

LEVEL 6 ASSESSMENT SPECIFICATION

Student name:	Rhodri Williams	Student ID number:	1910918
Programme:	School of Engineering Portfolio		
Module:	Structural & Fluid Analysis		
Module code:	ACMA6010	Contribution to Overall Module Assessment (%):	50%
Lecturer:	Kelvin Lake	Internal Verifier:	Rob Goodson
Assignment Title:	Designing a butterfly valve Fluid flow assessment	Word count (or equivalent):	3000
Submission deadline:	30/07/2021	Return date of provisional marks & written feedback:	20 Working days after the submission date
Submission method:	All written assessments, where practical and possible, must be submitted via Turnitin unless otherwise instructed by the Lecturer. (Please DO NOT put this assessment specification into Turnitin or it will match many similarities with other students' submissions.) Alternative submission method (if applicable): Late submission of the assessment will result in a late penalty mark. Penalties for late submission: Up to one week late, maximum mark of 40%. Over one week late, 0%. Only the Extenuating Circumstances Panel may approve a change to submission dates.		
Academic honesty / referencing:	Academic honesty is required. In the main body of your submission you must give credit to authors on whose research and ideas your work is based. Append to your submission a reference list that indicates the books, articles, etc. that you have used, cited or quoted in order to complete this assessment.		

Module Learning Outcomes (from module syllabus)

Critically analyse and evaluate techniques for structural and fluid problems and CAE results.

Table of Contents

	Page
Introduction	1
Part 1 - Reynolds Number	2-3
Part 2 - a. Boundary Conditions	4
b. Design	5
Part 3 – Glass-Fibre Reinforced Plastic	6
Part 4 – a. CDF Methods	7-8
b. Method Selection	8
Part 5 – Experiment	9-10
Conclusion	10
Bibliography	11-12
Appendix	13-14

Introduction

A butterfly valve is a fluid flow control device used widely in many industries. The purpose of this essay is to understand the importance of the Reynolds number and how it effects the flow around the valve and understanding the boundary conditions when the valve is opened.

An analysis of the design and the material used is to be carried out to gain an understanding of the purpose of the valve.

Modelling techniques for CFD play an important role in valve design by simulating fluid flow without having to make a scale model, saving time and money. Different methods to perform CDF are to be analysed with and the appropriate model selected to run the simulation. To confirm the simulation accurate, an experiment is needed to confirm the data is correct. How the experiment is set up and used is to be discussed.

PART 1

Reynolds Number

The Reynolds number is a dimensionless value used to predict whether fluid flow is laminar or turbulent and is the relationship between inertial and viscous forces in a fluid.

$$Re = \frac{\text{Inertial Force}}{\text{Viscous Force}} = \frac{\text{Fluid and Flow Properties}}{\text{Fluid Properties}}$$

The number can be calculated using the following equation: $Re = \frac{\rho VL}{\mu} = \frac{VL}{\nu}$

V = Velocity of flow (m/s)

L = Characteristic linear dimension (m)

ρ = Fluid density (kg/m³)

μ = Dynamic viscosity (Pa.s)

ν = Kinetic viscosity (m²/s)

By observing the above equations, we can see that the Reynolds number, Re, is directly proportional to the fluid's density, velocity, and length of travel, indicating that as any one of these increases, so will the Reynolds number. Conversely, the value of the viscosity, is inversely proportional to the Re, meaning that if this value is to increase, then the Reynolds number will decrease. (Reynolds Number , 2021)

A low Reynolds number (< 2000) indicates that the flow is laminar, meaning that viscous forces are dominant. Whilst these forces are dominant, the fluid flow is smooth and predictable, with flow lines moving at different velocities and running parallel to one another, with the slowest moving velocities at the boundary layer, with each layer moving slightly faster the further away from the boundary layer they get, with almost no mixing between layers.

A high Reynolds number (> 4000) implies that the flow is turbulent. In a turbulent flow the inertial forces are dominant. With dominant inertial forces in the fluid, the flow will become unpredictable with unsteady circular eddies mixing within the flow causing turbulence and energy loss.

Transitional flow is the area between flows is a mixture of both laminar and turbulent flows. A disturbance introduced into a laminar flow will be amplified, and as a result will exhibit flows that are not only orderly in parts but also show irregular fluctuations in others. A mix of both. (Durst, 2008)

The importance of the Reynolds number is well documented with applications in many different industries and has an important role in heat transfer and fluid dynamics calculations. For example, it is extremely important for a hydraulic system used for heat transfer or to provide necessary cooling to a nuclear reactor core to maintain a suitable flow rate. This flow rate is affected by the valves and bends in the pipes, as well as friction within

straight pipes. Other heat transfer calculations often depend on flow regimes, with the value of other characteristic numbers, such as the Nusselt number depending on whether the flow is laminar, turbulent or in transition.

(Glasstone, 1994)

As well as important heat transfer applications, the Reynolds number is used to predict fluid flow around an aeroplane's wing or a helicopter's propeller allowing engineers to design more efficient and safer vehicles. It is used to scale comparable flows, for example, a scale model of a race car can be used in a wind tunnel to obtain different desired flow scenarios, highlighting the flow separation over the car which in turn causes drag, allowing engineers to predict areas that will cause drag and make design alterations before scaling to the full-size version, saving time and money.

To estimate the Reynolds number around the valve firstly the fluid velocity is calculated using Bernoulli's equation.

Velocity = 9.9m/s

The dynamic viscosity of petrol, gasoline = 0.008 Pa.s

(Viscosity Guide For Paints, Petroleum & Food Products, 2021)

Density of petroleum, gasoline = 750 kg/m³

(Fuels - Densities and Specific Volume, 21)

By inserting these figures into the Reynolds equation, we get:

$$Re = \frac{750 \times 9.9 \times 0.194}{0.008} = 180,056.$$

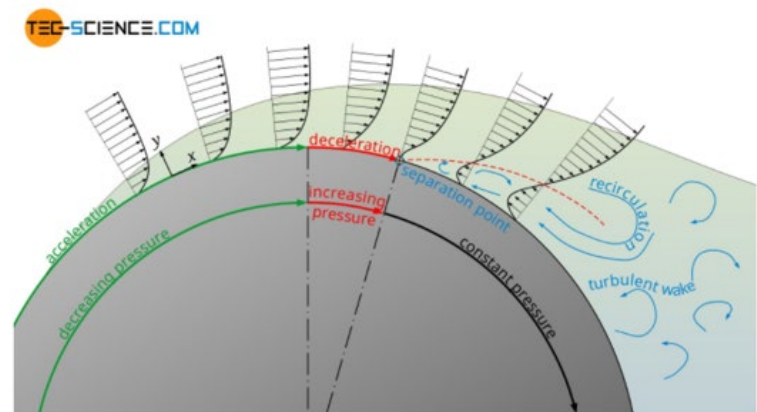
This shows the flow is turbulent.

PART 2

The boundary flow around the butterfly valve would differ at different degrees of opening. According to a paper by (Marimuthu & Chinnathambi, 2020) the angles of opening (AOP) have an important role in how the fluid behaves. For larger angles of opening the fluids velocity is distinctly lower than that of the smaller angles, indicating that the flow downstream will become less turbulent as the valve opens.

The boundary layer is a thin layer of fluid that is in contact with an object or surface that it is flowing over where velocity is zero. This is the area of the fluid flow where the viscous effects are significant. As the fluid moves over the front of the disc, the molecules in contact with its surface stick. (Schlichting Herrmann, 2000) Once opened, the fluid moves around the butterfly valve. The boundary layer on the surface of the valve is dependent on its angle of opening. When the valve is fully open at 90° there is minimal disturbance between the fluid layers, however due to the design of this specific valve with its two joining rivets on the disc face, and the protruding shaft to which the disc is joined on the back, the aerodynamic forces generated between the fluid and the valve would be greater than it would be on a more streamlined design.

When fully open, the fluid starts to move around the valve. The fluid moving over the face of the valve exhibits a lower velocity than the fluid flowing over the back side of the valve, this is due to the shape of the valve. The boundary layer on the front of the valve the flat profile until it encounters the rivets. The rivets protrude from the surface causing a sharp transition of fluid flow around them, at this point if the fluid is no longer able to follow the contour of the valve and begins to separate from the surface, this is commonly referred to as a boundary layer separation. At the separation point the fluid shears from the surface, and it is at this point where vortices form as recirculated fluid moves backwards between the surface and the separation streamline. This turbulent wake causes a loss in energy and turbulence within the fluid.



(Image: Flow separation (boundary layer separation) - tec-science, 2021)

When the valve is partially open, the effects of the rivets are decreased due to the angle the fluid flows over them. As the fluid contacts the rivets at a more favourable angle, the separation occurs closer to the following surface and there is less space for recirculation to occur. As the fluid flows through the smaller opening its velocity increases, which as we know is directly proportional to the Re value. The boundary layer remains in contact with the valve until it reaches its tip, and it is at this point the high velocity of the fluid being forced through these gaps shears from the surface. The jet of fluid rapidly spreads out, with the thickness of the mixing layer increasing as the fluid moves downstream. Similarly, to the separation caused by the rivets, the abrupt

transition leaves a separation streamline, and a large area for the reverse flow eddies to fill, causing drag and energy loss.

With the stem located behind the centreline, the contact between the disc and seat is minimised before full closure of the valve. Reducing the contact would imply that this valve has been designed with the goal of extending its service life by minimising the wear on the valve where it encounters the seat, or the seal between the valve and seat. By doing this there will be less wear on the edge of the valve and the seat/seal and therefore less chance of leakage, which is desirable when controlling fluid. Its design implies that manufacture of this valve would be relatively cheap and simple, with easily manufacturable components and a simple assembly. This, combined with the goal of extending its service life by minimising contact between its surface will mean the valve will be cost effective, providing cheaper and less frequent maintenance. This design however would be less effective in high pressure and temperature applications, where the selection of a triple offset valve would be more suitable. A triple offset valve would be more efficient and last longer than the single offset design seen in this essay, meaning that if it were to be used in these high temperatures and pressures then its initial cost effectiveness would be minimised do to more maintenance and repair.

PART 3

The butterfly valve is to be manufactured from glass fibre reinforced plastic (GFRP), which as a material is used to improve the corrosion resistance of properties of more traditional cast iron valves which are governed by BS3952:1965. However, BS EN ISO 16136:2006+A1:2019 dictates the requirements for butterfly valves made of thermoset plastic material such as GFRP must have a lifetime of 25 years which is not a requirement of more traditional metallic valves. (standardsdevelopment, 2021)

According to (Bai, 2013) the advantages offered by GFRP include a high strength to weight ratio and high corrosion resistance. GFRP is made from thermoset resins which offer the corrosion resistance properties infused with glass fibres wound or woven in certain angles or patterns to meet the strength requirements of the application. The polymer has good resistance to the compressive loading on the valve, and the glass fibres are good at resisting the tensile forces acting upon the valve. The corrosion resistance for GFRP coupled with the above-mentioned properties make for a good selection of material for this valve, however, (Schutte 1994; Ellyin and Maser 2004) state that high pressure, temperature, and moisture environments cause a degradation to the fibre matrix bond. This drawback however would seem not as important as first thought as the design of the valve implies use for a low temperature and pressure environment.

Traditionally corrosion of components in the Petro-chemical industry is a big problem. A study by the US Federal Highway Administration in 2002 showed that the cost of metallic corrosion in a wide range of industries was estimated in the US to be \$276 billion. (G.Koch, M. Brongers, N. Thompson, P. Virmani, J. Payer) Although this study is not directly addressing the corrosion of valves in use with petroleum, it does show a worrying trend. Historically metal valves were used until GFRP, and other thermoset resins were developed, and with the higher rate of corrosion within metal over these newer materials a switch to GFRP was needed to increase capital savings as the valves will last longer, decrease maintenance costs, and lower the risk of environmental accidents.

More recently, in a 2019 study carried out by (Marquéz MC) it was shown that the skin structure on the short fin mako shark reduces its skin friction drag by 8%. Inspired by this (Marimuthu & Chinnathambi, 2020) conducted a study on the effects of a hollow pattern biometric butterfly valve(HPBBV) compared to regular smooth surfaced valves and found that the HPBBV not only reduced skin friction and the turbulence kinetic energy, but also increased the net mass flow rate of the pipe.

Taking tips from nature is currently aiding in the development and efficiency in many different applications. This coupled with the increasing research of nano- reinforced thermoset plastics and composites based on renewable bio-resins indicates an exciting future for development of not only valves, but engineering in general.

PART 4

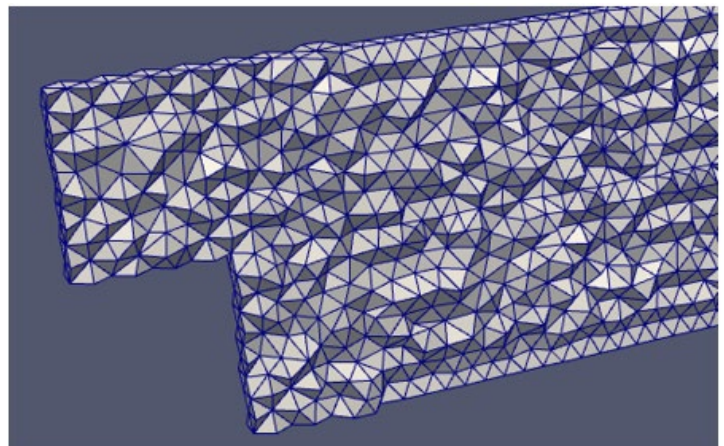
Computational Fluid Dynamics (CFD) is based on three main numerical approaches – the Finite-Element method (FEM), Finite-Difference method (FDM) and the Finite-Volume method (FVM).

Finite-Element Method

The finite element method (FEM) works by subdividing a CAD model into geometrically simple shapes. The collection of these small finite sized elements makes up the finite-element mesh.

The equations used if FEM depend on what physics is needed for modelling. When the scope of analysis is known, then a system of field equations that describe the physics is applied to each element. The fields within the elements are approximated as a simple function within the degrees of freedom of each. These equations describe the local physics around each element and when assembled into the larger matrix of the object can be solved by the CFD software to show an accurate representation of the system in question.

Different applications for FEM are tailored to use different physics, and therefore show unique structural imprints of the matrix.



Tetrahedral Mesh

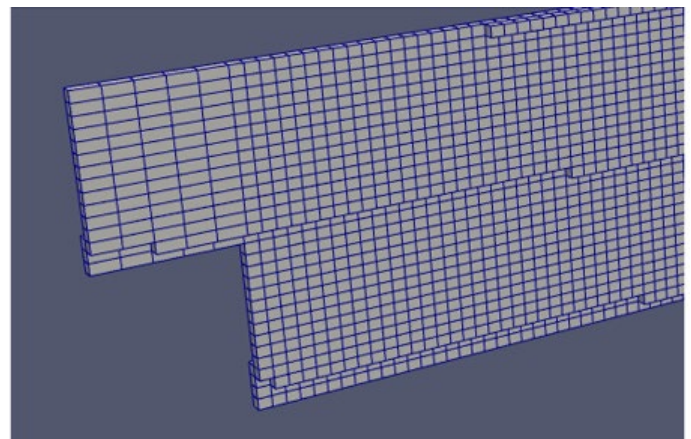
(Polyhedral, Tetrahedral, and Hexahedral Mesh Comparison | Symscape, 2021)

Finite-Difference Method

The most direct approach to discretizing partial differential equations is the finite-difference method (FDM). With an approximate difference formula, generally derived from a Taylor series expansion, replacing each derivative. By divided the computational area into hexahedral cells, FDM incorporates discrete equations in the place of continuum representations of the equations of the object to gain the solutions for each node added.

Finite-Volume Method

Like FEM, the Finite-Volume Method (FVM) divides a CAD model of an object into small, geometrically simple shapes of finite size. Firstly, FVM changes to concept of elements, instead referring to them as cells. This, along with utilising some physical laws



Hexahedral Mesh

(Polyhedral, Tetrahedral, and Hexahedral Mesh Comparison | Symscape, 2021)

known as conservation laws where what force is applied to one side of the cell will need to leave the other side of the cell. This idea lends to a formulation that is made of flux conservation equations, which are defined as an average over the cells (Manganih, 2016). Fluxes are important in FVM as the resulting equations require fluxes of the conserved variable, which only need to be calculated at the cell boundaries and not at every node. Due to the conservation laws involved, FVM can handle non-linear problems which cause discontinuities in the solutions. Simply, what goes in, must come out. FVM ability to solve fluid flow problems has been historically successful.

Method Selection

To select the best method to simulate the fluid flow around the butterfly valve we must consider the advantages of each. One of FEM's biggest advantages is that it can be used to combine different functions to approximate each elements solution. These mixed formulations allow for different functions such as heat transfer and structural analysis to be tightly combined within the matrix allowing for the combination of accurate solutions. These mixed formulations are relatively straightforward for FEM to model; however, they are very difficult or impossible with the other methods.

For time dependant simulations FD and FV methods have a clear advantage over FEMs. A previous study conducted by (Gohil T, McGregor RHP & Szczerba D) observed that the computing time for FVM was far lower than it was for FEM. Along with this fact, FEM can be more difficult to apply than the other two methods, this is due to having more complex mathematics to implement, compounding the time needed to run a simulation.

Various mesh types are used for accurate fluid flow, such as tetrahedron, hexahedron, prism, and polygon. FDM can only use a hexahedron mesh, this makes simulating basic geometry simple and efficient, however this type of mesh struggles with complex geometry, a problem that FEM and FVM doesn't have as it can utilise the other shape meshes. According to (Woowon Jeong a, 2013) FVM based software shows similar results whether tetrahedron or hexahedron meshes were used. This was not the same for FEMs, which would seem to have more variation when the mesh type is changed. As the cells do not need to be structured in FVM, complex geometry can be easy to model, with the ability to refine the mesh around areas of interest, in the same way as FEM does. However, once the approximations are made using FVM, the solutions cannot be easily made of a higher order, leaving it at a disadvantage to FEM.

The method to be selected to simulate the flow around the butterfly valve is the Finite-Volume Method. Whilst FEM has some practical advantages over other methods, the computational power and time needed is higher and increases with its complexity. FVMs faster computational time, coupled with its easy handling of complex geometry, simpler mathematics involved and historical success in solving fluid flow problems has resulted in it being the chosen method.

PART 5

To test the fluid flow around this valve a prototype is created to assess the accuracy of a CFD simulation.

Apparatus:

- Prototype valve
- Hydraulic bench
- Pitot tube
- Manometer
- Osborne Reynolds Apparatus
- Recording Device
- Clear Tubing

The prototype valve is to be fixed into position within the clear tubing and connected into a circuit containing the hydraulic bench. The hydraulic bench, equipped with a pump that can be calibrated so that the pressure of the fluid on the face of the valve matches the hydrostatic pressure of the petroleum that the final valve will experience.

An Osborne Reynolds Apparatus is set up for visual analysis. A tank containing a dye is connected to the upstream of the valve that when opened a constant flow of dye follows the fluids path around the valve.

A U-tube monometer containing mercury is to be fitted so that the first opening is before the valve in the upstream, and the second behind the valve in the downstream. This is to see the change in pressure of the fluid as the valve opens.

A Pitot Tube is to be fitted in the downstream where is the velocity of the fluid is likely to be at its highest. The position of the Pitot tube can be adjusted so once the reading is taken towards the top of the tube it can be moved to the middle and then the bottom, measuring the velocity of the fluid at different areas of the flow.

Once the experiment is set up, and the pump is set to match the hydrostatic pressure the valve can be opened at pre-determined angle. The Osborne Reynolds Apparatus releases the dye as the valve is opened, giving a visual indication of the flow type around the valve. Whist there are no readings to take with this visual test, a recording device is used to record the flow so that it can be compared to the simulation.

The opening of the valve and the subsequent change in pressure will be measured with a U-Tube monometer. By measuring the difference in height of the mercury contained in the monometer the pressure drop can be calculated and recorded. The equation used is:

$$P_1 - P_2 = \rho gh.$$

The Pitot tube is to be firstly placed at the top of the tube for the valve opening, this measures the difference between the static and total pressure, and once these values have been obtained, then a rearrangement of the below equation will result in a velocity of the fluid.

$$P_2 = P_1 + \frac{1}{2} \rho u_1^2$$

Once the velocity is measured and recorded the pitot tube can be adjusted towards the centre and then the bottom of the tube, recording the results of each.

Once all the measurements have been recorded, the experiment can be reset, and the valve opened to a different angle and repeated until the valve is fully open.

With all the measurements recorded for different degrees of opening, they are then plotted onto a graph, and compared to the results of the simulation. It is very important that the boundary conditions are the same for the experiment and the simulation. If the data correlates between both techniques, then the simulation results can be used. If the data does not match, then there is likely an issue with within the CFD program that will require adjusting.

Conclusion

The Reynolds number has been shown to be very important in fluid flow problem, with turbulence being unpredictable and hard to model. The number relating to the flow around the valve was estimated to be 180,056. This shows the flow to be turbulent, however there may be errors in the calculation due to the complexity of the hydraulic diameter. It has been shown that the angle of opening of the valve influences the flow of the fluid, with the downflow becoming less turbulent as the valve opens.

The benefits of GRFP have been shown to have good a good strength to weight ratio, and a high level of corrosion resistance, and will outlast more traditional metallic valves. The valve design implies use for low pressure and temperature applications which coincide wit the GFRP properties.

The Finite-Volume Method was selected to simulate the flow around the valve for its accuracy, speed of calculations and historical success with fluid flow problems. An experiment was set up to check the accuracy of the simulation, including what apparatus was needed and how the experiment should be conducted.

Bibliography

- Bai, J. (2013). Advanced fibre-reinforced polymer (FRP) composites for structural applications. Cambridge: Woodhead Publishing.
- Durst, F. (2008). Fluid mechanics. An introduction to the theory of fluid flows. Erlangen: Springer.
- El-Sherik, A. M. (2017). Trends in Oil and Gas Corrosion Research. Cambridge: Woodhead Publishing.
- Fuels - Densities and Specific Volume. (21, 7 15). Retrieved from Engineering Toolbox: https://www.engineeringtoolbox.com/fuels-densities-specific-volumes-d_166.html#:~:text=Sponsored%20Links%20Density%20-%20CF%81%20-and%20specific%20volume,%202014100%202017%20more%20rows%20
- Glasstone, S. (1994). Nuclear Reactor Engineering: Reactor Systems Engineering. 4th edition: Springer.
- Manganih, F. M. (2016). The Finite Volume Method. Beirut: Springer.
- Reynolds Number . (2021). Retrieved from Nuclear Power: <https://www.nuclear-power.com/nuclear-engineering/fluid-dynamics/reynolds-number/>
- Schlichting Herrmann, G. K. (2000). Boundary-Layer Theory. Berlin: Springer.
- standardsdevelopment. (2021). Retrieved from bsigroup.
- Viscosity Guide For Paints, Petroleum & Food Products. (2021, 7 15). Retrieved from REFINA: <http://pyrobin.com/files/Viscosity%20Chart.pdf>
- Woowon Jeong a, n. J. (2013). Comparison of effects on technical variances of computational fluid dynamics (CFD) software based on finite element and finite. International Journal of Mechanical Sciences, 24-26.
- Marquéz MC (2019a) Spilling the secret to a mako shark's speed. Forbes, 4 March. See <https://www.forbes.com/sites/melissacristinamarquez/2019/03/04/spilling-the-secret-to-a-makosharks-speed/> (accessed 04/06/2019).
- Abbas AA, Ghopa WAW, Mat S et al. (2018) Surface roughness effects on turbulent boundary layer structure of NACA 0026 Airfoil. International Journal of Engineering & Technology 7(3–17): 254–259.
- Schutte, C. L. (1994). "Environmental durability of glass fiber composites." Mater. Sci. Eng. R, 13(7), 265–323
- Ellyin, F., and Maser, R. (2004). "Environmental effects on the mechanical properties of glass-fiber epoxy composite tubular specimens." Compos. Sci. Technol., 64(12), 1863–1874.

G. Koch, M. Brongers, N. Thompson, P. Virmani, J. Payer, Corrosion Cost and Preventive Strategies in the United States, FHWA-RD-01-156, March 2002.

<https://standardsdevelopment.bsigroup.com/search?Term=butterfly+valve#1>

Gohil T, McGregor RHP, Szczerba D, Burckhardt K, Muralidhar K, Székely G. Simulation of oscillatory flow in an aortic bifurcation using FVM and FEM: a comparative study of implementation strategies. *Int J Numer Methods Fluids* 2010;66:1037–67.

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/#:~:text=%20Flow%20separation%20%28boundary%20layer%20separation%29%20%201,laminar%2C%20but%20also%20turbulent%20at%20increased...%20More%20>> [Accessed 21 July 2021].

Engineeringtoolbox.com. 2021. *Absolute or Dynamic Viscosity Online Converter*. [online] Available at: <https://www.engineeringtoolbox.com/dynamic-viscosity-d_571.html> [Accessed 22 July 2021].

Appendices

Chart

Below is a general list of commonly used industrial products and a guide to their viscosity nominally at room temperature; the viscosity is listed in Centipoise (cP)

Water has a viscosity of 1 cP at 20°

Product	Viscosity	Product	Viscosity
Acetate glue	1,200-1,400	Maple syrup	150-300
Alcohol, sopropyl	2	Mayonnaise	5,000-20,000
Alkyd resins	500-3,000	Methyl ethyl ketone	4
Anti-freeze	15	Milk	3
Beer	18	Milk condensed	40-80
Benzine	5	Milk evaporated	6,100
Blood	10	Milk condensed	6,000
Butter	30,000	Molasses	5,000-10,000
Butter cream, sour	550	Motor oil SAE 10	50
Butter fat	45	Motor oil SAE 20	125
Castor oil	500-1,000	Motor oil SAE 30	200
Caulking compound	5,000,000-10,000,000	Motor oil SAE 40	320
Caustic soda 50%	45	Motor oil SAE 50	540
Cheese, soft	30,000	Motor oil SAE 60	1,000
Chocolate syrup	2,250	Motor oil SAE 70	1,600
Cleaning emulsions	1,500	Oleic acid	40
Coconut oil	80-110	Olive oil	40
Com oil	50-100	Paint, water based	2,400
Corn syrup	15,000	Pancake batter	2,200
Cream, cheese	100,000	Paraffin	10
Cream, clotted	20,000	Paraffin emulsion	3,000
Cream, double	120	Peanut butter	150,000-250,000
Cream, single	50	Peanut oil	100
Cream, sour	15,200	Petrol, gasoline	8
Detergents	1,500	Petroleum jelly	64,000
Diesel	14	Plastisol	28,000
Diethylene	32	Polyester resin	3,000
Dipropyleneglycol	107	Polyglycerine caprinate	6,000-7,000
Epoxy resin	5,000-10,000	Polyisobutylene	12,500
Ethylene glycol	16	Polymer solution	20,000
Fuel oil no 4	12	Polyol, non-pigmented	500-5,000
Gelatine	1,200	Polypropylene	240,000
Glucose	4,300-6,800	Potassium hydroxide	67
Glue, gun grade	5,000-50,000	Propylene	52
Glue, pourable	3,000-5000	Putty	100,000,000
Glycerine	1,000-2,000	PVA resin	65,000
Glycol	20	Resin pourable	880-975
Hand cream	8,000	Resin solution	7,100
Hand lotion	5,000	Salad cream	1,300-2,600
Honey	1,800-3,000	Sesame oil	94
Hydraulic oil	30-120	Shampoo	3,000-5,000
Icing	10,000	Soap solution	82
Ink, printing, liquid	550-2,200	Spaghetti sauce	1,000
Ink, printers	20,000	Starch solution 25%	300
Jam	8,500	Sugar syrup, sucrose	1,210
Kerosene	10	Titanium dioxide	5,000
Lacquer, water-based	900	Toluol	6
Lacquers 25% pigments	3,000	Tomato juice	180
Lard	1,000,000	Tomato ketchup	1,000
Lard oil	3,800	Tomato puree	50,000
Latex emulsions	200	Tomato sauce	2,600
Latex neoprene	5,000	Toothpaste	70,000-100,000
Latex paint	750	Triethylene	40
Linseed oil	30-70	Turpentine	10
Liquid soap	85	Varnish	320
Liquid wax	500	Vegetable oil	40
Lubricating oil	60-200	Water	1
Machine oil, heavy	600	Wax, liquid	75
Machine oil, light	150	Whey	800-1,500
Malt extract	9,500	Yoghurt	3,000-8,000

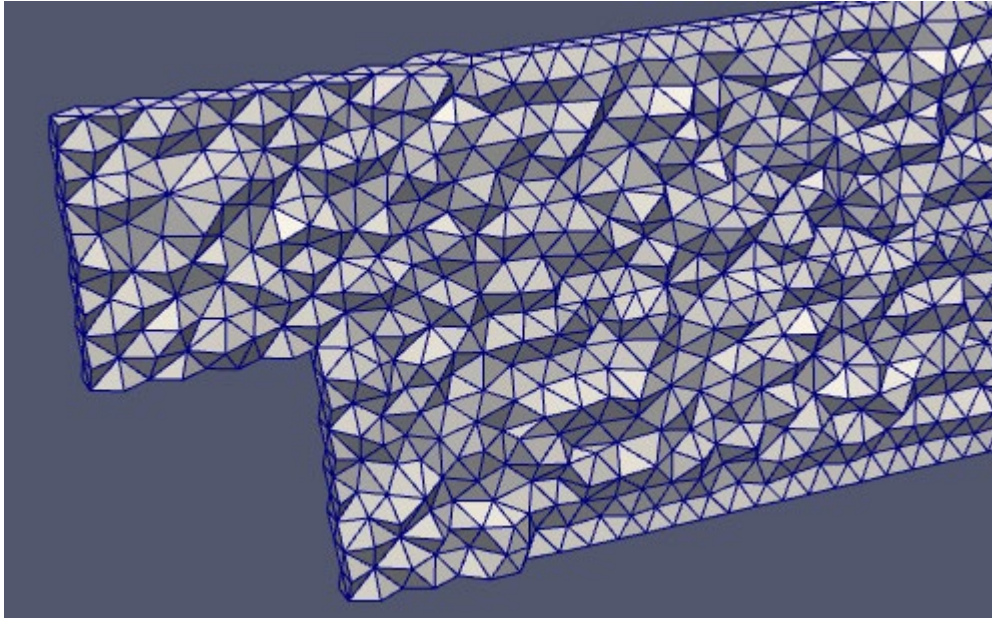
(Viscosity Guide For Paints, Petroleum & Food Products, 2021)

Convert from	Multiply by				
	Convert to				
	Poiseuille (Pa s)	Poise (dyne s/cm ²) (g/cm s)	centiPoise	kg/m h	kg _f s/m ²
Poiseuille (Pa s)	1	10	10 ³	3.63 10 ³	0.102
Poise (dyne s/cm ²) (g/cm s)	0.1	1	100	360	0.0102
centiPoise	0.001	0.01	1	3.6	0.00012
kg/m h	2.78 10 ⁻⁴	0.00278	0.278	1	2.83 10 ⁻⁵
kg _f s/m ²	9.81	98.1	9.81 10 ³	3.53 10 ⁴	1
lb _f s/inch ²	6.89 10 ³	6.89 10 ⁴	6.89 10 ⁶	2.48 10 ⁷	703
lb _f s/ft ²	47.9	479	4.79 10 ⁴	1.72 10 ⁵	0.0488
lb _f h/ft ²	1.72 10 ⁵	1.72 10 ⁶	1.72 10 ⁸	6.21 10 ⁸	1.76 10 ⁴
lb/ft s	1.49	14.9	1.49 10 ³	5.36 10 ³	0.152
lb/ft h	4.13 10 ⁻⁴	0.00413	0.413	1.49	4.22 10 ⁻⁵

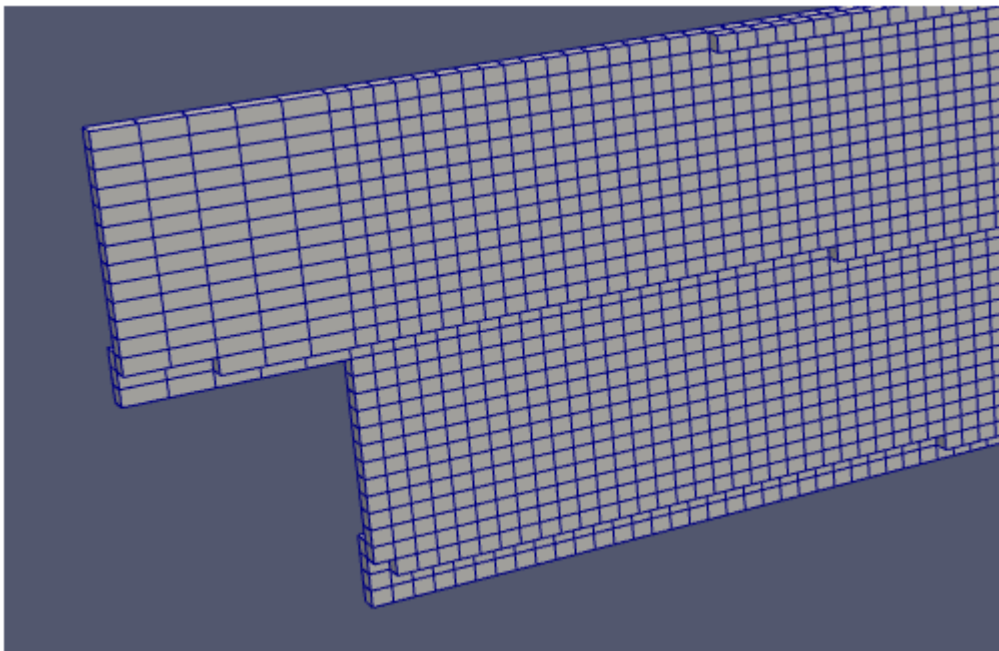
(Absolute or Dynamic Viscosity Online Converter, 2021)

Fuel	Density@15°C - ρ -		Specific Volume - v -	
	(kg/m ³)	(lb/ft ³)	(m ³ /1000 kg)	(ft ³ per ton)
Anthracite	720 - 850	45 - 53	1.2 - 1.4	42 - 50
Bituminous coal	690 - 800	43 - 50	1.2 - 1.5	45 - 52
Butane (gas)	2.5	0.16	400	14100
Charcoal, hard wood	149	9.3	6.7	240
Charcoal, soft wood	216	13.5	4.6	165
Coke	375 - 500	23.5 - 31	2.0 - 2.7	72 - 95
Diesel 1D ¹⁾	875	54.6	1.14	40.4
Diesel 2D ¹⁾	849	53	1.18	41.6
Diesel 4D ¹⁾	959	59.9	1.04	36.8
EN 590 Diesel ²⁾	820-845	51-53	1.18-1.22	42-43
Gas oil	825-900	51-56	1.1-1.2	36-43
Gasoline	715-780	45-49	1.3-1.4	45-49
Fuel Oil No.1 ³⁾	750-850	47-53	1.2-1.3	42-47
Fuel Oil No.2 ³⁾	810-940	51-59	1.1-1.2	38-44
Heavy fuel oil	800-1010	50-63	1.0-1.3	35-44
Kerosene	775-840	48-52	1.2-1.3	42-46
Natural gas (gas)	0.7 - 0.9	0.04-0.06	1110-1430	39200-50400
Peat	310 - 400	19.5 - 25	2.5 - 3.2	90 - 115
Propane (gas)	1.7	0.11	590	20800
Wood	360 - 385	22.5 - 24	2.5 - 2.8	90 - 100

(Fuels - Densities and Specific Volume, 21)



Tetrahedral Mesh



Hexahedral Mesh

(Polyhedral, Tetrahedral, and Hexahedral Mesh Comparison | Symscape, 2021)