

University of Wales Trinity Saint David

School of Engineering

Mathematical Modelling of Hydraulic Transient
Within a LNG Transfer Pipeline

James Neucker

P139577

BEng Mechanical Engineering

25/04/2017

Major Project

M6X00417



Computational Modelling of Hydraulic Transient

'This thesis is the result of the author's original research. It has been composed by the author and has not been previously submitted for examination which has led to the award of a degree.'

Signed:

Date:

Abstract

The motivation behind this project is to analyse the behaviour of a LNG transfer line during the event of a down-stream break-away coupling activating due to ship movement. The analysis in this report has been completed using explicit finite difference methods with a variety of initial conditions to indicate where up-stream breakaway coupling activation may occur, which may result in a potentially flammable atmosphere. The relevance of this analysis is increasing due to the growth in use of LNG as a fuel as industry moves away from the use of traditional fossil fuels. The results of the analysis show which initial conditions result in either partial separation or complete activation of an up-stream breakaway coupling which indicate that the situations in which these conditions exist require further safety precautions.

Contents

Introduction	1
Research.....	2
Hydraulic Transient	2
LNG Transfer Line.....	5
Break-away Coupling	9
Liquefied Natural Gas.....	12
Methodology.....	14
The Problem.....	14
Procedure.....	15
Finite Difference Method.....	15
Initial Conditions	19
MATLAB Code	20
Liquid Properties	20
Pipe Properties.....	20
Initial Calculations.....	22
Internal Pipe Friction Calculation.....	23
.....	28
Results.....	28
Test Conditions 1	29
Test Conditions 2	32
Test Conditions 3	35
Test Conditions 4	38
.....	39
Test Condition 5	41
Test Condition 6	44
Test Conditions 7	47
Test Condition 8.....	50
Test Conditions 9	53
Test Conditions 10	56
Analysis	59
Valve Closure	59
Pipe Internal Diameter.....	62
Wall Thickness.....	65
Increase in Volumetric Flowrate.....	67

Computational Modelling of Hydraulic Transient

Discussion.....	70
Conclusion.....	73
Evaluation	74
Works Cited.....	75
Appendices.....	76
Complete Code.....	76
Figure 1: LNG Vessel at Karratha	5
Figure 2 : Simplified Image of a LNG Transfer Line with Breakaway Couplings.....	5
Figure 3 : LNG comparison to CNG and Natural Gas	7
Figure 5: KLAW Cryogenic Break-away Coupling	10
Figure 6: Sequential closure of Flap Valve within Break-away Coupling:	10
Figure 7: Break Pin Side View.	11
Figure 8: LNG Tanker with Fuel Spheres	12
Figure 9: Initial Condition Graphs	27
Figure 10: Final Condition graphs	28
Figure 11 Test Condition 1 Final Plot	29
Figure 12 : Test Conditions 1 Pressure Surge Propagation.....	30
Figure 13 : Test Conditions 1 Decrease in Volumetric Flowrate.....	31
Figure 14 : Test Conditions 2 Final Plot.....	32
Figure 15: Test Condition 2 Pressure Surge Propagation	33
Figure 16: Test Conditions 2 Decrease in Volumetric Flowrate.....	34
Figure 17: Test Conditions 3 Final Plot.....	35
Figure 18: Test Conditions 3 Pressure Surge Propagation.....	36
Figure 19 : Test Condition 3 Decrease in Volumetric Flowrate	37
Figure 20: Test Conditions 4 Final Plot.....	38
Figure 21: Test Conditions 4 Pressure Surge Propagation.....	39
Figure 22: Test Conditions 4 Decrease in Volumetric Flowrate.....	40
Figure 23: Test Conditions 5 Final Plot.....	41
Figure 24: Test Conditions 5 Pressure Surge Propagation.....	42
Figure 25: Test Conditions 5 Decrease in Volumetric Flowrate.....	43
Figure 26: Test Conditions 6 Final Plot	44
Figure 27: Test Conditions 6 Pressure Surge Propagation.....	45
Figure 28: Test Conditions 6 Decrease in Volumetric Flowrate.....	46
Figure 29: Test Conditions 7 Final Plot.....	47
Figure 30 : Test Conditions 7 Pressure Surge Propagation.....	48
Figure 31 : Test Conditions 7 Decrease in Volumetric Flowrate.....	49
Figure 32: Test Conditions 8 Final Plot.....	50
Figure 33: Test Conditions 8 Pressure Surge Propagation.....	51
Figure 34: Test Conditions 8 Decrease in Volumetric Flowrate.....	52
Figure 35: Test Condition 9 Final Plot	53
Figure 36 : Test Conditions 9 Pressure Surge Propagation.....	54
Figure 37: Test Conditions 9 Decrease in Volumetric Flowrate.....	55
Figure 38: Test Conditions 10 Final Plot.....	56

Computational Modelling of Hydraulic Transient

Figure 39: Test Conditions 10 Pressure Surge Propagation.....	57
Figure 40 : Test Condition 10 Decrease in Volumetric Flowrate	58
Figure 41: Comparison of valve closures 0.01 vs 0.25	60
Figure 42: VC 0.25 vs VC 0.01 effect on flowrate.....	61
Figure 43: Pressure Surge 24" (Día 1) and 18" (Día 2)	63
Figure 44: Pressure Surge 24"(Dia 1) vs 12"(Dia 3).....	64
Figure 45: Effect of Wall Thickness on Pressure Surge 2" (Wt 1) and 4" (Wt 2).....	66
Figure 46: Effects of Increased Volumetric Flow Rate	68
Figure 47: Effect of Increased Volumetric Flow Rate.....	69
Figure 48: Axial loading on coupling	71

Equation 1 Joukowsky.....	3
Equation 2 Sonic Wave Speed	3
Equation 3 Continuity Equation.....	4
Equation 4 Momentum Equation	4
Equation 5: Taylors theorem (Stroud, 1993)	15
Equation 6: Forward Difference Approximation (Chaudry, 2014).....	16
Equation 7: Central Difference Approximation (Chaudry, 2014).....	16
Equation 8: Momentum equation with CDA (Chaudry, 2014).....	18
Equation 9: Pipe resistance.....	18
Equation 10: Continuity equation with CDA (Chaudry, 2014)	18
Equation 11: Stability criterion (Chaudry, 2014)	18
Equation 12:Reservoir piezometric head pressure.....	19
Equation 13: Reservoir flowrate	19
Equation 14:Valve piezometric head pressure	19
Equation 15:Valve flowrate	19
Equation 16:Valve flowrate after Breakaway coupling activation.....	19
Equation 18: Axial load calculation	71

Table 1 : Chemical Analysis of SSC Invar 36	6
Table 2: Thermal Expansion of SSC Invar 36	7
Table 3 : Mechanical Properties of SSC Invar 36.....	8
Table 4 : Table of 36 NiFe Alloy.....	8
Table 5 : Calculated Coupling Break Force.....	12
Table 6 : LNG Chemical Composition	13
Table 7 : Typical LNG Physical Properties	13
Table 8 : Test Conditions 1.....	29
Table 12 : Test Conditions 2.....	32
Table 14 : Test Conditions 3	35
Table 15 : Test Conditions 4.....	38
Table 16 : Test Conditions 5.....	41
Table 17 : Test Conditions 6.....	44
Table 18 : Test Conditions 7.....	47
Table 19 : Test Conditions 8.....	50

Computational Modelling of Hydraulic Transient

Table 20 : Test Conditions 9	53
Table 21 : Test Condition 10	56
Table 22 : Calculated Interface for Break-away Coupling	71
Table 23 : $P_i * a_2$ Calculation for Axial Loading	72
Table 24 : Axial Loading on Break-away Coupling	72

Introduction

The purpose for this project is to analyse the behaviour of Liquefied Natural Gas within a transfer pipeline when subjected to the conditions of an emergency shut down. During an emergency shut down the transfer of the fluid must be stopped as quickly as possible to prevent the loss of product from the system, the transfer can be stopped by a variety of mechanisms such as pneumatic Remotely Operated Shut Off Valves (ROSOV), the immediate shut down of transfer pumps or vacuum pumps and the activation of Break-away couplings. In the transfer of LNG both the ROSOV's and the pumps are connected to the emergency shut down (ESD) system that can be manually or automatically activated by a drop-in pressure in the air line that charges the ROSOV's, the detection of Hydrocarbons in the atmosphere by a Gas detection system or the activation of the local fire alarm system. Each of these changes in condition activate the ESD system which shuts the ROSOV's and cuts power to the transfer pumps, both can be calibrated to stop the transfer of fluid over several seconds which prevents the transfer pipe work from experiencing the effects of Hydraulic transient.

This project will analyse the result of Hydraulic transient caused by the activation of a Break-away coupling and the upstream changes in pressure and flow rate. Break-away couplings are used where the transfer pipework can come under forces applied from the movement of the loading or unloading vessel, this can occur because of isolated weather events, incorrect mooring or as the result of the break-away coupling failing due to material fatigue. Break-away couplings are fixed at points where pipework is connected to High pressure flexible hose and at the points where pipework is connected to loading arms. The purpose of the break-away coupling is that when the coupling comes under a force that is greater than the tensile strength of the joining pins the coupling separates and the butterfly valve within each of the halves of the coupling close in less than one second to prevent the loss of product.

The activation of a break-away coupling during transfer results in a rapid change in volumetric flow rate within the pipeline and causes the phenomena of Hydraulic transient. Because of a rapid change in volumetric flow rate the piezometric head pressure rapidly increases and a pressure wave travels at the speed of sound within the pipeline upstream. The propagation of this pressure wave upstream will be analysed using the finite difference method to calculate the piezometric head pressure and the volumetric flow rate within the pipe. The magnitude of the pressure wave as it moves upstream will be analysed, as with many transfer pipelines break-away couplings can be placed upstream to prevent joint failures resulting from pipe work movement. The pressure increase within the pipe will

be compared to the force required to separate the upstream breakaway coupling to establish if hydraulic transient is great enough to cause break away coupling activation upstream or even partial separation of the breakaway coupling which would result in loss of product and a potentially flammable atmosphere.

Research

Hydraulic Transient

Hydraulic transient is present when the velocity of a fluid is either reduced or increased, however the effects of Hydraulic transient are not always noticeable if the change in condition is gradually made. If the rate of change in fluid velocity is rapid this is when the effects of Hydraulic transient are noticeable. Hydraulic transient is commonly called Water hammer, this name is used to describe the noise that is emitted from domestic water pipes when a tap or valve that has a closure angle of ninety degrees is shut quickly reducing the velocity of the water resulting in a pressure wave propagating upstream of the valve. The pressure wave as it moves upstream causes the pipes to move causing the noise that is described as water hammer. In a domestic situation, the effects of Hydraulic transient are minor and rarely cause physical damage to pipe, in industrial situations the effects can be catastrophic as the volumetric flow rate and pressures are far greater with the effects of Hydraulic transient damaging pumping equipment, rupturing pipes and the release of Hazardous substances.

With a rapid change in volumetric flow rate because of valve closure the fluid that is traveling within the pipe must come to a stop. With this fluid stopping immediately the fluid compresses at the point where the volumetric flow rate is zero meters per second and travels upstream with velocity which is the same as the sonic speed of sound in the pipe. The compression that is caused by the rapid reduction in volumetric flow rate is related to the fluids Bulk modulus. Most fluids are thought of as incompressible which is incorrect, all fluids are compressible to varying degrees due to the molecules that make up the fluid moving closer together as the column of fluid behind decelerates.

The term bulk modulus usually means the reciprocal of compressibility and defines the slope of the curve when plotted against specific volume. (George, 2007)

The pipe system that will be analysed in this project will be a fully wetted LNG transfer line, with the pipe being fully wetted the pipe is assumed to be free of air and to be filled with LNG. The presence of air in transfer lines affects efficiency and can restrict flow.

Hydraulic transient has been formulated by many Physicists and Engineers in the past with slight variations, the most frequently used equation to determine the peak pressure resulting from hydraulic transient is the Joukowsky (Equation 1) shown below;

$$\Delta P = \rho c \Delta v$$

EQUATION 1 JOUKOWSKY

Where;

$$\rho = \text{Fluid Density}$$

$$\Delta v = \text{Change in Fluid Velocity}$$

$$c = \text{Sonic Wave Speed}$$

The Joukowsky equation uses the Sonic wave speed within the pipe to determine the peak pressure to calculate the sonic wave speed the pipe material properties, pipe dimensions and the fluid properties. To calculate the sonic wave speed within the pipe (Equation 2) is used which is a modified version of Hooks law.

$$c = \sqrt{\frac{1}{\rho \left(\frac{1}{K} + \frac{D}{E \cdot e} \right)}}$$

EQUATION 2 SONIC WAVE SPEED

Where;

$$K = \text{Bulk Modulus of Fluid (N/m}^2\text{)}$$

$$D = \text{Internal Diameter of Pipe (m)}$$

$$E = \text{Youngs Modulus of Pipe Material (N/m}^2\text{)}$$

$$e = \text{Wall Thickness of Pipe (m)}$$

When calculated the result of (Equation 2) is shown in meters per second.

This project requires the analysis of the change in conditions at each point within the pipe at each point in time until the pressure wave 'c' has travelled the length of the pipe. To do this the Continuity (Equation 3) and Momentum (Equation 4) must be used to calculate these changes in conditions.

$$\frac{\partial H}{\partial t} = - \frac{c^2}{g \cdot A} \cdot \frac{\partial Q}{\partial x}$$

EQUATION 3 CONTINUITY EQUATION

$$\frac{\partial H}{\partial x} = - \frac{1}{g \cdot A} \cdot \frac{\partial Q}{\partial t} - f(Q)$$

EQUATION 4 MOMENTUM EQUATION

Where;

$$H = \frac{P}{\rho \cdot g} + z = \text{Pressure} + \text{Potential Head (m)}$$

$$Q = \text{Volumetric Flowrate (m}^3/\text{s)}$$

$$c = \text{Sonic Wave Speed (m}^2\text{)}$$

$$A = \text{Cross Sectional Area of Pipe Cavity (m}^2\text{)}$$

$$g = \text{Gravitational Constant}$$

$$f(Q) = \frac{f_r Q |Q|}{2 D g A^2}$$

LNG Transfer Line



FIGURE 1: LNG VESSEL AT KARRATHA

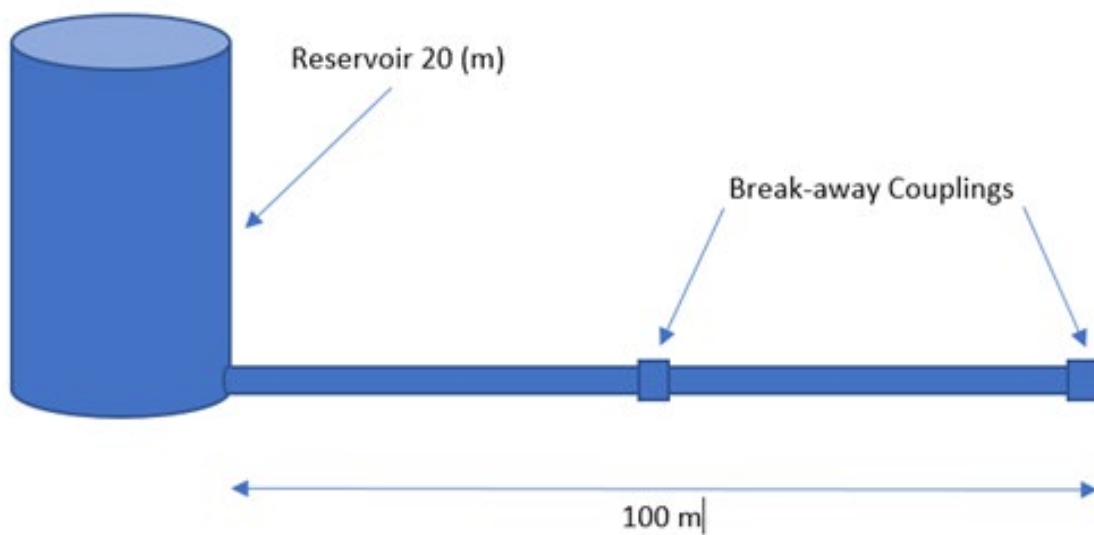


FIGURE 2 : SIMPLIFIED IMAGE OF A LNG TRANSFER LINE WITH BREAKAWAY COUPLINGS

The LNG transfer line that will be analysed will be a pipe with a uniform cross sectional area that is horizontal to the ground with zero gradient. At 50 meters and 100 meters from the reservoir will be Breakaway couplings with the Breakaway coupling that is furthest down-stream acting as the closing valve. This pipe layout is typical of the feed pipe system that is used to connect a reservoir of LNG to a loading arm that is used to connect a flexible cryogenic hose that loads or unloads a LNG Vessel. LNG when transferred or stored is cooled to a temperature of -160°C or 113.15°K . Cooling of LNG is required to liquefy the gas and to also reduce the volume, because of the cooling of the LNG to those temperatures the pipeline is required to be constructed out of a 36% Nickle Steel alloy. This alloy is chosen as it has a low Thermal expansion coefficient meaning the dimension of the pipe will not change by a large amount because of the low temperatures.

TABLE 1 : CHEMICAL ANALYSIS OF SSC INVAR 36

Chemical Analysis			
Nickel	36.00%	Chromium	0.25% max
Carbon	0.05% max	Manganese	0.60% max
Silicon	0.40% max	Cobalt	0.50% max
Phosphorus	0.015% max	Sulphur	0.015% max
Aluminium	0.10% max	Magnesium	0.10% max
Zirconium	0.10% max	Titanium	0.10% max

(Sandmeyer Steel Company, 2016)

Liquefying Natural gas is the preferred method of transporting Natural Gas as the volume of LNG is three times less than that of CNG (Compressed Natural Gas) and LNG when cooled below -160°C is liquid at atmospheric pressure which allows it to be transported in vessels that are not pressurised only insulated vessels. Chemical Analysis of SSC Invar 36

Liquefied Natural Gas (LNG) is simply natural gas that is cooled to minus 260°F , at which point it becomes a liquid that can be transported without high compression. Converting natural gas to LNG reduces its volume by about 600 times, so it can be moved by tanker trucks and in ISO containers on roads and highways, or on the water. Once delivered to its destination, the LNG is warmed and re-gasified so that it can be used just like existing natural gas. (LNG America, 2014)

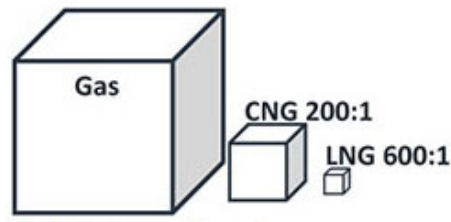


FIGURE 3 : LNG COMPARISON TO CNG AND NATURAL GAS

The specific pipe that will be analysed for this project will be the SSC INVAR 36 nickel alloy plate (UNS K93603) which is a commonly used cryogenic pipe.

The alloy has good fatigue and mechanical properties at cryogenic temperatures. It exhibits moderately high strength along with good ductility and toughness. These properties along with good fabrication characteristics make SSC INVAR 36 nickel alloy ideal for applications requiring precise dimensional stability. (Sandmeyer Steel Company, 2016)

TABLE 2: THERMAL EXPANSION OF SSC INVAR 36

Temperature Range		Total Expansion	Mean Linear Coefficient	
°C	°F	(10 ⁻³)	(10 ⁻⁶ /°C)	(10 ⁻⁶ /°F)
-200 - 20	-328 - 68	-0.33	1.5	0.8
-100 - 20	-148 - 68	-0.16	1.3	0.7

(Sandmeyer Steel Company, 2016)

Shown in Table 1 are the Thermal expansion coefficients of the SSC Invar 36 Alloy and for the temperature range -200 to -20 degrees Celsius the total expansion is -0.000033 metres showing that the material per metre will contract by 0.33 millimetres. When the Cryogenic piping is installed the piping is heavily insulated to reduce the amount of heat transfer from the atmosphere to the LNG this insulation has no effect on the mechanical properties of the pipe.

The volumetric flow rate for a 24" LNG line is typically 7500 gal/min = 0.5682613 m³/s, this will be taken as the standard transfer volumetric flowrate. The volumetric flowrate can change depending on the performance of the transfer pumps that are used.

TABLE 3 : MECHANICAL PROPERTIES OF SSC INVAR 36

Temperature		Yield Strength		Tensile Strength	
°C	°F	(MPa)	psi	(MPa)	psi
20	68	240	35.0	490	71.0

(Sandmeyer Steel Company, 2016)

TABLE 4 : TABLE OF 36 NiFe ALLOY

Material	Invar - Nickel Iron Alloy		
Property	Minimum Value (S.I.)	Maximum Value (S.I.)	Units (S.I.)
Atomic Volume (average)	0.0068	0.0071	m ³ /kmol
Density	8.1	8.2	Mg/m ³
Energy Content	50	200	MJ/kg
Bulk Modulus	106	112	GPa
Compressive Strength	240	725	MPa
Ductility	0.06	0.45	
Elastic Limit	240	725	MPa
Endurance Limit	185	405	MPa
Fracture Toughness	120	150	MPa.m ^{1/2}
Hardness	1200	2400	MPa
Loss Coefficient	0.0003	0.0011	
Modulus of Rupture	240	725	MPa
Poisson's Ratio	0.28	0.3	
Shear Modulus	54	58	GPa
Tensile Strength	445	810	MPa
Young's Modulus	137	145	GPa

(Woolger, 1996)

The Value for the Youngs Modulus of the pipe material will be taken as 141 Giga Newtons per meter square as this is the average between the maximum and minimum known values of the material.

Break-away Coupling

Break-away couplings are classed as process safety critical equipment when used in the transfer of Hazardous products. The process safety critical classification is needed as the loading or unloading vessel is never considered to be fixed and has the potential during product transfer to drift or break mooring. The movement of the vessel can subject the transfer pipe network to forces because of movement of the vessel. Without the use of break-away couplings these forces can result in pipe work fracturing and connection flanges rupturing leading to loss of product to atmosphere. To prevent these circumstances from arising the application of Break-away couplings is safety critical as the couplings are critical in potentially dangerous circumstances.

Applications and specialisms include Terminals, Ship-to-Ship, Ship-to-Shore, Ship-to-Rig, Road Tankers and Rail cars. (KLAW Products , 2017)

Previous models of breakaway couplings that were based on a poppet valve that when the coupling is disconnected the spring-loaded valve snapped shut worked well providing the disconnection was complete. However, in certain situations the force applied may only be great enough or applied for a limited time to cause a partial separation causing the coupling not to totally shut. This partial closure and the high volumetric flowrate of product transfer pipelines resulted in the loss of product to atmosphere.

Partial Break is when the coupling has been partially parted but not fully separated. This is due to the applied force being prematurely removed (e.g. due to an operator beginning to move the vehicle and then realising they have not disconnected, thus stopping). (KLAW Products , 2017)

Because of the requirement to remove the possibility of partial separation of break-away couplings the couplings have since been altered to stop this from occurring.

The KLAW unit ensures that under no circumstances can the seal between the two coupling body halves open to the atmosphere, before the internal flap valves have been released and provide a 100% shut off. (KLAW Products , 2017)

Because of the change in the design the Break-away couplings that are used in industry have a flap valve that is considered to shut immediately which is less than 0.5 seconds. The change in design of Breakaway coupling has eliminated the possibility of a partial break and product release but has resulted in greater negative effects of Hydraulic transient as the valve closure time has decreased leading to greater pressure rises in transfer lines.



FIGURE 4: KLAW CRYOGENIC BREAK-AWAY COUPLING

(KLAW Products , 2017)

The Break-away couplings used for LNG transfer must be cryogenic breakaway couplings and manufactured out of the same 36% Nickel Steel alloy as the material has a very low Thermal expansion coefficient.

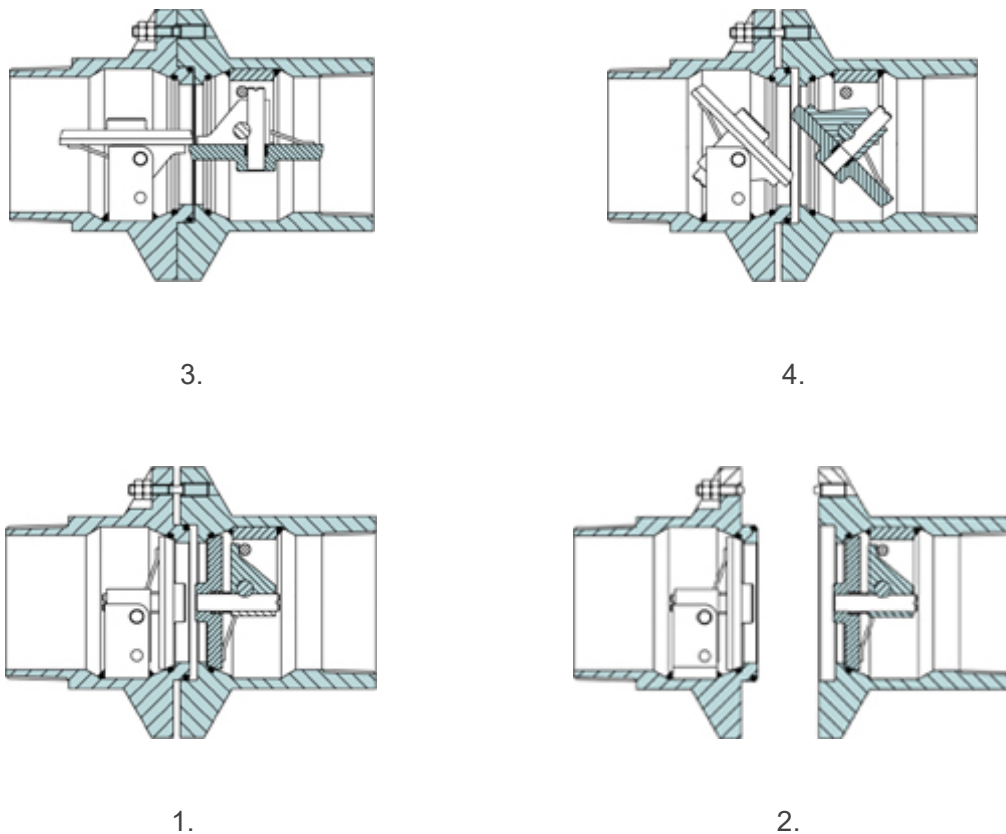


FIGURE 5: SEQUENTIAL CLOSURE OF FLAP VALVE WITHIN BREAK-AWAY COUPLING:

(KLAW Products , 2017)

1. Valves lock each other in open position
2. Coupling begins to separate, Flaps rotate
3. Flaps snap into the closed position
4. Coupling parts and hose system separates

This project will specifically analyse the upstream effects of a down-stream Break-away coupling activating and whether the pressure increase caused by the change in conditions is great enough to activate a Breakaway coupling located upstream at twenty metres distance from the reservoir.

To activate the upstream coupling the pressure within the pipe must exceed the ultimate tensile strength of the break-away couplings shear bolts.

Breaking Loads

Coupling break-loads can be varied to suit any application.

(KLAW Products , 2017)

When selecting the break-load for the coupling the force must not be great enough to damage the pipe but must be low enough to ensure the couplings activation. The break pins that are used for the transfer of cryogenic liquids must also have low rates of thermal expansion as the possible changes in temperature may be so great that partial separating may be caused by the break pin reducing in length and coming under loading thus when the temperature is increased the bolt could expand and the complete seal between couplings is not made.



FIGURE 6: BREAK PIN SIDE VIEW.

(intricoproducts, 2017)

Break pins diameters for couplings can range from 0.25mm to 9mm with the largest bore requiring the larger sized pin and each pair of couplings requiring four break pins for connection.

TABLE 5 : CALCULATED COUPLING BREAK FORCE

Pin Diameter (m)	Cross sectional area (m ²)	Tensile strength (MN/m ²)	Ultimate tensile strength of pin (N)	Break force for four pins (N)
0.005	0.000019634	490	9,621	38,484
0.0075	0.000044178	490	21,647	86,588
0.009	0.000063617	490	31,172	124,688

Liquefied Natural Gas

Natural gas that is extracted has a typical Methane content of 80% with the remaining 20% containing various other Hydro carbons such as Ethane, Propane and Butane. With dependence on Natural gas increasing year on year the need to extract gas from increasingly remote environments the practicality of using pipe networks to distribute the fuel is not great enough. This has led to the development of cooling the natural gas until the Natural gas changes phase, from gas to liquid. Cooling Natural gas to initiate liquefaction requires the temperature of the Natural gas to be reduced to -160°C, once the fuel has reached -160°C it is in the liquid state and as it has been liquefied by cooling the fuel can be stored at atmospheric pressure. The volume of the Gas when liquefied is reduced to 1:600th of the volume at 15°C, the reduction in volume allows for the fuel to be transported by ship or in some cases by road tanker.



FIGURE 7: LNG TANKER WITH FUEL SPHERES

The composition of Natural gas is dependent on where the natural gas has been extracted from this variation in composition will be considered when creating the code that will analyse the condition changes within the LNG pipeline. This will be done by the code that is written requiring the user to input the properties of the LNG as these change depending on the composition.

Liquefied Natural Gas has the following variations in composition;

TABLE 6 : LNG CHEMICAL COMPOSITION

Chemical	Chemical Formula	Low Composition	High Composition
Methane	CH ₄	87%	99%
Ethane	C ₂ H ₆	<1%	10%
Propane	C ₃ H ₈	>1%	5%
Butane	C ₄ H ₁₀	>1%	>1%
Nitrogen	N ₂	0.1%	1%
Other Hydrocarbons	Various	Trace	Trace

(GIIGNL, 2016)

Although the physical properties for LNG can change depending on the fuels composition the table below shows typical properties of the fuel;

TABLE 7 : TYPICAL LNG PHYSICAL PROPERTIES

Property	Value
Symbol	LNG
Operating Temperature	-160°C
Viscosity	0.157cP
Density	462 kg/m ³
Phase at Room Temperature (20°C)	Gas
Bulk Modulus	8.65 x 10 ⁷ kgf/m ²

(Syahrudin, 2011)

When LNG is transferred from one vessel to another the pipeline will be fully wetted as the vessels as purged before loading and are then gas charged with natural gas before being loaded with liquified natural gas. This process ensures that the transfer vessel contains only natural gas to be present within the pipeline. As the only compound in the transfer pipeline is LNG the system can be hydraulically stiff, meaning the presence of air pockets or bubbles in the system are not present however some dissolved air could potentially be present but would not lower the bulk modulus of the fluid.

Dissolved air is that in solution in a fluid. Because it is neither free nor entrained air, it does not behave according to Boyle's law. It does, however obey Henry's law: the weight of gas dissolved in a liquid is proportional to the pressure of the gas. It can be removed by two methods: subjecting the fluid to a reduced pressure and/or raising the fluid temperature. Its presence or absence does not affect fluid volume. (Hydraulics & Pneumatics, 2017)

Without the presence of air in the system the compressibility of the fluid can be taken as the bulk modulus of LNG, if air was present in larger volumes the transfer line greater energy would be required to transfer the product as the system would become 'spongy' which would decrease efficiency.

Methodology

The Problem

The reasoning behind completing this report is to analyse the flow conditions within a LNG transfer pipeline during the activation of a Break-away coupling downstream of the LNG reservoir. The activation of the Breakaway coupling results in the flow within the pipe decreasing to zero meters per second in a short period due to the shutting speed of the valve within the Break-away coupling reaching closure in less than one second. The rapid reduction in flow velocity results in a pressure rise as the column of fluid that is travelling towards the closed valve creates a pressure wave that proper gates upstream of the closed valve.

The results from the analysis can be used to predict loading on pipelines because of Hydraulic transient and to prevent against ruptured pipes, product loss, weakening of pipe material and damaged instrumentation.

Procedure

To model the proposed problem within MATLAB the Continuity equation (equation 3) and the Momentum equation (equation 4) must be modified into a form that allows the computer to solve. The pipe will be analysed as a one-dimensional problem along the central axis of the horizontal pipe, the dimension of the pipe will be discretised to a given number of nodes chosen by the user when the program asks for the user to choose a dimension for grid spacing. At the last node of the model will be the boundary condition for the valve where the fluid velocity becomes zero metres per second and at the first node of the model will be the reservoir node where the pipe joins the reservoir tank. Completing these steps will allow the computer to calculate the changes in volumetric flowrate and piezometric pressure at each node within the pipe at each time step for the duration taken for the pressure wave to propagate from the valve node upstream to the reservoir node.

Finite Difference Method

The Finite difference method is the chosen method of this problem as it can be used to solve partial differential equations without the need for analytical solutions which a computer cannot perform but by solving the equations using a numerical method based on approximations that contain a small error. This method could theoretically be completed by hand but would be far too time consuming to be worthwhile, however computers can perform these calculations at exceptional speed and by utilising the processing power of a computer the number of calculations based on approximations the solution to the partial differential equations can be solved numerically in a relatively short amount of time. The FD method has been developed from Taylors theorem.

$$f(x + h) = f(x) + h \cdot f'(x) + \frac{h^2}{2!} f''(x) + \frac{h^3}{3!} f'''(x) + \dots$$

EQUATION 5: TAYLORS THEOREM (STROUD, 1993)

In the explicit finite-difference method, the partial derivatives are replaced by the finite-difference approximations such that the unknown conditions at a grid point at the end of time step are expressed in terms of the known conditions at the beginning of the time step. (Chaudry, 2014)

The method that has been selected to find a solution is an explicit finite difference method that can calculate the changes in conditions because of changing boundary conditions using single equations for volumetric flowrate and Piezometric pressure and will result in values for the discretised location and time. Compared to implicit Finite Difference methods that would result in a set of simultaneous equations for flow rate and pressure that would then require further processing to result in values for flowrate and pressure at each location at each time step.

The Finite difference method used to discretise time will be the Forward Difference Approximation as the initial value for the time step is a known value;

$$f'(t) = \frac{f(t + \Delta t) - f(t)}{\Delta t}$$

EQUATION 6: FORWARD DIFFERENCE APPROXIMATION (CHAUDRY, 2014)

Where;

t = current time

Δt = time step

$$\frac{\partial Q}{\partial t} \quad \text{and} \quad \frac{\partial H}{\partial t}$$

The partial derivatives above will be discretised using the Forward difference approximation as they are both functions of time.

The Finite difference method used to discretise space will be the Central Difference Approximation shown below;

$$f'(xi) = \frac{f(xi + \Delta x) - f(xi - \Delta x)}{2\Delta x}$$

EQUATION 7: CENTRAL DIFFERENCE APPROXIMATION (CHAUDRY, 2014)

Where;

x_i = point of interest

Δx = grid spacing

$$\frac{\partial Q}{\partial x} \quad \text{and} \quad \frac{\partial H}{\partial x}$$

The partial derivatives above will be calculated using the Central Difference Approximation as they are a function of the discretised space.

Assume the conditions at time t_0 (i.e., j level) are known and we want to compute their values at time $t_0 + \Delta t$ (i.e., $j+1$ level). Thus, the values of H and Q are known at the j -time level (either known initial conditions or computed during the previous time step) and their values at $j+1$ time level are to be determined. (Chaudry, 2014)

$$\frac{\partial H}{\partial t} = \frac{H_i^{j+1} - \bar{H}_i}{\Delta t}$$

$$\frac{\partial Q}{\partial t} = \frac{Q_i^{j+1} - \bar{Q}_i}{\Delta t}$$

$$\frac{\partial Q}{\partial x} = \frac{Q_{i+1}^j - Q_{i-1}^j}{2\Delta x}$$

$$\frac{\partial H}{\partial x} = \frac{H_{i+1}^j - H_{i-1}^j}{2\Delta x}$$

Where;

$$\bar{H}_i = \text{abs}(0.5 \times H_{i-1}^j + H_{i+1}^j)$$

$$\bar{Q}_i = \text{abs}(0.5 \times Q_{i+1}^j - Q_{i-1}^j)$$

Using both the Forward difference approximation and the Central difference approximation the Governing equations can be altered to produced equations that fully rely on Explicit finite difference approximations;

$$Q_i^{j+1} = \frac{1}{2} (Q_{i-1}^j + Q_{i+1}^j) - \frac{1}{2} g \cdot A \frac{\Delta t}{\Delta x} (H_{i+1}^j - H_{i-1}^j) - R \cdot \Delta t \overline{Q_i} | \overline{Q_i} |$$

EQUATION 8: MOMENTUM EQUATION WITH CDA (CHAUDRY, 2014)

Where;

$$R = f_r / (2DA)$$

EQUATION 9: PIPE RESISTANCE

Above shows the momentum equation with the CDA method substituted into the original momentum formula

$$H_i^{j+1} = \frac{1}{2} (H_{i-1}^j + H_{i+1}^j) - \frac{1}{2} \frac{\Delta t}{\Delta x} \frac{c^2}{g A} (Q_{i+1}^j - Q_{i-1}^j)$$

EQUATION 10: CONTINUITY EQUATION WITH CDA (CHAUDRY, 2014)

When using the explicit finite difference method, a stability criterion must be obeyed, this will be ensured by restrictions put on the maximum and minimum time steps and the same restrictions will be applied to the grid spacing. Because of using the explicit finite difference method each calculation produces an error, if the stability criterion is not met the error that is produced is amplified as the iterations continue, however if the stability criterion is met then the error that is produced either remains the same and is neither amplified or reduced or the error is reduced as the iterations continue. Both situations where the error is not increased result in a stable system that will produce data that is accurate to an acceptable point and with an unstable system the data produced will be corrupted by the amplified error.

$$\frac{c \cdot \Delta t}{\Delta x} < 1$$

EQUATION 11: STABILITY CRITERION (CHAUDRY, 2014)

Initial Conditions

The initial conditions must be determined before the finite difference method can be used to produce data. For this problem, the initial conditions will be that of a LNG transfer pipe line during product transfer from reservoir to ship and that the transfer line is fully wetted without any pockets of trapped gas. These initial conditions are at time level zero which is the previous time step of any changes in boundary conditions.

The value for volumetric flowrate will be constant along the length of the pipe at time zero however the initial piezometric head pressure will not be constant along the pipe as there will be head loss due to friction. To calculate the head loss due to friction the Colebrook-White equation will be used providing the Reynolds number of the fluid which will be calculated is greater than 4500. For the boundary conditions for the nodes at the valve and at the reservoir the following equations will be used;

$$H_i = H_{res}$$

EQUATION 12: RESERVOIR PIEZOMETRIC HEAD PRESSURE

$$\text{Reservoir Flow rate } Q_{i1} = Q_{i2} + \frac{g \cdot A}{c} (H_{res} - H_{i2}) - R \cdot \Delta t \cdot Q_{i2} |Q_{i2}|$$

EQUATION 13: RESERVOIR FLOWRATE

$$\text{Valve Head Pressure } H_{i n+1} = H_{i n} - \frac{c}{g \cdot A} (Q_{i n+1} - Q_{i n}) - \frac{c}{g \cdot A} \cdot R \cdot \Delta t \cdot Q_{i n} |Q_{i n}|$$

EQUATION 14: VALVE PIEZOMETRIC HEAD PRESSURE

$$\text{Valve Flow Rate } Q_{i n+1} = Q_0 \left(1 - \frac{t}{T_{close}}\right) \text{ for } 0 \leq t \leq T_{close}$$

EQUATION 15: VALVE FLOWRATE

$$Q_{i n+1} = 0 \text{ for } t > T_{close}$$

EQUATION 16: VALVE FLOWRATE AFTER BREAKAWAY COUPLING ACTIVATION

(Chaudry, 2014)

These boundary conditions will change with respect to time with the boundary for the valve node for volumetric flowrate becoming a constant of zero meter per second once the time taken for valve closure has been exceeded.

MATLAB Code

Liquid Properties

To input the Liquid properties into the script the user is asked to input the properties of the LNG that is being transferred within the pipe line. This is required as the properties of LNG can vary as natural gas extracted from different environments can have varying compositions.

```
%%%Liquid Properties
Fluid = ('LNG');
rho = input('Fluid density in kg/m3 = ');
K = input('Bulk modulus in N/m2 = ');
Vis = input('Dynamic Viscosity of fluid in kg/ms = ');
Liquidpress = input('Gauge pressure of fluid in N/m2 = ');
```

All values that are inputted into the script must be in the S.I units, failure to enter the data in S.I units will result in incorrect results when the code is run.

Pipe Properties

The pipe properties will also require user inputs in S.I units as these can change depending on the problem being modelled.

```
%%%Pipe Properties
D = input('Pipe diameter in meters = ');
Wt = input('Pipe wall thickness in meters = ');
E = input('Pipe materials Youngs modulus in N/m2 = ');
L = input('Pipe length in meters = ');
Rough = input('Pipe internal Absolute surface roughness in meters = ');
while Rough <=0
    Rough = input('Pipe internal Absolute surface roughness in meters = ');
end
fc = input ('Pipe internal friction coefficient = ');
```

Using the inputted data the calculation for the internal cross sectional area can be made;

```
Area = (pi*(D/2)^2);
```

For the user input for Roughness of the internal pipe wall the script has a while loop with a logic statement to prevent the user from entering a negative value that may corrupt the values given in calculations that will be made as the code is run.

```
%%Valve Properties
Cl = input('Valve closure time in seconds = ');
%%Reservoir properties
Z = input('Reservoir height in meters = ');
%%Gravitational Acceleration
g = input('Gravitational acceleration in m/s2 = ');
```

The user is also asked to input the closure time for the downstream breakaway coupling, the Reservoir height and the gravitational constant.

Further calculations can be made using the inputted values

```
%%Sonic Waves Speed Within Pipe c
C = sqrt(1/(rho*((1/K)+(D/(E*Wt)))));
%%Piezometric Head Within Pipe
H = ((Liquidpress/(rho*g))+Z);
```

When the code is to ask the user to input a value for the grid spacing, the code will prevent the inputted value from being less than or equal to zero as inputted value of zero or less would cause the code to crash. It is therefore necessary to incorporate a while loop with a logical statement to prevent this from occurring.

```
%%Grid Spacing
X = input('Input Grid spacing in x direction = ');
while (X<=0) || (X>L) || rem(L,X)~=0
    X = ('Input Grid spacing in x direction = ');
End
```

The above while loop ensures that the inputted value is greater than zero, less than the total length of the pipe and that the length over the grids pacing results in no remainder.

A similar while loop is used when asking the user to input the time step for the calculations.

```
Ts = input ('Input Time step in seconds = ');  
while ((Ts*C)/X)>1 || ((Ts*C)/X)<=0  
    Ts = input ('Input Time step in seconds = ');  
End
```

If the value inputted into the above while loop is greater than one or less than zero the code will ask for another time step to be entered as the logic statement has not been fulfilled.

The final user input required for the code to run is the volumetric flow rate, this is done with a simple user input.

```
%%%Flow rate  
Q = input ('Input Volumetric flow rate in m3/s = ');
```

Initial Calculations

The code when run uses the inputted data to then complete the following calculations to correctly set up the model.

```
%%%Fluid dynamic velocity  
Fv = (Q/Area);  
%%%Reynolds Number of Fluid  
Re = ((rho)*(Fv)*(D))/(Vis));
```

Above shows how the calculations for the fluids dynamic velocity and Reynolds number are completed.

```
%%%Nodes within pipe  
Np=L/X+1;
```

The number of nodes within the pipe is the result of the length divided by the grid spacing plus one, the plus one is required to allow for the valve node to be assigned to the model.

The next calculations are completed to allow the plotting of the data in graph form to be done automatically and to scale, this ensures that when the problem that is being modelled has changed the user does not require altering the script. Completing these calculations are not necessary to the completion of the script but allow for users that are unfamiliar with the script to run it and produce accurate data. The calculation for the Maximum surge pressure within the pipe that is being

modelled will change if the initial conditions are altered allowing for the proportion of the resulting graphs being correct.

```
%%%Graph X axis
Xaxis = linspace (0,L,Np);
%%%Hydraulic Impedance
Hyd = (sqrt(rho*K))/Area;
%%%Maximum Surge Pressure of Fluid
Hmax = ((Hyd*Q)*0.0001019977334)*3)+H;
```

Internal Pipe Friction Calculation

This script has the capability to calculate the friction coefficient of the pipe that is being modelled using the inputted data. The formula used for this is the Colebrook-White equation which relies on the surface roughness of the internal pipe wall. However, using this equation for calculating the friction coefficient requires the fluid that is travelling through the pipe to be turbulent with a Reynolds number greater than four thousand. To ensure this is met a while loop is included to ensure that the fluid is turbulent, if the fluid is not turbulent the code will display the message 'Fluid is not turbulent' and the code will stop as the following calculation will not be accurate.

```
%%%Colebrook-White equation for friction coefficient (fc)
while Re < 4000
    disp('Fluid is not turbulent')
    break
end
dummy_variable=10;
while dummy_variable>1
    A = 1/sqrt(fc);
    B = -2*log10((Rough/(3.71*D))+(2.51/(Re*sqrt(fc))));
    if abs(A-B)<0.01 || fc>1
        dummy_variable=0;
    end
    fc=fc+0.00001;
end
```

This calculation starts with a while loop where the dummy variable is greater than one allowing the calculation to be performed until the absolute value of A-B is less than 0.01 or fc is greater than 1, if either of these conditions are met the dummy variable is then equal to zero and the loop is exited.

Where f_c is then increased after each iteration by 0.00001 until the while loop statement is satisfied resulting in a calculated value for ' f_c '.

Once the value for ' f_c ' has been calculated the overall resistance for the pipeline can be established also the head loss due to friction as the resistance along the pipe length is known.

```
%%%Pipe-resistance
Res = (fc/(2*D*Area));
%%%Darcy Weisbach Equation for Head Loss Due to Friction if Re>4000
Headloss=((fc)*(L/D)*((Fv^2)/(2*g)));
%%%Pipe Flow Rate Vector
PVQ=repmat(Q,1,Np);
%%%Pipe Head Pressure Vector
PVH=repmat(H,1,Np);
for i = 1:Np
    PVH(1,i) = H-(i-1)*((fc*L)/(2*g*D*(L/X)*Area^2));
end
```

The Head pressure within the pipe is calculated using a for loop with the counter starting at one and finishing at the last node of the pipe length, using this for loop and calculation the resulting vector that is generated with a uniform value for Piezometric head pressure has the value assigned to each node by the amount of friction that is a result of each nodes position along the pipe.

The nodes that are situated at either end of both the vector that will be used for Pressure calculations and the vector used for flow calculations each have specific calculations,

```
%%% Boundary Condition at Reservoir Node
Hvec(1,1) = PVH(1,1);
Qvec(1,1) = PVQ(1,2)+((g*Area)/C)*(PVH(1,1)-PVH(1,2))-
Res*Ts*PVQ(1,2)*abs(PVQ(1,2));
%%% Boundary Condition at Valve Node
Hvec(1,Np) = PVH(1,Np-1)-(C/(g*Area))*((PVQ(1,Np))-PVQ(1,Np))-
(C/(g*Area))*Res*Ts*PVQ(1,Np-1)*abs(PVQ(1,Np-1));
Qvec(1,Np) = Q;
```

Where 'Hvec' is the vector for Piezometric head pressure and 'Qvec' is the vector for volumetric flowrate. The first node of both is the point where the pipeline is connected to the reservoir and the last nodes of both vectors are the point where the downstream break-away coupling is located.

As the 'Hvec' has already been populated with data from the Head loss calculation and the initial boundary conditions are inputted no further calculations are required. For 'Qvec' only the boundary conditions have been calculated, 'Qvec' is populated with the volumetric flow rate across its length apart from the first and last nodes.

```
%%%Populate Flow Vector With Q
Qvec(1,2:Np-1) = Q;
```

Knowing the velocity of the sonic wave and the length of the pipe the time taken for the wave from to reach the reservoir node can be calculated, the result of the calculation can be used to determine how many iterations the code will complete. Two vectors are also generated to allow the calculated data for each node to be stored in until the next iteration is completed, this continues until the last iteration and the values in both vectors are the final values.

```
%%%Number of Iterations
t=0;
tmax=(L/C);
HP = PVH;
Flow = Qvec;
```

Up to this point the calculations that have been completed have allowed the initial conditions and the boundary conditions of the model to be established. The final calculations for the transient flow effects on pressure and volumetric flow are shown below.

```
while t < tmax
    for c = 2:Np-1
        Qvec(1,c) = 0.5*(Flow(1,c-1)+(Flow(1,c+1)) -
0.5*(g*Area)*(Ts/X)*(PVH(1,c+1)-PVH(1,c-1)) -
Res*Ts*(0.5*(Flow(1,c+1)+Flow(1,c-1)))*abs(0.5*(Flow(1,c+1)+Flow(1,c-1))));
        HP(1,c) = 0.5*(PVH(1,c-1)+PVH(1,c+1)) -
0.5*(Ts/X)*(C^2/(g*Area))*((Flow(1,c+1))-(Flow(1,c-1)));
    end
```

Computational Modelling of Hydraulic Transient

```

HP(1,1) = H;
Qvec(1,1) = Qvec(1,2) + ((g*Area)/C) * (HP(1,1) - HP(1,2)) -
Res*Ts*Qvec(1,2)*abs(Qvec(1,2));

HP(1,Np) = HP(1,Np-1) - (C/(g*Area)) * ((Qvec(1,Np)) - Qvec(1,Np)) -
(C/(g*Area)) * Res*Ts*Qvec(1,Np-1)*abs(Qvec(1,Np-1));
if (t+Ts)<Cl
    Qvec(1,Np) = Q*(1-((t+Ts)/Cl));
else
    Qvec(1,Np) = 0;
end
t=(t+Ts);
Flow=Qvec;
PVH=HP;

figure(1)
subplot(2,1,1);
plot(Xaxis,HP,'b')
axis([0 L 0 Hmax])
title('Piezometric Head Pressure')
subplot(2,1,2);
plot(Xaxis,Qvec,'b')
axis([0 L -(Q/10) Q+0.025])
title('Volumetric Flowrate')
end

```

The while loop to runs until 't' which starts at zero and increases by one after each iteration will exit when 'tmax' has been reached where 'tmax' is the value for the time taken for the front of the sonic wave to reach the reservoir node. Within the while loop is the for loop that is used to calculate the changes in Piezometric pressure and Volumetric flowrate at each node excluding the nodes representing the boundary conditions for the respective vectors.

Once the calculations for the internal nodes of the two vectors have been completed the change that occurs at the boundary nodes can be calculated starting with the reservoir nodes for each vector then moving on to the node representing the Break away coupling. For the node representing the Breakaway coupling an if loop is included in the calculations as once the Valve closure time has been met and or exceeded the value for Volumetric flow rate at the end node of the Flow vector becomes zero.

If this loop was not included the value that would be calculated would begin to enter a negative value once the valve closure time has been exceeded. Once the values for the two vectors have been calculated for that iteration the data is then stored in two storage vectors that then allow the original vectors to be used in the next iteration.

The chosen method for representing the changes in condition within the pipe during the hydraulic transient was to sub plot two graphs that both have the pipe length along the x axis and for the y axis for the flow graph the maximum value is taken as the initial volumetric flow rate plus 0.025 m^3 and the minimum value of a tenth of the initial volumetric flow rate to fully represent the data. For the Pressure graph the maximum y axis value is the maximum surge pressure that was calculated and the minimum value is zero.

Completing these calculations beforehand in the code allows for the data to be represented as soon as the code is run improving the usability of the script. Once the number of iterations has met the value for 'tmax' the code stops by exiting the outer while loop.

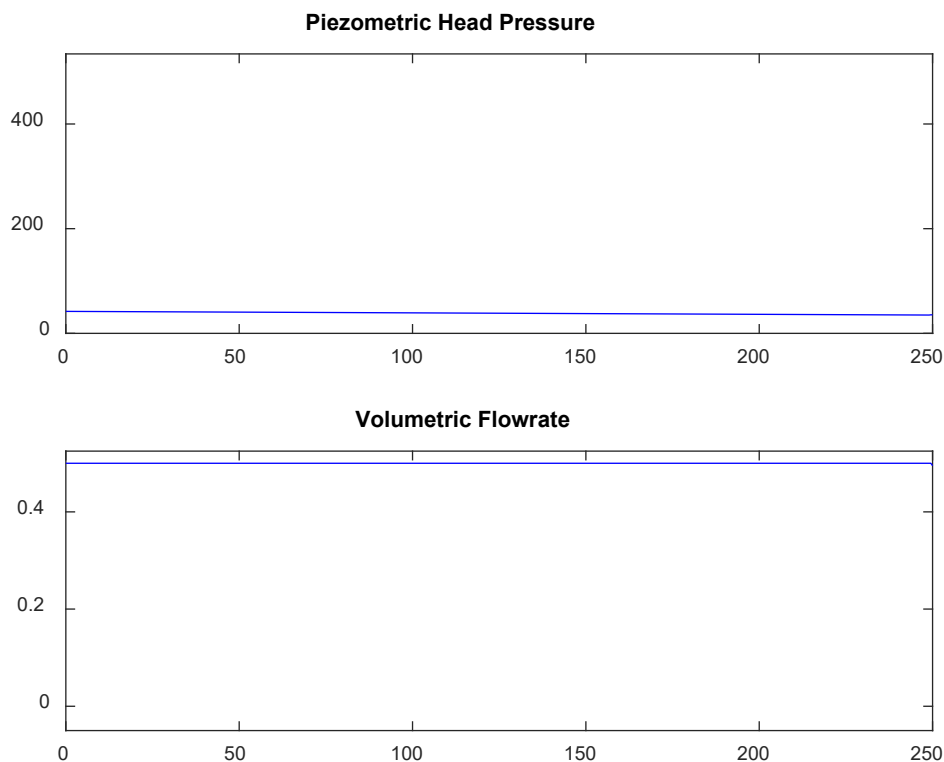


FIGURE 8: INITIAL CONDITION GRAPHS

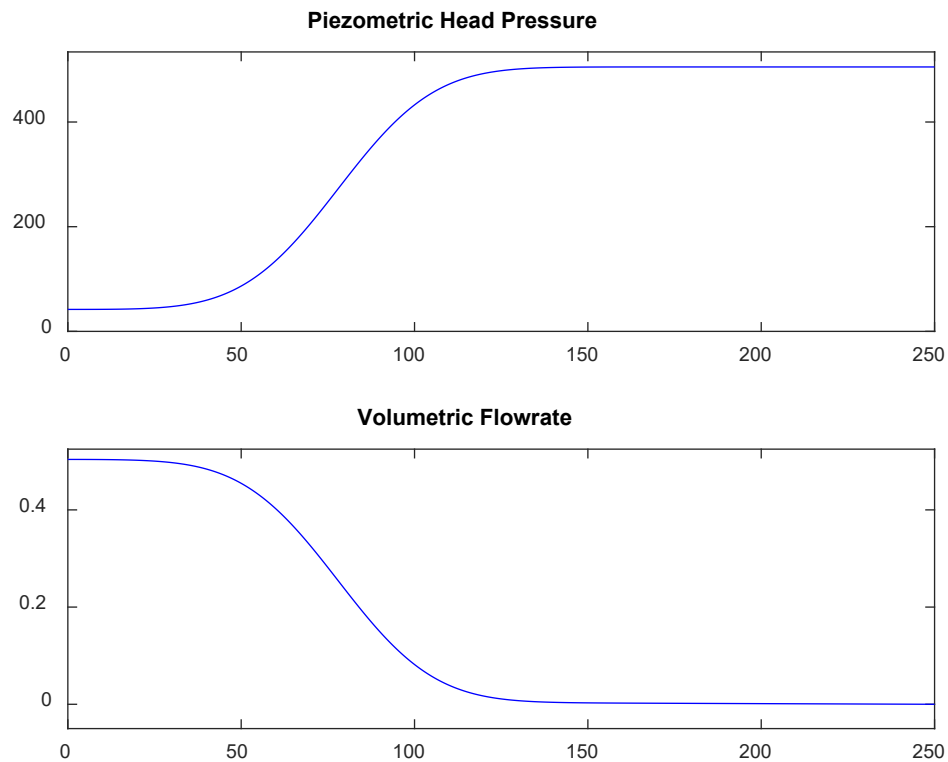


FIGURE 9: FINAL CONDITION GRAPHS

Results

The Code will be run to establish how changes in pipe properties and the fluids dynamic properties will affect the magnitude and speed to reach maximum surge pressure. This will allow the pipeline to be analysed and the effects that the pressure surge will have on the upstream break-away coupling. All models will be run with the reservoir assumed to be 80% full with a liquid height from outlet to be 20 meters.

Test Conditions 1

TABLE 8 : TEST CONDITIONS 1

Test Conditions 1	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.6096
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step (s)	0.0001
Fluid Height (m)	20

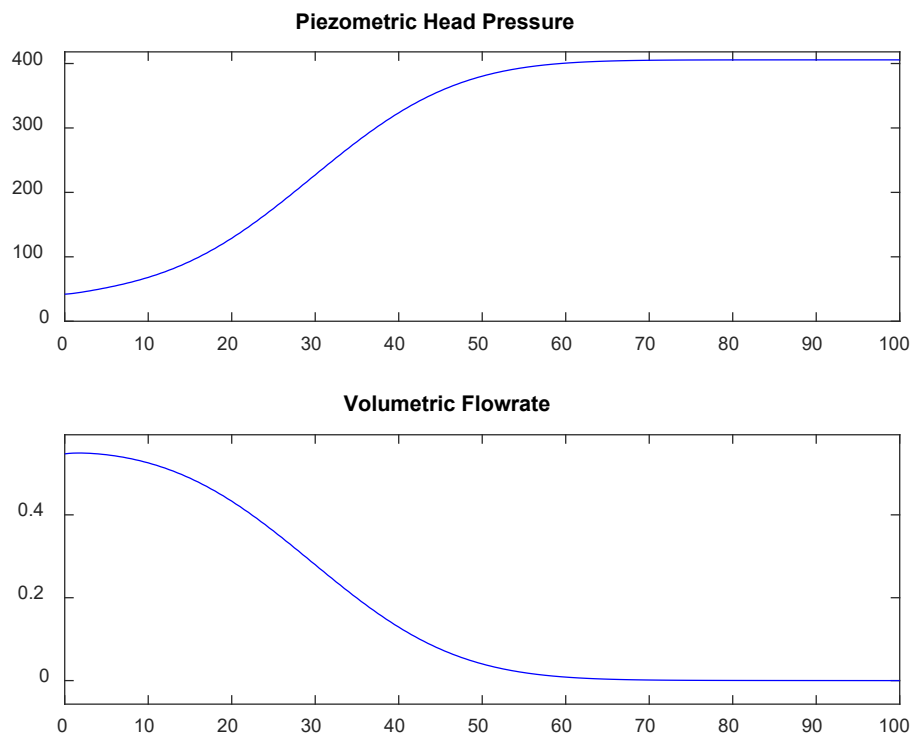


FIGURE 10 TEST CONDITION 1 FINAL PLOT

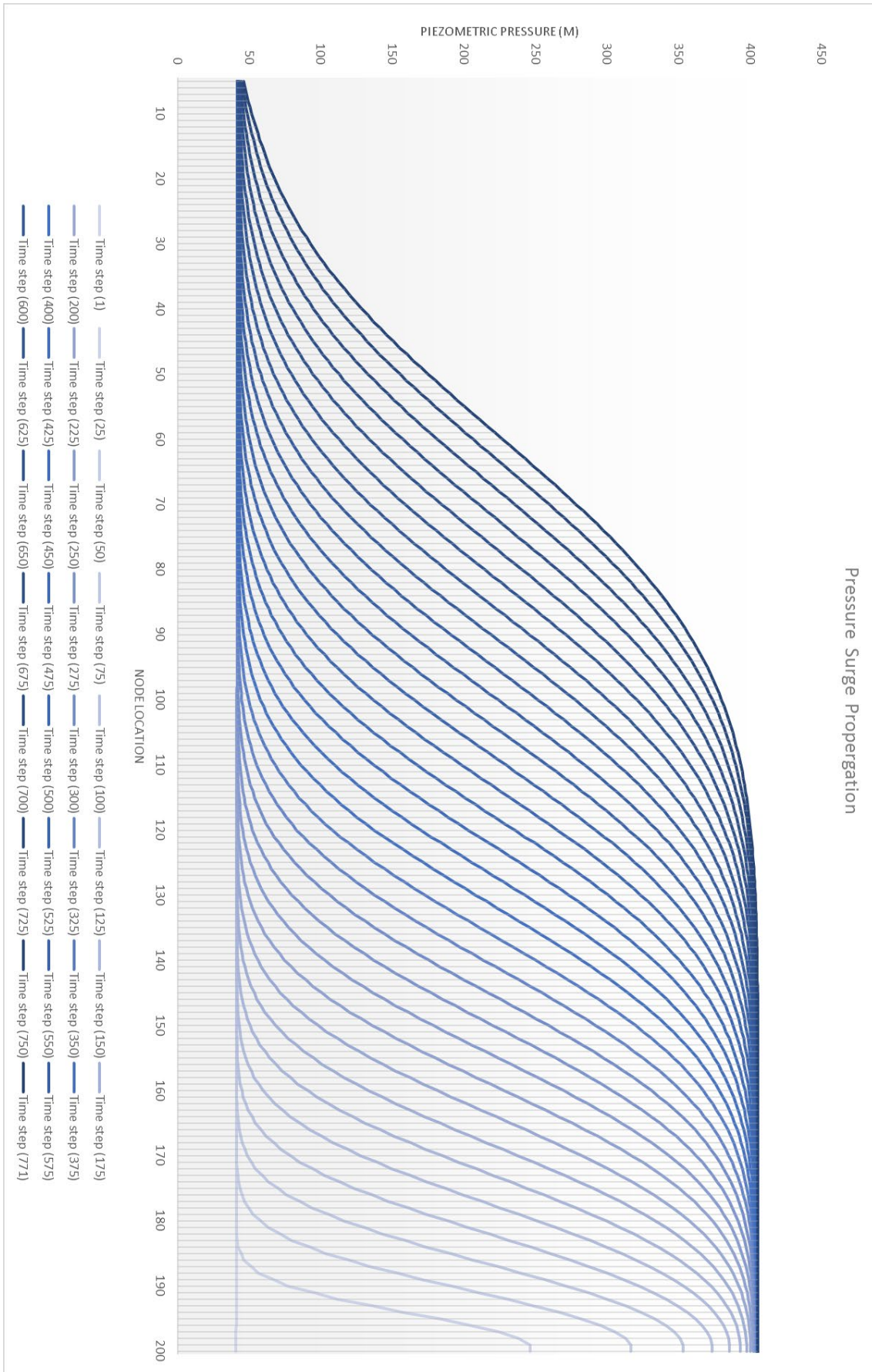


FIGURE 11 : TEST CONDITIONS 1 PRESSURE SURGE PROPAGATION

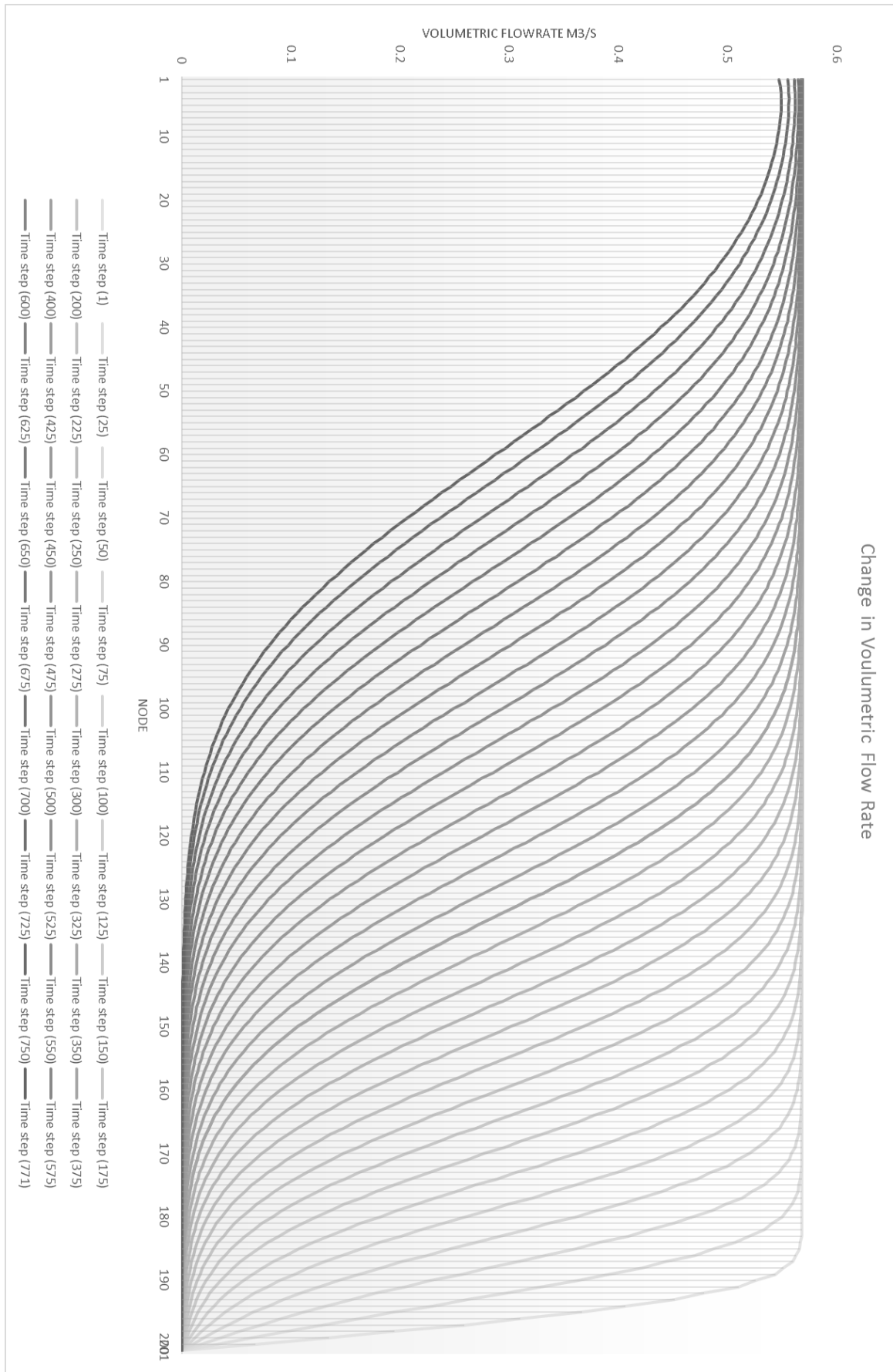


FIGURE 12 : TEST CONDITIONS 1 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 2

The model will be run with the same conditions as test one but the diameter will be the same as an 18" bore pipe which with the 24" diameter pipeline is the standard size used in industry.

TABLE 9 : TEST CONDITIONS 2

Test Conditions 2	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.4572
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

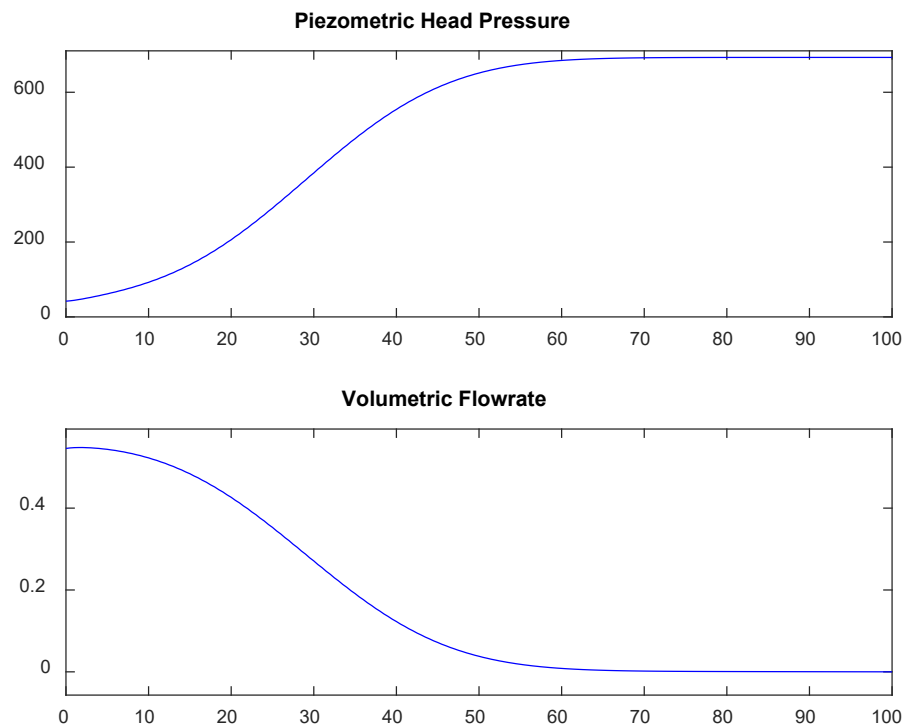


FIGURE 13 : TEST CONDITIONS 2 FINAL PLOT

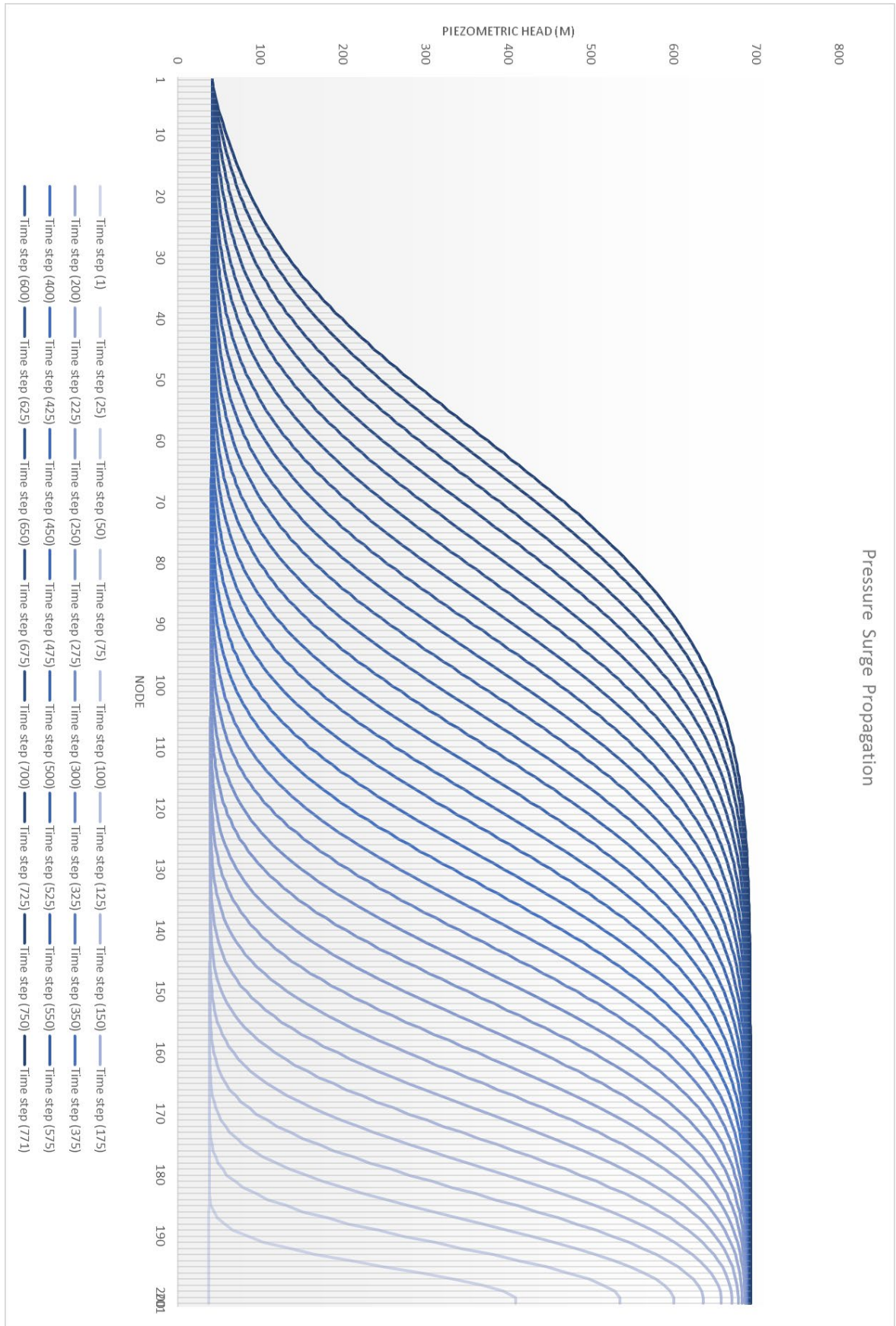


FIGURE 14: TEST CONDITION 2 PRESSURE SURGE PROPAGATION

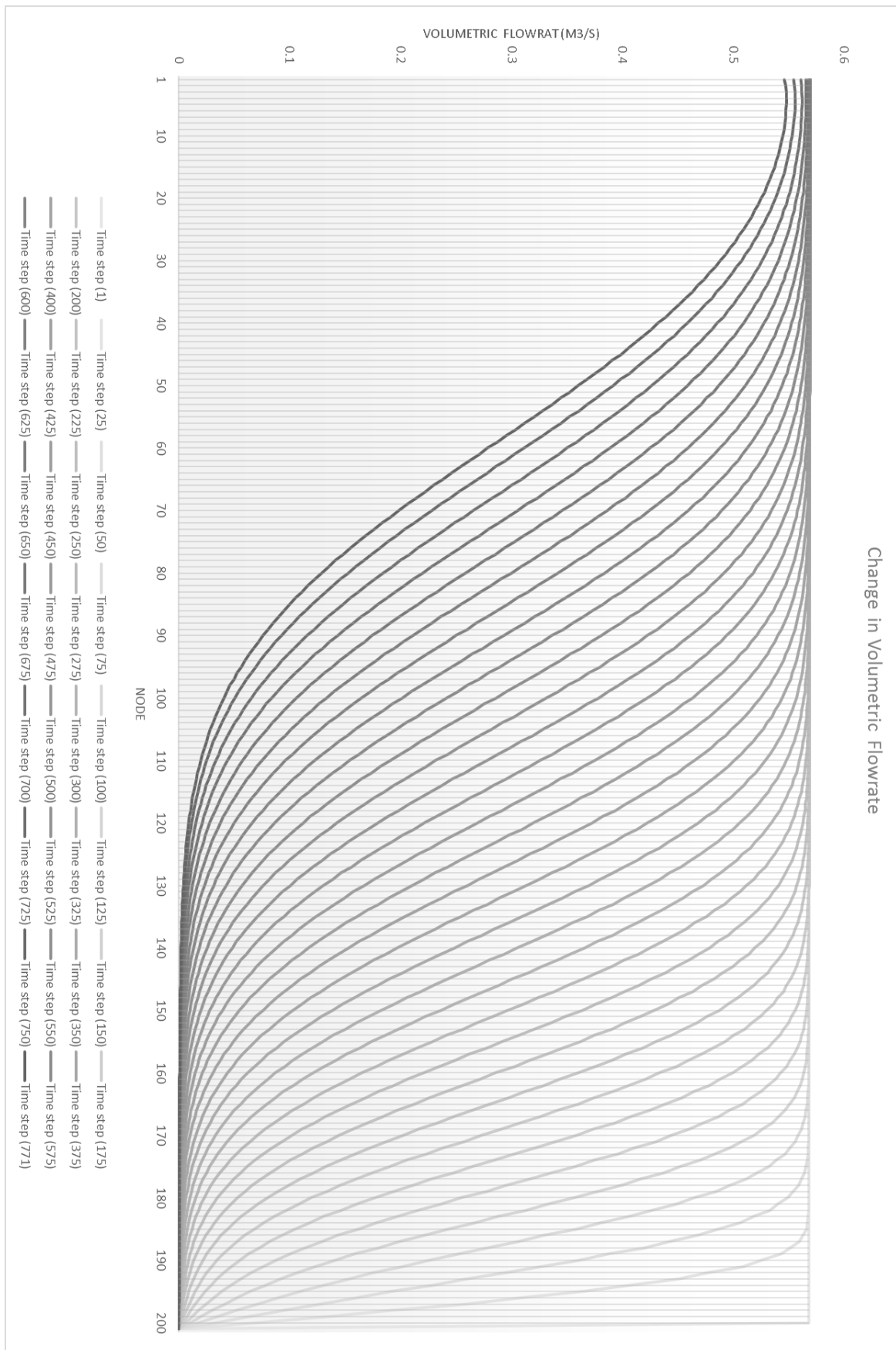


FIGURE 15: TEST CONDITIONS 2 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 3

Condition three will have the same properties as one and two however the internal diameter will be twelve inches which is the smallest diameter pipe used for ship to shore LNG transfer, this diameter pipe is the minimum diameter used as the power required to overcome the friction along the length would need to be increased and is not possible at every LNG plant.

TABLE 10 : TEST CONDITIONS 3

Test Conditions 3	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.3048
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ³)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step (s)	0.0001
Fluid Height (m)	20

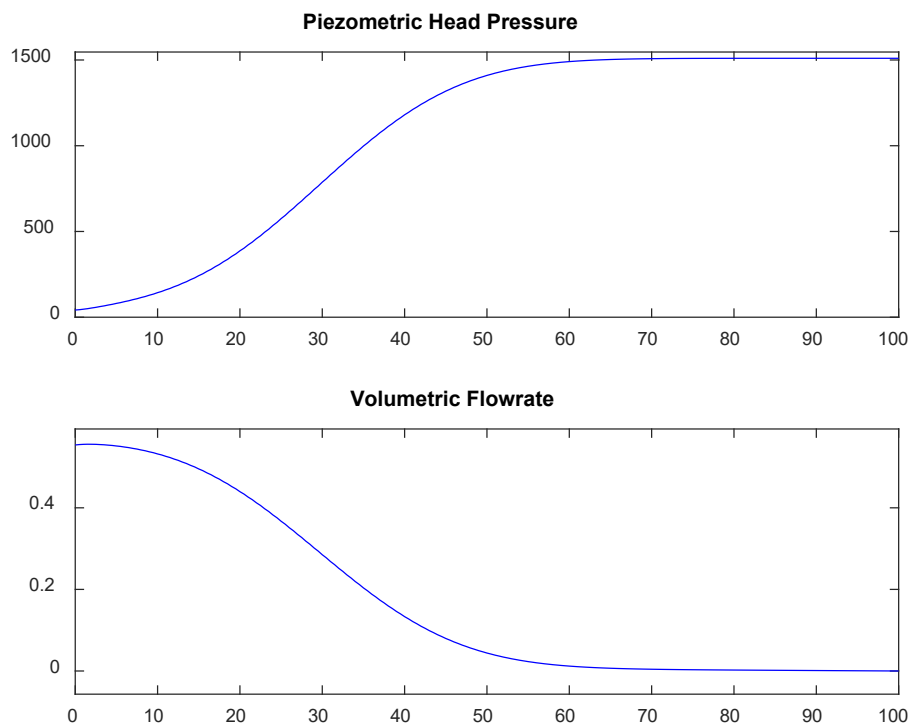


FIGURE 16: TEST CONDITIONS 3 FINAL PLOT

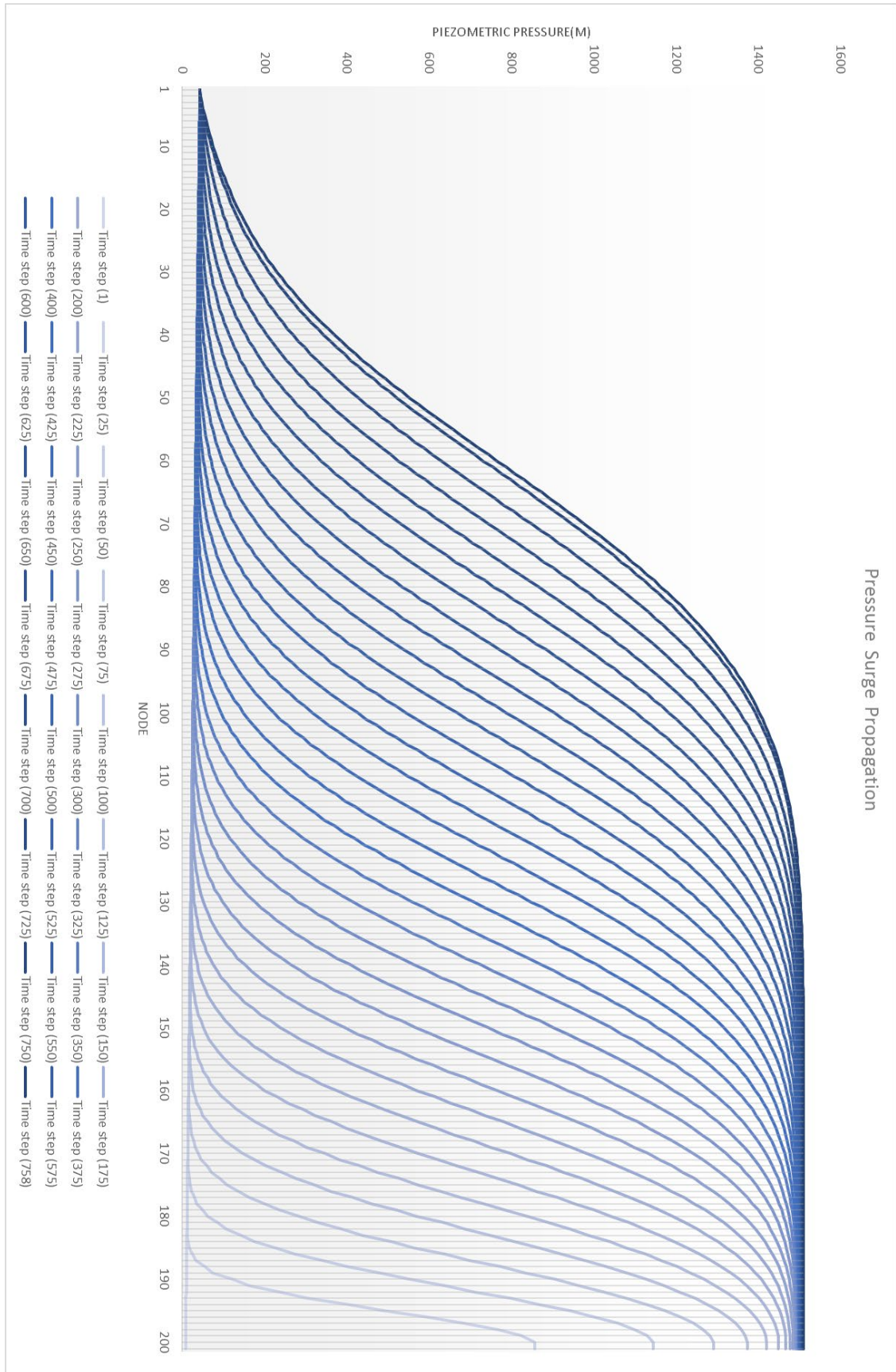


FIGURE 17: TEST CONDITIONS 3 PRESSURE SURGE PROPAGATION

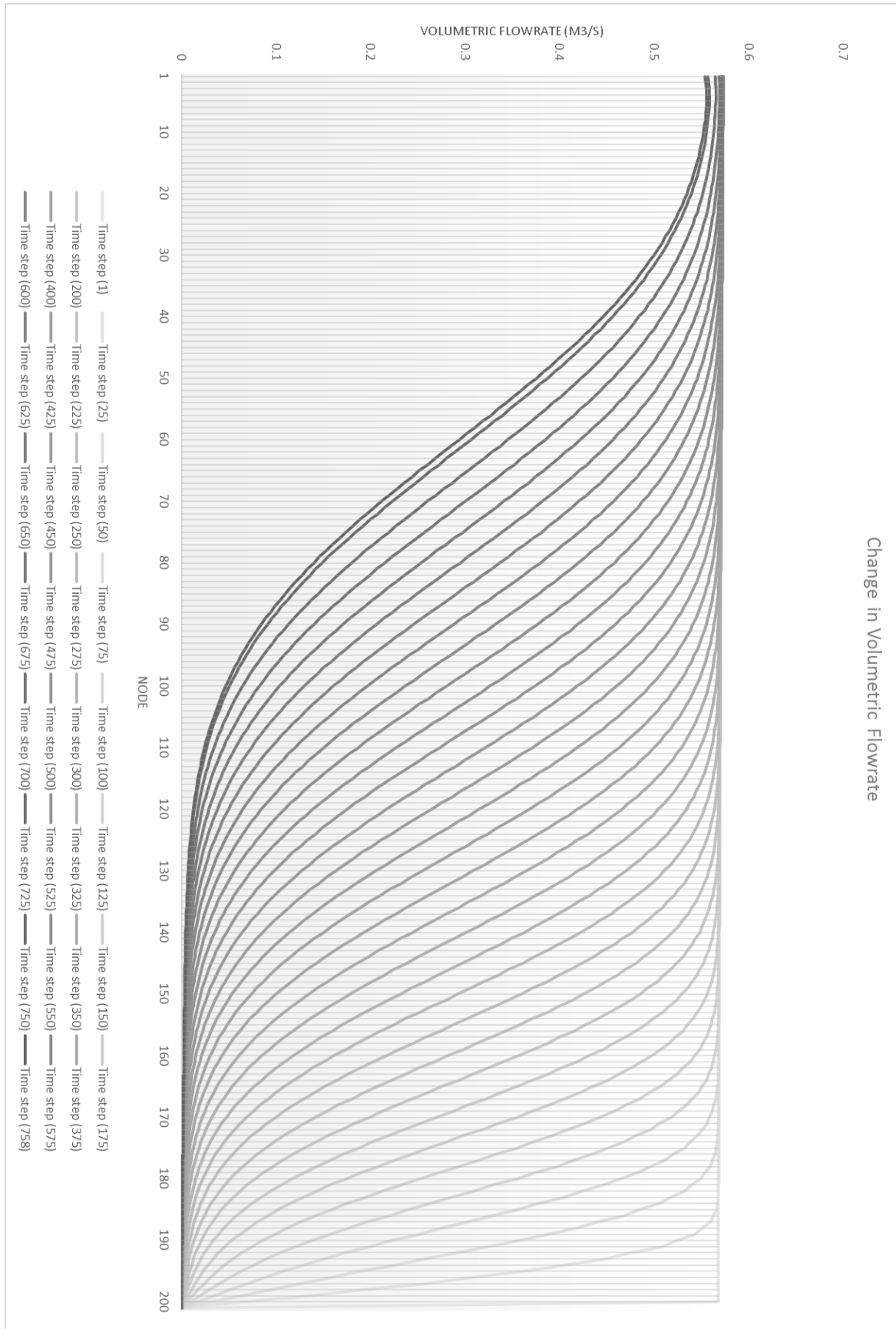


FIGURE 18 : TEST CONDITION 3 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 4

The condition for this model will be the same as those used in condition one, however the valve closure time will be increased to allow the effect of increasing the valve closure time on the pressure surge and the change in volumetric flowrate.

TABLE 11 : TEST CONDITIONS 4

Test Conditions 4	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.6096
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.1
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

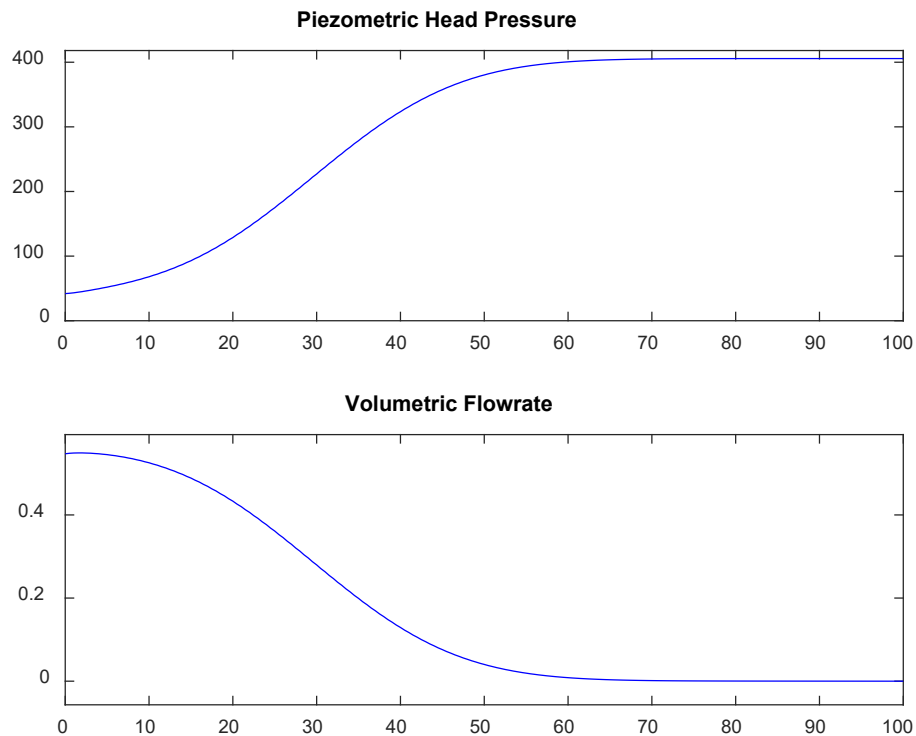


FIGURE 19: TEST CONDITIONS 4 FINAL PLOT

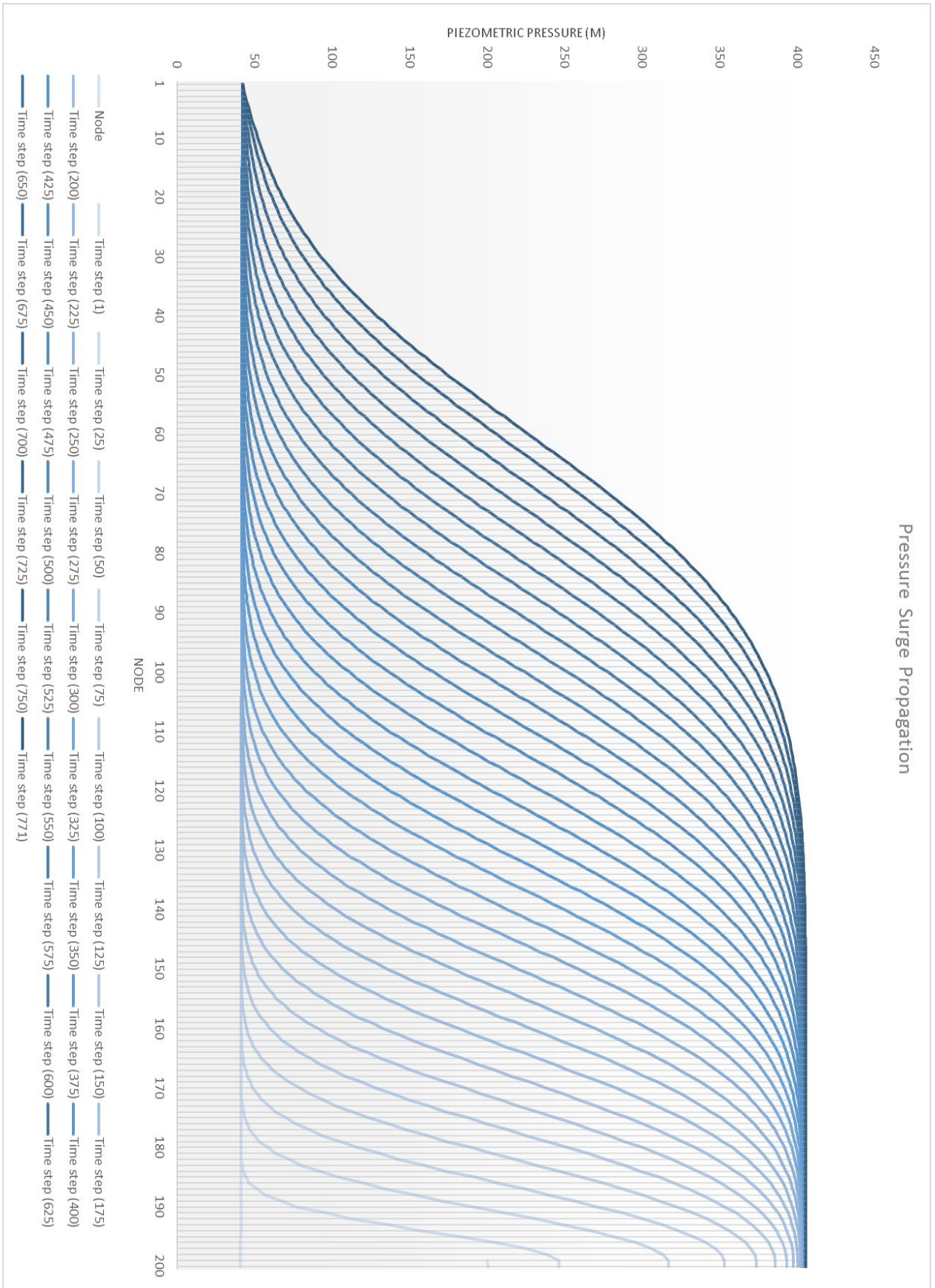


FIGURE 20: TEST CONDITIONS 4 PRESSURE SURGE PROPAGATION

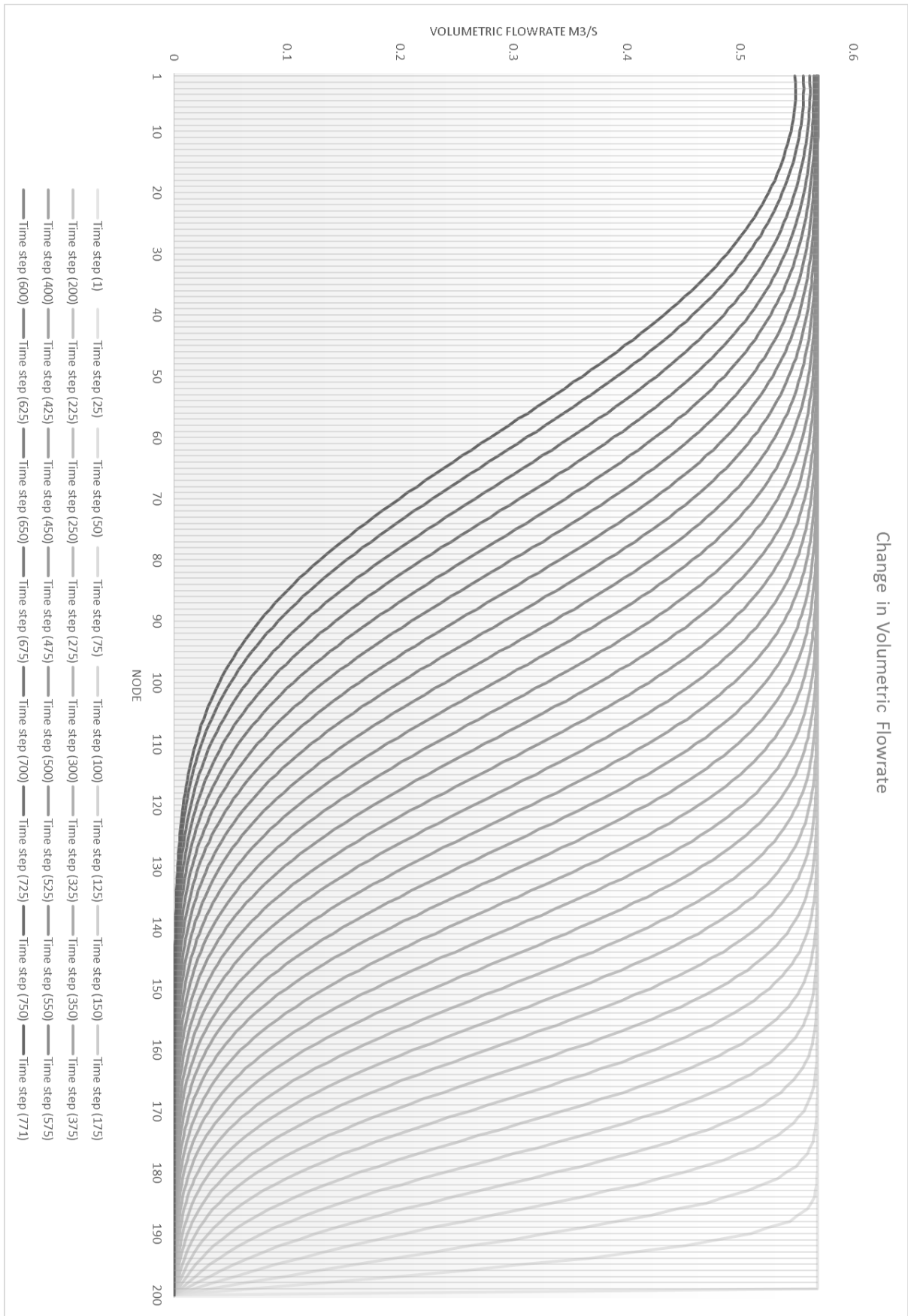


FIGURE 21: TEST CONDITIONS 4 DECREASE IN VOLUMETRIC FLOWRATE

Test Condition 5

The conditions of the pipe for this simulation will be the same as for condition two although the valve closure time will be 0.1 seconds the same as for condition 4.

TABLE 12 : TEST CONDITIONS 5

Test Conditions 5	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.4572
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.1
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

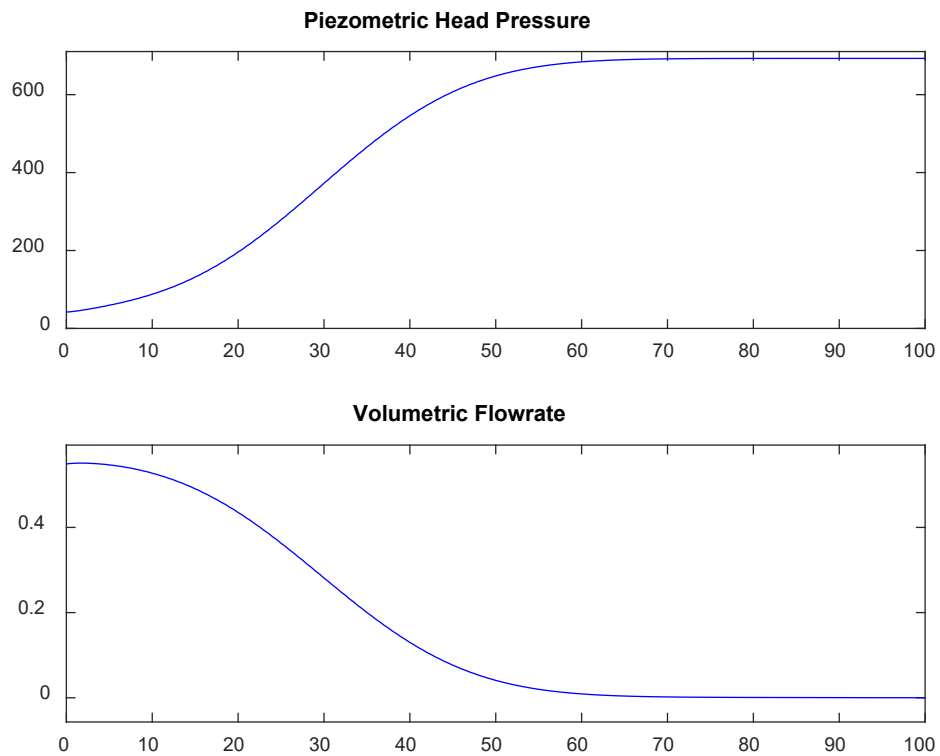


FIGURE 22: TEST CONDITIONS 5 FINAL PLOT

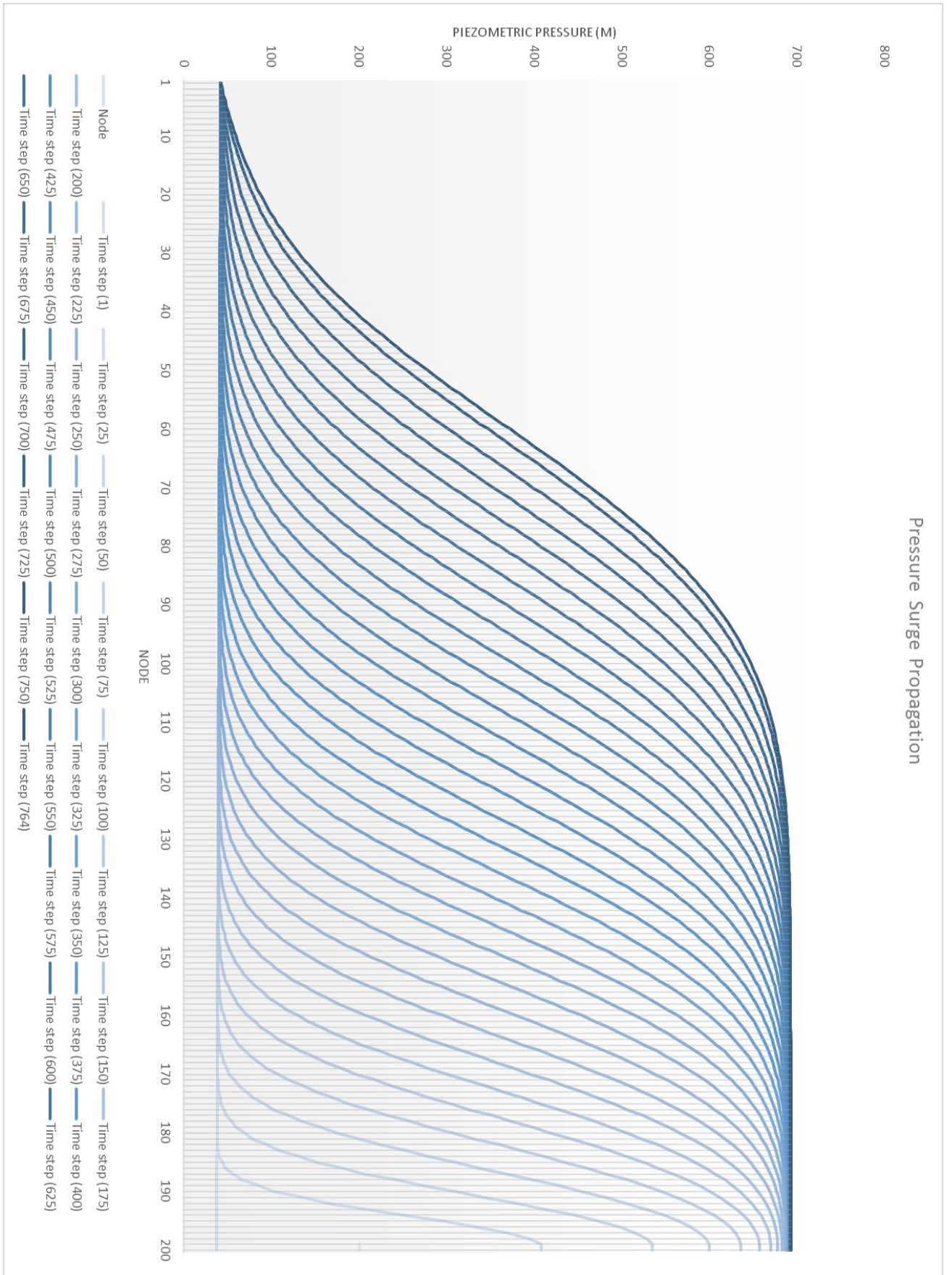


FIGURE 23: TEST CONDITIONS 5 PRESSURE SURGE PROPAGATION

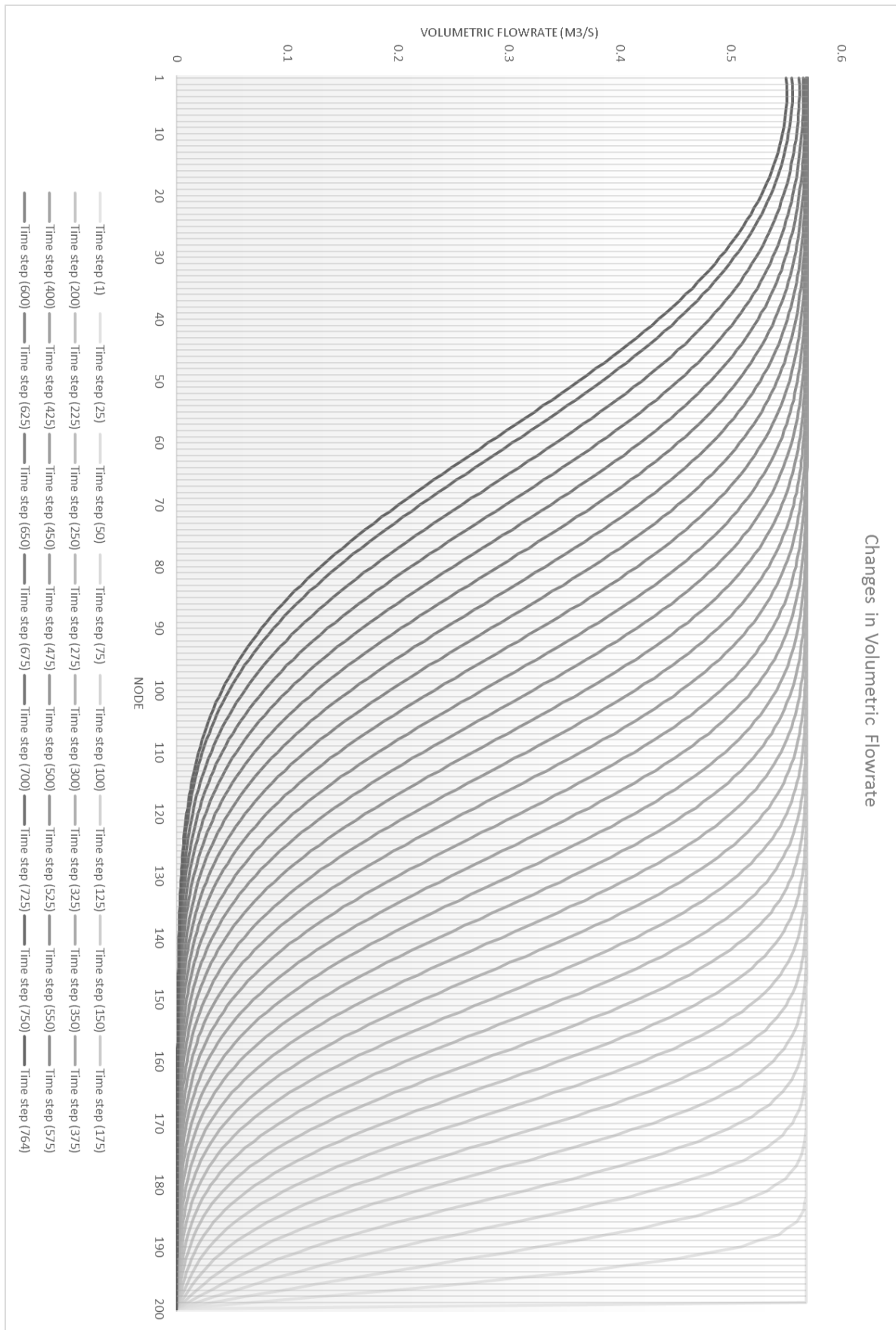


FIGURE 24: TEST CONDITIONS 5 DECREASE IN VOLUMETRIC FLOWRATE

Test Condition 6

Condition 6 will have the same pipe properties as condition 3 but with the same valve closure time as Conditions 4 and 5.

TABLE 13 : TEST CONDITIONS 6

Test Conditions 6	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.3048
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.1
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

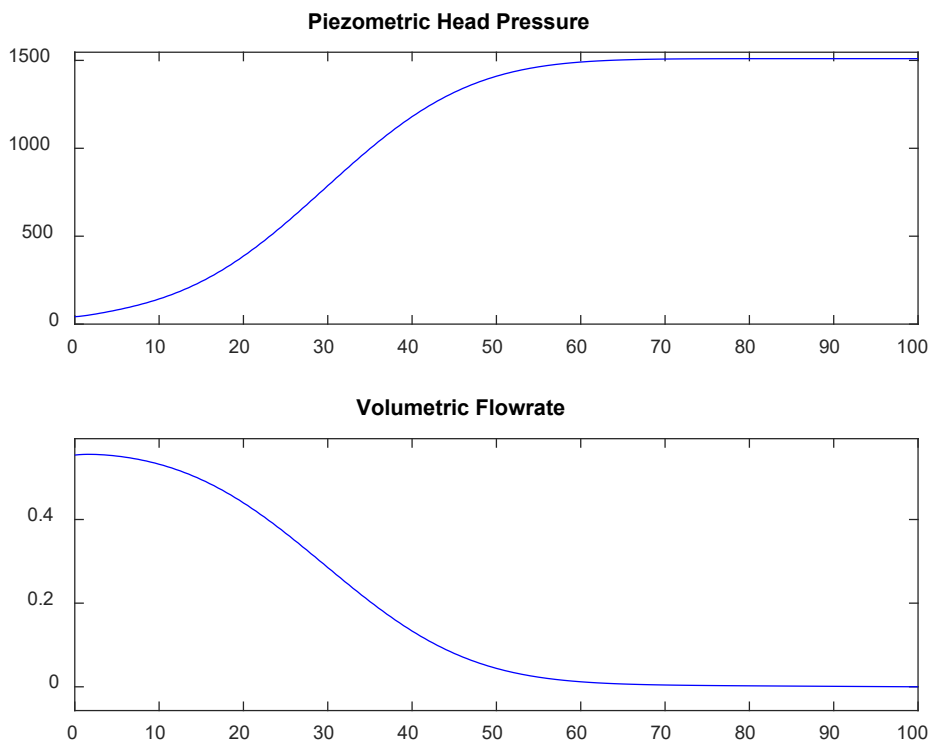


FIGURE 25:TEST CONDITIONS 6 FINAL PLOT

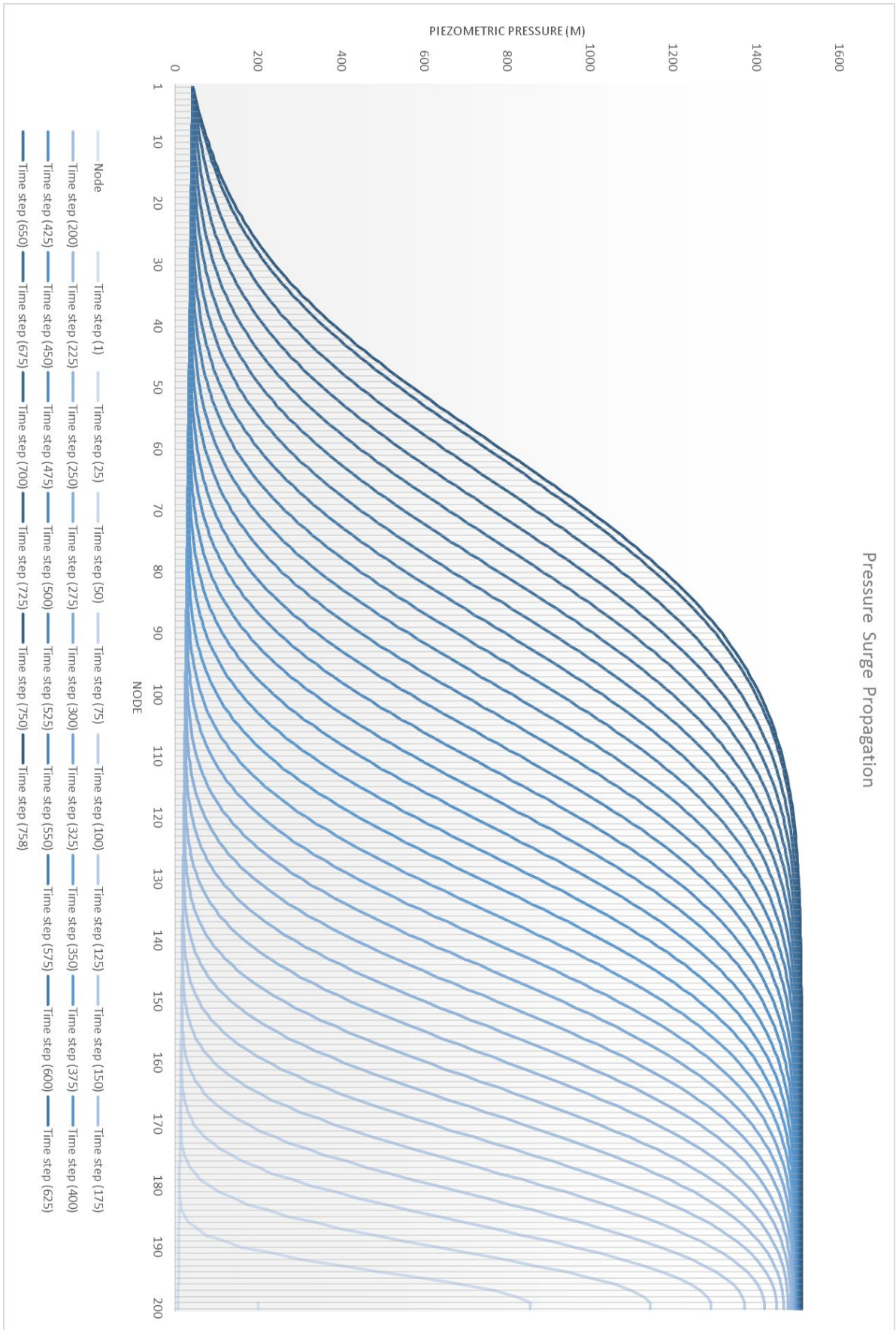


FIGURE 26: TEST CONDITIONS 6 PRESSURE SURGE PROPAGATION

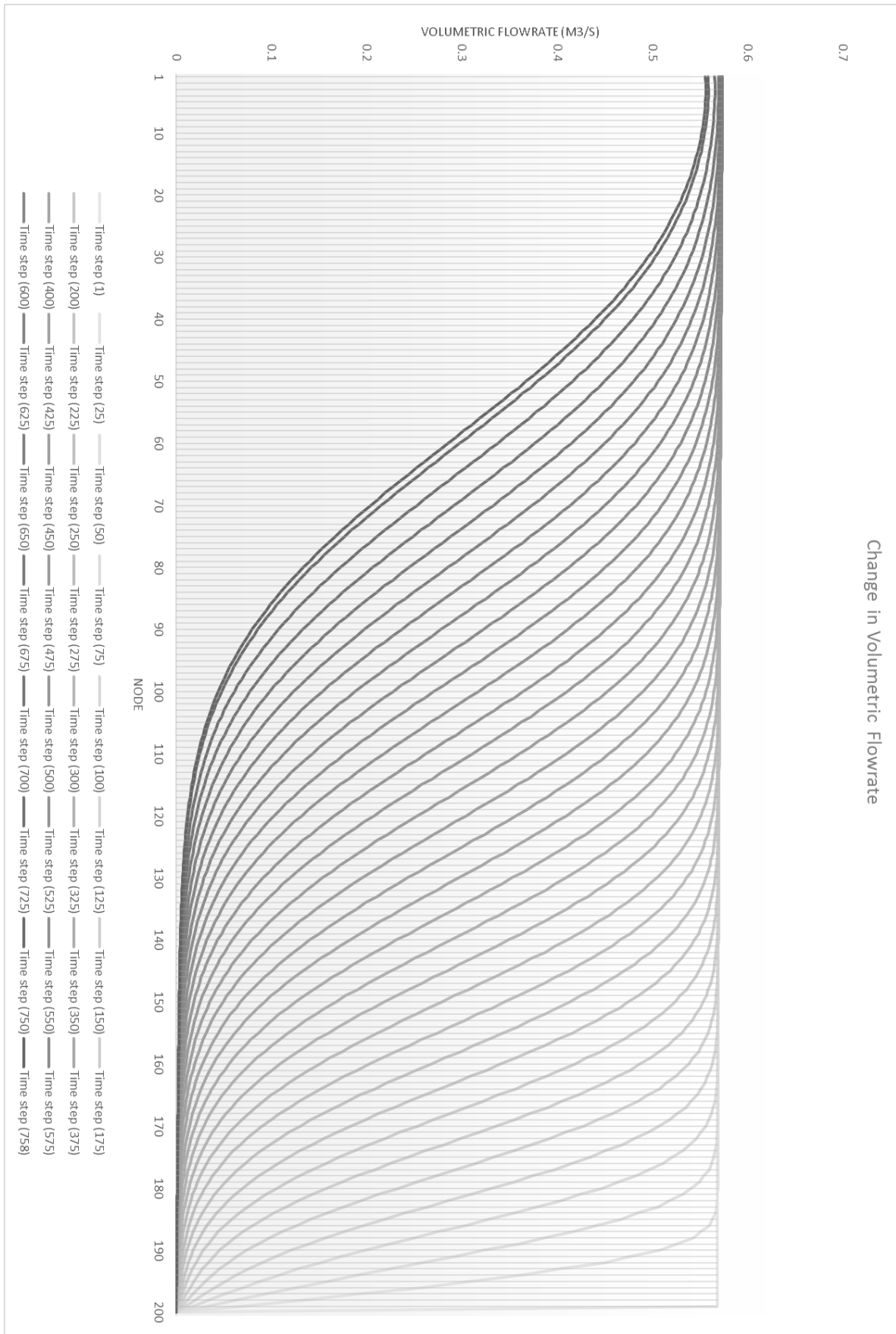


FIGURE 27: TEST CONDITIONS 6 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 7

For condition 7 the pipe properties will be the same as those in condition 1 with the same volumetric flow rate but the valve closure time will be 0.25 seconds.

TABLE 14 : TEST CONDITIONS 7

Test Conditions 7	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.3048
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.25
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

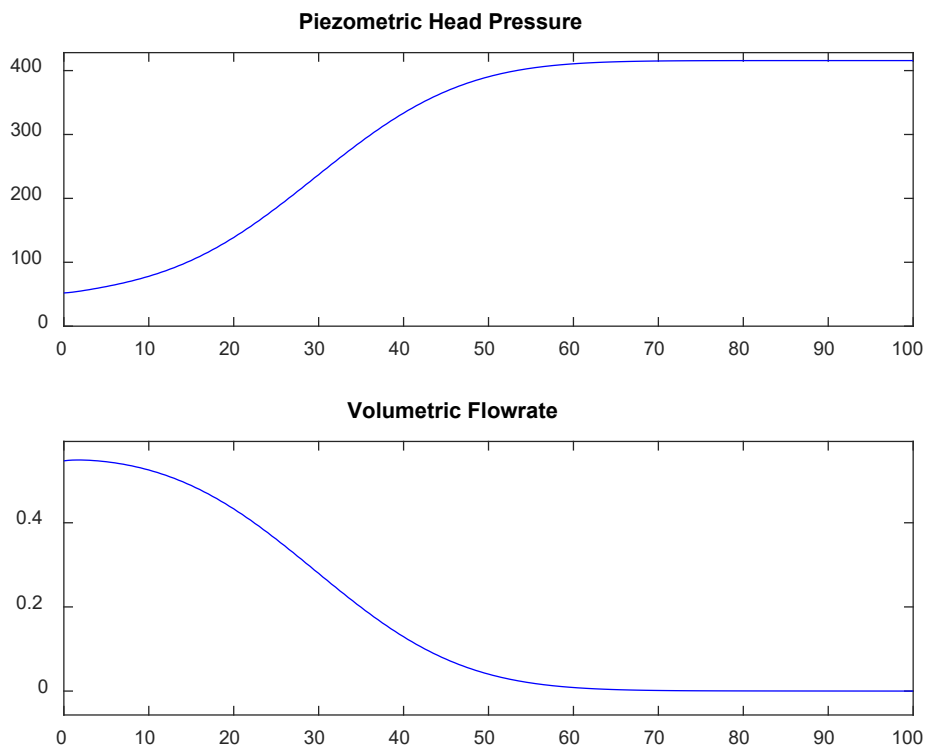


FIGURE 28: TEST CONDITIONS 7 FINAL PLOT

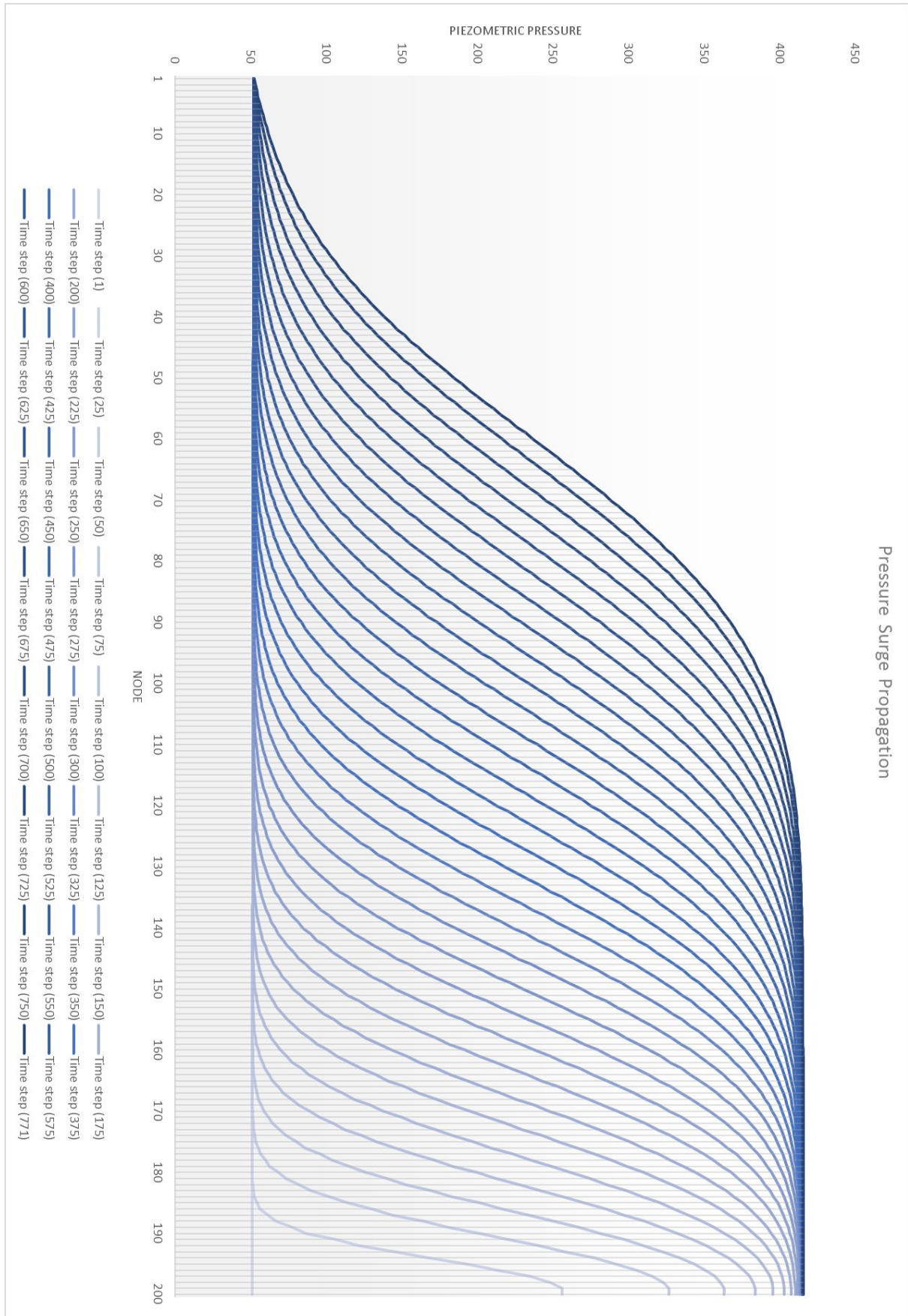


FIGURE 29 : TEST CONDITIONS 7 PRESSURE SURGE PROPAGATION

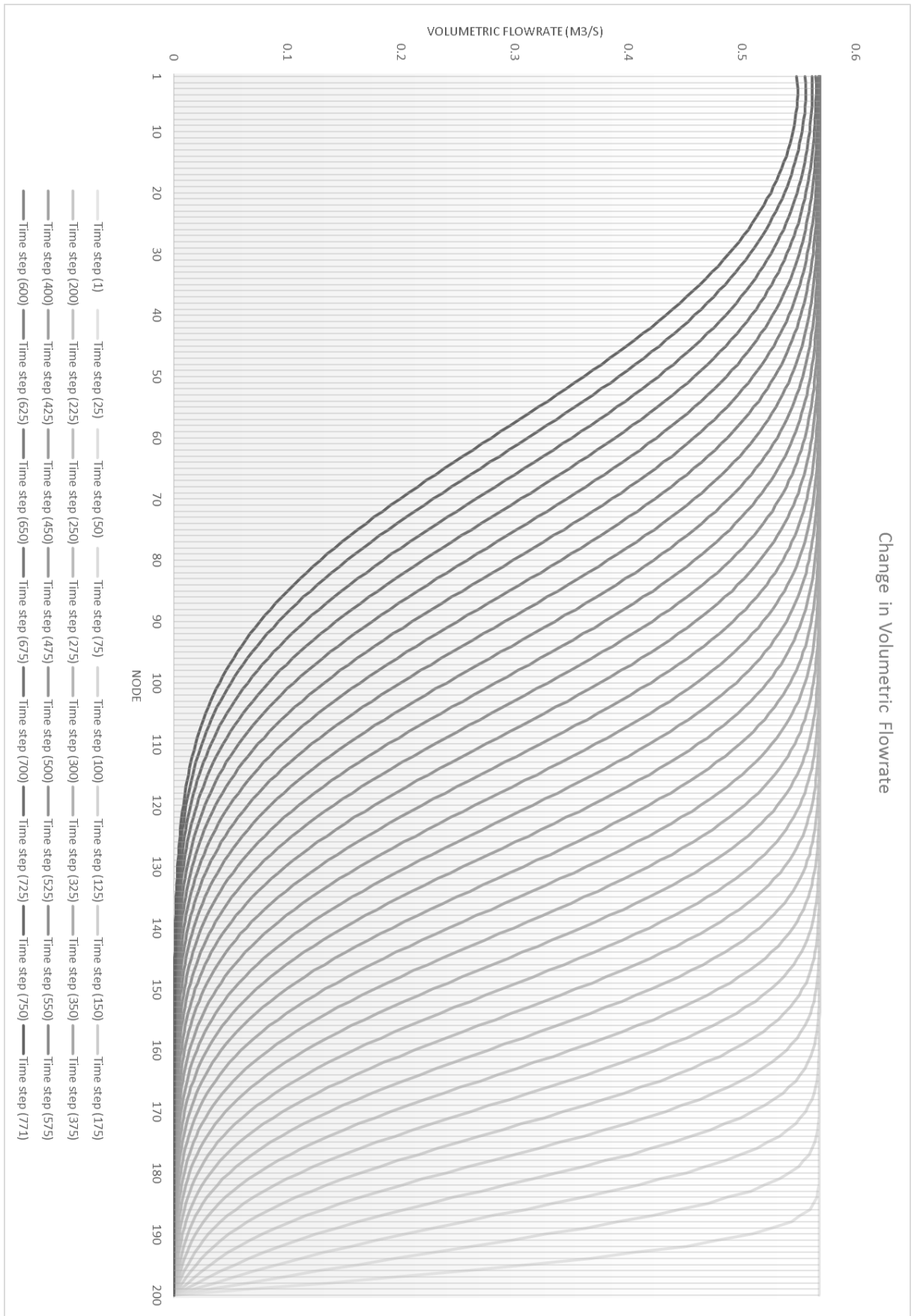


FIGURE 30 : TEST CONDITIONS 7 DECREASE IN VOLUMETRIC FLOWRATE

Test Condition 8

For the pipe properties for condition eight the wall thickness of the pipe will be doubled from the industry standard if two inches to four inches and the remaining properties will be the same as Condition 1.

TABLE 15 : TEST CONDITIONS 8

Test Conditions 8	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.6096
Pipe Wall Thickness (m)	0.1016
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

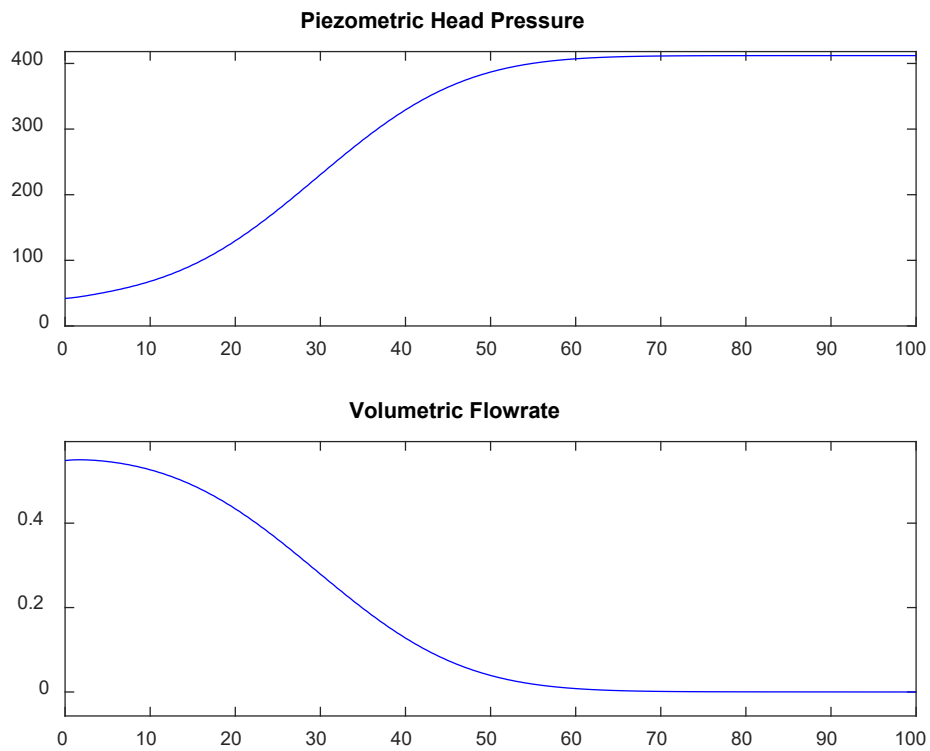


FIGURE 31: TEST CONDITIONS 8 FINAL PLOT

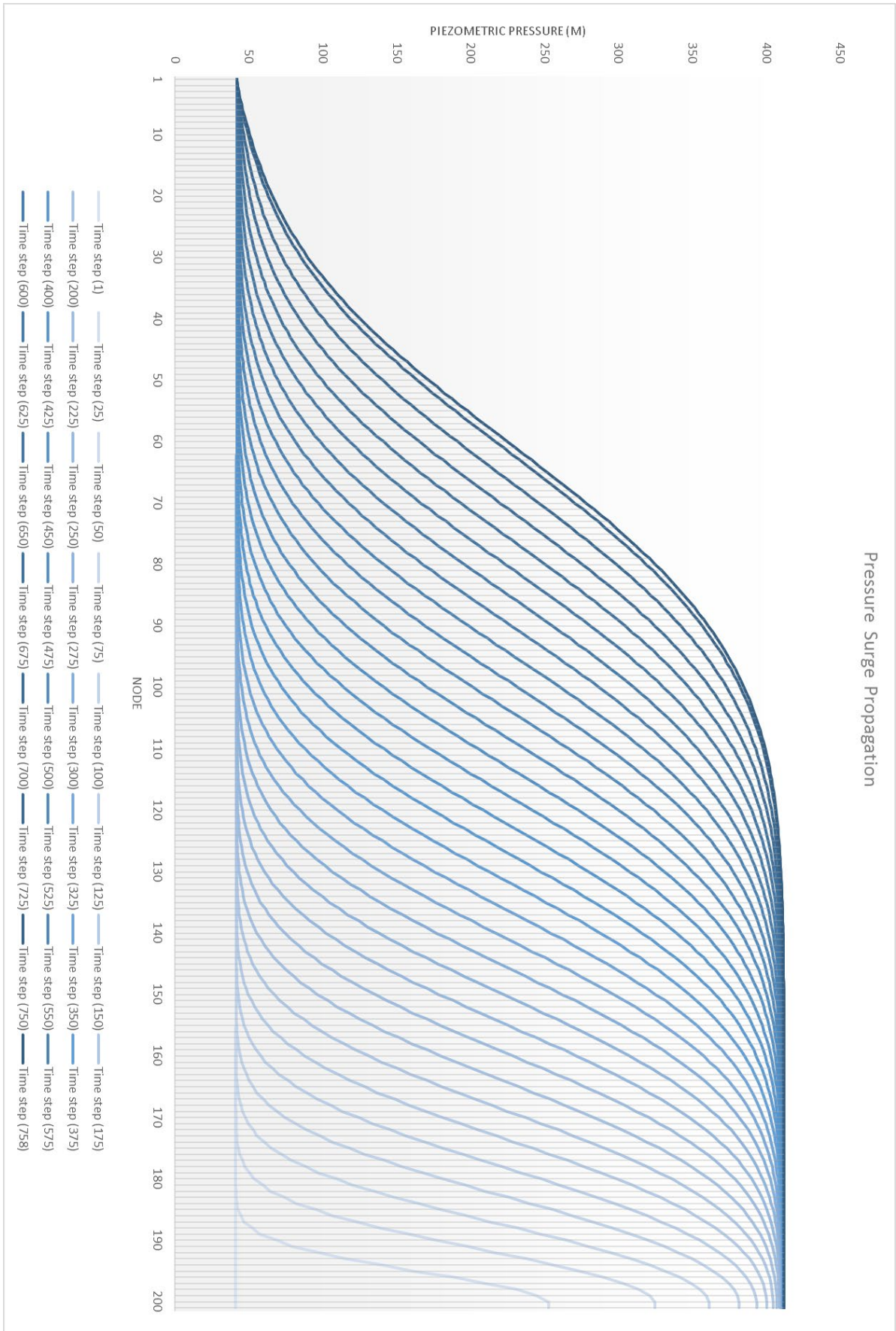


FIGURE 32: TEST CONDITIONS 8 PRESSURE SURGE PROPAGATION

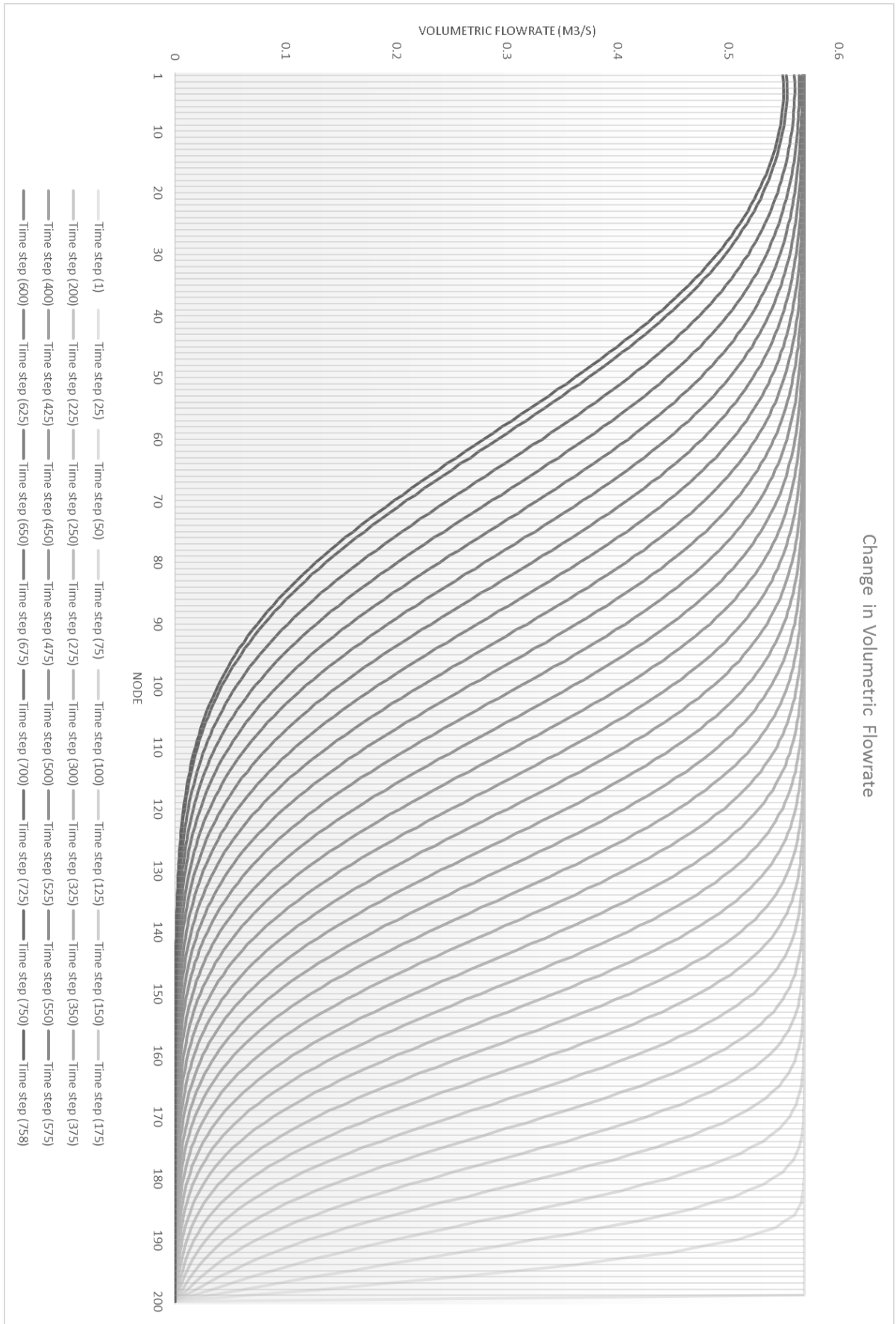


FIGURE 33: TEST CONDITIONS 8 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 9

Condition nine will have the same pipe properties as Condition one but the volumetric flowrate will be increased from 7500 gallons per minute to 15000 gallons per minute, this would represent an upgrade to transfer pumps used at LNG plant.

TABLE 16 : TEST CONDITIONS 9

Test Conditions 9	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.6096
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	1.1365226
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step	0.0001
Fluid Height (m)	20

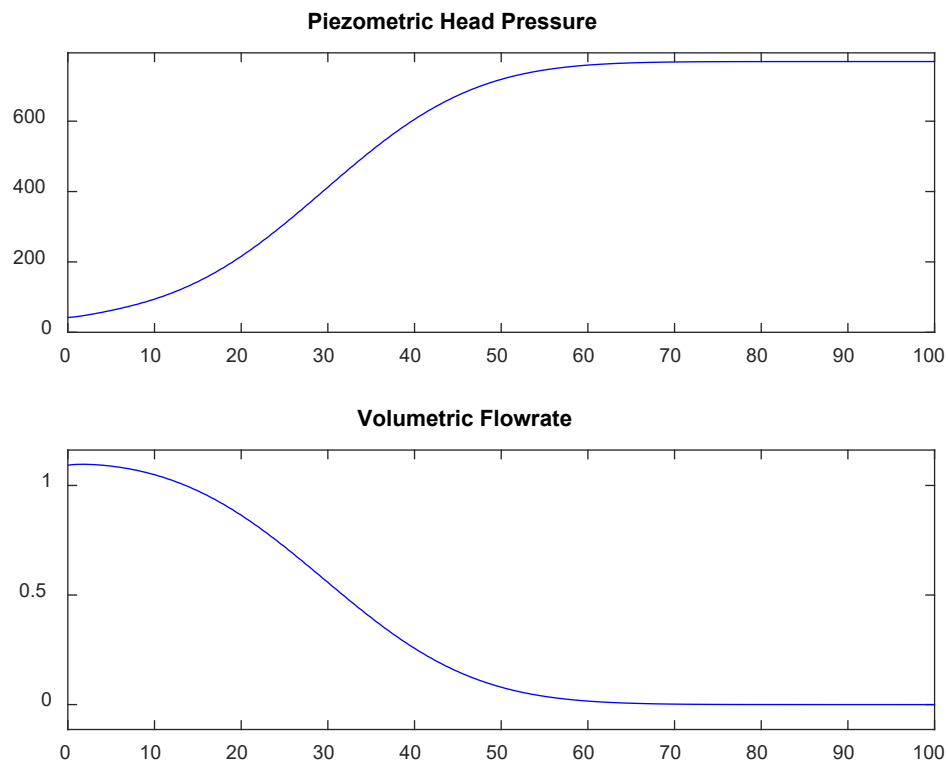


FIGURE 34: TEST CONDITION 9 FINAL PLOT

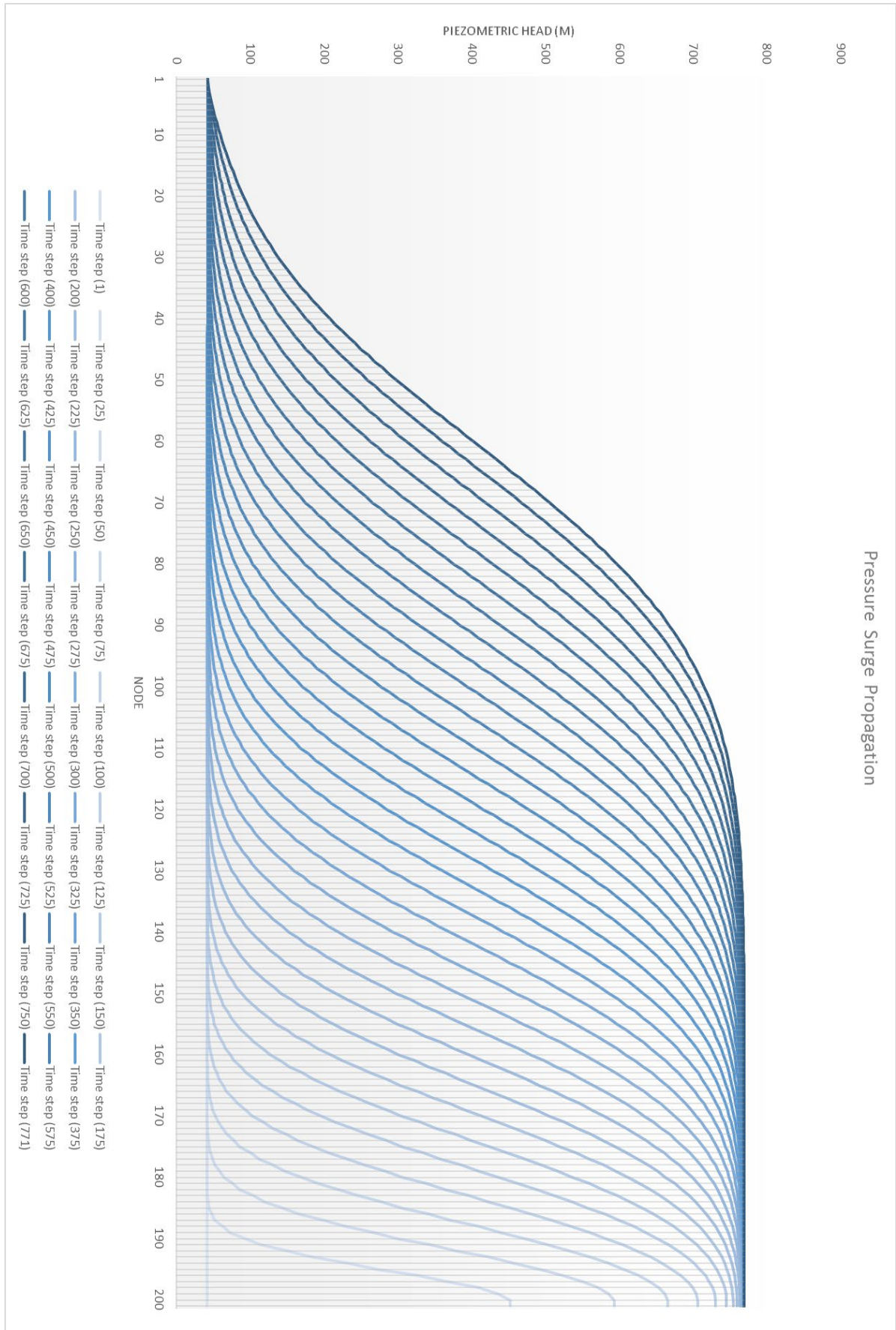


FIGURE 35 : TEST CONDITIONS 9 PRESSURE SURGE PROPAGATION

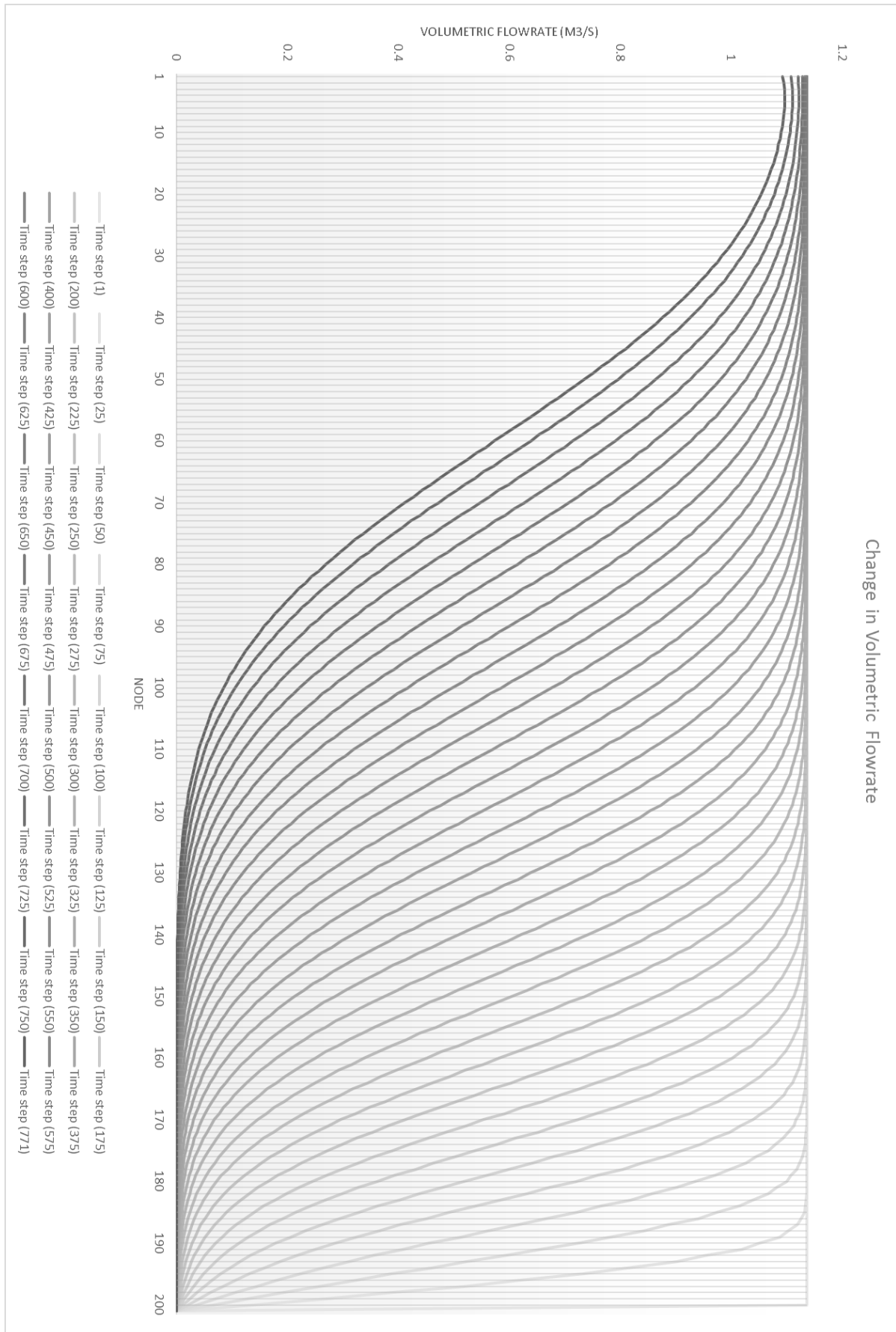


FIGURE 36: TEST CONDITIONS 9 DECREASE IN VOLUMETRIC FLOWRATE

Test Conditions 10

Condition ten will have the same pipe properties as condition one along with the same volumetric flowrate, the difference being is that the height of the fluid will be raised by ten meters to simulate a storage vessel that is 50% taller than the original.

TABLE 17 : TEST CONDITION 10

Test Conditions 10	
Pipe Length (m)	100
Pipe Internal Diameter (m)	0.6096
Pipe Wall Thickness (m)	0.0508
Pipe Material Youngs Modulus (GN/m ²)	141
Fluid Velocity (m ³ /s)	0.5682613
Valve Closure Time (s)	0.01
Grid Spacing	0.5
Total Number of Nodes	201
Time Step (s)	0.0001
Fluid Height (m)	30

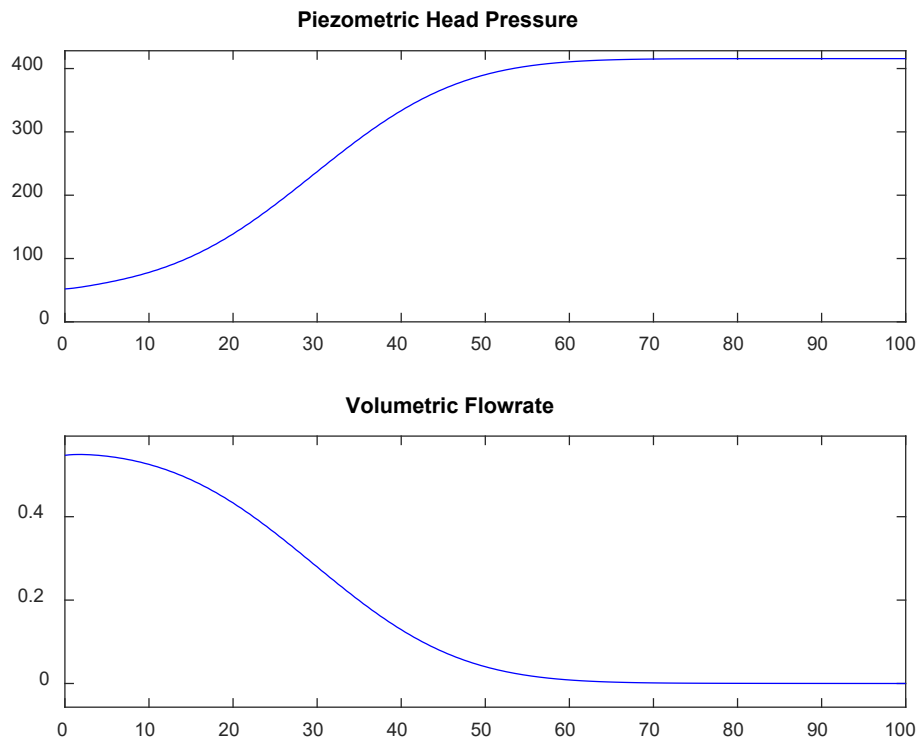


FIGURE 37: TEST CONDITIONS 10 FINAL PLOT

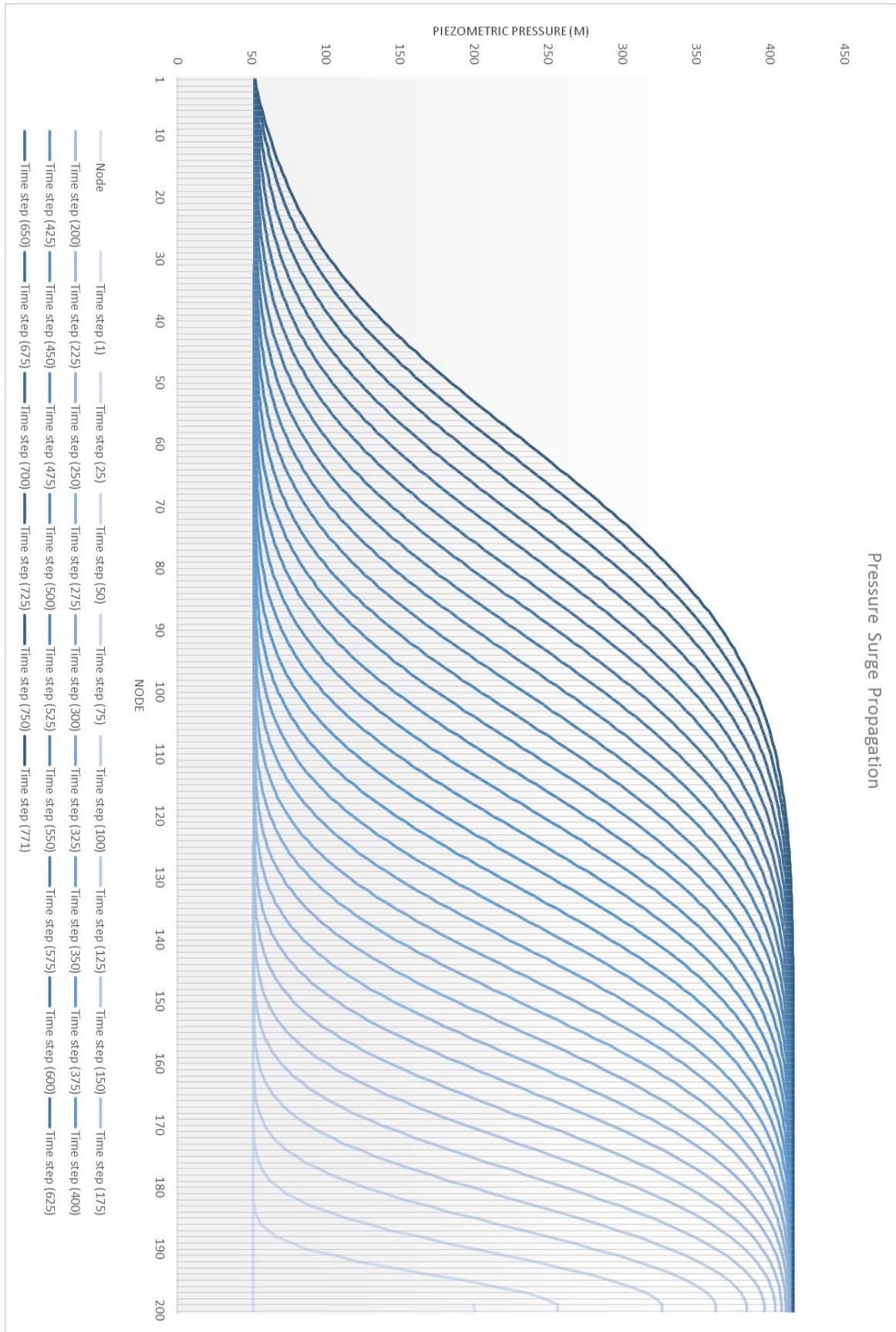


FIGURE 38: TEST CONDITIONS 10 PRESSURE SURGE PROPAGATION

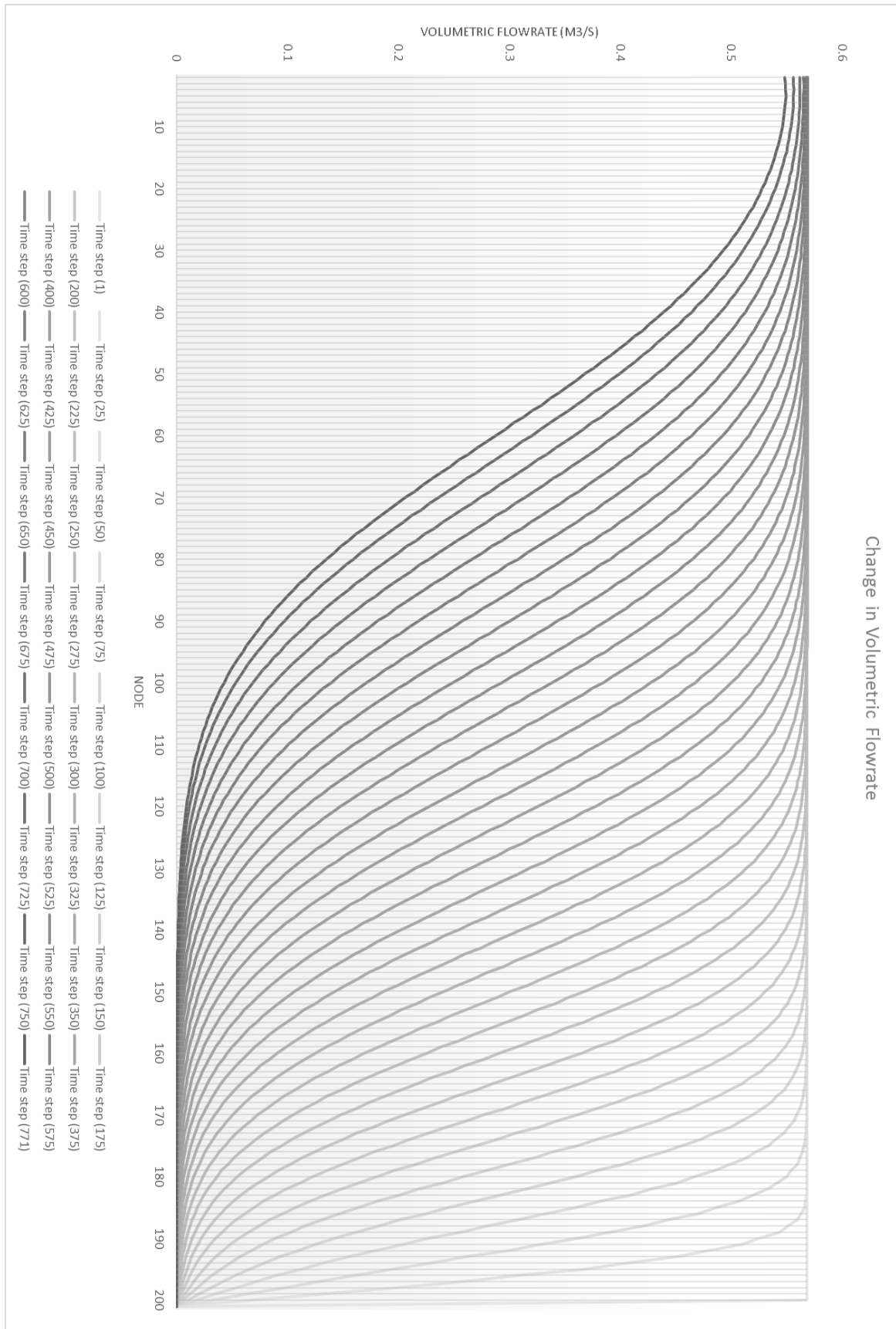


FIGURE 39 : TEST CONDITION 10 DECREASE IN VOLUMETRIC FLOWRATE

Analysis

The flow of the fluid through the pipeline with conditions that do not change is known to be steady flow, where the conditions change the flow is known to be unsteady. For the analysis section of this report comparisons of the effects of the various conditions will be made, this will highlight the changes in magnitude of the pressure wave because of different pipe and flow properties and how different conditions may result in the release of Hazardous substances into the environment through the partial or total separation of the upstream break-away coupling.

Valve Closure

Comparisons between the valve closure times of 0.25 seconds and 0.01 seconds show that the value given for the difference valve closure time was too small to result in variations in results and using a longer valve closure time for this problem would not be worthwhile as the break-away couplings that are used would not have a closure time greater than 0.25 seconds as their main application is to reduce the loss of product during the application of loading on transfer pipelines.

If the valve closure for the pipeline was to be somehow greater than 0.75 seconds then when the simulation is run the pressure wave would not reach the reservoir node as the time taken for the valve node to reach zero meters per second would be greater than the simulations run time and would require the code to be altered to accommodate this change in condition.

The rate at which the valve closes could only effect the time taken to reach the maximum surge pressure within the pipeline providing the initial pipe, piezometric head pressure and flow rate conditions are the same.

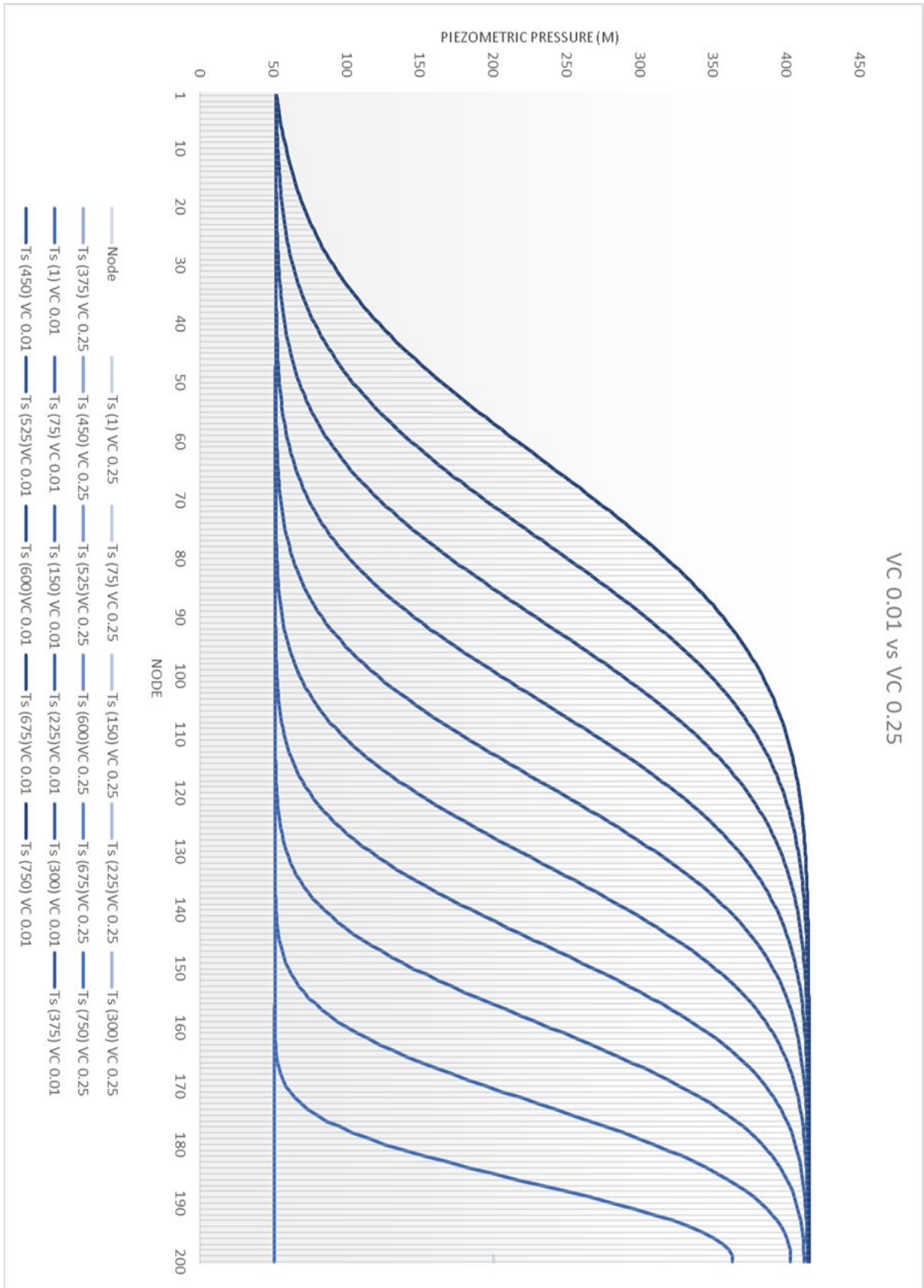


FIGURE 40: COMPARISON OF VALVE CLOSURES 0.01 VS 0.25

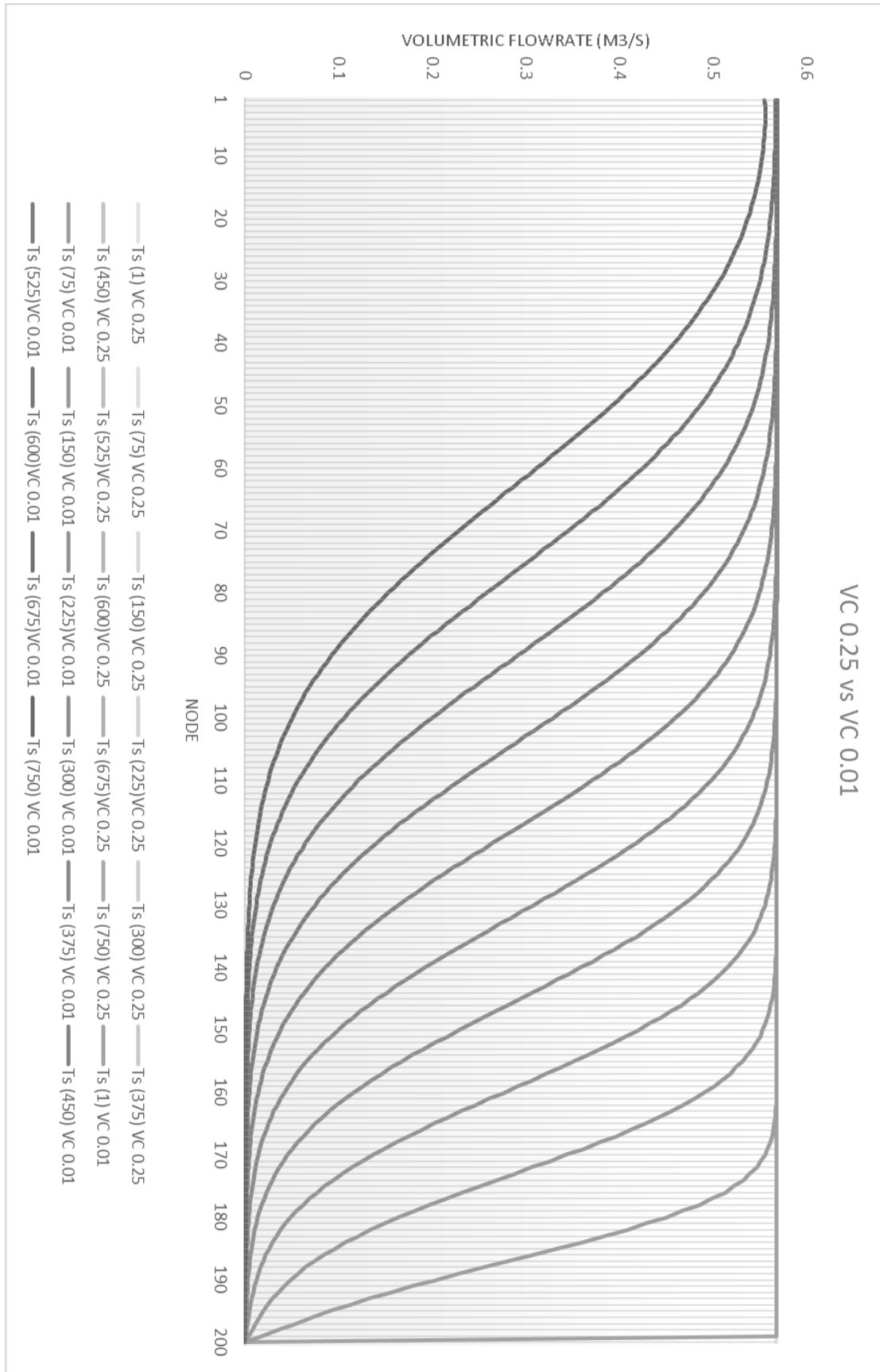


FIGURE 41: VC 0.25 vs VC 0.01 EFFECT ON FLOWRATE

Pipe Internal Diameter

The internal diameter of the LNG transfer can vary depending on the equipment in use and the transfer pumps that are in use at different LNG plants. The industry standards for 36%NiFe Cryogenic Piping is twenty-four inches, eighteen inches and twelve inches. The twelve-inch bore is generally only used for transfer lines that are no greater in length than fifty meters as the head loss due to friction can have a major effect on the efficiency of the transfer pumps and the overall cost. Whilst knowing this a bore of twelve inches was used for a simulation with a pipe length of one hundred meters to establish what effects the smallest internal diameter would have on the magnitude of the pressure surge.

Figure 33 has both pressure surges for a twenty-four-inch diameter and eighteen-inch diameter pipe plotted at time intervals of seventy-five tie steps across the length of the transfer pipeline. The graph shows that the pressure surge of the eighteen-inch pipe peaks at six hundred and seven meters where the twenty-four-inch pipe has a peak piezometric pressure of four hundred and five meters. What is also shown in figure 33 is that the speed at which the pressure surge peaks is almost the same at the 150th time step this is mostly likely due to the valve closure speed being the same for both simulations.

Figure 34 shows the comparison between a twenty-four-inch diameter transfer pipeline and a twelve-inch pipeline, the graph shows that the maximum surge pressure of the twelve-inch pipe is 1509m this shows along with the comparison shown in figure 33 that with decreasing pipe diameter the maximum surge pressure increases.

The pressure wave increase that is represented by the increasing piezometric pressure is a result of the column of fluid travelling in the pipeline towards the valve, when the valve is shut rapidly the fluid that is at the point at which the valve has closed compresses slightly. When the fluid has compressed in a pipe with a large diameter the fluid can absorb more of the kinetic energy because of the greater volume of liquid present, where a smaller diameter pipe is used and the fluid is stopped rapidly there is some compressibility but not as much as with a larger diameter pipeline and thus less energy is absorbed by the fluid and a greater amount is transferred into the pressure surge that travels upstream.

The change in Pipe diameter did not influence the change in volumetric flowrate as the rate of change was the same for each pipe diameter.

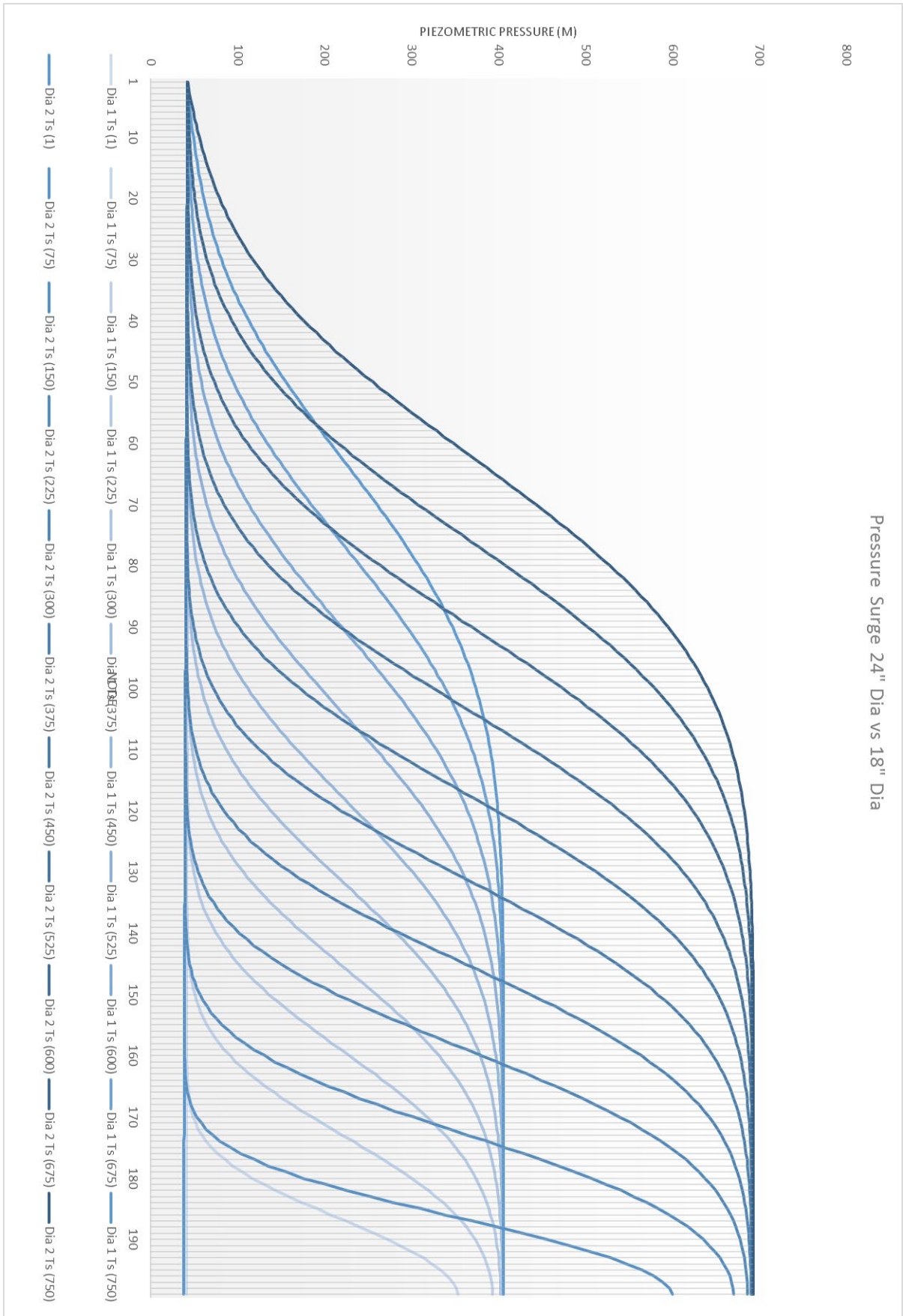


FIGURE 42: PRESSURE SURGE 24" (DÍA 1) AND 18" (DÍA 2)

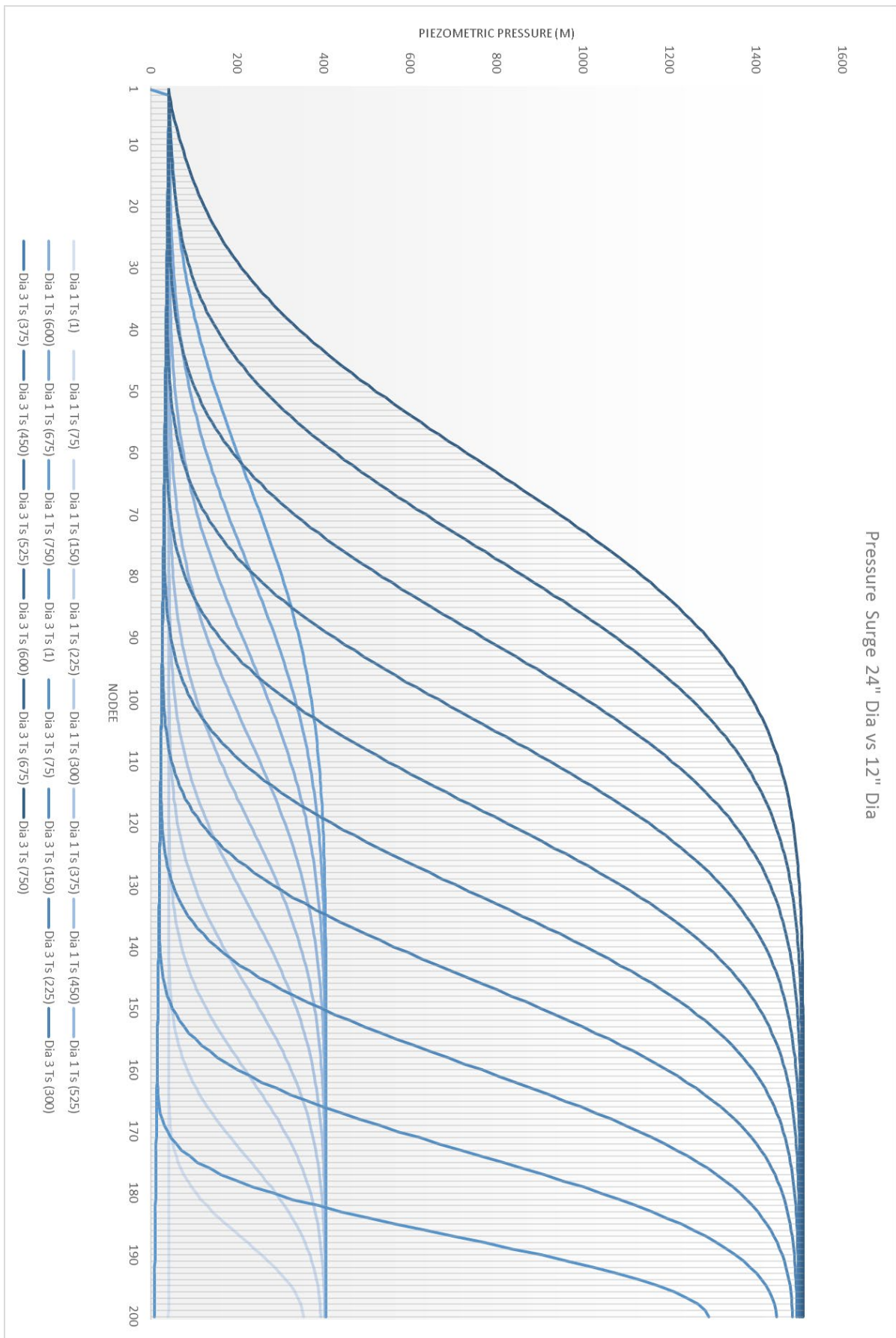


FIGURE 43: PRESSURE SURGE 24"(DIA 1) VS 12"(DIA 3)

Wall Thickness

The wall thickness for the LNG transfer pipe lines used in industry is two inches, this was doubled for a simulation to see what affect the wall thickness would have on the increase in surge pressure.

With the elastic properties of the fluid allowing for compression when there is a rapid change in volumetric flow rate and knowing from the simulations that have been run for different diameter pipes and the increase in pressure surge when the diameter of the pipe is reduced, the effects of how increasing the wall thickness to limit the elasticity of the pipe should be established.

When increasing, the wall thickness the internal diameter of the pipe remained at twenty-four inches this ensured that the volumetric flow rate remain the same allowing for comparison with the first simulation.

Figure 37 shows that when the simulations were run for a wall thickness double the original thickness of two inches the pressure surge maximum value increased from 406 meter to 412 meters, this increase is due to the pipe restricting the compression of the fluid during transient and resulting in the increase in pressure surge.

What is also noticeable from the comparison is the point at which the pressure surge was measured for Wall thickness two, the measurements were taken at the same intervals of seventy-five time steps, however the measurements taken for wall thickness two were further upstream than those of Wall thickness one. This is a result of the wall thickness being increased as the wall thickness when increased has a direct effect on the sonic wave speed within the pipe.

$$c = \sqrt{\frac{1}{\rho \left(\frac{1}{K} + \frac{D}{E \cdot e} \right)}}$$

Where e is the pipes wall thickness and E is the pipe materials modulus of elasticity and as shown through the comparison when the wall thickness is increased the sonic wave speed is increased.

Knowing that the diameter of the pipe affects the size of the pressure surge and that the thickness of the pipe wall can also affect the surge a combination of a thick high strength pipe wall with a small diameter would create a greater pressure surge than a pipe with a large diameter and thin flexible walls as the compression caused by hydraulic transient is transferred far more effectively into pressure when in a ridged and less flexible pipe than would occur in a flexible pipeline.

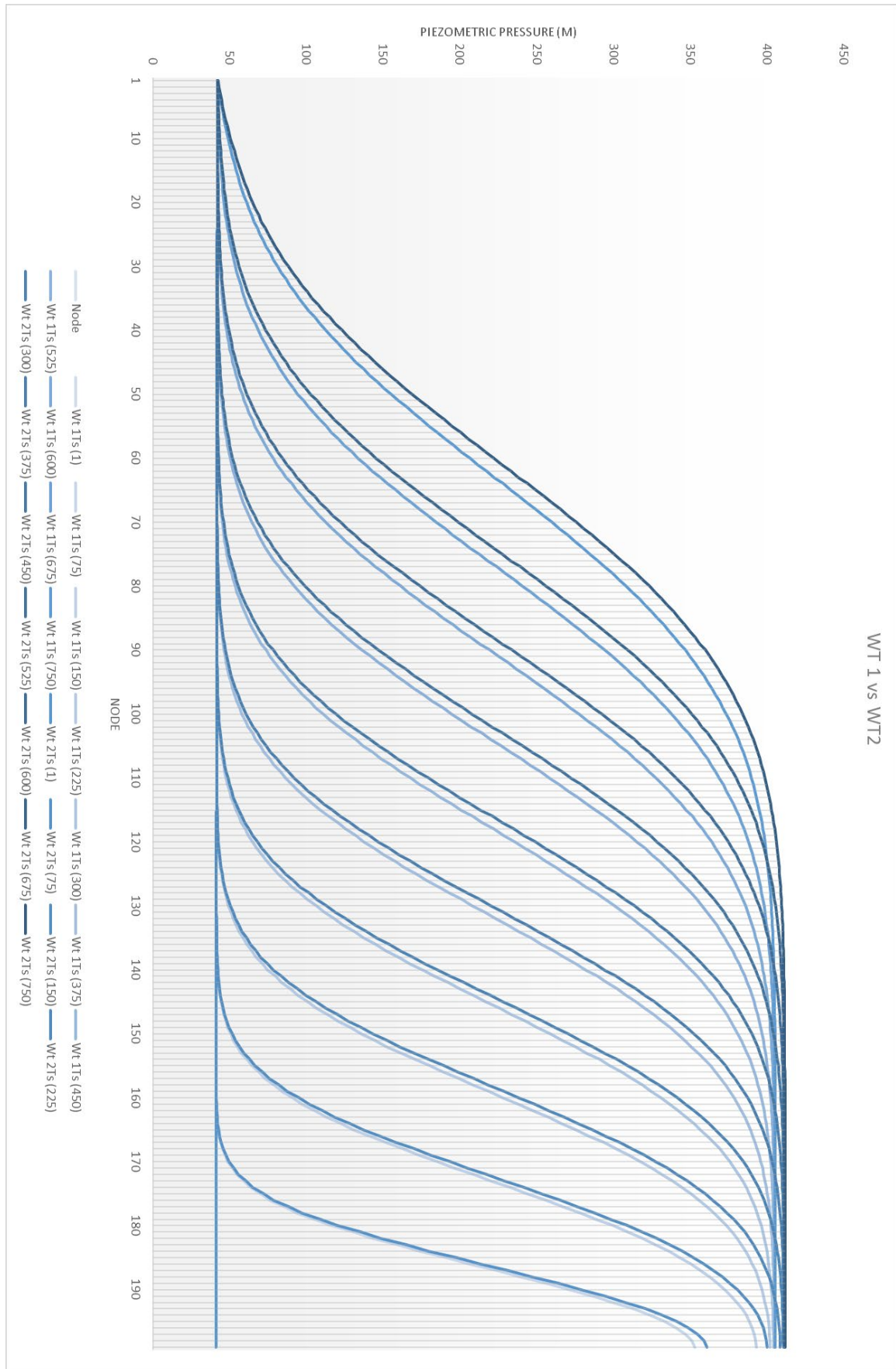


FIGURE 44: EFFECT OF WALL THICKNESS ON PRESSURE SURGE 2" (WT 1) AND 4" (WT 2)

Increase in Volumetric Flowrate

For the simulation run for condition nine the pipe properties and valve closure time were the same as those run for condition one, the volumetric flowrate was increased to 15000 gallons per minute from 7500. This increase in flowrate was used in the simulation to establish what effects an improvement on transfer rate would have on the pressures and rate of deceleration of the fluid within the transfer line.

Shown in the comparison between the two pressures resulting from both the simulations the pressure surge resulting from the increased flow rate of 15000 gallons per minute has a maximum value of 769 meters compared to the maximum of 405 meters from condition one. This would be expected as the greater the fluids volumetric flow rate the greater the fluid velocity and kinetic energy, assuming the pipe properties are the same.

The pressure surge is not double the surge seen with the initial volumetric flow rate this is due the fluid with the greater volumetric flow rate having a higher Reynolds number which results in a greater energy loss through turbulent flow and frictional losses, also because of an increased pressure surge the greater the amount the pipe walls flex which will absorb some of the energy from the pressure surge.

The graph representing the change in volumetric flowrate shows that the greater the volumetric flowrate the decrease in fluid velocity is far more rapid shown by the steeper lines representing deceleration. The graph also shows that the greater the volumetric flow rate the time taken for the fluid to decelerate is slightly greater, this can be linked to the Bulk modulus of the fluid which is the fluids ability to decrease in volume when pressure is applied.

Where the column of fluid is rapidly decelerating the fluid in front is compressed, with the greater volumetric flowrate the greater the fluids compression acting like a spring compressing when a larger load is applied. This increase in compression allows the fluid with the greater volumetric flowrate a greater distance to come to a stop.

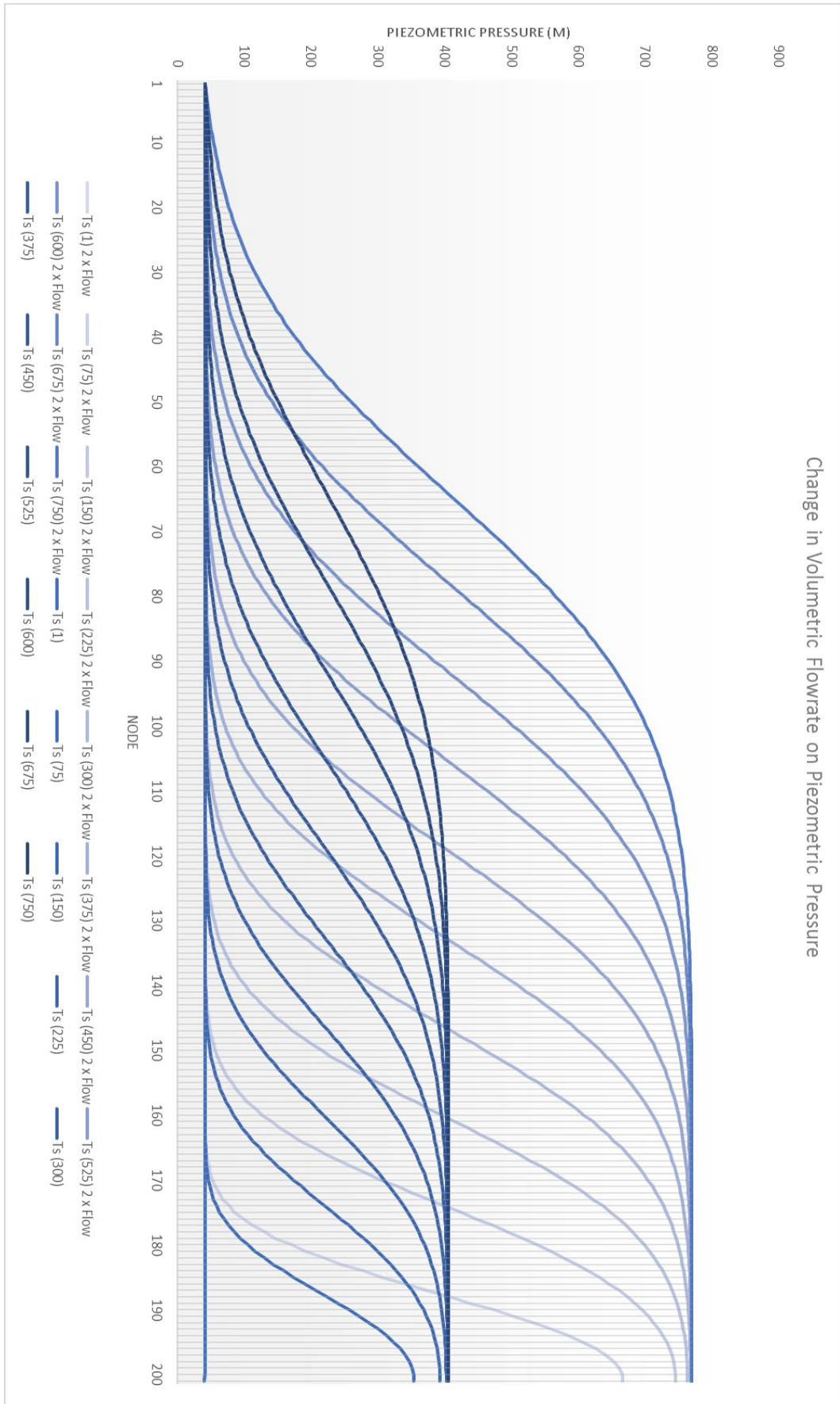


FIGURE 45: EFFECTS OF INCREASED VOLUMETRIC FLOW RATE

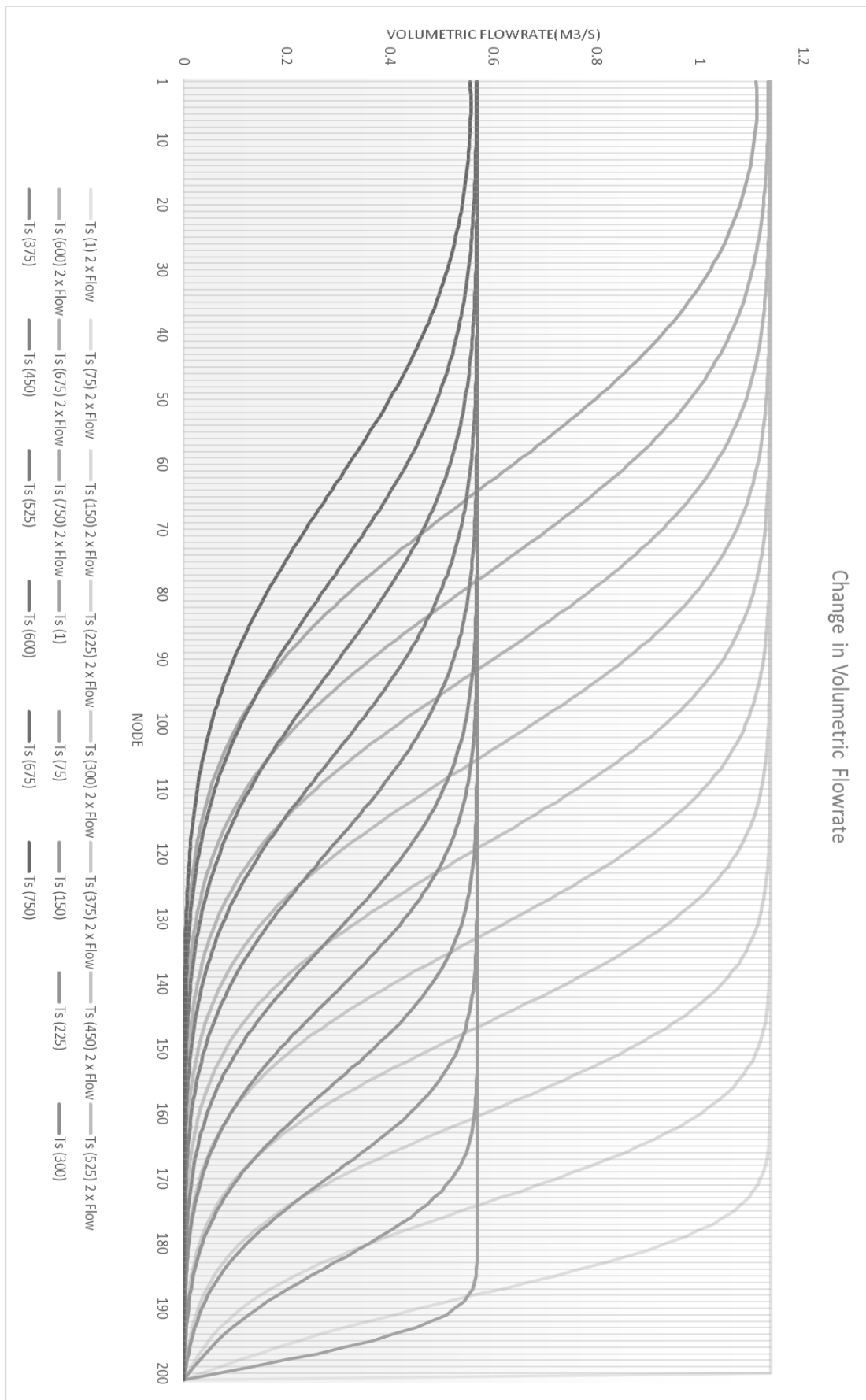


FIGURE 46: EFFECT OF INCREASED VOLUMETRIC FLOW RATE

Discussion

From the simulations run and the comparisons made between the results of the differing conditions it can be shown that the pipe properties have the greatest effect on the size of the pressure surge caused by a rapid change in volumetric flowrate. A decrease in internal diameter will result in a greater pressure surge with the same volumetric flow rate as the velocity of the fluid is increased. This fluid when subjected to rapid deceleration from valve closure causes a rapid change in the fluids density with the fluid nearest the closed valve compressing first then the compression of the fluid travelling upstream to the reservoir, this is the pressure wave. With a smaller diameter, the column of fluid has a smaller cross sectional area which is compressed by the same force as a pipe with a larger cross sectional area and the same volumetric flow rate.

An increase in wall thickness also resulted in an increase in the pressure surge this is a result of hoop stress applied onto the pipe during the pressure surge not causing as much elastic deformation of the pipe, where the pipe absorbs energy during the pressure surge.

The valve closure rate used for this project had to be below 0.25 seconds as the break-away couplings used to prevent loss of product to atmosphere close at this rate, this meant that it is not relevant to look at slower shutting valves as this would result in product release. The comparisons done on the differing valve closure times of 0.01, 0.1 and 0.25 seconds all induced pressure surges with very little difference in pressure surge size and the rate of deceleration of the fluid.

When the volumetric flowrate was doubled the pressure surge that resulted was almost double, it would not have resulted in exactly double the pressure surge as there would be an increase in the Reynolds number of the fluid traveling through the pipe which results in a greater amount of energy lost through friction, increasing the volumetric flowrate has a similar effect to the decrease in diameter as the velocity of the fluid in both situations is increased resulting in a greater change in the fluids density during hydraulic transient.

The aim of this project is to determine whether the pressure surge caused by the activation of a downstream breakaway coupling would cause the activation of a breakaway coupling located fifty meters upstream. Knowing that the fifty-meter point of the one-hundred-meter transfer line is represented by node 101 of the simulation the piezometric pressure can be determined.

To determine the loading the break studs that the breakaway couplings are subjected to the following formula was used;

$$W = \pi a^2 \times P$$

EQUATION 17: AXIAL LOAD CALCULATION

Where;

W = Axial force due to internal pressure

a^2 = Break away coupling interface area

P = internal pressure

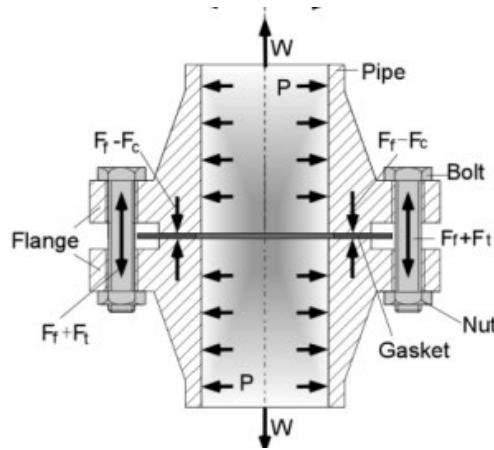


FIGURE 47: AXIAL LOADING ON COUPLING

TABLE 18 : CALCULATED INTERFACE FOR BREAK-AWAY COUPLING

Test Conditions	Break-away Coupling Interface Area (m ²)
1	0.1054
2	0.08108
3	0.056754
4	0.1054
5	0.08108
6	0.056754
7	0.056754
8	0.22701
9	0.1054
10	0.1054

TABLE 19 : $P_i * a_2$ CALCULATION FOR AXIAL LOADING

Test Conditions	$P_i * a_2$
1	0.0349
2	0.02065
3	1.0119
4	0.0349
5	0.02065
6	1.0119
7	1.0119
8	0.16186
9	0.0349
10	0.0349

TABLE 20 : AXIAL LOADING ON BREAK-AWAY COUPLING

Condition	$P_i * a_2$	Maximum Pressure (N/m ²)	Axial Load (N/m ²)	Pin Dia 0.005 (m) Break load	Pin Dia 0.0075 (m) Break load	Pin Dia 0.009 (m) Break load
1	0.0349	1,838,581	64,166	38,484	86,588	124,688
2	0.02065	3,141,142	64,864	38,484	86,588	124,688
3	1.0119	6,842,966	6,924,397	38,484	86,588	124,688
4	0.0349	1,838,581	64,166	38,484	86,588	124,688
5	0.02065	3,141,032	64,862	38,484	86,588	124,688
6	1.0119	6,842,966	6,924,397	38,484	86,588	124,688
7	1.0119	1,867,072	1,889,291	38,484	86,588	124,688
8	0.16186	1,883,903	304,928	38,484	86,588	124,688
9	0.0349	3,487,481	121,713	38,484	86,588	124,688
10	0.0349	1,883,903	65,748	38,484	86,588	124,688

The cells coloured red indicate that the upstream breakaway coupling will activate when the pressure surge propagates to node 101 of the system and the yellow cells represent the breakaway coupling not activating.

The blue cells represent where the loading on the pins is not great enough to cause breakaway coupling activation but may cause the break studs to lengthen or partially fracture.

Knowing that the break studs highlighted in blue have been subjected to loading that could cause partial activation and the release of product to atmosphere it would be recommended that after the downstream coupling has been activated both break away couplings are removed and have the break studs replaced with new studs to prevent partial separation.

Conclusion

From the analysis of the results gained from running the simulations with a variety of possible initial conditions of volumetric flow rate, pipe properties and valve shut off speeds and the calculation of axial loading on the upstream breakaway coupling it can be determined whether the upstream breakaway coupling requires immediate onsite maintenance to replace the break studs to prevent partial separation and product loss to atmosphere.

Providing the user of the code written in MATLAB has access to the correct values for the fluid, pipe, valve and reservoir properties then the simulation can be run to establish whether reactive maintenance will have to be completed on transfer pipeline carrying differing fluids to the fluid discussed in this project.

From an environmental view, understanding where the risks of product release may occur can greatly decrease the probability of product release but also improve the clean-up operation and the location of emergency equipment.

The information gained from using this script would also be of use to any site engineer when analysing the emergency shut down system as it would prove beneficial to position strain gauges on each of the breakaway couplings to shut down transfer equipment and to sound safety alarms to inform any personnel of the situation and to follow the sites safety plan.

A site Engineer could use the data from the simulations to establish the pressure surge force at set intervals where Break-away couplings are likely to be fitted. Along with axial loading calculations from possible ship movement these could be combined to establish break stud diameters for each coupling which would avoid unnecessary maintenance on every Break-away coupling after pressure surge event. Knowing what conditions Process safety critical equipment is potentially going to be exposed to can greatly improve the quality of maintenance and operational routines within a business resulting in greater preparedness for emergencies.

Evaluation

The main body of this project was creating the code in MATLAB, the code that was written allowed for the minimum amount of assumed values to be used. Using the Colebrook-White equation for determining the head loss based on the fluid properties and the roughness of the pipe material removed the need for using an approximated value for the head loss due to friction.

Gaining all the relevant properties for the LNG transfer line proved difficult as the equipment and pipe types that are industry standard do not have a wide range of sources as the equipment although when used is used in large quantities remains specialist.

The code itself is very usable providing the user is familiar with MATLAB and how to operate the software and analyse the data. The code is also versatile as it can be applied to many transfer lines providing they are horizontal and uniform in diameter and the code can be run for a wide range of fluids. The representation of the data as the code is run has also been successful as the sub plots do not need any modification regarding axis dimension as the maximum values are calculated when the code is run and these are used to give both the plots for pressure and flow rate accurate scale and allows the plots to be read easily without further user input.

Creating the model in a Computational Fluid Dynamics programme and running it with the same conditions would be a good test of how accurate the code is and how the results differ. Along with the CFD model further improvements could be made on the comparison of simulations as this proved time consuming.

Works Cited

- Technifab Products. (2015). <https://technifab.com/cryogenic-resource-library/cryogenic-fluids/liquefied-natural-gas-lng/>. Retrieved from <https://technifab.com>: <https://technifab.com/cryogenic-resource-library/cryogenic-fluids/liquefied-natural-gas-lng/>
- Chaudry, M. (2014). *Applied Hydraulic Transient Third Edition*. South Carolina: Springer New York Heidelberg Dordrecht London.
- George, H. F. (2007). *Bulk Modulus: What is it? When is it Important?* Hydraulics & Pneumatics.
- GIIGNL. (2016). *LNG Information Paper No. 1*. GIIGNL.
- Hydraulics & Pneumatics. (2017). <http://beta.hydraulicspneumatics.com/hydraulic-fluids>. Retrieved from <http://beta.hydraulicspneumatics.com>: http://beta.hydraulicspneumatics.com/hydraulic-fluids/effects-air-hydraulic-systems-0?utm_test=redirect&utm_referrer=https%3A%2F%2Fwww.google.co.uk%2F
- intricoproducts. (2017, 03). <http://www.intricoproducts.com/product/breakaway-couplings-industrial-breakaway-marine-breakaway>. Retrieved from <http://www.intricoproducts.com/product/breakaway-couplings-industrial-breakaway-marine-breakaway>: <http://www.intricoproducts.com/product/breakaway-couplings-industrial-breakaway-marine-breakaway?gclid=ClvVgfes7dICFUgW0wod7ggNEA>
- KLAW Products . (2017, 03). <http://www.klawproducts.com>. Retrieved from <http://www.klawproducts.com>: <http://www.klawproducts.com/>
- LNG America. (2014). <http://www.lngamerica.com/lng-facts.html>. Retrieved from <http://www.lngamerica.com>: <http://www.lngamerica.com/lng-facts.html>
- Sandmeyer Steel Company. (2016). <https://www.sandmeyersteel.com/ssc-invar36.html>. Retrieved from <https://www.sandmeyersteel.com>: <https://www.sandmeyersteel.com>
- Stroud, K. A. (1993). *Engineering Mathematics for Technicians*. Coventry: Palgrave and MacMillan.
- Syahrudin, N. (2011). *LNG Loading Lines Surge Analysis*. Hong Kong: IMECS.
- Woolger, C. (1996). Materials World. In C. Woolger, *Materials World* (pp. 332-33). London: The Institute of Materials, Minerals and Mining. Retrieved from http://www.nickel-alloys.net/invar_nickel_iron_alloy.html#_Physical_properties: http://www.nickel-alloys.net/invar_nickel_iron_alloy.html#_Physical_properties

Appendices

Complete Code

```

clc
clear all

%%%Liquid Properties
Fluid = ('LNG');
rho = input('Fluid density in kg/m3 = ');
K = input('Bulk modulus in N/m2 = ');
Vis = input('Dynamic Viscosity of fluid in kg/ms = ');
Liquidpress = input('Gauge pressure of fluid in N/m2 = ');

%%%Pipe Properties
D = input('Pipe diameter in meters = ');
Wt = input('Pipe wall thickness in meters = ');
E = input('Pipe materials Youngs modulus in N/m2 = ');
L = input('Pipe length in meters = ');
Rough = input('Pipe internal Absolute surface roughness in meters = ');
while Rough <=0
    Rough = input('Pipe internal Absolute surface roughness in meters = ');
end
Area = (pi*(D/2)^2);

%%%Valve Properties
Cl = input('Valve closure time in seconds = ');
%%%Reservoir properties
Z = input('Reservoir height in meters = ');
%%%Gravitational Acceleration
g = input('Gravitational acceleration in m/s2 = ');
%%%Sonic Waves Speed Within Pipe c
C = sqrt(1/(rho*((1/K)+(D/(E*Wt)))));
%%%Piezometric Head Within Pipe
H = ((Liquidpress/(rho*g))+Z);
%%%Grid Spacing
X = input('Input Grid spacing in x direction = ');
while (X<=0) || (X>L) || rem(L,X)~=0
    X = ('Input Grid spacing in x direction = ');
end
%%%Time Step
Ts = input ('Input Time step in seconds = ');
while (((Ts*C)/X)>1) || (((Ts*C)/X)<=0)
    Ts = input ('Input Time step in seconds = ');
end

%%%Initial conditions at time 0
%%%Flow rate
Q = input ('Input Volumetric flow rate in m3/s = ');
%%%Fluid dynamic velocity
Fv = (Q/Area);
%%%Reynolds Number of Fluid
Re = (((rho)*(Fv)*(D))/(Vis));
%%%Nodes within pipe
Np=L/X+1;
%%%Graph X axis
Xaxis = linspace (0,L,Np);

```

Computational Modelling of Hydraulic Transient

```

%%Hydraulic Impedence
Hyd = (sqrt(rho*K))/Area;
%%Maximum Surge Pressure of Fluid
Hmax = ((Hyd*Q)*0.0001019977334)*3)+H;

%%Colebrooke-White equation for friction coefficient (fc)
while Re < 4000
    disp('Fluid is not turbulent')
    break
end
dummy_variable=10;
fc = 0.00001;
while dummy_variable>1
    A = 1/sqrt(fc);
    B = -2*log10((Rough/(3.71*D))+(2.51/(Re*sqrt(fc))));
    if abs(A-B)<0.01 || fc>1
        dummy_variable=0;
    end
    fc=fc+0.00001;
end
%%Pipe-resistance
Res = (fc/(2*D*Area));
%%Darcy Weisbach Equation for Head Loss Due to Friction if Re>4000
Headloss=((fc)*(L/D)*((Fv^2)/(2*g)));
%%Pipe Flow Rate Vector
PVQ=repmat(Q,1,Np);
%%Pipe Head Pressure Vector
PVH=repmat(H,1,Np);
for i = 1:Np
    PVH(1,i) = H-(i-1)*((fc*L)/(2*g*D*(L/X)*Area^2));
end
PVH1=PVH;

%% Boundary Condition at Reservoir Node
Hvec(1,1) = PVH(1,1);
Qvec(1,1) = PVQ(1,2)+((g*Area)/C)*(PVH(1,1)-PVH(1,2))-
Res*Ts*PVQ(1,2)*abs(PVQ(1,2));
%% Boundary Condition at Valve Node
Hvec(1,Np) = PVH(1,Np-1)-(C/(g*Area))*((PVQ(1,Np))-PVQ(1,Np))-
(C/(g*Area))*Res*Ts*PVQ(1,Np-1)*abs(PVQ(1,Np-1));
Qvec(1,Np) = Q;

%%Populate Flow Vector With Q
Qvec(1,2:Np-1) = Q;
%%Time taken
Time = (L/C);
%%Number of Iterations
Iter = (Time/Ts);
tmax=(L/C);
t = 0;
HP = PVH;
Flow = Qvec;
while t < Iter
    t=t+1;
    for c = 2:Np-1
        Qvec(1,c) = 0.5*(Flow(1,c-1)+(Flow(1,c+1))-
0.5*(g*Area)*(Ts/X)*(PVH(1,c+1)-PVH(1,c-1))-
Res*Ts*(0.5*(Flow(1,c+1)+Flow(1,c-1)))*abs(0.5*(Flow(1,c+1)+Flow(1,c-1))));
        HP(1,c) = 0.5*(PVH(1,c-1)+PVH(1,c+1))-
0.5*(Ts/X)*(C^2/(g*Area))*((Flow(1,c+1))-Flow(1,c-1));
    end
end

```

Computational Modelling of Hydraulic Transient

```

    HP(1,1) = H;
    Qvec(1,1) = Qvec(1,2) + ((g*Area)/C) * (HP(1,1) - HP(1,2)) -
    Res*Ts*Qvec(1,2)*abs(Qvec(1,2));

    HP(1,Np) = HP(1,Np-1) - (C/(g*Area)) * ((Qvec(1,Np)) - Qvec(1,Np-1)) -
    (C/(g*Area)) * Res*Ts*Qvec(1,Np-1)*abs(Qvec(1,Np-1));
    if (t+Ts)<Cl
        Qvec(1,Np) = Q*(1 - ((t+Ts)/Cl));
    else
        Qvec(1,Np) = 0;
    end
    t=(t+Ts);
    Flow=Qvec;
    PVH=HP;

    figure(1)
    subplot(2,1,1);
    plot(Xaxis,HP,'b')
    axis([0 L 0 Hmax])
    title('Piezometric Head Pressure')
    pause(0.0000001)

    subplot(2,1,2);
    plot(Xaxis,Qvec,'b')
    axis([0 L -(Q/10) Q+0.025])
    title('Volumetric Flowrate')
    pause(0.0000001)

end

```