

Evaluation of 1D and 3D CFD Simulation Methods for Exhaust Backpressure

Master's thesis in Automotive Engineering

MUKUND RUDRAVAJHALA

MASTER'S THESIS 2018:30

Evaluation of 1D and 3D CFD Simulation Methods for Exhaust Backpressure

MUKUND RUDRAVAJHALA



Department of Mechanics and Maritime Sciences
Division of Combustion and Propulsion Systems
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2018

Evaluation of 1D and 3D CFD Simulation Methods for Exhaust Backpressure
MUKUND RUDRAVAJHALA

© MUKUND RUDRAVAJHALA, 2018.

Supervisors:

Anders Lind (97491) and Maria Dybeck (97684), Volvo Car Corporation

Examiner:

Sven B Andersson, Department of Mechanics and Maritime Sciences

Master's Thesis 2018:30

Department of Mechanics and Maritime Sciences

Division of Combustion and Propulsion Systems

Chalmers University of Technology

SE-412 96 Gothenburg

Telephone +46 31 772 1000

Cover: Flow visualization inside a rear muffler of an exhaust system showing the velocity profile of the gases in 3D space.

Typeset in L^AT_EX

Printed by Department of Mechanics and Maritime Sciences

Gothenburg, Sweden 2018

Abstract

An internal combustion engine generates exhaust gases that need to be transported away from the engine and also the passenger cabin. This is achieved by using an exhaust system that directs the flow of gases. Apart from directing the flow of gases, the exhaust system also has the function of reducing the sound that is generated due to the pressure pulsations that occur during the combustion of gases in the engine. The flow of gases through the exhaust system leads to some back pressure due to phenomenon such as friction, contraction and expansion of exhaust gases and the turbulence of gases.

Back pressure affects the engine performance as the gases in the exhaust system offer resistance to the gases that are trying to escape from the engine. Due to this resistance, the volumetric efficiency of the engine decreases, causing increased fuel consumption and loss of performance. Hence, it is very important to consider the back pressure of an exhaust system from the early stages of design. In the industry, it is not viable to make prototypes every time a small change is carried out to the design. Simulations are very useful for checking concepts and ensuring that the back pressure targets are met.

The aim of the master thesis is to evaluate 1D and 3D CFD simulation methods for exhaust back pressure that are currently in use at Volvo Cars. This evaluation was carried out by comparing the results from the simulations to those obtained from physical tests. Once the methods were evaluated, they were then modified to replicate the tests better.

Simulations with the modifications were then run, and the differences in results were observed. The modifications made an improvement for two out of the three geometries. It was found out that modelling small geometrical features inside the exhaust systems and calibrating the models with the right heat transfer coefficients contribute greatly to better the correlations between the results from the simulations and experimental tests.

The work in this thesis can be used as a reference for future modelling, as it calls for improvements to the methods that are currently in place at Volvo Cars.

Keywords: CFD, 1D, 3D, GT-Suite, StarCCM+, Exhaust Back pressure, Pressure Drop

Preface

This master thesis has been completed in partial fulfillment of the requirements for a Master's degree in Automotive Engineering. It is a project carried out in collaboration between the Engine - CAE Fluids Group (97684) and Exhaust System Group (97491) at Volvo Car Corporation, Gothenburg, Sweden. The primary work has been performed from January 2018 to June 2018 at Volvo Car Corporation.

Acknowledgements

I would like to express my sincere gratitude to everyone who has helped me through the course of this master thesis.

Firstly, I would like to thank Anders Lind and Maria Dybeck, my supervisors at Volvo Car Corporation, for being patient with me and providing guidance throughout the thesis. The thesis would not have been completed without their inputs and support.

I would also like to thank Anders Lindersson, my manager at Volvo Car Corporation, for ensuring that I had all the necessary resources on time, and also making sure that the working environment was always comfortable.

Furthermore, I would like to thank Christer Berg and Bo Augustsson at Volvo Car Corporation for being very accommodating in the test rigs, despite the tight schedules.

A special thanks goes to Sven B Andersson, my examiner at Chalmers, for his active interest throughout the course of the thesis.

Mukund Rudravajhala

Gothenburg, June 2018

Contents

Abstract	v
Preface	vii
Acknowledgements	vii
Contents	x
Nomenclature	xi
Abbreviations	xi
List of Figures	xi
List of Tables	xiii
1 Introduction	1
1.1 Volvo Car Corporation	1
1.2 Background	1
1.3 Aim and purpose	2
1.4 Thesis Scope	2
1.5 Delimitations	2
1.6 Specification of issues under interest	2
2 Theory and Background	3
2.1 Exhaust System	3
2.1.1 Exhaust System Cold End	3
2.2 Computational Fluid Dynamics (CFD)	5
2.2.1 What is CFD?	5
2.2.2 1D CFD Simulations	6
2.2.3 3D CFD Simulations	6
2.2.4 Equations Governing Fluid Dynamics	6
2.2.5 Reynolds Number Correlations	8
2.3 Computer Modelling	10
2.3.1 GEM3D	10
2.3.2 GT-ISE	10
2.3.3 ANSA	11
2.3.4 STAR-CCM+	11

2.4	Reference Values	12
3	Methods	13
3.1	Flow Bench Tests	13
3.1.1	Cold Flow Rig Tests	13
3.1.2	Hot Flow Rig Tests	14
3.2	1D Simulations	16
3.2.1	GEM3D	16
3.2.2	GT-ISE	19
3.2.3	GT-Post	22
3.3	3D Simulations	23
3.3.1	ANSA	23
3.3.2	STAR-CCM+	24
4	Results and Discussion	29
4.1	Overview of Simulation Settings	29
4.1.1	Base settings	29
4.1.2	Optimized settings	29
4.2	System 1	31
4.2.1	1D Simulations	32
4.2.2	3D Simulations	33
4.2.3	Summary	35
4.3	System 2	37
4.3.1	1D Simulations	37
4.3.2	3D Simulations	39
4.3.3	Summary	43
4.4	System 3	46
4.4.1	1D Simulations	46
4.4.2	3D Simulations	47
4.4.3	Summary	50
4.5	Hot Flow Test - Model Calibration	52
5	Conclusion	53
6	Future Work	54
	Bibliography	55
A	Appendix	I
A.1	Plots for System 1	I
A.2	Plots for System 2	II
A.3	Plots for System 3	V

Nomenclature

Parameter	Symbol	Unit
Area	A	m^2
Mass Flow	m_f	$kg.h^{-1}$
Static Pressure	p	Pa or kPa
Pressure	P	Pa
Density	ρ	$kg.m^{-3}$
Gas Constant	R_d	$J.kg^{-1}.K^{-1}$
Temperature	T	K or $^{\circ}C$
Volumetric Flow Rate	Q	$m^3.s^{-1}$
Mass Flow Rate	\dot{m}	$kg.s^{-1}$
Diameter	D	m
Velocity	u	$m.s^{-1}$
Dynamic Pressure	q	Pa
Dynamic Viscosity	μ	$Pa.s$
Kinematic Viscosity	ν	$m^2.s^{-1}$
Reynolds Number	Re	-
Static Pressure Drop Coefficient	C_p	-
Internal Energy	E	$J.kg^{-1}$
Heat Flux	ϕ_q	$W.m^{-2}$
Thermal Conductivity	κ	$W.m^{-1}.K^{-1}$
Time	t	s

Abbreviations

1D	One Dimensional
3D	Three Dimensional
ANSA	Pre-processing tool for 3D Simulations
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CATIA v5	CAD Modelling software, version 5
CFD	Computational Fluid Dynamics
GEM3D	Pre-processing tool for 1D Simulations
GT-ISE	Gamma Technologies Integrated Simulation Environment
GT-Post	Post-processing tool for 1D Simulations
NVH	Noise, Vibration and Harshness
ODE	Ordinary Differential Equation
PDE	Partial Differential Equation
PID	Property Identification
RANS	Reynolds-Averaged Navier-Stokes
STAR-CCM+	3D Simulation Environment
SUV	Sports Utility Vehicle

List of Figures

2.1	Typical Cold End	4
2.2	Baffles in a rear muffler	4
2.3	Baffle Leaks	5
2.4	Equation for frequency in a Helmholtz Resonator [2]	5
2.5	Pressure Drop Coefficient vs. Reynolds Number [5]	10
3.1	Experimental Setup of Cold Flow Rig	14
3.2	Schematic of the Hot Flow Rig	15
3.3	Experimental Setup of the Hot Flow Rig	15
3.4	3D Representation of a Front Muffler in GEM3D	18
3.5	Front Muffler in after conversion into a .gtsub file	19
3.6	1D Model in GT-ISE	20
3.7	1D Model of the Hot Flow Bench in GT-ISE	22
3.8	Cleaned Geometry ready for surface meshing	24
3.9	Cross sectional view of Rear Muffler after meshing in STAR-CCM+	27
3.10	3D Visualization of Flow velocity through a rear muffler	28
4.1	Isometric view of System 1	31
4.2	Rear Muffler Internals of System 1 System	31
4.3	Pressure Drop of System 1 at 20°C	32
4.4	Pressure Drop of System 1 at 775°C	32
4.5	Rear Muffler of System 1 at 20°C, Base Settings	33
4.6	Rear Muffler of System 1 at 20°C, Optimized Settings	33
4.7	Rear Muffler of System 1 at 775°C, Base Settings	34
4.8	Rear Muffler of System 1 at 775°C, Optimized Settings	34
4.9	Isometric view of System 2	37
4.10	Rear Muffler Internals of System 2	37
4.11	Pressure Drop of System 2, Closed Valve	38
4.12	Pressure Drop of System 2, Open Valve	38
4.13	Rear Muffler of System 2 at 20°C, valve closed, Base Settings	39
4.14	Rear Muffler of System 2 at 20°C, valve closed, Optimized Settings	39
4.15	Rear Muffler of System 2 at 775°C, valve closed, Base Settings	40
4.16	Rear Muffler of System 2 at 775°C, valve closed, Optimized Settings	40
4.17	Rear Muffler of System 2 at 20°C, valve open, Base Settings	41
4.18	Rear Muffler of System 2 at 20°C, valve open, Optimized Settings	41
4.19	Rear Muffler of System 2 at 775°C, valve open, Base Settings	42
4.20	Rear Muffler of System 2 at 775°C, valve open, Optimized Settings	42

4.21	Isometric view of System 3	46
4.22	Rear Muffler Internals of System 3	46
4.23	Pressure Drop of System 3 at 20°C	47
4.24	Pressure Drop of System 3 at 775°C	47
4.25	Rear Muffler of System 3 at 20°C, Base Settings	48
4.26	Rear Muffler of System 3 at 20°C, Optimized Settings	48
4.27	Rear Muffler of System 3 at 775°C, Base Settings	49
4.28	Rear Muffler of System 3 at 775°C, Optimized Settings	49
4.29	Pressure Drop of System 2 at an average of 775°C, Valve Open	52
A.1	Residuals for System 1 at 20°C, Base Settings	I
A.2	Residuals for System 1 at 775°C, Base Settings	I
A.3	Residuals for System 1 at 20°C, Optimized Settings	II
A.4	Residuals for System 1 at 775°C, Optimized Settings	II
A.5	Residuals for System 2 at 20°C Valve Open, Base Settings	II
A.6	Residuals for System 2 at 775°C Valve Open, Base Settings	III
A.7	Residuals for System 2 at 20°C Valve Closed, Base Settings	III
A.8	Residuals for System 2 at 775°C Valve Closed, Base Settings	III
A.9	Residuals for System 2 at 20°C Valve Open, optimized Settings . . .	IV
A.10	Residuals for System 2 at 775°C Valve Open, Optimized Settings . .	IV
A.11	Residuals for System 2 at 20°C Valve Closed, Optimized Settings . .	IV
A.12	Residuals for System 2 at 775°C Valve Closed, Optimized Settings . .	V
A.13	Residuals for System 3 at 20°C, Base Settings	V
A.14	Residuals for System 3 at 775°C, Base Settings	V
A.15	Residuals for System 3 at 20°C, Optimized Settings	VI
A.16	Residuals for System 3 at 775°C, Optimized Settings	VI

List of Tables

3.1	Templates used for 1D Simulations	20
3.2	Physics models used	26
4.1	Results for System 1 at 720kg/h, 20°C and base settings	35
4.2	Results for System 1 at 720kg/h, 20°C and optimized settings	35
4.3	Results for System 1 at 720kg/h, 775°C and base settings	35
4.4	Results for System 1 at 720kg/h, 775°C and optimized settings	36
4.5	Base results for System 2 at 720kg/h and 20°C, Open Valve	43
4.6	Optimized results for System 2 at 720kg/h and 20°C, Open Valve	43
4.7	Base results for System 2 at 720kg/h and 20°C, Closed Valve	43
4.8	Optimized results for System 2 at 720kg/h and 20°C, Closed Valve	44
4.9	Base results for System 2 at 720kg/h and 775°C, Open Valve	44
4.10	Optimized results for System 2 at 720kg/h and 775°C, Open Valve	44
4.11	Base results for System 2 at 720kg/h and 775°C, Closed Valve	44
4.12	Optimized results for System 2 at 720kg/h and 775°C, Closed Valve	45
4.13	Base results for System 3 at 720kg/h and 20°C	50
4.14	Optimized results for System 3 at 720kg/h and 20°C	50
4.15	Base results for System 3 at 720kg/h and 775°C	50
4.16	Optimized results for System 3 at 720kg/h and 775°C	51
4.17	Temperature measurements for hot flow bench	52

1

Introduction

The master thesis has been carried out at Volvo Car Corporation, at Torslanda, Gothenburg, Sweden, to evaluate the CFD methods currently in use to estimate the back-pressure in the cold end of the exhaust system. The simulation results were compared to those obtained from physical tests. The thesis work was carried out jointly between the Exhaust Systems Group (97491) and the Engine CAE - Fluids Group (97684) at Volvo Car Corporation, Gothenburg.

1.1 Volvo Car Corporation

Volvo Car Corporation is a Swedish vehicle manufacturer established in 1927. The company has its head quarters in Gothenburg, Sweden. Volvo Cars is a global company, with around 38000 employees worldwide. The company has production facilities in Sweden, China, Malaysia, Belgium and the United States of America. Volvo cars is a premium car brand with care for the planet and the customer and aims to deliver vehicles with focus on safety, sustainability, comfort. The range of vehicles includes wagons, SUVs and sedans. In 2017, Volvo Cars sold 571,577 vehicles, and had an operating income of 14.061 billion Swedish Krona.

1.2 Background

Exhaust back pressure is an important aspect that needs to be considered while designing a powertrain, as it can influence its performance significantly. Back pressure can cause difficulty for burnt gases to exit the cylinder after combustion. This causes reduced volumetric efficiency as the volume of combustible gases that are in the cylinder decreases. This in turn, leads to increased fuel consumption as more work needs to be done in order to get the same power output.

Apart from influencing the performance of the power train, it also affects the NVH characteristics of the vehicles, and this is a major factor to consider while designing premium cars. Back pressure can be measured physically in a test rig, but this is expensive and time consuming as new prototypes need to be built each time a parameter is changed. An alternative option is performing simulations to predict flow behaviour with a reasonable degree of accuracy.

Simulations can be run using either 3D CFD or 1D CFD approaches. 3D CFD simulations are more accurate, but require a lot of computational resources and time. 1D CFD simulations on the other hand are less accurate, but use far lesser computational resources and time.

1.3 Aim and purpose

The aim of the master thesis is to evaluate 1D and 3D CFD methods and investigate the accuracy in different parts of the exhaust system. The accuracy is investigated by comparing the values obtained from simulations to those obtained by physical tests. Once the accuracy of the simulations is evaluated, the current simulation methods are then optimized by changing certain geometrical and simulation parameters.

1.4 Thesis Scope

The thesis work has the following steps:

- Plan the physical tests and prepare the three different exhaust systems
- Obtain and interpret data from the physical rigs once the tests are performed
- Setup and run 1D and 3D simulations for the same exhaust systems, using existing methods to get a baseline
- Compare the results from the baseline simulations to the physical tests
- Investigate the simulation methods to identify differences in results
- Perform study by making changes to the simulations to reflect the differences
- Compile results from the study for future reference

1.5 Delimitations

The thesis has been carried out under the following delimitations:

- The study is limited to the cold end of the exhaust system
- Physical test data is obtained from a sample size of 1
- Cold flow tests are not representative of the actual application of the exhaust system
- Availability of test rigs at Volvo Cars
- All the details in CAD cannot be replicated in simulations
- All simulations are limited to steady state only
- 1D simulations tend to oversimplify some features
- 3D simulations take a long time to set up and run
- Simulations are very geometry specific, and optimization carried out on one system may not be applicable to other systems
- The thesis work is carried out by one student in a time frame of 20 weeks

1.6 Specification of issues under interest

- How are the physical tests performed?
- How accurate are the existing models?
- How are the models set up?
- What parameters influence the simulation results?
- How close can the simulations get to the test results?

2

Theory and Background

This chapter introduces the reader to exhaust systems, and gives a brief overview of its various components. Apart from that, the chapter also describes what CFD is and the theory behind the models that are used for the simulations. A brief introduction to the software that is used for the thesis is also included in this section.

2.1 Exhaust System

An exhaust system consists of all the parts that are used to guide the flow of gases after the combustion process in a cylinder of an internal combustion engine. A typical exhaust system consists of:

- Exhaust Manifold
- Down Pipe
- Turbocharger
- Catalytic Converter
- Cold End

2.1.1 Exhaust System Cold End

The thesis work is limited to evaluating the methods used for the Cold End of the exhaust system. The cold end includes all parts of the exhaust system after the catalytic converter. This includes:

- Front Muffler (if present)
- Connection Pipe
- Rear Muffler

Mufflers are mainly used to attenuate the noise originating from:

- Flow of gases
- Pulsations arising from combustion in the cylinders

The side effect of having mufflers is an increased back pressure, which is not desirable as it reduces the performance of the engine. The top view of a typical cold end is shown in Figure 2.1

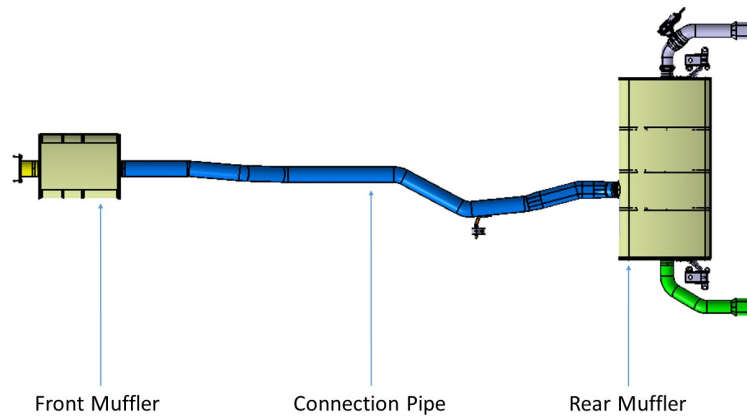


Figure 2.1: Typical Cold End

2.1.1.1 Baffles

Baffles are vertical walls that are inside the mufflers. These are either perforated, to help in reducing flow noise, or they may be solid to separate certain air volumes in the mufflers. Another reason for having perforations on baffles is to reduce the restriction to flow, as this leads to increased back pressure in exhaust concepts where the gases need to flow through multiple chambers in order to exit the exhaust system. The baffles inside of a rear muffler is shown in Figure 2.2

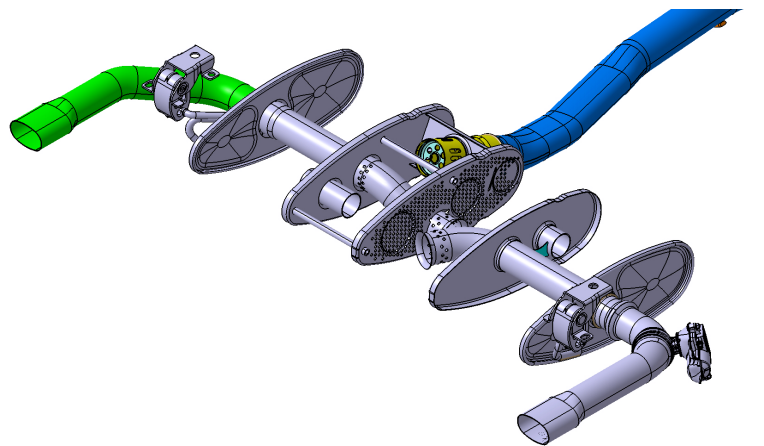


Figure 2.2: Baffles in a rear muffler

Usually, these baffles have some leakage between the various parts of the rear muffler. These leaks are made on purpose to help condensed water or other liquids to flow between the chambers, and eventually exit the exhaust system through the tailpipes. A side section view of rear muffler, highlighting the baffle leaks is shown in Figure 2.3

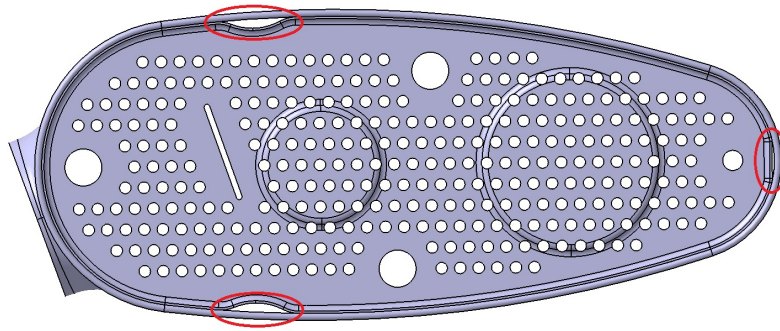


Figure 2.3: Baffle Leaks

2.1.1.2 Helmholtz Resonance Chambers

Helmholtz resonance or wind throb is the phenomenon of air resonance in a cavity, such as when one blows across the top of an empty bottle. The theory of Helmholtz resonators is used in exhaust systems to alter the sound of the exhaust note and for differences in power delivery by adding chambers to the exhaust. Exhaust resonators are also used to reduce potentially loud and obnoxious engine noise where the dimensions are calculated so that the waves reflected by the resonator help cancel out certain frequencies of sound in the exhaust [1].

Figure 2.4 shows the equation for calculating the resonance frequency of the sound that is created by the Helmholtz chamber.

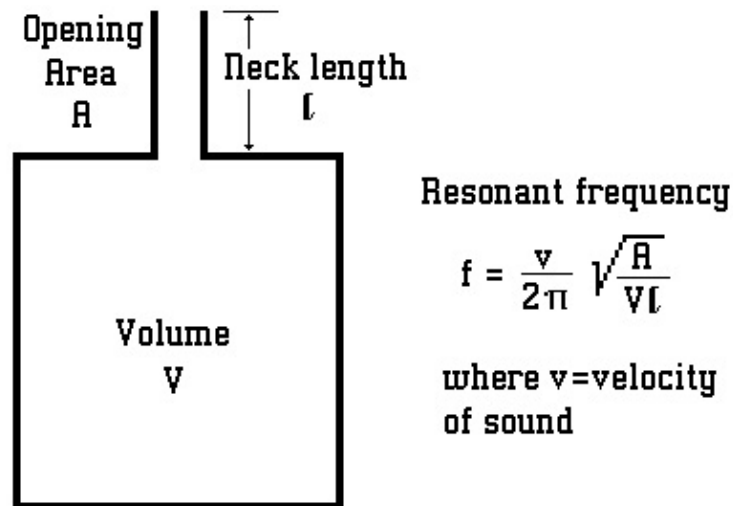


Figure 2.4: Equation for frequency in a Helmholtz Resonator [2]

2.2 Computational Fluid Dynamics (CFD)

2.2.1 What is CFD?

CFD is a branch of fluid mechanics that solves problems involving fluid flow using numerical analysis. The common types of CFD problems include simulating fluid

turbulence, chemical reactions, heat transfer and fluid flow. The calculations are performed using computers, by specifying certain boundary conditions.

CFD is especially useful in the automotive industry for applications such as:

- Aerodynamic analysis
- Combustion simulations
- Cooling simulations
- Air conditioning simulations

The advantages to using CFD methods are:

- Low costs - no prototypes or test facilities necessary
- Lower lead times than conventional testing
- Better visualization of flow in areas that are hard to see otherwise
- Multiple configurations and various setups can be tested and compared directly

The main disadvantage of CFD is that certain physical phenomenon cannot be modelled mathematically, and these may bring uncertainties to the results obtained from simulations.

2.2.2 1D CFD Simulations

In 1D CFD Simulations, the system dynamics are calculated as functions of time and one dimension. This is achieved by using ordinary differential equations or differential algebraic equations. Another important feature is that a 3D CAD model of the system that is being simulated is not always necessary. It is sufficient to have a model in which the connectivity between various components is specified correctly. Hence, knowing what sort of components need to be used in the simulation to represent the actual system, is extremely important to get reliable results from the simulations.

2.2.3 3D CFD Simulations

In 3D CFD Simulations, the system dynamics are calculated as functions of time and space in three dimensions. This is achieved by using partial differential equations. For 3D simulations to be accurate, good 3D CAD models and material properties of the various components of the system that is being simulated is necessary. It is not sufficient to have a model in which just the connectivity between various components is specified correctly. Knowing what sort of mesh grid that needs to be used in the simulation system is extremely important to get reliable results from the simulations.

2.2.4 Equations Governing Fluid Dynamics

Fluid dynamics is governed by three principles:

- Conservation of Mass
- Conservation of Momentum
- Conservation of Energy

These principles are expressed as mathematical equations and then solved numerically for a certain geometry and a certain set of boundary conditions that go along with the geometry.

2.2.4.1 Conservation of Mass

This law states that mass can neither be created nor destroyed for a closed system. It is also known as the continuity equation. Matter can change states but the overall mass before and after the change will remain constant. When it comes to fluid dynamics, the mass flow through elements in a grid is more commonly described. Thus, the continuity equation states that the net inflow of mass into a fluid element is equal to the rate of change of mass in the same element. Equation 2.1 describes the continuity equation in Cartesian form, for one dimension. [3]

$$\frac{\partial \rho}{\partial t} = -\frac{\partial \rho u}{\partial x} \quad (2.1)$$

Where:

ρ = Density of gas

u = Velocity of gas

2.2.4.2 Conservation of Momentum

This law states that for a closed system, momentum of a particle can only be changed by applying an external force, as described by Newton's Second Law. It is also known as the Navier-Stokes equation. When it comes to fluid dynamics, the Navier-Stokes equation states that the rate of increase of momentum of a fluid element is equal to the sum of the net influx of momentum and surface forces acting on the same element. Equation 2.2 describes the Navier-Stokes equation in Cartesian form, for one dimension. [3]

$$\frac{\partial u \rho}{\partial t} = -\frac{\partial u^2 \rho}{\partial x} - \frac{\partial P}{\partial x} \quad (2.2)$$

Where:

P = Pressure

The pressure here is the sum of the thermodynamic and viscous pressure. Viscous pressure is given by: $\mu(\partial u / \partial x)$, where μ is Dynamic Viscosity.

2.2.4.3 Conservation of Energy

This law states that in a closed system, energy can neither be created nor destroyed, but only transferred from one form to another. It is also known as the First Law of Thermodynamics. When it comes to fluid dynamics, the law states that the rate of change of energy of a fluid element is equal to the sum of heat flux into the element and the work done by external forces on the element. Equation 2.3 describes the Conservation of Energy equation in Cartesian form, for one dimension. [3]

$$\frac{\partial E \rho}{\partial t} = -\frac{\partial u \rho E}{\partial x} - \frac{\partial P u}{\partial x} - \frac{\partial \phi_q}{\partial x} \quad (2.3)$$

Where:

E = Internal Energy

ϕ_q = Heat Flux = $-\kappa \frac{\partial T}{\partial x}$, where κ is the Thermal Conductivity

2.2.5 Reynolds Number Correlations

Reynolds number correlations are used to recalculate the pressure drop over a component at various temperatures and diameters. In this thesis, these correlations are used to recalculate measured pressure drops at 20°C to represent pressure drops at 775°C.

For the scope of the thesis, the gas flowing through the exhaust system is assumed to be an Ideal Gas. Hence, by using the Ideal Gas Law, we get:

$$p = \rho R_d T \quad (2.4)$$

Where:

p = Static Pressure [Pa]

R_d = Gas Constant for air [$287.09 \text{ J.kg}^{-1}.\text{K}^{-1}$]

T = Temperature of gas [K]

Since the static pressure and temperature of gas is know, Equation 2.4 is used to calculate the density of the gas.

The next step is to calculate the Volumetric Flow Rate of gases through the exhaust system. The mass flow is a known parameter from the experimental setup. Equation 2.5 shows the relation to calculate the Volumetric Flow Rate.

$$Q = \frac{\dot{m}}{\rho} \quad (2.5)$$

Where:

Q = Volumetric Flow Rate [$m^3.s^{-1}$]

\dot{m} = Mass Flow rate [$kg.s^{-1}$]

Since the diameter of the various tubes is known, the cross sectional area can be calculated.

$$A = \frac{\pi D^2}{4} \quad (2.6)$$

Where:

A = Cross Sectional area [m^2]

D = Diameter of the tube [m]

The next step is to calculate the velocity of gases through the tube. Since the Volumetric Flow Rate and area is known, the velocity is calculated as:

$$u = \frac{Q}{A} \quad (2.7)$$

Where:

u = Velocity of gas [$m.s^{-1}$]

Once the velocity is known, the dynamic pressure can be calculated by using Equation 2.8

$$q = \frac{\rho u^2}{2} \quad (2.8)$$

Where:

q = Dynamic Pressure [Pa]

The next step is to calculate the dynamic viscosity of air at 775°C or 1048.15K and atmospheric pressure. For the sake of simplicity, this value is taken from a look-up table for properties of air at various temperatures [4]. The value is obtained is:

$$\mu = 43.8 \times 10^{-6} \text{ Pa.s}$$

Where:

μ = Dynamic Viscosity [$Pa.s$]

The kinematic viscosity of the gas is calculated next. This is done using Equation 2.9

$$\nu = \frac{\mu}{\rho} \quad (2.9)$$

Where:

ν = Kinematic Viscosity [$m^2.s^{-1}$]

The Reynolds number is calculated next. The relation for this is given in Equation 2.10

$$Re = \frac{uD}{\nu} \quad (2.10)$$

Where:

Re = Reynolds Number [Dimensionless]

The static pressure drop coefficient is then calculated at 20°C. This is done by using Equation 2.11.

$$C_p = \frac{P}{q} \quad (2.11)$$

The Reynolds number is then plotted vs the static pressure drop coefficient at 20°C. Once this is done, a 4th order polynomial curve is fitted through all these points on the plot. The Reynolds number is then calculated at 775°C, and then projected onto the same curve. The static pressure drop coefficient at 775°C is then noted by projecting this point onto the Y-axis of the plot. Figure 2.5 shows the plot from which the pressure drop coefficients are obtained.

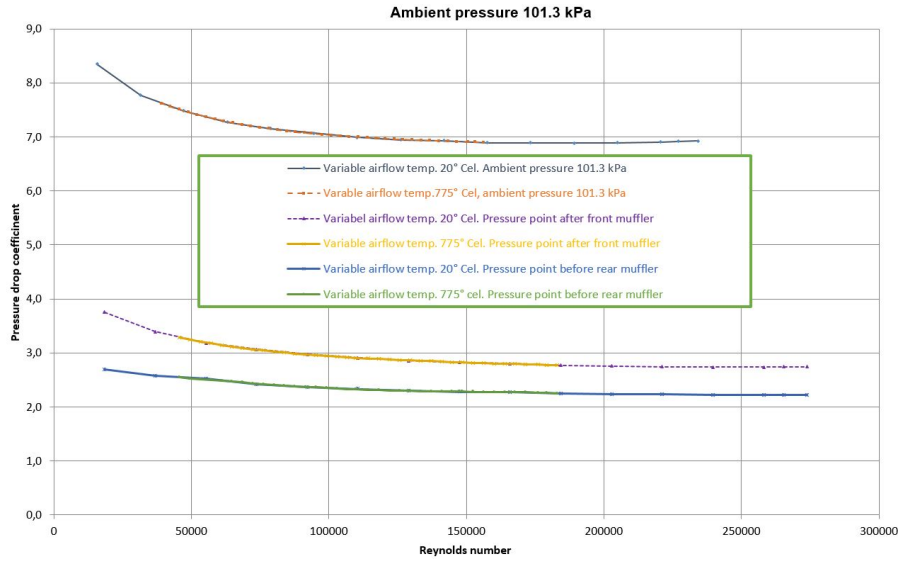


Figure 2.5: Pressure Drop Coefficient vs. Reynolds Number [5]

Once this is obtained, the static pressure at 775°C can then be calculated by using Equation 2.12.

$$P_{775} = q_{775} C_{p775} \quad (2.12)$$

Where:

P_{775} = Static Pressure drop at 775°C [Pa]

q_{775} = Dynamic Pressure at 775°C [Pa]

C_{p775} = Static Pressure drop coefficient at 775°C [Dimensionless]

2.3 Computer Modelling

2.3.1 GEM3D

GEM3D is a 3D graphical pre-processor that combines building and importing tools used to create 1D GT-Suite models from 3D geometries. It can be used for engine performance applications to build and characterize core components like manifolds, plenums, runners, and more. GEM3D can also be used for acoustic applications to build and characterize complete intake and exhaust systems including air boxes, resonators, plenums, mufflers, silencers, and more. Some of the applicable features include baffles (any orientation), perforates (on pipes and baffles), absorbing material (wool), sleeves (empty or filled with wool), and more. [6]

2.3.2 GT-ISE

GT-ISE is the main interface where models are built, simulation settings are declared, and simulations (both single and batch) are launched. It is an environment where various components are brought into a project map and connected together linking the entire model as a whole. [7]

For the purpose of this thesis, all simulations are carried out using the Explicit methods for time step calculations. The variables such as mass flow rate, density

and internal energy are calculated based on the governing laws of fluid dynamics. These variables are calculated by using values from the previous time steps. [8]

2.3.3 ANSA

ANSA is an advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment. [9]

2.3.4 STAR-CCM+

STAR-CCM+ is a CAE solution for solving multidisciplinary problems in both fluid and solid continuum mechanics, within a single integrated user interface. The STAR-CCM+ simulation environment offers all stages required for carrying out engineering analyses, including:

- Import and creation of geometries
- Mesh generation
- Solution of the governing equations
- Analysis of the results
- Automation of the simulation work-flows for design exploration studies
- Connection to other CAE software for co-simulation analysis [10]

2.3.4.1 Flow and Energy Modelling

The models that can be used in STAR-CCM+ for simulating Flow and Modelling are:

- **Segregated Model:**

In this model, the flow equations are solved separately to get the pressure and velocity. This type of modelling is computationally less intensive. However, the number of iterations needed to reach convergence is dependent on the mesh size [10].

- **Coupled Model:**

In this model, the flow equations are solved together, or in a coupled fashion to get the pressure and velocity. This type of modelling is computationally demanding, and requires a lot more memory than the Segregated model. However, the number of iterations needed to reach convergence is independent on the mesh size. It usually requires a much greater number of iterations to converge, especially when compared to the Segregated model [10].

2.3.4.2 Turbulence Modelling

The most common models used for turbulence modelling in STAR-CCM+ are:

- **$k - \epsilon$ Model:**

This is the most commonly used model in CFD, to simulate flow characteristics for turbulent flows. The turbulence is described by two PDEs. The variables used by the PDEs are:

- Turbulence Kinetic Energy (k)
 - Rate of Dissipation of Turbulence Energy (ϵ)
- **$k - \omega$ Model:**

This is a commonly used model in CFD, which also uses two PDEs to describe the turbulence. The variables used by the PDEs are:

 - Turbulence Kinetic Energy (k)
 - Specific Rate of Dissipation of Turbulence Kinetic Energy (ϵ)

Here, the kinetic energy is dissipated into internal thermal energy.

2.4 Reference Values

An engine is like an air pump, and is only capable of delivering a certain maximum mass flow of gases in its operating range. Based on these mass flows, the Engine Department at Volvo Cars sets a specific back pressure target for each engine. However, due to the variation in mass flows, the comparison between various exhaust systems gets difficult, and hence there are some reference values used by the Exhaust System Group to have comparability between various exhaust system concepts. There are also temperature variations due to the different mass flows, and this adds to the difficulty of comparing various concepts. The reference temperature is set based on data that is obtained from testing, and is the average temperature of the gases flowing through the exhaust system. The values that are used are:

- Reference Mass Flow Rate: **720 kg/h**
- Reference Gas Temperature: **775°C**

3

Methods

Based on the tasks that needed to be done, the thesis was divided into three distinct parts:

- Flow bench tests
- 1D Simulations
- 3D Simulations

The flow tests were performed first. The results obtained from the flow bench tests were considered as the benchmark for any comparisons. Simulations were then performed using both 1D and 3D CFD methods, and then the differences between the tests and simulations were observed. The optimization process was then started by varying certain geometrical and simulation parameters in the simulations, and observing the results. In the 1D simulations, calibration was performed based on the experimental data. The settings used for models were considered to be optimal once gave the closest results in comparison to the bench results.

3.1 Flow Bench Tests

There were two types of flow bench tests performed during the course of the thesis:

- Cold Flow Rig Tests
- Hot Flow Rig Tests

The basis for these tests was to have various mass flow rates at a given temperature, by performing a sweep of the flow rates. Once the flow reached a steady state, the static pressure measurements at various points were taken so that the pressure drop can be observed. More details of each rig and the test setup are shown in sections 3.1.1 and 3.1.2.

3.1.1 Cold Flow Rig Tests

Physical measurements on the different exhaust systems were carried out in the Cold Flow rig at Volvo Cars, Torslanda. The test setup included measurement points for measuring pressure at various points along the exhaust system. The main areas of interest for taking measurements include:

- Before the front muffler
- After the front muffler
- Before the rear muffler
- Points in the rear muffler
- Ambient pressure

The static pressure measurements were taken at different points so that the pressure drop over individual components can be calculated by subtracting the pressure values

obtained at various points. Also, the overall pressure drop can be calculated by subtracting the ambient pressure from the pressure before the front muffler. Each pressure measurement was an average of four individual pressure measurements taken at each point of measurement. This was done as all the cross sections are not perfectly circular, and taking an average gives a better estimation of the pressure drop across the cross section. The experimental setup is shown in Figure 3.1

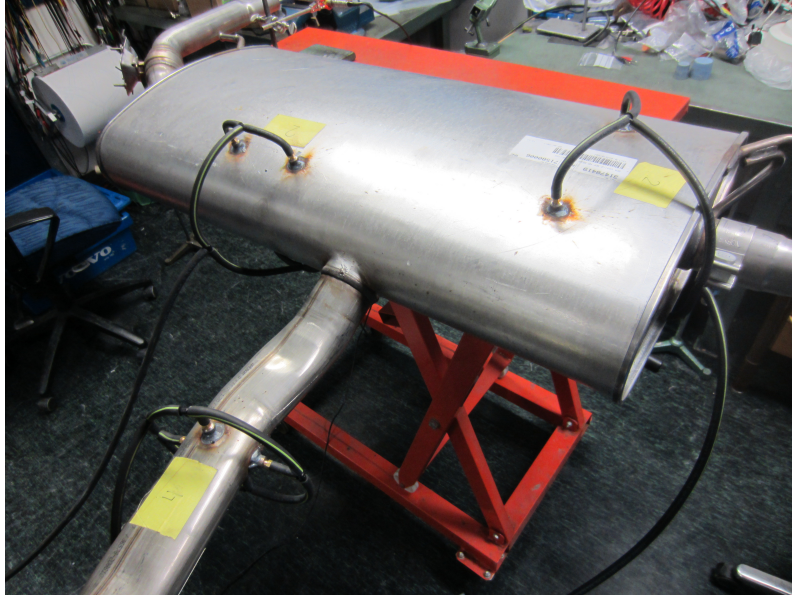


Figure 3.1: Experimental Setup of Cold Flow Rig

All tests were performed at 20°C, for various mass flow rates. The mass flow rate was increased in steps of 100 kg/h, until the maximum mass flow rate possible for each exhaust system was reached. The maximum flow rate that was possible for each system was dependent on the back pressure of the exhaust system. Measurements were taken when the flow attained stability at each step increment. The obtained static pressure measurement values were then converted to the pressures that are representative of pressures one could expect at 775°C, by using Reynolds number correlations, as shown in Section 2.2.5.

3.1.2 Hot Flow Rig Tests

Additional physical tests were carried out at a later stage in the thesis. These tests were performed at the STC Test Rig at Volvo Cars, Torslanda. The test rig generates hot gases at various flow rates by utilizing a burner. Due to the space constraints in the test room, the exhaust system cannot fit along the direction of the burner. Hence, additional piping is necessary to create a 180° bend, so it can be accommodated in the room. A schematic of the test rig is shown in Figure 3.2. In the cold flow rig, various pressure measurements were taken. The pressure drop over the various components was then calculated by subtracting the pressures from each other. However, in the hot flow rig, the pressure was measured only at the outlet of the burner. This was due to the unavailability of static pressure measurement equipment that could withstand the high exhaust gas temperatures. Also, the

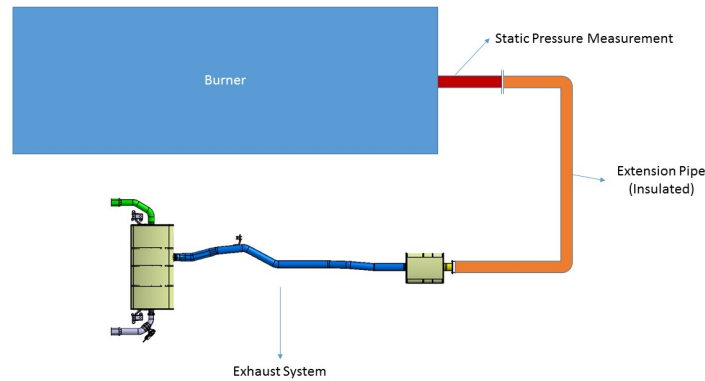


Figure 3.2: Schematic of the Hot Flow Rig

existing pressure measurement equipment in the test rig was directly attached to the outlet of the burner making it very hard to move or re-position.

Apart from this, temperature of the exhaust gas was measured at various points along the exhaust system using N-Type thermocouples. These temperature measurements were then used to calibrate the 1D models.

The extension pipe was insulated with thermal tape to avoid high temperature losses, so that the temperature of the exhaust gas is at the correct value at the inlet of the exhaust system. It was important to get the correct temperatures of the exhaust gases at the inlet into the exhaust system as they are used to calibrate the models for simulations. The temperature measurements and part of the experimental setup is shown in Figure 3.3



Figure 3.3: Experimental Setup of the Hot Flow Rig

The tailpipes of the exhaust system needed to be connected to the ventilation system to avoid contaminating the room with burnt gases. For the ventilation system to work correctly, there was a suction at the outlets of the exhaust system. This suction was an unknown under-pressure between 200-400 Pa. This is another variable that

was taken into account when modelling this test in the simulations. Since the suction pressure was not very high, for the purpose of the simulations, this under-pressure was neglected.

3.2 1D Simulations

1D simulations were carried out using GT-Suite. The software modules used under GT-Suite were:

- GEM3D v2018
- GT-ISE v2018
- GT-Post v2018

The 3D CAD model was first imported into GEM3D in order to perform a clean-up, so that the model can be converted into a representation that can be understood by the software. Once the clean-up is performed, the model can be generated and executed in GT-ISE. Once the model is executed, the results are then viewed and post-processed in GT-Post.

3.2.1 GEM3D

Before work on converting the CAD model to a 1D representation can be done, the CAD model was converted to a neutral format such as .stl, using CATIA v5. The .stl file was then imported into GEM3D for further processing. GEM3D was used for:

- Basic clean-up
- Discretization
- Creation of components from surfaces
- Creation of Muffler shells and Baffles
- Perforations
- Wool operations
- Specifying Ports
- Exporting into a GT Model

3.2.1.1 Basic clean-up

The CAD models of the exhaust systems will usually have a lot of features that are not necessary for simulations. These include: welds, mounting hooks, support structures, bolts and nuts. All such features that are not in the flow path were removed before further clean up can be done. This was done either in CATIA v5 (before importing), or in GEM3D (after importing).

3.2.1.2 Discretization

Discretization is the process of dividing the entire system into small sections. This is especially important to highlight details that are present in regions of sudden cross section change and really where smaller details are present. Since the exact geometry cannot be shown in a 1D model, the discretization needs to be done at

proper length intervals. If it is too large, then the geometry is approximated and some smaller details may be skipped altogether. The next step was to divide the exhaust system into sub-assemblies, so that the final model was modular in nature. Also, the inlet and outlet ports for each sub-assembly were specified, to enable the solver to understand the direction of flow of gases. The exhaust systems were divided into the following sub-assemblies:

- Front muffler (if present)
- Connection pipe
- Rear muffler

3.2.1.3 Creation of components from surfaces

Once this basic clean-up was done, there were a lot of shapes and surfaces left from importing the 3D CAD model. These surfaces were converted into components in GEM3D to give the software a clear understanding of the shape from the CAD model. The surfaces that are relevant in most cases are those that are directly in the flow path. This meant that the outer surfaces of most components could be deleted. Each surface was then classified into different types of components such as pipes and shell elements.

3.2.1.4 Creation of muffler shells and baffles

The shapes of mufflers and baffles are usually very complicated to be converted directly from surfaces to components. Hence, it was easier to create them using the tools in GEM3D. To get an accurate representation of the muffler shell, all the surfaces except the inner surface of the muffler shell were deleted. This surface was then used to create the muffler shell. GEM3D also has some useful features to identify the baffles.

3.2.1.5 Perforations

The holes on the baffles and pipes were created in GEM3D. During conversion of the imported shapes from CAD, a perforation option was used to specify the location and number of perforations on various components. In 1D simulations, the model considers all perforations as orifices, and just establishes links with other volumes around it. This means that the exact location of the holes is not important, but the overall open area of the holes is important as this influences how many volumes are connected to each other.

3.2.1.6 Wool operations

Some of the muffler chambers were filled with insulating glass wool. This is represented in the 1D model by adding some resistance coefficient to flow in certain directions.

All operations performed in Section 3.2.1.3 to Section 3.2.1.6 need some parameters that are used to specify things like initial pressures, temperatures and wall proper-

ties. These parameters were set as reference objects, as then be imported and edited in the case setup of the main model, as shown in Section 3.2.2.2

3.2.1.7 Specifying Ports

Each sub assembly needs to have inlet and outlet ports that specify the direction of flow. These were specified by utilizing sub-assembly connections. These connections were later used for connecting all parts in the final model. The ports in each assembly were assigned numbers that made them unique in the final model.

At this point, the clean up of all the components of the exhaust system is completed. Figure 3.4 shows a 3D representation of a Front Muffler in GEM3D after the clean up.

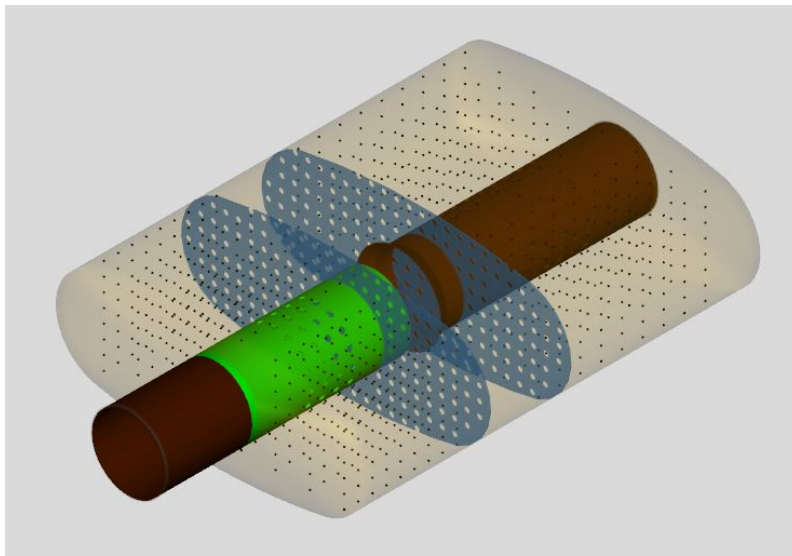


Figure 3.4: 3D Representation of a Front Muffler in GEM3D

3.2.1.8 Exporting into a GT Sub-Assembly

Once all the operations were finished, the GEM3D file was exported into a GT sub-assembly as a .gtsub file. This is the step which finally converts the 3D model into a 1D representation. The conversion is governed by specifying various parameters that instruct the software to divide the internal volume of a component into smaller sub volumes. These include:

- **Pipe Discretization Length** - Specifies how small each element of the pipe needs to be
- **Shell Discretization Lengths along X, Y and Z directions** - Specifies how big the volumes of the cubes that the inside of the rear muffler is divided into
- **Flowsplit Acceptance ratio** - Specifies the percentage of the cube's volume that must be present for the flow split to be retained

All these settings were decided based on the type of geometry, and how well it was represented in the discretized GT Model.

The 3D geometry was then converted into a representation. The 1D representation of the various components consists of volumes that are connected to each other by flow split elements and orifices. Figure 3.5 shows what a front muffler looks like after conversion of the GEM3D file to a .gtsub file.

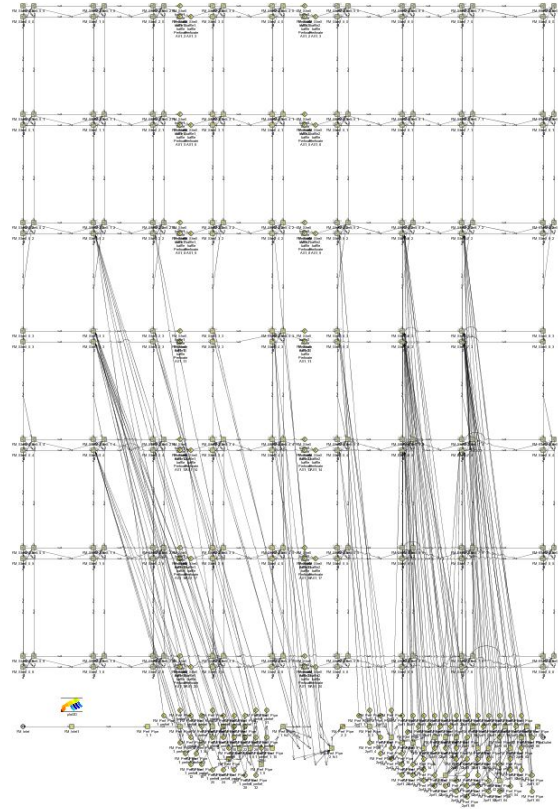


Figure 3.5: Front Muffler in after conversion into a .gtsub file

3.2.2 GT-ISE

Once each sub-assembly is converted into a .gtsub file in GEM3D, they can be imported into GT-ISE. GT-ISE is used for:

- Creating a model
- Boundary conditions
- Run Setup
- Plot setup

3.2.2.1 Creating a model

The first step was to create a blank model in GT-ISE. The required library templates were then imported into the model from the library of available templates in GT-ISE. The templates used for the purpose of setting up the simulations for back pressure are shown in table 3.1

Table 3.1: Templates used for 1D Simulations

Template	Function
EndEnvironment	Ambient Atmosphere at Outlet
EndFlowInlet	Inlet boundary condition
OrificeConn	Connections between components
MathEquation	Used for calculating pressure drop
RLTCreator	Recording pressure drop
SensorConn	Used to send values to MathEquation
SubAssemblyExternal	Input all parts of the exhaust system

The storage locations of the sub-assembly files were then mentioned so they can be imported into the model. Connections between the various sub-assemblies were established based on the order that they are supposed to be connected in.

An example of a model in GT-ISE is shown in Figure 3.6.

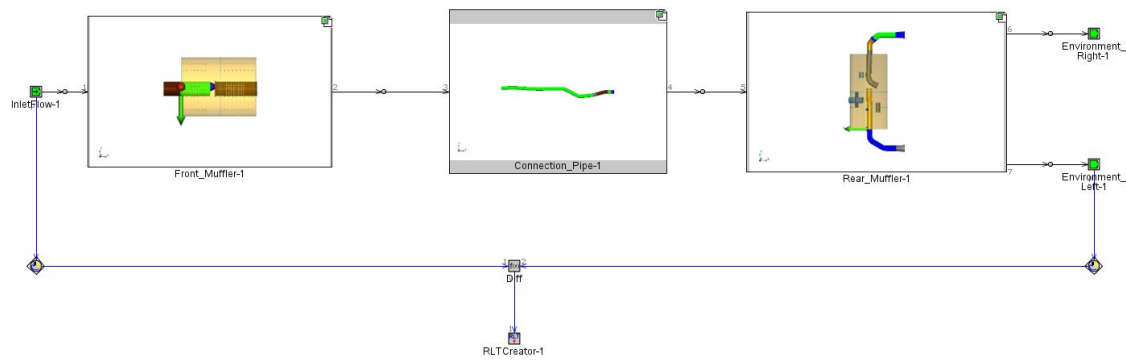


Figure 3.6: 1D Model in GT-ISE

This type of model is modular in nature, and the various components can easily be swapped with other components.

3.2.2.2 Boundary conditions

The boundary conditions were specified for the inlet and the outlet in each model. Most simulation parameters were set as reference objects, as it made it easier to setup the various cases in the simulations with one case setup.

- **Inlet Boundary Conditions:**

The inlet boundary was set as a mass flow inlet. For the simulations, air was used as the composition of the inlet gas as it was used in the physical tests. An array was used to represent the various mass flows. In this thesis, the mass flow values that were evaluated range from **100 kg/h** to **1200 kg/h**. For each mass flow, the simulations were run with gas temperatures of **20°C** and **775°C**.

- **Outlet Boundary Conditions:**

The outlet boundary was set as an end environment. For the simulations, air was selected as the composition of the environment. The temperature was set to **20°C**, which was the ambient temperature of the room in the test rig.

- **Wall Boundary Conditions:**

All components were assigned wall boundary conditions. The wall boundary conditions include:

- Assigning materials to walls
- Assigning the thermal conditions
- Changing friction multipliers for various components
- Assigning the wall temperatures
- Assigning initial fluid conditions

For the simulations, the walls were considered to be adiabatic. The initial temperature of the walls was set to the same as the inlet gas temperature to avoid numerical errors during the initial stages of simulation. Also, the wall material was assigned as steel. This ensures that the friction losses in the exhaust system are modelled correctly.

3.2.2.3 Run Setup

The next step was to perform the Run Setup. This included settings and options pertaining to running the simulation and post processing the results. Some of the parameters that were modified here include:

- Time Control: Continuous
- Maximum Simulation Duration: 10 seconds - This is set to give some hard to converge cases enough time to converge.
- Initialization: User Imposed
- Flow Control: Explicit
- ODE Solver: Explicit Runge Kutta - This is the recommended solver for Flow only simulations such as exhaust flow [8]

Once all these were set up correctly, the simulation was then run.

3.2.2.4 Simulating Hot Flow bench tests

Simulating of the Hot flow bench test has a few additional steps apart from the ones mentioned in Sections 3.2.2.1 to 3.2.2.3. In order to simulate the hot flow bench tests accurately, the model needed two additional components:

- Pressure measurement device
- Extension Pipe

This was done as there was only one static pressure measurement in the hot flow rig. Hence, modelling everything in the flow path will give a fair comparison with the bench results. Figure 3.7 shows the modified 1D model with the additional components from the hot flow bench.

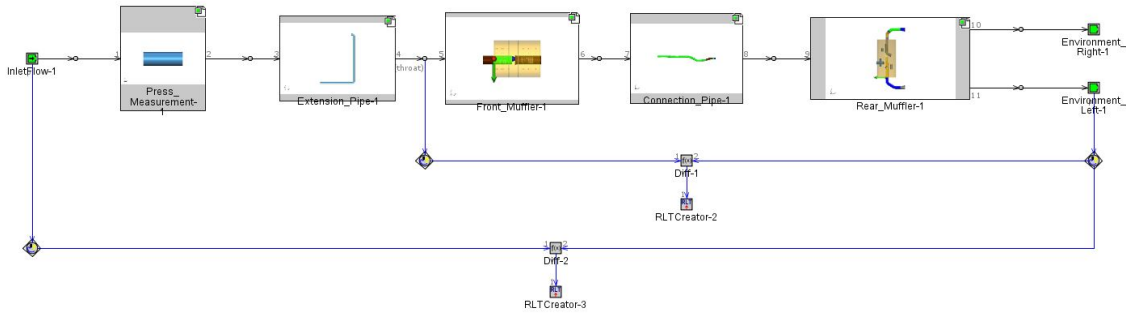


Figure 3.7: 1D Model of the Hot Flow Bench in GT-ISE

Since the exhaust gas temperatures at various positions were available from the hot flow bench tests, the 1D model was then calibrated to ensure that the temperatures correspond to the test results, at each of the measurement points. The pressure drop over the exhaust system was then noted from the simulation and compared with the results from the correlated cold flow test results.

3.2.3 GT-Post

GT-Post is the post processor in GT-Suite. It is used to:

- View results from the simulation
- Visualize various parameters in each component
- Create reports and plots

3.2.3.1 View results from the simulation

Results from the simulation were stored in the RLTCreator for each case. These are directly viewed by clicking on it in the GT-Post window to get a quick overview along with a plot of the pressure drop for various mass flows.

3.2.3.2 Visualize various parameters in each component

By clicking on any component on the simulation window, a detailed insight into the various thermal and flow properties was obtained. This was useful to look at as it gave a better understanding of the model, and also aided in spotting any anomalies in the simulation results. It was also useful during calibration of models as certain simulation parameters were changed, and the results were then observed to ensure that the calibration settings worked correctly.

3.2.3.3 Creating reports

Reports were used to store various sets of relevant data. Since the simulation results give a lot of data, it is a good practice to store the necessary data into a report file. For the thesis work, the data that was stored was inlet and outlet pressure at each major component. Plots were then created using this data and stored in the same report file.

3.3 3D Simulations

3D simulations are carried out in two stages:

- Pre-processing
- Simulations/Post Processing

Pre-processing, is carried out using ANSA, which is a 3D tool used to prepare the given geometry for simulations. ANSA is also used to generate the surface mesh.

Simulations and post processing are carried out in STAR-CCM+, which is a 3D CFD tool that can handle different types of fluid and thermal simulations.

3.3.1 ANSA

Pre-Processing for 3D simulations includes:

- Importing of the 3D geometry
- Assigning PIDs to all components
- Basic clean-up
- Deletion of duplicate surfaces
- Baffle processing
- Creating leak proof components
- Generating outlet volumes
- Creating additional surfaces for interfaces
- Generating a surface mesh
- Exporting the surface mesh

A brief description of the important steps during pre-processing for 3D simulations is mentioned below. A document with more detailed version of all the steps and settings for pre-processing was also created as part of the master thesis, but excluded from this report due to confidentiality.

Pre-processing for 3D simulations started with importing of the CAD Model with fine settings for resolution to ensure that no details are lost. After importing, the display settings were adjusted to ensure that all small components can be seen clearly. The next step was to assign names to all the parts in the assembly. Once all the parts were named, all unnecessary parts such as bolts and welds that are outside the flow path were deleted.

Since only the flow volume is necessary for the type of simulations performed in the thesis, all surfaces other than the flow surfaces were then deleted in a methodical way. Special attention was given to critical regions such as baffles and valves that are in the regions with high flows and velocities. These details are important as they lie directly in high flow paths, and can easily disturb the results from the simulations and cause variations in the results.

For the simulations to run correctly, it was crucial to make sure that all the components were leak proof. This was done by patching up small geometrical irregularities in the CAD model by deleting and/or creating new surface where necessary. Another modification that was performed was creating outlet volumes for the exhaust

system. This is to capture the effects of evacuation of the exhaust gases coming out of the system, and entering a large volume like in reality.

In order to track the pressure drop across individual components in the exhaust system, some surfaces are necessary. These were created next, marking the end of the cleanup. Figure 3.8 shows the cleaned geometry along with the outlet atmosphere volumes, prior to generating the surface mesh.

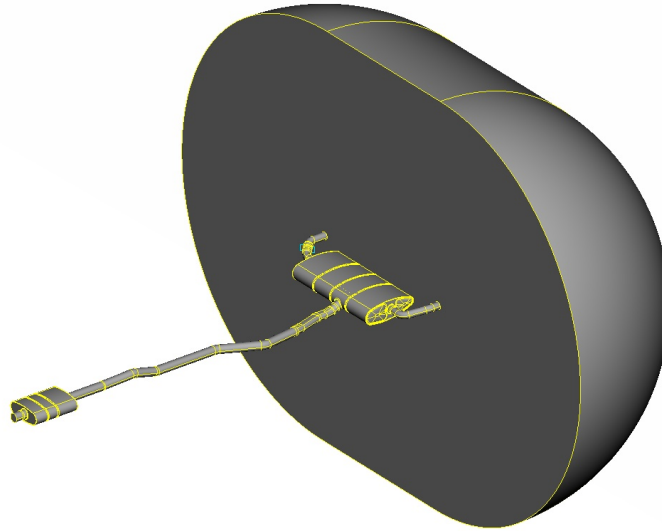


Figure 3.8: Cleaned Geometry ready for surface meshing

The next step was to prepare the perimeters for the generation the surface mesh. To ensure that there are no errors in the geometry, geometry checks were performed using the inbuilt functions in ANSA. The errors (if any) were fixed before proceeding further. Once the geometry was verified, the spacing of the points on the perimeters was set to the correct values to ensure that the settings are correct. To avoid excessive memory usage and computational time, coarser settings were used on non-critical surfaces like the atmosphere volume. The target perimeter lengths for the critical flow regions was set to be between **0.5 mm** and **1 mm**, whereas the perimeter lengths for the atmosphere was set to be between **2 mm** and **5 mm**.

The mesh was then generated. It is the final representation of the geometry that will be used to run the simulations. During meshing, the atmosphere and the exhaust system were meshed separately like when setting the perimeter spacing, to ensure good growth rate. The mesh was then checked to see if there are any odd cells, and necessary corrections were performed to ensure that the quality of the surface mesh was good. The mesh was then imported into a format (.inp) that can be interpreted by StarCCM+.

3.3.2 STAR-CCM+

The simulation and post processing is carried out using STAR-CCM+. The various steps involved are:

- Importing the surface mesh
- Creating parts from imported surfaces
- Imprinting
- Creating an extension before the inlet
- Assigning all parts to regions
- Specifying boundary type
- Setting up continua
- Creating shapes for volume refinements
- Meshing operation
- Creating Derived parts
- Scalar setup
- Report setup
- Monitor/Plot Setup
- Stopping criterion

A brief description of the important steps is mentioned below. A document with more detailed version of all the steps and settings for the 3D simulations was also created as part of the master thesis, but excluded from this report due to confidentiality.

Simulation setup began with importing the surface mesh that was generated after pre-processing. Once the mesh was imported, the various surfaces were then grouped up to form the various parts in the exhaust system. The next step was to compute the part curves, so the perimeters are identified correctly. The imprinting operation was performed next. Imprinting is done to connect various parts together and ensures that a conformal grid is obtained across interfaces. Conformal grids ensure a more stable simulation.

To ensure that the flow is fully developed at the inlet of the exhaust system, an extrusion was created before the inlet. The next step was to assign all the created parts to regions. Regions are necessary to assign wall and boundary conditions. The boundary conditions that were used are mass flow inlet, pressure outlet and walls for the exhaust system. The mass flow for all simulations was set as **720 kg/h** and the outlet pressure was set as **0 Pa**, as the reference pressure was set as **1 bar**. Also the temperatures of the inlet gas used was either **20°C** or **775°C** depending on which case was being run. The ambient temperature was set as **20°C**.

The next step was to set up the physics continua, so the software knows what models to use to calculate the flow characteristics. The models used for simulating the flow of gases in the exhaust system are shown in Table 3.2

Table 3.2: Physics models used

Physics Condition	Model Used
Space	Three Dimensional
Time	Steady
Material	Gas
Flow	Segregated Flow
Equation of State	Ideal Gas
Energy	Segregated Fluid Temperature
Viscous Regime	Turbulent
Reynolds-Averaged Turbulence	K-Omega Turbulence
Optional Models	Cell Quality Remediation
Wall Treatment	All y+ Wall Treatment Exact Wall Distance
Turbulence Model	SST (Menter) K-Omega
Equations of Motion	Reynolds-Averaged Navier-Stokes

Since all the regions in the exhaust system are not in the direct path of critical flow, different refinements can be used for different regions to reduce the cell count in the volume mesh and decrease computational time. Separate parts that represent these refinement regions were created so that the refinement settings can be applied to those regions easily.

The meshers used to represent the exhaust system in STAR-CCM+ are:

- Surface Remesher
- Polyhedral Remesher
- Prism Layer Mesher

The base mesh size was set as **10 mm**, and everything else was set in relation to this. In regions of high flow, such as pipes and mufflers, the value was set to 40% of the base size, i.e., **4 mm**. Regions that have more details in the high flow path (baffles and rear muffler) were set to have a value of 10% of the base size, i.e., **1 mm**. Less important regions like the atmosphere were set to a much larger size, i.e., 1000% of the base size, to reduce the cell count drastically. Another aspect was the prism layers. These are used on surfaces of pipes where there is flow interaction with high gradients, i.e., on walls of pipes. The base number of prism layers was set as 2, but for pipes and other regions of high gradients, the number of prism layers was either **8 or 5** depending on where it is located along the flow path.

These settings are only representative, and can vary a lot between different geometries. Special care must be taken to ensure small details like baffle leaks and perforations are captured effectively. In order to capture the baffle leaks, special settings on the prism layers need to be applied to the baffles as they tend to collapse, leading to numerical instabilities in the simulations.

Meshing usually takes between **3-5 hours** for the type of geometries that are used

in the thesis. The meshing time may vary depending on the size and complexity of the exhaust system. Figure 3.9 shows a cross section of the generated volume mesh in STAR-CCM+.

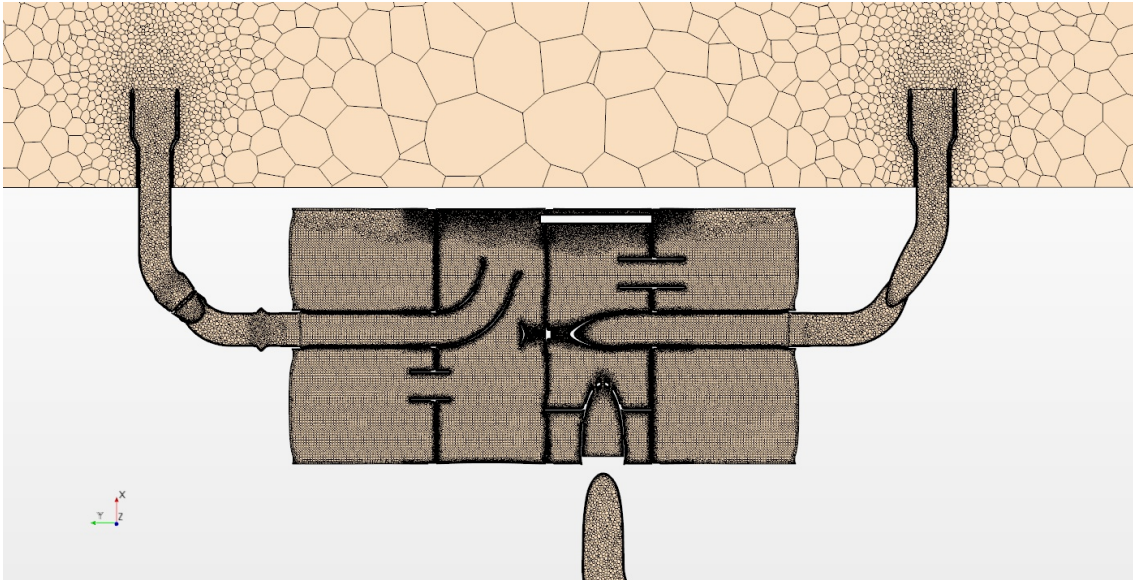


Figure 3.9: Cross sectional view of Rear Muffler after meshing in STAR-CCM+

Once the meshing operation was completed, the next step was to specify what parameters had to be recorded from the simulations, so that the results can be analyzed. This was done before running the simulations, so that the parameters can be monitored as the simulation is running. The parameters of interest during simulations are:

- Mach Number
- Velocity
- Pressure drops across various components
- Wall Y^+ values

In order to visualize the 3D flow, various scenes are created before the simulations were run. The visualizations for velocity and mach number are of most interest to observe flow paths. Also, plots were created so that the convergence of the residuals and other variables could be observed easily.

Finally, the simulations were run, and the monitors are observed for every iteration. The number of iterations for which the simulations runs was decided based on observing the residuals and seeing when they start converging. The results from the simulations are considered to be converged if the residuals and the pressure drops over various components do not change for at least 10000 iterations. Figure 3.10 shows a scene created in STAR-CCM+, which is a 3D visualization of flow velocity through a rear muffler.

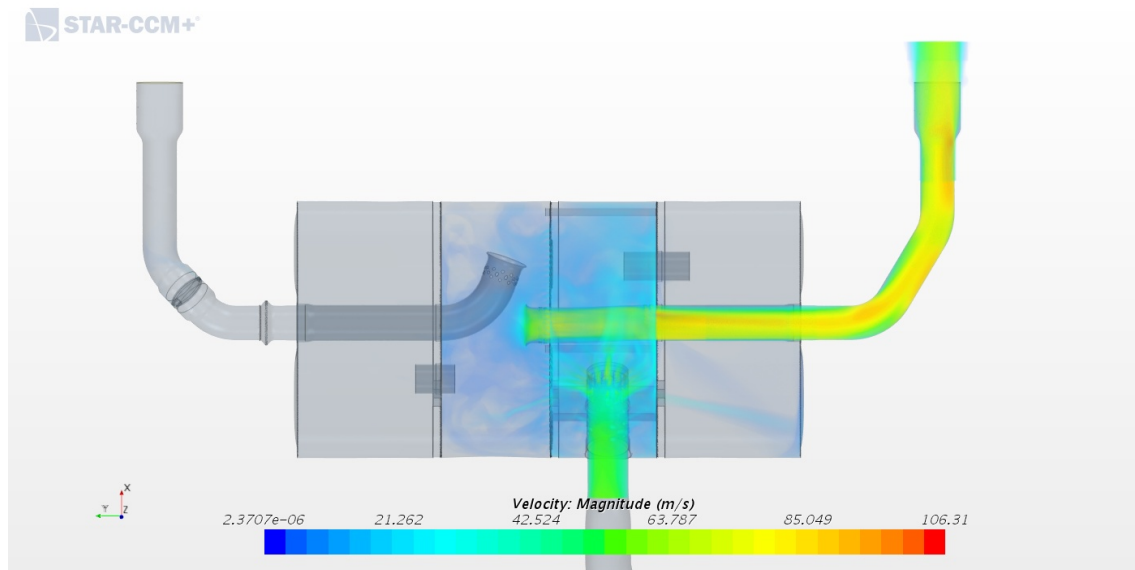


Figure 3.10: 3D Visualization of Flow velocity through a rear muffler

4

Results and Discussion

The results for each system are shown below. The results from the simulations are compared to those from the bench test, and the difference is specified in percentage. The results from the 1D simulations is the pressure drop at various mass flows, and is represented as a sweep. The 3D simulation results are an average of at least the last 10000 iterations after reaching stability.

4.1 Overview of Simulation Settings

4.1.1 Base settings

These are the settings that are currently in use at Volvo Cars. All the simulations are first run with these settings first to get a baseline from the simulation results. The settings used are:

- Baffle leaks not modelled in 1D or 3D
- **1D Simulations**
 - Muffler shell of similar volume is created in GEM3D
 - Smaller perforations are clubbed together into a perforation that has a similar area
 - Plates and non flow components inside muffler are ignored
 - Flares on pipes ignored
 - Tailpipe Valve ignored
 - Discretization length of 40 mm is used for pipes
 - Flow split acceptance ratio of 50% was used
- **3D Simulations**
 - Tailpipe Valve is simplified

4.1.2 Optimized settings

These are the settings that gave the closest results to the bench tests. Simulations were run with these settings and then compared to the baseline results to see the improvements in results.

- Baffle leaks modelled in both 1D and 3D
- **1D Simulations**
 - Exact geometry of Muffler shell is converted in GEM3D
 - Most perforations are retained in correct positions.
 - Support Plates and non flow components inside muffler are modelled
 - Flares on pipes modelled
 - Tailpipe Valve modelled as an orifice with a certain open area
 - Discretization length of 5 mm is used for pipes to capture more details

- Flow split acceptance ratio of 25% was used
- **3D Simulations**
 - Tailpipe Valve is very close to CAD model with very few simplifications

Apart from this, there were also several other changes on how the CAD was pre-processed and how the discretization was performed.

For 1D simulations, a study was made regarding including all the various small details, before deciding on the settings that were used for all systems. The studies were performed on Exhaust System 2, as the geometry was neither too complicated or too simple. These studies showed a very slight improvement in the back pressure estimates, and hence these modifications were included in the optimized settings for the final 1D model.

For 3D simulations, the settings that are currently in use are a right compromise between computational demand and accuracy. A study into the mesh independence for CFD simulations of cold end of the exhaust system was conducted at Volvo Cars previously [11]. Settings similar to those used in the study were used to run the simulations without any further investigation into the sensitivity of the mesh size.

The images shown in the results for the 3D simulations are mainly focused on the rear muffler, as most of the optimization was performed in the rear muffler. Also, the rear muffler contributes to the largest pressure drop, and hence, is critical to monitor more closely than the rest of the exhaust system.

4.2 System 1

This is the exhaust system used with the low performance variant of the 4 cylinder diesel engine that is currently produced by Volvo Cars. The isometric view of System 1 is shown in Figure 4.1

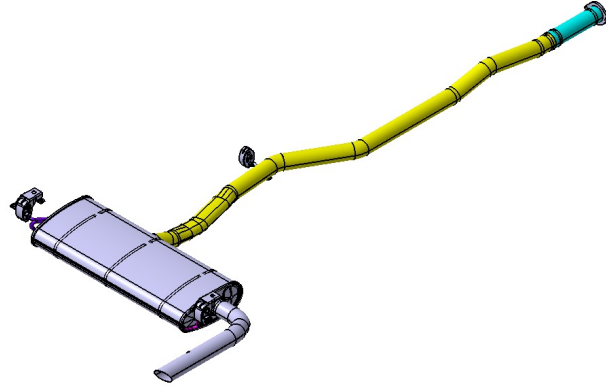


Figure 4.1: Isometric view of System 1

This exhaust system has the simplest rear muffler geometry among the three systems that were focused on during the thesis, as it has no front muffler and a simple rear muffler with just one single tailpipe. Figure 4.2 shows the internals of the rear muffler of System 1.

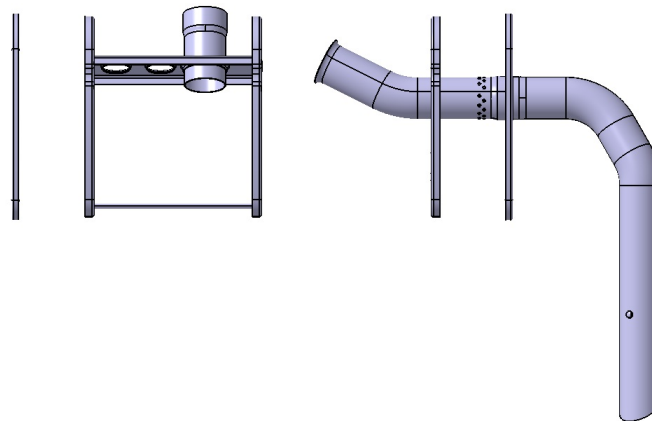


Figure 4.2: Rear Muffler Internals of System 1 System

4.2.1 1D Simulations

Figures 4.3 and 4.4 shows the pressure drop of System 1 at 20°C and 775°C at various mass flows, respectively.

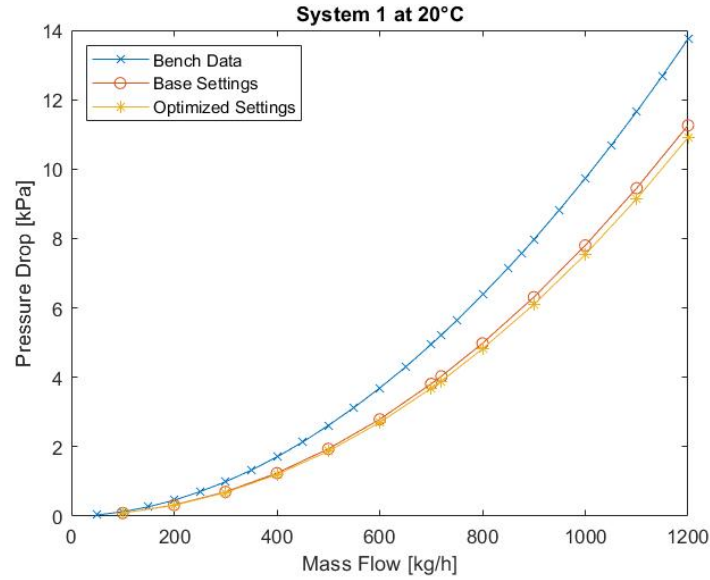


Figure 4.3: Pressure Drop of System 1 at 20°C

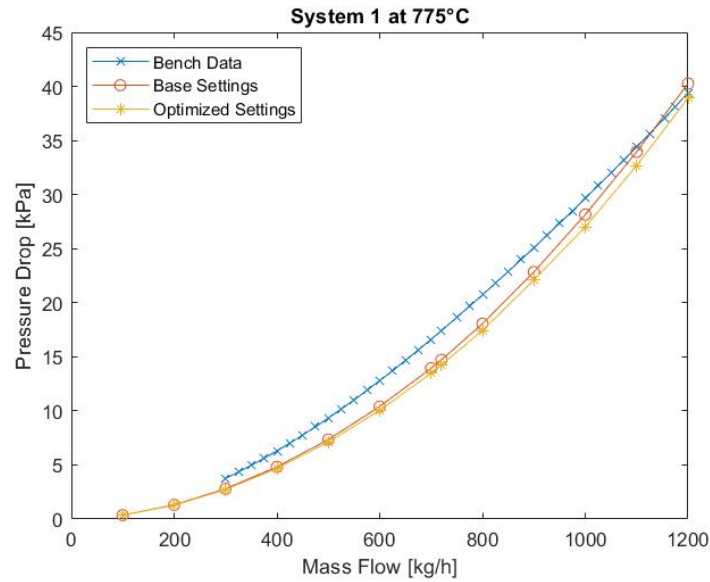


Figure 4.4: Pressure Drop of System 1 at 775°C

From these plots, it can clearly be observed that the results with the optimized settings under-predict the back pressure even more, as compared to when the base settings are used. It was thus concluded that including all the smaller geometrical details for this geometry is not beneficial for 1D Simulations.

4.2.2 3D Simulations

Figures 4.5 and 4.6 show the velocity profiles inside the rear muffler of System 1 at 20°C and a flow rate of 720kg/h, using the base and optimized settings respectively.

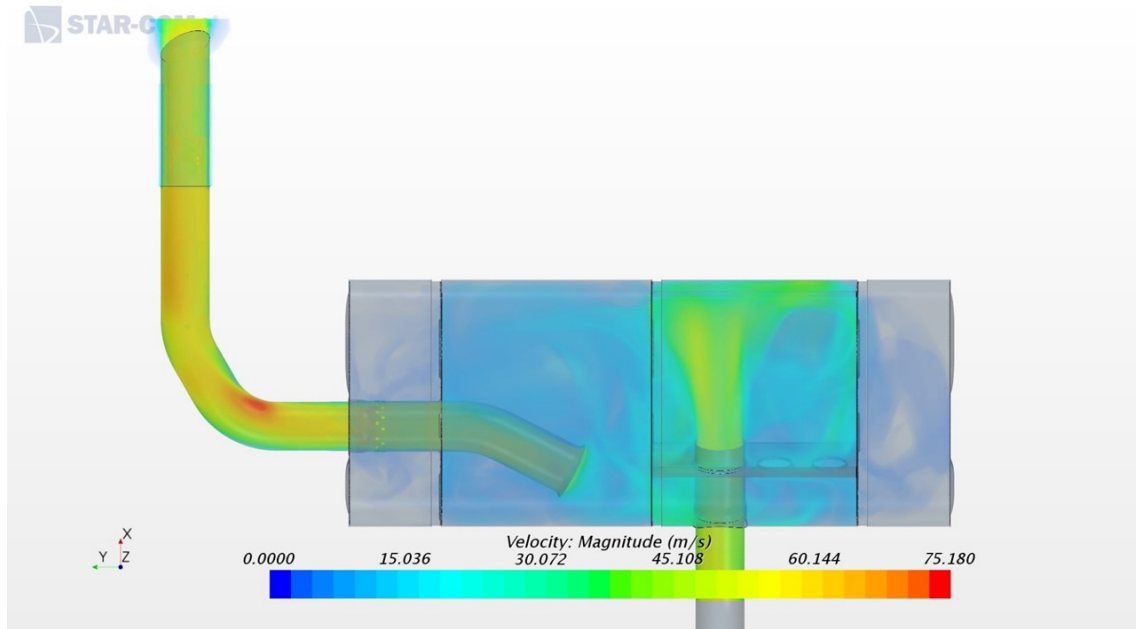


Figure 4.5: Rear Muffler of System 1 at 20°C, Base Settings

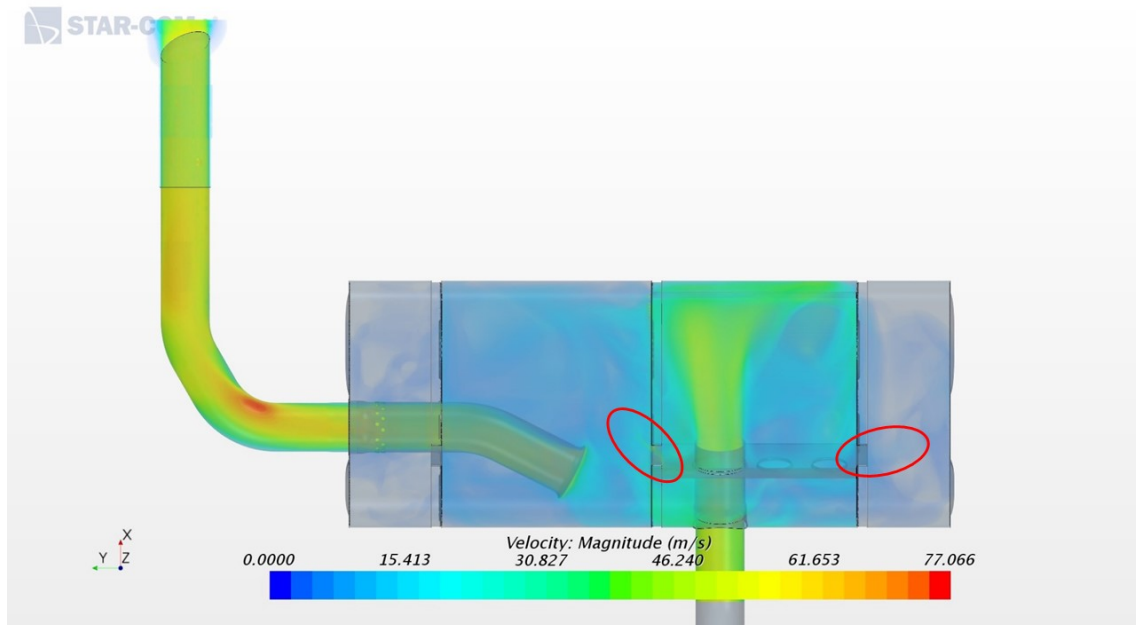


Figure 4.6: Rear Muffler of System 1 at 20°C, Optimized Settings

Figures 4.7 and 4.8 show the velocity profiles inside the rear muffler of System 1 at 775°C and a flow rate of 720kg/h, using the base and optimized settings respectively.

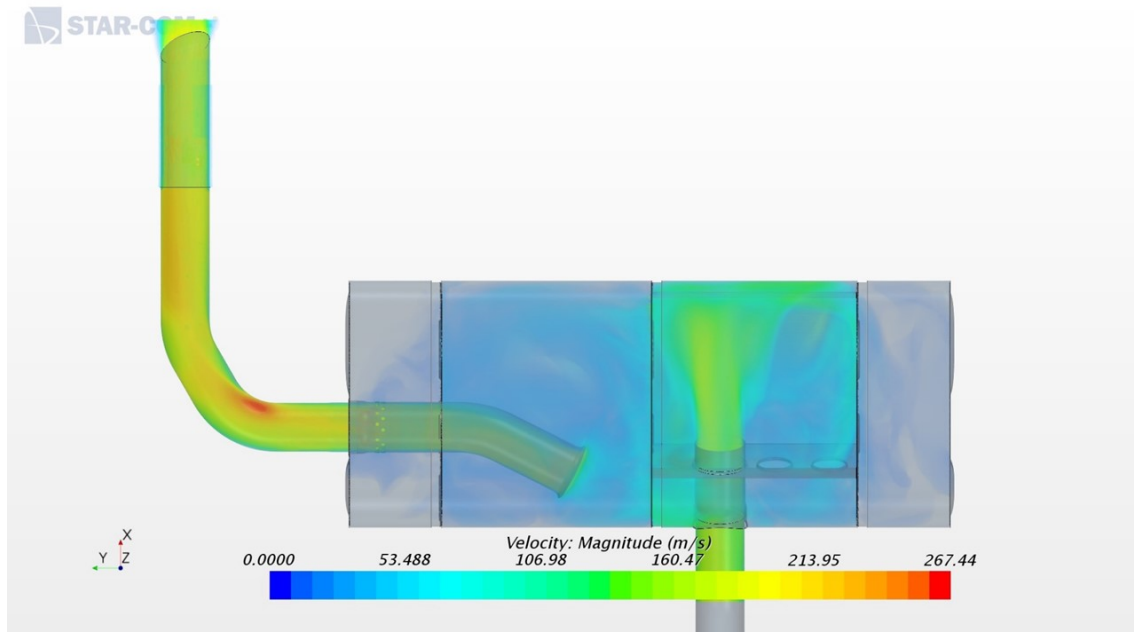


Figure 4.7: Rear Muffler of System 1 at 775°C, Base Settings

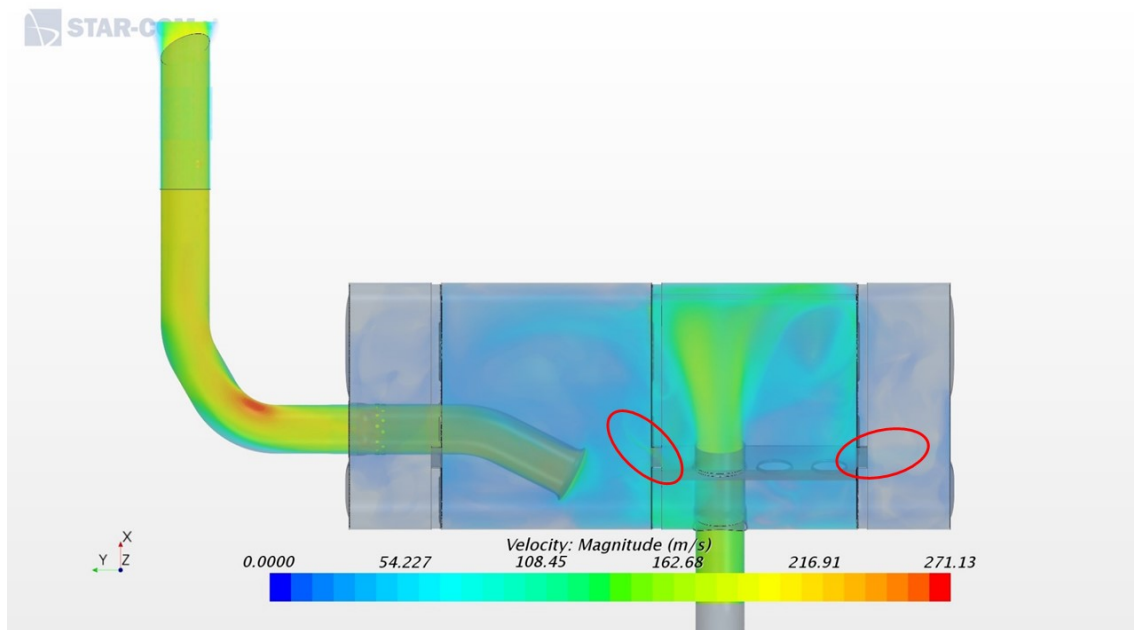


Figure 4.8: Rear Muffler of System 1 at 775°C, Optimized Settings

From these velocity profiles, we can see that there is not much difference between the profiles with and without optimization. Some slight differences in flow paths can be observed in the encircled regions, but the overall flow pattern remains very similar.

This can be attributed to the fact that there are no separated volumes in the rear muffler, as all the baffles have perforations. This in turn leads to a lesser pressure

differential between the chambers, and thus when the leaks are included, a significant change in flow pattern is not observed.

4.2.3 Summary

This section summarizes all the results in a tabular form to get an overview of the pressure drop over each component in System 1, and also the entire exhaust system.

4.2.3.1 20°C

The results for System 1 at 20°C using base and optimized settings are shown in Tables 4.1 and 4.2 respectively.

Table 4.1: Results for System 1 at 720kg/h, 20°C and base settings

	Bench	1D	3D
Connection Pipe	1.31 kPa	1.24 kPa	1.15 kPa
Rear Muffler	3.90 kPa	2.78 kPa	4.41 kPa
Overall Static Pressure Drop	5.21 kPa	4.02 kPa	5.56 kPa
% Difference	Ref	-22.8	+6.7

Table 4.2: Results for System 1 at 720kg/h, 20°C and optimized settings

	Bench	1D	3D
Connection Pipe	1.31 kPa	1.25 kPa	1.15 kPa
Rear Muffler	3.90 kPa	2.64 kPa	4.37 kPa
Overall Static Pressure Drop	5.21 kPa	3.89 kPa	5.52 kPa
% Difference	Ref	-25.3	+5.6

4.2.3.2 775°C

The results for System 1 at 775°C using base and optimized settings are shown in Tables 4.3 and 4.4 respectively.

Table 4.3: Results for System 1 at 720kg/h, 775°C and base settings

	Bench	1D	3D
Connection Pipe	4.24 kPa	4.44 kPa	3.88 kPa
Rear Muffler	13.17 kPa	10.26 kPa	15.43 kPa
Overall Static Pressure Drop	17.41 kPa	14.70 kPa	19.31 kPa
% Difference	Ref	-15.5	+10.9

Table 4.4: Results for System 1 at 720kg/h, 775°C and optimized settings

	Bench	1D	3D
Connection Pipe	4.24 kPa	4.48 kPa	3.88 kPa
Rear Muffler	13.17 kPa	9.82 kPa	15.33 kPa
Overall Static Pressure Drop	17.41 kPa	14.30 kPa	19.21 kPa
% Difference	Ref	-16.6	+10.3

From these result summaries, it can be observed that:

- 1D simulations get worse when the optimized settings are used
- 3D simulations are very close to bench results, but the results get slightly better when the optimized settings are used
- For this geometry, the optimization does not work as intended for 1D simulations, but make some improvements for 3D simulations

4.3 System 2

This is the exhaust system used for the high performance variant of the 4 cylinder petrol engine produced by Volvo Cars. The isometric view of the System 2 shown in figure 4.9

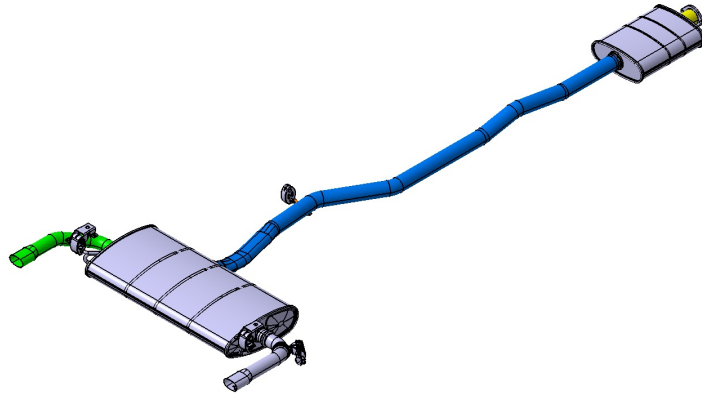


Figure 4.9: Isometric view of System 2

This exhaust system has a slightly more complicated rear muffler geometry than System 1, as it has a front muffler, two tailpipes, a valve on the tailpipe and Helmholtz Resonance chambers. Figure 4.10 shows the internals of System 2.

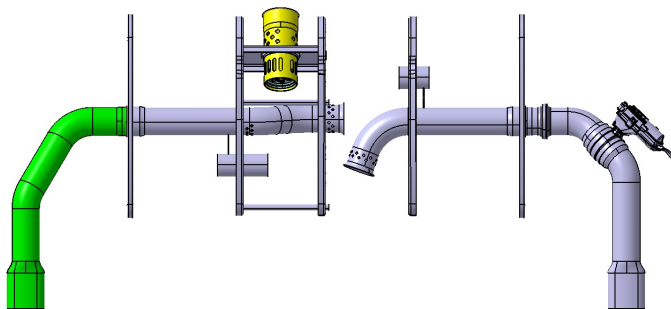


Figure 4.10: Rear Muffler Internals of System 2

4.3.1 1D Simulations

Figure 4.11 shows the pressure drop of System 2 at 20°C and 775°C with closed valve at various mass flows.

4. Results and Discussion

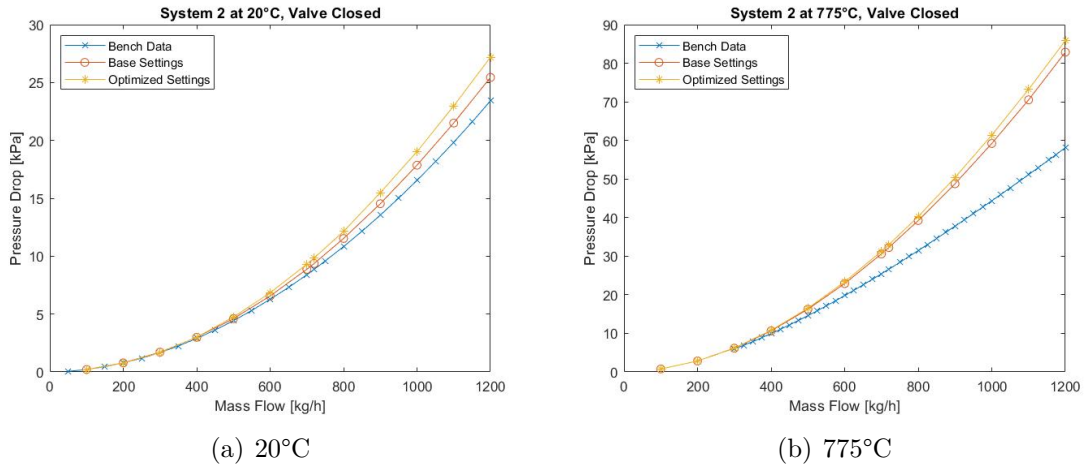


Figure 4.11: Pressure Drop of System 2, Closed Valve

From these plots, it can clearly be observed that the results on using the optimized settings over-predict the back pressure more than when the base settings are used. From this, it can be concluded that including all the smaller geometrical details for this geometry is not beneficial for the case where the exhaust valve is closed. This may be due to the numerical limitations of 1D calculations when calculating the overall pressure drop as one of the tailpipes is completely blocked.

Figure 4.12 shows the pressure drop of System 2 at 20°C and 775°C with open valve at various mass flows.

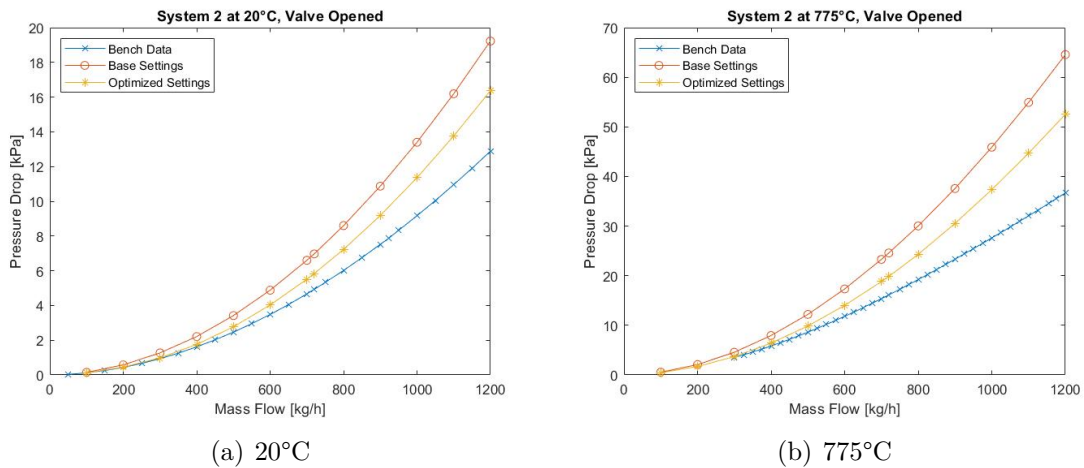


Figure 4.12: Pressure Drop of System 2, Open Valve

From these plots, it can clearly be observed that the results on using the optimized settings over-predict the back pressure lesser than when the base settings are used. From this, it can be concluded that including all the smaller geometrical details for this geometry is beneficial for the case where the exhaust valve is open.

4.3.2 3D Simulations

Figures 4.13 and 4.14 show the velocity profiles inside the rear muffler of System 2 at 20°C, valve closed and a flow rate of 720kg/h, using the base and optimized settings respectively.

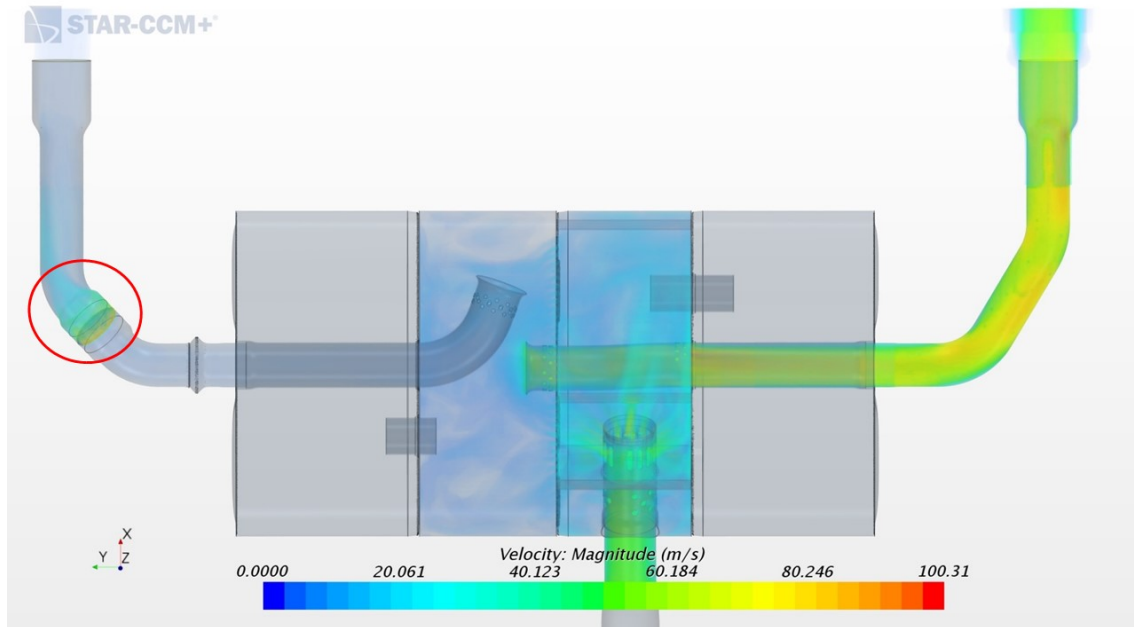


Figure 4.13: Rear Muffler of System 2 at 20°C, valve closed, Base Settings

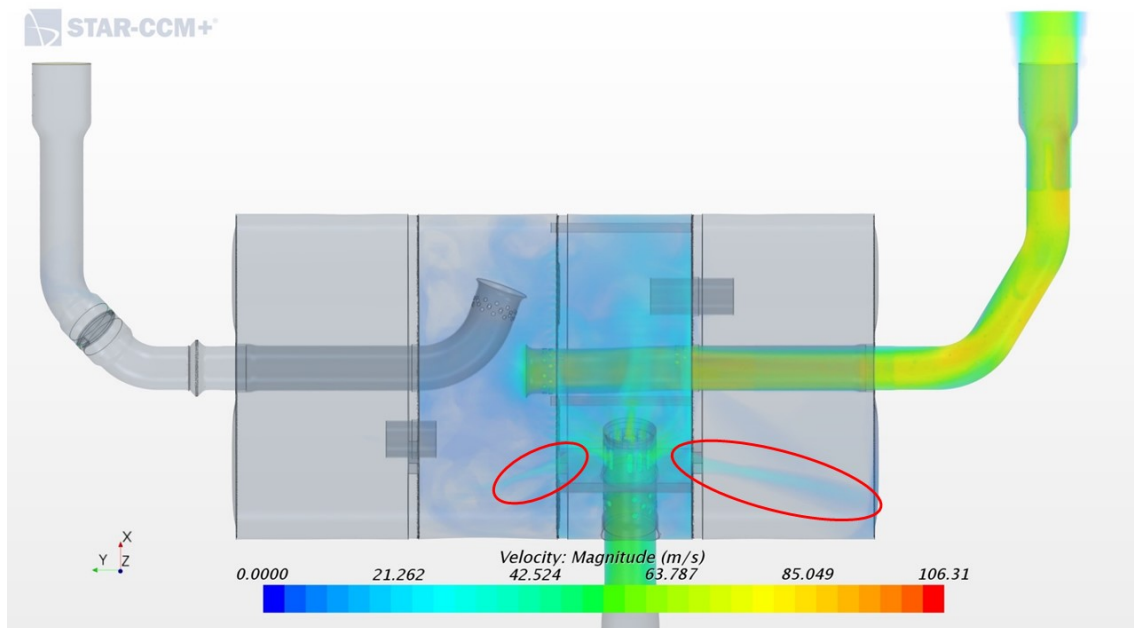


Figure 4.14: Rear Muffler of System 2 at 20°C, valve closed, Optimized Settings

Figures 4.15 and 4.16 show the velocity profiles inside the rear muffler of System

4. Results and Discussion

2 at 775°C, valve closed and a flow rate of 720kg/h, using the base and optimized settings respectively.

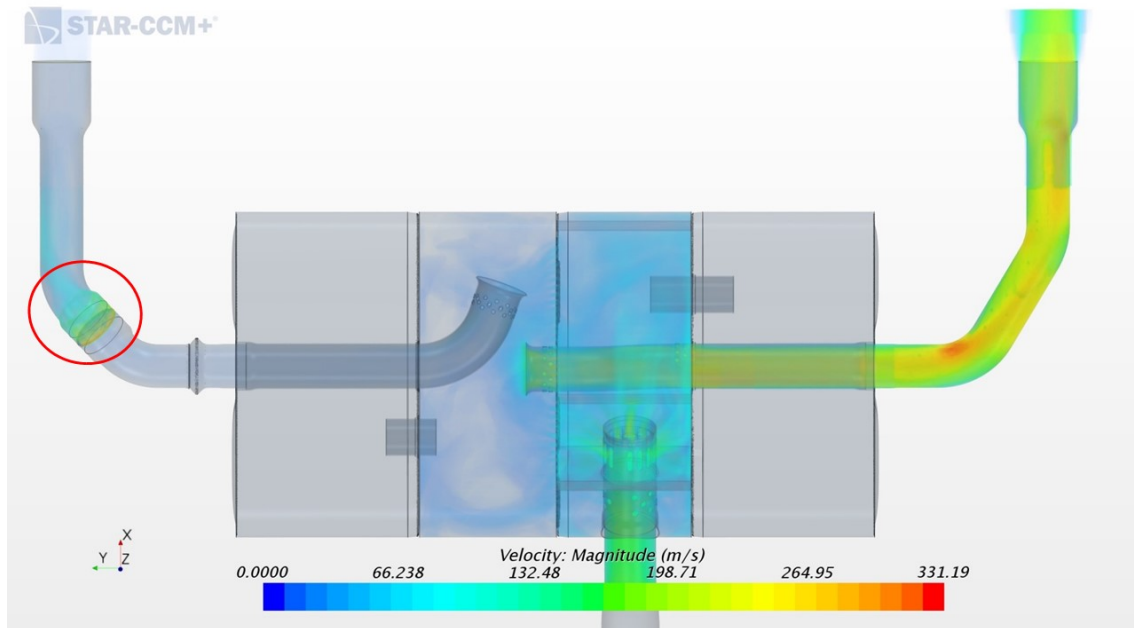


Figure 4.15: Rear Muffler of System 2 at 775°C, valve closed, Base Settings

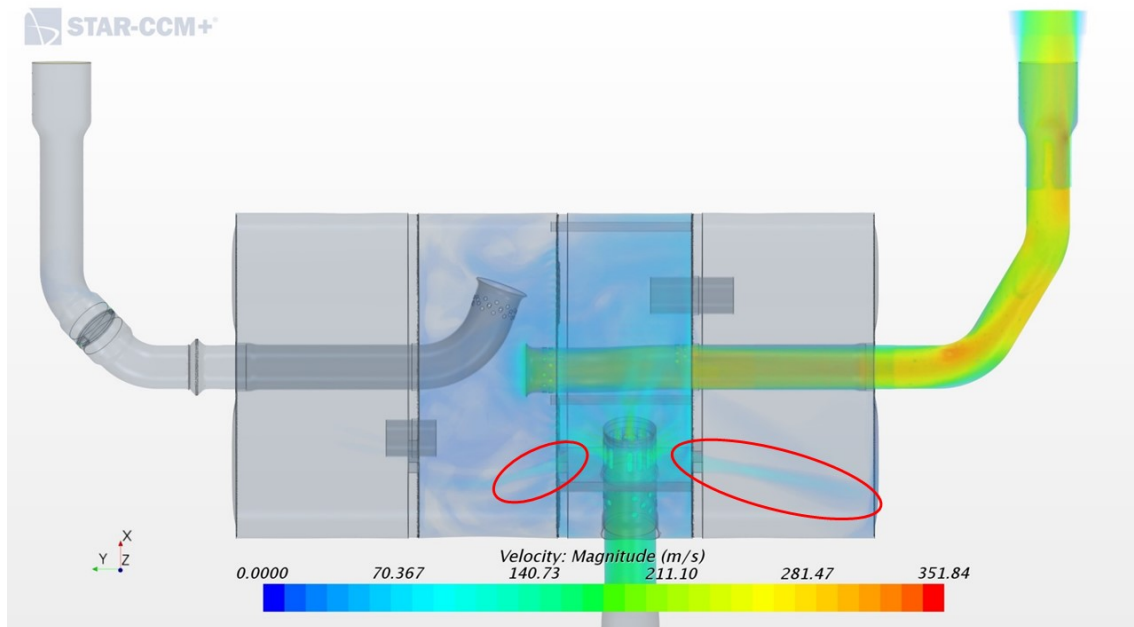


Figure 4.16: Rear Muffler of System 2 at 775°C, valve closed, Optimized Settings

Figures 4.17 and 4.18 show the velocity profiles inside the rear muffler of System 2 at 20°C, valve open and a flow rate of 720kg/h, using the base and optimized settings respectively.

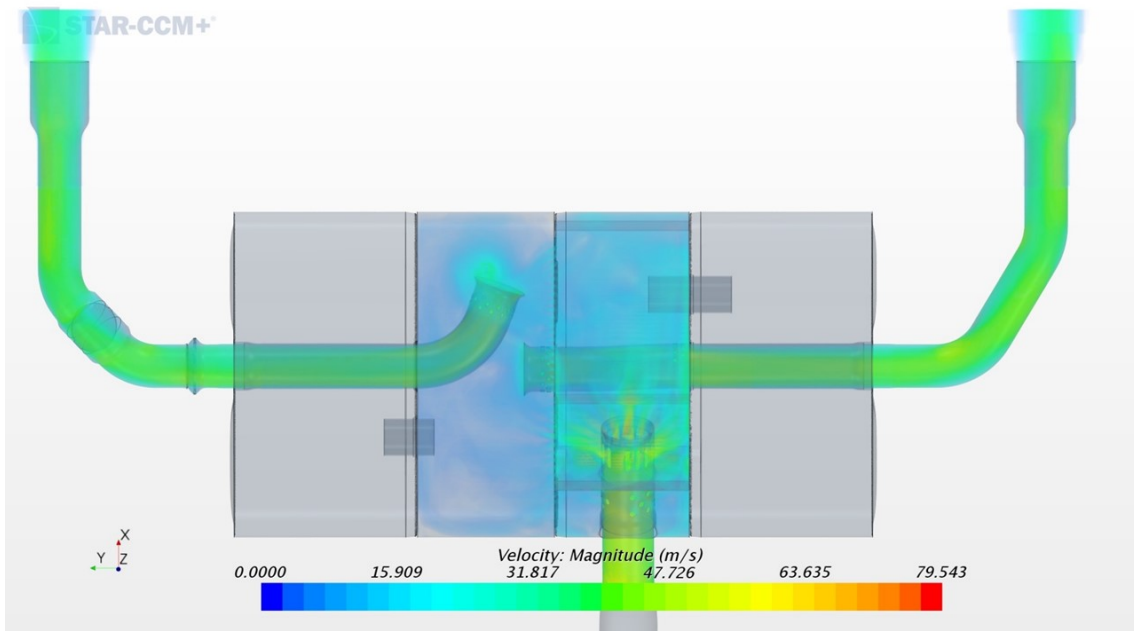


Figure 4.17: Rear Muffler of System 2 at 20°C, valve open, Base Settings

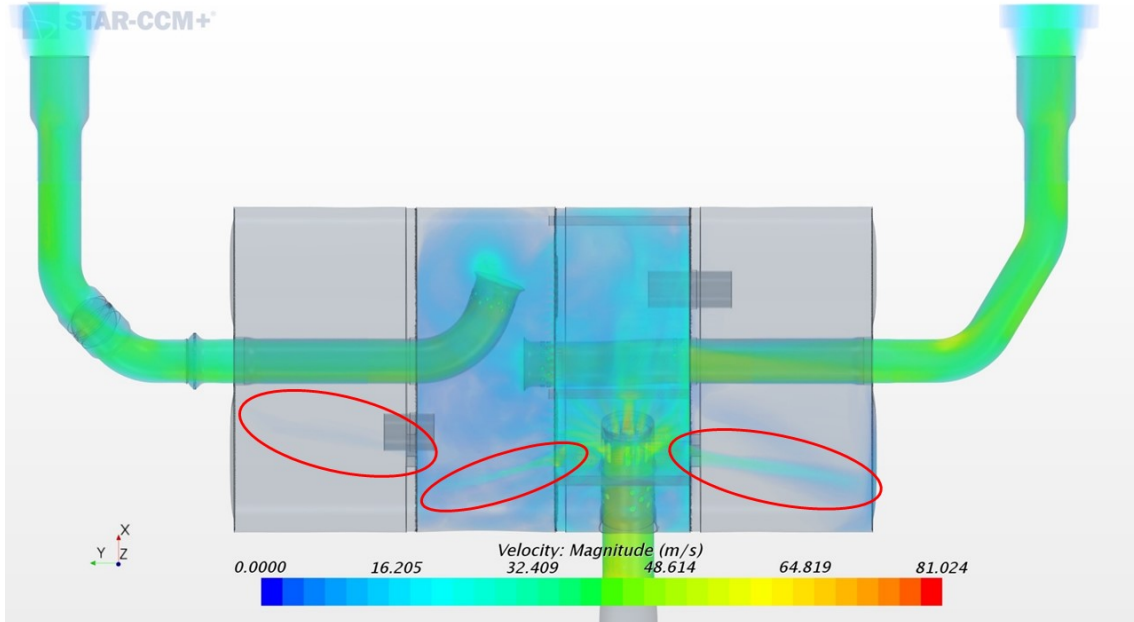


Figure 4.18: Rear Muffler of System 2 at 20°C, valve open, Optimized Settings

Figures 4.19 and 4.20 show the velocity profiles inside the rear muffler of System 2 at 775°C, valve open and a flow rate of 720kg/h, using the base and optimized settings respectively.

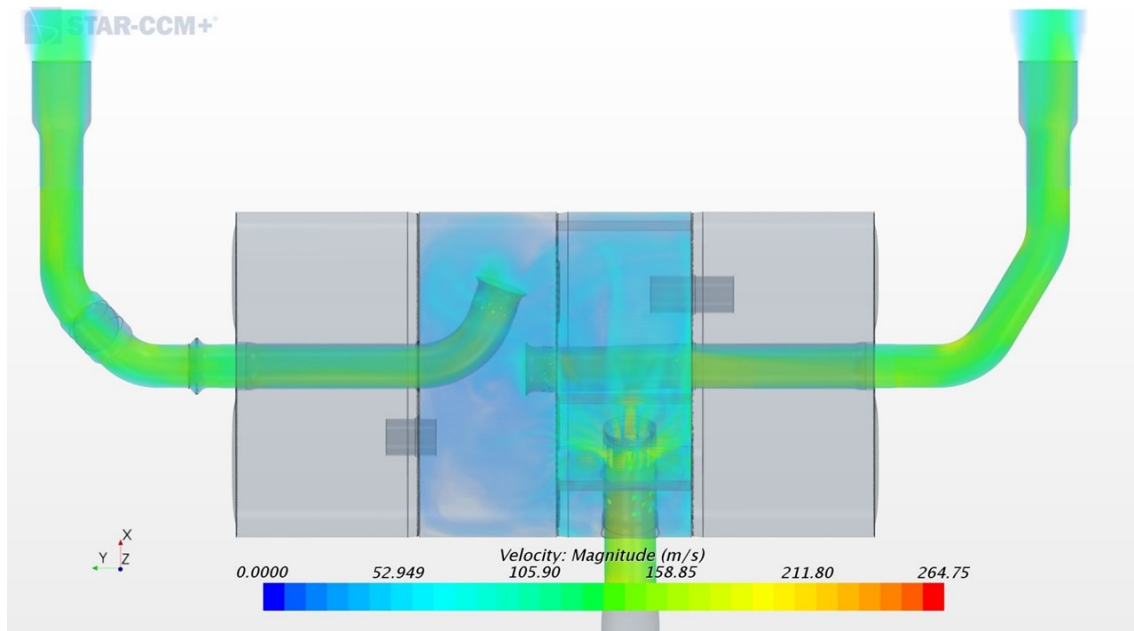


Figure 4.19: Rear Muffler of System 2 at 775°C, valve open, Base Settings

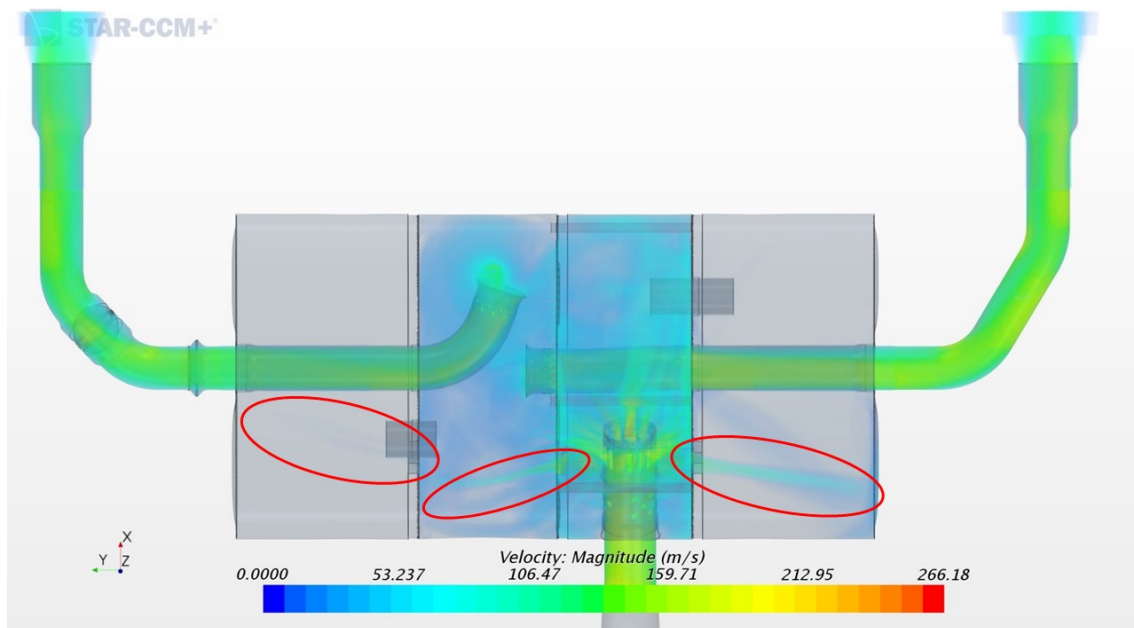


Figure 4.20: Rear Muffler of System 2 at 775°C, valve open, Optimized Settings

From these velocity profiles, we can see that there is a significant difference between the profiles with and without optimization. Clear differences in flow paths can be observed in the encircled regions. For cases where the exhaust valve remains closed, it can be noted that there are some numerical instabilities without optimization. These numerical instabilities vanish once the geometrical optimization is performed. These instabilities near the valve can be attributed to the fact that the mesh is not fine enough to capture the small gap between the valve and the tailpipe. Also, the

clear distinction in flow path may be due to separated volumes in the rear muffler, leading to a higher pressure differential between the chambers. Thus, when the leaks are included, a significant change in flow pattern is observed.

4.3.3 Summary

This section summarizes all the results in a tabular form to get an overview of the pressure drop over each component in System 2, and also the entire exhaust system.

4.3.3.1 20°C

The results for System 2 at 20°C with open valve, using base and optimized settings are shown in Tables 4.5 and 4.6 respectively.

Table 4.5: Base results for System 2 at 720kg/h and 20°C, Open Valve

	Bench	1D	3D
Front Muffler	0.60 kPa	1.93 kPa	0.38 kPa
Connection Pipe	0.96 kPa	1.18 kPa	1.15 kPa
Rear Muffler	3.37 kPa	3.87 kPa	3.13 kPa
Overall Static Pressure Drop	4.93 kPa	6.98 kPa	4.66 kPa
% Difference	Ref	+41.5	-5.48

Table 4.6: Optimized results for System 2 at 720kg/h and 20°C, Open Valve

	Bench	1D	3D
Front Muffler	0.60 kPa	1.93 kPa	0.37 kPa
Connection Pipe	0.96 kPa	1.18 kPa	1.15 kPa
Rear Muffler	3.37 kPa	2.72 kPa	3.19 kPa
Overall Static Pressure Drop	4.93 kPa	5.83 kPa	4.71 kPa
% Difference	Ref	+18.3	-4.4

The results for System 2 at 20°C with closed valve, using base and optimized settings are shown in Tables 4.7 and 4.8 respectively.

Table 4.7: Base results for System 2 at 720kg/h and 20°C, Closed Valve

	Bench	1D	3D
Front Muffler	0.57 kPa	1.86 kPa	0.35 kPa
Connection Pipe	0.91 kPa	1.18 kPa	1.11 kPa
Rear Muffler	7.38 kPa	6.33 kPa	6.81 kPa
Overall Static Pressure Drop	8.86 kPa	9.37 kPa	8.27 kPa
% Difference	Ref	+5.7	-6.7

Table 4.8: Optimized results for System 2 at 720kg/h and 20°C, Closed Valve

	Bench	1D	3D
Front Muffler	0.57 kPa	1.86 kPa	0.35 kPa
Connection Pipe	0.91 kPa	1.18 kPa	1.10 kPa
Rear Muffler	7.38 kPa	6.81 kPa	7.54 kPa
Overall Static Pressure Drop	8.86 kPa	9.85 kPa	8.99 kPa
% Difference	Ref	+11.2	+1.5

4.3.3.2 775°C

The results for System 2 at 775°C with open valve, using base and optimized settings are shown in Tables 4.9 and 4.10 respectively.

Table 4.9: Base results for System 2 at 720kg/h and 775°C, Open Valve

	Bench	1D	3D
Front Muffler	1.65 kPa	5.77 kPa	1.28 kPa
Connection Pipe	3.42 kPa	4.24 kPa	3.98 kPa
Rear Muffler	11.05 kPa	14.58 kPa	10.90 kPa
Overall Static Pressure Drop	16.12 kPa	24.59 kPa	16.16 kPa
% Difference	Ref	+52.5	+0.2

Table 4.10: Optimized results for System 2 at 720kg/h and 775°C, Open Valve

	Bench	1D	3D
Front Muffler	1.65 kPa	5.77 kPa	1.26 kPa
Connection Pipe	3.42 kPa	4.24 kPa	3.98 kPa
Rear Muffler	11.05 kPa	9.84 kPa	11.11 kPa
Overall Static Pressure Drop	16.12 kPa	19.85 kPa	16.35 kPa
% Difference	Ref	+23.1	+1.4

The results for System 2 at 775°C with closed valve, using base and optimized settings are shown in Tables 4.11 and 4.12 respectively.

Table 4.11: Base results for System 2 at 720kg/h and 775°C, Closed Valve

	Bench	1D	3D
Front Muffler	1.37 kPa	5.08 kPa	1.10 kPa
Connection Pipe	2.88 kPa	3.68 kPa	3.53 kPa
Rear Muffler	22.36 kPa	23.43 kPa	23.87 kPa
Overall Static Pressure Drop	26.61 kPa	32.19 kPa	28.50 kPa
% Difference	Ref	+20.9	+7.1

Table 4.12: Optimized results for System 2 at 720kg/h and 775°C, Closed Valve

	Bench	1D	3D
Front Muffler	1.37 kPa	5.06 kPa	1.09 kPa
Connection Pipe	2.88 kPa	3.68 kPa	3.46 kPa
Rear Muffler	22.36 kPa	24.22 kPa	26.52 kPa
Overall Static Pressure Drop	26.61 kPa	32.96 kPa	31.07 kPa
% Difference	Ref	+23.9	+16.8

From these result summaries, it can be observed that:

- When the valve is open:
 - 1D simulations improve a lot when the optimized settings are used
 - 3D simulations extremely close to bench results for all cases, and the optimized settings do not make much of a difference
- When the valve is closed:
 - 1D simulations get worse when the optimized settings are used
 - 3D simulations get worse when the optimized settings are used
- For this geometry, the optimization makes an overall improvement to the results in 1D simulations, whereas the 3D simulations are more or less accurate to begin with, and the deviations are within the margin of computational errors.

4.4 System 3

This is the exhaust system used for the medium performance variant of the 3 cylinder petrol engine produced by Volvo Cars. The isometric view of the System 3 is shown in figure 4.21

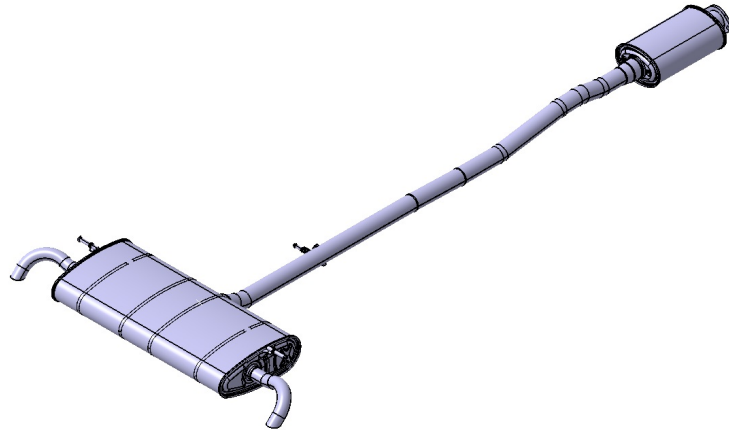


Figure 4.21: Isometric view of System 3

This exhaust system has the most complicated rear muffler geometry among the three systems that were focused on during the thesis, as there is a 'half-shell' construction of tubes to make manufacturing of the exhaust system easier. Also, there is glass wool in the rear muffler to further aid noise attenuation. Figure 4.22 shows the internals of System 3.

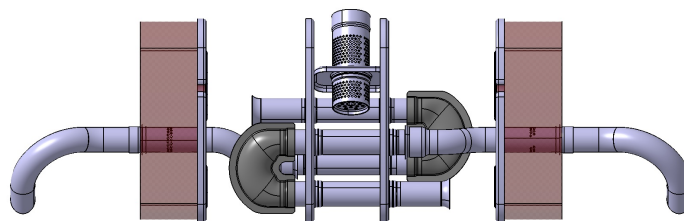


Figure 4.22: Rear Muffler Internals of System 3

4.4.1 1D Simulations

Figures 4.23 and 4.24 show the pressure drop of System 3 at 20°C and 775°C at various mass flow rates, respectively.

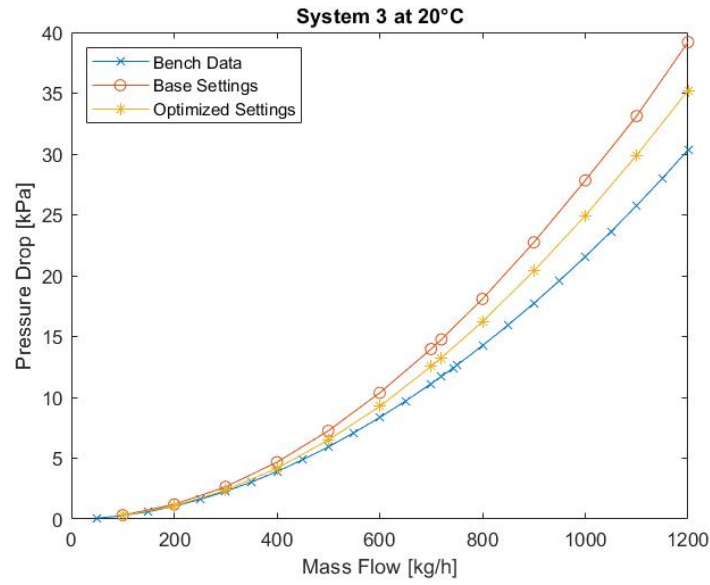


Figure 4.23: Pressure Drop of System 3 at 20°C

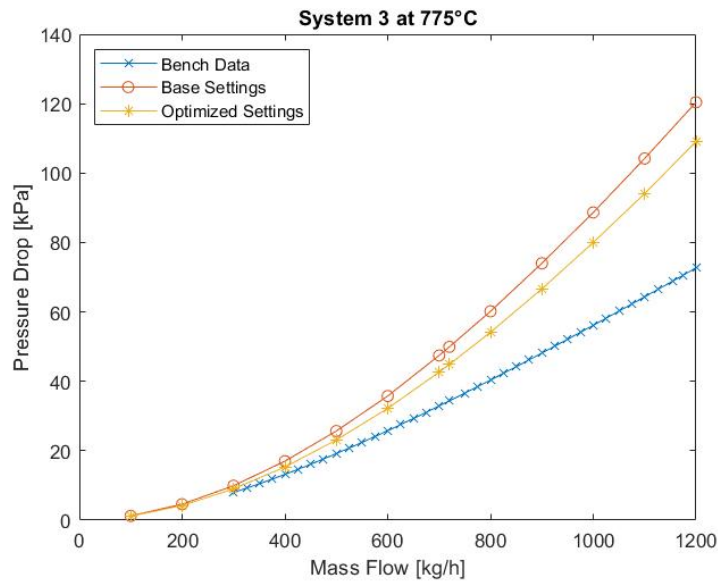


Figure 4.24: Pressure Drop of System 3 at 775°C

From these plots, it can clearly be observed that the results on using the optimized settings over-predict the back pressure lesser than when the base settings are used. From this, it can be concluded that including all the smaller geometrical details for this geometry is beneficial for 1D Simulations.

4.4.2 3D Simulations

Figures 4.25 and 4.26 show the velocity profiles inside the rear muffler of System 3 at 20°C and a flow rate of 720kg/h, using the base and optimized settings respectively.

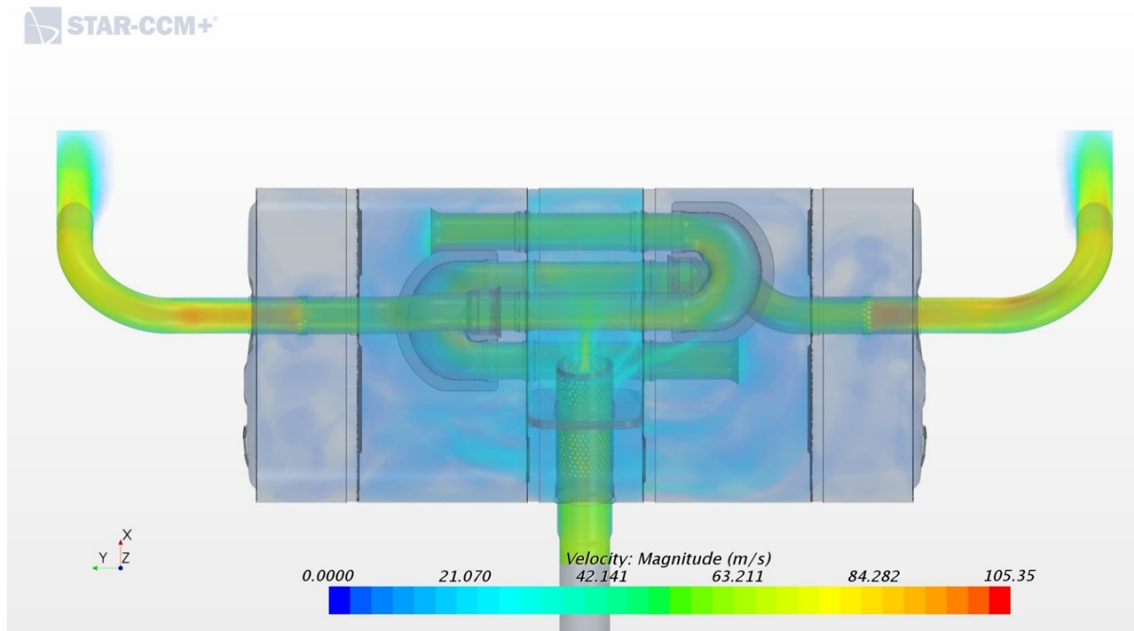


Figure 4.25: Rear Muffler of System 3 at 20°C, Base Settings

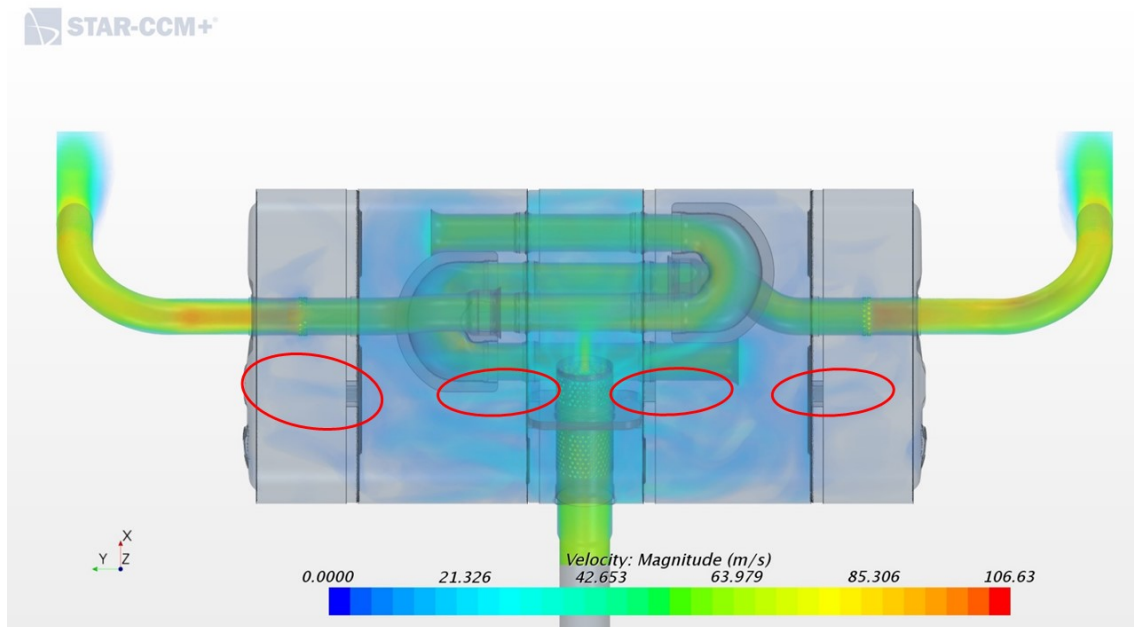


Figure 4.26: Rear Muffler of System 3 at 20°C, Optimized Settings

Figures 4.27 and 4.28 show the velocity profiles inside the rear muffler of System 3 at 775°C and a flow rate of 720kg/h, using the base and optimized settings respectively.

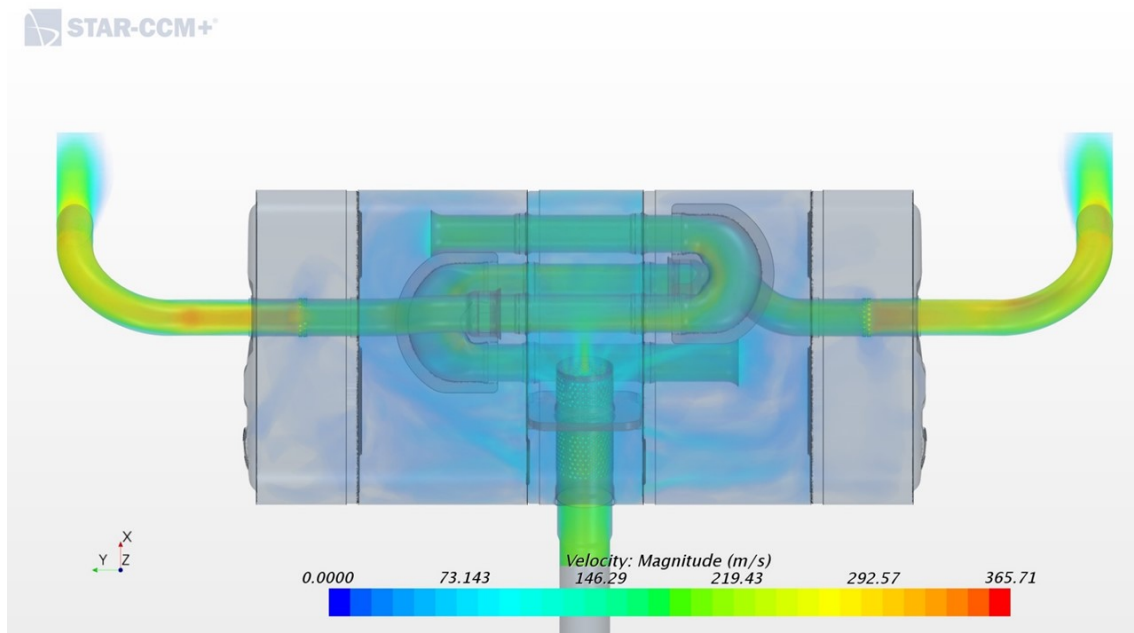


Figure 4.27: Rear Muffler of System 3 at 775°C, Base Settings

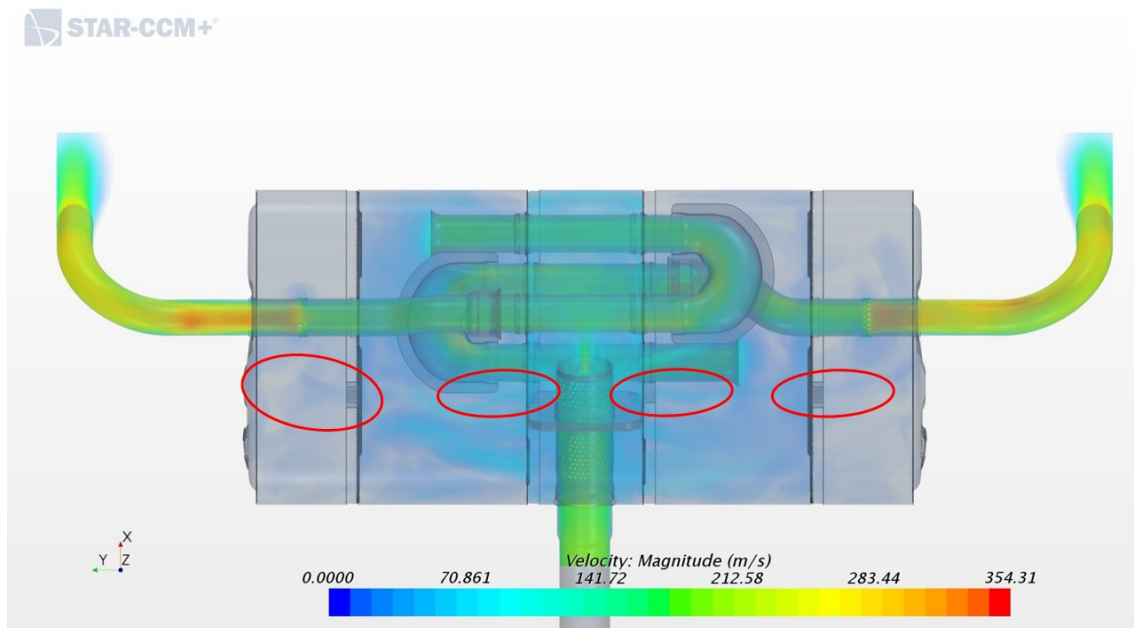


Figure 4.28: Rear Muffler of System 3 at 775°C, Optimized Settings

From these velocity profiles, we can see that there is a significant difference between the profiles with and without optimization. Clear differences in flow paths can be observed in the encircled regions, though they are not as pronounced as with System 2. These differences can be attributed to the fact that there are no separated volumes in the rear muffler, as all the baffles have perforations. However, unlike System 1, there is a high pressure differential between various parts in the rear muffler, and thus when the leaks are included, a significant change in flow pattern is observed.

4.4.3 Summary

This section summarizes all the results in a tabular form to get an overview of the pressure drop over each component in System 3, and also the entire exhaust system.

4.4.3.1 20°C

The results for System 3 at 20°C using base and optimized settings are shown in Tables 4.13 and 4.14 respectively.

Table 4.13: Base results for System 3 at 720kg/h and 20°C

	Bench	1D	3D
Front Muffler	2.80 kPa	0.15 kPa	0.30 kPa
Connection Pipe	1.57 kPa	2.55 kPa	2.50 kPa
Rear Muffler	7.35 kPa	12.06 kPa	9.70 kPa
Overall Static Pressure Drop	11.72 kPa	14.76 kPa	12.50 kPa
% Difference	Ref	+25.9	+6.7

Table 4.14: Optimized results for System 3 at 720kg/h and 20°C

	Bench	1D	3D
Front Muffler	2.80 kPa	0.16 kPa	0.30 kPa
Connection Pipe	1.57 kPa	2.68 kPa	2.73 kPa
Rear Muffler	7.35 kPa	10.41 kPa	9.71 kPa
Overall Static Pressure Drop	11.72 kPa	13.25 kPa	12.74 kPa
% Difference	Ref	+13.0	+8.7

4.4.3.2 775°C

The results for System 3 at 775°C using base and optimized settings are shown in Tables 4.15 and 4.16 respectively.

Table 4.15: Base results for System 3 at 720kg/h and 775°C

	Bench	1D	3D
Front Muffler	6.50 kPa	0.44 kPa	0.89 kPa
Connection Pipe	4.75 kPa	7.51 kPa	7.71 kPa
Rear Muffler	23.11 kPa	41.98 kPa	33.00 kPa
Overall Static Pressure Drop	34.36 kPa	49.93 kPa	41.60 kPa
% Difference	Ref	+45.3	+17.4

Table 4.16: Optimized results for System 3 at 720kg/h and 775°C

	Bench	1D	3D
Front Muffler	6.50 kPa	0.46 kPa	0.88 kPa
Connection Pipe	4.75 kPa	7.84 kPa	8.42 kPa
Rear Muffler	23.11 kPa	36.54 kPa	33.15 kPa
Overall Static Pressure Drop	34.36 kPa	44.84 kPa	42.45 kPa
% Difference	Ref	+30.5	+23.5

From these result summaries, it can be observed that the results for the pressure drops over the individual components are significantly different as compared to the bench results as they are computed at different pipe diameters. However, the overall static pressure drop is still correct as the correct pipe diameter is used during the simulations. Hence, the result summary only reflects the overall pressure drop, and the observations from those results are:

- 1D simulations improve greatly when the optimized settings are used
- 3D simulations are close to bench results, but the results get worse when the optimized settings are used
- For this geometry, the results in the 3D simulations at 20°C correspond very well to the bench values, but the results at 775°C start to deviate quite a bit. This may be attributed to the fact that all the complex features cannot be captured in the simulations.

4.5 Hot Flow Test - Model Calibration

Hot flow tests were performed as described in Section 3.1.2, and the 1D simulations were then carried out as described in Section 3.2.2.4.

The temperature measurements from various points along the exhaust system is shown in Table 4.17. The conditions for these results are:

- Mass Flow Rate: **720 kg/h**
- Ambient Temperature: **20°C**
- Average temperature through the exhaust system: **775°C**

Table 4.17: Temperature measurements for hot flow bench

Measurement Point	Temperature
Before Front Muffler	851.76 °C
After Front Muffler	842.49 °C
Before Rear Muffler	793.06 °C
Left Tailpipe	725.56 °C
Right Tailpipe	705.28 °C

Figure 4.29 shows the pressure drop of System 2 for an average temperature of 775°C, with the exhaust valve open, and at various mass flow rates.

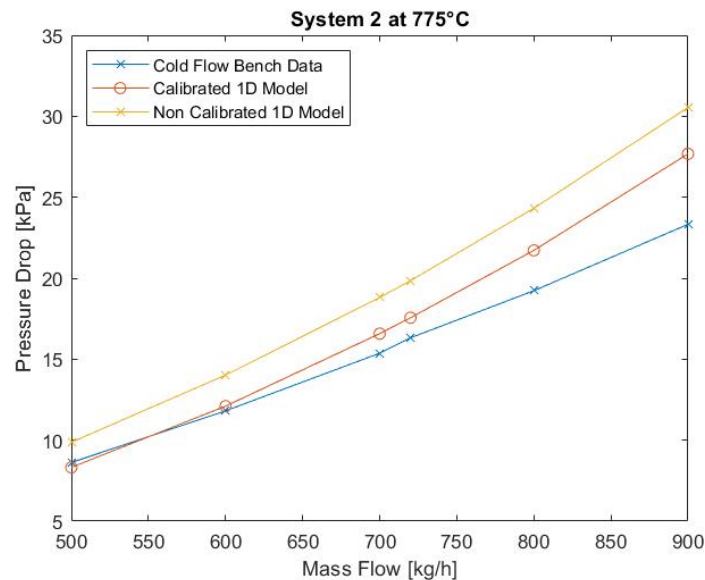


Figure 4.29: Pressure Drop of System 2 at an average of 775°C, Valve Open

From Figure 4.29, it can clearly be seen that the 1D model perform very well and predicts the pressure drop to a much closer value to the actual bench results. This shows the importance of calibration of models, as the simulations become more reliable.

5

Conclusion

Based on the evaluation carried out during the thesis, the following conclusions can be drawn:

- **Pre-processing**
 - Spending more time initially to clean up the geometry gives much better results
 - Baffle leaks make a lot of difference to the overall results
- **Physical Tests**
 - Necessary during the prototyping stages
 - Aid in gathering data to improve models
 - Tests results can be used to compare results from suppliers
- **1D Simulations**
 - Good for initial stages of concept selection to get a ballpark value
 - Multiple cases can be evaluated very quickly
 - Further calibration will yield quick and reliable results
- **3D Simulations**
 - Good once the geometry reaches a near final stage
 - Much better estimation of the pressure drop

Overall, the results from the thesis work call for future modifications to the methods already in place.

6

Future Work

The work that can be done in the future based on the study carried out in the thesis includes:

- More physical tests in the Hot Flow rig for obtaining calibration data
- Temperature measurements of both walls and gases need to be considered during experiments
- Calibration of more existing models to get an idea of what setting to use in 1D models
- Establishing a baseline setting that can work for multiple systems with reasonable margin of error
- Verification of baseline settings by using them on new concepts
- Including thermal losses into 3D model
- Running simulations with the full exhaust system and including an engine model

Bibliography

- [1] Helmholtz Resonance [Internet]. Wikipedia Article 2018. Available from: https://en.wikipedia.org/wiki/Helmholtz_resonance
- [2] Helmholtz Resonators - Basic Analytic Devices [Internet]. Article 2018. Available from: https://people.seas.harvard.edu/~jones/scie129/nu_lectures/lecture3%20/ho_helmholtz/ho_helmholtz.html
- [3] Andersson J, *Computational Fluids Dynamics - The Basics with Applications*, McGraw-Hill; 1995.
- [4] 6. Air - Dynamic and Kinematic Viscosity [Internet]. Engineeringtoolbox.com. 2018. Available from: https://www.engineeringtoolbox.com/air-absolute-kinematic-viscosity-d_601.html
- [5] Internal Test Report, (2018), *LFR_18-03-05_001_Exhaust*, Volvo Car Corporation
- [6] Gamma Technologies Official Website. <https://www.gtisoft.com/gt-suite/productivity-tools/gem3d-preprocessor/>
- [7] Gamma Technologies Official Website. <https://www.gtisoft.com/gt-suite/productivity-tools/gt-ise-integrated-simulation-environment/>
- [8] Gamma Technologies, (2018), *Flow Theory Manual*
- [9] BETA CAE Systemn Official Website. <https://www.beta-cae.com/ansa.htm>
- [10] Siemens Product Lifecycle Management Software Inc. (2017) STAR-CCM+ User Guide Version. 12.06
- [11] Hägglund J, (2017), *CFD Simulation of Cold End System*, Presentation presented at Volvo Cars

A

Appendix

A.1 Plots for System 1

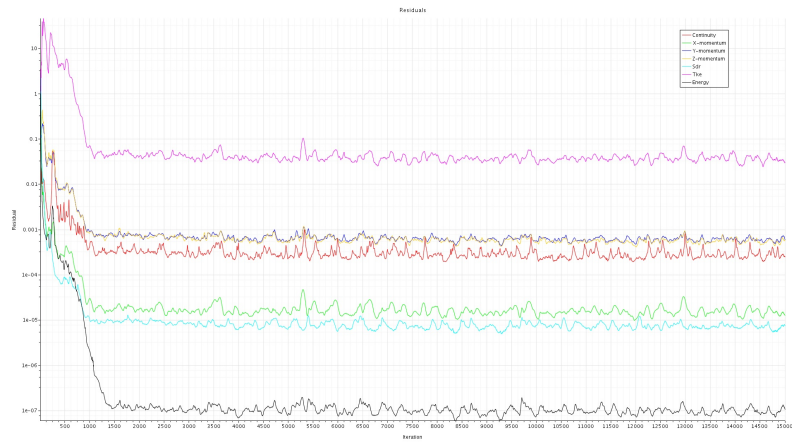


Figure A.1: Residuals for System 1 at 20°C, Base Settings

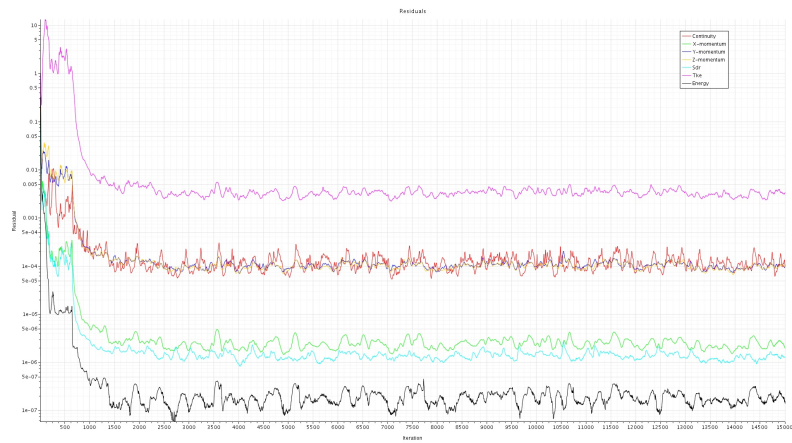


Figure A.2: Residuals for System 1 at 775°C, Base Settings



Figure A.3: Residuals for System 1 at 20°C, Optimized Settings

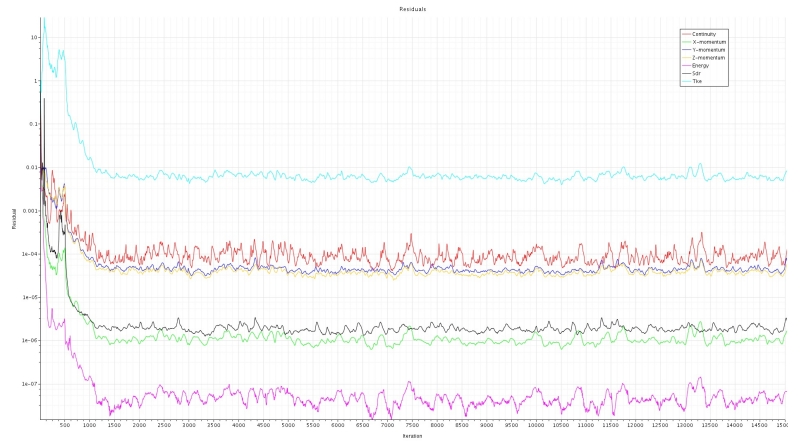


Figure A.4: Residuals for System 1 at 775°C, Optimized Settings

A.2 Plots for System 2



Figure A.5: Residuals for System 2 at 20°C Valve Open, Base Settings



Figure A.6: Residuals for System 2 at 775°C Valve Open, Base Settings

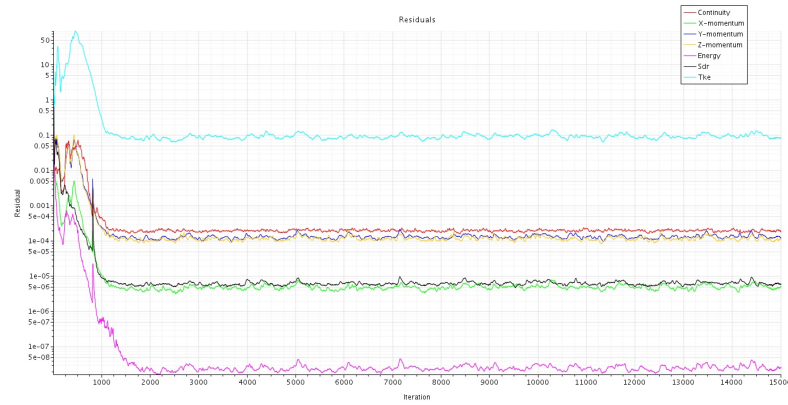


Figure A.7: Residuals for System 2 at 20°C Valve Closed, Base Settings

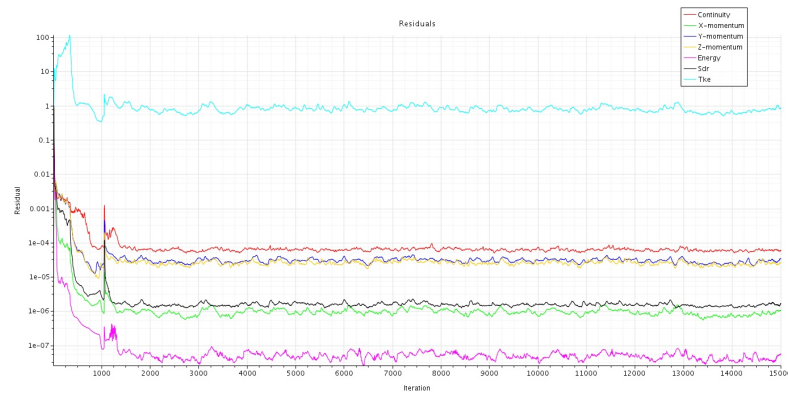


Figure A.8: Residuals for System 2 at 775°C Valve Closed, Base Settings

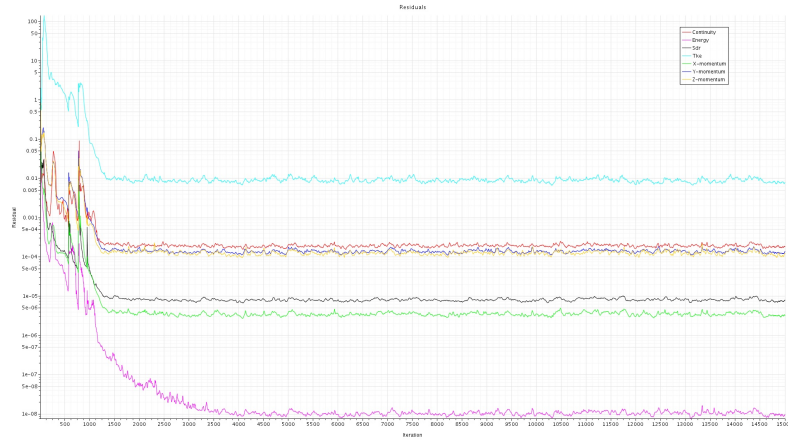


Figure A.9: Residuals for System 2 at 20°C Valve Open, optimized Settings

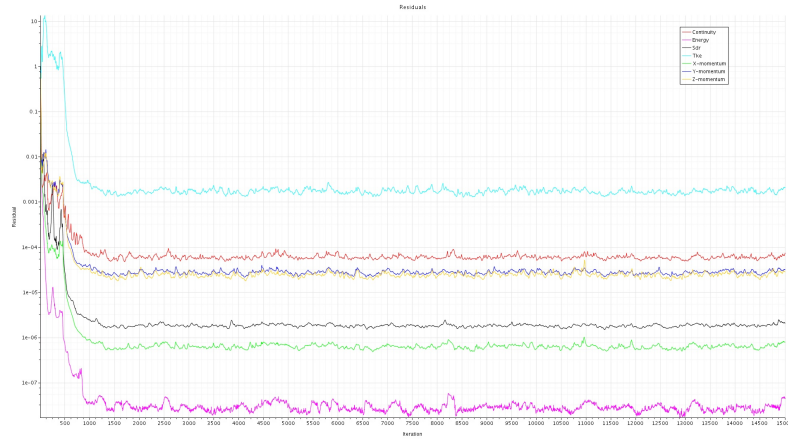


Figure A.10: Residuals for System 2 at 775°C Valve Open, Optimized Settings

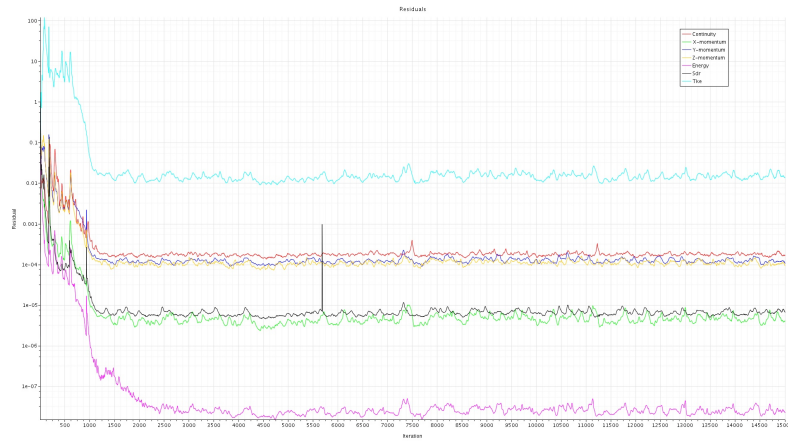


Figure A.11: Residuals for System 2 at 20°C Valve Closed, Optimized Settings

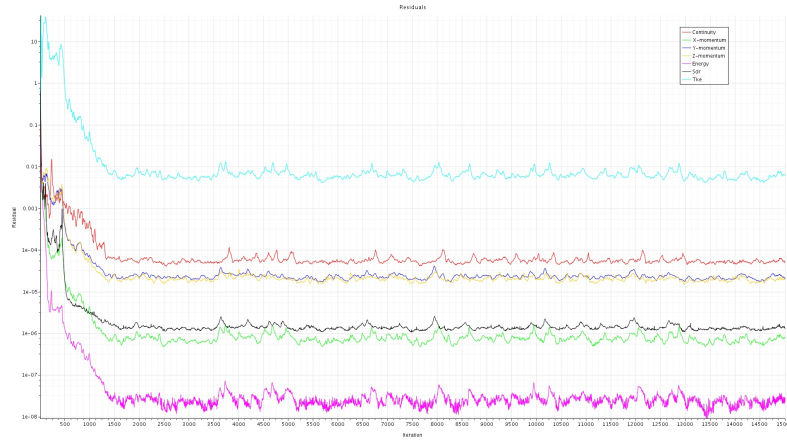


Figure A.12: Residuals for System 2 at 775°C Valve Closed, Optimized Settings

A.3 Plots for System 3



Figure A.13: Residuals for System 3 at 20°C, Base Settings

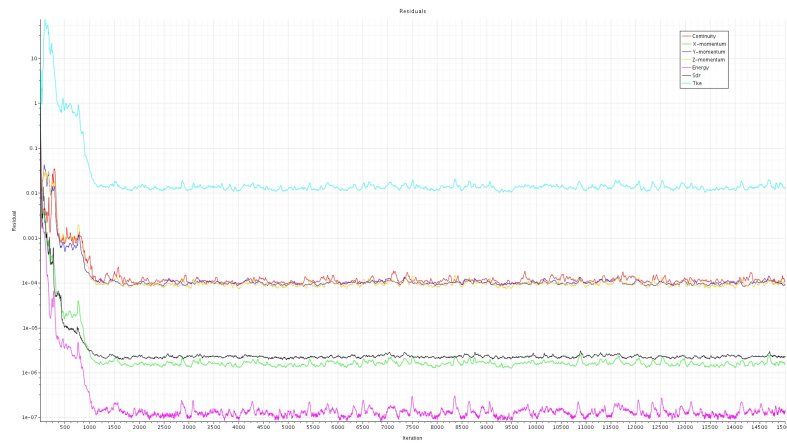


Figure A.14: Residuals for System 3 at 775°C, Base Settings

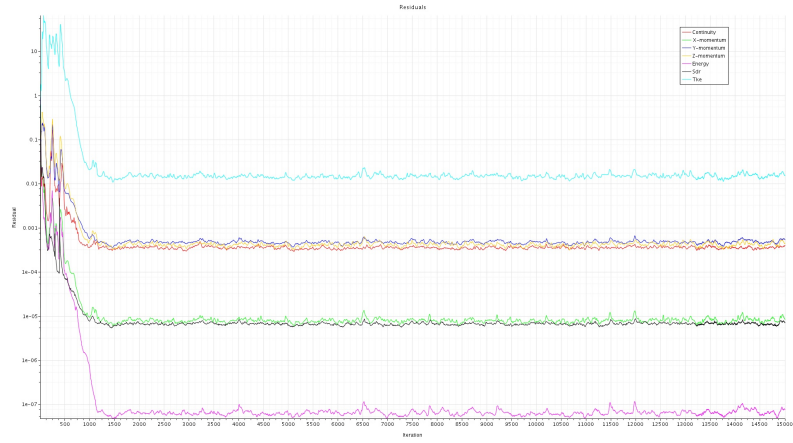


Figure A.15: Residuals for System 3 at 20°C, Optimized Settings

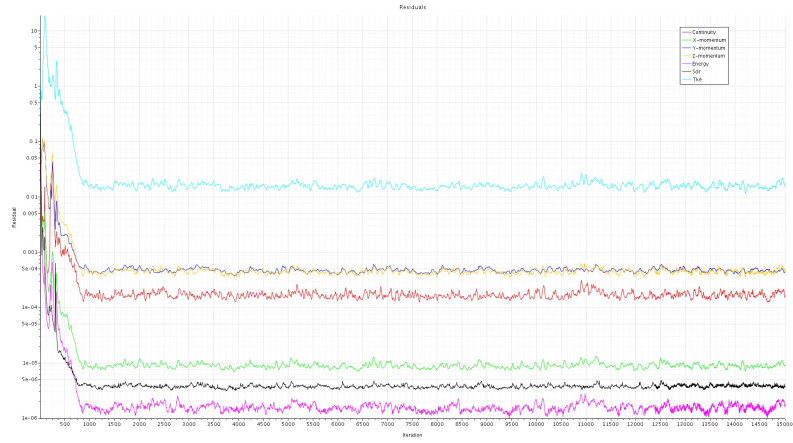


Figure A.16: Residuals for System 3 at 775°C, Optimized Settings