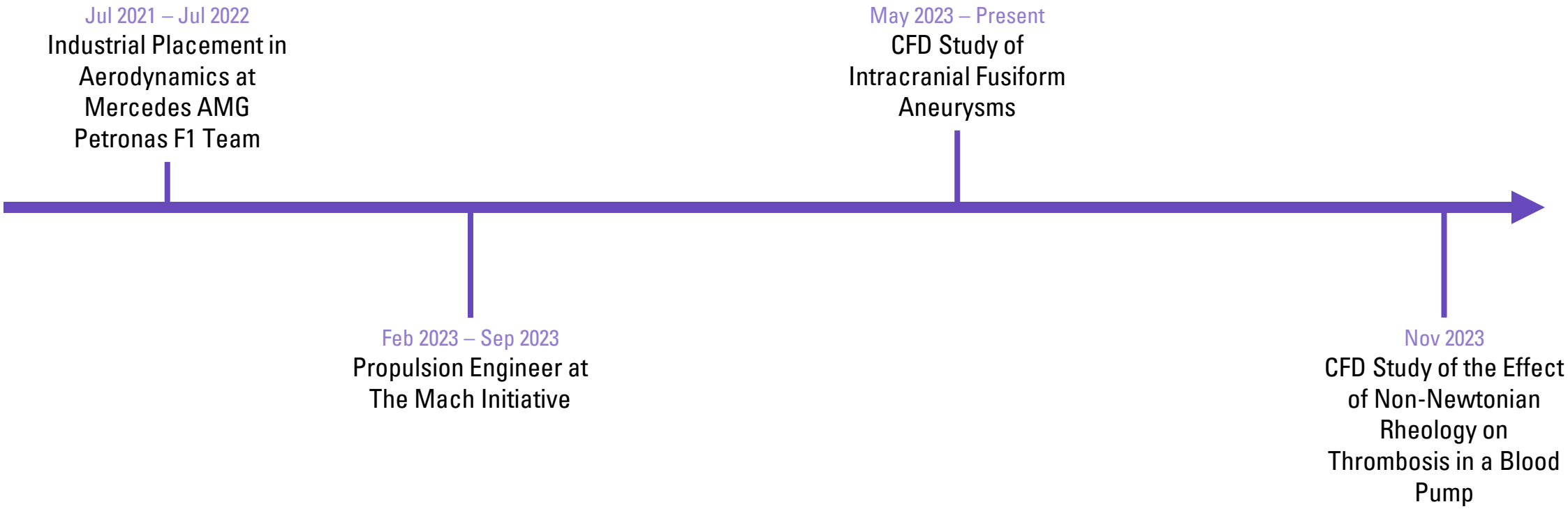
Romesh Thayalan

Project Portfolio

Key Projects Timeline

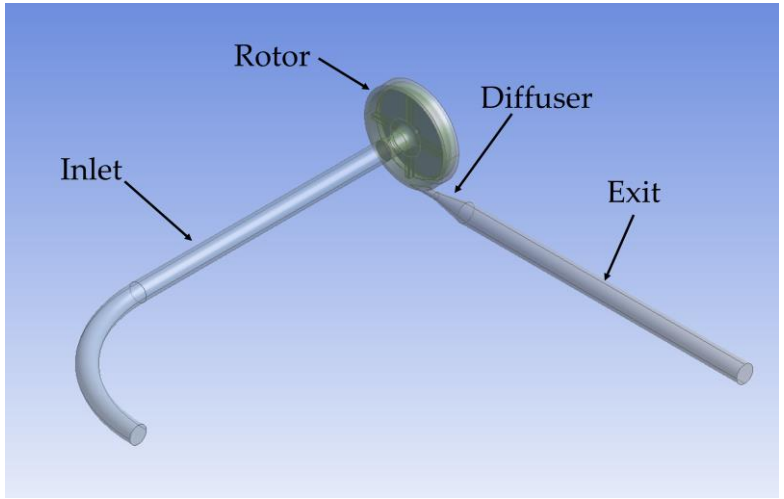


CFD Study of Thrombosis in a Blood Pump – 1st (83%)

CFD Study of Thrombosis in a Blood Pump

What?

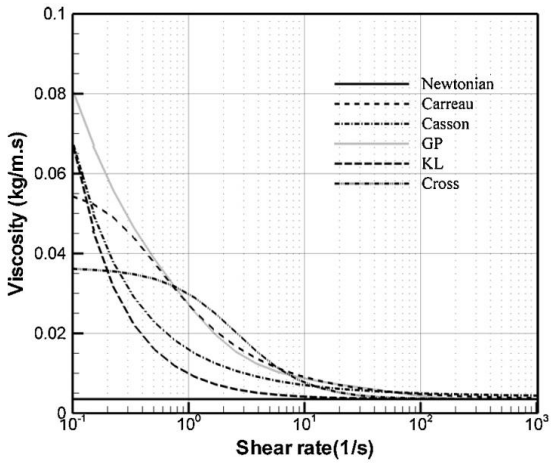
- Patients with heart failure may require Ventricular Assist Devices (VADs) as a bridge until a donor heart becomes available
- The FDA has provided benchmark tests for a simplified centrifugal blood pump model to validate computational models.



CFD Study of Thrombosis in a Blood Pump

Why?

- Rotor thrombosis is a potential issue that can lead to reduced VAD performance or thrombus being causing occlusion of vessels leading to stroke
- The high shear in the rotor activates platelets. Any areas distal (downstream) with low shear rates such as in recirculation zones, will lead to platelet deposition and so thrombosis.
- Blood is a non-Newtonian fluid exhibiting shear-thinning behaviour – its viscosity reduces with shear. Most studies do not account for this.



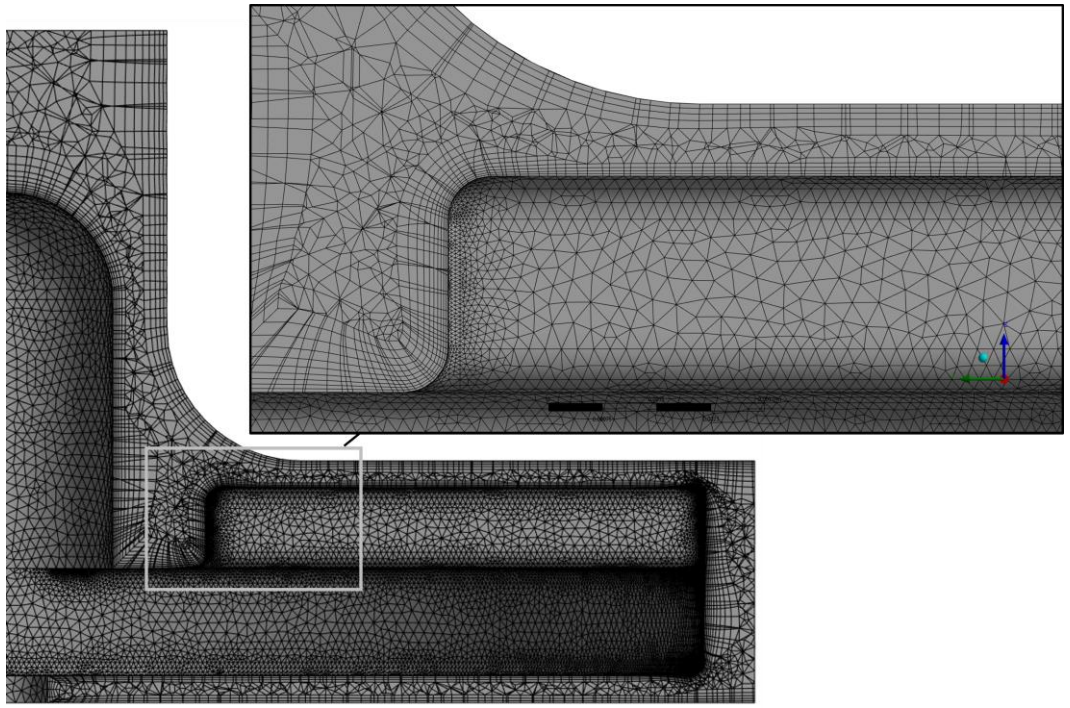
CFD Study of Thrombosis in a Blood Pump

How?

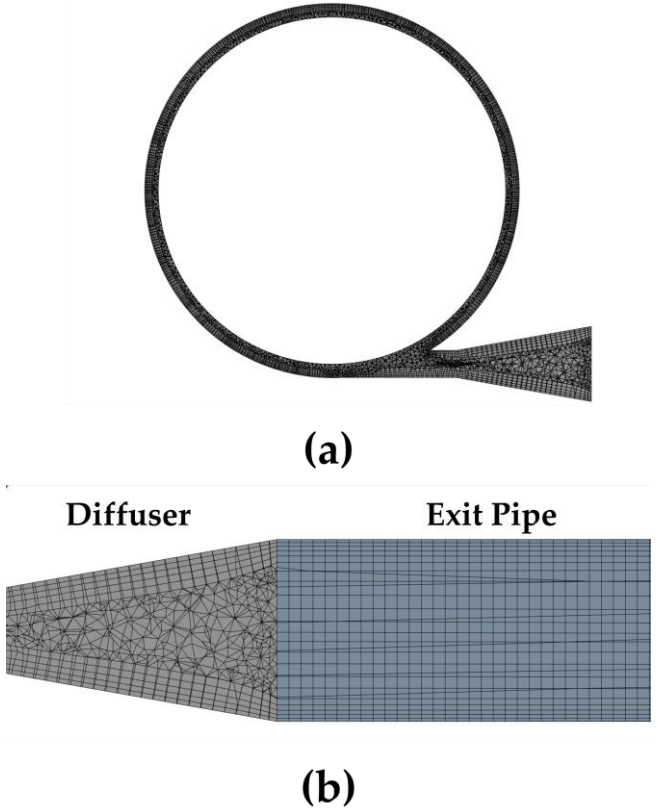
- A numerical model was created in Ansys (CFX) for a centrifugal Ventricular Assist Device (VAD) modelling the system as a steady state problem using a mixed reference frames (stationary and rotating)
- Grid independence studies were carried out and the model was validated against experimental data, including Particle Image Velocimetry (PIV) and pressure head data.
- A shear-thinning model was implemented (Carreau) and the volume of fluid below a critical shear-rate for thrombosis were compared.

CFD Study of Thrombosis in a Blood Pump

Discretisation



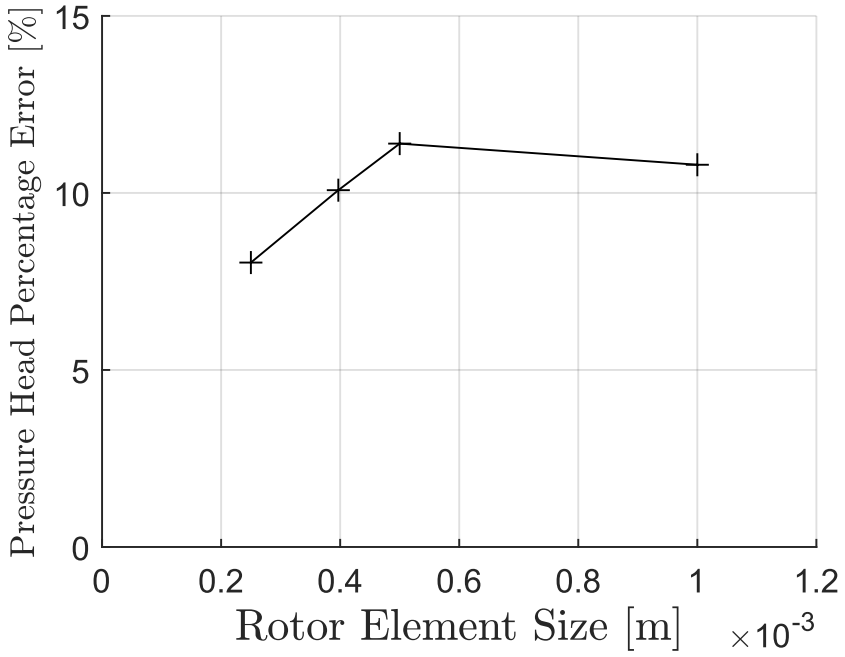
Rotor Grid



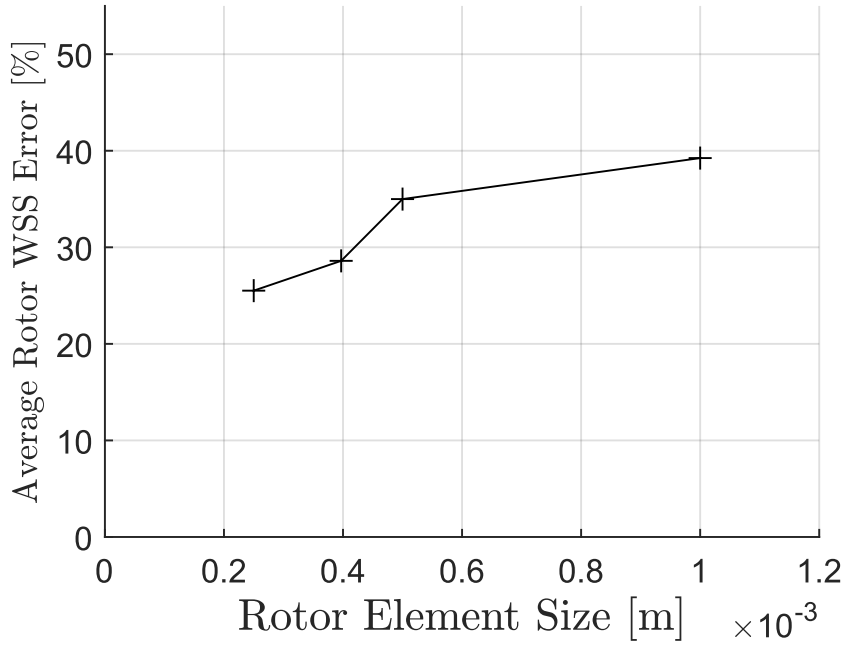
Exit Grid – conformal tetrahedral and hexahedral elements in the diffuser and exit respectively

CFD Study of Thrombosis in a Blood Pump

Discretisation



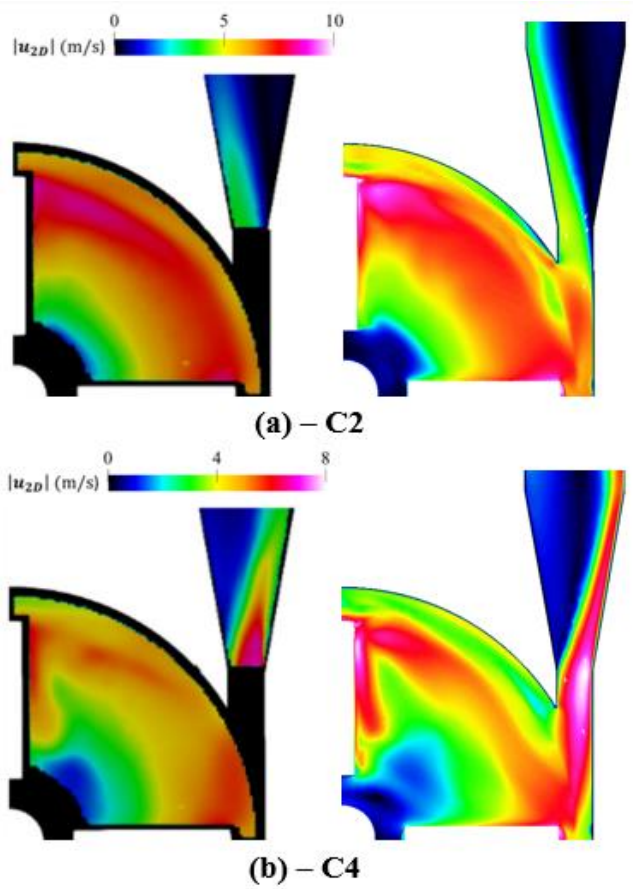
Grid Size Effect on Pump Pressure Head



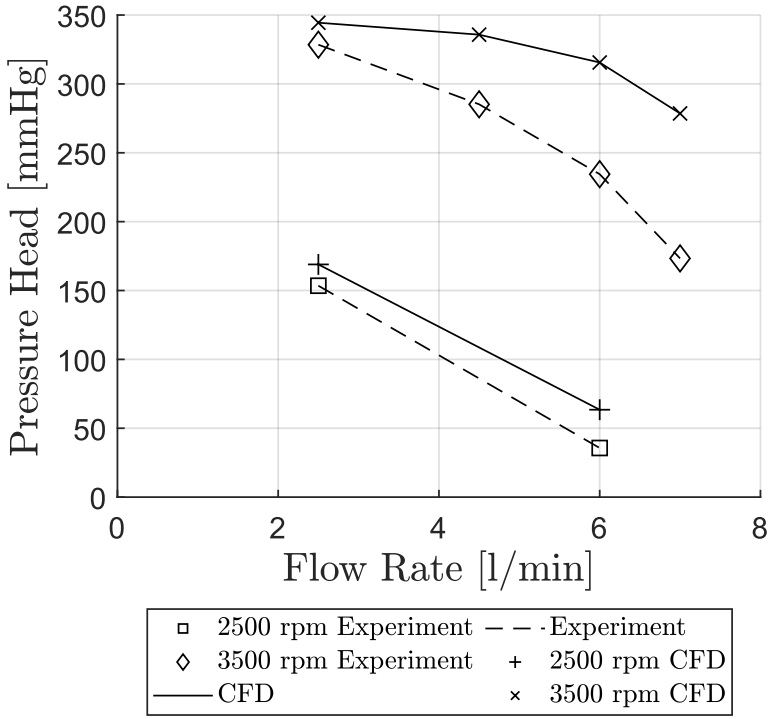
Grid Size Effect on Rotor Wall Shear

CFD Study of Thrombosis in a Blood Pump

Validation



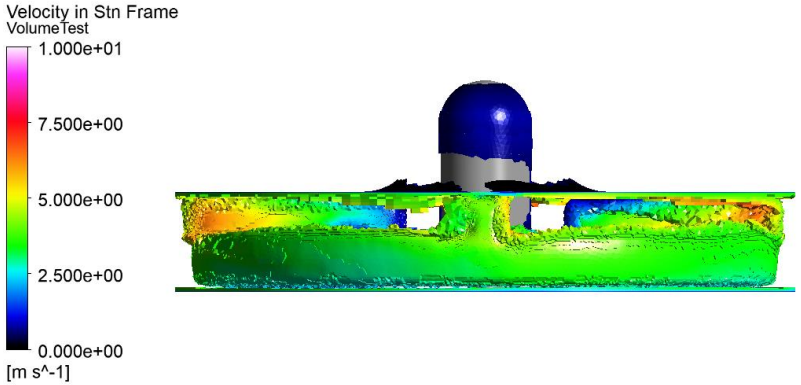
PIV CFD



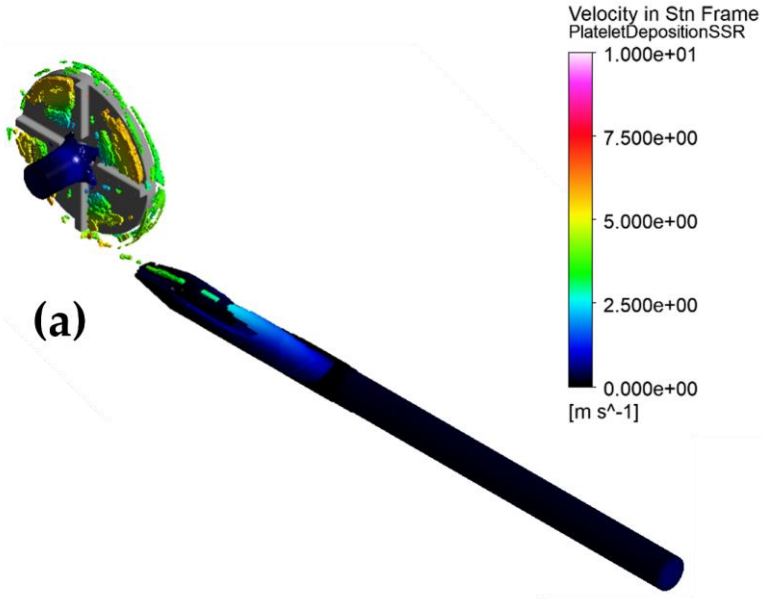
Pump Pressure Head against flow rate compared between experiment and CFD

CFD Study of Thrombosis in a Blood Pump

Results



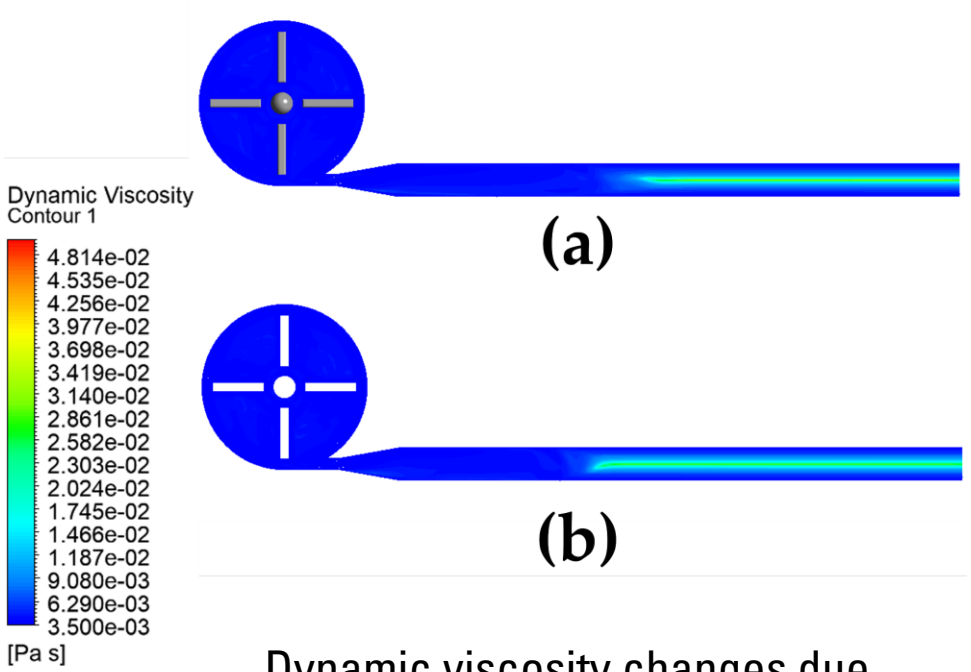
High shear regions concentrated around rotor – especially the blades



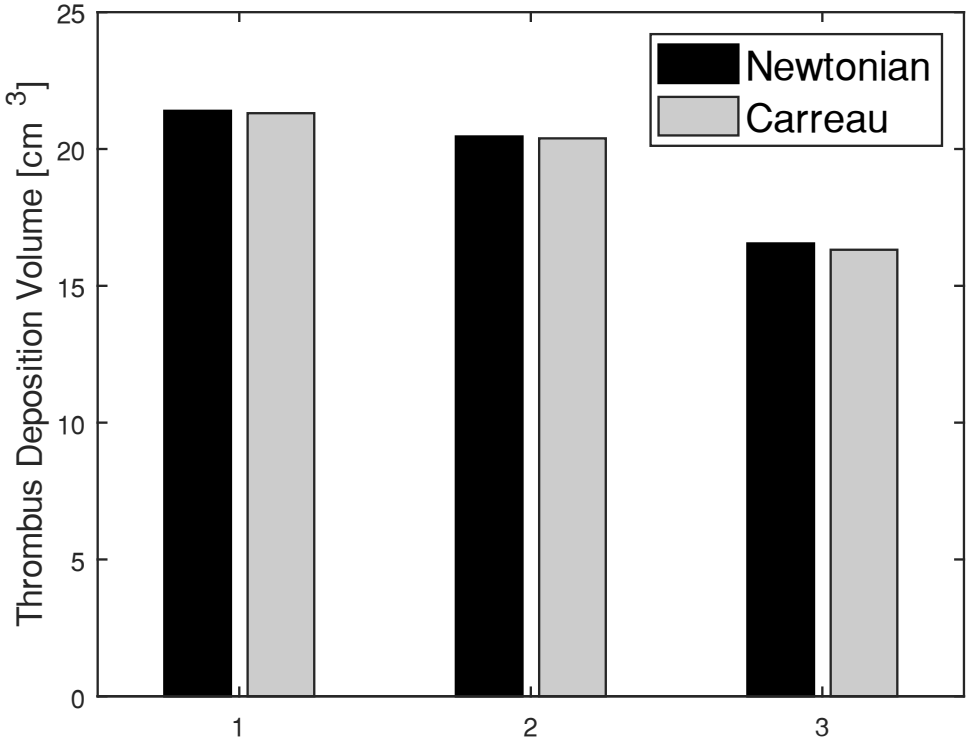
Low shear areas concentrated in the regions between rotor blades and in the exit

CFD Study of Thrombosis in a Blood Pump

Results



Dynamic viscosity changes due to non-Newtonian model concentrated in distal end of exit



For the conditions tested, the shear-thinning model had only a minor impact

Conclusion: Omission of shear-thinning models is justified for thrombosis potential prediction in centrifugal blood pumps

CFD Study of Intracranial Fusiform Aneurysms

May 2023 → Present

CFD Study Intracranial Fusiform Aneurysms

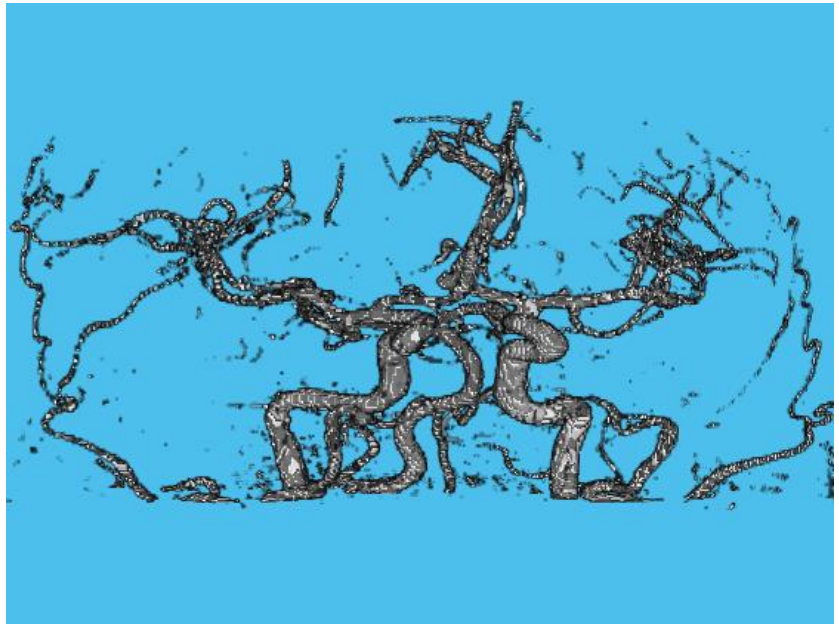
What?

- Intracranial fusiform aneurysms (IFAs) affect 3-13% of aneurysm patients.
- Flow instabilities in aneurysms may be the cause of growth which can lead to rupture and haemorrhage.
- Blood's shear-thinning behaviour may have an impact in supressing these flow instabilities.
- Accurate patient-specific simulations may one day be used clinically to determine a therapy plan.

CFD Study Intracranial Fusiform Aneurysms

So far...

- Processing of MRI scans of brain vessels using MATLAB.
- Literature review of current understanding of aneurysm haemodynamics
- Project plan for February – May 2024
- CFD study of blood pump to enhance non-Newtonian modelling knowledge



Processed MRI data of brain vessels

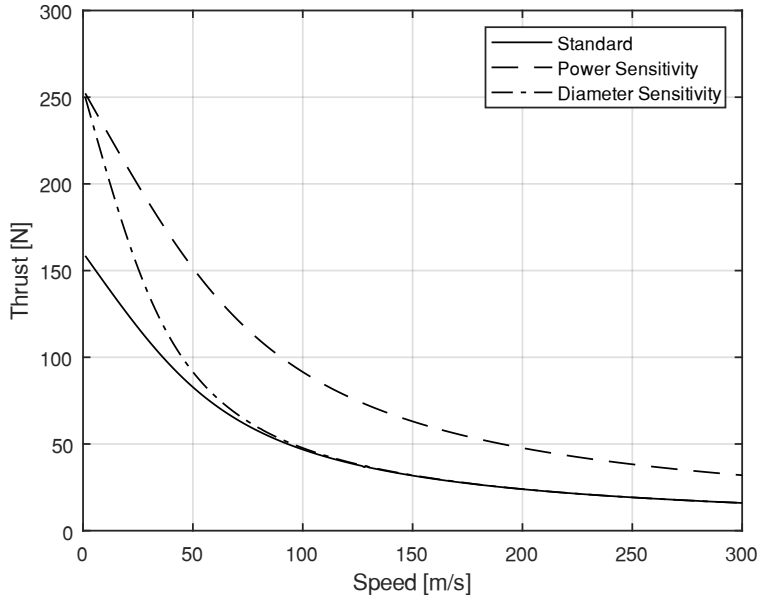
Propulsion System for a High-Speed Unmanned Aircraft 1st (71.4%)

Feb 2023 → Sep 2023

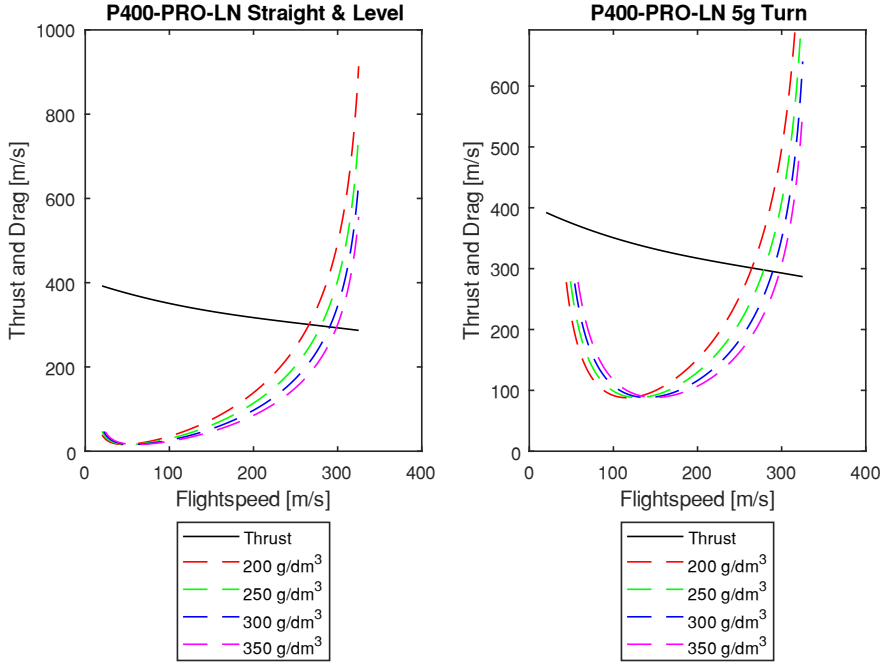
Propulsion System for a High-Speed Unmanned Aircraft

1D Modelling of Propulsion Systems

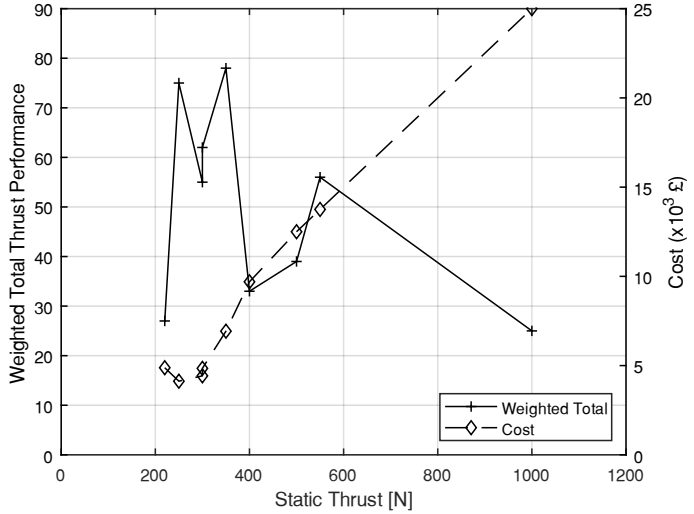
- Models of propellers and turbojets created from theory in MATLAB. Drag models from aerodynamics engineer were incorporated. These models were integrated into the performance models for mission profile planning.



Propeller thrust against speed assessing different diameters and power inputs



Turbojet model incorporating thrust (black) & drag estimates (colours)

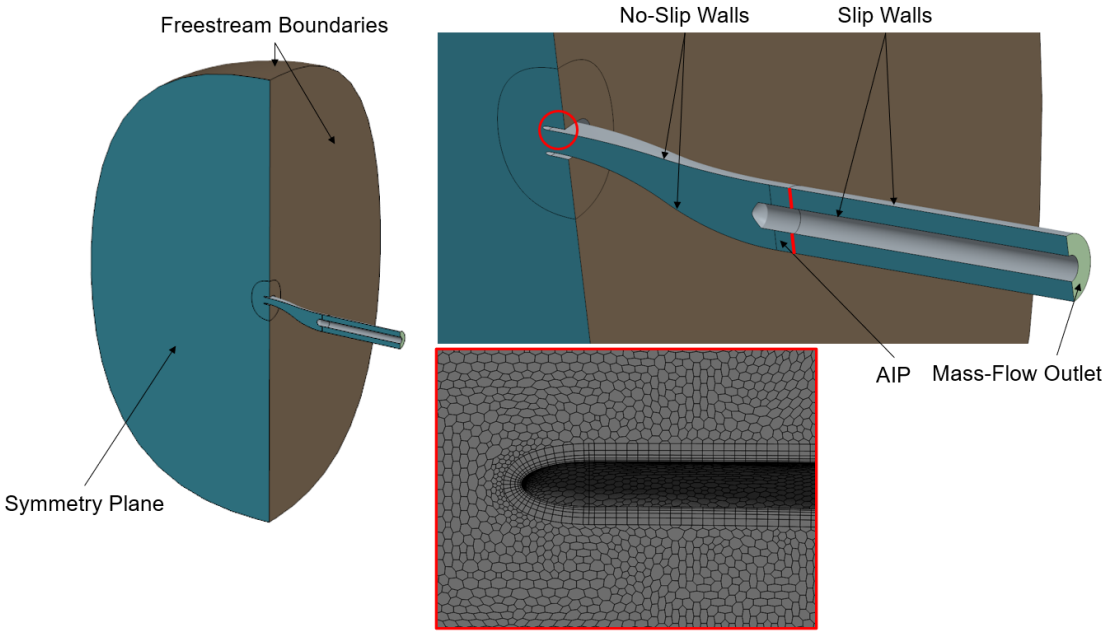


A weighted rank system was used to select the correct engine

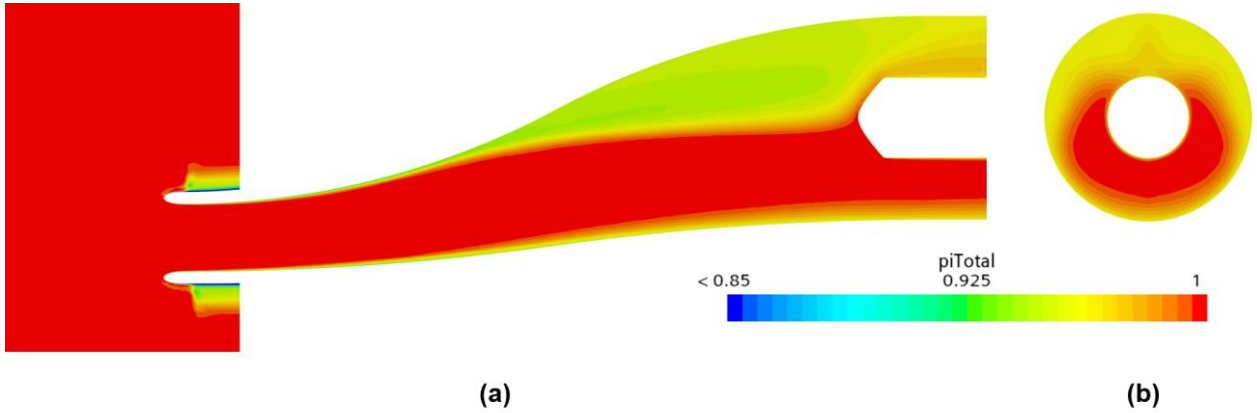
Propulsion System for a High-Speed Unmanned Aircraft

3D CFD Model of S-Duct Inlet

- The S-duct was designed in Autodesk Fusion 360 and discretised in Star-CCM+. RANS simulations were performed. A large separation was identified.



CAD and CFD model of the S-duct inlet

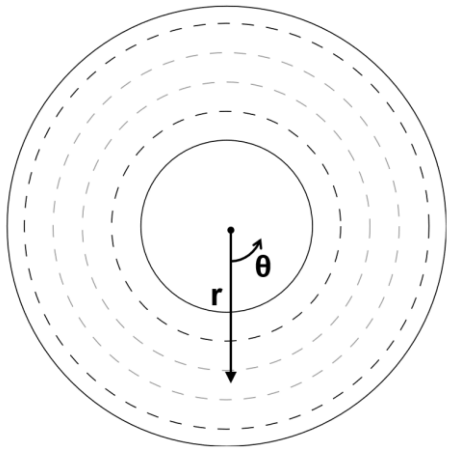


CFD revealed a large separation at the inflection point reducing total pressure at the engine face

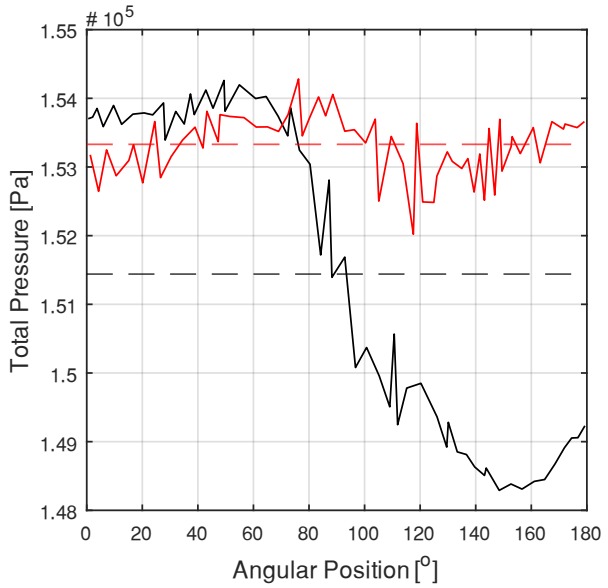
Propulsion System for a High-Speed Unmanned Aircraft

3D CFD Model of S-Duct Inlet

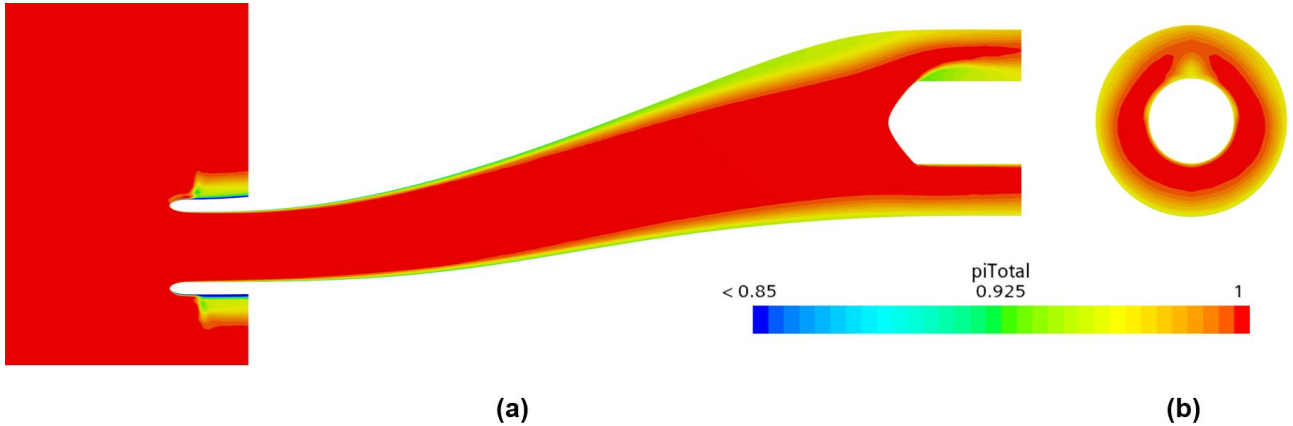
- The design of the s-duct curvature and area distribution was modified reducing the separation. The distortion of total pressure on the engine face was mapped on concentric rings showing more uniform onset.



Schematic of where rings on which distortion was measured



Total pressure becomes more uniform with the new design (red)



Improvement to the area and curvature distribution improved this separation increasing energy to the engine

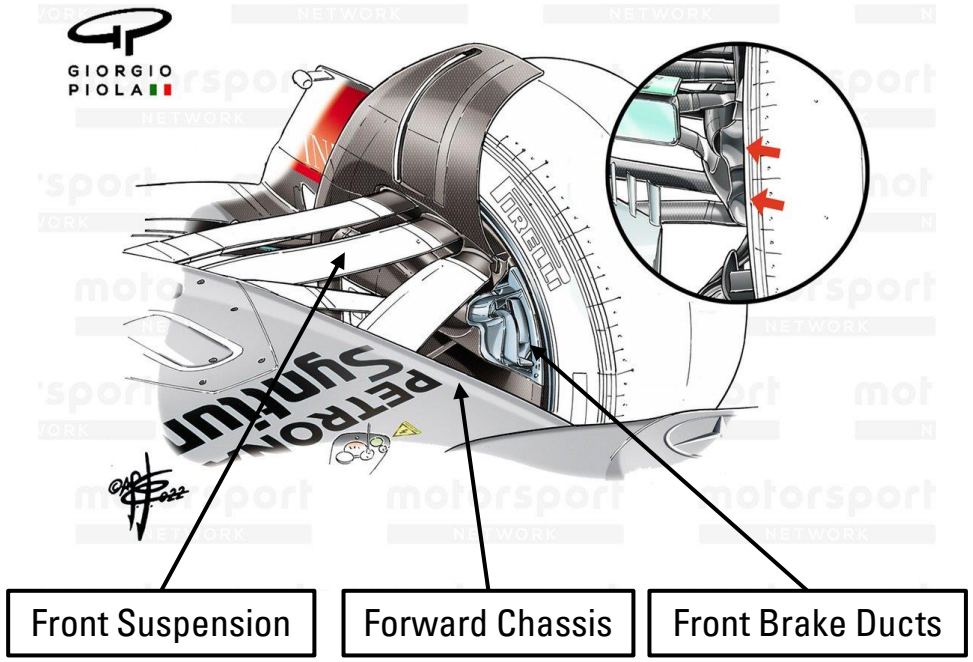
Industrial Placement in Aerodynamics at Mercedes AMG Petronas F1 Team

Jul 2021 → Jul 2022

Industrial Placement in Aerodynamics at Mercedes AMG Petronas F1 Team (Under NDA)

Front End Aerodynamics Development

- Contributed to development of front suspension, brake cooling ducts, and forward chassis.
- Used CFD to analyse options designed in CATIA 3DX.
- Validated CFD with wind tunnel tests tracking correlation.
- Created performance analytics trackers in Python to track how various flow field parameters related to performance.
- Independently ran wind tunnel test campaigns acting as the “Aero On Shift”
    - Ensured model and test quality
    - Oversaw and assisted in model assembly
    - Modified test-plan as needed
    - Presented test-plan to the group in the run up to wind tunnel sessions



Jul 2021 → Jul 2022