

Structural & Fluid Analysis AMCA6010

Steffan Rhys Tubb (1910092)

Table of Contents

Introduction	2
History	2
Material	3
Understanding Flow	4
Reynolds number	7
Computational fluid dynamics	9
Prototype testing	10
Conclusion	11
Bibliography	12

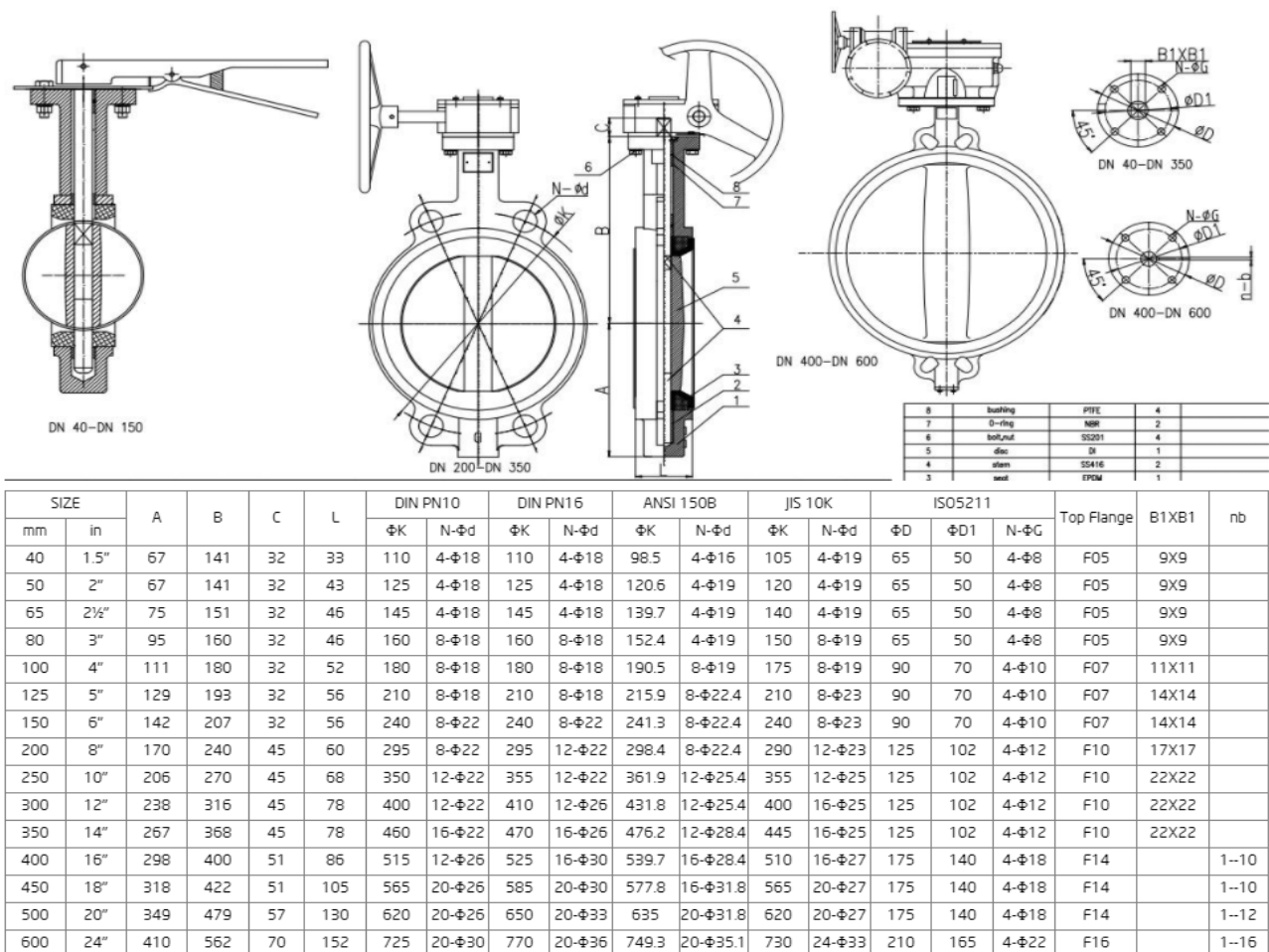
Introduction

I was tasked with looking into boundary flow through a butterfly valve that was made out of glass fibre reinforced plastic. We were given a CAD file of a butterfly valve to help us. To get a better understanding of the flow through the valve I decided to look into various methods of testing and reasoning behind the behaviours of the flow. I also wanted to have a look into the material selection to see why it was chosen for the valve. Here are my findings:

History

James Watt was the first noted person to use butterfly valves. He did so on the prototypes of his steam engine in the late 18th century. They were typically made up of a cast iron body and a metallic butterfly disk that could rotate by turning the handle attached to the body. After world war two, manufacturers started to develop the butterfly valves to make them more practical and efficient. They did this by adding rubber seals inside the lining of the body. This helped seal the valve entirely and minimising minute leakages. It also meant they could be used in a wider range of industries. For years one of the main applications of the valves was in the piping on oil fields. For many years the butterfly valve was one of the least standardised valves in use. This was down the fact that there were several types and different materials used in the production of them. Then, in 1982 the high performance butterfly valve joined the world of standardisation (*A complete guide to butterfly valves / Actuation Valve & Control, 2021*).

This meant that manufacturers had to make valves at standard sizes and materials so that customers could purchase like for like parts and replacements from suppliers. It also meant that the valves would have to meet pressure testing criteria such as the API standard 598 test which tests the leakage rates of the valves. This brought the quality and performance of the valves to a minimum level. Below is an example of the different standard sizes for the different components that make up a butterfly valve:



(One stem no-pin wafer butterfly valve, 2021)

Material

As the butterfly valve was going to be made out of glass fibre reinforced plastic. I decided to look into the reasoning behind the selection and how it might impact on the performance of the valve. I researched into the properties and characteristics of glass fibre reinforced plastic (GFRP). Here is the summary of my findings on GFRP and why it's advantageous to be used for the purpose of the butterfly valve:

Ease of fabrication - GFRP can be easily fabricated using standard hand tools such as circular saws, bandsaws and hacksaws. This is advantageous as you don't need expensive specialised tools to produce the valves and also it makes making alterations to the valves a lot easier.

High impact resistance - GFRP can resist severe and sudden point loading and avoids permanent distortion. If GFRP is deformed due to impact, it will return to its original shape without having to incur expensive replacements or repairs.

High corrosion resistance - GFRP offers exceptional corrosion resistance. This makes it highly tolerant of the most aggressive of environments. Its resistance against anions such as chloride demonstrates its ability to withstand harsh conditions and shows its durability. GFRP offers a long-term practical solution to overcome the corrosion challenges facing

many industries that use valves today. Materials like steel and cast iron can't compete with GFRP on corrosion resistance.

High strength - GFRP's superior tensile strength is equal to or greater than equivalent steel profiles. Despite its lightweight properties, GFRP offers impressive strength-to-weight load-bearing performance when compared to steels. The high glass-to-resin ratios used in GFRP outperforms and outlasts traditional materials such as steel and cast iron.

Lightweight - GFRP is approximately 75% lighter than its steel equivalents. The saving of weight eliminates the need for heavy lifting equipment and significantly reduces transport costs. As it can be easily cut and transported onsite, the risk of manual handling injuries is greatly reduced.

Non-conductive – GFRP is a highly effective electromagnetic and thermal insulator. It is electronically transparent and unaffected by radio wave frequencies and electromagnetic fields. The non-conductive properties of GFRP make it ideally suited for use on electrically hazardous sites. The non-sparking qualities of GFRP make it suitable for locations where combustible gases may be present such as gas power stations.

Low maintenance - GFRP is an incredibly resilient and durable material that delivers outstanding cost benefits when replacing other traditional materials such as steel and aluminium. GFRP has an average lifespan of around 50 years when used for industrial purposes which is a very long time. *(Why GRP? | FRP Rebar | Competitive Prices | Engineered Composites, 2021)*

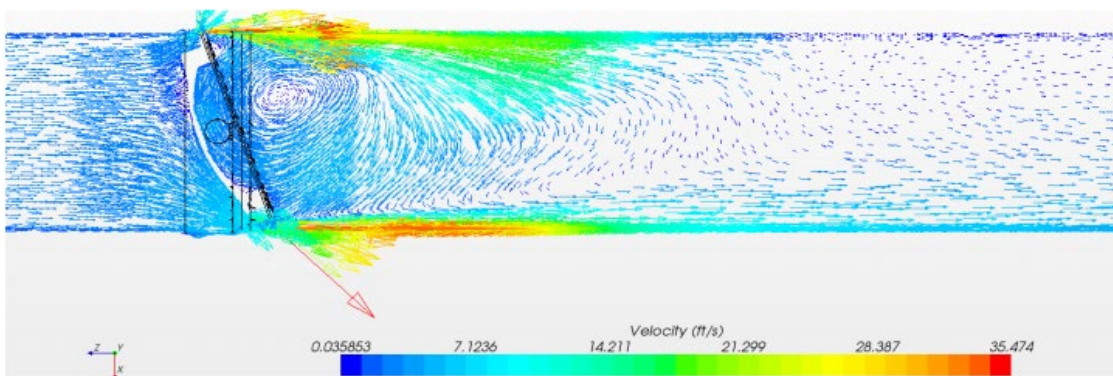
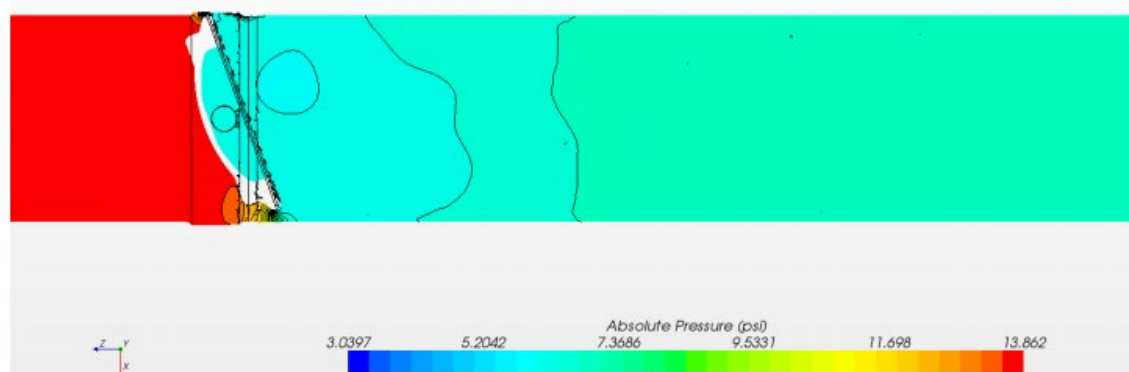
Understanding Flow

The flow of the fluid around the valve will change as the valve opens. This is due to a process called boundary layer separation. Boundary layer separation occurs when the flow can no longer follow the path of the body around which it flows and separates turbulently from it. When a fluid flows around blunt bodies or experiences sharp transitions, the boundary layer begins to detach itself from the body surface. Further down from the separation point, vortices begin forming which disturb the flow of the fluid and begin to make it turbulent.

Some scenarios can negate turbulence, but they must have the right conditions to do so. For example, a fluid with a high viscosity can do it. This is due to the internal cohesion being large enough to resist the inertial forces. Due to the intermolecular forces acting, the surface of the solid tries to bind itself to the fluid. This also leads to the fluid sticking to the surface. The layer above this adherent fluid layer will not be able to tear itself away from it, because intermolecular pressure forces act between the layers. As a result, any flow flowing around a body is tempted to follow the profile of the surface *(Flow separation (boundary layer separation) - tec-science, 2021)*.

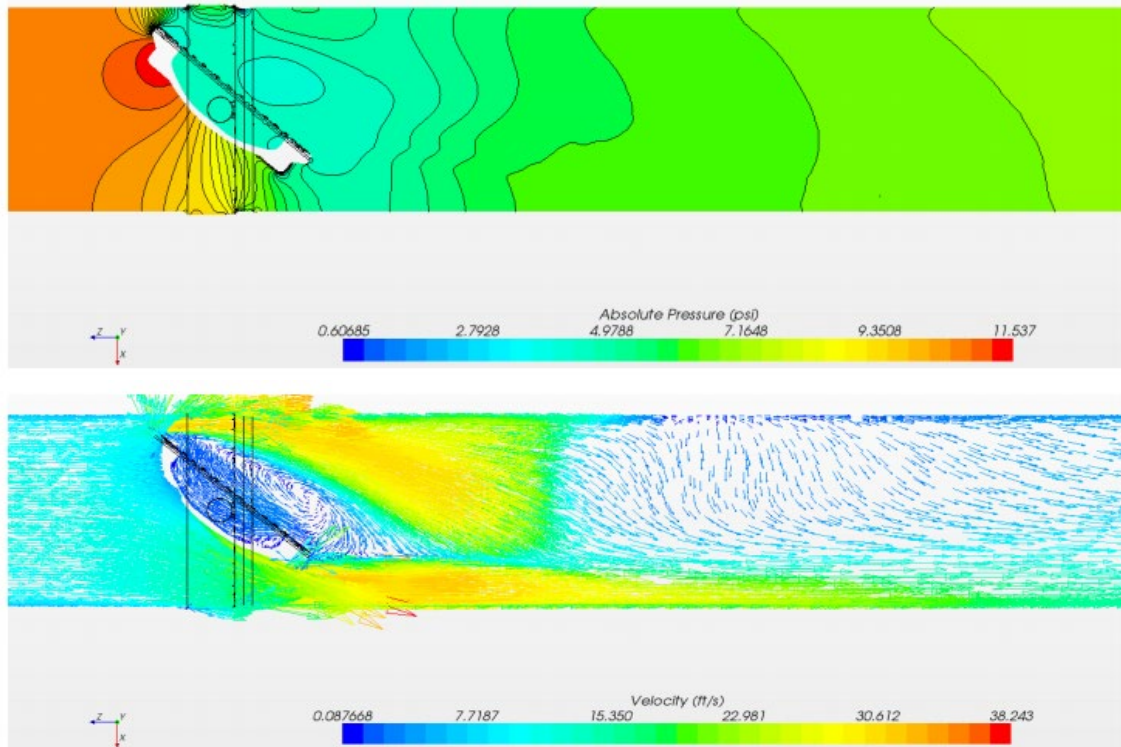
When the butterfly valve is fully closed the pressure of the fluid is extremely high at the back of the disk, this means there is little to no velocity as there is nowhere for the fluid to go. As the valve begins to open as little as 20 degrees a pressure drop will occur, resulting from the fluid beginning to flow around the gaps between the disk and the body. The

velocity of the fluid will begin to increase as it now has room to flow. This vastly increases the turbulence of the flow on the edges of the disk as the available path of flow is slowly changing causing disturbances in the flow pattern.



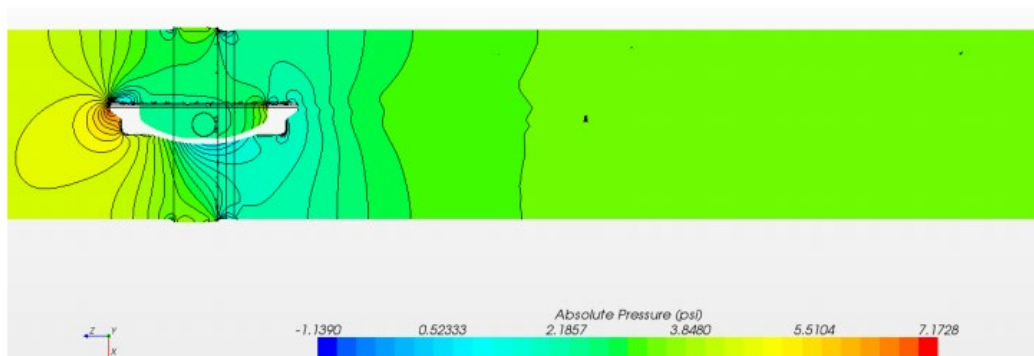
(Del Toro, 2012)

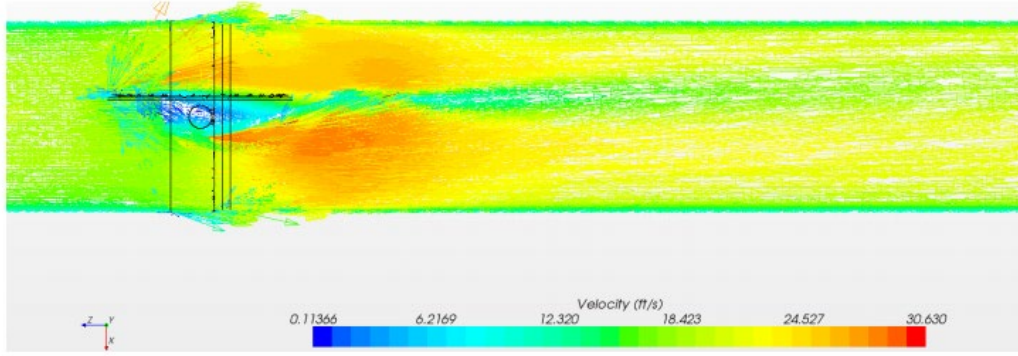
As the valve opens over the halfway point at 50 degrees the pressure is starting to change towards an equilibrium between the entry and exit sides of the valve. The area of higher velocity is now much larger than when the valve started opening, this is due to increased vortex movement after the initial boundary separation points of the opening of the valve. Now the rivets are becoming exposed but aren't making much of a difference as their current angle is favourable to the flow of the fluid.



(Del Toro, 2012)

At 90 degrees when the valve is fully open the pressure is now fairly similar on both sides, this suggest the fluid is flowing at a constant rate either side of the valve. The area of higher velocity at the exit side of the valve is at its biggest. This is because vortices are compounding with each other. The flat side of the disk creates a lower amount of turbulence than the domed side as fluid prefers to flow along straight lines as opposed to curved lines. The rivets are now causing more turbulence as they are running perpendicular to the flow of the fluid. This causes the fluid to change its direction of flow slightly. This in turn causes some boundary separation which further on down the pipeline turns into turbulence vortices.





(Del Toro, 2012)

I think the valve has been designed this way as its easier to hold the disk onto the rotating central shaft and makes repairs or replacement disks easier to undertake. From a cost point of view this would save time and money. As the flow will be experiencing turbulence anyway, I don't think having the rivets where they are will contribute to the effect too much more on top. If it was in a transitional range then I would suggest removing the rivets but as the Reynolds number was high and suggested turbulent flow then there would be no need to try and eliminate the rivets.

Reynolds number

The Reynolds number (Re) is a dimensionless number that predicts fluid flow by expressing the ratio of inertial forces to the viscous forces. Inertial forces are resistant to change or motion and are represented by the product of the density (ρ) times velocity (V) times the change in velocity ($\frac{dV}{dx}$). The viscous forces are represented by the dynamic viscosity coefficient (μ) times the second gradient of the velocity ($\frac{d^2V}{dx^2}$). The equation for the Reynolds number therefore can be expressed as:

$$Re = \frac{\rho \cdot V \cdot \frac{dV}{dx}}{\mu \cdot \frac{d^2V}{dx^2}}$$

The change in velocity is proportional to the velocity divided by the length (L), this could also be expressed as diameter (D). Equivalently, the second gradient of the velocity is proportional to the velocity divided by the length squared (L^2). The equation can then be written as:

$$Re = \frac{\rho \cdot V \cdot \frac{V}{L}}{\mu \cdot \frac{V}{L^2}}$$

$$\therefore Re = \frac{\rho \cdot V \cdot L}{\mu}$$

If we use kinematic viscosity (ν) instead of the dynamic viscosity (μ) then the equation can be simplified even further as the kinematic viscosity is equal to the dynamic viscosity divided by the density.

$$\nu = \frac{\mu}{\rho}$$

$$\therefore Re = \frac{V \cdot L}{\nu} \quad (\text{Reynolds Number, 2021})$$

The Reynolds number is used to rule whether a flowing fluid is in a turbulent or laminar state. It is comprised of both static and kinetic properties of fluids but is described as a flow property as dynamic conditions are being investigated. For turbulent flows, the movement of the fluid is caused by the inertial forces resisting changes in the velocity of an object. Diversely, for laminar flows the viscous forces are resistant.

To estimate the Reynolds number for the fluid flowing around the butterfly valve, I had to find out information regarding the viscosity of the petroleum and also calculate the velocity of the fluid exiting the storage tank and through the butterfly valve.

I started off by calculating the velocity of the petroleum using the information given to me. I knew that the storage tank had a 5m depth, contained petroleum and was open to atmospheric pressure. With this information I could use Bernoulli's equation solve the problem.

The pressures at the open surface in the tank and at the outlet are both the same as they are open to the atmosphere, $P_A = P_B = P_{atm}$. Assuming the height drops very slowly due to the flow, we may set the velocity at the upper surface $v_A = 0$.

Bernoulli's equation becomes:

$$\frac{P_{atm}}{\rho \cdot g} + \frac{0}{2 \cdot g} + h_A = \frac{P_{atm}}{\rho \cdot g} + \frac{v_B^2}{2 \cdot g} + h_B$$

$$\therefore v_B^2 = 2 \cdot g(h_A - h_B)$$

$$\therefore v_B = \sqrt{2 \times 9.81 \times 5} = 9.9 \text{ m/s}$$

From the design of the valve given, the diameter of the entrance and exit of the valve was 194mm which is equal to 0.194 m.

I had to research online to come to a conclusion for the kinematic viscosity. I took petroleum as crude oil. This was because petroleum has a broad spectrum of by-products and it was hard to determine which one to use. I found a table of kinematic fluid viscosities that had crude oil with an API gravity of 32.6 degrees at between 23.2 and 7.1 centistokes (Liquids - Kinematic Viscosities, 2021). I took the middle ground for my calculation which is 15.15 centistokes. To convert centistokes to m^2/s I had to multiply it 10^{-6} .

Now I had all the information I needed to plug into the calculation and get answer for the Reynolds number for the scenario:

$$Re = \frac{9.9 \times 0.194}{15.15 \times 10^{-6}} = 126,772$$

This is quite a high number and would indicate a strong turbulent flow. This is in line with what I was expecting from the result as its quite a high velocity flowing through a relatively small pipe when compared to industrial sized ones. The only thing that would change the number is if I had a more accurate kinematic velocity.

Computational fluid dynamics

To try and get an understanding of how the valve was going to perform I needed to simulate how the fluid was going to flow through the valve. As many aerodynamic and fluid flow processes aren't simply quantifiable during a concept phase of a project, the only way for an engineer to be able to predict the performance and optimize his designs is to simulate them using software or use prototypes to conduct physical tests. I decided to look into Computational Fluid Dynamics (CFD) modelling software. Historically, fluid flow patterns were solved using complex hand calculations to generate numerical solutions. Then with the emergence of computers, engineers were able to put together a mathematical model of the physical case as well as using a numerical method to solve the problem. This reduced and in some cases even eliminated the need to physical prototypes as you could gather accurate results through computational power alone.

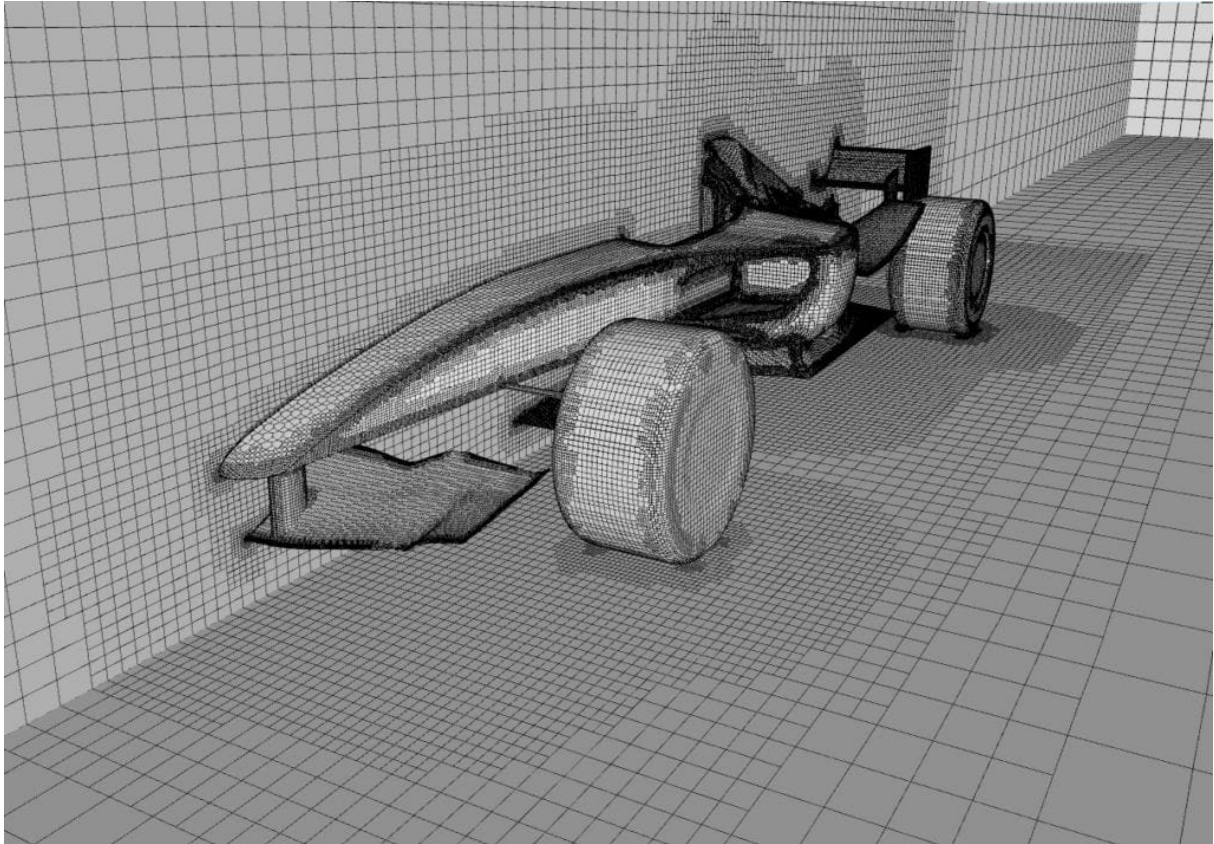
CFD software can conduct an examination of fluid flow in line with its physical properties such as velocity, temperature, pressure, density and viscosity. These properties of the fluid have to be considered simultaneously to generate an accurate picture of the flow of the fluid and offer solutions to any problems that could occur.

The reliability of the results of CFD software ultimately depend on the input of the user. If the mathematical model of the structure isn't created with the highest degree of accuracy then the numerical methods used to generate a reliable solution won't produce the desired results. Cost also plays a big factor when using CFD software. Licenses can be expensive for professional versions and they require costly high specification computers to run.

One innovative tool that CFD software uses is called meshing. Meshing is the process of splitting the mathematical model into small cells so that the software can read and interpret the simulation faster and easier than on a continuous point to point basis. Meshing is an important feature of CFD as it enables the software to generate a picture of the model using less computing power. Errors can occur resulting in the failure of the simulation while using meshing. This could come from the mesh being too coarse or not being fine enough. To eliminate problems like this the user of the software could manually alter the resolution of the mesh to the desired grade for analysis.

Another feature of CFD software is convergence. Convergence will create bigger cells on the basic structures of the model as they have less stress points to measure. Adversely, where there are more complex shapes and boundaries to the structure the convergence tool will split the cells into smaller groups for measuring as more stress data will be collected from these areas to create an accurate representation of the model. Below is a diagram example

of convergence on the mesh of a Formula 1 car which as you can see has coarser mesh around the more basic structures and finer mesh around the more subtle and intricate parts of the tyres and body of the car.



CFD software is used in many applications of testing. It can be used to determine whether a fluid has a compressible or incompressible flow. This is useful for when compressibility becomes a non negligible factor. It can also be used to determine whether a fluid is in laminar or turbulent flow conditions. When analysing turbulent flow, the software uses a lot of computing power to generate the simulation as turbulent conditions are constantly changing over time. The number of calculations required to solve turbulent flow patterns is far greater than that of laminar ones. CFD software can also simulate mass and thermal projections such as smoke propagation and gas distributions inside tanks and heat exchangers. (*What is Computational Fluid Dynamics (CFD)?* | SimScale | SimScale, 2021).

Prototype testing

It was proposed that a prototype was to be produced and tested in a laboratory to understand the fluid flow through the valve and to correlate it to the simulation software. To ensure that the test would be fair, the test would need to have the exact same set up as the CFD software. This would reduce the number of variables that could alter the results collected and possibly spoil the comparison between the two methods.

Before the testing could begin the butterfly valve would need to undergo a quality test to be tested for any leakages that could impact the data output. On the CFD simulation the software would assume a complete seal, so it would be required to ensure that this was also

the case when testing in the lab. This would be done using a specialised machine which seals both ends of the valve with plates. The valve would then be closed, and water would be pumped through at the same pressures that would be used in testing to ensure no leakages occurred.

The Laboratory test should be set up on a pipe manifold to hold the pipes in place. The diameter of the bore on the butterfly valve in the drawing provided was 7 and five eighths of an inch (193mm). The standard pipe that would fit this is a 7 inch pipe (*Pipe Sizes, 2021*). This would mean that the bore diameter of the pipe will be 7 inches, and the outer diameter would be 7 and five eighths of an inch. The pipes should be inserted either end of the prototype butterfly valve and secured using the 3 locator bolt holes on the body of the valve.

Firstly, a water inlet supply would need to be fitted so that the fluid could pass through the system. To keep the test as accurate as possible and to study the fluid flow, water would have to be used on the CFD simulation. Next, a flow meter would need to be inserted to control and measure the flow of the water passing through the valve in gallons per minute. The third component in the pipe chain would be the valve itself. This would need to be followed by two pressure gauges. The first of which would measure the pressure of the water exiting the valve in pounds per square inch (PSI). The second gauge would need to have measuring points before and after the valve to measure the head loss which is the difference in pressure before and after the water passes through the valve in inches of water. A collection tank would have to be put on the end of the system to collect the water exiting the system.

Once the system is set up, the tests should be run at designated flow rates and degrees of valve opening. Measurements should be recorded using the gauges accurately. If multiple measurements were to be used for the same sample conditions, then an average result must be calculated.

Once the results from the experiment have been collected and confirmed, the CFD simulation can take place. The CFD simulation must undergo the same conditions as the laboratory test. Once the data from the simulation is collected it should be compared with the data from the lab test. As long as the data matches within a pre-defined degree of accuracy then the simulation can be deemed accurate and can be used to visualise the fluid flow around the valve.

Conclusion

With the task completed, there were a few factors I feel I could have helped me further in understanding the fluid flow. The first being a defined fluid not just a general one such as petroleum. With this I could have got a more accurate Reynolds number instead of an estimated one. The second thing I feel would have helped me understand the flow a bit better would be to actually use the CFD software instead of researching it. Overall, I enjoyed the assigned task and found it beneficial to my understanding on fluid flow.

Bibliography

Actuation.co.uk. 2021. *A complete guide to butterfly valves | Actuation Valve & Control*. [online] Available at: <<https://www.actuation.co.uk/guide-butterfly-valves/#high-performance-types>> [Accessed 26 July 2021].

Hawstd.com. 2021. *One stem no-pin wafer butterfly valve*. [online] Available at: <<https://www.hawstd.com/products/butterfly-valve/one-stem-no-pin-wafer-butterfly-valve.html>> [Accessed 27 July 2021].

Engineered Composites - GRP Glass Reinforced Plastic. 2021. *Why GRP? | FRP Rebar | Competitive Prices | Engineered Composites*. [online] Available at: <<https://engineered-composites.co.uk/why-grp/>> [Accessed 27 July 2021].

tec-science. 2021. *Flow separation (boundary layer separation) - tec-science*. [online] Available at: <<https://www.tec-science.com/mechanics/gases-and-liquids/flow-separation-boundary-layer-separation/>> [Accessed 27 July 2021].

Del Toro, A., 2012. *Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors*. [online] Digitalcommons.usu.edu. Available at: <<https://digitalcommons.usu.edu/cgi/viewcontent.cgi?referer=&httpsredir=1&article=2454&context=etd>> [Accessed 28 July 2021].

Grc.nasa.gov. 2021. *Reynolds Number*. [online] Available at: <<https://www.grc.nasa.gov/www/BGH/reynolds.html>> [Accessed 17 July 2021].

Engineeringtoolbox.com. 2021. *Liquids - Kinematic Viscosities*. [online] Available at: <https://www.engineeringtoolbox.com/kinematic-viscosity-d_397.html> [Accessed 20 July 2021].

SimScale. 2021. *What is Computational Fluid Dynamics (CFD)? | SimScale | SimScale*. [online] Available at: <<https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/>> [Accessed 28 July 2021].

Norscreenfilters.co.uk. 2021. *Pipe Sizes*. [online] Available at: <http://www.norscreenfilters.co.uk/pipe_sizes.html> [Accessed 29 July 2021].