Omar Wannous

Pressure Loss Optimization of a Sprinkler Valve

Helsinki Metropolia University of Applied Sciences

Bachelor of Engineering

Mechanical Engineering

Bachelor's Thesis

29 May 2016

Author(s) Title Number of Pages Date	Omar Wannous Pressure Loss Optimization of a Sprinkler Valve 85 pages + 6 appendices 29 May 2016
Degree	Bachelor of Engineering
Degree Programme	Mechanical Engineering
Specialisation option	Energy and Environmental Engineering
Instructor(s)	Jarmo Helasuo, M.Sc. Engineering Manager Tobias Immler, M.Sc. Product Development Engineer Heikki Paavilainen, Senior Lecturer

The aim of this Bachelor's thesis was to optimize the pressure loss for an already built sprinkler valve for a high pressure water mist system. The purpose of optimizing the pressure loss through the sprinkler valve is to gain better functionality characteristics for the high pressure water mist system in both land and marine based applications.

In order to, obtain a good understanding of the high pressure water mist technology and why it is preferred especially in marine applications, the conventional sprinkler system is introduced and discussed briefly in the beginning. Also, the regulations and standards as well as the classification societies are introduced for the high pressure water mist system. It is important to develop a good understanding of fluid mechanics since it is widely used in the design of modern engineering systems. Therefore, the fluid mechanics for internal flow and the concepts regarding the pressure loss effects are discussed and explained comprehensively.

The research started by conducting an extensive feasibility study for a sprinkler valve, in order to allocate the problem as well as to compare the results with the results obtained from the development work. The experimental results unveiled the deviation of the theoretical based results, and this was due to a simplified approach and hypotheses made in theory.

The feasibility study resulted in several potential design concepts for the sprinkler valve, which were simulated by utilizing the Computational Fluid Dynamics (CFD) approach. Based on the obtained results from the simulation, one concept design was upgraded and field tested in the sprinkler valve.

In conclusion, the scope of the project was achieved by the final concept design. The development work concluded in high optimization level, which will in the future bring several enhanced functionality properties for the water mist system.

Keywords	Fluid mechanics, internal flow, pressure loss optimization, sprinkler system, computational fluid dynamics

Metropolia

Contents

Acknowledgements

Firstly, I would like to show my deepest gratitude for my parents who have supported me throughout my studies and stood beside me every single moment. I also want to thank my brothers for the immense support as well.

I would like to thank my project supervisor Jarmo Helasuo for giving me such an interesting and challenging subject as well as for the precious advises throughout my thesis work. I would like to thank especially Tobias Immler for the help and support as well as for the valuable tips and advices and the wonderful experience. I also would like to show my gratitude for the help and advices I received from the rest of the staff.

Finally, I would like to thank my work supervisor, senior lecturer Heikki Paavilainen, for the advices and comments for my thesis work.

Helsinki 29.05.2016

Omar Wannous

1 Introduction

Fluid mechanics has been an important topic for centuries for scientists since fluid transportation plays a major part in various engineering applications. Archimedes was the first to study fluid fundamentals by postulating the parallelogram law for addition of vectors and the laws of buoyancy and applied them to floating and submerging objects. There are also many other scientists who has contributed to the development of the fluid flow engineering from which we benefit today. There are numerous applications that comfort our daily life interactions which have developed by time into commonly used inventions. The presence of fluid technology can be noticed in all kinds of transportation systems, waste transport and treatment, power plant systems, pipelines, crude oil and gas transfer, designing tall buildings and bridges. All technologies where fluid dynamics is used cannot be all listed since there are so many different applications.

The basic concept of fluid is transporting the substance from place to another by exploiting its kinetic energy and transferring it to a form of mechanical energy.

Fluid flow studies are governed by two completely different substances, liquid and gas. The study of liquid flow is called hydrodynamics. Liquid includes different kinds of substances such as oil and water chemical solutions. In most fluid flow applications water is by far the most common liquid and respectively, most applications are involved in hydrodynamic fluid control.

Studies of fluid are classified into two research regimes, external flow and internal flow. External fluid flow studies concentrate on the behaviour of the flow and its profile on external surfaces e.g. the wings of an airplane. Respectively, internal fluid flow studies focus on the flow’s behaviour inside the control volume, for example in tubing systems.

Pressure loss, when transporting liquids in pipes, is indicated due to friction affect near the wall of the pipe because of the no slip condition. In practice, the study of this physical phenomenon is divided into two approaches, analytical and experimental, after which prototypes based on the obtained results are constructed and tested in laboratories. The analytical approach is practised in two different ways, numerical or computational. The numerical method is to apply mathematical and physical laws and equations based on simplified hypotheses in which, it is more convenient to classify them on the basis of fluid common characteristics to make the laws more feasible to study numerically. The computational analysis is implemented more in practise rather than the numerical approach for studies because of the technicality and accuracy of the method. Numerical solutions in highly demanding applications bring uncertainty to the results depending on the hypotheses and approximations made. Therefore, computational fluid dynamics, better known as CFD, is most likely to be applied to reduce uncertainties and to simulate the fluid flow in the application. The numerical methods are inexpensive and fast, whereas the experimental method is more expensive and time consuming and sometimes it can be justified as impractical.

1.1 Research Problem

Marioff Corporation Oy high pressure tubing system is applied in various application fields one of which is marine. It produces different kinds of pump units which come with different sizes of electrical motors. Permissible pressure loss in the system is determined by the pressure of the pump unit and the water amount that pump unit has to deliver to cover the most demanding section in the vessel. Essential component of the high pressure tubing system is a sprinkler. Each sprinkler type has a validated operating pressure which it needs in order to operate in designed features, therefore, pressure losses over the tubes has to be limited for achieving the validated operating pressure of a sprinkler. Thus, pressure loss over every component in the tubing system has to be controlled and especially valves which develops pressure losses the most, relatively.

Pressure loss over the section valve is considerable in high volume flow rates. In some cases, the pressure losses in the tubing system could exceed the permissible value which in return requires enhancing the pump unit or installing bigger tubes. Consequently, this might be a deal breaker for the company.

The research problem of this thesis is to reduce the pressure loss over the section valve by modifying design features inside the valve in order to allow more accessible path for the water to flow through. The water flow is assumed to go through an insufficient area between the piston and the body which consequently develops a huge pressure drop in that certain area.

1.2 Scope of Thesis

Introduction for fluid mechanics is to be discussed comprehensively e.g. Bernoulli equation, common characteristics of laminar flow and turbulent flow, viscosity affect and Darcy-Colebrook equation etc. HI-FOG high pressure water mist system will be introduced generally as well as the operational principle of the HI-FOG system and its applications. Also, the classification societies that governs on both land and marine fields as well as the regulations and standards set by them are introduced. A comprehensive overview of the operational principle of the sprinkler section valve is discussed as well as the interface design with water mist system.

In order to optimize the pressure loss over the sprinkler valve, a research study is conducted for the current configuration design. Research will be carried out comprehensively by using modern technical programs and well equipped test facilities. The results obtained from research will be analysed and product development will be carried out based on that. The aim is to optimize the pressure loss over the sprinkler valve, which will in return improve to optimize the pressure level in the system.

The aim is to develop several potential solutions for the sprinkler valve. All the design concepts will be studied and simulated, after which only the viable solutions will be upgraded and manufactured for final testing. In the end, recommendations for future work will be noted.

1.3 Limit of Research

In terms of the theory part, the HI-FOG high pressure water mist system will be discussed on the general level without taking a stand of conventional sprinkler system neither competitive solutions on the market. The basic concept of fluid mechanics will be discussed only for internal flow inside a circular pipe leaving out open flow studies and the characteristics of it. Classification societies will be introduced generally without bringing up any classification centre in particular, given that referential context might be noted from individual classification centre.

Due to installation purposes, any considerable redesigning to the valve body is not permissible. Therefore, the research will only focus on the cartridge inside the sprinkler valve. Primarily, the concentration in the development work will be either on redesigning the piston or replacing the spring for much softer spring which consequently enables the water to flow over a bigger area. The research will not include any other components of the HI-FOG system nor there will be cost analysis of the system.

2 Introduction to Water Mist

The HI-FOG water mist and fire suppression system was established in the early 1990s. At present, the high pressure water mist fire suppression system competes with other fire fighting systems such as conventional sprinkler and gas suppression systems.

The high pressure water mist fire suppression technology was initially developed after Halon, which is a general name to a hydrocarbon in which one or more hydrogen atoms have been replaced by atoms from the halogen series as like bromine or chlorine, were forbidden to be used in solid fire extinguishing systems due to their harmful impact on the environment. A solid fire extinguishing system is a system that cannot be replaced after installation /1/. In the early 1990s, IMO (International Maritime Organization) ordered that new passenger vessels carrying more than 35 passengers should be equipped with a sprinkler fire-fighting system or an equivalent corresponding system, and it was also decided that by the year 2005 the hazardous fire extinguishing systems of old vessels had to be upgraded to meet with the new regulations. /2/

The regulations set by IMO were considered very harsh because a conventional sprinkler fire extinguishing system demands very large pipes and thus more water. Consequently, this upgrade brought more weight and unstableness to vessels that had to improve their system. Thus, this point was considered a beginning for the high pressure water mist fire suppression technology. One of the advantages that the high pressure water mist suppression system brings is smaller pipes and lightness as well as smaller water consumption in comparison with conventional sprinkler systems. The main principle of the high pressure suppression system is that it sprays very small water droplets, i.e. fog, unlike conventional sprinkler with big water droplets. In addition, the high pressure water mist suppression system has a better extinguishing efficiency than conventional sprinkler systems, which has been reached by downsizing the water droplet size. /4/

A high pressure water mist system is defined by the water droplet size. The National Fire Protection Association (NFPA 750) states that the water droplet size has to be less than 1000 microns at the minimum operating pressure of the sprinkler in a high pressure water mist system. System pressure in some high pressure water mist systems can rise nearly up to 200 bar. In the high pressure water mist systems, the water mist coming out from an automatic activation sprinkler is achieved by high pressure water flowing through multiple nozzle orifices. The out coming mist obtains high momentum and it is extremely fine. /4/

2.1 HI-FOG Water Mist and Fire Protection System

Marioff water mist system was established at a time when the maritime industry was looking for a replacement to a conventional sprinkler fire extinguishing system. The founder of Marioff Corporation, Göran Sundholm, responded to the needs of the industry and introduced the high pressure water mist system which is known under the HI-FOG brand. HI-FOG brand was launched in the early 1990s. /3/

Figure 1 illustrates the HI-FOG water mist and fire protection system.

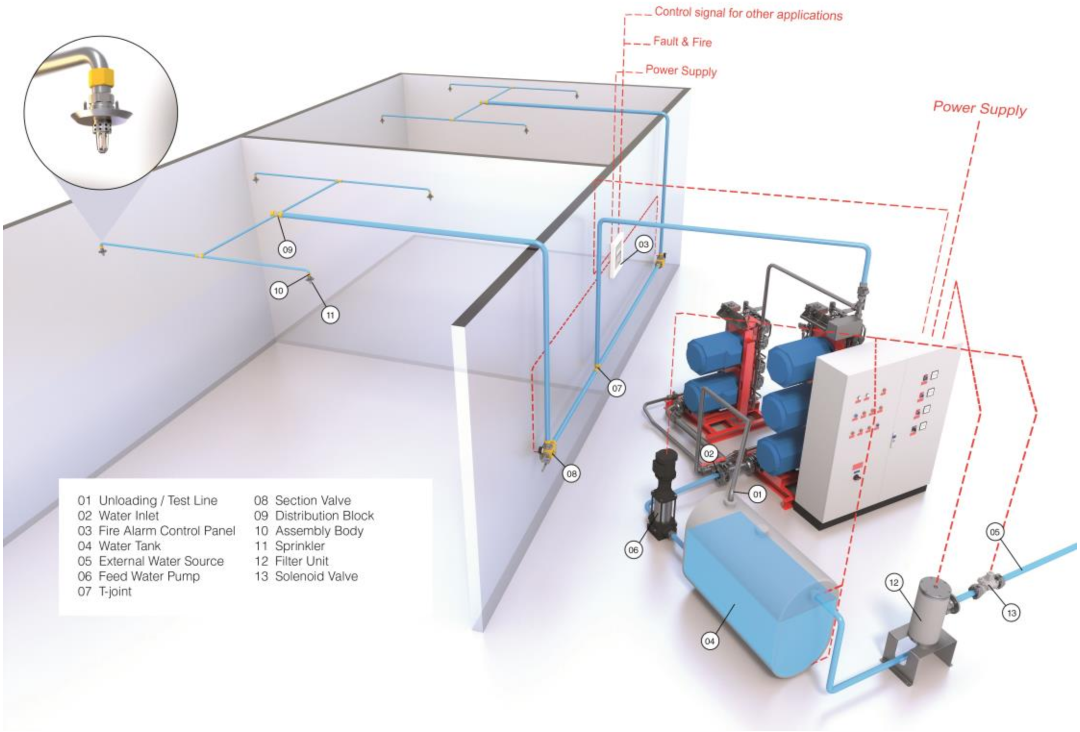

Figure 1. Marioff high pressure water mist and fire protection system. A wet pipe installation system.

The main components of Marioff high pressure water mist system are water supply, pump unit, section valve and a sprinkler. The water is pumped from a water tank by the pump unit into the high pressure pipes. The sprinklers are mounted behind every section valve. A section valve keeps the other section apart from each other, thus every fire protected section in a marine based application or a land based application has its own section valve. The sprinklers are mounted in the ceiling behind the section valve. Sprinklers, section valves and pump units vary between marine application and land applications. IMO has regulated the sections in marine application in regards to each sections fire risk. Therefore, in marine applications components, for example, sprinklers and section valves are designed to comply with the regulations set by IMO. The following chapters will introduce the main characteristics of different nozzle types of Marioff and two main installation systems.

2.1.1 Range of Nozzle Types

There are two types of sprinklers that are applied in a sprinkler system: automatic activation sprinklers and spray heads. /4/

An automatic activation sprinkler has a spraying head in which an activation element is integrated. The activation element will discharge the sprinkler when it reaches a certain temperature degree. The element can be, for example, an ampoule in which is a liquid and a small air bubble. The ampoule explodes when it reaches the designed temperature, which consequently discharge the sprinkler. There are various range of ampoules with a variable explosion temperature. The breaking temperature of an ampoule is chosen on the basis of the demand of the protected application. /4/

In figure 2, two Marioff automatic activation sprinklers from different product range are displayed.

Figure 2. Marioff automatic activation high pressure sprinklers. On the right is 2000-series sprinkler type and on the left is 1000-series sprinkler type.

A spray head is similar to an automatic activation sprinkler but without a discharge element in it. Figure 3 represents a model of spray head.

Figure 3. A model of Marioff many spray head types.

The amount of nozzles as well as the angle of the cone at the end of a spray head can differ from a protected application to another.

2.2 Installation Systems

Sprinkler installation systems can be classified to four main installation groups from which two mainly used are introduced in this chapter: wet pipe system and dry pipe system. Other main installation types are pre-action systems and section release systems. In addition of the main installation systems, there are various installation methods that are in the end more or less manufacturers own designs. /4, 5/

Therefore, two main Marioff installation systems are introduced. Due to undesirable information to be posted, installation systems are discussed in a general level.

2.2.1 Wet Pipe Installation System

In wet pipe installation, the pipe is filled in water up till the automatic activation sprinkler in standby state, as shown in figure 1. The standby pressure depends upon the system pressure class and the manufacturer. In case of fire, the generated heat breaks the ampoule of the automatic activation sprinkler which, consequently discharges the sprinkler. The water inside the pipe begins to flow through the sprinkler nozzles. Behind each section valve can be numerous amounts of automatic activation sprinklers, nevertheless the water flows through the discharged sprinklers and not necessarily from all the mounted sprinklers. In wet pipe installation, there is a section valve which is designed for the noted installation system. The section valve is normally open (NO) in standby state and it is closed for maintenance purposes or stopping the water flow e.g. after a fire. /4, 5/

Figure 4 illustrates a model of Marioff wet pipe system section valve.

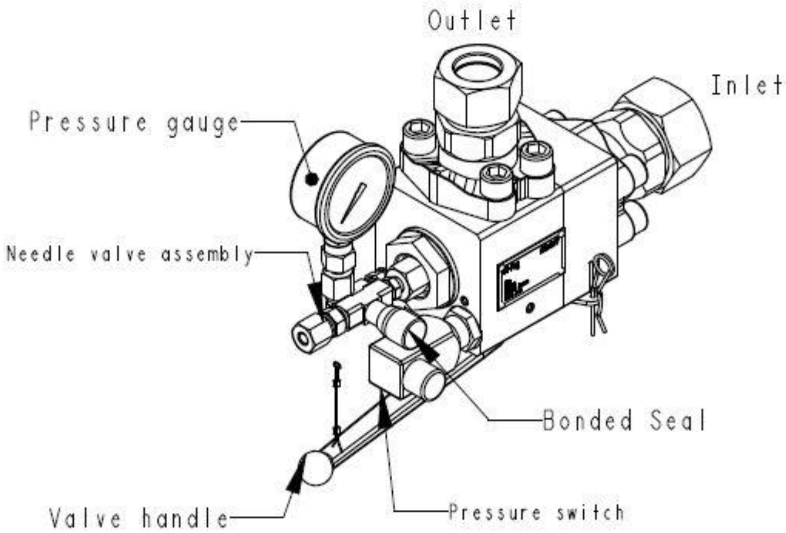

Figure 4. A Marioff wet pipe system section valve

In case of fire, a pressure difference switch will indicate a pressure drop due to water flow, which will send a signal to a local control panel. The control panel will set off an indication of the section valve in the control panel, in order to notify a pressure drop in that section. /4/

The wet pipe installation is the most utilized installation system. It has become popular due to its simplicity and reliability as well. Because it is a simple technology, it is the most cost saving installation system in terms of the purchase cost, installation cost and service cost as well. The wet pipe installation system is faster to extinguish a fire compared to other installation systems, that is because the pipe is filled with water all the way up till the automatic activation sprinkler. /4, 6/

The disadvantage of a wet pipe installation system is that they are not suitable as such to sub-freezing environments. It is not applicable for applications which ambient temperature may rise over 95 degrees Celsius as well. The system may leak if the piping parts are objected to a severe damage during handling or installation. /4, 6, 7/

Wet pipe installation system is more applicable for a protected application which requires an immediate start up. In general, suitable applications are therefore buildings, hotels, factories, warehouses, office facilities and common spaces in vessels. Respectively, it is not the most preferable solution for museums, libraries and facilities of electronics industry. /4/

2.2.2 Dry Pipe Installation System

Dry pipe sprinkler installation system uses automatic activation sprinklers and the pipe is surrounded by air or inert gas, compare to wet pipe installation system. Figure 5 shows the schematic of a dry pipe system.

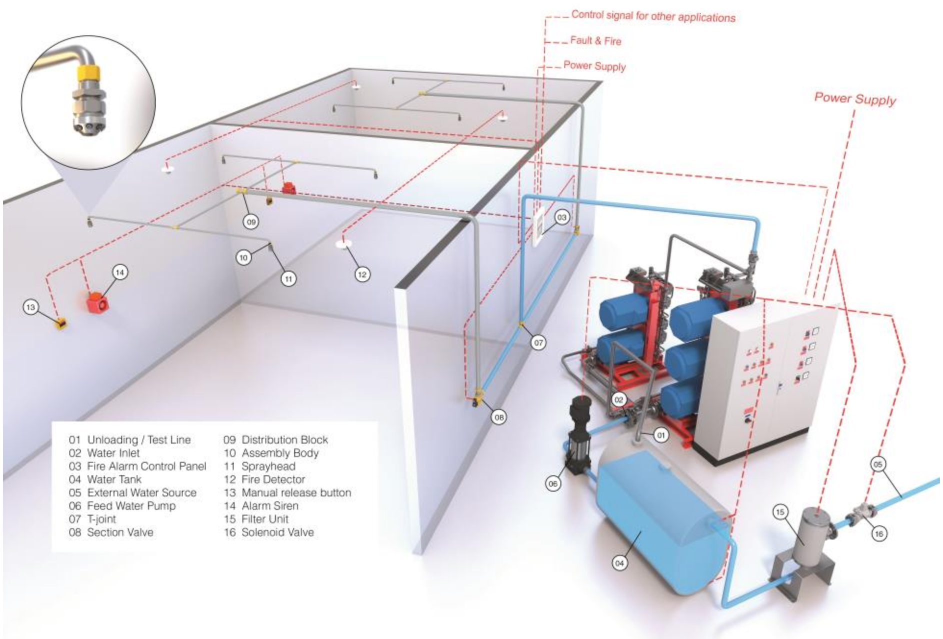

Figure 5. Machinery space system.

As shown in figure 5, the skeletal pipelines are filled with water in standby pressure. The water in the skeletal pipelines is separated from the section pipelines by a dry pipe system section valve. The section valve is remained closed by the pressurized air or the inert gas in the section pipeline, and it will open when the pressure in the dry pipe section decreases to a certain pressure level. When the dry pipe section valve reaches an opening pressure due to an activated sprinkler, the water begins to flow excessively into the dry pipeline and it will come out from the discharged sprinkler. It is necessary for the section valve to have some kind of detector attached to it, in order to indicate the release of the valve and, thus locating the fire. /4, 8, 9/

The dry pipe installation system has to have an inert gas or a pressurized air source for maintaining the pressure inside the dry pipe section. Usually, the system manufacturer recommends the appropriate maintainable pressure level in the dry pipe section. The pressure level is designed to sustain a minor leakage, thus the dry pipe section valve does not open due to minor leakage. The source has to be dimensioned in such way, in order for the system to activate in 60–90 second according to the industry standards. As the activation is typically detected from a decreased gas pressure level, the activation time depends on the volume of the dry pipe section and the k-factors of the sprinklers. /4, 9, 10, 11, 12/

The great advantage of the dry pipe installation system is that the system is applicable in applications of sub-freezing environments. The system is also applicable in sites above 95 degrees Celsius. There are also some restrictions for the rest of the system. The dry pipe section valve has to be installed in ambient temperature between 0–95 degrees Celsius. The dry pipe system is usually installed in cold stores, attics, loading docks, garages and freezers. It is often presumed that dry pipe installation system brings additional protection against accidental discharges, and thus they would be applicable for delicate protection sites e.g. museums or libraries. This is a misinterpretation, because the water in the dry pipe system will start flowing through the sprinkler nozzles within a minute after the activation of the sprinkler. /4, 10, 11/

There is also disadvantages in dry pipe system that have to be taking into account when choosing this particular installation type. The dry pipe installation system is more complicated in comparison with the wet pipe installation system. Therefore, the dry pipe installation system is more expensive, installation and service expenses are costly as well. As noted earlier, systems activation time is longer (60–90 seconds) than the wet pipe installation system. Any improvements for the system after the installation are restricted. The dry pipe section has to be well dried off from water droplets in pursuance of installation in order to avoid corrosion in pipes /4/. As a conclusion, the dry pipe system does not bring a clear advantage on the wet pipe system if the protected room is within the appropriate temperature range of 5–95 degrees Celsius.

2.3 Classification Societies

Maritime safety is constantly being improved to prevent accidents and pollutions by ships. Regulations regarding maritime are made by International Maritime Organization (IMO) which is the United Nations (UN) specialized agency with responsibility for the safety and security of shipping. Every vessel that has been built is registered under a classification society. Classification societies are private agencies that establish and update technical standards for construction and operation of ships as well as safety standards. /13/

When a high pressure water mist system is designed, it is bear in mind that the system has to pass a full scale fire test. Therefore, the entire system with its components has to prove their undisputable performance. Marioff has conducted thousands of full scale fire tests in order to prove the system efficiency in various applications and it has the most class approvals in the entire water mist industry /4, 14/. In addition of fire tests, most classification centres conduct separate tests for the system components. An American insurance company FM (Factory Mutual) conducts tests for every critical component in the system individually. Testing for a classification approval is a time consuming process and expensive as well, and this is the reason of slow growing appearance in the market for water mist systems. /4, 15/

Important operational parameters of a water mist system are distance between sprinkler heads, operational pressure and installation height of the sprinkler. These parameters are determined by the fire tests. The parameters which were agreed upon on and the system has passed on, are not to be modified afterwards. The system cannot be installed in a space with greater fire risk than the conditions it was tested in. The fire tests reflect the worst case scenario of what could be a potential forthcoming fire in the application. Other tests beside class society tests are based on standards. The system is built according to standards references, for instance, distance between sprinkler heads, installation height, sprinklers operational pressure etc. However, the latter fire test method includes certain risks e.g. does customer's ordered system comply with a conducted test that is based on a certain standard. /4, 15/

With standards, a neutral comparison is obtained between the authorities and the classification societies. Water mist standards are done by several authorities, including IMO, NFPA (National Fire Protection Association), ISO (International Organization of Standardization), UL (Underwriters Laboratories), VdS (Verband der Schadenversichen) and CEN (European Committee for Standardization). In addition, every country based local research centres gives their recommendations regarding installation and operational method. In Finland, VTT (Valtion Teknillinen Tutkimuskeskus) is the research centre. /4/

As earlier mentioned, IMO does set regulations regarding maritime safety and security, however, it does not conduct fire tests neither gives approvals for the system. IMO is the leader in developing standards for water mist systems in vessels. SOLAS (Safety of Life At Sea) has listed IMO's regulations and they have become SOLAS agreements and resolutions. Installation guidance and regulations are given in SOLAS and FSS Code (International Code for Fire Safety Systems) for equivalent sprinkler systems. The operational principal of an equivalent sprinkler system has to comply with the resolutions stated on SOLAS and this is a precondition for using the system. IMO has developed numerous fire test methods for an equivalent sprinkler system. It has determined comparison fire test methods for different fire risk sections in a vessel. In case of the equivalent sprinkler system passes the preconditioned tests, it is then allowed to replace the conventional sprinkler system. /4, 16, 17, 18/

3 Common Engineering Review for Fluid Mechanics

Fluid mechanics deals with liquids and gases in motion and at rest. The fluid flow studies are classified as being internal or external, depending on whether the fluid flows in a conduit or in an open space over a surface. In the beginning of a study, the system is defined. The region outside the system is called surroundings and it is left out from the studies as the interest concentrates upon the system. The surface of the imaginary line which separates the system from its surroundings is called the boundary. The boundary of the system can be fixed or movable. The systems can be considered to be open systems or closed systems. The classification is determined by the nature of the study, whether the chosen system is a fixed mass or a volume in space. A closed system has a fixed mass which does not fluctuate, in other words, the mass is constant throughout the system boundary. But energy, in the form of heat and work, can cross the boundary. In addition, the volume of the closed system is not necessarily fixed. An open system, or a control volume as it is often denoted for, is a selected region in space. A control volume retains a component or a device that involves a mass flow such as valves or nozzles. The control volume can be fixed or it may involve a moving boundary. Mass interactions as well as energy interactions may be involved. /19/

The terms steady and unsteady are frequently used in engineering. In the calculations both terms imply different things. The term steady means that there is no change of properties, velocity, temperature, etc., at a point with time. Respectively, the term unsteady is the opposite of steady. /19, 20/

A flow is classified as being compressible or incompressible, depending on the level of variation of density during flow. If the density remains nearly constant throughout the control volume, the flow is classified as incompressible. In general, the densities of liquids are essentially constant all the time, and thus the flow of liquids is typically incompressible. Therefore, liquids are referred to as incompressible substances. Compressible substances are gases, for example, a small pressure change causes a change in the density of atmospheric air. /19, 20/

3.1 Viscous and Inviscid Regions of Flow

A fluid viscosity value informs the magnitude of a fluids internal resistance to the flow. For instance, when two layers move relative to each other a friction force develops between them. The friction force develops at the contact surface and the direction of the vector is opposite to motion. All fluids consist of internal resistance effects, however, there are regions where the viscous effects are negligible, and they are called inviscid flow regions. For example, a uniform fluid streams over a flat plate normal to the fluid flow. The flat plate will separate the fluid as it hit the plate. The viscous affects are significant near the plate and insignificant on both sides away from the plate. The viscous and inviscid flow regions are shown in figure 6. /19/

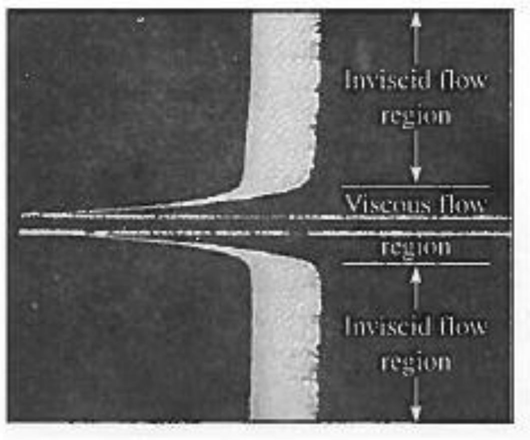

Figure 6. The regions of viscous and inviscid flow over a flat plate normal to the fluid stream. /19/

The velocity of a fluid stream near a stationary surface is zero, and this is because of the no-slip condition. When a uniform fluid approaches a surface, the fluid motion comes to a complete stop at the surface because the layer which is in direct contact with the surface sticks to it and, hence, the velocity at the surface is zero. The velocity profile, consequently, on the surface is not uniform anymore. Therefore, the flow region adjacent to the wall in which the viscous effects are significant is called the boundary layer. /19/

3.2 Overview to Cavitation

Temperature and pressure are fluctuating properties for pure substances during phase-change processes. The saturation temperature, also denoted as T_{sat}, is the temperature at which the pure substance begins to go under phase changes at given pressure. Likewise, saturation pressure, also denoted as P_{sat}, is the pressure at which the pure substance starts to go under phase changes at given temperature. The vapour pressure, usually denoted as P_v, is defined as “the pressure exerted by its vapour in phase equilibrium with its liquid at a given temperature” /19/.

The vapour pressure is an exclusive property for a pure substance and it is an equivalent to the saturation pressure of the liquid ($P_v = P_{sat}$). In liquid-flow systems, the pressure of a liquid may drop below vapour pressure at some locations and result in undesired vaporization. The vapour bubbles explode when they are swept away from the low pressure regions, generating high pressure waves. This destructive explosion of the vapour bubbles causes a drop in performance etc. This phenomenon is called cavitation and it is commonly encountered in applications where impellers and pumps are utilized. /19/

3.3 Mass and Volume Flow Rates

The conservation of mass principle for a control volume is simply, the net mass transfer to or from a control volume during a time interval Δt /19, 20/, and this expressed by:

$$\dot{m}_{in} - \dot{m}_{out} = \dot{m}_{CV} \qquad \text{(kg/s)}$$

Where $\dot{m}_{in}$and $\dot{m}_{out}$ are the total rates of mass flow into and out of the control volume, and $\dot{m}_{CV}$ is the rate of change within the boundaries of the control volume. During in steady flow process the total amount of mass contained in the control volume does not change with time. That is, the amount entering the control volume per unit time is equal to the amount exiting the control volume per unit time /19, 10/. Thus, the conservation of mass principle for a steady-flow system with multiple inlets and outlets is expressed by:

$$\sum_{in} \dot{m} = \sum_{out} \dot{m} \qquad \text{(kg/s)}$$

The amount of mass flowing through a cross-section per unit time is called mass flow rate, and it is denoted by $\dot{m}$. The dot over a symbol indicates a time rate of change. For incompressible flow or even for compressible flow where the density is approximated to be uniform across the cross-section are, the equation for mass flow rate becomes: /19, 20/

$$\dot{m} = \rho V_{avg} A_c \qquad \text{(kg/s)}$$

Where ρ is the density of the substance [kg/m^3], V_{avg} is the average velocity across the cross-section area [m/s] and A_c is the area of the cross-section [m^2]. The area of the cross-section for pipes and ducts is expressed by:

$$A_c = \frac{\pi d^2}{4} \qquad (1)$$

The conservation of mass principal can be also expressed by volume flow rate for steady incompressible flow:

$$\sum_{in} \dot{V} = \sum_{out} \dot{V} \qquad \text{(m}^3\text{/s)}$$

The volume flow rate is expressed by:

$$\dot{V} = V_{avg} A_c \qquad \text{(m}^3\text{/s)} \qquad (2)$$

Where V_{avg} is the average velocity across the cross-section area, and A_c is the area of the cross-section.

3.4 The Bernoulli Equation and its Limitations

The Bernoulli equation gives an approximate relation between pressure, velocity and elevation. The equation is applicable to steady and incompressible flow. The most commonly used Bernoulli equation in fluid mechanics for steady, incompressible flow along any two points on the same streamline in inviscid regions of flow /19, 21/, is expressed by:

$$\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2 \qquad (3)$$

Where P_1 and P_2 are the static pressure at the inlet and the outlet of the control volume [Pa], V_1 and V_2 are the velocity at the inlet and the outlet of the control volume [m/s], g is the acceleration of gravity [m/s^2], z_1 is the reference elevation level [m], and z_2 is the elevation relative to the reference level [m].

The mechanical energy is the form of energy that can be converted to a mechanical work completely by a mechanical device. The fluid systems are designed to transport fluid from one location to another at specified flow rate, velocity and elevation. The system may generate mechanical work in a turbine or it may consume mechanical work in a pump. A turbine extracts mechanical energy from a fluid by dropping the pressure of the fluid. Respectively, a pump transfers mechanical energy to a flowing fluid by raising the pressure of the fluid. Flow energy expresses the amount of energy in a flowing fluid. /19, 21/

Therefore, it is convenient to express the mechanical energy of a flowing fluid by fluid energy. Hence, the Bernoulli equation can be expressed as: P/ρ is the flow energy, $V^2/2$ is the kinetic energy, and gz is the potential energy of the fluid. /19, 20, 21/

In order to make Bernoulli equation clear and visible in terms of the flow energy, each term in the equation can be multiplied by the density ρ and it will yield to /19/:

$$P + \rho\frac{V^2}{2} + \rho gz = constant\ (along\ the\ streamline) \qquad (4)$$

Now, each term has pressure units, and thus each term represents some kind of pressure: P is the static pressure, and it represents the actual thermodynamic pressure of the fluid, $\rho\frac{V^2}{2}$ is the dynamic pressure and it represents the pressure rise when the fluid in motion is brought to a complete stop, and ρgz is the hydrostatic pressure. /19/

The sum of the static pressure and dynamic pressure is called stagnation pressure. The stagnation pressure represents the pressure at a point when the fluid in motion is brought to a complete stop. It can be noted from the conservation of mass principle, that when the flow area decreases the average velocity increases and vice versa. The static pressure, respectively, decreases at the orifice. Vena contracta is the point where the flow area reaches the minimum value. The Bernoulli equation (4) demonstrates the balance between static, dynamic and hydrostatic pressure. When the stream of a flow is level, there is no elevation differences, the hydrostatic pressure is assumed to be constant along the streamline, thus, it can be eliminated from the equation. As the velocity increases towards the vena contracta point the static pressure decreases. After the vena contracta point, the fluid velocity decreases and the static pressure increases. This phenomenon is called pressure recovery. The concept of vena contracta point is shown in figure 7. /19, 21/

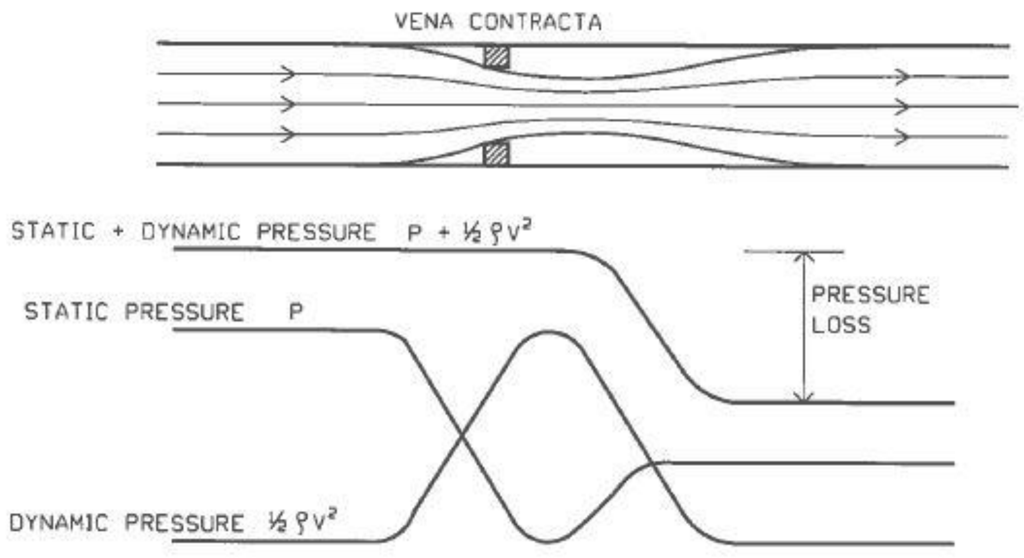

Figure 7. Static and dynamic pressure around vena contracta point. /21/

Care must be exercised when applying the Bernoulli equation. The equation is one of the most frequently used and misused because of the simplicity of the Bernoulli equation. In order to, make the Bernoulli equation applicable, some limitations and assumptions have to be made. The first limitation of the Bernoulli equation is that the flow has to be a steady flow, negligible viscous effects, any shaft work is not considered, the density of the fluid has to be approximated to be constant, incompressible flow, and the Bernoulli equation is applicable along a streamline. /19, 21/

3.5 Laminar and Turbulent Flow

Fluid flow applications are comprehensively used in cooling applications and fluid distribution networks etc. The friction affect is directly proportional to the pressure drop and head loss in pipes. In this regard, flow sections of circular cross section are referred as pipes. Pipes are widely used in fluid flow applications, especially when the fluid is liquid. This is due to their ability to withstand high pressure differences between the inside and the outside without affecting the measurements. At low velocities the flow is smoothly streamlined and this characterization is called laminar. When the velocity begins gradually to increase, the streamline of the flow starts to behave chaotically, a disordered motion, and this characterization is called turbulent. In between laminar and turbulent phase, there is a development phase which is called transition. The transition phase is characterized as a mixture of laminar and turbulent flow affects. The flow fluctuates between laminar and turbulent flow before it becomes fully turbulent. In the 1880s Osborne Reynolds discovered that the flow regimes depend on the ratio of inertial forces to viscous forces in the fluid, and afterwards each and every regime was expressed by Reynolds number values. Under most practical conditions, the flow in circular pipe is laminar for Reynolds number under 2300 and turbulent for over 4000. The transitional flow is in between Reynolds number 2300 and 4000.Figure 8 shows the flow regimes. /19, 20, 21/

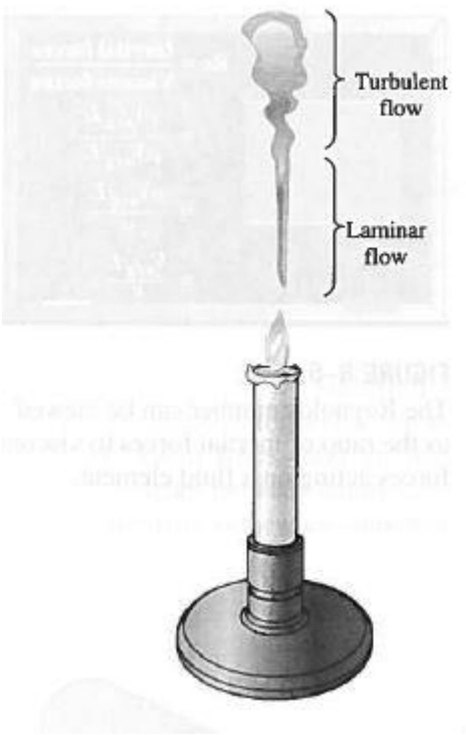

Figure 8. An outlining figure of laminar and turbulent flow regimes. /19/

Reynolds number is expressed for internal flow in a pipe as:

$$Re = \frac{\rho V_{avg} D}{\mu} \qquad (5)$$

Where ρ is the density [kg/m^3], V_{avg} is the average flow velocity [m/s], D is the diameter of the pipe [m] and μ is the dynamic viscosity [kg/m*s]. Reynolds number is a dimensionless quantity.

3.5.1 Entry Regions and Lengths

The velocity profile at the entrance of the pipe is uniform. However, due to the no-slip condition the fluid particles in the streamline next to the pipe wall come to a complete stop. This layer causes the fluid particles in the adjacent streamline to slow down as a result of friction. Thus, the velocity near the wall gradually decreases. In order to, keep the mass flow rate constant through the pipe, the velocity in the middle section of the pipe has to increase. Figure 9 shows the development of the velocity profile. /19, 20/

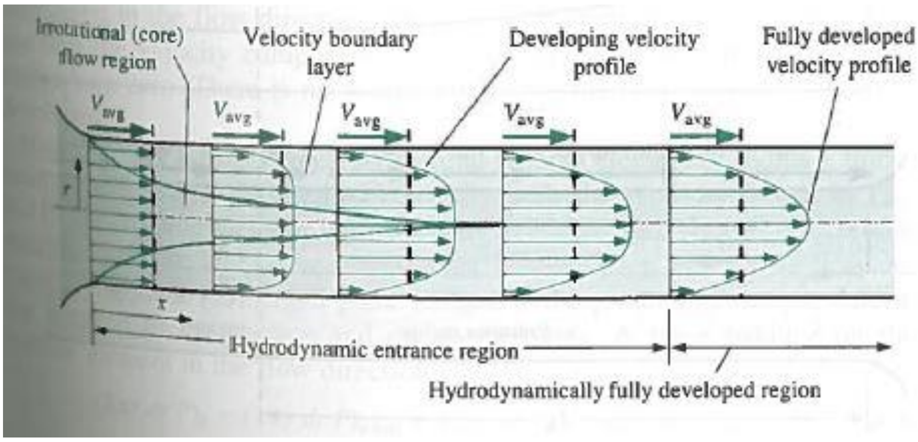

Figure 9. The development of the velocity profile. /19/

The layer in which the effects of the viscous shearing forces are felt is called boundary layer. The region in which the frictional effects are negligible and the velocity remains constant radially, is called irrotational flow region. The boundary layer continues to increase in the flow direction until it reaches the centre of the pipe. The region from the pipe inlet to the point at which the velocity profile becomes fully developed is called the hydrodynamic entrance region, and respectively the length of this region is called the hydrodynamic entry length. The region beyond the hydrodynamic entrance region in which the velocity profile remains fully developed is called hydrodynamically fully developed region. However, the flow is denoted by fully developed. /19, 20/

The hydrodynamic entry length is determined separately for laminar and turbulent flow due to their different behaviour. For laminar flow the dimensionless hydrodynamic entry length is determined as follows:

$$\frac{L_{h,laminar}}{D} \cong 0{,}05Re \qquad (6)$$

Where $L_{h,\ laminar}$ is the hydrodynamic entry length for laminar flow, D is the nominal diameter of the pipe and Re is the Reynolds number. The entry length for Reynolds number 2300 is 115D /19, 20/. Respectively, for turbulent flow the dimensionless hydrodynamic entry length is approximated as:

$$\frac{L_{h,turbulent}}{D} \approx 10 \qquad (7)$$

The entry length for turbulent flow is much lower than the laminar flow, because turbulent flow does not depend on the Reynolds number.

3.6 Pressure drop and Minor Loss

The pressure loss is expressed for all types of internal fully developed flows by /19/:

$$\Delta P_L = f\frac{L}{D}\frac{\rho V_{avg}^2}{2} \quad (8)$$

Where ΔP_L denotes for the pressure loss between two points in the streamline of the flow [Pa], f is the Darcy friction factor and $\rho V_{avg}^2/2$ is the dynamic pressure. This equation expresses the pressure loss for fully developed laminar and turbulent flows and for circular and non-circular pipes. /19/

For fully developed laminar flow in a pipe, the Darcy friction factor is expressed as /19/:

$$f = \frac{64}{Re} \quad (9)$$

The friction factor in fully developed laminar pipe flow depends on the Reynolds number. However, the friction factor in fully developed turbulent pipe flow depends on the Reynolds number as well but also on the relative roughness ε/D, which is the ratio of the mean height of roughness of the pipe to the pipe diameter /19/. The equation for calculating the friction factor for fully turbulent pipe flow is called Colebrook equation, which is named after Cyril F. Colebrook (1910–1997) /19/. The Colebrook equation:

$$\frac{1}{\sqrt{f}} = -2{,}0\log\left(\frac{\frac{\varepsilon}{D}}{3{,}7} + \frac{2{,}51}{Re\sqrt{f}}\right) \quad (10)$$

Where ε is the surface roughness of the pipe [mm]. Lewis F. Moody designed a diagram, which is nowadays known as the Moody chart, see appendix 1. The Moody chart represents the Darcy friction factor for pipe flow as a function the Reynolds number and ε/D over a wide range /19/. /19, 21/

Naturally, in a piping system the flow passes through bends, elbows, valves, expansions, contractions etc. These components and designs induces the flow to separate, and thus developing more pressure loss in the system. However, the most remarkable pressure loss comes from the straight sections of the piping system, and this is denoted as major losses. Thus, the pressure loss which develops due to the components in the piping system are called minor losses. The flow profile and behaviour inside a valve and fittings is very hard to anticipate and complex to determine by theoretical analysis. Therefore, minor losses are determined experimentally and, naturally the manufacturers conduct such tests for their components. /19/

Minor losses are expressed in terms of the loss coefficient K_L, defined as:

$$K_L = \frac{h_L}{\frac{V^2}{2g}} \qquad (11)$$

Where h_L is the additional irreversible head loss caused by insertion of the component and it is defined as $h_L = \Delta P/\rho g$ /19/.

In general, the loss coefficient strongly depends on the Reynolds number and the geometry of the component.