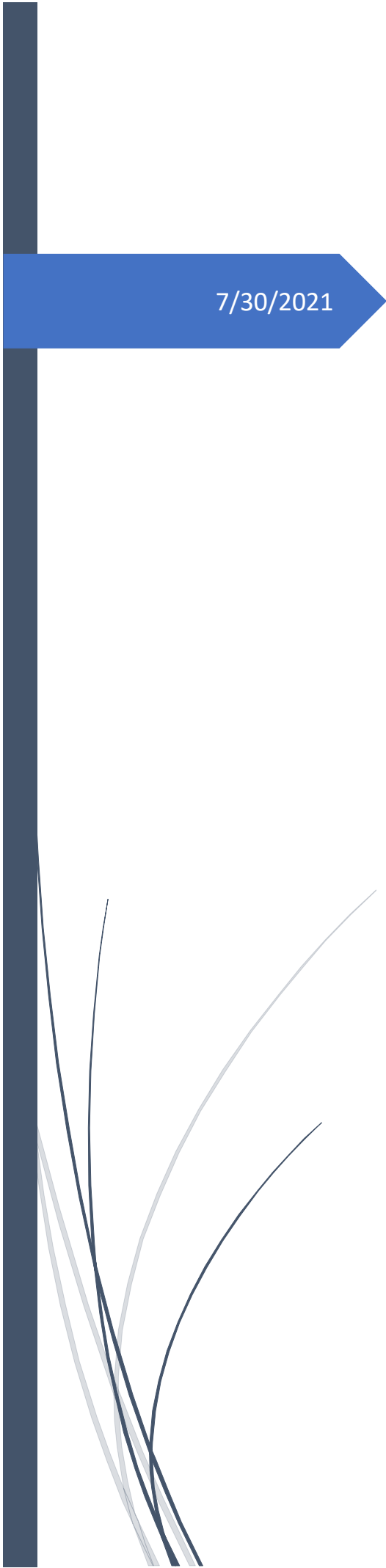




7/30/2021

Designing a Butterfly Valve Fluid Flow Assessment

Structural & Fluid analysis

Joseph Garland
1707750

Summary

The following report looks at a number of different fluid flow characteristics and how they affect engineering components such as Butterfly valves. Firstly, a study is conducted to find out the Reynolds number for the butterfly valve given in the assignment brief and the influence it has on the valve's performance.

A discussion into the effects of boundary flow through the butterfly valve and the impact it has on the valve's performance. An in-depth study into the design of the butterfly valve is also carried out in order to find out whether there are any possible implications caused due to the design.

The valve is intended to be made from glass fibre reinforced plastic so a study into the implications in terms of performance of the valve due to this was carried out. Performance factors such as strength, weight, durability, cost and corrosion resistance are all taken into consideration in order to understand the implications involved in using glass fibre reinforced plastic to produce the valve.

Modelling techniques are a crucial part of fluid flow analysis and help engineers better understand the behavioural characteristics of components before putting them into production. The study will look at both wind tunnel testing and computational fluid dynamic CFD modelling and discuss the pros and cons of both.

Finally, the report will look at what the best way of carrying out an experiment on a proposed prototype of the valve and how data should be gathered and compared to some of the modelling techniques used for fluid flow analysis.

Nomenclature

Re – Reynolds Number

CFD – Computational Fluid Dynamics

ρ – Density

l – Given Dimension (Diameter)

μ – Kinematic velocity

P – Pressure

M - volumetric mass

H – Height

G – Gravity

A - Area

P atm – Atmospheric pressure

PIV – Particle image velocimetry

m/s – Meters per second

°C – Celsius

GFRP – Glass fibre reinforced plastic

GRP – Glass reinforced plastic

FRP – Fibre reinforced plastic

UV – Ultra violet

Contents

Summary	2
Nomenclature	3
Introduction	5
Tasks.....	6
Reynolds Number.....	6
Reynolds number calculations for butterfly valve	7
Reynolds Number Calculations	8
Boundary Flow	10
Shearing forces in boundary flow	12
Manufacturing material	12
Fluid Flow Modelling Techniques	14
Prototype Testing.....	18
Bibliography	20

Introduction

The assignment will consist of a study into fluid flow through a butterfly valve. The study will aim to look at how the Reynolds number for flow of fluid through the valve and how the design of the valve may affect whether the flow is laminar or turbulent. The study will also look to determine whether the Reynolds number will be affected by boundary flow of fluid within the valve.

The material the valve is intended to be made of will be glass fibre reinforced plastic, research into whether there are any possible implications when using glass fibre reinforced plastic for the main components of the valve will also be carried out.

Further research carried out as part of the assignment tasks will also aim to prove the advantages and disadvantage of various different fluid flow modelling techniques and following this a discussion into the best suited form of fluid flow modelling technique for the valve.

Finally, a discussion into how experimentation of a prototype of the valve should be carried out and how the data gathered should be compared to data that would have been gathered in task 4.

Tasks.

Reynolds Number

The theory behind the now known as Reynolds number was first studied by Sir George Stokes in 1851, where when carrying out further research into a study carried out by Claude Louis Navier he was able to derive an equation for fluid in motion and called it the Navier-Stokes equation (What is Reynolds Number? | SimWiki | SimScale, 2021) . The Navier-Stokes equation was a dimensionless number used to determine viscous flow of fluid.

However, in 1883 Osbourne Reynolds was able to discover another dimensionless number whereby he was able to predict the flow of fluid using both its static and dynamic properties. Reynolds examined the relationship between velocity and fluid flow in liquids.

According to (Herrmann and Klaus, 2000) Reynolds number is made up of both static and dynamic fluid properties. Due to the Reynolds number being a study of dynamic conditions of fluid it can be specified as a flow property.

Reynolds number is used as a way of determining the behavioural characteristics of a given flow of fluid (Herrmann and Klaus, 2021). The Reynolds number commonly referred to as Re has no units of measurement and is therefore dimensionless. (Herrmann and Klaus, 2021) also state that, the two main characteristics of fluid flow recognised when looking at Reynolds number is laminar and turbulent flow, however fluid flow can also be recognised as in a transitional phase. Whether the fluid flow is laminar, turbulent or transitional is determined using the equation:

$$Re = \frac{\text{Inertia Force}}{\text{Viscous Force}}$$

Also seen as:

$$Re = \frac{\rho v l}{\mu}$$

Reynolds number is equal to the density multiplied by the velocity multiplied by the given size (diameter/length), all divided by the viscous force of the fluid. Using this equation, a Reynolds number of <2300 would classify the flow as laminar, if the Reynolds number ranges 2300<4000 then the fluid flow can be seen as transitional and if the Reynolds number is >4000 then the flow would be classed as turbulent.

The Reynold's number of a fluid is important as it can be used in a number of different industries to better understand the influence laminar and turbulent flow has on items.

Laminar or turbulent flow characteristics can have an impact of the performance factors of a butterfly valve. Factors such as Pressure loss, hydrodynamic torque, flow coefficient and torque coefficient are all effected by the type of flow the valve experiences (Herrmann and Klaus, 2000). For example, turbulent flow would mean the valve experiences high stress on its internal elements. Whereas laminar flow would have much less of an impact on the internal elements of the valve.

Reynolds number calculations for butterfly valve

The following assumptions were made prior to carrying out the workings out required to achieve a Reynolds Number for the butterfly valve:

- There are no losses due to friction experienced
- The viscosity of Petroleum is taken at 20°C
- Velocity is taken while valve is fully open
- Reynolds number is taken when valve is fully open

Analysis of the data presented in the assignment brief identified that the velocity of the flow of petroleum through the valve, must first be calculated in order find the Reynolds number.

Reynolds Number Calculations

$$P_1 = P_{atms}$$

$$P_2 = P_{atms}$$

$$\rho = 800 \text{ kg/m}^3$$

$$V_1 = 0 \text{ m/s}$$

$$V_2 = ?$$

$$\mu = 0.006 \text{ (Toolbox, 2021)}$$

$$H_1 = 5 \text{ m}$$

$$H_2 = 0 \text{ m}$$

$$l = 0.194 \text{ mm}$$

Using Bournoulli's equation for v:

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho v_2^2 + \rho g h_2$$

$$0 + 0 + g h_1 = 0 + \frac{1}{2}v_2^2 + 0$$

$$\frac{1}{2}v_2^2 = g h_1$$

$$v_2 = \sqrt{2 g h_1}$$

$$v_2 = \sqrt{2 \times 9.81 \times 5}$$

$$v_2 = 9.9 \text{ m/s}$$

In order to prove the figure obtained for v the potential energy and kinetic energy equations were also used:

Assumptions:

- Tank dimensions are 1m in length x 1m in width x 5m in height

Volumetric mass off a full tank of petroleum is:

Density x Area

$$800 \times 5 = 4000 \text{ kg/m}^3$$

$$PE = mgh$$

$$KE = \frac{1}{2}mv^2$$

$$PE = KE$$

Therefore:

$$mgh = \frac{1}{2}mv^2$$

$$4000 \times 9.81 \times 5 = \frac{1}{2} \times 4000 \times v^2$$

$$\frac{196200}{2000} = v^2$$

$$98.1 = v^2$$

$$\sqrt{98.1} = v$$

$$v = 9.9 \text{ m/s}$$

Now the value for v was substituted into Reynold Number Equation.

$$Re = \frac{\rho v l}{\mu}$$

$$Re = \frac{800 \times 9.9 \times 0.194}{0.006}$$

$$Re = 256080 \text{ or } 256.08 \times 10^3$$

From the calculations for the Reynolds number, it is clear that the butterfly valve will experience turbulent flow when fully open as the Reynolds number is greater than 4000.

Boundary Flow

When studying the flow of fluid through an object such as a butterfly valve, the effects of boundary flow should be taken into consideration in order to understand how the fluid will behave and valve will perform.

As stated by (Kudela, 2021) Boundary flow in fluid mechanics is the layer of fluid that is in direct contact with an object, for example the disc inside a butterfly valve. The fluid that comes into direct contact with an object first, will have the thinnest boundary layer and the boundary layer will thicken as the fluid travels over the object (Kudela, 2021). Flow in this manner is generally classed as behaving in a laminar fashion. Boundary flow will either be turbulent or laminar according to the Reynolds number (Boundary Layer, 2021). Fluids at the boundary layer are exposed to shearing forces as the velocity of the fluid that passes over it will vary the further away from the contact area the fluid is. The fluid closest to an object as it flows over it will have a lower velocity than fluid that

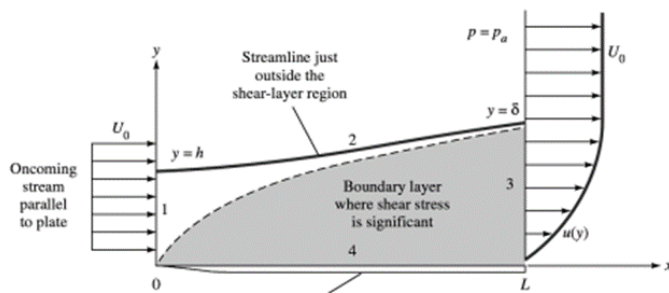
is further away (Boundary Layer, 2021). This velocity will increase the further away from an object the fluid gets as it experiences less surface drag.

A butterfly valve is typically made up of a disc of some material attached to a central shaft which acts as an axis at which the disc is able to revolve. A butterfly valve is stereotypically used in between sections of pipework as a way of controlling fluid flow. By rotation the disc within the valve through 0° (closed) to 90° (open) the resistance to flow will gradually decrease, with the valve experiencing the most resistance to flow when closed and the least when the valve is open. Butterfly valves are typically exposed to stresses and strains caused by high Reynolds numbers (Del Toro, 2021).

A butterfly valve would usually experience laminar fluid flow when entering to the valve and turbulent upon exiting the valve. The flow within the valve itself however may consist of both laminar and turbulent flow depending of the position of the disc i.e. fully open or fully closed.

The disc within a butterfly valve is typically cast symmetrically with the centre shaft connecting to the centre of the cast so that it does not interfere with flow either side of the disc. The design of the butterfly valve however sees the disc attached to the centre shaft using two rivets. Turbulent flow encountered through the valve results in its component experiencing high torque forces resulting in significant stress and strain being exerted on the valve. Due to boundary flow the valve is also likely to experience shearing forces (Choi, Seo and Kim, 2021). The rivets within the design of the butterfly valve create a weak point that over time could potentially fail as a result of the forces they are subjected to whilst in use. Should the rivets fail and shear the performance of the valve would be greatly affected rendering it incapable of being operated effectively and would require replacing.

Shearing forces in boundary flow



The valve may have been designed in this way as it was not intended to be used for flow with a high Reynolds number. For example, in between two sections of pipework where the valve is rarely operated and is either fully open or fully closed the majority of the time.

The value of butterfly valves varies widely from relatively low cost to extremely expensive; this could be due having a long-life expectancy. Should the rivet within the design fail it could prove to be costly as not only would it require replacing with a new one, would also incur labour cost for fitting and in engineering factories cost of unplanned downtime.

Manufacturing material

Glass fibre reinforced plastic more commonly known as GRP or FRP is used widely among the engineering industry for the production of valves. GFRP is used as a way of increasing the strength of standard plastics by mean of bonding glass fibres and setting them in resin (Technical, 2021). GFRP generally consists of two components, one being the glass fibres themselves and the other being a matrix to provide support and protection to fibres. It is the fibres that actually bear any load the material in subjected to with the strength of the bond being reliant of the length of the fibres used in the mixture (Technical, 2021).

The performance characteristics of GFRP make it extremely suitable for use in production of valves.

Some of the characteristics (GRP Advantages and Disadvantages | Pros & Cons | GripClad, 2021)

Characteristics such as:

Low maintenance

GFRP require very little to no maintenance. The low maintenance involved in GFRP makes it an ideal material for the butterfly valve to be made with as once it is fitted it will not need regular maintenance checks.

High Strength to Weight Ratio

The high strength to weight ratio GFRP possesses would mean the valve is capable of withstanding heavy loads caused by turbulent flow such as shearing forces, high stress and strain. The valve would therefore be able to perform repeat operation multiple times over a long period without being affected. The weight of GFRP would also make for better manoeuvrability in comparison to that of a steel/iron butterfly valve, aiding installation should the valve need to be installed overhead.

Affordability

Often materials traditionally used in the production of butterfly valve such as steel can be expensive due to the amount/weight of material required to make it. GFRP can also be expensive when used in the production of valves however, it's extremely low up keep and long life expectancy means that GFRP butterfly valves provide very good value for money in comparison to valves made of more traditional materials.

Resistance to Corrosion

GFRP Butterfly valves provide great resistance to corrosion from fluids with a wide PH level, unlike more traditional valve i.e steel valves. One of the benefits of being so corrosion resistant would be that unlike traditional steel valves the GFRP valve will not experience rust and seize rendering it unable to perform as intended.

UV Stability

The UV provided from the GFRP mean the valve would be capable of performing perfectly in environment where it is exposed to both high and low levels UV light.

Durability

The high durability of GFRP would make the valve very long lasting as it would be capable of withstanding the wear and tear, pressure and damage that incurs from regular use.

Industrial butterfly valves are required to meet a large numbers of different industry standards.

However, perhaps some of the more relevant industry standards to a GFRP butterfly valve would be:

- ISO 14992:2017
- ISO 12092:2000
- EN 1267:1999
- EN 12266-1:2003
- EN 12570:2000

(ISO 14692-1:2017, 2021) (ISO.org, 2021)

Fluid Flow Modelling Techniques

During the design phase and prior to installation of an item subjected to forces of fluid flow, it can be useful for manufacturers to carry out modelling of the item to get a better understanding of the type of flow the item will experience.

Types of modelling include precision laboratory wind tunnels; these wind tunnels allow manufactures to simulate real world conditions and gather a better understanding of how the fluid

flow will behave when passing over or through the item being tested. In order to carry out the test the model/item must be placed inside the wind tunnel where air is then blown using motorized fans to create air flow over the model.

By carrying out fluid flow modelling in a wind tunnel, manufacturing teams are able to gain an understanding of whether the flow around the object will be laminar or turbulent (Handojo, 2021). The engineering team will also be able to note the effects of the stress and strain the object will be subjected to as a result of the fluid flow. Wind tunnel modelling however is not completely accurate as it is not always capable of accurately recreating real-world conditions. For example, when using a wind tunnel to test a cars aerodynamics, if the model is stationary, it means the manufacturing team will be unable to visibly consider the effect of its wheels in motion. Further real-world testing of the car would then be necessary in order to gauge the effects the wheels in motion would have on fluid flow around the car.

According to (Handojo, 2021) Some of the advantages of wind tunnel testing include the following:

- Much more cost effective way of testing fluid flow for engineering components such as aeronautical equipment i.e aeroplane propellers.
- Widely adjustable factors of parameter such as wind speed, temperature, pressure and oxygen saturation.
- Capable of achieving extreme levels of laminar flow due to being carried out in a controlled environment.
- Provides engineers/members of the design team with a visual representation of the fluid flow around the model. The visualisation of the fluid flow can be achieved via the use of different testing methods such as mist in the fluid flow, PIV and schlieren.
- Allows more intensive testing to be carried out over a shorter period of time in comparison to testing the model in actual operation.

- Has ability to identify any faults in the design of the model in relation to its desired performance.
- Helps identify design faults that could potentially prove to be hazardous.

The use of wind tunnels to simulate the fluid flow behavioural patterns also has its disadvantages, including the following:

- There is a limitation to the size of the model intended to be test i.e aeroplane wings.
- Scaling down the size of a model can result in a change in the aerodynamic characteristics of fluid flow over the model, such as Reynold number.
- Inability to accurately recreate real world factors that may affect the fluid flow, e.g fluid flow testing of a car when stationary does not take into consideration the effect its wheels in motion. This can result in the gathering of somewhat misleading data if not taken into consideration.
- Depending on the size of the wind tunnel and the model size, the walls of the tunnel can sometimes interfere with the boundary flow around the object.

Due to wind tunnel modelling being best suited to aerodynamic studies, it would not be best suited to the fluid modelling needed for a butterfly valve.

CFD modelling otherwise known as computational fluid dynamics is the use of mathematical processes to physically model an object subjected to fluid flow.

(Computational Fluid Dynamics (CFD), 2021) states that, Computational fluid dynamics was first brought into use in 1910 when basic mathematic models and numerical methods were used to better understand fluid flow around or through an object. This allowed scientists to better study the effect the fluid flow had on any given object. Since its introduction CFD modelling has been developed continuously and between 1950 - 1960 was used by Los Alamos Nation Laboratory in an initial study using computers as a way of modelling fluid flow based of Navier – Stokes equations.

CFD is commonly used within engineering design teams when designing items such as sports cars, aeroplane parts or valves etc. CFD modelling has become broadly used throughout all industries, ranging from medical, aeronautical and heavy engineering. Within the engineering industry CFD modelling is applied when studying the performance of different engineering components, when exposed to different types of fluid flow. Whether it be fluid flowing through the components or the flow of a solid body passing through a fluid.

(Patel, 2021) states, Some of the advantages of using CFD modelling include the following:

- Capable of taking specific boundary conditions into consideration
- Ability to isolate certain parts of a model to gain area specific fluid flow data.
- Allows data to be measured through simulation at any point of the fluid flow pattern
- Capable of gathering data from many different flow parameters which may otherwise be inaccessible in experiments.
- Capable of gathering information from a wide variety of design prototypes while the model/product is in its infancy. This helps minimise the likelihood of experiencing unexpected fluid flow characteristics when testing.
- Allows for a better understanding of what problems may be encountered during the models operation over time.
- The costs associated with carrying out CFD modelling are comparatively lower than the costs of having to carry out a test/experiment on a physical model.

CFD modelling also comes with some disadvantages, these include:

- Room for user error such as wrong input of information.
- Error can become likely due to simple flow models and boundary flow conditions.
- Interpolation errors due to lack of information/values to compute.
- Time taken to input and compute data can sometime take a while when dealing with models bigger in size.

- Consulting errors may prove to be much more costly in comparison than when they are experienced in experiments.

CFD modelling would be best suited for use of gathering data prior to carrying out prototype testing of the butterfly valve. Due to the design of a butterfly valve (internal flow), it is relatively difficult to gather accurate data regarding performance without carrying out physical testing. CFD modelling of the butterfly valve would help gain a better understanding of the fluid flow, as it is capable of giving a visible representation of the flow that will occur virtually, as stated by (Team, 2021).

Prototype Testing

It is proposed a prototype of the butterfly valve is tested in a laboratory prior to being put into actual service in order to gather all the relevant performance data of the valve. Prior to testing the Butterfly valve, it would best to first carry out some computational fluid dynamic modelling (as discussed in task 4) of the valve as a means of gathering comparative data. Doing this will give the testing team a better understanding of how they should expect the valve to perform during testing throughout all operating motions i.e fully open, fully closed and when in transition (Team, 2021). Performance characteristics such as Reynolds number, velocity, flow rate and forces experienced by the butterfly valve should all be of close interest during its testing.

In an effort to gain as accurate test results/data as possible the prototype should be produced true to its design specification. As a result, conditions during the test will directly imitates conditions in its intended environment. By doing this the testing team should expect there be no anomalies in data caused by scaling down the valve.

If possible, the prototype of the butterfly valve should be fitted to pipework at the base of a 5-meter-high tank where it should be operated as intended numerous times. The fluid used in the

experiment should also be petroleum, as the fluid should contain the exact same properties to emulate real world results.

The valve should be fitted with flowmeters, pressure sensors and strain gauges as a means of gathering all relevant data needed for comparison with the expected data obtain from CFD modelling of the valve.

Analysis of the two sets of data will allow to test team to gauge whether the valve is performing as it should be and whether there are any faults or inaccuracies in the CFD modelling simulation.

Bibliography

- Grc.nasa.gov. 2021. Boundary Layer. [online] Available at: <https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html> [Accessed 28 July 2021].
- Encyclopedia Britannica. 2021. boundary layer | Definition, Characteristics, & Facts. [online] Available at: <https://www.britannica.com/science/boundary-layer> [Accessed 30 July 2021].
- Choi, S., Seo, H. and Kim, H., 2021. Analysis of Flow Characteristics and Effects of Turbulence Models for the Butterfly Valve. 1st ed. [ebook] Applied Sciences. Available at: <http://file:///C:/Users/Joseph%20Garland/Downloads/applsci-11-06319-v2.pdf> [Accessed 28 July 2021].
- Ifes-koeln.de. 2021. Computational Fluid Dynamics (CFD). [online] Available at: <https://www.ifes-koeln.de/en/services-20-years-project-experience/simulation-analysis/computational-fluid-dynamics-cfd.html> [Accessed 30 July 2021].
- Del Toro, A., 2021. Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors. [ebook] Utah State University, pp.1-12. Available at: <https://digitalcommons.usu.edu/cgi/viewcontent.cgi?referer=&httpsredir=1&article=2454&context=etd#:~:text=As%20a%20butterfly%20valve%20is,readily%20flow%20past%20the%20valve.&text=Another%20method%20wherein%20but%2D%20terfly,system%20around%20a%20butterfly%20valve.> [Accessed 29 July 2021].
- Gripclad. 2021. GRP Advantages and Disadvantages | Pros & Cons | GripClad. [online] Available at: <https://gripclad.co.uk/useful-information/grp-advantages-and-disadvantages/> [Accessed 30 July 2021].
- Handojo, V., 2021. What are advantages and disadvantages of wind tunnels? - Quora. [online] Quora.com. Available at: <https://www.quora.com/What-are-advantages-and-disadvantages-of-wind-tunnels> [Accessed 24 July 2021].
- Herrmann, S. and Klaus, G., 2021. Reynolds Number | Definition, Calculation & Examples | nuclear-power.com. [online] Nuclear Power. Available at: <https://www.nuclear-power.com/nuclear-engineering/fluid-dynamics/reynolds-number/> [Accessed 27 July 2021].
- ISO. 2021. ISO 14692-1:2017. [online] Available at: <https://www.iso.org/standard/62256.html> [Accessed 30 July 2021].
- Kudela, H., 2021. External Flows. Boundary Layer Concept. [ebook] Available at: http://fluid.itcmp.pwr.wroc.pl/~znmp/dydaktyka/fundam_FM/Lecture14.pdf [Accessed 29 July 2021].

- Patel, M., 2021. 4 Advantages and Disadvantages of CFD. [online] CFD Modeling Services. Available at: <<https://cfmodelingservices.wordpress.com/2013/06/12/4-advantages-and-disadvantages-of-cfd/>> [Accessed 27 July 2021].
- Team, S., 2021. 13 Key Applications of CFD Simulation and Modeling. [online] Blog.spatial.com. Available at: <<https://blog.spatial.com/cfd-modeling-applications>> [Accessed 30 July 2021].
- Technical, C., 2021. Fiber Reinforced Plastic | Types, Advantages, Disadvantages, Applications. [online] ClubTechnical. Available at: <<https://clubtechnical.com/fiber-reinforced-plastic>> [Accessed 30 July 2021].
- Toolbox, E., 2021. Liquids - Kinematic Viscosities. [online] Engineeringtoolbox.com. Available at: <https://www.engineeringtoolbox.com/kinematic-viscosity-d_397.html> [Accessed 30 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 30 July 2021].
- SimScale. 2021. What is Reynolds Number? | SimWiki | SimScale. [online] Available at: <<https://www.simscale.com/docs/simwiki/numerics-background/what-is-the-reynolds-number/#:~:text=In%201883%2C%20Irish%20scientist%20Osborne,characteristics%20of%20the%20fluid%5E2>> [Accessed 27 July 2021].

