



Investigation and implementation of a 1-D water hammer numerical model with liquid column separation

D Hitchcock

 **orcid.org/ 0000-0001-5700-3649**

Dissertation accepted in fulfilment of the requirements for the degree *Master of Engineering in Mechanical Engineering* at the North-West University

Supervisor:	Dr J Kruger
Co-supervisor:	Mr GM Benson
Assistant Supervisor	Mr AJK van der Walt

Graduation:	June 2025
-------------	-----------

ACKNOWLEDGEMENTS

First, I want to thank our Heavenly Father, God in the Trinity, for blessing me with this talent and the opportunities to live it every day. He provides me with hope, strength, and peace to face any challenge life throws my way. All glory to Him, His wisdom, and His presence.

“If I have seen further than others, it is by standing upon the shoulders of giants” – Sir Isaac Newton. I would like to thank the following giants in my life, whose support throughout this Master’s was much appreciated:

- My study leaders, Dr J Kruger, Mr GM Benson, and Mr AJK van der Walt, for their guidance and support throughout this dissertation. Thank you for your time and commitment.
- My colleague and mentor, Dawie Viljoen, thank you for all your advice, both professionally and personally. You are an inspiration to any young engineer, and I am thankful for having the opportunity to have worked alongside you.
- My friends, Marinus, Nicolené, Francois, Peter, Daniella, Paul, and Rahul, for their support, words of encouragement and laughs shared. Your friendship is cherished and priceless.
- My best friend, Wihan. Since we were 15 years old, we have shared our dreams and life goals. Thank you for always being there with me through the highs and lows and supporting me throughout this journey from high school to my master's degree (and all that will follow).
- My extended family, Oom Swannie, Tannie Petro, Anja, and Jana, for their support and words of encouragement. Anja and Jana, thank you for giving me a break from my studies with your contagious laughter and inviting personalities.
- My family, my mother Odette, sister Carmen, grandmother Sarie, uncle Stephan Sr., aunt Linna, and cousin Stephan Jr., for always believing in me every step of the way. For making it possible for me to pursue my dreams and ambitions and always encouraging me to be my best, try my hardest and never quit. You are all the cornerstone of the person I am today, and I deeply thank you all for that.
- My partner, Ria, for all your love, support, words of encouragement, patience, and sharing my highs and lows throughout life and throughout this master’s degree. Thank you for believing in my dreams, goals and ambitions, and for inspiring me to pursue them. Words are not enough to express my gratitude towards you.

This master’s degree is dedicated in the loving memory of my mentor, father figure, hero, and inspiration, Stephan Nel Sr.

ABSTRACT

Over the years, many advances have been made in the numerical prediction and simulation of hydraulic transients. This is especially true for water hammer with liquid column separation. The open literature presents numerous research articles, numerical models, and experiments on hydraulic transients. However, even with these advancements and mathematical tools, water hammer with liquid column separation are still hydraulic transients often overlooked by design and process engineers. This may lead to equipment damage and downtime of plants and catastrophic, and sometimes even fatal, consequences. The hydraulic transient causes high-pressure pulses in the pipe system much higher than the operating and water hammer pressures.

The open literature showed that the most popular numerical model for predicting and simulating water hammer with liquid column separation is the discrete gas cavity model (DGCM). Unfortunately, the open literature lacks information on implementing the DGCM model, especially regarding the free gas volume weighing factor's effect on the grid and time step independence and the proper value for the weighing factor when conducting simulations. Many researchers present conflicting proposed values for the weighing factor in the literature.

This study aims to address the gap in the literature by implementing the DGCM model into the EES software package, conducting a sensitivity analysis of the weighing factor, and comparing experimental results with simulation results for various weighing factor values. The sensitivity analysis of the weighing factor gives insight into the effect of different weighing factors on the grid and time step independence of the DGCM numerical model. The purpose of the comparison of the experimental and simulation results is to establish whether a baseline value for the weighing factor exists or if the weighing factor is system-dependent.

The sensitivity analysis results revealed that as the weighing factor value increased from 0.5 to 1, the grid and time step size for independence increased. This led to the conclusion that for higher values of the weighing factor, grid and time step independence was reached at a larger grid and time step size. If a weighing factor of 1 was used, the larger grid and time step size led to a shorter simulation time.

The comparison between the experimental and simulation results revealed that as the weighing factor value increased from 0.5 to 1, better comparisons were obtained with the experimental results, and numerical stability increased. In conjunction with the sensitivity analysis results, this

led to the conclusion that a weighing factor of 1 is strongly recommended for lower simulation time, higher numerical stability, and improved accuracy.

Key terms: hydraulic transient, liquid column separation, discrete gas cavity model (DGCM), free gas volume weighing factor, method of characteristic (MOC), CFD grid and time step independence, cavitation

OPSOMMING

Oor die jare is aansienlike vordering gemaak in the numeriese voorspelling en simulاسie van hidrouliese oorgange. Dit is veral waar vir waterhamer met vloeistofkolomskeiding. In die literatuur word 'n oorvloed van navorsingsartikels, numeriese modelle en eksperimente oor waterhamer met vloeistofkolomskeiding aangebied. Selfs met hierdie vooruitgang en wiskundige gereedskap tot ons beskikking, is waterhamer met vloeistofkolomskeiding egter steeds 'n hidrouliese oorgang wat dikwels deur ontwerp- en prosesingenieurs oor die hoof gesien word. Dit lei nie net tot skade aan toerusting en die onderbreking van aanlegte nie, maar kan ook katastrofiese, en soms noodlottige gevolge hê. Hidrouliese oorgang lei tot hoëdrukpulse in die pypstel. Hierdie hoëdrukpulse veroorsaak dat the bedryfdruk en selfs die waterhamer druk vêr oorskry word.

Uit die literatuur het dit duidelik geword dat die gewildste numeriese model wat gebruik word om waterhamer met vloeistofkolomskeiding te voorspel en te simuleer, die diskrete gasholte model (DGCM) is. Ongelukkig is daar 'n tekort in die oop literatuur aan inligting wat handel oor die implementering van die DGCM model, veral in terme van die effek van die vrye gasvolume weegfaktor op die rooster en tydstep onafhanklikheid, sowel as die korrekte waarde om vir die weegfaktor te gebruik wanneer simulاسies uitgevoer word. In die literatuur gee verskeie navorsers egter teenstrydige voorgestelde waardes vir die weegfaktor.

Hierdie studie het ten doel om die gaping in the literatuur aan te spreek deur die DGCM model in EES te implementeer, 'n sensitiwiteitsanalise van die weegfaktor uit te voer, en eksperimentele resultate met simulاسie resultate vir verskeie weegfaktorwaardes te vergelyk. Die doel van die sensitiwiteitsanalise van die weegfaktor is om insig te bied rakende die effek van die weegfaktor op die rooster en tydstep onafhanklikheid. Die doel van die vergelyking tussen eksperimentele en simulاسie resultate is om vas te stel of 'n basislynwaarde vir die weegfaktor bestaan en of die weegfaktor stelsel afhanklik is.

Die sensitiwiteitsanalise het aan die lig gebring dat namate die weegfaktor waarde van 0.5 tot 1 toeneem, die rooster en tydstep grootte vir onafhanklikheid ook toeneem. Dit lei tot die gevolgtrekking dat vir hoër waardes van die weegfaktor, rooster en tydstep onafhanklikheid verkry word op 'n groter rooster en tydstep grootte. Die gevolg hiervan is dat indien 'n weegfaktor van 1 gebruik word, die groter rooster en tydstep grootte tot 'n korter simulاسietyd lei.

Die eksperimentele en simulاسie resultate vergelyking het aan die lig gebring dat namate die weegfaktorwaarde van 0.5 na 1 toeneem, beter vergelykings met die eksperimentele resultate

verkry word, asook verhoogde numeriese stabiliteit. Dit, met die resultate van die sensitiviteitsanalise in ag geneem, lei tot die gevolgtrekking dat n weegfaktor waarde van 1 sterk aanbeveel word vir die DGCM vir laer simulasietyd, beter numeriese stabiliteit, en verhoogde akkuraatheid.

Sleuteltermes: hidrouliese oorgang, vloeistofkolomskeiding, diskrete gasholte model (DGCM), vrye gasvolume weegfaktor, metode van karakteristieke (MOC), CFD-rooster en tydstep-onafhanklikheid, kavitasie

LIST OF ABBREVIATIONS

1-D	One dimensional
3-D	Three dimensional
CFD	Computational fluid dynamics
CIS	Condensate-induced shock
DVCM	Discrete vapour cavity model
DGCM	Discrete gas cavity model
EES	Engineering Equation Solver
FSI	Fluid-structure interaction
CGI	Grid convergence index
GIVCM	Generalised interface vaporous cavitation model
HGL	Hydraulic grade line
IBC	Intermediate bulk container
LNG	Liquefied natural gas
MOC	Method of characteristics
NRV	Non-return valve
NWU	North-West University
ODE	Ordinary differential equation
PDE	Partial differential equation
PT	Pressure transducer
USA	United States of America

LIST OF SYMBOLS

α_0	Initial gas void fraction
α_v	Void fraction
Δ	Difference
Δt	Time step in algebraic form
Δx	Difference in axial direction
θ	Pipe angle from horizontal
ξ	MOC x -intercept of the curve
ρ	Fluid density
ρ_m	Density of vaporous mixture
τ_w	Wall shear stress
ψ	Relaxation/weighing factor
ω	Angular frequency of the system
a	Speed of sound
A	Cross-sectional pipe area
A_m	Hydraulic pipe area
a_s	Shock wave speed,
C	a Constant
C_1	Poisson ratio coefficient
D	Hydraulic pipe diameter
D_i	Pipe inner hydraulic diameter
e	Pipe wall thickness
E	Young's modulus of the pipe material
f	Friction factor
F_f	Friction force (from the O-ring) and
F_g	Force of gravity

$f_{h=0}$	Theoretical value of the numerical model simulation
F_p	Force of water pressure
F_s	Force of the spring
g	Gravitational constant
h	Grid spacing
H	Pressure head
$H.O.T$	Higher order terms
H_s	Liquid side hydraulic grade line
H_{sv}	Vaporous mixture hydraulic grade line.
i	Computational section
J	Friction term
k	Spring constant
K	Fluid bulk modulus
L	Length of the pipe
m	Mass
n	Amount of substance
p	Order of convergence
P	Pressure
P_{g0}^*	Reference pressure
P_g^*	Absolute gas pressure
Q	Volumetric flow rate
Q_u	Upstream discharge/volumetric flow rate
Q_d	Downstream discharge/volumetric flow rate
r	Refinement ratio
R	Gas constant and
t	Current time step
T	Temperature

T_c	Valve closure time
t_{in}	Time of cavitation inception
V	Flow velocity
V_0	Initial flow velocity
V_m	Velocity of the vaporous mixture
V_m^i	Inception velocity of the vaporous mixture
V_m^t	Terminal velocity of the vaporous mixture
$\forall$	Volume
$\forall_g$	Gas cavity volume
$\forall_m$	Computational cell volume
$\forall_{vap}$	Vapour cavity volume
w_t	Pipe wall thickness
x	Position in the axial direction
x_0	Initial length of the spring

Table of Contents

ACKNOWLEDGEMENTS	I
ABSTRACT	II
OPSOMMING	IV
LIST OF ABBREVIATIONS	VI
LIST OF SYMBOLS.....	VII
CHAPTER 1 INTRODUCTION	20
1.1 Introduction and background	20
1.2 Problem statement	24
1.3 Research questions.....	25
1.4 Research aims and objectives.....	25
1.4.1 Research aim	25
1.4.2 Research objectives	25
1.5 Chapter layout	26
CHAPTER 2 LITERATURE REVIEW.....	28
2.1 Introduction	28
2.2 Water hammer and liquid column separation.....	29
2.3 Vaporous and gaseous cavitation.....	33
2.4 Other types of hydraulic transients.....	35
2.4.1 Pulsating flow	35

2.4.2	Condensate hammer	35
2.4.3	Reverse flow hammer	36
2.4.4	Two-phase flow	36
2.4.5	Geysering	37
2.5	Need for modelling of water hammer with liquid column separation	39
2.6	Advances in the field.....	41
2.7	Popular numerical models	43
2.7.1	The discrete vapour cavity model (DVCM).....	43
2.7.2	The discrete gas cavity model (DGCM)	49
2.7.3	The generalised interface vaporous cavitation model (GIVCM)	54
2.8	Summary of past experiments on the DVCM, DGCM and GIVCM	60
2.9	Research voids and uncertainties.....	63
2.10	Conclusion.....	63
CHAPTER 3	THEORY AND MODEL IMPLEMENTATION	66
3.1	Introduction	66
3.2	Numerical model theory.....	67
3.2.1	Method of characteristics (MOC)	67
3.2.2	The classic water hammer equations.....	71
3.2.3	The discrete gas cavity model (DGCM)	72
3.2.4	Time step and grid size independence theory.....	76
3.3	Theory implementation verification and validation.....	79
3.3.1	Case 1: Pure water hammer case.....	80

3.3.2	Case 2: Low-flow water hammer with liquid column separation case.....	87
3.3.3	Case 3: High-flow water hammer with liquid column separation case	91
3.4	Conclusion.....	94
CHAPTER 4	EXPERIMENTAL METHODOLOGY AND DESIGN	96
4.1	Introduction	96
4.2	Experimental methodology.....	97
4.3	Experimental design.....	98
4.3.1	Fast closing valve design.....	99
4.3.2	Measuring equipment	105
4.4	Conclusion.....	112
CHAPTER 5	RESULTS COMPARISON	113
5.1	Introduction	113
5.2	Input parameters measurements and calculations	114
5.2.1	Flowrate measurement	114
5.2.2	Reservoir pressure measurement.....	114
5.2.3	Atmospheric and bulk water temperature measurement	115
5.2.4	Barometric pressure	116
5.2.5	Valve closure time calculation.....	117
5.2.6	Summary of EES model input parameters	118
5.3	Sensitivity analysis (effect of weighing factor) results	120
5.4	Experimental results and simulation results comparison	121
5.4.1	Experimental results findings	121

5.4.2	Experimental results and EES simulation results comparison.....	126
5.5	Conclusion.....	136
CHAPTER 6 CONCLUSIONS AND RECOMMENDATIONS.....		140
6.1	Introduction	140
6.2	Summary of the study and main findings.....	140
6.3	Contribution to the field of study	142
6.4	Limitations and areas of further research	143
6.4.1	Limitations and improvements of this research effort	143
6.4.2	Areas of further research	144
6.5	Concluding remarks	144
BIBLIOGRAPHY.....		146
ANNEXURE A: DGCM EES CODE		153
ANNEXURE B: SPRING-MASS MATHEMATICAL MODEL		166
ANNEXURE C: FAST-CLOSING VALVE DESIGN.....		169
ANNEXURE D: CALIBRATION CERTIFICATE.....		180
ANNEXURE E: TIEPIE OSCILLOSCOPE DATASHEET		185
ANNEXURE F: RESISTOR DATASHEET		186

LIST OF TABLES

Table 2.1: Calculated DVCM values: $V_0 = 0.3$ m/s from [50].	48
Table 3.1: Classic water hammer equations, Eq. 3.13 and Eq. 3.14, variable description.	71
Table 3.2: Wave speed equation, Eq. 3.15, variable description.	72
Table 3.3: Ideal gas equation, Eq. 3.13, variable description.	74
Table 3.4: Physical properties of the fluid (pure water), Case 1A.	81
Table 3.5: Material properties of the pipe (commercial steel), Case 1A.	81
Table 3.6: Physical properties of the fluid (pure water), Case 1B.	83
Table 3.7: Material properties of the pipe (commercial steel), Case 1B.	83
Table 3.8: Time step and grid size independency results, Case 1.	85
Table 3.9: Point difference (error) for Case 1A and Case 1B results comparison.	87
Table 3.10: Physical properties of the fluid (water), Case 2A.	88
Table 3.11: Material properties of the pipe (commercial copper), Case 2A.	88
Table 3.12: Point difference (error) for Case 2A and Case 2B results comparison.	91
Table 3.13: Point difference (error) for Case 3A and Case 3B results comparison.	93
Table 3.14: Grid and time step independence study results for the DGCM EES model (verification and validation cases).	95
Table 3.15: Verification and validation results summary of calculated point differences (errors).	95
Table 4.1: Calculated average spring constant.	102
Table 4.2: Point difference (error) calculated from reservoir pressure transducer calibration.	111
Table 5.1: Results of measured weight, calculated mass flow, and calculated flow rate.	114

Table 5.2: Results of measured reservoir pressure and calculated reservoir pressure head.....	115
Table 5.3: Results of measured atmospheric temperatures.	115
Table 5.4: Results of measured bulk water temperatures.	116
Table 5.5: Physical properties of general water at 14.12°C.	116
Table 5.6: Material properties of the pipe (galvanised steel).	116
Table 5.7: Barometric pressure sensitivity analysis results.	117
Table 5.8: EES model input parameters.	119
Table 5.9: Result comparison EES model grid refinement study results.	127
Table 5.10: Point difference (error) of results comparison ($\Psi = 0.5$).	128
Table 5.11: Point difference (error) of results comparison ($\Psi = 0.6$).	130
Table 5.12: Point difference (error) of results comparison ($\Psi = 0.7$).	131
Table 5.13: Point difference (error) of results comparison ($\Psi = 0.8$).	133
Table 5.14: Point difference (error) of results comparison ($\Psi = 0.9$).	134
Table 5.15: Point difference (error) of results comparison ($\Psi = 1.0$).	136
Table 5.16: Grid and time step size sensitivity analysis results summary.	137
Table 5.17: Point difference (error) summary of results comparisons.	138

LIST OF FIGURES

Figure 1-1: (a) Simple reservoir- valve system [5], (b) Simplified water hammer pressure – time trace [6].	21
Figure 1-2: Water Hammer with Liquid Column Separation Pressure – Time Trace [8].	22
Figure 1-3: Failed 12” pipe at the Theodore, AL Millard refrigeration plant [9].	23
Figure 2-1: Illustration of pressure and velocity of a water hammer event [14].	29
Figure 2-2: Illustration of Figure 2-1 on a pressure-time trace, adapted from [14].	31
Figure 2-3: Illustration of cavitation vapour “bubble” collapse [16].	32
Figure 2-4: Effect of water hammer and cavity collapse pressure pulse superpositioning [17].	33
Figure 2-5: Illustration of Local Vaporous Cavitation.	34
Figure 2-6: Illustration of Intermediate Vaporous Cavitation.	34
Figure 2-7: Illustration of Distributed Vaporous Cavitation (Bubble Flow) [21].	34
Figure 2-8: Illustration of condensate slug formation causing the condensate hammer [23]. ...	36
Figure 2-9: Illustration of basic two-phase flow regimes for horizontal pipes [24].	37
Figure 2-10: Haukadalsvegur geyser erupting in Iceland [25].	38
Figure 2-11: Pressure-time trace of pressure pulsations of geysering in geothermal well [26].	38
Figure 2-12: Schematic view of hydroelectric power plant [28].	39
Figure 2-13: Images from the 1950 Oigawa hydroelectric power plant accident [28].	40
Figure 2-14: Freeport Texas LNG plant accident 2022 [30].	41
Figure 2-15: Experimental test apparatus layout [50].	45

Figure 2-16: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 0.3$ m/s, $H_{T2} = 22$ m and $N = 32$ [50].	46
Figure 2-17: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 0.3$ m/s, $H_{T2} = 22$ m and $N = 128$ [50].	47
Figure 2-18: Comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 1.4$ m/s, $H_{T2} = 22$ m and $N = 32$ [50].	47
Figure 2-19: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 1.4$ m/s, $H_{T2} = 22$ m and $N = 128$ [50].	47
Figure 2-20: DGCM comparison of heads at $H_{v,1}$: (a) $V_0 = 0.3$ m/s (b) $V_0 = 1.4$ m/s, $H_{T2} = 22$ m [17].	51
Figure 2-21: Wave speed ratio as a function of the initial void fraction and absolute gas partial pressure [48].	53
Figure 2-22: Wave speed without limitation (left) and with limitation (right) on the initial gas void fraction ($a_{min} = 10^{-7}$) [55].	53
Figure 2-23: Two-phase flow equations control volume [58].	55
Figure 2-24: Shock equations control volume [58].	57
Figure 2-25: Experimental setup used by [3].	58
Figure 2-26: Experimental and calculated pressure head (GIVCM) [59].	59
Figure 3-1: A typical characteristic curve [12].	68
Figure 3-2: Discretization of a pipe into MOC staggered grid and rectangular grid, adapted from [51] and [59].	70
Figure 3-3: MOC characteristic lines in the x-t plane [52].	73
Figure 3-4: Benchmark Flownex® network, Case 1A.	82
Figure 3-5: Experimental setup used in [1], Case 1B.	84
Figure 3-6: Results Comparison between EES model and Flownex® Benchmark, Case 1A.	85

Figure 3-7: Results Comparison between EES model and experimental benchmark, Case 1B.	86
Figure 3-8: Experimental setup used by [17], Case 2B.	89
Figure 3-9: Experimental and DGCM model results ($V = 0.3 \text{ m/s}$) of [17].	90
Figure 3-10: Experimental and DGCM model results ($V = 1.4 \text{ m/s}$) of [17].	93
Figure 4-1: Overall experimental methodology.	97
Figure 4-2: P&ID of experimental test bench constructed at the NWU.	99
Figure 4-3: Designed valve closure over time.	101
Figure 4-4: Illustration of spring arrangement in fast closing valve.	101
Figure 4-5: MTS results for the three randomly selected springs.	102
Figure 4-6: Static pressure and total pressure measurement illustration [68].	103
Figure 4-7: Recommended static pressure wall probe design, $1.5 < L/d < 6$, [69].	104
Figure 4-8: Fast closing valve pressure transducer response time test bench.	105
Figure 4-9: Fast closing valve pressure transducer response time test results.	106
Figure 4-10: Experimental results of water hammer with liquid column separation [17].	106
Figure 4-11: TiePie oscilloscope measuring loop.	107
Figure 4-12: Load cell calibration, measured weight vs laboratory weight.	108
Figure 4-13: Load cell calibration, calculated point difference error.	109
Figure 4-14: Illustration of calibration procedure of the reservoir pressure transducer.	110
Figure 4-15: Reservoir pressure transducer calibration results.	111
Figure 5-1: Approximated valve closure time of the experimental test bench.	118
Figure 5-2: Sensitivity analysis on the effect of weighing factor results.	120
Figure 5-3: Preliminary experimental test run results.	122

Figure 5-4: Effects of entrapped air on pressure surges in a pipeline [73].	123
Figure 5-5: Spontaneous erratic behaviour in measured experimental results.	124
Figure 5-6: Illustration of the DOF of a pipe section [75].	125
Figure 5-7: Experimental results of FSI conducted by [76].	126
Figure 5-8: Comparison of experimental measurements with DGCM EES model results (Ψ = 0.5).	128
Figure 5-9: Comparison of experimental measurements with DGCM EES model results (Ψ = 0.6).	129
Figure 5-10: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.7$).	131
Figure 5-11: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.8$).	132
Figure 5-12: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.9$).	134
Figure 5-13: Comparison of experimental measurements with DGCM EES model results ($\Psi = 1.0$).	135
Figure 5-14: Point difference (error) summary of results comparisons.	139
Figure B - 1: Free body diagram of forces acting upon the fast-closing valve piston.	166

CHAPTER 1 INTRODUCTION

This chapter provides an introduction and background regarding water hammer with liquid column separation and the popular numerical models developed to simulate said hydraulic transient. It also gives context to the problem statement, research questions, aims, and objectives discussed in the section. This is followed by a summary of the chapter layout of this dissertation.

1.1 Introduction and background

Water hammer occurs when a liquid in motion is rapidly forced to change flow direction and flow velocity, or the flow is stopped abruptly [1]. This can be caused by the sudden closure of a valve, a sudden change in the slope of the pipe, bends in the pipe profile or an uncontrolled pump start/shutoff. If the flowing liquid is incompressible and has momentum, there is very little absorption of energy (only that by friction and wall expansion). Therefore, this sudden change in flow direction or stoppage of flow causes a high-pressure shockwave, which propagates through the piping system from the location of the disturbance [2].

The sequence of transient events that leads to the high-pressure water hammer shockwave is explained in [3] and [4] and can be summarised as follows: Figure 1-1 (a) below displays a simple reservoir-horizontal pipe-valve system. Initially, the pressure or hydraulic grade line (HGL) is constant. Fast closure of the valve at the end of the pipe causes a sudden rise in pressure at the valve. Referring to the simplified water hammer pressure time trace, in Figure 1-1 (b) below, this high-pressure wave propagates from the valve to the reservoir, where it reflects negatively and then propagates back to the valve. At the valve, the high-pressure wave will reflect positively from the valve and propagate back to the reservoir. The wave will reflect negatively again from the reservoir and propagate back to the valve.

The pressure wave caused by the water hammer will always reflect negatively off the reservoir and positively off the valve. This sequence of events will continue, causing a rapid fluctuation in pressure until the high-pressure wave dampens out and stabilizes.

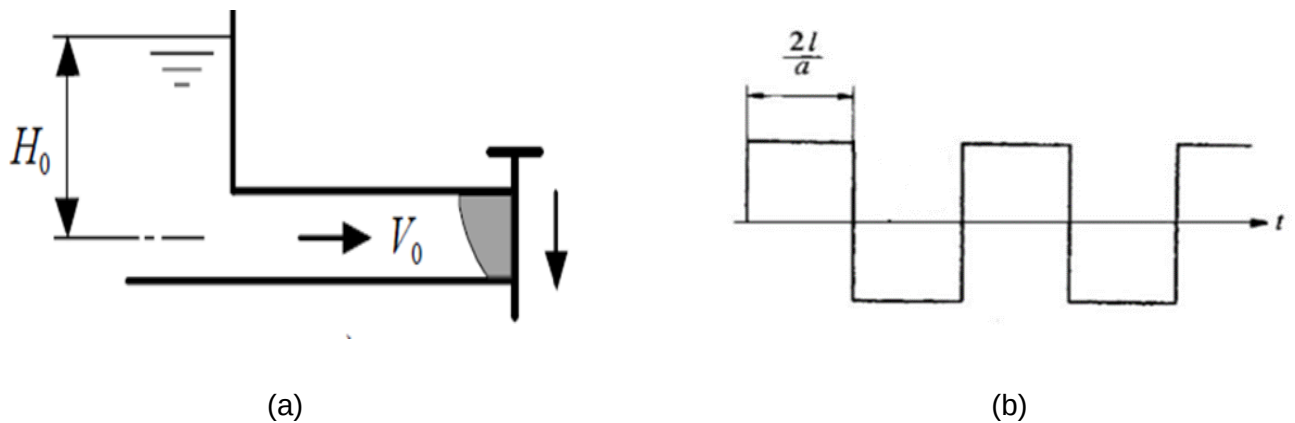


Figure 1-1: (a) Simple reservoir- valve system [5], (b) Simplified water hammer pressure – time trace [6].

A fluid transient commonly observed with water hammer is liquid column separation. Liquid column separation occurs when the negative pressure wave returns to the valve from the reservoir, and the liquid starts to flow towards the reservoir. When the local static pressure head drops to or below the vapour pressure head of the fluid a vapour cavity will start to form at the valve [7]. Since the flow velocity towards the reservoir is not immediately reduced to zero when the water hammer pressure wave reflects from the valve, the liquid detaches from the valve, and the cavitation-induced cavity grows, as indicated by the grey area next to the valve in Figure 1-1 (a). This phenomenon is known as liquid column separation.

With the water hammer pressure wave continuously reflecting off the vapour cavity, the velocity of the flow towards the reservoir will start to decrease. Eventually, the velocity will become positive (towards the valve). This causes the cavity to shrink and to collapse. The collapsing cavity causes a secondary pressure wave in the system which is less than that of the water hammer pressure wave. However, due to the superpositioning of both pressure waves, that of the water hammer and that of the cavity collapse, occurring at the same time a short-duration pressure pulse is observed [8]. This combined pressure pulse is significantly larger than the pressure wave of the water hammer alone (see encircled area of Figure 1-2 below).

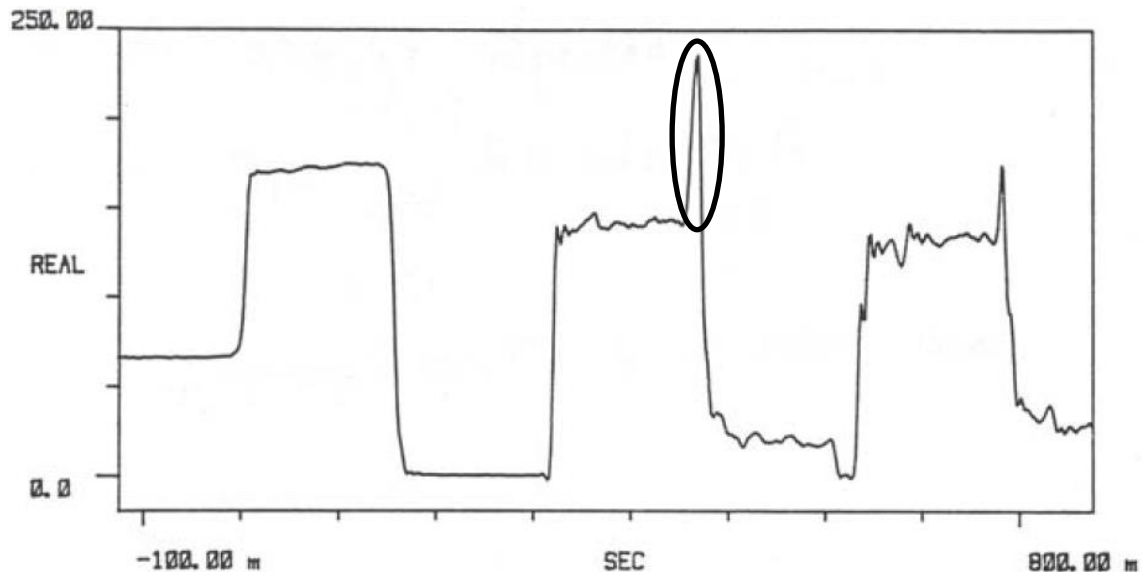


Figure 1-2: Water Hammer with Liquid Column Separation Pressure – Time Trace [8].

From Figure 1-2 it is observed that initially, only the high-pressure wave of the water hammer event is present. However, a short-duration pressure pulse is present at the second pressure peak. This short-duration pressure pulse is caused by the superpositioning of the water hammer and the cavity collapse pressure waves. The head of the short-duration pressure pulse is far higher than that of the original water hammer pressure wave. It should however be noted that this might not always be the case. For example, if the water hammer pressure wave was to arrive back to the valve at the moment of cavity collapse, the before-mentioned phenomena would not be observed [2].

From the above discussion, water hammer in general (including liquid column separation) is a transient hydraulic phenomenon that needs to be avoided as far as possible. This is because the pressure experienced by the system is far greater than the operating pressure of the system, see Figure 1-2. Failures caused by water hammer with liquid column separation can be catastrophic. Take for example the accident at the Millard refrigerated warehouse on August 23, 2010, in Theodore, AL [9]. A hydraulic shock caused a 12" pipe to fail, releasing 14.5 tonnes of anhydrous ammonia; see Figure 1-3 below. Around 152 offsite contractors and a ship crew near the incident reported ammonia exposure, while onsite roughly 8 million pounds of product were spoiled due to contamination [9].



Figure 1-3: Failed 12" pipe at the Theodore, AL Millard refrigeration plant [9].

Although the effects of water hammer with liquid column separation can be catastrophic, it is a hidden hazard seldom explored by design and process safety engineers, as pointed out by [10]. The effects of water hammer and liquid column separation can be mitigated in most cases. The author of [10] summarised various methods used in the industry to mitigate water hammer. These methods include eliminating gas/air accumulation in the system, installing pressure (surge) relief valves, installing anti-surge tanks in the system and reducing the rate of valve closure [10]. However, avoiding water hammer is not always possible, and in these cases, modelling this hydraulic transient is crucial to designing the pipeline system appropriately to avoid catastrophe.

Over the years, as technology advanced, many numerical models were developed to simulate water hammer with liquid column separation. These models can range from less accurate models, but easy to implement and solve to very accurate complex models. The open literature identified three different models to be widely used in the industry. These models are the discrete vapour cavity model (DVCM), the discrete gas cavity model (DGCM) and the generalized interface vapour cavity model (GIVCM). Of these three the DVCM is the simplest numerical model, whereas the GIVCM is the most complex. However, from the open literature, it is concluded that the DGCM numerical model is the most popular model used in the industry due to its balance between accuracy and complexity [11].

The DGCM numerical model is built upon principles from the DVCM, the main difference being that with the DGCM a permanent-infinitesimal gas cavity is added to each computational section. This allows the simulation to capture the column separation phenomena anywhere over the entire length of the pipe [12]. With the DGCM the discrete gas cavities replace the discrete vapour cavities from the DVCM. The DGCM is described by four sets of equations that are the classic water hammer equations, the gas cavity volume continuity equation and the ideal gas law [13]. To suppress the numerical oscillations a weighing factor/relaxation parameter (Ψ) is introduced to the discretised gas cavity volume continuity equation.

Considering all the above, this research effort will aim to address and answer the problem statement, research questions, aims, and objectives below.

1.2 Problem statement

When designing a pipeline system, one desires that the system can sustain the maximum pressure within it. Simply designing the strength of the pipeline system based on the operating pressure is not sufficient since pipelines are transient systems with many dynamic flow changes occurring frequently. When abrupt flow changes occur, they lead to transient flow events such as water hammer with liquid column separation.

The transient flow event mentioned above causes a pressure within the system that is far greater than that of the operating pressure and the pressure predicted by water hammer equations. The open literature concludes that the DGCM is the recommended numerical model to simulate water hammer with liquid column separation.

Although extensive research and experiments have been conducted on the DGCM in the past, little is presented in the open literature on implementing the numerical model. This is especially true regarding the free gas volume equation weighing factor (Ψ) value, and its effect on the grid and time step independence. The open literature also does not clarify whether the weighing factor is system-specific.

Therefore, the implementation of the DGCM needs to be investigated in terms of the free gas volume equation weighing factor and grid and time step independence.

1.3 Research questions

The primary research questions are whether a baseline exists for the DGCM's free gas volume weighing factor and what effect the weighing factor has on the grid and time step independence of the DGCM numerical model. The secondary research question is whether the said weighing factor depends on the system layout.

1.4 Research aims and objectives

1.4.1 Research aim

This research effort aims to investigate, verify, and validate the implementation of the DGCM water hammer with liquid column separation numerical model in terms of the free gas volume weighing factor and its effect on the grid and time step independence.

1.4.2 Research objectives

The objectives of this research effort are as follows:

- Implement the DGCM into Engineering Equation Solver (EES) software and verify and validate the EES model with published simulation and experimental data.
- Generate experimental data of water hammer with liquid column separation for a straight horizontal pipe system.
- Conduct a grid and time step independence study on the EES numerical model for various free gas volume weighing factors.
- Conduct a sensitivity analysis on the grid and time step independence study, determining the effect of the weighing factor.
- Simulate the experiment using a grid and time step independent EES numerical models for each of the various weighing factors.
- Compare the various weighing factor EES simulation results to the experimental results.
- Draw a conclusion based on the results and validate the primary and secondary research questions, see Section 1.3.

1.5 Chapter layout

This dissertation is divided into six chapters, the chapter layout is as follows:

Chapter 1: In Chapter 1, an introduction and background are given regarding water hammer with liquid column separation and the various numerical models developed to simulate said hydraulic transient, the most popular of which is the DGCM model. This provides context for the problem statement, followed by the research questions, research aims, and research objectives.

Chapter 2: In Chapter 2, in-depth background and introduction are given on water hammer with liquid column separation. Since liquid column separation is a cavitation-induced hydraulic transient, two types of cavitation are discussed: vaporous cavitation and gaseous cavitation. Other typical hydraulic transients are briefly discussed for the reader's interest, as it is not within the scope of this research. The need for modelling and simulating water hammer with liquid column separation is presented in this chapter to highlight the importance of using numerical models to predict water hammer and liquid column separation.

Furthermore, this chapter discusses the advances over the years within the field of water hammer and using numerical models for water hammer with liquid column separation. It highlights the three most popular modern numerical models used today, which are discussed in depth. A brief overview of past experiments from the open literature performed on the numerical models mentioned before is given. Lastly, this chapter discusses the research voids and uncertainties, sketching the background for the research problem and research questions.

Chapter 3: Chapter 3 presents the method of characteristics (MOC) theory as a take-off point, as the MOC is an integral part of the DGCM (and most water hammer numerical models). The classic water hammer equations derived for the MOC are given and discussed. This is followed by the discussion and derivation of the theory used for the DGCM model. After this, the theory used to calculate time step and grid size independence is discussed in depth. Lastly, the verification and validation process of implementing the DGCM numerical model into EES is outlined and discussed.

Chapter 4: In Chapter 4, the experimental methodology followed during this research effort to answer the research questions is outlined and discussed. This is followed by a discussion of the experimental test bench design, including the design of the fast-closing valve used in the bench and the various measuring instruments used and their calibration procedures.

Chapter 5: In Chapter 5, the comparisons of the results are discussed. First, the parameters measured and calculated from the experiment are presented. A discussion of the sensitivity analysis results on the effect of the weighing factor on the grid and time step independence of the DGCM numerical model follows this. Following are the key findings made from the experimental procedure, the comparison of the results between the simulation results for various weighing factors, and experimental results are presented and discussed.

Chapter 6: In Chapter 6, the conclusions and recommendations are presented. This includes a summary of the study with the main findings made from comparing the results. Following this is an overview of the contribution that this research effort makes to the field of study. After that, the limitations of this research effort are outlined, and improvements are recommended. Lastly, the areas for further research are provided, and the advantages thereof are discussed. Then, the final concluding remarks of this research effort are presented.

CHAPTER 2 LITERATURE REVIEW

This chapter provides an in-depth background and introduction to water hammer with liquid column separation. It discusses the types of cavitation and hydraulic transients, the need for modelling, popular numerical models, and past experiments. Lastly, it discusses the research voids and uncertainties and sketches the background of the research problem and questions.

2.1 Introduction

This chapter aims to provide an in-depth background on hydraulic transients, specifically water hammer with liquid column separation. It discusses the working principle of water hammer and, consequently, liquid column separation in detail. This is followed by discussing two types of cavitation that might occur during a liquid column separation event. The various other types of hydraulic transients are briefly discussed for the reader's interest.

Stressing the importance of modelling water hammer with liquid column separation, examples of industrial accidents caused by said hydraulic transient are presented. This is followed by an outline of the three most popular numerical models used for simulating said transient and a brief overview of past experiments conducted on the popular numerical models. Lastly, the research voids and uncertainties found in the open literature are discussed.

The outline of this chapter is as follows:

- Section 2.2: Provides an in-depth overview and discussion regarding water hammer, and water hammer with liquid column separation.
- Section 2.3: Discuss the two types of cavitation occurring with liquid columns separation hydraulic transients (vaporous and gaseous cavitation).
- Section 2.4: Provides a brief discussion on other types of hydraulic transients.
- Section 2.5: Stresses the need for modelling water hammer with liquid column separation modelling by providing an overview of industry-related accidents.
- Section 2.6: Outlines the advances made in the field of modelling hydraulic transient with the use of analytical and numerical methods.
- Section 2.7: Discusses the three most popular numerical models used in the industry today, that is, the DVCM, the DGCM, and the GIVCM.

- Section 2.8: Provides a brief discussion of the past experiments performed on the DVCM, DGCM and GIVCM numerical models.
- Section 2.9: This section summarises the research voids and uncertainties obtained from the open literature.

2.2 Water hammer and liquid column separation

As discussed in Section 1.1 above, water hammer occurs when an incompressible fluid in motion is either stopped or forced to change flow direction abruptly [1]. Since the fluid has momentum and little energy absorption is present, the sudden change in flow causes a pressure pulse to be generated within the piping system, from the location of the disturbance [2], known as water hammer.

Figure 2-1 below illustrates the pressure and velocity profile within a single-conduit frictionless pipeline following the sudden closure of a valve [14].

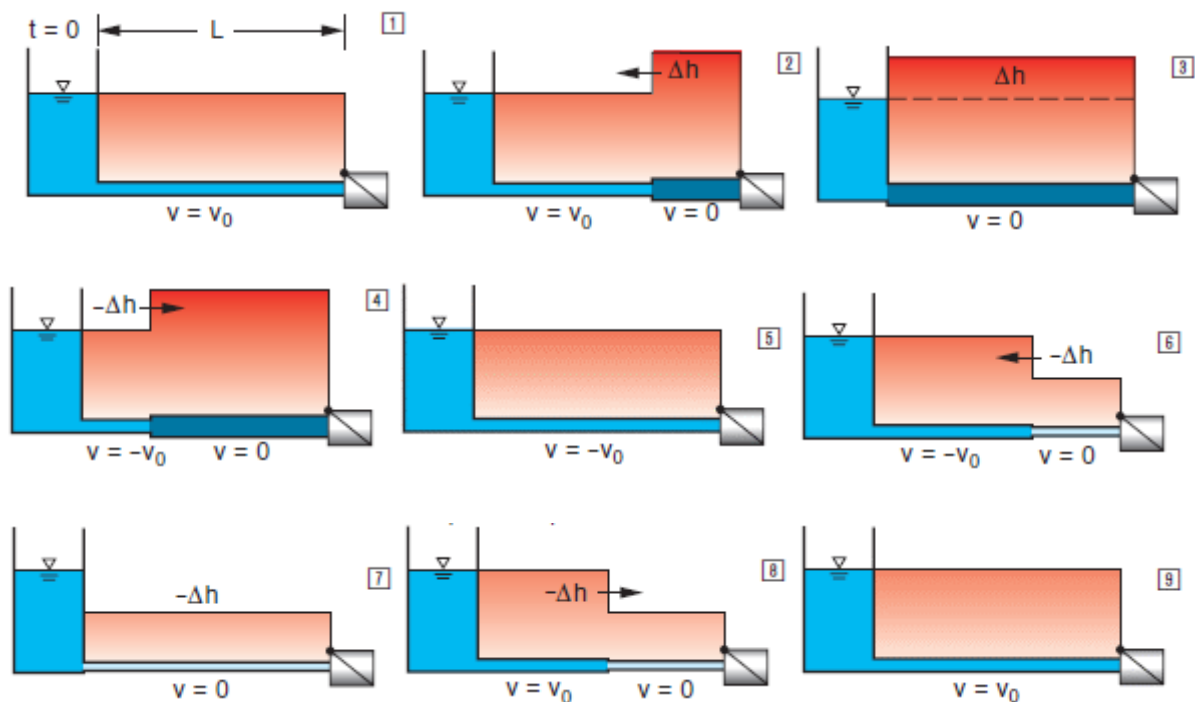


Figure 2-1: Illustration of pressure and velocity of a water hammer event [14].

Note that the steady-state pressure head is shaded medium orange, the increased pressure head is shaded dark orange, and the lower pressure head is shaded light orange. Additionally, the authors of [14] illustrate the expansion and contraction of the pipeline in Figure 2-1 above. Medium

blue represents the steady-state pipe, dark blue represents the expanded pipe, and light blue represents the contracted pipe.

The sequence of events during a water hammer event can be described, in Figure 2-1 above, as follows [14]:

1. Represents the steady-state conditions of the system; the pressure profile is constant, and the same as the reservoir head (due to no friction assumed) and the steady-state velocity is represented as V_0 .
2. The downstream valve is suddenly closed, which causes a high-pressure pulse Δh and the pipe to expand. The pressure pulse travels to the reservoir against the steady-state flow, at the speed of sound. This causes the velocity in the pipe, trailing the pressure pulse, to reduce to zero.
3. The pressure pulse arrives at the reservoir; however, since the reservoir pressure remains constant, an unbalanced condition occurs, causing the pressure pulse to reflect in the opposite direction of the reservoir. The flow velocity also changes direction at this point and flows to the reservoir.
4. The pressure pulse $-\Delta h$ travels to the downstream valve, whilst the flow velocity travels from the valve to the reservoir.
5. The pressure pulse $-\Delta h$ arrives at the closed downstream valve.
6. The arrival of the above-mentioned pressure pulse causes a sudden change in pressure. Since the valve is still closed, the pressure pulse travels upstream to the reservoir. This is accompanied by a change in flow velocity to zero and a contraction of the pipe.
7. The pressure pulse reaches the reservoir, and the pressure resumes the reservoir head.
8. The increased pressure (reservoir head) travels to the valve, and the flow velocity is restored in the direction from the reservoir to the valve.
9. The conditions are the same as described in the steady-state condition, and the whole process starts over.

The above-described sequence of events can be best illustrated using the pressure-time trace in Figure 2-2 below, adapted from [14].

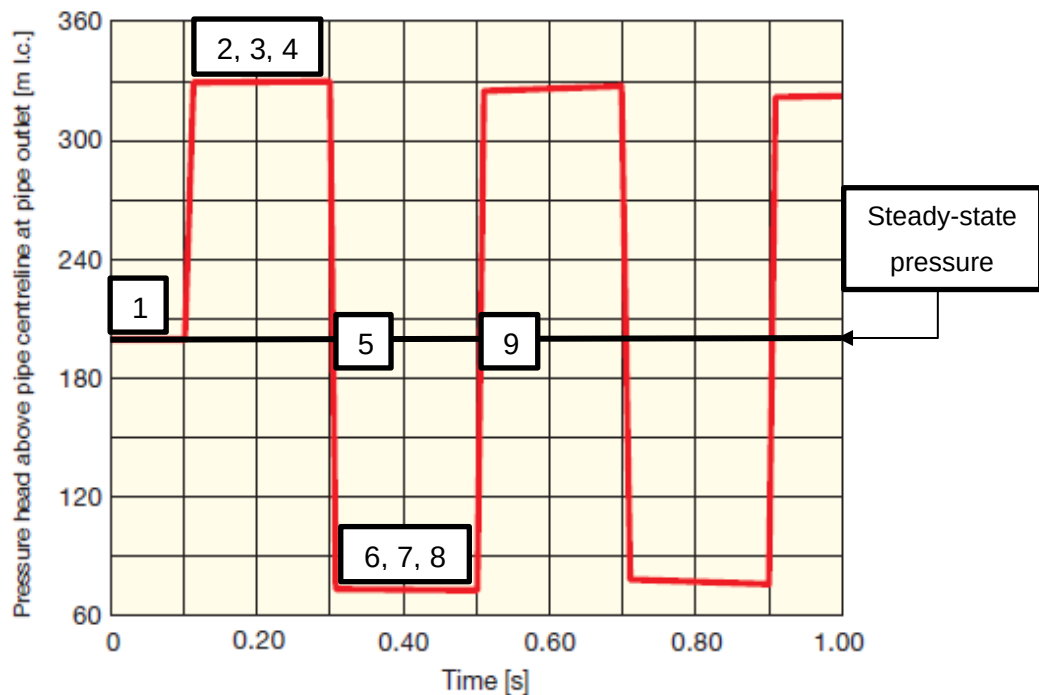


Figure 2-2: Illustration of Figure 2-1 on a pressure-time trace, adapted from [14].

The above-described scenario is only valid for a perfectly isolated system where no energy is lost or gained [14]. However, energy will be lost due to friction, heat, and vibration. Therefore, the sequence of events mentioned above will continue, causing a rapid fluctuation in pressure until the high-pressure pulse dampens out and stabilizes.

Figure 2-1 above, shows that initially, the flow velocity is from the reservoir to the downstream valve. However, the flow velocity changes direction during the transient, and flow occurs from the downstream valve to the reservoir. This sequence of events may lead to a common fluid transient event occurring with water hammer, namely liquid column separation (cavitation). Liquid column separation occurs when the local static pressure head drops to or below the vapour pressure head of the fluid [7]. The vapour pressure head can be defined as: “the pressure exerted by a vapour in thermodynamic equilibrium with its condensed phases at a given temperature in a closed system” [15].

When this happens, a vapour cavity will form at the valve. However, since the flow velocity is towards the reservoir, the cavity detaches from the valve, and a cavitation-induced cavity is formed [7]. When the flow velocity again changes direction, from the reservoir to the valve, the cavity will shrink and collapse due to the local static pressure increasing and surpassing the vapour pressure head of the fluid. This causes a secondary pressure pulse in the system [8].

Figure 2-3 below illustrates the collapse of a cavitation “bubble”.

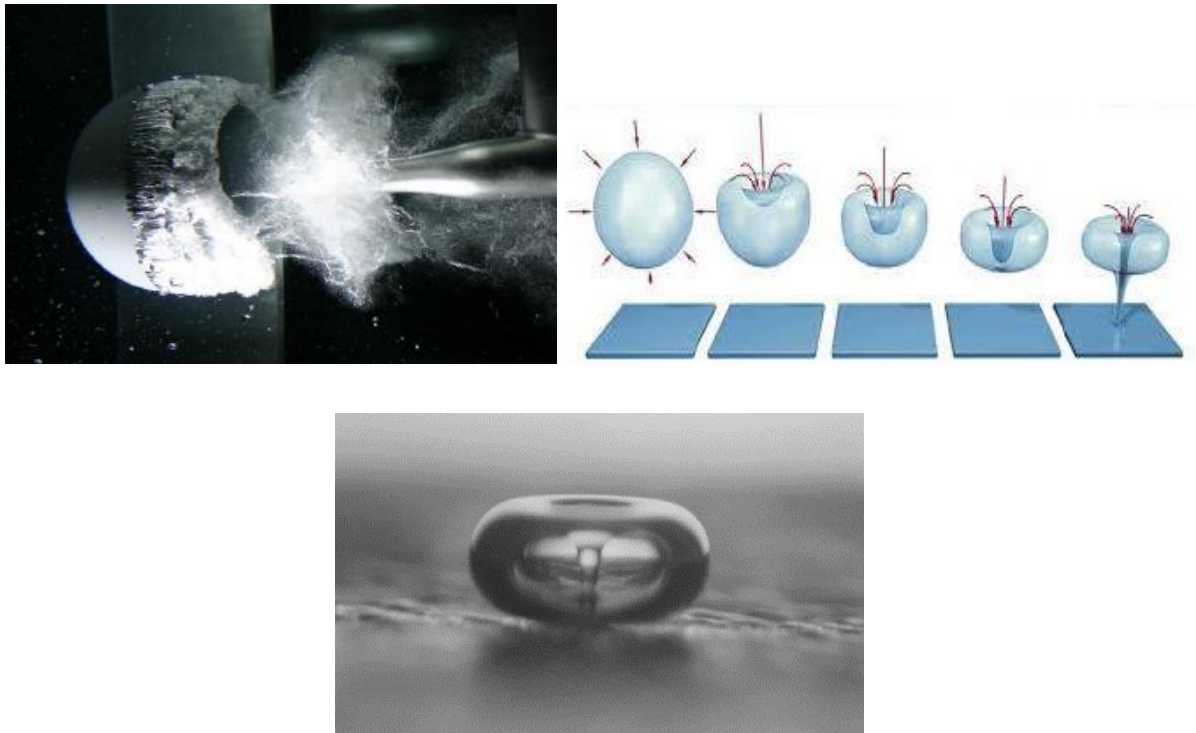


Figure 2-3: Illustration of cavitation vapour “bubble” collapse [16].

The process of cavity collapse occurs in a matter of microseconds (at the speed of sound). This produces shockwaves, noise, light emission, the pressure inside the imploding cavity of several 100 bars, and temperatures inside the imploding cavity reaching several thousand Kelvin [16]. However, these effects are local to the imploding cavity, as the pressure pulse generated by cavity collapse is less than that of the water hammer pressure pulse [8]. Under the right circumstances, the two pressure pulses, that of the water hammer and that of the liquid column separation can super-impose [2]. This causes a short-duration pressure pulse that is significantly larger than that of the water hammer pressure pulse.

The effect of superpositioning of the water hammer and cavity collapse (liquid column separation) pressure pulses can be seen from the experimental results of [17], in Figure 2-4 below:

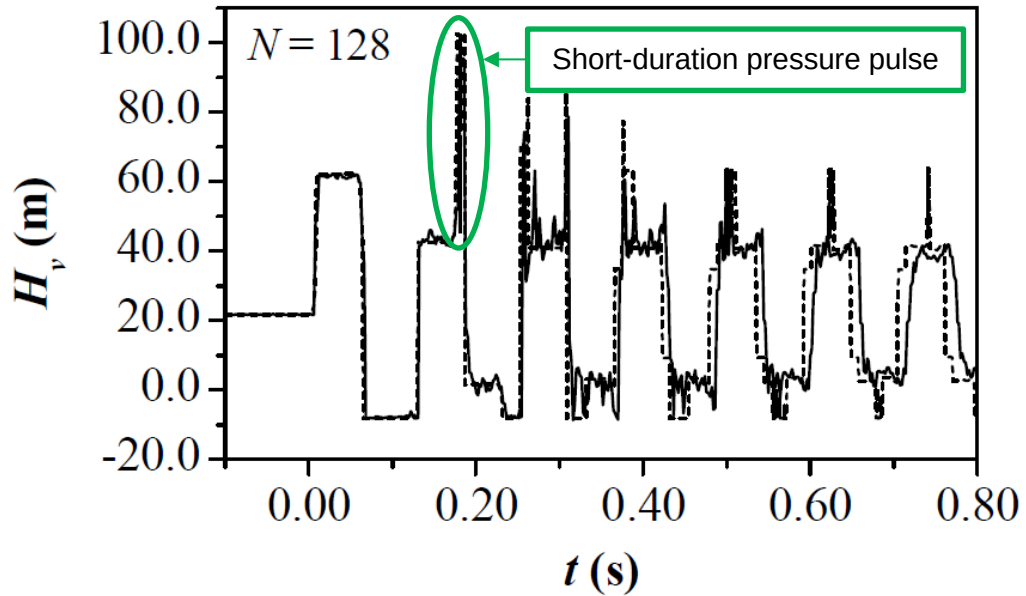


Figure 2-4: Effect of water hammer and cavity collapse pressure pulse superpositioning [17].

Figure 2-4 shows that the short-duration pressure pulse is far greater than that of only the water hammer pressure pulse, not to mention the system's steady-state (operating) pressure.

2.3 Vaporous and gaseous cavitation

From the above discussion, liquid column separation is caused by cavitation. Therefore, it is important to distinguish between the two types of cavitation that might occur during a water hammer with liquid column separation event.

The first type of cavitation is vaporous cavitation. Vaporous cavitation occurs when the local static pressure in the system drops below the liquid's vapour pressure. Once the static pressure is equal to or less than the liquid's vapour pressure, the liquid effectively starts to boil, causing a two-phase mixture (liquid and gas).

Vaporous cavitation can manifest itself in three primary forms within a pipeline system. The first of which is in the form of local vaporous cavitation. Common examples of local vaporous cavitation include cavity formation next to a valve, at a high point or a bend in the pipeline [18], i.e., where the water hammer originates.

An illustration of local vaporous cavitation can be seen in the following Figure 2-5 below:

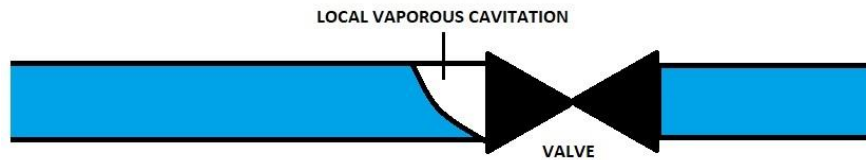


Figure 2-5: Illustration of Local Vaporous Cavitation.

The second form of cavitation is known as intermediate vaporous cavitation. Intermediate vaporous cavitation forms when two crossing low-pressure waves meet and superpositions anywhere along the length of the pipeline [19].

An illustration of intermediate vaporous cavitation can be seen in the following Figure 2-6 below.

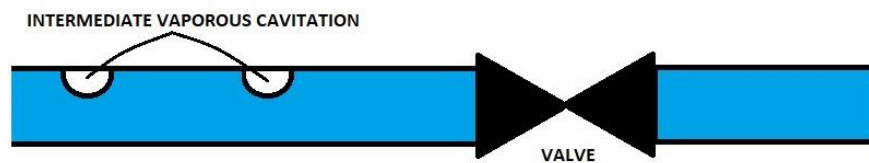


Figure 2-6: Illustration of Intermediate Vaporous Cavitation.

The third form of cavitation is distributed vaporous cavitation (bubble flow). Distributed vaporous cavitation forms when a negative pressure wave passes along the pipeline. This will cause the static pressure to decrease in the direction of wave propagation due to friction or the slope of the pipe. A variation in the velocity of the liquid between the two ends of the pipeline occurs, effectively causing the liquid to tear apart, which leads to a two-phase flow in the pipe [20].

An illustration of intermediate-distributed vaporous cavitation can be seen in the following Figure 2-7 below:

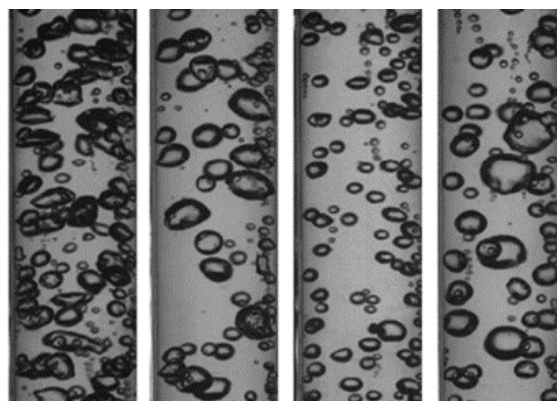


Figure 2-7: Illustration of Distributed Vaporous Cavitation (Bubble Flow) [21].

The second type of cavitation is gaseous cavitation. Gaseous cavitation occurs when the local static pressure in the system drops below the saturation pressure of the non-condensable gas dissolved within the liquid. Diffusion occurs once the local static pressure equals or is less than the saturation pressure of the non-condensable gas dissolved within the liquid. This diffusion process can be explained by Henry's Law which can be summarised as follows: gas dissolves into a fluid until equilibrium between the partial pressure of the gas and the pressure of the liquid is reached [15]. Liquids are notorious for their capability to absorb a certain amount of free air/gas. Therefore, a liquid can release the entrained gas once the saturation pressure of the said non-condensable gas is reached.

With vaporous cavitation, the vapour will almost immediately condense back into liquid form once the static pressure rises above the vapour pressure of the liquid. However, with gaseous cavitation, the released gas will not immediately dissolve back into the liquid. It is known that gas absorption is much slower than gas release [18]. The presence of free gas in the pipe system will reduce the speed of the pressure wave propagation, and therefore the subsequent pressure variations.

2.4 Other types of hydraulic transients

The concept of water hammer with liquid column separation is well known. However, it is important to note that this is not the only type of hydraulic transient phenomenon in fluid pipeline systems. Therefore, it is important to briefly mention the other prominent kinds of hydraulic transients that might occur.

2.4.1 Pulsating flow

According to [22] pulsating flow occurs due to the flow pulsations caused by reciprocating machinery, such as pumps and compressors. These pulsations cause vibrations within the piping system, resulting in undesirable results and may lead to equipment failure.

2.4.2 Condensate hammer

Condensate hammer, also called "steam hammer", is a hydraulic phenomenon that occurs when vapour is introduced to a condensate system or when condensation occurs in a pressurised vapour system [10]. Condensate slugs can form mainly in two ways (refer to Figure 2-8 below):

(1) first is when condensation fills the pipe's cross-sectional area, usually at a pipe trap, and (2) second is when the condensation partially fills the cross-sectional area of the pipe.

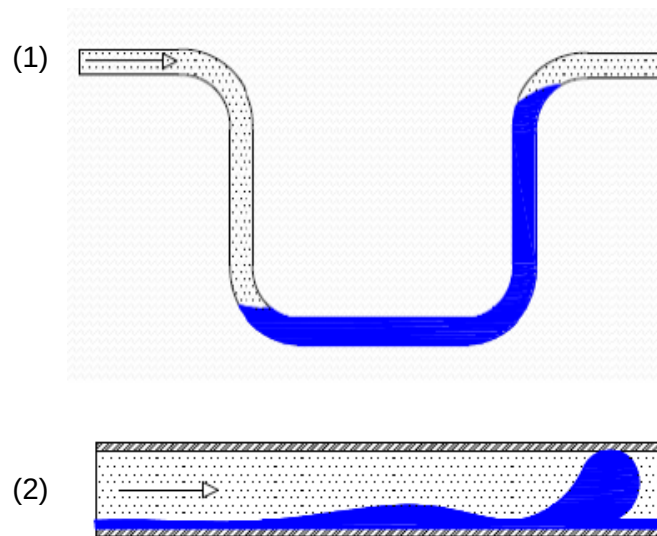


Figure 2-8: Illustration of condensate slug formation causing the condensate hammer [23].

The vapour in the system travels faster than the condensate. As the vapour reaches the condensate “slugs”, the condensate is provided with kinetic energy and is propelled throughout the system, commonly referred to as vapour-propelled slugs [23]. As the condensate is propelled through the system, any obstruction, such as valves or bends, causes the condensate to impact these obstructions, causing condensate-induced shock (CIS) [10].

2.4.3 Reverse flow hammer

Reverse flow hammer occurs when the rate of closure of a non-return valve (NRV) isn't sufficient [10]. This leads to flow reversing, in which the NRV shuts down, causing water hammer, as described in Section 2.1 above.

2.4.4 Two-phase flow

Two-phase flow refers to a piping system where both liquid and gas are present. It can take on many forms of flow regimes, as depicted by Figure 2-9 below.

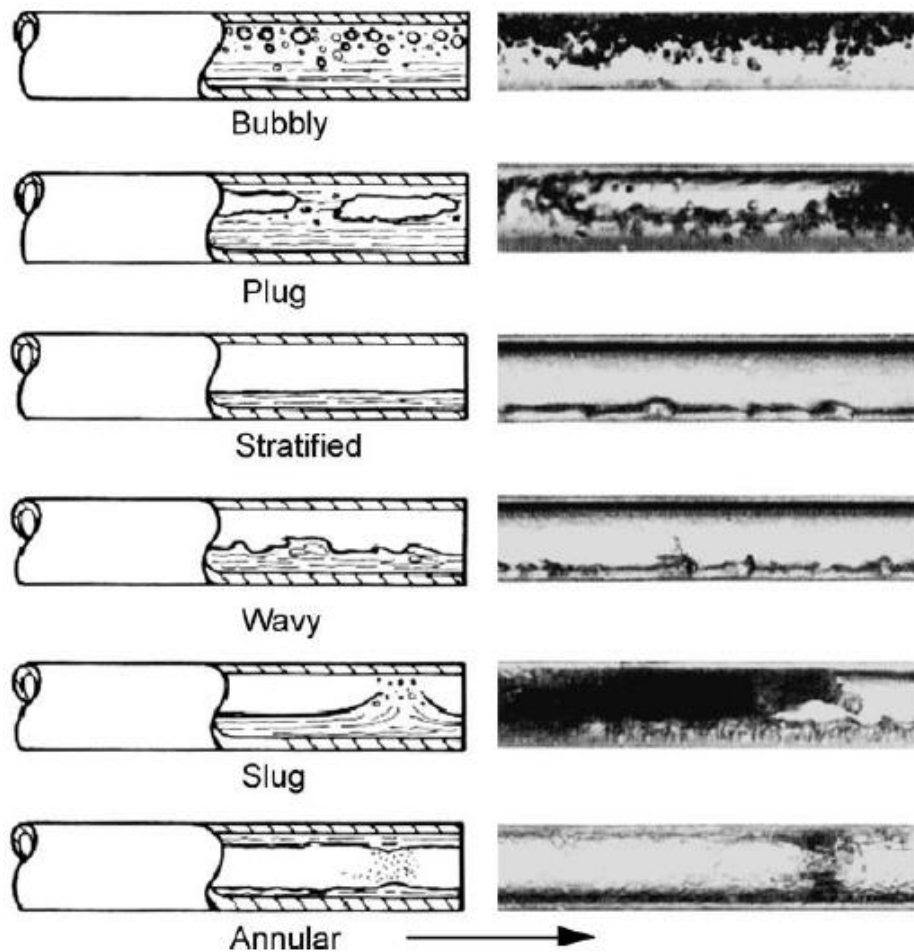


Figure 2-9: Illustration of basic two-phase flow regimes for horizontal pipes [24].

During a two-phase flow, the liquid and the gas may stratify, causing a temperature difference between the two phases. This leads to damaging pipe wall stresses, commonly referred to as “bowing” [10].

2.4.5 Geysering

According to [10] geysering is a hydraulic transient event that occurs in vertical or inclined pipes when the operating temperature of the liquid is at or near its boiling point. Rapid boiling causes the liquid to be propelled from the pipe, causing pressure pulses in the system.

A typical example of geysering is geothermal geysers seen in nature. Figure 2-10 below displays Iceland's Haukadalsvegur geothermal geyser erupting (Eysering).



Figure 2-10: Haukadalsvegur geyser erupting in Iceland [25].

Figure 2-11 below displays a pressure-time trace of the pressure pulsations at various depths caused by geysering in a geothermal well.

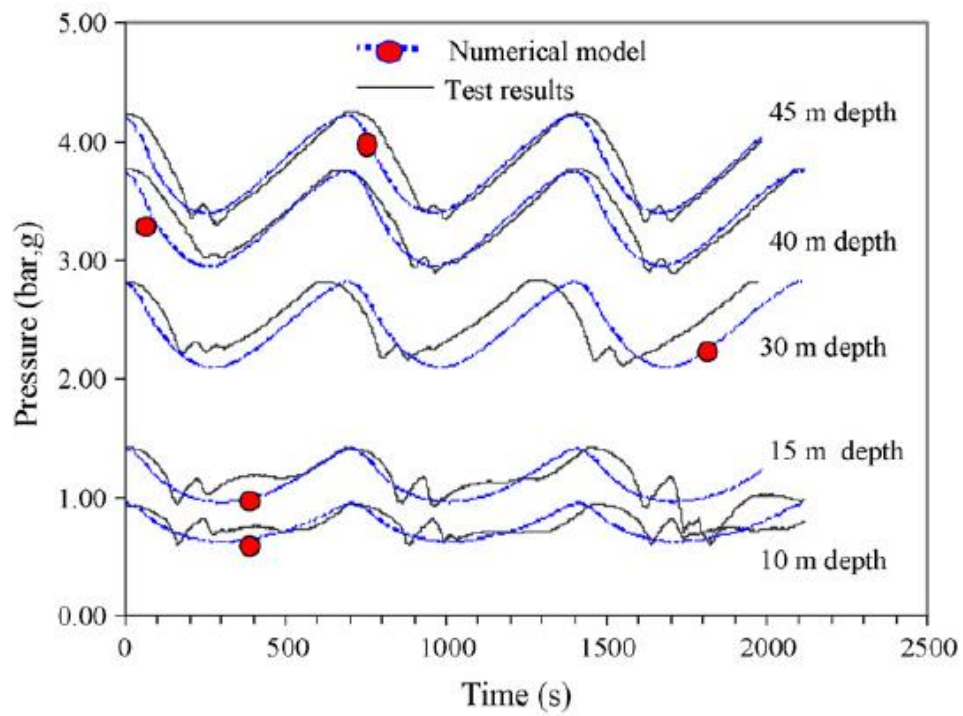


Figure 2-11: Pressure-time trace of pressure pulsations of geysering in geothermal well [26].

2.5 Need for modelling of water hammer with liquid column separation

The danger of water hammer with liquid column separation is that a hydraulic system experiences a far greater maximum pressure than the operating pressure and the pressure predicted by simplified water hammer equations, as observed in Figure 1-2 and Figure 2-4. If water hammer with liquid column separation cannot be avoided, its impact should at least be mitigated if possible [19].

The energy associated with water hammer can be immense; for example, in the incident that occurred in Ohio, USA, at a Sherwin-Williams plant, intermediate bulk containers (IBCs) were filled with diesel using a 2" stainless steel pipe via a pump bolted onto a concrete foundation. The operator suddenly closed the ball valve, causing water hammer pressure pulses [10]. These pressure pulses caused the 2" pipe to bend and rip the pump from the concrete pad.

Water hammer with liquid column separation not only causes damage to hydraulic components such as pipes, valves, and pumps but can also have devastating and even lethal consequences. Take, for example, the deadly accident at the Oigawa hydroelectric power plant in Japan in 1950. The accident can be summarised as follows according to the accounts of [27]: A butterfly valve shut down rapidly, causing extremely high-pressure waves. This caused the penstock to fail and burst at the valve.

See the location of the penstock framed in red in Figure 2-12 below from [28].

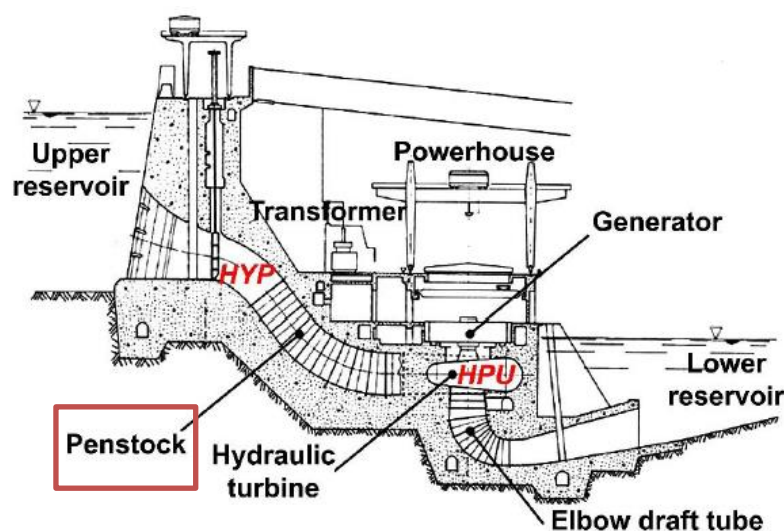


Figure 2-12: Schematic view of hydroelectric power plant [28].

The sudden release of water from the compromised penstock caused a low-pressure wave, thus resulting in liquid column separation. The difference in pressure inside the penstock and the atmosphere caused a section of the pipe to collapse. Figure 2-13 below displays the aftermath of the Oigawa power station accident. The left-hand side of Figure 2-13 displays the burst section of the penstock, and the right-hand side shows the collapsed section of the penstock.

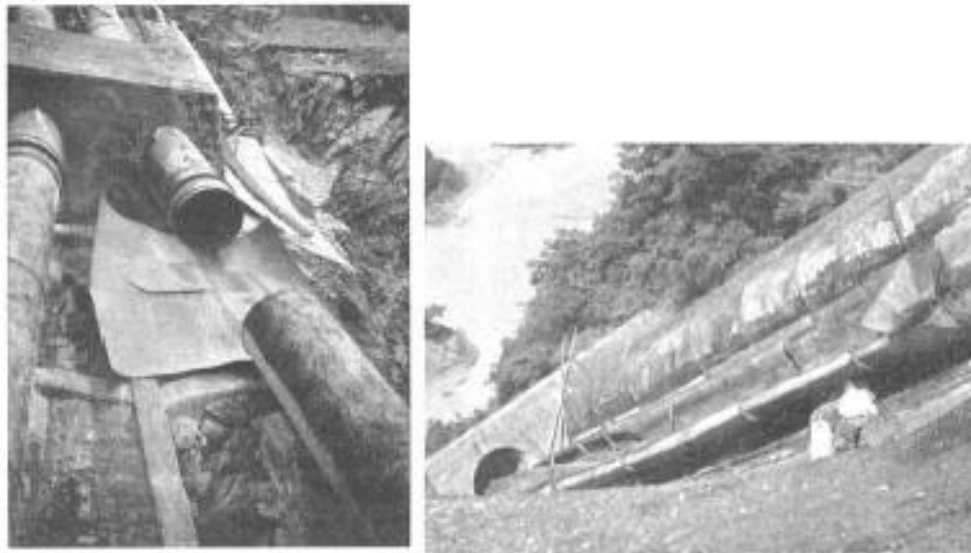


Figure 2-13: Images from the 1950 Oigawa hydroelectric power plant accident [28].

Water hammer, in general, is a significant concern with the transfer of liquified natural gas (LNG) [10]. LNG is transferred in liquid form by pressurising the gas to around 55-75 bar. This means that LNG is usually handled near its boiling point [29]. Water hammer with liquid column separation causes high-pressure waves in piping systems and low-pressure waves. These low-pressure waves can cause localised flashing of the LNG in the system. Due to the volatility of LNG, flashing can cause the LNG to combust. Since the whole pipeline system is filled with flammable natural gas, an explosion is evident [29].

Figure 2-14 below displays an accident at the Freeport LNG plant in Texas, USA.



Figure 2-14: Freeport Texas LNG plant accident 2022 [30].

From the above-discussed, water hammer with liquid column separation can have costly consequences in terms of damaged equipment and plant downtime; however, it can also be concluded that the consequences can be catastrophic and even lethal. Nonetheless, even with the potential hazards hidden with this hydraulic transient, it is a topic rarely explored by design and plant process engineers [10]. One reason for this can be attributed to the complexity of simulating the said hydraulic transient, which relies on computer processing power and numerical solving software to simulate the various numerical models developed over the years.

2.6 Advances in the field

Water hammer is not a modern hydraulic transient phenomenon. Since ancient times, water hammer have caused a multitude of concerns and issues for growing societies [28]. The first-ever documented approach to defining water hammer dates back to the 1st century B.C. in the work of Marcus Vitruvius Pollio [31]. Marcus explained the effects of water hammer on the lead and stone pipes of the Roman public water network and proposed mitigations.

The rapid growth in technology and manufacturing during the Industrial Revolution led to unprecedented scientific breakthroughs. This progress also led to the rapid development of hydraulic systems. This led to researchers developing new theories for deriving hydrodynamic

equations, ultimately used to create fluid models to analyse and predict water hammer [28]. In 1858, Menabrea noted that many engineers only considered the shock produced by water hammers but failed to consider the elasticity and brittleness of the pipe wall, as well as the compressibility of water [32]. Menabrea derived a formula to calculate the pipe wall's thickness considering the fluid's velocity and the system's net head. In 1878, Michaud published literature that proposed various methods to mitigate the pressure surges caused by water hammer [33]. The most prominent work done during this time frame was that of Nikolay Joukowsky when he introduced the “Joukowsky equation” used to calculate the maximum theoretical water hammer pressure in a system, which is still used extensively to this day [6].

See the “Joukowsky equation” below.

$$\Delta H = \frac{aV_0}{g} \quad (2.1)$$

where V_0 = initial flow velocity, a = speed of sound and g = gravitational constant.

Even though these early advancements were crucial to developing the numerical models we know today, these simplistic methods of calculating water hammer were insufficient by the turn of the 19th century. As more complex systems were being built, such as hydroelectric dams, design engineers were only left with these simple methods to account for water hammer, leading to a multitude of failures [28], as the concept of liquid column separation was not yet known. This led engineers to design systems and mitigation processes from accident case studies, such as in the works of [34] and [35].

By the early 20th century, water hammer events were still being investigated and solved using outdated and less accurate methods such as the graphical method and various other arithmetic methods. This is because, during the first half of the 20th century, the modelling of water hammer with liquid column separation was impossible due to the lack of computer processing power. It was only around the 1960s that the first modelling of water hammer with liquid column separation using the aid of computers began, as observed by [36] and [37]. During and after the 1960s, research on the said transient event accelerated along with technological advancements. By using numerical methods, researchers could better understand water hammer transient events, thus enabling them to create more accurate and robust numerical models.

2.7 Popular numerical models

Over the years, a multitude of numerical models have been developed, from relatively accurate and simple models to very accurate and complex models. From the literature, it became evident that three popular models are currently being used within the industry and for research purposes.

The three current popular models include:

1. The discrete vapour cavity model (DVCM).
 - Refer to [38], [39] and [40].
2. The discrete gas cavity model (DGCM).
 - Refer to [4], [41], [42], [43], [44], [45] and [46]
3. The generalized interface vaporous cavitation model (GIVCM).
 - Refer to [4], [47] and [48].

The numerical models describing water hammer with liquid column separation consist of a set of one-dimensional equations, each representing a different physical state of the liquid.

2.7.1 The discrete vapour cavity model (DVCM)

The DVCM is one of the most widely used numerical models and is used for most engineering fluid transient software packages [49]. This is because the DVCM can easily be implemented to reproduce various physical events occurring during column separation. The DVCM allows for the formation of vapour cavities at any computational section within the method of characteristics (MOC) when the pressure at the section falls to or below the vapour pressure of the fluid [50]. An in-depth discussion regarding the MOC is given in Section 3.2.1 of this dissertation.

The basic theory of the DVCM, as described by [49] is that the vapour region is concentrated over the entire cross-section of the pipe. When the pressure in any area along the length of the pipe drops to or below the vapour pressure of the fluid, a vapour cavity is formed. The pressure at this region is then equal to the vapour pressure of the fluid over the entire cross-section of the pipe and remains constant. The cavity is filled with steam, which inherently causes a discontinuity in the liquid. Therefore, the fluid is then divided into two independent and continuous fluid regions. The water hammer wave only propagates at a constant wave speed in the continuous fluid region.

For the DVCM, only the fluid vaporisation in the cavity is considered, and the release of dissolved gases is ignored. Since gas release is a relatively slow process, it can be ignored to keep the

model simple [49]. In other words, the DVCM is only capable of modelling vaporous cavitation. However, in most industrial systems, it is assumed that a negligible amount of free gas is released during column separation [43].

The assumptions regarding the vapour cavity in the DVCM are [50]:

1. The cavity pressure is equal to the vapour pressure of the fluid,
2. The volume of the cavity is small in comparison to the volume of the liquid in the reach,
3. The cavity is assumed to occupy the entire cross-section of the pipe; hence the pressure waves are reflected off the cavity,
4. The cavity is not allowed to move,
5. The cavity contains only pure liquid vapour (no released gasses),
6. The mass and momentum of the pure liquid vapour that occupies the cavity are neglected,
7. The cavity is assumed to exhibit isothermal behaviour,
8. The vapour condenses completely when the water columns re-join,
9. The friction coefficients of the pipe are not affected by the cavity.

The unsteady pipe flow is described by the continuity equation (conservation of mass) and equation of motion (conservation of momentum). This is known as the classic water hammer equations. For the classic water hammer equations, it is assumed that the flow in the pipe is 1-D, the pressure is greater than the vapour pressure of the fluid, unsteady frictional losses are approximated as steady friction losses, the pipe wall and the liquid behaves linearly elastically, and the amount of free gas is small and negligible [51].

The growth and decay of the discrete vapour cavity can be given by the continuity equation of the vapour cavity volume [50]:

$$\forall_{vap} = \int_{t_{in}}^t (Q_d - Q_u) dt \quad (2.2)$$

where t_{in} = time of cavitation inception, Q_d = downstream discharge, Q_u = upstream discharge, and dt = time step.

Utilising the Euler-backward method, Eq 2.2 is integrated numerically within the staggered grid of the MOC, and the following discretised equation is obtained for the continuity equation of the vapour cavity:

$$(\nabla_{vap})_{i,t} = (\nabla_{vap})_{i,t-2\Delta t} + \left[(1-\Psi) \left((Q_d)_{i,t-2\Delta t} - (Q_u)_{i,t-2\Delta t} \right) + \Psi \left((Q_d)_{i,t} - (Q_u)_{i,t} \right) \right] 2\Delta t \quad (2.3)$$

where $i = i^{\text{th}}$ computational section, $t = \text{current time step}$, $\Delta t = \text{MOC time step}$, $\nabla_{vap} = \text{Vapour cavity volume}$ and $\Psi = \text{relaxation weighting factor}$.

The cavity collapses when the calculated cavity volume is less than zero. The liquid phase is then re-established, and the classical water hammer equation is yet again valid [50]. The standard form of the continuity equations uses $\Psi = 0.5$, according to [52]. However, Ψ can take on values between 0 and 1, but the practical range for the weighing factor is 0.5 to 1, [17]. The author of [17] recommended that the maximum cavity volume not exceed 10% of the reach volume (control volume of an incremented pipe section).

In the experimental comparisons of [50], the experimental data from [17] were used. The experimental data of [17] were generated using an experimental apparatus from the Robin Hydraulic Laboratory at the University of Adelaide in Australia. The experimental test bench consisted of a straight sloping pipe with a diameter of $D = 0.022 \text{ m}$, a length of $L = 37.2 \text{ m}$ and a pipe slope of $\theta = 3.2^\circ$ connecting two reservoirs (pressurised tanks). Figure 2-15 below displays the experimental test apparatus used by [17].

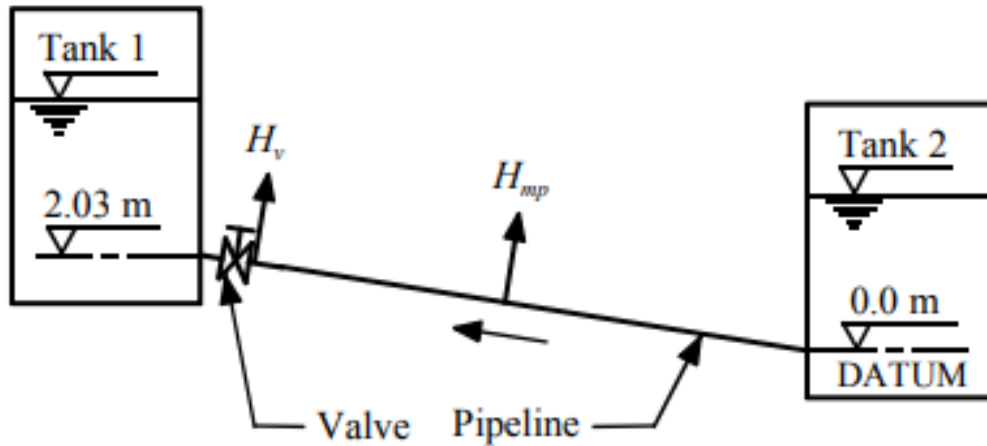


Figure 2-15: Experimental test apparatus layout [50].

A water hammer is induced by the rapid closure of a ball valve that can be located either in the middle of the pipeline (mp) or at either one of the reservoirs (v). A pressure control system regulates the steady-state flow and head. Five pressure transducers are located at central points along the pipeline's length. All experimental runs used demineralised water.

The paper of [50] presented pressure measurements at the valve (H_v) and the midpoint of the pipe (H_{mp}). During the experiment, the water temperature in Tank 1 was continuously monitored, the valve position was recorded with optical sensors, and the flow was estimated based on the volume change rate in Tank 1. A comparison between the computed DVCM results and the experimental measurements was made in the paper of [50] for two different steady-state flow velocities, $V_0 = 0.3$ m/s and $V_0 = 1.4$ m/s, respectively.

The computational and experimental runs were performed to rapidly close a valve positioned at the downstream end of the upward-sloping pipe (Tank 1, Figure 2-15). The static head of the reservoir (Tank 2, Figure 2-15) was kept constant at $H_{T2} = 22$ m. The valve closure time of $t_c = 0.009$ s was identical for both the low-flow and high-flow cases. The weighing factor was set to unity for all computational runs. To test the robustness and numerical convergence of the model, different numbers of reaches were selected for each computational run, that is, $N = 16, 32, 6, 128, 256$. However, only the results of $N = 32$ and $N = 128$ are presented in the paper [50].

Figure 2-16 and Figure 2-17 below present different computational reaches of $N = 32, 128$ for an initial flow velocity of $V_0 = 0.3$ m/s.

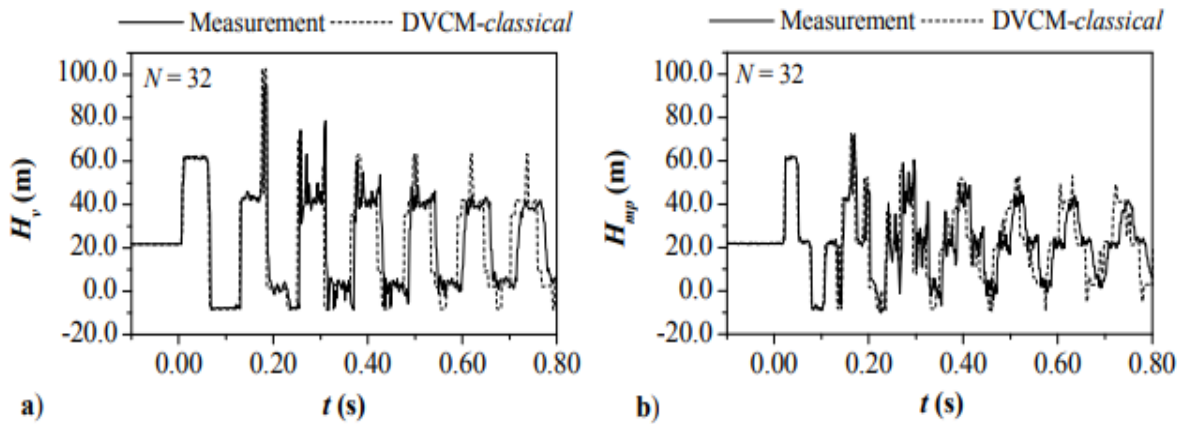


Figure 2-16: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 0.3$ m/s, $H_{T2} = 22$ m and $N = 32$ [50].

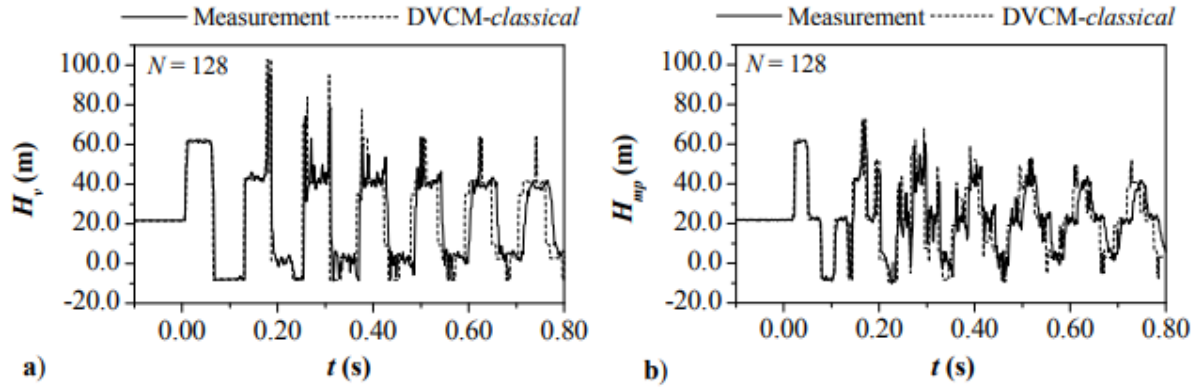


Figure 2-17: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 0.3$ m/s, $H_{T2} = 22$ m and $N = 128$ [50].

Figure 2-18 and Figure 2-19 below present different computational reaches of $N = 32, 128$ for an initial flow velocity of $V_0 = 1.4$ m/s.

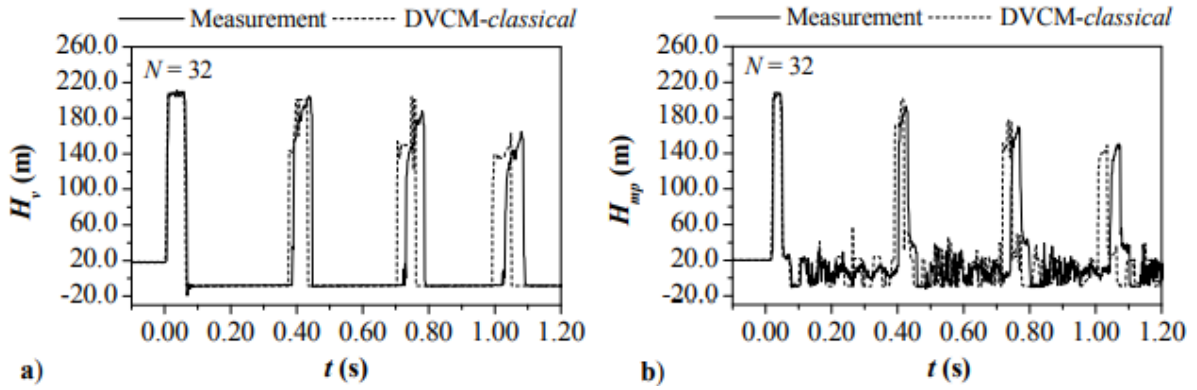


Figure 2-18: Comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 1.4$ m/s, $H_{T2} = 22$ m and $N = 32$ [50].

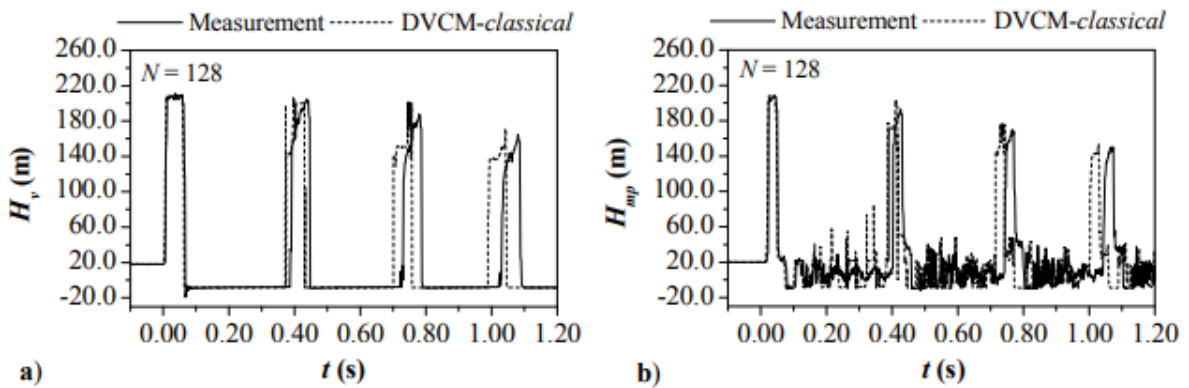


Figure 2-19: DVCM comparison of heads at (a) H_v and (b) H_{mp} : $V_0 = 1.4$ m/s, $H_{T2} = 22$ m and $N = 128$ [50].

For the low-initial flow velocity case, the rapid closure of the valve generated a moderate cavitation event [50]. However, the location, as well as the intensity of discrete vapour cavities, was governed by the hydraulic characteristics, piping layout and type of transient regime [46].

The maximum head measured at the valve in the experimental runs of [17] for $V_0 = 0.3$ m/s was $H_{v, \max} = 95.6$ m, which occurred as a short-duration pressure pulse. The maximum head calculated by the DVCM [50] is presented in Table 2.1 below.

Table 2.1: Calculated DVCM values: $V_0 = 0.3$ m/s from [50].

Number of reaches	$H_{v, \max}$ [m]
32	102.7
128	102.5

From Table 2.1 above, the DVCM overpredicted the maximum head by 7.43% and 7.22% for $N = 32$ and $N = 128$, respectively. In the paper of [50] It is concluded that although the DVCM may produce unrealistic pressure spikes, it can provide reasonably accurate results when the number of reaches is restricted. In the paper of [46], the same approach was followed, and similar results were obtained as described above.

Usually, the wave speed is assumed constant for the partial differential equations describing the transient liquid flow in pipelines. However, these equations are invalid when vaporous cavitation occurs, as pressure waves don't propagate through vaporous cavitation zones. [49]. Hence, the DVCM is only a simplistic approximation of the real-world phenomena occurring during water hammer with liquid column separation [17].

The disadvantages of the DVCM are illustrated below by analysing its main assumptions [49]:

1. The pressure wave speed is assumed to be constant throughout the transient process. In reality, it varies constantly throughout the process. The change in pressure will bring about a change in the gas content in the liquid, which will ultimately affect the pressure wave speed. According to Joukowsky, the pressure boosting of the water hammer is a function of the pressure wave speed.
2. Usually, a node is taken as the object of research. It is assumed that the flow into the node is not equal to the flow out of the node. This inherently causes a cavity to form. However, in reality, the fluid is elastic, and when the column of water is stretched within the range of its elasticity, no cavity will form.
3. The assumption is made that the cavity fills the entire cross-section of the pipe. In reality, the cavity can be elongated by passing pressure waves, and a condition may exist where the cavity only fills a portion of the cross-section.

Even though the DVCM is easily implemented and provides reliable results, it does, however, like any numerical approach, suffer from deficiencies [53]. This includes (i) unrealistic pressure spikes that may result from the restriction that avoids negative cavity sizes; (ii) vapour cavities are only allowed to form at computational sections, leading to biased results based on where these sections are located; (iii) the model is limited in its ability to model cavitation, due to fact that the size of the cavity and momentum of the cavity is ignored.

Given the deficiencies of the DVCM, this model can produce reliable and accurate results if the number of reaches (computational section size) is restricted [53]. As a rule, the ratio of maximum cavity size to the reach volume should never exceed 10% when using the DVCM [17]. Simpson and Bergant recommended that the maximum volume of the discrete cavity should be less than the reach volume [53].

In essence, this model allows the formation of vapour cavities to develop at any computational section of the MOC. It is assumed that a constant pressure wave speed persists within the liquid in between computational sections. When the pressure at the computational sections drops to or below the fluid's vapour pressure, a cavity is formed. When this occurs, the classic water hammer equations are no longer valid. The discrete vapour cavity can then be described by two water hammer compatibility equations, where $P = P_{\text{vapour}}$ and the upstream and downstream discharges can be obtained from the MOC compatibility equations. The discrete cavity is furthermore also described by the vapour cavity volume continuity equation. Once the vapour cavity collapses, a one-phase liquid flow is re-established, and the classic water hammer equations are yet again valid.

Since the introduction of the DVCM by the authors of [39], the problem of the DVCM with multi-cavity collapse and the effects thereof in producing unrealistic and spurious pressure pulses has been documented extensively in the literature; refer to [3], [20], [43], and [45]. The authors of [54] introduced the DGCM as a better alternative to the DVCM.

2.7.2 The discrete gas cavity model (DGCM)

The DGCM is built on the foundation of the DVCM, the only difference being that with the DGCM a permanent-infinitesimal gas cavity is added to each computational section in the MOC. This allows the simulation to capture the column separation phenomena anywhere over the entire length of the pipe [12], allows for gaseous cavitation to be considered and eliminates the possibility of multi-cavity collapse [46]. The use of the DGCM over the DVCM for the simulation

of column separation has been strongly recommended by researchers [11]. The DCGM model was first introduced by [54] and can perform consistently over a wide variety of parameters [46].

With the DGCM the discrete gas cavities replace the discrete vapour cavities from the DVCM, and the growth and collapse of the gas cavity are calculated using the classic water hammer equations, the gas cavity volume continuity equation, and the ideal gas equation [15]. With the DGCM a low gas void fraction (the ratio between the free-gas and the free-gas-liquid mixture volume) is used. The small amount of free gas is assumed to persist at each computational cell, and each isolated gas volume expands and contracts isothermally as the pressure varies, according to the ideal gas law [11]. This accounts for column separation over the entire length of the system [46]. It is recommended that the void fraction of $\alpha_g \leq 10^{-7}$ should be used. The DGCM allows for the formation of gas cavities at any computational section of the MOC grid. It is assumed that a single-phase liquid with a constant wave speed occupies the computational reach [53].

As discussed previously, the DGCM is described by four sets of equations. These include the classic water hammer equations, the gas cavity volume continuity equation, and the ideal gas law.

The integrated and discretised gas cavity volume continuity equation, as reported by [44], can be given as follows:

$$(\nabla_g)_{i,t} = (\nabla_g)_{i,t-2\Delta t} + \left[(1-\psi) \left((Q_d)_{i,t-2\Delta t} - (Q_u)_{i,t-2\Delta t} \right) + \psi \left((Q_d)_{i,t} - (Q_u)_{i,t} \right) \right] 2\Delta t \quad (2.4)$$

To suppress the numerical oscillations appearing in transients where the pressurisation of gas bubbles occurs, a weighing factor (Ψ) is introduced. Values of Ψ between 0 and 0.5 produce unstable results [43].

The general form of the ideal gas equation (gas equation of state) can be given as:

$$P\nabla = nRT \quad (2.5)$$

where P = pressure, ∇ = volume, n = amount of substance, R = gas constant and T = temperature.

Discretising Eq. 2.5 into the MOC, the following form of the ideal gas equation is obtained, as reported in [1]:

$$\nabla_g = \frac{P_{g0}^* \alpha_0 \nabla_m}{P_g^*} \quad (2.6)$$

where P_{g0}^* = reference pressure, α_0 = initial gas void fraction, ∇_m = computational cell volume and P_g^* = absolute gas pressure.

In the experimental comparison of [46] the experimental data from [17] were used; refer to Section 2.7.1, Figure 2-15. The computed DVCM, DGCM, and GIVCM results and the experimental measurements were compared for two different steady-state flow velocities: $V_0 = 0.3$ m/s and $V_0 = 1.4$ m/s, respectively. However, only the comparison regarding the DGCM will be presented in this section.

The computational and experimental runs were both performed for the rapid closure of a valve that is positioned at the downstream end of the upward-sloping pipe (Tank 1, Figure 2-15). The static head of the reservoir (Tank 2, Figure 2-15) was kept constant at $H_{T2} = 22$ m. The valve closure time of $t_c = 0.009$ s was identical for both the low-flow and high-flow cases. The numerical parameters for each model (DVCM, DGCM and GIVCM) are the number of reaches $N = 16$, a weighing factor of $\Psi = 1$ and an initial gas void fraction of $\alpha_g \leq 10^{-7}$ for the DGCM. Furthermore, a steady-state friction factor was used in all the computational runs.

The results comparison of the pressure head at the valve between the computed DGCM results from [46] and the measured experimental results from [17], for an initial flow velocity of (a) $V_0 = 0.3$ m/s and (b) $V_0 = 1.4$ m/s are displayed in Figure 2-20 below.

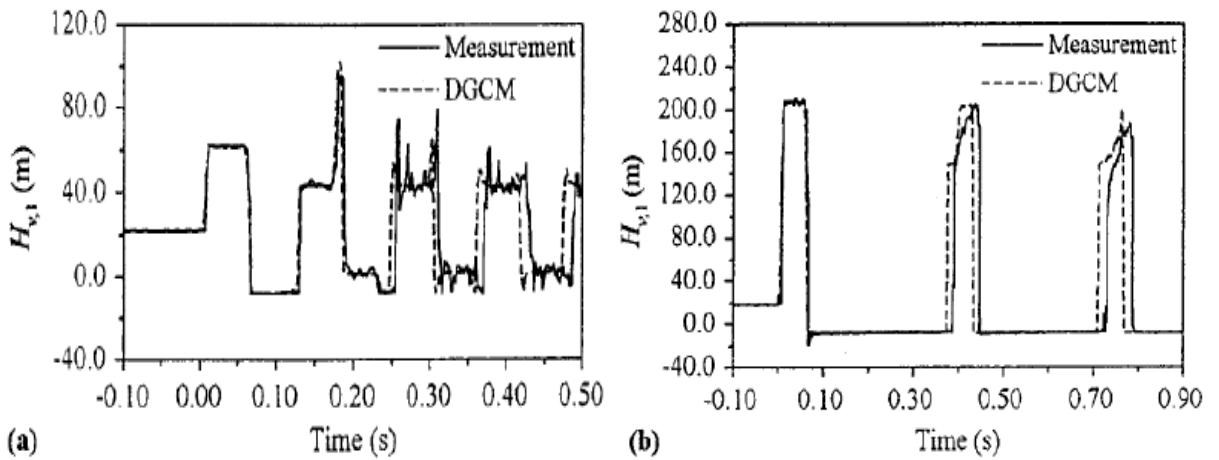


Figure 2-20: DGCM comparison of heads at $H_{v,1}$: (a) $V_0 = 0.3$ m/s (b) $V_0 = 1.4$ m/s, $H_{T2} = 22$ m [17].

From Figure 2-20 (a) above, it is observed that the low flow condition produces a column separation event in which the maximum pressure induced is more significant than that of the Joukowsky water hammer pressure rise (first peak) [46]. The high flow condition, Figure 2-20 (b), produces a column separation event with the opposite effect of the low flow condition, albeit the overall pressure rise is more considerable. It was also reported in [17] that the timing match between the DGCM and the experimental data correlates well until around 0.22 s, which corresponds to the observation of Figure 2-20 (a) and (b). The increased differences in timing after 0.22 s are likely due to unsteady friction effects [46], whereas the DGCM model used utilises a steady friction model.

For the low flow condition, it is reported in [46] that the maximum pressure head and the duration of the first cavity at the valve is measured by [17] to be 95.6 m and 0.0636 s, respectively. The maximum pressure head and duration of the first cavity at the valve for the low flow condition are calculated by [26] using the DGCM to be 101.9 m and 0.0635 s. Therefore, the point difference between the experimental measurements and the calculated DGCM results for the low flow condition is 6.59% and 0.16% for the maximum pressure head and duration of the first cavity, respectively.

For the high flow condition, it is reported in [46] that maximum pressure head and the duration of the first cavity at the valve are measured by [17] to be 210 m and 0.318 s, respectively. However, for the high flow condition, the maximum pressure head is the water hammer pressure rise, not the column separation event. The duration of the first cavity at the valve for the high flow condition is calculated by [46] to be 0.310 s. Therefore, the point difference between the experimental measurements and the calculated DGCM results for the high flow condition is 2.52% for the duration of the first cavity.

The DGCM is capable of reproducing many of the phenomena occurring during water hammer with liquid simulation. Moreover, the DGCM provides the investigator with reliable and reasonably accurate results [46].

However, the DGCM does suffer from highly non-linear numerical wave speed behaviour [48], and is therefore bounded to the initial void fraction. Also, the DGCM appears to be sensitive to the amount of free gas in the system [48] and produces pressure oscillations when the void fraction is uniform, and a single-phase liquid is present upstream from the cavity. The content of free gas significantly reduces the wave speed in pressurised pipes.

Figure 2-21 below displays the non-linear behaviour of the wave speed as a function of the initial void fraction and the absolute gas partial pressure.

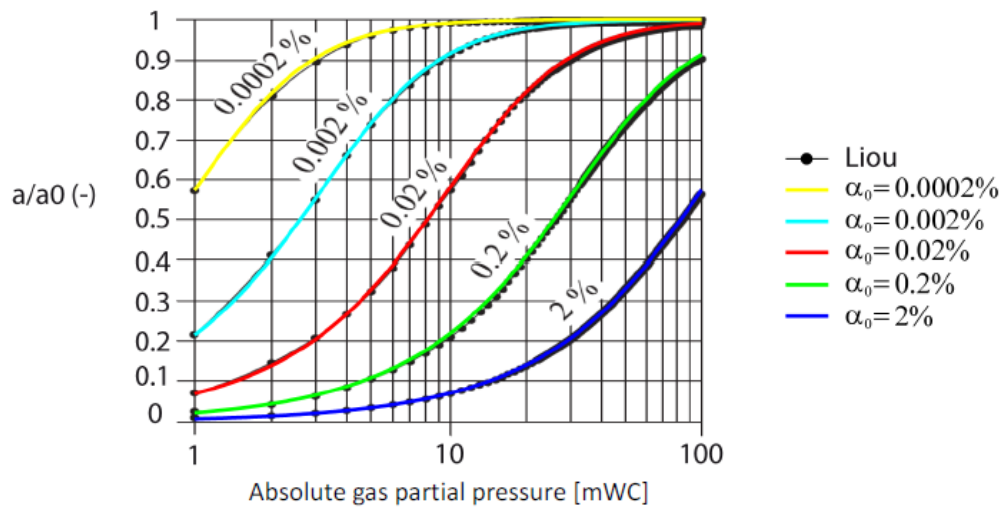


Figure 2-21: Wave speed ratio as a function of the initial void fraction and absolute gas partial pressure [48].

During water hammer with liquid column separation, the pressure in the system drops to or below the fluid's vapour pressure. The pressure might even drop to a negative value [55]. This is physically impossible as the intensity of cavitation will increase, leading to a larger volume of cavities and dampening the wave speed.

Figure 2-22 below displays the decrease and subsequent increase in the wave speed as the pressure drops (left) and the effect of a minimum initial void fraction on the wave speed (right).

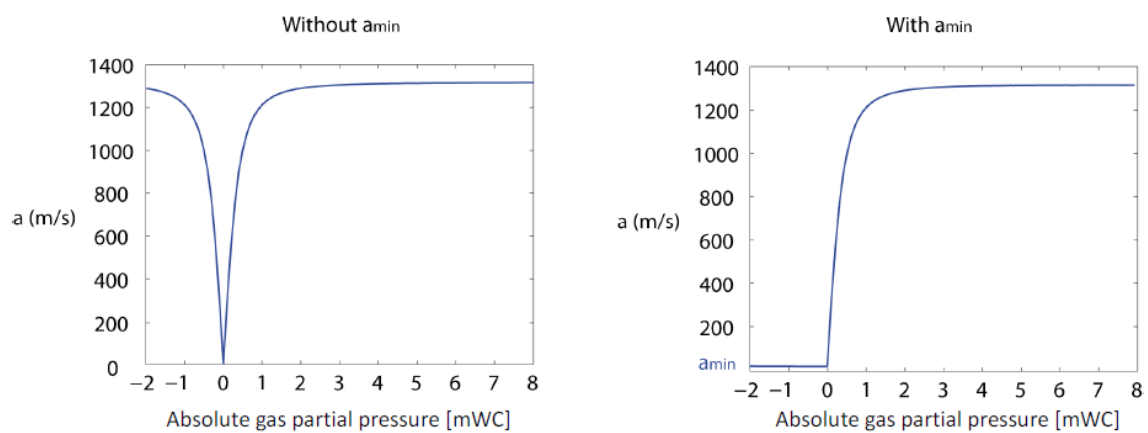


Figure 2-22: Wave speed without limitation (left) and with limitation (right) on the initial gas void fraction ($a_{\min} = 10^{-7}$ [55]).

Figure 2-22 shows that with a minimum initial gas void fraction (right), the wave speed behaves much more realistically than without one (left). Therefore, the wave speed is bounded to a minimum initial gas void fraction, where this value is defined *a priori* [55].

In essence, the DGCM model works on the same principle as the DVCM since the DVCM is used as a basis of the DGCM. The DGCM replaces the discrete vapour cavities from the DVCM with discrete gas cavities. Small amounts of free gas are assumed to persist at each computational cell within the MOC. It is assumed that each of the isolated gas volumes expands and contracts isothermally, according to the ideal gas law, with varying pressure. This enables the DGCM to account for gaseous cavitation and liquid column separation over the entire length of the pipeline system. An initial void fraction of $\alpha_g \leq 10^{-7}$ is used to simulate vaporous cavitation. This also ensures that only a small change in wave pressure occurs due to the presence of free gas in the system, even at low pressures [56].

The DGCM has proven to be very effective in numerically recreating many of the phenomena occurring during water hammer with liquid column separation. The DGCM provides reliable and satisfactory accurate results, although it suffers from non-linear wave speed behaviour. Therefore, the DGCM is widely used in many industrial numerical software packages [1], and is preferred above the DVCM since it is much more accurate and numerically stable.

2.7.3 The generalised interface vaporous cavitation model (GIVCM)

An interface type of model was first proposed by [57]. Improvements to the interface model have been developed by many researchers, such as [3] and [58]. However, it was [4] who used the DVCM as a basis to develop an interface model referred to as the generalised interface vaporous cavitation model (GIVCM).

The DVCM allows for the formation of cavities at each computational section within the MOC [45]. The GIVCM uses the DVCM as the basis and incorporates distributed cavitation, shock waves and various types of discrete cavities [46]. The GIVCM utilises the coupling of a set of analytical and numerical equations in one algorithm to solve water hammer with liquid column separation. A loop for the shock treatment at the appropriate computation sections is introduced in [46]. Further, a module for the computation of a discrete vapour cavity and a distributed vaporous cavitation zone is also introduced. The GIVCM allows for the correction of the physical behaviour between various phase interactions and allows for the identification of new interactions [47].

In the GIVCM, the various physical states of the fluid, such as the liquid state, homogeneous vaporous mixture, and the discrete cavity, can be described by the representative equations from [47]. The representative equations are (1) The classic water hammer equations that describe the liquid state, (2) Two-phase flow equations describing the homogenous vaporous mixture, (3) Equations describing the discrete cavity or vaporous cavitation regions, and (4) Shock equations that describe the condensation of a vaporous mixture back to a liquid state.

The classic water hammer equations describing the unsteady flow in the liquid, in its liquid state, are given by the continuity equation (conservation of mass) and equation of motion (conservation of momentum). These equations are only valid when the liquid is in the sub-cooled region, i.e. when the pressure at any specific point in the fluid is above the vapour pressure of the fluid [58].

The two-phase flow equations describing the homogenous vaporous mixture were developed by [58]. A homogenous vaporous mixture is assumed to occur when a negative pressure wave travelling in the pipe reduces the pressure over an extended length of the pipe to or below the fluid's vapour pressure. Since the pressure at an established vaporous cavitation region is assumed to be that of the vapour pressure of the fluid, no pressure wave can propagate through such a region [47].

The two-phase flow equations were developed by [58] for the control volume displayed in Figure 2-23 below:

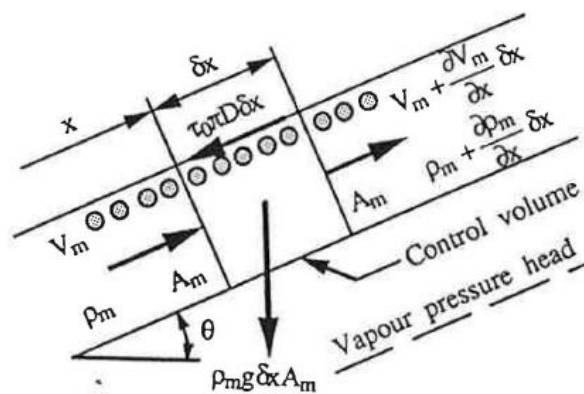


Figure 2-23: Two-phase flow equations control volume [58].

where A_m = hydraulic pipe area, ρ_m = density of vaporous mixture and τ_w = wall shear stress.

From the derivation of [58], Eq. 2.7 and Eq. 2.8 below gives the continuity equation and equation of motion, respectively.

$$\frac{\partial \alpha_v}{\partial t} + V_m \frac{\partial \alpha_v}{\partial x} - \frac{\partial V_m}{\partial x} = 0 \quad (2.7)$$

$$\frac{\partial V_m}{\partial t} + V_m \frac{\partial V_m}{\partial x} + g \sin \theta + \frac{f V_m |V_m|}{2D} = 0 \quad (2.8)$$

where α_v = the void fraction, g = gravitational constant, θ = pipe angle, f = friction factor, D = hydraulic pipe diameter, and V_m = the velocity of the vaporous mixture.

The continuity equation (Eq. 2.7) is used to solve the void fraction α_v , approximated via numerical integration [58]. The momentum equation (Eq. 2.8) is used to solve for V_m via numerical integration [58] and is dependent on the slope of the pipe, the terminal velocity V_m^t and the inception velocity of the vaporous mixture V_m^i . An inception velocity is calculated from the characteristic lines (C^+ and C^-) of the MOC, refer to Figure 3-2.

The equation for the terminal velocity, as derived by [47] is presented below.

$$V_m^t = + \sqrt{\frac{2gD|\sin \theta|}{f}} \quad (2.9)$$

Furthermore, in the paper of [47] the equations for V_m for the following conditions are given:

1. Upwards sloping pipe for a positive inception velocity ($\theta > 0$ with $V_m^i > 0$).
 - a. Before flow reversal.
 - b. After flow reversal.
2. Downwards sloping pipe for a negative inception velocity ($\theta < 0$ with $V_m^i < 0$).
3. Downwards sloping pipe for a positive inception velocity ($\theta < 0$ with $V_m^i > 0$).
4. Upwards sloping pipe for a negative inception velocity ($\theta > 0$ with $V_m^i < 0$).
 - a. Before flow reversal.
 - b. After flow reversal.
5. Horizontal pipe for a positive inception velocity ($\theta = 0$ with $V_m^i > 0$).
6. Horizontal pipe for a negative inception velocity ($\theta = 0$ with $V_m^i < 0$).

As mentioned, the void fraction α_v is approximated by the numerical integration of the continuity equation, Eq. 2.7. The numerical integration of Eq. 2.9 at time t is given by [43] as:

$$(\alpha_v)_j^t = (\alpha_v)_j^{t-\Delta t} + \left[\psi \left((V_m)_{i+1}^t - (V_m)_i^t \right) + (1-\psi) \left((V_m)_{i+1}^{t-\Delta t} - (V_m)_i^{t-\Delta t} \right) \right] \frac{\Delta t}{\Delta x} \quad (2.10)$$

where i = the upstream node and $i + 1$ = the downstream node of the computational reach j , respectively. With a computational reach length of $\Delta x = a\Delta t$.

When the weighing factor $\Psi = 1$, the solution is known as Simpson's solution [3]. The void fraction can also be calculated by analytical integration, as shown by [47]. However, the accuracy of calculating the void fraction is affected by the approach (numerical or analytical integration) used.

The growth and diminishment of the discrete vapour cavity, which commonly forms at a boundary, high point or intermediate point, can be described by either the water hammer equations or the two-phase flow equations [47].

As the liquid progresses into a vaporous mixture or the intermediate cavity in between two vaporous mixtures collapses, the vaporous mixture condenses back into a liquid state [47]. The shock equations describe the movement of the interface between the liquid and the two-phase homogenous mixture (liquid-vapour) and are derived for the control volume displayed in Figure 2-24 below:

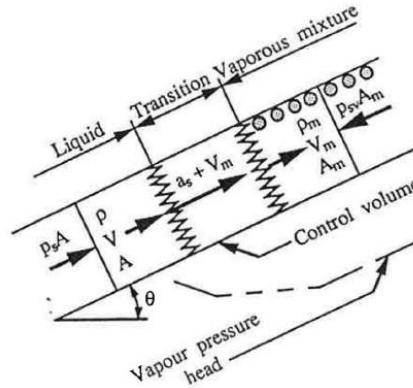


Figure 2-24: Shock equations control volume [58].

where p_s = pressure on the liquid side of the shock wave, p_{sv} = vapour pressure on the vaporous mixture side of the shock wave, A = hydraulic pipe area and ρ = liquid density.

The continuity equation and momentum equation of the shock equations derived from the control volume displayed in Figure 2-24 above are given as Eq. 2.11 and 2.12, respectively.

$$a_s \left(\alpha_v + \frac{g(H_s - H_{sv})}{a^2} \right) - (V - V_m) = 0 \quad (2.11)$$

$$g(H_s - H_{sv}) + (V - V_m)(V - V_m - a_s) = 0 \quad (2.12)$$

where a_s = shock wave speed, H_s = liquid side hydraulic grade line and H_{sv} = vaporous mixture hydraulic grade line.

In the experimental comparisons of [59], the experimental data from [3] were used to compare two different numerical models (DVCM and GIVCM). This section will display only the results regarding the GIVCM.

The experimental setup of [3] consisted of a straight sloping pipe with a length $L = 36$ m and an inner diameter of $D_i = 19.05$ mm. The pipeline does not consist of a long continuous pipe but two right-angled pipes. The upward-sloping pipe connects two pressure tanks (“upstream tank” and “downstream tank”). Figure 2-25 below displays the schematics of the experimental setup used by [3].

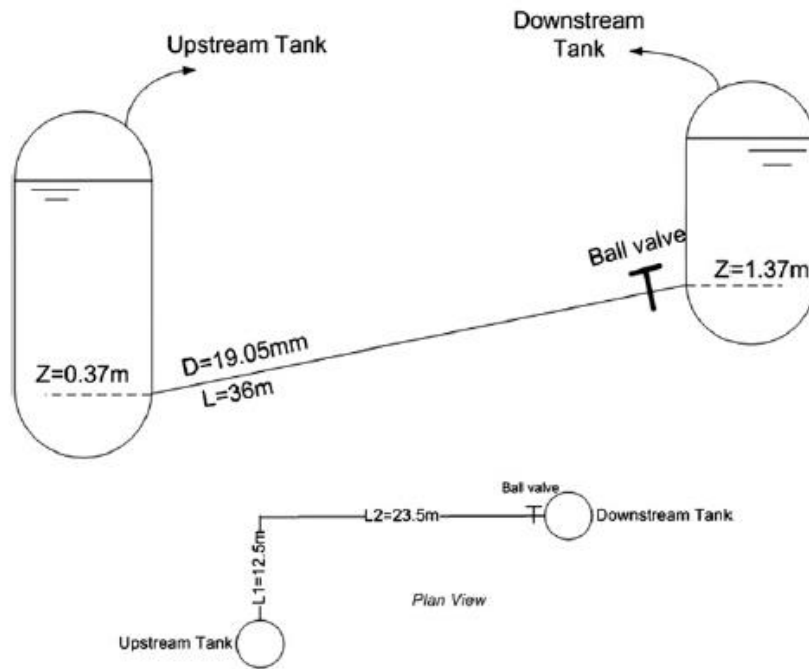


Figure 2-25: Experimental setup used by [3].

Water hammer with liquid column separation is induced when the ball valve is closed rapidly. That is when the valve closure time is $t_c \leq 2L/a$. The initial flow velocity in the pipe is 0.332 m/s, and

the pressure wave speed is 1280 m/s. The pressure head comparison of the GIVCM against the experimental data for 128 reaches/increments is displayed in Figure 2-26 below.

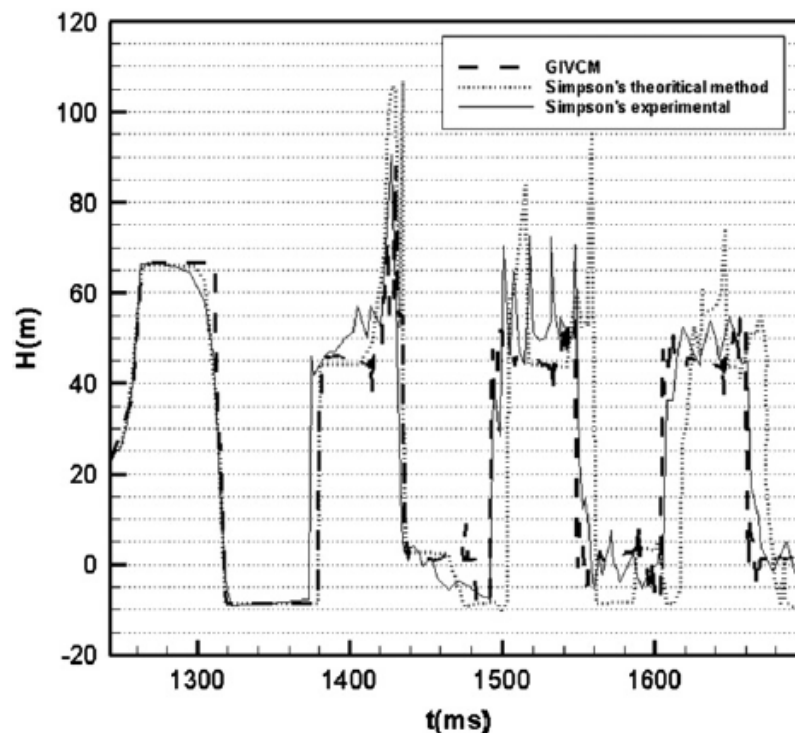


Figure 2-26: Experimental and calculated pressure head (GIVCM) [59].

From [59] it is reported that a cavity forms at the valve at around 1316 ms and collapses at 1375 ms. A significant increase in pressure follows at around 1431 ms due to the superposition of the water hammer pressure wavefront and the cavity collapse pressure wavefront.

The maximum pressure head measured by [3] was 91 m, and the maximum pressure head calculated with the GIVCM by [59] was reported to be 88.2 m after 1430 ms. This, in essence, means that the GIVCM model underpredicted the maximum pressure head of the experimental data by 3.08%. It was also reported by [59] that the GIVCM compares much more favourably to the experimental data than the DVCM in terms of the calculated pressure head and the timing of transient events.

The GIVCM can simulate different column separation phenomena: discrete cavities and distributed cavitation zones. The GIVCM can perform consistently over a wide range of parameters. The GIVCM is also capable of modelling an array of different pipeline configurations, the interactions between water hammer regions, distributed and intermediate cavitation zones, as well as discrete cavitation zones usually located at boundaries [46]. The improved performance of the GIVCM is credited to the more accurate treatment of distributed vaporous cavitation zones,

the introduction of shock waves and the various types of discrete cavities [46]. The main drawback of the GIVCM in comparison with the DVCM and DGCM is the complex nature of the model [46]. This leads to much longer computational solve times. One has to consider whether the increase in the accuracy of the results is worth the increase in the computational solving time.

2.8 Summary of past experiments on the DVCM, DGCM and GIVCM

The authors of [49] stated that the main weakness of the traditional DVCM is its numerical instability and non-convergence. The paper focused on introducing an improved DVCM by implementing a new wave speed equation and a new cavity model and using a floating grid instead of the traditional staggered grid. The results of the improved DVCM were compared to the conventional DVCM and the experimental data. The authors of [50] concluded that the improved DVCM provides more accurate results and is more numerically stable than the traditional DVCM. Adding the new wave speed equation that considered the gas holdup, liquid and pipe deformation. The wave speed changed as the gas holdup changed. The new cavity model considered the elasticity of the liquid. The floating grid method considered the influence of the cavity length.

In [50], the authors considered the advanced treatment of the growth and collapse of cavities within the DVCM. The authors implemented time adjustments to open and collapse cavities within the DVCM and compared the results of the improved DVCM to that of the traditional DVCM, as well as experimental results from [17]. The authors of [50] concluded that although the traditional DVCM provides reasonably accurate results, it is prone to spurious pressure spikes. The comparisons with the results of [50] showed that adjusting the timing of cavity collapse has a more significant influence than cavity opening. The adjustment to the timing of the cavity collapse did, however, result in a larger predicted maximum pressure head, but a much better timing of the transient event was achieved.

In the paper of [17] a sensitivity analysis of the DVCM was performed on changes in input parameters (wave speed, friction factor, initial velocity, pipe diameter, pipe length, and pipe slope). It was concluded by [17] that the DVCM's prediction of maximum pressure rise varied dramatically with only slight changes to the wave speed, friction factor, initial velocity, pipe diameter, and pipe slope. Furthermore, the authors recommended that a sensitivity analysis be performed to mitigate unrealistic pressure head predictions if the DVCM was used for water hammer studies.

The authors of [46] compared three numerical models (DVCM, DGCM, and GIVCM) in their ability to solve a generalised set of equations describing all conventional types of low-pressure zones. Water hammer zones, distributed and discrete vaporous cavitation zones, and shock zones. The authors compared the results of the aforementioned numerical models to the experimental results of [17]. It was concluded by [46] that the DVCM and DGCM involve a much simpler numerical code than that of the GIVCM. However, the GIVCM suffers from longer solving times than the latter. The GIVCM provided better insight into the transient event by its ability to track the actual column separation phenomena directly. Also, the discrepancies between the three numerical models can be attributed to the description of the vaporous cavitation zone, according to [46].

In the experimental comparisons of [51] the DGCM, considering the friction effects of unsteady flow, was tested. The tests were conducted using the experimental results of [17]. Usually, the DGCM is implemented, assuming that the friction factor is that of the steady-state Darcy-Weisbach friction factor. This is not accurate since the flow during water hammer transients is unsteady. The authors of [51] implemented two unsteady friction factor correlations into the DGCM, the quasi-steady friction factor model and the convolution-based unsteady friction model. Both improved models were tested against the experimental results. The authors were able to prove that the inclusion of these unsteady friction factor correlations significantly improved the performance of the DGCM. It was concluded by [51] that the convolution-based unsteady friction model better captures the behaviour of unsteady friction during water hammer transients. Furthermore, the authors concluded that the DGCM with the convolution-based unsteady friction model should be used for engineering practices.

The authors of [7] explored the idea of coupling the 1-D DGCM numerical model to a 3-D CFD software package to calculate liquid column separation. The 1-D numerical model is used for the transient analysis, and the 3-D CFD software (Ansys Fluent) is used to calculate the two-phase flow. The small portion of the pipe containing the cavity is calculated using the 3-D CFD software, and the rest of the transient is computed using the 1-D numerical model. The 1-D-CFD coupling was tested against experimental results from [17]. The authors of [7] concluded that the 1-D CFD and 3-D CFD both capture the transient event with almost identical accuracy. In terms of the 1-D-CFD coupling, although the results agree with the experimental results, some discrepancies still need to be explained.

In the paper of [8] the DVCM and GIVCM were both tested against the experimental results of a simple pipe-reservoir system. Furthermore, the authors tested the influence of various parameters on the calculated pressure head during a pump failure scenario for the before-mentioned

numerical models. These parameters are (1) the pipe diameter, (2) the flywheel effect and (3) the specific speed of the pump. The authors of [8] concluded that the GIVCM yielded better results than the DVCM compared to experimental results regarding the timing of the transient events and the maximum calculated pressure. Furthermore, the authors found that an increase in pipe diameter and pump inertia will reduce the maximum pressure in the pipeline. Also, the authors concluded that pumps with higher specific speeds may mitigate or prevent column separation.

The authors of [55] conducted water hammer with liquid column separation simulations using an electrical analogy. The authors compared both the DGCM and the electrical analogy model to experimental results from [17]. The electrical analogy modelling was based on a finite difference method with a centred scheme and a lax scheme for the discharge. The set of ordinary differential equations is represented as a T-shaped electrical equivalent scheme. The electrical analogy approach allowed the authors to have a pressure-dependent wave speed in the capacitance term and account for energy dissipation by adding the capacitance in series. The authors concluded that the electrical analogy method showed similar agreement to the DGCM, and both models compared well with the experimental results.

In the paper of [53], the authors investigated the modelling of vaporous cavitation in viscoelastic (polyethylene) pipes. The authors developed a model that incorporates the rheological-mechanical behaviour of viscoelastic pipes. The DVCM and DGCM were used to describe the cavitating flow in the pipes. The model's results were compared to an experimental test bench setup using high-density polyethylene. The authors found that the contribution of unsteady friction to the dissipation is small compared to the pipe's viscoelastic effect and that the pipe's movement and the additional dissolved air introduced additional dampening. Firstly, the authors concluded that the DVCM provided imprecise results and that the assumption of the ideal gas law in the DGCM is more appropriate. Secondly, the authors concluded that their developed model compared relatively well with the experimental results. However, future research and development is needed in this area.

The authors of [11] simulated water hammer with liquid column separation using 3-D-CFD based on the finite volume approach. The results obtained from the CFD, as well as the DVCM analysis, were compared to the experimental results of [17]. The authors concluded that the 3-D-CFD approach compared well with the experimental results and is far superior to the 1-D DVCM in predicting pressure fluctuations for turbulent fluid transients. The 3-D-CFD also better visualises the separation and re-joining of liquid columns, giving better insight into the system's dynamics.

However, one has to consider if the immense increase in processing power and simulation time is worth the increase in accuracy.

2.9 Research voids and uncertainties

Although an exhaustive amount of research has been conducted on water hammer with liquid column separation in the past, the open literature suggests that since the start of the 21st century, the research on water hammer with liquid column separation has significantly declined. There are still many uncertainties and voids within this specific area of study. While most fundamental research and development occurred during the 20th century, there is still much room for improvement in the different numerical models to increase their accuracy, robustness and ease of implementation.

From the open literature, it became evident that there is no explicit agreement on how to implement the numerical models, especially in terms of the weighing factor of the free gas volume equation in the DGCM model. For instance, according to [43] it is recommended that the weighing factor (Ψ) should be set to a value higher than 0.5; however, according to [48] it is strongly suggested that this weighing factor be set to unity only. The open literature also lacks information on the weighing factor's effect on the grid and the numerical model's time step independence.

2.10 Conclusion

This chapter provides an overview of water hammer and water hammer with liquid column separation, see Section 2.2. Water hammer occurs when a fluid in motion is stopped abruptly or is forced to change flow direction. Since the fluid possesses incompressible momentum, little energy absorption is present. The kinetic energy of the flowing liquid is transformed into pressure, which causes high-pressure pulses in the system, travelling back and forth from the point of disturbance to the reservoir (see Figure 2-1).

A common hydraulic transient observed with water hammer is liquid column separation. Initially, during a water hammer event with the rapid closure of a valve, the flow velocity is from the reservoir to the downstream valve. However, the flow velocity changes direction during the transient, and flow occurs from the downstream valve to the reservoir. The before-mentioned sequence of events may lead to the local static pressure head dropping to or below the vapour pressure head of the fluid. This leads to a vapour cavity to form at the valve. Since the flow velocity is towards the reservoir, the cavity grows and detaches from the valve (liquid column separation).

When the flow velocity again changes direction, now from the reservoir to the valve, the cavity will shrink and collapse. This causes a secondary pressure pulse in the system that is smaller than the water hammer pressure pulse. Under the right circumstances, the two pressure pulses, the water hammer and the liquid column separation, can superposition. This causes a short-duration pressure pulse significantly larger than the water hammer pressure pulse (refer to Figure 2-4).

Secondly, since liquid column separation is a cavitation-induced phenomenon, an overview of the two types of cavitation is presented in Section 2.3: vaporous and gaseous. Vaporous cavitation occurs when the system's local static pressure drops below the liquid's vapour pressure. Once the static pressure is equal to or less than the vapour pressure of the liquid, the liquid effectively starts to boil causing a two-phase mixture (liquid & gas). Gaseous cavitation occurs when the local static pressure in the system drops below the saturation pressure of the non-condensable gas dissolved within the liquid. A diffusion process occurs once the local static pressure is equal to or less than the saturation pressure of the non-condensable gas dissolved within the liquid.

Thirdly, a brief overview of other types of hydraulic transients is provided. These hydraulic transients are pulsating flow, condensate hammer, reverse flow hammer, and geysering (refer to Section 2.4).

Section 2.5 stresses the importance of modelling and simulating water hammer with liquid column separation from the design to the operating phase of a project. Relevant examples of industry accidents caused by the latter were given. These examples range from minor incidents like that of the Sherwin-Williams plant in Ohio to major lethal accidents like that of the Oigawa power station in Japan in 1950. A brief overview highlighted the dangers of liquid column separation and the need for modelling thereof in the LNG industry.

An overview of the advances in the water hammer and the liquid column separation field over the years is given in Section 2.6. Ranging from the first written concept of water hammer that dates to the 1st century B.C. in the work of Marcus Vitruvius Pollio, explaining the effects and proposed mitigations of water hammer on the lead and stone pipes of the Roman public water network, to the robust and accurate numerical models we use today. Today's most widely used numerical models are the DVCM, DGCM, and GIVCM. Section 2.7 provides an in-depth overview of the latter numerical models. Section 2.8 provides an overview of past experiments from the open literature performed on the numerical models presented in this chapter.

Lastly, this chapter provided an overview of the research voids and uncertainty found in the open literature in Section 2.9, giving background and context to the problem statement and research questions. The open literature concluded that the most widely used numerical model today, of the three models mentioned above, is the DGCM. However, although there has been an exhaustive amount of research and experiments done on the DGCM in the past, little is presented on implementing the said numerical model. This is especially true in terms of the weighing factor of the free gas volume equation in the DGCM model. According to [43] it is recommended that the weighing factor (ψ) should be set to a value higher than 0.5; however, according to [48] it is strongly suggested that this weighing factor be set to unity only. The open literature also lacks information on the weighing factor's effect on the grid and the numerical model's time step independence.

Therefore, the implementation of the DGCM in terms of the free gas volume equation weighing factor and its effect on the grid and time step independence needs to be investigated, validated, and verified.

Chapter 3 discusses the theory used to implement the DCGM model into EES and provides an overview of the verification and validation effort by comparing the EES simulation results to data obtained from the open literature.

CHAPTER 3 THEORY AND MODEL IMPLEMENTATION

This chapter provides an overview and in-depth discussion of the theory used in the DGCM numerical model. This includes the MOC, classic water hammer equations, derivation of the DGCM numerical model, and the theory used for conducting grid and time step independence study for CFD numerical models. Lastly, the verification and validation process for implementing the DGCM numerical model in EES are outlined and discussed.

3.1 Introduction

The previous chapter overviews hydraulic transients, specifically water hammer with liquid column separation. The need for modelling this specific hydraulic transient and the advances made in the numerical modelling of this hydraulic transient was discussed. The three main numerical models used to simulate this hydraulic transient were outlined (DCVM, DGCM and GIVCM), and an overview of past experiments was performed on the three aforementioned numerical models. The research voids and uncertainties identified in the open literature concerning the free gas volume weighing factor of the DCGM and its implementation are also discussed.

This chapter outlines the theory used in the DGCM numerical model. To establish whether the DGCM is implemented correctly into EES, verification and validation are performed on the EES model using the model and experimental results obtained in the open literature.

The outline of this chapter is as follows: Section 3.2 provides the theory used in the numerical modelling of water hammer with liquid column separation. This includes the MOC, the classic water hammer equations, the derivation of the DGCM model using the MOC grid, and the theory used for time step and grid size independence studies with CFD models. Section 3.3 provides the EES model's verification and validation results using data from the open literature.

Chapter 4 discusses this study's experimental methodology. This is followed by a discussion of the experimental test bench design, which includes the procedure used to design the fast-closing valve. Lastly, an overview and discussion of the measuring instruments used in the experimental test bench follows.

3.2 Numerical model theory

In the past, many researchers developed techniques and methods to simulate and describe the phenomena occurring during hydraulic transients, especially water hammer with liquid column separation, more accurately. Most of these numerical models utilise the well-known method of characteristics (MOC) to solve the equations describing the various flow regions in the pipe during fluid transients.

3.2.1 Method of characteristics (MOC)

The MOC, also known as the predictor-corrector method [1], is synonymous with fluid transient numerical models. Almost all numerical methods dealing with water hammer fluid transients utilise the MOC numerical solving scheme [1], due to its simplicity, minimal CPU usage, reliability and accuracy.

The reduction of partial differential equations (PDE) to ordinary differential equations (ODE) can be done using the MOC provided in [12]. Consider the theory of ordinary differential equations; it is stated that the equation (Eq. 3.1) can be solved uniquely if an initial condition u_0 is provided, only if f is a continuous function of both t and u . The following theory is explained in the paper of [12].

$$\frac{du}{dt} = f(t, u) \quad (3.1)$$

If the initial condition is allowed to be a function of x then the solution of u will also be a function of x . Therefore, u becomes a solution of:

$$\frac{du}{dt} = f(x, t, u) \quad \text{with} \quad u(x, 0) = F(x) \quad (3.2)$$

This can be considered an initial value problem for a PDE. If it is assumed that f and F are continuous functions of x then the solution $u(x, t)$ will also be a continuous function of x . Now consider the following simple one-dimensional linear advection equation, also known as the wave equation:

$$\frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} = 0 \quad (3.3)$$

$$u(x, 0) = F(x) \quad (3.4)$$

If the problem is reduced to an ODE along some arbitrary curve $x(t)$ in the (x, t) -plane of which the slope can be given as dt/dx , the curve $x(t)$ can be determined so that:

$$\frac{du(x(t), t)}{dt} = \frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} = 0 \quad (3.5)$$

Figure 3-1 below displays a typical characteristic curve, $x(t)$:

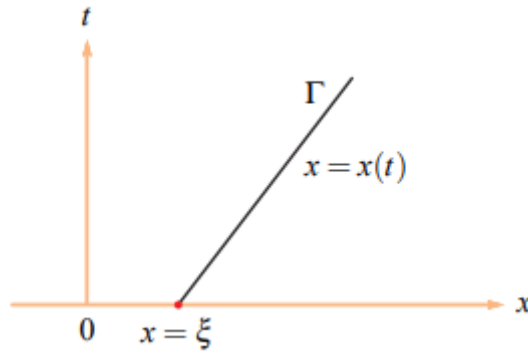


Figure 3-1: A typical characteristic curve [12].

Following the chain rule differentiation, the following equation is obtained from Eq. 3.5:

$$\frac{du(x(t), t)}{dt} = \frac{\partial u}{\partial x} \frac{dx}{dt} + \frac{\partial u}{\partial t} \frac{dt}{dt} = \frac{\partial u}{\partial t} + \frac{\partial u}{\partial x} \frac{dx}{dt} \quad (3.6)$$

From the observation of Eq. 3.5 and Eq. 3.6 it is clear that the ODE on the left-hand side of Eq. 3.5 is equivalent to the PDE on the right-hand side only if:

$$\frac{dx}{dt} = a \quad (3.7)$$

which is the reciprocal of the slope of $x(t)$. From this the PDE in Eq. 3.3 may consider the ODE:

$$\frac{du}{dt} = 0 \quad (3.8)$$

along any of the curves $x(t)$, Figure 3-1, which are the solutions to the curves given by Eq. 3.7.

A characteristic curve of the PDE (Eq. 3.3) is a curve in the (x, t) -plane given by $x = x(t)$, Figure 3-1, where $x(t)$ is a solution of the ODE (Eq. 3.5). From Eq. 3.7 it is clear that u is a constant along the characteristic curve. Integrating Eq. 3.7 provides the characteristic curves:

$$x(t) = at + \xi \quad (3.9)$$

where ξ is the x -intercept of the curve, see Figure 3-1. It shows that the characteristic curves are straight lines with a slope, $\frac{dt}{dx} = 1/a$. Furthermore, since u is constant along any given characteristic curve, its value can be determined from initial data. Therefore, along any given characteristic curve, we must have the following:

$$u(x, t) = u(\xi, 0) = F(\xi) \quad (3.10)$$

However, since $\xi = x - at$, the solution of the PDE can be given as:

$$u(x, t) = F(x - at) \quad (3.11)$$

The solution of PDE (Eq. 3.3) corresponds to transporting the initial profile $F(x)$ unaltered (preserving the shape of the waveform) along the characteristics with a speed $\frac{dx}{dt} = a$. Let Δx be the displacement in the x -direction during the time Δt . From Eq. 3.11, we obtain:

$$u(x, t + \Delta t) = u(x - \Delta x, t) \quad (3.12)$$

When discretising a pipe into the MOC grid to solve fluid transients, such as water hammer, the computational domain of the MOC is discretised into numerous computational cells with identical spatial and temporal increments, Δx and Δt , respectively. The unknowns at the computational points at timestep t_{N+1} can be calculated explicitly based on the known conditions from the previous timestep [1].

Figure 3-2 below displays a pipe discretised into the MOC grid.

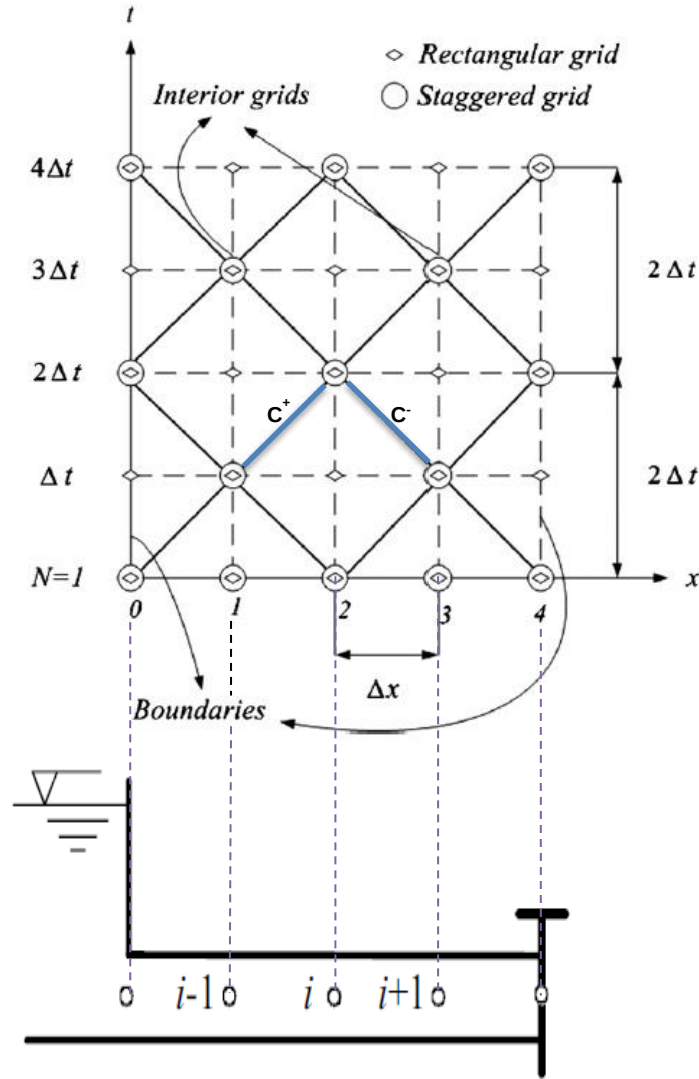


Figure 3-2: Discretization of a pipe into MOC staggered grid and rectangular grid, adapted from [51] and [59].

Figure 3-2 above displays a pipe discretised into the MOC computational grid in identical spatial and temporal increments. Two types of grid patterns appear: the rectangular grid and the staggered grid (Evangelisti grid) [17]. With the rectangular grid, the value of the pressure and flow is computed explicitly at each computational reach and each time step. However, with the staggered grid, the value of the pressure and flow is only calculated at every other time step for each computational reach. For example, take computational reach 0; the value at this reach is only calculated at $t = 0$ s, $t = 2\Delta t$, $t = 4\Delta t$ and so on. With the staggered grid, the computational effort and time are essentially half that of the rectangular grid.

It should be noted that the rectangular grid is more accurate for smooth solutions (i.e. the flow variables change gradually). Whereas, the staggered grid is more accurate for shock waves and discontinuities in the flow variables (i.e. water hammer).

The pressure and flow at time step $t = 0$ s are the initial conditions calculated by steady-state equations. At the boundaries, reach 0 and 4 in Figure 3-2, the values are computed using boundary-specific equations. In this case, that would be the head of the reservoir at reach 0 and a boundary-specific equation for the valve at reach 4. When a fluid transient event is initiated, the water hammer compatibility equations, that is, the classic water hammer equations, explicitly discretised for the MOC grid, are used to calculate the value at each interior grid of the MOC. The values at a specific interior grid point are calculated explicitly using the compatibility equations along the characteristic lines, C^+ and C^- in Figure 3-2, based on the known values of the grid points of the previous time step.

The MOC provides a simplistic yet reliable and accurate method to reduce PDEs to ODEs and solve the set of equations simultaneously. Through the comparisons of [50] where several column separation models were tested, the authors strongly recommended that the staggered grid (Evangelisti grid) be used with the MOC rather than the rectangular grid.

3.2.2 The classic water hammer equations

The equations describing water hammer transients are the continuity and momentum equations. Frequently referred to as the classic water hammer equations, the derivation of these equations from the conservation of mass and momentum equations can be seen in [45]. The classic water hammer equations, Eq. 3.13 and Eq. 3.14 below, are displayed as the continuity and momentum equations, respectively.

$$g \frac{\partial H}{\partial t} + \frac{a^2}{A} \frac{\partial Q}{\partial x} - g \frac{Q}{A} \sin(\theta) = 0 \quad (3.13)$$

$$g \frac{\partial H}{\partial x} + \frac{1}{A} \frac{\partial Q}{\partial t} + J = 0 \quad (3.14)$$

where:

Table 3.1: Classic water hammer equations, Eq. 3.13 and Eq. 3.14, variable description.

Variable	Description
g	Gravitational constant
H	Piezometric head
t	Time

Variable	Description
A	The cross-sectional area of the pipe
Q	Volumetric flow rate
x	Position in horizontal direction
J	Friction term
θ	Pipe angle from horizontal
a	Wave speed

The equations described above, Eq. 3.13 and Eq. 3.14, assume that the pressure wave speed (a) and the cross-sectional area of the pipe (A) are constant. Also, the convective terms may be neglected [13].

The equation used for the wave speed assumes that the flow is one-dimensional and that the compressibility and flexibility of the pipe are accounted for [13]. The wave speed equation is displayed in Eq. 3.15 below.

$$a^2 = \frac{K/\rho}{1 + \left[\left(K/E \right) \left(D/e \right) \right] c_1} \quad (3.15)$$

where:

Table 3.2: Wave speed equation, Eq. 3.15, variable description.

Variable	Description
ρ	Fluid density
K	Fluid bulk modulus
E	Young's modulus of the pipe
D_i	Pipe inner hydraulic diameter
e	Pipe wall thickness
c_1	Poisson ratio coefficient

3.2.3 The discrete gas cavity model (DGCM)

The following derivation of the DGCM theory and the implementation of the DGCM is described in the paper of [13]. The MOC transforms the set of PDEs (Eq. 3.13 and Eq. 3.14) into four ODEs, shown below as Eq. 3.16 and Eq. 3.17:

$$\frac{dQ}{dt} \pm \frac{gA}{a} \frac{dH}{dt} + AJ \mp \frac{g}{a} \sin(\theta) Q = 0 \quad (3.16)$$

$$\frac{dx}{dt} = \pm a \quad (3.17)$$

Eq. 3.16 can be solved along the characteristic lines, and Eq. 3.17 determines the slope.

Figure 3-3 below displays the characteristic lines within the MOC in the x, t -plane.

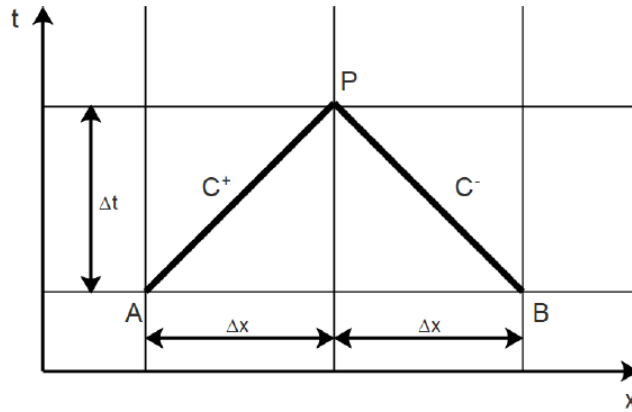


Figure 3-3: MOC characteristic lines in the x - t plane [52].

Approximating Eq. 3.16 with finite differences and integrating along the positive (C^+) and negative (C^-) characteristic lines yields the following:

$$C^+ : H_P = C_P - BQ_P \quad (3.18)$$

$$C^- : H_P = C_m - BQ_P \quad (3.19)$$

where:

$$C_P = H_A + BQ_A - \frac{a}{g} J_A \Delta t + \frac{\Delta x}{aA} \sin(\theta) Q_A \quad (3.20)$$

$$C_m = H_B - BQ_B + \frac{a}{g} J_B \Delta t + \frac{\Delta x}{aA} \sin(\theta) Q_B \quad (3.21)$$

and:

$$B = \frac{a}{gA} \quad (3.22)$$

Inserting Eq. 3.18 into Eq. 3.19 and solving for H_P and Q_P yields:

$$H_p = \frac{C_p + C_m}{2} \quad (3.23)$$

$$Q_p = \frac{H_p - C_m}{B} \quad (3.24)$$

The pressure head (H_p) and volumetric flow (Q_p) at point P, refer to Figure 3-3, are determined by the known values from points A and B. Therefore, the steady-state conditions must be known initially. The steady-state conditions are calculated using the well-known steady-state equations.

The ideal gas law is given by Eq. 3.25 below:

$$M_g R_g T = P_g \alpha V = P_{g,0} \alpha_0 V \quad (3.25)$$

where:

Table 3.3: Ideal gas equation, Eq. 3.13, variable description.

Variable	Description
M_g	Mass of free gas
R_g	Gas constant
T	Temperature
P_g	Free gas absolute partial pressure
$P_{g,0}$	Absolute partial pressure for initial void fraction
α_0	Initial void fraction

When Eq. 3.25 is discretised into the MOC, it takes on the form as presented in Eq. 3.26 below:

$$V_{g,p} = \frac{C_3}{H_p - z_p - H_v} \quad (3.26)$$

where:

$$C_3 = \frac{P_{g,0} \alpha_0 V}{\rho g} \quad (3.27)$$

and:

$$P_g = \rho g (H_p - z_p - H_v) \quad (3.28)$$

where V_g = the volume of the gas pocket, z_p = the height of node P, and H_v is the vapour pressure head of the fluid.

The volume of the gas pocket is calculated from the gas volume continuity equation, Eq. 3.29 below:

$$V_{g,P} = V_{g,P0} + \left[(1-\psi)(Q_{P0} - Q_{u,P0}) + \psi(Q_P - Q_{u,P}) \right] 2\Delta t \quad (3.29)$$

where Ψ is the relaxation weighing factor and subscript u denotes the upstream conditions. Subscript P denotes the conditions at time-step t and subscript P0 denotes the conditions at time-step $t - 2\Delta t$.

From Eq. 3.26 and Eq. 3.29, the following equation is obtained:

$$\frac{C_3}{H_p - z_p - H_v} = V_{g,P0} + \left[(1-\psi)(Q_{P0} - (Q_u)_{P0}) + \psi(Q_P - (Q_u)_P) \right] 2\Delta t \quad (3.30)$$

Rearranging Eq. 3.30 yields the following:

$$(H_p - z_p - H_v)^2 + 2B_1(H_p - z_p - H_v) - C_4 = 0 \quad (3.31)$$

where:

$$\begin{aligned} B_2 &= 0.25 \\ C_4 &= \frac{B_2 B C_3}{\psi \Delta t} \\ B_v &= \frac{0.5 \Delta t V_{g,P0} + (1-\psi)(Q_{P0} - Q_{u,P0})}{\psi} \\ B_1 &= -B_2(C_p + C_m) + B_2 B B_v + \frac{z_p + H_v}{2} \end{aligned} \quad (3.32)$$

Eq. 3.31 can be solved as a quadratic equation as described below:

$$H_p = \begin{cases} -B_1 \left(1 + \sqrt{1 + B_B}\right) + z_p + H_v & \text{if } B_1 < 1 \text{ and } B_B > 0.001 \\ -B_1 \left(1 - \sqrt{1 + B_B}\right) + z_p + H_v & \text{if } B_1 > 1 \text{ and } B_B > 0.001 \\ -2B_1 - \frac{C_4}{2B_1} + z_p + H_v & \text{if } B_1 < 1 \text{ and } B_B < 0.001 \\ \frac{C_4}{2B_1} + z_p + H_v & \text{if } B_1 > 1 \text{ and } B_B < 0.001 \\ \sqrt{C_4} + z_p + H_v & \text{otherwise} \end{cases} \quad (3.33)$$

The flow rate downstream and upstream are calculated using Eq. 3.31 and the equation provided below, respectively:

$$Q_{u,p} = \frac{C_p - H_p}{B} \quad (3.34)$$

3.2.4 Time step and grid size independence theory

Time step and grid size independence are crucial to the accuracy and stability of numerical CFD models. As the grid size is refined, i.e., the number of computational cells increases, and the time step size is refined, i.e., smaller time steps for each iteration, the spatial and temporal discretisation errors (excluding computer round-off errors) asymptotically reach zero [60].

Uncertainties and errors cause numerical simulation results to differ from their actual exact values. Discretisation errors, also known as numerical errors, are due to the formation and representation of the governing equations as algebraic equations in a discrete domain of space and time [61]. The spatial domain is known as the grid (mesh), and the temporal domain is known as the time step. As mentioned above, a consistent numerical model will approach the exact solution of the equation with zero error in the spatial and temporal domains as the grid size and time step are refined, respectively.

Theoretically, one can use infinitely small grid and time step sizes for the numerical simulation. However, this is impractical as the computational time will be infinitely large. Therefore, for any numerical simulation, a grid size and time step size exist for which the computational time and error are deemed acceptable. A method for examining the spatial and temporal convergence of a simulation was introduced by [62], thereby determining the ordered discretisation error in a

numerical simulation; see discussion in Section 3.2.4.1 below, as presented in [60] (obtained from [62]).

3.2.4.1 Order of convergence

A solution's error can be defined as the difference between the discrete and exact solutions (see Eq. 3.35 below).

$$E = f(h) - f_{\text{exact}} = Ch^p + H.O.T \quad (3.35)$$

where C = a constant, h = some measure of grid spacing, p = order of convergence, and $H.O.T$ = higher order terms. Suppose the solution is second-order, $p = 2$.

Neglecting the higher-order terms and taking the logarithm of both sides of Eq 3.35, leads to:

$$\log(E) = \log(C) + p \log(h) \quad (3.36)$$

Therefore, the order of convergence (p) can be obtained from the slope of the curve $\log(E)$ vs. $\log(h)$. However, a more accurate and direct evaluation of the convergence order (p) can be obtained from three numerical solutions (each with different grid size and time step values) and refinement ratio r .

$$p = \ln \left(\frac{f_3 - f_2}{f_2 - f_1} \right) / \ln(r) \quad (3.37)$$

The grid refinement ratio (r) is calculated as follows:

$$r = \frac{h_2}{h_1} \quad (3.38)$$

Solution f_1 , f_2 , and f_3 are obtained by three simulation runs of the numerical model at different grid sizes and time steps. For their corresponding solutions, 1 and 2, h = grid size.

The time step of each simulation run (i) is calculated by Eq. 3.39 below.

$$\Delta t_i = \frac{h_i CFL}{a} \quad (3.39)$$

where CFL = Courant number, h_i = grid size for simulation run i and a = speed of sound.

Once the order of convergence is established, the Richardson extrapolation method calculates the numerical simulation's theoretical value. The Richardson extrapolation equation is presented in Eq. 3.40 below.

$$f = f_{h=0} + g_1 h + g_2 h^2 + g_3 h^3 + \dots \quad (3.40)$$

where h = the grid size and the functions g_1 , g_2 , and g_3 are independent of the grid spacing. Therefore, $f_{h=0}$ is the theoretical value of the numerical model simulation. The Richardson extrapolation can be rewritten as $f_h = 0$ as follows.

$$f_{h=0} = f_1 + \frac{f_1 - f_2}{r^p - 1} \quad (3.41)$$

Once the Richardson extrapolation value for $f_{h=0}$ is known, a grid convergence index (GCI) can be calculated for each solution f_1 , f_2 , and f_3 . The GCI, referenced to the Richardson extrapolation solution, of the before-mentioned solutions can be calculated as follows.

$$GCI_i = 1.25 \left(\frac{|f_{h=0} - f_i|}{f_{h=0}} \right) / (2^p - 1) \quad (3.42)$$

where i = solution number, in this case.

With the GCI known for each of the three solutions, each with its grid size and corresponding time step size, linear interpolation or extrapolation may be used to calculate the required increment size for the desired mesh error specified. From this, using Eq 3.39, the corresponding time step size or the size of increment needed can be calculated.

If two or more parameters are being investigated for grid size and time step independence, the above-discussed mesh refinement study should be executed for all parameters. In this case, the

highest value for the required number of increments (finest grid) is used for the entire simulation [62], ensuring that all parameters of interest account for grid size and time step independence.

It is important to note that the calculated increment and time step size are unique to the specific numerical model setup. If any parameter in the numerical model changes, the above-described mesh refinement study is invalid and should be recalculated for yhr changed numerical model setup.

3.3 Theory implementation verification and validation

The purpose of this section is to elaborate on the verification and validation process followed to ensure that the DGCM theory, as described in Section 3.2.3, is implemented correctly into EES and that the EES model is fit for its intended purpose in predicting water hammer, as well as water hammer with liquid column separation. The complete EES code, used in Section 5.4.2, can be obtained in Annexure A.

The verification and validation process is divided into three cases, each with two subcases. Subcase A pertains to verification, and Subcase B pertains to validation. In the verification cases, the EES model is compared to theoretical results found by other authors. In the validation cases, the EES model is compared to experimental results found by other authors.

The verification and validation cases are as follows:

- **Case 1:** Pure water hammer case.
 - Case 1A: Verification by comparing the EES model results to that of Flownex®.
 - Case 1B: Validation by comparing the EES model results to the experimental results of [1].
- **Case 2:** Low-flow water hammer with liquid column separation case.
 - Case 2A: Verification by comparing the EES model results to the numerical model results of [17].
 - Case 2B: Validation by comparing the EES model results to the experimental results of [17].
- **Case 3:** High-flow water hammer with liquid column separation case.
 - Case 3A: Verification by comparing the EES model results to the numerical model results of [17].

- Case 3B: Validation by comparing the EES model results to the experimental results of [17].

The verification and validation parameters used are:

- **Case 1:** The maximum pressure head of the first pressure wave and the frequency of the first two pressure waves.
- **Case 2 and Case 3:** The maximum pressure head of the second pressure wave (cavity collapse-induced short-duration pressure pulse) and the time occurrence of said maximum pressure head (time of cavity collapse).

The above verification and validation parameters align with the results of the literature comparison. A normalised point difference is used to compare the verification and validation parameters of the EES model to those of the benchmark. In this case, the benchmark pertains to the values obtained from the literature.

The point difference (P.D) can be calculated as follows:

$$P.D = \frac{BNCH - EES}{BNCH} \quad (3.43)$$

The acceptance criteria for the calculated point differences are as follows: Point differences calculated in verification cases, i.e., EES compared to theoretical results found by other authors, are set at 5%. Point differences calculated in validation cases, i.e., EES compared to experimental results found by other authors, are set at 10%.

3.3.1 Case 1: Pure water hammer case

This case pertains to a pure water hammer event. Therefore, no liquid column separation occurs, which leads to an additional pressure spike in the system.

3.3.1.1 Case 1A verification and benchmark specification

3.3.1.1.1 Case 1A verification specification

This case consists of verifying a pure water hammer simulation in EES. The EES model results will be compared to the benchmark results from the simulation results of a similar Flownex® network.

The case is considered when a valve at the end of a horizontal pipe, filled with an incompressible liquid (water), is closed instantaneously. The horizontal pipe has a length of 10m, an internal diameter of 0.05 m, and a wall thickness of 0.002 m. The initial steady-state condition at the reservoir is reservoir pressure and temperature of 500 kPa and 15°C, respectively. A weighing factor of $\Psi = 1$ and an initial void fraction $\alpha_0 = 10^{-7}$ were used.

The material used in this verification is pure water. The pipe material is commercial steel. The properties mentioned above are displayed in the tables below.

Table 3.4: Physical properties of the fluid (pure water), Case 1A.

Property	Value	Unit
Density	998.66	kg/m ³
Viscosity	0.001135	kg/m.s
Specific Heat	1005	J/kg.K
Bulk Modulus	200	MPa

Table 3.5: Material properties of the pipe (commercial steel), Case 1A.

Property	Value	Unit
Young's Modulus	206	GPa
Poisson Ratio	0.29	-
Relative Roughness	30	μm

3.3.1.1.2 Case 1A benchmark specification

The benchmark, in this case, is a Flownex® network. Flownex® is a commercial 1-D systems CFD code containing a library of verified and validated component models that can be assembled to simulate flow in complex piping networks [63]. The Flownex® version used for this benchmark specification is the 2022 update 1 release (version 8.14.1.4845); see [64] for more information on this version. The Flownex® network input parameters are the same as discussed in Section 3.3.1.1.1 above. The upstream boundary condition has a pressure and temperature of 500 kPa and 15°C assigned to it, respectively. The downstream boundary condition consists of a flow

velocity of 0.051 m/s. The Alpha Crank-Nicholson value was set to 0.7, and the direct matrix inversion method was specified for the solver using the iterative solver.

The figure below displays the Flownex® network used for the benchmark results:

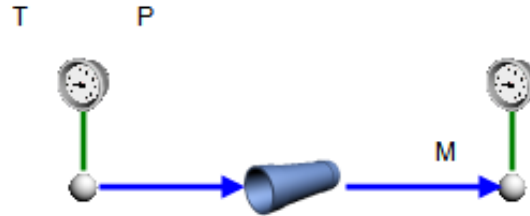


Figure 3-4: Benchmark Flownex® network, Case 1A.

This verification is done to confirm the EES model's ability to simulate pure water hammer.

3.3.1.2 Case 1B validation and benchmark specification

3.3.1.2.1 Case 1B validation specification

This case considers the validation of a pure water hammer simulation in EES. The EES model simulation results are compared to the experimental results obtained by [1]. The experimental results of [1] were generated at the Hydraulic Laboratory of the Institute of Water Supply and Water Engineering at the Warsaw University of Technology.

This validation case consists of an equal percentage valve (ball valve) at the end of a horizontal pipe filled with an incompressible fluid, closed at a recorded rate of 0.034 s. The length of the pipe is reported to be 41 m with an internal diameter of 0.042 m. The initial reservoir pressure head was 50 m (assumed from the steady state results; no mention was made of absolute pressure in the article). A weighing factor of $\Psi = 1$ and an initial void fraction of $\alpha_0 = 10^{-7}$ was used in the EES model.

Other input parameters mentioned in the literature of [1] includes the following:

- Pipe wall thickness $w_t = 3$ mm,
- Modulus of elasticity of the pipe material $E_{st} = 2.06 \times 10^{11}$ Pa,
- Bulk modulus of water $E_c = 2.19 \times 10^9$ Pa,
- Water temperature $T_w = 4.5^\circ\text{C}$.

However, the literature includes no other information on the properties of the pipe material or the properties of the fluid. Therefore, these properties are assumed as displayed in the tables below, that is, water at 4.5°C at the steady state pressure of 50 m static pressure head and commercial steel at 4.5°C:

Table 3.6: Physical properties of the fluid (pure water), Case 1B.

Property	Value	Unit
Density	998.66	kg/m ³
Viscosity	0.001135	kg/m.s
Specific Heat	1005	J/kg.K
Bulk Modulus	200	MPa

Table 3.7: Material properties of the pipe (commercial steel), Case 1B.

Property	Value	Unit
Young's Modulus	206	GPa
Poisson Ratio	0.29	-
Relative Roughness	30	μm

3.3.1.2.2 Case 1B benchmark specification

The benchmark used in this validation case is the experimental results obtained in [1]. The experimental setup in [1] consists of a pressure tank (2) connected to a horizontal pipe (1) and a ball valve (4) at the end of the pipe. The pressure tank is supplied with water from a supply system designed to maintain a constant pressure in the tank during unsteady flow (water hammer). A pressure transducer (5) is located just before the ball valve (4).

Figure 3-5 below displays the experimental setup used in the experiment of [1] in more detail:

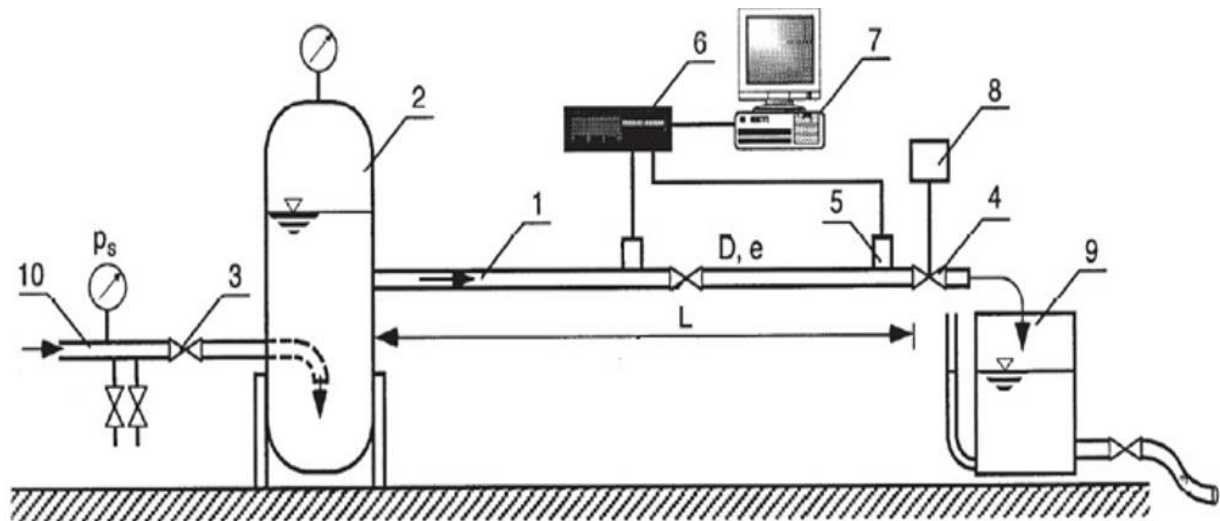


Figure 3-5: Experimental setup used in [1], Case 1B.

The components listed in Figure 3-5 above are described as follows.

1. Steel pipe,
2. Pressure tank,
3. Pressure reducing valve,
4. Ball valve,
5. Pressure transducer,
6. Extensometer amplifier,
7. Computer,
8. Digital time recorder,
9. Calibration tank,
10. Local water supply system.

The pressure transducer (5) used is a strain gauge, and the pressure was recorded in time intervals of 10^{-5} s.

This validation is done to confirm the EES model's ability to simulate pure water hammer.

3.3.1.3 Time step and grid size independence

The theory described in Section 3.2.4 was used to ensure time step and grid size independence. For Case 1A, a time step and grid size independence study was conducted for both the EES model and the Flownex® network and only for the EES model of Case 1B.

Table 3.8 below displays the time step and grid size calculated for Case 1. Note that increments refer to grid size. The convergence criteria for the above time step and grid size independency study was set to 0.0001% for Case 1A and 1% for Case 1B.

Table 3.8: Time step and grid size independency results, Case 1.

	Time Step (ms)	Increments
<i>Case 1A</i>		
EES Model	0.2679	104
Benchmark (Flownex)	0.4159	67
<i>Case 1B</i>		
EES Model	0.1731	161

3.3.1.4 Results comparison

Figure 3-6 and Figure 3-7 below displays the results of the EES model and the benchmark for Case 1A and Case 1B, respectively.

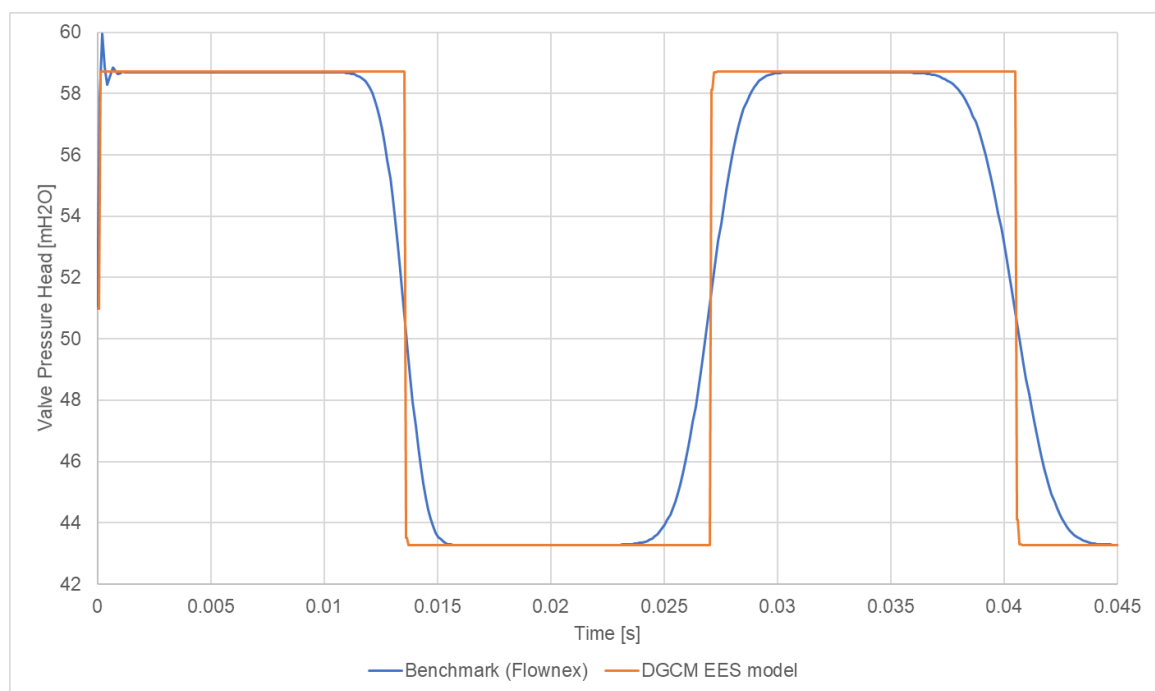


Figure 3-6: Results Comparison between EES model and Flownex® Benchmark, Case 1A.

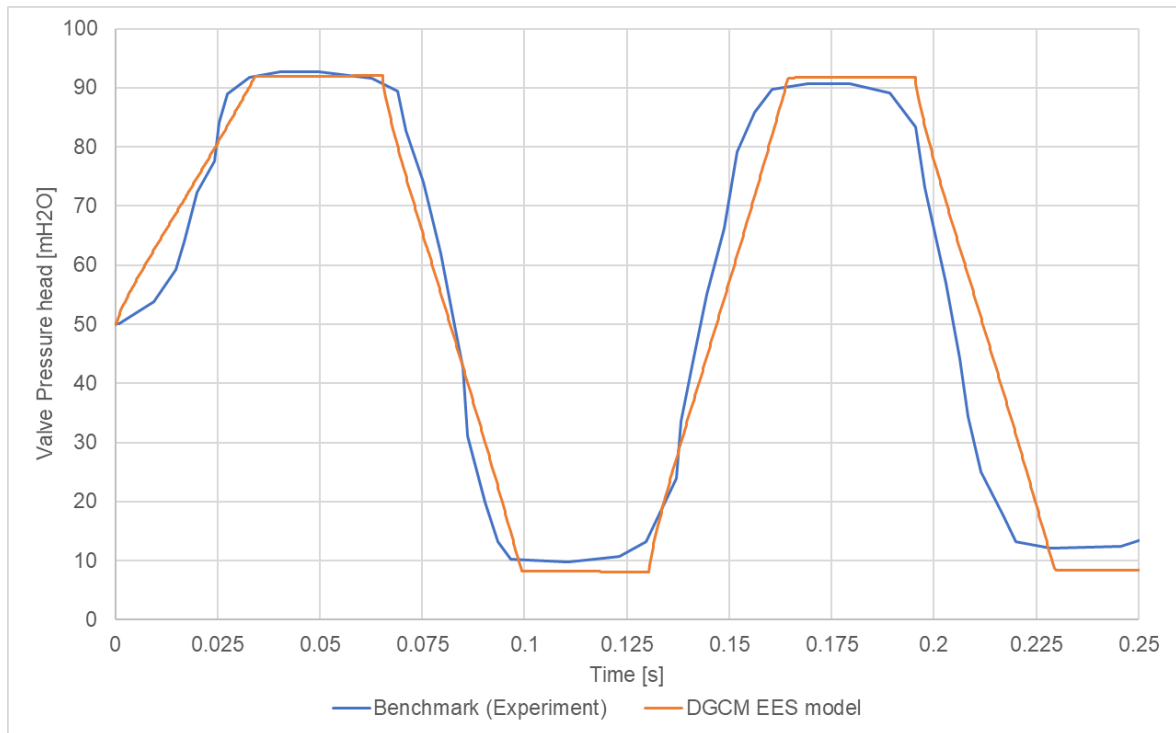


Figure 3-7: Results Comparison between EES model and experimental benchmark, Case 1B.

From Figure 3-6 and Figure 3-7 above, the EES model compares well to the Flownex® benchmark and experimental benchmark results regarding the verification parameters. The calculated point difference (error) of the maximum pressure head and the frequency for both Case 1A and Case 1B are displayed in Table 3.9.

For Case 1A, the maximum pressure head of the Flownex® benchmark and the EES simulation was calculated to be 58.6917 mH₂O and 58.7100 mH₂O, respectively. Similarly, the benchmark and the EES simulation's frequencies were calculated to be 36.7713 Hz and 37.0131 Hz, respectively.

For Case 1B, the maximum pressure head of the experimental benchmark and the EES simulation was calculated to be 92.7551 mH₂O and 92.0123 mH₂O, respectively. Similarly, the benchmark and the EES simulation's frequencies were calculated as 12.4363 Hz and 12.2622 Hz, respectively.

Table 3.9 below lists the point difference (error) calculated for each verification and validation parameter of Case 1.

Table 3.9: Point difference (error) for Case 1A and Case 1B results comparison.

Verification and validation parameter	Point difference (error)	
	<i>Case 1A</i>	<i>Case 1B</i>
Maximum pressure head	0.0312%	-0.7785%
Frequency	0.6576%	-1.3994%

Table 3.9 shows that the DGCM EES model passes the verification and validation requirements in terms of pure water hammer. The highest point difference was calculated as 1.3994% for the frequency of the validation case, Case 1B, which is lower than the acceptance criteria stipulated in Section 3.3.

3.3.1.5 Conclusion

According to the comparison of the results displayed in Figure 3-6 and Figure 3-7, and the differences calculated for the verification and validation parameters, see Table 3.9, this case serves as a successful verification and validation of the EES model in predicting pure water hammer.

3.3.2 Case 2: Low-flow water hammer with liquid column separation case

This case pertains to water hammer with liquid separation with “low flow”. Hence, the effect of the superpositioning of the water hammer and cavity collapse pressure pulse is magnified and not dominated by the water hammer pressure pulse.

3.3.2.1 Case 2A verification and benchmark specification

3.3.2.1.1 Case 2A verification specification

This case considers a transient water hammer with liquid column separation simulation with low flow in the EES model. The EES model simulation results will be compared to the benchmark results from the simulation results obtained from the DGCM model coupled with quasi-steady friction developed by [17].

The case considers a ball valve at the end of an inclined copper pipe connecting two pressure tanks filled with an incompressible liquid (water) that is closed instantaneously. The copper pipe has a length of 37.23 m, an internal diameter of 0.0221 m and a wall thickness of 0.00163 m. The inclined copper pipe has a constant slope of 5.45% or 3.12°, and the valve closure time is set to 0.009s. The initial steady-state conditions are; reservoir pressure head and flow velocity of 22 m

and 0.3 m/s, respectively. A weighing factor of $\Psi = 1$ and an initial void fraction of $\alpha_0 = 10^{-7}$ was used.

Since no mention is made of the fluid's temperature or the exact specification of the pipe material given, it is assumed that the fluid has the properties of pure water and the pipe has the properties of commercial copper pipe at 25°C.

Table 3.10: Physical properties of the fluid (water), Case 2A.

Property	Value	Unit
Density	998.66	kg/m ³
Viscosity	0.001135	kg/m.s
Specific Heat	1005	J/kg.K
Bulk Modulus	200	MPa

Table 3.11: Material properties of the pipe (commercial copper), Case 2A.

Property	Value	Unit
Young's Modulus	130	GPa
Poisson Ratio	0.34	-
Relative Roughness	15	μm

3.3.2.1.2 Case 2A benchmark specification

The benchmark, in this case, consists of the results found in the literature of [17], using their DGCM model coupled with a quasi-steady friction model. Hence, the input parameters of the model used by the authors of [17] are the same as described in Section 3.3.2.1.1 above, except for the parameters assumed in Table 3.10 and Table 3.11.

This verification is done to verify the EES model's ability to simulate water hammer with liquid column separation at low-flow conditions.

3.3.2.2 Case 2B validation and benchmark specification

3.3.2.2.1 Case 2B validation specification

This case considers the validation of a water hammer with liquid column separation simulation in EES. The EES model simulation results will be compared to the experimental results obtained by [17]. The experimental results are generated at the Robin Hydraulics Laboratory at the University of Adelaide, Australia.

The published experimental results and the EES model input parameters are the same as those specified in Section 3.3.2.1.1 above.

3.3.2.2.2 Case 2B benchmark specification

The benchmark used in this validation case is the experimental results obtained in [17]. The experimental setup in [17] consists of a ball valve (H_{v1}) at the end of an inclined copper pipe connecting two pressure tanks (T_1 and T_2) at a constant incline of 5.45% or 3.12°.

Figure 3-5 below displays the experimental setup used in the experiment of [17] in more detail:

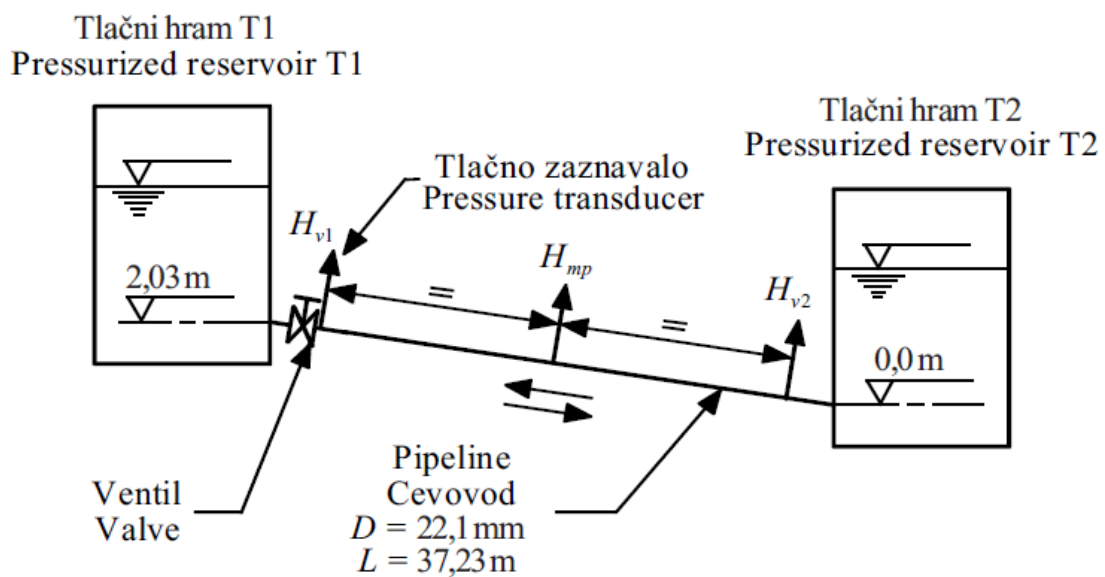


Figure 3-8: Experimental setup used by [17], Case 2B.

This validation is done to confirm the EES model's ability to simulate water hammer with liquid column separation at low-flow conditions.

3.3.2.3 Time step and grid size independence

The theory described in Section 3.2.4.1 was used to ensure time step and grid size independence. Since the same EES code is used in both the verification and validation cases, a single time step and grid size independence study was conducted for the EES model of both Case 2A and Case 2B.

The grid size (number of increments) calculated for Case 2 is 177, and the time step size is 0.1574 ms.

3.3.2.4 Results comparison

In the literature of [17], the maximum pressure head, obtained by the DGCM numerical model, is 99.6 mH₂O, occurring at a time step of 0.1881 s. Similarly, the maximum measured (experimental) pressure head at the valve is 95.6 mH₂O, occurring at a time step of 0.1858 s.

The experimental and DGCM numerical model results used for this verification and validation case of [17] are displayed in Figure 3-9 below. The solid line indicates the experimental results of [17], and the dotted line indicates the simulation results of [17].

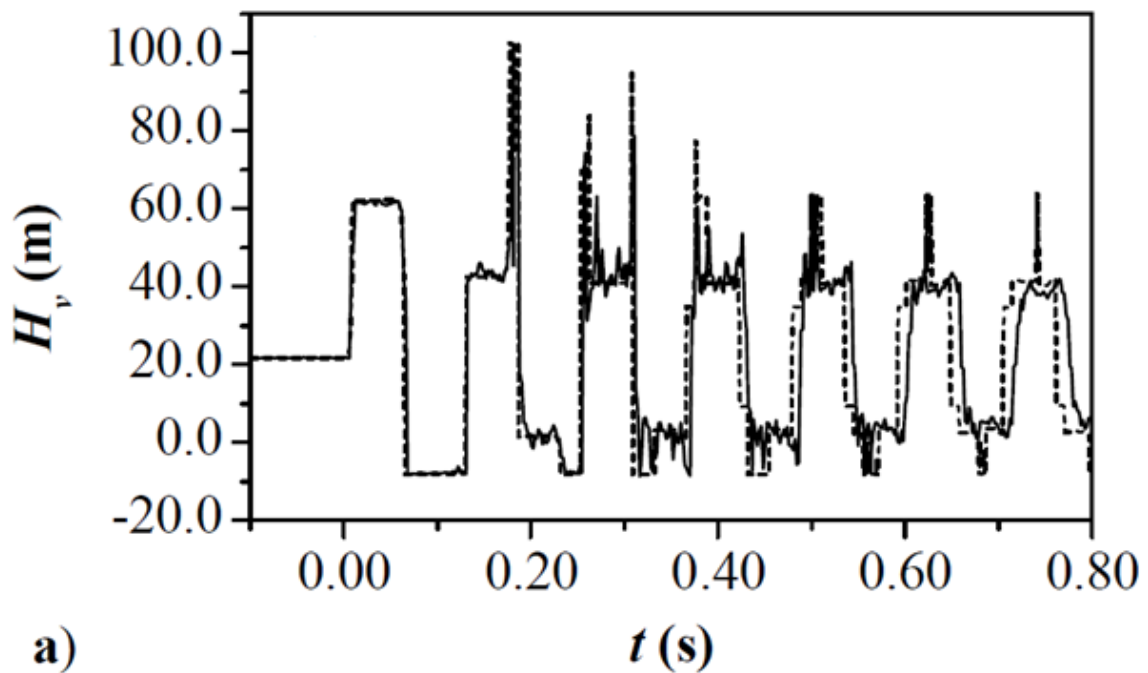


Figure 3-9: Experimental and DGCM model results ($V = 0.3$ m/s) of [17].

From the simulation results of [17], it is calculated that the maximum pressure head is 100.9027 mH₂O occurring at a time step of 0.1828 s.

Case 2 consisted of two subcases, Case 2A and Case 2B. Where Case 2A pertains to the verification by comparing the EES model results to the numerical model results of [17]. Case 2B pertains to the validation by comparing the EES model results to the experimental results of [17].

The point difference (error) calculated for each verification and validation parameter of Case 2 is listed in Table 3.12 below.

Table 3.12: Point difference (error) for Case 2A and Case 2B results comparison.

Verification and validation parameter	Point difference (error)	
	Case 2A	Case 2B
Maximum pressure head	1.3052%	5.5439%
Time of cavity collapse	-2.8463%	-1.6593%

Table 3.12 shows that the DGCM EES model passes the verification and validation requirements in terms of water hammer with liquid column separation at low flow. The highest point difference was calculated as 5.5439% for the maximum pressure head of the validation case, Case 2B, which is lower than the acceptance criteria stipulated in Section 3.3.

3.3.2.5 Conclusion

According to the point differences (errors) calculated for the verification and validation parameters (see Table 3.12), this case successfully verified and validated the EES model in predicting water hammer with liquid column separation at low flow.

3.3.3 Case 3: High-flow water hammer with liquid column separation case

This case pertains to water hammer with liquid separation with “high flow”. Hence, the effect of the superpositioning of the water hammer and cavity collapse pressure pulse is suppressed and dominated by the water hammer pressure pulse.

3.3.3.1 Case 3A verification and benchmark specification

3.3.3.1.1 Case 3A verification specification

This case verifies a transient water hammer with liquid column separation simulation and high flow in the EES model. The EES model simulation results will be compared to the benchmark results discussed in Section 3.3.2.1.1, except the initial flow velocity is 1.4 m/s instead of 0.3 m/s.

3.3.3.1.2 Case 3A benchmark specification

The benchmark used for this verification case is the same as that discussed in Section 3.3.2.1.1, except the initial flow velocity is 1.4 m/s instead of 0.3 m/s.

This verification is done to confirm the EES model's ability to simulate water hammer with liquid column separation at high-flow conditions.

3.3.3.2 Case 3B validation and benchmark specification

3.3.3.2.1 Case 3B validation specification

This case involves validating a transient water hammer with liquid column separation simulation and high flow in the EES model. The EES model simulation results will be compared to the experimental benchmark results discussed in Section 3.3.2.2.1, except the initial flow velocity is 1.4 m/s instead of 0.3 m/s.

3.3.3.2.2 Case 3B benchmark specification

As discussed in Section 3.3.3.2.1, the experimental benchmark used for this validation case is the same as in Section 3.3.2.2.1, except the initial flow velocity is 1.4 m/s instead of 0.3 m/s.

This validation is done to confirm the EES model's ability to simulate water hammer with liquid column separation at low-flow conditions.

3.3.3.3 Time step and grid size independence

The theory described in Section 3.2.4.1 was used to ensure time step and grid size independence. Since the same EES code is used in both the verification and validation cases, a single time step and grid size independence study was conducted for the EES model of both Case 3A and Case 3B.

The grid size (number of increments) calculated for Case 3 is 90, and the time step size is 0.3096 ms.

3.3.3.4 Results comparison

In the literature of [17] it is reported that the maximum pressure head, obtained by the DGCM numerical model, is 198.2290 mH₂O, occurring at a time step of 0.4535s. Similarly, the maximum measured (experimental) pressure head at the valve is 210.9 mH₂O, occurring at a time step of 0.4375s.

Case 3 consisted of two subcases, Case 3A and Case 3B. Case 3A pertains to verification by comparing the EES model results to the numerical model results of [17]. Case 3B pertains to validation by comparing the EES model results to the experimental results of [17].

The experimental and DGCM numerical model results used for this verification and validation case of [17] are displayed in Figure 3-10 below.

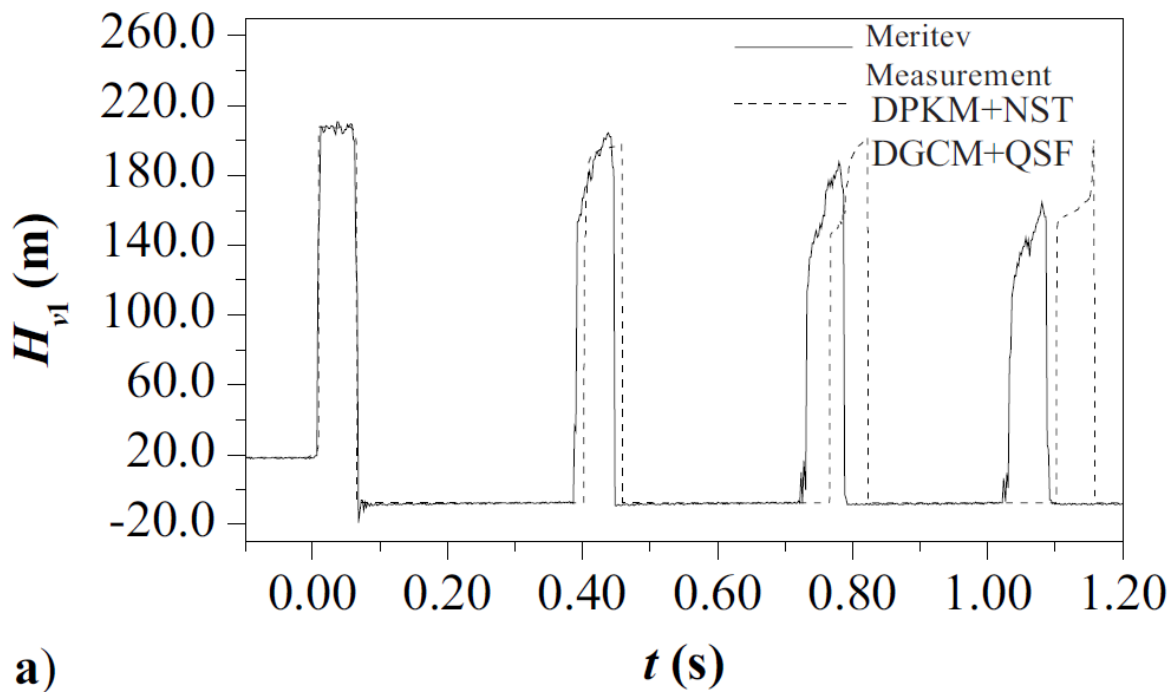


Figure 3-10: Experimental and DGCM model results ($V = 1.4$ m/s) of [17].

The DGCM EES model simulation results calculate that the maximum pressure head is 205.2109 mH₂O, occurring at a time step of 0.4341 s.

The point difference (error) calculated for each verification and validation parameter of Case 3 is listed in Table 3.13 below.

Table 3.13: Point difference (error) for Case 3A and Case 3B results comparison.

Verification and validation parameter	Point difference (error)	
	Case 3A	Case 3B
Maximum pressure head	-3.5221%	-2.7027%
Time of cavity collapse	-4.2783%	-0.8490%

Table 3.13 shows that the DGCM EES model passes the verification and validation requirements in terms of water hammer with liquid column separation at low flow. The highest point difference

was calculated as -4.2783% for the time of cavity collapse of the verification case, Case 3A, which is lower than the acceptance criteria stipulated in Section 3.3.

3.3.3.5 Conclusion

According to the point differences (errors) calculated for the verification and validation parameters (see Table 3.13), this case successfully verified and validated the EES model in predicting water hammer with liquid column separation at high flow.

3.4 Conclusion

Chapter 3 outlines the theory used in the numerical modelling of a water hammer with liquid column separation transient event. This chapter presents an overview and discussion of the MOC theory. This is followed by an overview of the DGCM numerical model theory derivation, starting with the classic water hammer equations. Lastly, a grid refinement study is crucial to ensure that a numerical simulation is time-step and grid size-independent. Therefore, this chapter presents the theory used for a time-step and grid-size independence study in CFD modelling.

Following the discussion of the theory, this chapter provides an overview of the implementation of the theory into EES. This is done by discussing the verification and validation effort by comparing the DGCM EES model simulation results to data from open literature. Following the verification and validation effort, it was concluded that the DGCM EES model is fit for its intended purpose of simulating pure water hammer cases (Case 1), water hammer with liquid column separation at low flow cases (Case 2), and water hammer with liquid column separation at high flow cases (Case 3).

During the grid and time step independence study of the DGCM EES numerical models used in the verification and validation effort, it was noted that for the water hammer with liquid column separation at the low flow case (Case 2), the smallest grid and time step size was needed to obtain independence, out of the three V&V cases. This can be attributed to the intensity of the numerical calculations occurring at the short-duration pressure pulse with the low flow case, showcasing the importance of grid and time step independence studies when using CFD numerical models.

Table 3.14 below displays the results of the grid and time step independence study conducted on the DGCM EES model for all verification and validation cases.

Table 3.14: Grid and time step independence study results for the DGCM EES model (verification and validation cases).

Case Number	Number of increments [-]	Time step size [ms]
Case 1A	104	0.2679
Case 1B	161	0.1731
Case 2	177	0.1577
Case 3	90	0.3096

Table 3.15 below summarises the calculated point differences (errors) during the verification and validation effort of the DGCM EES numerical model. Subcases A and B refer to verification cases.

Table 3.15: Verification and validation results summary of calculated point differences (errors)

Case number	Point difference (error)		
	<i>Maximum pressure head</i>	<i>Frequency</i>	<i>Time of cavity collapse</i>
1A	0.0312%	NA	0.6576%
1B	-0.7785%	NA	-1.3994%
2A	1.3052%	-2.8463%	NA
2B	5.5439%	-1.6593%	NA
3A	-3.5221%	-4.2783%	NA
3B	-2.7027%	-0.8490%	NA

Chapter 4 follows with a discussion of the experimental methodology used for the research effort. It also discusses the experimental design of a test bench capable of producing water hammer with liquid column separation events. A more detailed discussion is provided on the design of the fast-closing valve used in the test bench and an overview of the measuring instruments used in the test bench.

CHAPTER 4 EXPERIMENTAL METHODOLOGY AND DESIGN

This chapter outlines and discusses the experimental methodology of this research effort. It then discusses the design of the experimental test bench, focusing on the design of the fast-closing valve used in the test bench and the measuring equipment used during experimental data acquisition.

4.1 Introduction

The previous chapter discussed the theory used in the numerical modelling of water hammer with liquid column separation. It also discussed the derivation of the DGCM numerical model theory and the theory used for time step and grid size independence studies of CFD models.

This study aims to assess whether a baseline value for the weighing factor of the DGCM free gas volume equation exists and to establish whether the said weighing factor affects the grid and time step independence of the DGCM numerical model. The open literature alluded to the lack of information on implementing the DGCM numerical model, especially regarding the weighing factor. Some authors, such as [43] suggest that the weighing factor should be set between 0.5 and 1, whereas the authors of [48] suggest that the weighing factor should be as close to unity as possible.

Therefore, to fully control experiment parameters involved with validating a numerical investigation into the weighing factor, an experimental test bench must be constructed to recreate water hammer with liquid column separation events. The experimental data can then be used to compare various simulation runs of the DGCM numerical model, programmed in EES, using different weighing factors. After the results are compared and assessed, a conclusion can be drawn, and suggestions for future research can be made.

This chapter outlines the experimental methodology for this research effort. It then discusses the experimental design of the test bench, which generates water hammer with liquid column separation results. This includes a brief overview of the design of the fast-closing valve used in the test bench, along with a discussion of the measuring equipment used and its calibration procedures.

This chapter's outline is as follows: Section 4.2 provides the experimental methodology used in this study. Section 4.3 outlines the experimental design. More specifically, Section 4.3.1 provides

a brief discussion of the specifically designed fast-closing valve used in the experimental test bench, and Section 4.3.2 discusses the measuring equipment used in the experimental test bench with its calibration procedures (where applicable).

Chapter 5 discusses comparing the results. This includes the measured and calculated parameter inputs from the experimental procedure used in the EES simulation. The sensitivity analysis results on the effect of the weighing factor on the grid and time step independence of the DGCM numerical model follow. Lastly, the crucial findings made during experimentation and comparing the experimental results and simulation results are discussed.

4.2 Experimental methodology

Figure 4-1 below describes the experimental methodology followed during this research effort. The main research aim of this study is to identify whether a baseline value exists for the free gas volume weighing factor used in the DGCM. The secondary aim is to determine the effect of the weighing factor on the grid and time step independence of the DGCM numerical model.

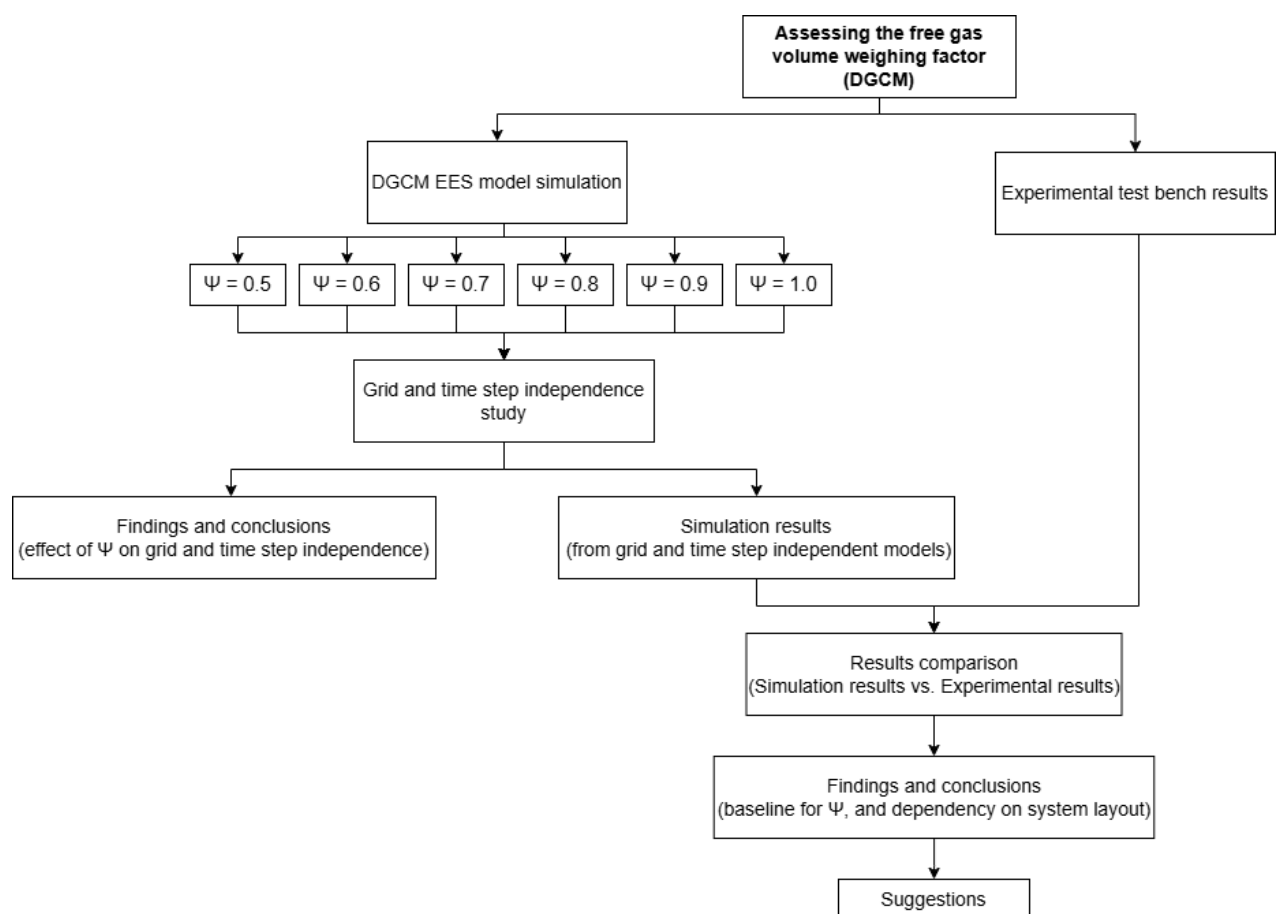


Figure 4-1: Overall experimental methodology.

If it can be concluded from the comparison of the results that the best simulation results are obtained when the weighing factor $\Psi = 1$, it further strengthens the suggestion made by [48] that the weighing factor should be as close to unity as possible. Also, it will explain why all results comparisons of the DGCM in the open literature use a weighing factor equal to unity; however, if the best simulation results are obtained when $\Psi \neq 1$, it will prove that no baseline for the DGCM weighing factor exists. Also, it will open the possibility of whether the weighing factor is system-specific, i.e. if it depends on other factors such as system layout, type of fluid, pipe material, etc.

If the grid and time step independence studies (sensitivity analysis) prove that an increase in the weighing factor increases the grid size and, consequently, the time step size, then it can be concluded that as the weighing factor increases, so does the stability of the DGCM numerical model. If not, then the weighing factor does not affect the grid and time step independence of the DGCM numerical model.

4.3 Experimental design

An experimental test bench for investigating water hammer with liquid column separation was constructed at the Faculty of Mechanical Engineering, North-West University (NWU), Potchefstroom, South Africa.

The experimental test bench comprised an open container tank connected to a local water source through a float valve to ensure the water level remained constant throughout the experimental procedure. The water temperature was measured with the use of a thermocouple. A 1" flexi hose from the open container led down to a collection tank, where the reservoir pressure was measured using a pressure transducer. From the collection tank followed a straight 1" galvanised steel pipe connected to a specifically designed fast closing valve, where the pressure was measured at the valve inlet using a pressure transducer. Downstream from the fast closing valve followed a flow control valve (gate valve), leading to a calibration tank calibrating the flow rate. Eventually, the water used during the experiment was dumped into a local water sink.

Figure 4-2 below displays a P&ID schematic of the experimental test bench constructed.

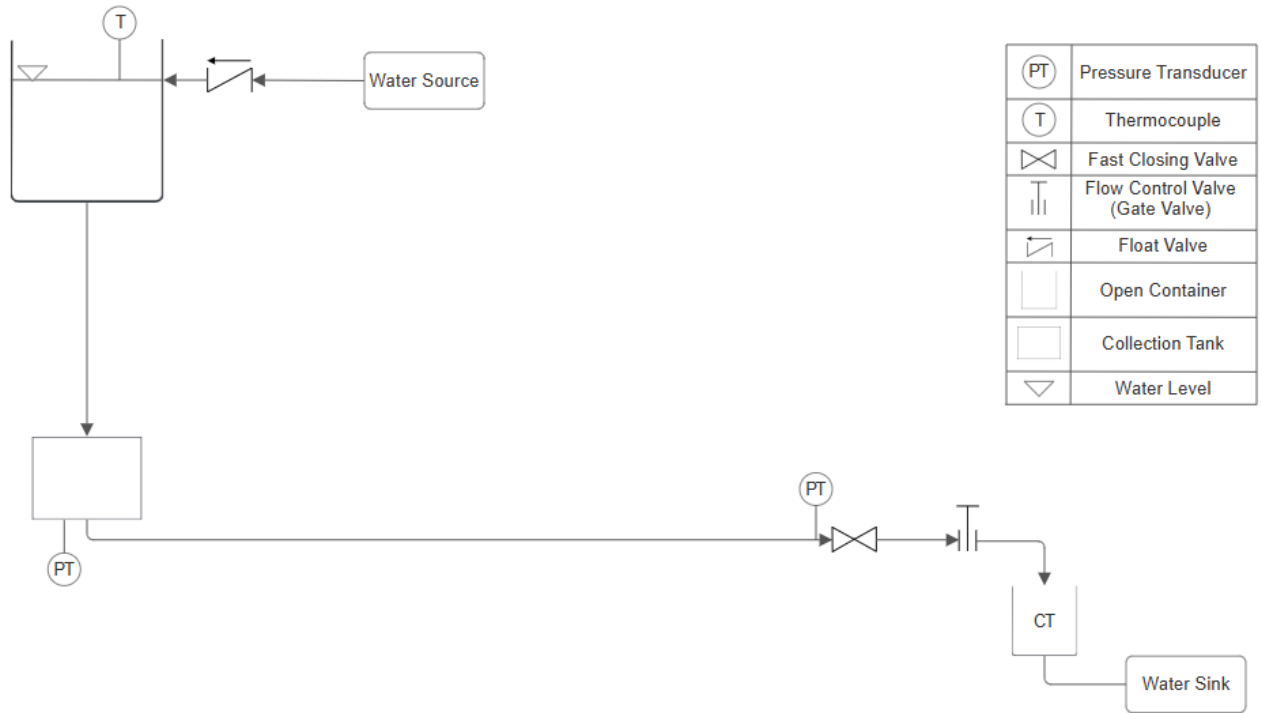


Figure 4-2: P&ID of experimental test bench constructed at the NWU.

The horizontal galvanised steel pipe has a length of $L = 15.3$ m, an internal diameter of $D_i = 27.4$ mm and a wall thickness of $w_t = 2.7$ mm. The reservoir pressure head measured from the water level at the outlet of the float valve to the bottom of the collection tank is $H = 8$ m.

4.3.1 Fast closing valve design

4.3.1.1 Valve spring mechanism design

From the literature, it is defined that a fast valve closure occurs when the valve closure time is faster than the period of the pipe, as illustrated in the equation below [2].

$$T_c \leq \frac{2L}{a} \quad (4.1)$$

Where T_c = valve closure time, L = length of the pipe and a = speed of sound.

However, in the literature, it is observed that for water hammer with liquid column separation to occur, the valve is usually closed around four times faster than the period of the pipe. For the experimental test bench presented in Section 4.3, the speed of sound in water is calculated to be 1384 m/s; using the equation above, the period of the pipe is calculated as 22.5997 ms. Therefore,

utilising the observations from the literature, the fast-closing valve used in the experimental test bench (see Figure 4-2) should have a valve closure time of at most 5.6499 ms.

Since a valve with the specified valve closure time of less than 5.6499 ms is commercially unattainable, the need to design and manufacture a fast-closing valve for the experimental test bench arose. To keep the valve design simple and financially feasible, a spring-actuated piston valve was chosen to design the fast-closing valve used in the experimental test bench.

The valve is designed to be spring-loaded to obtain the fast closing action. A mass-spring mathematical model was developed to determine the spring constant needed for the valve to close within the specified time of 5.6499 ms, see Annexure B. Four forces act upon the piston: the force of the spring, the force of gravity, the frictional force on the O-ring, and the force of the water pressure. The O-ring is sized based on the O-ring manufacturer, ERIKS, sealing elements guide [65]. A video link of the valve during the commissioning of the test bench and manufacturing drawings are included in Annexure C.

The frictional force of the O-ring is calculated using the theory described in [66]. The force of the water pressure increases as the valve closes. The verified and validated EES model is used to simulate the experimental test bench as presented in Section 4.3, with approximated inputs, using a specified maximum flow velocity of $V = 1 \text{ m/s}$ or a flow rate of $Q = 5.8965\text{E-}04 \text{ m}^3/\text{s}$ and an assumed valve closure of $T_c = 5.6499 \text{ ms}$.

This provides a pressure-time trace, and the theory of Annexure B can be utilised to calculate the piston position over time, changing the spring constant (k) until the desired valve closure time is met. The maximum flow velocity used for designing the valve is 1 m/s (i.e., a flow rate of $5.8965\text{E-}04 \text{ m}^3/\text{s}$), ensuring that for any flow velocity lower than the design, the valve will close faster as the pressure rise is less while still ensuring the occurrence of liquid column separation.

According to the CAD design, the piston's mass is 0.977 kg . The spring's initial displacement is $x_0 = 0.043 \text{ m}$, with a target end displacement of $x_1 = 0.016 \text{ m}$. Figure 4-3 below displays the displacement time trace of the mass-spring model derived in Annexure B.

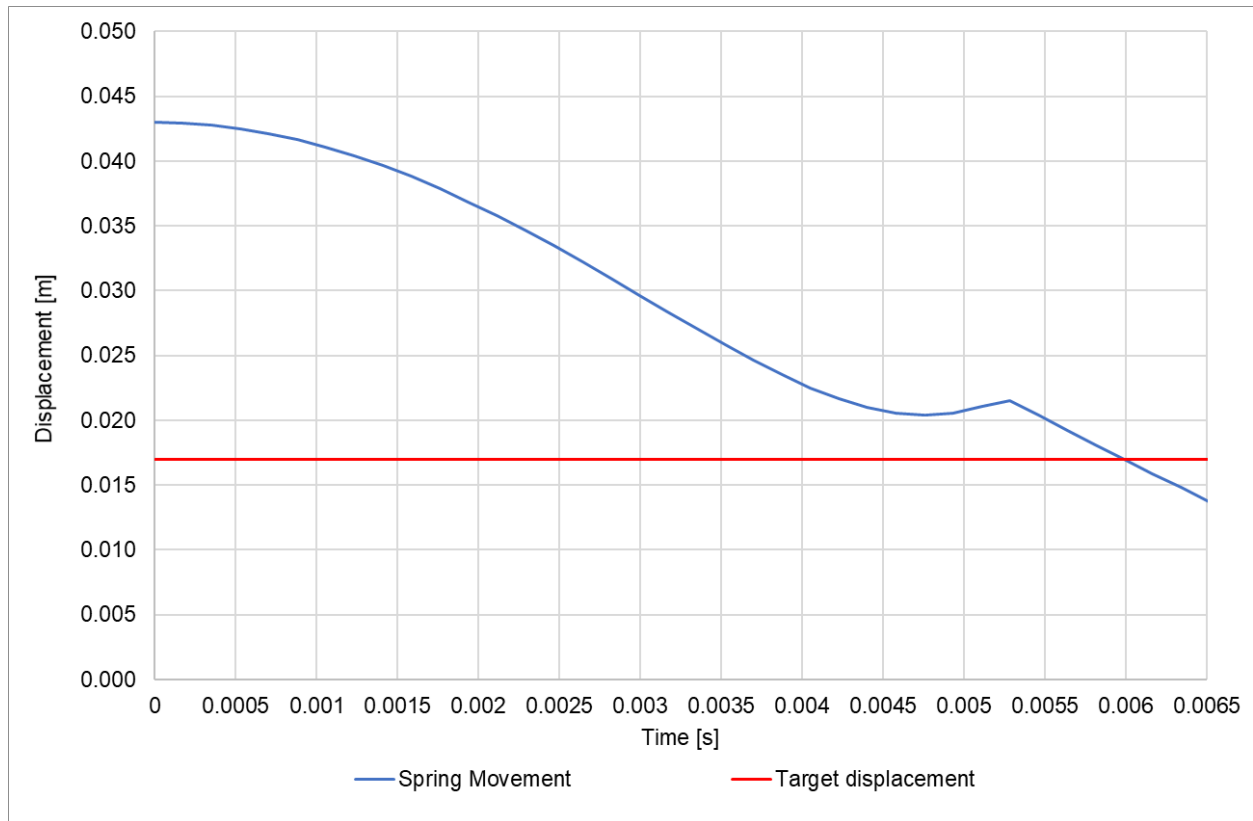


Figure 4-3: Designed valve closure over time.

Using a spring constant of $k = 25.78 \text{ kN/m}$, it is observed from Figure 4-3 above that the valve can theoretically close within $T_c = 5.9914 \text{ ms}$ at a flow velocity of 1 m/s . However, since a spring with the specified spring constant is commercially unattainable, springs of $k = 12.9 \text{ kN/m}$ are placed in a configuration of four parallel sets of two springs in series, displayed in Figure 4-4 below.

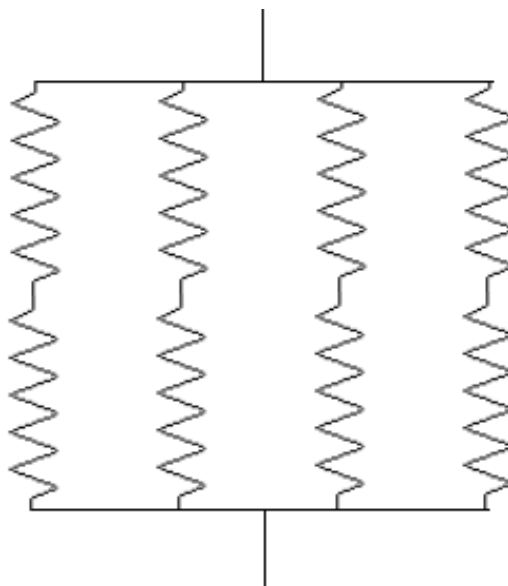


Figure 4-4: Illustration of spring arrangement in fast closing valve.

Utilising the series and parallel rules of springs, the effective spring constant can be calculated as:

$$k_{eff} = 2k = 25.8 \text{ kN/m} \quad (4.2)$$

Three random springs were selected in the batch and tested on a calibrated MTS hydraulic press to ensure that the spring behaviour and constant for all the springs in the batch were consistent. Figure 4-5 below displays the load versus displacement results of the three selected springs.

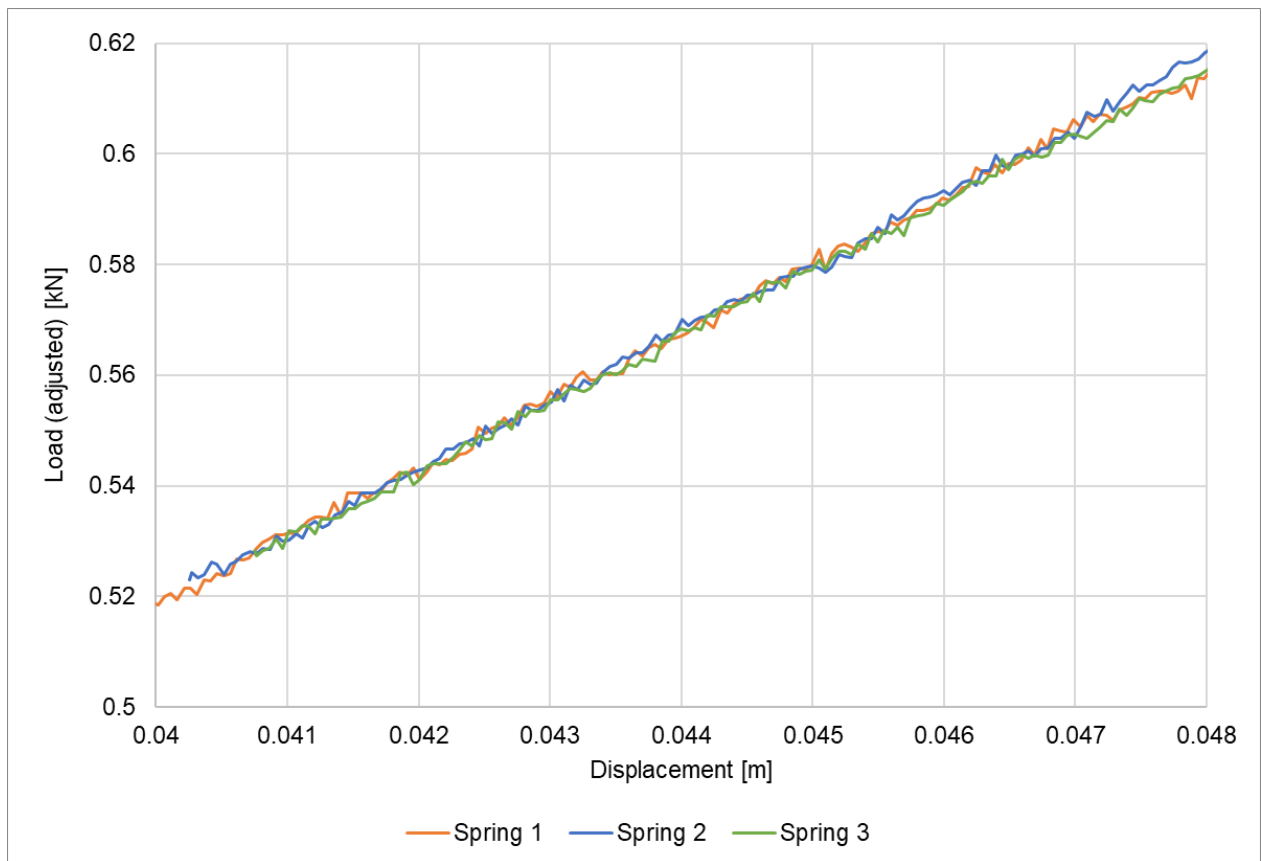


Figure 4-5: MTS results for the three randomly selected springs.

Figure 4-5 above shows that all three randomly selected springs compare well with each other and exhibit the same behaviour. The average spring constant calculated from the MTS results is:

Table 4.1: Calculated average spring constant.

Spring number	Average spring constant [kN/m]
Spring 1	12.91
Spring 2	12.90

Spring number	Average spring constant [kN/m]
Spring 3	12.87

The average spring constant of the three test springs is calculated as $k_{avg} = 12.89$ kN/m. It can be concluded that the springs used in the valve are all consistent and have a spring constant of 12.9 kN/m, as used in the design.

4.3.1.2 Fast closing valve static pressure probe design

The static pressure at the inlet of the fast-closing valve is of interest to this research effort, as the DGCM model is derived based on the static pressure head, and all experimental comparisons used in the open literature use the static pressure results for comparison. The static pressure is the actual thermodynamic pressure of a flowing fluid and is measured perpendicular to the flow field [67]. The total pressure of a flowing fluid is the sum of the static pressure and dynamic pressure and is measured parallel to the flow field [67].

Figure 4-6 below illustrates the difference between static and total pressure.

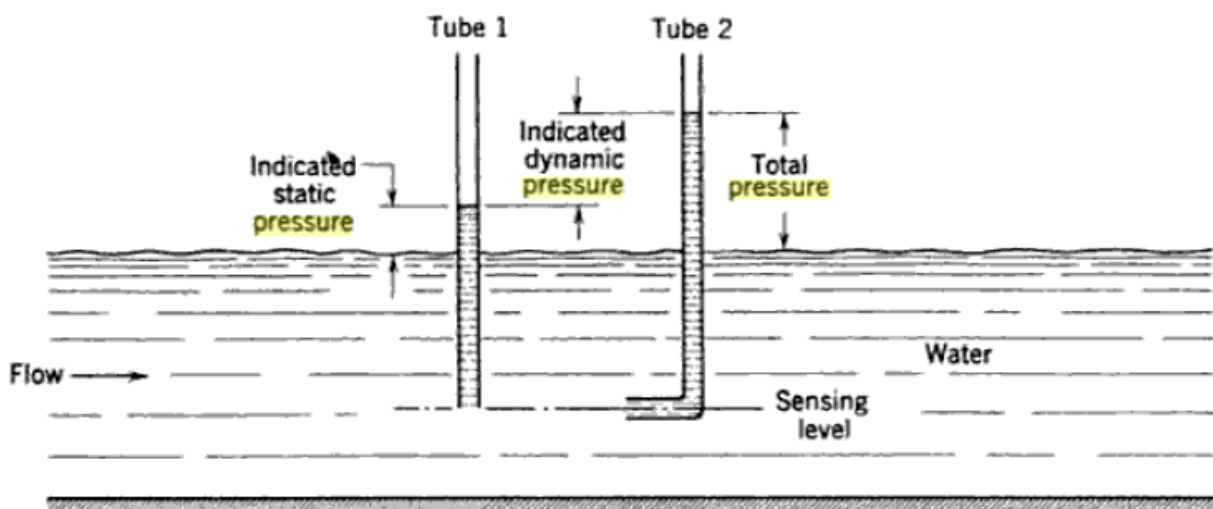


Figure 4-6: Static pressure and total pressure measurement illustration [68].

A static pressure probe is designed so that the pressure transducer connects to the valve and measures the pressure perpendicular to the flow field, measuring the static pressure at the inlet of the fast-closing valve. However, doing so creates space for errors to occur in the measurements [67]. Common sources of errors using a static pressure probe are eddies developing in cavities, fluid turbulence, Mach number effects, and stagnation of the fluid in the hole.

Eddies are formed when the shear stress of the flow boundary layer passes over the static pressure probe, forming recirculating flow within the cavity. This results in a higher static pressure measured in the cavity [69]. Burrs on the edge of the static pressure probe hole from manufacturing processes have a negative impact on the measurements and should be avoided if possible. Removing the burrs causes the fluid to separate more cleanly into the static pressure probe, and only minor deflections of the streamlines are present.

Following the recommendation of [69], the static pressure probe upstream of the fast-closing valve is designed based on the guidelines in Figure 4-7 below.

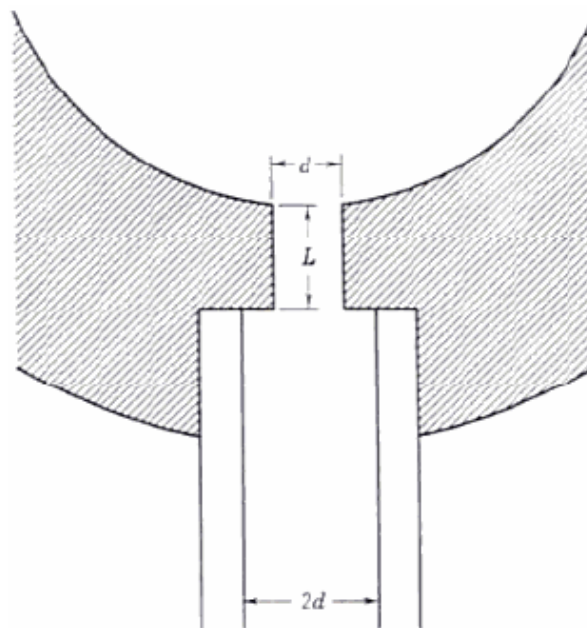


Figure 4-7: Recommended static pressure wall probe design, $1.5 < L/d < 6$, [69].

Doing this will ensure that the error in the pressure measurements using the static pressure probe is minimised if not mitigated altogether.

Annexure C contains selected manufacturing drawings and a video link of the manufactured valve.

4.3.2 Measuring equipment

4.3.2.1 Fast-closing valve pressure transducer

The pressure transducer located at the inlet of the fast-closing valve measures the water hammer with liquid column separation, which is used to compare the results of the EES model. The pressure transducer mentioned before is a WIKA 891.13.500 model; see the calibration certificate attached in Annexure D. The WIKA pressure transducer used at the inlet of the fast-closing valve ranges between 0 and 16 bar gauge pressure. According to the calibration certificate (Annexure D), the accuracy of this pressure transducer is reported to be 0.02 bar.

Since the water hammer with liquid column separation is a rapid transient event, the response time of the pressure transducer at the inlet of the fast-closing valve must be determined. This is done by connecting the transducer (PT) to a pressure chamber filled with water and producing an instantaneous force (F) on the plunger, creating an instantaneous pressure pulse.

Figure 4-8 below displays the test bench used to determine the response time of the pressure transducer.

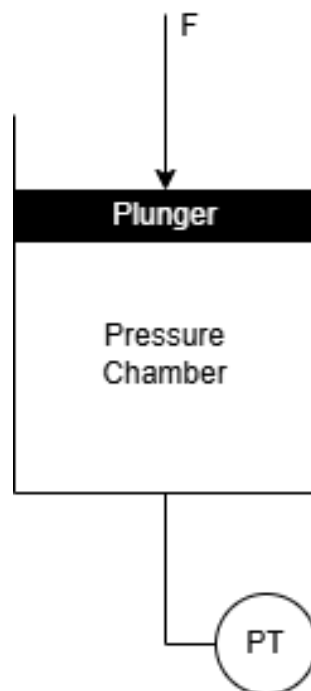


Figure 4-8: Fast closing valve pressure transducer response time test bench.

The results of the pressure response time test are presented in Figure 4-9 below.

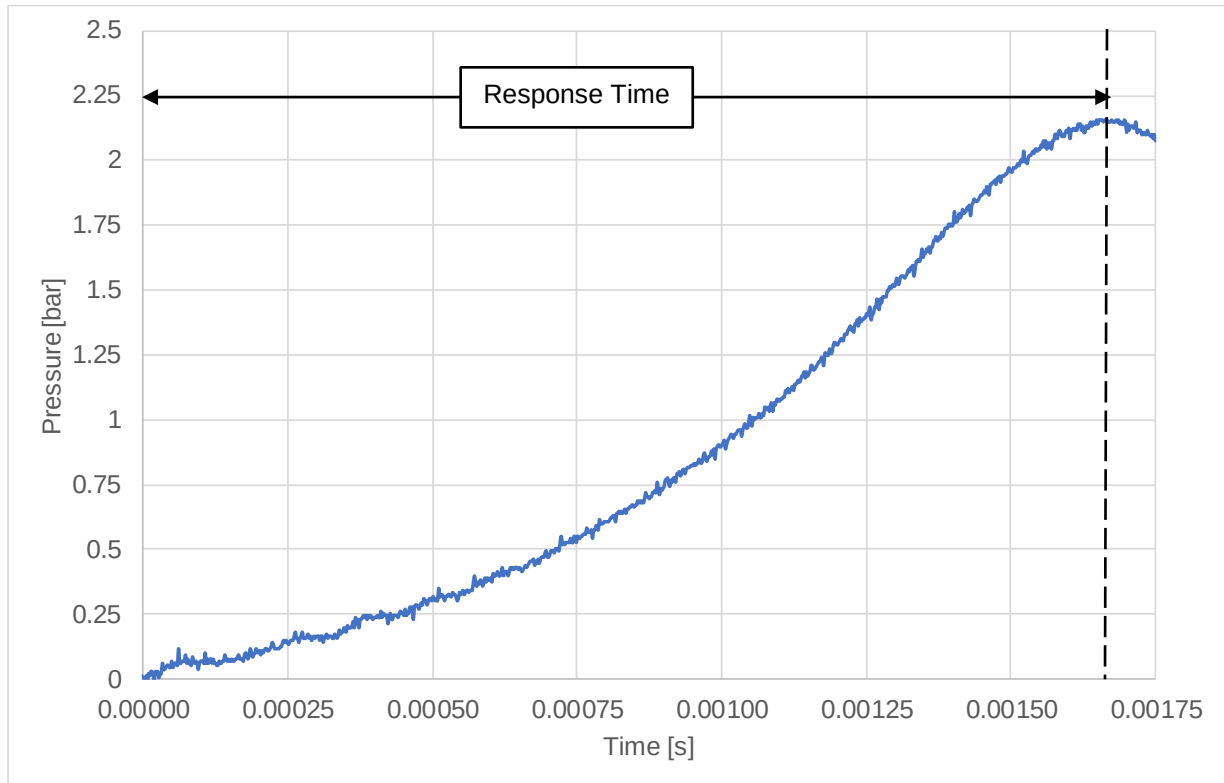


Figure 4-9: Fast closing valve pressure transducer response time test results.

From the test results presented in Figure 4-9 above, the response rate of the pressure transducer is measured to be around 1.66 ms. Figure 4-10 below displays the experimental results of [17], the red dashed lines indicate the time duration of a typical instantaneous pressure spike.

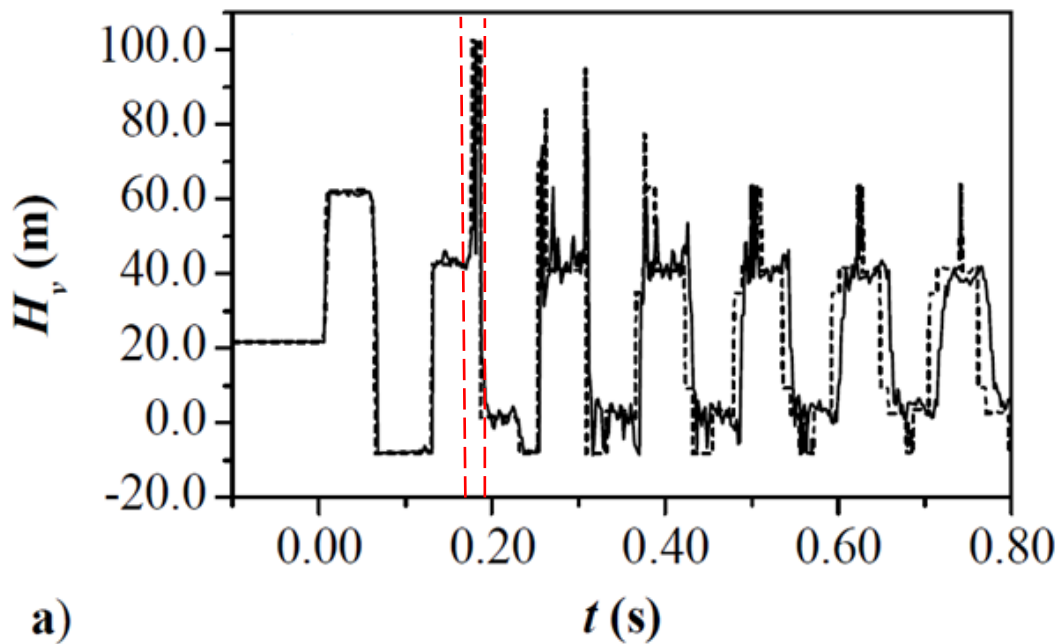


Figure 4-10: Experimental results of water hammer with liquid column separation [17].

From Figure 4-10 above, the instantaneous high-pressure pulse event, caused by the superpositioning of the water hammer pressure wave and cavity collapse pressure wave, lasts around 0.4 s (or 40 ms) from start to finish (see red dashed lines). Therefore, it can be concluded that the pressure transducer has a sufficient response time to capture a water hammer with liquid column separation event.

4.3.2.2 TiePie oscilloscope measuring loop.

The pressure transducer (PT) located at the inlet of the fast-closing valve (see Section 4.3.2.1 above) is connected to an oscilloscope used for data capture. The oscilloscope used is a TiePie Handyscope HS4-DIFF model (see datasheet attached in Annexure E), with a built-in sampling rate of 0.002 ms sufficient to capture a water hammer with liquid column separation event (see Section 4.3.2.1).

However, the scope only measures Volts, whereas the output of the pressure transducer is a current (4 mA – 20 mA). Therefore, a resistor of $R = 250\ \Omega$ is placed in the electrical circuit, ensuring that the 4 mA – 20 mA transducer output current is converted to a Volts output of 1V – 5V.

Figure 4-11 below displays the measuring loop of the TiePie oscilloscope used to capture data on the fast-closing valve pressure transducer.

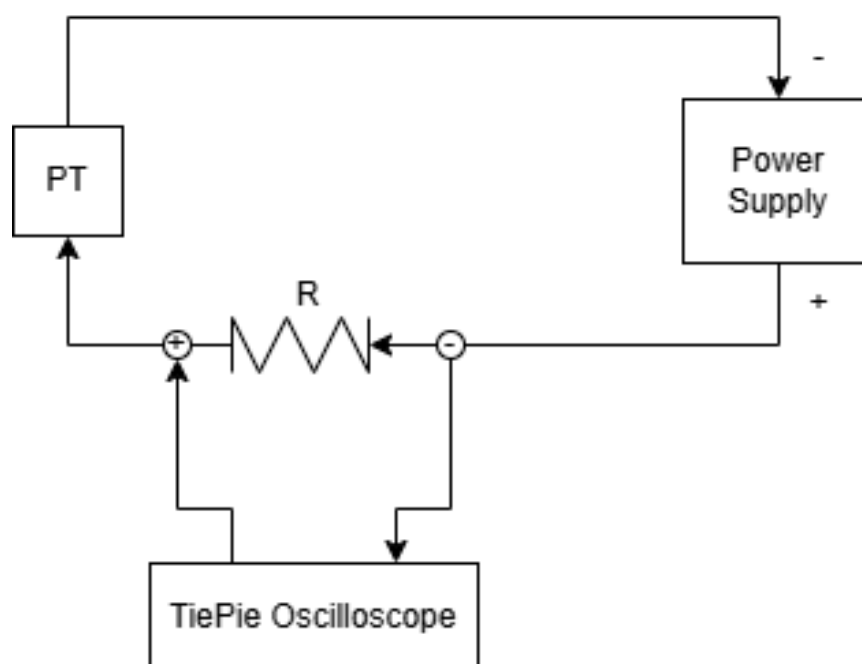


Figure 4-11: TiePie oscilloscope measuring loop.

The resistor (R) used is a Vishay 250Ω metal film resistor with a ±0.1% tolerance (see the data sheet attached in Annexure F). The Voltage output measured is post-processed to current, using Ohm's law, which can then be converted to pressure: 4 mA = 0 bar to 20 mA = 16 bar.

4.3.2.3 Flow rate calibration

A flow control valve controls the experimental test bench's flow rate (see Figure 4-2 above). A load cell calibrates the flow rate by measuring the total weight of water collected in the calibration tank over a certain time.

The load cell is calibrated by comparing laboratory weights to the weight measured by the load cell. Figure 4-12 displays the calibration results of the measured load cell weight versus the laboratory weight.

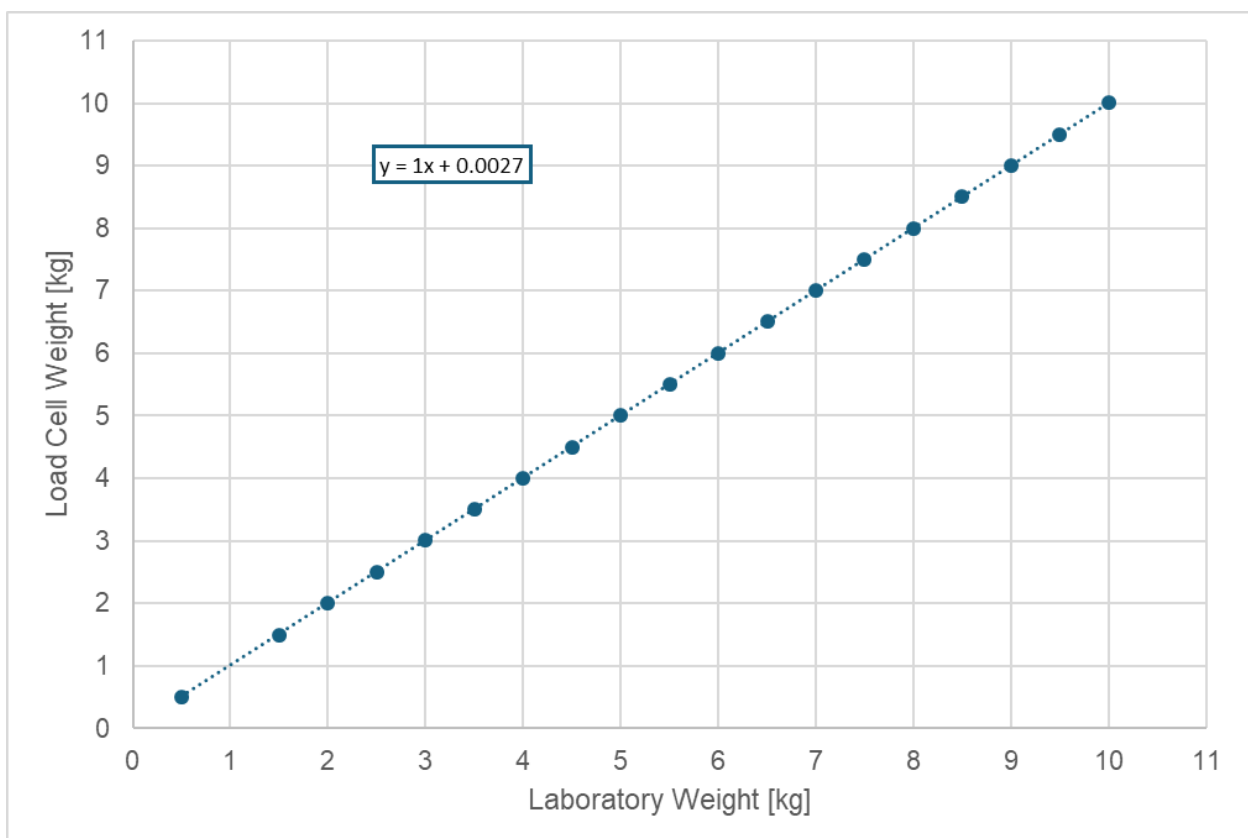


Figure 4-12: Load cell calibration, measured weight vs laboratory weight.

From Figure 4-12 above, the measured load cell weight compares well to the laboratory weight across the range tested (0.5 kg - 10 kg). The error of the estimated weight is calculated to analyse the load cell calibration analytically.

The calibration error is calculated using a normalised point difference, displayed in Figure 4-13 below, calculating the point difference between the measured weight and the laboratory weight.

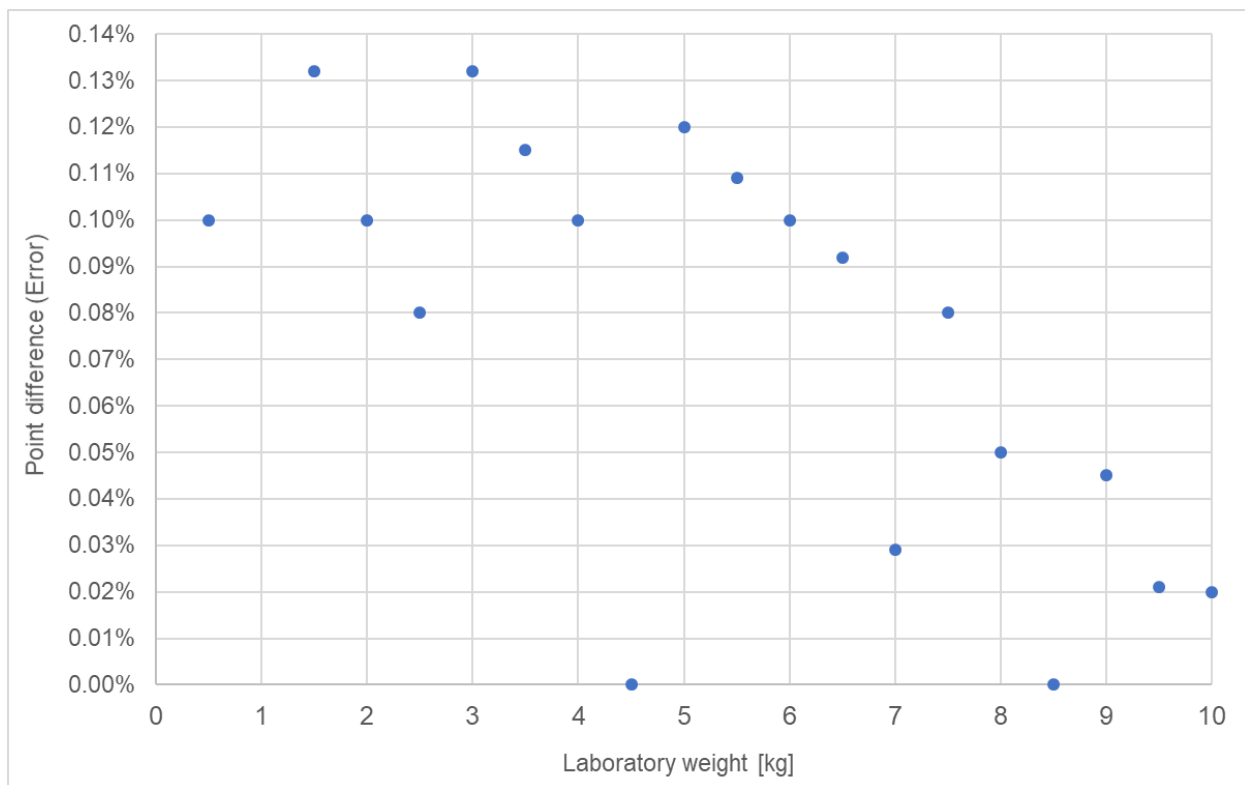


Figure 4-13: Load cell calibration, calculated point difference error.

From Figure 4-13 above, the maximum point difference error is calculated as 0.1333%. Therefore, it can be concluded that the load cell is accurate for its intended purpose. However, the linear best-fit curve in Figure 4-12 above will be used as a post-processing step.

4.3.2.4 Reservoir pressure transducer

A pressure transducer measures the reservoir pressure head at the straight galvanised steel pipe's inlet (see Figure 4-2). The pressure transducer used at the reservoir is a DRUCK pressure transducer, PTX1400 model, displayed through an Endress + Hauser monograph, M RSG40 model. The DRUCK pressure transducer used at the reservoir ranges between 0 and 6 bar gauge pressure.

To ensure the accuracy of the reservoir pressure transducer, it is calibrated on a laboratory deadweight test bench setup, using the calibrated pressure transducer of Section 4.3.2.1 as a reference. Deadweight test benches are known for superior accuracy and precision in calibrating

pressure measurement devices. The basic working principle of a deadweight test bench is that a known force (weight) is applied to the piston of an oil-filled chamber. The distributed force over the surface area of the piston generates pressure in the chamber.

Figure 4-14 below illustrates the calibration bench setup for calibrating the reservoir pressure transducer. PT_1 refers to the calibrated pressure transducer, and PT_2 refers to the uncalibrated reservoir pressure transducer.

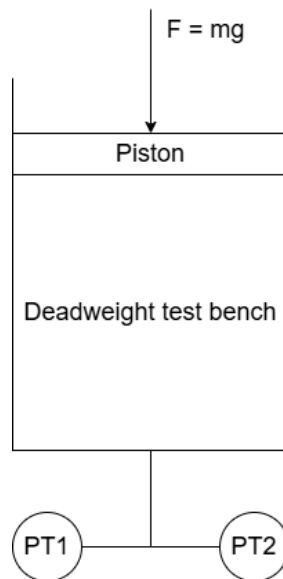


Figure 4-14: Illustration of calibration procedure of the reservoir pressure transducer.

The measurements from the calibration procedure of the reservoir pressure transducer are displayed in Figure 4-15 below.

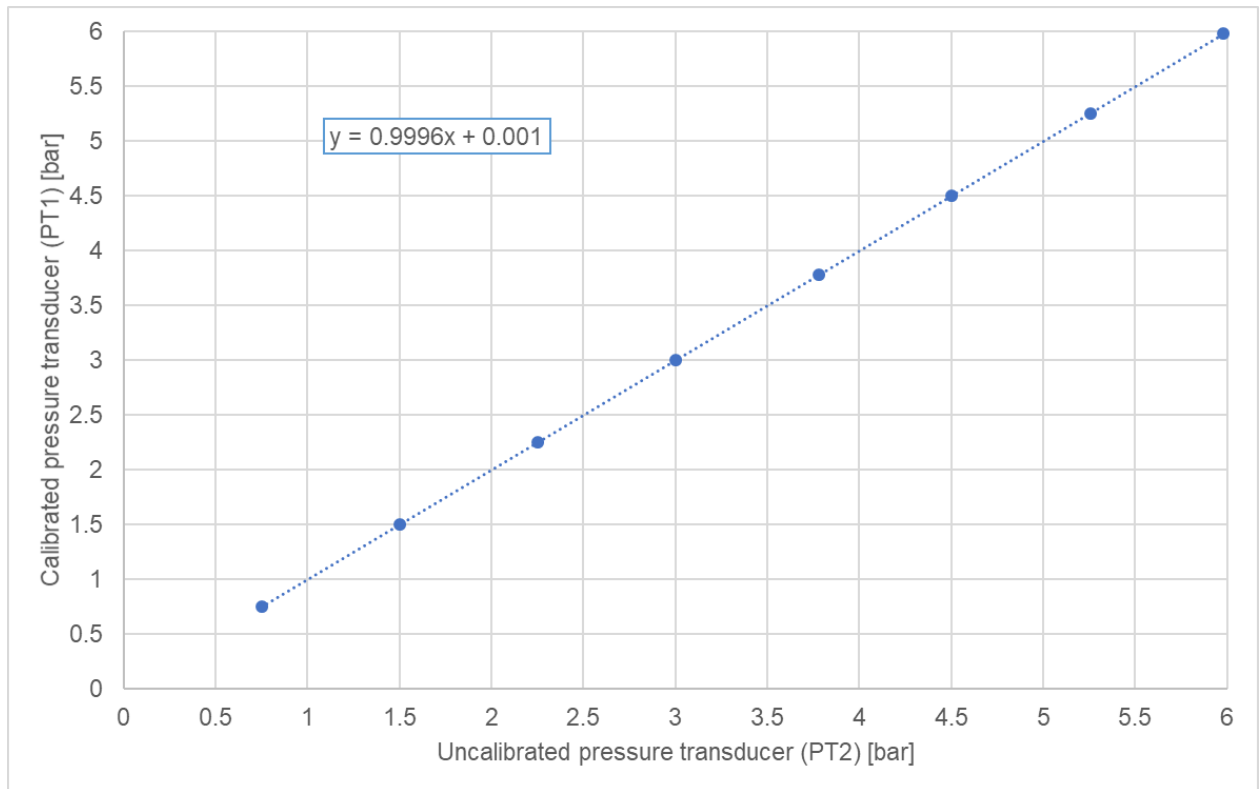


Figure 4-15: Reservoir pressure transducer calibration results.

From Figure 4-15 above, the measured pressure of the uncalibrated reservoir pressure transducer matches well with the results of the calibrated pressure transducer. The calculated point differences (errors) of the calibration measurements are displayed in Table 4.2 below.

Table 4.2: Point difference (error) calculated from reservoir pressure transducer calibration.

Measurement number	Point difference (error)
1	-0.0265%
2	-0.0492%
3	0.0263%
4	0.0828%
5	0.0365%
6	-0.0413%
7	-0.0437%
8	-0.0187%

From Table 4.2 above, the maximum error between the calibrated and uncalibrated pressure transducer is 0.0828%. Therefore, it can be concluded that the reservoir pressure transducer is accurate for its intended purpose. However, the linear best-fit curve in Figure 4-15 will be used as a post-processing step.

4.3.2.5 Barometric pressure transducer

The barometric pressure transducer is an uncalibrated WIKA S-10, with an absolute range of 0.8 – 1.2 bar. The pressure is displayed through an Endress + Hauser monograph, M RSG40 model. A sensitivity analysis of the effect of the barometric pressure on the simulation results is outlined and discussed in Section 5.2.4 below, justifying the use of the uncalibrated barometric pressure measurement.

4.3.2.6 Thermocouples

Six uncalibrated thermocouples measured the atmospheric and bulk water temperatures, respectively. The temperature measurements were displayed using an Endress + Hauser monograph, M RSG40 model. To mitigate the uncertainty of the uncalibrated thermocouples, the average of the measured water temperatures was used to determine the water's physical properties.

4.4 Conclusion

Chapter 4 outlines the experimental methodology and design for this research effort. The methodology addresses this dissertation's problem statement and research questions. To proceed with the proposed experimental methodology, an experimental test bench was constructed. This chapter discusses the design of the experimental test bench, more specifically, the design of the fast-closing valve and the measuring equipment used in the test bench, along with their calibration procedures.

In Chapter 5, the comparisons of the results are discussed. First, the parameters measured and calculated from the experiment are presented. A discussion of the sensitivity analysis results on the effect of the weighing factor on the grid and time step independence of the DGCM numerical model follows. This is followed by the key findings from the experimental procedure, after which the comparison of the simulation results for various weighing factors and experimental results are presented and discussed.

CHAPTER 5 RESULTS COMPARISON

In this chapter, the results are compared and discussed. This includes the measurements and calculations of the input parameters, followed by the sensitivity analysis conducted on the grid and time step size independence for various DGCM free gas volume weighing factors. Lastly, the crucial findings made in the experimental results and the comparison between the measured experimental results and the DGCM EES model results for various weighing factor values are provided and discussed.

5.1 Introduction

The previous chapter outlines the experimental methodology for this research effort. The methodology was set up to address the problem statement and research questions posed in Sections 1.2 and 1.3, respectively. This is followed by a discussion of the design of the experimental test bench used to generate experimental results. More specifically, the design of the fast-closing valve is discussed in detail, followed by an overview of the measuring equipment used and the calibration procedure thereof.

This chapter aims to outline and compare the results of this research effort. First, the calculations and measurements made from the experimental procedure used for the input parameters of the EES simulation are presented. The sensitivity analysis on the effect of the weighing factor follows this. Lastly, the crucial findings from the experimental results are outlined and discussed, after which a comparison of the experimental results and EES simulation results is presented and discussed.

The chapter outline is as follows: Section 5.2 presents the calculated and measured parameters used as input of the EES model. Section 5.3 follows with a discussion of the sensitivity analysis on the effect of the weighing on the EES simulation results. Lastly, Section 5.4 compares the experimental and simulation results, Section 5.4.1 discusses the crucial findings made from the experimental results, and Section 5.4.2 presents and compares the experimental and simulation results.

Chapter 6 summarizes the study and its main findings, makes recommendations, and presents the study's concluding remarks.

5.2 Input parameters measurements and calculations

For parameter measurements where multiple measurements are taken, the average of the set of measurements is taken as the value for the parameter. The tolerance around the average is calculated using the student-t method described in [70] using a confidence level of 95%.

5.2.1 Flowrate measurement

During the experimental procedure, six weight measurements were made for 30 seconds once the system was in steady flow conditions. The average of these weight measurements was taken and converted to the flow rate.

Table 5.1 below presents the results of the measured weight, the adjusted weight from the load cell calibration, the calculated mass flow, and the flow rate during the experimental procedure.

Table 5.1: Results of measured weight, calculated mass flow, and calculated flow rate.

Measurement	Actual weight measurement [kg.30s]	Adjusted weight measurement [kg.30s]	Calculated mass flow [kg/s]	Calculated flow rate [m ³ /s]
1	11.387	11.3897	0.3797	0.0003797
2	11.398	11.4007	0.3800	0.0003800
3	11.368	11.3707	0.3790	0.0003790
4	11.415	11.4177	0.3806	0.0003806
5	11.378	11.3807	0.3794	0.0003794
6	11.408	11.4107	0.3804	0.0003804
Average	11.392 ± 0.019	11.3950 ± 0.0137	0.3798 ± 0.0005	3.7983E⁻⁰⁴ ± 2.5249E⁻⁰⁶

The calculated average flow rate during the experimental procedure, using the method described in Section 4.3.2.3, was calculated as 3.7983E⁻⁰⁴ m³/s, with a computed tolerance of 2.5249E⁻⁰⁶ m³/s.

5.2.2 Reservoir pressure measurement

During the experimental procedure, five reservoir pressure measurements were made over several minutes when the system was in steady flow conditions. The actual reservoir pressure measurements, the adjusted reservoir pressure from the pressure transducer calibration, and the calculated reservoir head are displayed in Table 5.2 below.

Table 5.2: Results of measured reservoir pressure and calculated reservoir pressure head.

Measurement	Actual pressure measurement [kPa]	Adjusted pressure [bar]	Calculated pressure head [mH ₂ O]
1	78.4	0.7847	8.0016
2	78.3	0.7837	7.9914
3	78.1	0.7817	7.9710
4	78.5	0.7857	8.0118
5	78.4	0.7847	8.0016
Average	78.3400 ± 0.1883	0.7841 ± 0.0005	7.9955 ± 0.0485

The average calculated reservoir head from the pressure measurements is 8.0143 mH₂O, with a computed tolerance of 0.0481 mH₂O.

5.2.3 Atmospheric and bulk water temperature measurement

During the experimental procedure, six independent thermocouples were used to measure the atmospheric and bulk water temperatures. The average of the measured water temperature is used to determine the water's physical properties. Meanwhile, the pipe's material mechanical properties are assumed to be constant.

The results of the measured water temperatures are depicted in Table 5.3 below.

Table 5.3: Results of measured atmospheric temperatures.

Thermocouple	Temperature [°C]
T ₁	27.2
T ₂	27.0
T ₃	26.6
T ₄	27.2
T ₅	26.9
T ₆	27.4
Average	27.0500 ± 0.2950

The average measured atmospheric temperature is 27.05°C, with a calculated tolerance of 0.2950°C.

The results of the measured water temperatures are depicted in Table 5.4 below.

Table 5.4: Results of measured bulk water temperatures.

Thermocouple	Temperature [°C]
T ₁	14.2
T ₂	14.1
T ₃	13.9
T ₄	14.3
T ₅	14.4
T ₆	13.8
Average	14.1167 ± 0.2432

The average bulk water temperature is 14.1167°C, with a calculated tolerance of 0.2432°C.

The physical properties of the water used during the experimental procedure are assumed to be that of general water at 14.12°C from [71]. Table 5.5 below presents the assumed physical properties of the water used in the experimental test bench.

Table 5.5: Physical properties of general water at 14.12°C.

Property	Value	Unit
Density	999.2	kg/m ³
Viscosity	0.001168	kg/m.s
Bulk Modulus	2.14	GPa

The mechanical properties of the galvanised steel pipe used during the experimental procedure are assumed to be constant. Table 5.6 below presents the assumed material properties of the galvanised steel pipe used in the experimental test bench obtained from [72].

Table 5.6: Material properties of the pipe (galvanised steel).

Property	Value	Unit
Young's Modulus	200	GPa
Poisson Ratio	0.29	-
Relative Roughness	15	µm

5.2.4 Barometric pressure

A calibrated absolute pressure transducer was not available during the testing. Hence, the uncalibrated pressure transducer was used to measure the barometric pressure, and the results were compared to the barometric pressure from a weather report [73]. The pressure transducer is a WIKA S-10, with a 0.8 – 1.2 bar absolute range. The pressure is displayed through an Endress + Hauser monograph, M RSG40 model.

Using the measured barometric pressure as a baseline, the difference between the measured and reported pressure is calculated. From this, a sensitivity analysis was conducted on the DGCM EES model to determine the effect of the atmospheric pressure on the simulation results, where the calculated difference serves as the sensitivity analysis range.

The measured barometric pressure was 87.9 kPa, and the reported barometric pressure on 10/10/2024 was 656 mmHg or 87.5 kPa, [73]. The calculated difference is 0.4 kPa or 0.5%. The sensitivity analysis compared the results of three simulation runs, using the input parameters of the experimental test bench, at a barometric pressure of 87.5 kPa, 87.9 kPa, and 88.3 kPa. The parameters used for the sensitivity analysis are the water hammer pressure head (H_1), the cavity collapse pressure head (H_2), and the time of cavity collapse (t_{collapse}).

The barometric pressure sensitivity analysis results are displayed in Table 5.7 below.

Table 5.7: Barometric pressure sensitivity analysis results.

Parameter	Barometric pressure			Point difference (lower range)	Point difference (upper range)
	87.5 kPa	87.9 kPa	88.3 kPa		
H_1 [mH ₂ O]	98.7839	98.7839	98.7840	0.0000%	0.0000%
H_2 [mH ₂ O]	118.8920	118.8624	118.8442	0.0252%	-0.0153%
t_{collapse} [s]	0.15711	0.15711	0.15711	0.0000%	0.0000%

The sensitivity analysis results from Table 5.7 revealed that the only notable difference is the cavity collapse pressure head, with a maximum point difference of 0.0252% calculated for the lower range. Therefore, the barometric pressure does not significantly impact the simulation results, and the barometric pressure measured with the uncalibrated absolute pressure transducer can be used. Although it is noted that this is not the most accurate approach, from the sensitivity analysis, the impact of the atmospheric pressure is negligible on the simulation results, within reason.

5.2.5 Valve closure time calculation

Since the spring-mass mathematical model discussed in Annexure B requires all dynamic forces acting upon the piston during the valve closure (including the pressure rise during valve closure), an estimated pressure-time trace must be obtained using the EES model. To obtain the estimated

pressure-time trace, using the input parameters outlined in Section 5.2.6, the initial valve closure time was estimated by dividing the period of the pipe by a factor of four, see discussion in Section 4.3.1.1. From this, an actual valve closure time could be approximated.

Figure 5-1 below displays the displacement time trace of the mass-spring model using the inputs of Table 5.8 and an initial valve closure time of $T_c = 5.9914$ ms.

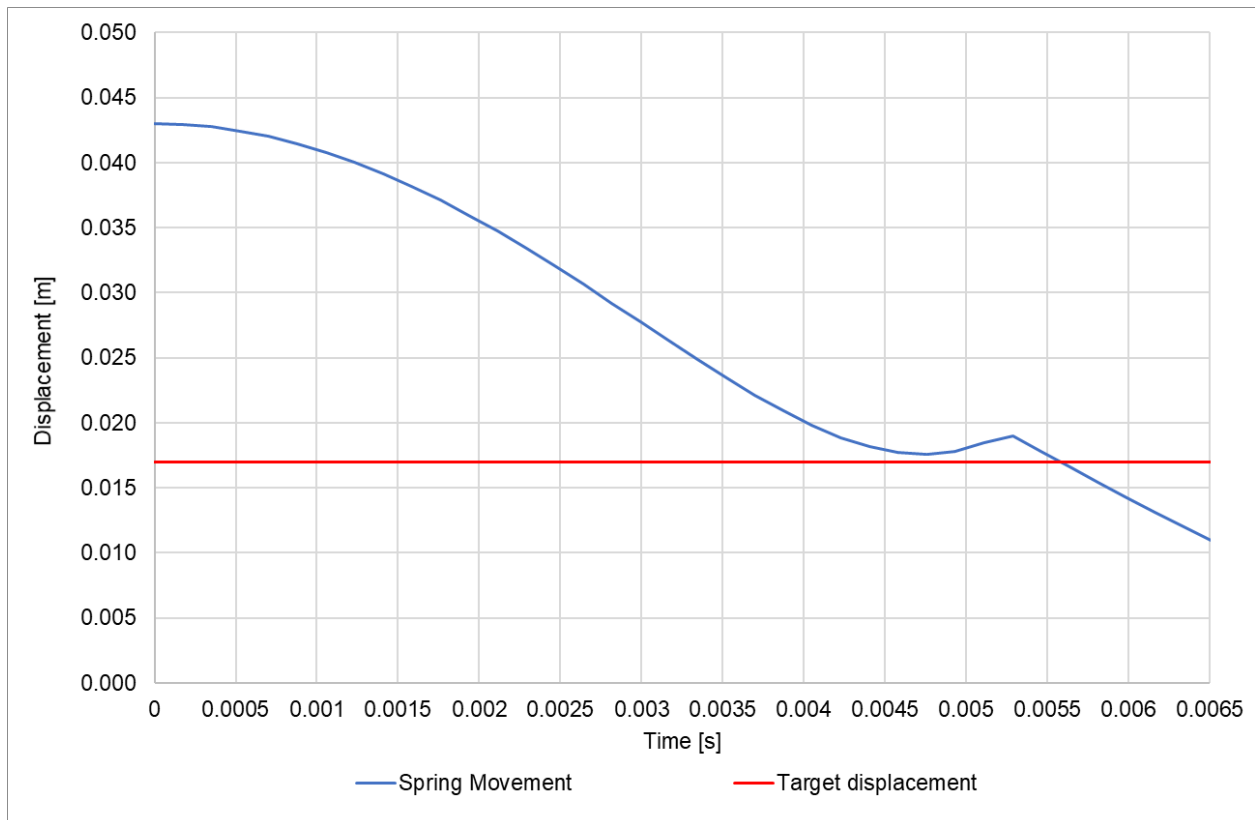


Figure 5-1: Approximated valve closure time of the experimental test bench.

From Figure 5-1 above, the actual valve closure time can be approximated as $T_c = 0.0053$ s.

5.2.6 Summary of EES model input parameters

The EES model input parameters are categorised into the following categories:

- Measured parameters: These include parameters measured using the measuring equipment presented in Section 4.3.2.

- Assumed parameters: Includes the fluid physical properties and pipe mechanical properties (assumed constant) at the measured temperatures (see Section 4.3.2), using [71] and [72] respectively.
- Calculated parameters: These include theoretically calculated parameters, such as the pipe slope (through the design and manufacturing of the pipe supports) and the valve closure time (through the theory discussed in Section 4.3.1.1).
- Numerical model-specific parameters include parameters specific to the DGCM numerical model. The suggested values for these parameters are obtained from the literature.

The input parameters of the EES model are presented and categorised in Table 5.8 below.

Table 5.8: EES model input parameters.

Parameter	Value	Unit
<i>Measured parameters</i>		
Pipe length	15.3 ± 0.0046	m
Pipe internal diameter	27.4 ± 0.03	mm
Pipe wall thickness	2.7 ± 0.03	mm
Volumetric flowrate	$3.7983\text{E-}04 \pm 5.9951\text{E-}07$	m^3/s
Reservoir head	7.9955 ± 0.0485	m H ₂ O
Barometric Pressure	87.9 ± 0.4	kPa
<i>Assumed parameters</i>		
Pipe wall roughness	1.5E-04 m	m
Pipe Young's modulus	200 GPa	GPa
Pipe Poisson ratio	0.29	-
Fluid Density	999.2 kg/m ³	kg/m^3
Fluid bulk modulus	2.14 GPa	GPa
Fluid viscosity	11.68E-02 kg/ms	kg/ms
Fluid saturation pressure	0.1630	m
<i>Calculated parameters</i>		
Valve closure time	0.0053	s
Pipe slope	0°	°
<i>Numerical model-specific parameters</i>		
Valve closure characteristic	1	-
Void fraction	1E-07	-
Courant number	1	-

5.3 Sensitivity analysis (effect of weighing factor) results

As discussed in Section 3.2.4, time step and grid size independence are crucial to the accuracy and stability of numerical CFD models. The effect of the DGCM free gas volume weighing factor (Ψ) on the grid and timestep independence of the DGCM numerical model is mostly alluded to in the open literature.

To address the research question, in Section 1.3, of the effect of the DGCM free gas volume weighing factor on the grid and time step size independence; a grid refinement study (sensitivity analysis) was conducted on the EES model for simulation runs of various free gas volume weighing factors ($\Psi = 0.5, 0.6, 0.7, 0.8, 0.9$ and 1.0), using the grid and time step independence theory presented in Section 3.2.4.1.

The parameters used for the grid refinement study are the maximum pressure head of the second pressure wave (cavity collapse-induced short-duration pressure pulse) and the time occurrence of said maximum pressure head (time of cavity collapse). Assuming a mesh error of 1%, the grid refinement study results are displayed in Figure 5-2 below.

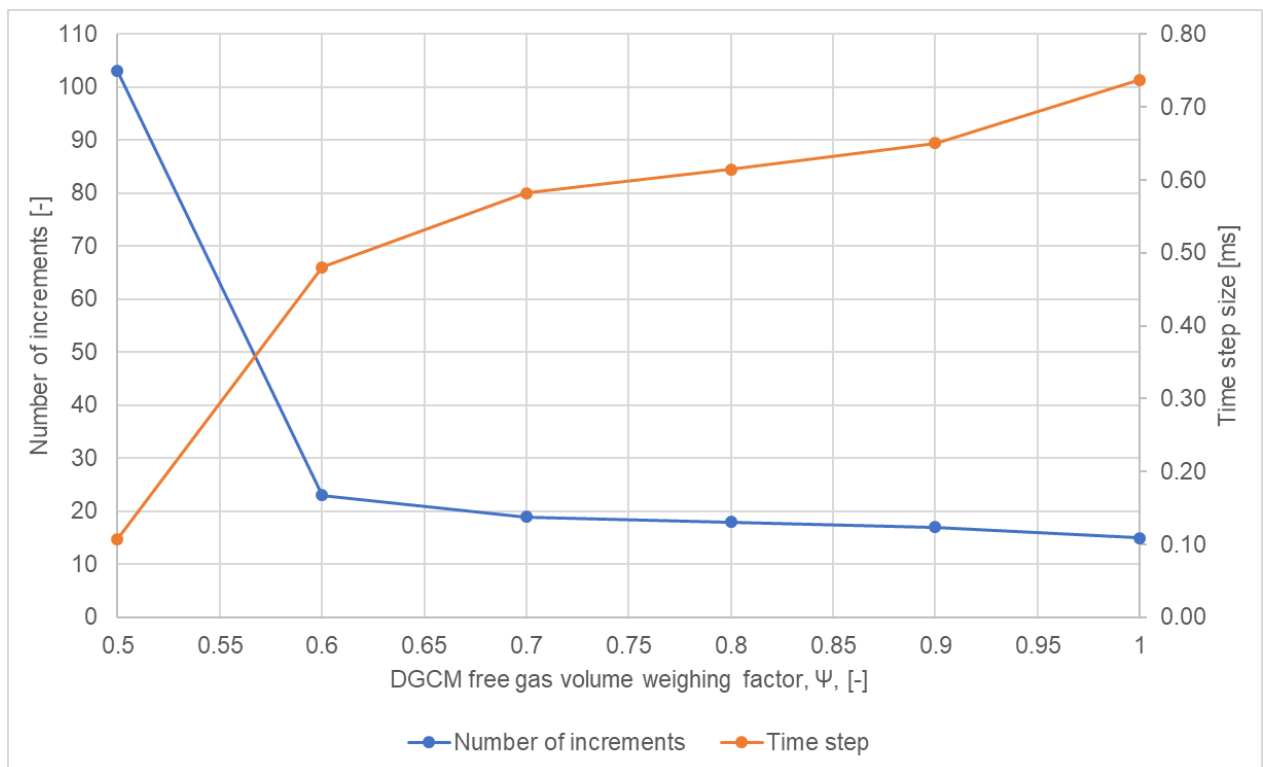


Figure 5-2: Sensitivity analysis on the effect of weighing factor results.

From Figure 5-2, the grid refinement study concludes that as the DGCM free gas volume weighing factor increases, so does the stability of the numerical model. For higher values of the weighing factor, the model reaches grid and time step independence at a larger grid size (fewer increments) and a larger time step size.

As discussed in Section 3.2.4, larger grid and time step sizes directly correlate to faster numerical model solving times. Therefore, a larger grid and time step size can be used for higher weighing factor values while maintaining model stability and leading to faster solving time.

5.4 Experimental results and simulation results comparison

This section discusses the critical findings made during the experimental procedure. After this, the results comparison of the experimental results and simulation results are presented and discussed.

5.4.1 Experimental results findings

5.4.1.1 Effects of entrapped air on measured data (unpurged system)

During the commissioning phase of the experimental test bench, the system was filled with water from top to bottom. During the filling procedure, it was observed that air bubbles became entrapped in the clear 1" flexi hose. This is due to a force balance achieved by the water flowing down and the buoyancy force of the air pushing the air against the flow direction. However, eventually, it seemed that all the air was purged from the system. A preliminary experimental test run was conducted to determine the success of the commissioning phase.

Figure 5-3 below displays the results of the preliminary experimental test run.

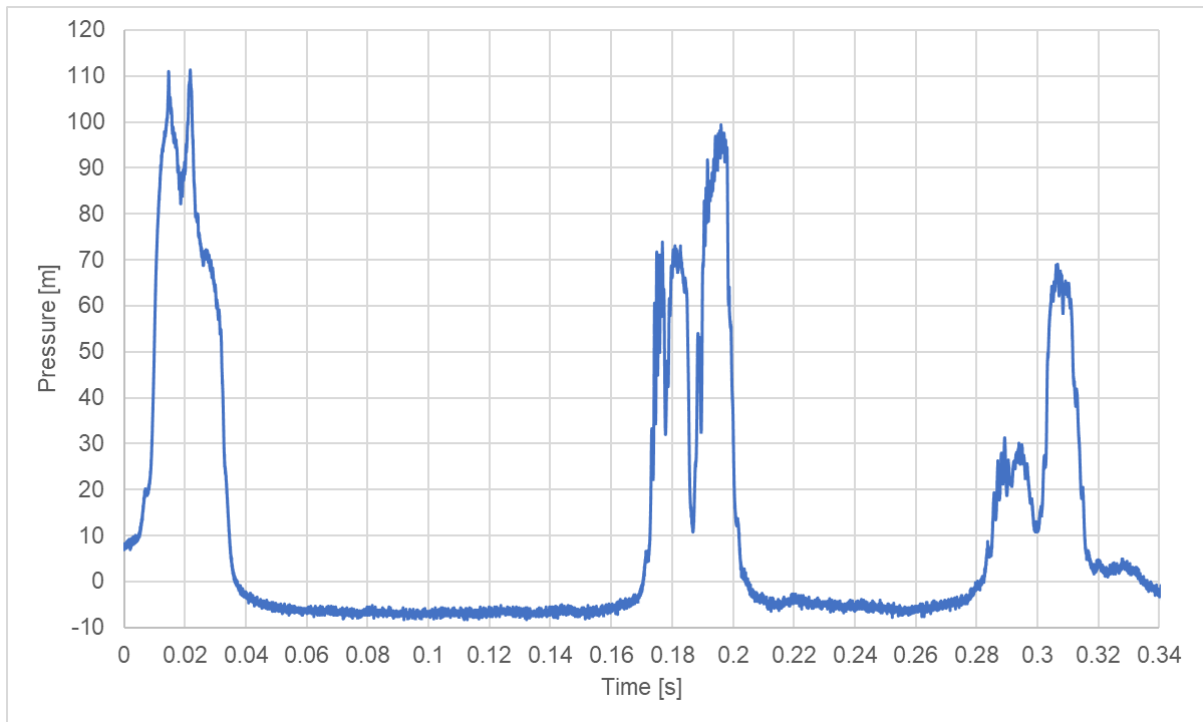


Figure 5-3: Preliminary experimental test run results.

Upon analysing the results (Figure 5-3 above), it is evident that the preliminary experimental results do not exhibit the same behaviour as what is expected from a water hammer with liquid column separation event, as seen in the literature. Considering that air became entrapped during the filling phase, it was hypothesised that large quantities of entrapped air remain in the system.

Figure 5-4 below displays the results of [74], where the authors tested the effects of entrapped air on pressure surges for various volumes (v) of entrapped air.

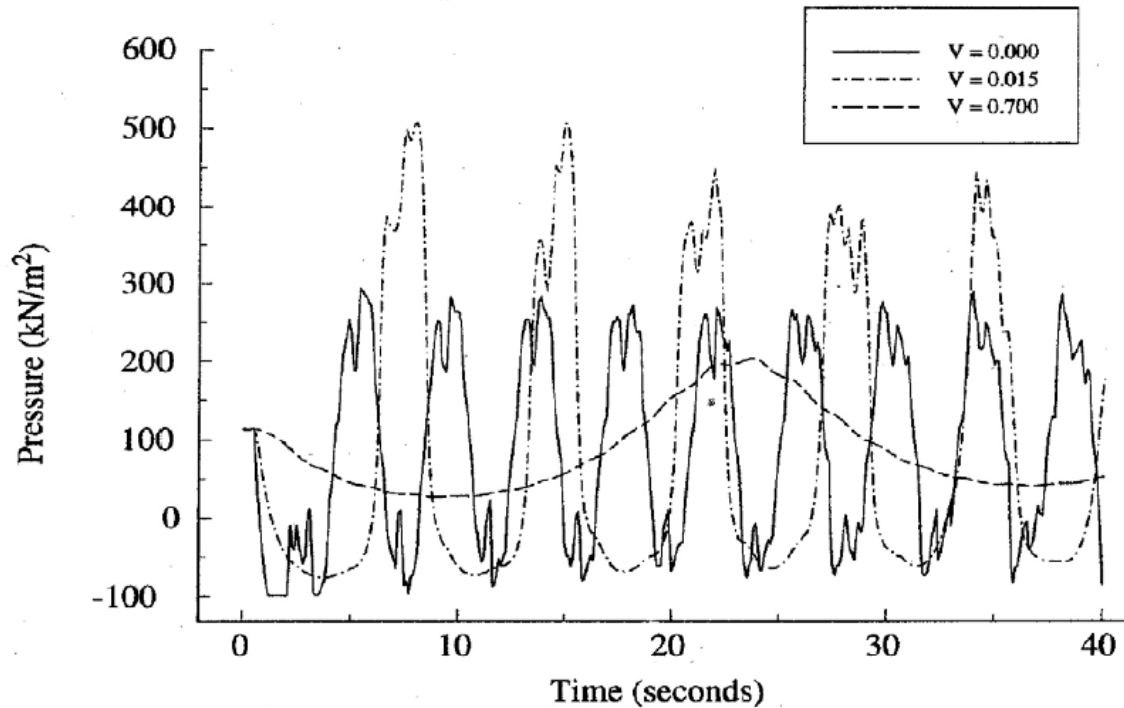


Figure 5-4: Effects of entrapped air on pressure surges in a pipeline [73].

The results from Figure 5-4 above exhibit the same behaviour as that found in the preliminary experimental results, shown in Figure 5-3 above. According to [73] the presence of air within a pipeline system can significantly impact pressure surges, either beneficially or detrimentally. Entrapped air increases the elasticity of the water column, considerably affecting the speed of the pressure wave. Upon investigating the comparison between the preliminary experimental results and the illustration of [73], it was concluded that a different technique should be used when filling the system to purge it of entrapped air pockets.

This was done by adding an air bleed valve on the top of the bottom collection tank, refer to Figure 4-2, and filling the system from the bottom (i.e. at the outlet) to the top (i.e. the open container where the float valve is located). This ensures that (1) little to no air is entrapped within the bottom collection tank and (2) the buoyancy force of the air aids in the flow of air pockets in the direction of the flow.

Figure 5-5 in Section 5.4.1.2 below discusses the experimental results obtained after the abovementioned changes were made and the system was correctly purged of entrapped air pockets.

5.4.1.2 Effects of fluid-structure interaction (FSI) on measured data

Favourable experimental results were obtained after the experimental test bench was adequately purged of air. However, spontaneous erratic behaviour was noticed in the measured experimental results (see red encircled area in Figure 5-5 below).

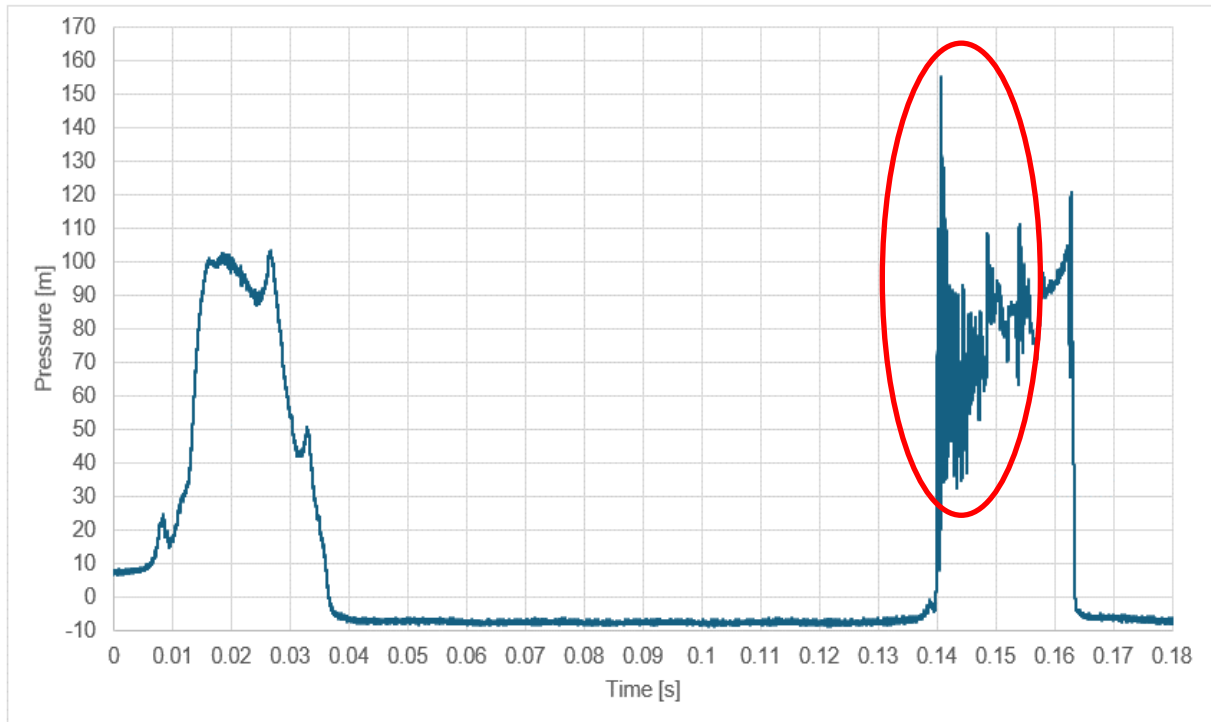


Figure 5-5: Spontaneous erratic behaviour in measured experimental results.

First, note from Figure 5-5 above that even when mitigation measures were put in place to purge the system of free air (see Section 5.4.1.1), the results indicate that there was still some residual free air left in the system when experimenting (refer to the first pressure wave of Figure 5-5). From the results, it can be deduced that the air in the system after mitigation is far less than that presented in Figure 5-4 (before mitigation). Therefore, it can be concluded that although the residual air in the system impacts the experimental results, it is not significant.

The spontaneous, erratic behaviour exhibited in the red encircled area of Figure 5-5 above is synonymous with that of the effects of dynamic FSI on hydraulic transients, as seen in the literature [75]. Dynamic FSI can be defined as “the transfer of momentum and forces between the pipe wall and the contained fluid during unsteady flow” [75]. The effects of FSI are not accounted for in the classic water hammer equations and subsequently also not accounted for in the numerical models derived from them, such as the DGCM model.

The degrees of freedom (DOF) of a mechanical system describes the position or configuration of the system at any time in a 3-D space. The DGCM numerical model assumes a 1-D approach for the fluid, i.e. pressure propagates axially for the fluid only in its derivation. This is a reasonable approach since pipes are slender elements [75]. In reality, transient forces within a pipe cause the pipe system to move in 3-D space. The transient forces on the pipe wall during a hydraulic transient event can cause the pipe to move in the translation DOF (heaving, swaying, and surging) and in the rotation DOF (pitching, yawing, and rolling).

Figure 5-6 below displays the DOF mentioned above of an infinitesimal control volume of a pipe.

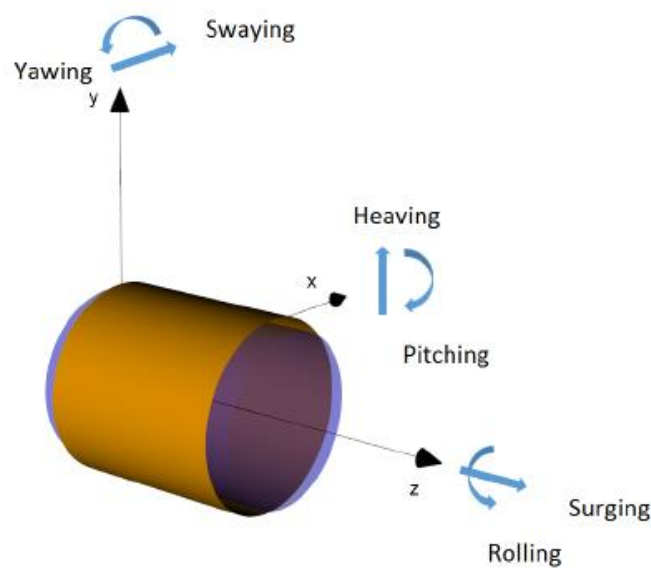


Figure 5-6: Illustration of the DOF of a pipe section [75].

According to [76] the pressure variations cause circumferential stresses in the pipe wall and, due to the Poisson effect, also cause longitudinal stresses in the pipe wall. These longitudinal stresses propagate along the length of the pipe much faster than the transient pressure wave [76]. This causes the pipe to move within its DOF, leading to high-frequency pressure oscillations in the system.

Figure 5-7 below displays the experimental results conducted by [76] on dynamic FSI.

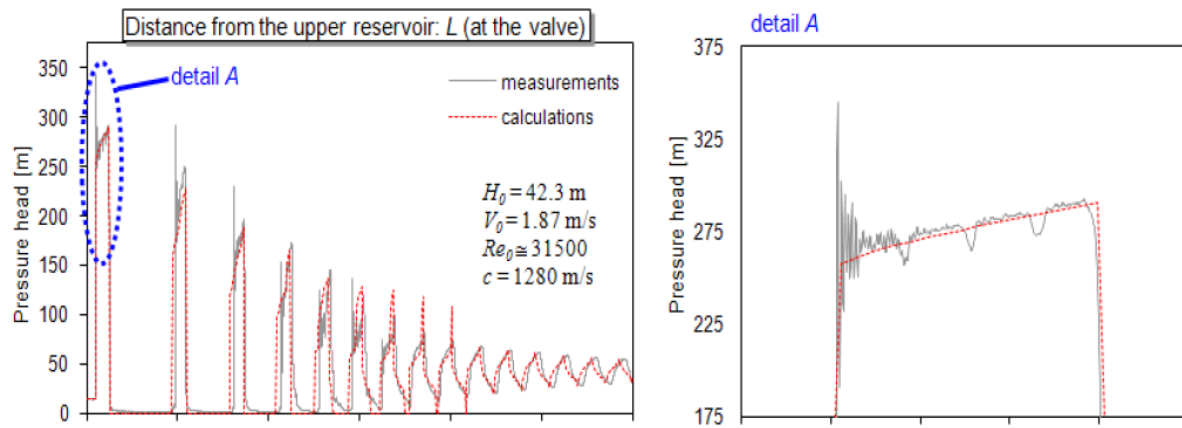


Figure 5-7: Experimental results of FSI conducted by [76].

Figure 5-7 shows that the FSI experimental results of [76] display the same characteristics as the experimental results obtained from the test bench, see Figure 5-5. Therefore, it can be concluded that the spontaneous erratic behaviour seen in the experimental results can be attributed to the dynamic FSI effects of the test bench structure.

It should be noted that FSI is an entire research field on its own, and numerical models exist in the open literature for which these FSI effects are accounted. However, it is not in the scope of this research effort and will therefore not be accounted for in the numerical model or mitigated on the experimental test bench. It is, however, essential to note the existence of FSI in the experimental results obtained from the test bench.

5.4.2 Experimental results and EES simulation results comparison

To address the research question posed in Section 1.3 about whether a baseline exists for the DGCM free gas volume weighing factor, experimental and simulation results were compared. The experimental results obtained from the constructed experimental test bench (refer to Section 4.3) were compared to the simulation results of the EES model for various weighing factors.

5.4.2.1 EES model Grid size and time step size

Table 5.9 below lists the grid size and time step size used for each of the EES models used to compare results. These grid and time step sizes follow the grid refinement (sensitivity analysis) conducted in Section 5.3.

Table 5.9: Result comparison EES model grid refinement study results.

Mesh Error [%]	Number of increments [-]	Time step size [ms]
$\Psi = 0.5$		
1E+00	103	0.1073
$\Psi = 0.6$		
1E-04	76	0.1455
$\Psi = 0.7$		
1E-04	68	0.1626
$\Psi = 0.8$		
1E-05	108	0.1024
$\Psi = 0.9$		
1E-05	68	0.1626
$\Psi = 1.0$		
1E-05	66	0.1675

Note from Table 5.9 above that as the DGCM free gas volume weighing factor increases, the mesh error decreases, where the number of increments and the time step size are of the same order. Larger desired mesh errors were used for smaller values of the weighing factor to avoid the number of increments getting too large and the time step size getting too small, as EES only has a limited memory it can work with.

5.4.2.2 Experimental results and EES simulation results

Using the parameter inputs listed in Table 5.8 above and setting the DGCM free gas volume weighing factor to $\Psi = 0.5, 0.6, 0.7, 0.8, 0.9$ and 1.0 , the simulation results are obtained as displayed below, respectively. The grid and time step size used to generate the simulation results is that of the grid and time step independent model, as portrayed in Table 5.9 above.

Figure 5-8 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 0.5$.

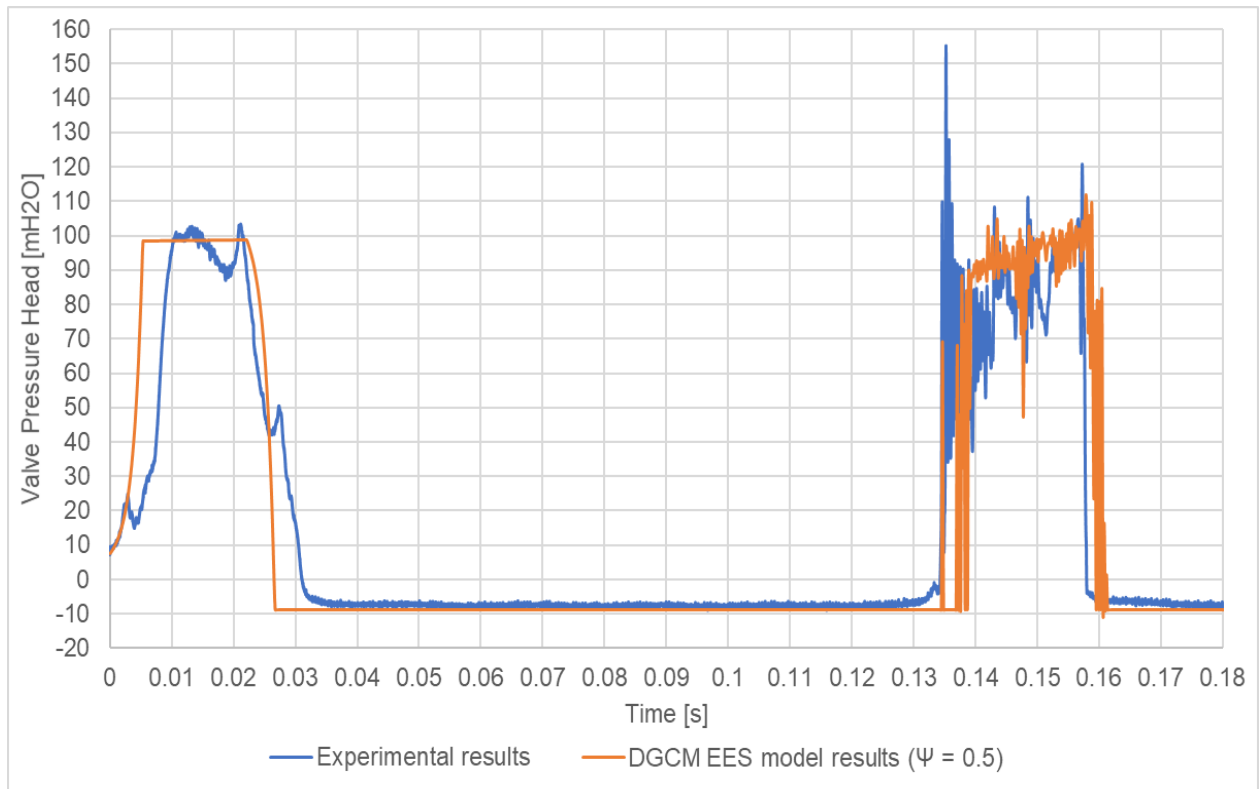


Figure 5-8: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.5$).

From Figure 5-8 above, although the simulation results follow the same trend as the experimental results, they appear to exhibit numerical instability at the second pressure wave, especially at the cavity collapse region. The maximum pressure head of the cavity collapse (second wave) of the experimental results is calculated as 120.7944 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is 0.1573 s.

From the simulation results for a weighing factor of $\Psi = 0.5$, the maximum pressure head of the cavity collapse (second wave) is calculated as 111.9105 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is 0.1579s. A point difference (error) is calculated and displayed in Table 5.10 below.

Table 5.10: Point difference (error) of results comparison ($\Psi = 0.5$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	111.9105	mH ₂ O

Validation parameter	Value	Unit
Time (cavity collapse)	0.1579	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-7.3546	%
Time (cavity collapse)	0.3803	%

From the point differences calculated in Table 5.10, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 7.3546%, and overpredicts the time of cavity collapse of the experimental results by 0.3803%.

Figure 5-9 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 0.6$.

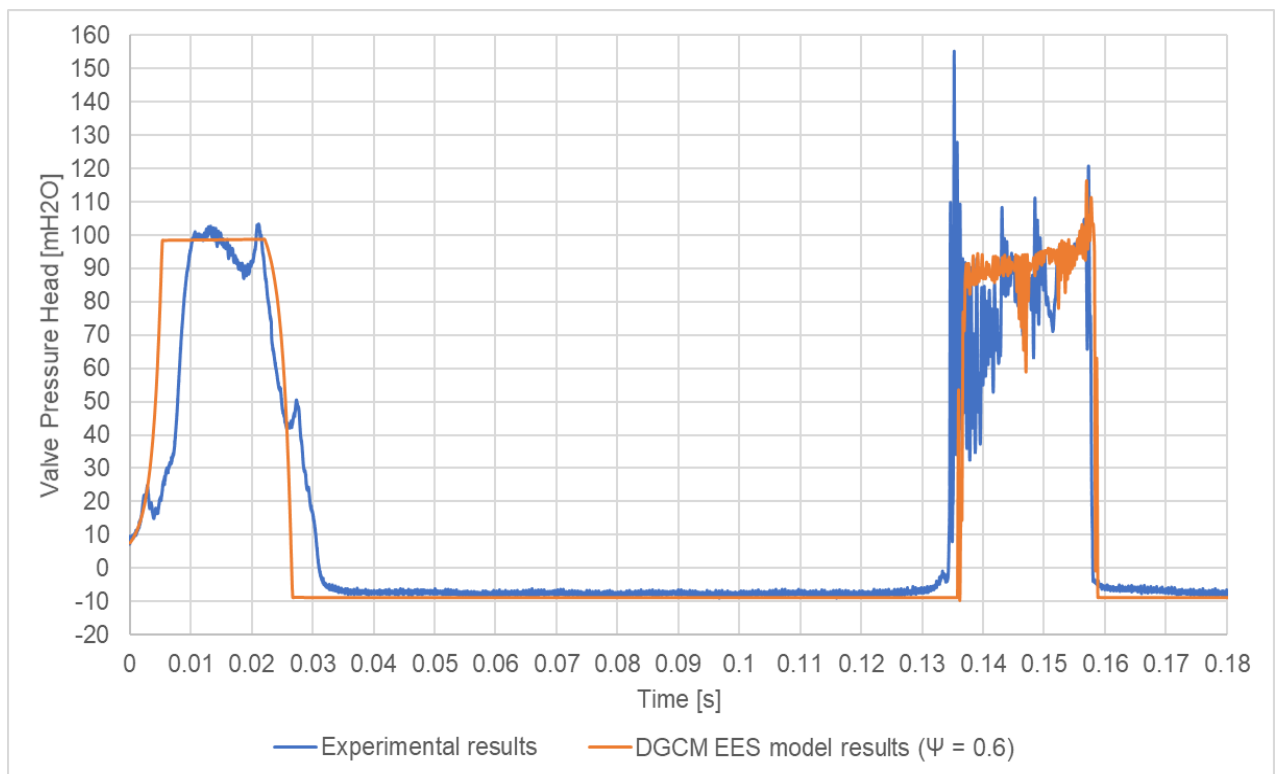


Figure 5-9: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.6$).

Figure 5-9 above shows that the simulation results follow the same trend as the experimental results. From the simulation results for a weighing factor of $\Psi = 0.6$, the maximum pressure head of the cavity collapse (second wave) is calculated as 116.4036 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is calculated as 0.1570 s.

A point difference (error) for each parameter mentioned above is calculated and displayed in Table 5.11 below.

Table 5.11: Point difference (error) of results comparison ($\Psi = 0.6$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	116.4036	mH ₂ O
Time (cavity collapse)	0.1570	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-3.6349	%
Time (cavity collapse)	-0.2045	%

From the point differences calculated in Table 5.11, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 3.6349%, and underpredicts the time of cavity collapse of the experimental results by 0.2045%. This is an improvement on the simulation results of $\Psi = 0.5$.

Figure 5-10 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 0.7$.

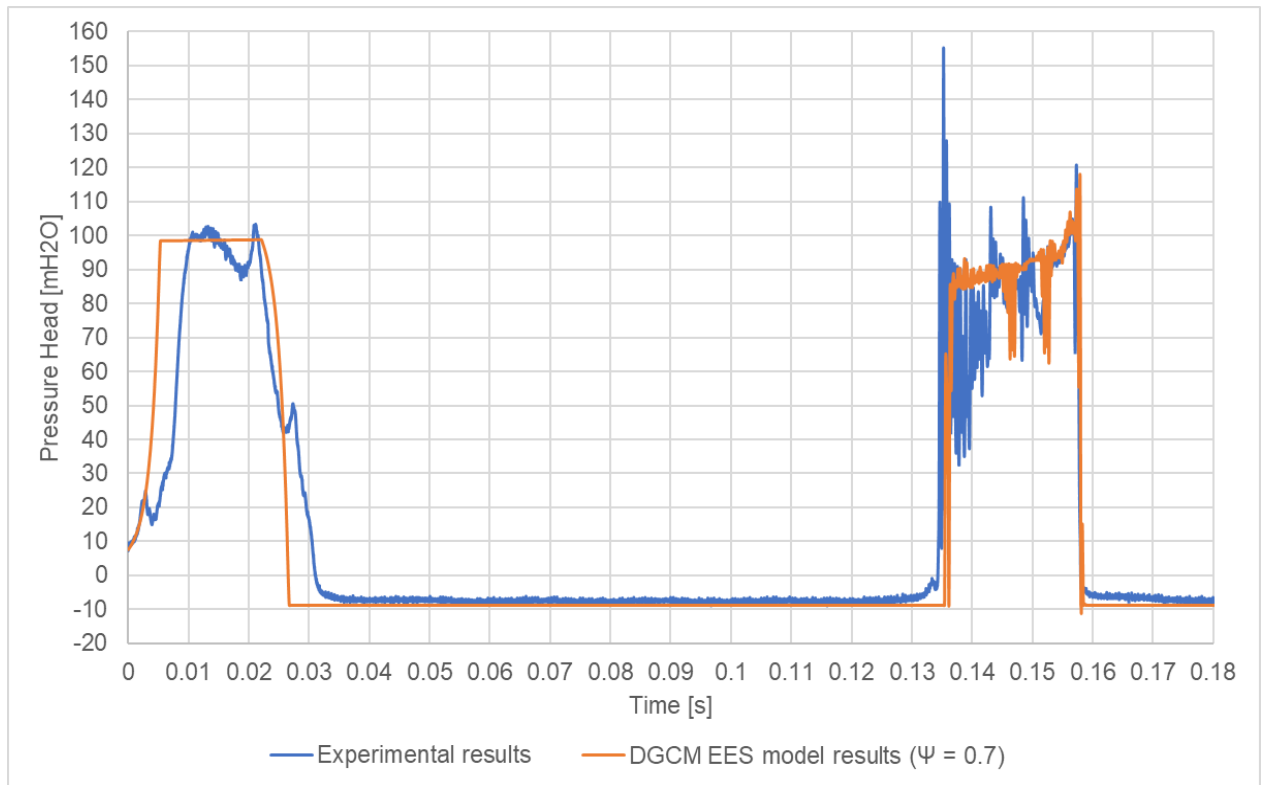


Figure 5-10: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.7$).

Figure 5-10 above shows that the simulation results follow the same trend as the experimental results. For a weighing factor of $\Psi = 0.7$, the maximum pressure head of the cavity collapse (second wave) is calculated as 116.9587 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is calculated as 0.1579 s.

A point difference (error) for each parameter mentioned above is calculated and displayed in Table 5.12 below.

Table 5.12: Point difference (error) of results comparison ($\Psi = 0.7$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	116.9587	mH ₂ O
Time (cavity collapse)	0.1579	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-3.1754	%

Validation parameter	Value	Unit
Time (cavity collapse)	0.3803	%

From the point differences calculated in Table 5.12, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 3.1754% and overpredicts the time of cavity collapse of the experimental results by 0.3803%. This is an improvement on the simulation results of $\Psi = 0.6$ in terms of the maximum pressure head, where the time of cavity collapse is the same as that of $\Psi = 0.5$.

Figure 5-11 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 0.8$.

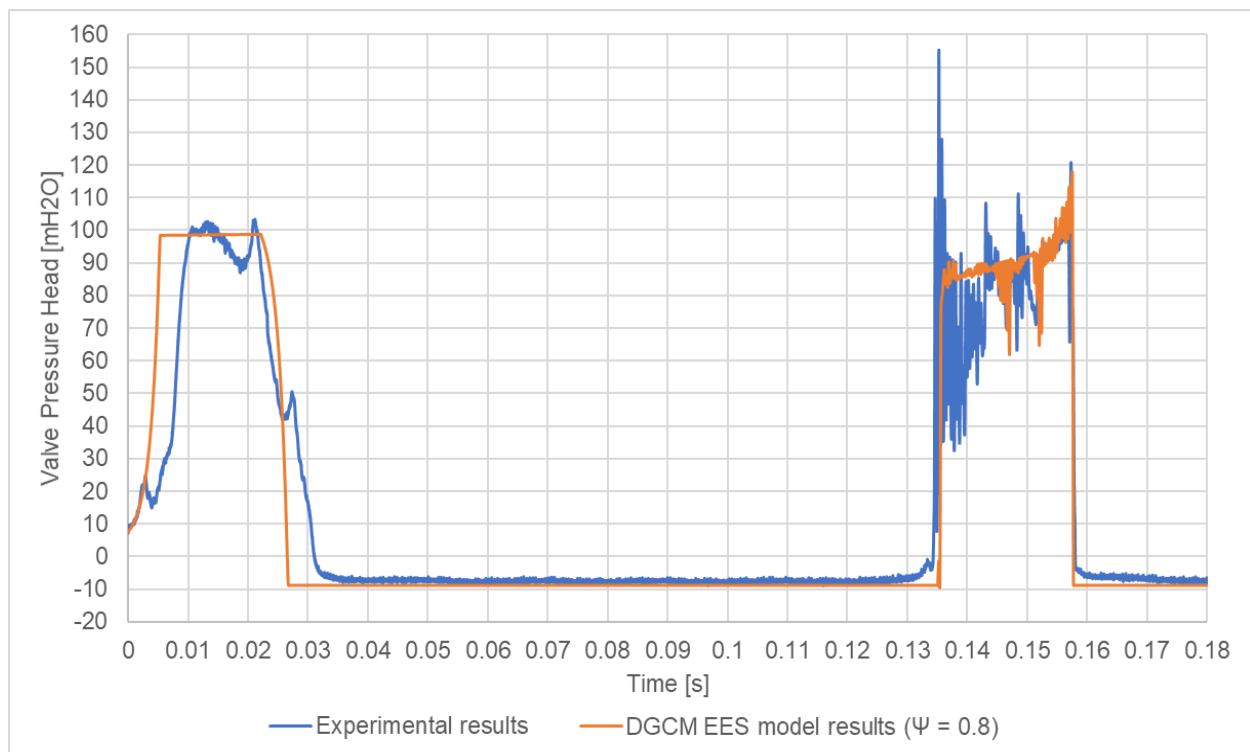


Figure 5-11: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.8$).

Figure 5-11 above shows that the simulation results follow the same trend as the experimental results. For a weighing factor of $\Psi = 0.8$, the maximum pressure head of the cavity collapse (second wave) is calculated as 117.2716 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is calculated as 0.1575 s.

A point difference (error) for each parameter mentioned above is calculated and displayed in Table 5.13 below.

Table 5.13: Point difference (error) of results comparison ($\Psi = 0.8$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	117.2716	mH ₂ O
Time (cavity collapse)	0.1575	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-2.9164	%
Time (cavity collapse)	0.1460	%

From the point differences calculated in Table 5.13, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 2.9164% and overpredicts the time of cavity collapse of the experimental results by 0.1460%, an improvement on the simulation results of $\Psi = 0.7$.

Figure 5-12 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 0.9$.

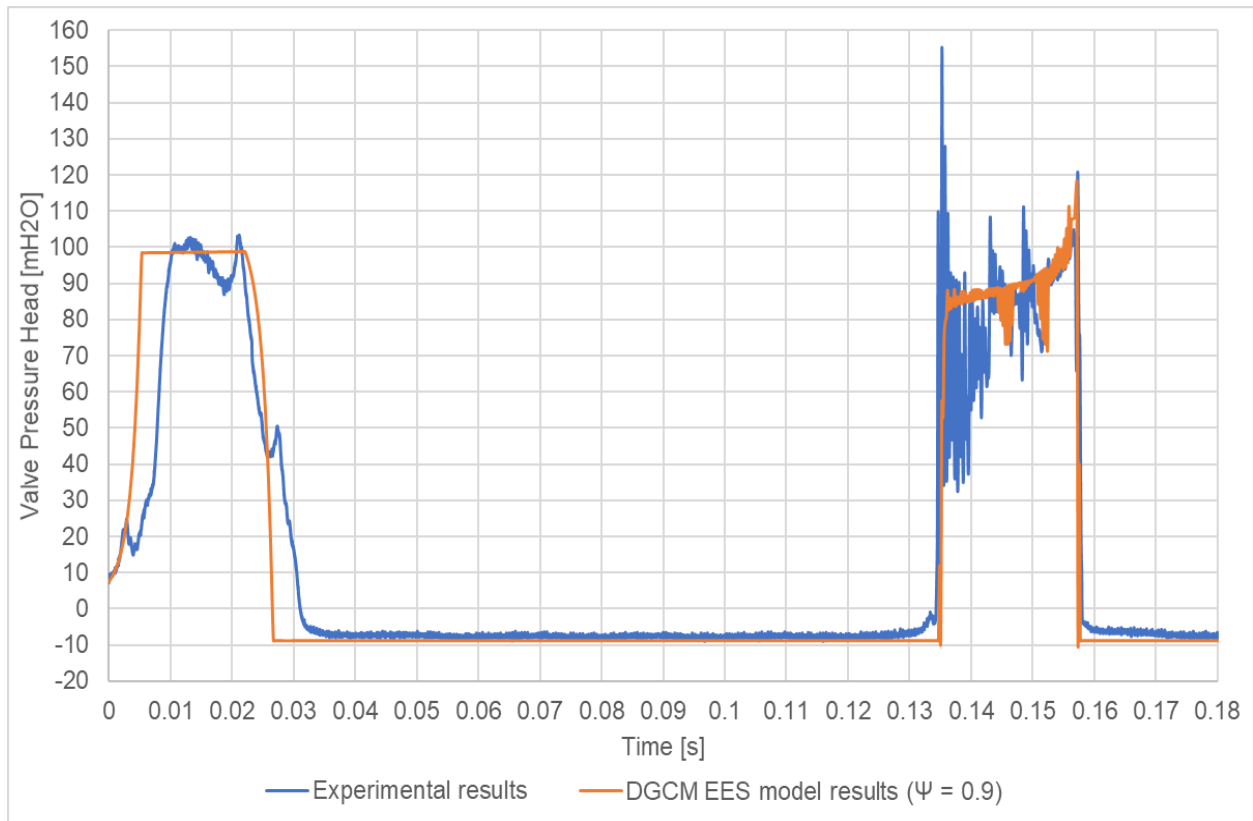


Figure 5-12: Comparison of experimental measurements with DGCM EES model results ($\Psi = 0.9$).

Figure 5-12 above shows that the simulation results follow the same trend as the experimental results. From the simulation results for a weighing factor of $\Psi = 0.9$, the maximum pressure head of the cavity collapse (second wave) is calculated as 117.4321 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is calculated as 0.1572 s.

A point difference (error) for each parameter mentioned above is calculated and displayed in Table 5.14 below.

Table 5.14: Point difference (error) of results comparison ($\Psi = 0.9$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	117.4321	mH ₂ O
Time (cavity collapse)	0.1572	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-2.7835	%

Validation parameter	Value	Unit
Time (cavity collapse)	-0.0883	%

From the point differences calculated in Table 5.14, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 2.7835%, and underpredicts the time of cavity collapse of the experimental results by 0.0883%. This is an improvement on the simulation results of $\Psi = 0.8$.

Figure 5-13 below compares the experimental results with the DGCM simulation results for a weighing factor of $\Psi = 1.0$.

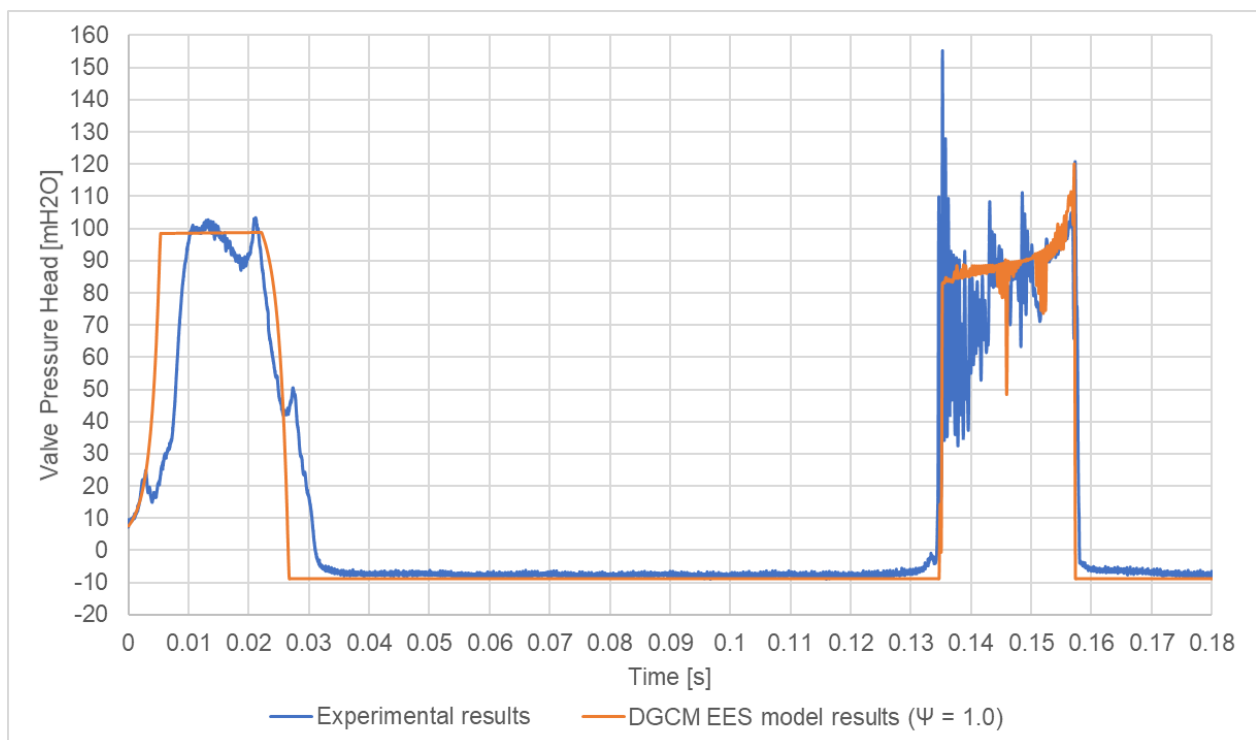


Figure 5-13: Comparison of experimental measurements with DGCM EES model results ($\Psi = 1.0$).

The simulation results from Figure 5-13 above follow the same trend as the experimental results. For a weighing factor of $\Psi = 1.0$, the maximum pressure head of the cavity collapse (second wave) is calculated as 118.3247 mH₂O, and the time occurrence of the said maximum head (cavity collapse) is calculated as 0.1572 s.

A point difference (error) for each parameter mentioned above is calculated and displayed in Table 5.15 below.

Table 5.15: Point difference (error) of results comparison ($\Psi = 1.0$).

Validation parameter	Value	Unit
<i>Experimental results</i>		
Pressure head (cavity collapse)	120.7944	mH ₂ O
Time (cavity collapse)	0.1573	s
<i>DGCM EES model results</i>		
Pressure head (cavity collapse)	118.3247	mH ₂ O
Time (cavity collapse)	0.1572	s
<i>Point difference (error)</i>		
Pressure head (cavity collapse)	-2.0446	%
Time (cavity collapse)	-0.0883	%

From the point differences calculated in Table 5.15, the DGCM EES simulation underpredicts the maximum pressure head of the experimental results by 2.0446% and the time of cavity collapse of the experimental results by 0.0883%. This is an improvement on the simulation results of $\Psi = 0.9$ in terms of the maximum pressure head, where the time of cavity collapse is the same as that of $\Psi = 0.9$.

A common trend observed in the comparison of the results, see Figure 5-8 to Figure 5-13, is that as the DGCM free gas volume weighing factor increases, so does the stability of the numerical simulation results at the second pressure wave, especially at the cavity collapse region.

5.5 Conclusion

Chapter 5 discusses the comparison of the results. Section 5.2 presents the calculated and measured parameters used as input for the EES model. Section 5.3 follows with a discussion of the sensitivity analysis on the effect of the weighing on the EES simulation results. Lastly, Section 5.4 compares the experimental and simulation results. Section 5.4.1 discusses the crucial findings made from the experimental results, and Section 5.4.2 presents and discusses the comparison of the experimental and simulation results.

To determine the effect of the DGCM free gas volume equation on the grid and time step independence of the DGCM numerical model, a grid refinement study is conducted in Section 5.3. This is done using the theory in Section 3.2.4.1 for various weighing factor values of $\Psi = 0.5, 0.6, 0.7, 0.8, 0.9$ and 1. The grid refinement study results in Figure 5-2 conclude that as the weighing factor value increases, the DGCM numerical model reaches grid and time step

independence at a larger grid and time step size. Therefore, as the weighing factor increases, so does the stability of the DGCM numerical model, leading to a faster solving time of the numerical model.

Table 5.16 below summarises the grid refinement (sensitivity analysis) results, displayed in Figure 5-2, conducted on the DGCM numerical model.

Table 5.16: Grid and time step size sensitivity analysis results summary.

Weighing factor (Ψ)	Number of increments [-]	Time step size [ms]
0.5	103	0.1073
0.6	23	0.4806
0.7	19	0.5818
0.8	18	0.6142
0.9	17	0.6503
1.0	15	0.7370

Following the discussion and analysis of Section 5.4.1.1, it was concluded that entrapped air in the system can significantly impact the experimental results. From this, was determined that the filling strategy of the experimental test bench is to be changed and mitigations to be implemented to avoid or at least minimise the amount of trapped air in the system. This was done by adding an air bleed valve on the top of the bottom collection tank, refer to Figure 4-2, and filling the system from the bottom (i.e. at the outlet) to the top (i.e. the open container where the float valve is located). Referring to Figure 5-5, it is evident that implementing the mitigation strategies proved to be successful in removing most of the entrapped air. However, some entrapped air remained in the system.

The results of Figure 5-5 display erratic behaviour in the measured experimental results, especially at the start of the second pressure wave. The discussion in Section 5.4.1.2 concludes that these erratic behaviours can be attributed to the effects of dynamic FSI in the experimental test bench structure. The transient forces on the pipe wall during a hydraulic transient event can cause the pipe to move in the translation DOF (heaving, swaying and surging) and in the rotation DOF (pitching, yawing and rolling), see Figure 5-6. The pressure variations cause circumferential stresses in the pipe wall, causing the pipe to move within its DOF and leading to high-frequency pressure oscillations in the system. It should be noted that numerical models exist in the open literature for which these FSI effects are accounted for. However, it is not in the scope of this

research effort and is therefore only crucial for this research effort to note the existence of FSI in the experimental results obtained from the test bench.

Section 5.4.2 compares the experimental results to the simulation results of the DGCM numerical model for various DGCM free gas volume weighing factors to determine whether a baseline for the weighing factor exists and to conclude whether the weighing factor depends on the system layout. Analysing the comparison of results, it is deduced that as the weighing factor increases, better results comparison is obtained, with $\Psi = 1.0$ exhibiting the best results comparison. The calculated point difference (error) for the maximum cavity collapse pressure head improved from -7.3546% for $\Psi = 0.5$ to -2.0446% for $\Psi = 1.0$. Regarding the time of cavity collapse, a downward trend is observed in the calculated point difference as the weighing factor increases.

Table 5.17 and Figure 5-14 below summarise the point differences calculated by comparing the results conducted.

Table 5.17: Point difference (error) summary of results comparisons.

DGCM free gas volume weighing factor (Ψ)	Point difference (error)
<i>Pressure head (cavity collapse)</i>	
0.5	-7.3546%
0.6	-3.6349%
0.7	-3.1754%
0.8	-2.9164%
0.9	-2.7835%
1.0	-2.0446%
<i>Time (cavity collapse)</i>	
0.5	0.3803%
0.6	-0.2045%
0.7	0.3803%
0.8	0.1460%
0.9	-0.0883%
1.0	-0.0883%

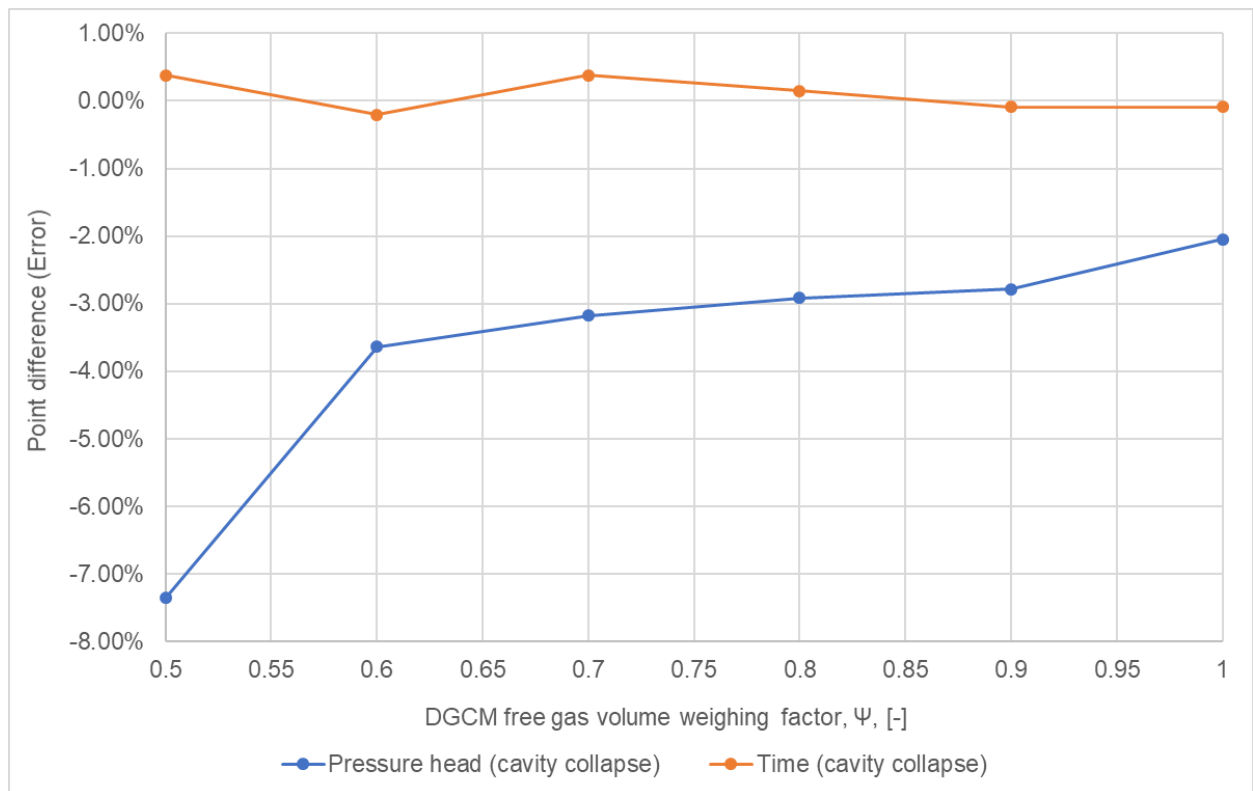


Figure 5-14: Point difference (error) summary of results comparisons.

The comparison of results also concluded that the simulation stability improved as the weighing factor increased (see Figure 5-8 to Figure 5-13), especially at the cavity collapse region of the second pressure wave.

CHAPTER 6 CONCLUSIONS AND RECOMMENDATIONS

This chapter provides a summary of the study and main findings, followed by a discussion of the contribution to the field of study. Next, the limitations of this study and further research are provided. Lastly, the concluding remarks are outlined and discussed.

6.1 Introduction

Chapter 5 presents and compares the numerical predictions and experiment measurements used to address the research problem and questions posed in this research effort. This includes the sensitivity analysis of the grid and time step size for various DGCM free gas volume weighing factors. The crucial findings from the discussed experimental results follow this and include the effect of entrapped air in the system and the effect of dynamic FSI on the system. Lastly, the experimental and EES simulation results for various weighing factors are compared and discussed, establishing whether a baseline for the weighing factor can be suggested.

In this chapter, a brief overview of the previous chapters is provided. The need for the study is highlighted first, followed by the research objectives endeavoured to answer the research questions. Following this, the main findings and conclusions made during this research effort are emphasised. This is followed by a short discussion on the contribution of this research effort to the field of study, after which the limitations posed during the research effort are stressed, and the areas of further research are discussed. Lastly, the concluding remarks on this dissertation are given.

6.2 Summary of the study and main findings

Hydraulic transients, especially water hammer with liquid column separation, pose a significant risk to liquid transport piping systems. Piping systems are naturally transient systems with flow variations occurring often. Any sudden variation in the flow where an incompressible fluid is transported causes rapid energy transfer, leading to water hammer. Under the right circumstances, liquid column separation occurs, leading to a short-duration pressure pulse much higher than the system's operating pressure. From the literature, it became evident that process and design engineers often overlook hydraulic transients. If not adequately accounted for, this sudden increase in pressure can cause damage to the system components; however, in some cases, the damages can be costly, dangerous and even lethal.

Many advances have been made in analytically predicting hydraulic transients and numerically simulating hydraulic transients. These numerical tools prove to be vital in the design and operation of modern pipeline systems. From the literature, it was concluded that the most popular modern numerical model used in simulating water hammer with liquid column separation is the DGCM model. The DGCM model is improved upon the DVCM model, which is used as a foundation by adding a permeant-infinitesimal gas cavity to each computational section in the MOC. To suppress the numerical oscillations appearing in transients where the pressurisation of gas bubbles occurs, a weighing factor (Ψ) is introduced to the discretised free gas volume equation.

Unfortunately, the open literature does not provide information on implementing the DGCM model. This is especially true in terms of the said weighing factor as [43] recommends that the weighing factor should be set to a value higher than 0.5, but according to [48] it is strongly suggested that this weighing factor should be set to unity only. It is also unknown if the weighing factor depends on the system's layout or whether it affects the grid and time step independence of the DGCM numerical model.

The above-discussed void in the research has led to the primary aim of this research effort: to establish whether the weighing factor influences the grid and time step independence of the DGCM numerical mode and whether a baseline for the weighing factor value exists. The secondary aim of this research is to determine the possibility that the weighing factor depends on the system layout.

The specific objectives of this study were the following:

- Implement the DGCM into an equation solver software (EES) and verify and validate the EES model with published simulation and experimental data,
- Generate experimental data of water hammer with liquid column separation for a straight horizontal pipe system,
- Conduct a grid and time step independence study on the DGCM EES numerical model for various free gas volume weighing factors,
- Conduct a sensitivity analysis on the grid and time step independence study, determining the effect of the weighing factor,
- Simulate the experiment using a grid and time step independent DGCM EES numerical models for each of the various weighing factors,

- Compare the various weighing factor DGCM EES simulation results to the experimental results,
- Draw a conclusion based on the results and validate the primary and secondary research questions.

The sensitivity analysis provides insight into the effect of the weighing factor on the grid and time step independence of the DGCM numerical model. The results show that as the weighing factor value reaches unity, the number of increments decreases, and the time step size increases. Therefore, as the weighing factor value increases, the DGCM numerical model reaches numerical stability at a larger grid time step size. Hence, at a weighing factor value of unity, the DGCM numerical model maintains numerical stability at a larger grid time step size, leading to less simulation time and computing process power.

The comparison of the experimental results to the DGCM EES simulation results provides insight into the effect of the weighing factor value on the stability of the numerical model, as well as the ability of the numerical model to simulate and predict water hammer with liquid column separation. The comparison of the results shows that as the weighing factor value increases, the numerical model's stability increases, and better agreement between the experimental results and simulation results is obtained. This continues the claim made by [48] who strongly suggests that a weighing factor of unity is to be used for numerical stability and accuracy.

From this research effort, it cannot be concluded rightly that the weighing factor depends on the system layout. However, given the fact that most of the research conducted on the DGCM in the open literature (where various piping layouts exist) uses a weighing factor value of unity, as well as the fact that the best agreement for this research effort is obtained using a weighing factor of unity, it so far supports the hypothesis that the weighing factor does not depend on the layout of the system. Therefore, a baseline for the weighing factor value of unity is strongly recommended and should be used for numerical stability and accuracy.

6.3 Contribution to the field of study

This research effort contributes to the literature on the DGCM numerical model used to simulate water hammer with liquid column separation. This is primarily due to the lack of information in the open literature regarding implementing the DGCM numerical model in terms of grid and time step independence and the value of the free gas volume weighing factor. From the open literature, various authors provide conflicting propositions about the value of the weighing factor when

implementing the DGCM numerical model. To the author's knowledge, no study exists in the open literature where the effect of the weighing factor on the grid and time step independence is studied, as well as if specific research exists to establish whether a baseline for the weighing factor exists and whether the weighing factor is dependent on the layout of the system. This research effort, therefore, sheds light on the effect of the weighing factor on grid and time step independence, as well as the impact of the weighing factor on the stability and accuracy of the DGCM numerical model.

6.4 Limitations and areas of further research

6.4.1 Limitations and improvements of this research effort

During this research effort, several assumptions were made, and various limitations were discovered. These assumptions and limitations should be reconsidered and improved upon:

- It was assumed from the literature that the DGCM numerical model coupled with a steady state friction model is sufficient in predicting and simulating water hammer with liquid column separation. An improved friction model should be coupled with the DGCM numerical model to increase the model's accuracy and obtain a better dampening rate of the high-pressure pulse.
- The spring-release mechanism for the fast-closing valve worked satisfactorily for this research effort. However, it is recommended that the valve be redesigned using an electrical actuating system to ensure better control over its opening and closing and a more accurate valve closure time.
- During experimentation, best efforts were made to remove entrapped air from the system. However, some entrapped air remained in the system, and it was assumed that it had a limited effect on the experimental results. Better purging techniques should be investigated and implemented. Such as welding the pipe sections to avoid air entrapment in the unions at the pipe joints.
- From the experimental results, it was discovered that dynamic FSI was present in the experimental test bench. It was assumed that this does not significantly affect the timing of the cavity collapse and the maximum pressure caused by cavity collapse. However, the pipe supports of the experimental test bench should be stiffened to mitigate, if not remove, the effect of dynamic FSI. Also, it should be considered if an FSI model should be implemented in the DGCM numerical model, such as that of [75].

- From the comparison of results and observations made in the open literature, it was concluded that the weighing factor does not depend on the system's layout. However, to confirm if this assumption is valid, the experimental test bench should be modified to accommodate various pipeline configurations (i.e. pipes at certain angles) for further testing.

It should be noted that the DGCM is suited for a wide range of flow rates. However, it is best suited for flow rates within the turbulent regime, especially the fully developed turbulent regime. Whilst the DGCM is still valid for low flow rate applications some considerations might be applicable; at very low flow rates the viscous effects become significant which is not specifically modelled in the DGCM, in a nearly stagnant system transient effects are dominated by elastic wave reflections rather than column separation, and the model might require additional damping for numerical oscillations. On the other hand, at extremely high flow applications the DGCM is still valid, but with some limitations. These limitations include; compressibility effects and shock waves playing a more significant role, supersonic conditions might occur which are not accounted for in the DGCM, and additional turbulence modelling might be needed.

6.4.2 Areas of further research

This research effort focused on implementing the DGCM numerical model for a rigid pipe material (galvanised steel) using water as the transporting fluid on a straight horizontal experimental test bench. However, there is still a void in the research on implementing the DGCM numerical model regarding the weighing factor for various pipe materials, types of transporting fluids, and system configurations. Therefore, further research can be conducted using this framework for different pipe materials, various transport fluids, and various system configurations. This will either cement or challenge the conclusions made during this research effort that unity is the recommended baseline value for the weighing factor to be used.

6.5 Concluding remarks

This research effort endeavours to identify and assess the effect of the DGCM free gas volume weighing factor on the time and grid size independence of the DGCM numerical model and whether a baseline for the weighing factor exists.

The study from the research this effort shows that a weighing factor of unity, grid and time step stability is obtained at a larger grid and time step size, leading to faster simulation solving time

and reducing computer resource requirements. Furthermore, this research effort shows that the best agreement with the experimental results is obtained when the weighing factor value is set to unity. This suggests that the baseline for the weighing factor value is unity, and further strengthens the claim made by [48] who strongly suggested that a weighing factor of unity is to be used.

This research effort can be continued by implementing better purging methods to remove entrapped air, improving the experimental test bench to mitigate the effect of dynamic FSI, and coupling an FSI model to the DGCM numerical model. A more accurate dynamic friction model can also be implemented instead of relying on the steady-state friction model. Further research should be conducted on the effect of the weighing factor value for various pipe materials, types of transporting fluids, and system layout.

The simulation and prediction of hydraulic transients, especially water hammer with liquid column separation, are crucial to designing and operating a pipeline system. This research effort could help engineers correctly implement the DGCM numerical mode and conduct further research on the topic.

BIBLIOGRAPHY

- [1] R. Wichowski, "Hydraulic Transients Analysis in Pipe Networks by the Method of Characteristics (MOC)," *Archives of Hydro-Engineering and Environmental Mechanics*, vol. 53, no. 3, pp. 267–291, 2006.
- [2] H. Cao, M. Mohareb, and I. Nistor, "Mechanical response of buried and covered pipes under water hammer," *International Journal of Pressure Vessels and Piping*, vol. 190, pp. 104–310, Apr. 2021, doi: doi:10.1016/j.ijpvp.2021.104310.
- [3] A. R. Simpson, "Large water hammer pressures due to column separation in sloping pipes," *The University of Michigan, Ann Arbor, USA*, 1986.
- [4] A. Bergant, "Kavitacijski tok med prehodnimi rezimi v cevni sistemih. (Transient cavitating flow in pipelines.)," 1992.
- [5] V. Streeter and E. Wylie, *Hydraulic Transients*. New York: McGraw-Hill, 1967.
- [6] F. Stojkovski and V. Stojkovski, "Influence of the selected pipe material over the water hammer effect," in *International Metallurgical Congress*, 2016.
- [7] D. Khani, Y. H. Lim, and A. Malekpour, "Calculating Column Separation in Liquid Pipelines Using a 1D-CFD Coupled Model," *Mathematics*, vol. 10, no. 12, Jun. 2022, doi: doi:10.3390/math10121960.
- [8] C. Martin, "Experimental investigation of column separation with rapid closure of downstream valve," in *Proceedings of the 4th International Conference on Pressure Surges*, 1983, pp. 77–88.
- [9] L. Tyler, "Preventing hydraulic shock: Millard refrigerated service, Inc. anhydrous ammonia release," *U.S. Chemical Safety Board, GCAP Ammonia Safety Day*, May 2015.
- [10] L. G. Britton and R. J. Willey, "Avoiding water hammer and other hydraulic transients," *Process Safety Progress*, vol. 43, no. 1, pp. 101–112, Mar. 2024, doi: 10.1002/prs.12517.
- [11] H. A. Warda, E. M. Wahba, and M. S. El-Din, "Computational Fluid Dynamics (CFD) simulation of liquid column separation in pipe transients," *Alexandria Engineering Journal*, vol. 59, no. 5, pp. 3451–3462, Oct. 2020.

- [12] A. Salih, "Method of Characteristics," *Department of Aerospace Engineering Indian Institute of Space Science and Technology, Thiruvananthapuram*, Jul. 2016.
- [13] R. K. Jensen, J. K. Larsen, K. L. Lassen, M. Mandø, and A. Andreassen, "Implementation and validation of a free open source 1D water hammer code," *Fluids*, vol. 3, no. 3, Oct. 2018.
- [14] "Water Hammer," *KSB know how, Volume 1*.
- [15] Aquatherm.com.au, "Vaporous and Gaseous Cavitation."
- [16] "Vaporous and Gaseous Cavitation," *Technews*, vol. 2, 2017.
- [17] A. Bergant and A. R. Simpson, "Water Hammer and Column Separation Measurements in an Experimental Apparatus," *Report No. R128, Department of Civil and Environmental Engineering, The University of Adelaide, Adelaide, Australia*.
- [18] J. LeConte, "Experiments and calculations on the resurge phase of water hammer.," *Experiments and calculations on the resurge phase of water hammer*, pp. 691–694, 1937.
- [19] H. Lupton, "Graphical analysis of pressure surges in pumping systems," *Journal of the Institution of Water Engineers*, vol. 7, pp. 87–125, 1953.
- [20] C. Kranenburg, "Transient cavitation in pipelines," *Communications on Hydraulics*, vol. 73, no. 2, 1974.
- [21] M. Narcy, E. de Malmazet, and C. Colin, "Flow boiling in tube under normal gravity and microgravity conditions," *International Journal of Multiphase Flow*, vol. 60, pp. 50–63, Apr. 2014.
- [22] J. N. Tilton, *Perry's Chemical Engineers' Handbook (Section 6)*, 8th ed. McGraw-Hill, 2008.
- [23] "Understanding Hydraulic Shock," *The Electronic Newsletter of The Industrial Refrigeration Consortium*, vol. 9, no. 1, 2009.
- [24] B. Monachan, R. J. Thomas, D. Steaphen, M. Skaria, and K. A. Shafi, "Simulation of Stratified Two-phase Flow Regime using Air-Water Model in ANSYS Fluent ®," in *Journal of Physics: Conference Series*, Institute of Physics Publishing, Nov. 2019. doi: 10.1088/1742-6596/1355/1/012014.

- [25] “Geysers,” GeologyScience, Available on: <https://geologyscience.com/geology-branches/volcanology/geysers/>.
- [26] X. Lu, A. Watson, A. V. Gorin, and J. Deans, “Experimental investigation and numerical modelling of transient two-phase flow in a geysering geothermal well,” *Geothermics*, vol. 35, no. 4, pp. 409–427, 2006, doi: 10.1016/j.geothermics.2006.07.001.
- [27] C. C. Bonin, “Closure to ‘Discussions of “Water-Hammer Damage to Oigawa Power Station” (1960, ASME J. Eng. Power, 82, pp. 116–118),” *Journal of Engineering for Power*, vol. 82, no. 2, pp. 118–119, Apr. 1960.
- [28] S. I. Lupa, M. Gagnon, S. Muntean, and G. Abdul-Nour, “The Impact of Water Hammer on Hydraulic Power Units,” Feb. 01, 2022, *MDPI*. doi: 10.3390/en15041526.
- [29] R. P. Saunderson, “End-Flash is totally cool,” *Air Products, USA*.
- [30] E. Nilsen, “Natural gas export vapor explosion,” CNN, Available on: <https://edition.cnn.com/2024/03/20/climate/natural-gas-export-vapor-cloud-explosion/index.html>.
- [31] M. H. Morgan, “Vitruvius: The Ten Books on Architecture (Translated),” *Harvard University Press*, 1914.
- [32] L. F. Manabrea, “Note sur les effects de choc de l’eau dans les conduites,” *C.R. Hebd. Seances Acad. Sci*, vol. 47, pp. 221–224, 1858.
- [33] J. Michaud, “Coups de belier dans les conduites—Etude des moyens employees pour en atteneur les effects,” *Bull. Soc. Vaud. Ing. Arch.*, vol. 43, pp. 56–64, 1878.
- [34] A. Bouchayer, “Établissement Des Conduites Forcées,” *Houille Blanche*, vol. 1, pp. 212–218, 1902.
- [35] C. De Sparre, “L’étude des Coups de Béliér Dans les Canalisations Métalliques Sous Pression,” *Houille Blanche*, vol. 16, pp. 57–58, 1919.
- [36] “International Symposium on Water Hammer in Pumped Storage Projects,” in *ASME Winter Annual Meeting*, 1965.

- [37] V. L. Streeter, "Paper 1: Computer Solution of Surge Problems," *Proceedings of the Institution of Mechanical Engineers, Conference Proceedings*, vol. 180, no. 5, pp. 62–82, 1965.
- [38] C. Thibessard, "La simulation du coup de bélier sur calculateur numérique. (The simulation of water hammer on a numerical calculator.)," in *Proceedings of the 9th IAHR Convention*, 1961, pp. 881–890.
- [39] V. L. 1959 Streeter, "Water Hammer Analysis," *Journal of the Hydraulics Division*, vol. 95, no. 6, pp. 1959–1972, 1969.
- [40] T. Tanahashi and E. Kasahara, "Analysis of the Waterhammer with Water Column Separation," *Bulletin of JSME*, vol. 12, no. 50, pp. 206–214, 1969.
- [41] A. de Vries, "'Cavitation due to waterslag in horizontale leidingen. (Cavitation due to waterhammer in horizontal pipelines.)', Delft Hydraulics Laboratory, Report M 1116," 1972.
- [42] R. J. Brown, "Closure to "Discussion of 'Water-Column Separation at Two Pumping Plants,'" *ASME Journal of Basic Engineering*, vol. 90, no. 4, pp. 531–531, 1968.
- [43] E. B. Wylie, "Simulation of Vaporous and Gaseous Cavitation," *J Fluids Eng*, vol. 106, no. 3, pp. 307–311, 1984.
- [44] E. B. Wylie, "Low void fraction two-component two-phase transient flow," *Unsteady Flow and Fluid Transients*, pp. 3–9, 1992.
- [45] E. B. Wylie and V. L. Streeter, *Fluid transients in systems*. New Jersey: Prentice Hall, Englewood Cliffs, NJ, 1993.
- [46] A. Bergant and A. R. Simpson, "Pipeline Column Separation Flow Regimes," *Journal of Hydraulic Engineering*, vol. 125, no. 8, pp. 835–848, 1999.
- [47] A. Bergant and A. R. Simpson, "Interface model for transient cavitating flow in pipelines," *Unsteady Flow and Fluid Transients*, pp. 333–342, 1992.
- [48] J. C. P. Liou, "Numerical Properties of the Discrete Gas Cavity Model for Transients," *J Fluids Eng*, vol. 122, no. 3, pp. 636–639, Sep. 2000.
- [49] L. Zhao, Y. Yang, T. Wang, L. Zhou, Y. Li, and M. Zhang, "A simulation calculation method of a water hammer with multipoint collapsing," *Energies (Basel)*, vol. 13, no. 3, Mar. 2020.

- [50] A. Bergant, J. P. Vítkovský, A. R. Simpson, and M. F. Lambert, "Discrete Vapour Cavity model with improved timing of prediction of growth and collapse of cavities," 2007.
- [51] A. Bergant, J. P. Vítkovský, A. R. Simpson, and M. F. Lambert, "A discrete gas-cavity model that considers the frictional effects of unsteady pipe flow Acoustic methods for leak detection in water networks View project Water Distribution Networks Optimization View project," 2015.
- [52] V. L. Streeter, "Unsteady flow calculations by numerical methods," *ASME Journal of Basic Engineering*, vol. 94, pp. 457–466, 1972.
- [53] A. K. Soares, D. I. C. Covas, and N. J. G. Carrio, "Transient vaporous cavitation in viscoelastic pipes," *Journal of Hydraulic Research*, vol. 50, no. 2, pp. 228–235, Apr. 2012.
- [54] G. A. Provoost and E. B. Wylie, "Discrete gas model to represent distributed free gas in liquids," *Proceedings of the 5th International Symposium on Water Column Separation*, 1981.
- [55] C. Nicolet, "Simulation of Water Hammer Induced Column Separation through Electrical Analogy," *4-th International Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, 2011.
- [56] L. E. Doo, A. R. Ngus, A. R. Simpson, and A. S. Tijsseling, "Water hammer with column separation: a review of research in the twentieth century".
- [57] J. P. Kalkwijk and C. Kranenburg, "Cavitation in horizontal pipelines due to water hammer," *ASCE Journal of the Hydraulics Division* 97(HY10), 1971.
- [58] V. L. Streeter, "Transient cavitating pipe flow," *ASCE Journal of Hydraulic Engineering* 109(HY11), pp. 1408–1423, 1983.
- [59] M. Safadi, A. Riasi, and S. A. Nourbakhsh, "Cavitating flow during water hammer using a generalized interface vaporous cavitation model," *J Fluids Struct*, vol. 34, pp. 190–201, Oct. 2012.
- [60] "Examining Spatial (Grid) Convergence," NPARC Alliance CFD Verification and Validation Web Site, Available on: <https://www.grc.nasa.gov/www/wind/valid/tutorial/spatconv.html> .

- [61] "Uncertainty and Error in CFD Simulations," *NPARC Alliance CFD Verification and Validation Web Site*, Available on: <https://www.grc.nasa.gov/www/wind/valid/tutorial/errors.html>, Feb. 2021.
- [62] P. J. Roache, K. Ghia, and F. White, "Editorial Policy Statement on the Control of Numerical Accuracy," *ASME Journal of Fluids Engineering*, vol. 108, no. 1, p. 2, 1986.
- [63] "Flownex Product Info," Available on: <https://flownex.com/product-info/>.
- [64] "FLOWNEX SE (2022 – UPDATE 1)," Available on: <https://flownex.com/news/flownex-se-2022-update-1/>.
- [65] ERIKS group, "O-ring gland design," *Sealing Elements*.
- [66] B. E. Suisse, "Research for dynamic seal friction modeling in linear motion hydraulic piston applications," Master of Science, The University of Texas at Arlington, Arlington, 2005.
- [67] P. M. Gerhart, A. L. Gerhart, and J. I. Hochstein, *Munson's Fluid Mechanics*, Global Edition. Hoboken, New Jersey: Wiley & Sons, 2017.
- [68] S. H. Chue, "Pressure probes for fluid measurements," *Prog. Aerospace Sciences*, vol. 16, pp. 147–223, 1975.
- [69] R. Shaw, "Influence of hole dimensions on static pressure measurements," *J Fluid Mech*, vol. 7, pp. 550–564, 1960.
- [70] D. C. Montgomery and G. C. Ruge, *Applied Statistics and Probability for Engineers*, 6th ed. Wiley & Sons.
- [71] M. J. McFarland, "Physical Properties of Water and Air," in *Biosolids Engineering*, 1st ed., McGraw-Hill, 2001, ch. Appendix D.
- [72] "ASTM A525 Galvanized Steel."
- [73] "Wheater forecast for September 10 in Potchefstroom," Meteum, Available on: <https://meteum.ai/weather/en-US/date/september/10?lat=-26.712708&lon=27.076938&lang=en>.
- [74] Manuela. Escameia, *Air problems in pipelines : a design manual*. HR Wallingford, 2005.

- [75] D. Ferras, P. A. Manso, A. J. Schleiss, and D. I. C. Covas, "One-dimensional fluid-structure Interaction models in pressurized fluid-filled pipes: A review," Oct. 08, 2018, *MDPI AG*. doi: 10.3390/app8101844.
- [76] A. Adamkowski, S. Henclik, and M. Lewandowski, "Experimental and numerical results of the influence of dynamic Poisson effect on transient pipe flow parameters," *IOP Conf Ser Earth Environ Sci*, vol. 12, p. 012041, Aug. 2010, doi: 10.1088/1755-1315/12/1/012041.

ANNEXURE A: DGCM EES CODE

```

$Tabstops 1 2 3 4 5 6 7 8 9 10 11 12 13 14 cm
"-----Donovan Hitchcock-----"
"-----27389502-----"
"-----The Discrete Gas Cavity Model (DGCM) with a Steady Flow Friction Model-----"

"-----Functions & Procedures-----"

"-----Friction Factor-----"

"Function to calculate the Darcy-Weissbach Friction Factor based on the calculated Reynolds Number"

Function frictionfactor(Re, epsilon, D)

"For laminar flow"

    If ( Re < 2000) Then

        frictionfactor = 64/Re

    Else

"For turbulent flow"

    If ( Re > 4000) Then

        frictionfactor = 0.25*(log10( (0.27*epsilon/D) + (5.74/Re^0.9) ))^(-2)

"For transitional flow"

    Else

        f_1 = 64/2000
        f_2 = 0.25*(log10( (0.27*epsilon/D) + (5.74/4000^0.9) ))^(-2)

        frictionfactor = ((f_1 - f_2) / ((2000-4000)*(Re-4000))) + f_2

    Endif

Endif

End

"-----Lookup Tables-----"

"Procedure to Write LOOKUP TABLES for Head, Volumetric Flowrate and Gas Volume for the Interior Nodes of the MOC"

Procedure writeinteriornodes(H, Qu, Qd, Vg_P, f, r, c: temp)

    temp = 1

    lookup('H', r, c) = H           //Writing Head
    lookup('Qu', r, c) = Qu        //Writing Computational Node Upstream Volumetric Flowrate
    lookup('Qd', r, c) = Qd        //Writing Computational Node Upstream Volumetric Flowrate
    lookup('Vg_P', r, c) = Vg_P    //Writing Gas Volume
    lookup('f', r, c) = f          //Writing Friction Factor

```

End

"Procedure to Write LOOKUP TABLES for Head, Volumetric Flowrate and Gas Volume for the Exterior Nodes of the MOC"

Procedure writeexteriornodes(H_r, Qu_r, Qd_r, Vg_P_r, f_r, H_v, Qu_v, Qd_v, Vg_P_v, f_v, r, c: temp)

temp = 1

"Upstream Reservoir Side"

```
lookup('H', r, 1) = H_r           //Writing Head
lookup('Qu', r, 1) = Qu_r          //Writing Computational Node Upstream Volumetric Flowrate
lookup('Qd', r, 1) = Qd_r          //Writing Computational Node Upstream Volumetric Flowrate
lookup('Vg_P', r, 1) = Vg_P_r      //Writing Gas Volume
lookup('f', r, 1) = f_r            //Writing Friction Factor
```

"Downstream Valve Side"

```
lookup('H', r, c) = H_v           //Writing Head
lookup('Qu', r, c) = Qu_v          //Writing Computational Node Upstream Volumetric Flowrate
lookup('Qd', r, c) = Qd_v          //Writing Computational Node Upstream Volumetric Flowrate
lookup('Vg_P', r, c) = Vg_P_v      //Writing Gas Volume
lookup('f', r, c) = f_v            //Writing Friction Factor
```

End

"Procedure to Write LOOKUP TABLES for Pressure at Steady State for the Interior Nodes of the MOC"

Procedure writep0interior(P_0, r, c: temp)

temp = 1

```
lookup('P_0', r, c) = P_0          //Writing Pressure at Steady State
```

End

"Procedure to Write LOOKUP TABLES for Pressure at Steady State for the Exterior Nodes of the MOC"

Procedure writep0exterior(P_0_r, P_0_v, r, c: temp)

temp = 1

"Reservoir"

```
lookup('P_0', r, 1) = P_0_r        //Writing Pressure at Steady State at the Reservoir
```

"Valve"

```
lookup('P_0', r, c) = P_0_v        //Writing Pressure at Steady State at the Valve
```

End

"-----Steady State Calculations-----"

"Procedure for the Steady State Calculations"

Procedure steadystate(H_r, H_v, rho, Q_0, Re_0, epsilon, D, g, A, dx, dx_int, c, theta, Reaches, alpha_0, Volume_int, Volume_ext, j : H_P_v, Q_Pd_v, Vg_P_v, t)

"Time"

t = 0

"Steady State Friction Factor"

f_0 = **frictionfactor**(Re_0, epsilon, D)

"Steady State Resistance Coefficient"

R_0 = (f_0*dx_int)/(2*g*D*A^2)

"Steady State Head at Reservoir End"

H[1] = H_r
H_r[1] = H[1]

"Steady State Volumetric Flowrate at Reservoir End"

Q[1] = Q_0

Qu_r[1] = Q[1]
Qd_r[1] = Q[1]

"Elevation from Datum"

z[1] = (1-1)*dx_int*sin(theta)

"Pressure at Steady State"

P_0[1] = rho*g*(H[1] -z[1] - H_v)

P_0_r = P_0[1]

"Steady State Gas Cavity Volume at the Reservoir End"

Vg_P_r[1] = alpha_0*Volume_ext

"Steady State Friction Factor at the Reservoir End"

f_r[1] = f_0

"Duplicate Function to Calculate Steady State Head at each Computational Node"

inc = Reaches + 1

Duplicate i = 2, inc-1

"Steady State Volumetric Flow"

Q[i] = Q[i-1]

$Q_u[i] = Q[i]$
 $Q_d[i] = Q[i]$

"Elevation from Datum"

$z[i] = (i-1)*dx_int*\sin(\theta)$

"Steady State Head"

$H_0[i] = H[i-1] - dx_int*\sin(\theta)$

$H[i] = H_0[i] - R_0*Q[i]*\mathbf{abs}(Q[i]) - (dx/(c*A))*\sin(\theta)*Q[i]$

"Pressure at Steady State"

$P_0[i] = \rho*g*(H_0[i] - z[i] - H_v)$

"Steady State Gas Cavity Volumes at the Interior Nodes"

$Vg_P[i] = \alpha_0*Volume_int$

"Steady State Friction Factor"

$f[i] = f_0$

"Calling and Writing Interior Node Values into LOOKUP TABLES"

Call writeinteriornodes(H[i], Qu[i], Qd[i], Vg_P[i], f[i], j, i : temp[i])

Call writep0interior(P_0[i], j, i: temp[i])

End

"Steady State Volumetric Flowrate and Head at the Valve End"

"Volumetric Flowrate at Valve End"

$Q[inc] = Q[inc-1]$

$Q_u_v[inc] = Q[inc]$
 $Q_d_v[inc] = Q[inc]$

$Q_Pd_v = Q_d_v[inc]$

"Elevation from Datum"

$z[inc] = (inc-1)*dx_int*\sin(\theta)$

"Head at Valve End"

$H_0[inc] = H[inc-1] - dx_int*\sin(\theta)$

$H_v[inc] = H_0[inc] - R_0*Q[inc]*\mathbf{abs}(Q[inc]) - (dx/(c*A))*\sin(\theta)*Q[inc]$

$H_P_v = H_v[inc]$

"Pressure at Steady State"

$P_0[inc] = \rho*g*(H_0[inc] - z[inc] - H_v)$

```

P_0_v = P_0[inc]

"Steady State Gas Cavity Volume at the Valve End"

Vg_P_v[inc] = alpha_0*Volume_ext

Vg_P_v = Vg_P_v[inc]

"Steady State Friction Factor at the Valve End"

f_v[inc] = f_0

"Calling and Writing Interior Node Values into LOOKUP TABLES"

Call writeexteriornodes(H_r[1], Qu_r[1], Qd_r[1], Vg_P_r[1], f_r[1], H_v[inc], Qu_v[inc], Qd_v[inc],
Vg_P_v[inc], f_v[inc], j, inc: temp)
Call writetopexterior(P_0_r, P_0_v, j, inc: temp)

End

"-----Transient Calculations-----"

"Procedure for the Transient Calculations"

Procedure transient(Reaches, H_r, c, g, dx, dx_int, dt, dt_int, D, A, mu, epsilon, psi, alpha_0,
Volume_int, Volume_ext, theta, rho, B, H_v, t_c, Q_0, x, j : H_P_v, Q_Pd_v, Vg_P_v, t, Z_P_v)

"Duplicate Function to Calculate Interior Nodes in the Transient"

Duplicate i = 2, 60

"Elevation"

Z_P[i] = (i-1)*dx_int*sin(theta)

"Fetching Downstream Node Head and Volumetric Flowrate from LOOKUP TABLES of the Previous Time Step"

H_Ld[i] = lookup('H', j-1, i-1)
Q_Ld[i] = lookup('Qd', j-1, i-1)

"Fetching Upstream Node Head and Volumetric Flowrate from LOOKUP TABLES of the Previous Time Step"

H_Ru[i] = lookup('H', j-1, i+1)
Q_Ru[i] = lookup('Qu', j-1, i+1)

"Fetching Node i Head and Volumetric Flowrate from LOOKUP TABLES of the Previous Two Time Steps"

Q_Pu0[i] = lookup('Qu', j-2, i)
Q_Pd0[i] = lookup('Qd', j-2, i)

"Fetching Node i Gas Volume from LOOKUP TABLES of the Previous Two Time Steps"

Vg_P0[i] = lookup('Vg_P', j-2, i)

```

"Fetching Upstream & Downstream Node Friction Factor from LOOKUP TABLES of the Previous Time Step"

```
f_Ld[i] = lookup('f', j-1, i-1)
f_Ru[i] = lookup('f', j-1, i+1)
```

"Pressure at Steady State"

```
P_0[i] = lookup('P_0', 1, i)
```

"Positive Characteristic Equation"

```
C_p[i] = H_Ld[i] + B*(Q_Ld[i]) - ((c*dt_int)/g)*((f_Ld[i]*Q_Ld[i]*abs(Q_Ld[i]))/(2*D*A^2)) +
(dx_int/(c*A))*sin(theta)*Q_Ld[i]
```

"Negative Characteristic Equation"

```
C_m[i] = H_Ru[i] - B*Q_Ru[i] + ((c*dt_int)/g)*((f_Ru[i]*Q_Ru[i]*abs(Q_Ru[i]))/(2*D*A^2)) +
(dx_int/(c*A))*sin(theta)*Q_Ru[i]
```

"Constants Used to Calculate the Head"

```
B_2[i] = 0.25
```

```
C_3[i] = (P_0[i]*alpha_0*Volume_int)/(rho*g)
```

```
B_v[i] = ((Vg_P0[i]/(2*dt_int)) + (1 - psi)*(Q_Pd0[i] - Q_Pu0[i]))/psi
```

```
if B_v[i] <= 0 Then
```

```
    B_v[i] = 0
```

```
Endif
```

```
C_4[i] = (B_2[i]*B*C_3[i])/(psi*dt_int)
```

```
B_1[i] = -B_2[i]*(C_p[i] + C_m[i]) + B_2[i]*B*B_v[i] + 0.5*(Z_P[i]+H_v)
```

```
B_B[i] = C_4[i]/B_1[i]^2
```

"Head Calculation"

```
If (B_1[i] < 0) AND (B_B[i] > 0.001) Then
```

```
    H_P[i] = -B_1[i]*(1 + sqrt(1 + B_B[i])) + Z_P[i] + H_v
```

```
Else
```

```
If (B_1[i] > 0) AND (B_B[i] > 0.001) Then
```

```
    H_P[i] = -B_1[i]*(1 - sqrt(1 + B_B[i])) + Z_P[i] + H_v
```

```
Else
```

```
If (B_1[i] < 0) AND (B_B[i] < 0.001) Then
```

```
    H_P[i] = -2*B_1[i] - C_4[i]/(2*B_1[i]) + Z_P[i] + H_v
```

```
Else
```

```
If (B_1[i] > 0) AND (B_B[i] < 0.001) Then
```

$$H_P[i] = C_4[i]/(2*B_1[i]) + Z_P[i] + H_v$$

Else

If B_1[i] = 0 Then

$$H_P[i] = \text{sqrt}(C_4[i]) + Z_P[i] + H_v$$

Endif

Endif

Endif

Endif

Endif

If H_P[i] < Z_P[i] + H_v Then

$$H_P[i] = Z_P[i] + H_v$$

Endif

"Volumetric Flow Calculations"

"Node Upstream Volumetric Flowrate"

$$Q_Pu[i] = (C_p[i] - H_P[i])/B$$

"Node Downstream Volumetric Flowrate"

$$Q_Pd[i] = (H_P[i] - C_m[i])/B$$

"Velocity, Reynolds Number & Steady State Friction Factor Calculation"

"Calculating Average Velocity of Flow Through the Node i"

$$V_avg[i] = 0.5*\text{abs}(Q_Pu[i] + Q_Pd[i])/A$$

"Calculating the Reynolds Number of Node i"

$$Re[i] = (\rho*V_avg[i]*D)/\mu$$

"Calling and Calculating the Friction Factor Procedure"

$$f[i] = \text{frictionfactor}(Re[i], \text{epsilon}, D)$$

"Gas Cavity Volume Calculation"

$$Vg_P[i] = Vg_P0[i] + (\text{psi}*(Q_Pd[i] - Q_Pu[i]) + (1 - \text{psi})*(Q_Pd0[i] - Q_Pu0[i]))*2*dt_int$$

if Vg_P[i] < 0 Then

$$Vg_P[i] = C_3[i]/(H_P[i] - Z_P[i] - H_v)$$

Endif

"Calling and Writing Into Interior Nodes LOOKUP TABLE"

Call writeinteriornodes(H_P[i], Q_Pu[i], Q_Pd[i], Vg_P[i], f[i], j, i : temp[i])

End

"Reservoir End Algorithm"

"Fetching Upstream Node Head, Volumetric Flowrate, SS Friction from LOOKUP TABLES of the Previous Time Step"

$H_{Ru} = \text{lookup}('H', j-1, 2)$
 $Q_{Ru} = \text{lookup}('Q_u', j-1, 2)$

$f_{Ru} = \text{lookup}('f', j-1, 2)$

"Reservoir Head"

$H_{\text{reservoir}} = H_r$

"Reservoir Elevation"

$Z_{P_r} = (1-1) \cdot dx_{\text{int}} \cdot \sin(\theta)$

"Negative Characteristic Equation"

$C_m = H_{Ru} - B \cdot Q_{Ru} + ((c \cdot dt_{\text{int}})/g) \cdot ((f_{Ru} \cdot Q_{Ru} \cdot \text{abs}(Q_{Ru})) / (2 \cdot D \cdot A^2)) + (dx_{\text{int}} / (c \cdot A)) \cdot \sin(\theta) \cdot Q_{Ru}$

"Pressure at Steady State and C_3 Constant for Reservoir End"

$P_{0_r} = \text{lookup}('P_0', 1, 1)$

$C_{3_r} = (P_{0_r} \cdot \alpha_0 \cdot \text{Volume}_{\text{ext}}) / (\rho \cdot g)$

"Calculated Volumetric Flowrate at the reservoir"

$Q_{Pd_r} = (H_{\text{reservoir}} - C_m) / B$

$Q_{Pu_r} = Q_{Pd_r}$

"Velocity, Reynolds Number & Steady State Friction Factor Calculation"

"Calculating Average Velocity of Flow Through the Node i"

$V_{\text{avg}_r} = 0.5 \cdot \text{abs}(Q_{Pu_r} + Q_{Pd_r}) / A$

"Calculating the Reynolds Number of Node i"

$Re_r = (\rho \cdot V_{\text{avg}_r} \cdot D) / \mu$

"Calling and Calculating the Friction Factor Procedure"

$f_r = \text{frictionfactor}(Re_r, \epsilon, D)$

"Gas Cavity Volume Calculation"

$Vg_{P_r} = C_{3_r} / (H_{\text{reservoir}} - Z_{P_r} - H_v)$

"Valve End Algorithm"

"Fetching Upstream Node Head, Volumetric Flowrate, SS Friction from LOOKUP TABLES of the Previous Time Step"

H_Ld = **lookup**('H', j-1, 60)
 Q_Ld = **lookup**('Qd', j-1, 60)

f_Ld = **lookup**('f', j-1, 60)

"Valve Elevation"

Z_P_v = (Reaches + 1 - 1)*dx_int***sin**(theta)

"Fetching Node i Head and Volumetric Flowrate from LOOKUP TABLES of the Previous Two Time Steps"

Vg_P0 = **lookup**('Vg_P', j-2, 61)

Q_Pd0 = **lookup**('Qd', j-2, 61)

Q_Pu0 = **lookup**('Qu', j-2, 61)

"Positive Characteristic Equation"

C_p = H_Ld + B*Q_Ld - ((c*dt_int)/g)*((f_Ld*Q_Ld***abs**(Q_Ld))/(2*D*A^2)) +
 (dx_int/(c*A))***sin**(theta)*Q_Ld

"Pressure at Steady State and C_3 Constant for Valve End"

P_0_v = **lookup**('P_0',1,61)

C_3_v = (P_0_v*alpha_0*Volume_ext)/(rho*g)

"Transient Valve Closure"

t = dt_int*(j-2)

If t < t_c **Then**

tau = 1 - (t/t_c)^x

Else

tau = 0

Endif

H_0_v = **lookup**('H', 2, 61)

C_v = (Q_0*tau)^2/(2*H_0_v)

Q_Pd_v = -B*C_v + **sqrt**((B*C_v)^2 + 2*C_v*C_p)

"Constants Used to Calculate the Head"

"Constants"

B_2 = 0.5

B_v = ((Vg_P0/(2*dt_int)) + (1 - psi)*(Q_Pd0 - Q_Pu0))/psi

If B_v <= 0 **Then**

B_v = 0

Endif

```

C_4 = (B_2*B*C_3_v)/(psi*dt_int)
B_1 = -B_2*(C_p - B*Q_Pd_v) + B_2*B*B_v + 0.5*(Z_P_v + H_v)
B_B = C_4/B_1^2

```

"Head Calculation"

```

If (B_1 < 0) AND (B_B > 0.001) Then

```

```

    H_P_v = -B_1*(1 + sqrt(1 + B_B)) + Z_P_v + H_v

```

```

Else

```

```

If (B_1 > 0) AND (B_B > 0.001) Then

```

```

    H_P_v = -B_1*(1 - sqrt(1 + B_B)) + Z_P_v + H_v

```

```

Else

```

```

If (B_1 < 0) AND (B_B < 0.001) Then

```

```

    H_P_v = -2*B_1 - C_4/(2*B_1) + Z_P_v + H_v

```

```

Else

```

```

If (B_1 > 0) AND (B_B < 0.001) Then

```

```

    H_P_v = C_4/(2*B_1) + Z_P_v + H_v

```

```

Else

```

```

If B_1 = 0 Then

```

```

    H_P_v = sqrt(C_4) + Z_P_v + H_v

```

```

Endif

```

```

Endif

```

```

Endif

```

```

Endif

```

```

Endif

```

```

If H_P_v < Z_P_v + H_v Then

```

```

    H_P_v = Z_P_v + H_v

```

```

Endif

```

"Volumetric Flow Calculations"

```

Q_Pu_v = (C_p - H_P_v)/B

```

"Velocity, Reynolds Number & Steady State Friction Factor Calculation"

"Calculating Average Velocity of Flow Through the Node i"

```

V_avg_v = 0.5*abs(Q_Pu_v + Q_Pd_v)/A

```

"Calculating the Reynolds Number of Node i"

```

Re_v = (rho*V_avg_v*D)/mu

"Calling and Calculating the Friction Factor Procedure"

f_v = frictionfactor(Re_v, epsilon, D)

"Gas Cavity Volume Calculation"

Vg_P_v = Vg_P0 + (psi*(Q_Pd_v - Q_Pu_v) + (1 - psi)*(Q_Pd0 - Q_Pu0))*2*dt_int

If Vg_P_v < 0 Then

    Vg_P_v = C_3_v/(H_P_v - Z_P_v - H_v)

Endif

"Setting the last column number"

r = 61

"Calling and Writing Into Interior Nodes LOOKUP TABLE"

Call writeexteriornodes(H_reservoir, Q_Pu_r, Q_Pd_r, Vg_P_r, f_r, H_P_v, Q_Pu_v,
Q_Pd_v, Vg_P_v, f_v, j, r: temp)
End

"-----Fixed Input Variables-----"

"Pipe Geometry"

L = 15.3           [m]           //Length
D = 0.0274         [m]           //Diameter
wt = 0.0027        [m]           //Wall Thickness
epsilon = 1.5E-04  [m]           //Absolute roughness

A = 0.25*pi*D^2    //Area

B = c/(g*A)        //Pipe Constant

Period = (2*L)/c   //Period of the Pipe

"Pipe Material Properties"

E = 200E9          [Pa]          //Young's Modulus
nu = 0.29          [-]           //Poisson Ratio

"Fluid Properties"

rho = 999.2         [kg/m^3]     //Density
K = 2.14E9          [Pa]         //Bulk Modulus
mu = 1.168E-3       [kg/ms]     //Dynamic Viscosity

H_sat = 0.1630      [m]           //Saturation Pressure Head in Piezometric Head
H_v = H_sat - H_b   //Saturation Pressure Head in Gauge Pressure Head

"Initial Boundary Conditions"

Q_0 = 0.00037983   [m3/s]       //Initial Volume Flowrate

```

$H_r = 7.9955$ [m] //Reservior Head
 $P_b = 87.9$ [kPa] //Barometric Pressure
 $H_b = (P_b \cdot 1000) / (\rho \cdot g)$ //Barometric Pressure Head
 $V_0 = Q_0 / A$ //Initial Velocity
 $Re_0 = (\rho \cdot V_0 \cdot D) / \mu$ //Initial Reynolds Number
 $f_0 = \text{frictionfactor}(Re_0, \epsilon, D)$ //Initial Friction Factor
 $g = 9.81$ [m/s²] //Gravitational Constant
 $\theta = 0$ //Pipe Slope
 $Z_{P_v_test} = L \cdot \sin(\theta)$

"Valve Properties"

$t_c = 0.0053$ [s] //Valve Closure Time
 $x = 1$ [-] //Valve Closure Characteristic

"DGCM Properties"

$\psi = 1.0$ [-] //Weighing Factor
 $\alpha_0 = 1E-7$ [-] //Void Fraction @ Reference Pressure
 $c_1 = 2 \cdot (wt/D) \cdot (1 + \nu) + D \cdot (1 - \nu^2) / (D + wt)$ //Poisson Ration Coefficient
 $c = \sqrt{(K/\rho) / (1 + ((K/E) \cdot (D/wt)) \cdot c_1)}$ //Speed of Sound in Water

"Numerical Solver Properties"

$Reaches = 66$ [-] //Desired Number of Increments
 $Oscillations = 2$ [-] //Number of Peaks and Troughs to be Modelled - One
 Oscillation is 4x the Travelling Time of the Pressure Wave

$t_{trav} = L/c$ //Travel Time of Pressure Wave from Valve to Reservoir
 $dt_{int} = t_{trav}/Reaches$ //Intermediate Calculated Time Step
 $t_{max} = 4 \cdot Oscillations \cdot t_{trav}$ //Calculated Maximum Simulation Time
 $dx_{int} = Courant \cdot c \cdot dt_{int}$ //Intermediate Calculated Increment Size

$n = \text{round}((L/dx_{int}) + 1)$ //Calculated Number of Increments
 $m = \text{round}((t_{max}/dt_{int}) + 1)$ //Calculated Number of Time Steps

$dx = L/n$ //Calculated Increment Size
 $dt = t_{max}/m$ //Calculated Time Step Size

$Volume_{int} = A \cdot dx_{int}$ //Interior Reach Volumes
 $Volume_{ext} = 0.5 \cdot A \cdot dx_{int}$ //Exterior Reach Volumes

$Courant = 1$ //The Courant number is a measure of how much information
 traverses (c) a computational grid cell (dx) in a given time-step (dt), Courant < 1

"-----Initializing & Calling SteadyState and Transient Procedures-----"

"Calling the SteadyState Procedure to Run Through the First Two Timesteps"

Duplicate j = 1, 2

Call steadystate(H_r, H_v, rho, Q_0, Re_0, epsilon, D, g, A, dx, dx_int, c, theta, Reaches, alpha_0, Volume_int, Volume_ext, j : H_P_v[j], Q_Pd_v[j], Vg_P_v[j], t[j])

End

"Calling the Transient Procedure to Run Through the Transient"

Duplicate j = 3, 1500

Call transient(Reaches, H_r, c, g, dx, dx_int, dt, dt_int, D, A, mu, epsilon, psi, alpha_0, Volume_int, Volume_ext, theta, rho, B, H_v, t_c, Q_0, x, j : H_P_v[j], Q_Pd_v[j], Vg_P_v[j], t[j], Z_P_v[j])

End

"-----END-----"

ANNEXURE B: SPRING-MASS MATHEMATICAL MODEL

The fast-closing valve used in the experimental test bench (see Figure 4-2) utilises a spring loading mechanism enabling the valve to close within the specified time of $T_c \leq 5.53$ ms (refer to Section 4.3.1). A spring-mass mathematical model is derived to determine the spring stiffness necessary to obtain said valve closure time.

Referring to the figure below, the spring-mass mathematical model was derived as follows.

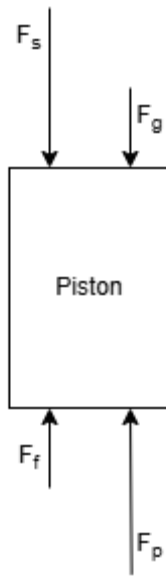


Figure B - 1: Free body diagram of forces acting upon the fast-closing valve piston.

Where F_s = force of the spring, F_g = force of gravity, F_f = friction force (from the O-ring) and F_p = force of water pressure.

Assuming that the spring has an uncompressed (natural) length of L , a spring constant of k , the spring is ideal (i.e. massless with no damping), and the mass attached (piston) has a mass of m , Hooke's Law can be used to describe the motion of the mass.

Initially, the spring is compressed at a distance x from its natural length (L), so the initial length of the spring is $x_0 = L - x$. The opposing forces, F_f and F_p , can be grouped into one force, namely F_o . Using Newton's Law, the force balance (equation of motion) of the system depicted in the figure above can be written as follows:

$$F_g + F_s = F_o \quad (B.1)$$

$$ma(t) + kx(t) = F_o \quad (B.2)$$

where $a(t)$ is the acceleration of the piston at time t , and $x(t)$ is the system's displacement at time t . Since the opposing force is constant at a specific time step t , a solution for the displacement can be assumed as:

$$x(t) = A\cos(\omega t) + B\sin(\omega t) \quad (B.3)$$

where A and B are constants and ω is the angular frequency of the system. Taking the first and second derivatives of $x(t)$, the following equations are obtained:

$$v(t) = -A\omega\sin(\omega t) + B\omega\cos(\omega t) \quad (B.4)$$

$$a(t) = -A\omega^2\cos(\omega t) - B\omega^2\sin(\omega t) \quad (B.5)$$

Substituting the above equations of $x(t)$, $v(t)$ and $a(t)$ into equation B.2 and simplifying, the following equations are obtained:

$$-m\omega^2 A + kB = 0 \quad (B.6)$$

$$-m\omega^2 B - kA = F_o \quad (B.7)$$

Solving for A and B follows that:

$$A = \frac{-F_o}{m\omega^2 B} = 0 \quad (B.8)$$

Therefore, the displacement equation for a constant opposing force is:

$$x(t) = \frac{F_o}{k}(1 - \cos(\omega t)) + x_0\cos(\omega t) \quad (B.9)$$

where x_0 is the initial displacement of the piston. Similarly, the velocity and acceleration equations can be obtained by taking the time derivatives of the displacement equation:

$$v(t) = -\omega x_0\sin(\omega t) + \frac{F_o}{k}\sin(\omega t) \quad (B.10)$$

$$a(t) = -\omega^2 \left[\left(x_0 - \frac{F_0}{k} \right) \cos(\omega t) + \frac{F_0}{k} \right] \quad (B.11)$$

The angular frequency of the system (ω) can be calculated by.

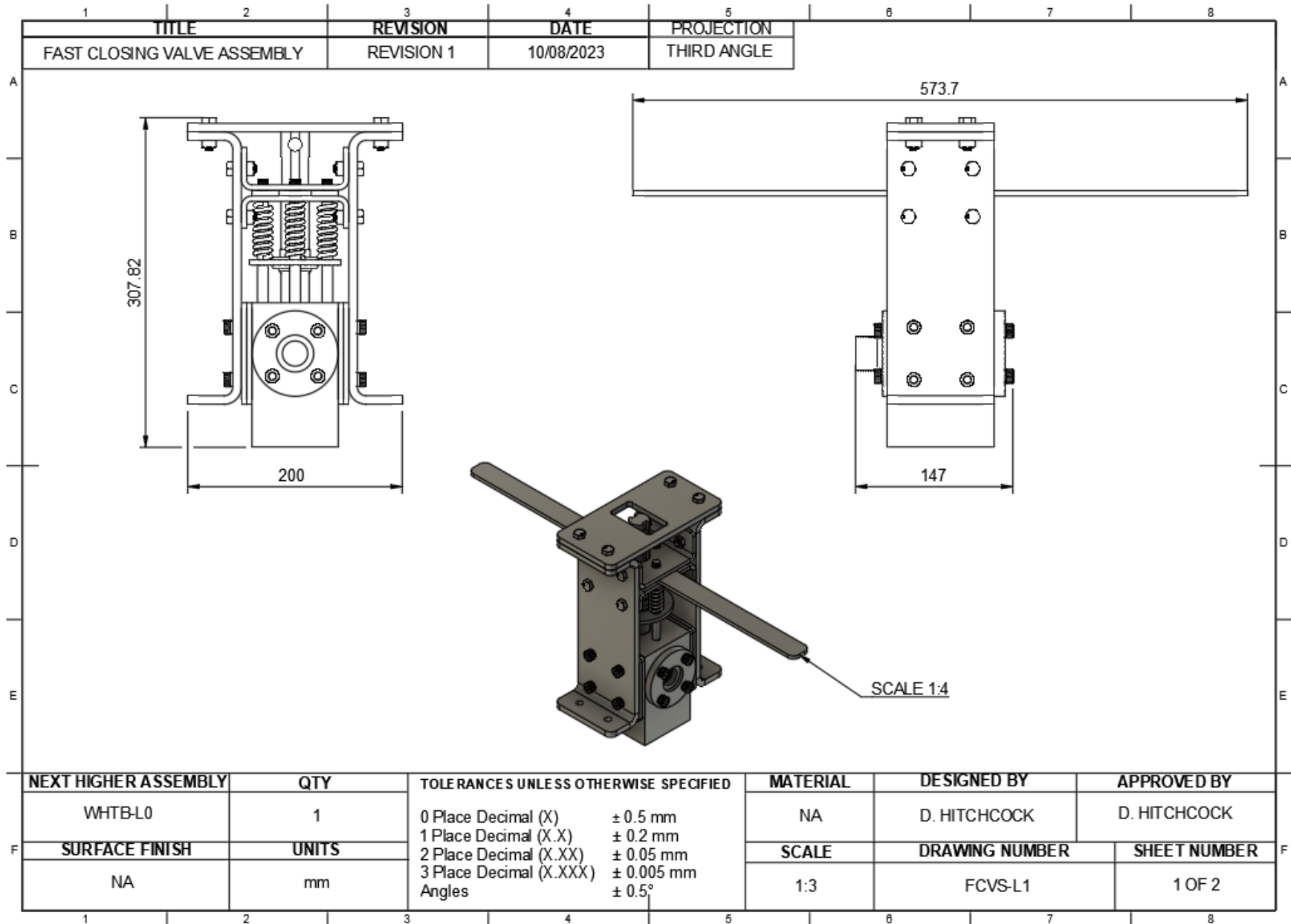
$$\omega = \sqrt{k/m} \quad (B.12)$$

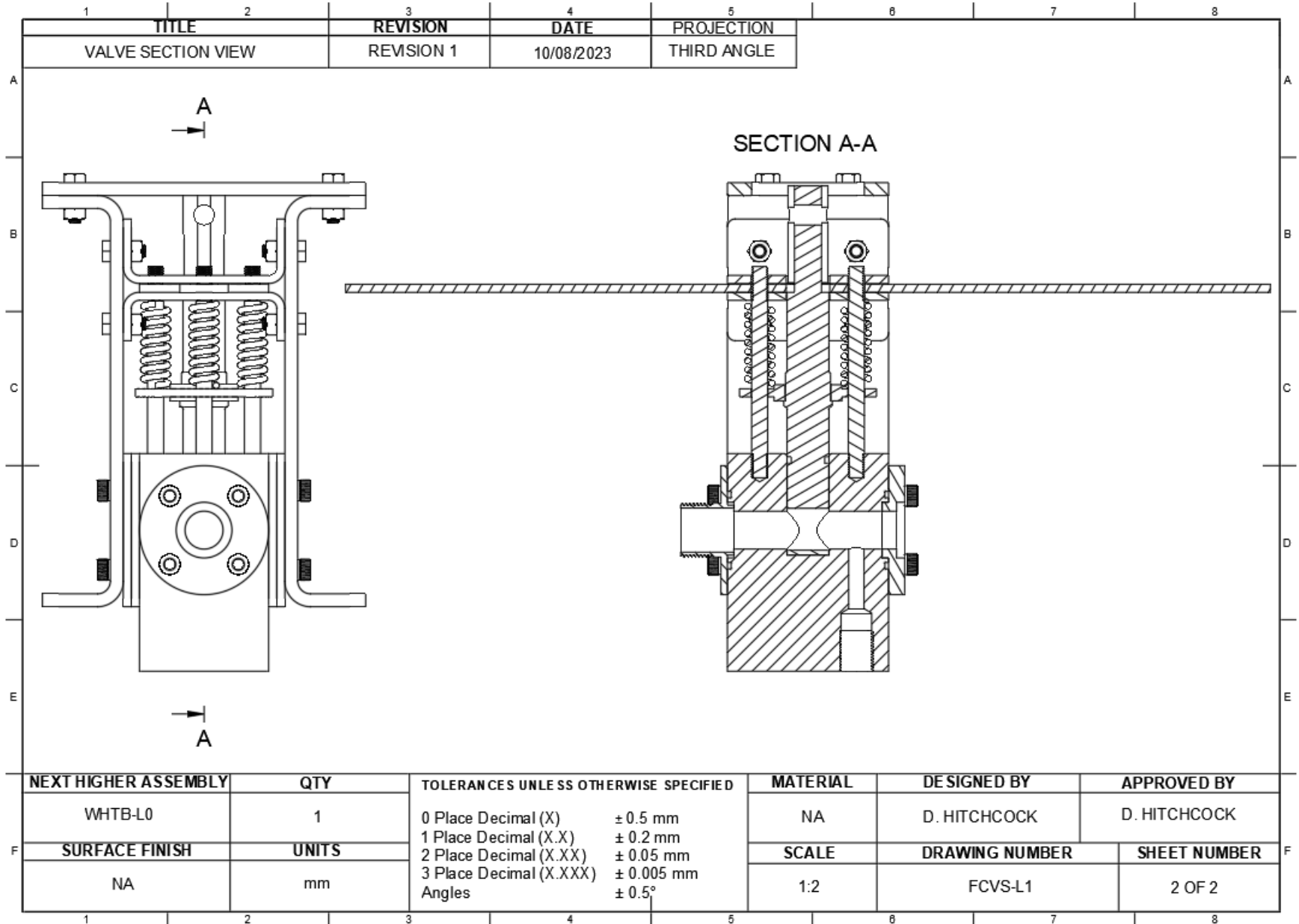
ANNEXURE C: FAST-CLOSING VALVE DESIGN

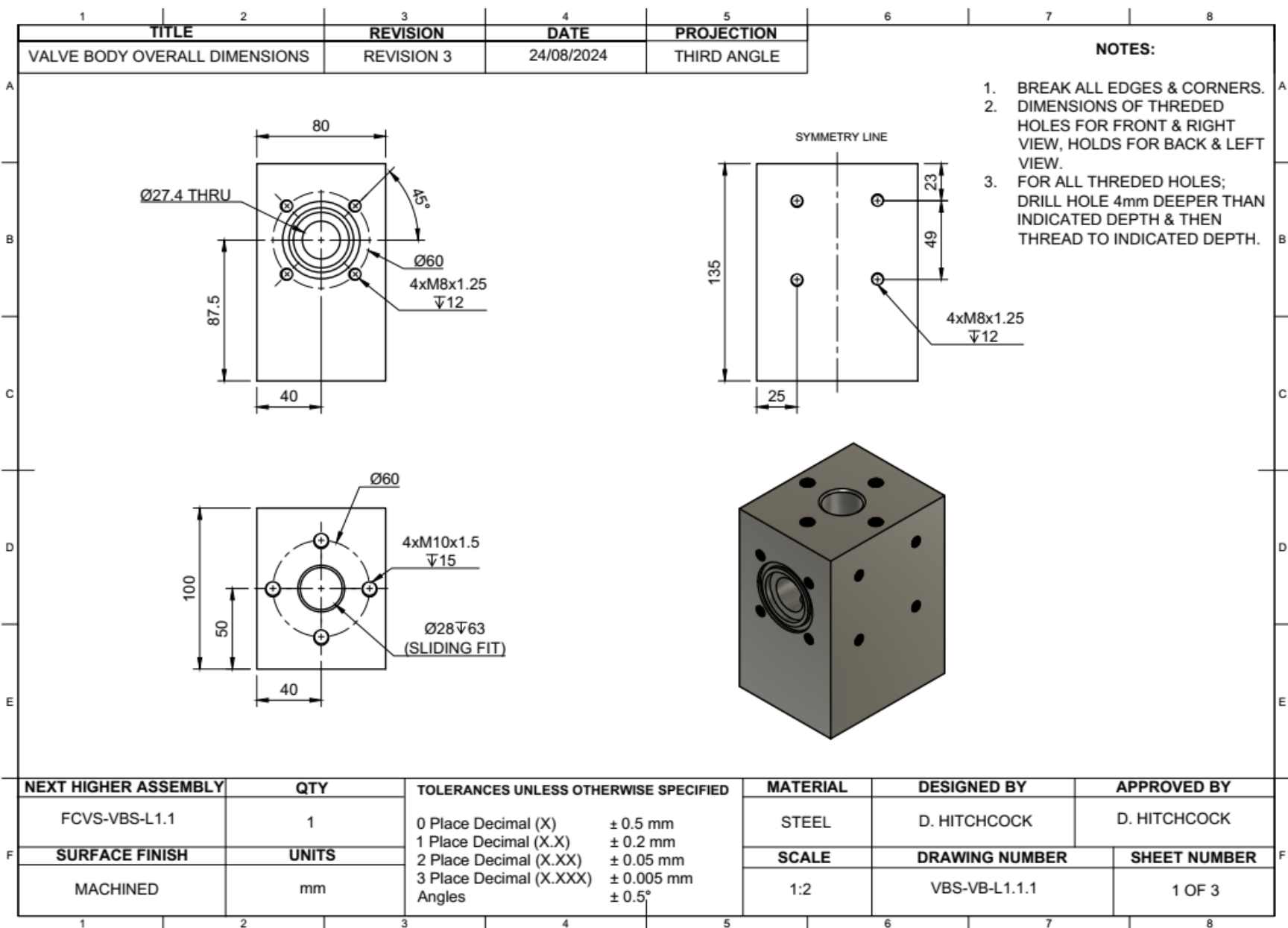
The video of the valve closing during the commissioning and testing stage can be seen here:

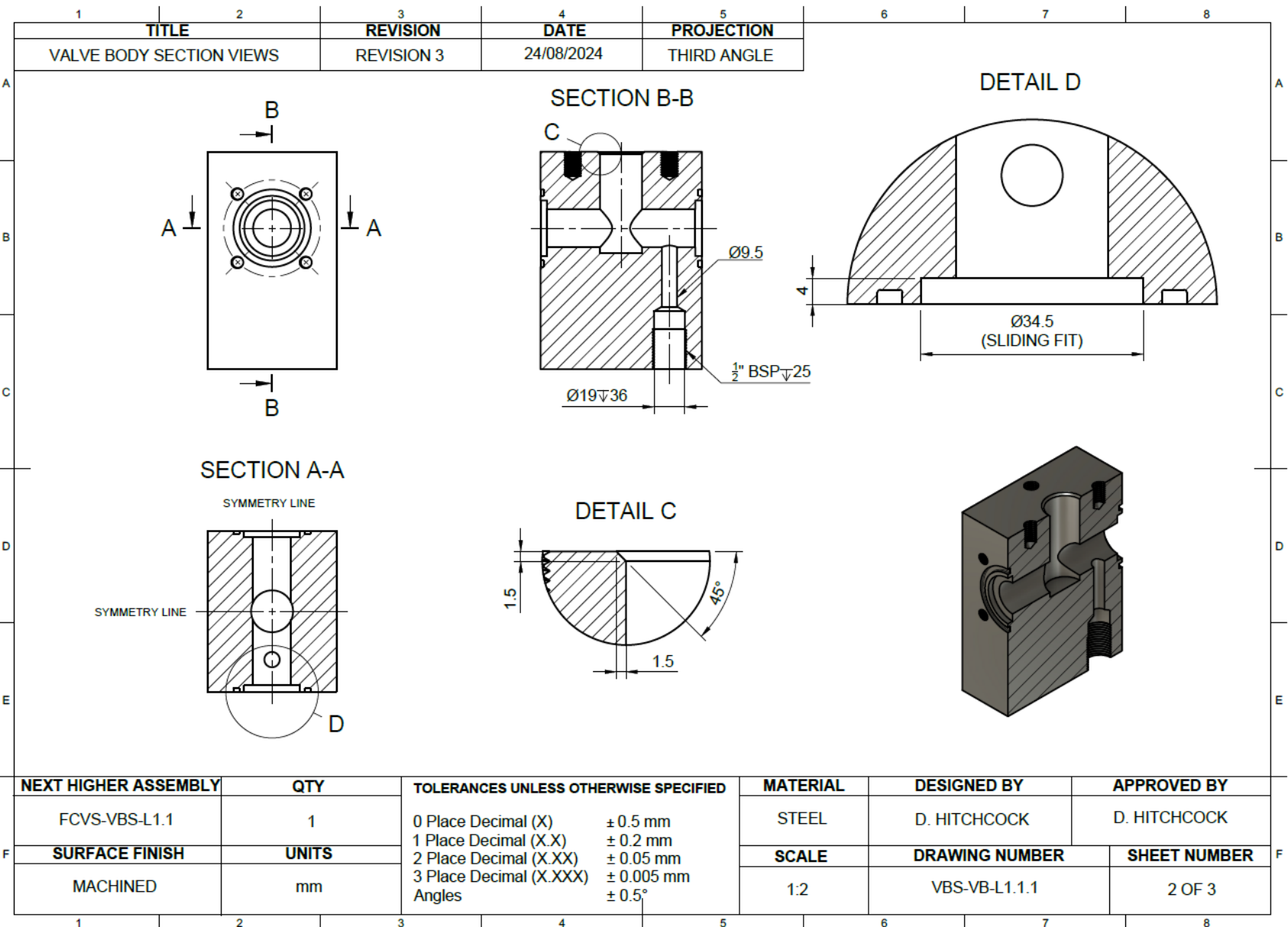
https://drive.google.com/file/d/1wYTKGqugFU3jwSipRsSRoVRRdw4wShr/view?usp=drive_link

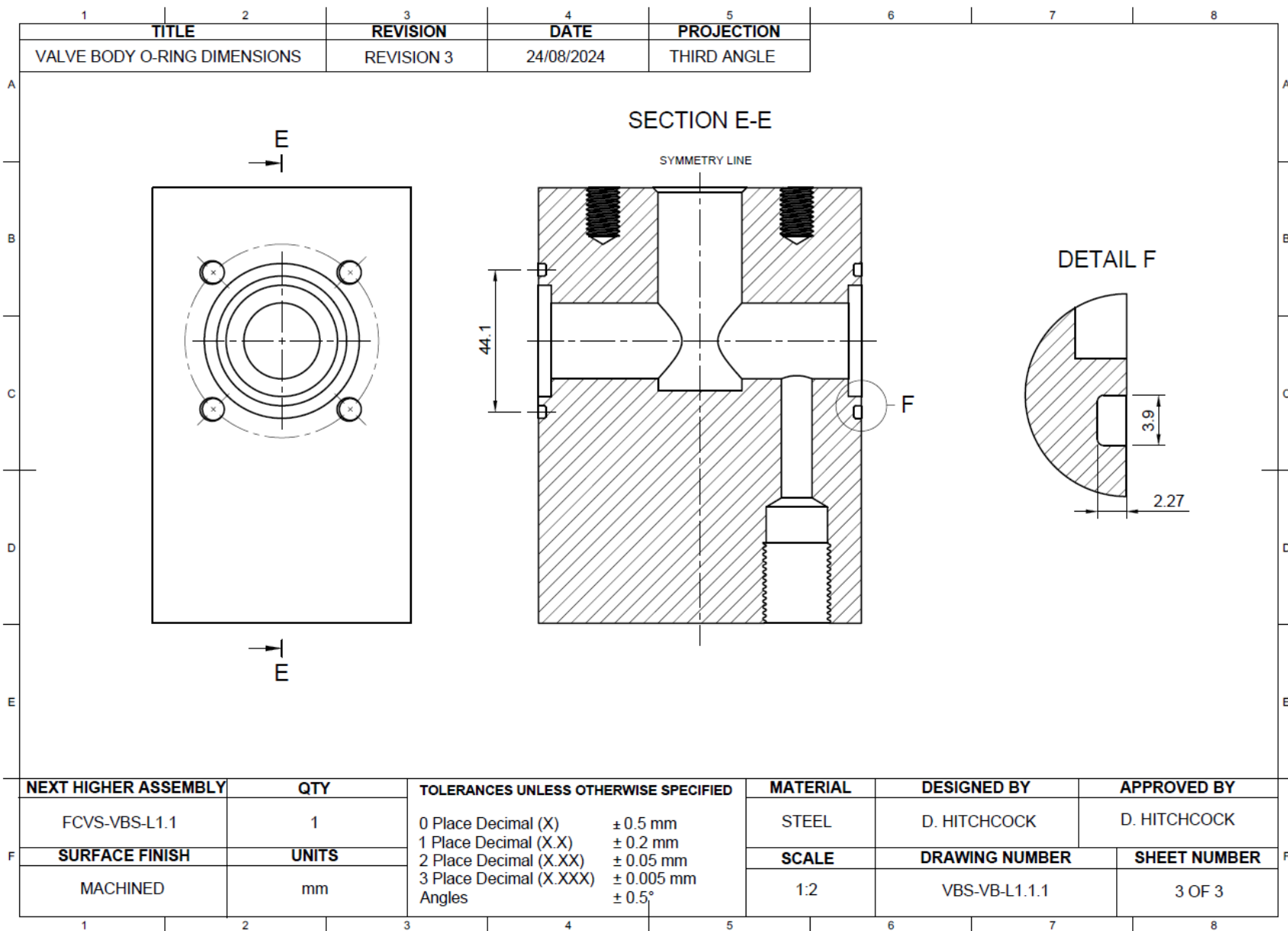
Some manufacturing drawings of the fast-closing valve follow on the next page.

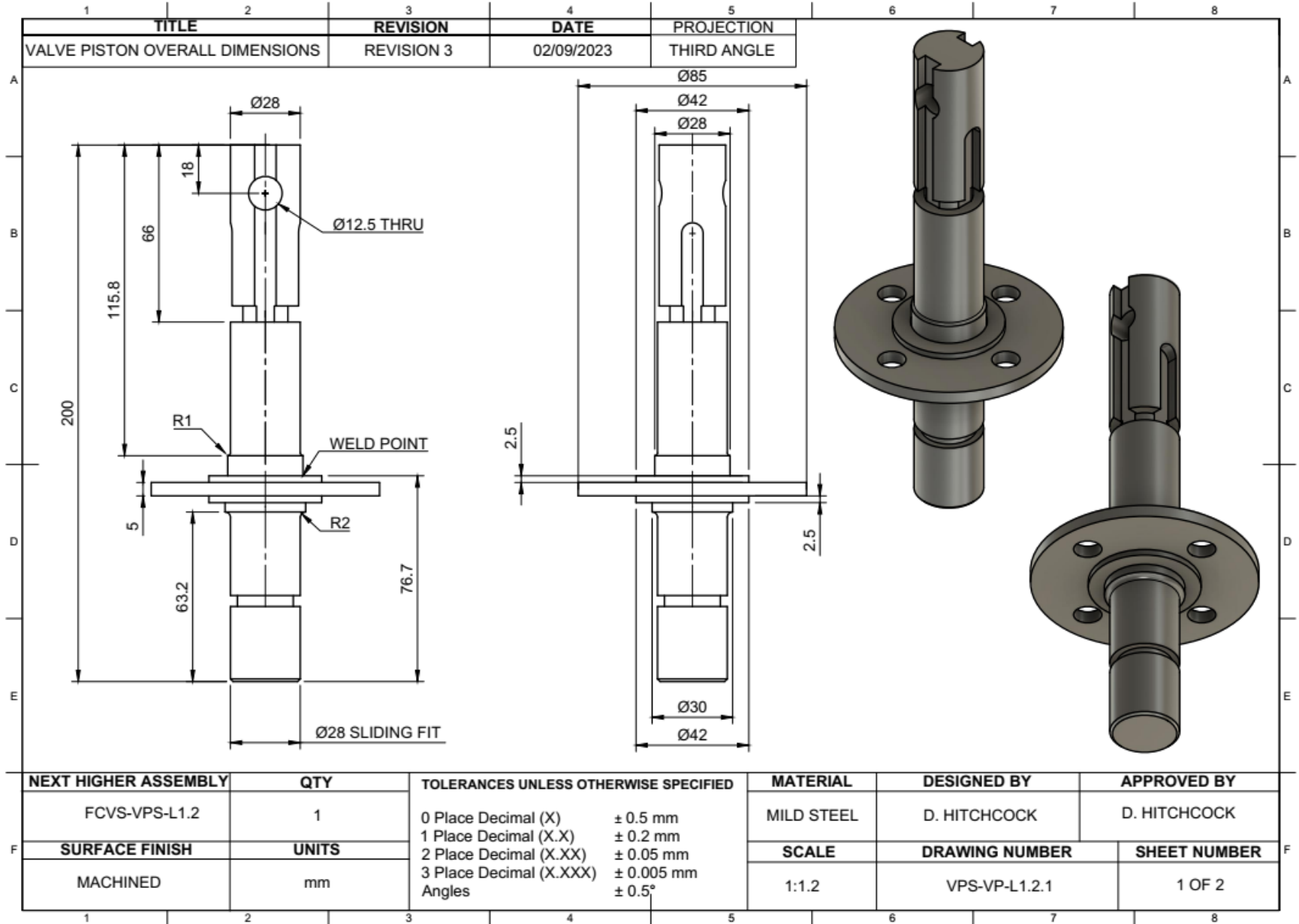


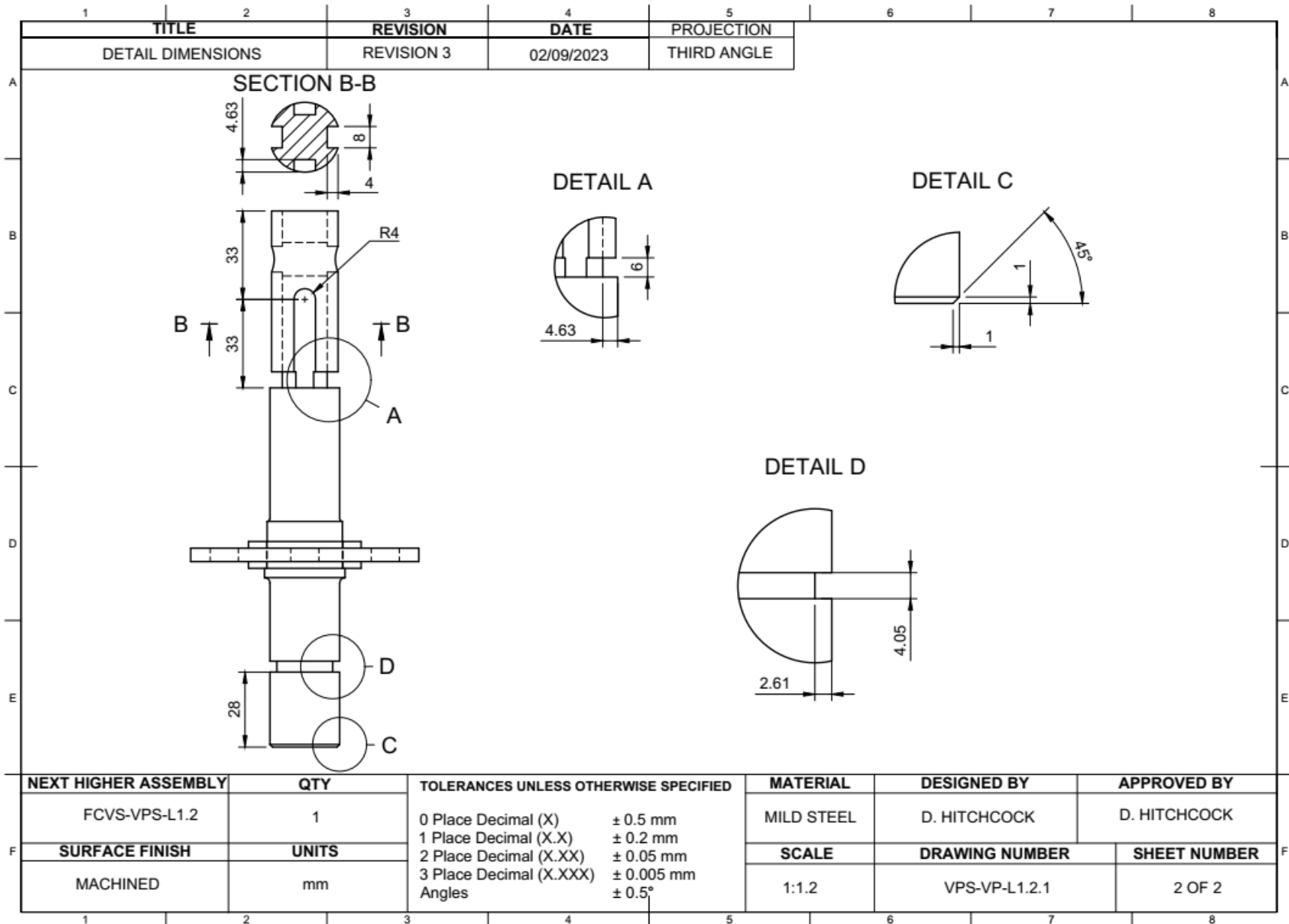


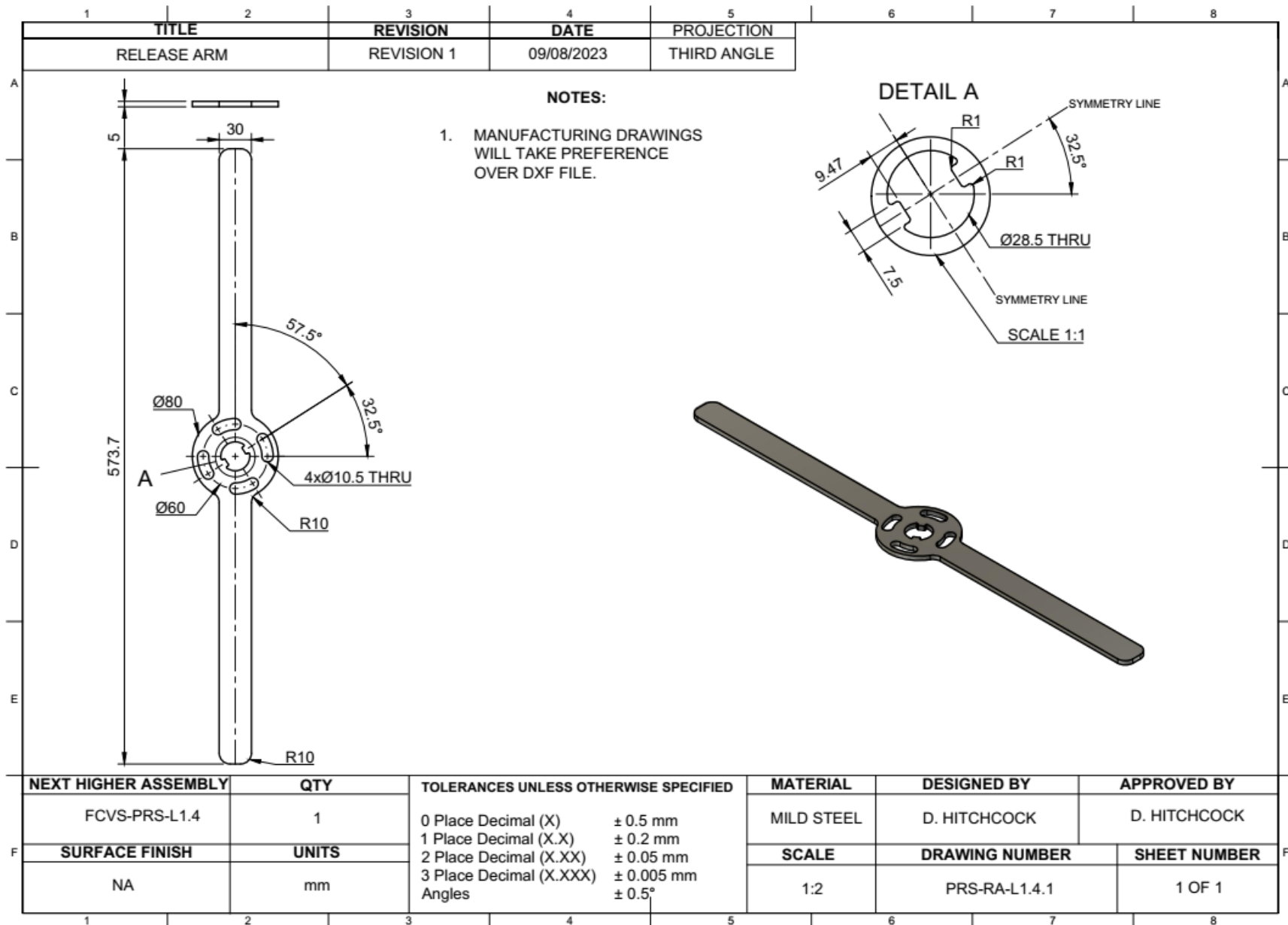


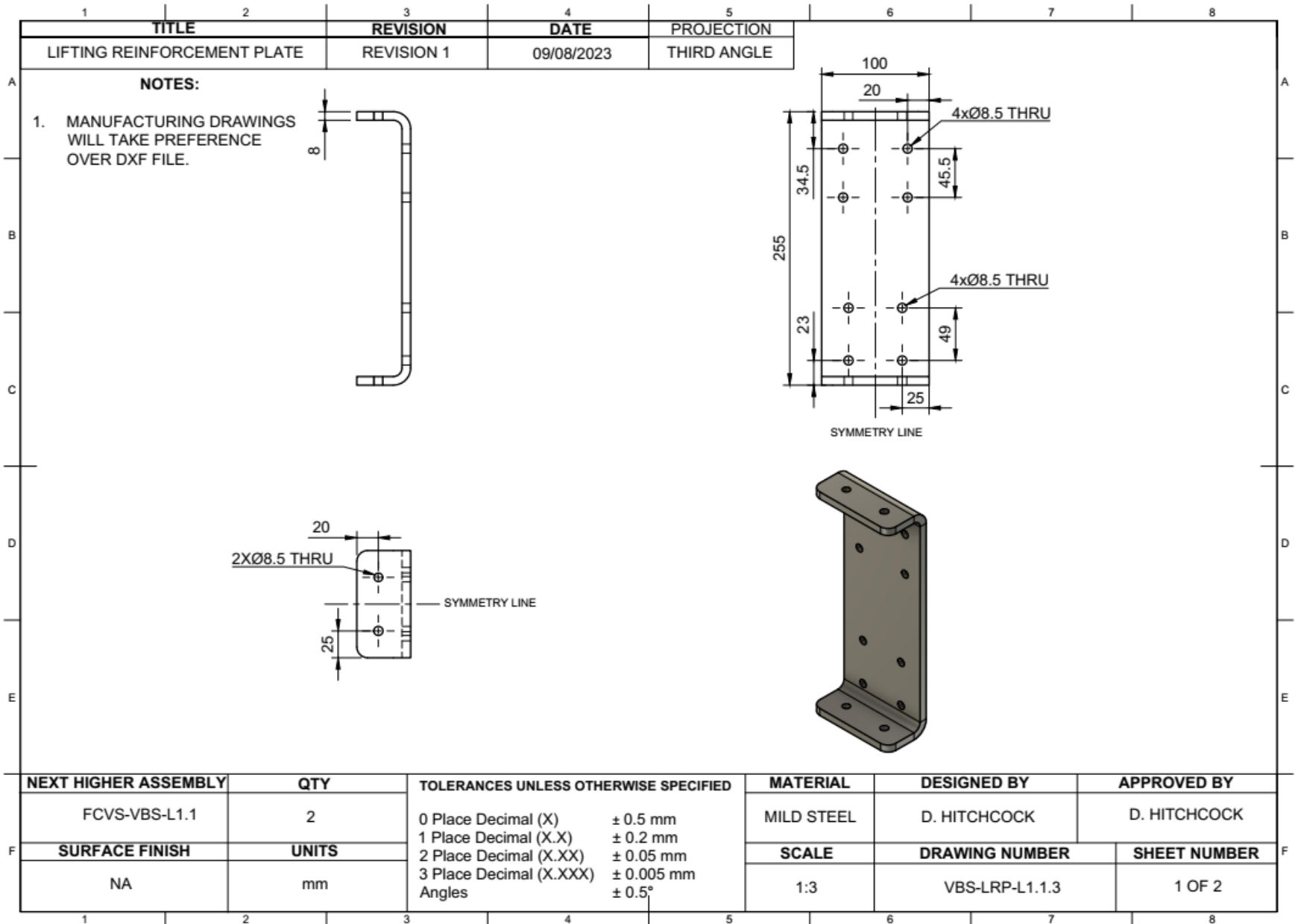


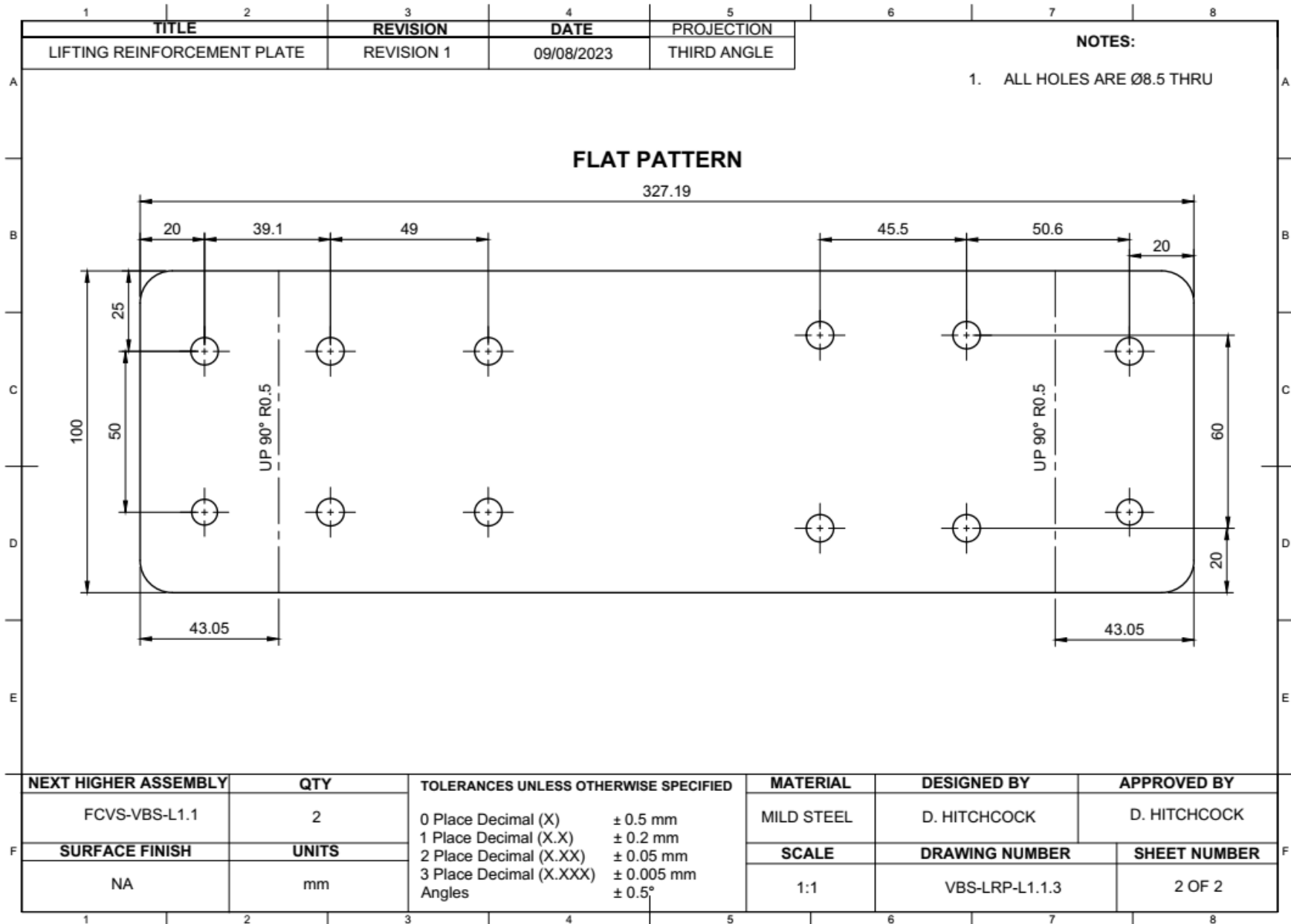












ANNEXURE D: CALIBRATION CERTIFICATE

**WIKA Instruments (Pty) Ltd.
South Africa**

**SANAS Accredited Laboratory
Number 246
for
Pressure Metrology**



Certificate Number

02-13451

CALIBRATION CERTIFICATE

Calibration of a	Pressure transmitter	This calibration certificate documents the traceability to national standards and international standards, which realise the units of measurement according to the International System of Units (SI).
Manufacturer	WIKAI	
Type	891.13.500	The South African National Accreditation System (SANAS) is a member of the International Laboratory Accreditation Cooperation (ILAC) Mutual Recognition Arrangement (MRA). This arrangement allows for the mutual recognition of technical test and calibration data by member accreditation bodies worldwide. For more information on the arrangement please consult www.ilac.org
Serial number	2363591	
Customer	North-West University PC-G15B Room #G01 11 Hoffman Street Potchefstroom 2531	The measurement results recorded in this certificate were correct at the time of calibration and the results are only valid for the instrument calibrated. The subsequent accuracy of the instrument will depend on factors such as care, handling, frequency of use, operating temperature and use under conditions other than specified by the manufacturer and or conditions of calibration. It is recommended that recalibration be undertaken, by the user, at an interval that will ensure that the instrument remains within the desired limits.
Order No.	347899	
Number of pages of the calibration certificate	5	Legal liability shall be limited to the cost of recalibration and or certification, but the applicant indemnifies WIKAI INSTRUMENTS against any consequential or other loss.
Date of calibration	11/08/2023	

This calibration certificate may not be reproduced other than in full except with the permission of the issuing laboratory. Calibration certificates without Technical Signatory's signature are not valid.

Date of issue	Calibrated by Calibration Technician	Technical Signatory
11/08/2023	 Lebogang Rabaloyi Laboratory Assistant	 Peresh Wellcome Metrologist and Laboratory Manager

WIKAI Instruments (Pty) Ltd.
Chivers Street, Denver, 2094
Johannesburg, South Africa
PO Box 75225, Gardenview, 2047

Tel: (011) 621 0000
Fax: (011) 621 0059
email: sales.za@wika.com

Device Under Test (DUT)

Pressure transmitter

Model	891.13.500		
Serial No.	2363591		
Scale range	(0 ... 16) bar g		
Signal	(4 ... 20) mA		
Accuracy	0.1 % FS		
Resolution (scale division)	0.001	mA	0.001 bar
Tag No.	N/A		

Calibration Method

Calibration was carried out according to DKD-R 6-1 "Calibration of Pressure Gauges" (Edition 03/2014) calibration sequence A, by comparing the measured values of the DUT with those of the Reference or Working Standard.
LW-007 r8 & LW-012 r9

Place of calibration

WIKSA SA Jhb Cal Lab

$g = (9.7855 \pm 0.000005) \text{ m/s}^2$

Measurement conditions

Working standard (WS) Pressure controller

Model	CPC8000 Z80513C
Calibration mark	218749 A2LA 11/22
Scale range	(-1 ... 60) bar g
Accuracy	0.00541 % FS

WIKSA Instruments (Pty) Ltd.
Chilvers Street, Denver, 2094
Johannesburg, South Africa
PO Box 75225, Gardenview, 2047
Tel: (011) 621 0000
Fax: (011) 621 0059
email: sales.za@wika.com



Measurement conditions

Digital multimeter

Model	Keithley 2000 DMM
Calibration mark	124504-1 SANAS 05/23

Assembly of the DUT at the time of calibration

Pressure medium	Dry air
Reference height (for reference value)	Connection of object DUT
Position during calibration	vertical

Environmental conditions

Temperature	21.9 ± 0.50 °C
Atmospheric pressure	841.0 ± 0.80 hPa
Rel. Humidity	31.0 ± 3.00 %

Measurement results

Calc. reference value	Reading DUT		
mA	mA		
	M1 ↑	M2 ↓	M3 ↑
4.000	4.000	4.000	4.000
6.000	5.999	5.994	5.999
8.000	7.995	7.987	7.995
10.000	9.992	9.984	9.992
12.000	11.991	11.983	11.991
14.000	13.992	13.984	13.992
16.000	15.993	15.987	15.993
18.000	18.000	17.994	18.000
20.000	20.004	20.004	20.004

Measurement results

$p \text{ in bar} = (\text{Reading} - I_{\min}) \cdot ((P_{\min} - P_{\max}) / (I_{\min} - I_{\max})) + P_{\min}$
 Alternative calculation: $p \text{ in bar} = \text{Reading} \cdot 1 \text{ bar/mA} - 4 \text{ bar}$

Reading Reference WS bar	Calculated pressure DUT bar		
	M1 ↑	M2 ↓	M3 ↑
0.0000	0.000	0.000	0.000
2.0000	1.999	1.994	1.999
4.0000	3.995	3.987	3.995
6.0000	5.992	5.984	5.992
8.0000	7.991	7.983	7.991
10.0000	9.992	9.984	9.992
12.0000	11.993	11.987	11.993
14.0000	14.000	13.994	14.000
16.0000	16.004	16.004	16.004

Evaluation of measurement results

The deviation must be subtracted algebraically from the DUT reading to obtain the correct value.

Reading Reference WS bar	Mean Value bar	Deviation bar	Repeatability bar	Hysteresis bar	Uncertainty of Measurement bar
0.0000	0.000	0.000	0.000	0.000	0.0064
2.0000	1.997	-0.003	0.000	0.005	0.0064
4.0000	3.991	-0.009	0.000	0.008	0.0067
6.0000	5.988	-0.012	0.000	0.008	0.0067
8.0000	7.987	-0.013	0.000	0.008	0.0067
10.0000	9.988	-0.012	0.000	0.008	0.0068
12.0000	11.990	-0.010	0.000	0.006	0.0064
14.0000	13.997	-0.003	0.000	0.006	0.0064
16.0000	16.004	0.004	0.000	0.000	0.0064

The reported expanded uncertainty of measurement is valid after a correction of the reading value with the systematical deviation (see table "evaluation").

The reported expanded uncertainty of measurement is stated as the standard uncertainty of measurement multiplied by the coverage factor $k=2$, which for a normal distribution corresponds to a coverage probability of 95,45%, the uncertainty of measurement has been estimated in accordance with the principles defined in the GUM, 'Guide to Uncertainty of Measurement, ISO, Geneva, 1993'.

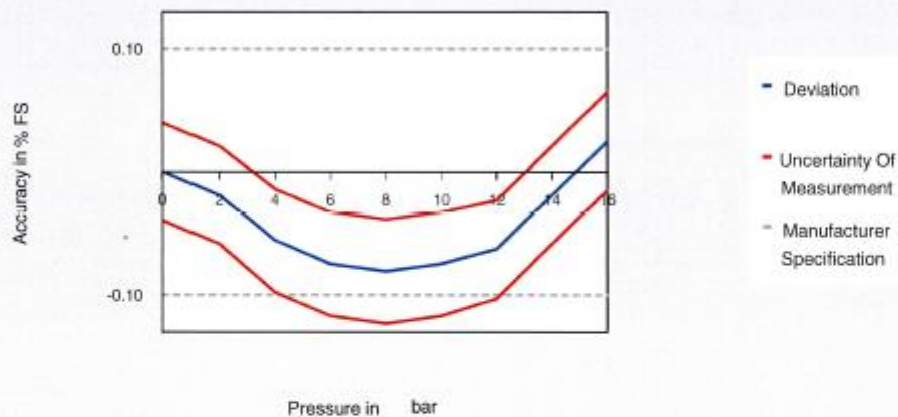
To convert in official-unit -kPa- use multiplier : 100 kPa/bar

M1 & M3 Indicates Rising Pressure, M2 Indicates Falling Pressure

NOTE: All readings entered on computer directly.

Evaluation of measurement results

Graphical representation of measurement results



The accuracy of the DUT is

0.02 bar

Statement of compliance

Additional information

-----END OF CERTIFICATE-----

WIKI Instruments (Pty) Ltd.
Chilvers Street, Denver, 2094
Johannesburg, South Africa
PO Box 75225, Gardenview, 2047
Tel.: (011) 621 0000
Fax: (011) 621 0069



ANNEXURE E: TIEPIE OSCILLOSCOPE DATASHEET

The datasheet of the TiePie Handyscope HS4-DIFF oscilloscope can be viewed here:
<https://www.tiepie.com/en/usb-oscilloscope/handyscope-hs4-diff>

ANNEXURE F: RESISTOR DATASHEET



www.vishay.com

CMF (Military RN and RL)

Vishay Dale

Metal Film Resistors, Military, MIL-R-10509 Qualified, Precision, Type RN and MIL-PRF-22684 Qualified, Type RL



FEATURES

- Very low noise (- 40 dB)
- Very low voltage coefficient (5 ppm/V)
- Controlled temperature coefficient
- Flame retardant epoxy coating
- Commercial alternatives to military styles are available with higher power ratings. See CMF Industrial data sheet: (www.vishay.com/doc?31018)

STANDARD ELECTRICAL SPECIFICATIONS

GLOBAL MODEL	MIL STYLE	MIL SPEC. SHEET	POWER RATING $P_{70^{\circ}\text{C}}$ W	POWER RATING $P_{125^{\circ}\text{C}}$ W	MAX. WORKING VOLTAGE ⁽¹⁾ V	RESISTANCE RANGE Ω MIL-R-10509 $\pm 100 \text{ ppm}/^{\circ}\text{C}$ (D)	RESISTANCE RANGE Ω MIL-R-10509 $\pm 50 \text{ ppm}/^{\circ}\text{C}$ (C)	RESISTANCE RANGE Ω MIL-R-10509 $\pm 25 \text{ ppm}/^{\circ}\text{C}$ (E)	RESISTANCE RANGE Ω MIL-PRF-22684	TOL. ⁽²⁾ $\pm \%$	DIELECTRIC STRENGTH V_{AC}
CMF50	RN50	08	-	0.05	200	-	10 to 100K	10 to 100K	-	0.1, 0.25, 0.5, 1	450
CMF55	RN55	07	0.125	0.10	200	10 to 301K	49.9 to 100K	49.9 to 100K	-	0.1, 0.25, 0.5, 1	450
CMF60	RN60	01	0.25	0.125	300	10 to 1M	49.9 to 499K	49.9 to 499K	-	0.1, 0.25, 0.5, 1	500
CMF65	RN65	02	0.50	0.25	350	10 to 2M	49.9 to 1M	49.9 to 1M	-	0.1, 0.25, 0.5, 1	900
CMF70	RN70	03	0.75 ⁽²⁾	0.50	500	10 to 2.49M	24.9 to 1M	24.9 to 1M	-	0.1, 0.25, 0.5, 1	900
CMF07	RL07	01	0.25	-	250	-	-	-	51 to 150K	2, 5	450
CMF20	RL20	02	0.50	-	350	-	-	-	4.3 to 470K	2, 5	700

Notes

⁽¹⁾ Continuous working voltage shall be $\sqrt{P \times R}$ or maximum working voltage, whichever is less.

⁽²⁾ Formerly rated at 1 W and is the direct replacement for RN70 of MIL-R-10509 rev. D.

⁽³⁾ Tolerances of $\pm 0.1 \%$, $\pm 0.25 \%$ and $\pm 0.5 \%$ are not applicable to characteristic D.

TECHNICAL SPECIFICATIONS

PARAMETER	UNIT	CONDITION
Voltage Coefficient	ppm/V	5 when measured between 10 % and full rated voltage
Insulation Resistance	Ω	$\geq 10^{10}$ min. dry; $\geq 10^8$ min. after moisture test
Operating Temperature Range	$^{\circ}\text{C}$	- 65/+ 175 (see derating curves for military range)
Terminal Strength	lb	5 pound pull test for RL07/RL20; 2 pound pull test for all others
Solderability		Continuous satisfactory coverage when tested in accordance with MIL-R-10509 and MIL-PRF-22684



GLOBAL PART NUMBER INFORMATION

New Global Part Numbering: RN60D3483FR36 (preferred part numbering format)

MIL STYLE	CHARACTERISTIC	RESISTANCE VALUE	TOLERANCE CODE	PACKAGING	SPECIAL
RN50 RN55 RN60 RN65 RN70	E = 25 ppm C = 50 ppm D = 100 ppm	3 digit significant figure, followed by a multiplier Use "R" for values < 100 Ω 10R0 = 10 Ω 2152 = 21.5 kΩ 2494 = 2.49 MΩ	B = ± 0.1 % C = ± 0.25 % D = ± 0.5 % F = ± 1 %	B14 = Tin/lead, bulk BSL = Tin/lead, bulk, single lot date code R36 = Tin/lead, T/R (full) RE6 = Tin/lead, T/R (1000 pieces) RSL = Tin/lead, T/R, single lot date code	Blank = Standard (Dash number) 88 = Hot solder dip 143 = Non-magnetic

Historical Part Number example: RN60D3483F (will continue to be accepted)

RN60	D	3483	F	R36
MIL STYLE	CHARACTERISTIC	RESISTANCE VALUE	TOLERANCE CODE	PACKAGING

New Global Part Numbering: RL07S471JR36 (preferred part numbering format)

MIL STYLE	LEAD MATERIAL	RESISTANCE VALUE	TOLERANCE CODE	PACKAGING	SPECIAL
RL07 RL20	S = Solderable	2 digit significant figure, followed by a multiplier Use "R" for values < 10 Ω 4R3 = 4.3 Ω 202 = 2.0 kΩ 474 = 470 kΩ	G = ± 2 % J = ± 5 %	B14 = Tin/lead, bulk BSL = Tin/lead, bulk, single lot date code R36 = Tin/lead, T/R (full) RE6 = Tin/lead, T/R (1000 pieces) RSL = Tin/lead, T/R, single lot date code	Blank = Standard (Dash number) 88 = Hot solder dip 143 = Non-magnetic

Historical Part Number example: RL07S471J (will continue to be accepted)

RL07	S	471	J	R36
MIL STYLE	LEAD MATERIAL	RESISTANCE VALUE	TOLERANCE CODE	PACKAGING

Note

- For additional information on packaging, refer to the Through Hole Resistor Packaging document (www.vishay.com/doc?31544).

MATERIAL SPECIFICATIONS

Element	Nickel-chrome alloy
Coating	Flame retardant epoxy, formulated for superior moisture protection
Core	Fire-cleaned high purity ceramic
Termination	Standard lead material is solder-coated copper. Solderable and weldable.

APPLICABLE MIL-SPECS

MIL-R-10509 and MIL-PRF-22684: The CMF models meet or exceed the electrical, environmental and dimensional requirements of MIL-R-10509 and MIL-PRF-22684.

Noise: Vishay Dale metal film resistors have exceptionally low noise level. Average for standard resistance range is 0.10 μV per V over a decade of frequency, with low and intermediate resistance values typically below 0.05 μV per V.

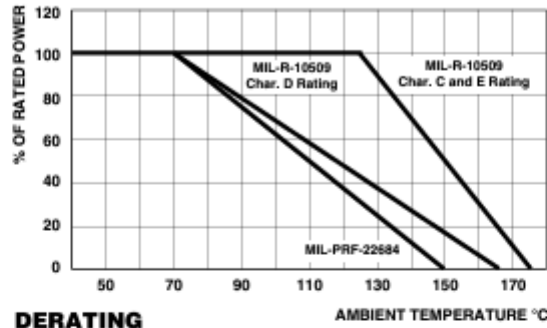
CAGE CODE: 91637

ENVIRONMENTAL SPECIFICATIONS

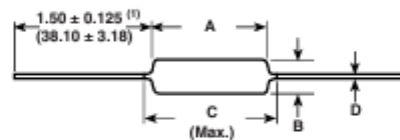
General	Environmental performance is shown in the Environmental Performance table. Test methods are those specified in MIL-R-10509 and MIL-PRF-22684.
Shelf Life	Resistance shifts due to storage at room temperature are negligible.



Vishay Dale CMF resistors have an operating temperature range of - 65 °C to + 175 °C. They must be derated according to the following curves:



DIMENSIONS in inches (millimeters)



VISHAY DALE MODEL	A	B	C (MAX.)	D
CMF50	0.150 ± 0.020 (3.81 ± 0.51)	0.065 ± 0.015 (1.65 ± 0.38)	0.244 (6.20)	0.016 ± 0.002 (0.41 ± 0.05)
CMF55	0.240 ± 0.020 (6.10 ± 0.51)	0.090 ± 0.008 (2.29 ± 0.20)	0.290 (7.37)	0.025 ± 0.002 (0.64 ± 0.05)
CMF60	0.344 ± 0.031 (8.74 ± 0.79)	0.145 ± 0.015 (3.68 ± 0.38)	0.425 (10.80)	0.025 ± 0.002 (0.64 ± 0.05)
CMF65	0.562 ± 0.031 (14.27 ± 0.79)	0.180 ± 0.015 (4.57 ± 0.38)	0.687 (17.45)	0.025 ± 0.002 (0.64 ± 0.05)
CMF70	0.562 ± 0.031 (14.27 ± 0.79)	0.180 ± 0.015 (4.57 ± 0.38)	0.687 (17.45)	0.032 ± 0.002 (0.81 ± 0.05)
CMF07	0.240 ± 0.020 (6.10 ± 0.51)	0.090 ± 0.008 (2.29 ± 0.20)	0.290 (7.37)	0.025 ± 0.002 (0.64 ± 0.05)
CMF20	0.375 ± 0.040 (9.53 ± 1.02)	0.145 ± 0.015 (3.68 ± 0.38)	0.425 (10.80)	0.032 ± 0.002 (0.81 ± 0.05)

Notes

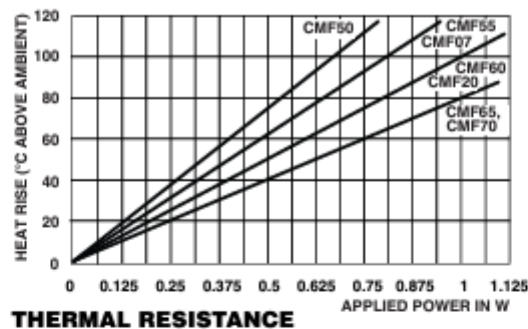
(1) Lead length for product in bulk pack. For product supplied in Tape and Reel, the actual lead length would be based on the body size, tape spacing and lead trim.

MILITARY POWER RATING			
WATTAGE	MILITARY QUALIFIED		
	MIL-R-10509		MIL-PRF-22684
	AT + 70 °C (D)	AT + 125 °C (C and E)	AT + 70 °C
0.05	-	RN50	-
0.10	-	RN55	-
0.125	RN55	RN60	-
0.25	RN60	RN65	RL07
0.50	RN65	RN70	RL20
0.75 (2)	RN70	-	-

Notes

• Commercial equivalents of military styles are available with higher power ratings. Consult factory.

(2) Formerly rated at 1 W and is the direct replacement for RN70 of MIL-R-10509 rev. D.


MARKING (per MIL-PRF-10509)

Characteristics: D = 100 ppm, C = 50 ppm, E = 25 ppm Tolerance: F = 1 %, D = 0.5 %, C = 0.25 %, B = 0.1 % Value = Three significant figures and multiplier J = JAN (Joint Army - Navy) brand			
RN50: (3 lines)		RN55, RN60, RN65, RN70 (4 lines)	
J50D	JAN, type, characteristic	DALE	Company logo
1211	Value	0137J	4 digit date code and JAN brand
F137	Tolerance and 3 digit date code	RN55D	Type and characteristic
		1211F	Value and Tolerance

Note

- RL series are color banded per MIL-PRF-22684.

PERFORMANCE

REQUIREMENT	MIL-R-10509			MIL-PRF-22684
	CHARACTERISTIC D	CHARACTERISTIC C	CHARACTERISTIC E	
MIL Temperature Coefficient	+ 200 ppm/°C - 500 ppm/°C	± 50 ppm/°C	± 25 ppm/°C	± 200 ppm/°C
Applicable Vishay Dale Temperature Coefficient	± 100 ppm/°C	± 50 ppm/°C	± 25 ppm/°C	± 200 ppm/°C
TEST	MIL_{max}	MIL_{max}	MIL_{max}	MIL_{max}
Thermal Shock	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 1.00 % ΔR
Short Time Overload	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 0.50 % ΔR
Low Temperature Operation	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 0.50 % ΔR
Moisture Resistance	± 1.50 % ΔR	± 0.50 % ΔR	± 0.50 % ΔR	± 1.50 % ΔR
Shock	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 0.50 % ΔR
Vibration	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 0.50 % ΔR
Load Life	± 1.00 % ΔR	± 0.50 % ΔR	± 0.50 % ΔR	± 2.00 % ΔR
Dielectric Withstanding Voltage	± 0.50 % ΔR	± 0.25 % ΔR	± 0.25 % ΔR	± 0.50 % ΔR
Effect of Solder	± 0.50 % ΔR	± 0.10 % ΔR	± 0.10 % ΔR	± 0.50 % ΔR