Modelling of a ducted fan and its contribution to drag on a PAV hull

Laurens Koers s1252089
University of Twente
Faculty of Mechanical Engineering, Thermal and Fluid Engineering
supervisor: Prof. H.W.M Hoeijmakers

RMIT University department of engineering , Melbourne Australia
external supervisor: Prof. C. Bil
period: 3/9/18-3/1/19

January 2019

Contents

Chapter 1: Summary

In this internship report, it will be discussed how the main research question "What is the contribution of the ducted propellers' wake on the aerodynamic drag experienced by the PAV" was answered. The PAV, or Personal Air Vehicle, has been the subject of earlier research and design. Different geometrical models of the PAV and parts were made in Solidworks, based on earlier designs, to be used later in aerodynamic simulations. The amount of drag on the PAV when in flight is determined first. After which different propeller setups, with either 3 or 4 blades, and with or without a duct, were simulated to find out how the propeller performance differs for these setups. Due to certain limitations in computational power, these results are a mere indications and not necessarily accurate. The 4-bladed, non-ducted propeller would turn out to provide the highest amount of thrust for a given angular velocity. Finally, the wake on the PAV was modelled to be a as a constant velocity airflow coming from a disk located at the point where the propeller would be. This velocity was calculated by a trust equation and the estimated amount of aerodynamic drag is $2.6 \cdot 10^2 N$. It is important to note that these results are not as accurate as could be and need refining for definitive values.

Chapter 2: Introduction

This report is written to finalise the internship conducted by me at RMIT University in Melbourne, Australia for the duration of 4 months. The internship is part of the curriculum of the second year of the masters study of 'Mechanical Engineering' and the 'Thermal and Fluid Mechanics' track at the University of Twente, Enschede, The Netherlands.

The objective of this internship was to investigate the influence of the wake of a ducted fan, which is used as the propulsion system of a Personal Air Vehicle (PAV), on the hull of this PAV. The PAV, which is still in the design phase, is meant to provide a new form of futuristic transportation for a single person. Other students have conducted earlier work with respect to the design of the PAV and ducted fan designs based on a set of requirements for this new form of transport. In addition, models were made of the PAV in different flying modes with the intention to determine its performance in flight and calculate the experienced drag while in flight. This design, both the hull and propulsion system, was recreated using CAD-software (Solidworks) and Computational Fluid Dynamics solver (Ansys Workbench/Fluent).

Chapter 3: Problem definition

The main goal of this internship was to get some insight in how the wake of the ducted propeller contributes to the total amount of drag the PAV experiences. The answer to this question will hopefully shed some light on the previously made design and perhaps conclude if it was well done or needs adjusting. Thus, the main question that needs to be answered is:

> "What is the contribution of the ducted propellers' wake on the aerodynamic drag experienced by the PAV?"

For this question to be answered a geometric model of the PAV has to be created. This includes the hull, propellers and ducts and will have to be based on the sketches and a table of dimensions provided by Chang Ryan Jia Meng[1]. Also, it is necessary to determine the drag force on the PAV in flying mode before the additional drag from the propeller can be calculated

In addition to this, it will also be investigated/verified if the presence of a duct around the propeller indeed results in a higher amount of lift achieved by a non-ducted propeller at the same angular velocity. Also, as is seen in Meng[1], the initial PAV design uses a 3 bladed propeller which was concluded as the optimal amount of blades for this particular purpose and a maximum angular velocity was given as well. These findings should be be tested to see if they were correct. So the tasks that need to be done before the main question can be answered are:

- create a model of the PAV
- create a model of the propeller and duct
- calculate the aerodynamic drag on the PAV while in flight
- verify the correctness and/or accuracy of blade design and operating rotational velocity
- verify that the presence of a duct around the fan increases its lift

Chapter 4: Methodology

The geometric models of the PAV, propeller and duct were made using Solidworks. The models were based on the given dimensions and sketches given by Meng[1] in which a preliminary design was conducted of this PAV based on its requirements. For the parts of the PAV for which the dimensions were not readily available, their dimensions were estimated using a tool in the Solidworks software which uses a known dimension of a sketch to estimate other dimensions. The propellerblade cross-section that will be used is a NACA 2412 profile. This airfoil shape was selected by Xu[2] because of its suitable properties. A graph plotting the pitch angle and chord length to the radial position of the blade was included and this together with the chosen airfoil and some insights from Anderson[3] was used to create a model of the propeller. For the duct, a cross section was picked from Xu[2] for the sole reason it was the only one in the paper with provided dimensions that made the modelling of it easier.

After the models of the PAV and its part are finished, models of the domain will have to be made that will be suitable for the Ansys Fluent solver to use. Hale[4] gave some good insights on how to do this. Basically, the Fluent solver needs a domain in which the fluid (air in this case) is represented as a 'solid' and the actual solid (PAV and parts) are subtracted from this domain creating a 'negative' of the solids. To create this domain, it has to be kept in mind that certain dimension have to be such that the appropriate boundary conditions can be selected on the edges of this domain.

These domains will then need to be discretized to create many tiny volumes in the entire domain. The discretization is done by using the Ansys meshing tool in which volumes and areas in the domain can be subdivided into these little volumes. Different volumes, areas and surfaces were given an appropriate name so boundary conditions can be easily applied in the fluid solver.

A few different type of simulations were then performed. A time steady simulation of the PAV in flying mode with the intention of calculating the aerodynamic drag when in flight. A transient simulation of the rotor to investigate the amount of lift it experiences and also to retrieve an exit flow velocity profile which can then be inserted in the 3^{rd} simulation. A transient simulation in which the contribution of the propeller exit flow on the amount of drag is determined.

Chapter 5: Results

5.1 PAV and propeller model

The model of the PAV and propeller can be seen below. The duct is not included for now to not overcrowd the paper with redundant figures. This PAV model was made by using the dimensions and sketches of the earlier paper, as can be seen in figure(5.1):

PAV Single mode

Propulsion system	Ducted Fans
Weight (Airframe including fans & shroud)	N/A
Width (Cockpit only)	800 mm
Width (Whole Aircraft)	1800 mm
Height (Cockpit only)	1120 mm
Height (Whole Aircraft)	1535 mm
Length	1520mm

Figure 5.1: table of PAV dimension

Using Solidworks, the PAV geometry was recreated and often reworked until the shape was achieved that resembled the required design, figure(5.3). It was stated in the requirements by Meng[1] that the PAV should only be able to carry one single person and fit into a regular garage. The design of the PAV has been adjusted slightly to be a little less cumbersome and more aerodynamic looking at its front.

While the modeling of the PAV hull went relatively smooth, making the model of the propeller proved surprisingly difficult. The main obstacles were the smooting of the blade tips and creating a curved trailing edge that the meshing programm would accept to use. After quite a few trials, the result can be seen in figure(5.3). The propellers are attached to the two cylindrical rods and were designed according to the following chord length and pitch angle distribution:

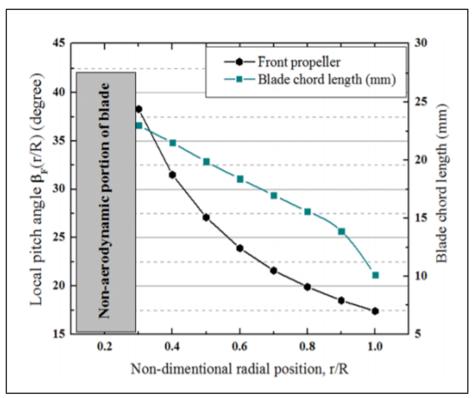


Figure 5.2: pitch angle and chord length distribution

The propeller has a radius of $r = 0.54m$ and a hub has a radius of 0.06. So the final design does differ slightly from Figure(5.2)

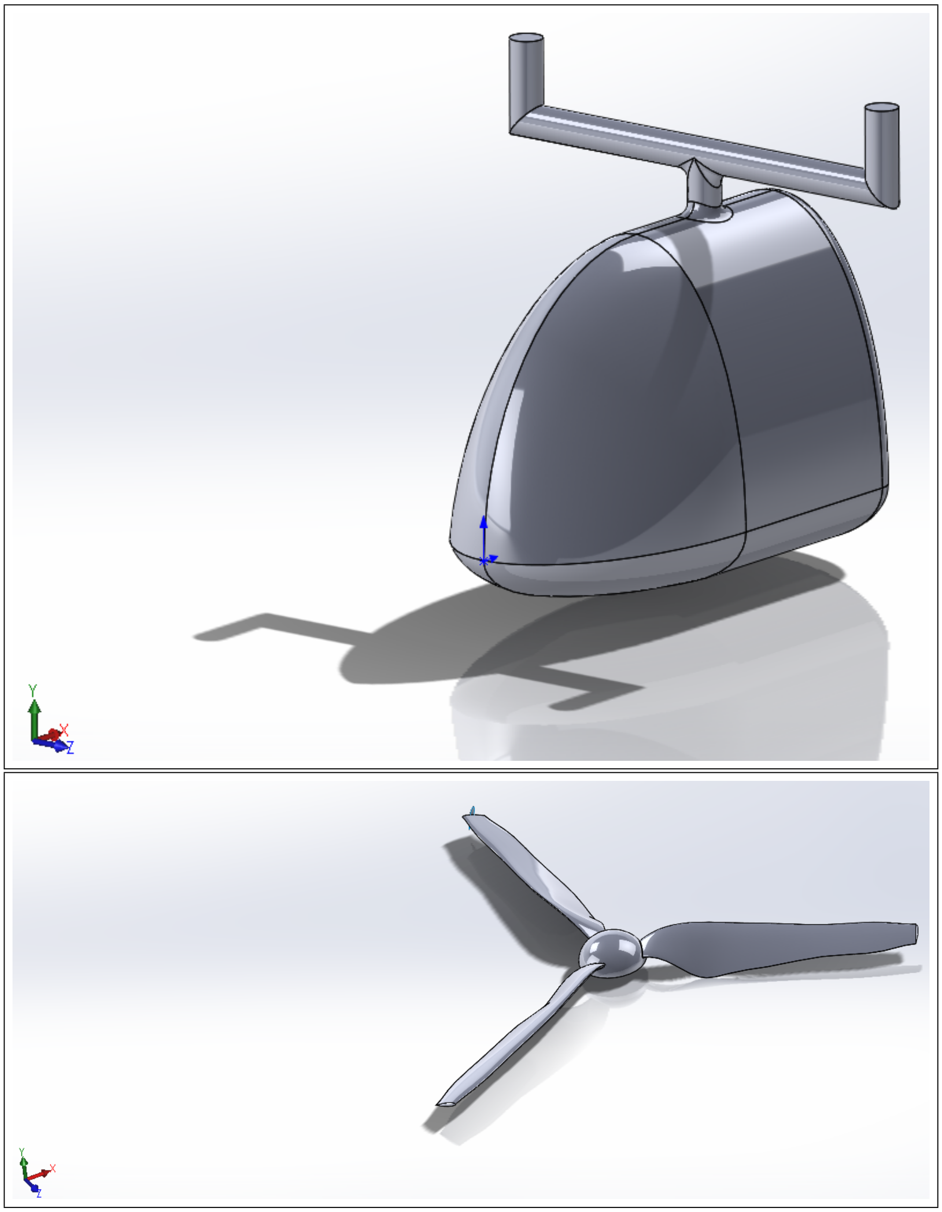


Figure 5.3: model of PAV and propeller

5.2 Performance calculations

Before the actual simulations were performed, some initial calculations were done to find out the required thrust the propeller needs to achieve for hovering and flying modes and also required angle of the propeller when in flight. Also this is done to check if the given angular velocity of the propeller is able to deliver this required thrust. It is given that the PAV has an estimated weight of $486.6kg$ or $4774N$. Since the PAV uses two propellers, each propeller should provide half of this amount of thrust for the PAV to hover, $2387N$. Since no requirement for the acceleration was given, it was assumed that the PAV should be able to accelerate with $5\frac{m}{s^2}$, about the same as an average car. Then, according to Newtons' second law, the amount of thrust one propeller needs to deliver to accelerate the PAV in x-direction is:

$$F_x = \frac{m \cdot a}{2} = \frac{486.6 \cdot 5}{2} = 1216.5N \tag{5.1}$$

Meaning the total amount of thrust needed by a single propeller to keep the PAV in the air and accelerate it is:

$$F_{total} = \sqrt{2387^2 + 1216.5^2} = 2679N \tag{5.2}$$

In reality, the total amount of thrust in x-direction will be a bit more to cope with the air resistance during acceleration. In the next section, this drag force will be calculated at cruising speed.

The angle at which the propeller needs to be tilted to accelerate will then be:

$$\theta = sin^{-1}(\frac{F_x}{F_{tot}}) = sin^{-1}(\frac{1216.5}{2679}) = 27^{\circ} \tag{5.3}$$

along the z-axis according to the right hand rule and the coordinate system as seen in figure(5.3).

5.3 Drag force on PAV in cruising flight

The first simulation that was performed is done to calculate the amount of drag on the PAV when it flies at cruising speed. The cruising speed is desired to be $80km/h$ or $22.22m/s$. For this simulation the domain was split along the symmetry line since this reduces computational time and to retain accurate solutions while using software limited in discrete elements. A mesh inflation layer was included on the PAVs surface in an attempt to capture boundary layer effects more realistically and a curvature size function was used with a small maximum angle of $6°$ was used to more accurately mesh curved surfaces. For this simulation, the PAV was angled $15°$ downwards to simulate a more realistic orientation while flying. Please note that in figure(5.5b) the inside of the PAV seems meshed but this is actually the surface mesh and not the interior. A time steady simulation was performed in with a laminar viscosity model. Other, more advanced, viscosity models are available but the main task of this simulation is to calculate the amount of drag force on the PAV while in flight and therefor the laminar model should do. The total amount of drag during flight was calculated to be $F_d = 70.8N$, and the convergence process for this value can be seen in Figure(5.4):

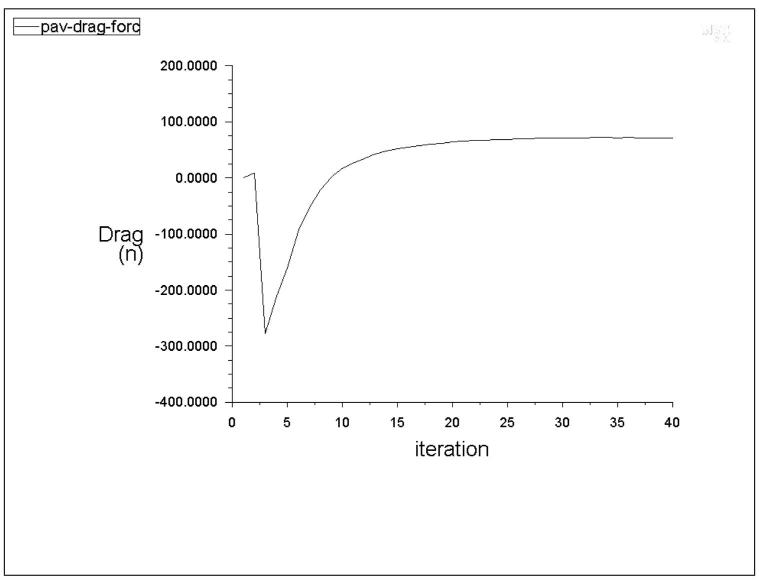
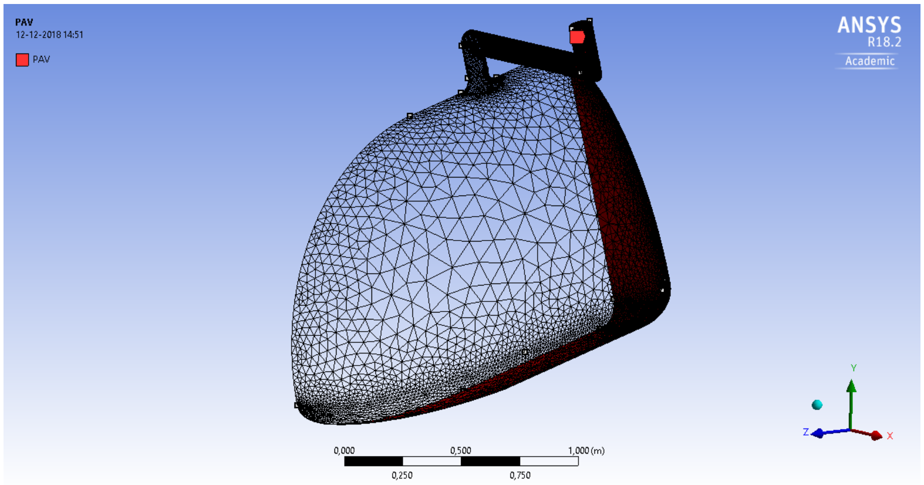
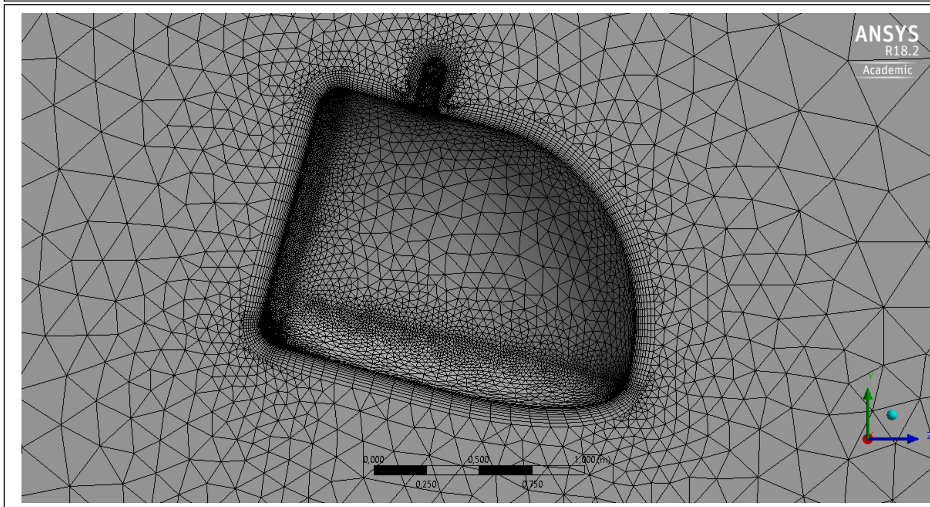


Figure 5.4: Convergence of viscous drag force on PAV

The convergence of the residuals can be seen in Appendix(A).



(a) PAV surface mesh



(b) Mesh around PAV

Figure 5.5: mesh on and around PAV

5.4 Comparison of propeller performances

In this section, different propeller setups will be compared. The goal is to find out if the presence of a duct around the propeller does indeed provide extra lift it is intended to as well as to check if the proposed 3 blades is indeed the optimum amount of blades for the PAV. To do this, the *Moving Reference Frame*-method (MRF) was used. This means that the propeller is part of a cylindrical cell zone which is given an angular velocity equal to that of the design velocity. This relatively small cylindrical cell zone rotates in a stationary larger cylindrical cell zone and these cell zone together make up the entire domain. A cross-section of the mesh of the 4-bladed propeller is shown in figure (5.6):

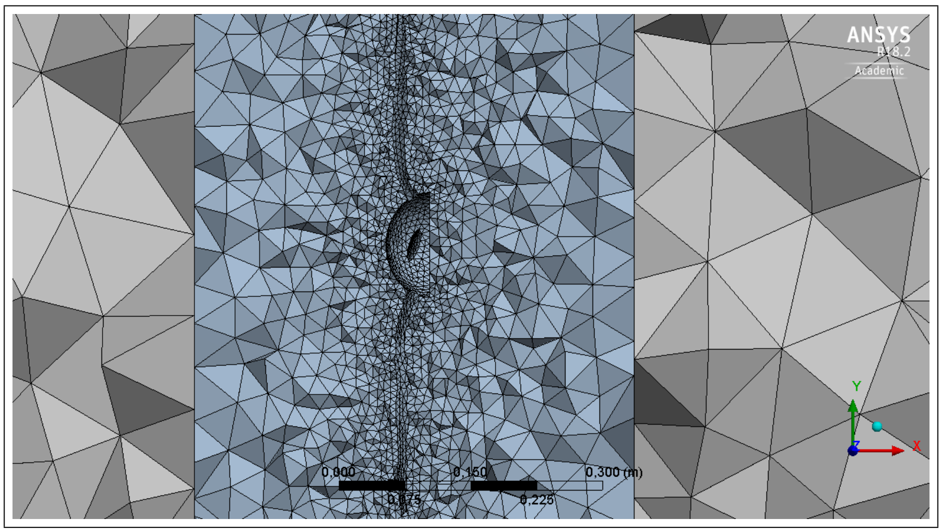


Figure 5.6: cross section of 4-bladed propeller mesh

The rotational domain is shaded blue and the stationary domain is shaded in grey. Inside of the blue area, the cross section of the propeller is visible. For accurate results, the surface of the propeller should have been covered in a mesh inflation layer which will accurately capture the flow boundary layer over the propeller and in turn, result in more accurate values of lift. An estimate of the inflation layer higher was made the following way.
The Reynolds number for flow over an airfoil an airfoil is given by:

$$Re = \frac{u_\infty c}{\nu} \tag{5.4}$$

in which u_∞ is the free stream velocity, c the chord length and ν the kinematic viscosity of the air. For a rotational velocity of $\omega = 4116$ rpm, radial position of $r = 0.54m$, chord length $c = 0.045m$ at this radial position and a kinematic viscosity of $1.568 \cdot 10^{-5} \frac{m^2}{s}$ the Reynolds number is $Re = 6.68 \cdot 10^5$. This means the flow over the tip of the blade is turbulent so the following formula for turbulent boundary layer thickness over a flat plate should be used:

$$\delta \approx \frac{0.37x}{Re_x^{1/5}} = \frac{0.37 \cdot 0.045}{(6.68 \cdot 10^5)^{1/5}} = 1.45 \cdot 10^{-3} m \quad (5.5)$$

Unfortunately, it would turn out that implementing an inflation layer of this height would exceed the allowed amount of elements in the fluid solver. Thus the simulation had to be performed with a mesh of less than satisfactory quality.

With the use of this mesh the transient simulation will then be carried out. The small cylinder that contains the propeller geometry rotates with an angular velocity of 4116 rpm or 431 $\frac{rad}{s}$. This simulation start by using very small time steps of $10^{-3}s$ so that at each time steps, the propeller makes $\frac{1}{14}$th of a rotation. It is important to start a simulation like this with small rotations at first for more accurate results. During each time step, the solver makes between 50 and 100 iterations, depending on what was observed during the simulation, to let the residuals (an indication of the error) converge to a hopefully small value. The smaller the value of the residual, the more accurate the result. After some time, the amount of lift the propeller experiences approaches a certain value that will then be the end result of the simulation. For the propeller simulations, the ideal gas law:

$$PV = nRT \quad (5.6)$$

will be considered to account for the compressibility effects that will occur at these high velocities. The tip of a propeller blade reaches velocities of:

$$v = \omega R = 431 \cdot 0.54 = 233 \frac{m}{s} \quad (5.7)$$

Which is equal to a Mach number of:

$$M = \frac{v}{c} = \frac{233}{340.3} = 0.685 \quad (5.8)$$

In which M is the Mach number, v the speed of the flow and c the speed of sound at sea level for this case. This is well above a Mach number of 0.3. Flows below this speed may be considered incompressible so for these simulations, the air must be considered compressible. For the viscosity model, the $k - \epsilon$ model was used which is a standard choice for modelling propellers.

Now that the most important parameters have been justified and/or explained, 4 simulations were run.

- A simulation of a 3-bladed propeller
- A simulation of a 4-bladed propeller
- A simulation of a 3-bladed propeller with duct
- A simulation of a 4 bladed-propeller with duct

For the first simulation of the 3-bladed propeller without the duct, the convergence process will be shown here, the convergence processes of the other simulations will be shown in Appendix(A).

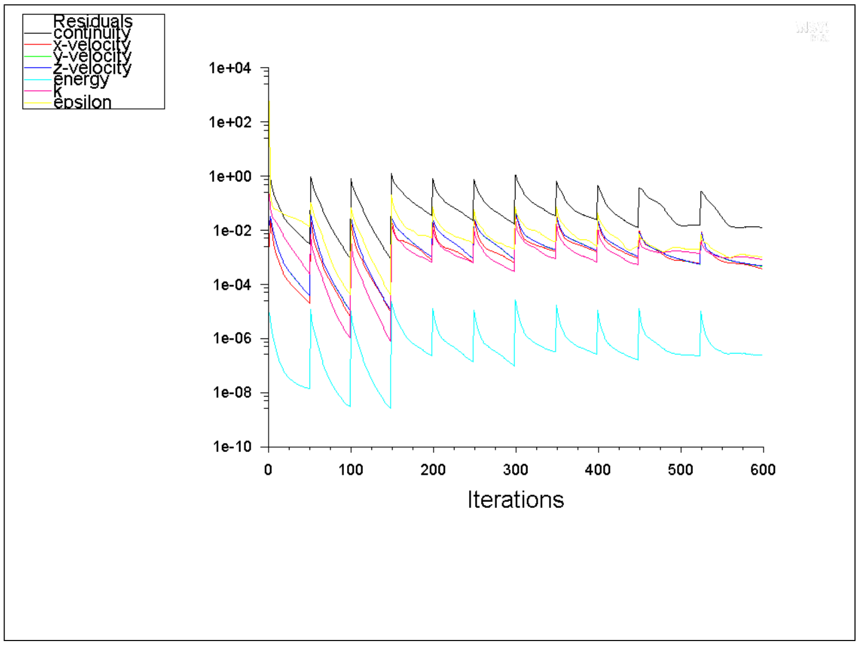
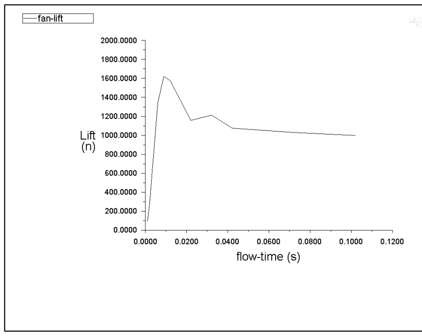
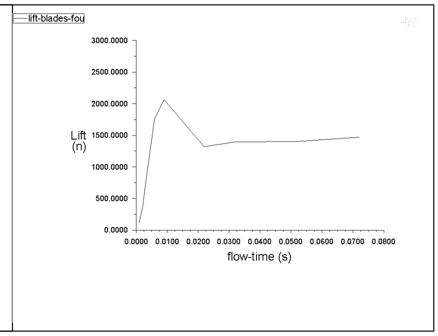


Figure 5.7: Convergence of residuals for 3-bladed propeller simulation

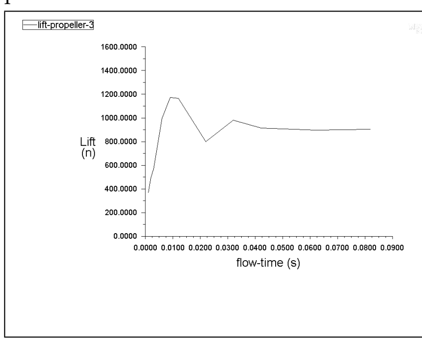
And in the figures below, the convergence of the amount of lift for the 4 simulations can be seen:



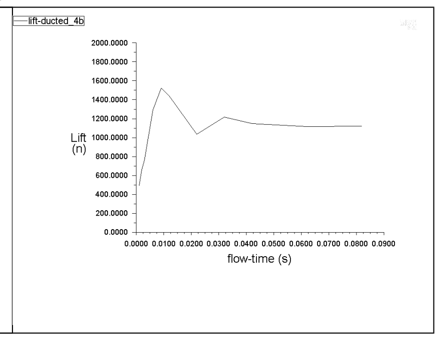
(a) Lift convergence of 3-bladed propeller



(b) Lift convergence of 4-bladed propeller



(c) Lift convergence of 3-bladed, ducted, propeller



(d) Lift convergence of 4-bladed, ducted, propeller

Figure 5.8: plots of lift convergence.

The exact results for the amount of lift achieved were not read from these plots but were printed on screen by the solver. These exact values can be found in Figure(5.9):

-	3 blades	4 blades	3 blades + duct	4 blades + duct
Lift(N)	997N	1500N	905N	1207N

Figure 5.9: table of lift results

These results show that in these simulations, the adding of a duct has had a negative influence on the amount of lift which is contradicting theory. The 4-bladed propeller also appeared to achieve 50% more lift than its 3-bladed counterpart.

For illustrative purposes, a figure of the air velocity in axial direction around the 4-bladed ducted propeller is shown in Figure (5.10). This figure shows the velocity field on in $z - y$-plane at the last time instant of the simulation.

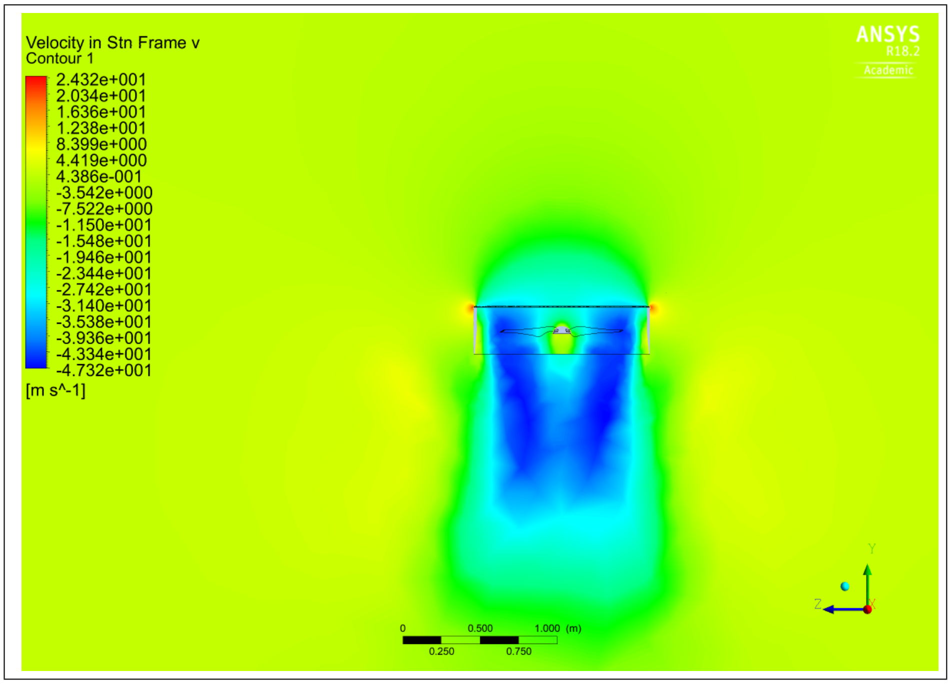


Figure 5.10: ducted propeller exit flow

Even though these simulations were performed with a coarse mesh, there are still some remarkable points to be seen in this figure that would have been seen in a fine mesh. For one, this figure shows that the air at the inlet part of the duct has a very high velocity gradient, transitioning quickly from the green to the red part. It also flows in positive y-direction, which means the air comes up from underneath the top part of the duct, which is an unfavourable result. The airflow around the hub and duct appears to have very little velocity in negative y-direction, which is what was expected. In the wake of the propeller, the air was accelerated up to $47.32m/s$ and smaller velocity gradients can be observed in front of the propeller and in the wake.

5.5 Drag on PAV

To determine the total amount of aerodynamic drag on the PAV, a time-steady simulation needs to be run in which the exit flow from the duct needs to be present. Because of the limit in elements, the idea was to get a time average velocity profile at the duct exit by performing a transient simulation and insert this velocity profile into the time steady simulation with the PAV. The velocity profile at the duct exit of the 4-bladed propeller, the one seen in Figure(5.10) is shown in Figure(5.11):

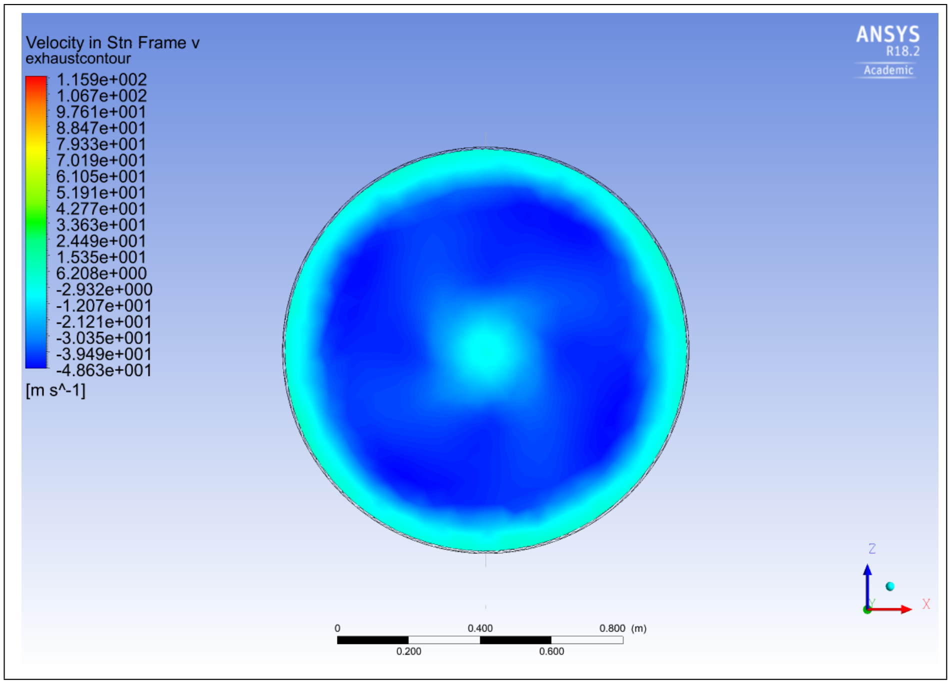


Figure 5.11: ducted propeller exit velocity profile

Unfortunately, it was not managed to take the time average of the exit profile during the simulation. However, the exit profile in Figure(5.11) seems to be quite axisymmetric, which was needed. This figure shows again how the flow near the hub and duct have a very small velocity. Inserting this velocity profile into a time steady simulation with the PAV proved difficult and was unfortunately not achieved. To circumvent this problem and to get some insight in the total amount of drag on the PAV, a simulation was performed in which the propeller exit flow was modelled by using a velocity inlet with a fixed amount of mass flow. This however did not include the duct. It was derived that for the hovering mode, a single propeller had to be able to provide $2387N$ of thrust. To calculate the required flow velocity, the following equation was used:

$$F_{thrust} = T = \frac{d}{dt}(mv) \tag{5.9}$$

Which for the case of accelerating air from standstill can be rewritten as:

$$T = \rho A v^2 \tag{5.10}$$

In which ρ is the air density, A the cross section of the duct and v the required velocity. Rewriting this equation and using the known values for T, ρ, A, v is found as:

$$v = \sqrt{\frac{T}{\rho A}} = \sqrt{\frac{2387}{1.225 \cdot 0.54^2 \pi}} = 46.1 m/s \tag{5.11}$$

Trying to simulate the drag in a steady-simulation turned out to be problematic as it did not converge properly. This was likely due to the fact that some problems are implicitly unsteady and thus a transient simulation was performed. In the same way as the propellers were simulated, the transient simulation started with small time steps that were gradually increased until a constant amount of force on the PAV was achieved. A laminar viscosity model was used and the air was considered incompressible. Unfortunately, this transient simulation suffered from poor continuity convergence. So the result of this simulation is can not be considered to be very accurate. It can however provide a small indication of the magnitude of drag.

Figures of the streamlines, contour of the amount of drag on the PAV, and the convergence of the drag force can be seen in figures (5.12), (5.13) and (5.14):

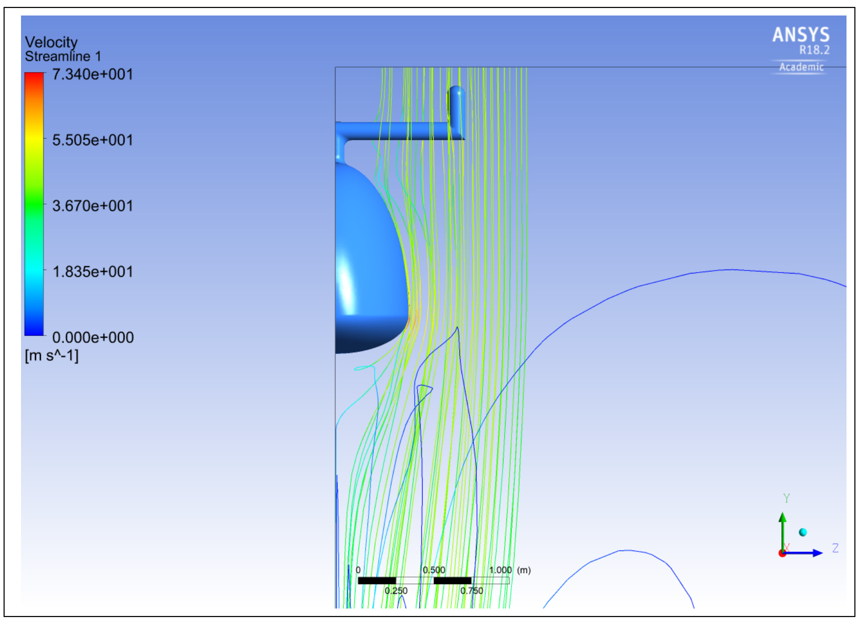


Figure 5.12: front view of propeller streamlines

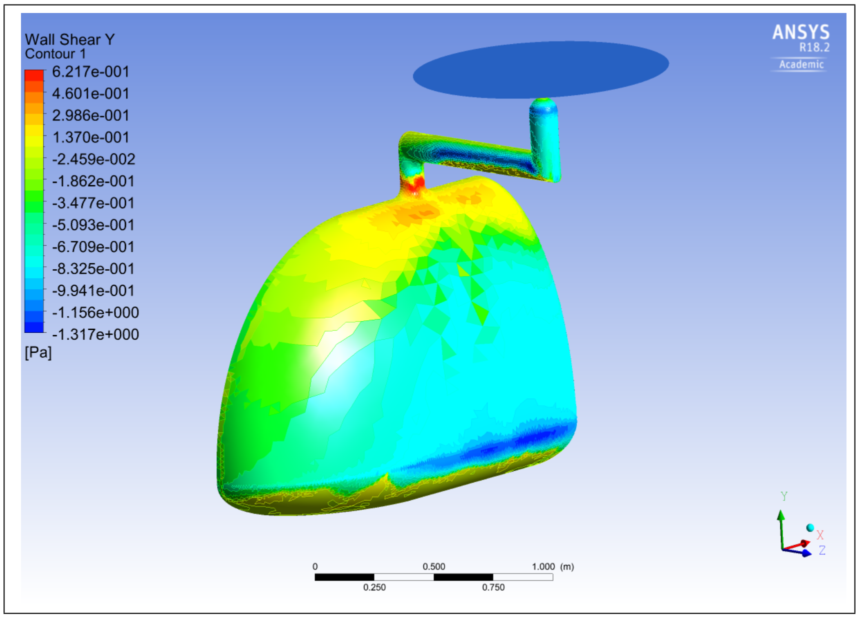


Figure 5.13: side view of surface contour of viscous drag

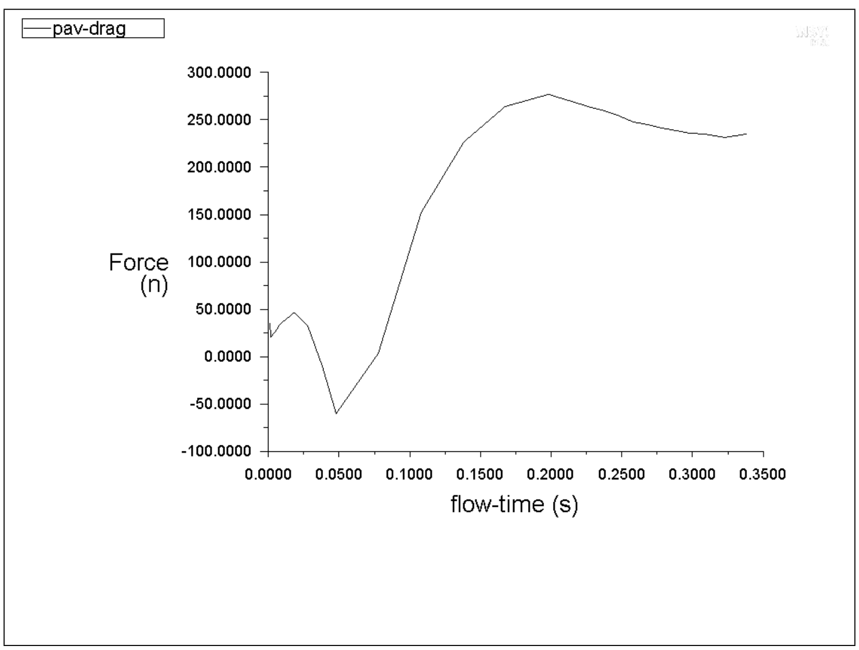


Figure 5.14: side view of surface contour of viscous drag

Figure(5.12) shows the streamlines, the path of a very small piece of air, around the PAV hull. The included legend shows a maximum absolute velocity of $73.4m/s$. It may not be quite visible in this plot but this maximum velocity occurs around the horizontal beam that supports the propellers. The air is accelerated briefly up to this velocity after which is decelerates again moving downwards to the PAV hull.

Figure(5.13) provides an image of where on the PAV the most amount of drag occurs. The horizontal beam, over which the air flows fastest, experiences the highest amount shear stress. On the PAV hull, the part just underneath the propeller has the highest shear stress. The red areas indicate a shear stress in positive y-direction, which is strange. This could be the consequence of a choice of boundary conditions that should have been chosen differently. Figure(5.14) shows a path to convergence. Quite likely due to the bad convergence of the continuity. In the final iterations, the amount of drag seems to converge to $2.6 \cdot 10^2 N$ which will be taken as the final result. The plot of residuals for this simulation can be found in the Appendix(A). It was not manage to perform a drag-simulation of the PAV in cruising light mode unfortunately. In conclusion, the answer to the main question:

> “What is the contribution of the ducted propellers’ wake on the aerodynamic drag experienced by the PAV?”

Will be about $\boxed{2.6 \cdot 10^2 N}$

Chapter 6: Conclusion

The geometrical models of the PAV and its parts that were based on earlier designs proved to be well suitable for the flow simulations that were done. After redesigning some part to make them more appropriate for the simulations, the final parts were used for all simulations that were discussed. Modelling the propeller, especially the blade tip, was the most challenging part of the modelling phase of this internship. In the end however, the propeller model was used for all propeller/duct-simulations and worked fine.

A few performance calculations were perfomed to get insight in what amount of thrust a propeller has to be able to generate for the PAV to hover and cruise. In addition, the angle at which a propeller has to be tilted to accelerate the PAV was determined. A propeller should provide a maximum of $2679N$ and should e tilted $27°$ degrees during acceleration.

The viscous drag simulation of the PAV was probably the most accurate of them all. Because of symmetry in the problem, the domain was cut in half reducing the amount of computational elements needed for a simulation. Even with the limit in computational elements, a decent inflation layer was created on the surface of the PAV to capture boundary layer effects more accurately. The final amount of drag on one half of the PAV in cruising flight was determined at $70.8N$.

For the propeller simulations, a Moving Reference Frame (MRF) method was used to make a transient simulation. This means the propeller is part of a small rotating domain set to rotate at a predetermined angular velocity. An estimate of the boundary-layer thickness was made using theory of boundary layer development over a flat plate and the thickness was estimated at $1.45 \cdot 10^{-3}m$. Due to limitations, the inflation layer to capture the boundary layer could not be included and it therefor become clear that the following simulations would not be accurate and could merely be used to give estimations of performance. The $k - \epsilon$-viscosity model was used, a model suitable for propeller simulations. However, because of the lack of the inflation layer, the applicability of this model is debatable. The propeller simulations showed that an additional blade provided more lift. A remarkable result since earlier work determined 3 blades was optimal. Adding a duct around the propeller led to less generated thrust. These results are, again, debatable because of lack in accuracy. The simulation of a 4-bladed, ducted propeller was able to provide some insight in the flow field in and around the duct.

To determine the total amount of drag on the PAV it was tried to insert a time-averaged propeller exit velocity profile. While the time averaged solution was not created, the velocity profile at a particular time instant at the end of the simulation appeared quite axisymmetric.

No definitive answer to the main question could be given. It was only managed to acquire a rough estimate of the total amount of drag the PAV experiences in hovering mode. The estimated amount of drag on the PAV hull in hovering mode is $2.6 \cdot 10^2 N$, which is roughly 11 percent of the needed thrust to hover.

Chapter 7: Recommendations

Having concluded this internship research, a few topics have come to mind in which other students might conduct further investigation to eventually come up with a great PAV design:

- Propeller design

  During this internship, a propeller was used for simulations that was modelled after a certain chord-length and pitch angle distribution. Some aspects that were not included were the modelling of the blade tip (which was somewhat blunt in this model), the fixation of a blade to the hub and the hub itself. for further research perhaps a comparison can be made in hub, blade fixation and blade tip design as well as different chord length and pitch angle distributions to find better propeller designs suitable for PAVs.

- Duct design

  There is some literate that discusses duct design but this internship did not go into that deeply. Perhaps in the future it cna be investigated what shapes of duct or cross-section have a positive influence on the amount of thrust that can be generated. A comparison of different expansion angles or how duct can have curved edges at its inlet to improve the flow field to have smaller velocity gradients, improving performance.

- Propeller location

  Since the initial PAV design has its propeller located above the hull it became clear the wake of the propellers would cause additional drag. One might ask if the propellers could be placed below the hull to circumvent this. However this could lead to practical problems so one might investigate just how feasible and practical this design would be.

Chapter 8: Acknowledgements

I would like to thank Prof. Hoeijmakers for making this internship possible and putting me into contact with Prof. Bil. At the start of it all he offered me many different locations around the world at which my internship could be conducted and it his willingness and enthusiasm for arranging internships abroad for students is greatly appreciated.

Much of gratitude to Prof. Bil as well and most importantly for inviting me over to Melbourne for this internship. It has been a fun, educational, challenging and sometimes a little stress full (nothing unhealthy) experience to do this internship at RMIT. It was fun to see how the theories of 'numerical methods' was put into practise. Also it was a great opportunity to refine my Solidworks skills. You always had a few minutes for me every time I had a few questions or you pointed me to some other people to help me out which is very much appreciated. On top of that there usually was time for a cup of coffee and a casual talk as well.

I would also like to thank Dr. Robert Carrese for his tips and counseling regarding the Ansys software. You definitely helped me a lot along the way and although I didn't manage to try out some of your suggestions, some of your recommendations made things easier along the way.

Chapter 9:   Literature

[1]: Meng(2018), *Final Report*
[2]: Xu(2015), *CFD Investigation into Propeller Spacing and Pitch Angle for a Ducted Twin Counter Rotating Propeller System*
[3]: Anderson(2016), *Introduction to flight*, 8^{th} edition.
[4]: Hale, *Development of a Reduced Order Model for Ducted Fan Modelling in CFD*

Chapter 10: Appendices

A: Plots and figures

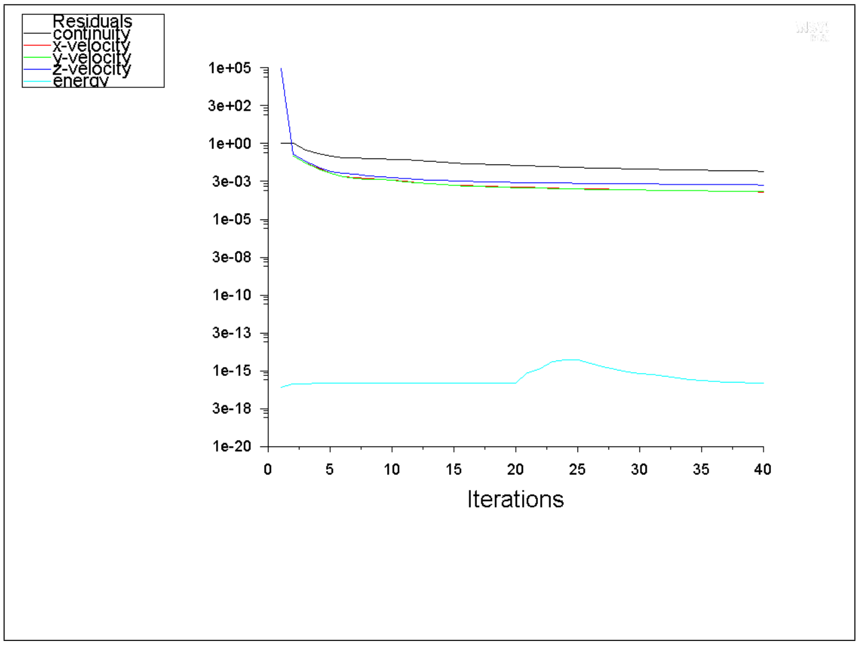


Figure 1: residuals of PAV drag in flight

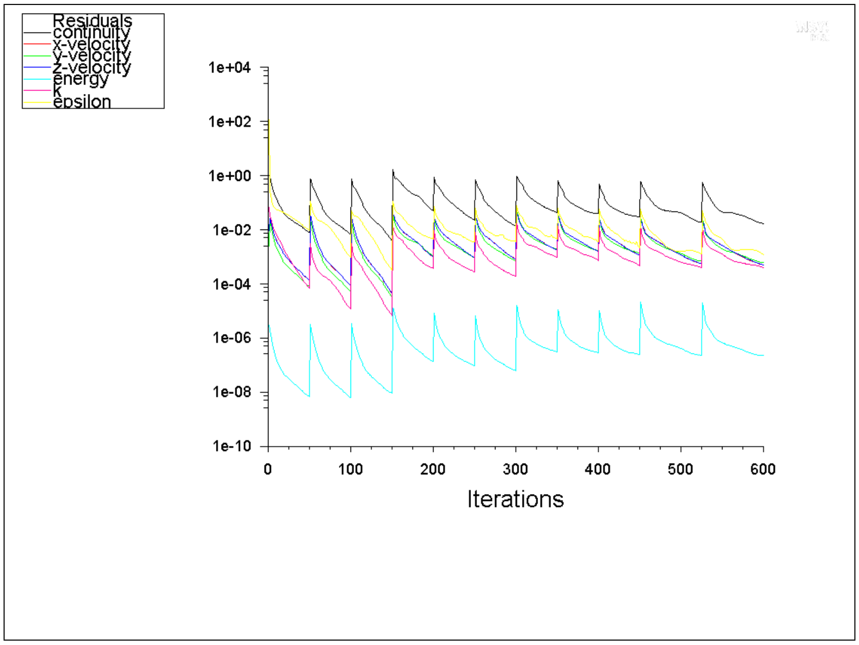


Figure 2: residuals of 3 bladed propeller with duct

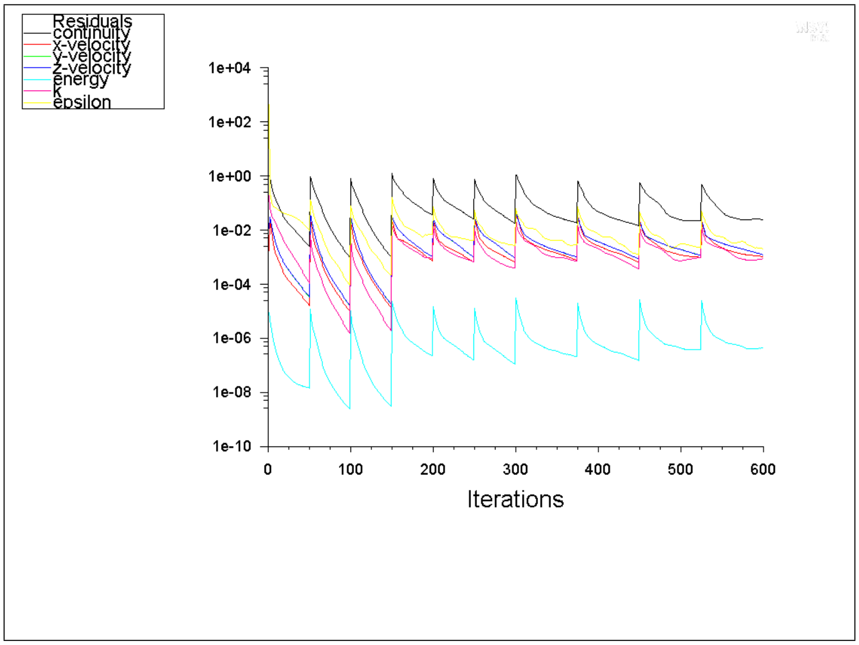


Figure 3: residuals of 4 bladed propeller

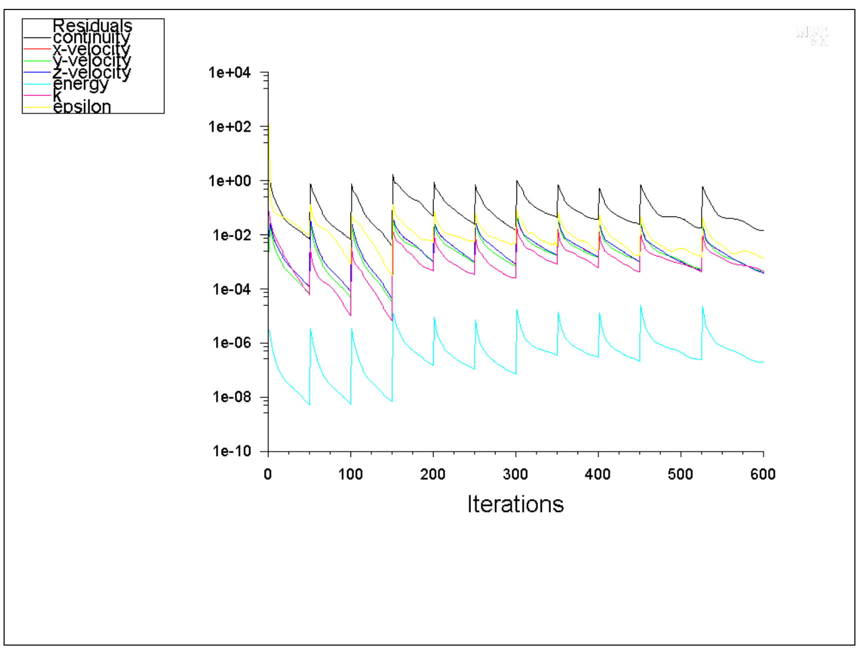


Figure 4: residuals of 4 bladed propeller with duct

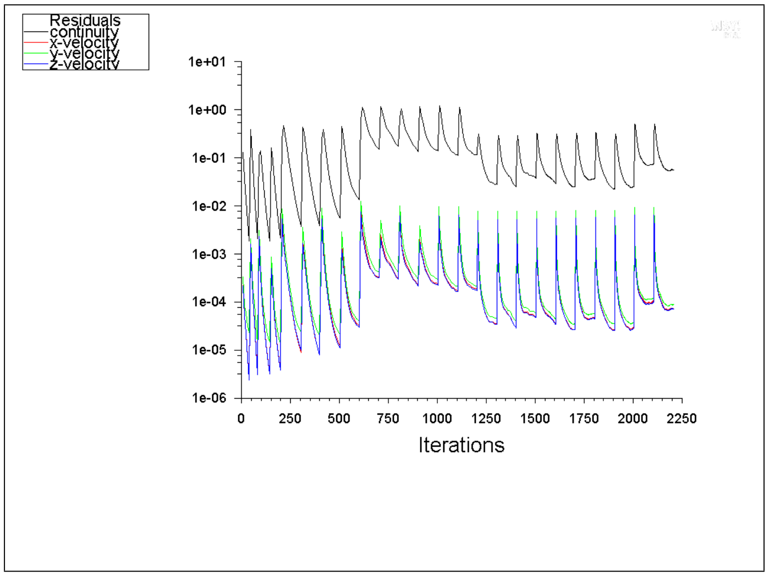


Figure 5: residuals of drag on hovering PAV

B: Reviews

My employer, Prof. Bil, has been a great supervisor during my time at RMIT University. He is a very approachable man who always found the time for any questions whenever I needed help, or putting me into contact with people who could help me further. On top of that I found it good that he checked up on me to see how things were going from time to time. He has been very welcoming on my first day and I greatly appreciate him for letting me come over to Australia to do my internship here.

On my own functioning during the internship. I thoroughly enjoyed working on the task that was given to me. I given to me even though it definitely was stress full and challenging from time to time. I was intrigued by the fluid-simulation software which was a very impressive piece of software engineering and I hope to find a master-thesis assignment in which I can continue to improve my skills with these kind of computer programs. I did learn about myself that I should be more systematic in the way that I approach an assignment like this, make a better planning from the start and perhaps ask for some help sooner since that could have saved me some time. In hindsight I feel like I should have arranged some other accommodation closer to the internship since the long commute to work did have a slight negative effect on my performance on some days. All in all I feel like like I've done my best during this internship and even though I did not manage to get as good of a result as I wished I learned a lot and it has been a great experience.