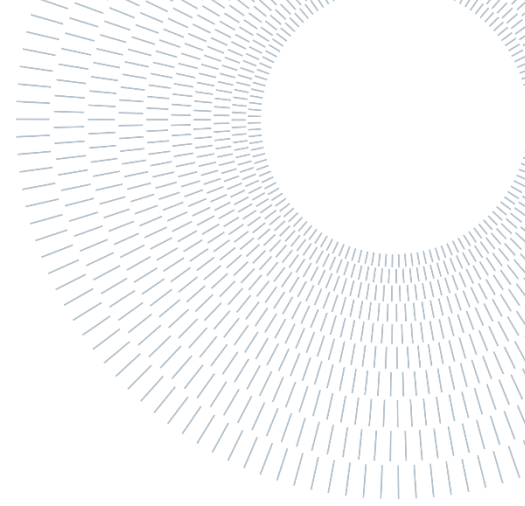




**POLITECNICO**  
MILANO 1863

SCUOLA DI INGEGNERIA INDUSTRIALE  
E DELL'INFORMAZIONE



# Steady-state and unsteady analysis of cylindrical bodies in hypersonic flow

TESI DI LAUREA MAGISTRALE IN

SPACE ENGINEERING – INGEGNERIA SPAZIALE

**Epaminonda Athos, 967538**

---

**Advisor:**

Prof. Luigi Vigevano

**Academic year:**

2024-2025

**Abstract:** Predicting the behavior of re-entry objects has gained a lot of interest in recent years. Calculating the aerodynamic coefficients of such objects as well as characterizing the flow around them is crucial in predicting their trajectory. In this project, both steady-state as well as unsteady analysis of a pitched circular cylindrical body at hypersonic flow re-entering the Earth's atmosphere is simulated. Steady-state simulations are run using SU2 CFD software aiming to validate the results against previous experimental and computational ones. Additionally, unsteady simulations with three different angular velocities around the pitch axis are carried out to assess the influence the angular velocity of the body has on both the aerodynamic coefficients as well as the flow around the body. The project proved the capabilities of SU2 in modeling compressible hypersonic flows by giving results similar to the experimental and computational ones. Its multi-zone capabilities were utilized in the quasi-stead simulations where it was concluded that the angular velocity of the cylinder does affect the aerodynamic coefficients of the body with more pronounced differences at higher angular velocities.

**Key-words:** CFD simulation, hypersonic flow, cylindrical bodies aerodynamics, atmospheric re-entry flow characterization

# 1. Introduction

With the ever-increasing space debris created by humanity, but also the danger of large meteorites and other space objects, there is a strong interest in the study and characterization of such bodies when they re-enter the Earth's atmosphere. Being able to predict the behavior of such objects by knowing their aerodynamic coefficients, will give an advantage to humans to know in advance how they will behave when they enter the atmosphere and whether or not they can pose a potential threat. Early identification of space debris/objects is possible through the use of either ground or space telescopes however their re-entry behavior largely depends on their shape and size. Being able to model in advance how such objects behave can give an insight on their expected trajectory and decide whether or not they can be a possible threat to humanity.

Nowadays, the use of Computational Fluid Dynamics (CFD) simulations have played a key role in the study and understanding of re-entry objects. Since this technology has matured sufficiently enough, complex flow phenomena that occur at hypersonic speeds can be captured with great accuracy, and the objects' trajectory can hence be predicted. However, such simulations require a lot of computational power and time and usually require clusters of computers in order to model everything precisely.

A research carried out by Seltner et al. [2], aimed to study the influence of an inclined cylindrical body in hypersonic flow regime. Both experimental and numerical analysis of a cylinder free-flying in a hypersonic flowfield with varying pitch angle from 0 to 90 degrees were carried out. For their experimental setup, the hypersonic wind tunnel H2K at the German Aerospace Center (DLR) in Cologne was used, whereas the computational study was carried out using two different NASA software. These were the inviscid solver Cart3D, and the US3D for the viscous simulations. Additionally, the aerodynamic coefficients of the cylindrical body were calculated. The above-mentioned study was unique in this field as researchers were the first to see the effect the pitch angle has on the flow around the body, getting a step closer to simulating real-life scenarios of tumbling bodies re-entering the atmosphere.

The results of this research were used as the basic literature for the current thesis project, where a similar approach is to be used but using the open-source CFD software Stanford University Unstructured 2 (SU2), which is a powerful tool primarily designed for compressible flows but has since expanded into other areas as well [3],[4]. The present project aims to replicate the steady-state simulation results by Seltner and hence validate the use of SU2 in such problems, and furthermore, use the multi-physics capabilities of the software to create an unsteady/time-dependent simulation of a rotating cylinder in hypersonic flow. Finally, the effect of different angular velocities will be analyzed and contrasted against the steady state results.

## 2. Aim and Objectives

As outlined in the introduction, this research will mainly focus in performing a 3-Dimensional simulation of a cylindrical body entering the Earth's atmosphere and calculating its aerodynamic coefficients at different pitch angles using the open-source SU2 CFD software. These results will then be contrasted and validated with both numerical and experimental ones provided by the research paper by Seltner et al. [2]. Once the results and setup of the steady-state simulations are validated, three more unsteady simulations will be run. At this point, the multi-physics capabilities of SU2 will be used to introduce a fixed angular velocity around the pitch axis of the cylinder. Three different angular velocities will be tested with the aim of seeing the influence they have on the results.

Through the following sections, the setup parameters for the simulation will be examined and some of the flow features expected to be seen at hypersonic re-entry will be briefly presented. Moving on, the mesh discretization procedure follows in combination to a reduction of the overall domain so as to make the simulations more power efficient and at the same time more accurate. A discussion then follows of both visual characteristics of the flow field around the cylinder and aerodynamic coefficients obtained from steady-state simulations with the aim of validating the SU2 results with the ones of the literature. Finally, the unsteady results at three different angular velocities around the pitch angle are discussed. The paper concludes with some limitations identified throughout the project as well as some recommendations for further research in this topic.

## 3. Simulation Environment Setup

For the following simulations, the flow conditions as well as the characteristics of the cylinder were kept the same as the paper by Seltner et al. [2]. These are a circular cylindrical body of 0.1 m in length and 0.05 m in diameter in Mach 7 hypersonic flow field. The remaining flow conditions are shown in Table 1.

Table 1: Flow conditions for experimental and numerical simulation

Quantity	Experimental H2K	Numerical simulation
Free-stream Static Pressure [Pa]	126	116
Free-stream Temperature [K]	56	56
Free-stream Density [ $g/m^3$ ]	7.8	7.3
Free-stream Velocity [m/s]	1046	1046

The aerodynamic coefficients were calculated based on the reference system illustrated on Figure 1 below, where the drag, lift and pitching moment coefficients are as shown [2]. The positive pitch angle ( $\theta$ ) is taken counter-clockwise around the y-axis (i.e. nose-up).

For the solver in SU2, the Reynolds-Averaged Navier-Stokes' equation (RANS) with Spalart-Allmaras (SA) turbulence model is used. Both of these options were chosen due to their robustness and ability to capture complex flow phenomena while keeping the computational power required relatively low compared to solving the full Navier Stokes equations.

An ideal gas was used for the gas model and the reference area for the calculation of force coefficients was calculated as  $A_{ref} = \pi * (d^2/4)$  which corresponds to the base area of the cylinder. The coefficient of drag ( $C_d$ ) was set as a convergence criterion for the steady state simulations, with an error between two iterations set to be less than  $1E - 8$ , which is sufficient enough for accurate results. As boundary conditions, a constant inflow condition with flow characteristics as table 1 was used, and the outer walls of the domain were set as farfield so as to not interact with the flow. The walls of the cylinder were treated with a slip condition, as it will be discussed in sub-section 5.1.1.

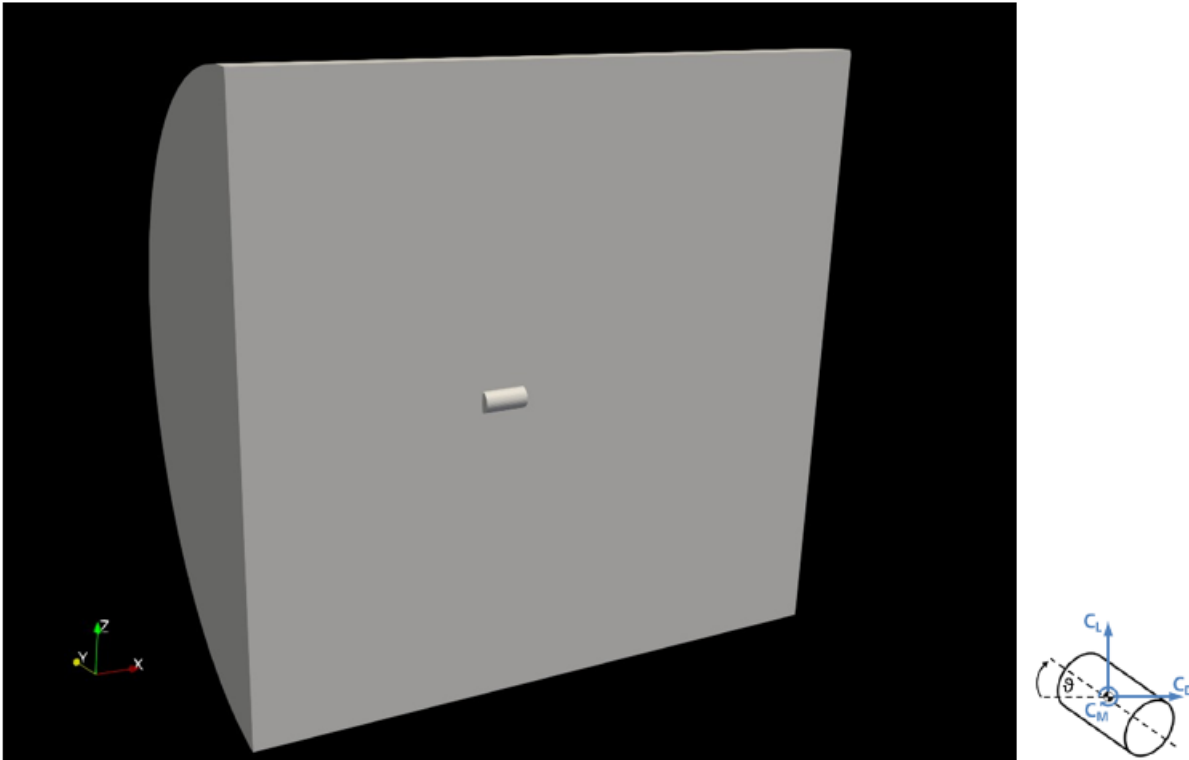


Figure 1: Cross section of the whole simulation domain and reference axis for the coefficients

## 4. Discussion on Expected Flow Characteristics and Mesh Quality

### 4.1 Expected Hypersonic Flow Characteristics

In a hypersonic flow, the aerodynamic and thermal phenomena around a cylinder become quite complex due to the high Mach number and the associated shock dynamics. Under such conditions, there are certain flow characteristics that are expected to be observed. These include a bow shock occurring at a distance (stand-off distance) from the frontal face of the cylinder, a separation bubble at the leading edge of the cylinder, as well as some expansion fans and a wake region at the end of the body.

For all the shocks to be captured accurately it is noted that a very fine mesh is required. Shock capturing is therefore not in the scope of this report due to the computational power required as only a single computer was available, but as can be noticed later, even with a single core, the most pronounced shocks are indeed present. Figure 2 below showcases the above-mentioned phenomena.

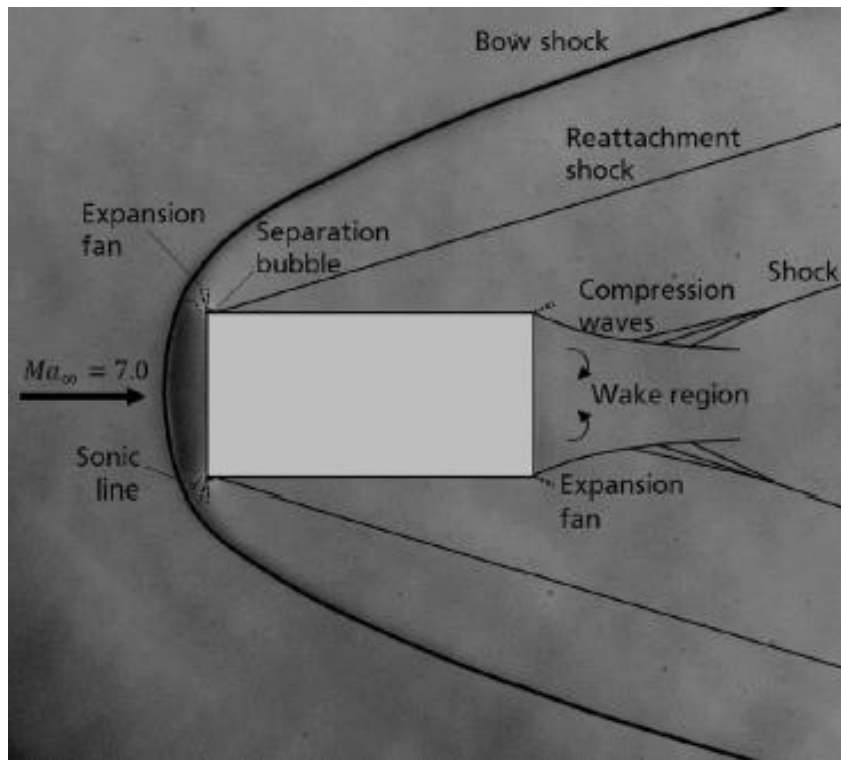


Figure 2: Hypersonic flow characteristic feature [2]

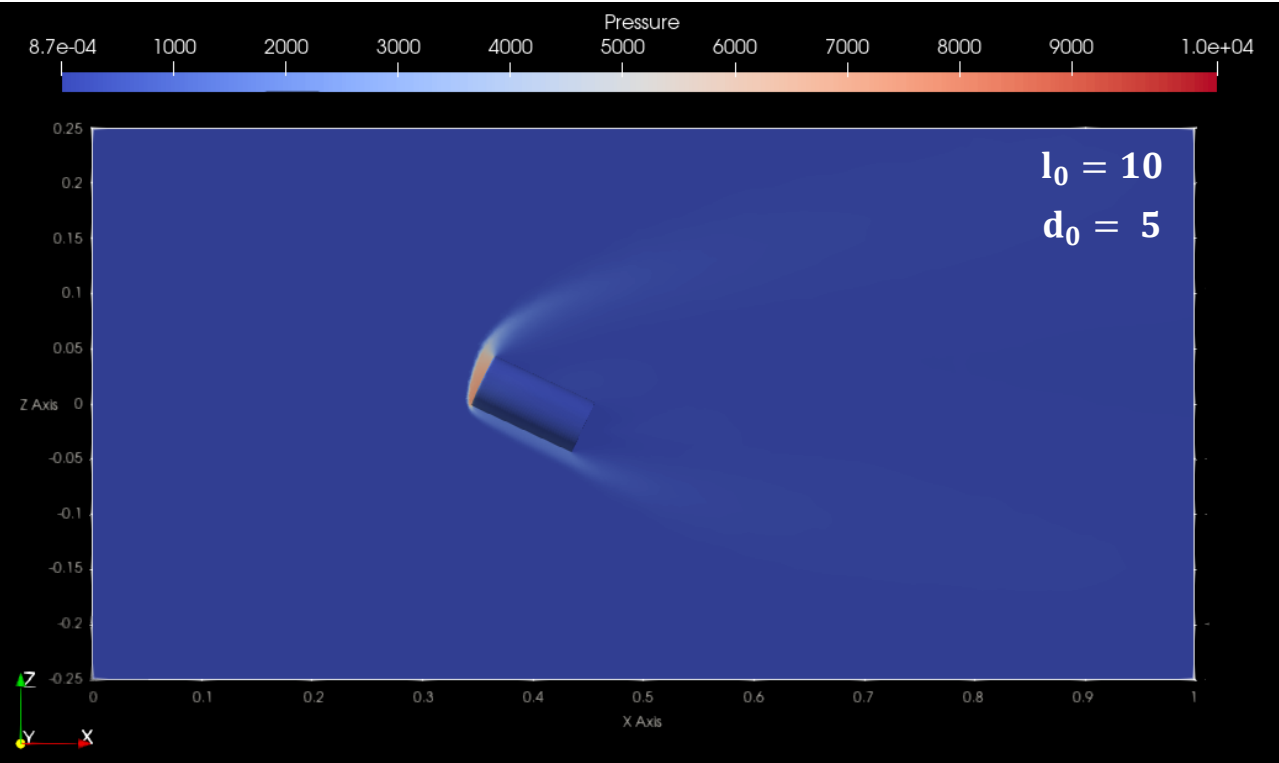
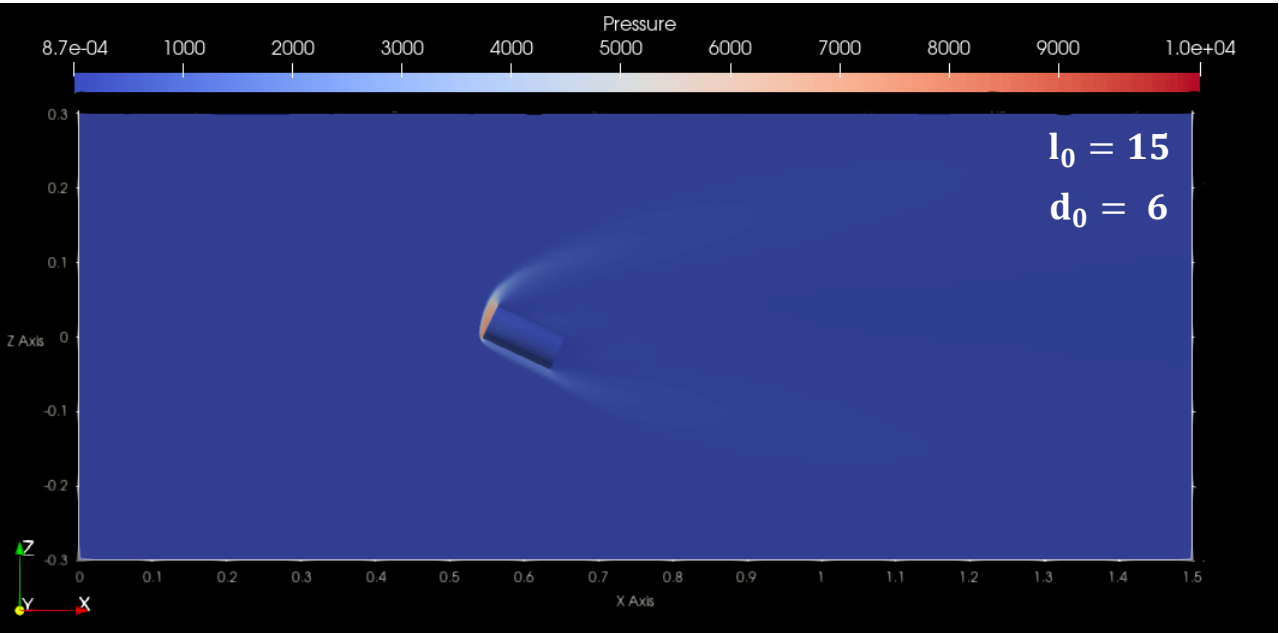
## 4.2 Mesh Discretization

The first step in any CFD simulation is to perform some analysis in the mesh and do a mesh discretization. This is a crucial step in every CFD simulation as the quality of the mesh will dictate how much computational power and time is required to reach convergence, but most importantly how well physics are captured. For the purpose of this study, an unstructured mesh was used with a higher number of elements around the cylinder in order to capture more accurately the aerodynamic coefficients.

Initially a coarse mesh was used and gradually a greater number of elements were added until either the results were satisfactory enough or the limit of the computer was reached. While moving gradually towards a finer mesh it was noticed that the limit of computational power was reached quite fast. For this reason, a different approach was followed to allow for further refinement of the mesh in order to get more accurate results.

Instead of having the whole domain so as to replicate exactly the experimental wind tunnel geometry, the domain around the cylinder was gradually reduced. A study regarding how much the domain could be reduced was also carried out in order to ensure that this reduction in the overall domain did not influence the results. Generally, at hypersonic flows, upstream conditions do not really influence the rest of the flow, therefore much of the domain in front of the cylinder could be reduced. Care should be taken however that there should be sufficient distance from the wall of the cylinder; greater the stand-off distance, so that the bow shock is captured correctly. By reducing the overall domain, many more smaller cells can be placed around the cylinder's body which is the main area of this research and therefore increase the accuracy of the aerodynamic coefficients. To help illustrate the problem, Figure 3 shows the different types of domain sizes used and concludes with

the one that was used for the remaining simulations. The normalized length  $l_0$  ( $\frac{\text{domain length}}{\text{cylinder length}}$ ) and normalized diameter  $d_0$  ( $\frac{\text{domain diameter}}{\text{cylinder diameter}}$ ) of each domain, are also shown.



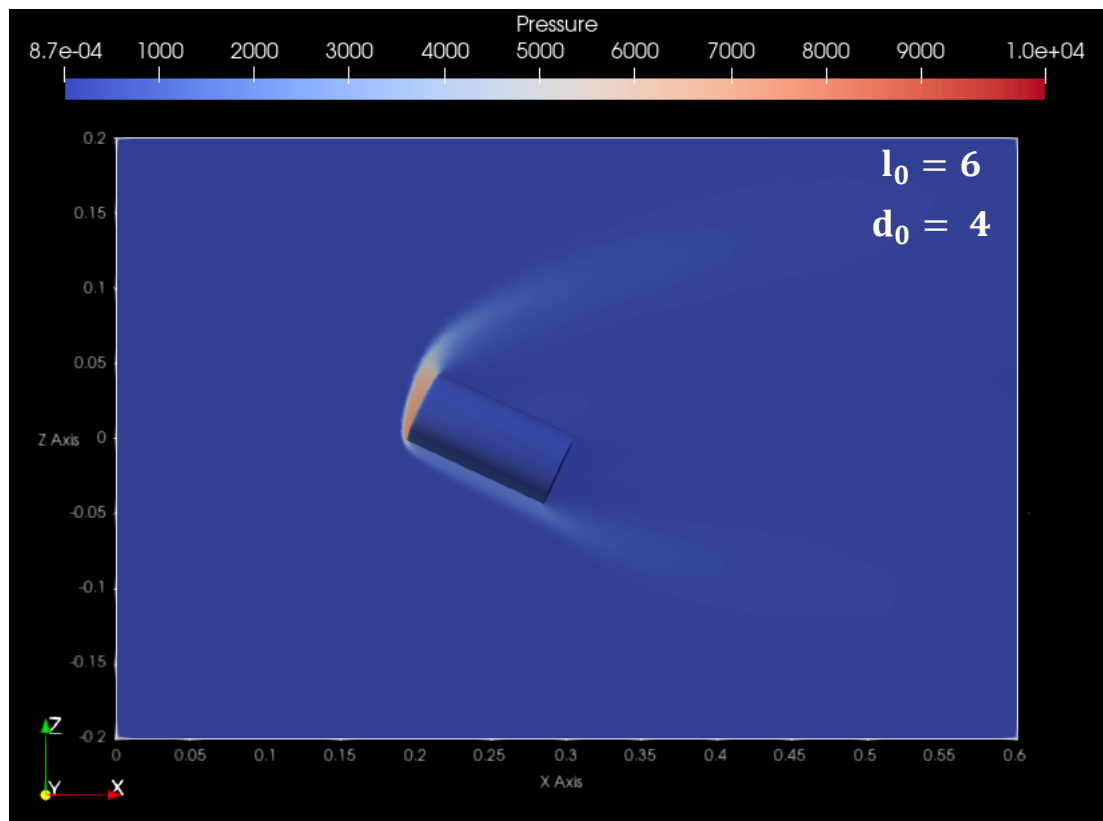
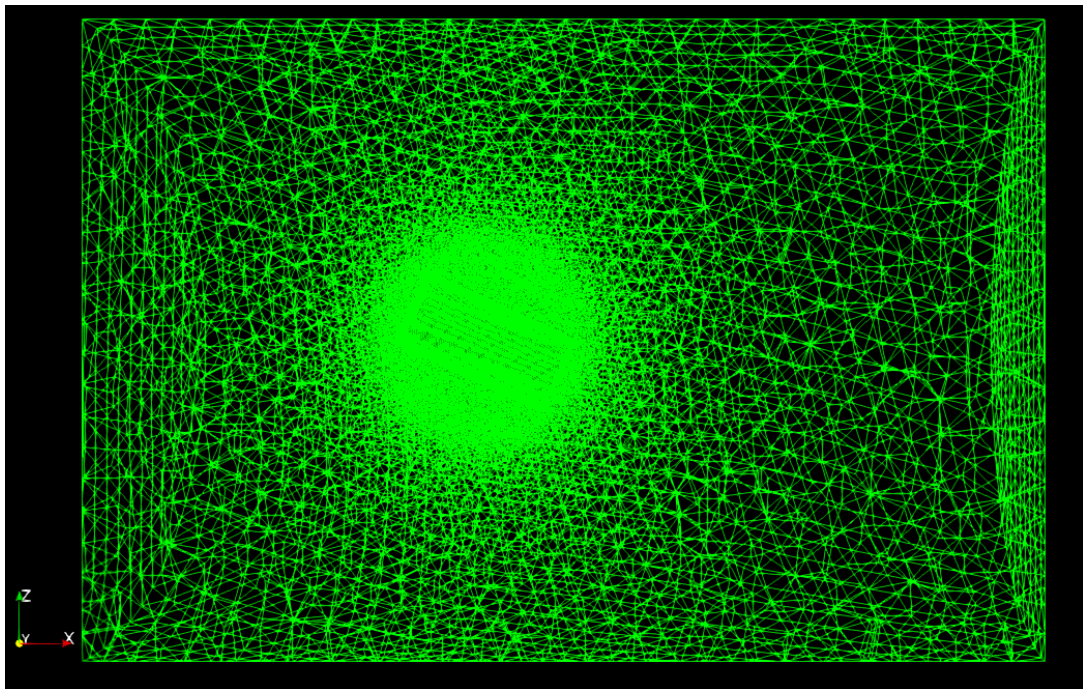
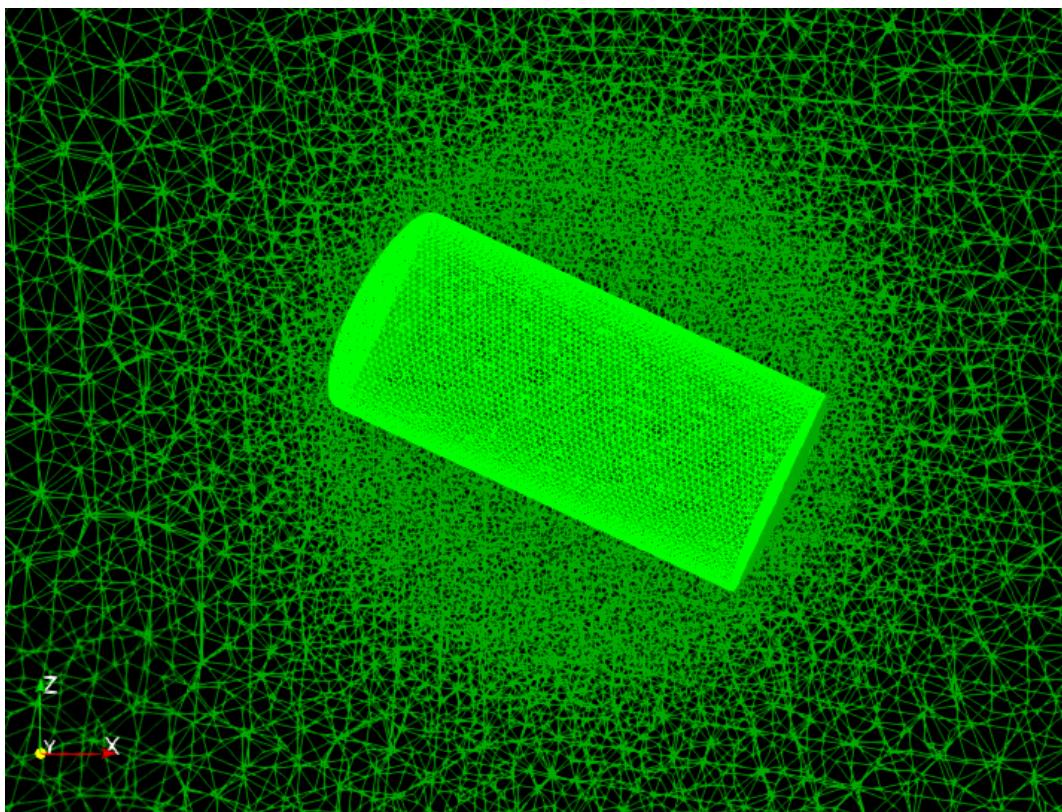


Figure 3: The three different domains tested with their dimensions

As can be seen from Figure 3, moving from the first to the last image, the domain around the cylinder decreases but there is no alteration of the results. Once the domain could not be further reduced as the results would be altered, smaller elements could then be placed around the body. In Appendix A, the results of all three different meshes can be seen plotted against the literature results emphasizing the importance of a fine mesh with smaller elements. On the next section, the discussion of the steady state simulations will be shown, with the finest mesh able to be achieved by the computer. The mesh in discussion has 753,700 grid points and 4,703,940 elements as illustrated in Figure 4.



a) The whole meshed domain



b) A close-up of the meshed cylindrical body

Figure 4: The final meshed domain

## 5. Results

The following sections will firstly go through the steady-state results. Flow characteristics as well as the aerodynamic coefficients calculated by the simulations will be compared with the reference results and then the unsteady ones will be presented.

### 5.1 Steady State Simulations

The steady-state simulations were visualized in Paraview which is an open-source software. Looking at the case with pitch angle  $\theta = 0$  degrees, it can be seen that the results closely match the research by Seltner et al. [2]. In more detail, the bow shock at the front part of the cylinder seems to be captured correctly and at the same distance (stand-off distance) from the body. It is noted that the smearing out of the bow shock seen further away from the body in the SU2 results, is due to the lower mesh refinement at that area. Other features of the flowfield like the wake/recirculation region at the trailing edge of the cylinder as well as the expansion fan at its base are only slightly visible in the pseudo-schlieren images obtained by computational results by Seltner et al. There are two reasons for not capturing these flow phenomena. The first being that there is no proper way of creating a Schlieren-like image similar to the reference image in Paraview (here the density is used with an x-ray like color palette), hence a lot of detail is missing, but the most important reason is attributed to the quality of the mesh. As stated previously, the mesh is very fine close to the body and gets coarser further away as the main focus of the project is to capture accurately the aerodynamic coefficients of the body. Therefore, abrupt changes in any type of quantity cannot be captured accurately with the current mesh, leading to many of the flow features to be missing.

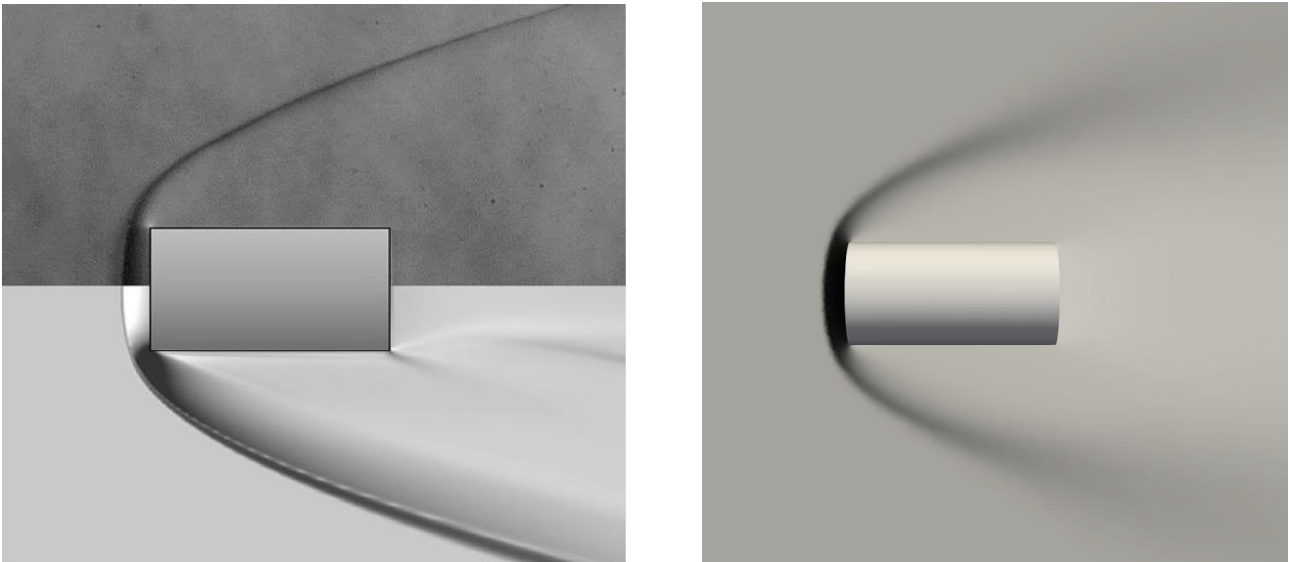
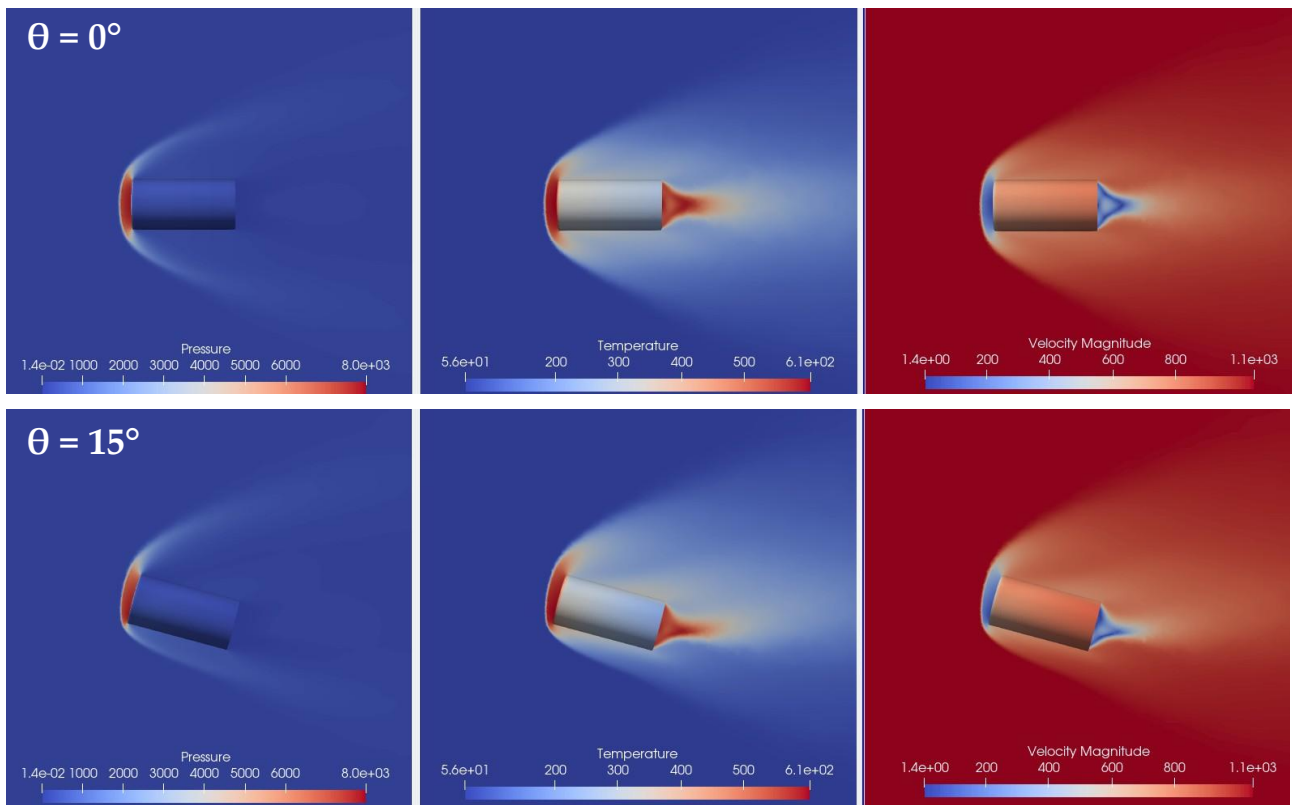
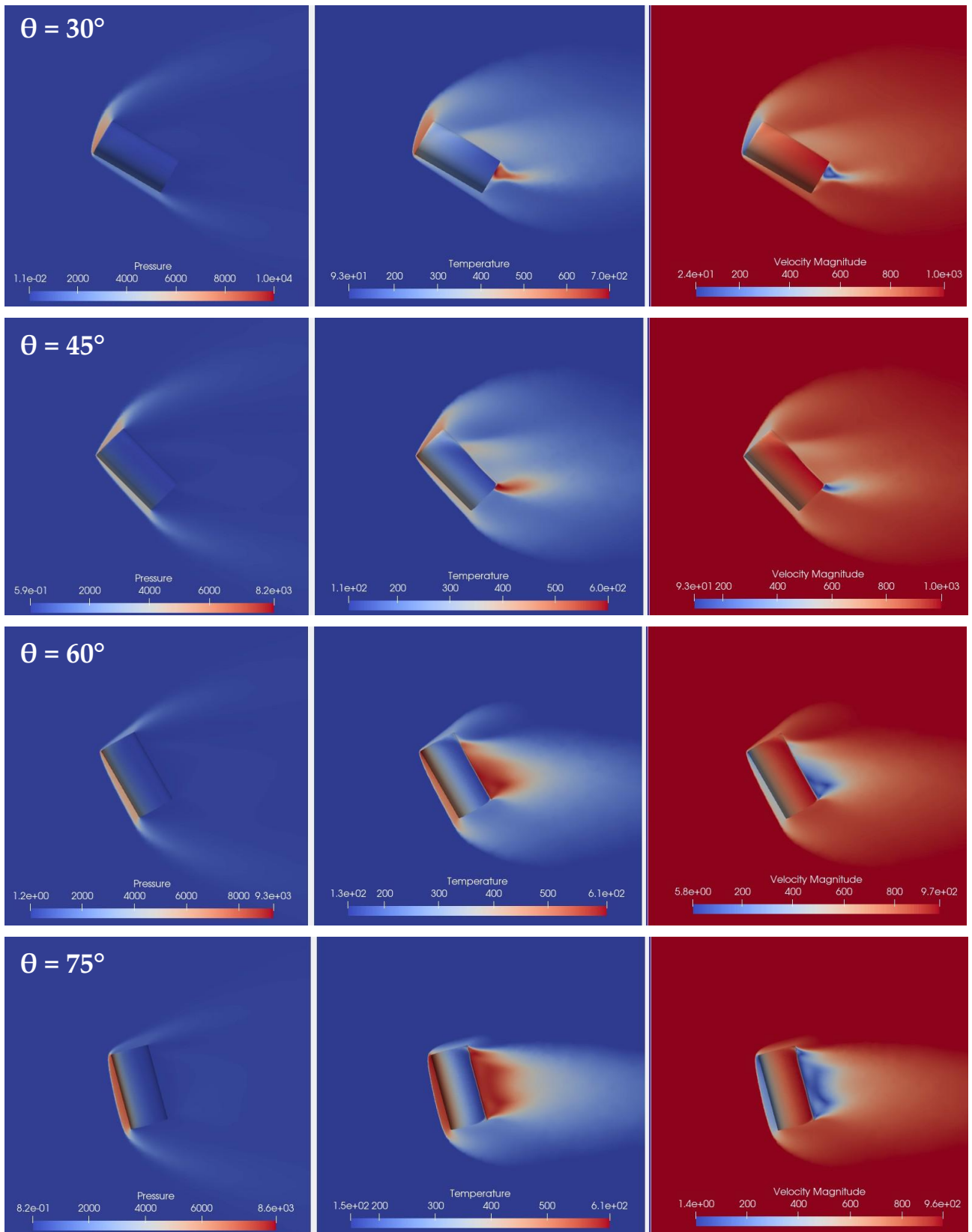


Figure 5: The experimental and pseudo-schlieren image by [2] on the left. On the right the one obtained by SU2

However, this is not the case when it comes to quantities like Temperature, Pressure and Velocity. For these, as can be seen from Figure 6, the current mesh is sufficient enough, as the results follow the expected evolution of the flow with changing pitch angles. In particular, the Temperature around the cylinder is almost identical to the reference case, with two regions having the highest values; the first being the initial bow shock region where due to the high velocity and compressibility effect the temperature rises and the second is the wake region. The same applies for the Mach number, where as expected it drops drastically from Mach 7 where the bow shock is formed and also outlines the wake region. Interestingly, from these figures some of the flow phenomena can be seen like the expansion fans at the trailing edge while in Figure 5 were not visible. Lastly, the pressure is captured well and matches in terms of magnitude the one in the literature mentioned before.

As can be seen from Figure 6, as soon as the pitch angle deviates from zero degrees, the flow around the cylinder immediately becomes more complex. It is no longer symmetrical and quantities like Temperature and Pressure change drastically. The bow shock changes shape and the stand-off distance appears to be reducing up to 45 degrees of pitch angle. From the velocity figures, it is obvious that the flow behind the cylinder becomes more turbulent and the wake region appears to first decrease up to 45 degrees but then increase again, with a maximum when the cylinder passes at 90 degrees through the flow field. These results demonstrate the reason that the studies of inclined bodies are so crucial, as the behavior of the flow changes drastically as soon as the cylinder rotates even slightly in pitch.





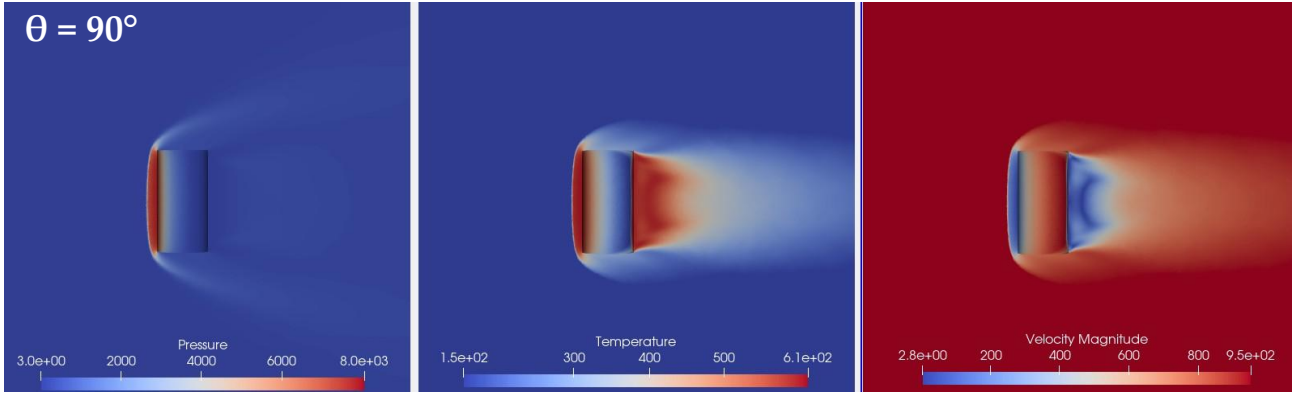


Figure 6: Pressure, Temperature and Velocity of the results obtained

Having briefly analyzed the results visually, the characterization of the aerodynamic coefficients, which is the main objective of this project is going to be discussed next. In this section, the coefficient of drag and lift as well as pitching moment coefficient will be calculated and compared directly to experimental results in H2K, analytical ones using a modified Newtonian method [5] and numerical results using US3D, all of which were directly extracted from the literature paper [2]. Results by Cart3D were omitted as they were similar to US3D ones and would only clutter the graphs. Since the cylinder is axisymmetric, it is expected that all coefficients will show periodicity. For this reason, the simulations were run in a 5-degree interval from 0 degrees up to 90 degrees. The three coefficients were calculated directly by SU2 by the equations (1) – (3) below:

$$C_d = \frac{D}{\frac{1}{2} * \rho * V^2 * A_{ref}} \quad (1)$$

$$C_l = \frac{L}{\frac{1}{2} * \rho * V^2 * A_{ref}} \quad (2)$$

$$C_m = \frac{M}{\frac{1}{2} * \rho * V^2 * A_{ref} * L_{ref}} \quad (3)$$

where  $M$  is the moment around the y-axis (pitch axis) as shown in Figure 1 and  $L_{ref}$  is the reference length taken as the diameter of the cylinder equal to 0.05 m.  $D$  and  $L$  are the drag the lift forces acting on the body.  $C_d$ ,  $C_l$  and  $C_m$  are the coefficients of drag, lift and pitching moment respectively and  $\rho$  and  $V$  are the free-stream air density and air velocity.

Looking first at the drag coefficient in Figure 7, the SU2 results from 0 up to 30 degrees of pitch and then from 70 to 90 degrees, are almost identical to the US3D ones. The biggest difference is observed at 45 degrees where it is calculated to be just 3.4%. Overall, it appears to follow very closely the trend of the other CFD software and outperform the analytical Newton method for the calculation of drag coefficient as it provides more accurate results. Both simulations, however, seem to underestimate the actual drag coefficient as measured in the H2K wind tunnel for all pitch angles.

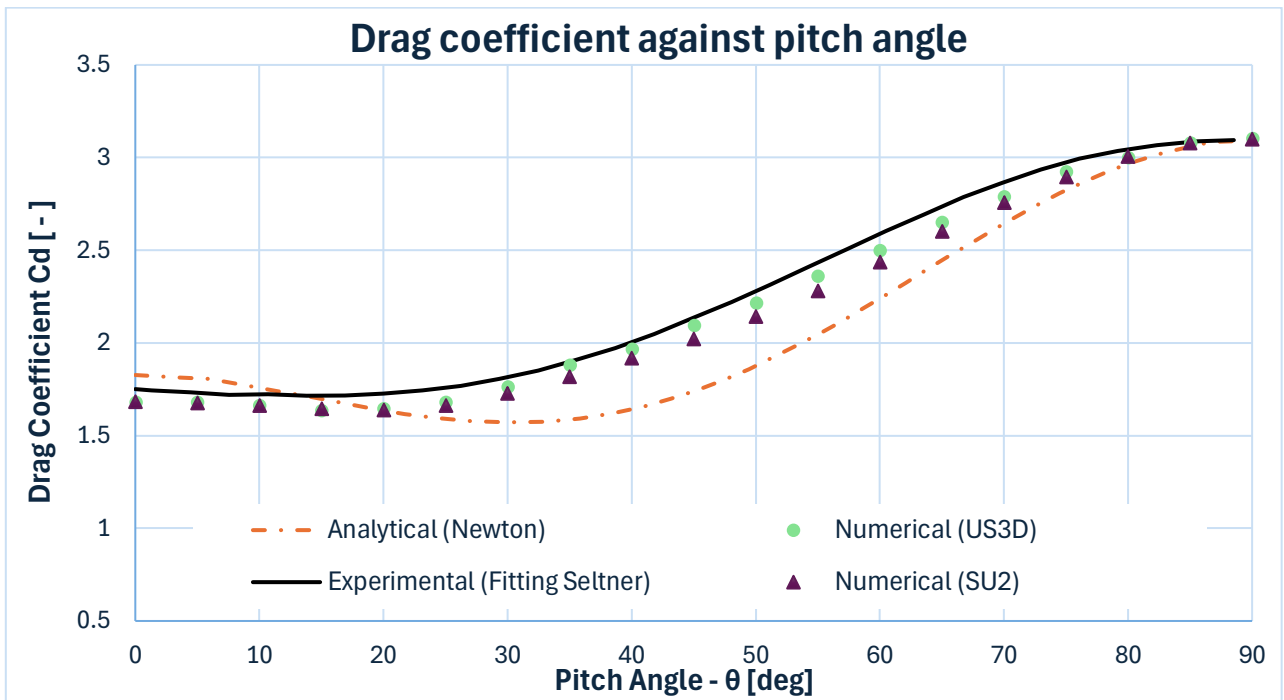


Figure 7: Drag coefficient against pitch angles

Moving on to the coefficient of lift shown in Figure 8, the results are even better as both solvers outperform the analytical Newtonian approach and are very close to the actual experimental ones. Regarding SU2 results, they are always very close to US3D ones and in some pitch angles they may even provide results closer to the experimental ones.

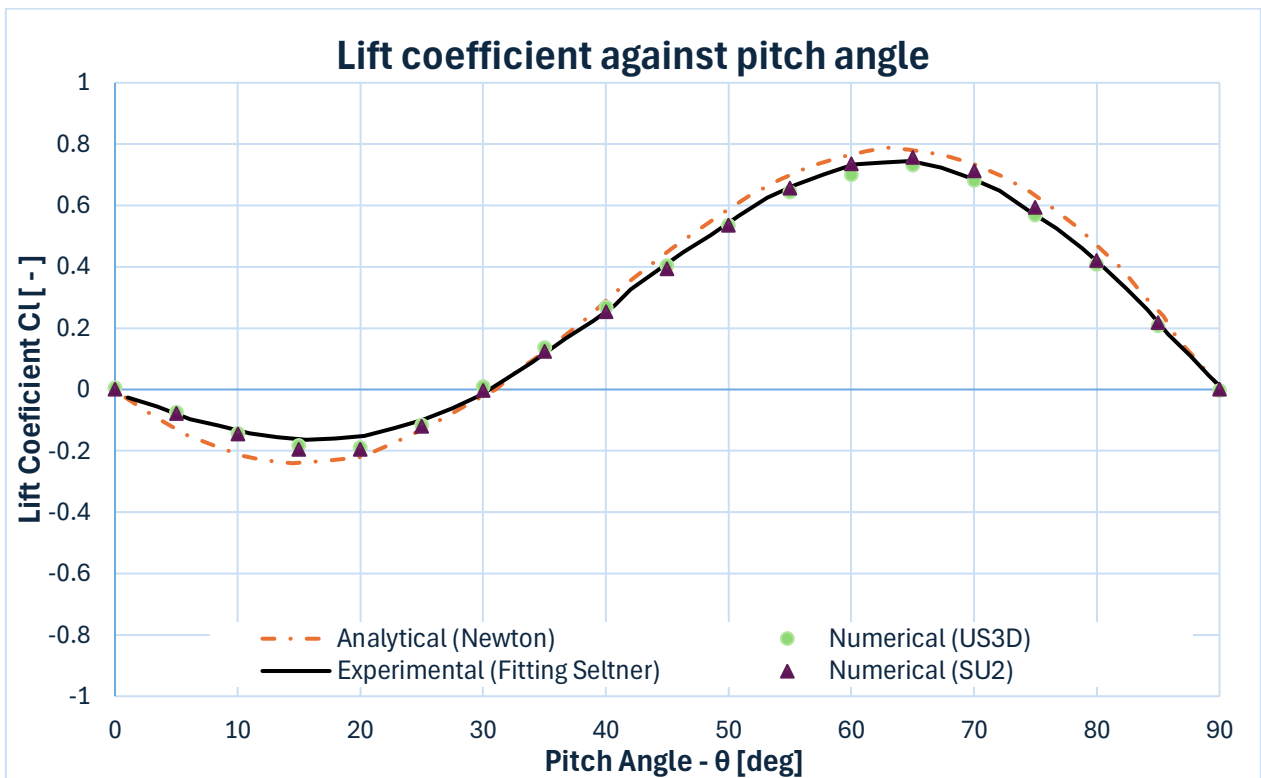


Figure 8: Lift coefficient against pitch angle

Similarly for the pitching moment coefficient as seen in Figure 9, both solvers appear to follow the same trend as the experimental data. Figure 10 shows the three different coefficients illustrating the static stability of the cylinder. From the results above it is concluded that SU2 gives comparable results with other software while not too much computational power is needed. It is also believed based on the analysis of the mesh refinement, that an even finer mesh would give even better results, closer to the experimental ones and would also be able to capture all the flow phenomena discussed in the previous section.

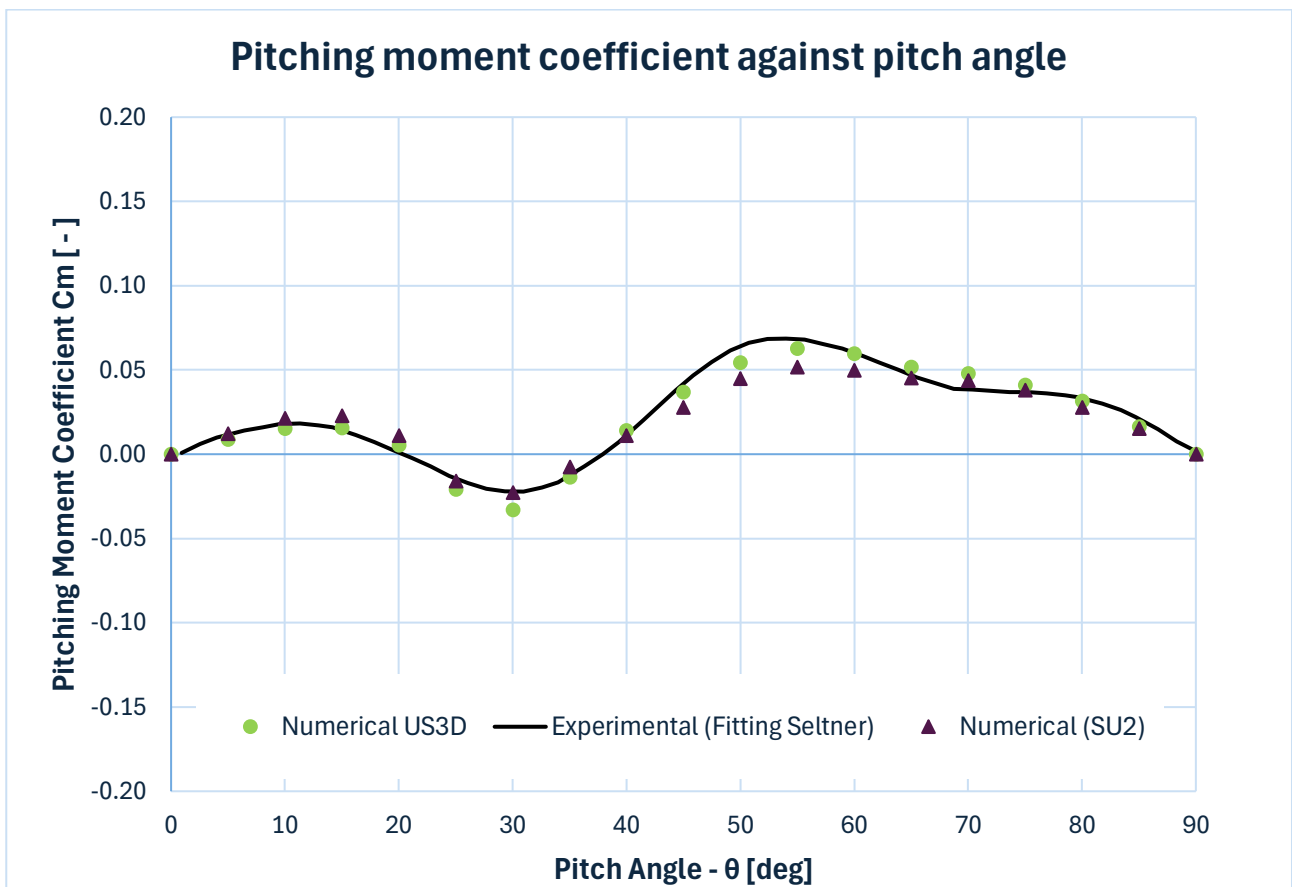


Figure 9: Pitching moment coefficient against pitch angle

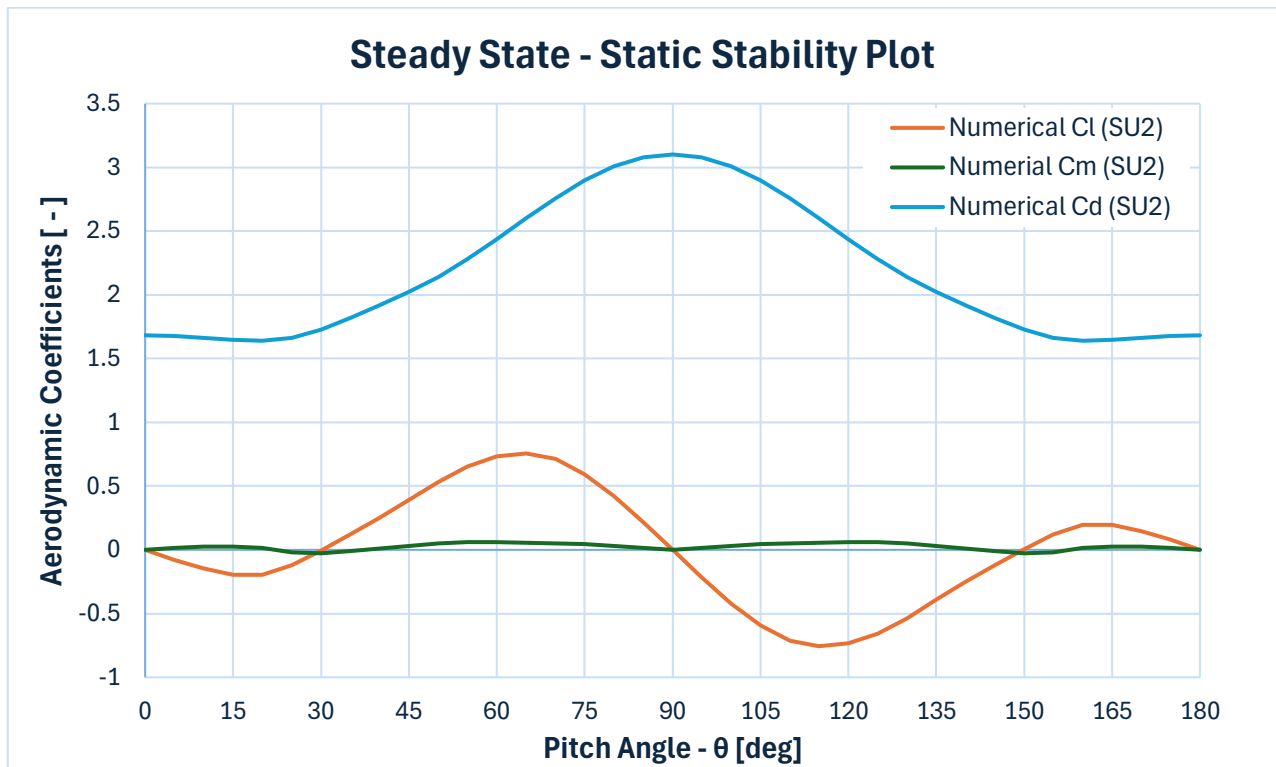


Figure 10: Static stability plot for steady-state simulations

### 5.1.1 Effect of Wall Boundary Condition

Visualizing the results as in Figure 6, one of the most important differences identified between the present study and the one by Seltner et al is the wall treatment. Unlike the reference paper [2] where the wall of the cylinder was treated as isothermal, with a no-slip condition and zero velocity prescribed at the wall, the current study did not prescribe a zero velocity at the wall as it was considered slightly strict of an assumption due to the rarefied medium the cylinder is in. Instead, tangential velocity at the wall was allowed, meaning that a slip condition was used. When it comes to rarefied flows as the case of this study, special attention should be taken on the choice of wall boundary. Depending on the Knudsen number, proper velocity-slip as well as temperature and concentration-jump boundary conditions should be used as studied and explained by Zade et al. [6]. Therefore, simply prescribing zero velocity at the wall, or using the standard slip condition available in SU2, might not be the most appropriate choice for the conditions near the wall and more research should be done before determining the most appropriate wall boundaries.

A few simulations were carried out using the no-slip condition for comparison, available also in SU2 by using an isothermal wall. These can be seen on Figure 11-13. Overall, it was noticed that the results using the no-slip condition are closer to the experimental values when it comes to drag coefficient but when it comes to the other two coefficients they do not perform so well, as illustrated in Figure 11-13. On the other hand, using a slip condition and allowing for tangential velocity seems to offer results closer to both the numerical as well as the experimental ones. For this reason, it was chosen to continue the study with the slip condition applied, however as pointed out before, future research could further study the effect of the wall boundary used, as from the presented results it appears that a wall treatment that blends between the no-slip and slip conditions would give even better results.

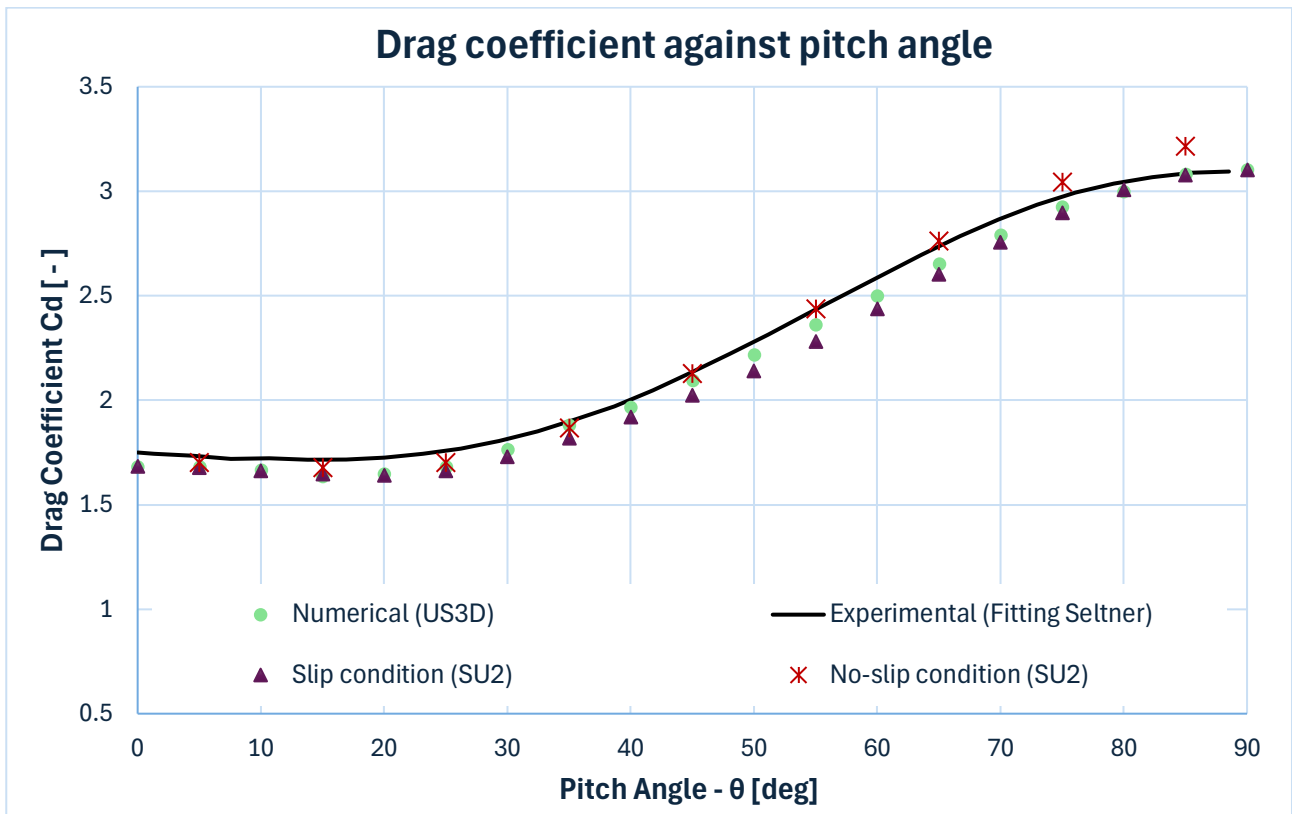


Figure 11: Drag coefficients against pitch angles for different wall treatment

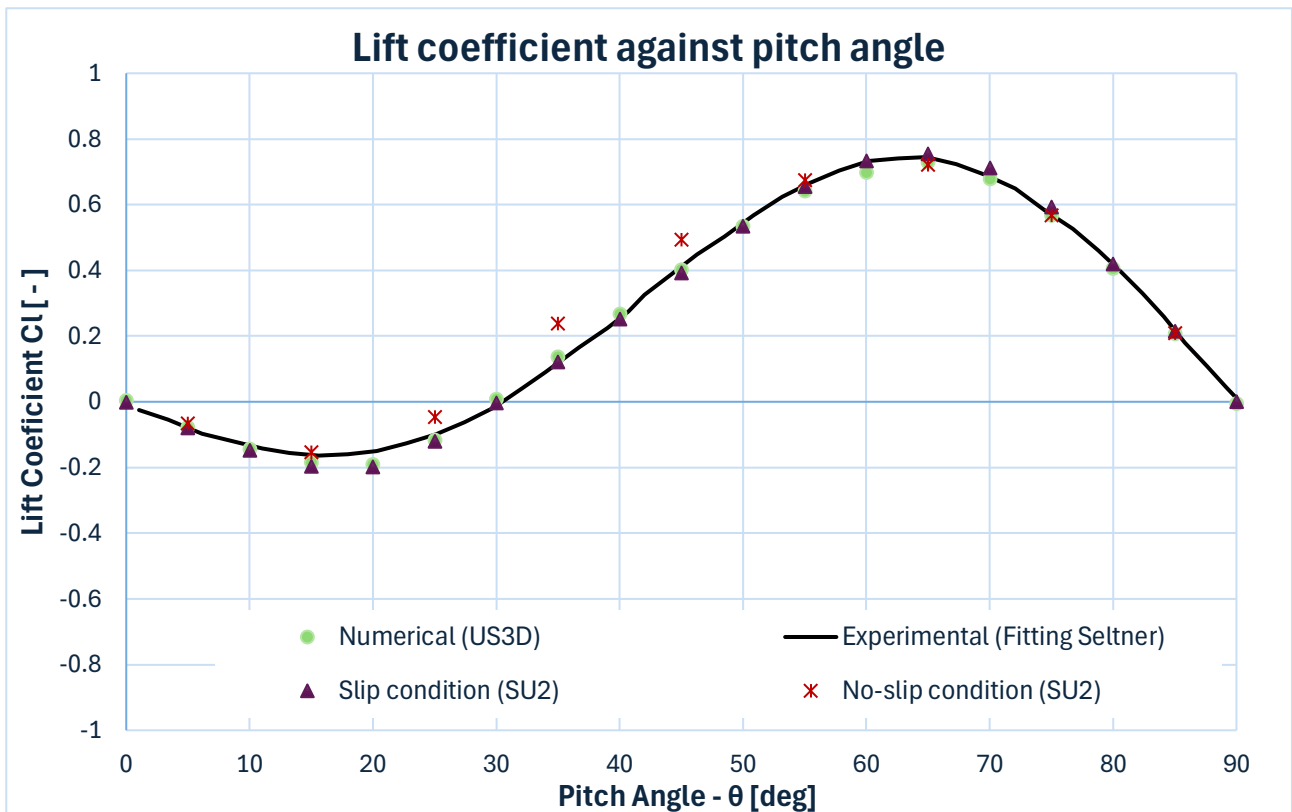


Figure 12: Lift coefficients against pitch angles for different wall treatment

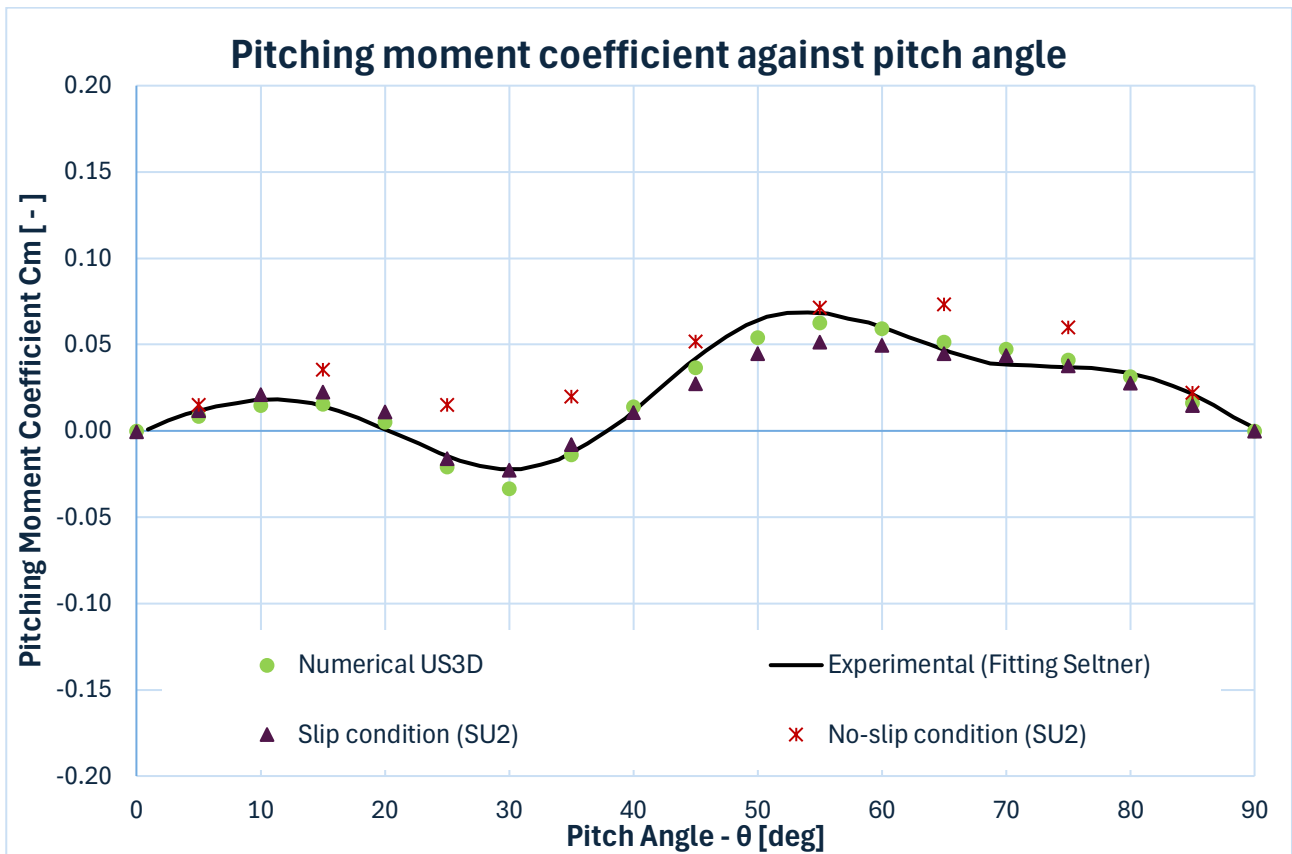


Figure 13: Pitching moment coefficient against pitch angles for different wall treatment

## 5.2 Unsteady Simulations

The final step of the present project was to create an unsteady simulation and compare these results with the steady-state case in order to see the effect the angular velocity of a rotating body has on its aerodynamic coefficients. For these simulations three random angular velocities were chosen as 45, 150 and 300 deg/sec. SU2 has the capability of allowing multi-zone / multi-physics simulations which are linked with a common interface. In this framework, the whole domain of the simulation can be split into smaller sections where each one can have its own mesh, physics or prescribed motion and therefore perform a dynamic/time-dependent simulation. Here, a sliding mesh approach is used with Fluid-Fluid coupling between the two zones [7].

For the purposes of this research, the mesh was split into two domains, Zone1 and Zone2 as can be seen on Figure 14. Zone1 represents the overall wind tunnel domain, whereas Zone2 is a spherical mesh around the cylinder where a prescribed rotation around the Y-axis is set, to simulate the cylinder rotating in pitch. Note that for the dynamic simulations, the mesh used was not as fine as the steady-state ones. The reason is again computational power available as multi-physics problems are generally more power demanding. Therefore, a slightly coarser mesh was used compared to the steady-state results discussed in the previous sections. For this reason and to allow for valid comparison between steady-state and unsteady simulations, the results for the steady-state plotted in the following figures have the same mesh quality as the unsteady simulations.

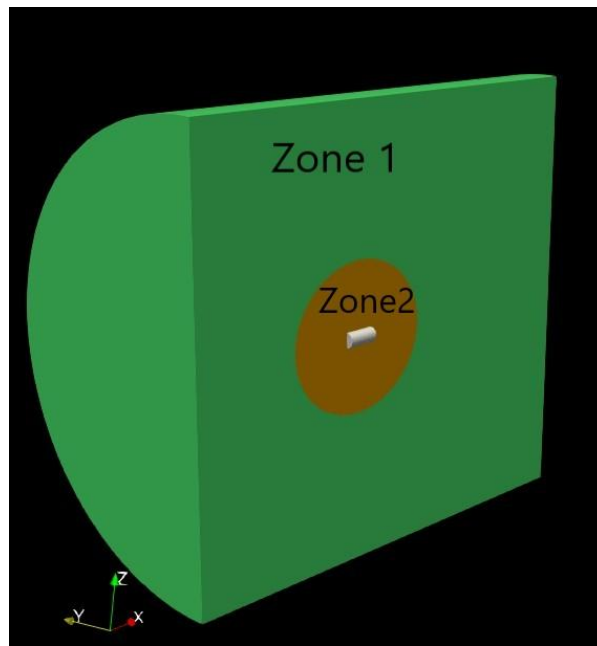


Figure 14: The two distinct zones used for the unsteady simulations

In order to control an unsteady simulation, care must be taken into selecting the appropriate inner iterations. Unlike the steady-state case where the convergence was set by a monitoring parameter or when the number of maximum iterations was reached, in a time dependent simulation there are two different time loops; an outer time loop referred to as the physical/real-time and the inner-time loop which is a pseudo-time. The selection of the number of pseudo-time iterations (inner iterations) in each physical/real-time iteration plays an important role in the quality of the results. Having too few inner iterations per real-time iteration leads to a flow that has not converged at all in pseudo-time leading to incorrect results, while on the other hand increasing the number of inner-iterations is directly linked to an increase in computational time. For this reason, a study had to be carried out in order to find out the appropriate number of inner iterations that would give the best results possible while also ensuring computational efficiency.

Inner iterations of 1, 3 and 10 were selected and a 45 deg/sec rotation around the y-axis was prescribed on Zone2. Figure 15 and Figure 16 illustrate the coefficient of drag and lift for the 3 different inner iteration steps used for the angular velocity of 45 deg/sec. The effect the inner iteration has on the results of the simulations is clearly seen by comparing the results of a single inner iteration with the rest. With only 1 inner iteration the flow does not get a chance to converge at all, leading to very different behavior compared to the other two results. On the other hand, increasing just slightly the inner iteration to 3, leads to much better final result as the results of both drag and lift coefficient, have similar behavior to the steady-state ones but are not exactly the same. This small difference is expected and is due to the unsteady flow caused by the movement of the cylinder. Increasing the number of inner iterations to 10, shows only a slight improvement when it comes to the lift coefficient. However, this insignificant increase in results accuracy comes at a cost in computational efficiency as the higher the number of inner iterations the more computationally expensive the simulation becomes. Therefore, for the abovementioned reason, the unsteady multi zone simulations were run with 3 inner iterations per iteration step.

It is noted that due to limited computational power, 3 inner iterations were used also for the 150 and 300 deg/sec cases. As the angular velocity of the body increases, the flow around it experiences faster and more abrupt changes. In order to capture these, either the number of inner iterations (pseudo-time) should be increased, or the physical step (real-time) should be reduced. In the present study however, a further reduction in physical time-step or an increase in inner-iterations was not possible due to limited computational power. Therefore, the results discussed for the three different angular velocity cases are all using the same physical time step with the same inner iterations number.

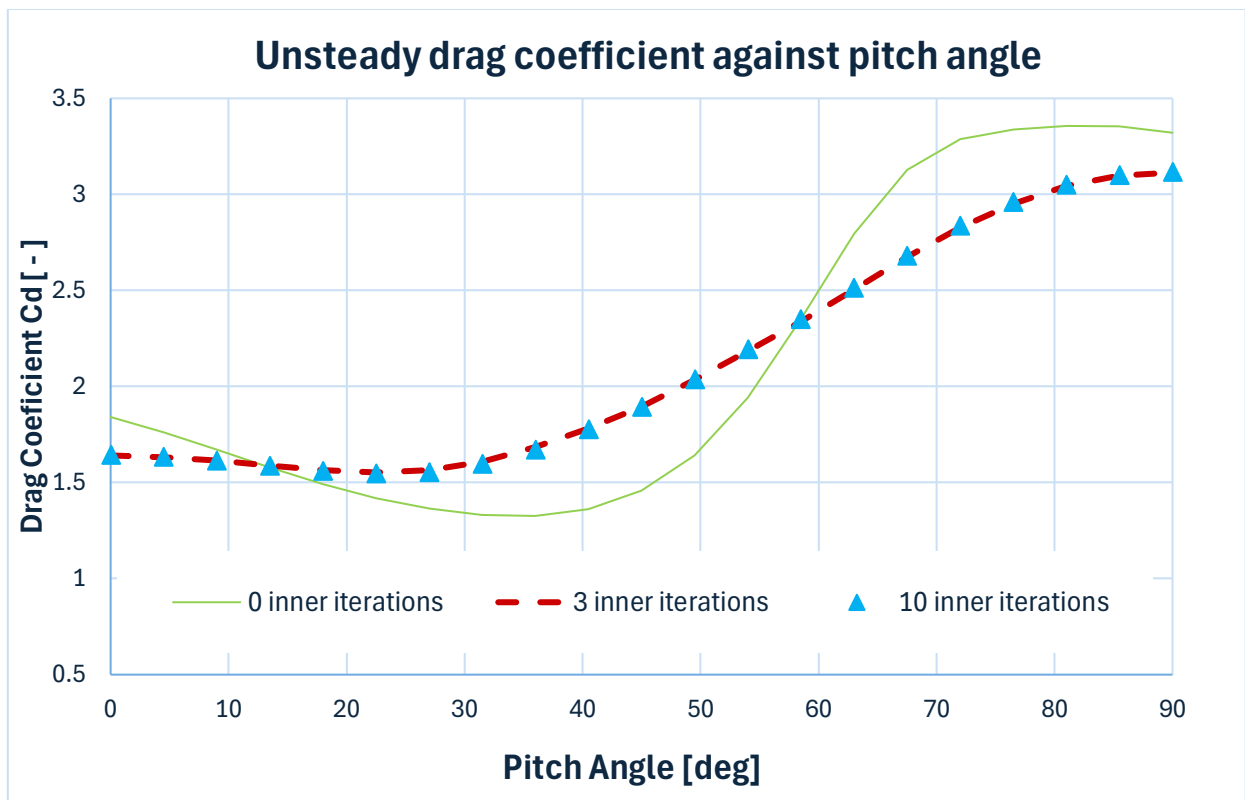


Figure 15: Coefficient of drag with different inner iterations

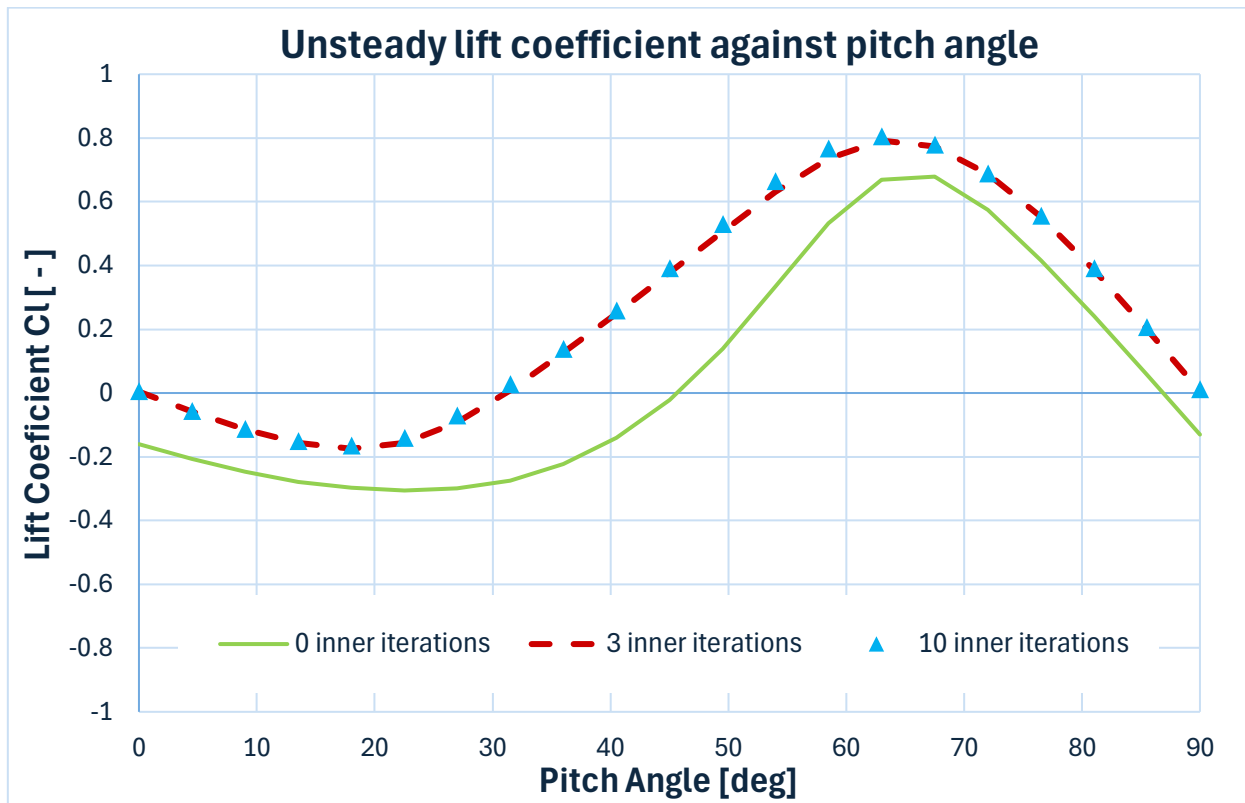


Figure 16: Coefficient of lift with different inner iterations

Three different angular velocities were then simulated in order to see the effect of the angular velocity on the overall aerodynamic coefficients. These were chosen as 45, 150 and 300 degrees/second, mapping a wide variety of angular speeds from slow to fast. From the results of these final simulations shown in Figure 17 and Figure 18 and plotted against steady-state for comparison, it can be seen that the angular speed of the cylinder does have an influence on both aerodynamic coefficients. While the slow speeds show similar trend and are quite close to the steady case, increasing the angular speed alters significantly not only the trend but also the magnitude of the coefficients. Most obvious difference for the 300 deg/sec case is the shift of the maximum drag coefficient from 90 degrees of pitch angle to 85 degrees. This is owed to the fact that with increased angular velocity, the flow around the cylinder becomes more complex and turbulent, and moves further away from the steady-state case. Appendix B plots the steady-state aerodynamic coefficients against the unsteady ones at the three different angular velocities for a clearer comparison.

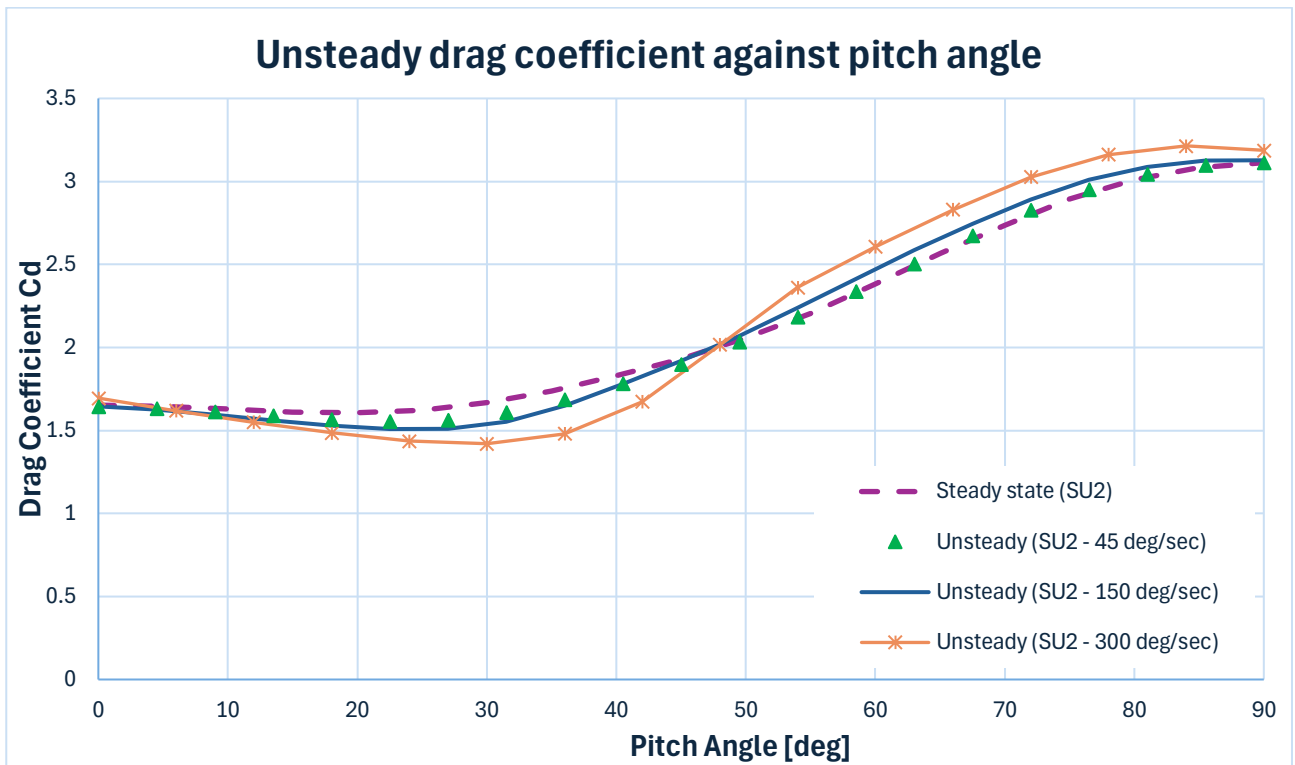


Figure 17: Unsteady drag coefficient at different angular velocities

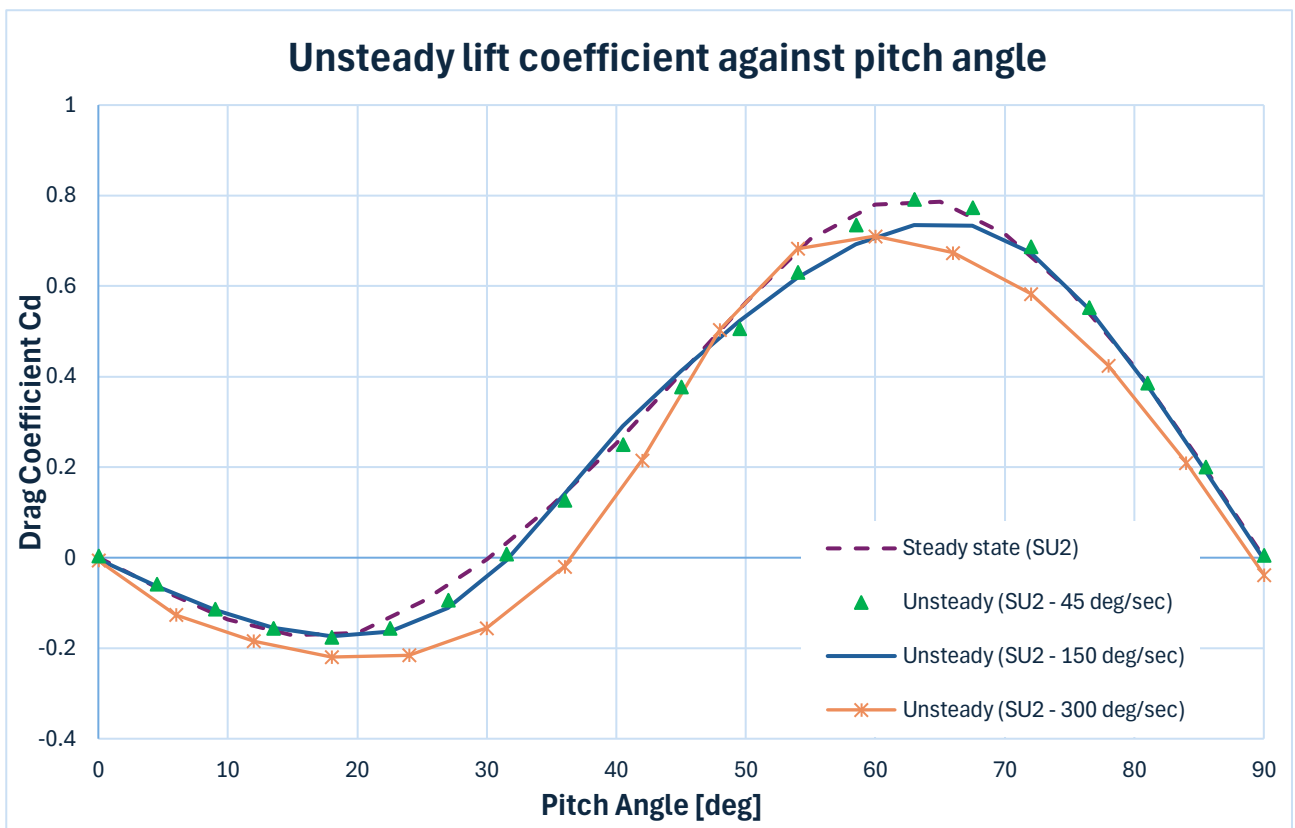
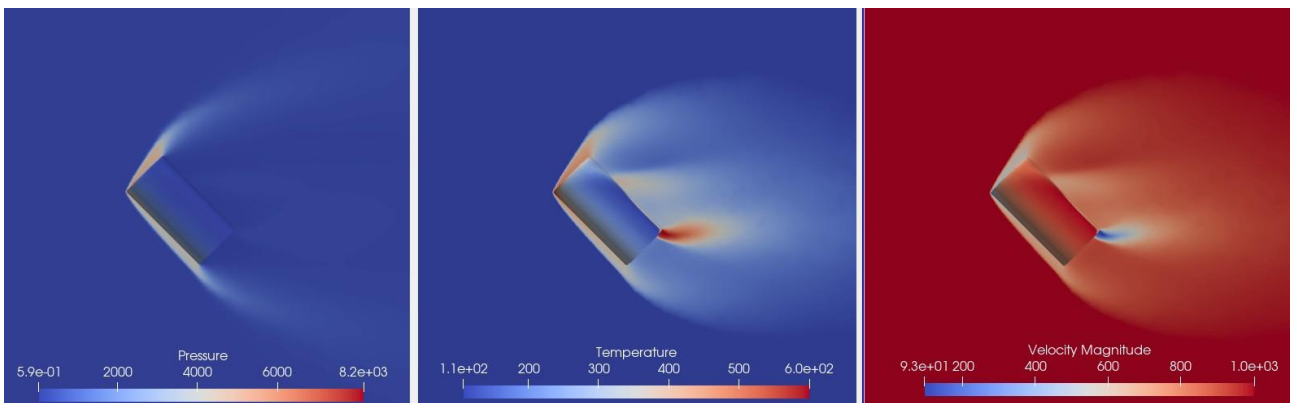
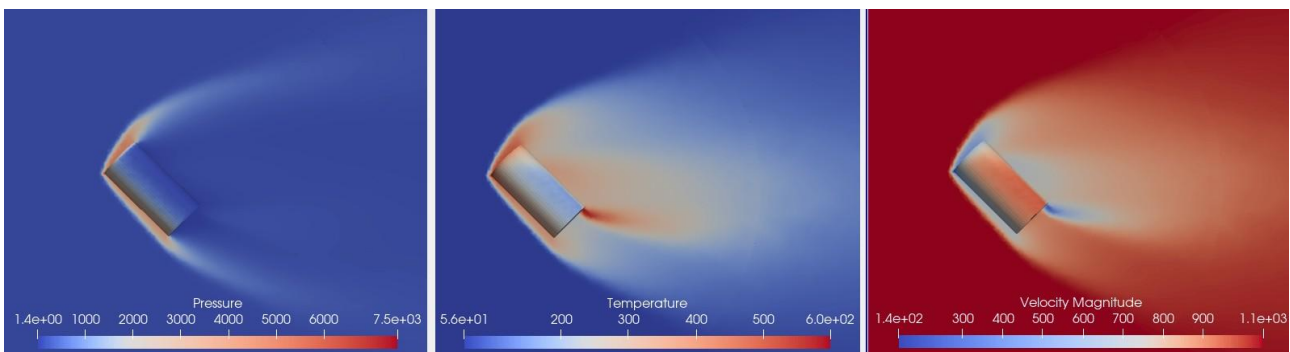


Figure 18: Unsteady drag coefficient at different angular velocities

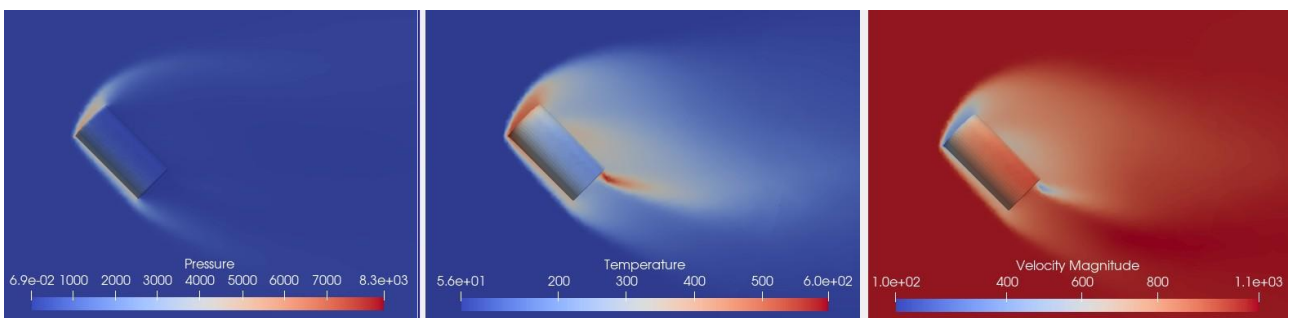
The above differences in the aerodynamic coefficients are also evident by observing at a snapshot of the cylinder rotating and visualizing the flow around it. Figure 19 takes a look at when the cylinders at each angular velocity reach a 45-degree pitch angle. The steady-state result is again illustrated here to help compare the different unsteady cases. It is evident that different angular velocities have an effect on the flow around the cylinder. While the 45 deg/sec and the 150 deg/sec look quite similar, the difference is more pronounced when viewing the 300 deg/sec. From the temperature plot it seems that while at slower angular velocities as well as at the steady-state simulation a thermal boundary layer seems to be forming at the lower surface of the cylinder, when the angular velocity is increased, this layer is not as strong. The same is also observed when it comes to the pressure. While schlieren images cannot be formed that can help better visualize the shocks created, from what can be observed by these figures is that the bow shock formed is not as pronounced, indicating that angular velocities can have a significant effect on the behavior of the shocks formed. Finally, the velocity plot can clearly show how much more turbulent the air around the body becomes and how the recirculation region is affected.



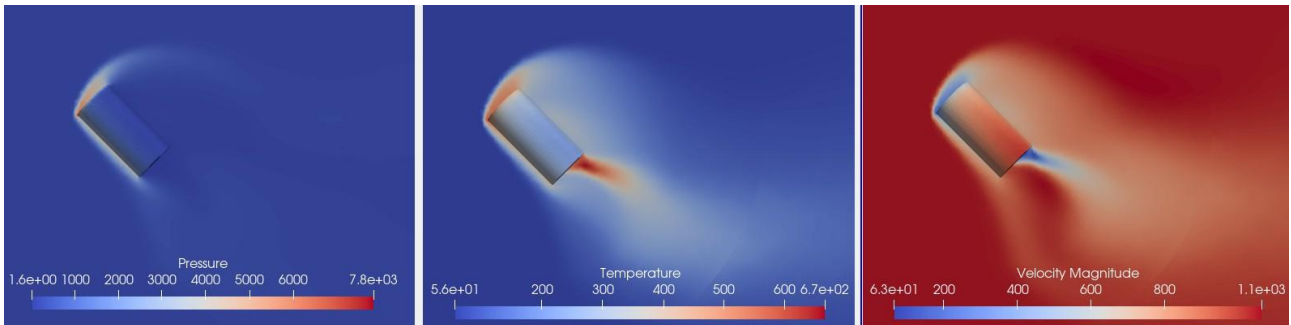
a) Steady state simulation at 45 degrees pitch angle



b) Snapshot of unsteady simulation with 45 deg/sec angular velocity at 45 degrees pitch angle



c) Snapshot of unsteady simulation with 150 deg/sec angular velocity at 45 degrees pitch angle



d) Snapshot of unsteady simulation with 300 deg/sec angular velocity at 45 degrees pitch angle  
 Figure 19: Pressure, temperature and velocity for steady state simulation and unsteady ones at 45 degrees of pitch angle.

## 6. Limitations and Suggested Future Work

The findings in this research are subject to certain limitations. One of the most important limitation of the present research as already pointed out in previous sections, was the fineness of the mesh. Due to the limited amount of computational power available, the mesh could not be as fine as the specific problem requires in order to capture all of the hypersonic flow characteristics. Even though the error in the aerodynamic coefficients which was the main objective of the present study is insignificant, when one wants to compare the visual effects of the flow, more computational power is required. Separation bubbles, reattachment shock and expansion fans, are some of the flow characteristics that would have been captured had the mesh been finer in those areas.

Another limitation identified is the wall treatment. While this is an assumption that the researcher has to make, better wall treatment would allow for more accurate calculation of aerodynamic coefficients. This is evident by the simulations carried out where one was using the slip condition and the other the no-slip condition. As pointed out in the text before, when it comes to rarefied flow, these two standard assumptions may be a bit too strict and special care needs to be taken when treating the quantities near the wall.

This research has thrown up some questions in need of further investigation. First of all, a much finer mesh should be used along the whole domain, and not just around the cylinder. This would prove the capabilities of SU2 in capturing shocks and other hypersonic flow phenomena. In addition, a different solver could be utilized in SU2. In particular, NonEquilibrium MOdeling (NEMO) solver could be used. SU2-NEMO has the capability of capturing high Mach number non-equilibrium flows and can be either used with the native thermochemical library or with Mutation++, allowing for any mixture of gas to be simulated. Performing such simulations with this solver would theoretically improve the results further as in real scenarios of re-entry, chemical reactions that take place will significantly influence the flow around the body. [8]

Further research could be done to investigate the different aspect ratios, sizes and shapes of the cylinder and examine whether the increase in velocity has a similar effect as to the present cylinders' geometry. Finally, the multi-zone capabilities of the software can be further studied by also having a angular speed not only in pitch but also in yaw and roll axis at the same time. Therefore, with 3 degrees of freedom, the simulation would be better in describing a cylindrical body tumbling into the atmosphere, as usually the case of re-entry objects includes random rotations along all three axes.

## 7. Conclusion

The present study was designed to firstly perform steady-state simulations of inclined cylindrical bodies in hypersonic flow, proving the validity of SU2, and secondly exploit the multi-physics capabilities of the software to perform unsteady simulations and observe the effect the angular velocity of the body has on its aerodynamic coefficients.

By the end of the study, SU2 was validated to be a powerful tool that can capture with great accuracy real-life flow phenomena like re-entry of objects into the atmosphere at hypersonic speeds. The steady-state aerodynamic coefficients calculated are almost identical to other CFD software like US3D and follow closely the experimental ones with negligible error. When it comes to visualizing the flow, some of the features expected to be there were not very pronounced, however this is not a limitation of the software but rather of the computational power available for this project.

With its multi-zone capabilities, it was possible to simulate an unsteady simulation with a 1 degree of freedom around the y-axis, simulating a rotation in pitch. Doing so, it was possible to compare the steady-state simulations with the unsteady ones. Three different angular velocities were tested; 45 deg/sec, 150 deg/sec and 300 deg/sec. The aerodynamic coefficients as well as the flow characteristics around the cylinder seemed to be varying when compared with the steady-state results and especially for the 300 deg/sec case. The results of this project support the idea that the unsteady nature of the flow around the rotating cylinder does have an influence on its aerodynamic coefficients, however this greatly depends on the angular velocity.

The current research was not specifically designed to evaluate and capture the hypersonic flow characteristics around the cylinder as the computational power was limited. As pointed out further mesh refinement and also a more appropriate wall boundary condition would help improve the results and are suggested for future research. In addition, different types of solvers are encouraged to be used and in particular SU2-NEMO while also different geometries of cylindrical bodies to study the effect this has on the aerodynamic coefficients.

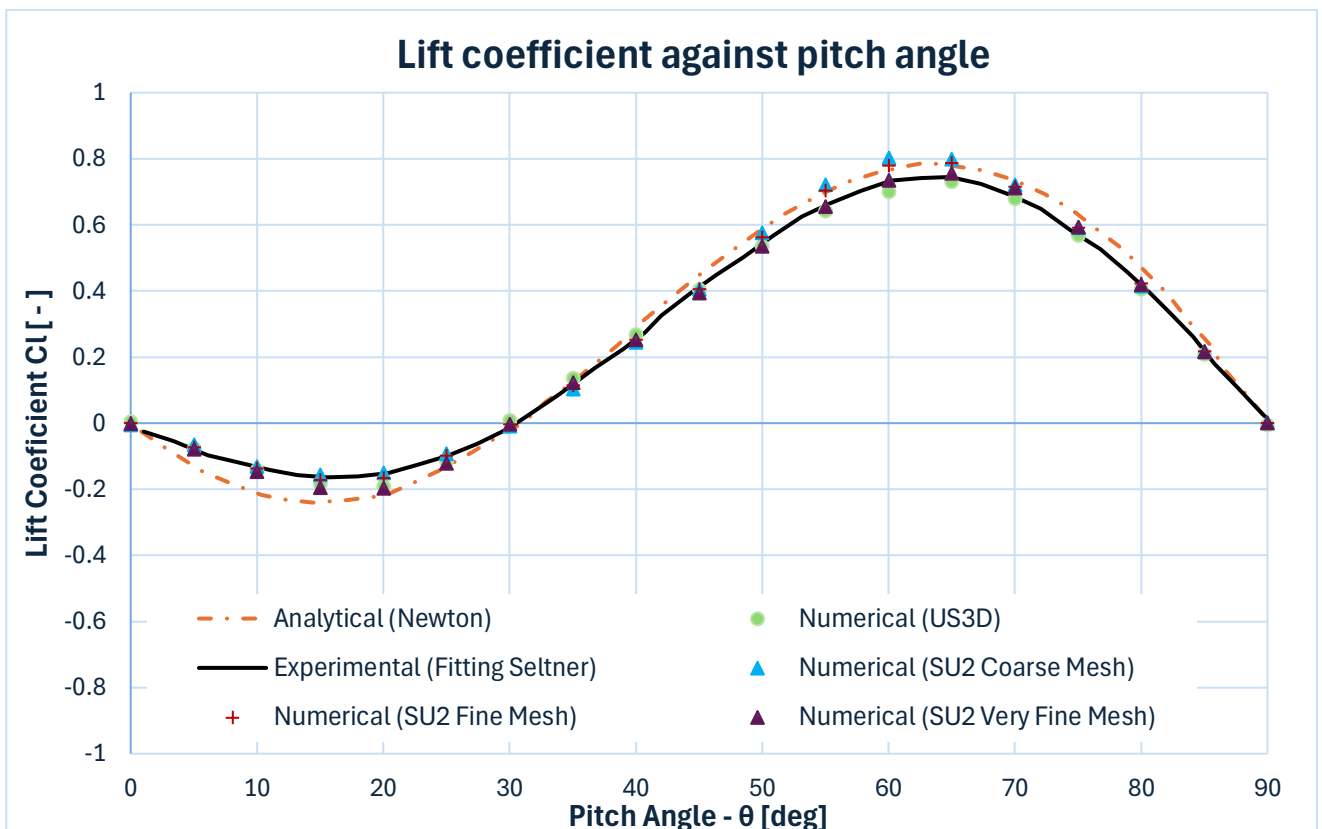
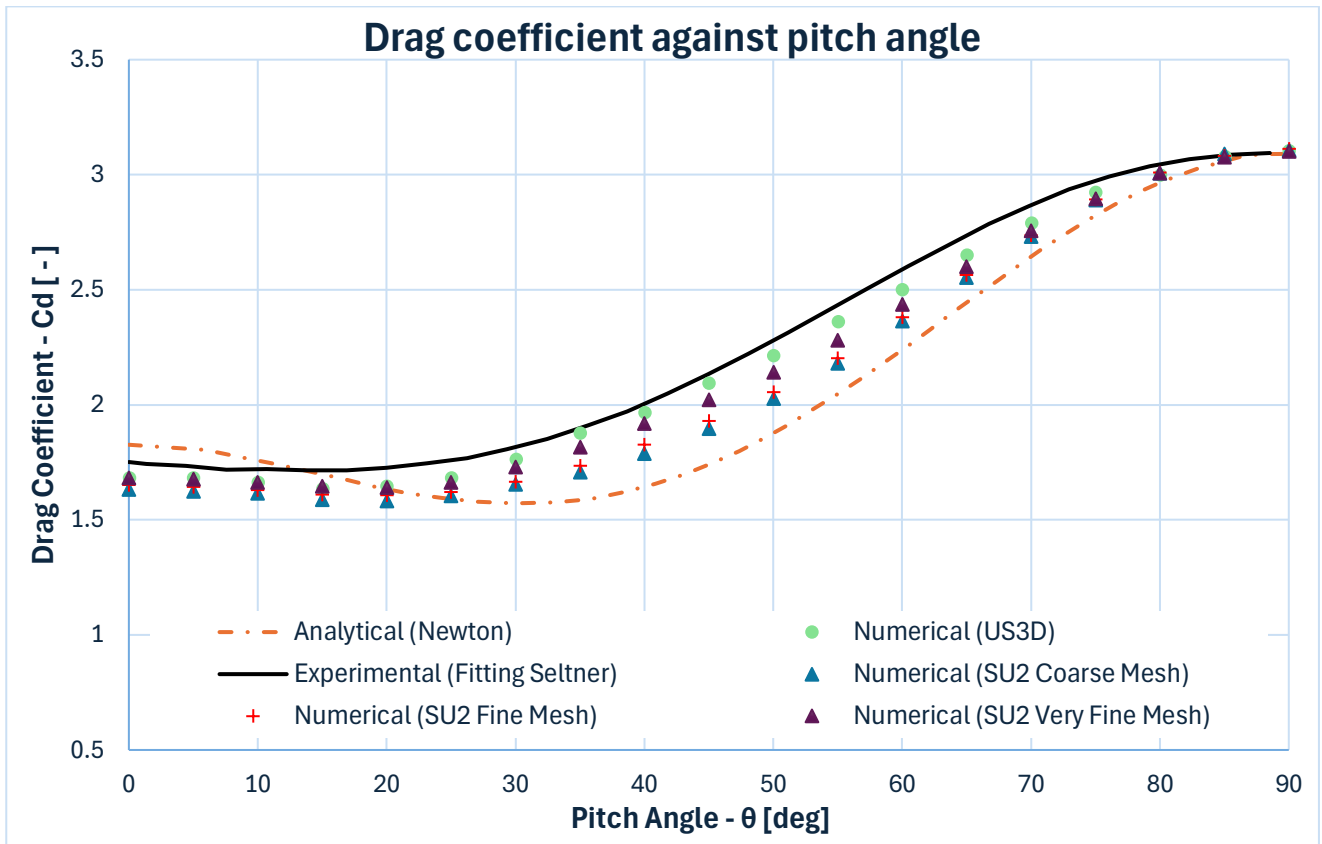
## Acknowledgements

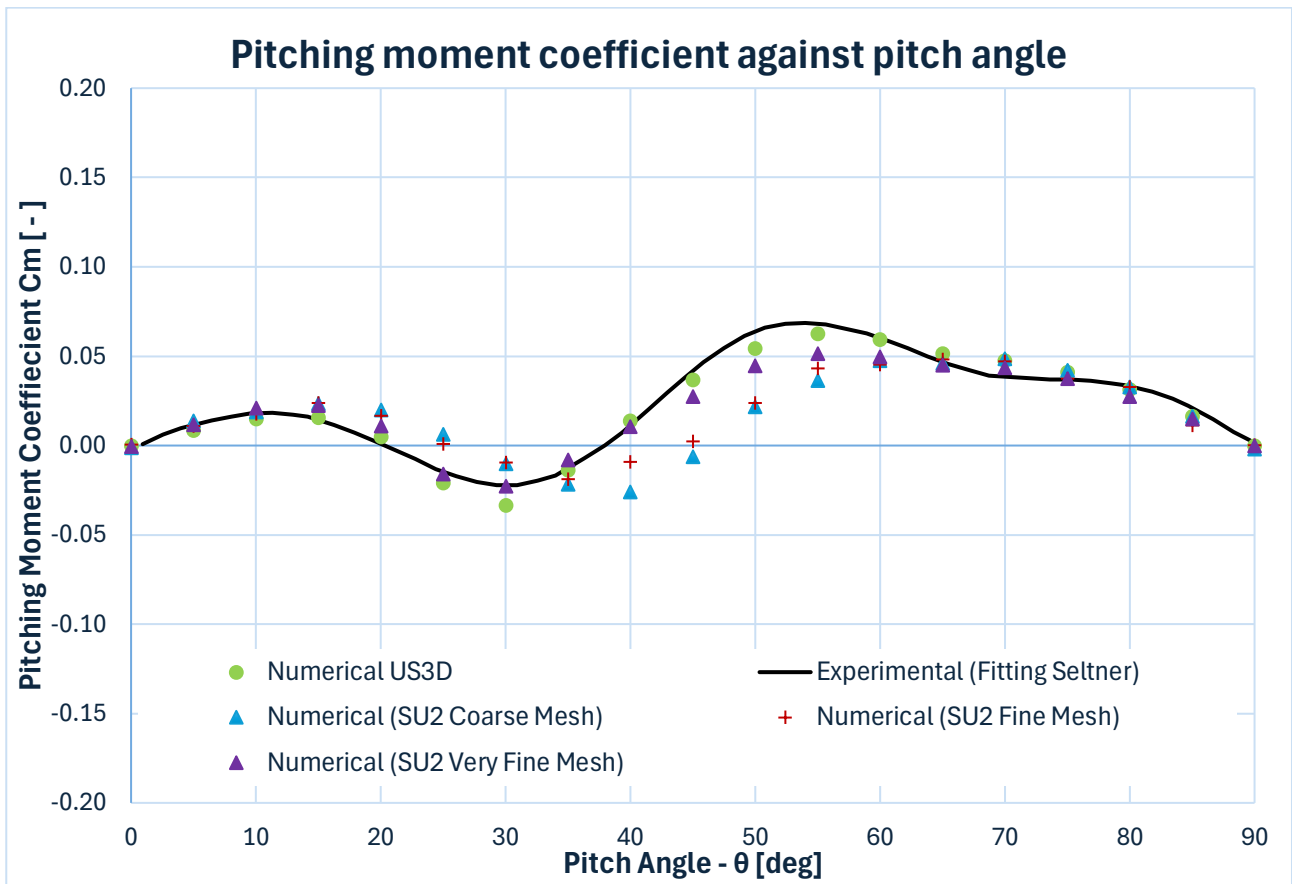
I would like to express my gratitude to Professor Vigeveno, my supervisor, for his enduring support and encouragement throughout my project. Professor Vigeveno has constantly provided invaluable guidance and assistance in my work and offered me the confidence I needed in my research.

## References

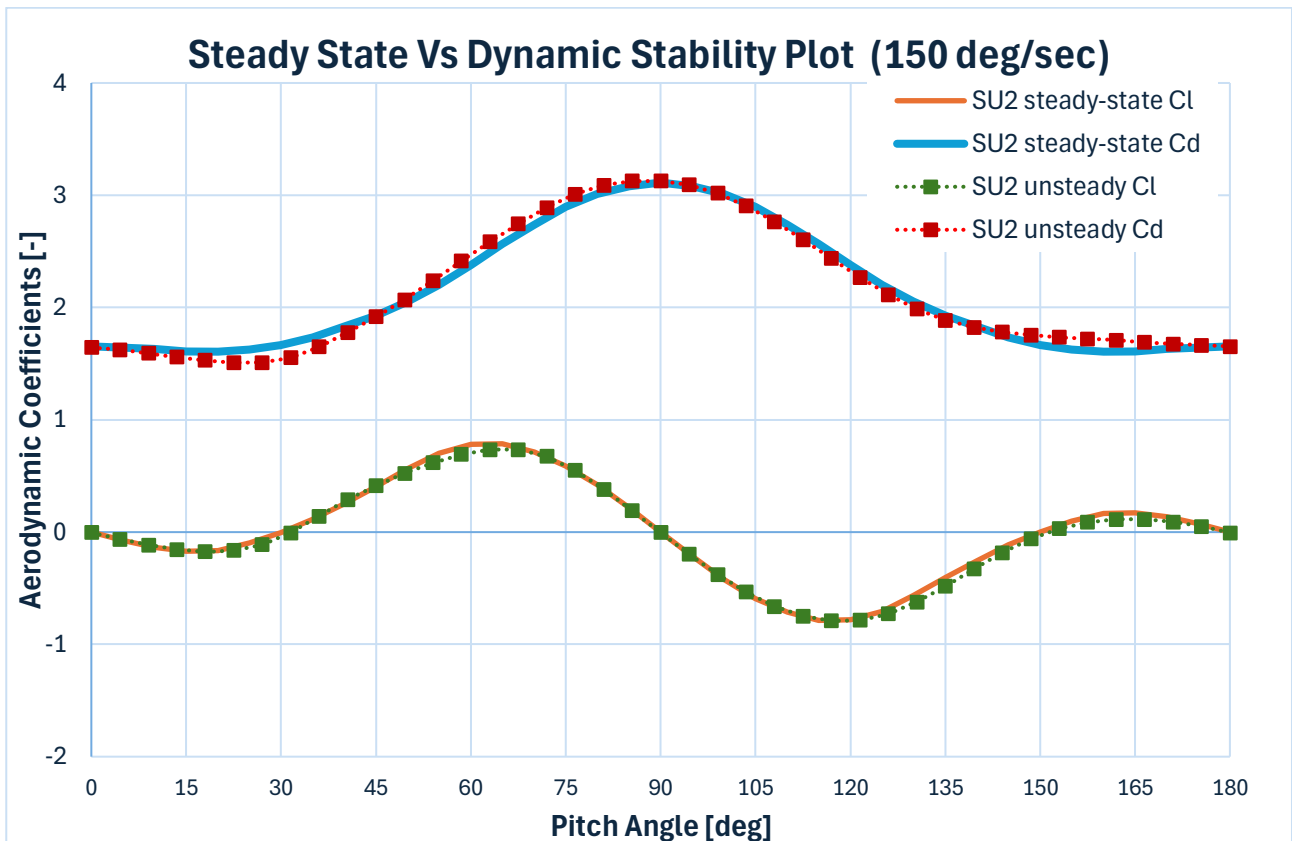
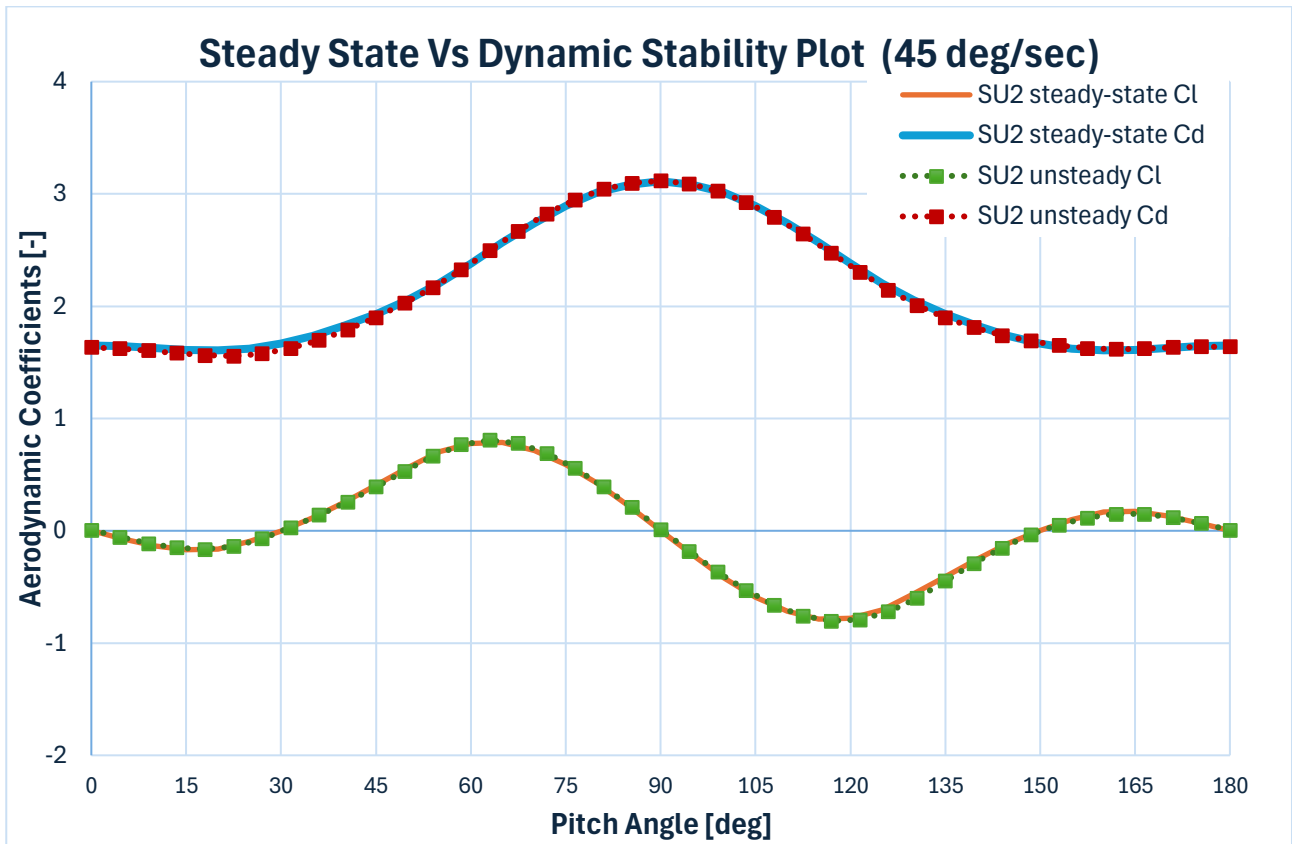
- [1] Matney, M. "Measuring small debris – what you can't see can hurt you"  
Available at: <https://ntrs.nasa.gov/api/citations/20160011226/downloads/20160011226.pdf>  
(Accessed 10<sup>th</sup> January 2025)
- [2] Seltner, P. M., Willems, S., Gülhan, A., Stern, E. C., Brock, J. M. & Aftosmis, M. J., 2021. Aerodynamics of inclined cylindrical bodies free-flying in a hypersonic flowfield. *Experiments in Fluids*.  
Available at: <https://doi.org/10.1007/s00348-021-03269-6>  
(Accessed 10<sup>th</sup> September 2024)
- [3] Economon, T., D., Palacios, F., Copeland, S., R., Lukaczyk, T., W. & Alonso, J., J., 2016. SU2: An Open-Source Suite for Multiphysics Simulation and Design, *AIAA Journal*, vol. 54, No. 3.  
Available at: <https://doi.org/10.2514/1.1053813>  
(Accessed 10<sup>th</sup> September 2024)
- [4] Palacios, F., Economon, T., D., Aranake, A., C., Copeland, S., R., Lonkar, A., K., Lukaczyk, T., W., Manosalvas, D., E., Naik, K., R., Padron, A., S., Tracey, B., Variyar, A. & Alonso, J., J., 2014.  
Stanford University Unstructured (SU2): Open-source Analysis and Design Technology for Turbulent Flows. *AIAA SciTech*  
Available at: <https://doi.org/10.2514/6.2014-0243>  
(Accessed 10<sup>th</sup> September 2024)
- [5] Lees, L., 2003 Hypersonic flow. *Journal of Spacecraft and Rockets*, vol. 40(5):700–735.  
Available at: <https://doi.org/10.2514/2.6897>  
(Accessed 12<sup>th</sup> January 2025)
- [6] Zade, A., Q., Renksizbulut, M. & Friedman, J., 2008.  
Slip/jump boundary conditions for rarefied reacting/non-reacting multi-component gaseous flows  
*International Journal of Heat and Mass Transfer*  
Available at: <https://doi.org/10.1016/j.ijheatmasstransfer.2008.02.044>  
(Accessed 20<sup>th</sup> November 2024)
- [7] SU2 User-Guide, Basics of Multi-Zone Computations  
Available at: [https://su2code.github.io/docs\\_v7/Multizone/](https://su2code.github.io/docs_v7/Multizone/)  
(Accessed 20<sup>th</sup> September 2024)
- [8] Maier, W., T., Needles, J., T., Garbacz, C., Morgado, F., Alonso, J., J. & Fossati, M., 2021. SU2-NEMO: An Open-Source Framework for High-Mach Nonequilibrium Multi-Species Flows, *MDPI*  
Available at: <https://doi.org/10.3390/aerospace8070193>  
(Accessed 10<sup>th</sup> September 2024)

## Appendix A: Aerodynamic coefficients with different mesh refinement

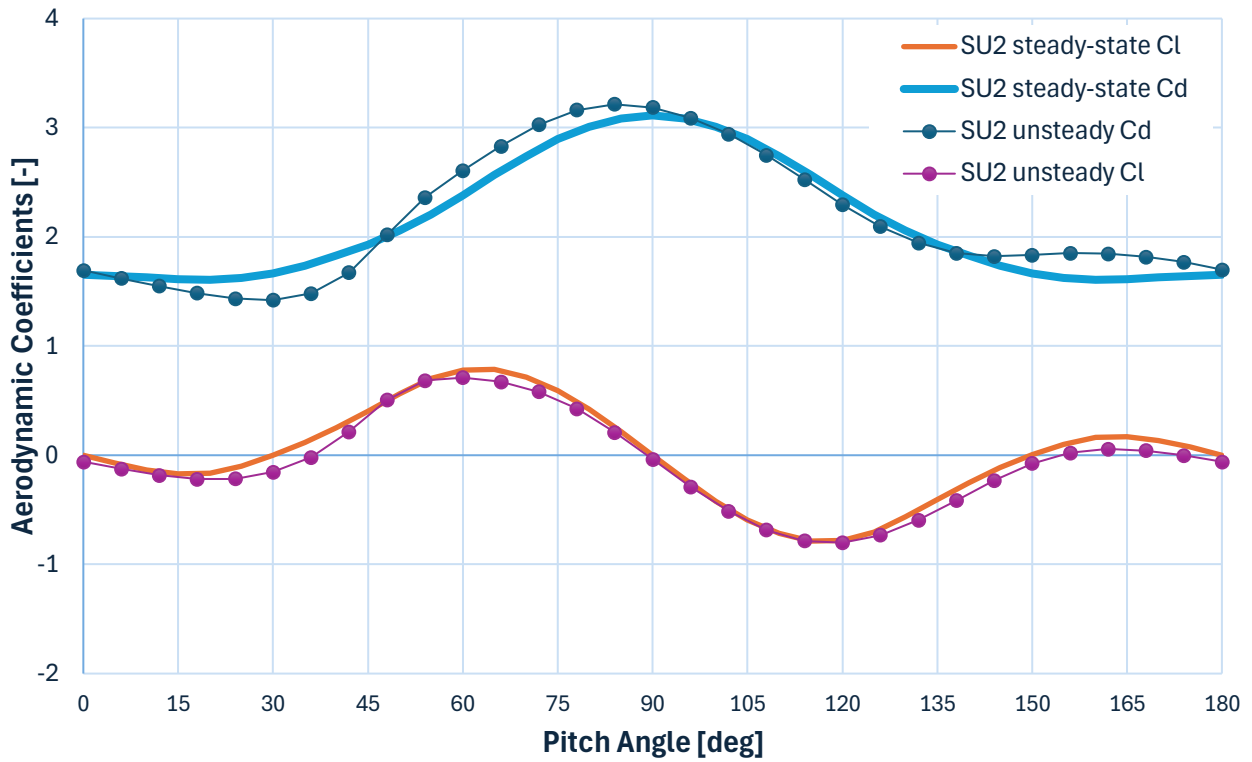




## Appendix B: Steady-state against unsteady aerodynamic coefficients for the three angular velocities



### Steady State Vs Dynamic Stability Plot (300 deg/sec)



## Abstract in lingua italiana

La previsione del comportamento degli oggetti di rientro ha suscitato molto interesse negli ultimi anni. Il calcolo dei coefficienti aerodinamici di tali oggetti, così come la caratterizzazione del flusso attorno a essi, è fondamentale per prevederne la traiettoria. In questo progetto, vengono simulate sia l'analisi stazionaria che quella non stazionaria di un corpo cilindrico circolare inclinato in un flusso ipersonico durante il rientro nell'atmosfera terrestre. Le simulazioni stazionarie vengono eseguite utilizzando il software SU2 CFD, con l'obiettivo di convalidare i risultati rispetto a quelli sperimentali e computazionali precedenti. Inoltre, vengono condotte simulazioni non stazionarie con tre diverse velocità angolari attorno all'asse di beccheggio per valutare l'influenza della velocità angolare del corpo sia sui coefficienti aerodinamici che sul flusso attorno al corpo. Il progetto ha dimostrato le capacità di SU2 nella modellazione di flussi ipersonici comprimibili, fornendo risultati simili a quelli sperimentali e computazionali. Le sue capacità multizona sono state sfruttate nelle simulazioni non stazionarie, dove si è concluso che la velocità angolare del cilindro influisce sui coefficienti aerodinamici del corpo, con differenze più pronunciate alle velocità angolari più elevate.

Parole chiave: Simulazione CFD, flusso ipersonico, aerodinamica dei corpi cilindrici, caratterizzazione del flusso di rientro atmosferico