



FLUID FLOW ASSESSMENT

ACMA6010

Samuel Thomas Melmoth (1809093)

180909@student.uwtsd.ac.uk

Abstract

CFD is a crucial tool that can be used on new products before implementation into systems in the design phase as well as optimising existing objects. Its uses are wide ranging from allowing designers to produce more efficient systems to improve safety.

Table of Contents

ABSTRACT	1
INTRODUCTION	1
BODY	2
1	2
2	4
3	5
4	6
FINITE DIFFERENCE METHOD	6
FINITE VOLUME METHOD.....	6
FINITE ELEMENT METHOD	6
5	7
CONCLUSION	8
WORKS CITED	9

Introduction

Fluid flow dynamics is the study of the forces acting on fluids whilst they are in motion. This assignment is to assess and answer questions on how petroleum flows through a butterfly valve. To do so I have had to apply knowledge of the material that has been taught to me over the duration of this module. I have also had to research on the topic whilst looking for suitable material to back up my statements and improve my understanding of the subject. In the first question I explain what the Reynolds number is, what figures come together to give that value and how it influences fluid flowing through a system. Calculations are then performed to calculate the Reynolds number for our scenario. Secondly the design characteristics are discussed and how they play a part in the fluid flow around the valve disk and mechanism. Thirdly the material choice was specified and compared to other possible options that are in the market to compare both advantages and disadvantages. In the fourth question I was tasked to investigate and compare the different computational modelling techniques available to assess the fluid flow around the valve. This compared the Finite Difference, Volume and Element methods. Lastly, I gave my opinions on how the component should be tested in a laboratory to back up the computational analysis that was done in question 4.

Body

1

The Reynolds number is the correlation between inertial forces (density x velocity x diameter) and viscous forces (Dynamic viscosity) as a fluid flows. It is also a dimensionless quantity; this means that there are no physical dimensions such as 'mm' or 's' allocated to this number. The Reynolds number is used most often to classify flows between Laminar and turbulent. A value between 0-2000 is considered a laminar flow whereas $Re > 2000$ would be a turbulent flow. (Menon, 2015)

Calculating the Reynolds number is important as it determines which type of flow there is through the valve. Getting this number as low as possible is crucial for keeping the liquid in laminar flow. This is preferable due to there being less friction and heat build-up in the system of transporting the liquid. In our scenario there is no consideration on how far the petroleum must travel but as there is no pump mentioned then there is only the hydrostatic head of the liquid forcing the liquid through and in order to maximise this advantage of gravity, keeping friction as minimal as possible is key.

Laminar flow occurs when the liquid stays in layers as it flows through the pipe and doesn't cause eddies or turbulences. As a liquid passes through a valve, it will become turbulent, and the Reynolds number will increase due to the increased surface area of the butterfly valve and the disruption it causes to the layers of the liquid.

The Reynolds number for this scenario can be found by dividing the total inertial forces by the total viscous forces. In order to use this equation, I have neglected the central plate and assumed that the diameter of the valve has no obstructions.

Equation for Reynolds number:

$$Re = \frac{\rho v d}{\eta}$$
$$\rho = 800 \text{ kg/m}^3$$
$$v = ?$$
$$d = 194 \text{ mm}$$
$$\eta = \frac{\tau}{\frac{\Delta u}{\Delta y}}$$

To find v use Bernoulli's equation

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

$$\rho g h_1 = \frac{1}{2} \rho v_2^2$$
$$v_2^2 = \frac{2(\rho g h_1)}{\rho}$$
$$v_2^2 = 2 g h_1$$
$$v_2 = \sqrt{2 g h_1}$$
$$v_2 = \sqrt{(2)(9.81)(5)}$$
$$v_2 = 10 \text{ m s}^{-1}$$

Coefficient of viscosity

$$\rho v = \eta$$

Kinematic viscosity x Density = Dynamic viscosity

$$2.6 \times 10^{-5} \times 800 = \eta$$

$$\eta = 0.0208$$

$$Re = \frac{(800)(10)(0.194)}{0.0208}$$

$$Re = 74615.4$$

2

Flow through a butterfly valve is controlled by a round plate connected to a shaft that is located in the centre of the cylindrical channel in the valve housing. This device allows control over the fluid flow through it. The full range of the valve is within 45° , full flow when the plate is at a right angle to the direction of flow through the cylinder. No flow when the disk is rotated

Surfaces that the fluid is in contact with are the valve wall and disk. When fluid passes by the disk it will be parallel to the flow. As fluid hits the leading edge of the disk the boundary layer is at its thinnest then as it travels along the length to the trailing edge it expands. Design implications, by using rivets to connect the disk to the shaft this allows nonskilled workers to be used to assemble the valve. If there was to be a permanent connection between the disk and shaft such as a weld this would require a skilled worker that costs more to manufacture.

By riveting the two components together this allows for individual components to be replaced if they were to fail.

by adding rivets to the disk will have an adverse effect on the pressure gradient on the back side of the rivet and will induce boundary layer separation. Causing turbulent flow. (M. Constantini, 2019) Also, the more that the rivets protrude from the smooth surface of the disk and shaft the worse the effect on the flow disruption. If the rivets were to be countersunk this would improve the design and limit disruption.

As the valve has flange connections on either side this allows it to keep the diameter consistent between the pipe and the valve. Due to this there are no step changes or changes in diameter that will cause disruption to the flow causing turbulent flow. If the valve were to have threaded connections there would be a step up at the input and step down at the output, increasing turbulent flow. Increasing frictional losses.

The first use of GRP to make a butterfly valve was back in 1980 by Boring and supplied to Fawley Power Station with two to be used on a cooling water line. They were replacing the two existing cast iron valves as they were prone to corrosion due to the proximity to the seaside. (M.H. Bryan_Brown, 1983)

There are many advantages to using Glass fiber reinforced plastics or GRP's. The first major benefit is that this material will not corrode like other more classic valve materials such as steels and other metals. This is evident as there is plenty of evidence of widespread use of GRP pipelines in civil engineering projects that have lifetimes of at least 60 years. (Suiith Bobba, 2018) Petroleum is a very corrosive substance so this is a crucial factor. (Monika Ziolkowska, 2015) Secondly GRP has a much higher strength to weight ratio in comparison to most metals this material characteristic will reduce the parts overall weight making it easier for manual handling and reducing the need for structural supports around the installation sight at the bottom of the tank. The fluid being stored is petroleum which is a flammable liquid and electricity, and static are known to be ignition sources. Therefore, by eliminating these would increase the safety of the system. GRP is a non-conductor and perfect for this role.

It is possible for liquid to be absorbed by GRP due to the epoxy not binding and perfectly creating a seal with the glass strands within the material. (Jones, 2012) Due to the valves constant contact with fluid this is a characteristic that is not desirable as it would cause the material to swell and could cause the valve to fail in several ways. Such as cracking, leaking fluid or not being able to rotate the valve disk because of material distortion. Due to this problem only parts of a high standard can be used in critical areas such as flammable fuel storage. BS 8018-2.8 (British standards, 1989)

GRP is a hard but brittle solid. If the valve disk were to be made from this material, then it is more likely for the edges of the valve to chip away with constant use as the tolerances are so close in order to make a seal that if material were to build up and contact was made between two surfaces, due to the force of the actuator moving the disk it would be more likely to cause damage than if a metal valve was used. (Imad Shakir Abbood, 2021)

Common simulation techniques used for fluid analysis are Finite difference, volume and element analysis. These three are all computational fluid dynamics techniques that use computing power to calculate long matrices that are formed from complex differential equations. (LeVeque, 2002)

Finite Difference Method

This technique solves differential equations by taking smaller steps and giving the equations finite boundaries approximating other nearby points to make it possible to solve the equations with a numerical values and get back a usable figure to quantify your results.

(Autar Kaw, 2008) The SBP-SAT method within this is an accurate technique for discretizing equations and placing boundary conditions of partial differential equations using high order finite differences. This method is most suited for rectangular geometries or for larger scale simulations for applications such as seismology or meteorology. This is due to it not being able to deal with curved edges when building it's meshes on parts. But if the grids of the mesh can be refined then it offers very accurate results for analysis.

Finite Volume Method

When using the Finite Volume Method for analysing it begins by dividing the CAD model into smaller cells of known finite dimensions that are geometrically simple. This method uses conservation laws, knowing this it creates equations that are traditionally very good at solving fluid flow problems. An advantage for the volume method is that it only needs to do evaluations of the cell boundaries and not the nodes in each corner as the flux conservation does not require it. This makes the calculations quicker on the computer. A disadvantage for the volume method is that whilst the cell mesh can be edited and refined to be more accurate in desirable locations such as corners of interest, the functions that approximate the cells cannot easily be made of a higher order, whereas this is possible in the other two methods. (Sjodin, 2016)

Finite Element Method

The Finite Element Method divides up a CAD model into smaller simple geometrical shapes that form a mesh of finite shapes, most commonly triangles for 2D shapes and tetrahedra's for 3D shapes with a node in each corner, or for more accuracy and detail nodes can be added on the boundaries as well. Finite element analysis is most used for stress analysis due to its similarity to well-known efficient methods of physical and electromagnetics analysis. It also has the capability to increase the order of its elements to include second- or third-degree polynomials to give very accurate results. As well as this it allows differing concentration of nodes within the mesh that allows greater accuracies in certain key areas where stress is known to be a problem. This allows the operator to use their personal experience to create a model that'll take less time and computing power to solve whilst still gathering accurate and important data from critical areas. This lends itself well to big scale analysis such as structural analysis of buildings. It is possible to combine multiple types of functions in the element method. This is called multi mixed formulations. A Disadvantage for element analysis is that it is the hardest to implement due to its mathematical complexity and therefore needs expertise and a good understanding of the field in order to use it effectively and access the advantages such as the mixed formulations.

To conclude for assessing fluid flow through a butterfly valve I believe that the finite volume method is the most abdicate due to its proven success in that area.

The computational analysis should always be backed up by real world testing. In order to verify the results. By testing the fluid flow over the valve disk the key measurable is how long it takes for the fluid flow to return to laminar after becoming turbulent through contact with the valve.

The equipment I would use for conducting this experiment would be:

A pump to force the petroleum through the valve at the desired flow that would match the hydrostatic head of the 5-meter tank. Getting rid of the tank is necessary in order to be able to conduct the test in a lab. As well as negating the risk of a leak and needing bunding for the large amount of petroleum as it is a hazardous fluid and causes health and safety concerns. Also, this allows control over the flow rate through the valve. As this pump would cause turbulent flow it is key that there is enough distance of pipe in-between the pump and valve so that the flow can return to laminar before contacting the valve in order to control the variables.

Clear pipe exiting the valve in order to be able to see the flow and record results along the length. This Would then feedback around to the pump in order to use as little petroleum as possible to reduce the risk of having large quantities of flammable liquids in the lab. As well as reducing the cost of the experiment.

Slow motion camera equipment to aid visual analysis of the flow on its return to laminar.

Die, as petroleum is transparent this will help to show the flow in the pipe.

Data will be gathered from this experiment by pumping petroleum through the valve at 10ms^{-1} as that is the velocity calculated by Bernoulli's equation for the hydrostatic head within the tank and the aperture of the valve both specified in the brief. Die would then be placed in the stream of fluid in order to observe the flow emerging from the other side through a slow-motion camera in order to locate how far from the valve does the flow return to laminar. This data can then be compared to the computational analysis to test its accuracy. Variables on the test can be changed such as velocity and positioning on the valve disk in order to see how this affects the flow as it travels out of the valve and on down the rest of the system. Installation of the valve in the correct direction has been emphasised in many installation guides from various companies. Using this test and changing this variable, would allow a quantifiable statistic to be found in order to back up these statements in terms of performance.

Conclusion

To conclude the Reynolds number for this scenario is calculated to be ____ . As well as this I believe that the designer used rivets to attach the valve disk to the shaft in order to reduce the production skill level for assembly as well as making the parts individually replaceable to aid repair. The butterfly valve was made from glass fiber reinforced plastic, this is because it outperforms metal valves regarding corrosion as petroleum is a very corrosive substance. As well as benefits in safety as the valve is made from an electrical insulating material. Out of the three modelling techniques evaluated I found the Finite Volume Method to be the most desirable for this application as it is the most used for CFD. The experiment that I described in the final question can be used to record the distance taken for the turbulent flow to stop after hitting the valve this will allow for improvements in design by reducing this distance and proving the computational models. During this assessment I have learnt to apply the skills and methods taught to me during this module and apply them to a real-life scenario. The research has also lead to a greater understanding of the subject.

Works Cited

- Autar Kaw, E. E. K., 2008. *Numerical methods with applications*. 1 ed. s.l.:University of South Florida .
- British standards, 1989. *BS 8010-2.5*. London : British Standards .
- Imad Shakir Abbood, S. A. O. K. F. H. M. A. J., 2021. Properties evaluation of fiber reinforced polymers and their constituent materials used in structures. *Materials today proceedings* , Volume 43, pp. 1003-1008.
- Jones, M., 2012. *Valve Amplifiers*. 4 ed. s.l.:Newnes.
- LeVeque, R. J., 2002. *Finite-Volume Method for Hyperbolic Problems*. 1 ed. s.l.:Cambridge University Press .
- M. Constantini, S. R. S. K. C. F. U. G. S. H. C. K., 2019. Experimental study of bump effects on boundary layer transition in compressible high Reynolds number flow. *Experimental thermal and fluid science*, Volume 106, pp. 234-254.
- M.H. Bryan_Brown, D. W., 1983. Large GRP Butterfly Valves . In: *Composite Structures 2*. Dordrecht: Springer, pp. 214-234.
- Menon, E. S., 2015. *Transmission Pipeline Calculations and Simulations Manual*. s.l.:Elsevier Inc. .
- Monika Ziolkowska, D. W., 2015. Corrosiveness of fuel during storage processes. In: K. Biernat, ed. *Storage stability of fuels* . s.l.:intech open .
- Sjodin, B., 2016. *MachineDesign*. [Online]
Available at: <https://www.machinedesign.com/3d-printing-cad/fea-and-simulation/article/21832072/whats-the-difference-between-fem-fdm-and-fvm>
[Accessed 30 July 2021].
- Suiith Bobba, Z. L. E. Z. S. S., 2018. *Fluid Flow and Static Analysis of E-glass Fiber Reinforced Pipe Joints Versus S-glass Fiber Reinforced Pipe Joints*, Putra: University Putra Malaysia.