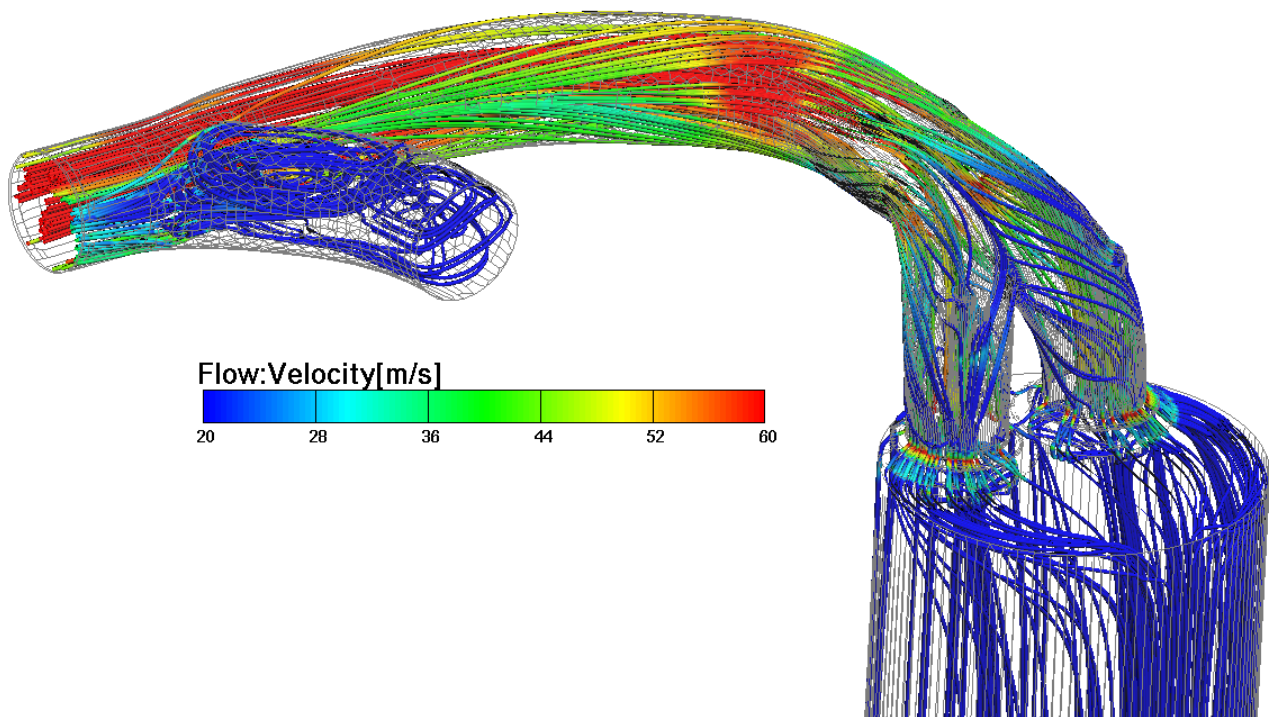




CHALMERS



Investigation of parametric exhaust port of a heavy-duty diesel engine

BASHAR FATAHALLA

TECHNICAL REPORT IN APPLIED MECHANICS

Investigation of parametric exhaust port of a heavy-duty diesel engine

BASHAR FATAHALLA

Department of Applied Mechanics

Division of Fluid Mechanics

CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2016

Investigation of parametric exhaust port of a heavy-duty diesel engine
BASHAR FATAHALLA

© BASHAR FATAHALLA, 2016

Technical report 2016:09
ISSN 1652-8549
Department of Applied Mechanics
Division of Fluid Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:

Front view of the exhaust port and manifolds at cylinder position 6 in the engine. Streamlines are velocity. A pressure boundary condition defined as flow rig test at Scania. The mesh on the computational domain boundaries is made transparent.

Chalmers Reproservice
Göteborg, Sweden 2016

Investigation of parametric exhaust port of a heavy-duty diesel engine
Technical report in Applied Mechanics
BASHAR FATAHALLA
Department of Applied Mechanics
Division of Fluid Mechanics
Chalmers University of Technology

ABSTRACT

Exhaust port developed today at Scania is based on a steady state approach in both flow rig test and simulation. A new method is required to investigate the transient effects to simulate real engine like conditions for engine exhaust port.

A parametric model of an exhaust port was created by using CATIA V5 and various geometries of the port were generated using the Latin hypercube sampling approach to perform a design of experiment (DOE) study. Steady-state and transient simulations, carried out using AVL Fire, are used as two different approaches to investigate the flow characteristics between different geometries. In the steady state simulation, three different valve lifts are investigated as well as two type of boundary conditions; one from steady-state flow rig test and one similar to real engine test. Another aim of the investigating the two approaches is to develop a methodology for further optimization.

Additionally, the impact on exhaust port design for different cylinder positions are investigated. Design of experiment study for each cylinder position showed a unique optimal port design. The exhaust port having two symmetric channels was not performing compared to the optimal port where the channels were asymmetric. The transient simulation showed complex flow phenomena at low valve lift compare to the high valve lifts. Additionally, transient state approach was validated against the steady state simulation. In this validation, the steady state simulation, tested for different pressure boundary conditions, showed only small discrepancies in discharged coefficient at both low and high valve lifts.

Moreover, the optimal geometry in transient state was not same as the optimal geometry for steady state at low valve lift. Instead, this optimal geometry from transient simulation was same as the one in steady state at high valve lifts. However, transient simulation was very computationally expensive compared to steady state. Longer simulation time and is not suitable for larger design of experiments investigations.

Keywords: Exhaust Port, Diesel Engine, Parameter Model, Steady state, transient state, Simulation method, Optimization, Surface response, Latin Hypercube Design, Computational Fluid Dynamics (CFD), AVL FIRE, Manifolds, Design of Experiment (DOE)

SAMMANFATTNING

Avgasporten utvecklas idag på Scania är baserad på en steady state metod både flödesrigg test och simulering. En ny metod krävs för att undersöka de transienta effekterna för att simulera verkliga motorn liknande förhållanden för motor avgasporten.

En parametrisk modell av en avgasporten har skapats med hjälp av CATIA V5 och olika geometrier av avgasporten genererades med det latinska hypercube urvalsmetoden för att utföra en försöksplanering (DOE) studie. Steady-state och transient simuleringar utföras med hjälp av AVL Fire, används som två olika metoder förundersöka flödesegenskaper mellan olika geometrier. I det steady state simulering, tre olika ventillyft undersöks liksom två typer av randvillkor , en från steady state flödesrigg test och ett som liknar verklig motortest. Ett annat syfte med utrednings de två metoderna är att utveckla en metod för ytterligare optimering

Dessutom är effekten på avgasporten design för olika cylinderpositioner undersökts. Försöksplanering studie för varje cylinder position visade en unik optimal port design. Avgasporten som har två symmetriska kanaler inte utför jämfört med den optimala exhaust port där kanalerna var asymmetriska. Den transient simulering visade komplexa flödesfenomen vid låg ventillyft jämfört med de höga ventillyft. Dessutom var transient state metod valideras mot steady state simulering. I denna validering, den steady state simulering, testades med avseende på olika tryckrandvillkor, uppvisade endast små skillnader i urladdat koefficient vid både låga och höga lyft ventil.

Dessutom optimala geometri transient state var inte densamma som den optimala geometrin för steady state på låglyftande ventil. Istället denna optimala geometri från transient simulering var densamma som den i stabilt tillstånd vid hög ventil lyfter. Men transient simulering var mycket beräknings dyrt jämfört med steady state. Längre tid simulering och lämpar sig inte för större försöks utredningar.

ACKNOWLEDGEMENTS

I would like to thank my direct supervisor Kim Petersson at Scania for his guidance, help and assistance through the project and answer my questions during the project time to understand the fundamental of the problem, combustion simulation and trust me from the first time and giving me opportunity of pursuing on this interesting and challenging thesis work.

Special thanks to my supervisor Full Professor of Computational Fluid Dynamics Siniša Krajnović at Chalmers university for his excellent supervision.

I would also like to thank everyone at Scania and especially the NMTD group at Scania for their help, encouragement, guidance, cooperation and for providing a friendly welcoming work environment.

Last but not least, I want to thank my parents, brother and his family, which always have been support me during my time at the thesis work. The support has helped me a lot and it was always a wonderful and relaxing time, when I spoke to them.

Göteborg March 2016

Bashar Fatahalla

NOMENCLATURE

Abbreviations and Acronyms

NMTD	The fluid and Combustion Simulation group
CFD	Computational Fluid Dynamics
DOE	Design Of Experiments
CAD	Computer Aided Design
FEM	Finite Element Method
AVL	Anstalt für Verbrennungskraftmaschinen List
CAE	Computer Aided Engineering
CPU	Central Processing Unit
RANS	Reynolds-Averaged Navier-Stokes
RSM	Response Surface Methodology
NO _x	Nitrogen oxides ($NO + NO_2$)
BDC	Bottom Dead Center
TDC	Top Dead Center
CAD	Crank Angle Degree
PANS	Partially Averaged Navier-Stokes
DNS	Direct Numerical Simulation
STL	STereoLithography
CATIA	Computer Aided Three dimensional Interactive Application
MATLAB	Matrix laboratory

Symbols and Mathematical Notation

ρ	Density
τ	Shear Stress
Re	The Reynolds number
μ	Dynamic Viscosity
U	Character velocity scale
L	Character length scale
ε_u	Dissipation
k_u	Kinetic energy
v_u	Turbulent viscosity
$C_L, C_{f1}, C_{f2}, C_1, C_2, C_\eta, C_\mu, C_\tau$	Constants for the $\xi - f$ model
f_k	Unresolved-to-total ratio of turbulent kinetic energy
f_ε	Unresolved-to-total ratio of dissipation
k	Turbulent kinetic energy
P_u	Production Term
U	Filtered velocity in PANS
T_u	Time scale for the PANS method
L_u	Length scale for the PANS method
$\sigma_{ku,eu,\xi u}$	Coefficient for the PANS model
Δ	Grid cell dimension
ξ	Velocity scale ratio
C_d	discharge coefficient
A_o	Outlet area of the channel
Δp	The pressure difference
τ	Turbulent time scale
$\tau(V_i, V_j)$	Residual stress tensor for the PANS method

Abstract	i
Sammanfattning	ii
Acknowledgements	iii
Nomenclature	v
1 Introduction	1
1.1 Background	1
1.2 Purpose & Goals	2
1.3 Limitations	2
2 Theory	4
2.1 Diesel engine	4
2.1.1 Piston movement	5
2.2 Flow through the exhaust port	6
2.2.1 Valve and exhaust port geometry	6
2.2.2 Governing equations of Exhaust flow field	7
2.2.3 Total pressure loss and the Bernoulli's equation	7
2.2.4 Friction losses	8
2.2.5 Total pressure loss due to obstructions	8
2.2.6 Compressibility of exhaust flow	9
2.3 Usage of exhaust gas energy	9
2.4 Turbulence	10
2.4.1 Turbulence modelling	10
2.5 Discharge coefficient theory	13
2.6 Parametric modelling	14
2.7 Design of Experiment	15
2.7.1 Full Factorial Designs	17
2.7.2 Latin Hypercube Design	17
2.7.3 Random sampling	18
2.8 Response surface methodology	18
2.9 Mesh type	19
2.9.1 Hexa mesh	19
2.9.2 Hybrid mesh	19
3 Method & Implementation	20
3.1 Pre-Setup Geometry	20
3.1.1 Design of Parameter model	20
3.1.2 Generating of DOE	21
3.2 CFD - simulation	22
3.2.1 Mesh independence	22
3.2.2 Steady state simulation	22
3.2.3 Transient-state simulation	25
3.3 Post-processing	27
3.3.1 Surface response and Optimization	27
4 Results & Discussion	28

4.1	Parameter modelling	28
4.2	CFD - simulation	29
4.2.1	Mesh independence	29
4.2.2	Steady state simulation	31
4.2.3	Transient state simulation	33
4.3	Optimization and surface response	35
4.4	Computational time	37
4.5	Source of error	37
4.5.1	Geometry	38
4.5.2	Arbitrary contact	38
4.5.3	Manifolds	38
4.5.4	Mesh type	38
4.5.5	Turbulence modeling	38
5	Conclusions & future work	39
5.1	Conclusions	39
5.2	Future work	39
	References	41
	Appendix A Requirement structure matrix	42
	Appendix B Parameter limitation structure matrix	43
	Appendix C Concepts matrix	44
	Appendix D Range of parameter	45
	Appendix E Computational time	47

1 Introduction

This thesis work has been carried out at Scania CV AB in Södertälje, Sweden. Scania has founded in 1891 in Malmö and is a global automotive industry manufacturer of commercial vehicles – specifically heavy trucks and buses. It also manufactures diesel engines for motive power of heavy vehicles, marine, and general industrial applications [1]. Today, Scania has sales and service organisation in more than 100 countries and has approximately 42000 employees. Of these, some 18400 work with sales and services. About 12400 people work at production units in seven countries and regional product centres in six emerging markets. Scania head office is located in Södertälje where a total of 5000 people work with sales as well as administrative and research and development operations with 3500 employs [2].

The Fluid and Combustion Simulation group NMTD has started this thesis work. The group uses Computer Fluid Dynamics - CFD as a tool to simulate and analyze the fluid phenomena for different engine component and provides detailed information to other research and development group at Scania. The group consists of 10 employs and 2 consultants with a wide range experience and one of the largest CFD group in Scandinavia focusing on engine development.

The thesis work is a continuation of thesis work performed during the spring at NMTD, where one focused on creating parametric model for inlet and other a parametric model for turbine volute [3, 14]. Both thesis works create a methodology for building a parametric model. This thesis work uses a part of the previous work as a foundation on which assumption and theories as made. The thesis work focuses more in the exhaust port to create a parametric model, generate a design of experiments matrix (DOE) , optimization of the exhaust port and create a method for steady and transient state simulation for the port to be easier for CFD calculation engineers to use it in the department.

1.1 Background

Scania CV AB is one of the company that are looking to have better designs of the internal combustion engine with respect to lower pollutant emissions and lower fuel consumption. This made the product life cycle shorter, therefore time to market has been more important for all parties related to the product development process. To increase focuses on shortened development phase of products, several companies have invest to create efficient product development process. Lead time, is not only important parameter; quality and cost are examples of other parameters that can be influences by the product development process [18]. By using parametric CAD - model which can easily and quickly be geometrically modified, development time and cost can be reduced; this will produced a better quality product. The parameter model can use of FEM, CFD engineer in their work and can change the models themselves and perform simulations or various geometric designs. This process builds a good process of connection between the calculation engineers to develop the model. On way to utilize simulation driven design is through Design of Experiment (DOE).

The design of experiment step started from the simulation group at Scania. Many of them were interested in doing DOE to visualize the design space for good concept, close the goal and fine - tune the models to discover geometries which give a good achievement. A group NMTD is one of the groups who started to search and develop the process of the parametric model. The idea is not only to save time in design, is also to build a communication between

the designer and the simulation engineer to communicate results, thus the process became very manual and not at all as efficient as originally intended.

The process is continually to Toyota lead system, Japanese uses the word Jidoka to describe how quality is achieved in production [12]. Jidoka consist of several different steps: automatic stops, error proofing, and in-station quality control. In collaboration with NMTD group at Scania, the aim of the thesis is to create a parametric model for the exhaust port to be able; the parameter model should be robust. Simulate the model by using AVL FIRE a computational fluid dynamic program to analysis the fluid inside the geometry. Study time, efficiency and quality of which type of simulation method is better to use for the exhaust port.

1.2 Purpose & Goals

The purpose of this thesis is to create a robust parametric model for the exhaust port in CATIA V5, this model will help to generate different design points by using a design of experiment approach. The model will be sufficient to represent many points in the design volume, study every parameter and correlations between the parameters, study the fluid movement through a exhaust port. Additionally, a method for steady state, study the effect of various cylinder position, manifolds exhaust, different Valves lift opening and pressure in the optimal exhaust port. Create a method to run a DOE for transient state. Studying which of the two methods are more accuracy and efficiency to use it. The goals of this thesis are listed under:

1. Create a parametric model for exhaust port in CATIA V5.
2. Develop and provide a clear methodology for steady state and transient simulations. Proof which method is the best of these two, to use it for the simulation of the exhaust port.
3. Validate the method on selected optimized geometries.

1.3 Limitations

The boundaries of the project are defined under:

1. The time of the project is limited to the twenty weeks. It starts 16 September 2015, week 38 and finish until 4 March 2016, week 9. The Christmas holiday takes place during week 51 and 52.
2. CATIA V5 program will be use as a CAD software to design the parametric model, because it is a primarily program at Scania. These parameters will not be include in this study: valves, valves seat, combustion chamber, inlet and exhaust diameter.
3. The engine consist of sex cylinder, It will only study the exhaust in cylinder number 6, 5 and 4, because of symmetry with cylinder number 1, 2 and 3.
4. The mass flow rate and the Discharge coefficient C_d will be the mean parameter measure when evaluate the exhaust port efficiency for steady state simulation. While the drop pressure for transient state simulation. No swirl studying will be done during the project.
5. Two interfaces will be considered as a limit for the exhaust port. The inlet interface will be the combustion chamber and outlet interface will be the outlet of exhaust port from cylinder head.

6. The computational fluid dynamics CFD analysis will be performed exclusively in the software AVL FIRE version 14.1.
7. The system assumed adiabatic, which means that there is no heat transfer from water jacket to the exhaust port.
8. The project concerns only numerical studies and parameter study of the exhaust port hence, no prototype will be manufactured.
9. The design of experiments (DOE) matrices will be generated by using an Excel file. This file developed at Scania.
10. The optimization process is performed by using a MATLAB code, used to study every parameters correlation, individual and surface response.

2 Theory

In this chapter, an overview of the theory related to this thesis is described. The section subtitle include some basic of Diesel engines, parameter model, design of experiments, surface response and mesh type. Incompressible flow as described by the Navier-Stokes equation, the PANS equation using $\zeta - f$ turbulence model are introduced, Computational Fluid Dynamic CFD, Design of Experiments (DOE), Optimization. This chapter only presents the equations solved in this numerical study and does not cover the solving methods or the derivations of these equations.

2.1 Diesel engine

The diesel engine is invented and patented by Rudolf Diesel in 1892 [19], 16 years after the Otto engine was invented by Nikolaus Otto which is used in today's petrol cars. The difference between the diesel and Otto engine, is a spark ignite a premixed mixture of fuel and air in Otto while in Diesel engine, the fuel is directly injected into the combustion chamber is started by the high temperature which a gas achieves when the greatly compressed.

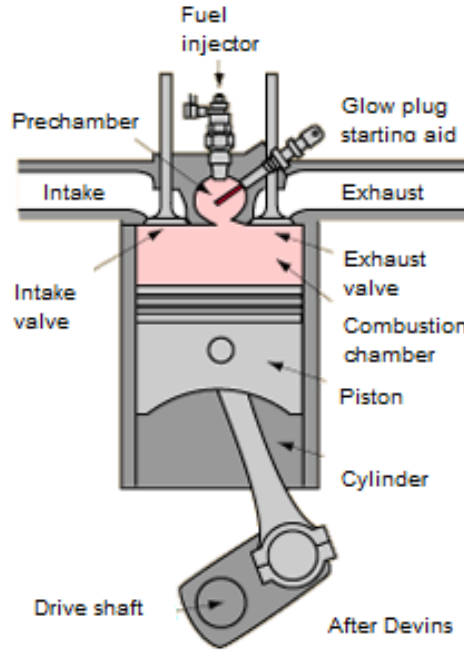


Figure 2.1.1: *Diesel engine* [16]

The diesel is also known as compression ignition (CI) engine and there are various configuration of Diesel engine. The advantage of diesel is a good efficiency, low fuel consumption and low emission while the disadvantage produced large amount of soot and high combustion temperature which increases the NOx formation. The diesel engine is an internal combustion engine which converts chemical by using a combustion process to mechanical energy is followed by many steps of losses. Assume the total of energy is presents 100 percent, after the combustion process it is only 15 to 30 will be defined for moving the engines crankshaft [20]. The losses are placed in many different factors for instance friction and parasitic losses, coolant and exhaust

gas. The largest energy losses is through the exhaust port. Both of Diesel and Otto engine can be divided into a sequence of separate process: intake, compression, combustion, expansion and exhaust. The cycle of the diesel engine illustrated in Figure 2.1.2a, and the pressure volume diagram in Figure 2.1.2b.

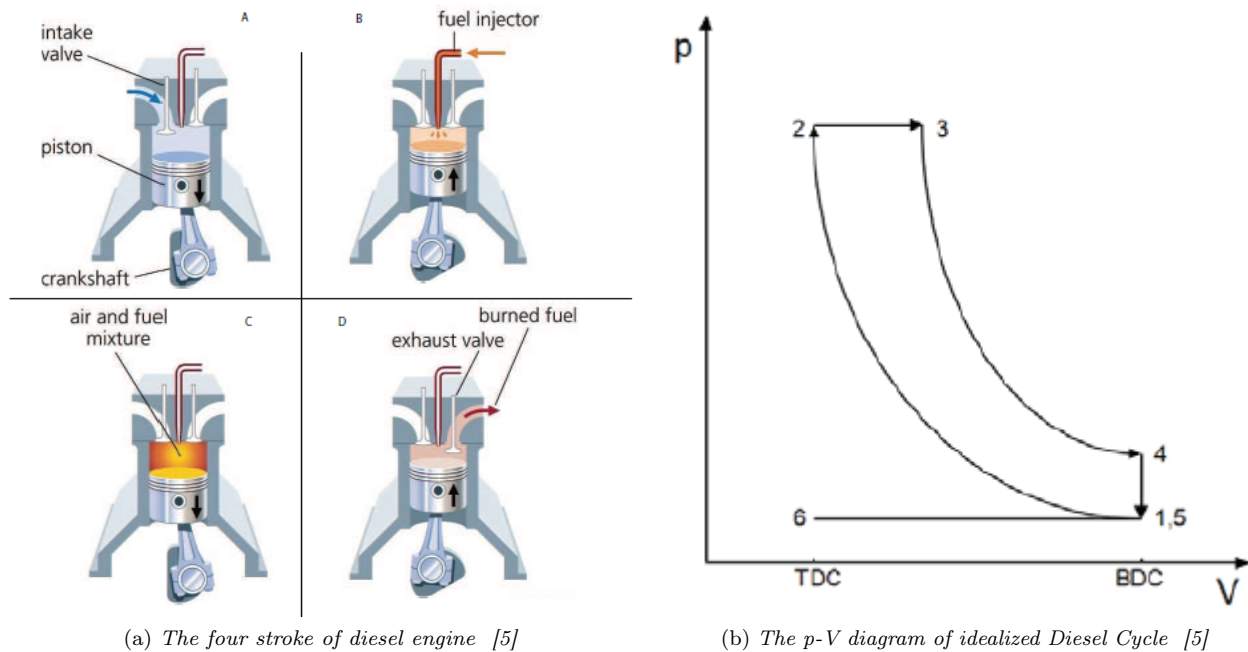


Figure 2.1.2: The four stroke of Diesel engine

The P-V steps listed under:

1. Intake the cold air enters the cylinder through the intake valves and the valves closes when the lowest point of the piston is reached. The air is compressed adiabatically in process 6 to 1.
2. Compression stroke the air by the upward movement of the piston which raises the temperature in process 1 to 2. Inlet and outlet valves are closed.
3. Combustion and expansion, the fuel injected during this step in process 2 to 3 and ignites due to the temperature of the compressed air in process 3 to 4. The piston draws down by the kinetic energy during combustion. The injector is placed at the center of the cylinder head.
4. Exhaust, before reaching the lowest piston position, the exhaust valves opens and the exhaust gas moves out of the combustion chamber while the piston moved up in process 4-5-6.

2.1.1 Piston movement

In Figure 2.1.3, a schematic drawing view of the piston is illustrated. Top dead center (TDC) and bottom dead center (BDC) represents the highest and lowest position of the piston movement. One stroke of the engine cycle is defined when the piston moves between (TDC) and (BDC). A crank angle degree CAD is a unit which used to measure the piston travel to

adjust ignition. The crank angle is zero at the (TDC) position and 180 degree at the (BDC) position. One stroke is represents by 180 CAD of the whole engine cycle 720 CAD, thus four strokes. The (TDC) position is when the air intake the cylinder and between compressed the air, injected the fuel and expansion.

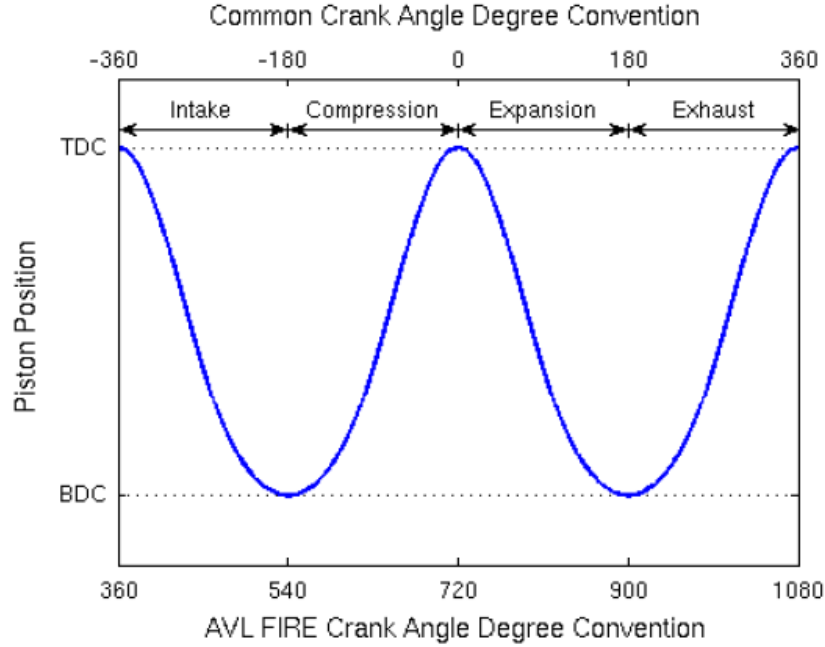


Figure 2.1.3: *Piston movement* [5]

2.2 Flow through the exhaust port

The stroke of exhaust gas is the most important flow step of any of four stroke cycle engine. This section describe the flow characteristics through the exhaust port.

2.2.1 Valve and exhaust port geometry

A valve and exhaust port geometry is illustrated in Figure 2.2.1. The figure shows the geometry design of a heavy duty engines, the difference between the geometry in this thesis and in the figure below, the two channels of exhaust port are symmetric. There are five parts in this geometry: Cylinder, valves, exhausts port and manifolds. There are some simplification consider in this geometry: the piston bowl doesn't include and the intake port is not either considered in this study. It is not considered either the parameter in the valves only valves lift. The manifolds consists of sex input pipes from the engine to the turbo charger. Three of these sex are symmetrical. It used only one side of the manifolds, consists of three inlet from cylinder position 6, 5 and 4. The inlet diameter of the exhaust port and the outlet of the exhaust port are fixed in this case. The manifolds have already a parameter model but in this thesis will use it as a fixed body to study only the exhaust port.

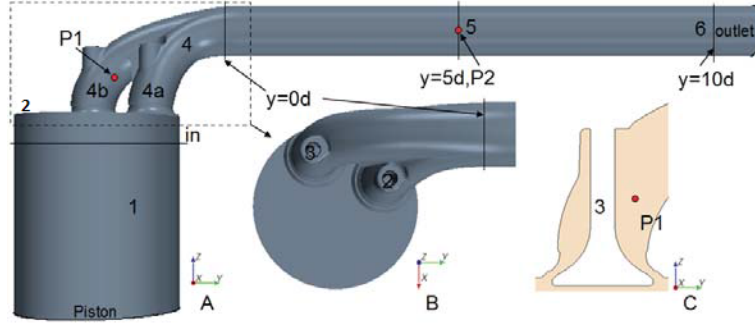


Figure 2.2.1: Details of the exhaust valves and ports geometries. A: The front view; B: The top view; C: Longitudinal section of valve. 1: Cylinder; 2: Chamber; 3: valves; 4: Exhaust port; 5: Exit pipe or manifolds [20]

2.2.2 Governing equations of Exhaust flow field

The compressible flow in the exhaust port has characterized as being turbulent, unsteady, high compressible, non-isothermal, transonic [20]. The flow in the exhaust port and manifolds is very important to calculate the efficiency of the engine. The fluid in the exhaust port follows the three fundamental physical principles:

1. Mass is conserved, in any steady state process the rate of mass enters the system is equal to the rate of mass leaves the system, as equation 2.2.1.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} \quad (2.2.1)$$

2. Momentum is conserved, which means that when two objects collide the total momentum of the objects before the collision is the same as the total momentum of the objects after the collision. The total momentum is summation of the momentum of object 1 plus the momentum of object 2. Applying Newton second law $F = ma$ for fluid motion, as equation 2.2.2.

$$\frac{(\rho u_i)}{\partial x_j} + \frac{\partial(u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} \quad (2.2.2)$$

3. Energy is conserved, energy can be transformed from one form to another without losing in energy, as equation 2.2.3.

$$\frac{\partial}{\partial t}(\rho(e + \frac{1}{2}u_i u_i)) + \frac{\partial}{\partial x_j}(\rho u_j(h + \frac{1}{2}u_i u_i)) = \frac{\partial}{\partial x_j}(u_i \sigma_{ij}) - \frac{\partial q_j}{\partial x_j} \quad (2.2.3)$$

$$p = \rho RT \quad (2.2.4)$$

2.2.3 Total pressure loss and the Bernoulli's equation

The Bernoulli equation for an ideal fluid of a non-conducting fluid, an increase in the speed of the fluid occurs at the same time with a decrease in pressure or a decrease in the fluid's

potential energy. The principle of Bernoulli equation is valid of the incompressible flow and more advanced form my applied to compressible form at higher Mach numbers, which represent the ration of flow velocity past a boundary to the local speed of sound. The total pressure loss is normally used for judging the total energy loss in the system. The losses in energy or total pressure can be divided into two reasons, firstly the friction losses and secondly energy losses due to obstructions.

2.2.4 Friction losses

The friction always exists between the fluid and the pipes wall when the fluid moves in an pipe system. The friction losses due to wall shear stress. In experiments, the wall shear stress is very hard to measure it directly. The pressure drop effect by friction can be calculated in a straight pipe of a certain cross section as in Figure 2.2.2. Where p_1 is the pressure in the first section, p_2 the pressure in the second section, v is the velocity of the fluid, τ_w is the shear stress of the wall.

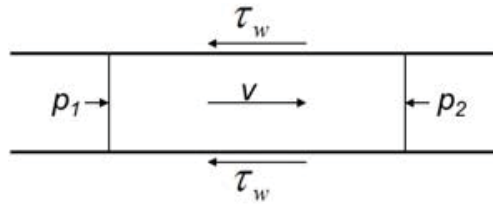


Figure 2.2.2: Flow force in the wall in a pipe which creare shear steress [20]

2.2.5 Total pressure loss due to obstructions

The change of the cross section of the flow can also lead to energy and total pressure losses [20] . If there is no changing in the cross section, the sum of total pressure will be conserved from location 1 to location 2. It can be changing in cross section like expansion, contraction or tube bending.

Contraction : is when the area of the pipe reduces suddenly along the length of the pipe as Figure 2.2.3, the downstream velocity will be higher than the upstream velocity. The flow accelerated at the region of the throat which separated and recirculation the flow. In the turbulent flow, this region can cause loss of energy, which occurs due to turbulence dissipation.

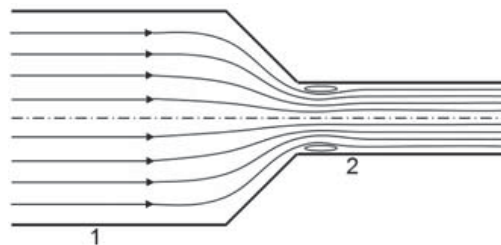


Figure 2.2.3: Flow through a sudden contraction [20]

Expansion : is when the area of the pipe increase suddenly along the length of the pipe as Figure 2.2.4, the downstream velocity will be lower than the upstream velocity. The expansion case is similar to the contraction case. The regions, which separated and recirculated the flow, depend on the cone angle and the diameter ration.

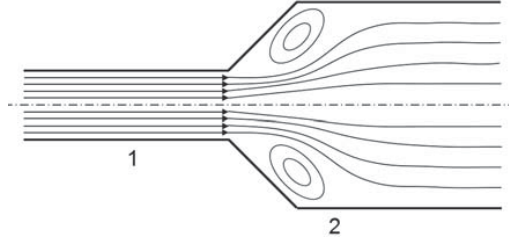


Figure 2.2.4: *Flow through a sudden expansion* [20]

The bended pipe geometry: is more than contraction and expansion, the exhaust pipe includes also bends as Figure 2.2.5. The total pressure in the tube bending can categorize to wall friction loss and total pressure loss due to momentum exchanges in the direction.

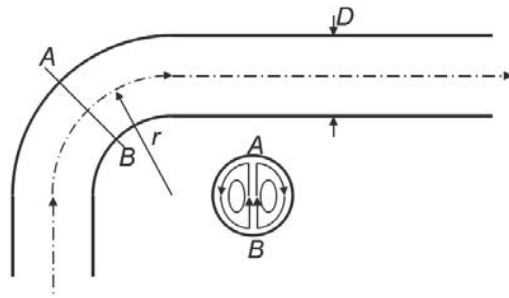


Figure 2.2.5: *Pipe bend with 90°* [20]

2.2.6 Compressibility of exhaust flow

The Bernoulli equation is valid only for the incompressible flows or compressible flow in low Mach number than 0.3. The flow in the exhaust flow can become transonic and supersonic. The compressible flow divided into regimes according to mach number: Subsonic flow with mach number between 0.3 to 0.8, Transonic flow with mach number between 0.8 to 1.2. Sonic flow with mach number 1, Supersonic flow with mach number 1.2 to 5.0, Hypersonic flow with mach number 5.0 to 10.0 and High-Hypersonic flow with mach number larger than 10.

2.3 Usage of exhaust gas energy

The energy of the exhaust gasses can have a large effect on the total effect on the total efficiency of the engine. A good way to use the energy is by using turbocharging. In turbo charging application, a turbine driven by force due to increase an internal combustion engines efficiency and power output by forcing extra air into the combustion chamber Figure 2.3.1.

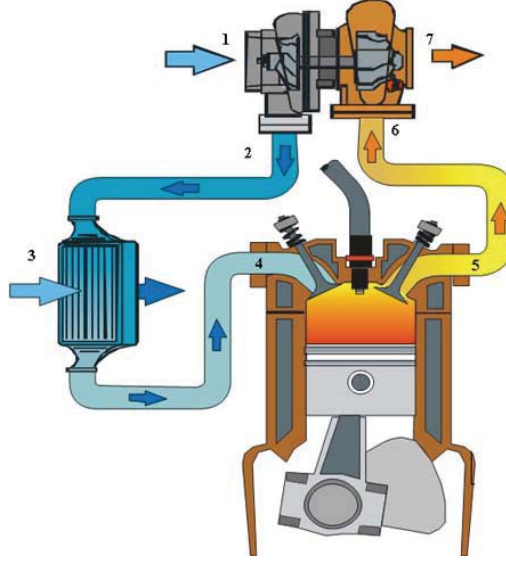


Figure 2.3.1: *Turbocharging system 1:Compressor inlet; 2:Compressor Discharge; 3:Charge air cooler; 4:Intake Valve; 5:Exhaust Valve; 6:Turbine Inlet and 7:Turbine Discharge [20]*

The gas pass through the resistance of the exhaust port. They expand through the turbine to drive the compressor. The compressor function is to compress the incoming air, which increase the pressure and density. The temperature rises during the compressor and affect the density of the air. A charge air cooler used to decrease the temperature into intake and increase the density of the air. The benefit of using turbo charger allow to have a smaller engine with the same engine output.

2.4 Turbulence

The flow in the exhaust port and manifolds are dominated by turbulence. To study the character of the flow, laminar or turbulence, need firstly to study the Reynolds number Re , is a dimensionless quantity [9], defined as equation 2.4.1:

$$Re = \frac{\rho U L}{\mu} \quad (2.4.1)$$

Where μ is the dynamic viscosity of the fluid, U Character velocity scale, L is Character length scale and ρ is the fluid density. The Reynolds number is defined as the ratio of momentum forces to viscous forces.

The flow is laminar in pipe when the Reynolds number is about (2000 – 3000), when the Reynolds number increase to 4000, the flow become fully turbulence. A critical Reynolds number, it is the transition between laminar to the turbulence flow.

2.4.1 Turbulence modelling

PANS is Partially Averaged Navier-Stokes (PANS) , is a method which build a bridge between Direct Numerical Simulation (DNS) and Rynolds-Averged Navier-Stokes (RANS) [7]. The advantage of this method is from resolving all scales of motion to fully modeling the turbulent

scale of motion. PANS involves solving evolution equations for unresolved kinetic energy k_u and unresolved dissipation ε_u . The RANS model used in this project is the $\xi - f$ model presented in Hanjalic [9]. For incompressible flow, the continuity equation 2.4.2 and momentum equation 2.4.3:

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \quad (2.4.2)$$

$$\frac{\partial \bar{v}_i}{\partial t} + \frac{\partial(\bar{v}_i \bar{v}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (v \frac{\partial \bar{v}_i}{\partial x_j} - \tau_{ij}) \quad (2.4.3)$$

The exact or instantaneous field is decomposed into a filtered field and a residual field:

$$V_i = U_i + u_i \quad (2.4.4)$$

Where $\tau(V_i, V_j)$ is the sub-filter stress closure which resulting from partially averaging. In the $\xi - f$ the modeled transport equations for dissipation ε and turbulent kinetic energy k are solved along the transport equation for the velocity scale ratio. The unresolved kinetic energy and dissipation ratio are defined respectively as:

$$f_k = \frac{k_u}{k} \quad (2.4.5)$$

$$f_\varepsilon = \frac{\varepsilon_u}{\varepsilon} \quad (2.4.6)$$

All dissipation is modeled meaning that f_ε and $\varepsilon_u = \varepsilon$, The total turbulent viscosity ν_u is calculated as :

$$\nu_u = C_\mu \xi u \frac{k_u^2}{\varepsilon_u} \quad (2.4.7)$$

The sub-filter stress closure is modeled by using the Boussinesq-type approximation:

$$\tau(V_i, V_j) = -\nu_u \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \frac{2}{3} K_u \delta_{ij} \quad (2.4.8)$$

where S_{ij} is the resolved rate of strain tensor

$$S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (2.4.9)$$

The modeled transport equations for unresolved turbulent kinetic energy k_u and the unresolved dissipation ε_u is:

$$\frac{Dk_u}{Dt} = P_u - \varepsilon_u + \frac{\partial}{\partial x_j} \left[\left(v + \frac{\nu_u}{\sigma_{k_u}} \right) \frac{\partial k_u}{\partial x_j} \right] \quad (2.4.10)$$

$$\frac{D\varepsilon_u}{Dt} = C_{\varepsilon 1} P_u \frac{\varepsilon_u}{k_u} - C_{\varepsilon 2}^* \frac{\varepsilon_u^2}{k_u} + \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_u}{\sigma_{\varepsilon u}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] \quad (2.4.11)$$

Where P_u is the production term:

$$P_u = \tau(V_i, V_j) \frac{\partial U_i}{\partial x_j} \quad (2.4.12)$$

$$P_u = 2v_\mu S_{ij} \frac{\partial U_i}{\partial x_j} \quad (2.4.13)$$

The modeled transport equation for the unresolved velocity scale ratio is :

$$\frac{D\xi_u}{Dt} = f_u - \frac{\zeta_u}{k_u} P_u + \frac{\zeta_u}{k_u} \varepsilon_u (1 - f_u) + \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_u}{\sigma_{\zeta u}} \right) \frac{\partial \xi_u}{\partial x_j} \right] \quad (2.4.14)$$

Equation is only valid when $f = 1$. Equation is solved for f_u :

$$L_u^2 \nabla^2 f_u - f_u = \frac{1}{T_u} (c_1 + C_2 \frac{P}{\varepsilon}) (\xi_u - \frac{2}{3}) \quad (2.4.15)$$

$$P = \frac{1}{f_k} (P_u - \varepsilon_u) + \frac{\varepsilon_u}{f_\varepsilon} \quad (2.4.16)$$

L_u and T_u , the the length and time scales defined from the unresolved kinetic energy and dissipation.

$$T_u = \max\left[\frac{k_u}{\varepsilon}, C_T \left(\frac{c}{\varepsilon}\right)^{\frac{1}{2}}\right] \quad (2.4.17)$$

$$L_u = C_L \max\left[\frac{k_u^{\frac{3}{2}}}{\varepsilon}, C_\eta \left(\frac{v^3}{\varepsilon}\right)^{\frac{1}{4}}\right] \quad (2.4.18)$$

Quantities $C_{\varepsilon 1}$, $C_{\varepsilon 2}^*$, $\sigma_{ku,eu,\xi u}$ are determined according to:

$$\sigma_{ku,eu,\xi u} = \sigma_{k,\varepsilon} \frac{f_k^2}{f_\varepsilon} C_{\varepsilon 2}^* = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} - (C_{\varepsilon 2} - C_{\varepsilon 1}) \quad (2.4.19)$$

The parameter that determines f_k is based upon the grid spacing, thus:

$$f_k = \frac{1}{\sqrt{C_\mu}} \left(\frac{\Delta^{\frac{2}{3}}}{\Lambda} \right) \quad (2.4.20)$$

Where Δ is the grid cell spacing and $\Lambda = \frac{k^{\frac{3}{2}}}{\varepsilon}$ is the turbulent length scale. This f_k parameter is recalculated at the end of each time step and that value used for next time step to determine the filter width. Table 2.4.1 shows the values of the $\xi - f$ model constants used.

Constants used in the $\xi - f$ model.									
C_μ	$C_{\varepsilon 2}$	c_1	C_2	σ_k	σ_ε	σ_ξ	C_τ	C_L	C_η
0.22	1.99	0.4	0.65	1	1.3	1.2	6.0	0.36	85

Table 2.4.1: Constants used in the $\xi - f$ turbulence model

2.5 Discharge coefficient theory

Discharge called C_d , is a dimensionless flow coefficient defined as a ratio of actual efficiency mass flow, which is the same as mass flow rate in inlet, and theoretical mass flow [13] as equation 2.5.1. Discharge magnitude gives always same order magnitude order as mass flow rates, when choose the optimal design. This relation defines how much losses in a certain seat and port in addition to the pressure drop losses, which controlled by pressure drop in Figure 2.5.1

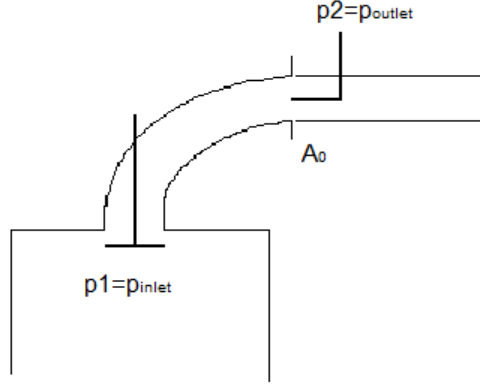


Figure 2.5.1: Definition for Discharge coefficient for exhaust port

As we see from equation 2.5.1, the ideal is to find a port and seat value that gives C_d of 1 that actual design is free from losses and a ideal mass flow is obtained. This means that the value of C_d is close to 1 preferred as possible. The theoretical mass flow is defined as:

$$C_d = \frac{\dot{m}_{actual}}{\dot{m}_{theoretical}} \quad (2.5.1)$$

$$\dot{m}_{theoretical} = A_o * w_2 * \rho \quad (2.5.2)$$

$$w_2 = \sqrt{\frac{2 * \Delta p}{p_m}} \quad (2.5.3)$$

where A is outlet area of the channel for the exhaust port out of cylinder head. Δp is the pressure difference driving the flow between inlet and outlet. ρ define the density in inlet of

the system and ρ_m is a mean density which calculated from values measures at the inlet and outlet as:

$$\rho_m = \frac{\rho_{inlet} + \rho_{outlet}}{2} \quad (2.5.4)$$

The value of density will vary from different tests at the experiments, In CFD simulation the inlet density is a constant while the density in the outlet is vary due to compressibility.

2.6 Parametric modelling

The basic of a parametric model is the complete defining of the geometry through variable parameters and associative links or conditions [4]. Theses parameter are dimensions, in CAD program, dimension drive the geometry. It can change the dimension value, this will due to change the geometry. The parameters may be user-named of their names are assigned automatically by the system. Parameter can be divided to constant or variables. This part has to do with the software itself [11] . Today, almost CAD program has a parametric function to quick modifying the design. Customer searching always after short time, quality in design, automated and functionality, which increase requirement for the parametric model in the market. In the beginning of the design, the designer should decide the most important parameter of the geometry. The parameter should be define and predict the movement of the geometry, define the transformation.

Morphological, study the shape, of the geometric can be divided in four different levels as Figure 2.6.1:

- **Fixed Object:** cannot change the shape. The object doesn't allow transforming.
- **Parameterized Object:** each parameter represents a dimension and change every parameter will change the shape. This is working well of non-complex geometry. Otherwise, it should use relations between the parameter.
- **Equation Based Relation:** an effective way to decrease the number of the parameter in the complex geometry by creating relations between them.
- **Script Based relation:** using a programming language in CAD program to create the relation between the parameter. Both equation and script are a kind of rule - based.

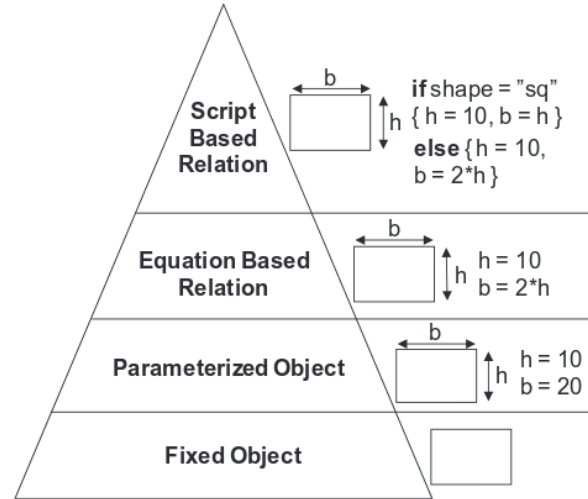


Figure 2.6.1: *Different stages of morphological transformation [10]*

Each a parametric model consists of three mainly iterations [11], listed below:

1. Definition stage, defined all parameter of the geometry by knowing the space of the design.
2. Design stage, design the model and defined every parameter, relation or script language.
3. Evaluation stage, evaluate the model through the user and the computer by understanding and analyzing the result.

2.7 Design of Experiment

The word experiment, used under controlled condition to discover an unknown or known effect. Experiments as often used to evaluate which process inputs, for example variable, have a significant effect on the process output, and what the target level of those inputs should be to achieve a desired result [15]. In the (DOE) theory, these design variables are also sometimes referred to as factors, a particular value of a factor is called level and a certain value of an output parameter corresponding to a design point is known as response. Experiments can be solve or designed in various ways to document the database. Experimental design, Design Experiments and Design of Experiments (DOE) all these word refer to the same meaning. Experimental designs used in the industry, to reduce design cost and speeding the design process, through reduce the time for engineer design changes. It takes a large number of iteration to optimize the geometry which is expansive, using the (DOE) explore a number of geometry during short time which explored finding a good design during short time. It can also use for optimization of quality and performance of product, screening of important factors and Robustness testing of products and process [6]. Design of experiments uses in many different area for example: Chemical industry, Car manufacturing industry, marketing and preference. DOE is a good way to explore a bigger design space also makes it possible to study the relations between design variable and target quantities. The method is a step to study which variable has big effect in the model and what is the best range of this variable. DOE is a methodology used to collect a maximum amount of database while reducing the number of experiments. DOE can generate a number of design points, design points means a number of geometries. These generated point or design send to simulation, the result gives a

general study over every parameter in the design.

There are many different methods of generating a DOE, depend on the number of factors in the task. Here will be present some of the method that has been used

1. Full Factorial Designs.
2. Latin Hypercube Design.
3. Random sampling.

There is some negative experience of using DOE, many of thesis point effect continue working with this method, the first relates to technical problem and second to non-technical problem [15]:

1. Choose an unreasonably large or small design.
2. Choose a wrong level for the factor or variable.
3. Using an uncontrolled variables or factor which can influence the output of the process.
4. Lacking of data analysis because of lacking in experience.

Some of the non-technical problem:

1. Choosing the wrong factors for the experiment.
2. Choosing a small number of design point which can be risk of understanding and studying the model.
3. Lack of experimental planning and document the information.

The number of samples you choose is up to person, but more samples will give you more reliable results and will also help avoid correlation between variables. If it expects that your effect size is going to be small relative to noise, then choose a larger sample size.

To prepare the Designed Experiment [8], There are some steps which need to take during starting work with design of experiment:

1. **Define the study object:** a general study of the object or geometry, function of the model, volume, possibility for changing.
2. **Choosing the responses to reach the goal:** choose the first response to study the object. For example, the swirl effect in the turbo, the mass flow rate in the exhaust port. Both of the swirl and mass flow can play a big role in the efficiency tube or exhaust port.
3. **Defining factor that may influence the response:** define which factor or parameter that can affect, for example the mass flow rate in the exhaust port. By listing out the factors, will be easy to study deeply the model.
4. **Defining the Factor-levels that may influence the response:** defining the high and low level of the parameter that been defined earlier in defining factor step.
5. **Choosing the experimental Design:** choose one of the design methods that generate a many design points and used most of the factor-level, this will be giving a clear details over every parameter in the model.

2.7.1 Full Factorial Designs

Contains all possible combination of a set of factors [6], for the two-level factorial design points, each of them contains multiple levels. For k factors, each of k has n factor-levels. There are n^k factor-level combination. For example, if there are 2 factors with a factor-level for each is 2. A full factorial design has 2^2 run, which is equal to 4 design points. A general full factorial design is employed when the number of levels for the factors is more than two. For example two factors $Y1$ and $Y2$ have 4 and 2 level respectively. A general full factorial in this case is $X1$ multiply with $X2$. A design point is then defined corresponding to every level in $Y1$ and $Y2$ as Figure 2.7.1. The full factorial design results gives a clear details of the model, but it is also the most costly in experiment. The size of the experiment or simulation grows up with increasing the number of factors. A fractional of full factorial is based on the full Factorial Design. It is only subset the fraction of full factorial experiment from the factor levels, the equation become n^{k-p} . Through the experiment, start to know which variable or factors has a main effect on the model.

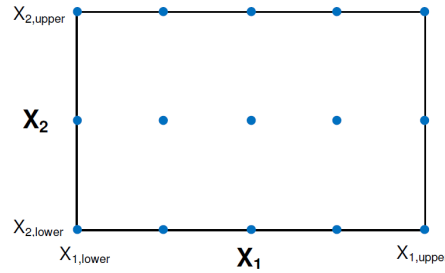


Figure 2.7.1: An example of full factorial Design for factor $X1$ and $X2$ with 5 and 3 levels

2.7.2 Latin Hypercube Design

The sampling method uses to identify firstly a function with a number of variables N with, each variables divided into equal probable intervals M . The method selected randomly according to the sample point that has been chosen. Latin hyper cube has the advantage of generating a set of samples that more precisely reflect the shape of a sampled distribution than pure random samples. One of the benefit points of this sampling method, there exists only one sample for each row and column in the hypercube of N intervals. This avoids the possibility of design points, which is problem in some random sampling method. A Latin Hypercube design for two factors and two intervals 4 and two in Figure 2.7.2. Second benefit point of this method, it is not expansive and does not need a long time as the full factorial design. The good things with the Latin Hypercube design is: it is optimal under the underlying statically model, It can serve various numbers of runs and input variables and it can be easily generated with a reasonable computing time.

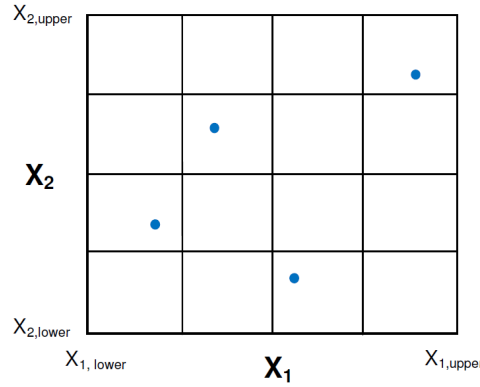


Figure 2.7.2: An example of Latin hyper cubic for two factors with 4 requested samples

2.7.3 Random sampling

It is a sampling method, a new simple point are generated without taking into account the previously generated sample points. This method does not need to calculate how many simple points is need before it started. It can also define as unbiased surveying technique, simplest of the probability sampling technique.

2.8 Response surface methodology

Response surface methodology , RSM, is a name of several mathematical and statical techniques used to model a problem in which the goal parameter is affected by many variables and optimizing the response. It uses data from design of experiments used to model a problem in which the goal parameter is affected by many variables and optimizing this response. It is usually unknown the relationship between the response and independent variables.

If the response is linear from dependent then this approximation can be done by first order polynomial, define as equation 2.8.1:

$$y = C_o + C_1x_1 + C_2x_2 + + C_nx_n \quad (2.8.1)$$

Where x_i is each independent variable and C_i is the polynomial constants [8]. This is usually a good guess if the form on the response surface is unknown as a good approximation in many cases.

It needs to use a polynomial of higher degree to create the response surface as in the second

order model as equation 2.8.2:

$$\begin{aligned}
y(x) &= y_{linear}(x) + y_{interaction}(x) + y_{quadratic}(x) \\
y_{linear}(x) &= C_o + \sum_{i=1}^k C_i x_i \\
y_{interaction}(x) &= \sum_{i=1}^k \sum_{j=1}^k C_{ij} x_i x_j \\
y_{quadratic} &= \sum_{i=1}^k C_{ii} x_i^2
\end{aligned} \tag{2.8.2}$$

Where k = the number of variables in the second - degree polynomial There are three ways to evaluate and visualize the effect of variables , the surface response the result by looking at the correlation between the parameters. Which used to understand, how two parameters are correlated to each other, or individual response which study every variable effect on the target quantities.

2.9 Mesh type

There are three different type of mesh, two type of mesh used in this thesis work. Here a short explanation of both type of mesh:

2.9.1 Hexa mesh

It is a structure computational grids because it is identified by regular connectivity. The benefit of structure grid over unstructured grid are better convergence and higher resolution. It can be difficult to apply for complicated geometry. This type of mesh used for the transient state simulation in this thesis work, because it was already used. The program called FAME-HEXA program, which produce hexa mesh in AVL FIRE program [17].

2.9.2 Hybrid mesh

The hybrid computational grids contains both structure and unstructured mesh. This gives a good possibility for those parts of the geometry to have structure grids, the complex parts can have unstructured grid. This type of mesh called hybrid assistant mesh in AVL FIRE [17].

3 Method & Implementation

In this chapter, detailed about the method described and the most important parameter, the process visualize in flow chart diagram as Figure 3.0.1 , show a flow chart of the primary and secondary points of method involved in this project:

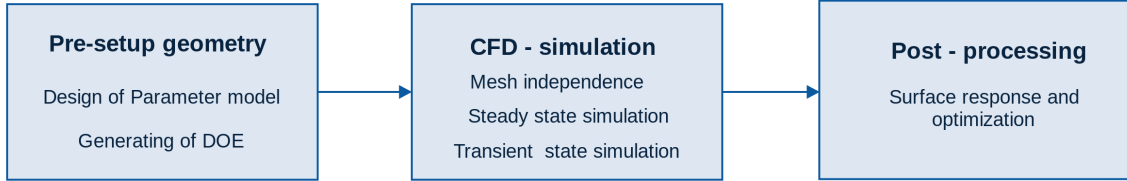


Figure 3.0.1: *Flow chart of method and implementation*

3.1 Pre-Setup Geometry

Pre-Setup geometry consisted of two main steps, create a parameter model after drawing the geometry and generating a design of experiments (DOE), the steps explains more in details below:

3.1.1 Design of Parameter model

CATIA V5program used to draw the exhaust port geometry, the dimension of all parts in the geometry measured directly from engine, the list belows holds a quick overview of the steps involved in this point:

1. Defined the interfaces of the geometry, means the boundary of the geometry: inlet and outlet of the exhaust port. Defined the parts which include to the exhaust port geometry: inlet, outlet, chamber, Cylinder, valves, exhaust seats, channels and manifolds as Figure 3.1.1.

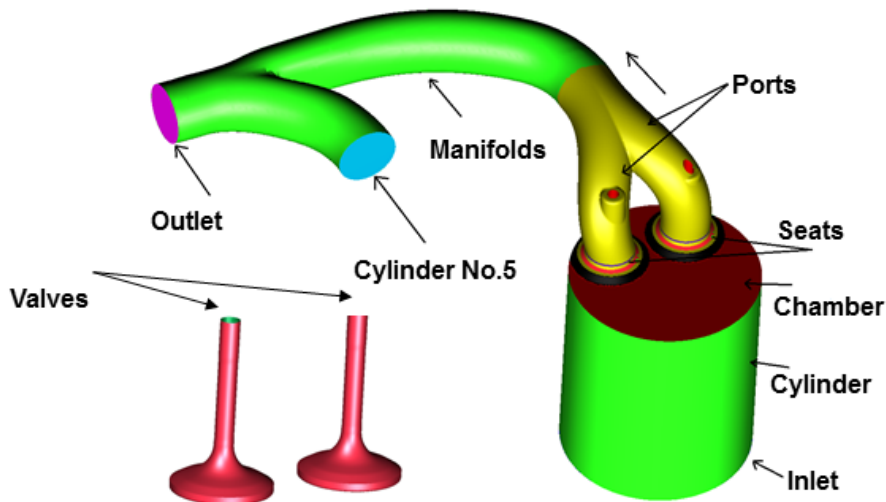


Figure 3.1.1: *Interface of exhaust, component belong to the exhaust*

2. Defined the requirements, design constraint and limitation after discussion with the designer of the exhaust port concept, as the table in Appendix A. Discussion gave a good detailed picture about the design space of the exhaust port and possibility of area movement in space.
3. Defined the dimension that would not be parametrize in this project in Appendix B.
4. Defined the name of parameters in CATIA V5. Additionally, several concepts were studied individually, thereafter compare with the main concept to decide which concept will be taken to simulation, the concepts matrix attach in Appendix C. Concept number one consist of eleven parameters, while concept number nine has five parameter more than the concept number one, as Figure 3.1.2

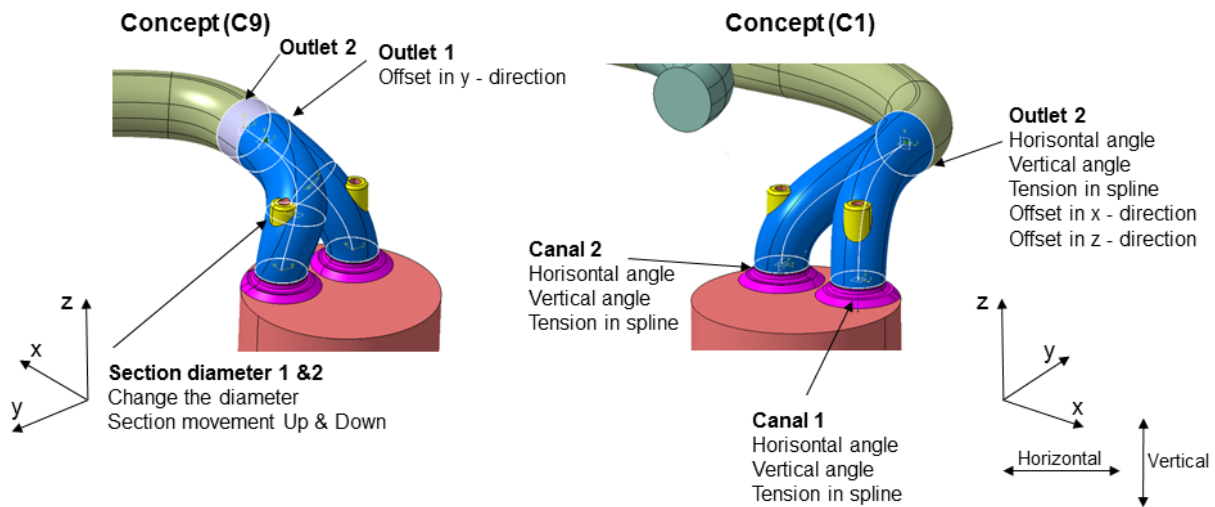


Figure 3.1.2: Paramater definition in concept 1 and 9

5. The CAD model exported to STL file for CFD-simulation, because this STL extension uses a simple triangular mesh to approximate the bounding surface of the part. The reason was that this file gave possibility to define the boundary selection in AVL FIRE and mesh the surface. The resolution of STL increased in CATIA V5 to export a finer STL file to AVL FIRE.

3.1.2 Generating of DOE

A file in Excel program was an interface to CATIA V5 program used to generate the design of experiments (DOE), the list below holds a quick overview of the steps involved in this points:

1. In Excel file chose the Latin hyper cubic method to generate the design of experiments (DOE).It didn't use the full factorial method in this project because of the time consuming.
2. Defined the range for every parameter, It defined a big range, when studied the concepts while it decreased the range for other studied in this thesis work. Depended on the range and number of design points generated the design och experiments (DOE) as in appendix D. This process will be saves the geometry as a STL file.

3.2 CFD - simulation

CFD - simulation consists of three main steps, mesh independence, steady state and transient state simulation, all simulation submit after an automatic process in cluster by using bash script, is a Unix shell and command language written, the process included a macro for defining the boundary selection in AVL FIRE, meshing and setup solver setting thereafter submit the file to cluster, to solve it, here explained more detailed the main steps:

3.2.1 Mesh independence

This step started before the steady state simulation process, the reason was to have an good quality mesh with low number of cell, low time consuming for CFD - simulation, to use it for a design och experiments (DOE). The type of mesh in this study was hybrid mesh, by using hybrid assistant mesh in AVL FIRE. Mesh Independence didn't make for the port in transient state simulation because it was already done a mesh study with FAME-HEXA program, this step consist of three steps:

1. Defined the number of cell that was sufficient to simulate and had a good convergence. The number of cells studying with the convergence of the mass flow rate [kg/s].
2. Studied the quality mesh according to AVL FIRE mesh , number of cells, first criteria aspect ration smaller, Skewness smaller than 0.85, twisted faces, warpage, no cell with negative volume, means the jacobian ratio is not minimum than -1 .
3. Studied the number of closure level in the mesh, the closure level defined, the value specifies the number of cell layers below the surface layer that should be included in the refinement. The mesh refinement starts at surface cell layers of the volume mesh that are intersected by the active surface and edge mesh selections. The refinement continues into the volume mesh from the surface cells along the number of layers specified [17]. It tested with high closer level two and three , due to increase the number of cell,which increased the time consuming of simulation, therefor the closure level defined as one becuae it was sufficient for this studying.
4. Studied the Y-plus, to avoid values larger than 100 ,Y-plus should be between (30-100). The effect of High Y-plus means the kinetic energy and the pressure drop increase unrealistic. This study had relation with the number of cell study. It started to increase the number boundary condition according to hybrid mesh program from 1 to 3. Boundary condition defined in Valves, face, cylinder, exhaust port and manifolds.

3.2.2 Steady state simulation

The simulation performed at three different time points or three different valves lift position: minimum lift opening 1 mm, lift opening 10 mm, lift opening 14.6 mm. The setting file defined the initial mode as uniform initialization, which means that the velocity here three different mode of initialization mode according to AVL FIRE program [17]. Or potential Flow: The velocity field is initialized based on the results of a potential flow calculation throughout the domain. This initialization can speed up the convergence considerably, especially for a steady calculation [17]. Defined the derivative equation as Least Sq and defined the compressibility as compressible, which means the density is a function of temperature only $M < 0.3$. And used a standard wall function as a default.

AVL- FIRE solver settings	
Fluid	Air
Derivatives	Least Square Fit
Turbulens Model	k- ε
Compressible	compressibility
Wall treatment	standard wall function
<hr/>	
Momentum	Differencing schemes
Continuity	MINMOD
Turbulence	Central differencing
Enegy	upwind
	upwind

Table 3.2.1: Definition of boundary condition for steady state

Study different exhaust port concepts

To study the difference between 8 different concepts of CAD models, generated a design of experiments (DOE) for 50 design points from every model with the same range and run simulation. From the result studied the mass flow rate in inlet for every matrix and plotted for every concept. The initial condition for whole the domain defined to 100 kPa and the iteration number for 2000. The convergence criteria defined to 1.10^{-4} for both pressure and momentum. The temperature was 295 K, a cold air as steady state flow rig test. The boundary condition defined as Table 3.2.2 below:

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	104.910 kPa	295 K
Outlet	Static pressure	100 kPa	

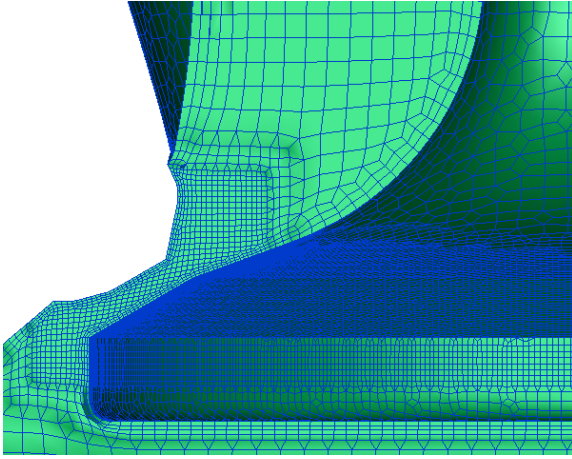
Table 3.2.2: Definition of boundary condition, valves lift opening 10 mm

Study valves lift opening 1 mm, 10 mm and 14.6 mm, low pressure

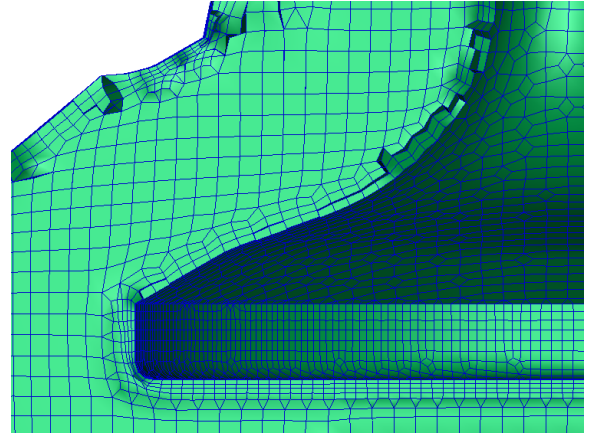
To study the different between valves lift opening 1mm, 10mm and 14.6mm, low pressure as steady state flow rig test. Run design of experiments of 20 design points of three different valves lift with the same pressure boundary condition as Table 3.2.3. The initial condition defined 101 kPa for whole the domain and the initialization mode is potential flow. Set the number for iteration to 3000 and convergence criteria 1.10^{-6} for both pressure and momentum. It decreased the convergence criteria to have better convergence, to study the mass flow rate. The pressure was also ramp to get convergence. It started with pressure 101 kPa, increase at iteration 500 to 105 kPa. The mesh refined between the valves and valves seat to capture the fluid phenomena at valves lift 1 mm as Figure 3.2.1a. The temperature changed to 726.6 K as transient engine test to study the different between the methods.

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	105 kPa	726.6 K
Outlet	Static pressure	100 kPa	

Table 3.2.3: Definition of boundary condition, valves lift opening 1 mm, low pressure



(a) Mesh for 1 mm Valves lift



(b) Mesh for 10 mm Valves lift

Figure 3.2.1: Two different mesh refinement, hybrid mesh, for two various Valves lift, valve cut section

Study valves lift opening 1 mm, high pressure

The reason of this step was the same as the case above 3.2.2. The pressure boundary condition defined according to transient state simulation or transient engine test at crank angle 870 as Table 3.2.4. Normally, the pressure is very high at the stroke expansion, started to decrease at the exhaust, when the valves lift opened gradually. The pressure was ramped to have better convergence from 351 kPa to 815 kPa at iteration 4000. Set the number of iteration to 6000 and the convergence criteria was to 0.000001 for both pressure and momentum. The initial condition defined to 352 kPa and temperature to 726.6 K.

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	815.8 kPa	726.6 K
Outlet	Static pressure	351 kPa	

Table 3.2.4: Definition of boundary condition, valves lift opening 1 mm, low pressure

Study valves lift opening 10 mm, high pressure

The pressure boundary condition defined as transient state simulation test at crank angle 915 as Table 3.2.5. The pressure at inlet was also ramped from 430 kPa to 462.340 kPa at iteration 4000. Initial condition defined to 430 kPa. The number of iteration set to 6000 and convergence criteria was 0.000001 in pressure and momentum. It also studied the different between having high temperature 726.6 K and cold temperature 295 K by running a (DOE) of 20 design points. The mesh wasn't refinement as lift 1 mm as Figure 3.2.1b.

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	460 kPa	726.6 K
Outlet	Static pressure	430 kPa	

Table 3.2.5: Definition of boundary condition, valves lift opening 10 mm, high pressure

Study valves lift opening 14.6 mm, high pressure

The boundary condition defined as the transient engine test, at crank angle 957, as Table 3.2.6 and run a simulation for steady state. The initial condition, convergence criteria, mesh settings and temperature applied at the same way as lift 10 mm.

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	340.3 kPa	726.6 K
Outlet	Static pressure	308.2 kPa	

Table 3.2.6: Definition of boundary condition, valves lift opening 14.6 mm, high pressure

3.2.3 Transient-state simulation

The transient state consisted normally of four stroke as mentioned in the theory. The transient simulation simplified to only one strokes in this study and run only for the exhaust stroke. That is because of the exhaust port was the focus in this project, saving time of simulation. The boundary condition defined as transient engine test data in this simulation. The simulation started at crank angle degree 845 deg (expansion), temperature 726.6 K, pressure in Cylinder defined as 12 bar and continue until the gave a way out of exhaust to crank angle degree 1072 deg (exhaust), the boundary defined as the Table 3.2.7. Convergence criteria defined for various crank angle degree to convergence the problem. Hybrid wall treatment used as a function between laminar and turbulent part of the boundary layer is derived from the relation of the friction velocity. Refer to section generalized Wall Treatment [17]. Three smoothing area created to smooth the flow, when it came out of the exhaust.

AVL- FIRE solver settings			
Boundary	Type	Value	Temperature
Inlet	Total pressure	1200 kPa	295 K
Outlet	Static pressure	254 kPa	
Fluid	Air		
Derivatives	Least Square Fit		
Turbulens Model	$k-\xi - f$		
Wall treatment	Hybrid wall treatment		
	Differencing schemes		
Momentum	MINMOD		
Continuity	Central differencing		
Turbulence	upwind		
Energy	upwind		

Table 3.2.7: Definition of boundary condition for transient state

The pressure boundary condition in cylinder as Figure 3.2.2a and out of manifolds as Figure 3.2.2b used in these three different valves lift position 1 mm , 10 mm and 14.6 mm at three different crank angle 870 , 915 and 957. The piston position defined in two different position, (BDC) defined as the bottom dead center and (TDC) define as the top dead center. Two phase in this stroke for pressure, firstly Blow down phase , when the valves started to open and the pressure decrease, the valves lift below 10 mm. Secondly Pumping phase, when the valves lift

reach 10 mm until it close.

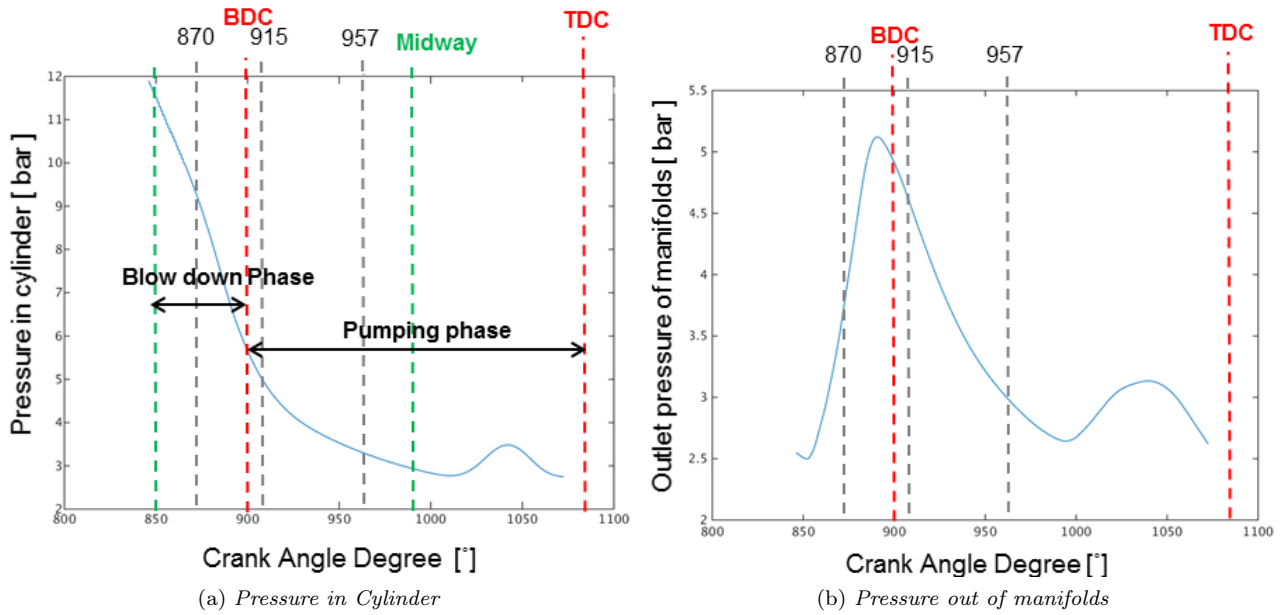


Figure 3.2.2: Pressure in Cylinder, out of manifolds in transient state simulation

At Scania, they had already a method of transient state moving mesh by using ESE - Engine program, Chamber modeller - ESE Engine is a novel approach to generate computational grids for simulating the thermo - fluid flow in internal combustion engines, here can modelling Mesh movement, 2D/3D Valve seat meshing and assembling of combustion chamber and ports [17]. A method created to run a (DOE) for transient moving mesh as Figure 3.2.3. The process consisted of three steps; A: have an old moving mesh, B: created an arbitrary contact plus an a (DOE). The arbitrary contact consist of node to fix node contact of the new mesh with the rest from the old mesh. This arbitrary contact must be at the same position where the port would be cut and changed with the new port. C : Assemble the mesh of new port with the rest mesh of moving mesh in piston, in the system to build a moving mesh. FAME Hexa used to mesh the new exhaust port in this in the transient simulation. The transient state run only for the four best geometries from the steady state.

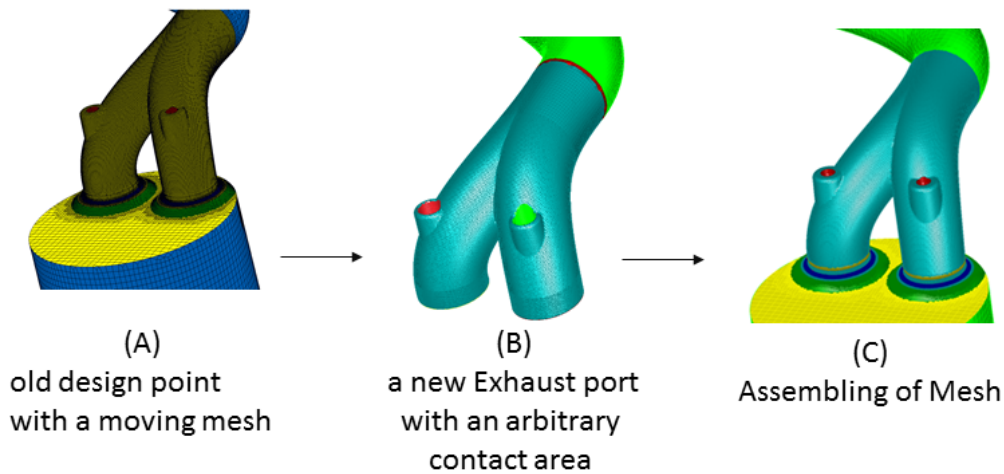


Figure 3.2.3: Assembling process for new port in transient simulation

3.3 Post-processing

After every simulation visualized the result by using AVL FIRE post processing to study the fluid, a bash script implemented, to read the mass flow rate for every exhaust port , thereafter calculated the discharge. The MATLAB code used to study the effect of the every parameter in the target parameter mass flow rate and study the design of experiment (DOE). In steady state simulation, was discharge coefficient C_d and mass flow rate were used to evaluate present indications of the best design point in the design of experiment. In transient state, used the pressure drop between the cylinder and outlet of the exhaust, to evaluate which port is the optimal in the (DOE).

3.3.1 Surface response and Optimization

The result of mass flow rate from the AVL FIRE was implemented in MATLAB code to study the design of experiments, studied the effect of parameter in main parameter mass flow rate, these results gave a clear picture about every parameter in the model. The steps of this section represented under:

1. Studied the sampling method Latin hyper cubic by plotting the parameter range, the reason is to study the sampling method.
2. Plotted normalize parameter correlation bar chart. This explain the correlation between every parameter or effect every parameter in itself and with other parameter in the model.
3. Plotted the normalize individual response, this visualize individual effect of every parameter in the mean parameter mass flow rate.
4. Plotted the normalize surface response to visualize the correlation of two parameter in the mean parameter mass flow rate.

4 Results & Discussion

In this chapter, represented and discussed the result from the parameter modelling, mesh independence, steady state and transient simulation. Visualized also the result of testing various valves lift and pressure boundary condition from steady state flow rig test and transient engine test. Discussed the difference between steady state and transient state for the exhaust port simulation.

4.1 Parameter modelling

The exhaust port had always losing in pressure in the geometry because of changing diameter from the inlet to outlet, valves, valves seat and two channel in inlet. In this study, used the same mesh setting after mesh independence studying. Creating this parameter model to have better control over the geometry. Eight various concepts created as mentioned in the method 3.1.1. The first concept gave the best result of mass flow rate according to steady state simulation in Figure 4.1.1a and robustness parameter model in Figure 4.1.1b according to CATIA V5 criteria. Robustness defined if the concept is intelligent, means robotic, or not, It can also means the combination of parameter in the concept accept to create a number of geometry without minimum number of failing, CATIA program has own criteria when it created a geometry, Some dimension can be difficult to create. Robustness decreased when it made the geometry more complex with high number of parameter. This study did after running a design of experiment of 50 design points or geometries. It chose a big range for every parameter to test the robustness of every concept, the first concept gave high robustness as Figure 4.1.1b and this concepts consist of three sections, two in inlet and one in outlet as shown in the method 3.1.1 . While the other concepts contents of a section between inlet and outlet. This section between the inlet and outlet increased the losing in pressure because it created contraction between inlet and the middle, some times created expansion, between the section and outlet.

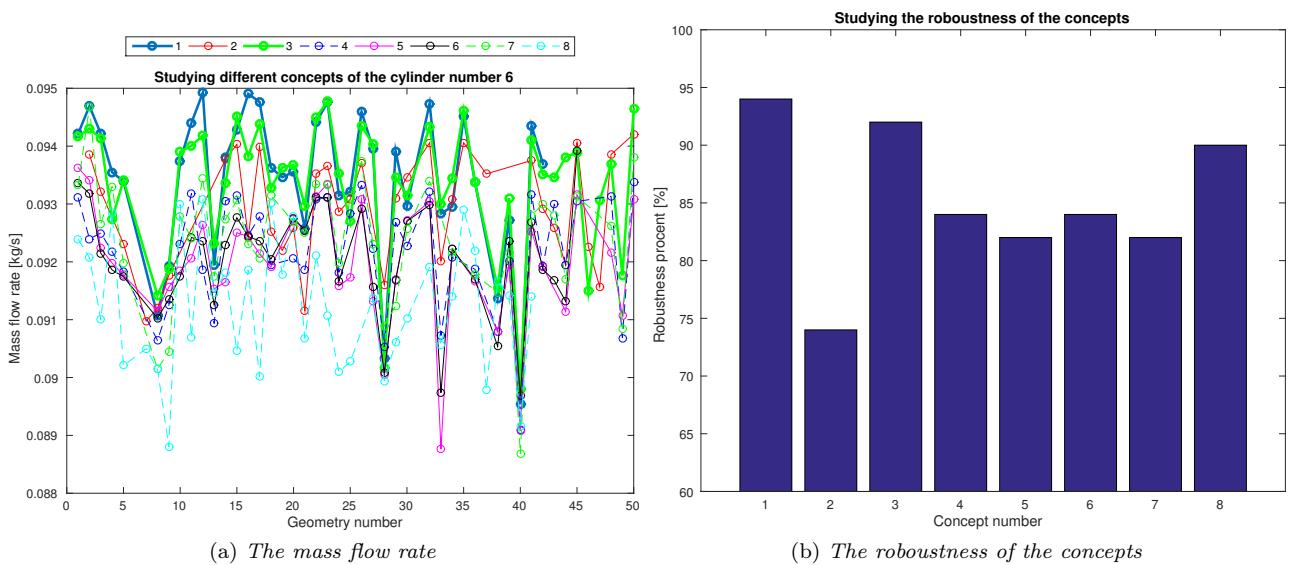


Figure 4.1.1: Show the mass flow rate studying of 8 various concepts and robustness, for 50 design points, Number 1 o 8 represent the number of concept in CATIA

It created a concept number 9 after first comparing between the concepts . Run a simulation of 200 geometries to see the different between concept 9 and 1 according to mass flow rate as Figure 4.1.2 . Concepts 9 created according to concepts 1 with five more parameter, The results shows that concepts 1 has 95 percent robustness than the concept 9 with 85 percent, the steady state simulation showed also that concept number 1 has better result than 9. This different has parallel relation with the range because of using a big range, which effect the efficiency of the model. From this result choose concept number one as a concept for further simulation in the project.

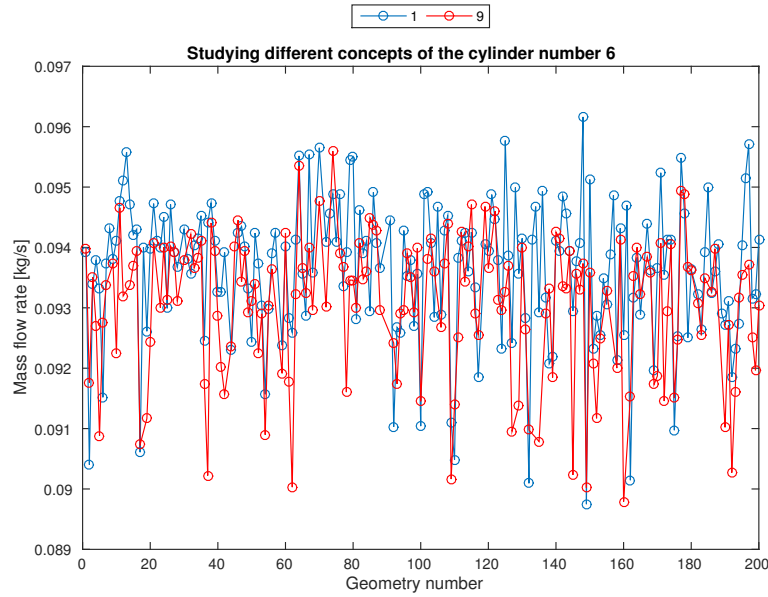


Figure 4.1.2: Compare between concept 1 and 9 by studying the mass flow rate [kg/s]

4.2 CFD - simulation

The result of CFD simulation consisted of both mesh studying, steady and transient state simulation. Simulation are evaluated by using the mass flow rate and discharge coefficient C_d for steady state simulation. The transient simulation evaluated by using the mean pressure drop, the difference between the pressure in cylinder and out of manifolds.

4.2.1 Mesh independence

A mesh Independence study was carried out of the exhaust port for steady state simulation. The reason of this study was to have a good quality mesh without any negative volume cell, which can apply for many design points, with a sufficient number of cells . Therefore it was running 6 different types of mesh setting as Figure 4.2.1. It started from coarse to fine mesh, the studying process continue until getting better convergence. The mesh setting number 6 was used as mentioned before for concept studying, different valves lift opening and different pressure boundary condition. This mesh was sufficiently because it started to convergence according the mass flow rate curve studying, the case number 6 has also relation with the number of boundary condition in Y-plus value studying.

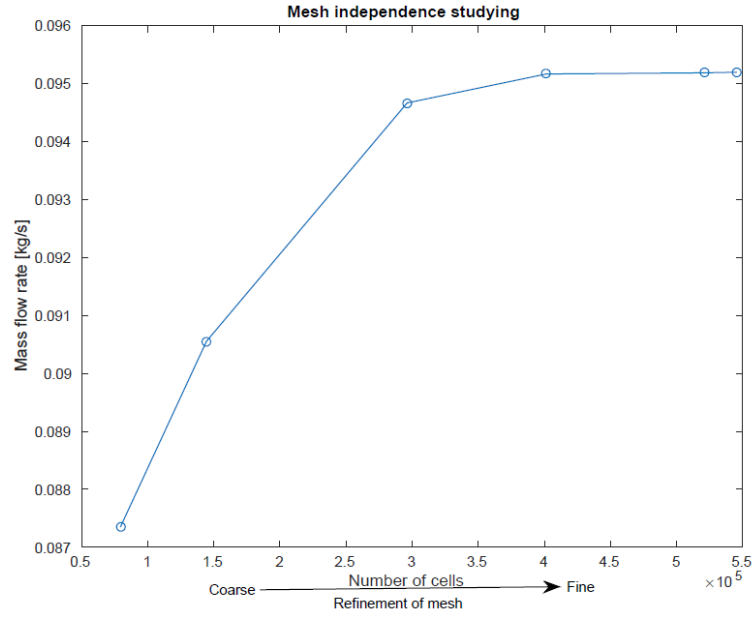


Figure 4.2.1: Mesh independence for steady state simulation

The quality of mesh started before the Y-plus to have a good quality mesh, according to to AVL FIRE program criteria, with low number of negative cell , have a convergence solution with low number of cell for time consuming as Table 4.2.1. The table shows some of the important criteria during the mesh Independence study. The mesh number 6 was the best according also to the number of boundary condition.

Mesh charatersitics for exhaust port							
No.	Number of cells	aspect ratio <10	skewness < 0.85	twisted faces	warpage 60-180	negative volume	mass flow kg/s
1	79534	0	1	1	15	0	0.0873528
2	144477	0	1	1	12	0	0.0905522
3	296173	0	1	0	0	0	0.0946614
4	401428	0	2	0	0	2	0.0951603
5	521350	0	1	0	0	0	0.0951805
6	545129	0	1	0	0	0	0.0951907

Table 4.2.1: Mesh quality and independence for exhaust port

Y - plus is good to study, to capture the increase in turbulence viscosity and calculated by the relation between the shear velocity divided by the kinematic viscosity. The Y-plus study for these component involved in this study show in the Table 4.2.2. The Y-plus value is normally between 30 - 100 but it also possible to have 120, which could found at the manifolds wall. The flow studying phenomena was in the exhaust port, valves, chamber and cylinder. The thickness of the boundary was 0.00236 mm according to the Y-plus thickness for the wall studying. It could made more finer mesh at the wall, due to increase the time of simulation and will cost more time, when it run a design of experiment (DOE).

Y- plus studying for exhaust port			
Name of Component	No. Boundary 1	No.Boundary 2	No.Boundary 3
Exhaust port	83	60	48
Chamber	75	55	45
Valves	62	41	30
Manifolds	180	140	120
Cylinder	75	55	45

Table 4.2.2: Y- plus studying with number of boudary

4.2.2 Steady state simulation

The result from steady state is for three different valve lifts, when the valves lift opening position was equal to 1 mm, 10 mm and 14.6 mm, used two different pressure boundary condition, at flow test rig at Scania and at engine test simulation.

Different cylinder position

Compared between different position of cylinder, as cylinder number 6 ,5 and 4, in the engine, to study if it needed to have an optimal exhaust port for every cylinder position, or it is sufficient with one optimal exhaust port for all positions. It run two different design of experiment of 20 design points and the one of the result show in Figure 4.2.2a because both running giving the same result, the table show the sort of the best five geometries in the simulation according to mass flow rate [kg/s] Table 4.2.3. From the result in the the table and figure, can see clearly that there showed an unique optimal exhaust port fits for every cylinder position. Both the tests showed the same result. The flow didn't treated at the same way for every cylinder because the result depended about the position of outlet. Cylinder 1, 2, 3 want investigate due to symmetry.The result showed shows also that the two symmetric inlet channels wasn't performing the best.

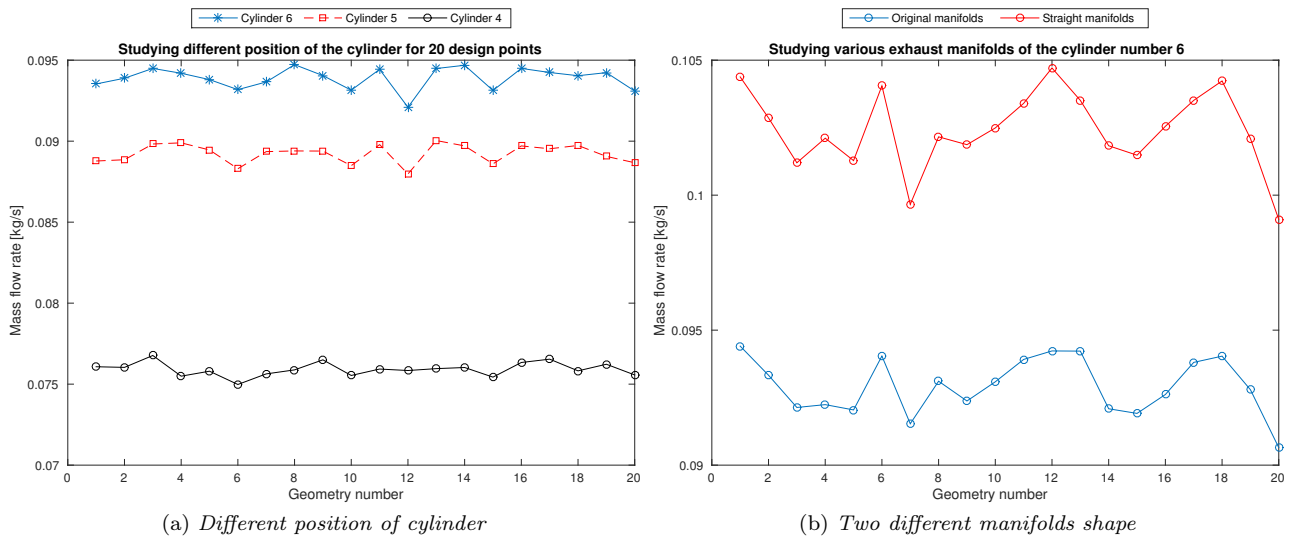


Figure 4.2.2: Show the mass flow rate for different position of cylinder and manifolds shape

Cylinder No. 6	Mass flow rate	Cylinder No. 5	Mass flow rate	Cylinder No. 4	Mass flow rate
8	0.0947	13	0.0900	3	0.0768
14	0.0947	4	0.0899	17	0.0765
16	0.0945	3	0.0898	9	0.0765
3	0.0945	11	0.0898	16	0.0763
13	0.0945	18	0.0897	19	0.0762

Table 4.2.3: The result for the first best five of 20 geometries, according to mass flow rate [kg/s]

Different manifolds shape

Compared also between two different manifolds exhaust. The first manifolds was like the original in the engine like a bend pipe and the second manifolds had a straight shape. The reason of this test was to see the effect of manifolds in the optimal exhaust port. At flow test rig laboratory is still using a straight manifolds when tested an optimal exhaust port. To tested both of them, run design of experiment for 20 geometries as Figure 4.2.2b. The result showed that an unique optimal exhaust ports for the original and straight manifolds. A higher pressure lossing in bend (orginal mainfolds) compare with the straight manifolds. Today, CFD tools gives possibility to test engine internal components. The best five geometry of the design experiment showed in the Table 4.2.4, sorting after the mass flow rate [kg/s].

Original mass flow	Mass flow rate	Straight manifolds	Mass flow rate
1	0.0944	12	0.1047
12	0.0942	1	0.1044
13	0.0942	18	0.1042
6	0.0940	6	0.1041
18	0.0940	17	0.1035

Table 4.2.4: The result for the first best design points in the design of experiment from different exhaust manifolds

Different Valves lift and pressure boundary condition

It made sex design of experiments (DOE) as mentioned in method 3.2.2. It used two different pressure boundary condition in this study, low pressure as steady state flow rig test and high pressure as transient engine test. First case was for valves lift 1 mm low pressure, second for valves lift 1 mm high pressure. Third for valves lift 10 mm low pressure, fourth for Valves lift 10 mm high pressure. Fifth for valves lift 14.6 mm low pressure and Sixth for valves lift 14.6 mm high pressure.

The reason of this test to study the effect of the pressure and valves lift in the optimal exhaust port in steady state simulation. The result illustrated in Figure 4.2.3, and the table sort the result from the first five best of 20 geometries in the matrix according to discharge coefficient C_d as Table 4.2.5. The result showed different optimal port for 1 mm valves lift and 10 mm valves lift. Because of the fluid has two different fluid phenomena at these two valves lift opening. While the result was similar for valves lift 10 mm and 14.6 mm. Which means that the fluid has the same fluid phenomena for valves lift above 10 mm.

It run also a design of experiment for case 3 and 4, with cold temperature 295 K as flow rig test.

To study the effect of temperature in the optimal port. The result for the five best geometries from this test was the same as 726.44 K. The difference was that the mass flow rate decrease because the density decrease in cold temperature.

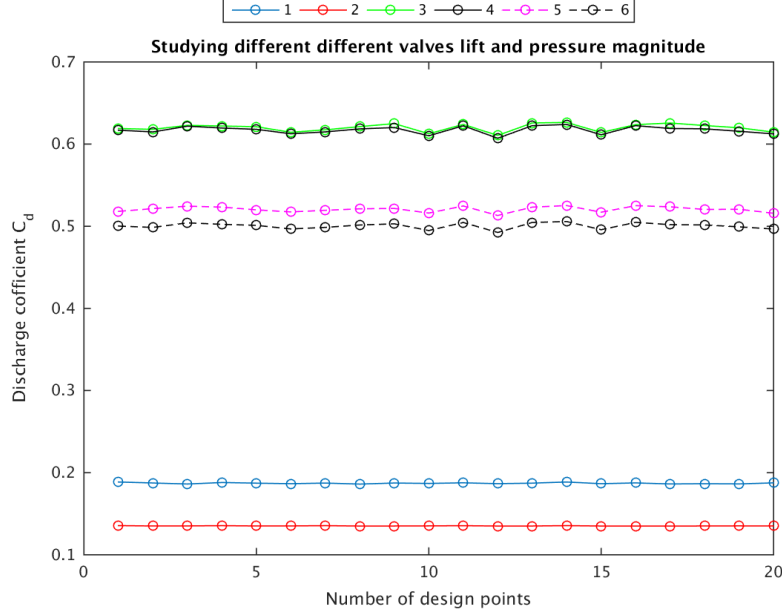


Figure 4.2.3: Discharge coefficient C_d for six different tests. 1:valves lift 1 mm, low pressure. 2:valves lift 1 mm, high pressure. 3: valves lift 10 mm, low pressure. 4:valves lift 10 mm, high pressure. 5:valves lift 14.6 mm, low pressure. 6:valves lift 14.6 mm, high pressure

1	C_d	2	C_d	3	C_d	4	C_d	5	C_d	6	C_d
1	0.1886	1	0.1355	14	0.6264	14	0.6239	14	0.5251	14	0.5057
14	0.1886	14	0.1355	13	0.6257	16	0.6225	16	0.5251	16	0.5047
4	0.1880	7	0.1354	17	0.6257	13	0.6224	11	0.5247	13	0.5046
11	0.1877	4	0.1354	9	0.6249	11	0.6223	3	0.5245	11	0.5046
20	0.1876	11	0.1354	11	0.6240	3	0.6218	17	0.5236	3	0.5041

Table 4.2.5: The result of first five geometries in the DOE from six different cases

4.2.3 Transient state simulation

Run a transient simulation for four different geometries from the Table 4.2.5. First and second of the geometries was the best in valves lift 1 mm , 10 mm , with pressure boundary condition from engine test simulation. The third and forth geometry wasn't of the five best geometries in the table. Calculated the means pressure for these four geometries as Figure 4.2.4a. From the result can see that the geometry number 16 is better than 7, because it has minimum mean pressure drop. It plotted the curve for these two geometries at three crank angles as Figure 4.2.4b. These three different crank angles represented three different valves lift opening as mentioned in the method 3.2.3.

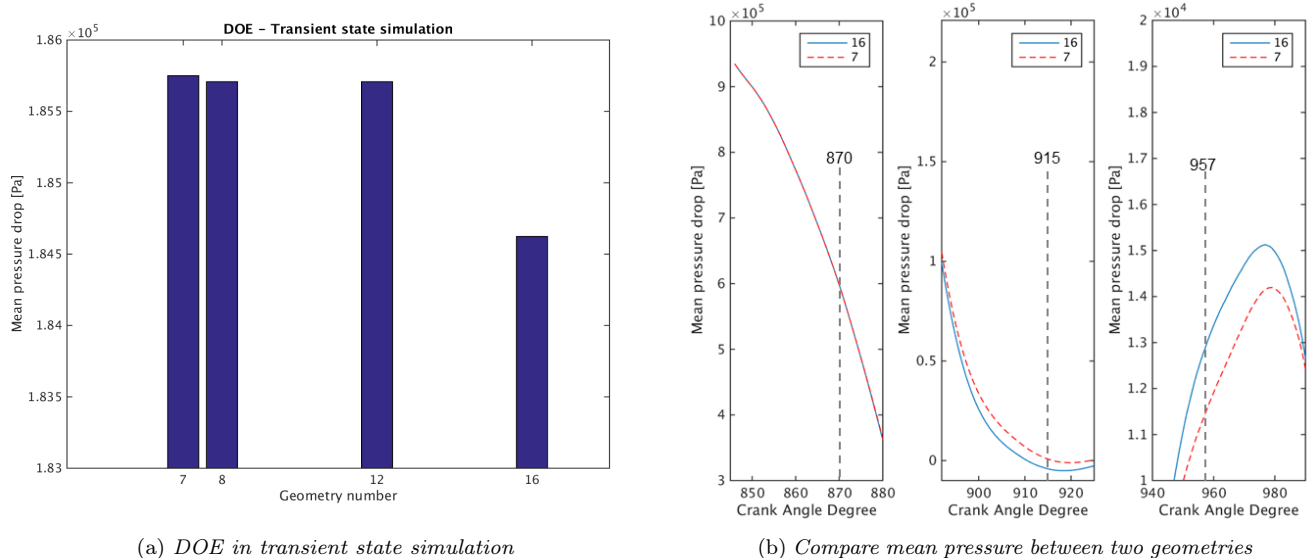


Figure 4.2.4: Compare between different geometries in the mean pressure drop, transient state simulation

From the result, there wasn't different in mean pressure for first crank angle at 870, when the valve was at 1 mm, the difference started after that. It visualized the result from AVL FIRE by taking picture for three different crank angle, two different geometries 16 and 7. It visualized the result from transient state and steady state at valves lift 1 mm in Figure 4.2.5 and 10 mm for geometry number 16 in Figure 4.2.6. The pressure boundary condition at steady state is from transient engine test simulation. Different in some parts in geometry: Valves, valves seat and piston. The different is also in mesh type, it applied hybrid mesh in steady state, while it applied FAME HEXA in transient state simulation. The result from steady state showed the same from valves lift above 10 mm with transient simulation.

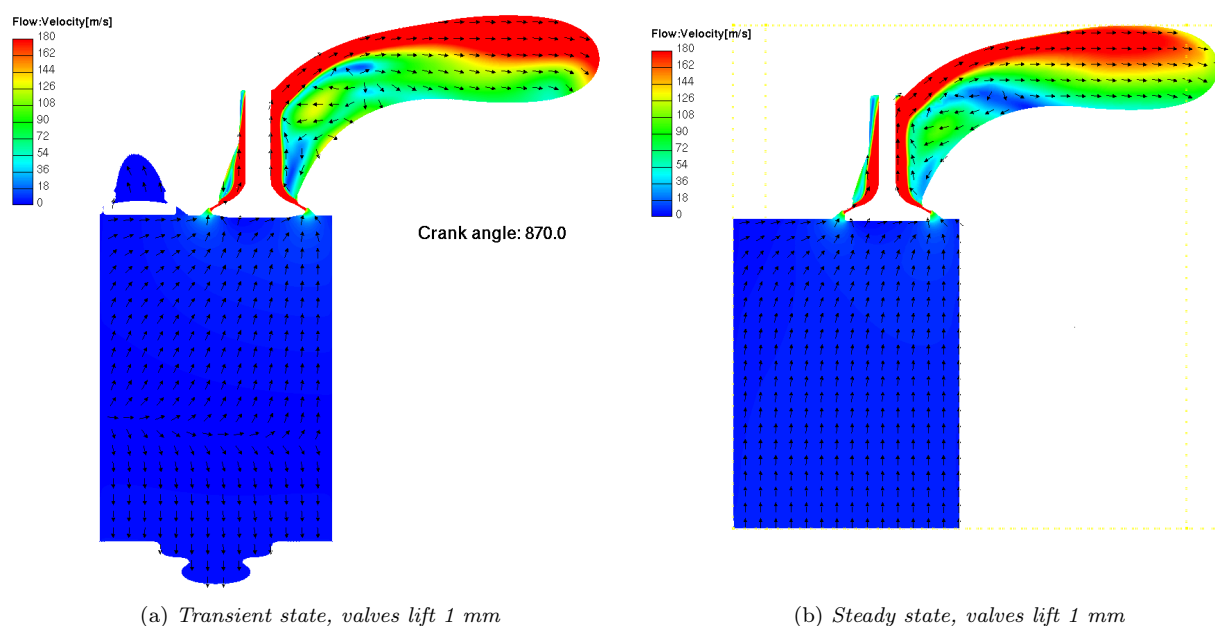


Figure 4.2.5: Compare between two approaches simulation, geometry number 16, valves lift 1 mm

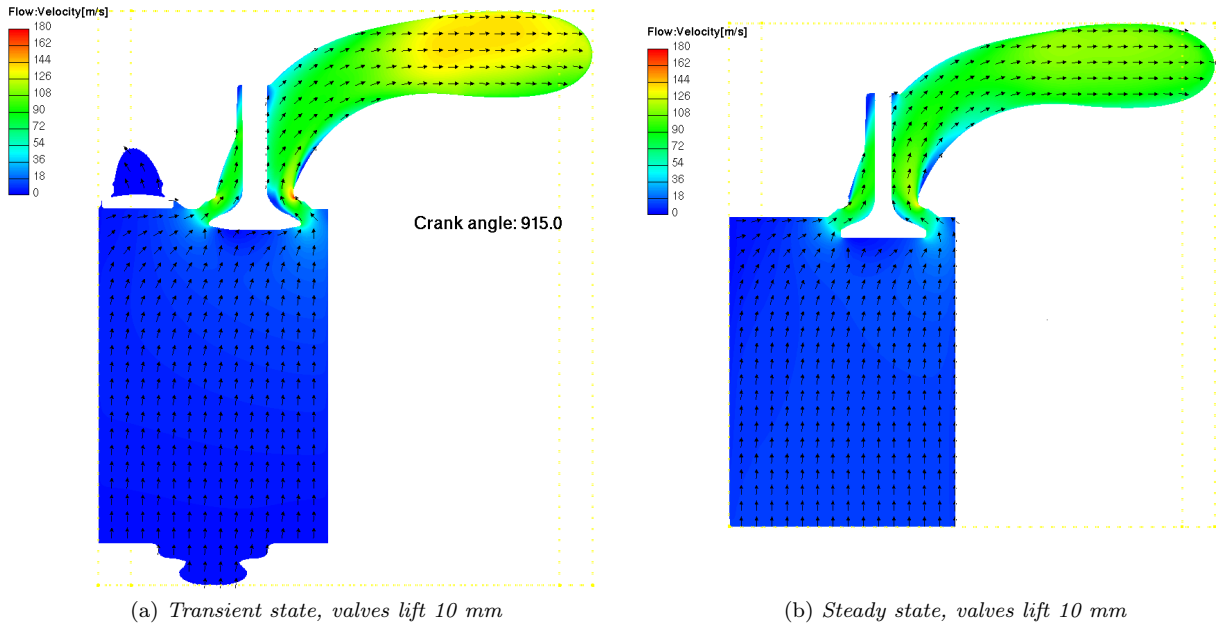


Figure 4.2.6: Compare between two approaches simulation, geometry number 16, valves lift 10 mm

4.3 Optimization and surface response

During the simulation, surface and individually response was studied to analysis the effect of every parameter in the main parameter. The sampling method Latin Hyper Cubic studied by plotting for every parameter, in Figure 4.3.2a show only one of the parameter. It plotted the surface response to optimize the process. Normally, used this step to study the effect of parameter in main parameter for example mass flow rate. In Figure 4.3.1, shows four different curves. First curve show the design of experiment for 200 design points, tested and studied three different types of optimization process.

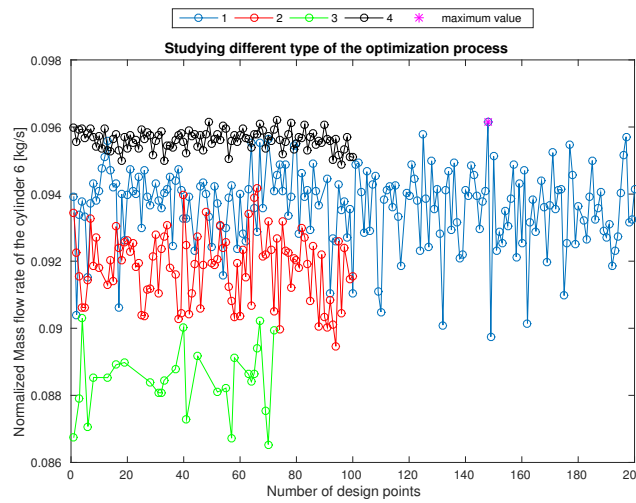


Figure 4.3.1: four different type of optimization, 1: DOE of 200 design points for cylinder 6 ;2: Optimization after looking at parameter correlation ;3: Optimization after looking at individual response; 4: Optimization after looking at optimal port with maximum mass flow rate in first curve

The first test was number 2 in the figure above created after studying the bar chart for every parameter, there could study the effect of every parameter in each other and with other parameter as Figure 4.3.4. Can see the correlation value, coefficient of determination R^2 was 0.8029 according to equation number 2.8.2 which is second order model. This value should be higher to explain the good correlation between the parameter. This value can change depend on the type of equation, number of design points and number of parameter. The curve number 3 created after studying the maximum value from the individual response as Figure 4.3.2b and this show a bad result. It showed clear that the optimization depended on effect of all parameter and not only effect of one of them. The curve number 4 gave the best result and this created after choosing the best model from first curve and built a new dimension range by multiplying the value by 10 percent up and down thereafter run a design of experiment of 100 design points.

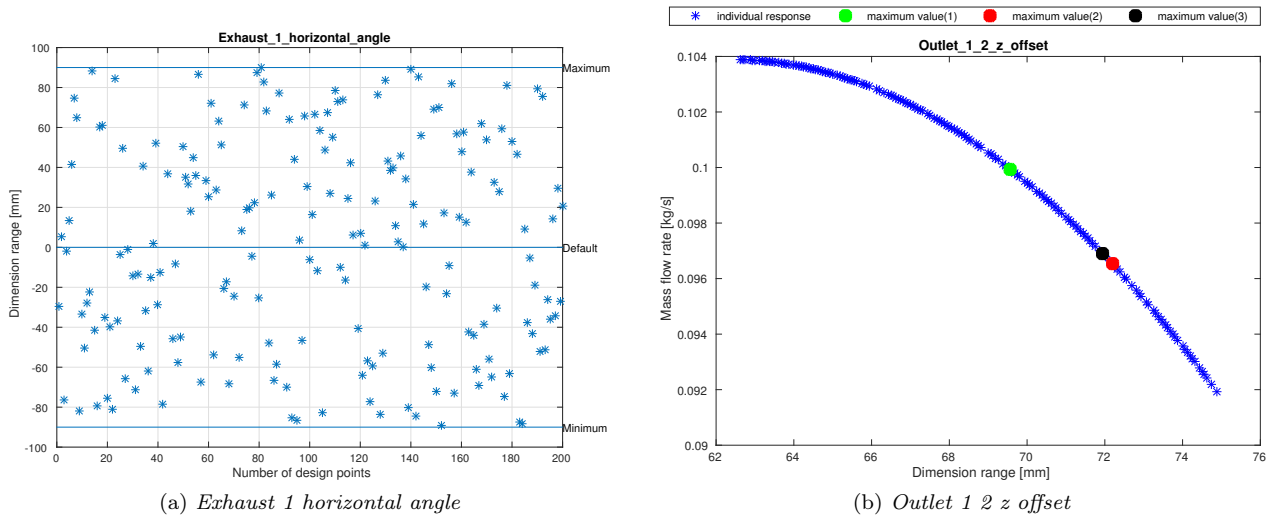


Figure 4.3.2: Latin Hyper cubic sampling method and denormalized individual response for one of the parameter, cylinder number 6, after running 200 design points result of concept 1 in Figure 4.1.2

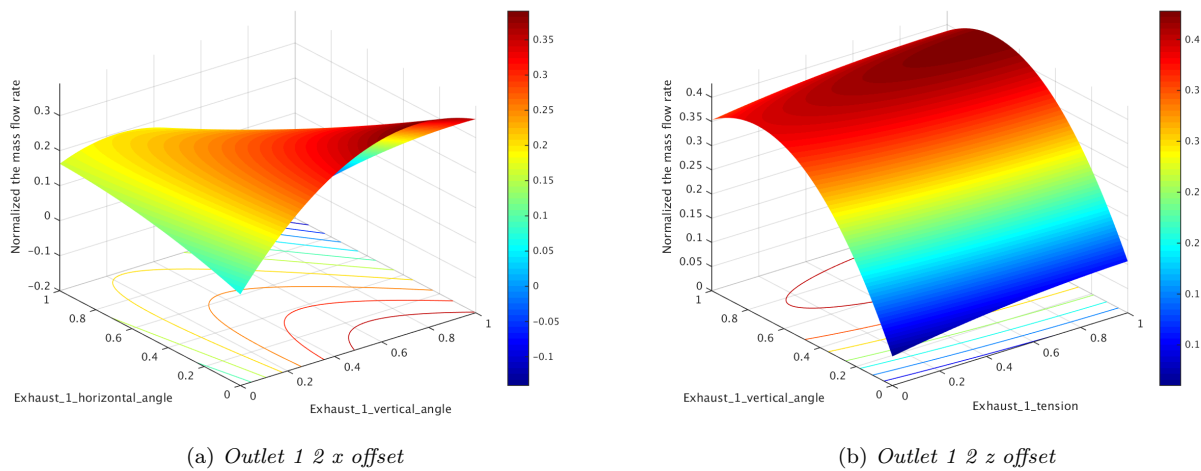


Figure 4.3.3: Normalized coefficients of surface response for two different parameter, after running 200 design points for cylinder 6, result of concept 1 in Figure 4.1.2

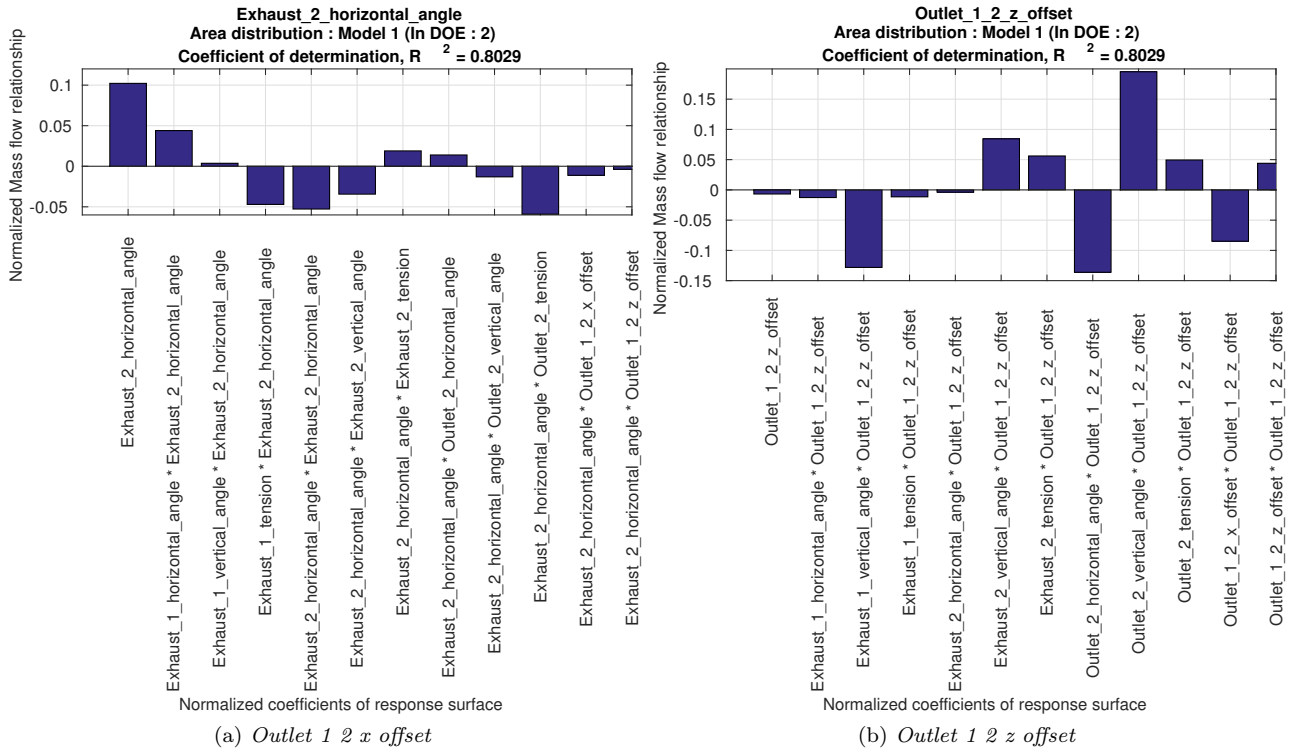


Figure 4.3.4: Normalized coefficients of response for two different parameter, after running 200 design points for cylinder 6 result of concept 1 in Figure 4.1.2

4.4 Computational time

The time for these three main processes in the project calculated to know consuming time for every process: Pre-CAD , Steady state and run a design of experiment for 100 design points in steady state , Transient state and run 10 design points in transient state simulation. The result showed in the Table 4.4.1. The result from the transient state was really a time consuming compare with the steady state. It cost more than 45 hours for every case in transient than steady state. In this study, didn't calculate the time for a moving mesh for the piston by using ESE-engine Chamber program for transient state simulation. The detailed calculation of every step in Appendix E.

Process name	CPU	Time for process per minute
Pre-CAD		1220
Steady state	48	2230
Transient state	120	6750

Table 4.4.1: Computational time table for the three process

4.5 Source of error

Errors originating from deviations and simplifications made during modelling may have occurred. This chapter describes some sources of errors while running the project.

4.5.1 Geometry

The valves and valves seat used in steady state was not the same as the valves in transient state. Therefore it made approximation in this study by accepting that different in valves because of project time. This approximation didn't effect the result, when it studied the phenomena of fluid direction at valves lift 1 mm , 10 mm and 14.6 mm.

4.5.2 Arbitrary contact

The arbitrary contact was important to have at the same position there the port cut. This contact was between the new mesh port and the rest. Otherwise, there was no connection between the new mesh port and the rest. The connection did by using a edge node, a circle contents of nodes which build a connection between the mesh, or by using a surface node which consist of whole surface filled with nodes to connect the meshes..

4.5.3 Manifolds

Cut the manifolds exhaust gave bad convergence in the end of the exhaust. This showed high velocity in the end of the manifolds, to decrease it therefore it extruded the outlet of manifolds by 200 mm. The extruded pipe didn't effect the optimal exhaust port process because our studying area was the exhaust port. The extruding pipe improved the convergence in simulation.

4.5.4 Mesh type

It used to different type of mesh. Hybrid mesh for steady state and FAME HEXA for transient state. This can be effect the result during the comparison between two different method. The whole picture of fluid direction for every valves lift position is the same in every mesh. Therefor the focus was to study the position of every valves and see the efficiency of each method.

4.5.5 Turbulence modeling

It used also two different type of turbulence modeling for the steady state and transient state. The steady state used k-epsilon while used k-zeta-f for the transient state. It studied the different between them by running one case. The difference was very little, when it studied the mass flow rate.

5 Conclusions & future work

In this thesis work, exhaust port simulations were performed using AVL FIRE CFD tool. The following points are studied:

5.1 Conclusions

The conclusion of this project can be described in following points:

- A parametric robust CAD model found to be useful for studying the exhaust port.
- Steady state simulation showed similarities to transient method at high valves lift after studying the fluid phenomena and optimal port.
- An optimal unique exhaust port for every cylinder position and manifolds shape.
- Various optimal unique exhaust port for low valves lift 1 mm and high valves lift 10 mm.
- An automatic method created for transient state simulation.
- No difference for an optimal exhaust port with different pressure boundary condition and temperature.

5.2 Future work

During analysing and visualizing the the result of this project , several points for further investigation are described below:

- Studying the swirl inside the exhaust the port. By plotting cut section of the model found there are swirl when the fluid after coming out from the two inlet channels. This swirl increase the pressure losing inside the exhaust and effect the life time of the turbo blade negative.
- Evaluate the transient state simulation by using the turbine energy or pump work. This can be done by using a porosity in AVL-FIRE.
- Apply method to optimize port design, compare the result.
- Integrate the optimal components and run simulation for whole the system: (Inlet, cylinder, exhaust port) & (manifolds and turbo). Because it can found the optimal port, exhaust and cylinder, to check if they gave same result, when it run the optimal components together.
- Optimize the parametric exhaust port with manifolds. It observed during the project that the outlet of the exhaust created edge with the inlet of the manifolds, because the manifolds was an dead object, without any parameter in this project.
- Optimize the valves seat , at 1 mm valves lift by using a parametric model. Because it observed that the most losing in pressure at blow down face, at small lift of the valves. That is because the valves seat has an angle 30 which stopped streaming the fluid, It can be a good to test with angle 45.

- Develop the optimization process with Matlab code. The code is a good step for product development, need to optimize it to be able to study the new range of every parameter for the step two of port optimization.

References

- [1] S. AB. History of Scania AB. *Wikipedia entry* (December 2012). URL: <https://sv.wikipedia.org/wiki/Scania>.
- [2] S. AB. Scania in brief. *Wikipedia entry* (February 2015). URL: www.scania.com/scania-group/scania-in-brief/.
- [3] M. Blomberg. Development of a parametrization methodology to aid simulation driven design. *Master's thesis* (2015), 58. DOI: LIU-IEI-TEK-A--15/02261SE.
- [4] K. Dvorak. Management of parametric CAD model by external tools. *Book* **390** (2013), 617–620. DOI: 10.4028/www.scientific.net/AMM.390.616.
- [5] C. J. Fredrik Jones. Diesel Combustion Modeling and Simulation for Torque Estimation and Parameter Optimization. *Master's thesis* (2008), 1–10. DOI: LiTH-ISKY-EX--08/4072SE.
- [6] C. Gatti. Design of experiments. *Doctor thesis* (2011), 68–72. DOI: 978-3-319-12197-0.
- [7] S. S. Girimaji. Partially-Averaged Navier-Stokes Model for Turbulence: A Reynolds-Averaged Navier-Stokes to Direct Numerical Simulation Bridging Method. *Journal article of Applied Mechanics* (February 2004), 1–9. DOI: 10.1115/1.2151207.
- [8] L. C. Jacques Goupy. Introduction to Design of Experiments with JMP Examples. *Book* (2007). DOI: 978-1599944227.
- [9] M. H. K. Hanjalic´ M. Popovac. A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD. *Journal article* (April 2004), 1047–1051. DOI: 10.1016/j.ijheatfluidflow.2004.07.005.
- [10] M. T. Kristian Amadori. Flexible and robust CAD models for design automation. *Research Article* **29** (2012), 180–194.
- [11] G. Kukec. Decreasing Engineering Time with Variable CAD Models: Parametric Approach to Process Optimization. *Technical journal* (2015). DOI: 10.14733/cadconfP.2014.172-174.
- [12] J. Liker. The Toyota Way. *Book* **350** (2014), 150–250. DOI: 9780071392310.
- [13] B. Lindgren. Formula for calculation in the channel evaluation of Swirl. *Scania Technical Paper* (2003), 1–15. DOI: 7015417.
- [14] V. Luu. Methodology development for parametric CAD modeling in CATIA V5 to aid simulation driven design using turbine volute as a case study. *Master's thesis* 67 (2015). DOI: LIU-IEI-TEK-A--15/02147SE.
- [15] D. C. Montgomery. Design and Analysis of Experiments. *Book* (2001), 10–50. DOI: 978-1118146927. URL: <https://www.moresteam.com/toolbox/design-of-experiments.cfm>.
- [16] H. physics. Diesel engine picture. *encyclopedia entry* (2010). URL: <http://hyperphysics.phy-astr.gsu.edu/hbase/thermo/diesel.html>.
- [17] A.-f. program. AVL-fire program manuel (2014).
- [18] D. S. SARA JOHANSSON. Simulation Driven Product Development How it can be combined with Lean Philosophy to achieve increased product development efficiency. *Master's thesis* 67 (2012), 1–5. DOI: E2011:021.
- [19] R. Stone. Introduction to internal combustion engines. *Book* (2012), 1–25, 291–321. DOI: 978-0-230-57663-6.
- [20] Y. Wang. Numerical Studies of Flow and Associated Losses in the Exhaust Port of a Diesel Engine. *Doctoral thesis* (2013), 1–73.

A Requirement structure matrix

Requirement Structure Matrix					
	Description	Requirement/Constraint	Parameter	Info (Why)	Value
1	Minimal offset below in z direction from the outlet of the exhaust port.	requirement	Outlet_1_2_z_offset	This is because of the water jacket volum is below the canales, it is important to cool the canales	0 mm
2	Minimal offset above the outlet section in z direction.	Optional	Outlet_1_2_z_offset	There is an empty volume , that it can be use.	15 mm
3	Minimum value for how much material is left between holes in the geometry	requirement	Min_material_thickness	This is for manufacturing process	5,5 mm
4	Minimal distance to the left and right of the outlet of the exhaust port	requirement	Outlet_1_2_x_offset	Because there is two screws in cylinder blocks	5 mm
5	Minimal offset distance for outlet in the y direction to move back	requirement	Outlet_1_2_y_offset	The distance from the cylinder to the cylinder head is 120 mm but it can be change the distance	30 mm
6	maximum opening of valves, distance to combustion plane.	requirement	Valve_opening_dist	This will be apply only for steady state case	10 mm
8	Distance from the center of the cylinder where the fasteners are positioned to cylinder head	requirement	Outlet_1_2_y_offset	This is fixed distance	120 mm
9	maximum opening of valves, distance to combustion plane.	requirement	Valve_opening_dist	This is maximum value of valves lift opening	15 mm

B Parameter limitation structure matrix

Parameter limitation Structure Matrix		
	Description	Requirement/Constraint
1	No parameter for the dimension of the valves and change geometry	requirement
2	No parameter for the inlet 1 diameter of the exhaust port	requirement
3	No parameter for the inlet 2 diameter of the exhaust port	requirement
4	No parameter for the outlet diameter of the exhaust port	requirement
5	No parameter for changing the diameter of the valves seat	requirement
6	No parameter to control the distance between inlet 1 and inlet 2	requirement
7	Used only one type of valves C2	requirement
8	No parameter for the cylinder height	requirement

C Concepts matrix

Concepts Matrix												
No	Dimension name	Type	Number of concepts									
			1	2	3	4	5	6	7	8	9	
1	Exhaust_1_horizontal_angle	Parameter	x	x	x	x	x	x	x	x	x	
2	Exhaust_1_vertical_angle	Parameter	x	x	x	x	x	x	x	x	x	
3	Exhaust_1_tension	Parameter	x	x	x	x	x	x	x	x	x	
4	Exhaust_2_horizontal_angle	Parameter	x	x	x	x	x	x	x	x	x	
5	Exhaust_2_vertical_angle	Parameter	x	x	x	x	x	x	x	x	x	
6	Exhaust_2_tension	Parameter	x	x	x	x	x	x	x	x	x	
7	Outlet_1_section fixed	Fixed		x								
8	Outlet_1_horizontal_angle	Parameter				x	x	x	x			
9	Outlet_1_vertical_angle	Parameter					x	x	x			
10	Outlet_1_tension	Parameter					x	x	x			
11	Outlet_2_horizontal_angle	Parameter	x	x	x						x	
12	Outlet_2_vertical_angle	Parameter	x	x	x						x	
13	Outlet_2_y_tension	Parameter	x	x	x							
14	Outlet_1_x_offset	Parameter	x	x	x	x	x	x	x	x		
15	Outlet_1_y_offset	Parameter		x		x	x	x	x	x		
16	Outlet_1_z_offset	Parameter	x	x	x	x	x	x	x	x		
17	Outlet_2_x_offset	Parameter	x	x	x		x	x	x	x	x	
18	Outlet_2_y_offset	Parameter									x	
19	Outlet_2_z_offset	Parameter	x	x	x	x	x	x	x	x	x	
20	Valves opening	Fixed	x	x			x	x		x	x	
21	Section diameter in canal 1 & 2	Fixed					x	x	x	x	x	
22	Section diameter in canal 1 & 2	Parameter						x				
23	Section movement in canal 1 & 2	Parameter							x	x	x	

D Range of parameter

Range of parameter						
DOE Parameters						
Parameter name	Parameter unit	Minimum value	Default value	Maximum value	Step size	
Exhaust 1 horizontal_angle	deg	-90	0	90		
Exhaust 1 vertical_angle	deg	70	100	120		
Exhaust 1 tension		0,7	1,36	1,9		
Exhaust 2 horizontal_angle	deg	-90	0	90		
Exhaust 2 vertical_angle	deg	70	80,5	100		
Exhaust 2 tension		0,9	1,4	2		
Outlet 3 horizontal_angle	deg	0	4,75	5		
Outlet 3 vertical_angle	deg	0,5	1,4	2		
Outlet 3 tension		0,5	1,52	2,2		
Outlet 1 2 x offset	mm	-5	0	5		
Outlet 1 2 z offset	mm	62,5	62,5	75		

No	Exhaust_1_horizontal_angle	Exhaust_1_vertical_angle	Exhaust_1_tension	Exhaust_2_horizontal_angle	Exhaust_2_vertical_angle	Exhaust_2_tension	Outlet_3_horizontal_angle	Outlet_3_vertical_angle	Outlet_3_tension	Outlet_1_2_x_offset	Outlet_1_2_z_offset
1	-25.2	81	0.92	36	84.4	1.43	0.4	1.43	0.87	-0.2	71
2	61.2	101	1.08	-79.2	80.2	1.47	1.5	1.5	0.59	1.4	74.25
3	-7.2	114	0.84	-86.4	76	1.78	1.2	0.89	1.18	-2.6	65.25
4	-21.6	96	1.23	75.6	81.4	1.25	1.8	1.25	0.92	3.6	63.25
5	-82.8	77	1.06	39.6	71.8	1.01	3.3	1.28	0.98	4.2	64
6	-86.4	118	0.87	79.2	72.4	1.21	2.3	1.21	2	2.4	70.25
7	7.2	119	0.82	-43.2	98.8	1.82	3	1.19	1.28	-4.8	68.25
8	3.6	102	0.94	-18	82.6	1.65	4.3	1.46	1.42	-0.8	68
9	-54	76	0.8	18	83.2	1.05	2.6	1.01	1.15	1.2	73.25
10	-18	120	1.37	-39.6	95.8	1.03	3.5	1.49	1.11	-4.4	68.75
11	57.6	106	1.54	32.4	89.8	1.1	0.8	1.59	0.8	-3.8	70
12	-39.6	94	1.85	-21.6	79	1.58	3.6	1.31	2.06	-4.6	74.75
13	14.4	113	1.32	28.8	98.2	0.94	2.8	1.7	1.45	3.4	67.5
14	-3.6	90	1.88	14.4	74.8	1.69	1	1.43	1.01	-1.2	71.25
15	0	86	1.42	-10.8	78.4	1.34	3.8	1.61	1.04	-2.2	63
16	21.6	100	1.56	-35.2	97.6	1.41	2.1	1.67	0.94	4.4	72
17	-68.4	73	1.28	-61.2	88	1.32	1.1	1.82	1.21	3.2	69.75
18	32.4	117	1.13	7.2	89.2	1.54	2	0.62	1.89	0.2	72.5
19	-10.8	80	1.01	21.6	82	1.14	2.2	1.22	1.38	-3.2	69.5
20	50.4	75	1.3	-46.8	85	1.38	0.2	1.34	1.52	-0.6	73.5
21	90	97	1.59	-50.4	91.6	1.8	3	0.95	1.96	3	65.5
22	64.8	72	1.25	-14.4	88.6	1.36	3.4	1.04	1.04	-4	64.5
23	-28.8	74	1.8	57.6	85.6	1.56	1.3	1.52	1.83	-4.2	66.5
24	-43.2	92	0.89	54	77.2	1.49	1.4	0.86	1.76	-0.4	66.25
25	10.8	79	1.47	25.2	70.6	1.98	4	1.07	1.93	2	64.75
26	36	88	1.76	-7.2	80.8	1.74	4.4	1.13	1.08	2.6	71.75
27	-79.2	78	1.52	0	94.6	1.89	2.4	0.83	2	2.2	72.75
28	-57.6	109	0.72	-68.4	94	0.99	3.9	1.1	2.17	-3.6	70.5
29	-61.2	93	1.73	-7.2	95.2	1.32	0.7	0.77	1.32	4.8	74
30	43.2	116	1.2	3.6	73.6	1.67	4.7	1.73	1.66	4.6	70.75
31	28.8	107	1.04	-64.8	91	0.97	3.2	1.4	2.03	-1	66
32	54	83	0.77	10.8	86.8	1.96	4.1	0.56	1.62	-3.4	75
33	68.4	85	1.35	-3.6	90.4	1.27	5	1.25	0.84	1	65.25
34	-14.4	103	1.64	43.2	79.6	1.08	0.1	0.68	1.69	0.4	73.75
35	-7.2	105	1.61	-38.8	73	1.63	0.5	1.97	1.79	0	64.25
36	-36	112	1.9	72	93.4	1.76	3.7	1.94	0.7	-1.8	65.75
37	39.6	108	1.71	-54	92.8	1.87	4.5	1.76	1.72	1.6	63.5
38	82.8	111	1.78	64.8	86.2	1.23	1.7	1.64	2.1	-2	73
39	79.2	95	1.49	46.8	83.8	1.85	0.3	1.37	0.91	-3	65
40	-64.8	99	1.44	-32.4	71.2	1.71	1.6	1.16	0.67	-2.4	67.25
41	7.2	110	0.99	-36	87.4	2	4.9	1.79	1.35	2.8	71.5
42	75.6	91	1.68	-57.6	76.6	1.12	0.8	1.58	0.77	-2.8	66.75
43	-46.8	104	1.66	61.2	96.4	1.6	3.1	0.53	2.13	5	68.5
44	25.2	89	1.16	50.4	92.2	1.52	1.9	1.91	0.64	4	72.25
45	-32.4	82	1.18	68.4	97	1.93	0.9	0.71	0.53	-1.4	62.75
46	86.4	115	1.4	82.8	74.2	1.16	4.2	0.74	1.86	0.6	63.75
47	46.8	87	1.11	-75.6	75.4	1.91	4.8	1.85	1.49	-1.6	74.5
48	-75.6	98	0.96	86.4	100	1.19	4.6	0.65	1.55	3.8	67
49	-50.4	84	0.75	-82.8	77.8	1.88	2.7	1.88	2.2	1.8	67.75
50	18	71	1.83	90	99.4	1.3	2.9	0.98	0.6	0.8	69

E Computational time

Studying for Preparation of CAD for DOE of 100 design points				
Time	Type	Phase	Activity	Description
1	hour	Pre - CAD	Gather information	Meeting with designer at Scania and collect information about the exhaust port, Interface of the exhaust port, gathering information to determine requirements and design boundaries.
1	hour	Pre - CAD	Define the parameters	Meeting with supervisor and defined all parameter that require for this design to build the first concept design.
7	hours	Pre - CAD	Modelling a construction geometry	Designs the geometry by using Catia program, measure all dimensions from the engine Dw6, started with building a sketch model thereafter surfaces.
2	hours	Pre - CAD	Define the parameters in Catia	Defined all parameters in the geometry that required for this design that defined the area of the design and gives robustness model.
20	minutes	Pre - CAD	Generating a design of experiments matrix	Used the interface between the Excel file and Catia to generate a matrix of design points. An example for 100 points in this study used in this time studying.
Total time for proces		11 hours and 20 minutes	1220 minutes	

Studying for CFD simulation of DOE of 100 design points by using steady state method				
Time	Type	Phase	Activity	Description
30	minutes	CFD - simulation	Build a macro	Build a macro to define all boundary selection in the geometry: inlet, outlet, exhaust port, manifolds, valves, valves seat and chamber.
3	hours	CFD simulation	Mesh dependency study	Study the mesh with the minimum number of cell, that gives a good result and convergence. Good quality mesh without any negative volume cell and define the number of boundary layer, thickness and y-plus should be between 30-100.
2	hours	CFD - simulation	Define simulation setting file	Defined all parameter that need in simulation setting file, Turbulence model, boundary selection, number of iteration, under relaxation, Total pressure in inlet and outlet, Smoothing in the model by defining initial condition.
30	minutes	CFD - simulation	Scripting and prepare a mapp for CFD simulation	Used a bash script in linux to simulate in CFD. This step consist of a map which include, DOE design points, script files and a template with definition of the simulation setting file.
13	hours	CFD - simulation	Simulation	The time for simulation in steady state is about 32 minutes for every geometry in this case, by running four at the same times
1	hours	Post - processing	Matlab code	Analyzing the result by using the Matlab code definition to study every parameter by using the surface response, individual response method to visualize the effect of every parameter in the mean parameter mass flow.
30	minutes	Post - processing	Parameter studying of discharge coefficient	Study the discharge coefficient for every design point in the matrix.
Total time for proces		20 hours and 30 minutes		
		1230 minutes		

Studying for CFD simulation of DOE of 10 design points by using transient state method				
Time	Type	Phase	Activity	Description
30	minutes	CFD - simulation	Build a macro	Build a macro to define all boundary selection in the geometry: inlet, outlet, exhaust port, manifolds, valves, valves seat and chamber of the cutting exhaust port.
2	hours	CFD simulation	Mesh dependency study	Study the mesh with the minimum number of cell, that gives a good result and convergence. Good quality mesh without any negative volume cell and define the number of boundary layer, thickness and y-plus should be between 30-100. Used the FAME HEXA meshing. The number of cells is about one and half million cells.
1	hour	CFD - simulation	Define the arbitrary contact	Defined the arbitrary contact area position, And defined it by using the edge nodes or surface nodes.
30	minutes	CFD - simulation	Run a DOE	Run a DOE of best 10 geometry from steady state simulation for all cutting port.
17	hours	CFD - simulation	Scripting and prepare a mapp for CFD simulation	Used a bash script in linux to simulate in CFD. This step consist of a map which include, DOE design points, script files and a template with definition of the simulation setting file, thereafter run scripting for meshing with FAME HEXA. A script include also a step for assembling of moving mesh. It needs 1 and half hour for every design points. Builds also a mapp for runing the files.
90	hours	CFD - simulation	Simulation	The time for simulation in steady state is about 50 hours for every geometry in this case, by running five at the same times. It used a cluster with the number of processor is 84 CPU to solve it.
1	hours	Post - processing	Matlab code	Analyzing the result by using the Matlab code definition to study every parameter by using the surface response, individual response method to visualize the effect of every parameter in the mean parameter mass flow.
30	minutes	Post - processing	Parameter studying of discharge coefficient	Study the discharge coefficient for every design point in the matrix.
Total time for proces		112 hours and 30 minutes		
		6750 minutes		