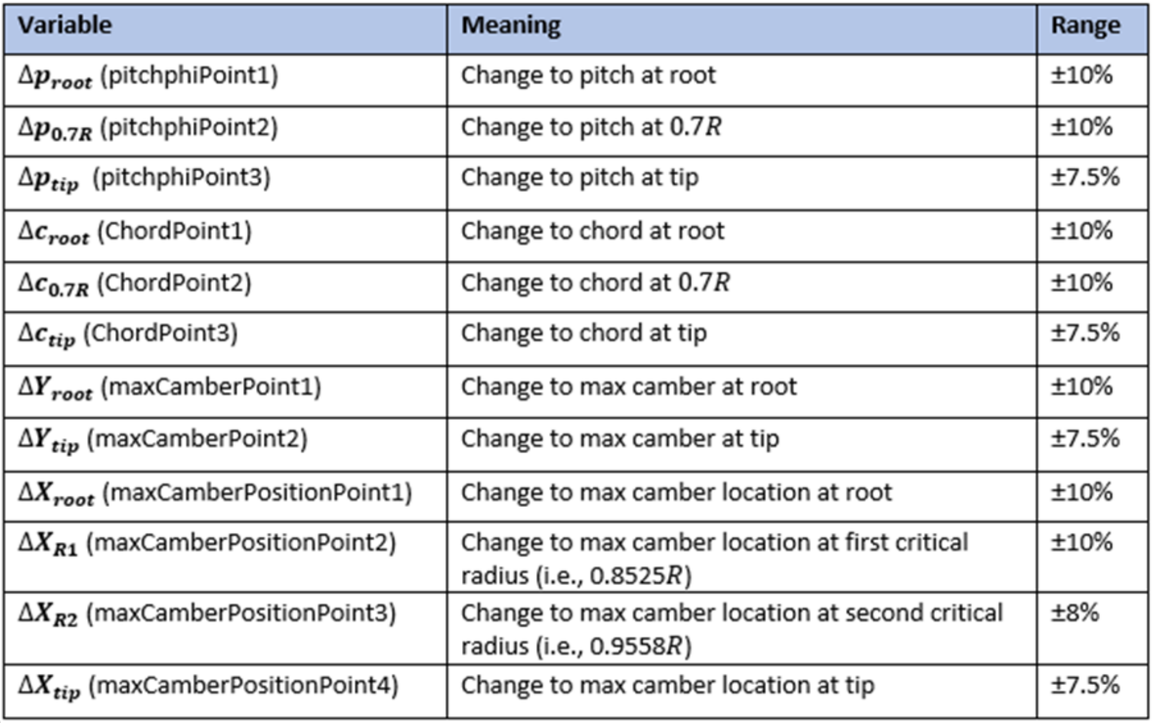




Red (Parametric) vs Black (Baseline)

Design Variables

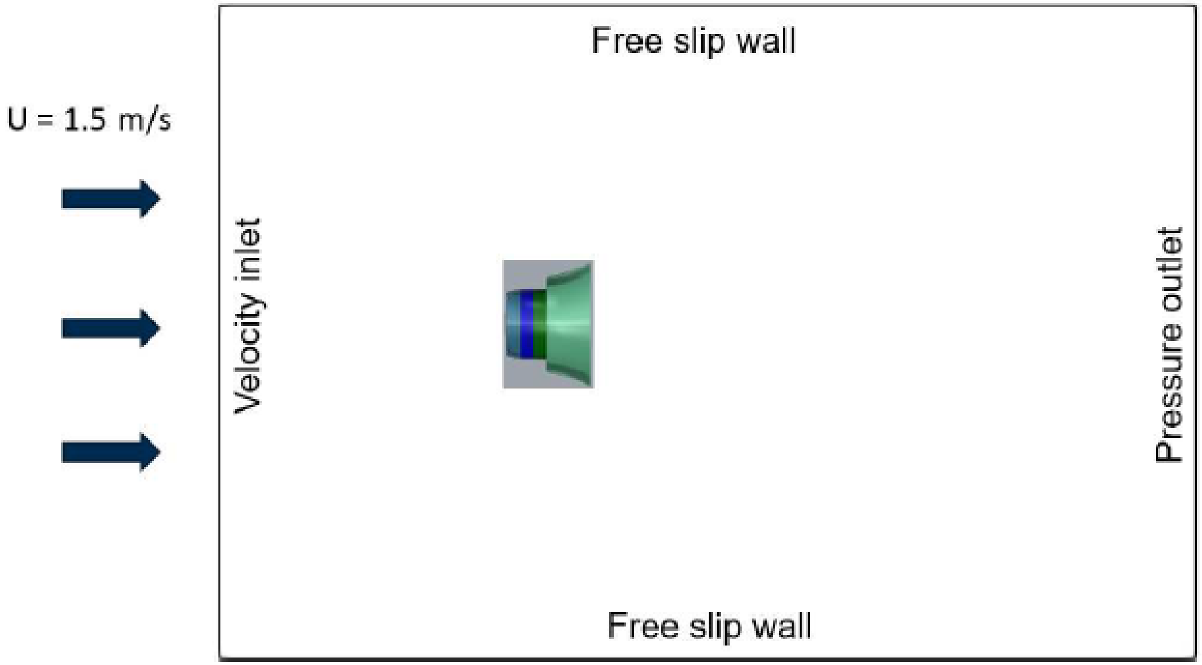


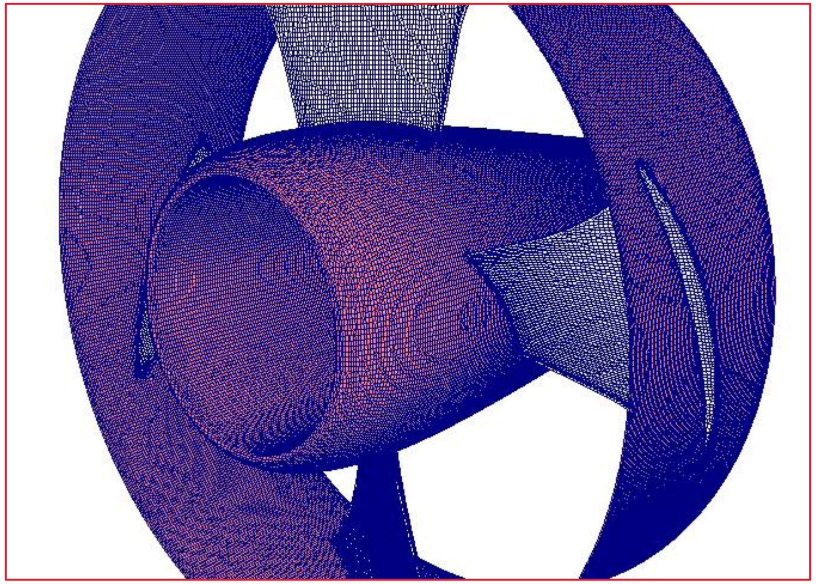


Variable	Meaning	Range
Δp_{root} (pitchphiPoint1)	Change to pitch at root	±10%
$\Delta p_{0.7R}$ (pitchphiPoint2)	Change to pitch at 0.7R	±10%
Δp_{tip} (pitchphiPoint3)	Change to pitch at tip	±7.5%
Δc_{root} (ChordPoint1)	Change to chord at root	±10%
$\Delta c_{0.7R}$ (ChordPoint2)	Change to chord at 0.7R	±10%
Δc_{tip} (ChordPoint3)	Change to chord at tip	±7.5%
ΔY_{root} (maxCamberPoint1)	Change to max camber at root	±10%
ΔY_{tip} (maxCamberPoint2)	Change to max camber at tip	±7.5%
ΔX_{root} (maxCamberPositionPoint1)	Change to max camber location at root	±10%
ΔX_{R1} (maxCamberPositionPoint2)	Change to max camber location at first critical radius (i.e., 0.8525R)	±10%
ΔX_{R2} (maxCamberPositionPoint3)	Change to max camber location at second critical radius (i.e., 0.9558R)	±8%
ΔX_{tip} (maxCamberPositionPoint4)	Change to max camber location at tip	±7.5%

Each curve was modeled by Bspline Curve. Except Max Camber Position, Bspline with minimal points was able to model baseline very accurately.

CFD Model Setup

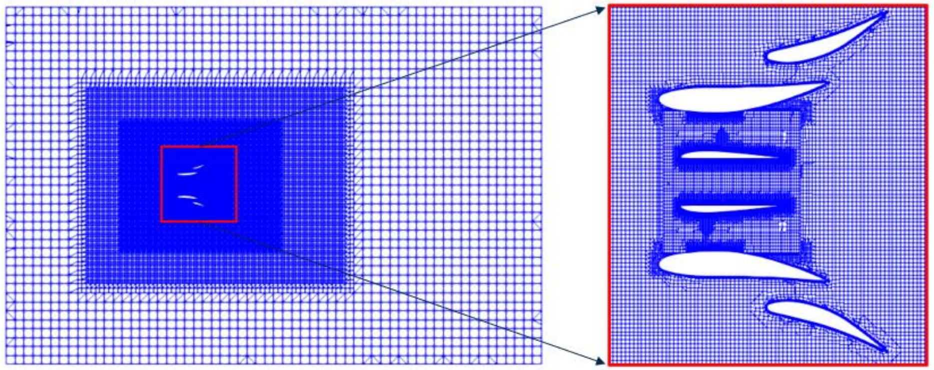


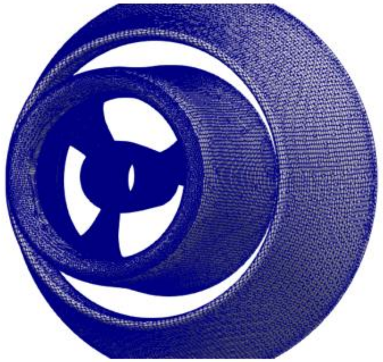



3 blades


4 blades

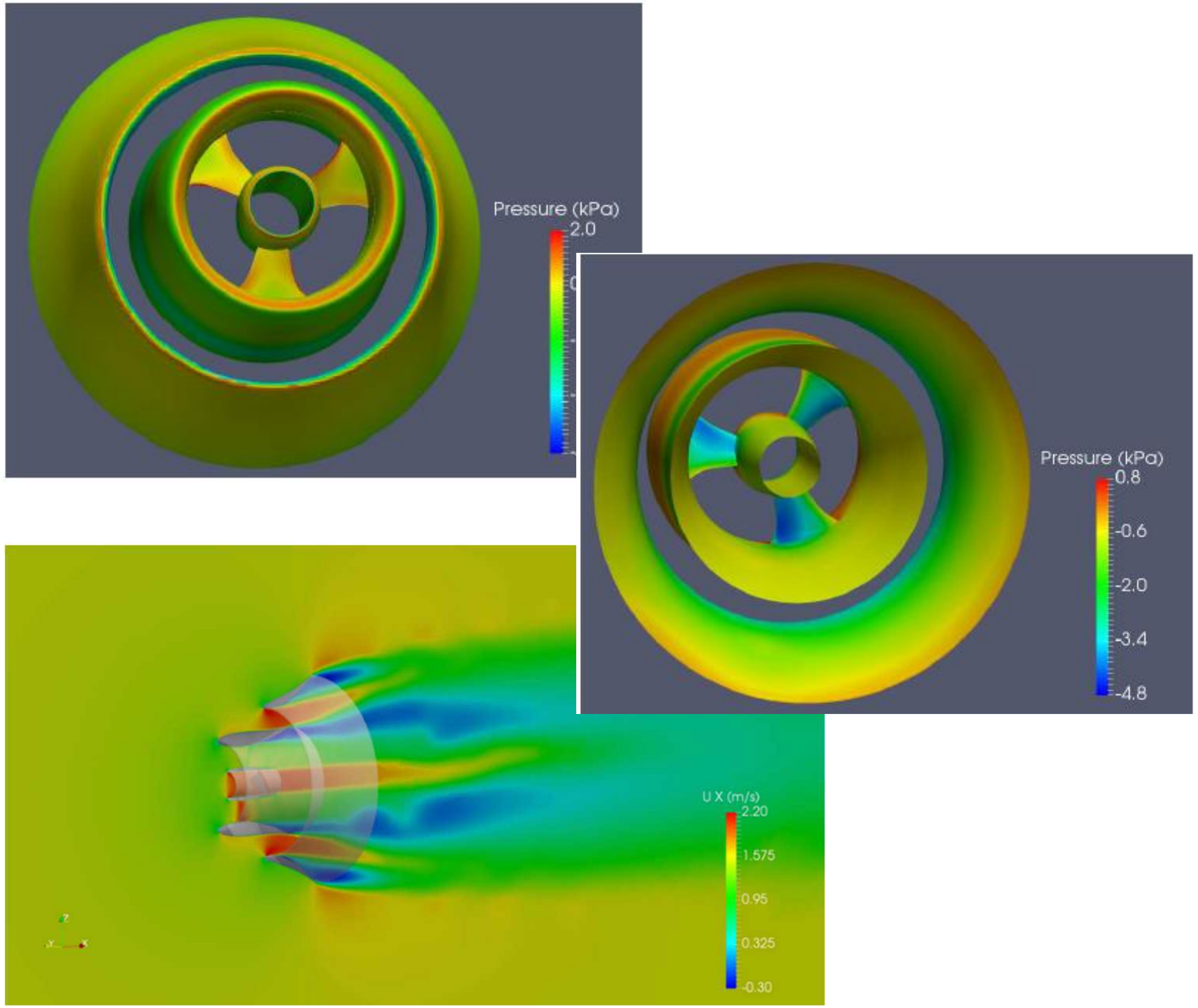

5 blades

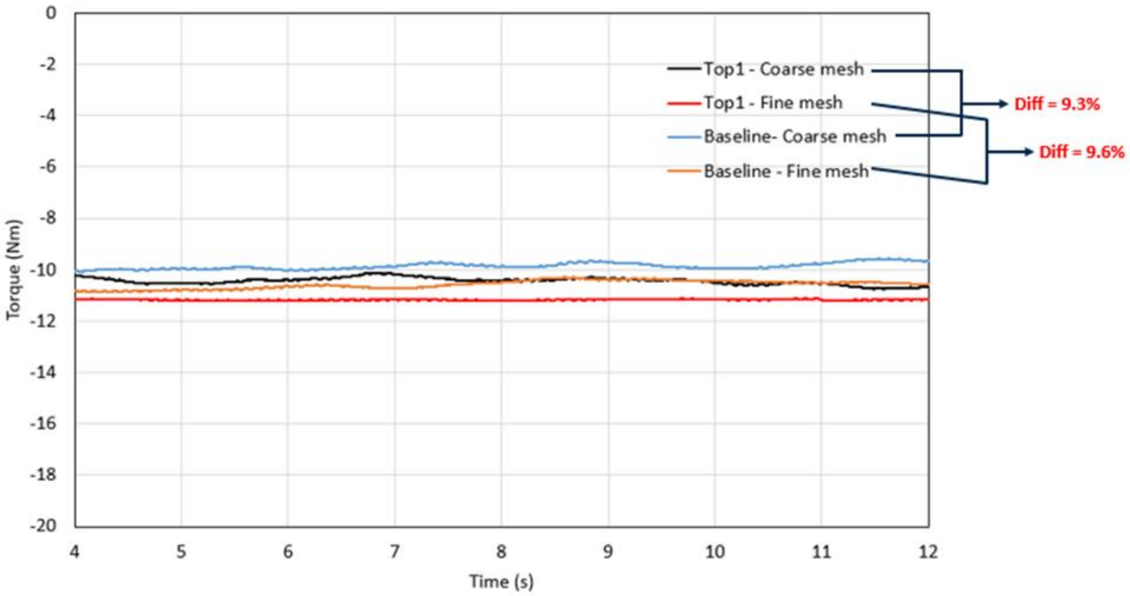




OpenFOAM AMI (Arbitrary Mesh Interface) Method

Discussion of Some CFD Outputs





CFD outputs are used to understand the physics and analyze changes to the design, but it has to be done very carefully.

Summary

- ABS has been using CAESES to help improve the energy efficiency of commercial ships such as tankers, bulk carriers, containerships, and LNG carriers.
- The ABS Technology department is involved in various types of research projects with different stakeholders in the marine and offshore industries.
- The HEC project helped further improve a hydrokinetic turbine with CFD tools. The CFD-assisted optimization helped improve the output power efficiency by 15% at the peak RPM.
- The trend in power efficiency increases when it has three blades.
  - A shroud with max. camber reduced by 10% and with its location moved downstream by 10%
  - A diffuser with max. camber reduced by 10%, its location moved downstream by 10%, maximum thickness reduced by 10%, diameter increased by 10%, and expansion angle increased by 4^o
  - A hub with max. camber reduced by 10% and maximum thickness reduced by 10%
- TEAMER (Testing Expertise and Access for Marine Energy Research) RFTS Round 3 funding (2021) is acknowledged.
- Tidal turbine design and design methodology belongs to © HYDROKINETIC ENERGY CORP. Any reproduction of the technology described herein is only allowed with the written consent and permission from Hydrokinetic Energy Corp.

For More Information

CFD-Assisted Power Efficiency Optimization of a Current Turbine

Zhongfu Ge
Technology
American Bureau of Shipping
Spring, Texas, USA
zhge@eagle.org

Deguang Yan
Technology
American Bureau of Shipping
Singapore
dyan@eagle.org

Walter Schurtenberger
Hydrokinetic Energy Corp.
Key West, Florida, USA
WS@Hydrokinetic-Energy.com

Abstract—**Marine renewable energy is becoming increasingly important amid the global effort to reduce greenhouse gas emissions. Current turbines have the potential of harnessing the energy in ocean currents and tidal streams and adding power to the grid without emissions. The turbine design studied in this paper applies a Flow Acceleration Technology (FAT), which is comprised of a shroud, a diffuser, and a hollow hub. This configuration helps regulate the incident flow and maximize the current kinetic energy passing through the turbine rotor. In order to achieve the highest hydrodynamic power efficiency for the rotor, a parametric study was conducted for the key geometric parameters for the shroud, the diffuser, and the hub. A robust computational fluid dynamics (CFD) model and gridding strategy were established prior to the application of the setup to more than 100 study cases. From the simulation results, it was found that the key to improving power output of the current turbine design is to reduce the obstruction of the turbine structures to the incident flow. Among baseline designs with different numbers of blades, a three-bladed configuration was found to be most energy efficient. Among all three-bladed design variates, smaller thicknesses, smaller maximum cambers, and maximum camber locations shifted downstream of the shroud, the diffuser, and the hub resulted in increased hydrodynamic power outputs of the rotor. The best design variant, included in the present study, accomplished a 15% increase in power output compared to the baseline case. These findings revealed the capability of CFD in optimizing power efficiency for marine renewable energy devices. This parametric study also laid a solid foundation for a more sophisticated blade optimization to be performed in the next phase.**

Keywords—energy, tidal, ocean, rotor, simulation

I. INTRODUCTION

Amid the global campaign for reduction of greenhouse gas emissions, new technologies have been developed and existing technologies modernized. Many of those technologies extract renewable energy from the ocean while minimizing the environmental impact. The use of ocean current energy is identified as a major category of ocean electricity energy technologies [1]. Ocean current energy is created by the movement of water bodies following the global or local circulations. The kinetic energy carried along with the ocean currents is ubiquitous, practically unlimited, and highly predictable. While often placed in the same category, tidal current energy from tidal streams could have to deal with the reversal of the flow direction. A nontidal current energy site has primarily a unidirectional current flow. A common device used for harnessing current energy is the current turbine, including horizontal-axis or vertical-axis configurations as well as more sophisticated ones with multiple rotors [2]. The focus of current turbine design for improving its hydrodynamic power efficiency was optimization of the turbine blades. A large body of literature has been dedicated to this aspect [2-6]. A new idea to draw extra current energy toward the turbine is to use the Flow Acceleration Technology (FAT), such as the Venturi effect devices. Those devices add a converging-diverging duct (or shroud) around the turbine rotor; the rotor will be at the throat of the duct. The geometrical shape of the duct often needs careful design and optimization. There have been commercial implementations of such shrouded turbines in the last two decades.

The current turbine investigated in the present work is an in-stream horizontal-axis, axial-flow turbine enhanced by Venturi-effect FAT with a shroud around the rotor, as shown in Fig. 1. The novelty of the current turbine, as compared with the common Venturi effect devices, is the use of an extra diffuser and a hollow hub. This FAT with multiple components has not been seen in commercially available applications. It has been verified by preliminary testing and numerical simulations that the combination of the rotor-hub-shroud-diffuser system could achieve increased volumetric flux of water through the rotor plane of the turbine [7]. With the FAT, this new design concept could be suitable for many low-velocity environments, where other hydrokinetic turbines are not capable of generating electricity. Environments with low current flows include ocean, lake, and tidal currents, and river streams. This turbine concept is also envisioned to be dynamic: it can be scaled up or down to fit the target deployment site.

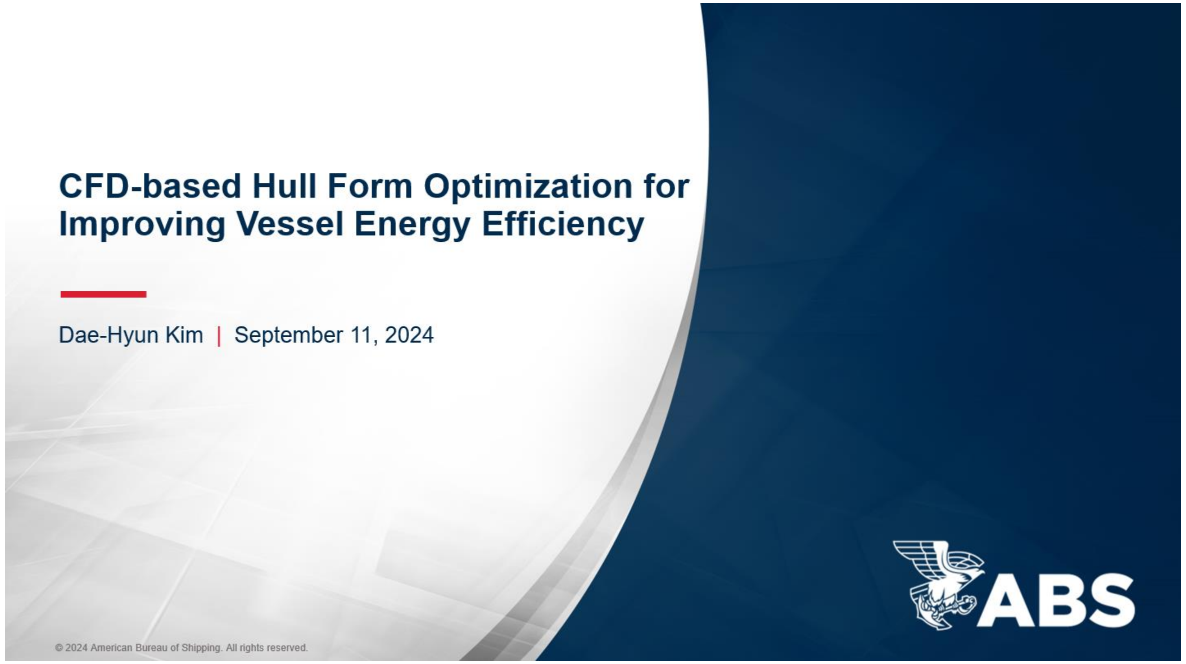
The objective of the present work was twofold: 1) to prove the FAT concept as applied to the turbines studied here using computational fluid dynamics (CFD) simulation and 2) to optimize the FAT components of the baseline turbine geometry, including the shroud, diffuser, and hub. Since the baseline turbine was designed with different numbers of blades, the number of blades was also a parameter for optimization. Since we focused on the FAT, optimization of the rotor blade was not attempted in the present work. This would be the subject of a follow-up study.

This work was funded by the TEAMER (Testing Expertise and Access for Marine Energy Research) RFTS Round 3 (2021).

Intellectual Property, Design and Design Methodology © HYDROKINETIC ENERGY CORP.

XXX-X-XXXX-XXXX-X/XX/$XX.00 ©20XX IEEE

Z. Ge, D. Yan and W. Schurtenberger, "CFD-Assisted Power Efficiency Optimization of a Current Turbine," *OCEANS 2024 - Halifax*, Halifax, NS, Canada, 2024, pp. 1-9, doi: 10.1109/OCEANS55160.2024.10754086.
keywords: {Geometry;Renewable energy sources;Parametric study;Computational fluid dynamics;Blades;Rotors;Hydrodynamics;Fats;Turbines;Optimization;energy;tidal;ocean;rotor;simulation},



CFD-based Hull Form Optimization for Improving Vessel Energy Efficiency

Dae-Hyun Kim | September 11, 2024

PowerPoint Presentation

Thank You

www.eagle.org