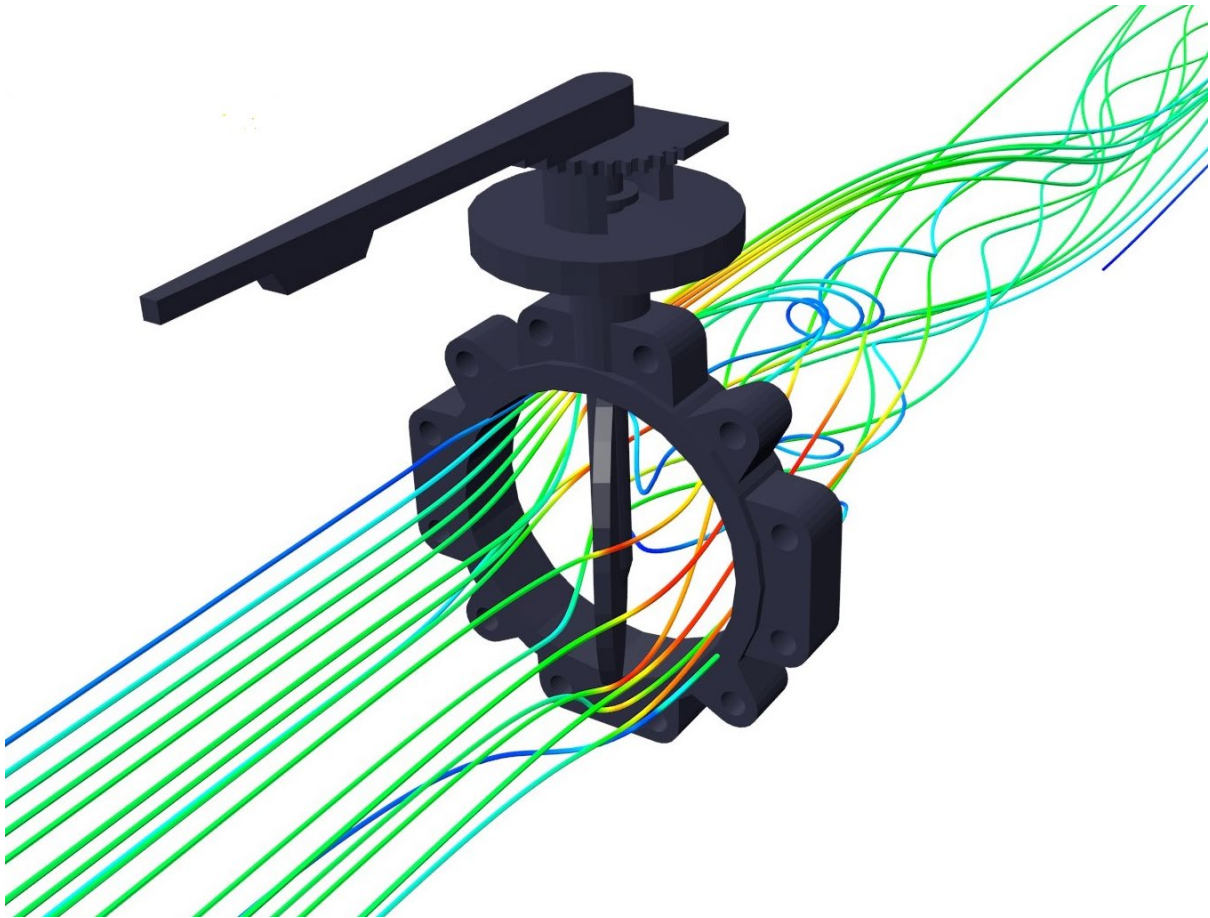


Matthew Ford

Structural and Fluid Dynamics Assignment



Structural and Fluid Analysis

Module code: ACMA6010

Student Number: 1903400

Contents	Pages
Task1	3-5
Task 2	6-8
Task 3	9-9
Task 4	10-11
Task 5	12-13
Bibliography	14

Task 1.

Discuss Reynolds number. Why is it important, how does it influence the flow around the butterfly valve and estimate what the Reynolds number is of the fluid flowing around the butterfly valve?

Is a number used to denote whether a fluid is flowing in a laminar state, turbulent state or combination of the two. It was created by British physicist Osbourne Reynolds in 1883 (Rogers, n.d.). It is a number that has no dimensions so is considered to be a pure number. A flow with a Reynolds number less than 2000 is generally considered to be laminar, and a flow with a Reynolds number greater than 3500, is considered to be turbulent, flow between these figures is referred to as the range of transition. These figures can vary considerably depending on situation, but those quoted are generally considered to be reasonable.

The equation used to describe the Reynolds number can be written as below:-

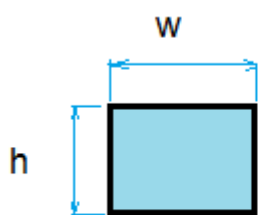
$$Re = \frac{\text{Initial Forces}}{\text{Viscous Forces}}$$

Or:-

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

Where v = velocity
 μ = viscosity of fluid
 ρ = density of fluid
 d = diameter of relative pipe of ducting

Note, that when dealing with square/rectangle cross section ducting the formula below:-



A diagram of a rectangle with a light blue fill and a black border. The width is labeled 'w' with a double-headed arrow above the rectangle. The height is labeled 'h' with a double-headed arrow to the left of the rectangle.

$$A = w \cdot h$$
$$P = w + w + h + h$$
$$Deq = \frac{4 \cdot A}{P}$$

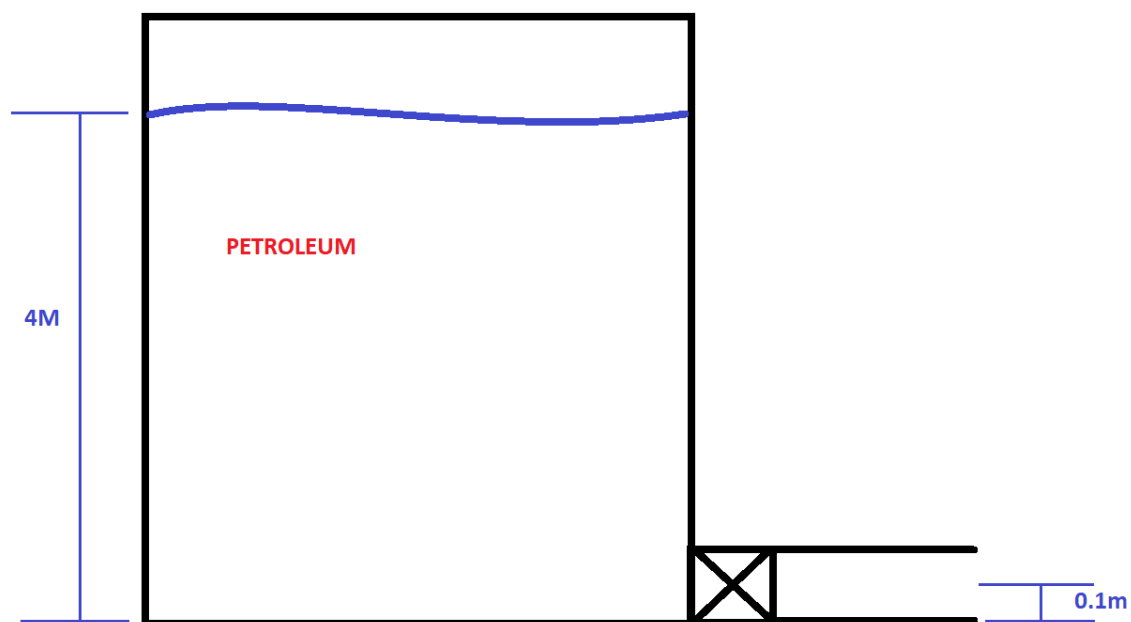
is used to determine an equivalent diametric value (hydraulic diameter) (Valvias, n.d.).

The Reynolds number is particularly important, in scale modelling of fluid flows such as those done in wind tunnels. It can be used to ensure that a scale model truly reflects the behaviour of a full sized item. The flow does not behave in the same way around scale models, this is because the laminar flow over a small component will transition to turbulent flow at the same distance along the component no matter what scale. The Reynolds number can be used to counteract this by altering the velocity or pressure to counteract the change in L value, thus you can maintain the same Reynolds Number value for the scale component, this concept is known as the Reynolds Principle of similarity (Stokes, n.d.)

The Reynolds number for the flow of fluid around the butterfly valve in question will be estimated when the valve is fully open, and in order to estimate this, several assumptions must be made. These assumptions and explanations are listed below:-

- Petroleum in the tank has a density value of 750 kg/m^3 , petroleum density varies from product to product, but 750 kg/m^3 is a reasonable assumptions in relation to an ambient air temperature of 20°C (toolbox, n.d.).
- That the petroleum has a viscosity value of 0.000006 Ns/m^2 (Unknown).
- The level of liquid in the tank will be the same for both estimates, and that height will be 4m.
- There are no frictional losses due to pipe wall imperfections
- The pressure downstream of the valve is at atmospheric, therefore there is no back pressure in the system.
- The valve dimension is 194mm, therefore the assumption is that it is designed to be installed into a 4" diameter pipe ie 200mm nominal bore
- When fully open, the valve internals (disc and stem) will cause no restriction

A schematic of the system can be seen below:-



In order to estimate the Reynolds number, the velocity of the fluid at the butterfly valve must first be calculated:-

$$\begin{aligned}V_{\text{out}} &= \sqrt{2g\Delta h} \\&= \sqrt{2 \times 9.81 \times (4 - 0.1)} \\&= 8.747\text{m/s}\end{aligned}$$

Using the above calculations, the Reynolds number when the valve is 100% open can be estimated as:-

$$100\% \text{ opening, } Re = \frac{8.747 \times 0.194}{0.000006} = 282819$$

As this number is significantly greater than 3500, it is therefore reasonable to assume that the flow is turbulent in nature.

Task 2. Discuss boundary flow around the butterfly valve and relate this to the separation and the turbulence around the valve, particularly the rivets that hold the valve onto the central shaft. Discuss why you think that valve was designed like this and the cost implications of such a design.

Boundary flow (also known as boundary layer) is formed when molecules flowing in a fluid, stick to the surface of an object. The molecules further away from the surface are slowed by the stationary molecules by means of them colliding with each other.

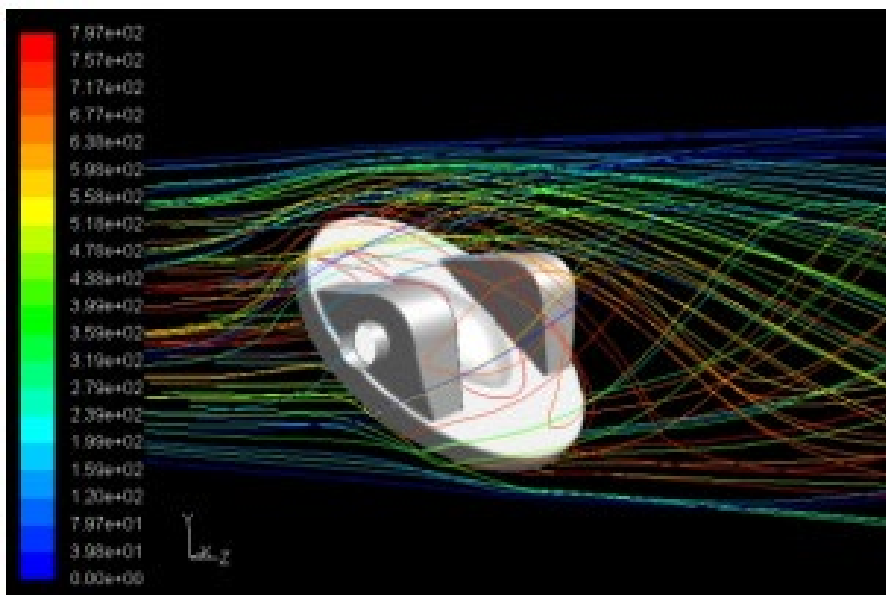
This process continues over and over as you move further and further from the surface, the further away from the surface, the less collisions occur. As a result of this, there is a region that begins at the object surface and extends away, where the fluid velocity changes from zero at the surface, to full stream velocity. This region is known as the Boundary Layer (Hall, 2021).

Boundary flow can be affected by obstacles before the fluid flow even enters the valve. Things such as upstream non return valves, thermowells, pipe stabblings, poor pipe wall finishes will all effect boundary flow. This aside, once the fluid reaches the valve, the boundary flow will be affected by any/all components of the valve sitting within the fluid flow. This would include the valve disc, disc rivets, and to a miniscule amount, the valve stem depending on clearances. The boundary flow will also be very different depending on the positioning of the disc.

For the purpose of this report, the boundary flow will be described when the valve is fully closed, 45 degrees open, and 90 degrees open (100% open).

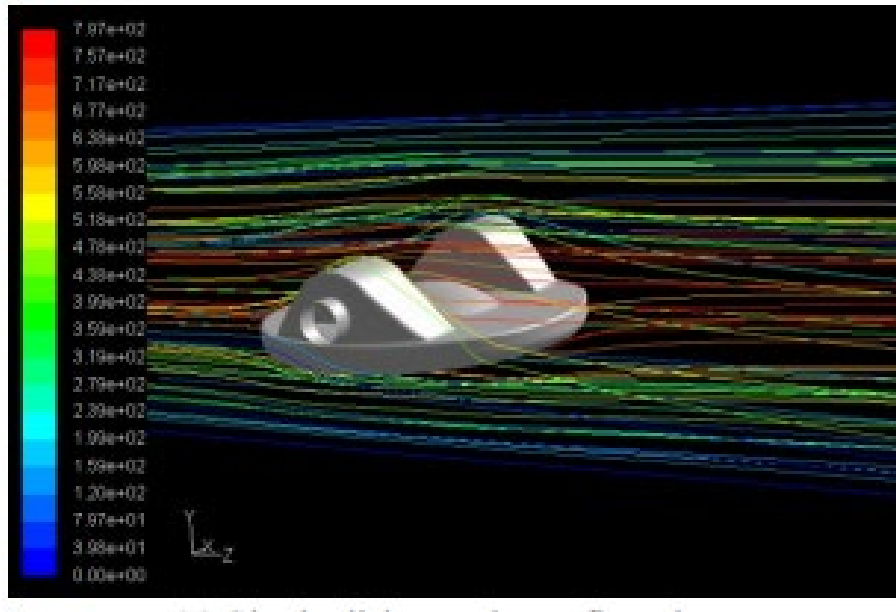
When the valve is fully closed, it is considered to achieve a 100% seal, therefore there is no flow to observe.

As the valve opens to a position of 45 degrees,



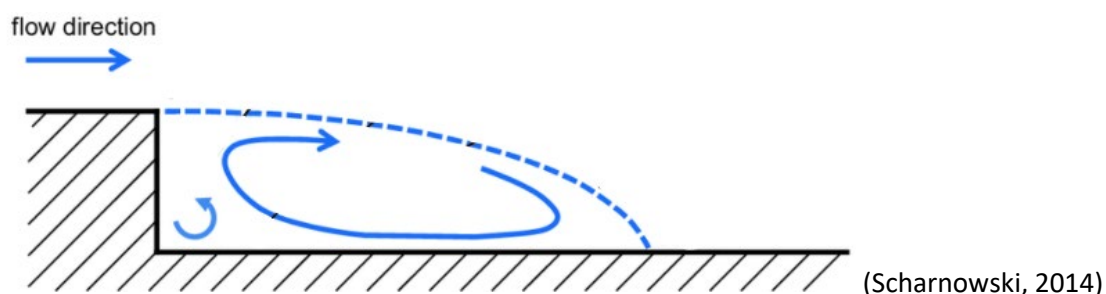
The boundary flow along the pipe walls will narrow, and when this happens the fluid velocity increases. Boundary flow around the disc is almost non-existent, a massive turbulence can be seen in the area directly downstream of the disc.

As the valve opens fully, to 90 degrees,



The boundary flow will revert back to that of the boundary flow of the pipe wall upstream of the valve. Boundary flow is also evident over the valve disc, with separation occurring from the disc before the flow has reached the mid point of the disc and turbulence occurs. This flow soon returns to laminar flow downstream of the disc, but if laminar flow were critical in the system, the disc could be profiled in the same way as an aeroplane wing is shaped, except in a neutral plane, providing neither lift or drag.

Boundary flow will also be impacted by the positioning of the valve within the pipeline. Due to necessary assembly clearances in the valves boltholes, the valve is highly unlikely to be sat perfectly in the pipeline bore. These misalignments will cause both forward and back facing steps in the pipe wall, causing boundary flow separation shown below.



The valve has been designed making use of rivots as it is the most basic way to achieve the desired goal. The goal being to secure the disc to the valve stem in such a way that ensures that the torque applied to the valve stem, is transferred to the valve disc. As well as being the most basic way to secure the disc to the stem, it is also the cheapest from a manufacturing point of view, requiring minimal machining. There are other ways that this can be achieved, but the key observation that must always be taken into account is that the stem and disc must be designed so that they are two separate items to ensure that the unit can be constructed and if required, maintained.

Other methods of securing the disc to the stem might be as follows :-

1. Countersunk rivets - *so that the peened head fills the countersunk whole so there is no protrusion*
2. Interference fit - *expensive high tolerance machining, difficult to assemble and maintain*
3. Roll Pin – *these are very cheap and easy to assemble/maintain, but unless a blind hole is used, the pin centre will allow fluid flow, thus full shut off could never be achieved*
4. Key and receiver design - *the valve stem is designed in a way that a male component engages with a female component on the valve disc, an example might be splines, or a simple roll pin engaging into a cut out. It is worth noting that this type of design is more expensive than others discussed, but is often used on valves intended for duty in corrosive environments, where the components exposed to the fluid are coated in a resistant material such as PTFE (polytetrafluoroethylene) or PVDF (polyvinylidene difluoride).*

Task 3. This butterfly valve is to be manufactured from glass fibre reinforced plastic. Discuss the implications of this in terms of the performance of the valve. Relate this to the historical, current, and future developments of such components. You should include reference to technical literature, industry standards and codes of practice.

The valve is to be manufactured from glass fibre reinforced plastic (gfrp). From a performance point of view, this will give the valve a strength (although other materials are stronger such as carbon FRP), corrosion resistance as well as being less brittle than some alternatives. One major advantage of using GFRP is that it is cheap, but only if costs are considered over a period of time. Initial costs of the component are slightly higher than equivalent items, but the trade off is that the GFRP will have a far greater longevity.

However an added advantage of using GFRP is from an environmental view point. As mentioned above the material is lightweight therefore more friendly to transport in bulk, and has a long life cycle, meaning (as mentioned above) that it will need to be replaced less often. There are also valid studies currently being undertaken that could mean that GFRP can be widely recycled, although it should be said that currently this is not economically viable (Wait, 2010) but is worth noting.

From a physical performance point of view, GFRP is an excellent choice of material, offering acceptable levels of surface finish thus not effecting fluid flow, and as the material composition itself can be manipulated, its characteristics can be adjusted to suit the application of the valve.

Historically, the butterfly valve has been in use since the beginning of the late 19th early 20th century, where they were used in steam engines and automobile throttle valves, it is only over the last twenty years that material and construction advancements have allowed butterfly valves to be used in a far greater spectrum, i.e. higher temperatures and pressures.

There are numerous British Industry standards, that are relevant to valving and that relate to handling petroleum. These are listed below:-

- **BS EN ISO 16136:2006+A1:2019** - Butterfly valves of thermoplastics materials
- **BS 4504** - Flanges and bolting for pipes, valves, and fittings, metric series. Part 1 (1969), ferrous. Part 2 (1974), copper alloy and composite flanges.
- **BS 4371**: -Fibrous gland packings.
- **BS EN 736-1** : 1995 covers valve terminology and definitions
- **BS EN 736-2** : 1997 as above
- **BS EN 736-3** : 2008 revision of above
- **ISO 29001** an industrial standard that may be relevant depending on ultimate usage.

Task 4 Discuss what modelling techniques could be used to simulate the fluid flow through this valve. Discuss the advantages and disadvantages of the tools highlighted.

The idea of modelling fluid flow can be dated back as far as the 1500's when Leonardo da Vinci wrote notebooks on turbulent flow in nature (See drawing below).



(Kemp, 2019)

There are numerous modern modelling techniques currently available to simulate fluid flow in general. These can be divided into two categories, practical experiments, similar to that described below in Task 5, or by using computers with substantial processing power, that perform multiple calculations and analysis based around known equations and logic. The more processing power a computer system has, the more calculations it can make, which ultimately leads to more accurate results being given. This technique is known collectively as Computer Fluid Dynamic (CFD). CFD is not limited to just fluid flow through a valve, it can be used in a wide range of fields, such as aerodynamics, heat transfer, combustion, weather simulation.

CFD can be broken down further into sub groups, that use slightly different equations and analysis methods, but they are all based around Navier-Stokes equations. These are differential equations that describe the relationship between temperature, pressure and density in a dynamic fluid (Hall, 2021).

The further CFD subgroups are listed below:-

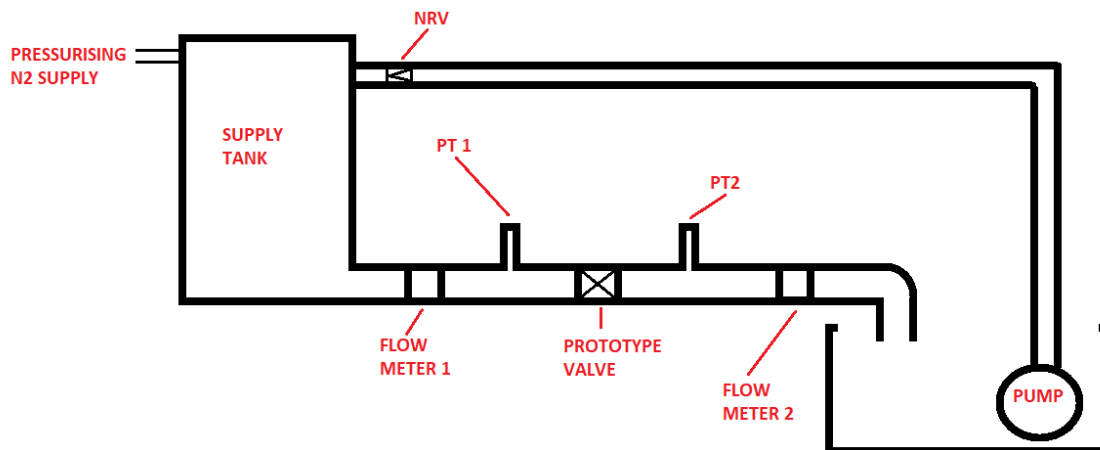
Reynolds-averaged Navier-Stokes (RANS) this method takes Navier-Stokes equations and averages them out to obtain mean equations of fluid flows. Reynolds stress terms are added to these equations to give results. RANS modelling is currently the most used technique in industry (Unknown, Turbulence Models in CFD, 2020).

Large eddy simulation (LES) this method separates out small scales of turbulence, and concentrates on the larger ones, resolving them principally. LES modelling can be applied to a large number of scenarios and is highly accurate, but as with most things, this comes with an increased price tag. There are also some modelling techniques that combine both RANS and LES, these are known as RANS-LES Hybrid models (Unknown, Turbulence Models in CFD, 2020). An example of a RANS-LES Hybrid is Detached Eddy Simulation

Direct Numerical Simulation DNS is a method that solves Navier-Stokes equations without modelling turbulence. Turbulence is resolved within the computational mesh. DNS requires huge amounts of computational resource and therefore is very expensive.

Task 5. During the development of this component, it is proposed that a prototype is produced and tested in a laboratory to understand the fluid flow through this valve and also to correlate it to the simulation carried out in task 4. Discuss how you think this experiment should be carried out and how the acquisition of data should be obtained and compared to the simulation.

Below is a schematic of a proposed test rig that would be used to undertake the experiment: -



The experiment should be set up with flow meters upstream and downstream, plus either independent pressure transmitters (or basic manometers) up and downstream or one meter measuring differential pressure across the valve, ideally these should be outside of the fluid flow, but if this is not possible, their presence must be accounted for in any results.

Rig needs to be set up so as none of the above effect the flow or pressure (i.e. they must be a suitable distance away from the valve) also the rig must be set up so that there is a straight pipe run into the valve and also exiting the valve.

You must ensure that the correct fluid medium is used i.e. petroleum, but if this is deemed too dangerous, a more inert fluid should be sourced with the exact same properties as petroleum nb if petroleum is to be used, the system must be sealed in its entirety, and earthed as and when required.

Also, it will be impractical to set up a 5m high storage tank, therefore the test rig should be set up so that a pressure can be applied to a more suitably sized reservoir on the inlet side of the butterfly valve, equivalent to a 5m head. In order to gain a range of data, this pressure should be easily adjusted to simulate different tank levels (nb if petroleum is to be used, an inert gas such as nitrogen must be used to pressurise the inlet reservoir).

There should also be a pump set up to return the “spent” fluid back to the reservoir once it has exited the butterfly valve. It should be noted that there is no need to use any Reynolds number calculations as the experiment will be conducted using 1:1 scale prototypes.

All pipework should be made in Perspex or similar high grade transparent material, ensuring that it is industry standard for the transportation of petroleum. In addition, this non-conductive pipework

must be bonded and earthed accordingly to ensure there is no static electricity build up. The ability to see the flow of material through the pipe walls will help analyse the flow through the valve, however, this will be limited by the colour of the petroleum. Ideally as mentioned above, a similar fluid should be used that is clear, allowing the addition of a stream of dyes that would enhance the ability to see fluid flow and turbulence.

The experiment as described above, would provide three main streams of data noted below:-

1. Pressure differential across the valve via the manometers
2. Flow differential across the valve via the flow meters
3. Indication of flow characteristics via the clear pipe and introduction of dye.

These three real world measurables could then be directly compared to the results given by the simulation. These results are expected to be similar in most ways, small variations are unavoidable, but larger variations should be investigated. These could be caused by input errors in the simulation, or read errors from the operator, or they could be caused by a design flaw in the test rig set up.

Bibliography

- Hall, N. (2021, May 13). *National Aeronautics and Space Administration*. Retrieved from Navier-Stokes Equations: <https://www.grc.nasa.gov/www/k-12/airplane/nseqs.html>
- Kemp, M. (2019, July 16). *Leonardo da Vinci's laboratory: studies in flow*. Retrieved from nature: <https://www.nature.com/articles/d41586-019-02144-z>
- Rogers, K. (n.d.). *Britannica Reynolds Number*. Retrieved from Britannica: <https://www.britannica.com/science/Reynolds-number>
- Scharnowski, S. (2014, March 26). *Investigation into turbulent shear flows with resolution PIV*. Retrieved from Researchgate: https://www.researchgate.net/figure/Backward-facing-step-flow-field_fig3_261090536
- Stokes, N. (n.d.). *How does the Reynolds number affect my CFD model*. Retrieved from leapaustralia: <https://www.computationalfluidynamics.com.au/reynolds-number-cfd/>
- toolbox, E. (n.d.). *Fuels - Densities and Specific Volume*. Retrieved from Engineering ToolBox, (2003): https://www.engineeringtoolbox.com/fuels-densities-specific-volumes-d_166.html
- Unknown. (2020). *Turbulence Models in CFD*. Retrieved from udeal simulations: <https://www.idealsimulations.com/resources/turbulence-models-in-cfd/>
- Unknown. (n.d.). *Viscosity of liquids and gases*. Retrieved from Hyperphysics: <http://hyperphysics.phy-astr.gsu.edu/hbase/Tables/viscosity.html>
- Valvias. (n.d.). *Valvias Reynolds Number*. Retrieved from Valvias.com: <http://www.valvias.com/reynolds-number.php>
- Wait, C. F. (2010). *The Reuse and Recycling of Glass Fibre Waste*. Birmingham: University of Birmingham. Retrieved from https://etheses.bham.ac.uk/id/eprint/1342/1/Wait11MRes_A1b.pdf