

Designing a Butterfly Valve Fluid Flow Assessment

Adam Sheppard / 1909576

Structural & Fluid Analysis

30/07/2021

Contents

Introduction.....	2
Task 1: Reynolds Number	3
What is Reynolds Number?	3
Flow Types	3
Reynolds number applications	4
Calculating the Reynolds Number of a Butterfly Valve.....	5
Assumptions.....	5
Calculations.....	6
Task 2: Boundary Flow.....	8
The Boundary Layer	8
Laminar vs Turbulent Boundary Layers.....	8
Separation	8
Butterfly Valve Boundary flow	9
Task 3: Valve Material	10
Glass Fibre Reinforced Plastic (GRP)	10
GRP Valve development.....	10
Implications of Valves Performance	11
Task 4: Modelling Techniques	12
Physical Flow Modelling and Computational Fluid Dynamics (CFD) Modelling.....	12
Advantages of CFD Modelling Vs Physical Modelling.....	12
Disadvantages of CFD Modelling	13
Valve CFD Modelling	13
Task 5: Prototype Testing	14
Butterfly Valve Physical Flow Modelling Schematic	14
Physical Flow Test Description	14
Modelling Comparison.....	15
Summary	16
Appendices.....	17
References	20

Introduction

During this assignment I will be calculating the Reynolds number of petroleum as it passes through the body of a butterfly valve at the base of a 5m tank, using the Reynolds number to state the type of flow present. I will also be discussing boundary flow around the butterfly valve, highlighting the causes of turbulent flow and separation.

I will be discussing the implications of the valve being manufactured from glass fibre reinforced plastics. Also debating the modelling techniques that can be used to simulate fluid flow through the valve and discussing prototype testing.

Task 1: Reynolds Number

What is Reynolds Number?

According to (Whittlesey, 2017) Reynolds number is used to describe the ratio of inertial forces to viscous forces. This is demonstrated in the Reynolds number equation, as shown below:

$$Re = \frac{\text{Inertial forces}}{\text{Viscous forces}}$$

$$\therefore Re = \frac{\rho u d}{\eta}$$

Inertial forces are represented on the top half of the equation in the form of ' $\rho u d$ ' and the viscous force is represented on the bottom half by ' η ', Where:

$Re = \text{Reynolds Number}$

$\rho = \text{Density} \left(\frac{kg}{m^3} \right)$

$u = \text{Velocity} \left(\frac{m}{s} \right)$

$d = \text{Diameter (m)}$

$\eta = \text{Dynamic Viscosity} \left(\frac{\text{Pascal}}{s} \right) \text{ or } \left(\frac{kg}{m \cdot s} \right)$

Flow Types

There are three different types of flow that are calculated using Reynolds number, Laminar flow, transitional flow, and turbulent flow. According to technology development company (Vapourtec Ltd, 2021), Laminar and Turbulent flows are defined as below:

- Laminar flow, also known as streamline flow in pipes, occurs when fluid layers run parallel to one another with no disruption between them. The motion of the particles is very orderly with the particles moving in straight lines, parallel to the inner pipe wall. When in the laminar flow condition, the fluid flows without lateral mixing. Flows with Reynolds numbers up to 2000.
- Turbulent Flow is the opposite of laminar flow, with the flow tending to be more 'chaotic' with many property changes. Unlike Laminar flows parallel layers, turbulent flow mixes within the pipe and has rapid variation in both pressure and flow velocities. Turbulent flow is present when Reynolds number is above 4000.

- Transitional flow is a mixture of both Laminar and Turbulent flows and contains characteristics from both, it is said to occur when the “viscous and Reynolds stresses are of approximately equal magnitude” (The McGraw-Hill Companies Inc, 2003).

Type of Flow	Reynolds Number Boundaries
Laminar	0 – 2000
Transitional	2000 – 4000
Turbulent	4000 >

Reynolds number applications

The ability to calculate the Reynolds number for the fluid flow is essential when it comes to fluid analysis. This calculation is vastly used in most, if not all engineering disciplines to predetermine flow patterns in different fluid flow situations and which type of flow is required is dependant on the application and the desired effect.

For instance, maintaining laminar flow is critical when it comes to the design of an aircraft aerofoil. Laminar flow is predictable, and pressures are less likely to alter by significant amounts where laminar flow is present. As an aircraft aerofoil works using differential pressure for elevation, it is essential that Reynolds number is understood and how the different types of flow may affect the flight of the aircraft. Therefore, Reynolds number is a critical tool for the designing, development, and testing of aircraft aerofoils.

Alternatively, Turbulent flow can also have a desired affect where needed. For example, High turbulent flows have been used in the civil engineering sector to assess how winds with high Reynolds numbers (and therefore turbulent flows) impact tall buildings (G.L. Larose, 2012). They are also purposely achieved in some heat exchanging process, the turbulent air creates an environment for friction to thrive and therefore accelerate the heat changing process.

In the following section, this theory was used to predetermine the type of flow that would be present around a predesigned valve relative to the Reynolds number calculated.

Calculating the Reynolds Number of a Butterfly Valve

Assumptions

When estimating the Reynolds number for provided Butterfly Valve, the following assumptions were made:

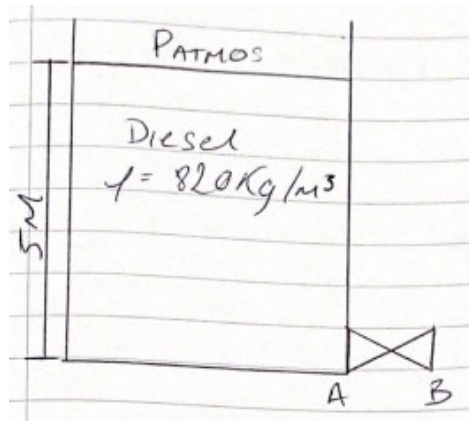
1. The type of petroleum contained in the tank was Diesel Fuel. This fuel was selected as it is one of the most common refined types of petroleum used in the world. Also, the British Standard EN 590 gives a set of specifications that the Diesel fuel must meet for it to be deemed safe to sell within the United Kingdom and Europe (Highland Fuels, 2017). Using these specifications to dictate the figures used for my own calculations gives a more accurate representation of the actual density and viscosity of the fluid that is likely to be present in this scenario. For this reason, I have assumed the following:

- Density (ρ) = 820 kg/m^3
- Kinematic Viscosity = 2cst

This is the lowest Kinematic Viscosity specification for diesel, which when calculated should give the most extreme conditions of flow which will be seen by the valve during its operation.

2. There is a smooth transition between the exit of the tank and the valve, with the entry of the valve and the butterfly valve itself having a smooth surface finish and therefore not generating any friction/energy losses due to heat transfer.
3. The butterfly valve is in the fully open position, also assuming that when the valve is in the 'open' position then all restrictions and resistance to flow are negligible and therefore will not greatly affect the calculations.
4. There is nothing connected on the opposite side of the butterfly valve and therefore the pressure at point B is equal to P_{Atmos} .
5. 3D CAD drawing of the butterfly valve is to scale and therefore the internal diameter of the pipe is 194mm or 0.194m.
6. The pressure head is 5m, despite the actual pressure head being equal to $5 - \frac{0.194}{2} \text{ m}$.

Calculations



Diesel Tank Diagram (Taken from 'Handwritten notes: Page 1')

To calculate Reynolds number:

$$Re = \frac{\rho u d}{\eta}$$

To calculate unknown velocity, Bernoulli's equation is required:

$$P_A + \rho g z_A + \frac{1}{2} \rho u_A^2 = P_B + \rho g z_B + \frac{1}{2} \rho u_B^2$$

There is no difference in height and therefore z_A and z_B and therefore both = 0

As the Area at A is significantly greater than the Area at B, I have assumed that the velocity at A is negligible and therefore: $\frac{1}{2} \rho u_A^2 = 0$.

$$P_A = P_B + \frac{1}{2} \rho u_B^2$$

$$P_A - P_B = \frac{1}{2} \rho u_B^2$$

$$(\rho g h + P_{Atmos}) - P_{Atmos} = \frac{1}{2} \rho u_B^2$$

Assuming that the pressure at point B is 0:

$$\rho g h = \frac{1}{2} \rho u_B^2$$

$$\therefore u_B = \sqrt{2gh}$$

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

$$u_B = 9.90 \frac{m}{s}$$

Velocity calculated at point B is used to calculate Reynolds Number, where:

Dynamic viscosity= Density x Kinematic Viscosity

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

$$\therefore Re = \frac{u d}{\nu}$$

$$Re = \frac{9.9 \times 0.194}{2 \times 10^{-6}}$$

$$Re = 960,300$$

As Reynolds number for flow is greater than 4,000, flow is Turbulent.

Task 2: Boundary Flow

The Boundary Layer

NASA describes the boundary layer as a thin layer of fluid near the surface of an object (or in this scenario, the butterfly valve inner wall). In this area, the flow velocity is affected by the shear stresses that exist between the fluid layers, this causes the velocity of the fluid to change from zero at the surface and increase to a free stream value away from the surface (National Aeronautics and Space Administration, 2021).

The size of the boundary layer is highly dependent on the viscosity of the fluid, fluids with lower viscosities will reach the free stream value closer to the surface, whereas higher viscosity fluid reach the free stream velocity at a greater distance from the surface. This is because fluids higher in viscosity exert stronger forces of attraction and have a stronger influence on the other fluid streams (Tec-Science, 2020).

Laminar vs Turbulent Boundary Layers

As Mentioned previously, the type of boundary layer that is created by the fluid depends highly on the how viscous the fluid is, when the fluid comes in to contact with the surface of the valve, a laminar sublayer is created. The more viscous the fluid, the bigger the sublayer created. The flow is likely to be laminar when the fluid is of a higher viscosity, as when the sublayer is larger, imperfections in the surface are covered by the sublayer, this creating a hydraulically smooth surface.

When Reynolds number is lower than 2000, the boundary layer is laminar. A laminar boundary layers increase in velocity uniformly as the layers move further away from the surface area and in to the 'free stream'.

Whereas, the turbulent boundary layer where Reynolds number exceeds 4000, has an unsteady flow due to the high amount of mixing through layers. Turbulent flows are more sensitive to wall roughness, especially when the Reynolds number is far higher, as the roughness elements that may protrude through the laminar sublayer completely modifies the flow across the whole boundary layer (E.L.Houghton, 2012).

Separation

Separation occurs when a fluids velocity in the boundary layer is slowing down, and the pressure is increasing. For example, when an obstacle is placed in a fluid flow a boundary layer is created around that obstacle. As the obstacle creates a resistance to flow, a high-pressure point is created at the point of fluid impact expending the fluids kinetic energy, thus causing is to decelerate to zero. Once the velocity is at zero, the adverse pressure gradient begins to push the boundary layer flow backwards creating an eddy (a circular current of fluid).

Butterfly Valve Boundary flow

As mentioned previously, for Reynolds number of the butterfly valve it is assumed that when the butterfly valve is fully open, the 'butterfly' itself has no effect on the flow passing through the valve, realistically this is not the case. As flow passes the butterfly another boundary layer would be created around the butterfly itself. Depending on the geometry of the butterfly, would depend on which type of boundary created. However, looking at the design, the butterfly has several areas where a turbulent boundary layer is likely to develop and therefore an increase of flow separation.

The rivets that are used to attach the butterfly to the central shaft will influence the surface roughness, as there are 2 appendages likely to protrude through the laminar boundary layer and therefore increase the already turbulent flow. The rivets protrude quite substantially, thus creating areas for pressure points to generate and rapidly increase separation. This would have a drastic effect on the process, the desired flow may no longer be able to be obtainable. The pressure drop in the system also creates an atmosphere for cavitation to develop which is common on butterfly valves.

It's likely the butterfly has been designed with rivets for future maintenance (changing of the butterfly) in mind and to keep production costs down. However, with the implications these rivets create in the process, further cost implications would occur in the future, such as:

- Downtime
- Spare parts
- Maintenance costs
- Process losses

Task 3: Valve Material

Glass Fibre Reinforced Plastic (GRP)

It is suggested that the butterfly valve is to be manufactured from glass fibre reinforced plastic (GRP). Over the years in industry, there has been extensive research and development into the manufacturing of GRP so that it can be used in more mechanical processes. The material itself is highly desirable due its mechanical properties, according to composite product specialist (Evergrip, 2021)these are as follows:

- Non-corroding
- Highly durable
- Electrically nonconductive
- Light weight
- High strength to weight ratio

However, historically it has been proven that GRP naturally has a rough surface area due to the way it is manufactured. GRP is traditionally a moulded product, meaning that it is placed in or around the shape of the object it is desired to take, thus making it difficult to acquire the smooth surface finish desired.

GRP Valve development

Historically butterfly valves have been bad performers, throughout the years they have proven to be susceptible to cavitation due to the nature of their operation and design. Constant improvements have been attempted including the manufacturing of butterfly valves out of GRP materials.

The manufacture of a 60inch GRP butterfly valve in Fawley Power station was one of the first attempts to be recorded in the development of the butterfly valve in 1978 (WALKER, 1983). The butterfly valve was manufactured to replace a cast iron valve of similar design, the cast iron valves lifespan was drastically decreased due to the 'seaside' conditions surrounding the power plant.

Despite this seeming to be a success, the development of butterfly valves has taken another step forward in recent years, with composite reinforced components offering better pressure values, due to a smoother surface finish as provided by (Pentair, 2021).

Modern day valves must be designed and tested to the British Standard EN 736, where the design must be appropriately validated through cycle prototype testing before being certified for manufacture (The British Standards Institution, 2021).

Implications of Valves Performance

As previously mentioned in the Boundary layer section, materials with rough surface finishes provide additional difficulties when it comes to the flow of the fluid. When rough surface finishes are present turbulent flow is almost inevitable and therefore so are the implications associated with it. Even if the design of the valve was altered to eradicate current flow issues (rivets and butterfly design) then using GRP as the manufacturing material would still promote high turbulence and separation. This would cause a large amount of pressure droppage and therefore would have a detrimental effect on the maximum mass flow rate achievable, AKA, the 'choking of the valve'.

Task 4: Modelling Techniques

Physical Flow Modelling and Computational Fluid Dynamics (CFD) Modelling

There are two main types of Modelling techniques commonly used across all engineering disciplines, Physical modelling, and Computational Fluid Dynamics modelling (otherwise known as CFD modelling).

The Physical modelling technique is primarily the development and testing of prototypes. Once a concept design is created, it is developed either to scale, or downscaled for ease of testing as a prototype. The prototype can then set up in a test rig (a wind tunnel for instance) and the fluid flows around/in the prototype can heavily assessed for relative engineering data. The Data collected can then be used for further design improvements.

CFD is a modelling technique that uses 3 dimensional representations to simulate interactions between fluids and surfaces that are defined by boundary conditions. This method uses the principles of fluid mechanics, using algorithms and numerical methods to solve fluid flow problems (David A. Tillman, 2012).

CFD is one of the most utilized modelling techniques in the world giving users the ability to carry out fluid flow experiments during the designing stage of a product, this is massively beneficial for manufacturers.

Advantages of CFD Modelling Vs Physical Modelling

According to the product research and engineering company (PRE Technologies, 2021), advantages of CFD modelling are as follows:

- Relatively inexpensive – Prior to the development of CFD, prototypes and physical tests would have been carried out to retrieve further engineering data required for the development of a design. However, it is now possible to carry out simulations of the tests to highlight obvious problem areas of design during the development and design stages. Therefore, unnecessary costs are incurred on prototypes.
- Time Saving – Depending on what CFD software is being used and what is being calculated, CFD simulations can be executed rapidly, with more complex taking a little longer. Waiting for physical prototypes to be developed can be extensive and waste time.
- Comprehensive information – when physical experiments are carried out, data can only be taken from the points where sensors and gauges have been strategically placed. Whereas, CFD allows the data to be extracted from any area of interest.
- Alternate conditions can be simulated – CFD allows the designer to imitate different physical conditions that may impact the design. This is much harder to test in physical flow modelling.

- CFD models are always represented at full size – When prototypes are used as a form of modelling, larger designs usually must be downsized for physical testing which may have an affect on results.

Disadvantages of CFD Modelling

Although there are many clear advantages to the use of CFD, there are always going to be some downfalls. Below are the disadvantages listed by Simulation and Analysis specialists (IFES, 2021):

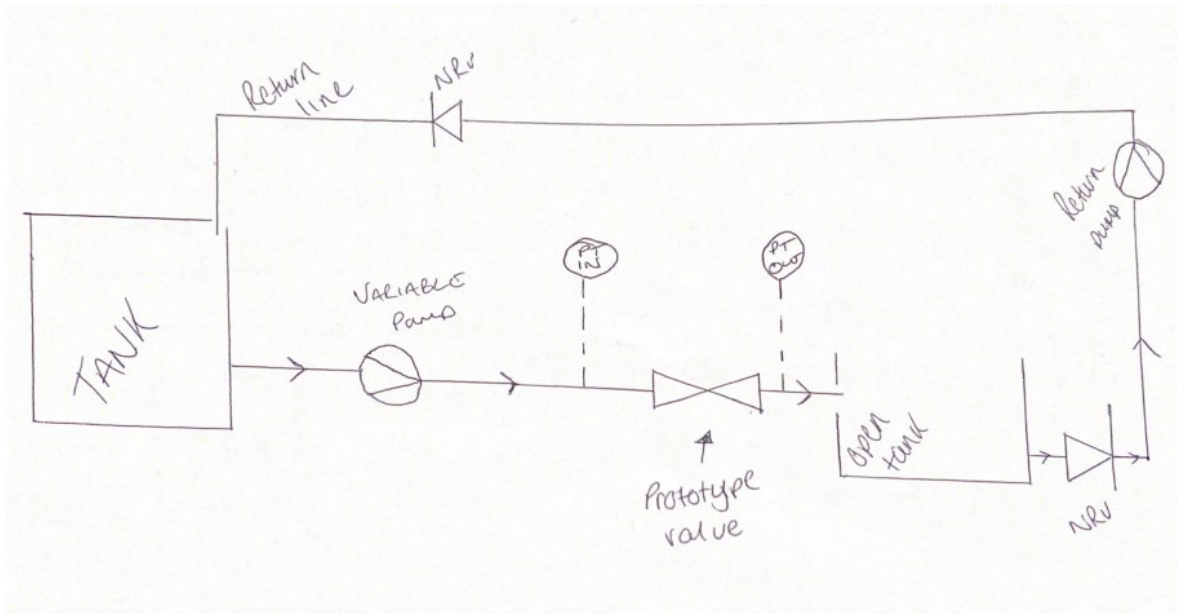
- Errors can occur because of simplified boundary layers or simplified flow models.
- Large models may increase computation time
- If a mistake is made in the calculation, it can be more costly in the long run, in comparison to physical experiments.

Valve CFD Modelling

Using CFD modelling it is possible to highlight problematic design issues that increase turbulent flow within the valve, whilst giving a representation of the valve efficiency in the terms of pressure drop across the valve.

Task 5: Prototype Testing

Butterfly Valve Physical Flow Modelling Schematic



Hand drawn Schematic

Physical Flow Test Description

A simple rig will be created as shown in the schematic above mimicking the makeup of the process the valve is to be inserted. With the use of a return feed line the physical flow test can be done over a long period of time and not pausing during the filling of the tank.

Ideally the exact fluid (diesel) would be used in the testing process so to create accurate results, if due to health and safety reason this could not be accomplished, a fluid with the same viscosity and density would have to be used if possible.

For this test to return accurate readings that will correlate with CFD testing, it is important that all the conditions are the same or as near as possible. It would be impractical to create the 5m tank for the sake of the test, therefore a solution is needed to artificially create the same fluid velocity at the entrance of the valve. For this reason, a variable speed pump will be inserted up stream of the flow.

Pressure sensor 'IN' can be used to ensure correct pressure is being achieved using the variable speed pump on entry to the valve. Whereas the pressure 'OUT' sensor is used to record data of the pressure drop across the body of the valve.

Modelling Comparison

Once physical flow modelling has been completed then the results of the test can be compared with that of the CFD testing from task 4. If the testing on both parts was completed correctly with the same conditions used for each test, the results should correlate and represent similar figures.

Using both CFD and physical flow testing provides a valuable tool for pinpointing design faults. For instance, after testing using a physical flow model, I would expect the pressure drop across the valve to be the same, if not near the pressure drop highlighted in CFD modelling. If there is an obvious correlation between the data matching, the areas of high frictional losses simulated in the CFD are the obvious design issues that need addressing. Which in this scenario I would anticipate being the butterfly itself, and the rivets fastening to the centre shaft.

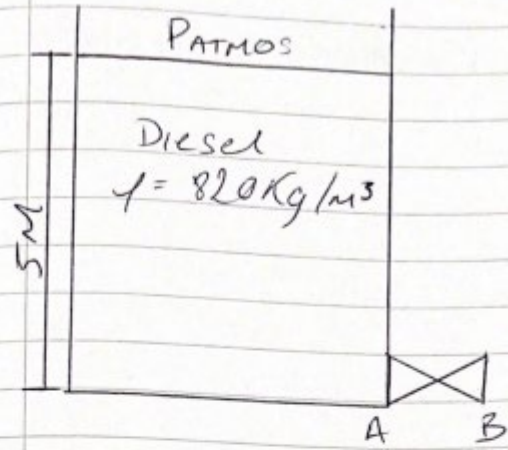
Summary

To Conclude with this assignment, I have successfully calculated the Reynolds number of petroleum passing through the body of a valve, stating the type of flow present. I have examined Boundary layer flow around the body of the valve and discussed the effects of turbulence and separation.

I have debated the use of GPR as a material for the butterfly valve and its implications. I have also successfully reviewed different modelling techniques and prototype testing.

Appendices

Reynold's number calculations



$Re = \frac{\rho u d}{\eta}$
 $Re = \text{reynolds number}$
 $\rho = \text{density}$
 $u = \text{velocity}$
 $d = \text{diameter}$

$d = 194 \text{ mm} \therefore d = 0.194 \text{ m}$

to calculate velocity use Bernoulli's

$$P_A + \rho g z_A + \frac{1}{2} \rho u_A^2 = P_B + \rho g z_B + \frac{1}{2} \rho u_B^2$$

$$A_A > A_B \therefore \frac{1}{2} \rho u_A^2 = 0$$

$$P_A = P_B + \frac{1}{2} \rho u_B^2$$

$$P_A - P_B = \frac{1}{2} \rho u_B^2$$

$$\therefore \rho g h = \frac{1}{2} \rho u_B^2$$

$$u_B = \frac{\sqrt{2 g h}}{\sqrt{2 \times 9.81 \times 5}}$$

$$\therefore u_B = 9.90 \text{ m/s}$$

Calculate mass flow rate $\dot{m}$

$$\dot{m} = \dot{V} \times \rho \quad \text{Where } \dot{V} = \text{Volumetric flow rate}$$

$$\dot{V} = U_B \times A_{\text{valve}}$$

$$A_{\text{valve}} = \pi r^2$$

$$= \pi \left(\frac{0.194}{2} \right)^2$$

$$A_{\text{valve}} = 0.03 \text{ m}^2$$

$$\dot{V} = U_B \times A_{\text{valve}}$$

$$\therefore \dot{V} = 9.9 \times 0.03$$

$$\dot{V} = 0.293 \text{ m}^3/\text{s}$$

$$\therefore \dot{m} = 0.293 \times 820$$

$$\dot{m} = 239.96 \text{ Kg/s}$$

Taking viscosity of Diesel from the same British Standard chart.

$$\text{Diesel Kinematic viscosity} = 2 \text{ cSt} = \text{mm}^2/\text{s}$$

$$\therefore 2 \times 10^{-6} \text{ m}^2/\text{s}$$

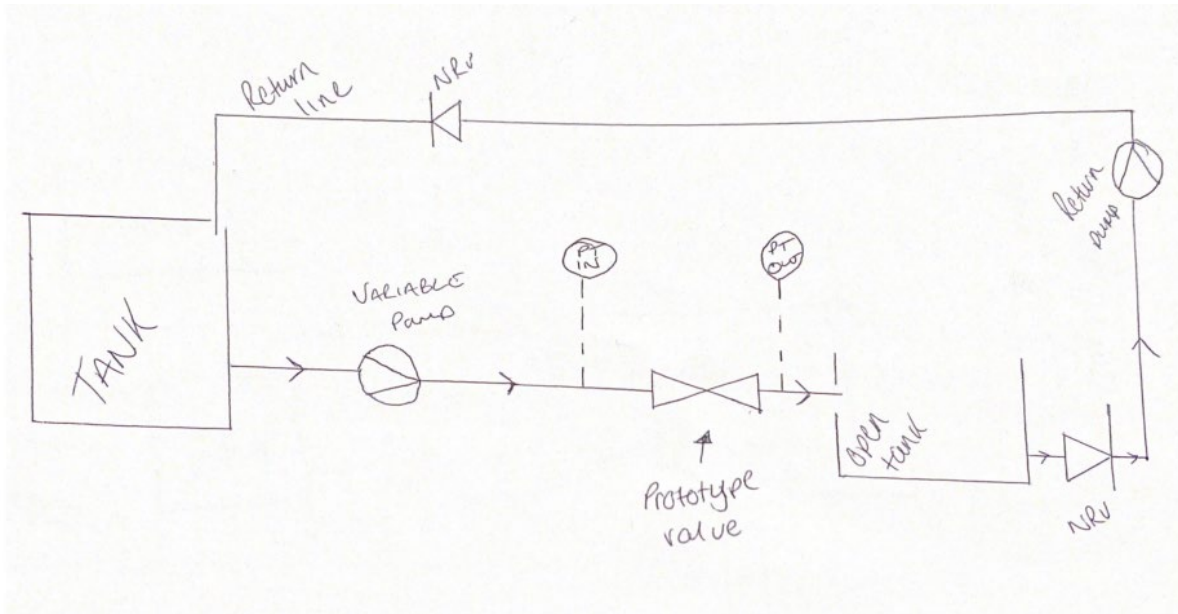
$$Re = \frac{\rho U_B r}{\mu}$$

$$Re = \frac{9.9 \times 0.194}{2 \times 10^{-6}}$$

$$Re = 960,300 \therefore \text{Flow is turbulent.}$$

$$Re > 4000.$$

Handwritten calculations: Page 2



Hand Drawn Schematic

References

- David A. Tillman, D. N. (2012). *Solid Fuel Blending*. Waltham: Butterworth-Heinemann.
- E.L.Houghton, P. S. (2012). *Aerodynamics for Engineering Students (Sixth Edition)*. Oxford: Butterworth-Heinemann.
- Evergrip. (2021, July 28). *What is GRP*. Retrieved July 28, 2021, from Evergrip: <https://evergrip.com/what-is-grp/>
- G.L. Larose, A. W. (2012). Sectional model investigation at high Reynolds number for a super tall building. *Journal of Wind Engineering and Industrial Aerodynamics*, 49-55.
- Highland Fuels. (2017, December 12). *Diesel UK Marketing Specification*. Retrieved from www.highlandfuels.co.uk: <https://www.highlandfuels.co.uk/wp-content/uploads/2018/03/Diesel-EN590-Specification.pdf>
- IFES. (2021, July 29). *COMPUTATIONAL FLUID DYNAMICS (CFD)*. Retrieved July 29, 2021, from ifes-koeln: <https://www.ifes-koeln.de/en/services-20-years-project-experience/simulation-analysis/computational-fluid-dynamics-cfd.html>
- National Aeronautics and Space Administration. (2021, May 13). *Boundary Layer*. Retrieved July 27, 2021, from NASA.gov: <https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html>
- Pentair. (2021, July 30). *Keystone Composeal Resilient Seated Butterfly Valves*. Retrieved July 30, 2021, from mjwilsongroup: <https://www.mjwilsongroup.co.uk/dynamic/pdfs/Keystone-Composeal-Butterfly-Valves.pdf>
- PRE Technologies. (2021, July 29). *Advantages of computational fluid dynamics*. Retrieved July 29, 2021, from pretechnologies.com: <http://www.pretechnologies.com/services/computational-fluid-dynamics/advantage>
- Tec-Science. (2020, April 26). *The hydrodynamic boundary layer of a flow has a decisive influence on heat and mass transport*. Retrieved July 27, 2021, from tec-science.com: <https://www.tec-science.com/thermodynamics/heat/boundary-layer-and-dimensionless-similarity-parameters/>
- The British Standards Institution. (2021, July 30). *BS EN 736-1:2018*. Retrieved July 30, 2021, from bsigroup.com: <https://shop.bsigroup.com/ProductDetail?pid=000000000030344096>
- The McGraw-Hill Companies Inc. (2003, January 1). *Transitional Flow*. Retrieved July 27, 2021, from The Free Dictionary: <https://encyclopedia2.thefreedictionary.com/transitional+flow>
- Vapourtec Ltd. (2021, July 22). *Laminar and Turbulent Flow*. Retrieved from www.vapourtec.com: <https://www.vapourtec.com/flow-chemistry/laminar-turbulent/#>
- WALKER, M. H.-B. (1983). *Large GRP Butterfly Valves*. Bristol: Springer, Dordrecht.
- Whittlesey, R. (2017). *Wind Energy Engineering*. Los Angeles: Elsevier Inc.