

# AHMED H. HANFY

Aerodynamic specialist | Scientific computing  
Engineer | Mechanical Engineer

Gdansk, Poland  
[ahmed.hady.hanfy92@gmail.com](mailto:ahmed.hady.hanfy92@gmail.com)  
(+48) 505 288 804  
[www.linkedin.com/in/engahmedhady/](https://www.linkedin.com/in/engahmedhady/)  
[github.com/EngAhmedHady](https://github.com/EngAhmedHady)



## Summary

Doctoral Researcher in the Marie Curie fellowship program with 3 years of experimental aerodynamics expertise. Holds Master's in Applied Mathematics and Scientific Computing, with prior experience as a junior mechanical designer. Proficient in Python, MATLAB, and C++; skilled in data/image analysis. Familiar with Autodesk Inventor and Siemens NX for CAD modeling. Seeking to contribute expertise in aerodynamics and pursue professional growth.

## Aerodynamic specialist [2020-2023]

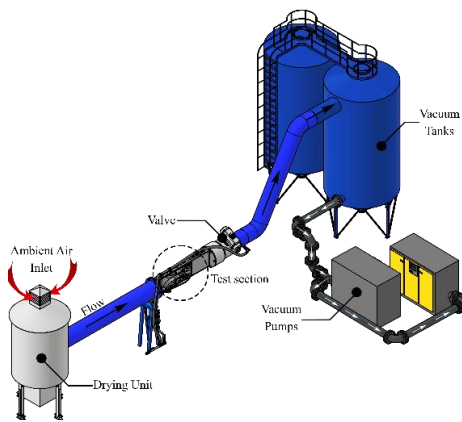


Figure 1 Full model of wind tunnel

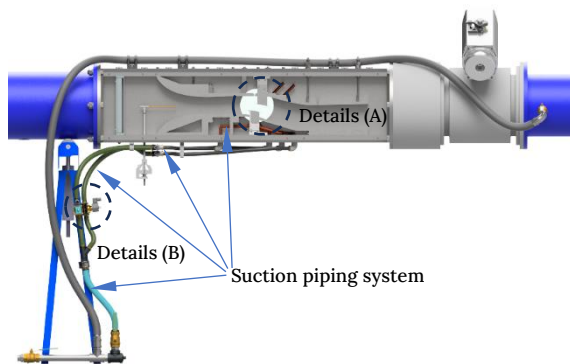


Figure 2 A render for the transonic test section



Figure 3 Modelled solenoid valve [Details (B)]

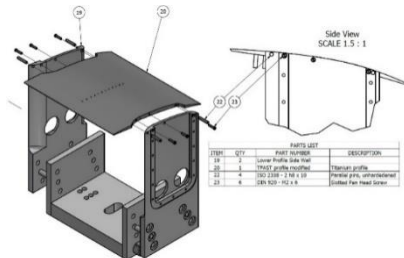


Figure 4 Lower profile with new pressure taps [Details (A)]

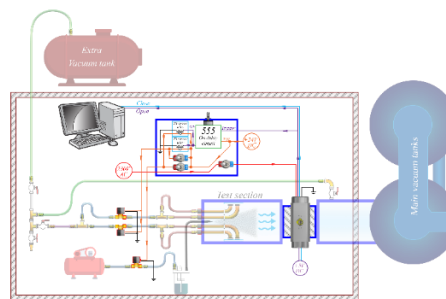


Figure 5 Wind tunnel electronics SID

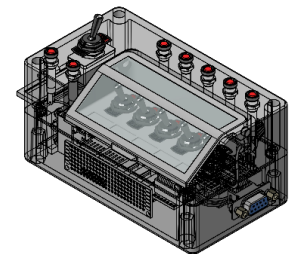


Figure 6 Full control panel model

During my employment as an Aerodynamic specialist at IMP PAN in Poland, one of my responsibilities was to re-model the test section (see Figure 1 and Figure 2). Therefore, modifying the lower profile pressure taps (as in Figure 3) also improves the suction piping system. The piping system was modeled on the CAD with a very high level of detail (see Figure 4) to reflect the real world, model 3D printed holing parts to fix and organize the hoses and pipes.

Acquiring the feedback from the wind tunnel valve was crucial to synchronize the speed camera and measuring devices as shown in Figure 5, a compact model of a control panel was fully designed on the CAD with the wiring system and electronic components as shown in Figure 6.

## Junior Mechanical Designer [2018]

- Sheet metals design and modeling solar tanks.
- Design and supervise the building of a 20k Liter water tank (Figure 8).

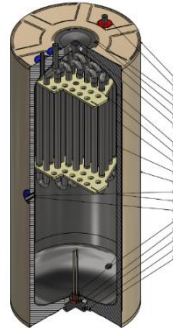


Figure 7 Solar tank with 300 Liters

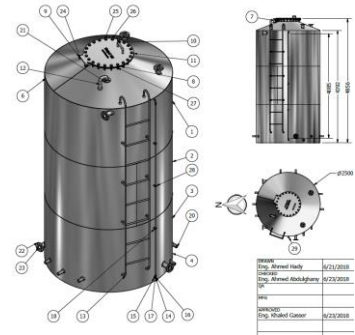


Figure 8 Clean Water tank with 20k Liters

## Volunteer at Torpedo robotics team [2014-2016]



Figure 9 Bismarck ROV Model, 2015  
Memorial University, Canada



Figure 10 Triton ROV Model, 2016  
Neutral Bouncy Lab (NBL), NASA, US

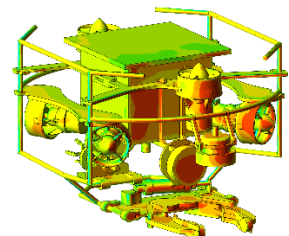


Figure 11 Latro ROV CFD results, 2017  
Long Beach, US

## Other working Designs

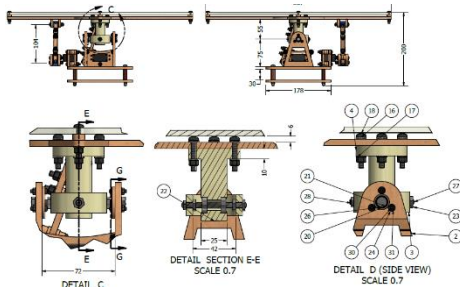


Figure 12 Ball on plate (Auto-balancing system with 3 rotational degrees of freedom), 2018

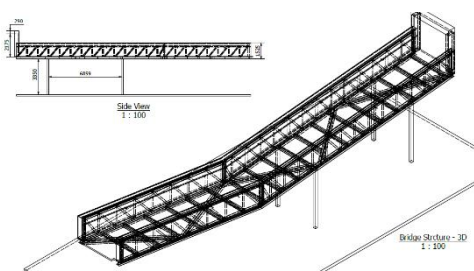


Figure 13 Bridge structure model, 2018

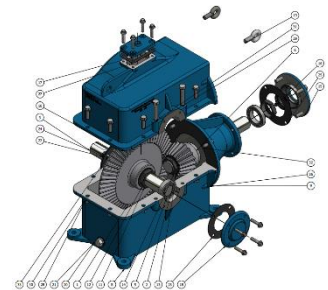


Figure 14 bevel gear exploded view (15kW-GR1/3-1.5krpm), 2015



Figure 15 Detailed model of car diaphragm pump, 2016

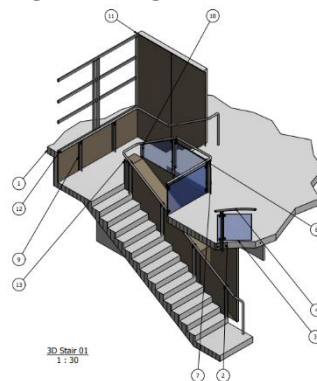


Figure 16 Full 3D model of stairs rails and glass work, 2018

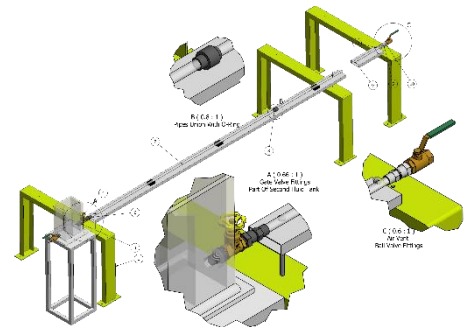


Figure 17 Full testing model of bachelor's degree experiment (for multi-product pipelines), 2016

## CFD works (using Ansys Fluent):

### Supersonic flows:

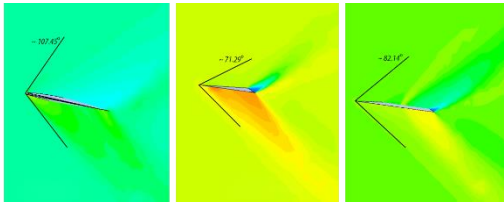


Figure 18 Delta-wings in supersonic ( $M=2$ ) regime (a-transonic profile, b-half diamond profile, c-half diamond double-delta), 2015

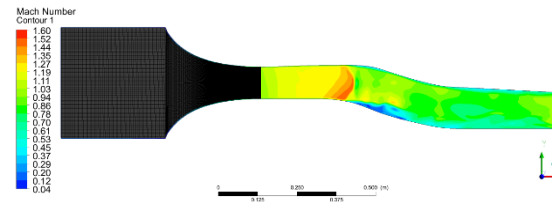


Figure 19 Nozzle transonic Flow (structured mesh with 3.7M element), 2022

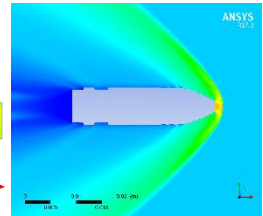


Figure 20 Bult 2D simulation, 2016

### Rotating devices (Sliding meshing):

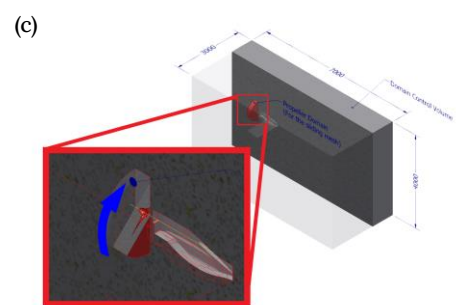
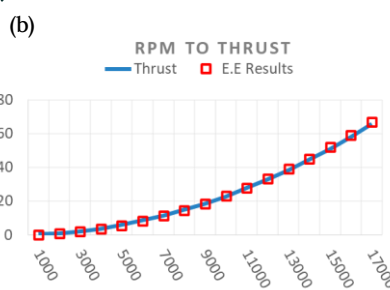
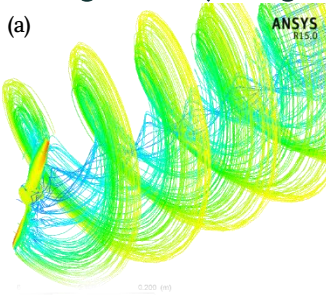


Figure 21 Propeller simulation, [2015-2020]

(a-Free propeller, b- validate the results with empirical equations, c- fixing the propeller on a wing)

### Multi-phase flows:

As a Bachelor graduation project, it was aimed to study:

#### 1-Multiproduct pipeline:

- The focus is on the interface length and shape in relation to pipe length and inclination.
- The results were validated with a lab experiment.

#### 2-Two-phase Horizontal separator:

- CFD simulation on both Horizontal and vertical separators.

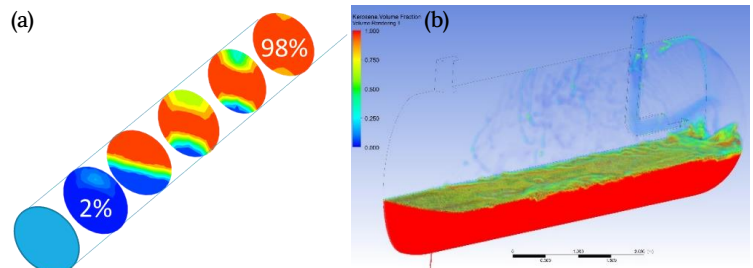


Figure 22 CFD applications in oil and gas industry, 2016 (a-multi-product pipelines, b- Two-phase horizontal separator)

### Other CFD projects:

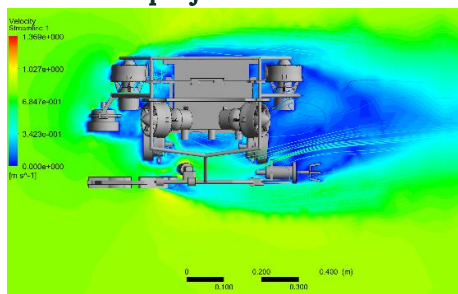


Figure 23 CFD simulation on ROV, 2017 (water flow of 1m/s)

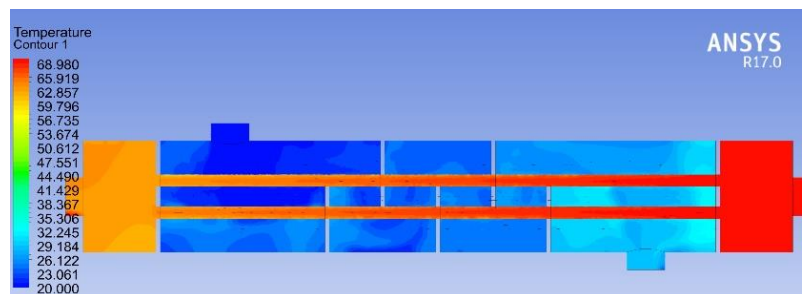


Figure 24 Heat exchanger, Baffle arrangements study, 2018