DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

COMPUTATIONAL FLUID DYNAMICS APPLICATION FOR DETERMINING FLOW CHARACTERISTICS OF VALVES

by
Erce KOYUNBABA

January, 2008
İZMİR

COMPUTATIONAL FLUID DYNAMICS APPLICATION FOR DETERMINING FLOW CHARACTERISTICS OF VALVES

A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of Dokuz Eylül University
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Mechanical Engineering, Energy Program

by
Erce KOYUNBABA

January, 2008
İZMİR

M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled **"COMPUTATIONAL FLUID DYNAMICS APPLICATION FOR DETERMINING FLOW CHARACTERISTICS OF VALVES"** completed by **ERCE KOYUNBABA** under supervision of **PROF. DR. İSMAİL TAVMAN** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Prof. Dr. İsmail TAVMAN

Supervisor

Prof. Dr. Ali GÜNGÖR

(Jury Member)

Assoc. Prof. Dilek KUMLUTAŞ

(Jury Member)

Prof.Dr. Cahit HELVACI
Director
Graduate School of Natural and Applied Sciences

ACKNOWLEDGMENTS

I would like to thank to my supervisor Prof. Dr. İsmail TAVMAN for his valuable guidance and understanding throughout my project.

I am also grateful to my family and colleagues for their advices and unlimited supports.

Most of all, I would like to thank to my fiancée for her incomparable helps, unlimited support and absolute love.

Erce KOYUNBABA

COMPUTATIONAL FLUID DYNAMICS APPLICATION FOR DETERMINING FLOW CHARACTERISTICS OF VALVES

ABSTRACT

The valves used in different fields of industry are the most indispensable elements of the flow control. The expectations of the users differ from one another according to the aim of the usage like having a longer lifeline, supporting a more sensitive flow control and having a minimum resistance to flow if necessary; that’s why, the engineers improve the existing valves and look for new models.

Nowadays, computers are the greatest helpers of engineers. Computer support in the analysis of difficult problems like solving flow problems is a big facility. After drawing a valve in three dimensional with computers, the working conditions of valves can be analyzed using finite element method and computational fluid dynamics codes. The results of the changes in the geometry can be observed without producing a prototype via this analysis.

In this study, the flow characteristic curves of the ball and butterfly types of valves, which are widely used in industry, are analyzed by ANSYS CFX and the results are compared with the catalogue data of the valve producing firms.

Keywords: CFD, ball valve, butterfly valve, flow characteristic

HESAPLAMALI AKIŞKANLAR DİNAMİĞİ UYGULAMASI İLE VANALARIN AKIŞ KARAKTERİSTİKLERİNİN BELİRLENMESİ

ÖZ

Endüstride birçok alanda kullanılan vanalar, akışkan kontrolünün vazgeçilmez elemanlarıdır. Vananın kullanıldığı amaca göre, kullanım ömrü daha uzun olan, akışkan kontrolünü daha hassas sağlayan, gerektiğinde akışa minimum direnç gösteren vanalar, kullanıcıların beklentileri arasındadır. Bundan dolayı mühendisler mevcut vanaları sürekli olarak geliştirmekte ve yeni model arayışına gitmektedirler.

Günümüzde bilgisayarlar mühendislerin en büyük yardımcıları konumundadırlar. Akışkan problemleri gibi çözümlemesi zor problemlerin analizlerinde bilgisayar desteği, büyük kolaylıklar sağlamaktadır. Bir vananın üç boyutlu çizimi bilgisayarlar aracılığı ile hazırlandıktan sonra, vananın çalışma koşulları sonlu elemanlar yöntemi ve hesaplamalı akışkanlar dinamiği kodları kullanılarak analiz edilebilir. Bu analizler sonucunda vana ilk örneği üretilmeden geometrideki değişimlerin sonuçları hızlı bir şekilde izlenebilir.

Bu çalışmada endüstride yaygın olarak kullanılan vana tipleri olan küresel ve kelebek vananın akış karakteristik eğrileri ANSYS CFX programı yardımıyla analiz edilmiş ve vana üreten firmaların katalog verileri ile karşılaştırılmıştır.

Anahtar sözcükler: HAD, küresel vana, kelebek vana, akış karakteristiği

CONTENTS

Page

CHAPTER ONE
INTRODUCTION

Control of the volumetric flow rate in a piping system is one of the functions of valves. In order to linearly control the volumetric flow rate in valves, several devices were designed. The effects of the control devices on the fluid flows, however, were not very clear, because the fluid flows could not be observed. The devices may linearly control the volumetric flow rate very well, but they could cause other side effects such as cavitations and large energy loss. Therefore, the investigation of fluid flows of valves with control devices is required when a control device is employed or designed. The main purpose of this study is to provide enough information of flow characteristics of valves by using commercial package, ANSYS CFX.

Fluid flows through a control valve received attention in the past decade. It was difficult to know the flow variation in a valve, because valves were not transparent. Due to the fast progress of the flow visualization and measurement techniques, it became possible to observe the flows inside a valve and to estimate the performance of a valve. For example, Mertai (2001) established a close-loop water tunnel system to conduct a performance test of a V-sector ball valve. They used a Laser Doppler Velocimetry (LDV) measuring system to investigate flows in ball valves. They also used a high-speed camera to capture the three-dimensional vertical structure at the downstream region. Davis & Stewart (2002) employed a close-loop piping system to test and to observe flows passing through a globe control valve.

An alternative approach to study fluid flows in a ball valve is to conduct computational simulation. Since the last century, computational fluid dynamics (CFD) has become an important tool for design of fluid machinery. For example, Computational analysis is done by Kerh (1997) et al. for the transient interaction of fluid and structure in a central valve by finite element method. Mertai (2001) et al. used the computer program FLUENT to define the flow around a V-sector ball valve and did experiments to conduct the performance test of a V-sector ball valve by building up a water tunnel system. van Lookeren Campagne (2002) used a commercial package, AVL-Fire, to simulate flows containing bubbles in ball valves. Davis & Stewart (2002) adopted FLUENT to study the flows in globe control valves. For three-dimensional analysis, Huang & Kim (1996) utilized FLUENT to simulate turbulent flows in a butterfly valve, in which the k-ε model was employed for turbulence consideration.

Chern and Wang (2006) worked on flow samples and cavitation phenomena experimentally. They visualized the flow for different openings via PTFV -particle tracking flow visualization method. In Figure 1.1, the necessary mechanism for displaying the flow and deriving the valve performance is seen.

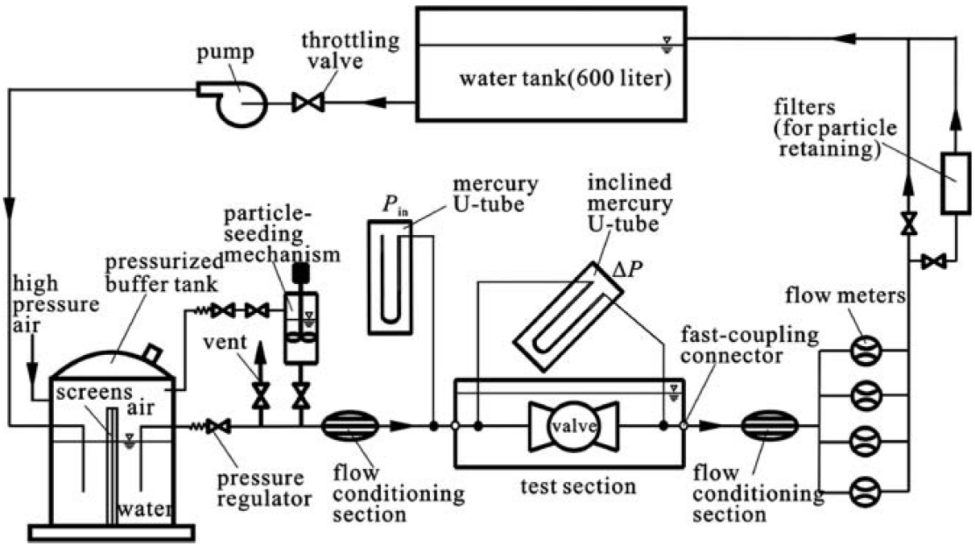


Figure 1.1 Schematic diagram of experimental apparatus

Chern and Wang (2006) used the ANSI/ISA-75.02-1996 “Control Valve Capacity Test Procedures” seen in Figure 1.2 for testing the valve performance. The test section is a transparent mechanism made up of Plexiglas tube and acrylic ball valve with a diameter of DN 50. Sample photos are created via a digital camera using Particle tracking flow visualization method (PTFV).

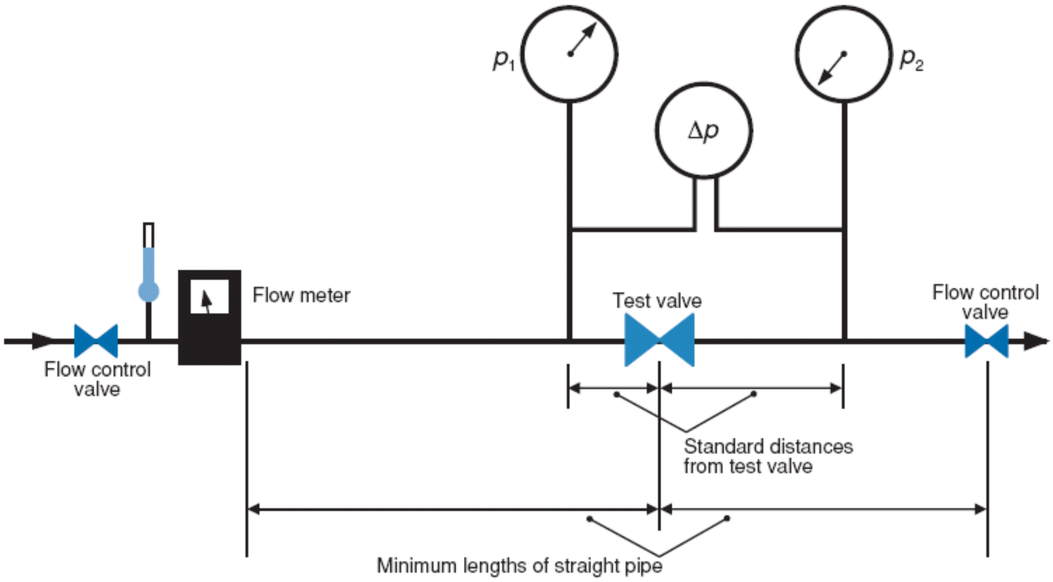


Figure 1.2 Valve manufactures determine flow coefficients by testing the valve with water using a standard ISA test method

They studied the performance characteristics and the inside valve flow patterns by employing the particle tracking flow visualization method and the flow rate/pressure measurement technique.

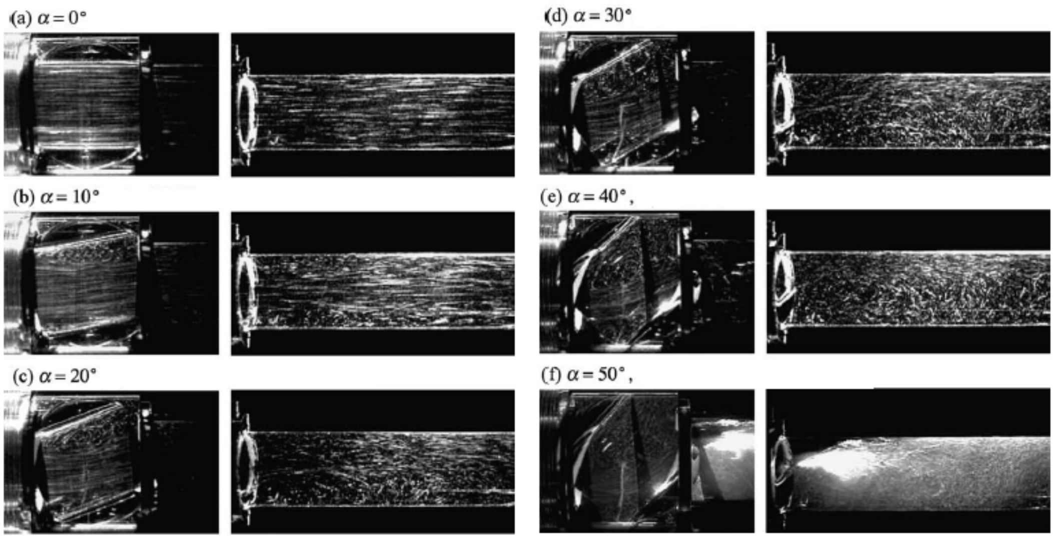


Figure 1.3 Results of flow visualization at Re= 0.64 x 10^5

Huang and Kim(1996) did numerical analysis in butterfly valves with FLUENT program using CFD codes. They compared the results they gained by FLUENT for different disc angles with the ones they gained from experimental studies. They also calculated the optimum pipe length at downstream of the valve for a disc angle of 45º.

Chern and Wang(2004) studied V-ports with angles of 30º, 60º and 90º numerically and experimentally. They concluded that V-ports having angles of 30º and 60º make the flow rate proportional to the valve opening. The pressure loss between the inlet and the outlet of a ball valve is increased in V-ports. V-ports with a small angle like 30º increase the possibility of cavitations when compared with flows without V-ports. The decreases in the angle of V-ports cause the increase in pressure loss.

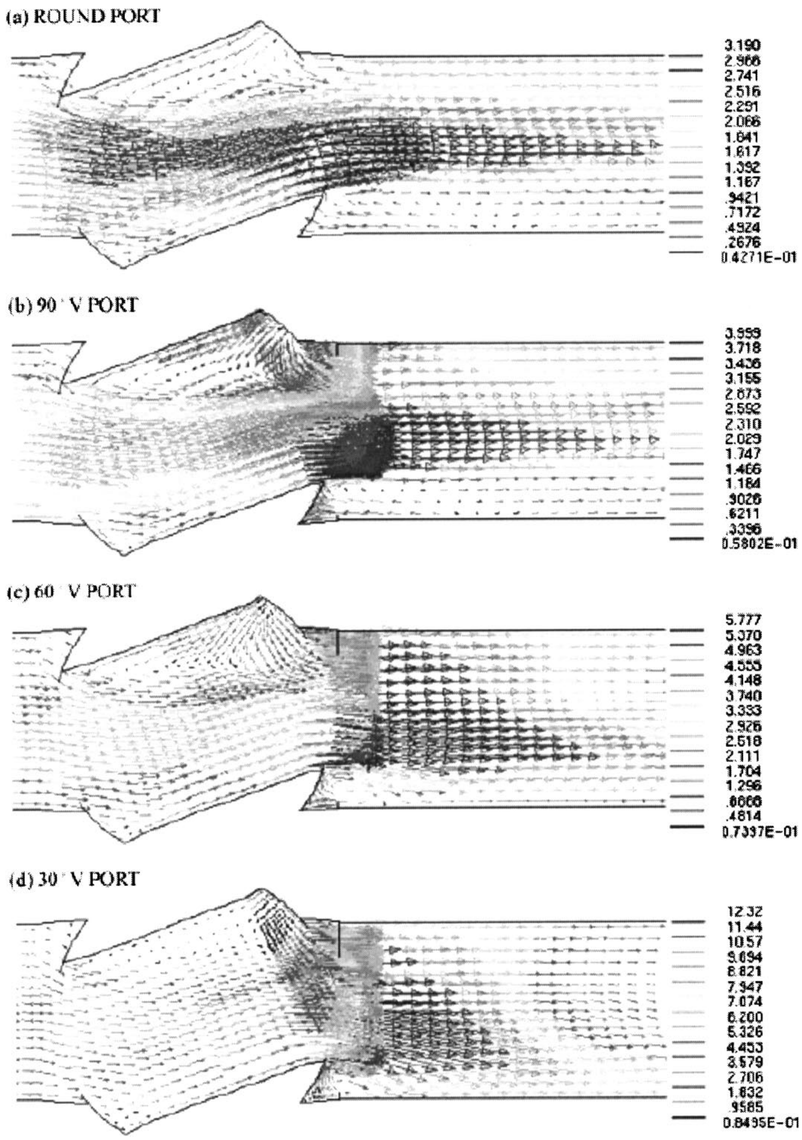


Figure 1.4 Flow patterns of computational results at Re= 0.64 x 10^5

CHAPTER TWO
FUNCTIONS OF VALVES

2.1 Introduction

Valves are integral components in piping systems they are the primary method of controlling the flow, pressure and direction of the fluid. Valves may be required to operate continuously e.g. control valves, or they may be operated intermittently e.g. isolation valves, or they may be installed to operate rarely if ever e.g. safety valves. A valve can be an extremely simple, low cost item or it may be and extremely complicated, expensive item. In piping design the valves probably require more engineering effort than any other piping component.

2.2 Valve Classifications

2.2.1 Isolation - Stop valve

The isolation of the downstream system from the upstream system by use of and isolation-stop valve is a critically important function. The prime requirements of this valve are tight shut off when closed and minimum restriction to flow when opens.

Valves used for this function include gate valves, globe valves, ball valves, plug valves, butterfly valves, diaphragm valves and pinch valves.

2.2.2 Regulation of flow

Many applications require the flow of the fluid be regulated (throttled) at some fixed or variable level between fully zero and maximum flow limits. This is achieved by introducing resistance to flow, or by bypassing flow or by changing the direction of the flow. An important feature for control valves is that the output variable (flow) is related to the input variable (valve position). An ideal operating characteristic of a hand operated flow control valve is that the flow is directly proportional to the position of the hand wheel.

Valve types for this function include globe, needle, ball, and butterfly. Globe and needle valves are best suited for this duty but ball valves are also easily adapted to give reliable flow control.

2.2.3 Back flow prevention

In some circumstances it is important to prevent reversed fluid flow. The type of valve for this duty is a non-return-valve (NRV) or check valve. The important criteria when selecting these valves are tight shut off against reverse flow, low resistance to flow for forward flow, fast response. The valve can be operated to close by gravity, fluid flow, or spring. Two main valve types are available for this function lift check valves or swing check valves.

2.2.4 Pressure Regulation

In many applications, more generally associated with gases, there is a need to reduce the supply press to a set fixed value. It is also necessary to maintain this reduced pressure over a range of fluid flow conditions. The pressure regulator valve is engineered for this application. The valve is basically a globe valve biased open by an adjustable spring force with the feedback pressure tending to move the valve to a closed position such that at the set pressure the feedback pressure force just exceeds the spring force.

The pressure regulator valve operates using the downstream fluid pressure as feedback. This is mostly taken from within the valve (self acting). For more accurate control a feedback connection can be taken from the downstream piping.

The pressure regulation at low near zero flows is difficult and it is often necessary to include internal or external relief valve functions to ensure no high pressures in the downstream system.

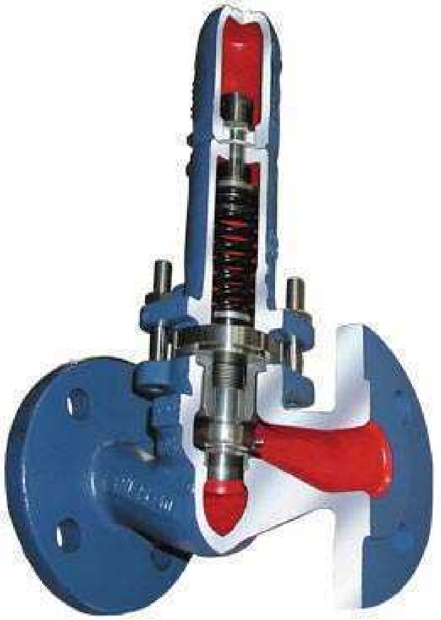


Figure 2.1 Pressure reducing valve

2.2.5 Pressure Relief Valves- Safety valves

A very important valve for safety is the pressure relief valve. This valve is used in applications where excessive pressure in the system can cause damage or failure or can introduce a safety risk. Uncontrolled excessive pressures can result in disastrous accidents e.g. when potentially explosive gases are being controlled. Relief valves are mainly spring loaded but they can also be gravity operated and other more specialized designs are available.

The bursting /rupture disc must be included under the general heading of safety valves. This is simply a disc which ruptures when a set pressure is exceeded. The fluid then escapes through the ruptured disc. If the bursting disc operates the system has be closed down and vented and the bursting disc is then replaced.

Relief valves when used for safety applications are engineered in line with safety regulations and require regular inspections to confirm the settings and the operation. An important part of the relief valve installation is the routing of the relieved fluid. This pipe route must be to a safe location and must be engineered such that it is always fully open.

Figure 2.2 Spring loaded safety valve

2.3 Fluid Properties and operating conditions

The properties of the fluid be controlled have a major impact on the design and materials of construction of the valve. The piping industry, over the years, had developed a wide range of valve designs and material to handle virtually all of the fluids being handled. The selection of the valve should take into account fluid viscosity, temperature, density and flow rate. The valve must be suitable to withstand resulting corrosion and erosion and if necessary the valve may have to be design for no internal hold up of fluids.

Important considerations include for absolute internal and external leak tightness when handling toxic or explosive fluids. There regulations also include for the need for a fire safe valve to maintain its internal and external integrity when the valve surrounded by flames from a fire.

2.4 Valve Connections

There are a number of methods of connecting valves into the piping systems- as follow,

- Flanges: The valve is provided with suitable rated flanges.
- Wafer: The valve is provided with suitable sealing faces and is trapped between line flanges.
- Butt Welded: The valve is provided with butt weld end and welded into the piping system using high integrity joints.
- Socket Welded: Socket welds allow and welded into the piping system using fillet welds.
- Screwed Ends: Ends can be provided with female or male screwed ends. The threads can be taper or parallel
- Compression Fittings: Ends can be provided with compression fittings

2.5 Valve Containment

An important requirement in valve design is to minimize the leakage of fluids into the surrounding environment. This is very important in the nuclear industry and when transferring toxic or flammable fluids. The possible leakage points on valves are listed below,

- The end connections with the piping.
- The spindle gland seals -allowing axial and rotary motion.
- For top entry valves the sealed top closure joint
- For three piece ball valves - the two split joints
- Valve drain connections and vent connections

The best option for minimizing risk of leakage from the pipe connections is to use butt welded joints which can be verified by non-destructive-testing (NDT). This option obviously eliminates the valve types which have to be removed for maintenance.

The options for eliminating risk of gland leakage is to use bellows sealed valves. The risk can also be reduced by incorporating dual seals with a test point between. Pinch valves and diaphragm valves do not include gland sealing and are therefore not at risk of gland leakage.

2.6 Valve Descriptions

2.6.1 Gate Valves

Gate valves are generally used in the process industry for on-off service. The design is not suitable for throttling duty because the sealing surfaces can easily suffer from wire drawing (erosion) when low flows are being maintained against high differential pressures and the design give very poor flow control characteristics.

The gate valve can be manufactured in a wide range of sizes from 5 mm to above 2000 mm diameter. The designs are proven and well tested. There is a tendency to move to butterfly valves as a lower cost option.

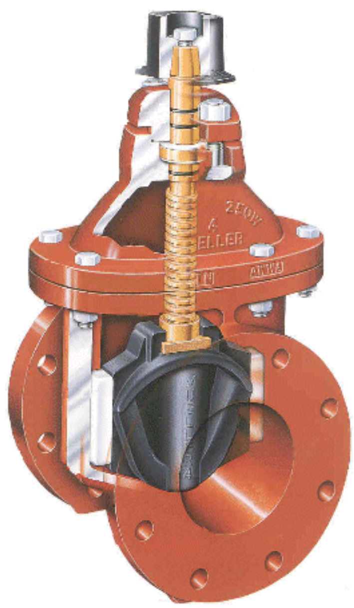


Figure 2.3 Wedge gate valve

The valve can be based on a solid wedge, a wedge which can adjust to suit the seal faces, or a parallel faced based on two discs which slide between parallel sealing faces with a mechanism form forcing the discs out on the last part of the spindle travel. The valve can be based on a simple rising spindle design or a fixed spindle which screws into the gate. There are a large number of gate valve variations including slide valves, knife valves, penstock valves, sluice valves, and venturi valves.

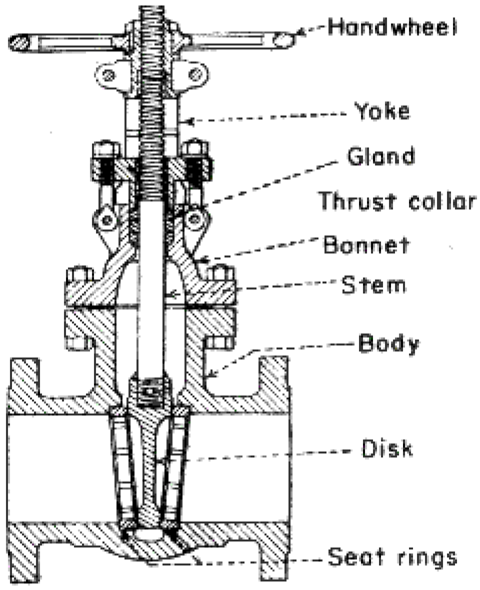


Figure 2.4 Rising spindle gate valve

2.6.2 Globe Valves

The globe valve includes an orifice set into the body through which the fluid flows. A disc located on the end of the spindle is engineered to move in and out along the axis of the orifice. When the disc is moved to sit in the orifice the flow path is shut-off. The flow path is progressively increased as the disc is moved away from the orifice.

The surface of the orifice (seat) is generally engineering as a replaceable item made from erosion resistant material with a polished surface finish. The disc can be fitted with a soft seat if a tight shut-off is required. For flow control duties the disc is supplied with an engineered shape often with a contoured skirt.

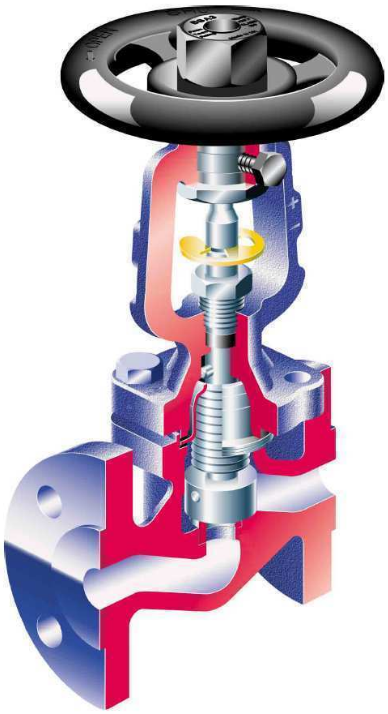


Figure 2.5 Bellows sealed globe valves

For manually operated valves the spindle screwed so that rotation of the handle moves the disc in and out. For actuated control valves the spindle is moved in and out using a linear actuator which can be pneumatic, hydraulic or electric.

The fluid flow path through globe valves is such that there is normally a high fluid head loss through the valve. The inline body design has the highest head loss; the angle pattern body design has a lower head loss. There are certain designs of globe valves which have been engineered to have low head loss characteristics. Globe valves are supplied in sizes from 3 mm bore through 400 mm and can be used, size limiting at pressures up to 450 barg. Depending on the sealing systems the valves can be used at temperatures the 600 ^{o}C.

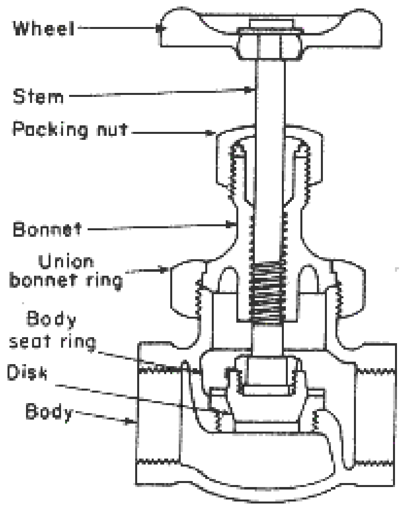


Figure 2.6 Small size screwed globe valve

2.6.3 Needle Valves

The needle valve is used specifically for accurately controlling the flow of fluids at low flows. The valve is basically a globe valve without the disc. It is generally used provided in small sizes of up to 20 mm bore.

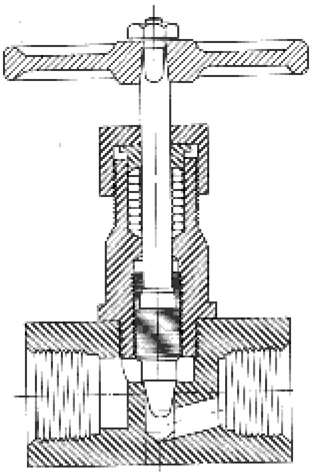


Figure 2.7 Needle valve

2.6.4 Plug Valves

The plug valve is the oldest of the valves. Plug valves have been in use for over 2000 years. This valve has been in continuous development over recent years. The plug valve is basically and on-off valve based on a plug with a rectangular hole through which the fluid flows. The plug is either tapered or cylindrical and is located in the valve body and can be rotated through a quarter turn to line the hole up with the pipe when open or across the pipe when closed. The plug can be adapted for multi-port use allows the valve to be used for diverting flow. The valve can be engineered with a lubricated plug which uses the lubricant to enable convenient operation over a wide range of pressures. The lubrication film also provides a seal.

Figure 2.8 Manuel plug valve

The unduplicated design includes seals in the plug and requires plastic bearing systems. The valve can include a cage between the plug and the body which includes the bearing a sealing system and allow convenient maintenance. These valves have be specially developed for use in industries requiring high performance operation under arduous conditions and allowing remote maintenance e.g. the nuclear industry.

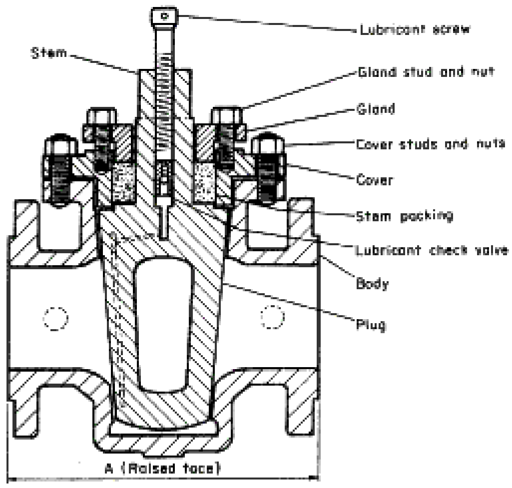


Figure 2.9 Lubricated plug valve

2.6.5 Ball Valves

The ball valve is basically a plug valve with a spherical plug and a round hole. Over recent years the materials of construction of the ball valve have been developed such that the ball valve is becoming the most popular valve for most process applications. There are two primary options for the ball valve design,

- Floating Ball Design- This is low cost option for the lower duties
- Trunnion Ball Design- This is a more costly option for the higher duties

The ball valve is generally provided as a reduced bore design allowing a smaller body but still with relatively low head loss compared to most other valve options e.g 25nb valve has a 20 mm reduced bore. The full bore option has a larger body but provides zero restriction to flow. The valve can be supplies as a multi-port design for flow diverting but only with the reduced bore option.

The engineering of the ball valve has to include for fitting and removing the ball and seat system. Ideally this has to be engineered to enable the valve to be maintained inline. One method of achieving this is to use the top-entry version - all of the internals are accessible by removing the top flange. Another method is to use a three piece body based on a central piece sandwiched between two pieces connecting the valve to the pipe work. The central piece can be released and pivoted away from the two outer pieces allowing access to all of the valve components.

Figure 2.10 Flanged ball valves

The ball valve can be engineered as a multi-port valve for flow diverting duties.

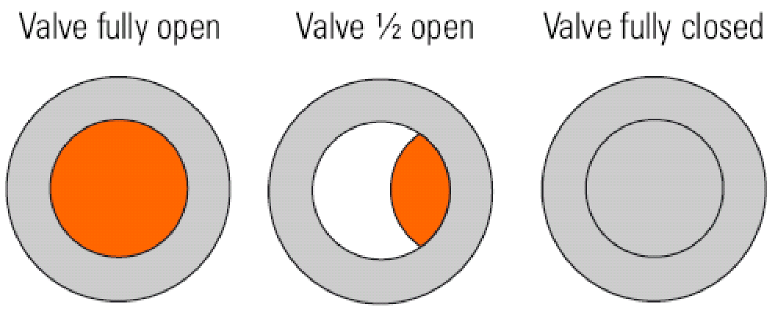


Figure 2.11 End view of the ball within the ball valve at different stages of rotation

An important advantage of all full bore valves is that the valve allows certain pipe cleaning operations e.g rodding. Ball valves can also be used on branches to enable instruments to be fed into pipe systems during operating periods.

Ball valves are available in all materials in sizes from 5 mm to over 600 mm. The valves can be used at pressures up to 700 bar. The main components limiting the performance of ball valves are the ball seals and valves are available with metallic seals.

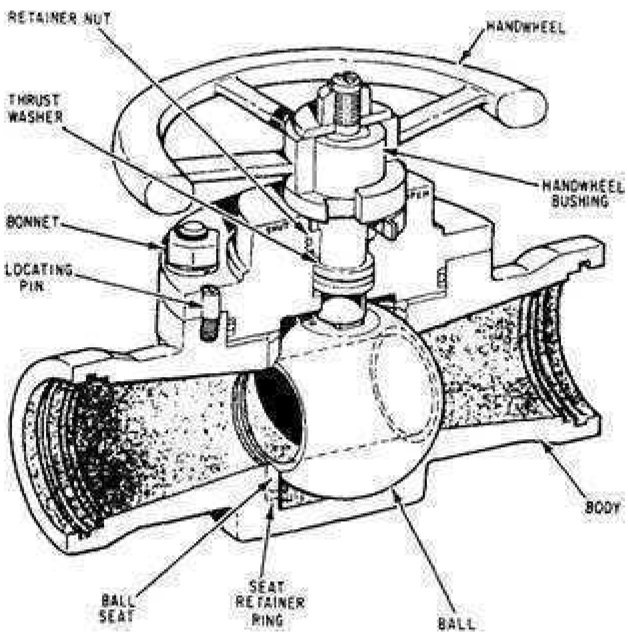


Figure 2.12 Typical seawater ball valve.

2.6.6 Butterfly Valves

The butterfly valve has head loss characteristics of a full bore valve. The design is based on use of an engineered disc of the same diameter as the bore of the pipe arranged to pivot such that when it is across the bore is closes off the flow path. When turned through 90^{o} the disc provides minimum resistance to the flow. The valve is a quarter turn valve.

The main variations for this valve are the methods of sealing the perimeter of the disc in its closed position. The simplest variation is to use an elastomer lined bore which is an interference fit on the disc. The other variations are based on offsetting the disc plane from the axis of rotation allow the disc to close against a circular face seal such that the fluid pressure increases the seal effect. Metallic seals are available allowing the valve to be used for a wide range of fluids at high temperatures.

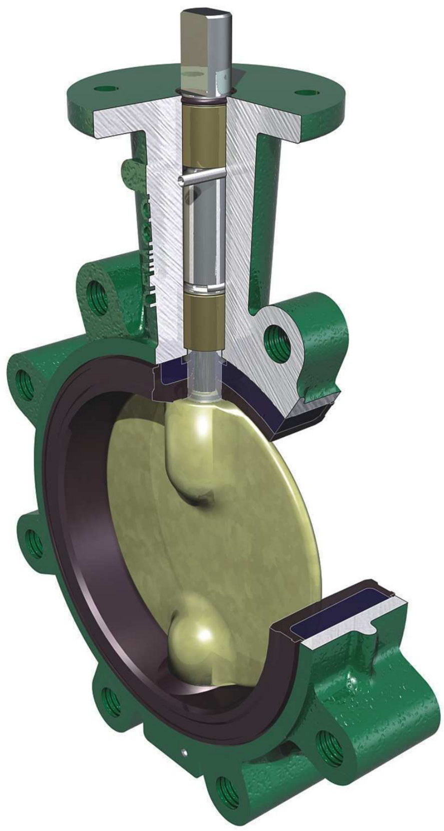


Figure 2.13 Butterfly valve

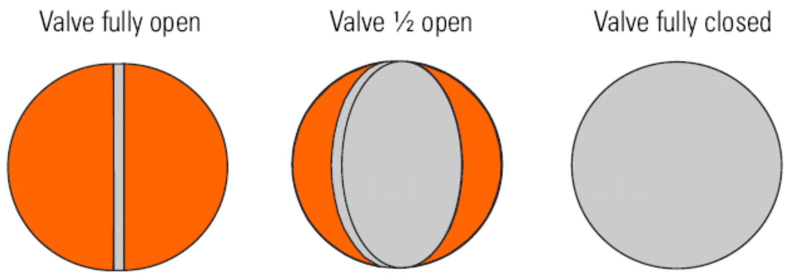


Figure 2.14 End view of the disc within the ball valve at different stages of rotation

The butterfly valve has been developing such that for many duties it now provides optimum solution for a leak tight on-off valve supplanting the gate valve. The butterfly valve can be engineered as a small valve of 25 mm bore and can be made for extremely large sizes above 5000 mm bore. Depending on the valve size working pressures up to 100 bar can be handled.

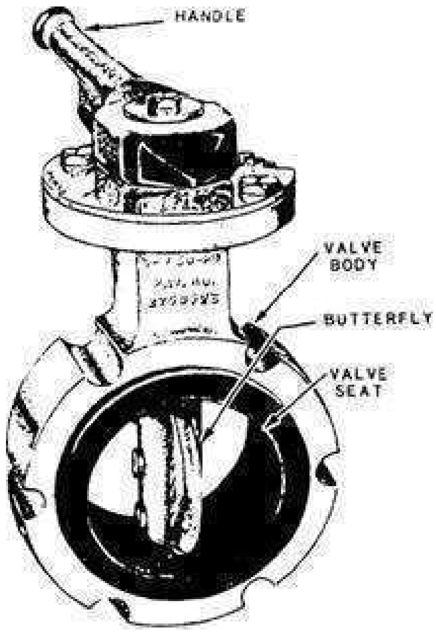


Figure 2.15 Butterfly valve

2.6.8 Diaphragm Valves

The diaphragm valve has a significant advantage over most of the other available designs, apart from the pinch valve, in that there is no gland seal requirement. The fluid flows straight through the valve via a chamber over which is an elastomer diaphragm. This diaphragm is normally arranged to provide no resistance to the flow.

The perimeter of the diaphragm is simply clamped to a seal face of the valve body as a static seal.

To close off the valve the diaphragm is simply forced down into the chamber to block off the flow. The chamber can include a weir across the flow path against which the diaphragm can be pressed to affect a more efficient seal with reduced diaphragm distortion.

The straight through variation is effectively a full bore valve design with all the associated benefits. However this option results in a much more arduous duty on the diaphragm which has to be a softer material.

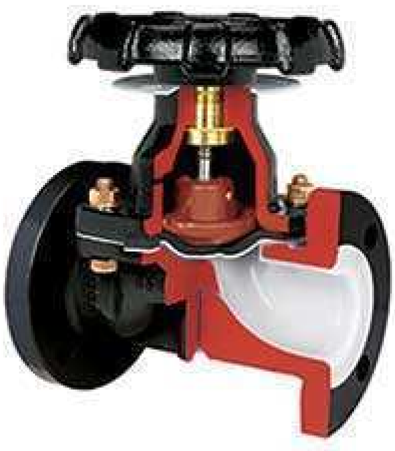


Figure 2.16 Diaphragm valve

This type of valve is manufactured in sizes from 6 mm to 400 mm and is generally limited to relatively low fluid pressures (less than 7 barg). However in the smaller sizes (up to 50 mm) valves can be specially engineered for use at pressures up to 30 barg. The diaphragm must be chosen to be compatible with the fluid. Whatever the fluid the diaphragms must be replaced at regular intervals and it is advisable to operate the valves frequently.

These valves are often used for duties which require a high degree of cleanliness as they can be supplied lined, and polished and can be very conveniently cleaned.

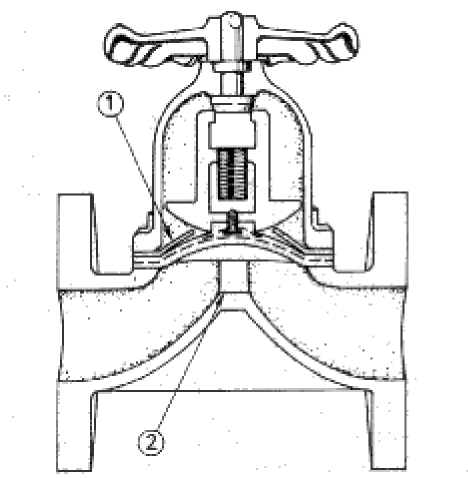


Figure 2.17 Diaphragm Valve

2.6.9 Pinch Valves

The pinch valve is a theoretically ideal solution for fluid on-off duties. The valve is simple a length of pipe made from an elastomeric material with a mechanical system for squeezing the tube closed when a shut off is required. The valve is a true full bore valve - there are no mechanical parts in contact with the fluid- The operation of the valve is ideally simple- The valve can be easily engineered as a tight-shut off valve.

The valve is often supplied with the pinch tube contained within a outer pipe between the end flanges. This option provides a method of monitoring for tube leaks and provides a degree of secondary containment.

The valve has similar limitations to the diaphragm valve. The diaphragm valve is really a variation on the pinch valve principles. Pinch valves are supplied is for diameters 25 mm - 1000 mm, temperatures -50 C - +160 C, and pressures 0 - 100 bar.

Figure 2.18 Pinch valve

2.6.10 Check Valves

Check valves are automatic in operation and designed to prevent reversal of flow in fluid piping systems. The valves are maintained open by the flow of fluid in the forward direction and are closed by back pressure of the fluid or by the weight of the closing mechanism or by a spring force. Various designs are available as listed below..

- Swing check
- Tilting disc
- Ball lift type
- Disc lift type
- Piston check
- Stop check

The range of check valve sizes range from 6mm to massive units of 3000 mm diameter and more.

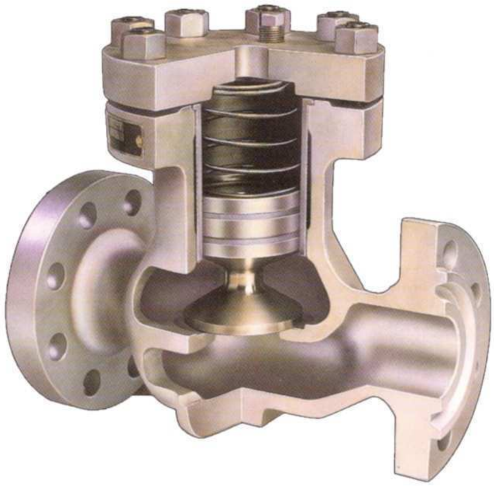


Figure 2.19 Piston check valve

The swing check variation is a low pressure drop unit based on a hinged disc. This type of valve is suitable for low velocity applications with infrequent velocity reversals. The valve can be fitted with external weights to allow faster closure to reduce water hammer or shock pressure on flow reversal. External systems can also be included to force the valve closed in the event of a local fire.

The lift check valve and piston check variations are used for higher duty applications. The valve is forced open by the fluid flowing up through the valve and is closed on fluid reversal by gravity, back pressure or by spring force.

The tilting disc variation on the swing check valve provides improve speed of operation and pressure performance and is probably the most popular design of check valve used in the process industry.

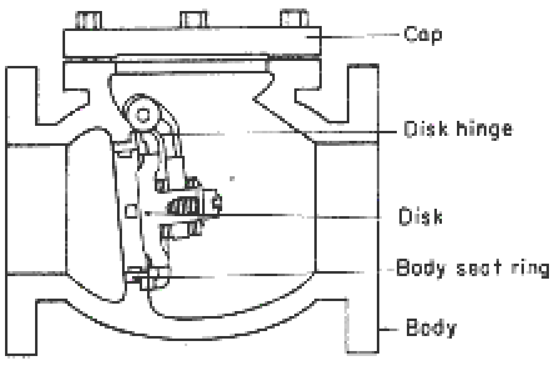


Figure 2.20 Swing check valve

CHAPTER THREE
FLOW THROUGH VALVES

3.1 Coefficients

Valves may be regarded as analogous to control orifices in which the area of opening is readily adjustable. As such, the friction loss across the valve varies with flow, as expressed by the general relationship,

$$v \propto (\Delta h)^{1/2} \qquad \ldots(3.1)$$

$$v \propto (\Delta p)^{1/2} \qquad \ldots(3.2)$$

where

v = flow velocity
Δh = headloss
Δp = pressure loss

For any valve position, numerous relationships between flow and flow resistance have been established, using experimentally determined resistance or flow parameters. Common parameters so determined are the resistance coefficient ζ and, dependent on the system of units, the flow coefficients C_v, K_v, and A_v. It is standard practice to base these parameters on the nominal valve size.

3.1.2 Resistance Coefficient ζ

The resistance coefficient ζ defines the friction loss attributable to a valve in a pipeline in terms of velocity head or velocity pressure, as expressed by the equations

$$\Delta h = \zeta \frac{v^2}{2g} \quad \text{(coherent SI or imperial units)} \qquad \ldots(3.3)$$

and: $\Delta h = \zeta \dfrac{v^2 \rho}{2}$ (coherent SI units) …(3.4)

or: $\Delta h = \zeta \dfrac{v^2 \rho}{2g}$ (coherent imperial units) …(3.5)

where

ρ = density of fluid
g = local acceleration due to gravity

The equations are valid for single-phase flow of Newtonian liquids and for both turbulent and laminar flow conditions. They may also be used for flow of gas at low Mach numbers. As the Mach number at the valve inlet approaches 0.2, the effects of compressibility become noticeable but are unlikely to be significant even for Mach numbers up to 0.5

Valves of the same type but of different manufacture, and also of the same line but different size, are not normally geometrically similar. For this reason, the resistance coefficient of a particular size and type of valve can differ considerably between makes. Table 3.1 can therefore provide only typical resistance-coefficient values. The values apply to fully open valves only and for $Re \geq 10^4$.

The Engineering Sciences Data Unit deals more comprehensively with pressure losses in valves. Their publication also covers correction factors for $Re < 10^4$ and shows the influence of valve size on the ζ-Value and scatter of data, as obtained from both published and unpublished reports and from results obtained from various manufacturers.

In the case of partially open valves and valves with reduced seat area, as in valves with a converging/diverging flow passage, the energy of the flow stream at the vena contracta converts partially back into static energy.

Table 2.1 Approximate resistance coefficients of fully open valves under conditions of fully turbulent flow

GLOBE VALVE, STANDARD PATTERN:	
• Full bore seat, cast.	$\zeta = 4.0–10.0$
• Full bore seat, forged (small sizes only).	$\zeta = 5.0–13.0$
Globe valve, 45° oblique pattern:	
• Full bore seat, cast.	$\zeta = 1.0–3.0$
Globe valve, angle pattern:	
• Full bore seat, cast.	$\zeta = 2.0–5.0$
• Full bore seat, forged (small sizes only).	$\zeta = 1.5–3.0$
Gate valve, full bore:	$\zeta = 0.1–0.3$
Ball valve, full bore:	$\zeta = 0.1$
Plug valve, rectangular port:	
• Full flow area.	$\zeta = 0.3–0.5$
• 80% flow area.	$\zeta = 0.7–1.2$
• 60% flow area.	$\zeta = 0.7–2.0$
Plug valve, circular port, full bore:	$\zeta = 0.2–0.3$
Butterfly valve, dependent on blade thickness:	$\zeta = 0.2–1.5$
Diaphragm valve:	
• Weir type.	$\zeta = 2.0–3.5$
• Straight-through type.	$\zeta = 0.6–0.9$
Lift check valve (as globe valve):	
Swing check valve:	$\zeta = 1.0$
Tilting-disc check valve:	$\zeta = 1.0$

3.1.3 Flow Coefficient C_v

The flow coefficient C_v states the flow capacity of a valve in gal(U.S.)/min of water at a temperature of $60F^0$ that will flow through a valve with a pressure loss of one pound per square inch at a specific opening position, as defined by the equation:

$$C_v = Q\left(\frac{\Delta p_0}{\Delta p} \times \frac{\rho}{\rho_0}\right)^{1/2} \qquad \ldots(3.6)$$

Where

Q = U.S. gal/min

Δp_0 = reference differential pressure = 1 lb/in.^{2}

Δp = operating differential pressure in lb/in.2

ρ_0 = density of reference fluid, water = 62.4 lb/ft^3

ρ = density of operating fluid in lb/ft^3

Because ρ/ρ_0 = specific gravity and the numerical value of Δp_0 is unity, Equation 3.6 is normally presented in the form:

$$C_v = Q\left(\frac{G}{\Delta p}\right)^{1/2} \qquad \ldots(3.7)$$

Where

G = specific gravity

3.1.4 Flow Coefficient K_v

The flow coefficient K_v is a version of coefficient C_v in mixed SI units. It states the number of cubic meters per hour of water at a temperature between 5◦ and 40◦C that will flow through the valve with a pressure loss of one bar at a specific opening position, as defined by the equation:

$$K_v = Q\left(\frac{\Delta p_0}{\Delta p} \times \frac{\rho}{\rho_0}\right)^{1/2} \qquad \ldots(3.8)$$

Where

Q = m^3/hour

Δp_0 = reference differential pressure = 1 bar

Δp = operating differential pressure, bar

ρ_0 = density of reference fluid (water = 1,000 kg/m^3)

ρ = density of operating fluid, kg/m^3

Because ρ/ρ_0 = specific gravity and the numerical value of Δp_0 is unity, Equation 3.8 is normally presented in the form:

$$C_v = Q\left(\frac{G}{\Delta p}\right)^{1/2} \qquad \ldots(3.9)$$

Where

G = specific gravity

3.1.5 Flow Coefficient A_v

The flow coefficient A_v is a version of the flow coefficient K_v in coherent SI units. Av states the number of cubic meters per second of water at a temperature between 5^0 and 40^0 C that will flow through the valve with a pressure loss of one Pascal at a specific opening position, as defined by the equation:

$$A_v = Q\left(\frac{\rho}{\Delta p}\right)^{1/2} \qquad \ldots(3.10)$$

Where

Q = flow rate, m^3/s
Δp = operational differential pressure, Pa
ρ = density of Newtonian liquid, kg/m^3

A_v is derived from Equation 3.10, which may be presented in the following forms:

$$Q = A\left(\frac{2}{\zeta}\right)^{1/2}\left(\frac{\Delta p}{\rho}\right)^{1/2} \qquad \ldots(3.11)$$

$$A = \left(\frac{2}{\zeta}\right)^{1/2} = Q\left(\frac{\rho}{\Delta p}\right)^{1/2} \qquad \ldots(3.12)$$

Where

v = flow velocity of fluid, m/s

A= cross-sectional area, m^2

The expression $A\left(\frac{2}{\zeta}\right)^{1/2}$ is replaced in Equation 3.10 by a single expression Av.

3.2 Interrelationships Between Resistance and Flow Coefficients

Resistance and flow coefficients are interrelated. If one coefficient is known, the other coefficients can be calculated. These are the interrelationships in which:

d_{inch} = reference pipe bore, inches

d_{mm} = reference pipe bore, mm

$$\zeta = \frac{889 d_{inch}^4}{C_v^2} = \frac{2.14 d_{mm}^4}{10^3 C_v^2} \qquad C_v = \frac{29.8 d_{inch}^2}{\zeta^{1/2}} = \frac{4.62 d_{mm}^2}{10^2 \zeta^{1/2}} \qquad \ldots(3.13)$$

$$\zeta = \frac{665.2 d_{inch}^4}{K_v^2} = \frac{1.6 d_{mm}^4}{10^3 K_v^2} \qquad K_v = \frac{25.8 d_{inch}^2}{\zeta^{1/2}} = \frac{39.98 d_{mm}^2}{10^3 \zeta^{1/2}} \qquad \ldots(3.14)$$

$$\zeta = \frac{512 d_{inch}^4}{10^9 A_v^2} = \frac{1.23 d_{mm}^4}{10^{12} A_v^2} \qquad A_v = \frac{716 d_{inch}^2}{10^6 \zeta^{1/2}} = \frac{1.11 d_{mm}^2}{10^6 \zeta^{1/2}} \qquad \ldots(3.15)$$

$$\frac{K_v}{C_v} = 865 \times 10^{-3} \qquad \ldots(3.16)$$

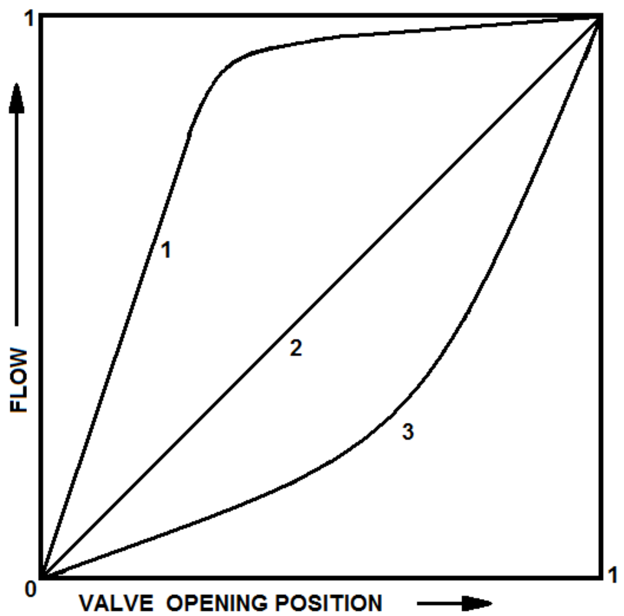
$$\frac{A_v}{C_v} = 23.8 \times 10^{-6} \qquad \ldots(3.17)$$

$$\frac{A_v}{K_v} = 27.8 \times 10^{-6} \qquad \ldots(3.18)$$

3.3 Relationship Between Resistance Coefficient and Valve Opening Position

The relationship between fractional valve opening position and relative flow through the valve is denoted as flow characteristic. When flow at all valve opening positions is taken at constant inlet pressure ion, the flow characteristic thus determined is referred to as inherent. Figure 3.1 shows such inherent flow characteristics that are typical for flow control valves.

In the most practical applications, however, the pressure loss through the valve varies with valve opening position. This is illustrated in Figure 3.2 for a flow system incorporating a pump. The upper portion of the figure represents the pump characteristic, displaying flow against pump pressure, and the system characteristic, displaying flow against pipeline pressure loss. The lower portion of the figure shows the flow rate against valve opening position. The latter characteristic is referred to as the installed valve flow characteristic and is unique for each valve installation. When the valve has been opened further to increase the flow rate, the pressure at the inlet of the valve decreases, as shown in Figure 3.2. The required rate of valve opening is, therefore, higher in this case than indicated by the inherent flow characteristic.



1 – Quick opening

2 - Linear

3 – Equal Percentage

Figure 3.1 Inherent flow characteristics of valves

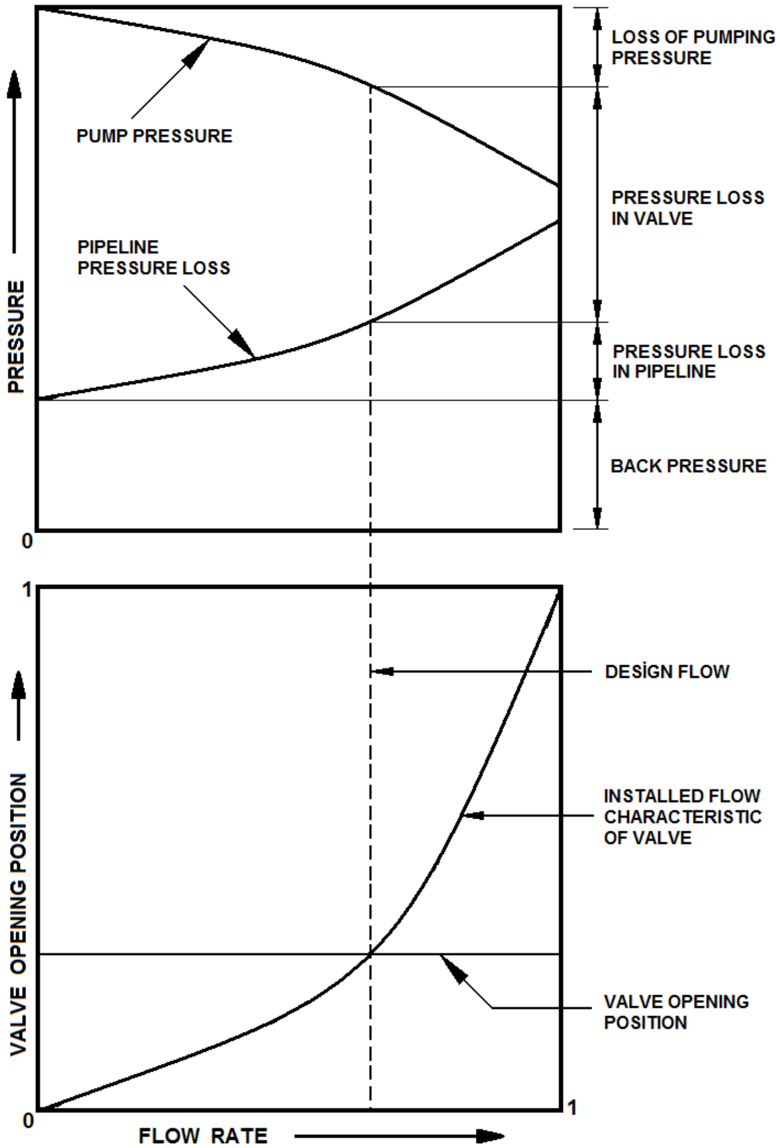


Figure 3.2 Relationship between flow rate, valve opening position, and pressure loss in a pumping system.

If the pump and system characteristic shows that the valve has to absorb a high-pressure drop, the valve should be sized so that the required pressure drop does not occur near the closed position, since this will promote damage to the seatings from the flowing fluid. This consideration leads frequently to a valve size smaller than the adjoining pipe.

3.4 Control Valves and Flow Characteristics

The relationship between control valve capacity and valve stem travel is known as the flow characteristic of the valve.

Trim design of the valve affects how the control valve capacity changes as the valve moves through its complete travel. Because of the variation in trim design, many valves are not linear in nature. Valve trims are instead designed, or characterized, in order to meet the large variety of control application needs. Many control loops have inherent non linearity's, which may be possible to compensate selecting the control valve trim.

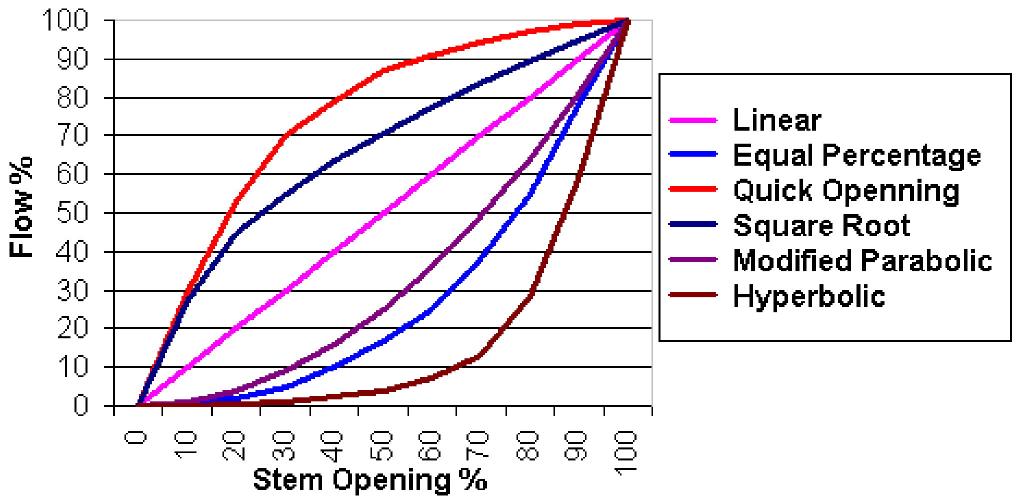


Figure 3.3 Control valve flow characteristics

The most common characteristics are shown in the figure 3.3. The percent of flow through the valve is plotted against valve stem position. The curves shown are typical of those available from valve manufacturers. These curves are based on constant pressure drop across the valve and are called inherent flow characteristics.

- **Linear** flow capacity increases linearly with valve travel.
- **Equal percentage** flow capacity increases exponentially with valve trim travel. Equal increments of valve travel produce equal percentage changes in the existing C_v.

- **A modified parabolic characteristic** is approximately midway between linear and equal-percentage characteristics. It provides fine throttling at low flow capacity and approximately linear characteristics at higher flow capacity.
- **Quick opening** provides large changes in flow for very small changes in lift. It usually has too high a valve gain for use in modulating control. So it is limited to on-off service, such as sequential operation in either batch or semi-continuous processes.
- **Hyperbolic**
- **Square Root**

The majority of control applications are valves with linear, equal-percentage, or modified-flow characteristics.

Installed Control Valve Flow Characteristics

When valves are installed with pumps, piping and fittings, and other process equipment, the pressure drop across the valve will vary as the plug moves through its travel.

When the actual flow in a system is plotted against valve opening, the curve is called the Installed Flow Characteristic**.**

In most applications, when the valve opens, and the resistance due to fluids flow decreases the pressure drop across the valve. This moves the inherent characteristic:

- A linear inherent curve will in general resemble a quick opening characteristic
- An equal percentage curve will in general resemble a linear curve

CHAPTER FOUR
COMPUTATIONAL FLUID DYNAMICS

4.1 What is CFD

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. More accurate software that can accurately and quickly simulate even complex scenarios such as transonic or turbulent flows are an ongoing area of research. Validation of such software is often performed using a wind tunnel. (Anonymous, 2007)

4.1.1 The History of CFD

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980's and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD was a tool used almost exclusively in research.

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time.

As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

4.1.2 The Mathematics of CFD

The set of equations which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations were derived in the early nineteenth century and have no known general analytical solution but can be discredited and solved numerically.

Equations describing other processes, such as combustion, can also be solved in conjunction with the Navier-Stokes equations. Often, an approximating model is used to derive these additional equations, turbulence models being a particularly important example.

There are a number of different solution methods which are used in CFD codes. The most common, and the one on which ANSYS CFX is based, is known as the finite volume technique.

In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discredited and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behavior of the flow.

4.1.3 Uses of CFD

CFD is used by engineers and scientists in a wide range of fields. Typical applications include:

- Process industry: Mixing vessels, chemical reactors
- Building services: Ventilation of buildings, such as atriums
- Health and safety: Investigating the effects of fire and smoke
- Motor industry: Combustion modeling, car aerodynamics
- Electronics: Heat transfer within and around circuit boards
- Environmental: Dispersion of pollutants in air or water
- Power and energy: Optimization of combustion processes
- Medical: Blood flow through grafted blood vessels

4.2 CFD Methodology

CFD can be used to determine the performance of a component at the design stage, or it can be used to analyze difficulties with an existing component and lead to its improved design.

For example, the pressure drop through a component may be considered excessive:

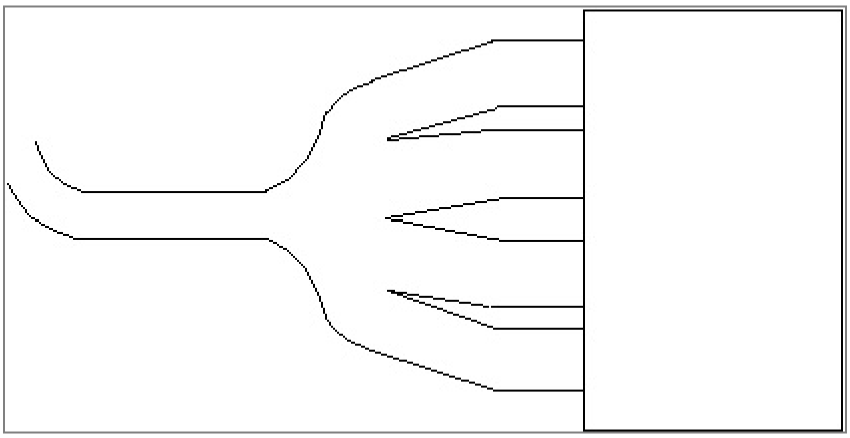


Figure 4.1 Existing component

The first step is to identify the region of interest:

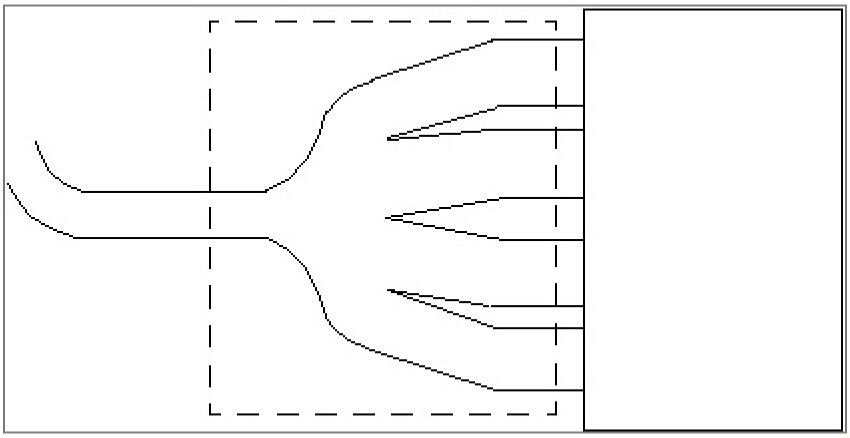


Figure 4.2 Defining the geometry to be analyzed

The geometry of the region of interest is then defined. If the geometry already exists in CAD, it can be imported directly. The mesh is then created. After importing the mesh into the pre-processor, other elements of the simulation including the boundary conditions (inlets, outlets, etc.) and fluid properties are defined.

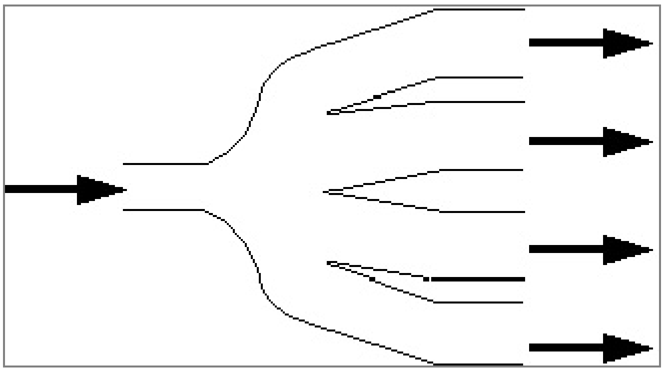


Figure 4.3 Defining boundary conditions

The flow solver is run to produce a file of results that contains the variation of velocity, pressure and any other variables throughout the region of interest.

The results can be visualized and can provide the engineer an understanding of the behavior of the fluid throughout the region of interest.

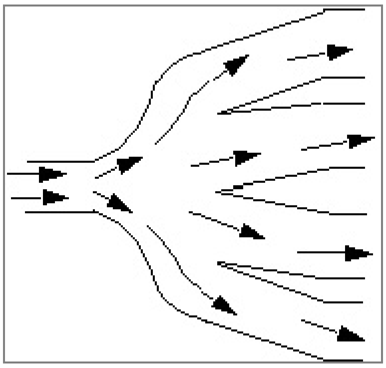


Figure 4.4 Flow results

This can lead to design modifications which can be tested by changing the geometry of the CFD model and seeing the effect.

The process of performing a single CFD simulation is split into four components.

4.2.1 Creating the Geometry/Mesh

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in CAD2Mesh or any of the other geometry/mesh creation tools. The basic steps involve:

- Defining the geometry of the region of interest.
- Creating regions of fluid flow, solid regions and surface boundary names.
- Setting properties for the mesh.

This pre-processing stage is now highly automated. In ANSYS CFX, geometry can be imported from most major CAD packages using native format, and the mesh of control volumes is generated automatically.

4.2.2 Defining the Physics of the Model

This interactive process is the second pre-processing stage and is used to create input required by the Solver. The mesh files are loaded into the physics pre-processor, ANSYS CFX-Pre.

The physical models that are to be included in the simulation are selected. Fluid properties and boundary conditions are specified.

4.2.3 Solving the CFD Problem

The component that solves the CFD problem is called the Solver. It produces the required results in a non-interactive/batch process. A CFD problem is solved as follows:

- The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law (for example, for mass or momentum) to each control volume.
- These integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations.
- The algebraic equations are solved iteratively.

An iterative approach is required because of the non-linear nature of the equations, and as the solution approaches the exact solution, it is said to converge. For each iteration, an error, or residual, is reported as a measure of the overall conservation of the flow properties.

How close the final solution is to the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals. Complex physical processes, such as combustion and turbulence, are often modeled using empirical relationships. The approximations inherent in these models also contribute to differences between the CFD solution and the real flow.

The solution process requires no user interaction and is, therefore, usually carried out as a batch process.

The solver produces a results file which is then passed to the post-processor.

4.2.4 Visualizing the Results in the Post-processor

The post-processor is the component used to analyze, visualize and present the results interactively. Post-processing includes anything from obtaining point values to complex animated sequences.

Examples of some important features of post-processors are:

- Visualization of the geometry and control volumes
- Vector plots showing the direction and magnitude of the flow
- Visualization of the variation of scalar variables (variables which have only magnitude, not direction, such as temperature, pressure and speed) through the domain
- Quantitative numerical calculations
- Animation
- Charts showing graphical plots of variables
- Hardcopy output.

4.3 Overview of ANSYS CFX

ANSYS CFX is a general purpose Computational Fluid Dynamics (CFD) software suite that combines an advanced solver with powerful pre- and post-processing capabilities.

ANSYS CFX is capable of modeling:

- Steady-state and transient flows
- Laminar and turbulent flows
- Subsonic, transonic and supersonic flows
- Heat transfer and thermal radiation
- Buoyancy
- Non-Newtonian flows
- Transport of non-reacting scalar components
- Multiphase flows
- Combustion

- Flows in multiple frames of reference
- Particle tracking.

ANSYS CFX includes the following features:

- An advanced coupled solver that is both reliable and robust.
- Full integration of problem definition, analysis, and results presentation.
- An intuitive and interactive setup process, using menus and advanced graphics.

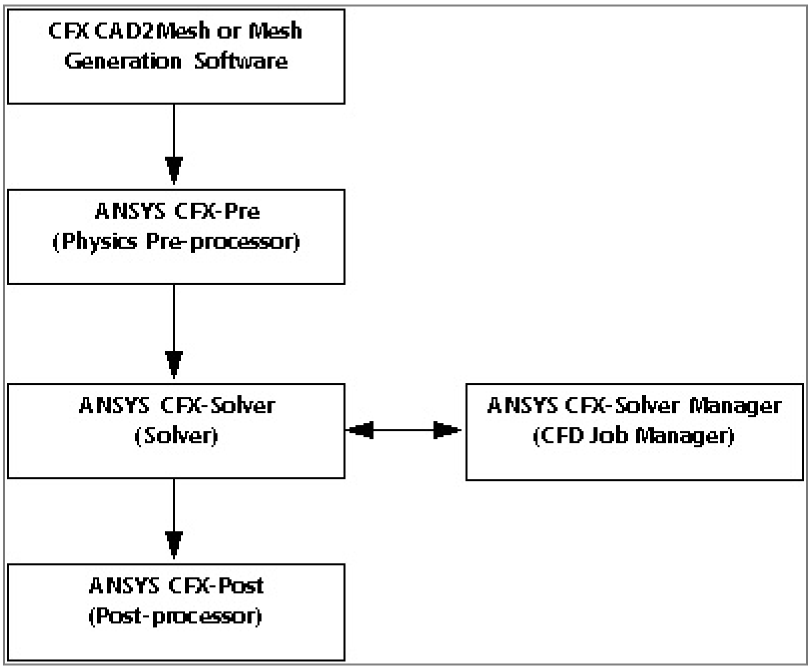


Figure 4.5 The Structure of ANSYS CFX

ANSYS CFX consists of five software modules that pass the information required to perform a CFD analysis.

4.3.1 ANSYS CFX-Mesh

CFX-Mesh is a mesh generator aimed at producing high quality meshes for use in computational fluid dynamics (CFD) simulations. CFD requires meshes that can resolve boundary layer phenomena and satisfy more stringent quality criteria than structural analyses.

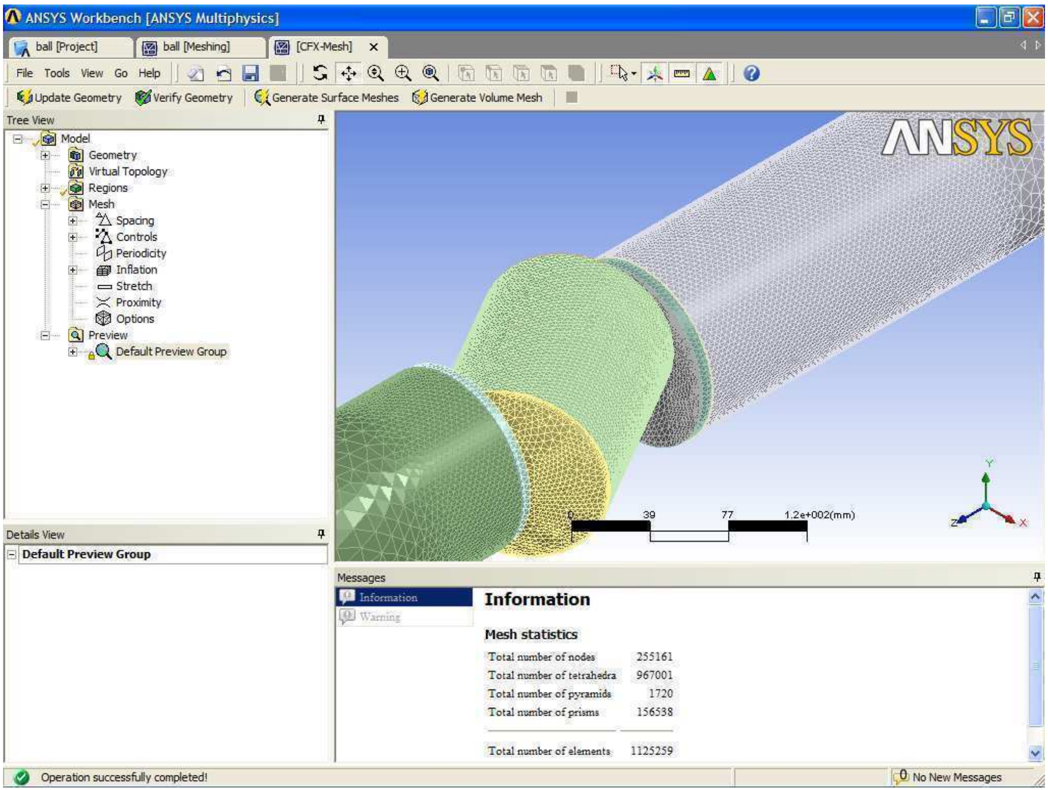


Figure 4.6 CFX Mesh Module Screen

4.3.2 ANSYS CFX-Pre

The next-generation physics pre-processor, ANSYS CFX-Pre, allows multiple meshes to be imported, allowing each section of complex geometries to use the most appropriate mesh.

Flow physics, boundary conditions, initial values and solver parameters are specified in ANSYS CFX-Pre. A full range of boundary conditions, including inlets, outlets and openings, together with boundary conditions for heat transfer models and periodicity.

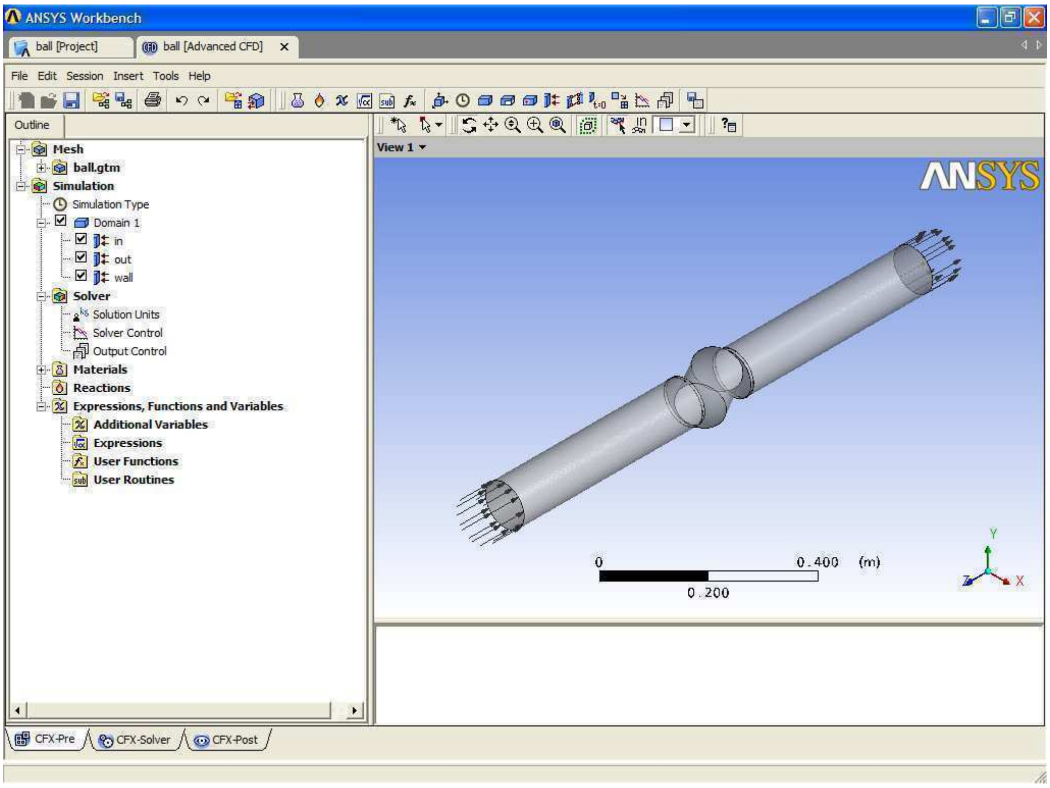


Figure 4.7 CFX-Pre Module Screen

4.3.3 ANSYS CFX-Solver

ANSYS CFX-Solver solves all the solution variables for the simulation for the problem specification generated in ANSYS CFX-Pre.

One of the most important features of ANSYS CFX is its use of a coupled solver, in which all the hydrodynamic equations are solved as a single system. The coupled solver is faster than the traditional segregated solver and fewer iterations are required to obtain a converged flow solution.

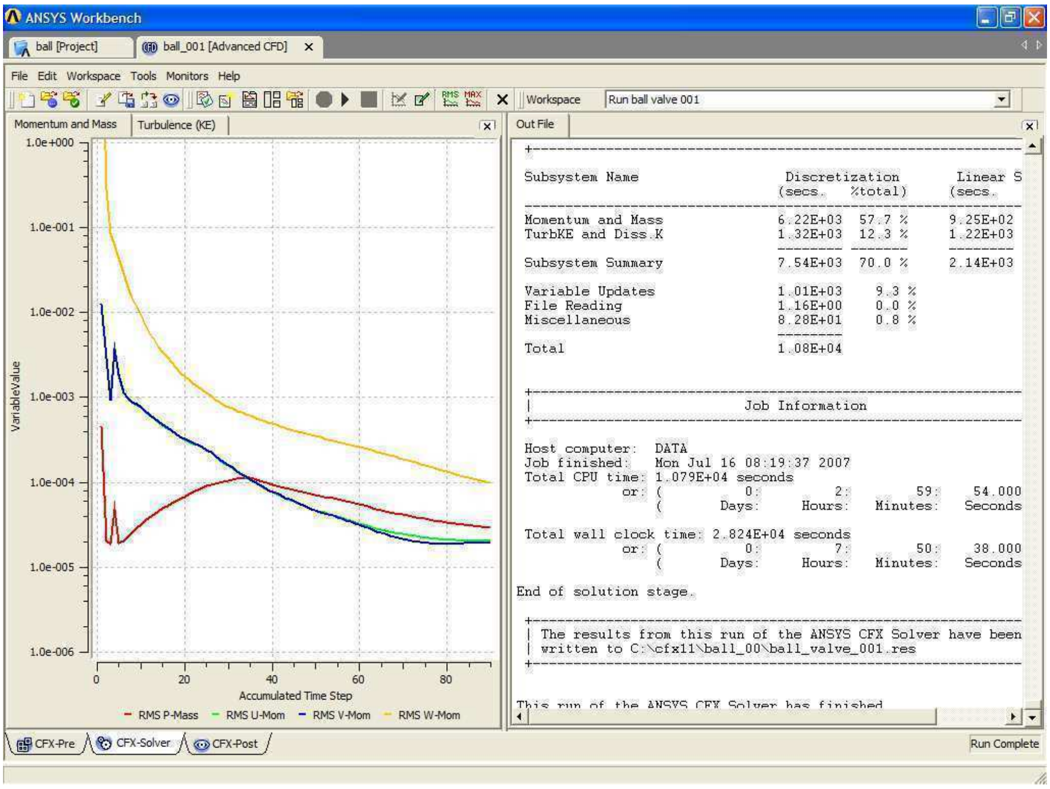


Figure 4.8 CFX-Solver Module Screen

4.3.4 ANSYS CFX-Solver Manager

The ANSYS CFX-Solver Manager module provides greater control to the management of the CFD task. Its major functions are:

- Specify the input files to the ANSYS CFX-Solver.
- Start/stop the ANSYS CFX-Solver.
- Monitor the progress of the solution.
- Set up the ANSYS CFX-Solver for a parallel calculation.

4.3.5 ANSYS CFX-Post

ANSYS CFX-Post provides state-of-the-art interactive post-processing graphics tools to analyze and present the ANSYS CFX simulation results.

Important features include:

- Quantitative post-processing
- Report generation
- Command line, session file or state file input
- User-defined variables
- Generation of a variety of graphical objects where Visibility, Transparency, Color and Line/Face rendering can be controlled
- 'Power Syntax' to allow fully programmable session files

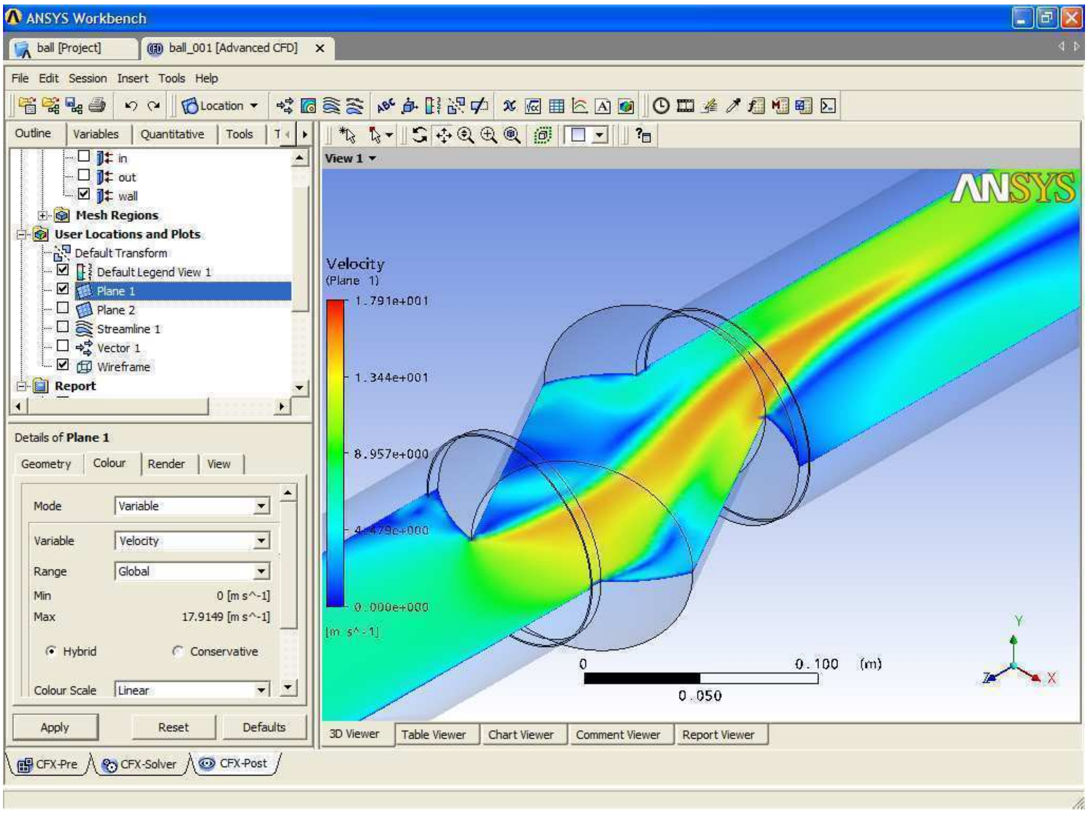


Figure 4.9 CFX-Post Module Screen

CHAPTER FIVE
NUMERICAL STUDY

5.1 Geometry and Mesh

As is well-known, the fluids take the shape of the container they are put in; that’s why, to create the volume of the fluid in the valve, the solid model of the valves were prepared. Three dimensional valve pieces are drawn with the CAD software SolidWorks by using the section drawings taken from the valve producers. These pieces are gathered together with the same software to prepare the solid model of the valves.

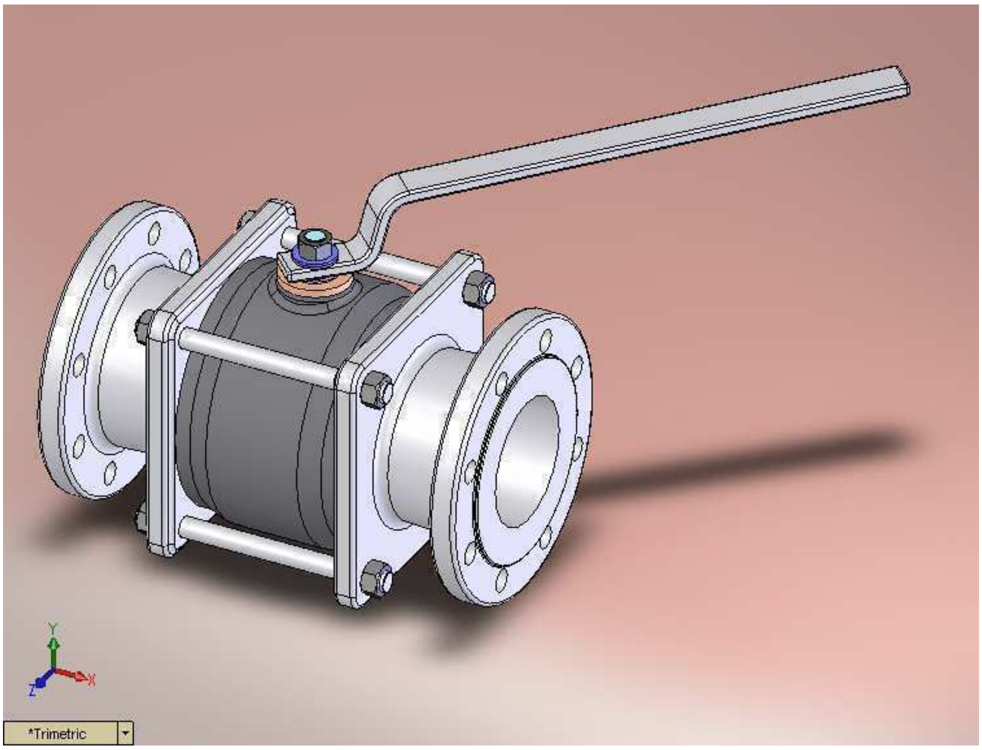


Figure 5.1 Solid model of ball valve

Valve drawings having a diameter of DN 100 were created because butterfly valves cannot be produced with a diameter smaller than DN 40. DN 100 diameter was defined because the disc width in butterfly valves with a small diameter prevents the flow more and the ball valves with a big diameter are rarely produced.

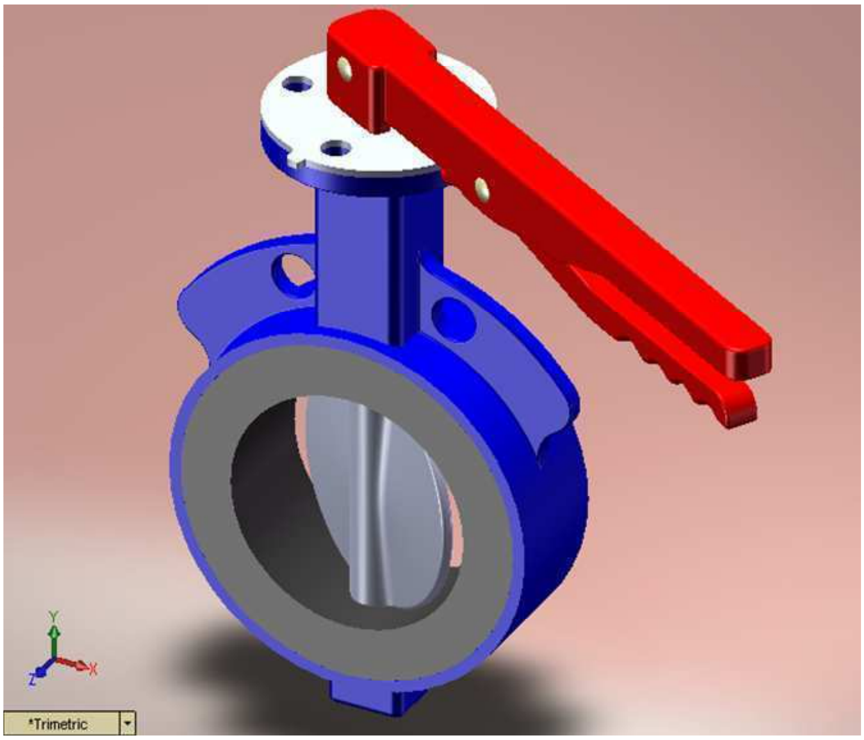


Figure 5.2 Solid model of butterfly valve

The angle between the valve axis with sphere axis and normal of the disc are changed for every 10º to create new geometries. The three dimensional drawings of valves are imported to ANSYS Design Plotter Module. The flow volumes the analysis will be done are created by this programme.

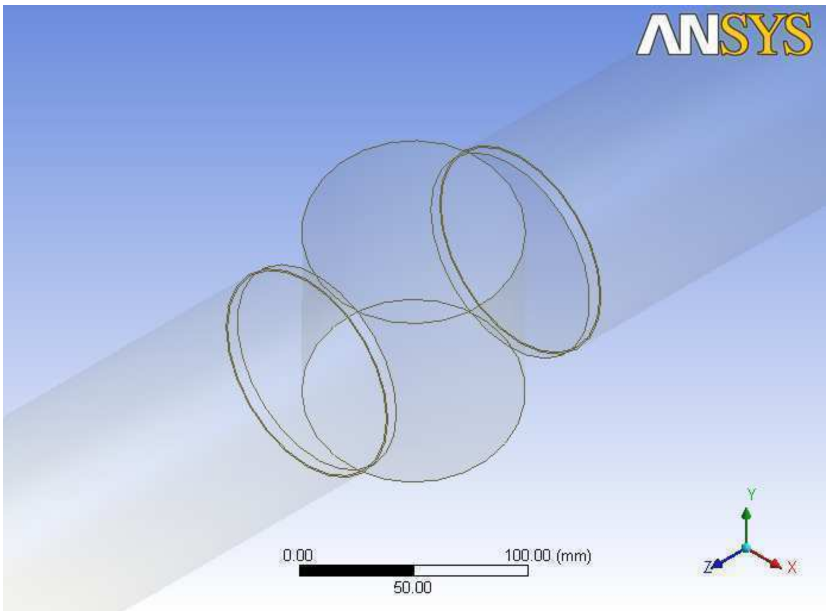


Figure 5.3 Ball valve's flow volume

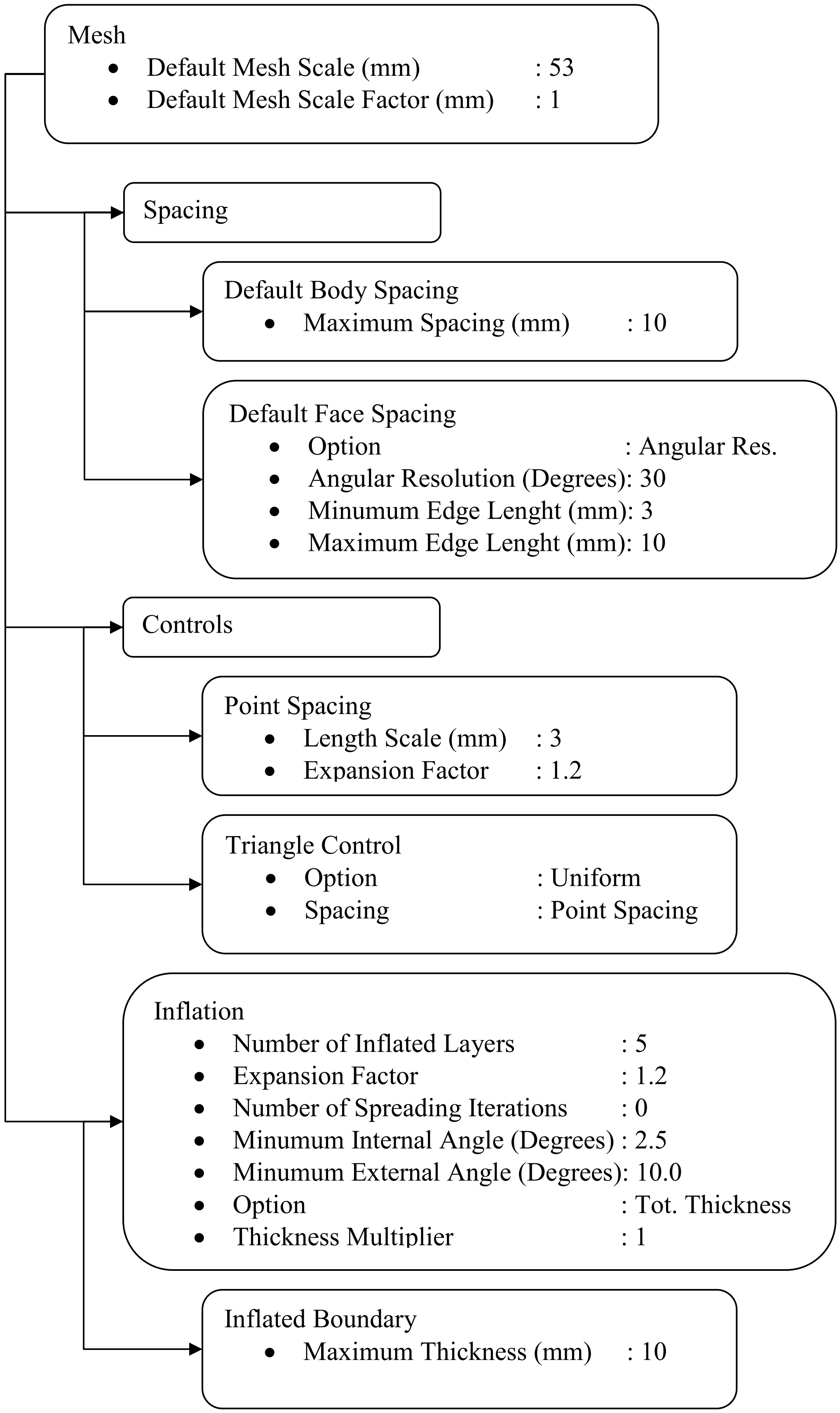


Figure 5.4 Mesh structure

The flow volumes are divided into finite difference elements with the programme ANSYS WORKBENCH CFX Mesh module. The knowledge about the mesh structure are shown in Figure 5.4. Even though it doesn’t affect the results of the analysis much, the mesh sizes around the smaller ball in ball valves, around the disc in butterfly valves and after the downstream of the valve are chosen in small scale.

The aim is to provide better quality in the visual scenes taken on the valves. When the mesh statistics are analyzed, it was seen that more than 1 000 000 finite elements are created in every model.

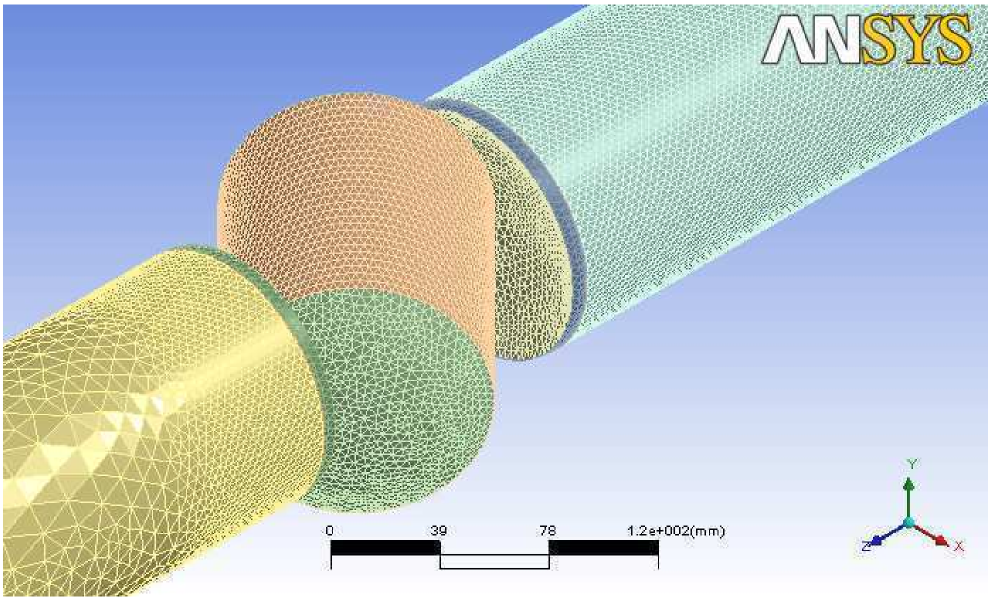


Figure 5.5 Ball valve’s flow volume mesh

5.2 Boundary Conditions

The mesh file, ANSYS CFX can recognize, is opened in the programme CFX and the boundary conditions of the problem are defined in CFX-Pre module. Water is chosen as the fluid for the domain. The reference pressure and temperature were chosen as the standard values of 1 atm and 25°C. The flow is assumed to be steady and incompressible. K-epsilon model was chosen as the turbulence model. One of the most prominent turbulence models, the *k*-epsilon model, has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the k-epsilon model offers a good compromise in terms of accuracy and robustness.

Within ANSYS CFX, the k-epsilon turbulence model uses the scalable wall-function approach to improve robustness and accuracy when the near-wall mesh is very fine. The scalable wall functions allow solution on arbitrarily fine near wall grids, which is a significant improvement over standard wall functions.

Due to the definition of Kv value, the static pressure at the inlet of the valve was taken as 2 bars and at the outlet, it was taken as 1 bar. Thus, the pressure difference was taken as ΔP=1 bar.

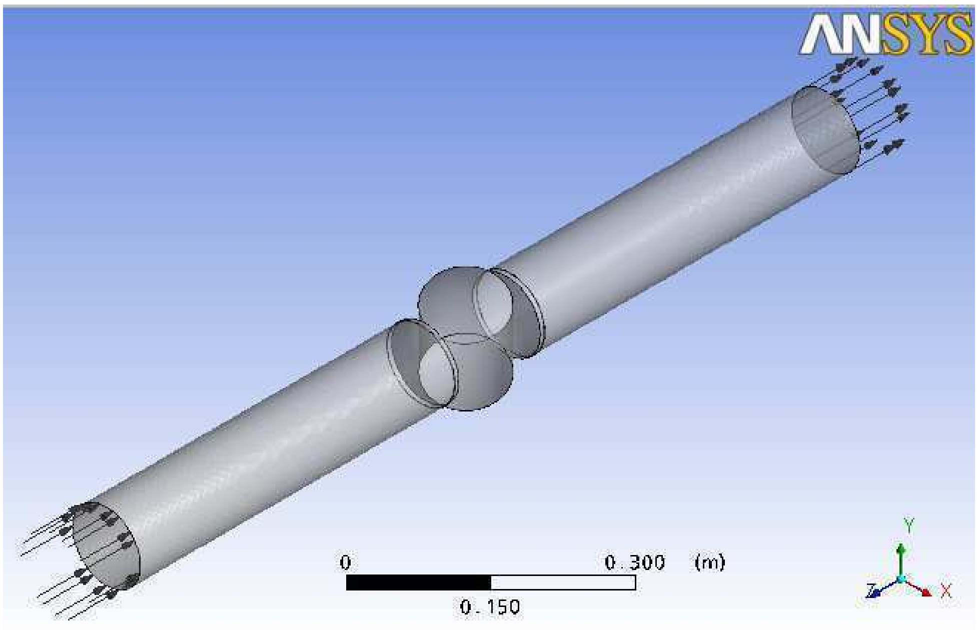


Figure 5.6 Ball valve's boundary conditions

The problem created in defining the other boundary conditions is solved in CFX Solver module. The problem solved is operated by CFX-Post module to create the streamlines, the velocity and the pressure values visually. The flow rate values for valves having different angles are solved via this module.

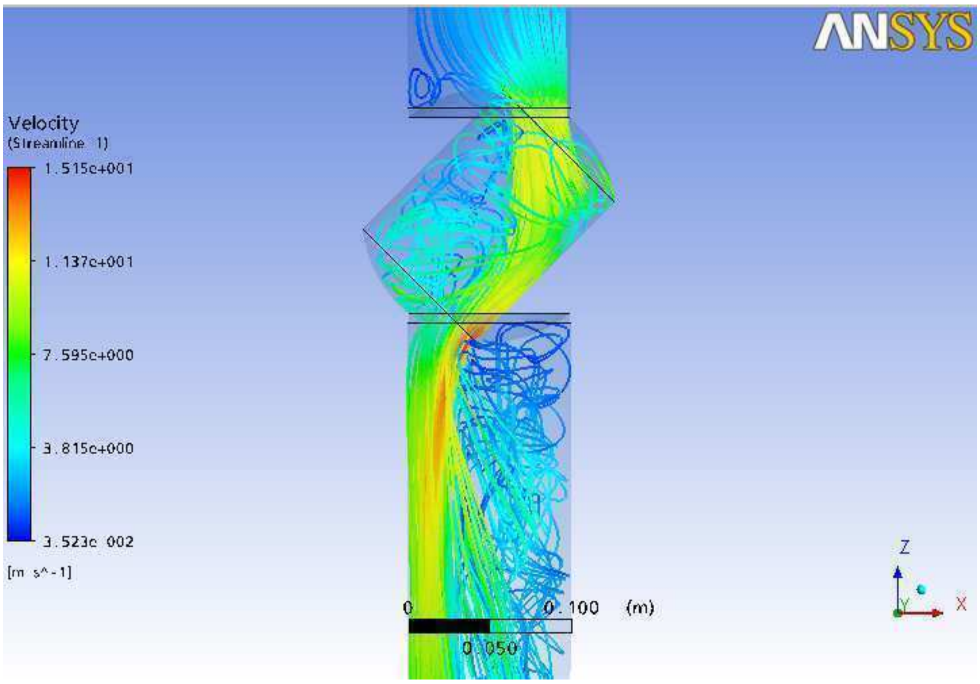


Figure 5.7 +Y view of flow velocity on streamline at 45^0 of valve angle

The angles of the ball and discs at the valve geometries were taken between 20° up to the fully open position which is 90° with a difference of 10°s each. The same boundary values were used in every model prepared and ANSYS CFX Solver Module was used to gain the results. Figures showing streamlines and speed values were created on the modules solved by ANSYS Post processor. The flows of the valves were calculated by the function “massFlow” in the part “Table Viewer”.

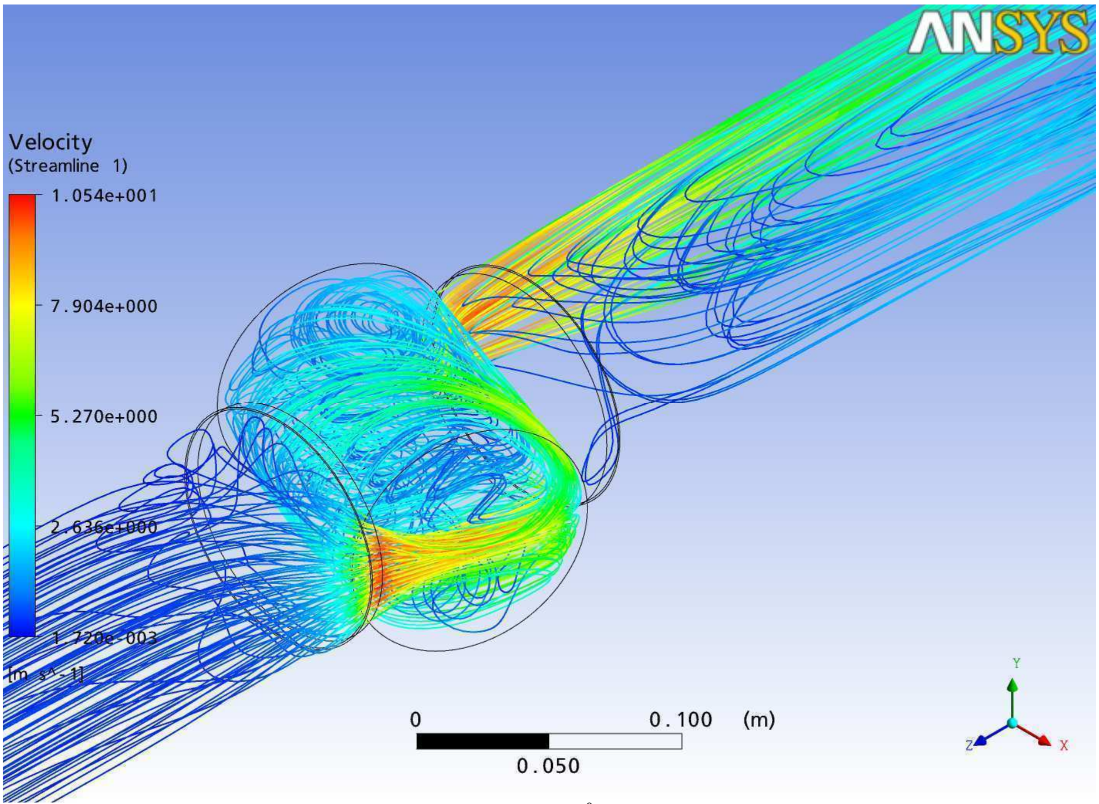


Figure 5.8 Results of flow velocity on streamline at 20^0 of ball valve angle

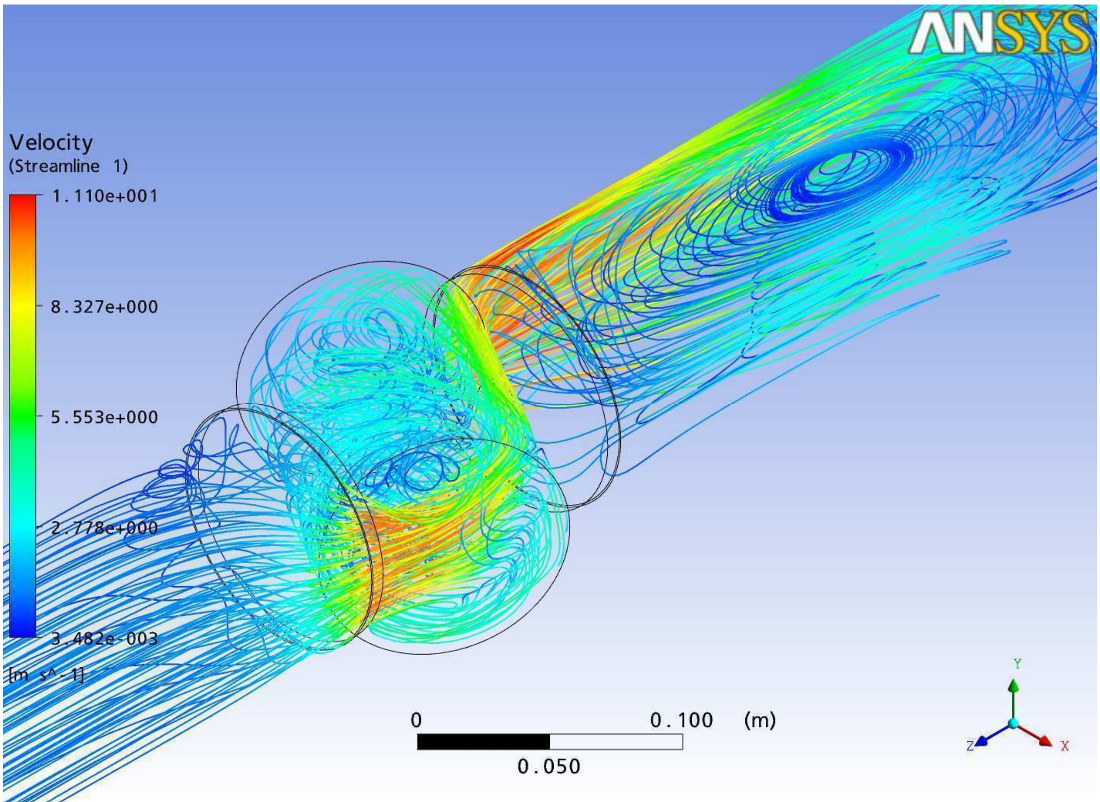


Figure 5.9 Results of flow velocity on streamline at 30^0 of ball valve angle

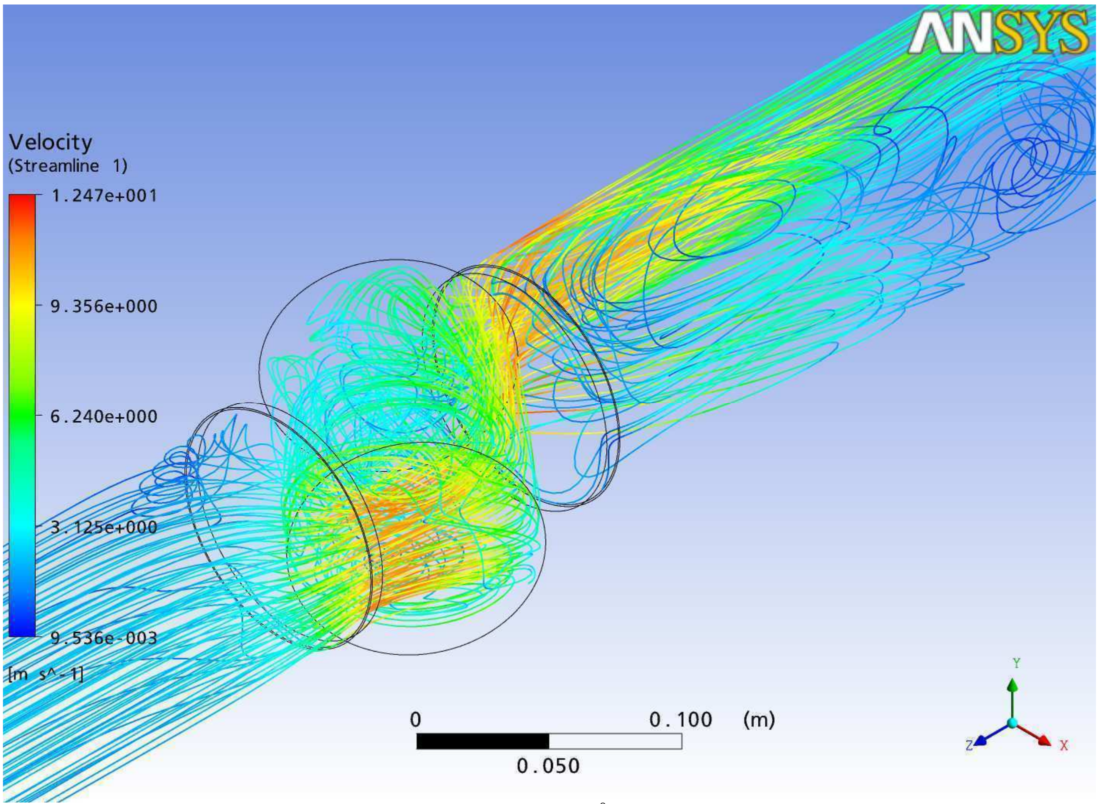


Figure 5.10 Results of flow velocity on streamline at 40^0 of ball valve angle

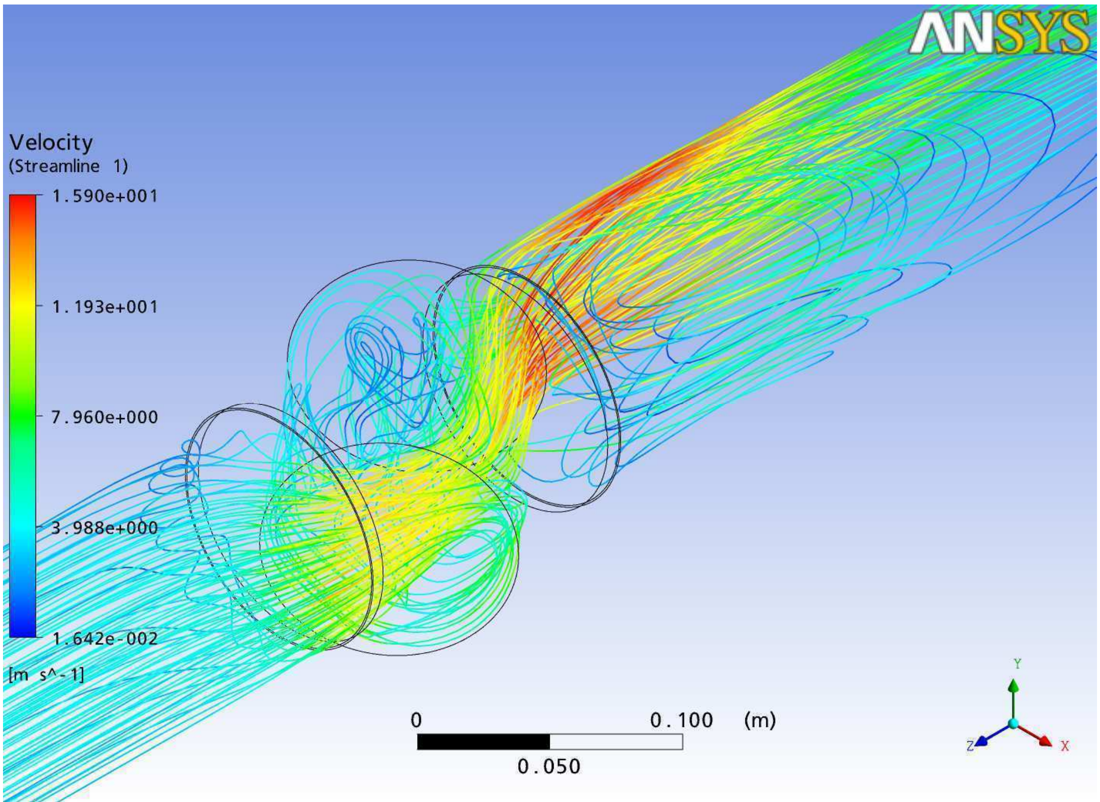


Figure 5.11 Results of flow velocity on streamline at 50^0 of ball valve angle

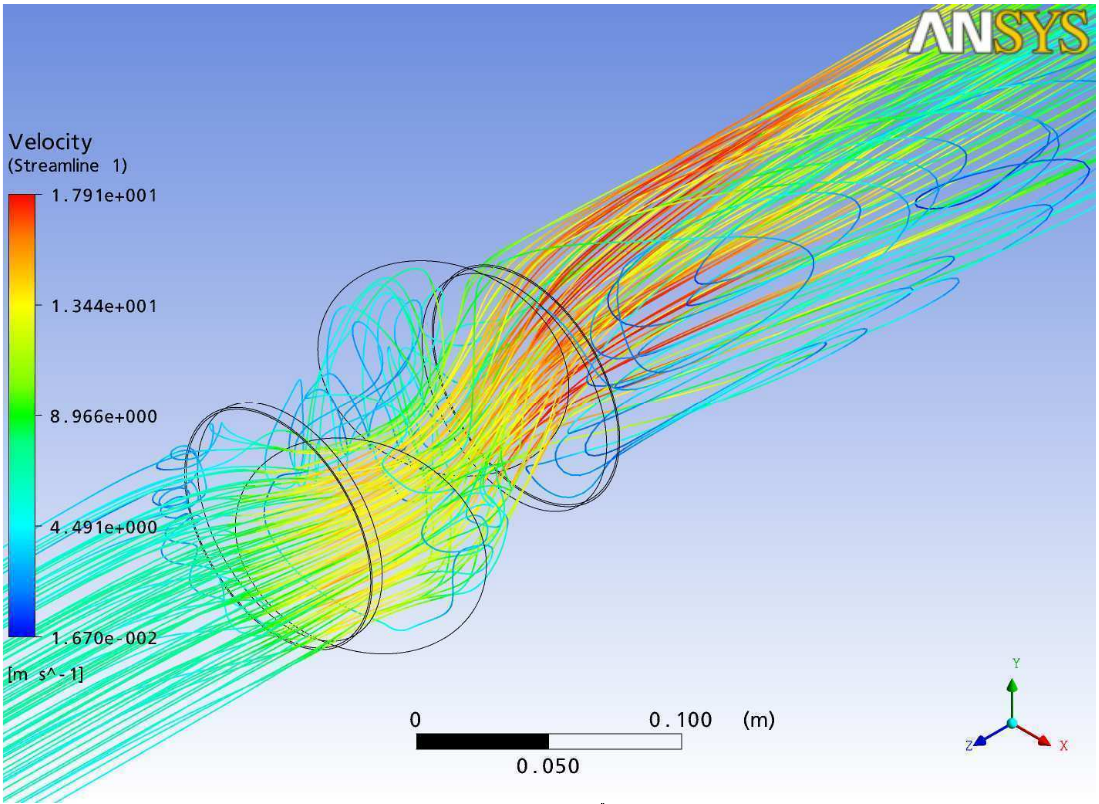


Figure 5.12 Results of flow velocity on streamline at 60^0 of ball valve angle

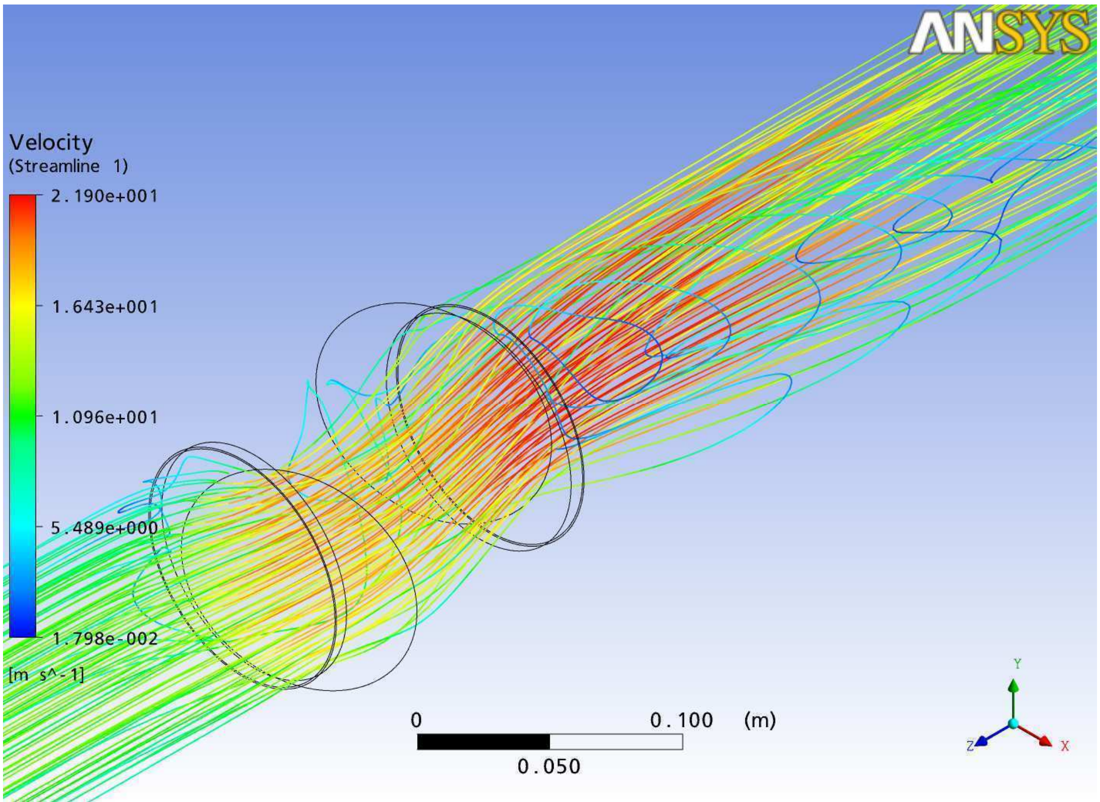


Figure 5.13 Results of flow velocity on streamline at 70^0 of ball valve angle

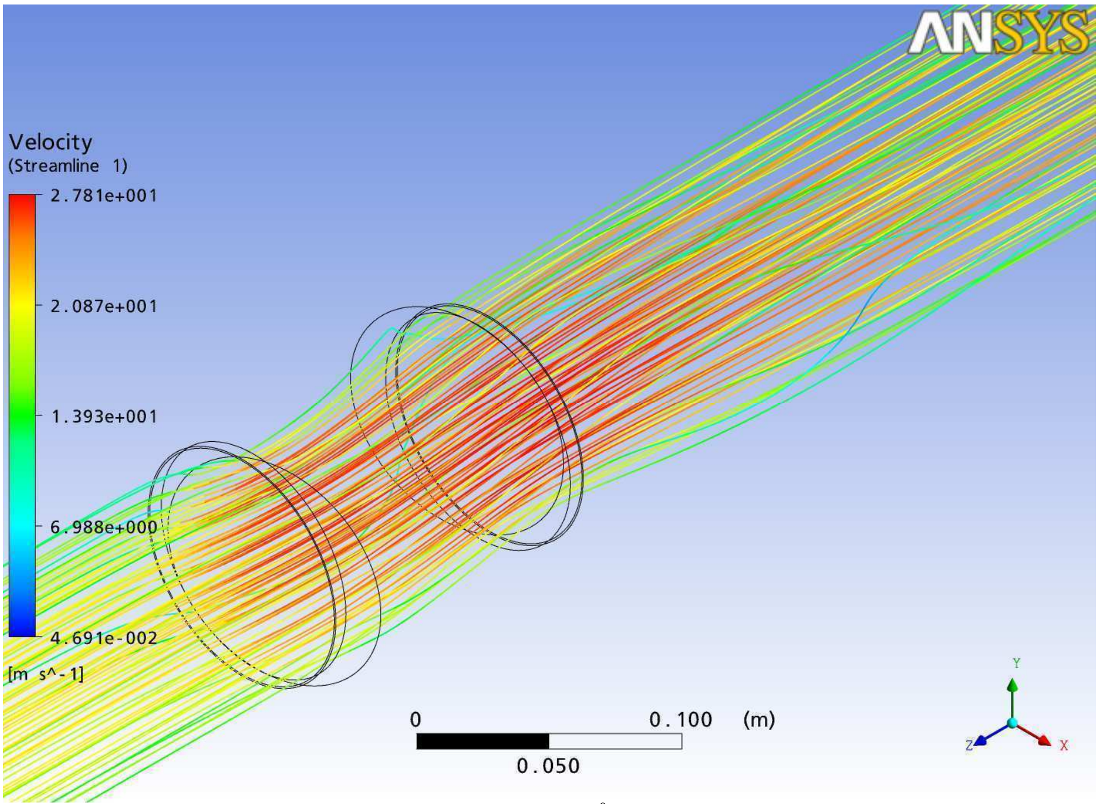


Figure 5.14 Results of flow velocity on streamline at 80^0 of ball valve angle

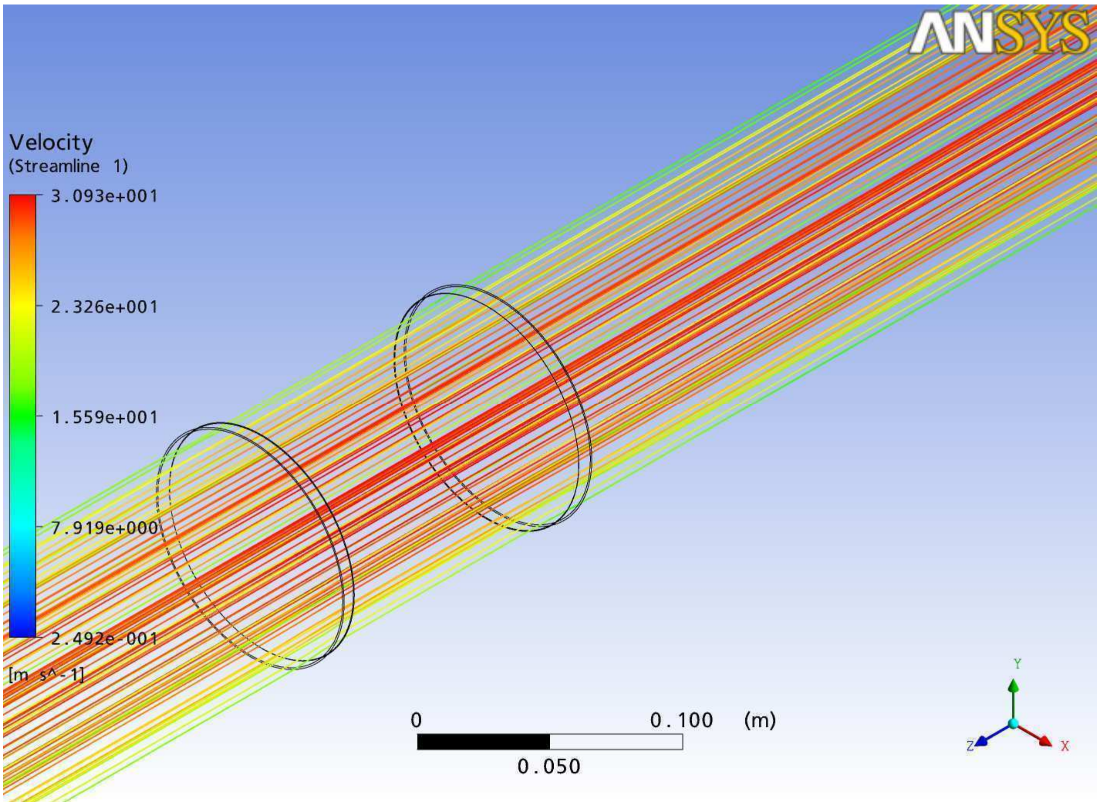


Figure 5.15 Results of flow velocity on streamline at 90^0 of ball valve angle

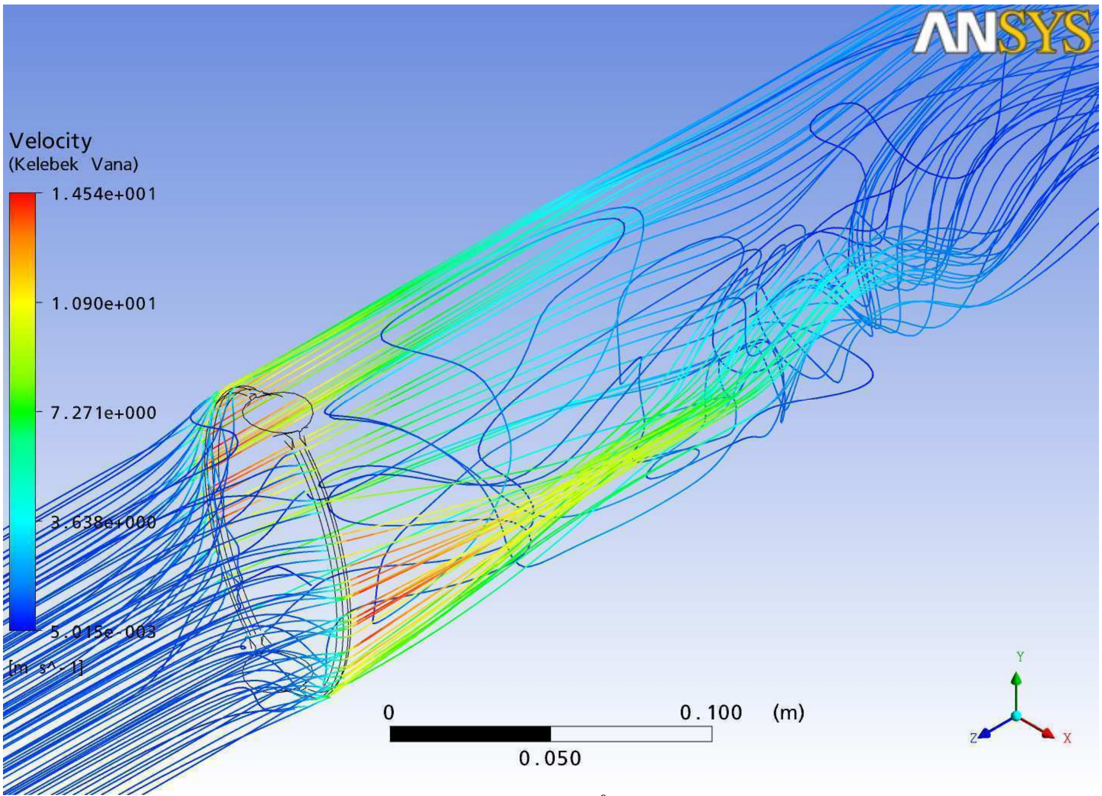


Figure 5.16 Results of flow velocity on streamline at 20^0 of butterfly valve angle

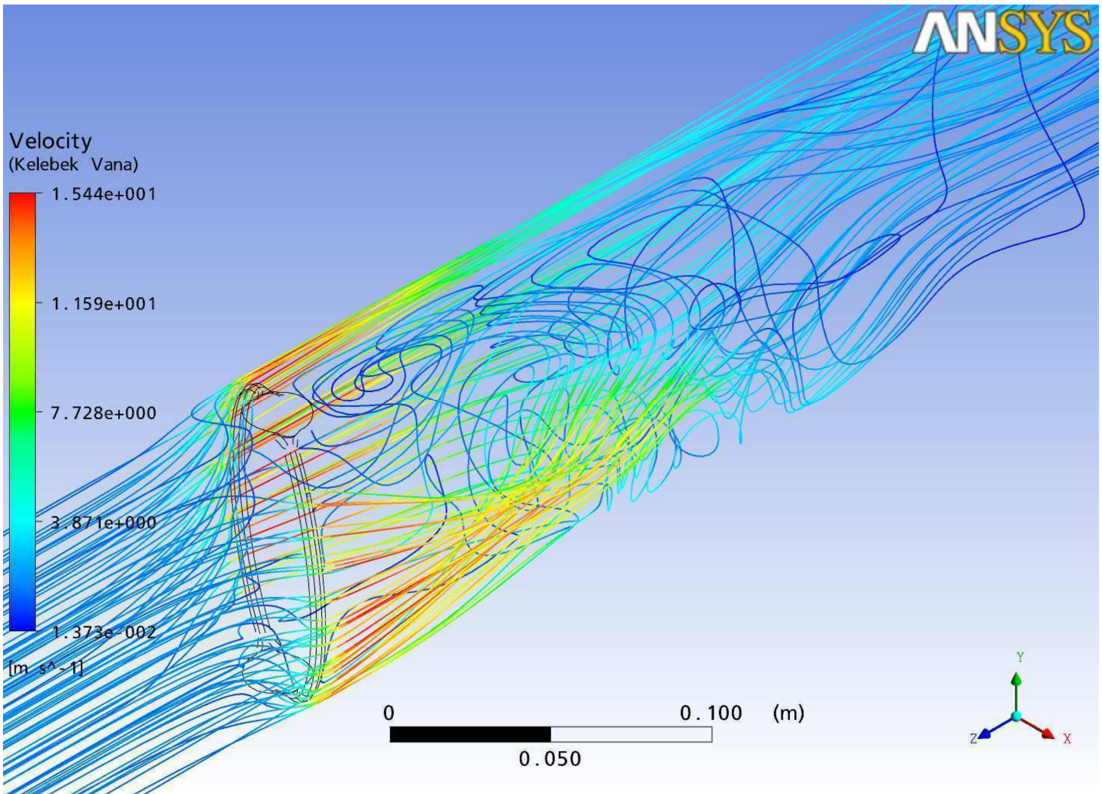


Figure 5.17 Results of flow velocity on streamline at 30^0 of butterfly valve angle

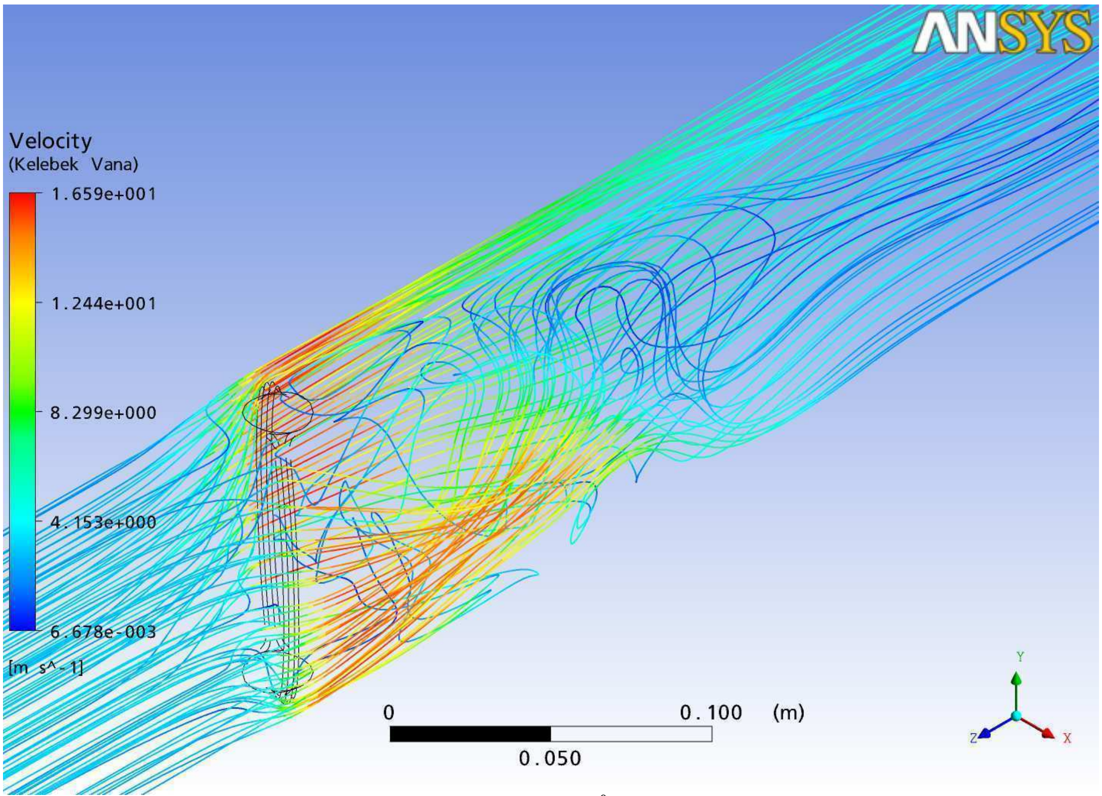


Figure 5.18 Results of flow velocity on streamline at 40^0 of butterfly valve angle

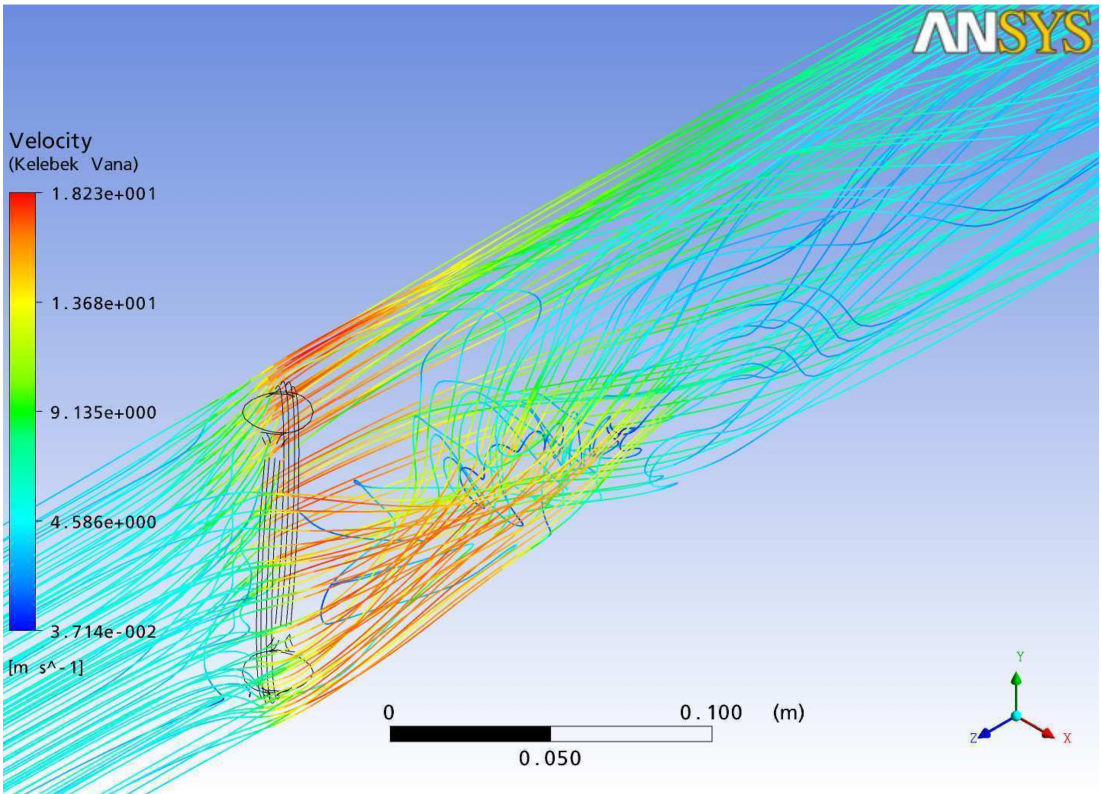


Figure 5.19 Results of flow velocity on streamline at 50^0 of butterfly valve angle

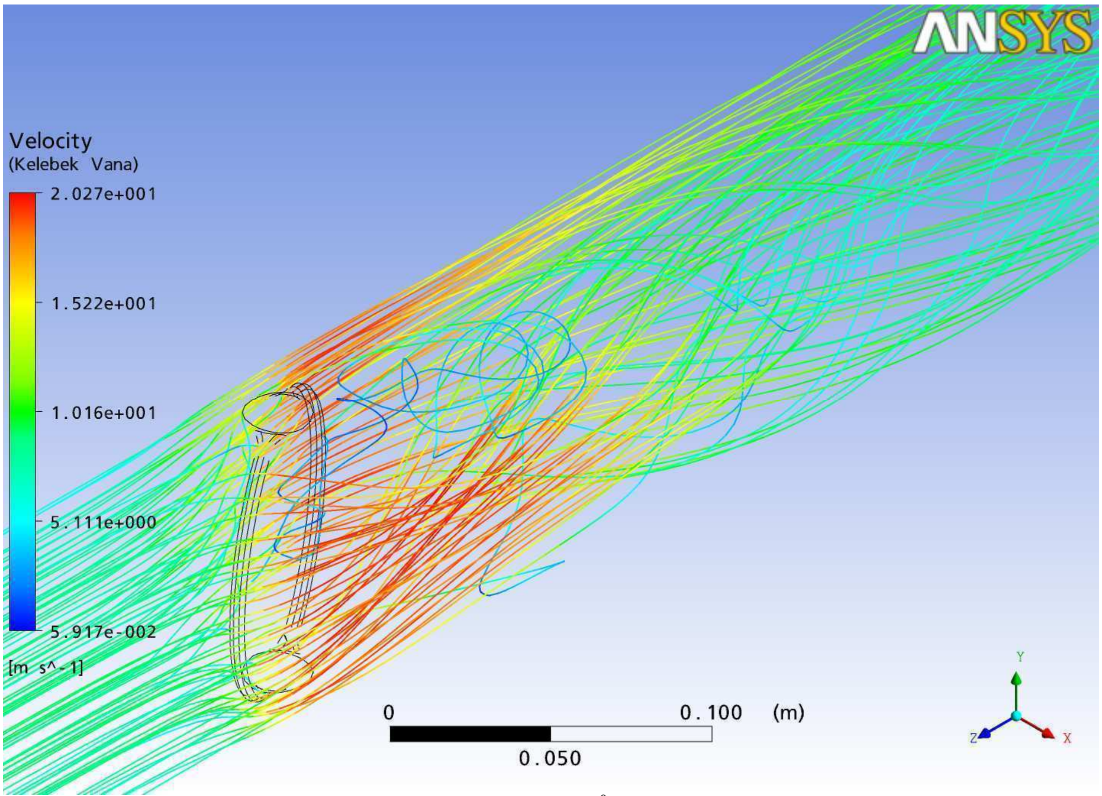


Figure 5.20 Results of flow velocity on streamline at 60^0 of butterfly valve angle

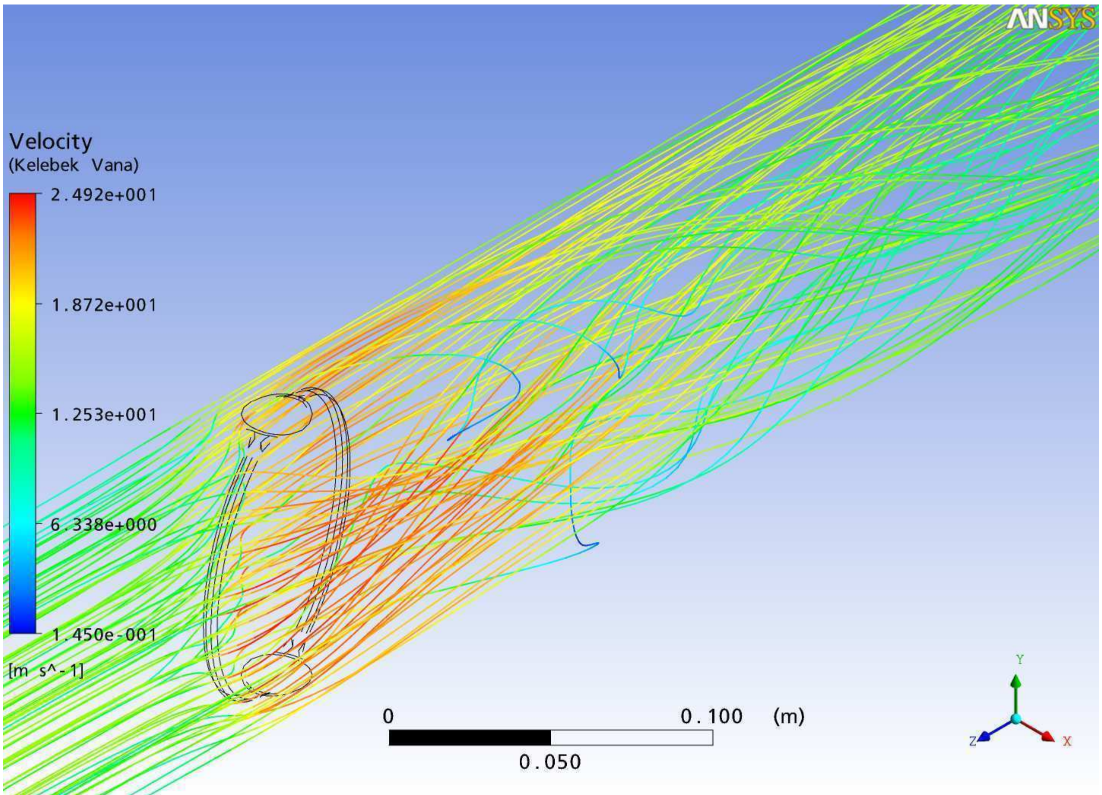


Figure 5.21 Results of flow velocity on streamline at 70^0 of butterfly valve angle

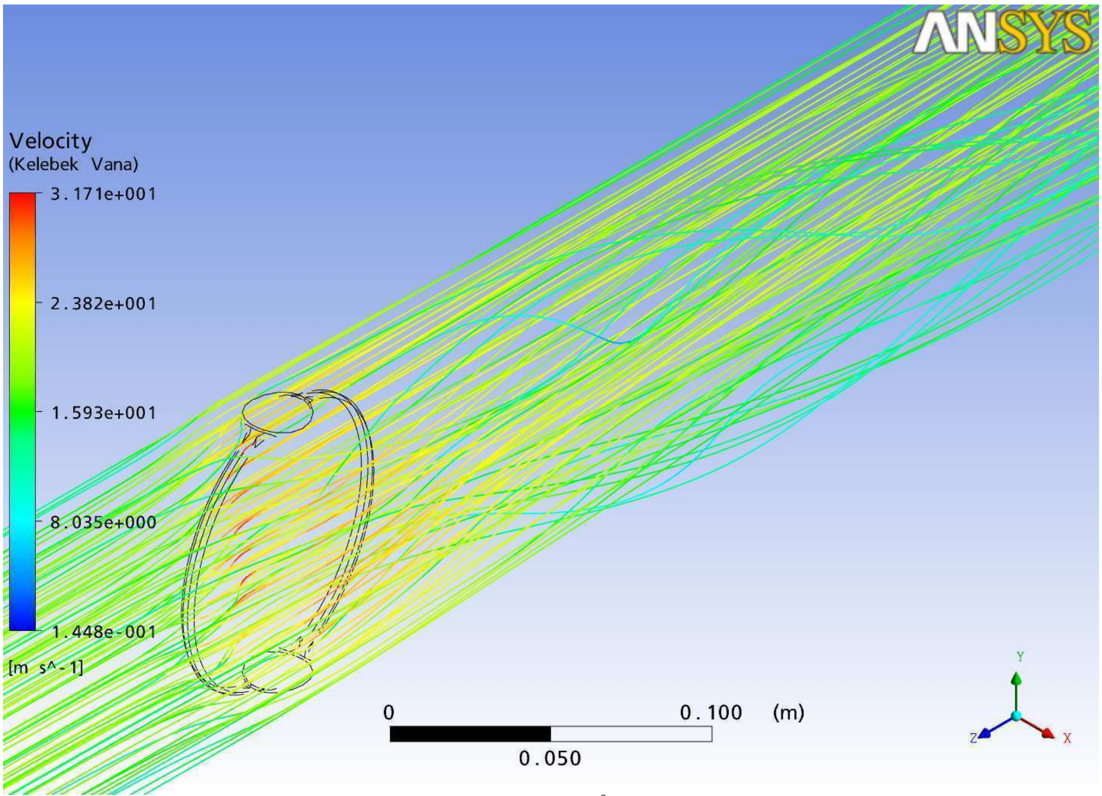


Figure 5.22 Results of flow velocity on streamline at 80^0 of butterfly valve angle

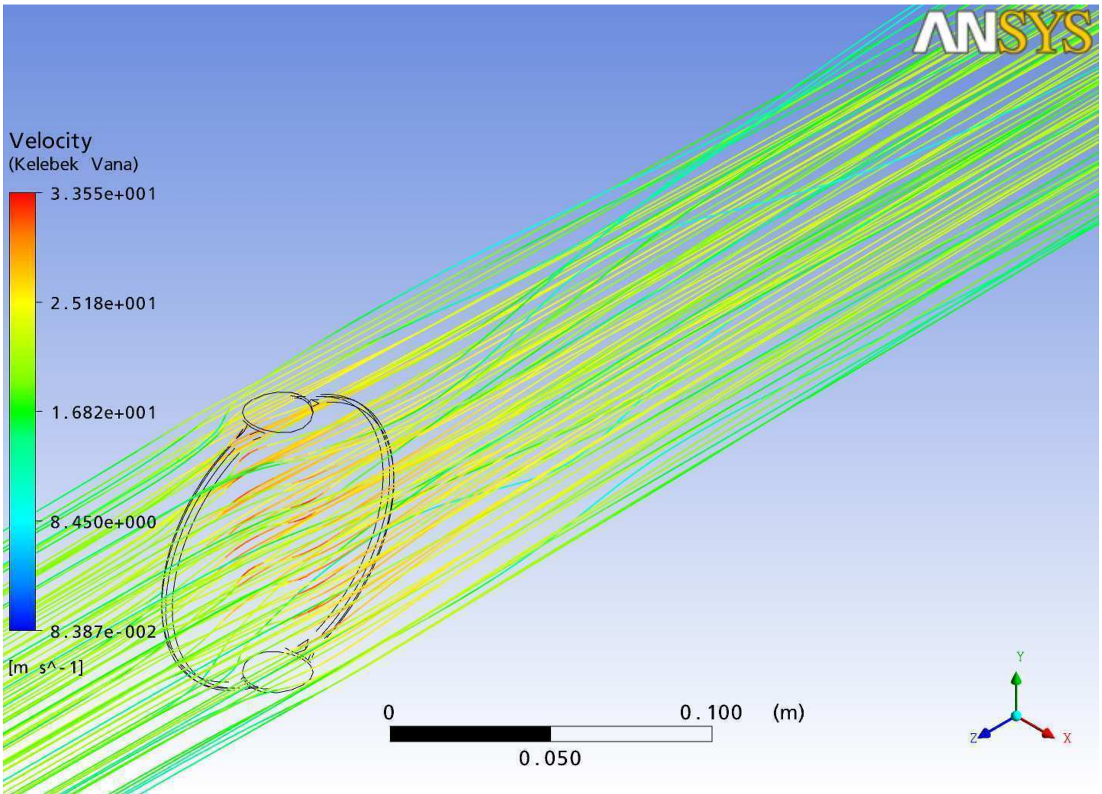


Figure 5.23 Results of flow velocity on streamline at 90^0 of butterfly valve angle

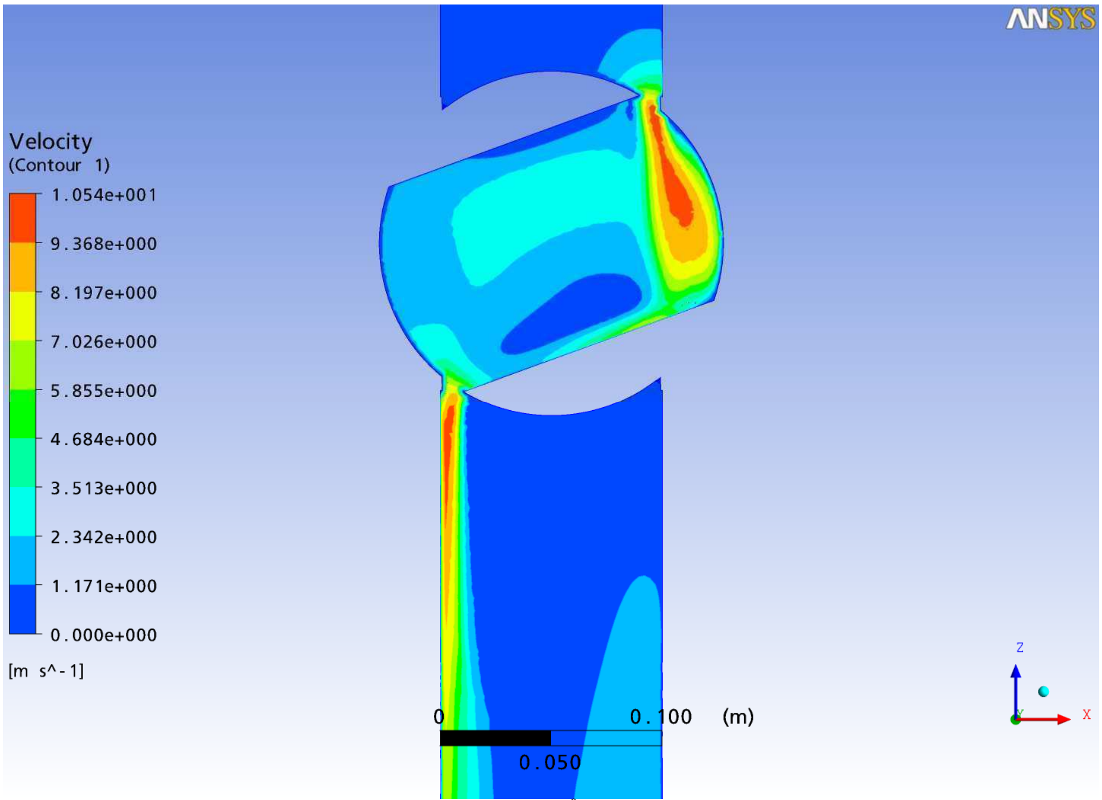


Figure 5.24 Results of flow velocity on contour at 20^0 of ball valve angle

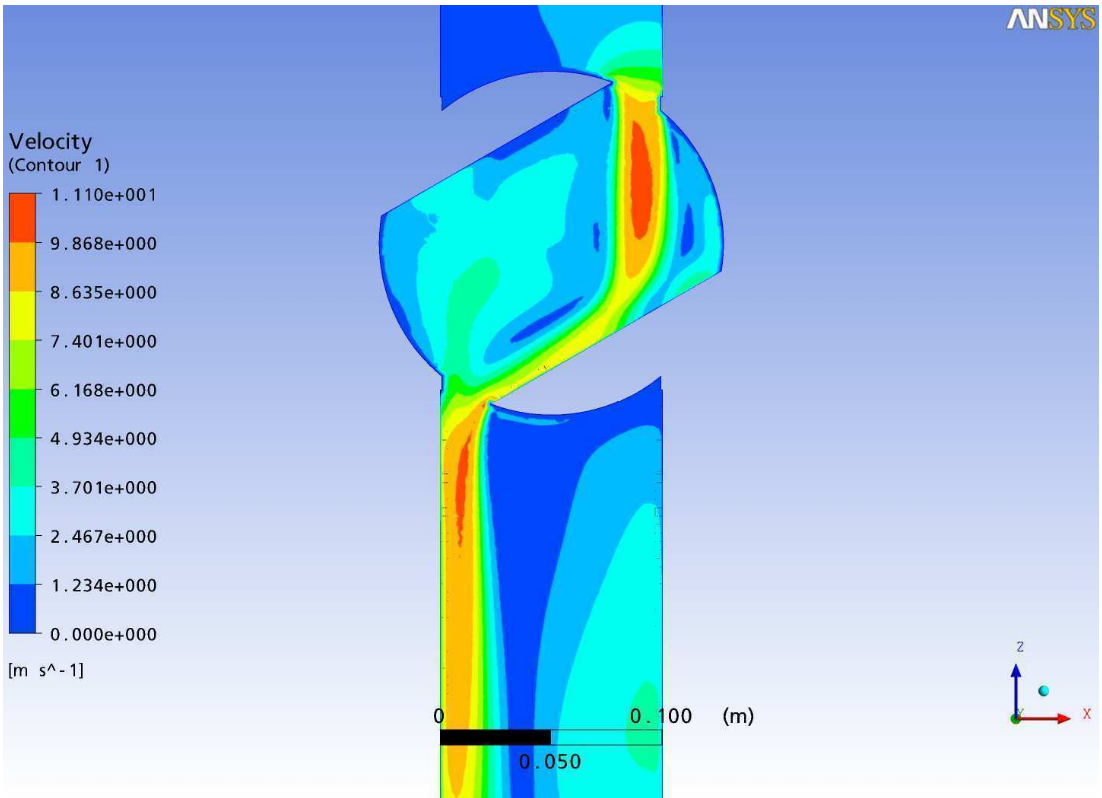


Figure 5.25 Results of flow velocity on contour at 30^0 of ball valve angle

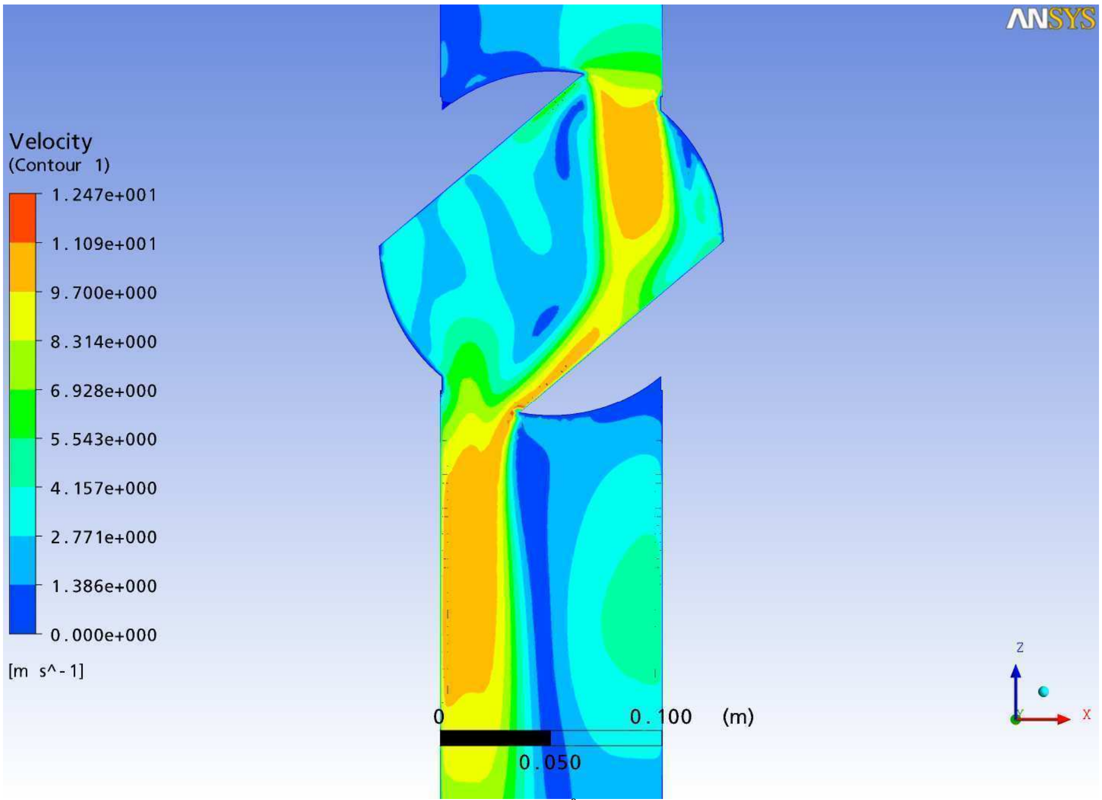


Figure 5.26 Results of flow velocity on contour at 40^0 of ball valve angle

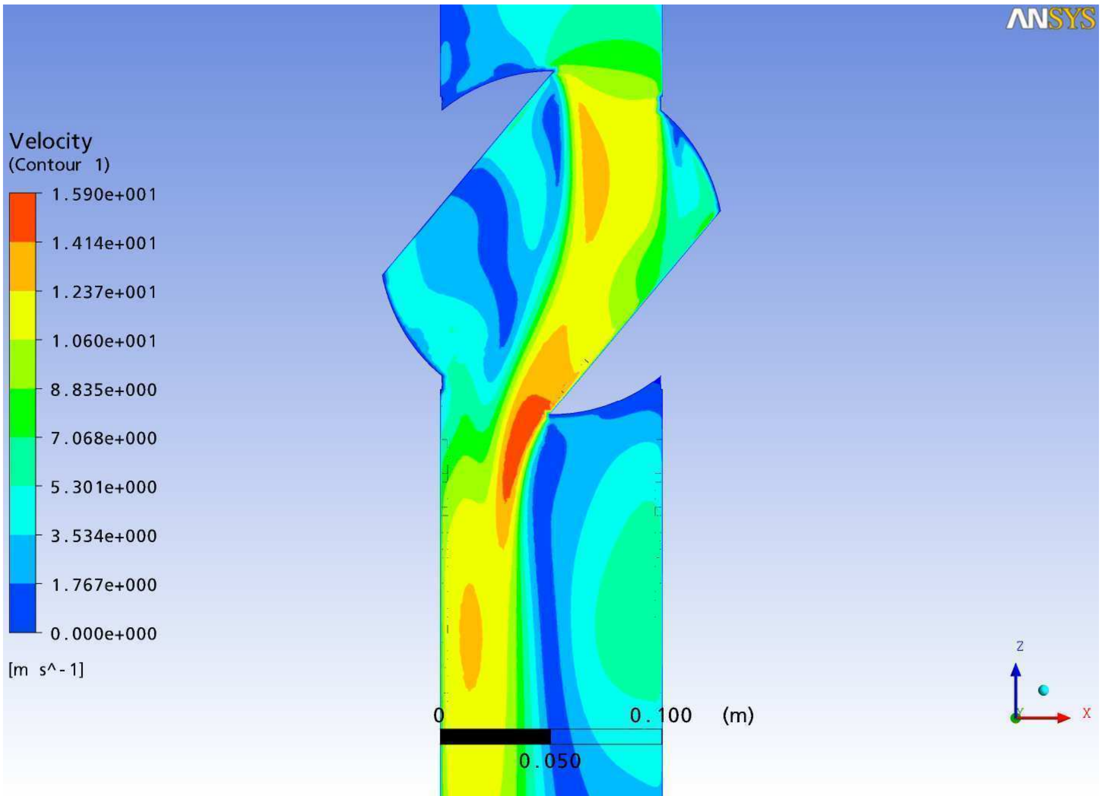


Figure 5.27 Results of flow velocity on contour at 50^0 of ball valve angle

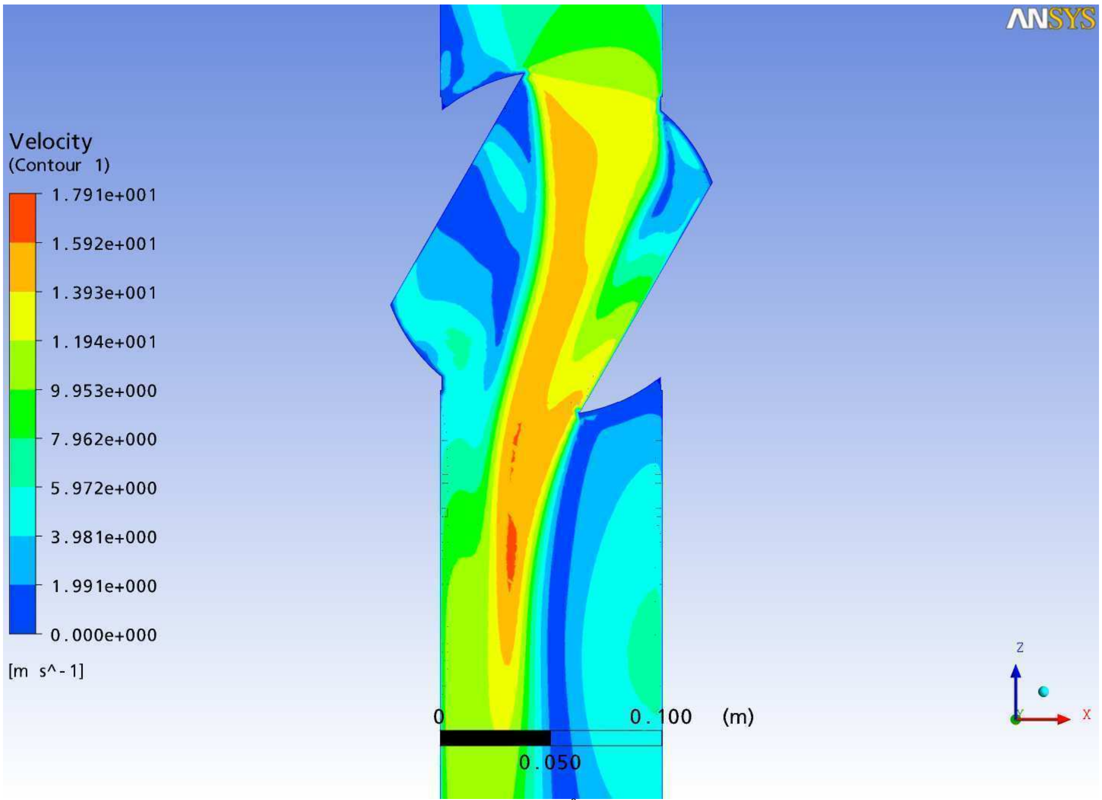


Figure 5.28 Results of flow velocity on contour at 60^0 of ball valve angle

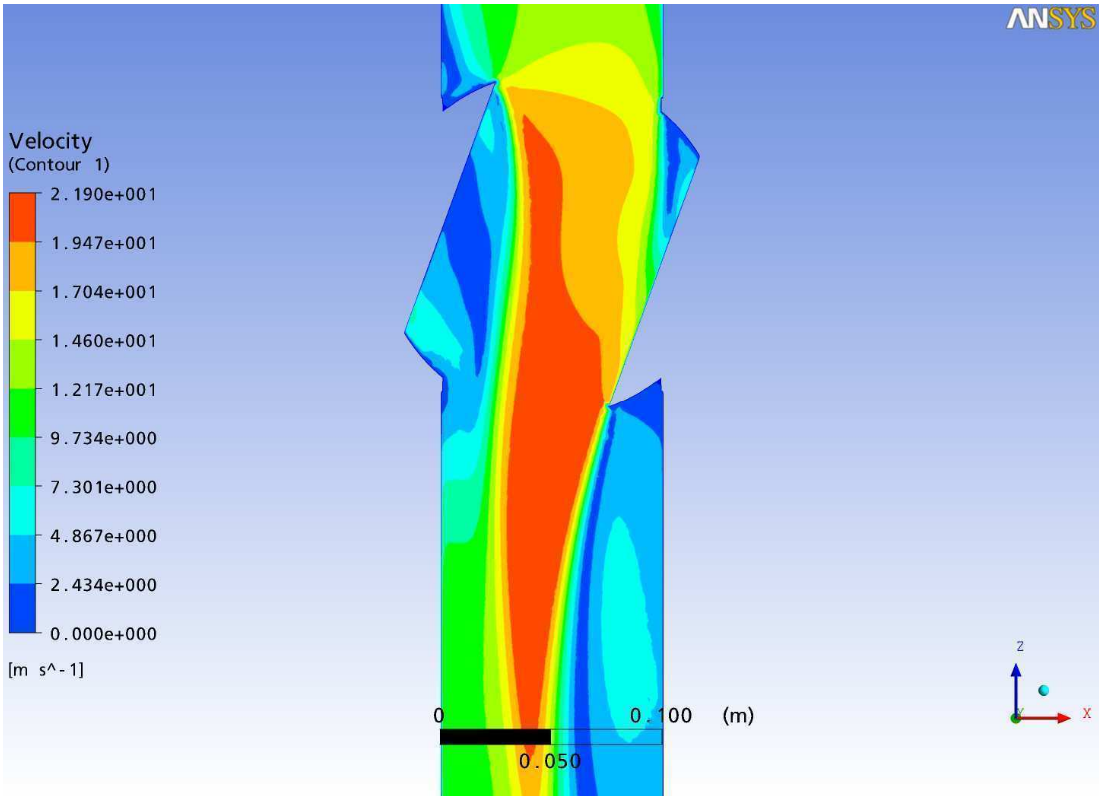


Figure 5.29 Results of flow velocity on contour at 70^0 of ball valve angle

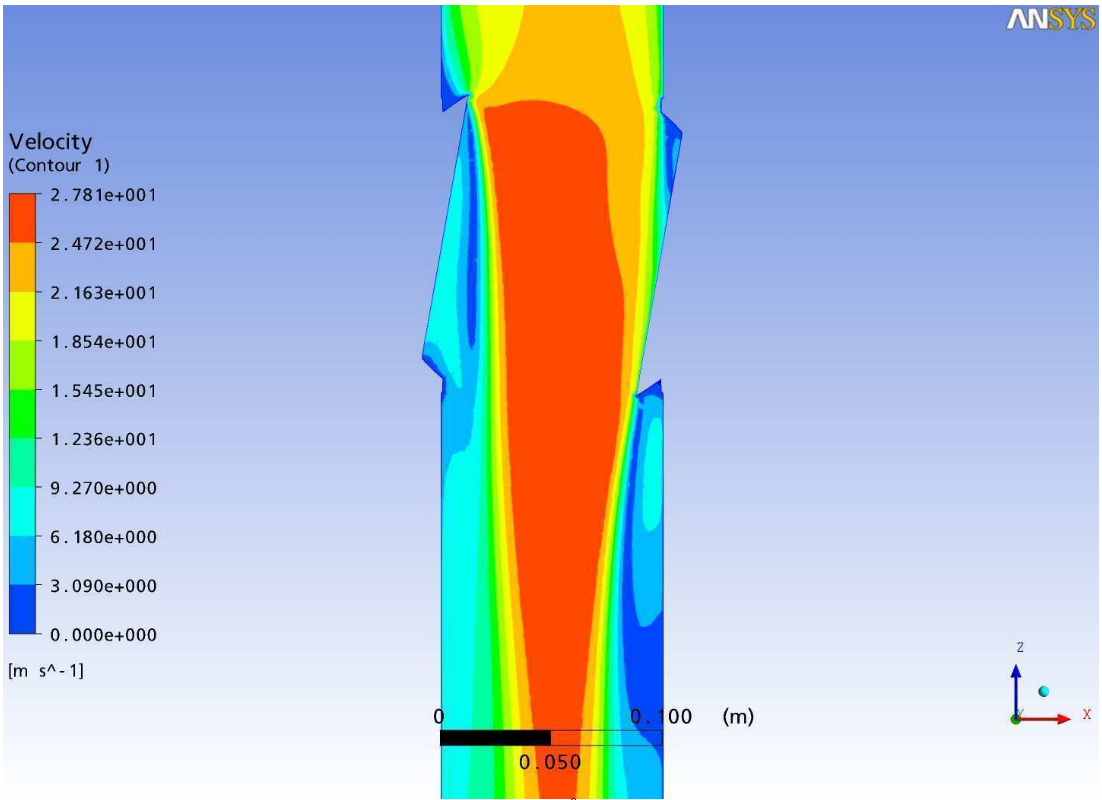


Figure 5.30 Results of flow velocity on contour at 80^0 of ball valve angle

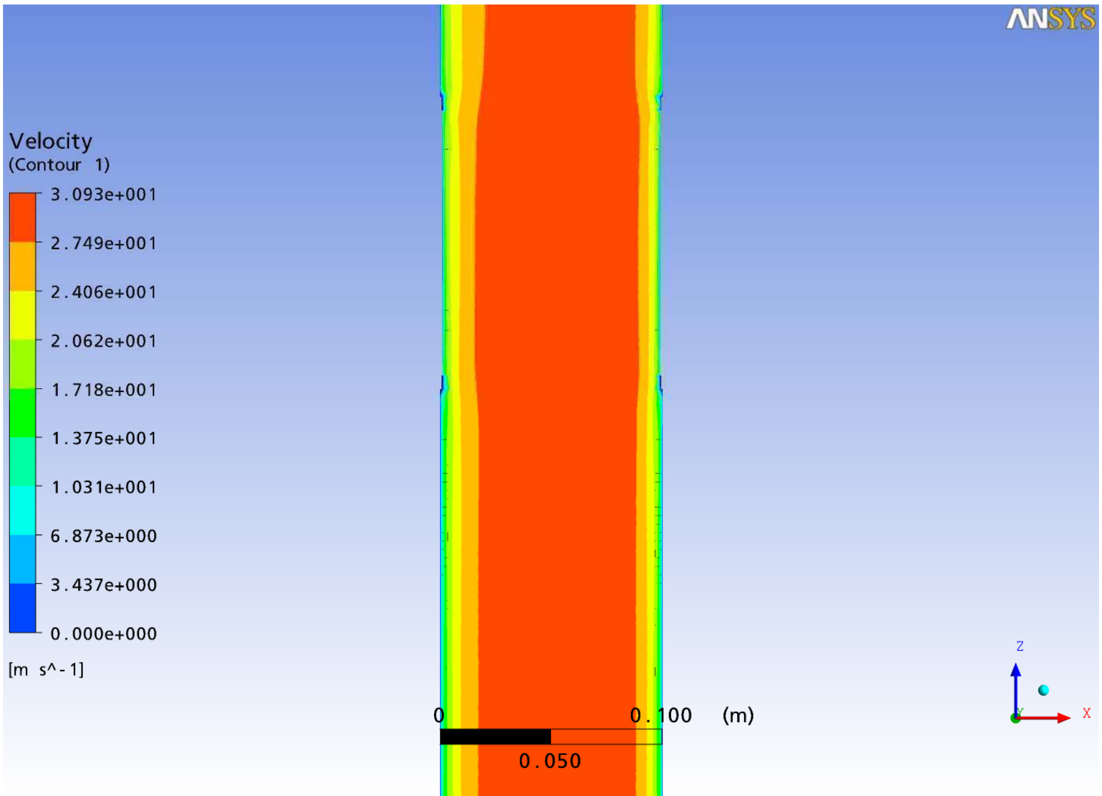


Figure 5.31 Results of flow velocity on contour at 90^0 of ball valve angle

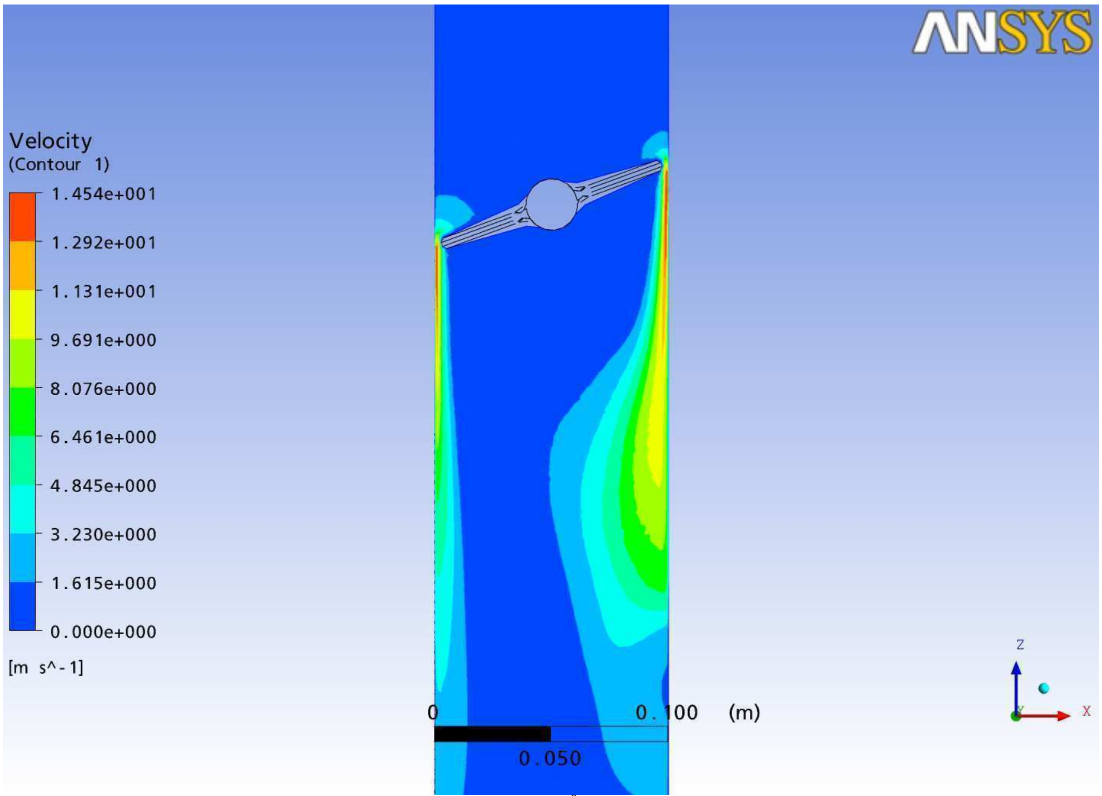


Figure 5.32 Results of flow velocity on contour at 20^0 of butterfly valve angle

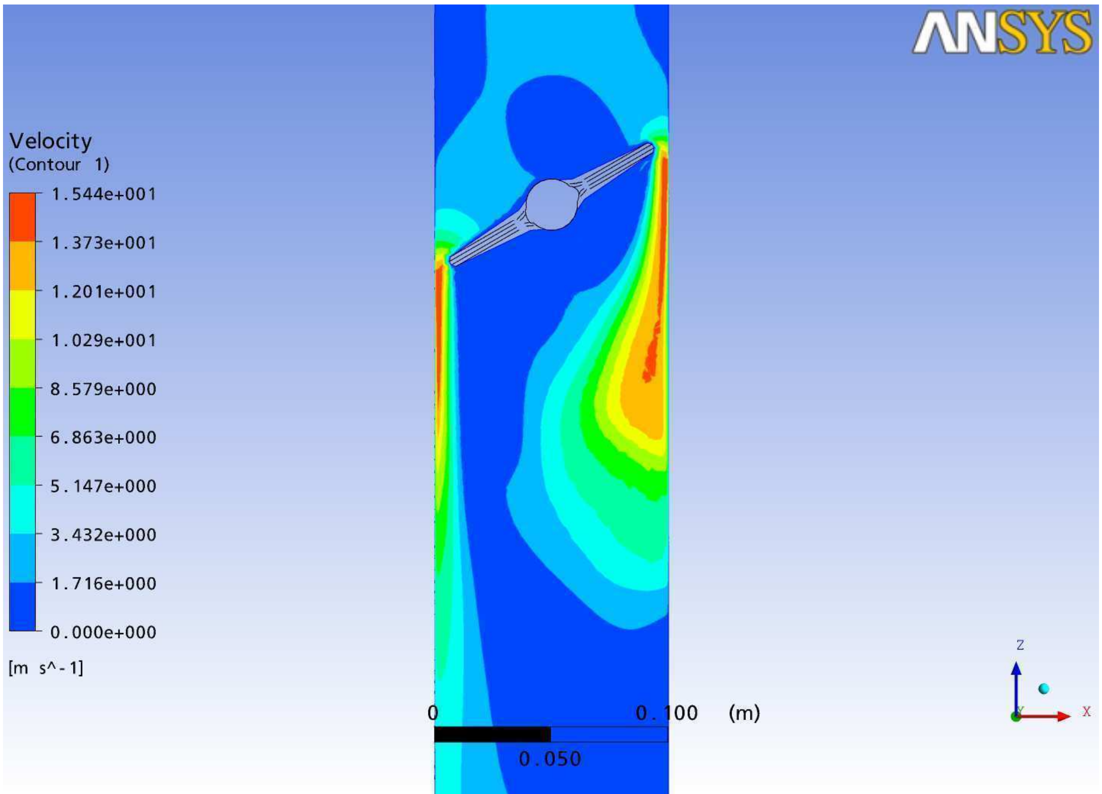


Figure 5.33 Results of flow velocity on contour at 30^0 of butterfly valve angle

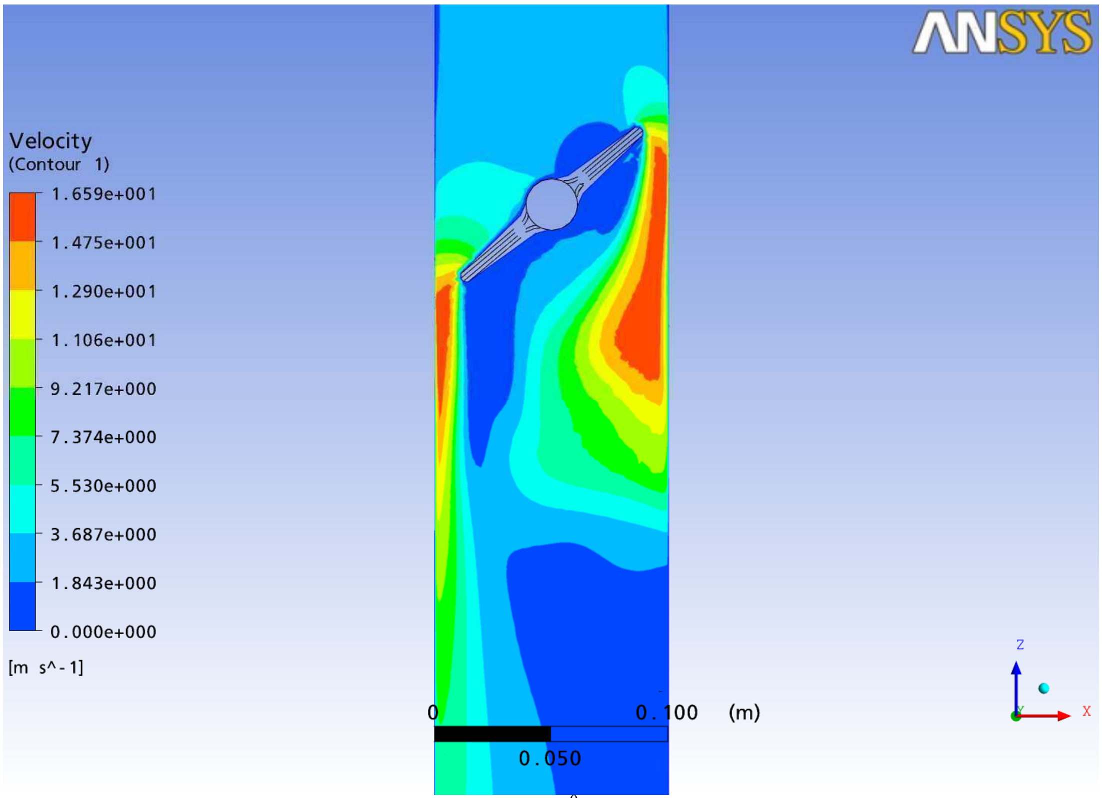


Figure 5.34 Results of flow velocity on contour at 40^0 of butterfly valve angle

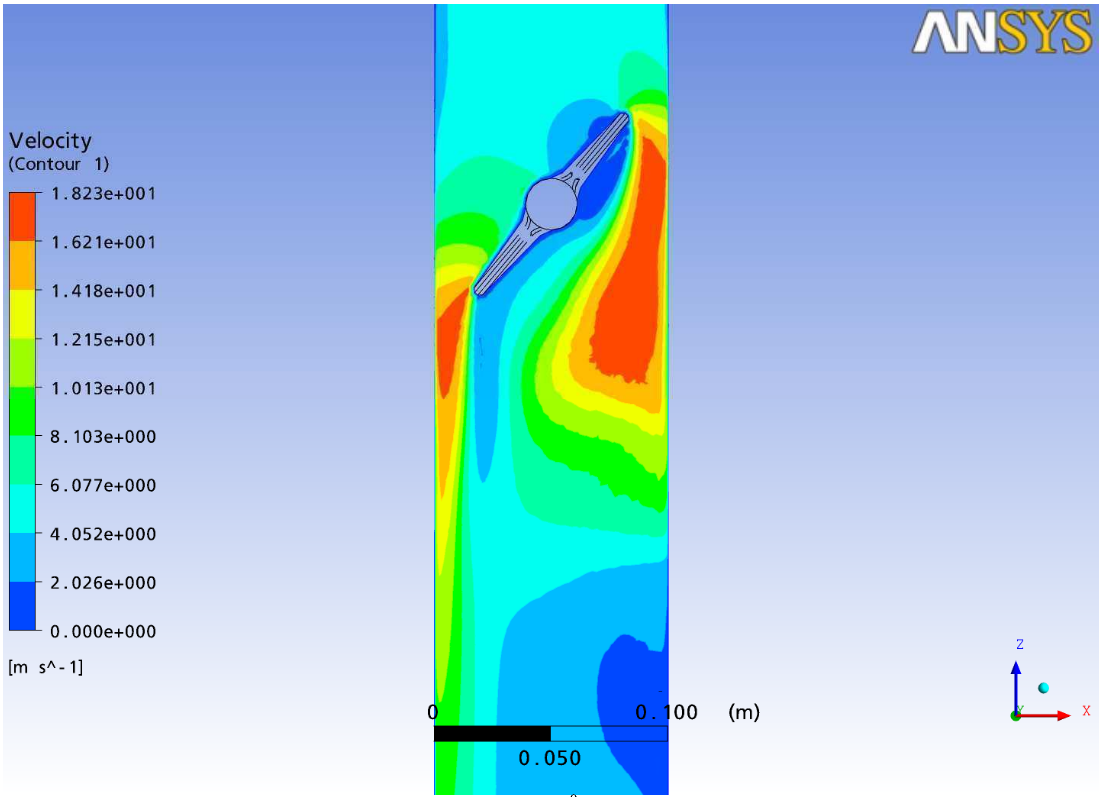


Figure 5.35 Results of flow velocity on contour at 50^0 of butterfly valve angle

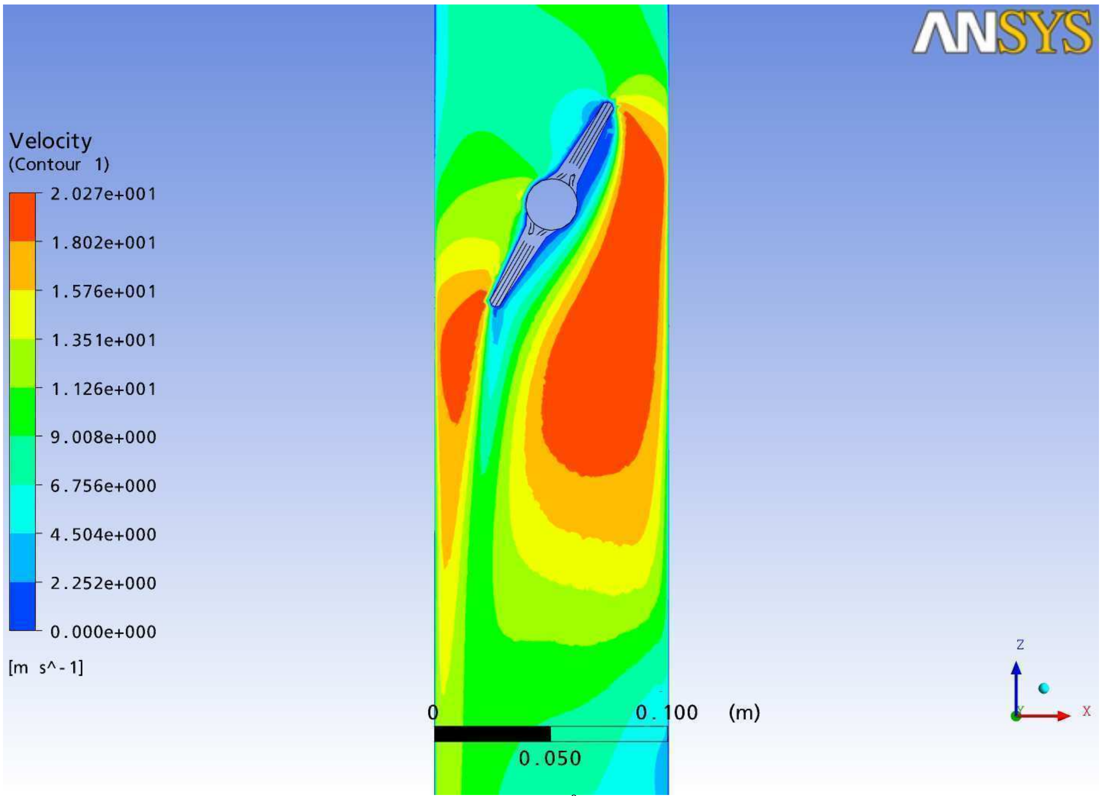


Figure 5.36 Results of flow velocity on contour at 60^0 of butterfly valve angle

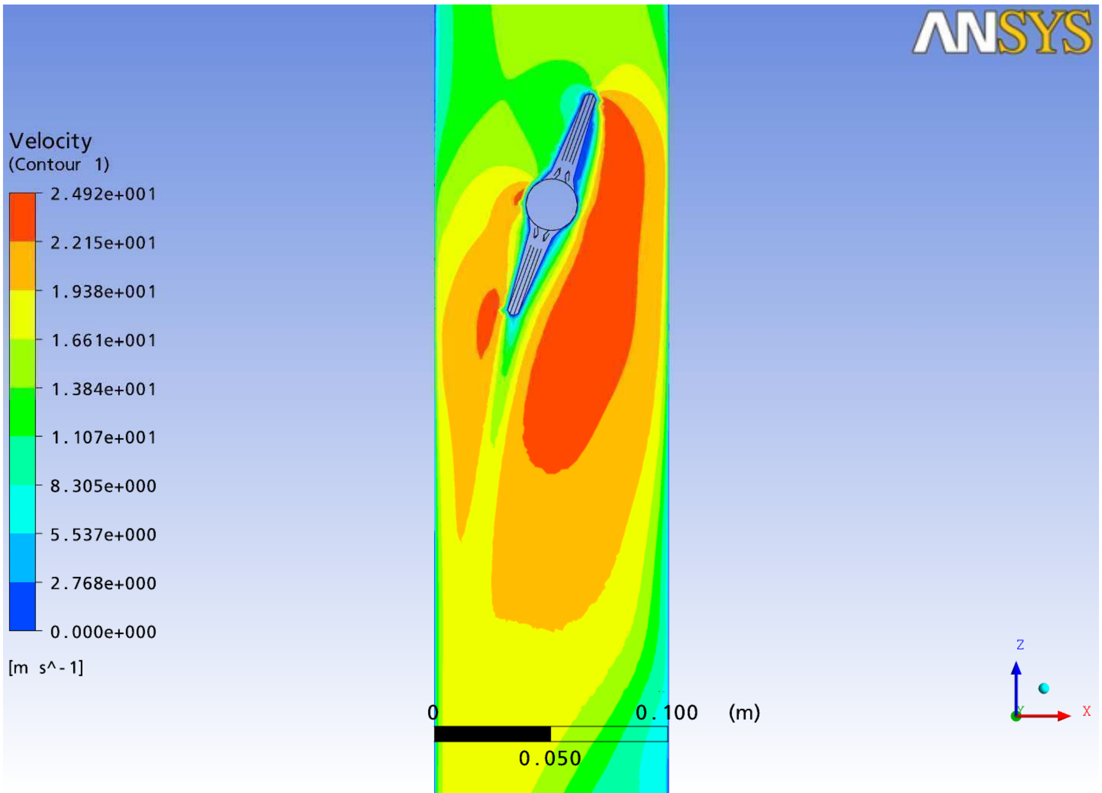


Figure 5.37 Results of flow velocity on contour at 70^0 of butterfly valve angle

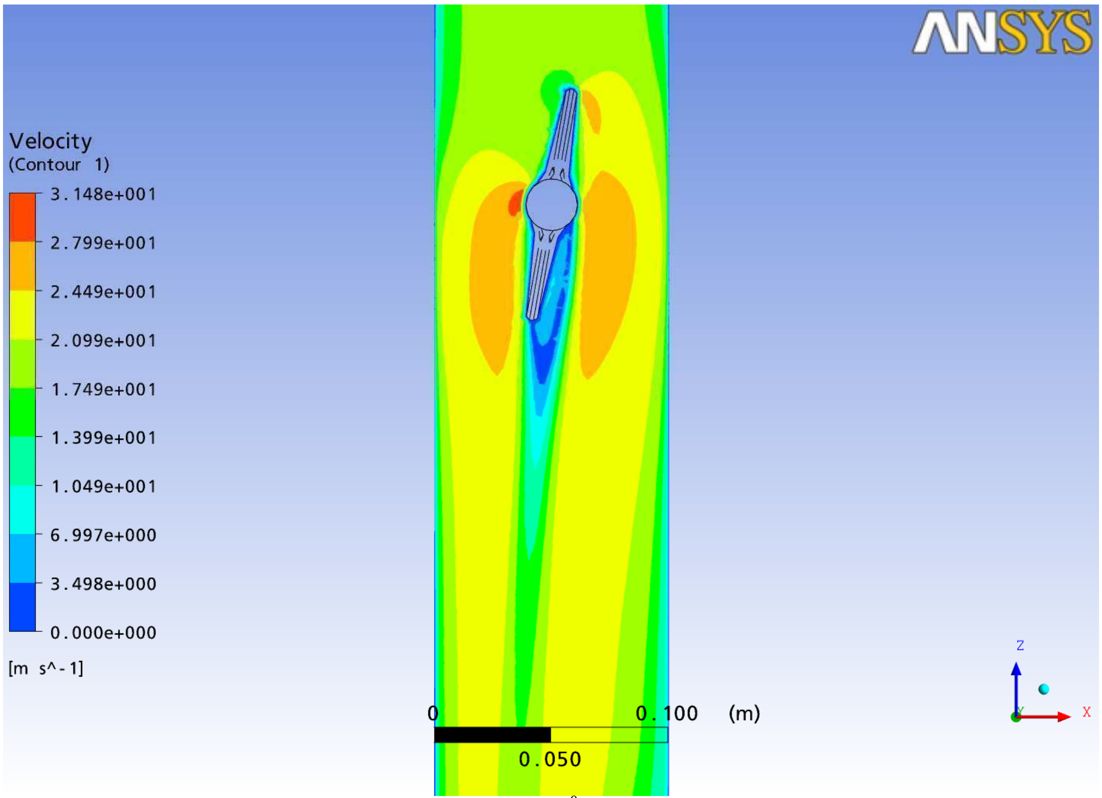


Figure 5.38 Results of flow velocity on contour at 80^0 of butterfly valve angle

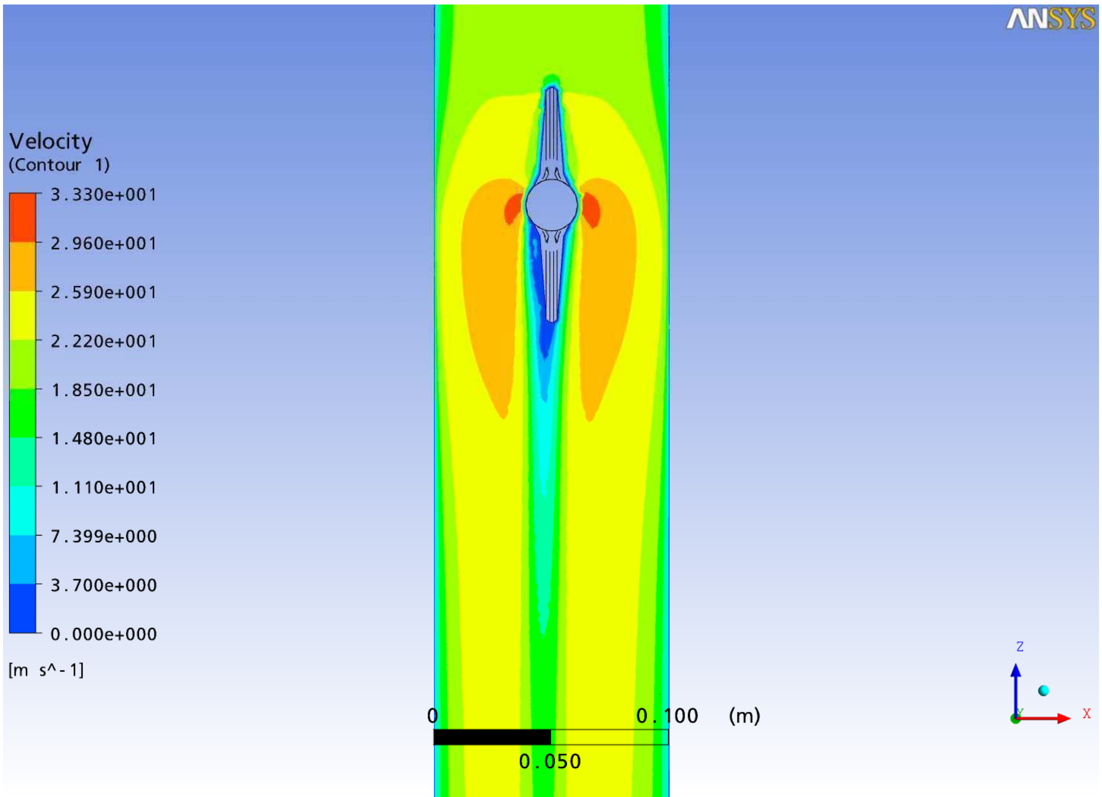


Figure 5.39 Results of flow velocity on contour at 90^0 of butterfly valve angle

CHAPTER SIX
RESULTS AND DISCUSSION

In this study, the production methods of flow characteristics in valves were analyzed. The literature about this subject was studied. (The literature review about this subject was studied.) The Kv value defining the flow characteristic of a valve was considered. The ball and butterfly valves that are widely used in industry were investigated deeply. The flow characteristic diagrams that were created by the valve producing firms using experimental methods were gained by a package programme using CFD codes.

The reason why ANSYS CFX was used is one of the many computer-aided engineering (CAE) tools available within the ANSYS Workbench platform, ANSYS CFX takes advantage of data and information common to many simulations. This begins with common geometry: Users can link to existing native computer-aided design (CAD) packages as well as create and/or modify CAD models in an intuitive solid modeling environment. Complementing the common geometry model is a suite of meshing tools, designed to ensure easy generation of the most appropriate mesh for the given application. ANSYS CFX tools then guide the user through the setup of operating conditions, selection of materials and definition of models.

The ANSYS CFX solver uses the most modern solution technology with a coupled algebraic multi-grid solver and extremely efficient parallelization to help ensure that solutions are ready for analysis quickly and reliably. Solution analysis with the ANSYS CFX post-processor then gives users the power to extract any desired quantitative data from the solution; it also provides a comprehensive set of flow visualization options.

The drawings of 3-dimensional butterfly and ball valves differing in a range of the fully open position which is 90° to 20° with an angle difference of 10° each were drawn with SolidWorks program. These drawings were used to create flow volumes with ANSYS Workbench Design Plotter program. These 3-dimensional flow volumes were separated into finite elements with ANSYS Workbench CFX Mesh program. Necessary boundary conditions were defined to flow volume with CFX-Pre module in CFX. The pressure difference between inlet and outlet of valves for every position was taken as 1 bar. Every model whose boundary conditions are defined were analyzed with CFX-Solver module. The modules analyzed were operated with CFX-Post module to produce figures showing streamlines and speed values. The flow values of the valves under pressure difference were gained by the command “massFlow” in CFX-Post module. The flow values according to the angle were created for every valve. New graphs were prepared for these values according to percentage ratios. The values of flow characteristics on the graphs and in the tables of catalogues of firms were arranged to make a comparison with the values gained from ANSYS CFX. The values found with CFX have similarities with the values given by firms as curve characteristics. The values belonging to the firms VIZA and FNW for butterfly valves; Burçelik and PBM for ball valves were used. In Figures 6.1 and 6.2, the flow valves for 10° of change; in Figures 6.3 and 6.4, the translation of these values into percentage graphics are seen.

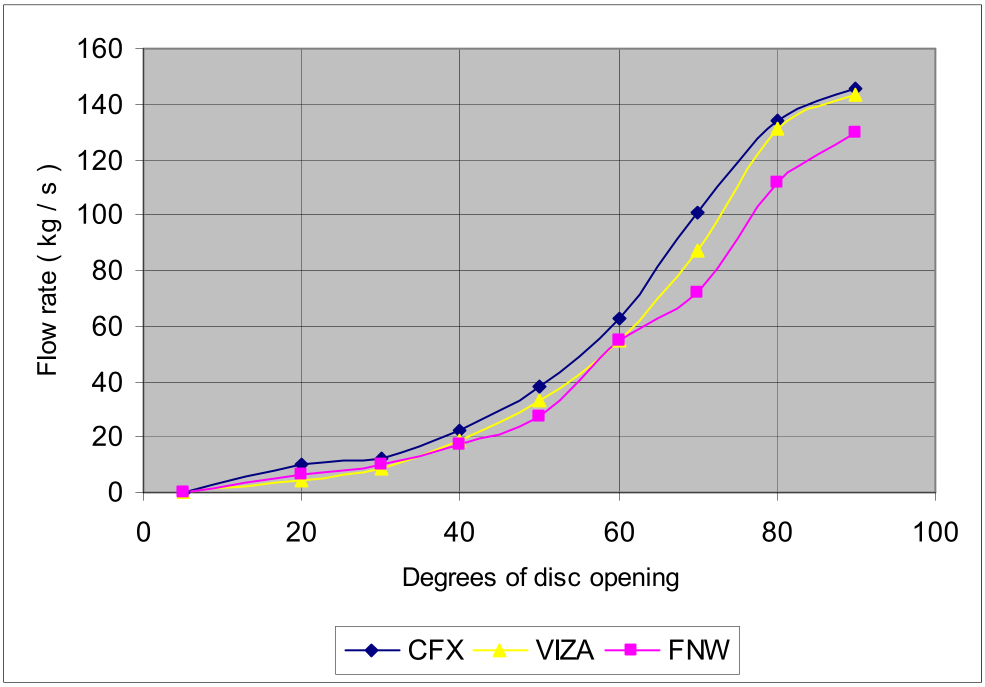


Figure 6.1 The flow rate values of butterfly valves according to angles

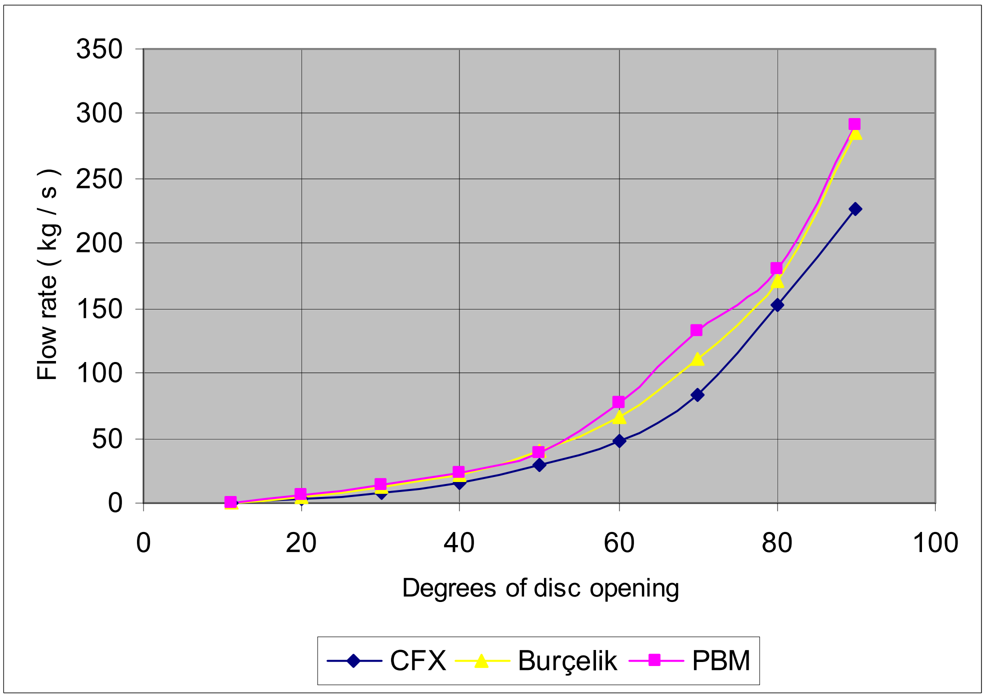


Figure 6.2 The flow rate values of ball valves according to angles

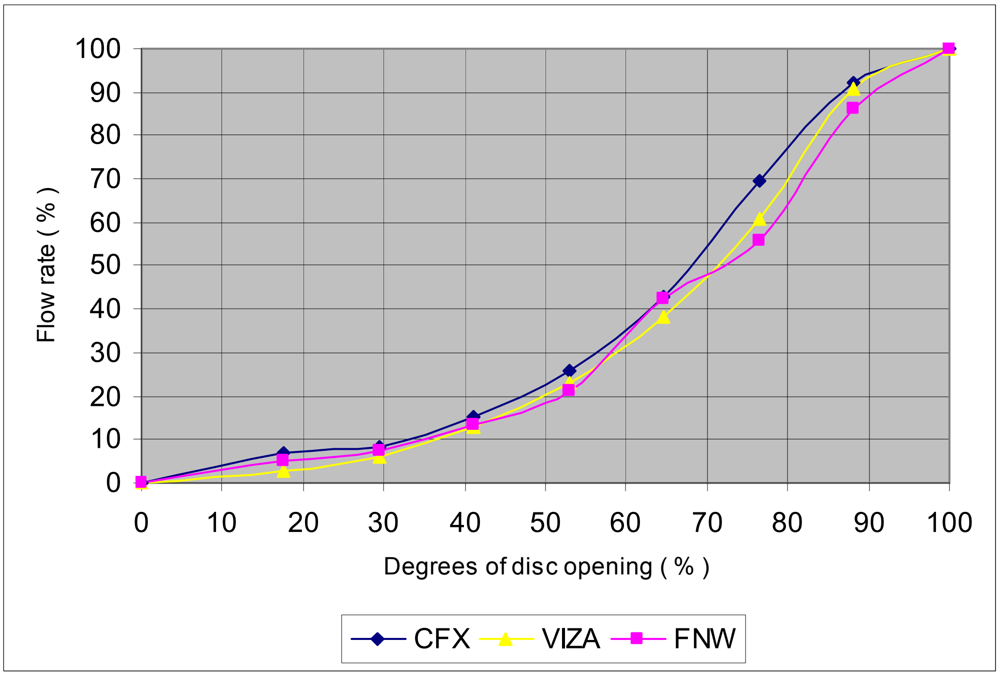


Figure 6.3 The flow characteristics of butterfly valves

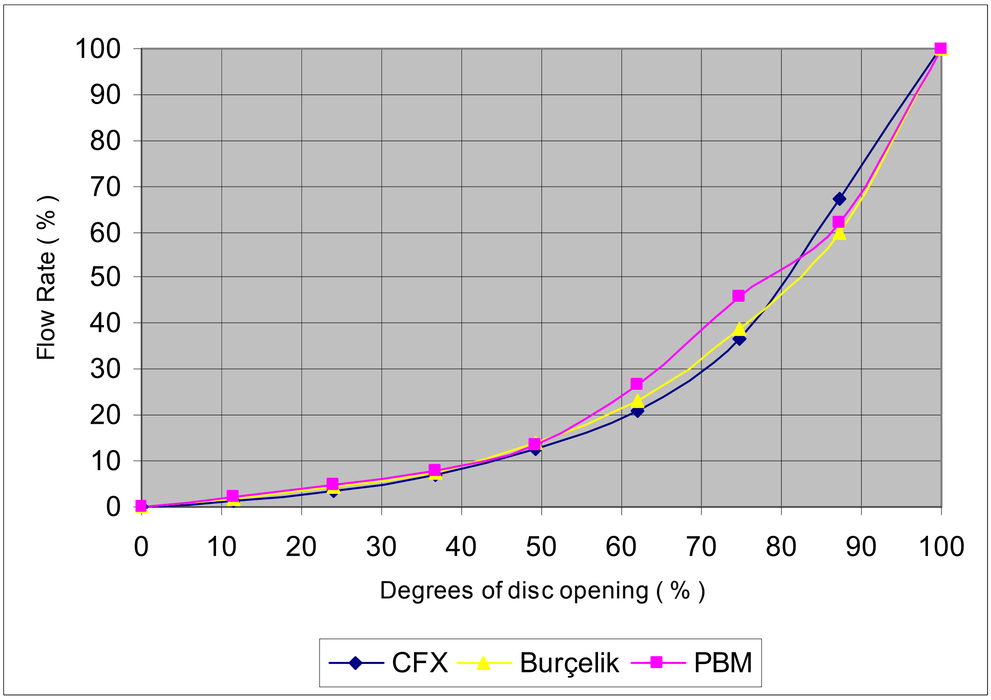


Figure 6.4 The flow characteristics of ball valves

Defining the flow characteristics of valves via CFD codes will be a big facility for the valve producers in producing new type valves. This can be used in defining the flow characteristics of V-ball and R-ball control valves that are widely used in industry. This method will give an idea to design engineers before the valves are produced.

One of the most important facts that give the valve its property is the disc design for the butterfly valve producers. In ball valve, when the valve is open, the flow geometry is in straight pipe geometry while the disc is an obstacle against flow in butterfly valve; that's why, when the disc geometry shows a minimum resistance to the flow, the maximum flow rate value the valve permits increases and the moment effect by the disc decreases. The effect of new disc designs to valve flow characteristics can be set by CFD codes.

The Analysis of Pressure Drop in Butterfly Valve

As an addition to all the studies done in this thesis, the disc angle in butterfly valves was changed and the pressure changes under constant flow rate were observed for 2 different flow rates.

In the analysis, the butterfly valve model having a diameter of DN 100 was used. In the first study, the speed value was taken as 0,01m/s so it was kept under the critical Reynolds number which is 2300 in circular pipes. The Reynolds number and the flow rate were calculated as 1120 and 0,283m^3/h in boundaries of laminar flow respectively.

In the second study, the flow rate and Reynolds number were increased to 28,274 m^3/h and 112035 respectively by taking the speed value as 1m/s. The pressure drop is shown in Figure 6.5 for such situations.

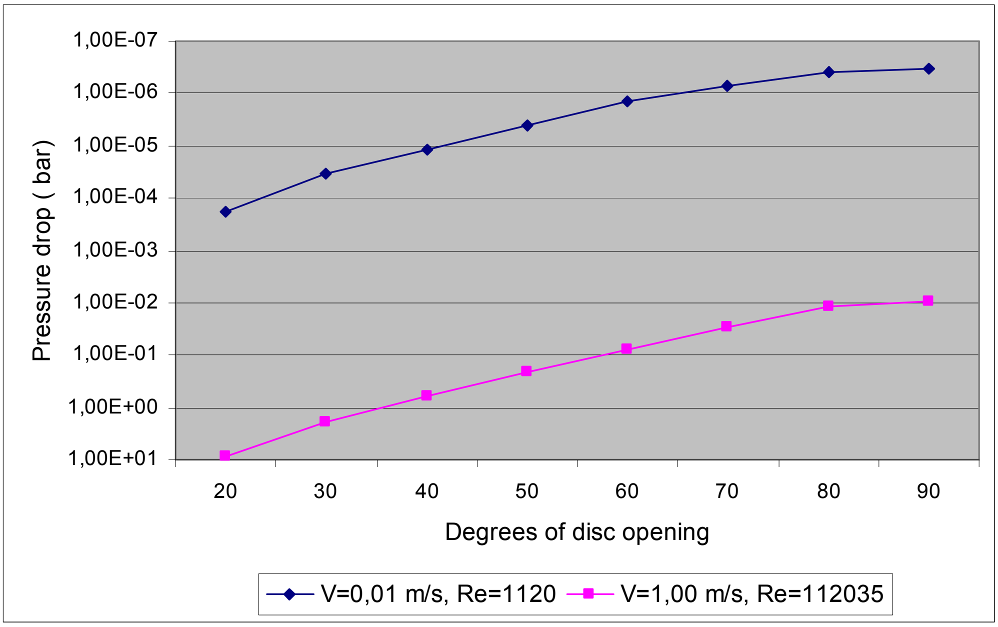


Figure 6.5 of Pressure Drop in Butterfly Valve

This logarithmic graph shows that the pressure drop increases very fast as we start turning down the valve slowly. However, it shows the same characteristic even if the flow is turbulent or laminar.

REFERENCES

ANSYS - CFX, *Master Contents*, 2007

ANSYS, *Documentation for ANSYS Workbench*, 2007

Burçelik Kelebek Vanalar. (n.d.). Retrieved August 10, 2007, from http://www.burcelikvana.com/Turkce/katalog/Kelebek.pdf

Chern, M.J. and Wang, C.C. (2004). Control of volumetric flow-rate of ball valve using V-port. *Journal of Fluids Engineering*, 126, 471-481.

Chern, M.J. ,Wang, C.C. and Ma, C.H.(2006). Performance test and flow visualization of ball valve. *Experimental Thermal and Fluid Science*, 31,505-512

Computational fluid dynamics, (n.d.) Retrieved August 12, 2007, from http://en.wikipedia.org/wiki/Computational_fluid_dynamics

Control Valve Flow Characteristics, (n.d.) Retrieved August 12, 2007, from http://www.engineeringtoolbox.com/control-valves-flow-characteristics-d_485.html

Davis, J.A., Stewart, M. (2002). Predicting globe control valve performance-part I: CFD modeling. *Journal of Fluids Engineering*, 124, 772-777.

Davis, J.A., Stewart, M. (2002). Predicting globe control valve performance-part II: CFD modeling. *Journal of Fluids Engineering*, 124, 772-777.

FNW, Resilient Seated Butterfly Valve. (n.d.). Retrieved August 10, 2007, from http://www.fnwvalve.com/FNWValve/assets/images/PDFs/FNW/FNW_Fig.731.pdf

Huang, C. and Kim, R.H. (1996). Three-dimensional analysis of partially open butterfly valve flows, *Journal of Fluids Engineering*, 118, 562-568.

Kerh, T., Lee, J.J. and Wellford, L.C. (1997). Transient fluid-structure interaction in a control valve. *Journal of Fluids Engineering*,119,354-359.

Merati, P., Macelt, M.J. and Erickson, R.B. (2001). Flow investigation around a v-sector ball valve. *Journal of Fluids Engineering*, 123, 662-671.

PBM Valve Solutions (n.d.). Retrieved August 10, 2007, from www.pbmvalve.com/pdf/VBall_control_valve.pdf

Smith, P.,& Zappe, R. W. (Eds.). (2004). *Valve selection handbook.* (5th ed.) United States of America: Elsevier, Inc.

Van Lookeren Campagne, C., Nicodemus, R., de Bruin, G.J. and Lohse, D. (2002). A method for pressure calculation in ball valves containing bubbles. *Journal of Fluids Engineering*, 124, 765-771.