Flow simulation with SolidWorks

Akvile Jonuskaite

Bachelor Thesis

Plastics Technology

2017

DEGREE THESIS	
Arcada	
Degree Programme:	Plastics Technology
Identification number:	16576
Author:	Akvile Jonuskaite
Title:	Flow Simulation with SolidWorks
Supervisor (Arcada):	Mathew Vihtonen
Commissioned by:	Arcada University of Applied Sciences

Abstract:

The idea of this thesis is to design a pipe system and run a flow simulation for the observation of the flow of fluids in pipes and compare it with the results obtained in the laboratory.

First, a pipe system was modelled in SolidWorks software. Separate parts were designed, and then brought together into the final assembly.

Secondly, an experimental analysis was performed in Heat Transfer laboratory. Volumetric flow rate was obtained using flow meter. This value was used in a velocity calculation.

Finally, fluid flow simulations were performed using FloXpress and Flow Simulation add-ins. Different velocity and pressure magnitudes were observed along the pipeline.

The average velocity in experimental analysis was found to be 0.531 m/s while the average velocity from Flow Simulation depending on the boundary conditions were 0.532 m/s and 1.375 m/s respectively. Head loss was also calculated for experimental and Flow Simulation values. Head loss from laboratory experiment was calculated to be 2.446 m. Head loss calculated from Flow Simulation values depending on boundary conditions were 0.409 m and 2.428 m respectively.

Keywords:	SolidWorks, Turbulent flow, Laminar flow, Bernoulli's equation, Velocity, Pressure, Head loss
Number of pages:	49
Language:	English
Date of acceptance:	

CONTENTS

1 INTRODUCTION 7

1.1 Background 7

1.2 Objectives 7

2 LITERATURE REVIEW 8

2.1 Bernoulli's Equation 8

2.2 Types of flow 9

2.2.1 Laminar flow 10

2.2.2 Turbulent flow 10

2.2.3 Transitional flow 11

2.2.4 Reynolds number 11

2.3 Entrance Region 12

2.4 Entry length 13

2.5 Head Loss in piping systems 14

2.5.1 Laminar flow 15

2.5.2 Turbulent flow 15

2.5.3 Major Head Loss 15

2.5.4 Minor Head Loss 16

2.5.5 Factors that affect head loss 16

2.6 Navier-Stokes equations 17

2.7 Friction factor 18

2.7.1 The laminar friction 20

2.7.2 The turbulent friction 20

2.5.2.1 The Colebrook equation 20

2.5.2.2 Moody chart 20

2.8 SolidWorks 21

2.8.1 Simulation add-ins 21

2.8.2 Standard parts 22

3 Method 23

3.1 The design of the pipe system 23

3.1.1 Weldments 23

3.1.2 Custom parts 23

3.1.3 The assembly of the pipe system 26

3.2 FloXpress Analysis 27

3.2.1 Simulation 1 .... *28*
3.2.2 Simulation 2 .... *28*
3.3 Flow Simulation 28
3.3.1 Simulation 1 .... *30*
3.3.2 Simulation 2 .... *32*
3.4 Laboratory experiment 34
4 RESULTS 36
4.1 Laboratory experiment calculation 36
4.2 FloXpress Analysis 38
4.3 Flow Simulation 39
5 DISCUSSION 42
6 CONCLUSION 45
References 47

Figures

Figure 1 Laminar flow [5] 10
Figure 2 Turbulent flow [5] 11
Figure 3 The development of the velocity boundary layer in a pipe [6] 13
Figure 4 The variation of wall shear stress in the flow direction for flow in a pipe from the entrance region into the fully developed region [7] 14
Figure 5 Relative roughness for various pipes [15]... 19
Figure 6 Moody Chart [14].. 21
Figure 7 Pipe.. 23
Figure 8 On the left: opened valve. On the right: closed valve 24
Figure 9 Exploded view of the valve.. 25
Figure 10 Elbow 26
Figure 11 The complete pipe system 26
Figure 12 Lid. Extruded Base feature.. 27
Figure 13 Inlet and outlet boundary conditions... 28
Figure 14 Mesh.. 30
Figure 15 Pressure contour 31
Figure 16 Velocity magnitude 31
Figure 17 Flow trajectories... 32
Figure 18 Pressure contour 33
Figure 19 Velocity magnitude 33
Figure 20 Flow trajectories... 34
Figure 21 Velocity magnitude 39
Figure 22 Cut plots. Velocity 40
Figure 23 Chart graph of average velocities obtained from Experimental and SolidWorks simulation 42

Tables

Table 1 Values for calculation 36
Table 2 FloXpress simulation results 39
Table 3 Flow Simulation results 41
Table 4 Average velocities obtained from Experimental and SolidWorks simulation .. 42
Table 5 Head loss 43
Table 6 The velocities obtained from experimental and COMSOL simulation [26] 43

Equations

Equation 1: Conservation of energy 8
Equation 2: Work relating force and distance 8
Equation 3: Work relating pressure, area, and distance 8
Equation 4: Work relating pressure and volume 8
Equation 5: Work done 8
Equation 6: Kinetic energy 8
Equation 7: Potential energy 8
Equation 8: Bernoulli's equation 9
Equation 9: Extended Bernoulli's equation 9
Equation 10: Reynold's number for circular pipes 11
Equation 11: Entry length. Laminar flow 13
Equation 12: Entry length. Turbulent flow 13
Equation 13: Total head loss 14
Equation 14: Head loss. Laminar flow 15
Equation 15: Head loss. Turbulent flow 15
Equation 16: Major head loss 15
Equation 17: Minor head loss 16
Equation 18: Continuity 18
Equation 19: Navier-Stokes 18
Equation 20: Reynold's number for laminar flow 20
Equation 21: The Colebrook equation for transitional and turbulent flows 20

List of symbols

	Name	Symbol	Unit
1.	Acceleration due to gravity	g	m/s^2
2.	Velocity	v	m/s
3.	Pressure	p	Pa
4.	Density	ρ	kg/m^3
5.	Mass	m	kg
6.	Volume	V	m^3
7.	Length	L	m
8.	Diameter	D	m
9.	Area	A	m^2
10.	Height	y	m
11.	Distance	d	m
12.	Temperature	T	K
13.	Volumetric flow rate	$\dot{V}$	m^3/s
14.	Dynamic viscosity	μ	Pa·s
15.	Kinematic viscosity	ν	m^2/s
16.	Head loss	H_L	m
17.	Reynolds number	Re_D	-
18.	Work	W	J
19.	Force	F	N
20.	Energy	E	J
21.	Friction factor	f	-
22.	Minor head loss coefficient	k	-

FOREWORD

I would first like to thank you my thesis supervisor Mathew Vihtonen for advice and excellent supervision throughout this thesis.

I would also like to express my gratitude to all the professors and technical staff for guidance during my studies.

Lastly, I would like to thank you my friends and family for their continuous support, and unlimited encouragement, particularly during the completion of this thesis.

1 INTRODUCTION

The purpose of this study is to simulate flow in pipes utilizing SolidWorks software. Fluid flow may be very hard to predict and differential equations that are used in fluid mechanics are difficult to solve. SolidWorks add-ins enable you to simulate flow of liquids and gases and efficiently analyse the effects of fluid flow.

1.1 Background

The motion of a fluid is usually very complex. The observed fluid flow behaviour becomes much more understandable after defining the flow into laminar or turbulent regimes. The momentum equation provides one of the most recurring tools to be used in understanding fluid flows. Another fundamental tool for fluid flow analysis is continuity equation, both in its volumetric and its more widely applicable mass flow form.
Along with the energy equation, the aforementioned equations, momentum and continuity, are also known as Navier – Stokes equations. Newtonian fluid flow is incompressible when the density is constant. In such case Navier - Stokes equations can be simplified.

1.2 Objectives

The objectives of this thesis are as follows:

1. Model a pipe system using SolidWorks.
2. Simulate and analyse the fluid flow in pipes using the SolidWorks Flow Simulation add-in and FloXpress.
3. Compare experimental and simulated results obtained from SolidWorks and COMSOL.

2 LITERATURE REVIEW

2.1 Bernoulli's Equation

The application of the principle of conservation of energy leads to a relation between pressure, elevation and flow velocity in a fluid. This relation is called Bernoulli's equation. [1] It is one of the best-known and widely-used equations in fluid mechanics.

Bernoulli's equation can be viewed as a conservation of energy law for a flowing fluid.

$Work\ done = Kinetic\ Energy + Potential\ Energy$

$\Delta W = \Delta(KE + PE)$ *(1)*

Work done equals force multiplied by distance:

$W = Fd$ *(2)*

We can plug in the formula that relates pressure and force, which gives us:

$W = pAd$ *(3)*

Where A represents area.

Volume is derived by multiplying area and height (distance), thus:

$W = pV$ *(4)*

Work done is equal to:

$\Delta W = p_1V_1 - p_2V_2$ *(5)*

Kinetic energy is the energy of mass in motion:

$KE = \frac{mv^2}{2} = \frac{\rho V v^2}{2}$ *(6)*

Where V represents volume.

Potential energy is dependent on height:

$PE = mgy = \rho V g y$ *(7)*

Where y represents height.

Substituting gives:

$$p_1V - p_2V = \frac{\rho V v_2^2}{2} + \rho V g y_2 - \frac{\rho V v_1^2}{2} - \rho V g y_1$$

Divide by V:

$$p_1 - p_2 = \frac{\rho v_2^2}{2} + \rho g y_2 - \frac{\rho v_1^2}{2} - \rho g y_1$$

Rearranging the formula to put the terms that refer to the same point on the same side of the equation:

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 \qquad (8)$$

Bernoulli’s equation has some restrictions:

- Steady flow
- Incompressible flow (which also means density is constant)
- Frictionless flow
- Flow along a streamline [2]

In practical situations, problems may be analysed using extended Bernoulli’s equation:

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 + H_L \qquad (9)$$

Where,

H_L – head losses due to friction or viscosity.
y_1 and y_2 – heights of inlet and outlet

2.2 Types of flow

There are three flow regimes. When a flow moves on in a tranquil fashion it is said to be streamline or laminar flow, because the various axial layers in the fluid remain intact as the flow proceeds. The so-called turbulent flow is chaotic, because layers in the flow conduit do not remain intact but are constantly being mixed due to turbulence, that is, chaotic motions in the flow. [3] Transitional flow is a mixture of laminar and turbulent flows.

2.2.1 Laminar flow

Laminar flow is characterized by smooth streamlines and highly ordered motion. If the pipe is sufficiently long (relative to the entry length) then the entrance effects are negligible and therefore the flow is fully developed. Laminar flow occurs when the fluid flows in parallel layers without mixing. The velocity of the fluid is constant at any given moment. Since the flow is steady, there is no acceleration. The flow is laminar for cylindrical pipes when Reynolds number is less than 2300. [4]

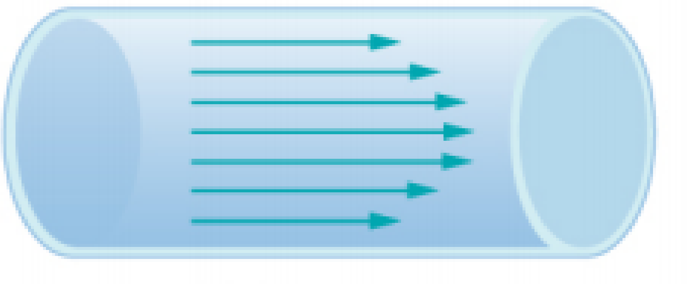

Figure 1 Laminar flow [5]

2.2.2 Turbulent flow

Turbulent flow is characterized by velocity fluctuations and highly disordered motion. Most flows encountered in practice are turbulent. Turbulent flow occurs when streamlines of the liquid are irregular and change over time. The paths of the fluid flow are also irregular and form tiny whirlpool regions. [1] The flow is turbulent when Reynolds number is greater than 4000. In practice, most flows in engineering are turbulent. However, the theory of turbulent flow remains underdeveloped since this flow is a very complex mechanism dominated by fluctuations. Therefore, turbulent flow is analysed by applying experimental measures. [4]

Figure 2 Turbulent flow [5]

2.2.3 Transitional flow

The transition from laminar to turbulent flow is not sudden. It occurs over some region in which turbulent flow in the centre of the pipe and laminar flow is near the edges of the pipe. The flow fluctuates between laminar and turbulent flows before it becomes fully turbulent. The flow is considered transitional when Reynolds number is in between 2300 and 4000. [4]

2.2.4 Reynolds number

Reynolds number can reveal whether flow is laminar or turbulent. The transition from laminar to turbulent flow depends on the surface roughness, flow velocity, geometry, surface temperature, and type of fluid, among others. Flow regime mainly depends on the ratio of inertial forces to viscous forces in the fluid. This ratio is called Reynolds number and is expressed as

$$Re_D = \frac{Inertial\ forces}{Viscous\ forces} = \frac{V_{avg}D}{v} = \frac{\rho V_{avg}D}{\mu} \qquad (10)$$

Where,

Re_D = Reynolds number for cylindrical pipe. Reynolds number is a dimensionless number
V_{avg} = average flow velocity (m/s)
D = diameter (m)
v = kinematic viscosity of the fluid (m^2/s)
μ = dynamic viscosity (Pa·s)

Critical Reynolds number Re_{cr} is the number at which the flow becomes turbulent. The value of this number is different for different geometries and flow conditions. The generally accepted value of the critical Reynolds number for internal flow in a circular pipe is $Re_{cr} = 2300$.

Under most practical conditions, the flow in a circular pipe is:

- Laminar when $Re \leq 2300$
- Transitional when $2300 \leq Re \leq 4000$
- Turbulent when $Re \geq 4000$ [4]

2.3 Entrance Region

Entrance region is a region where the fluid enters a pipe at uniform velocity. The fluid particles that are in contact with the surface of a pipe comes to a complete stop because of the no-slip condition. Because of friction, fluid particles in the adjacent layers gradually slow down. To make up for this velocity reduction, the velocity of the fluid at the midsection of the pipe increases to keep the mass flow rate through the pipe constant. The area of the flow in which the effects of the viscous shearing forces due to viscosity are felt is known as the velocity boundary layer. The hypothetical boundary layers can be divided into:

- The boundary layer region, where viscous effects and the velocity are considerable.
- Irrotational flow region, where frictional effects are negligible and velocity is constant in radial direction.

The region of boundary layer increases in the flow direction until it merges with the layer from the opposite side at the centreline. The region starting from the pipe inlet to the meeting point of the two boundary layers is known as hydrodynamics entrance region and the length of this region is called hydrodynamic entry length. Flow in the entrance region is called hydrodynamically developing flow. The region at the merging point of boundary layers is called hydrodynamically fully developed region. The flow is considered to be fully developed until change in temperature in the fluid occurs. [4]

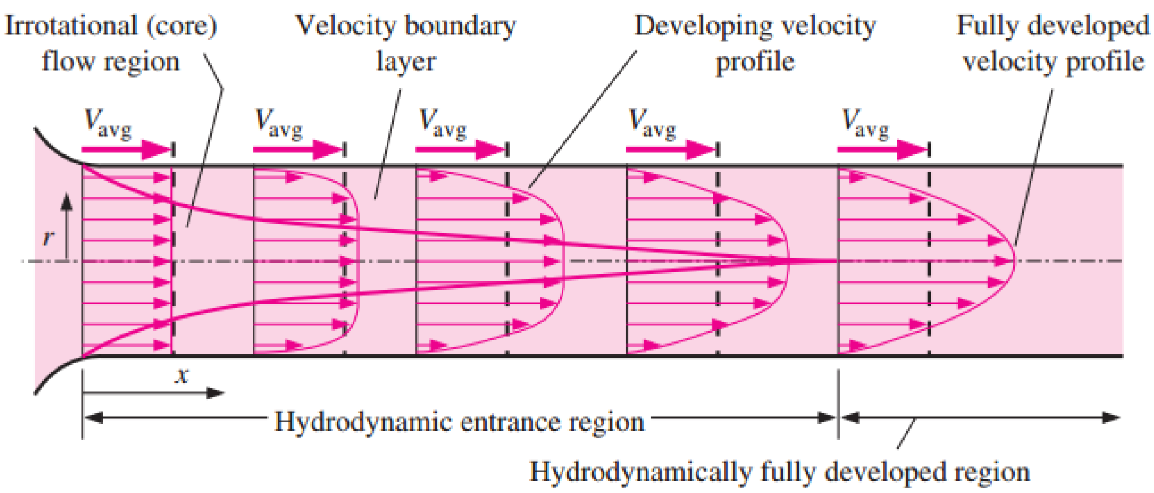

Figure 3 The development of the velocity boundary layer in a pipe [6]

2.4 Entry length

The length of hydrodynamics entrance region is called entry length. It may also be taken as the distance from the fluid entrance to 2% of the fully developed wall shear stress value. In laminar flow the hydrodynamic entry length is given as:

$$L_{h-laminar} \cong 0.05ReD \qquad (11)$$

In turbulent:

$$L_{h-turbulent} \cong 10D \qquad (12)$$

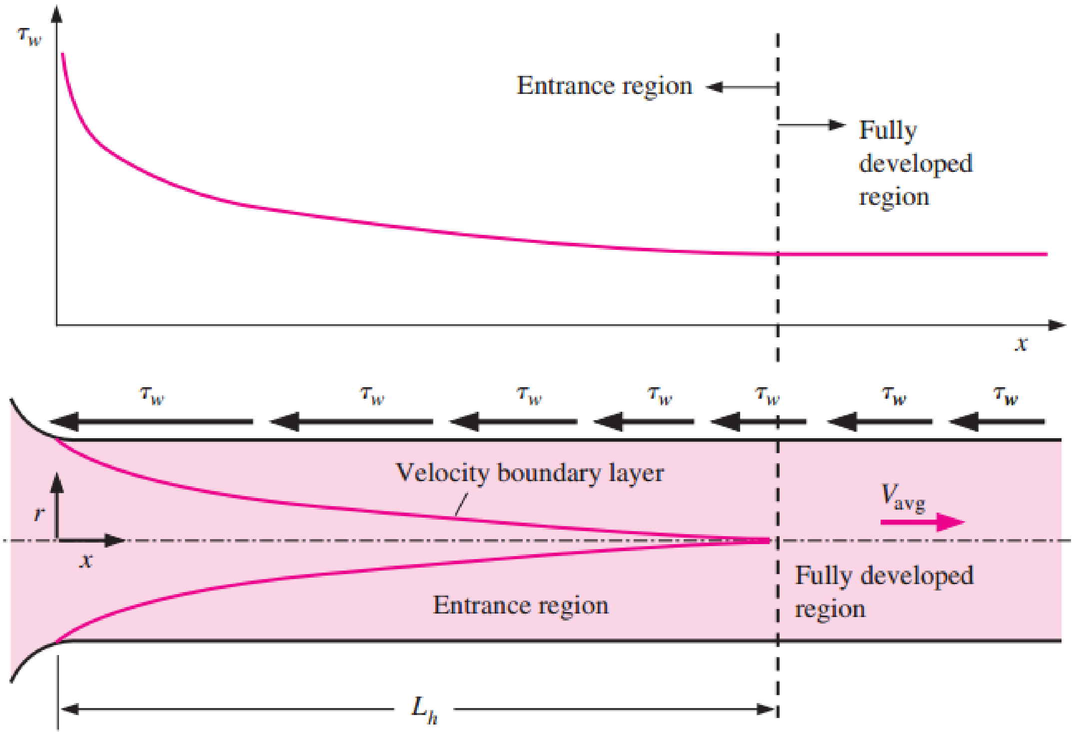

Figure 4 The variation of wall shear stress in the flow direction for flow in a pipe from the entrance region into the fully developed region [7]

2.5 Head Loss in piping systems

When fluid flows inside a pipe, friction occurs between the moving fluid and the stationary pipe wall. Some of the fluid's hydraulic energy is converted to thermal energy due to this friction. This process is irreversible therefore the fluid experiences a drop in pressure. This conversion and loss of energy is known as head loss. [8]

Total head loss can be expressed as:

$$H_L = \sum H_{major\ losses} + \sum H_{minor\ losses} \qquad [9] \qquad (13)$$

2.5.1 Laminar flow

The following equation is called Hagen-Poiseuille's equation and is used for head loss calculation of steady laminar flow of incompressible fluid.

$$H_L = \frac{\Delta p}{\rho g} \qquad (14)$$

Where,

H_L = head loss (m)
p = pressure (Pa)
ρ = density (kg/m^3)
g = acceleration due to gravity (g = 9.81 m/s^2) [4]

2.5.2 Turbulent flow

In turbulent flow, whirlpools and wakes make the flow unpredictable. The formula below is called Darcy's equation and is used for the calculation in fully developed flow:

$$H_L = f \frac{L}{D} \frac{V_{avg}^2}{2g} \qquad (15)$$

Where,

f = friction factor related to the roughness inside the pipe
L = length of the pipe (m)
D = internal diameter of the pipe (m)
V_{avg} = average liquid velocity (m/s)
g = acceleration due to gravity (g = 9.81 m/s^2) [4]

2.5.3 Major Head Loss

Major losses are associated with energy loss per length of pipe. It is caused by friction in pipes and ducts.

$$H_{L-major} = f \frac{L}{D} \frac{V_{avg}^2}{2g} \qquad (16)$$

Where,

f = friction factor

L = pipe length (m)

V_{avg} = average velocity (m/s)

D = internal diameter (m)

g = acceleration due to gravity (g = 9.81 m/s^2) [10]

2.5.4 Minor Head Loss

Minor losses are associated with technological equipment. It is caused by components such as valves, bends, tees, etc. Minor losses can easily exceed major losses in relatively short pipe systems with a relatively large amount of bends and fittings. [10]

$$H_{L-minor} = k\frac{V_{avg}^2}{2g} \qquad (17)$$

Where,

k = minor head loss coefficient [11]

2.5.5 Factors that affect head loss

1. **Flow Rate.**

   The velocity of the liquid increases at the same rate as the flow rate. Due to viscosity, the resistance to flow also increases. The head loss is proportional to the square of the velocity therefore the increase in loss is very rapid.

2. **Inside diameter of the pipe.**

   The velocity of the liquid is reduced when the flow area increases, which happens when the inside diameter is larger. Head loss due to friction is reduced when velocity decreases. However, the flow area decreases if the inside diameter of the pipe is reduced, in such case the velocity of the liquid increases and the head loss due to friction increases.

3. **Roughness of the pipe wall.**

The roughness of the inside pipe wall increases with the thickness of non-moving boundary layer increase. The resulting reduction in flow area causes the rise of the velocity of the liquid which in turn increases the head loss due to friction.

4. **Corrosion and Scale Deposits.**

Scale deposits and corrosion both increase the roughness of the inside pipe wall and thus increases head loss.

5. **Viscosity of the liquid.**

More energy is needed to move high viscosity liquid. The higher the viscosity of the liquid is, the more friction occurs.

6. **Length of the pipe.**

Head loss due to friction occurs all along a pipe. Therefore, head loss would be constant along the pipe at a given flow rate.

7. **Fittings.**

Fittings disrupt the smooth flow of the liquid. When the disruption occurs, head loss due to friction occurs. However, elbows, tees, valves, and other fittings are necessary to a piping system.

8. **Straightness of the pipe.**

Due to momentum, liquid travels in a straight line. Curved or crooked pipe disturbs straight flow and thus increases the head loss due to friction. [12]

2.6 Navier-Stokes equations

Navier-Stokes equations are the basis for nearly all CFD (Computational Fluid Dynamics) flow modelling. Solving these equations predicts the fluid velocity and its pressure in a given geometry. The Navier-Stokes equations are always solved together with the continuity equation. The Navier-Stokes equations serves as the conservation of momentum, while the continuity equation represents the conservation of mass. [13] These equations apply to any point in the flow and thus all details of the flow can be solved everywhere in the flow domain. However, most differential equations in fluid mechanics are very difficult to solve and therefore often require help from a computer. These equations in certain cases may need to be combined with additional equations, such as energy equation. [4]

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho v) = 0 \qquad (18)$$

Navier-Stokes:

$$\rho \frac{\delta v}{\delta t} = -\nabla p + \rho g + \mu \nabla^2 v \qquad (19)$$

Where,

ρ = fluid density

v = fluid flow velocity

p = fluid pressure

μ = fluid dynamic viscosity

∇ = del operator [4]

2.7 Friction factor

The friction factor is a dimensionless factor that depends primarily on the fluid velocity, pipe diameter, fluid density, and viscosity. It can also be a function of wall roughness which depends on the size e. Thus, the general formula can be written as:

$$f \propto Re, \frac{e}{D}$$

Where e/D = relative roughness, which is the ratio of the mean height of roughness of the pipe to the pipe diameter. [14]

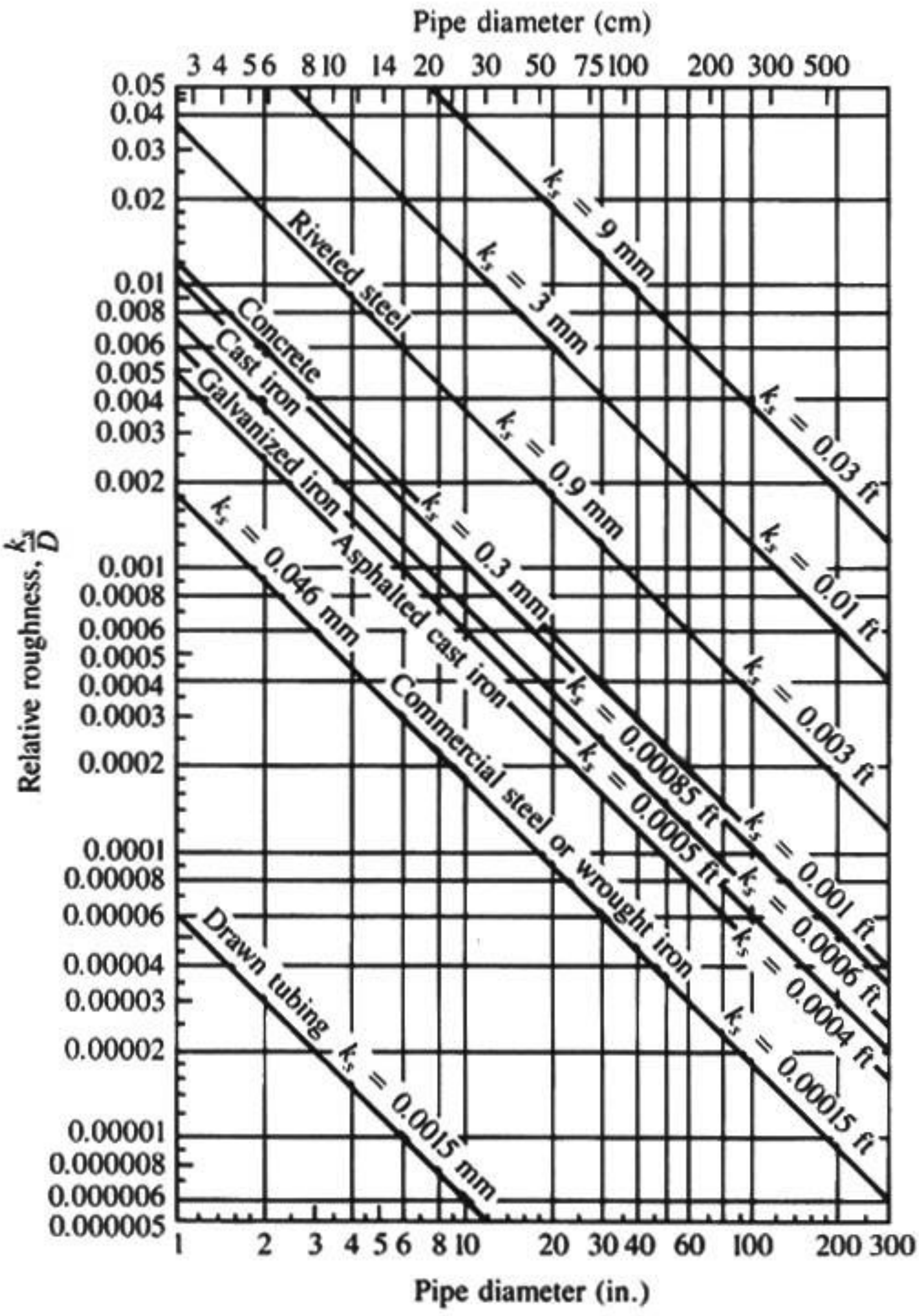

Figure 5 Relative roughness for various pipes [15]

2.7.1 The laminar friction

The laminar friction factor is a function of Reynolds number alone and is independent of any other factor.

$$f = \frac{64}{Re} \qquad [3] \qquad (20)$$

2.7.2 The turbulent friction

The turbulent friction factor is influenced by both Reynolds factor and wall roughness. To determine friction factor, Colebrook equation and Moody chart are used.

2.5.2.1 The Colebrook equation

In 1939, Cyril F. Colebrook combined the available data for transition and turbulent flow in smooth and rough pipes into the following formula known as the Colebrook equation:

$$\frac{1}{\sqrt{f}} = -2\, log_{10}\left(\frac{e/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \qquad [4] \qquad (21)$$

2.5.2.2 Moody chart

Moody chart is one of the most accepted and used charts in engineering. It relates the Darcy friction factor, Reynolds number and relative roughness. Moody friction factor can be used in Darcy-Weisbach major loss equation. [4]

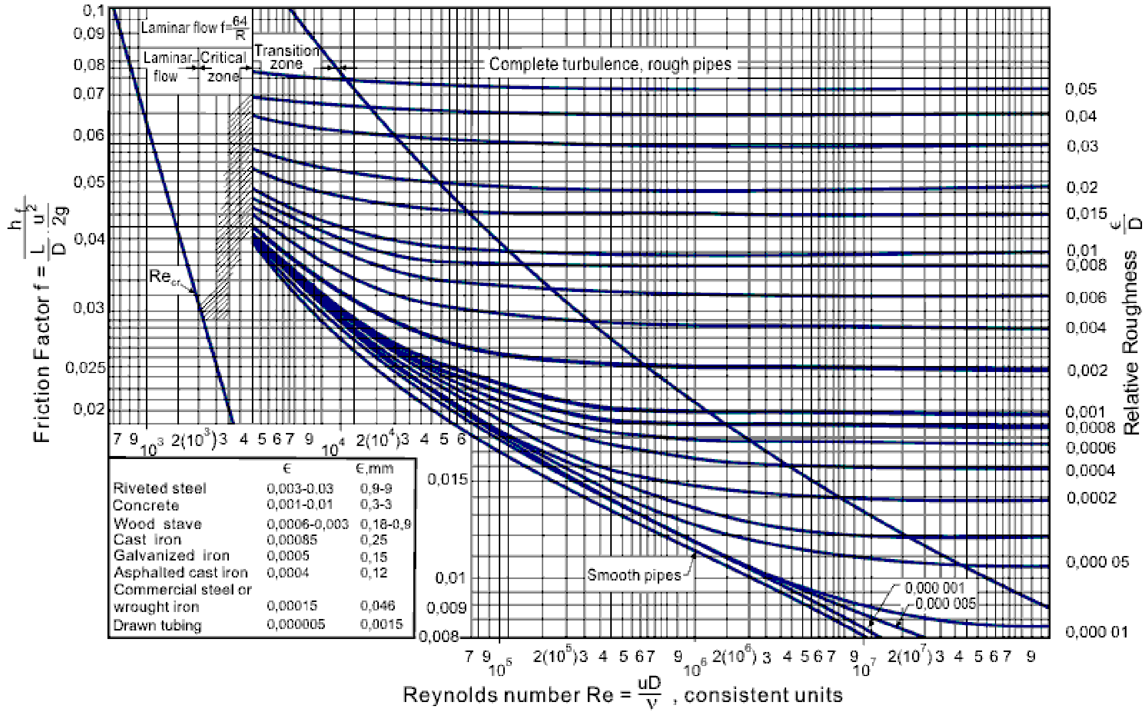

Figure 6 Moody Chart [14]

2.8 SolidWorks

SolidWorks is a computer-aided design (CAD) software, which is used to create 2D or 3D models. This simple and yet very powerful computer program enables designers to create highly detailed parts and assemblies as well as production-level drawings. SolidWorks is an excellent tool to cover a lot of stages of product development. Not only this software provides with tools needed to generate complex surfaces, structural welded assemblies, and others, it also allows you to test your design before manufacture using broad range of tools: fluid dynamics, static and dynamic response, heat transfer to name a few. [16]

2.8.1 Simulation add-ins

SolidWorks has tools that enable you to simulate liquid and gas flow in real world conditions, run “what if” scenarios, and efficiently analyse the effects of fluid flow, heat transfer, and related forces on immersed or surrounding components. [17]

FloXpress is a basic fluid flow analysis tool. It calculates how water or air flows through part or assembly models. It comes with all SolidWorks 3D CAD software packages. [18]

More advanced, SolidWorks Flow simulation uses CFD (computational fluid dynamics) analysis, which simulates fluid passing through or around an object. The analysis may contain unsteady and compressible flows, heat transfer, etc. in one calculation only. Such complicated analysis may be very costly and time consuming without some form of simulation tool. [19]

2.8.2 Standard parts

SolidWorks provides with library toolbox of standard parts which helps the user to speed up the design process, increase productivity, and save both time and development costs. The components can be customized to meet your needs or can be used “as is”. Items can be easily dragged and dropped into the assembly for further design processes. SolidWorks toolbox includes machine components and hardware – bolts, screws, nuts, bearings, washers, structural members, and others. [20]

3 METHOD

This segment is divided into the following sections:

1. The design of the pipe system
2. FloXpress Analysis
3. Flow Simulation
4. Laboratory experiment

3.1 The design of the pipe system

A typical pipe system consists of control systems, pipes, pipe connections, fittings (elbows, branches, diffusors, reducers, valves, etc.), support elements, expansion joints, pipe clamps, pumps, and compressors. Piping can be high-scale and extremely complex. The pipe system analysed in this thesis is rather simple and consists of pipes, valves, and elbows.

3.1.1 Weldments

Weldments are structural sections held together by the welding process. They are made using 2D or 3D sketches and then creating structural members that in turn contain groups of sketch segments. [21]

3.1.2 Custom parts

3.1.2.1 Pipes

The pipes were designed using DN 20 mm standard (Nominal Bore ¾ inch), with an outside diameter of 26.67 mm.

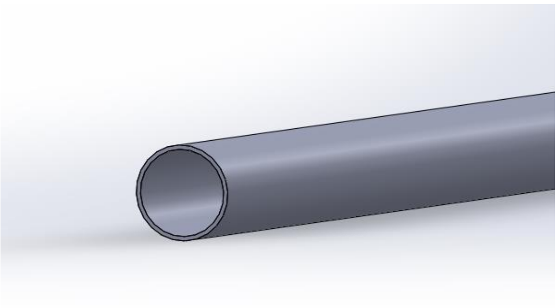

Figure 7 Pipe

3.1.2.2 Valves

The standard used in the design of valves is MSS SP-72 for Ball valves with flanged or butt - welding ends for general service. A ball valve controls flow using a hollow, perforated, and pivoting ball.

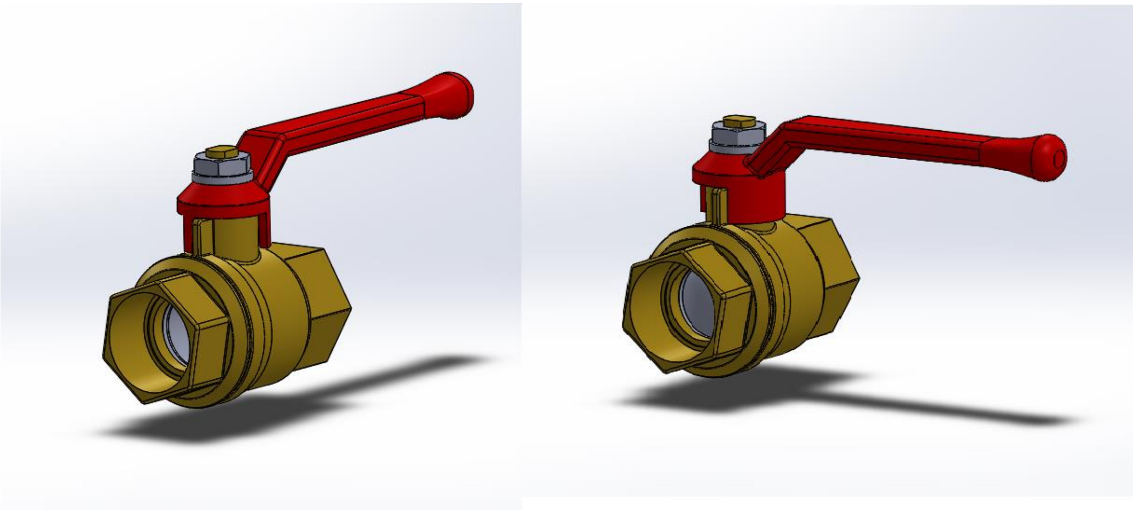

Figure 8 On the left: opened valve. On the right: closed valve

The valve was made by designing all parts separately and then combining them into assembly. The parts are as follows:

Body is the outer casing of the valve containing internal parts.

Handle is used to control the flow within the valve. Ball valves usually have handles with quarter-turn motion.

Hex nut and Clevis Pin Washer were derived from Toolbox provided by SolidWorks software. These fasteners secure the valve’s handle. The parts are classified per ISO 8675 and ISO 8738 standards respectively.

Ball has some freedom to move along with the axis of the pipeline. Quarter-turn motion moves the ball to fully open or fully close the valve.

Steam (sometimes referred to as screw) is a part of the valve that transmits the motion from the handle to the ball (within the body).

Gland nut is an independent bearing that provides a bearing surface for rotary applications. **[22]**

Valve Seat (sometimes called O-Ring) is integral part of a valve. It has a slot which provides to relieve the pressure and prevents the upstream seat from being forced against the ball. It also reduces wear and helps to achieve lower torque. **[23]**

Ports are passages for fluid to flow. In ball valves, ports are obstructed by ball to control the flow. Most ball valves have 2-3 ports.

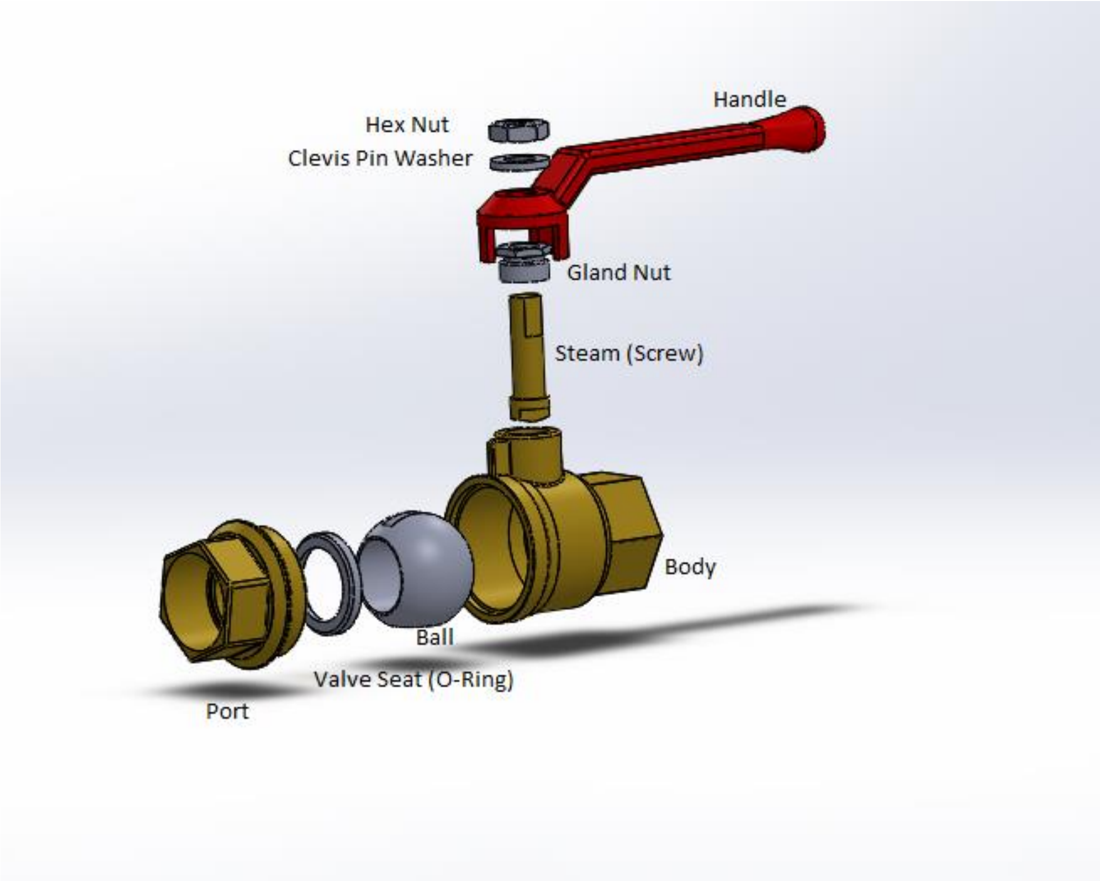

Figure 9 Exploded view of the valve

3.1.2.3 Elbows

The elbows of the piping system have nominal size of 20 mm (¾ inch) and 90º angle. They are based on ASME B16 for Pipes and Fittings standard.

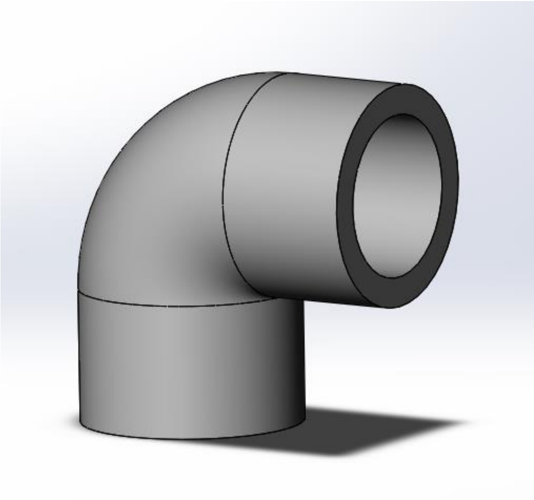

Figure 10 Elbow

3.1.3 The assembly of the pipe system

The figure below shows the assembled pipe system. It is a complete assembly that can be readily used for simulations and flow analysis.

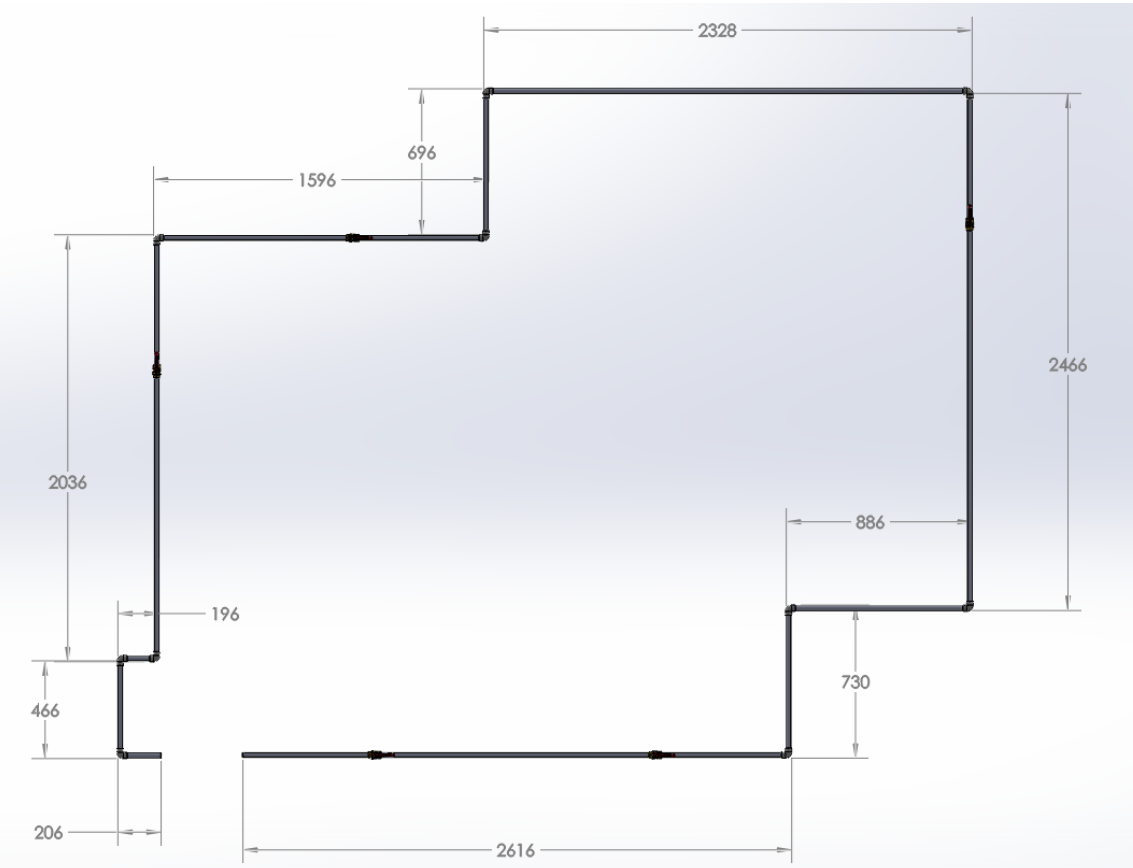

Figure 11 The complete pipe system

3.2 FloXpress Analysis

Based on the calculated flow trajectories, you can find problem areas in your design and improve them before you manufacture your parts. FloXpress add-in measures how fluid flows through a model. It helps to identify problems areas in the design and improve them before manufacturing any parts. FloXpress specifically analyses fluid flow in a fully enclosed volume. [18]

FloXpress add-in uses at least one inlet and one outlet, which is precisely what was used in the simulation of this model. To run a simulation, the lids had to be created to close the pipes. They define the boundary conditions. Extruded base feature was used to make the lids, choosing mid plane as a direction, because most problems require surface contact rather than a line contact. The base was made with the thickness of 2 mm. The lids were not merged with the pipes, because we want separate bodies.

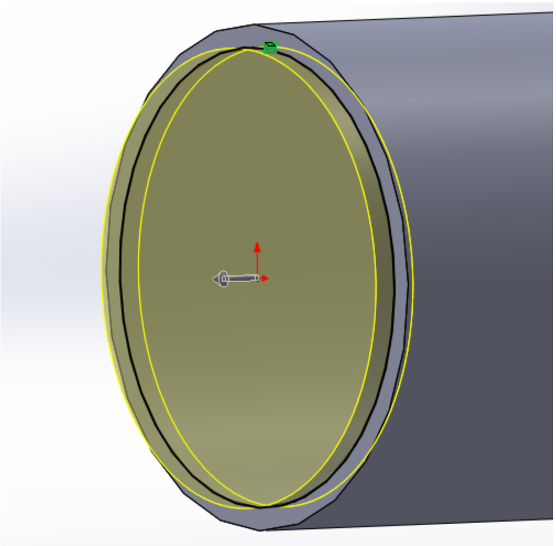

Figure 12 Lid. Extruded Base feature

FloXpress checks the geometry and if it is correct, it shows the fluid volume; then you can run the simulation. Water was chosen as the default fluid. The solid body-fluid contact surface was chosen as inlet and outlet. Two simulations with different boundary conditions were carried out.

3.2.1 Simulation 1

The inlet boundary condition was set to be volume flow, which was calculated using velocity obtained from the laboratory. i.e. 0.0002 m^3/s. The outlet boundary condition was pressure with the value of 224 kPa. The model is solved after the software performs meshing.

3.2.2 Simulation 2

The inlet pressure was chosen to be 248 kPa with ambient temperature of 293.20 K. The outlet pressure was chosen as 224 kPa. After setting boundary conditions, the software starts meshing and solving the model.

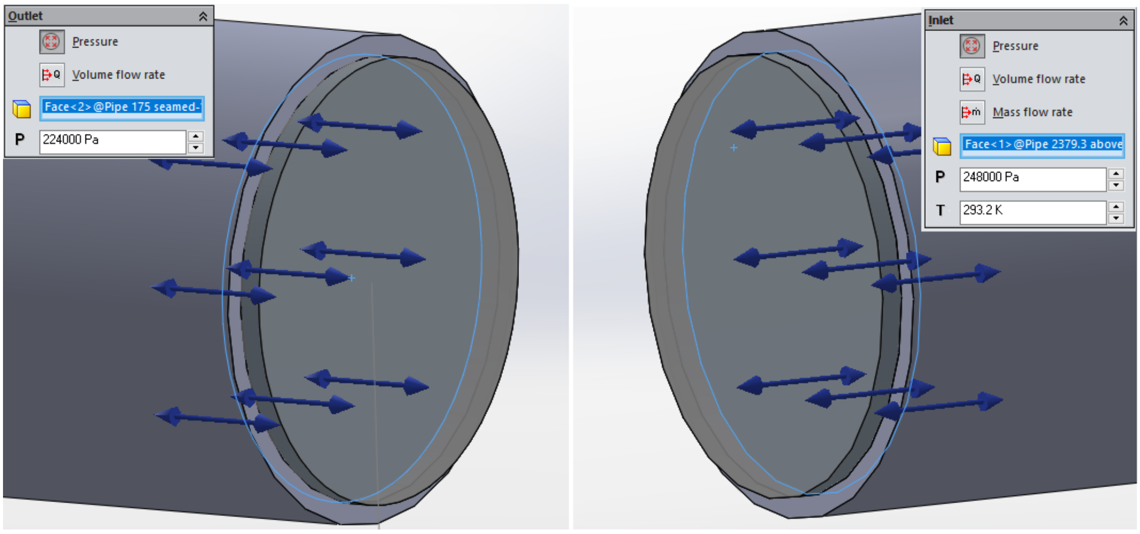

Figure 13 Inlet and outlet boundary conditions

3.3 Flow Simulation

Flow Simulation add-in is based on computational fluid dynamics. CFD is a branch of fluid dynamics that analyses and visualizes fluid flow using numerical analysis and algorithms. CFD simulates fluid passing through or around an object. Flow Simulation performs calculations based on Navier-Stokes equations to simulate the interaction of fluids with surfaces.

- Configuration

To bring the model to the flow environment, the wizard option was chosen. A new configuration was created. The chosen type of unit system to be used during the analysis was SI. The next step lets you choose the type of analysis: internal or external. The former was selected. The next step is selecting the default fluid type. Since the water flow in pipes is simulated, water was selected. To define the wall conditions, the wall was assumed to be perfectly smooth. In the next step, the default initial conditions were kept. After the configuration is finished, the flow simulation analysis appears in property manager. Flow simulation automatically selects computational domain, which, however, can be modified if needed. Computational domain is simply the boundaries, where the simulation happens. Then the boundary conditions were chosen.

- Boundary conditions

Two simulations with different boundary conditions were performed.

  - Simulation 1. For the inlet, the boundary condition type was chosen to be velocity. Inlet velocity was 0.531 m/s, as calculated in the laboratory experiment. Outlet boundary condition was chosen to be pressure, with the value of 224 kPa as in the laboratory.
  - Simulation 2. Both inlet and outlet boundary condition types were chosen to be pressure. As in the laboratory, inlet pressure was 248 kPa, outlet pressure was 224kPa.

- Goals

Goals guide the software towards an accurate and desired answer. The most important goals in this simulation were pressure drop and velocity: their average, as well as maximum and minimum values.

- Meshing

The final step before the simulation is meshing. Meshing is a representation of a given model expressed as finite set of geometric shapes. There are two types of Mesh to choose from: Global Mesh and Local Mesh. The former is used for the entire model; the latter only within a selected region. In this flow analysis, meshing was automatically generated

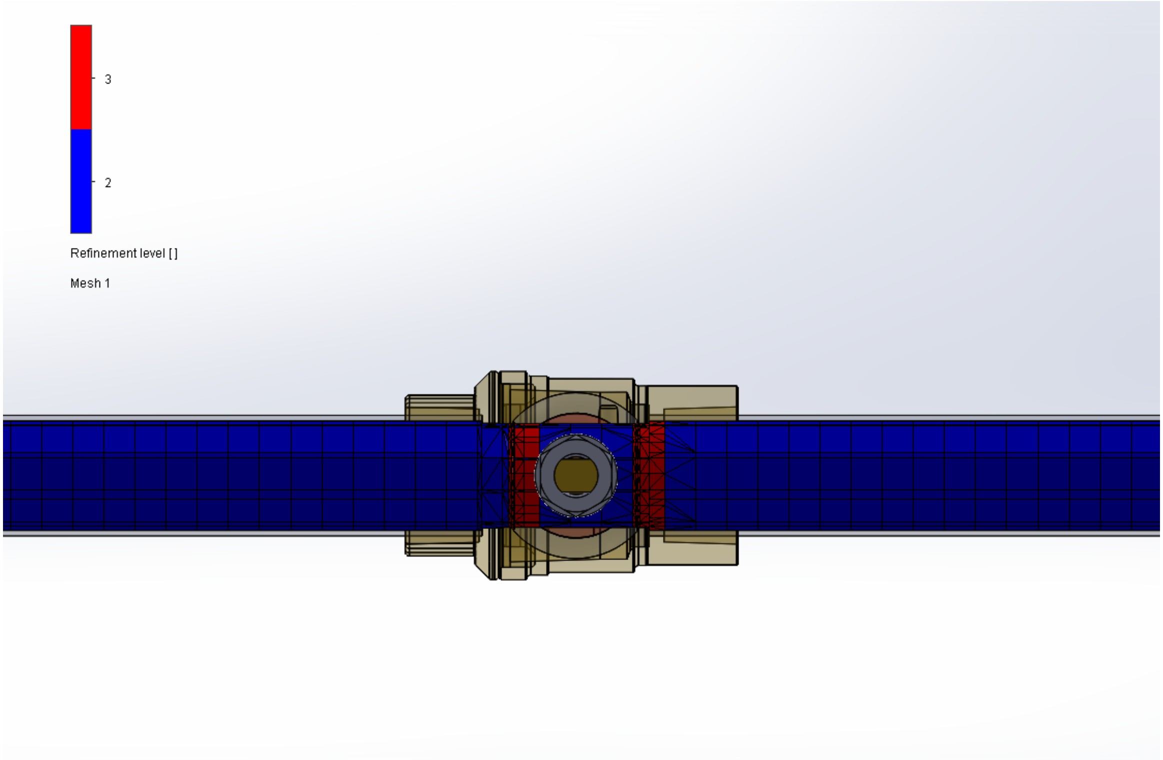

Figure 14 Mesh

3.3.1 Simulation 1

Inlet velocity was 0.531 m/s, while outlet pressure was 224 kPa. In the post processing, simulation automatically loaded the results. *Flow Trajectories* were selected in the *Results* menu. The surface of the inlet was selected as a *Starting Point*; 20 points were plotted. In the *Appearance* tab *Pipes* were chosen to illustrate the flow. The contour was coloured by either *Pressure* or *Velocity* in separate result displays.

- Pressure. Flow simulation calculated the pressure throughout the whole pipe system. As expected, pressure is continuously dropping along the system. Clearly, inlet pressure was the highest, with the pressure at the outlet being the lowest in the system. The figure below illustrates the pressure contour.

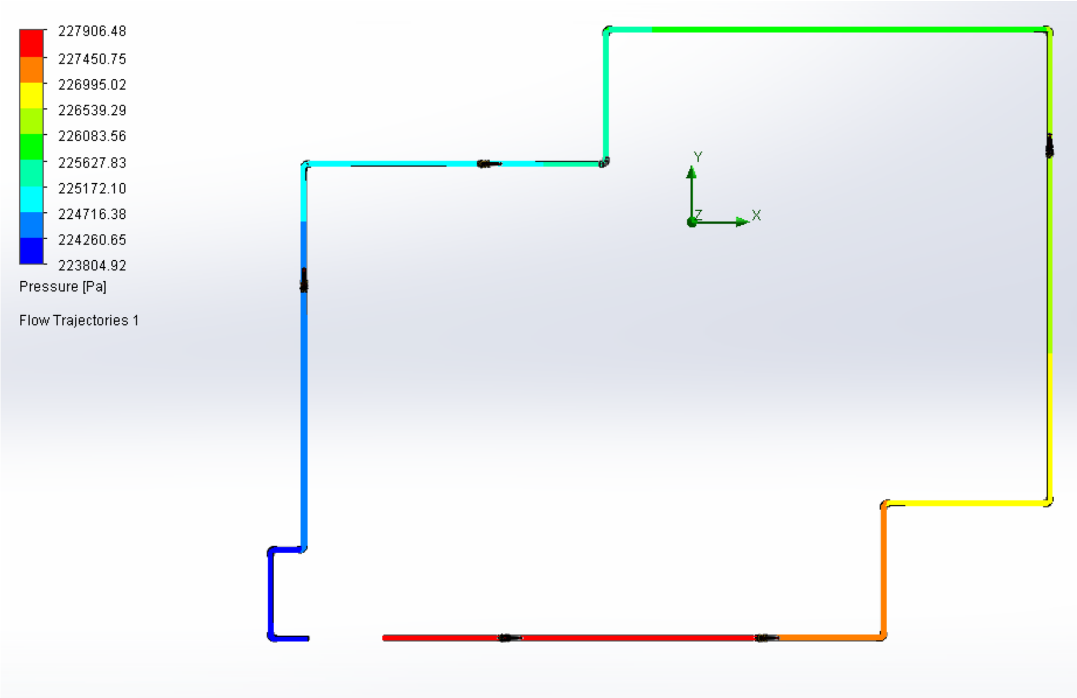

Figure 15 Pressure contour

➢ Velocity. The differences in colour in the figure below represent different velocity magnitudes.

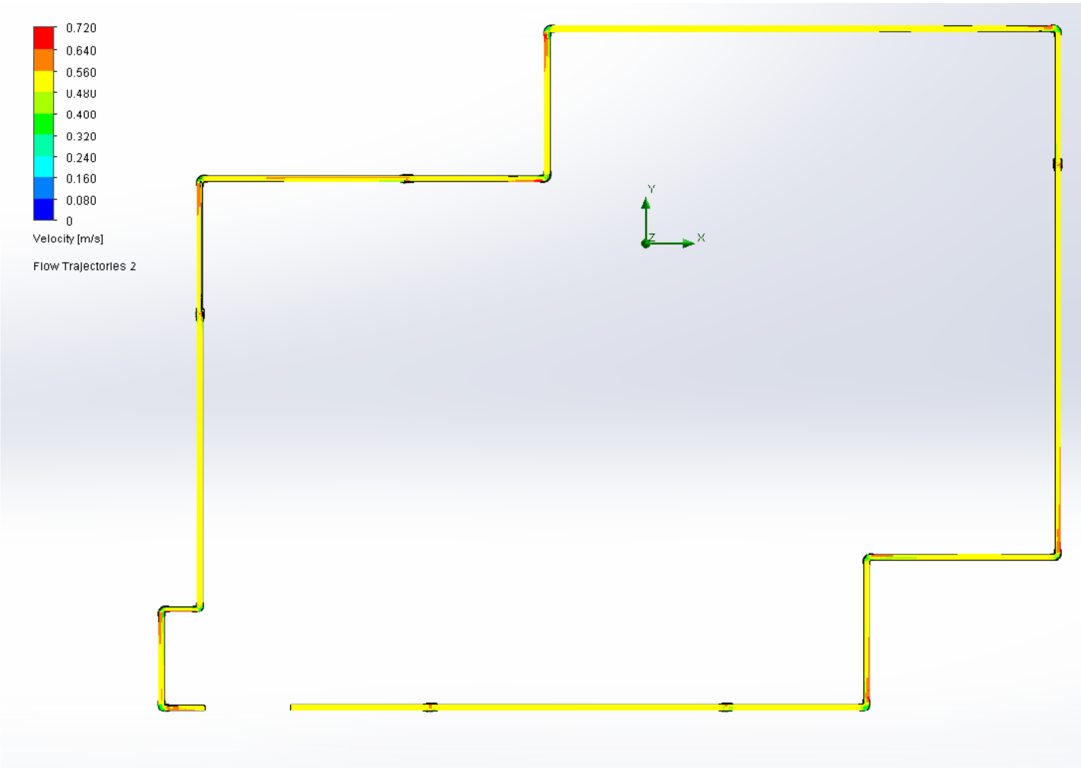

Figure 16 Velocity magnitude

Highest differences occur in the elbows and valves, as well as the ends of pipes at and next to the conjunctions.

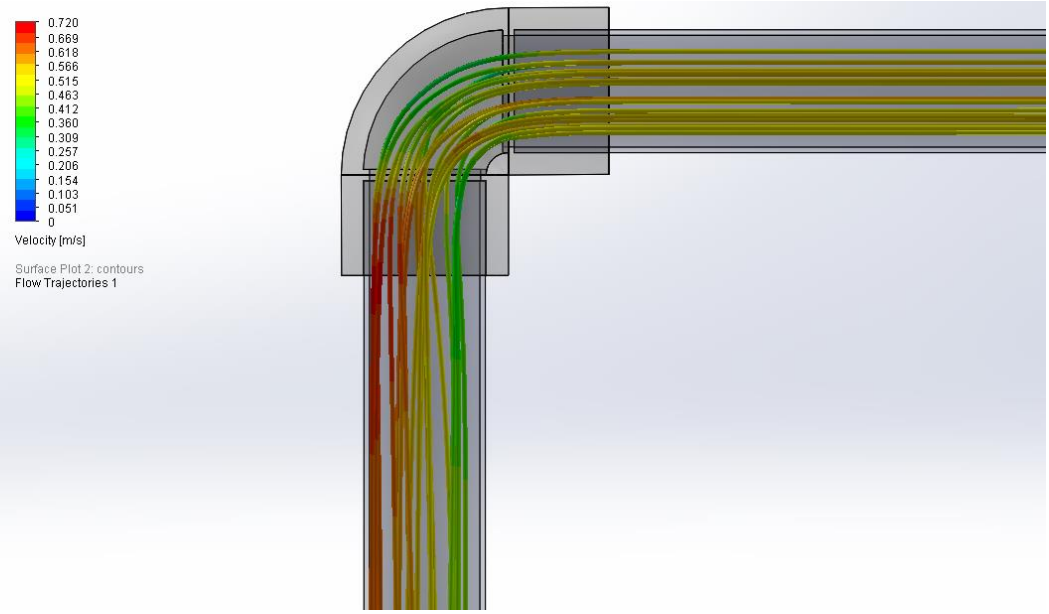

Figure 17 Flow trajectories

3.3.2 Simulation 2

The boundary condition values were as indicated in the laboratory experiment: inlet pressure 248 kPa; outlet pressure 224 kPa. In the *Results* tab, *Flow Trajectories* were plotted and displayed.

- Pressure. Flow simulation calculated the pressure throughout the whole pipe system. The variation in the colour represents the pressure in the different region of pipeline. The figure below illustrates the decreasing pressure contour starting from inlet towards the outlet.

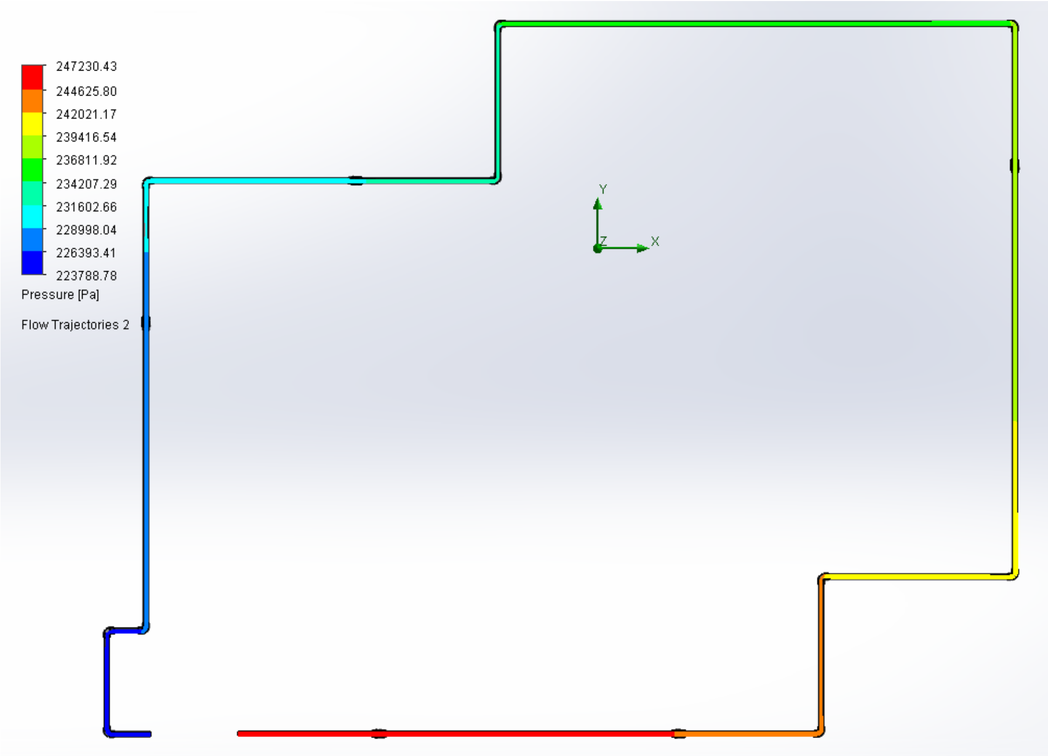

Figure 18 Pressure contour

➢ Velocity. The velocity magnitude is shown in the figure below. Contrasting colours represent the differences in the velocity magnitude.

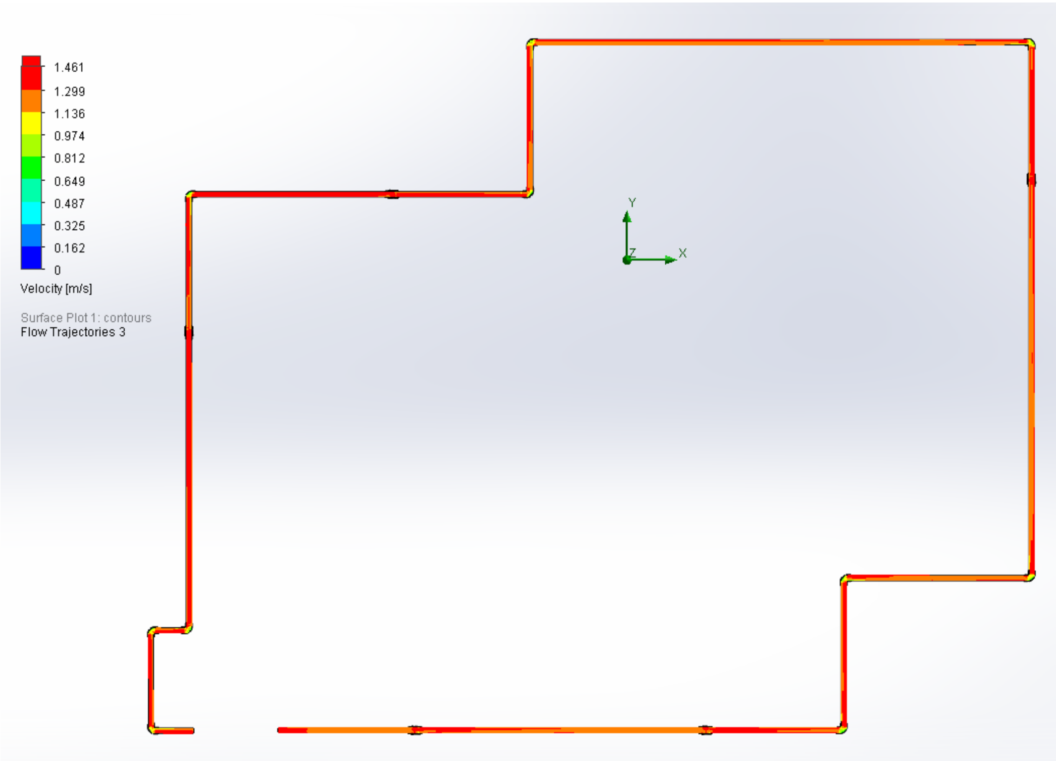

Figure 19 Velocity magnitude

As in the previous simulation, the highest differences occur in the elbows and valves, as well as the ends of pipes at and next to the conjunctions. Velocity magnitudes, however, differ. Flow trajectories in the elbow are shown in the figure below.

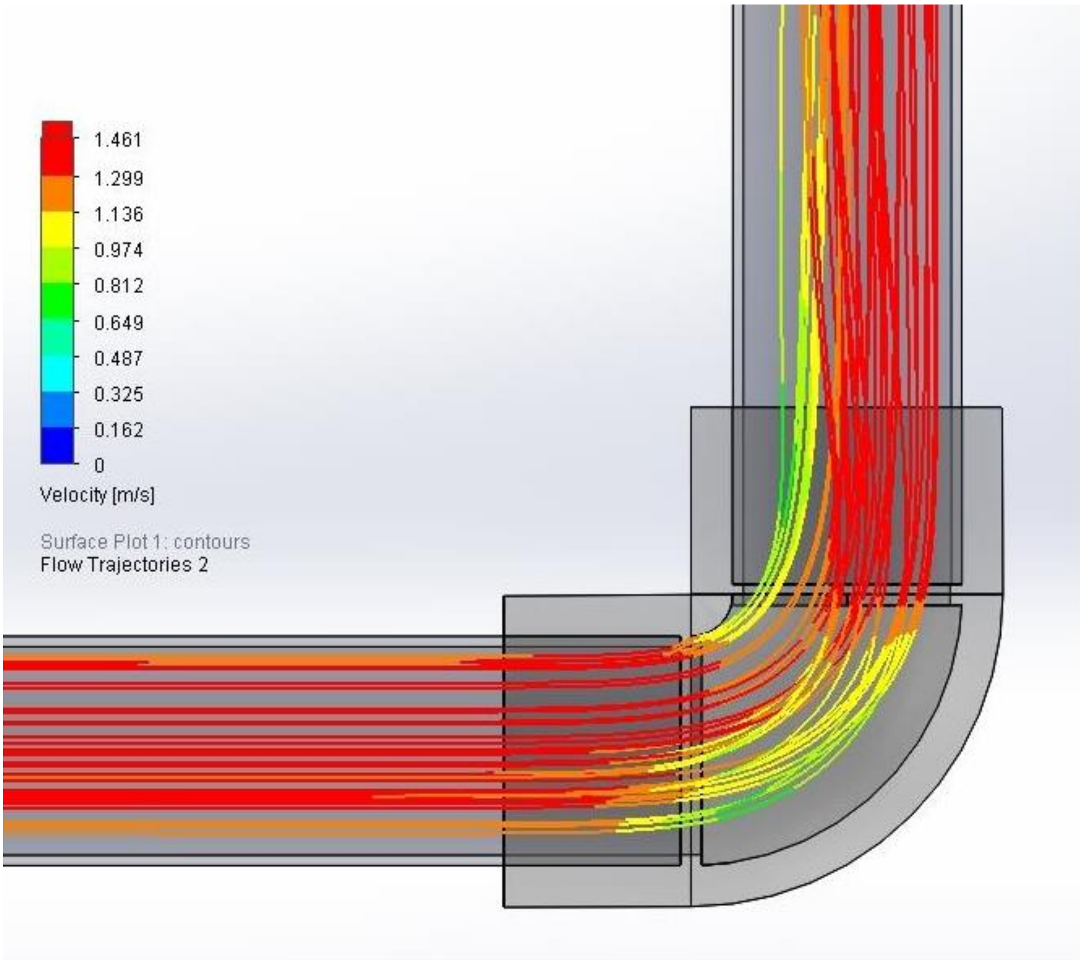

Figure 20 Flow trajectories

3.4 Laboratory experiment

The laboratory experiment was performed at the Heat Transfer laboratory in Arcada University of Applied Sciences. A fluid flow experiment allows us to have a better understanding of the applications of engineering equations in real life situations where fluid flow is involved. The analysis was mostly conducted to find the velocity of fluid flow in a pipeline, and calculate the head loss. The fluid flow in the laboratory is a cyclic process. By using a pump, water from the reservoir flows in the stream channel. The water flows through the whole pipe network and until it reaches the reservoir.

Equipment:

- Reservoir. It is a water storage, through which the process of incoming and outgoing flow continues.
- Pump. It pumps the water in a pipe system. The pump reads 22 W.
- Pressure Gauge. It is an instrument to measure and display pressure. It is connected at the starting and ending point of the flow in the pipeline. Inlet pressure gauge reads 248 kPa; outlet pressure gauge reads 224 kPa.
- Rotary Flow Meter. A device to measure flow. It has a dial of 10 units. The flow meter made 5 revolutions in 30 s.
- Vernier Calliper. It is a scale device used to measure the diameter of the pipe.

4 RESULTS

4.1 Laboratory experiment calculation

In the laboratory experiment velocity, flow rate, and head loss were calculated.

The values used for the calculation are:

Table 1 Values for calculation

Inlet Pressure	248 kPa
Outlet Pressure	224 kPa
Inside diameter of the pipe	20 mm = 0.02 m
Outside diameter of the pipe	22mm = 0.022 m
Power of the Pump	22W

Calculation of the flow rate

The flow meter made 5 revolutions in 30 s, and it has 10 divisions.

Thus,

$$1rev = 0.0001\ m^3$$

$$\dot{V} = \frac{Flow\ meter\ unit \times Number\ of\ revolution}{Time} = \frac{10^{-4}\times 50}{30} = 1.667\times 10^{-4}\ m^3/s$$

Calculation of velocity

$$V_{avg} = \frac{\dot{V}}{A}$$

Where,

A = cross-sectional area of a pipe

$$A = \frac{\pi d^2}{4} = \pi \times \frac{(0.02)^2}{4} = 3.1416\times 10^{-4}\ m^2$$

Then,

$$V_{avg} = \frac{1.667 \times 10^{-4}\ m^3/s}{3.1416\times 10^{-4}\ m^2} = 0.531\ m/s$$

Calculation of head loss

- Using Bernoulli equation

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 + H_L$$

The inlet and outlet height is the same. i.e. $y_1 = y_2$. The flow is fully developed, therefore the velocity at the ends of a pipe is the same. i.e. $v_1 = v_2$.

Therefore,

$$H_L = \frac{(p_1 - p_2)}{\rho g}$$

$$H_L = \frac{(248 \times 10^3 - 224 \times 10^3)}{10^3 \times 9.81} = 2.446\ m$$

- Calculation of Reynolds number

$$Re_D = \frac{\rho V_{avg} D}{\mu}$$

Where,

$\rho = 1000\ kg/m3$

$V_{avg} = 0.531\ m/s$

$D = 0.02\ m$

$\mu = 1.002 \times 10^{-3}\ Pa.s$; Dynamic viscosity of water at 20° C temperature [24]

$$Re_D = \frac{\rho V_{avg} D}{\mu} = \frac{1000 \times 0.531 \times 0.02}{1.002 \times 10^{-3}} = 10598.8$$

- Major head loss due friction

The formula below is called Darcy's equation and is used for the calculation in fully developed flow:

$$H_{L-major} = f \frac{L}{D} \frac{V_{avg}^2}{2g}$$

Where,

f = friction factor = 0.03; the value was taken from Moody Diagram

L = pipe length (m) = 14.2 m

V_{avg} = average velocity (m/s) = 0.531m/s

D = internal diameter (m) = 0.02 m

g = acceleration due to gravity (g = 9.81 m/s^2)

$$H_{L-major} = 0.03 \frac{14.2}{0.02} \times \frac{(0.531)^2}{2 \times 9.81} = 0.306\, m$$

- Minor head loss due to bends

$$H_{L-minor} = k \frac{V_{avg}^2}{2g}$$

Where,

k = minor head loss coefficient = 0.3; for regular 90^o flanged elbow. [25]

$$H_{L-minor} = 0.3 \frac{(0.531)^2}{2 \times 9.81} = 4.31 \times 10^{-3}\, m$$

The total head loss:

$$H_L = \sum H_{major\ losses} + \sum H_{minor\ losses} = (0.306 + 4.31 \times 10^{-3}\, m) = 0.31\, m$$

4.2 FloXpress Analysis

Maximum velocity obtained from FloXpress Analysis was 0.837 m/s in Simulation 1. While Simulation 2 attained the maximum velocity with the value of 1.735 m/s.

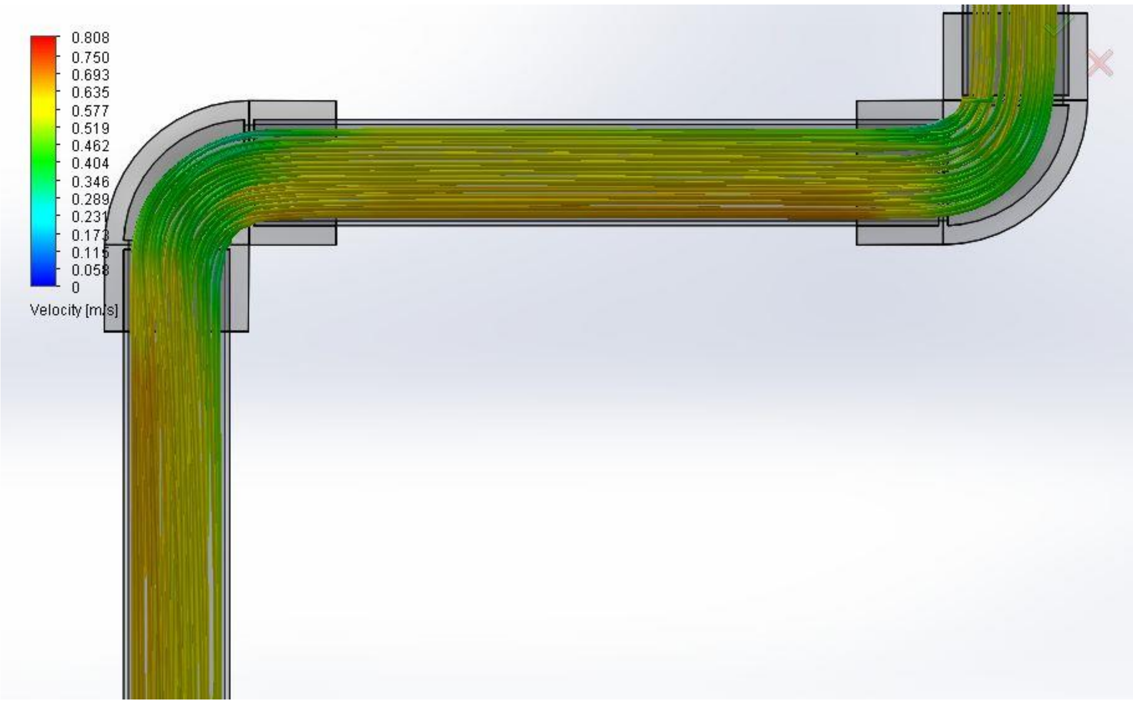

Figure 21 Velocity magnitude

Table 2 FloXpress simulation results

FloXpress:	**Input data:**	**Simulated results:**
Simulation 1	Inlet volumetric flow rate: 0.0002 m^3/s Outlet pressure: 224 kPa	Maximum velocity throughout the pipeline: 0.837 m/s
Simulation 2	Inlet pressure: 248 kPa Outlet pressure: 224 kPa	Maximum velocity throughout the pipeline: 1.735 m/s

FloXpress visualizes flow and calculates maximum velocity. It does not calculate neither average velocity nor pressure.

4.3 Flow Simulation

The type of flow was found to be turbulent based on the calculation of Reynolds number with a value of 10598.8. The turbulent flow simulation was used in Flow Simulation to find the magnitude of velocity. As seen in the figure below, minimum velocity occurs in the inside the elbow indicated by light blue colour. Maximum velocity occurs in the bottom of a pipe right before a fluid enters the elbow.

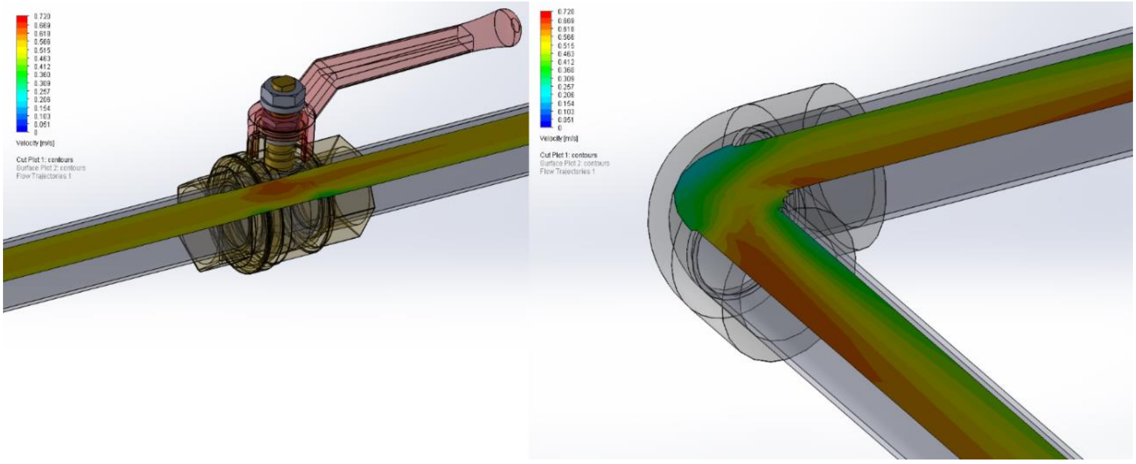

Figure 22 Cut plots. Velocity

The outlet velocity was found to be 0.542 m/s respectively in the Simulation 1. While the values obtained in Simulation 2 for inlet and outlet were 1.275 m/s and 1.378 m/s respectively. Simulation 1 calculated pressure for inlet 228029.65 Pa.

Head loss calculation is done using those values obtained from Flow Simulation. Bernoulli's equation can be used for this calculation:

$$\mathrm{p_1} + \frac{1}{2}\rho \mathrm{v}_1^2 + \rho g \mathrm{y}_1 = \mathrm{p}_2 + \frac{1}{2}\rho \mathrm{v}_2^2 + \rho g \mathrm{y}_2 + \mathrm{H_L}$$

Head loss calculation using values from Simulation 1:

$$\mathrm{H_L} = \frac{(\mathrm{p}_1 - \mathrm{p}_2)}{\rho} + \frac{(\mathrm{v}_1^2 - \mathrm{v}_2^2)}{2} = \frac{(228.03 - 224.01)\times 10^3}{10^3} + \frac{(0.531^2 - 0.542^2)}{2}$$

$$= 4.02 + 0.141 - 0.147 = 4.014\ \mathrm{m}^2/s^2$$

Now, this value is divided by acceleration of gravity:

$$H_L = \frac{4.014\ m^2/s^2}{9.81\ m/s^2} = 0.409\ \mathrm{m}$$

Head loss calculation using values from Simulation 2:

$$H_L = \frac{(p_1 - p_2)}{\rho} + \frac{(v_1^2 - v_2^2)}{2} = \frac{(248.93 - 224.98)\times 10^3}{10^3} + \frac{(1.275^2 - 1.378^2)}{2}$$

$$= 23.95 + 0.813 - 0.949 = 23.814 \text{ m}$$

This value is divided by acceleration of gravity:

$$H_L = \frac{23.814 \ m^2/s^2}{9.81 \ m/s^2} = 2.428 \text{ m}$$

Table 3 Flow Simulation results

Flow Simulation:	**Input data:**	**Simulated results:**	**Calculated results:**
Simulation 1	Inlet velocity: 0.531 m/s Outlet pressure: 224.00 kPa	Inlet pressure: 228.03 kPa Outlet velocity: 0.542 m/s Average velocity throughout the pipe: 0.532 m/s	Head loss (H_L): 0.409 m
Simulation 2	Inlet pressure: 248.00 kPa Outlet pressure: 224.00 kPa	Inlet velocity: 1.275 m/s Outlet velocity: 1.378 m/s Average velocity throughout the pipe: 1.375 m/s	Head loss (H_L): 2.428 m

5 DISCUSSION

The average velocity of fluid from the laboratory experiment was calculated to be 0.531 m/s. To determine the flow type, Reynolds number was calculated. The value was found to be 10598.8, indicating that the flow is turbulent because the value is above 4000. The simulation was done using turbulent flow to find the velocity magnitude. The average velocity throughout the pipeline from Flow Simulation was found to be 0.532 m/s (Simulation 1) and 1.375 m/s (Simulation 2). The maximum velocity throughout the pipeline using FloXpress was found to be 0.837 m/s and 1.735 m/s (Simulation 2). FloXpress, however, does not calculate average velocity and pressure.

The table below shows that the average velocity obtained by SolidWorks, where inlet and outlet boundaries were velocity and pressure respectively, is approximately the same as the average velocity calculated in laboratory experiment.

Table 4 Average velocities obtained from Experimental and SolidWorks simulation

1	Average velocity obtained from laboratory experiment	0.531 m/s
2	Average velocity obtained by SolidWorks. Simulation 1	0.532 m/s
2	Average velocity obtained by SolidWorks. Simulation 2	1.375 m/s

The velocity obtained in the second SolidWorks simulation, where inlet and outlet boundary conditions where pressure, is more than twice higher than in the previous calculations. This is illustrated in the graph below.

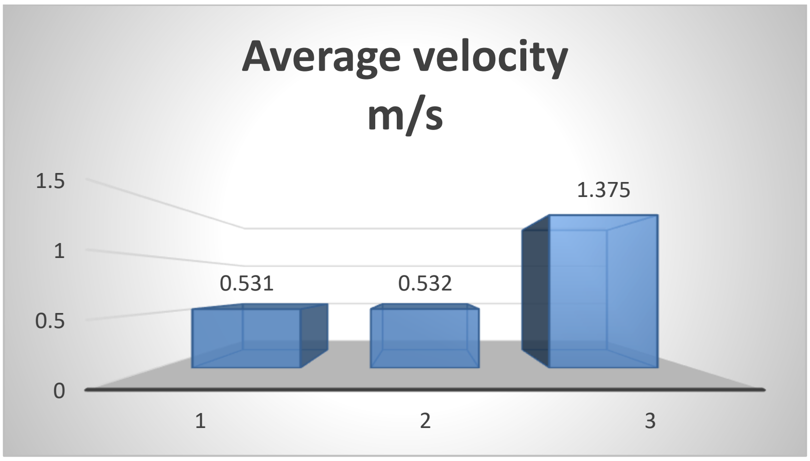

Figure 23 Chart graph of average velocities obtained from Experimental and SolidWorks simulation

The difference can be explained by head loss. Flow Simulation assumes that the wall is adiabatic, which means that there is no heat transfer. Software also assumes that the walls have zero roughness; neither of which is realistic. It also neglects minor head loss due to bends. All these factors cause pressure drop.

The head loss was calculated using Bernoulli's equation. The values used for the calculations were obtained from the laboratory experiment and simulations. Since FloXpress add-in calculates only maximum velocity, only values from Flow Simulation were used. Calculated head loss values are given in the table below.

Table 5 Head loss

	Head loss (H_L)
Laboratory experiment	2.446 m
Flow Simulation. Simulation 1	0.409 m
Flow Simulation. Simulation 2	2.428 m

As seen in the table, the head loss was greater in the laboratory experiment compared to the simulations. This correlates with the aforementioned reasons explaining the differences in the obtained velocity values, since head loss is proportional to the square of the velocity.

In *Analysis and FEM Simulation of Flow of Fluids in Pipes* by Saroj Acharya, COMSOL simulation was performed to obtain average velocity for the same pipe system design.

Table 6 The velocities obtained from experimental and COMSOL simulation [26]

1	Average Velocity obtained from laboratory experiment	0.532 m/s
2	Average turbulent Velocity from the inlet and outlet obtained by COMSOL simulation for the same design	0.529 m/s
3	Average laminar Velocity from the inlet and outlet obtained by COMSOL simulation for the same design	0.114 m/s

As seen in the table above, the average velocity from the COMSOL simulation was found to be 0.529 m/s. The velocity found experimentally and from COMSOL simulation is roughly the same. However, average laminar velocity was 0.114 m/s. The value for laminar flow had a high difference compared to experimental value. Thus, the flow could not be laminar. The simulations were performed before the calculation of Reynolds number. Later, average velocity was used to calculate the Reynolds number, which determined that the flow was turbulent. In experimental calculation of head loss, average velocity was used, however, the COMSOL simulation provided the different velocity and pressure at inlet and outlet. [26]

Moreover, SolidWorks simulation also provided different velocity and pressure at inlet and outlet. It is clear, that the different value of velocity and pressure had a direct impact on the head loss result obtained by both softwares. It is the main reason behind the differences in the values obtained.

6 CONCLUSION

The basic idea of this thesis is to design a pipe system and run a flow simulation for the observation of the flow of fluids in pipes and compare it with the results obtained in the laboratory.

Navier-Stokes equations predict the fluid velocity and its pressure in a given geometry. The Navier-Stokes equations are momentum equation, energy equation, and continuity equation. These equations apply to any point in the flow and thus all details of the flow can be solved everywhere in the flow domain. Sometimes Navier-Stokes equations can be simplified, however, most problems in fluid mechanics are complicated in nature and are very difficult to solve. Thus, it often creates differing opinions in laboratory experiments, and simulation evaluations.

The laboratory experiment was performed in Heat Transfer laboratory. Using flow rate, the velocity was calculated; obtained value was 0.531 m/s. The average velocity from the SolidWorks simulation was found to be 0.532 m/s or 1.375 m/s depending on the boundary conditions.

The head loss was calculated using the values obtained from the laboratory experiment and simulations. Calculated head loss from the laboratory experiment was found to be 2.446 m, which is greater head loss value than the result from the simulations i.e. 0.409 m and 2.428 m respectively.

An adiabatic wall is a theoretical concept; it is a wall that does not allow heat transfer from one side to another. However, in real life situations any thermal insulation allows some transfer of heat, which causes drop in pressure. Unless indicated otherwise, softwares assume that the walls of pipes, or any other geometry, are adiabatic. Moreover, head loss is also caused by friction in pipes, fittings, corrosion, etc. These factors may distort the results that softwares simulate.

Meshing is a crucial step in design analysis. Mesh allows user customization; however extensive knowledge is required for such a task to yield successful simulation. The automatic mesh generates a mesh considering model’s geometry, volume, surface area, and other specifications. This is by far a superior option for new users. Meshing is difficult for low/medium end workstations. If possible, a fast computer with many gigabytes RAM is highly recommended; this kind of simulation can require many hours or even days for large and complex models.

SolidWorks is a powerful tool for new and advanced users. It allows the design and simulation within the same software. For the same type of fluid flow simulation, COMSOL is recommended as an alternative software. NASTRAN is another powerful FEA (Finite Element Analysis) program that can be used to analyse flow.

Fluid flow, heat transfer and other problems almost always are initially performed using analytical tools. Often, the results obtained are much more detailed, sometimes even more accurate, than experimental analysis. Simulation is a less expensive and often a faster way to analyse and solve various problems.

REFERENCES

[1] OpenStax, University Physics Volume 1, OpenStax, 2016.

[2] R. W. Fox, A. T. McDonald and P. J. Pritchard, Introduction to Fluid Mechanics, Wiley, 2004.

[3] D. C. Rennels and H. M. Hudson, Pipe Flow : A Practical and Comprehensive Guide (1), New Jersey: Wiley, 2012.

[4] Y. A. Cengel and J. M. Cimbala, Fluid Mechanics. Fundamentals and Applications. Third Edition, New York: McGraw Hill, 2014.

[5] OpenStax, "Characteristics of Flow," in *University Physics Volume 1*, OpenStax, 2016, p. 728.

[6] Y. A. Cengel and J. M. Cimbala, "The Entrance Region," in *Fluid Mechanics. Fundamentals and Applications. Third Edition*, New York, McGraw Hill, 2014, p. 325.

[7] Y. A. Cengel and J. M. Cimbala, "Fluid Mechanics. Fundamentals and Applications. Third Edition," in *Entry Lengths*, New York, Mc Graw Hill, 2014, p. 326.

[8] R. Hardee, "Calculating Head Loss in a Pipeline," Pumps & Systems, [Online]. Available: http://www.pumpsandsystems.com/pumps/april-2015-calculating-head-loss-pipeline. [Accessed 29 10 2016].

[9] N. Power, "Classification of Head Loss," [Online]. Available: http://www.nuclear-power.net/nuclear-engineering/fluid-dynamics/bernoullis-equation-bernoullis-principle/head-loss/classification-of-head-loss/. [Accessed 30 10 2016].

[10] N. Power, "Head Loss – Pressure Loss," [Online]. Available: http://www.nuclear-power.net/nuclear-engineering/fluid-dynamics/bernoullis-equation-bernoullis-principle/head-loss/. [Accessed 30 10 2016].

[11] F. M. White, Fluid Mechanics. 7th edition, NY: McGraw-Hill, 2011.

[12] Pentair, “Head Loss in Piping Systems,” [Online]. Available: http://www.hydromatic.com/ResidentialPage_techinfopage_headloss.aspx. [Accessed 02 11 2016].

[13] Comsol, “Navier-Stokes Equations,” [Online]. Available: https://www.comsol.com/multiphysics/navier-stokes-equations. [Accessed 02 11 2016].

[14] Thermopedia, “Friction Factors for Single Phase Flow in Smooth and Rough Tubes,” 14 02 2011. [Online]. Available: http://www.thermopedia.com/content/789/. [Accessed 23 10 2016].

[15] B. C. Encyclopedia, “Applied Process Design for Chemical and Petrochemical Plants,” [Online]. Available: http://chempedia.info/info/147184/. [Accessed 06 04 2017].

[16] SolidWorks, “3D CAD Design Engineering Software Tools,” Dassault Systemes, [Online]. Available: https://www.solidworks.com/sw/3d-cad-design-software.htm. [Accessed 11 10 2016].

[17] SolidWorks, “SOLIDWORKS Flow Simulation,” Dassault Systemes, [Online]. Available: https://www.solidworks.com/sw/products/simulation/flow-simulation.htm. [Accessed 11 10 2016].

[18] SolidWorks, “SOLIDWORKS FloXpress,” Dassault Systemes, [Online]. Available: http://www.solidworks.com/sw/products/simulation/floxpress.htm. [Accessed 12 10 2016].

[19] SolidWorks, “Computational Fluid Dynamics (CFD),” Dassault Systemes, [Online]. Available: http://www.solidworks.com/sw/products/simulation/computational-fluid-dynamics.htm. [Accessed 12 10 2016].

[20] SolidWorks, “CAD Library: SOLIDWORKS Toolbox,” Dassault Systemes, [Online]. Available: http://www.solidworks.com/sw/products/3d-cad/solidworks-toolbox.htm. [Accessed 05 01 2017].

[21] SolidWorks, “Weldments,” Dassault Systemes, [Online]. Available: http://help.solidworks.com/2016/English/SolidWorks/sldworks/c_Weldments_Overview.htm. [Accessed 03 03 2017].

[22] T. Bearings, “Bushes,” Townsend Bearing and Transmission Ltd, [Online]. Available: http://www.townsendbearings.co.uk/index.php/bushes/. [Accessed 07 03 2017].

[23] N. V. a. Automation, “High Performance Ball Valve,” NIVZ, [Online]. Available: http://www.nivzvalves.com/Products/Ball-valve/High-Performance-Ball-Valve. [Accessed 03 03 2017].

[24] I. A. f. t. P. o. W. a. S. (IAPWS), “Release on the IAPWS Formulation 2008 for the Viscosity of Ordinary Water Substance,” 21 September 2008. [Online]. Available: http://www.iapws.org/relguide/viscosity.html. [Accessed 26 03 2017].

[25] “Minor loss coefficients for common used components in pipe and tube systems,” Engineering Toolbox, [Online]. Available: http://www.engineeringtoolbox.com/minor-loss-coefficients-pipes-d_626.html. [Accessed 10 03 2017].

[26] S. Acharya, “Analysis and FEM Simulation of Flow of Fluids in Pipes,” Arcada University of Applied Science, Helsinki, 2016.