

14 November 2019



The Effect of Intake Tumble Flow on Efficiency and Emissions for a GDI Production Engine

YOUSOUF BOUKRA / 1700055

Major Project Seminar Paper

Submitted to the Faculty of engineering

Automotive/Motorsport Department | University of Wales trinity St David

Table of Contents

List of Abbreviations	1
General Abstract:.....	2
Technical Abstract:.....	2
1. Introduction:.....	2
I. The main aim	3
II. The second aim.....	3
2. Overview:	3
IN-Cylinder Charge Motion:.....	4
3. Methodology:	6
4. Results:	8
5. Conclusion:	9
References.....	9
Appendices.....	11

List of Abbreviations

GDI: Gasoline direct injection
CIDI: Compression ignition direct injection
CAE: Computer-aided engineering
IC: Internal Combustion
ICE: Internal combustion engine
CAD: Computer-aided Design
RE: Reverse Engineering
CFD: Computational fluid dynamics
CAM: Computer-aided manufacturing
BSFC: Brake specific fuel consumption
PM: Particulate Matter
VVT: Variable valve timing
PFI: Port fuel injection
SI: Spark ignition
VVT-i: variable intake and exhaust valve timing
CC: Combustion Chamber

Table of Figures

Figure 1 Classification of charge motions (Kim et al. 2019).	4
Figure 3 CFD flow of filling and tumble ports (Schogl et al. 2011).	5
Figure 2 different charge motion Port Design (Schogl et al. 2011).	5
Figure 4 Structured Development Process Plan	6
Figure 7 NextEngine 3D scanner HD (cc mold scan)	7
Figure 6 CC molding process.....	7
Figure 5 Intake port molding process	7
Figure 8 Intake port CAD model (Solidworks).....	8

General Abstract:

In order to protect human health and the environment, automotive manufacturers around the world need to develop new technology to reduce exhaust pollutant gases emitted by vehicles. Numerous inventions have been introduced to this industry, such as small size engines, turbochargers, and direct injection which significantly enhances fuel economy (Adomeit et al. 2011). Exhaust gases are the result of a combustion between fuel and air and consequently, air flow must be improved to allow the engine to achieve better emissions and a higher efficiency rate. Specifically, the inducted air passing through the intake port of cylinder-head must be developed further.

This paper will discuss the development of cylinder-head intake port, by the utilisation of computer-aided engineering (CAE) tools, which are crucial to modern-day engineering. An intake port will be analysed for further optimisation, by running experimental testing and investigation. Aiming to present a better cylinder head port design to increase engine efficiency and reduce emission gases.

Technical Abstract:

Automotive manufacturers around the world have been and still frequently researching for suitable techniques to improve IC engines efficiency, in order to reduce exhaust pollutant gases, and meet the upcoming emission legislation. The application of technologies such as downsizing, turbocharging, and direct injection, has led to a significant evolution of petrol engines with regards to emissions and thermal efficiency (Adomeit et al. 2011). Combustion and emission performance are connected to the intensity of turbulence, as well as the quality of mixture formation or the distribution of residual gases, which depends on in-cylinder charge motion. Therefore, different intake and exhaust ports geometry has been identified as a potential solution to

enhance turbulence intensity and achieve better scavenging (Adomeit et al. 2005).

In this paper, a gasoline direct injection (GDI) cylinder-head intake port will be modelled and analysed by means of CAE tools, which combines reverse engineering methods, CAD modelling and CFD analyses. Critical optimisation will be followed by investigating the effect of tumble ratio on turbulence intensity and burn rate.

This study will aim to present a developed intake port geometry that enhances tumble ratio, while maintaining charge coefficient allowably high. Thus, improving the overall IC engines emissions and efficiency.

1. Introduction:

Computer-aided engineering (CAE) methods, such as computer-aided design (CAD), and computational fluid dynamics (CFD), have become crucial to modern-day engineering. The efficiency of system and component development has been significantly improved through the application of the CAE methods, because these tools provide a deeper insight into system behaviour (Adomeit et al. 2005).

Engine and combustion process development is strictly related to cylinder-head port design. Optimising the geometry of the intake port generate the effect of first tumble peak to increase in-cylinder turbulence which highly effects the combustion process, increase the burning rate, and improves engine efficiency. Due to the intake port geometry, the development of tumble motion during intake phase is the main factor that controls the level of turbulence inside the combustion chamber, which creates the rotating flow structure of the incoming charge. Predominantly, port's top/side profiles and valve angles with respect to port angle highly contributes to the tumble enhancement for intake port. (Singh et al. 2019).

In line with the above objectives, many studies have been conducted to develop on existing engines and improve in-cylinder charge motion as well as increasing exhaust gas velocity, by focusing on cylinder-head intake and exhaust ports. Currently, this paper will critically analyse the previous designs of intake port geometries, while carrying a complete investigation on an intake port of a 1.6 GDI engine cylinder-head. The investigation will be predominantly based on CAE methods, utilising CFD simulation analysis. However, the study will also combine an experimental steady-state-flow testing on SF-300/600 Flowbench for validation process. Reverse engineering techniques which includes 3D scanning technologies and computer-aided design software, will be used to convert the base intake port geometry into a numerical entity (CAD model), for further analysis and optimisation.

- I. The main aim of this work is to understand the intake flow characteristics, and the effect of tumble ratio on turbulence intensity and burn rate, to be able to define an optimised intake-port geometry, that allows a GDI engine to operate efficiently and limit the level of emission gases.
- II. The second aim is to establish knowledge on the different ways of implementing the new reverse engineering technology, CAD modelling, and CFD simulations, to solve the engineering problem regarding port geometry design and tumble flow analyses.

2. Overview:

In the Automotive/Motorsport Industry, future emission control legislations and fuel economy are urging for a further advanced combustion concepts. Diesel engines have been displaying high efficiency in term of fuel economy, but with the cost of higher NO_x and PM (Particles) emission level, which obliged companies to use complex and expensive after-treatment systems. In contrast, gasoline engines have shown high capacity in term of meeting the strict emission standards, but with the drawback of lower efficiency than diesel engines (Berntsson et al. 2011).

As the emphasis on achieving substantial improvement in automotive fuel economy and emission level has increased, technology today's targets the combination of high efficiency engines, with ultra-low emission levels. Engineers are pushing the limits to develop engines having enhanced BSFC (brake specific fuel consumption), with the capability to comply with the future stringent emission requirements. The fuel economy of the CIDI (compression ignition direct injection) diesel engine, is widely superior compared to that of the gasoline port fuel injection engine, primarily due to diesel engine ability to operate under a significant high compression ratio, coupled with unthrottled operation as a result of direct injection. Thus, the development of SI engines that are designed to directly inject the fuel inside the combustion chamber similar to diesel engine, is an important worldwide drive of the automotive industry. Such engines has shown a great thermodynamic potential for fuel economy, transient response and cold-start HC (hydrocarbon) emission levels, which led to a large number of research and development projects aiming to understand and optimise gasoline direct injection (GDI) combustion system. That technology introduced variety of research area such as, direct fuel injection processes, spray atomization and vaporization, charge cooling, mixture preparation, and in-cylinder air charge motion (Zhao et al. 1999).

Moreover, the development of GDI system certainly benefits from the combination of other fuel efficient technologies such as, Downsizing and turbocharging, and the application of variable effective compression ratio by VVT system (variable valve timing), resulting to GDI engines reaching even higher efficiency level (Wirth et al. 2000). In 2006 a study has been completed at AVL List GmbH, the paper outlined the potential of the GDI-Turbo concepts, the key elements, technical issues, and the solutions for such achievements in details. PFI engines (port fuel injection) with turbocharging technology offers high power output and nominal torque, but they have disadvantages regarding fuel economy, exhaust emissions, and transient response as a result of their conceptual layout. In the other hand, turbocharged GDI engines with VVT technology make it possible to implement synergy effects, which aid on the achievement of a high fuel efficient engine concept suitable for low emission (Bandel et al. 2006).

GDI fuel system provides the capability of utilising an increased scavenging without sacrificing hydrocarbon emissions, due to the separated control of fuel and air. It also provides the benefit of charge cooling that reduces knock, which is necessary for turbocharging applications (Kirwan et al. 2010). Cylinder-head design for an SI engine is important as it impacts performance, fuel economy and emissions. Specifically, designing an effective intake port is being widely investigated, since it determines the volumetric efficiency and in-cylinder charge motion which effects the thermodynamic properties. More mass flowing into the engine can improve volumetric efficiency, whereas increasing intake air motion will enhance combustion rate in order to develop a higher thermal efficiency and a better emission control (Selvaraj et al. 2011).

IN-Cylinder Charge Motion:

Changing the port design geometry will change the flow characteristics such as flow coefficient and charge movement or charge motion, which is what this research paper will be directly focusing on.

Toyota has started on developing the intake flow charge movement on the GDI engine almost 15 years ago, and same in the previously mentioned studies, Berntsson et al has positively influenced many combustion parameters by mainly developing on the intake flow charge movement in project completed in 2011.

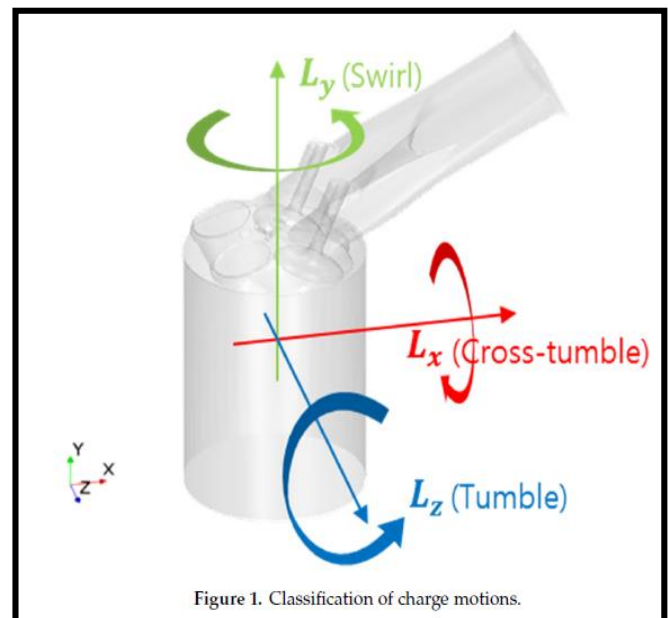


Figure 1. Classification of charge motions.

Figure 1 Classification of charge motions (Kim et al. 2019).

Figure 1, shows the different in-cylinder charge movement across the three X – Y – Z axis, as described by a number of engineers in this field. The blue arrow represent the tumble flow-charge movement direction as it rotate around the Z axis, the green arrow represent the swirl movement as the air flow around the Y axis which identify the centreline of the cylinder. Whereas the cross-tumble is a movement similar to the tumble charge but it flow around the X axis, which is parallel to the port design, therefore the flow around this Axis is not considered (Kim et al. 2019).

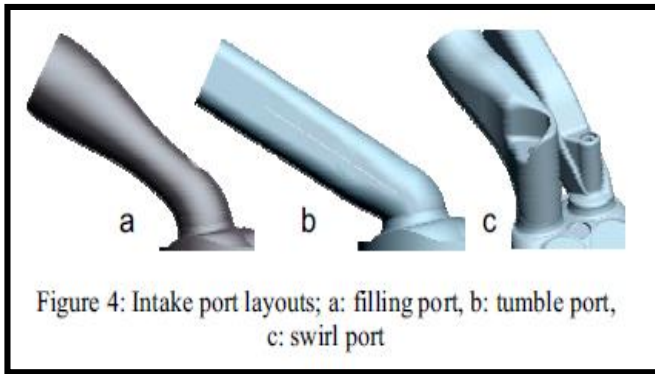


Figure 3 different charge motion Port Design (Schogl et al. 2011).

Controlling the charge movement into different direction inside the cylinder, directly depend on the port design layout or port geometry. Figure 2 shows the different port geometries with relation to the charge movement, where port layout 'a' represent the filling port which is efficient on delivering a high flow coefficient compared to other port layout, that is due to the smooth curvature of the port which allow the air to flow in high velocity and all around the port/valve edges. The disadvantage of this design is that it does not provide in-cylinder rotating movement because the flow descend directly from the top of the cylinder to the bottom of the cylinder as it is simulated in the CFD flow figure 3 'left photo' (Schogl et al. 2011).

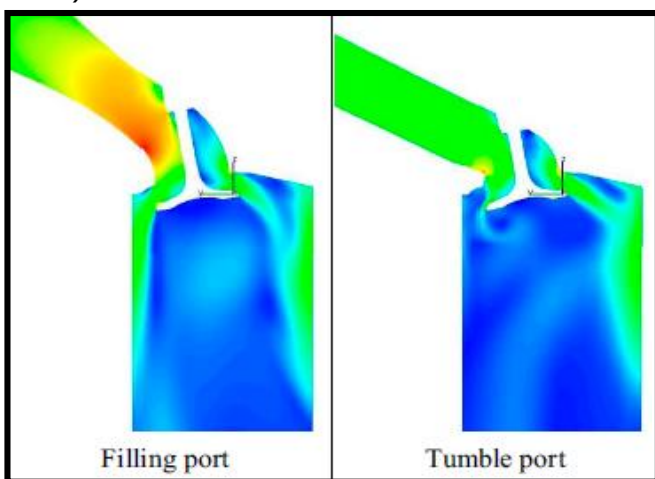


Figure 2 CFD flow of filling and tumble ports (Schogl et al. 2011).

Layout 'b' represent the tumble orientated port-design. In opposite to the filling port, the tumble port design introduce sharp angles instead of a smooth radius, and that is to be able to control flow charge into different intensity and velocity from top side to the bottom opposite side of the

port. Orientating the charge movement on one direction as shown in the CFD figure of the tumble port 'right photo, figure 3' will create the rotating movement called the tumble movement which is the charge motion discussed in this research paper (Kim et al. 2019).

The port layout 'c' represent the swirl movement, this design is not studied critically because it is mainly used on the compression ignition diesel engines, whereas this paper look at the gasoline direct injection system.

Numerous studies has already investigated intake port flow characteristics, by both experimental bench testing and CFD calculations. In 2000, The Mitsubishi GDI engine adopted a pair of upright intake ports to activate rotating in-cylinder flow, reverse tumble, and control air/fuel mixture. The optimization of intake port design for the Mitsubishi GDI engine was the key to achieve a high intensity of the reverse tumble while maintaining the charge coefficient high. Also, the refinement of port design enhances the flow in the intake sides of the intake valves, which maximized the reverse tumble motion inside the cylinder (Kuwahara et al. 2000). On the aim to develop a new V-6 stoichiometric GDI engine, Toyota highlighted the optimization of the intake port as one of the special technologies that contributed to the achievement of that successful high performance engine. Designing an intake-port that obtains both of a high flow coefficient and high tumble ratio, was a key to improve power performance. Dual VVT-i (variable intake and exhaust valve timing) systems were employed to decrease pumping loss and increase the expansion ratio, in order to improve fuel consumption (Sadakane et al. 2005).

Schogl's et al. (2011) completed a study on an intake-port of high performance three-cylinder gasoline engine, the investigation was implemented in the development of a GDI engine concept where different intake port

geometries were developed and analysed , utilising CAD and 3D CFD simulation methods. The primary objective was to present the ideal possible charge movement without losses in the cylinder charge, therefore the port layout was designed to generate tumble flow. The study predicted an emission reduction as a result of combustion behaviour improvement, and a better performance with the optimised cylinder charging through intake port geometry. It was clearly stated that the turbulence level influences the combustion behaviour.

Another research has been done in attempt to enhance fuel efficiency and meet the emission reduction requirements, it mentioned that direct gasoline injection combined with turbo charging and downsizing is a cost effective technology to meet the above objectives. The turbulence induced by the in-cylinder air motion can influence efficiency, therefore, the intake-generated flow field was varied for a direct injected turbo charged concept, aiming to evaluate if further increase in tumble will potentially lead to higher efficiency when compared with baseline model. The study combined two series of testing, one aiming to evaluate tumble in low to medium load and engine speed by mainly focusing on efficiency, and the second test was performed at high load region to examine any negative consequences of increased tumble (Berntsson et al. 2011).

A transient study was conducted on a straight shape intake port geometry by adding an upper vane, the modified intake port generated a much larger peak value for tumble ratio compared to the original peak. This proved that airflow is well organized and the momentum provided by the modified intake port is also well preserved. The development of that tumble ratio, break up through the end of compression and transfer the energy stored as a tumble into kinetic turbulence energy. This revealed that the turbulence kinetic energy in the modified design is twice as high compared to the original model when tumble breaks up. Thus, the

combustion quality will be greatly improved (Qi et al. 2010).

3. Methodology:

The focus of this project will be on the generation of tumble flow by optimising the intake port geometry. Critical analyses of previous similar papers will be conducted prior and while working on the development of the GDI cylinder-head intake port, which will be approached using a structured development process plan shown in Figure 4.

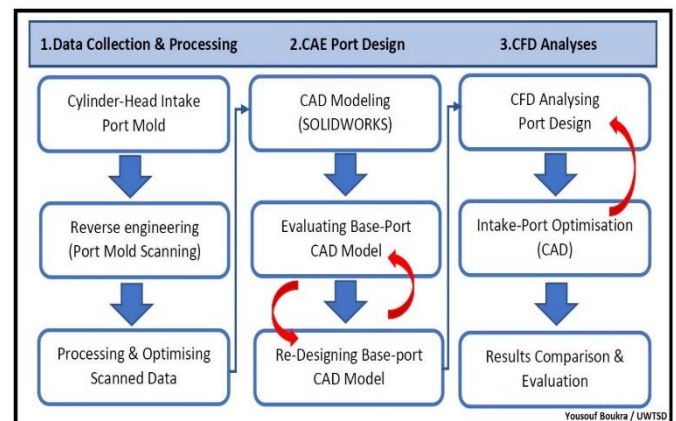


Figure 4 Structured Development Process Plan

The methodology performed in order to complete this project consisted of three main phases:

First stage is obtaining an internal geometry of the intake port and combustion chamber (CC), in order to proceed to the reverse engineering work and allow physical scanning, this phase is referred to as *data collection & processing* in the development plan. The work started by forming a silicon model to represent the negative geometry of the intake port and the combustion chamber as shown in figure 5 and figure 6, After 24 hours, the silicon parts were ready to be removed and scanned. To transform the silicon shapes to a digital entity, NEXTENGINE 3D SCANNER was used (figure 7). Post scanning the parts, the points cloud scanned data were used to develop a clean surface by importing the points cloud files into Geomagic_Studio_12 Software (Appendix A, B, and C). However, due to the limited capacity

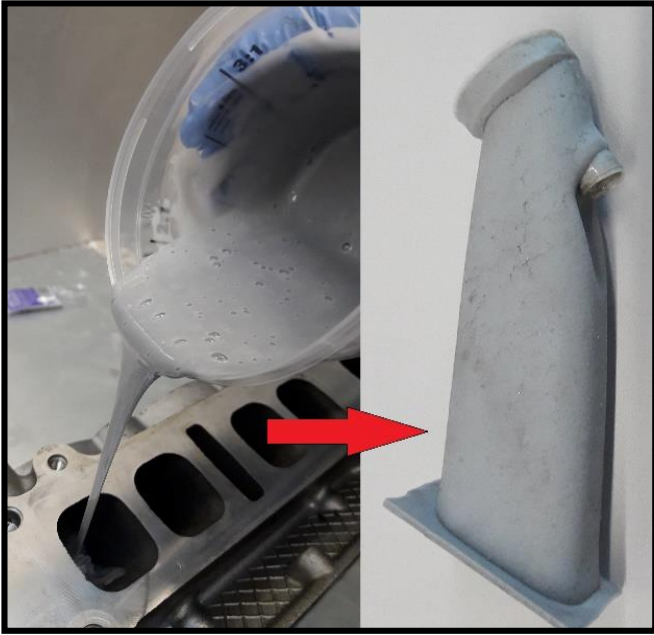


Figure 7 Intake port molding process

of the utilised scanner on capturing critical points in the silicon parts, some of the curvatures were optimised using multiple rectifying techniques available inside Geomagic_Studio_12 such as, hole filling, bridging... etc (Appendix D, E, F, G, H, and I).

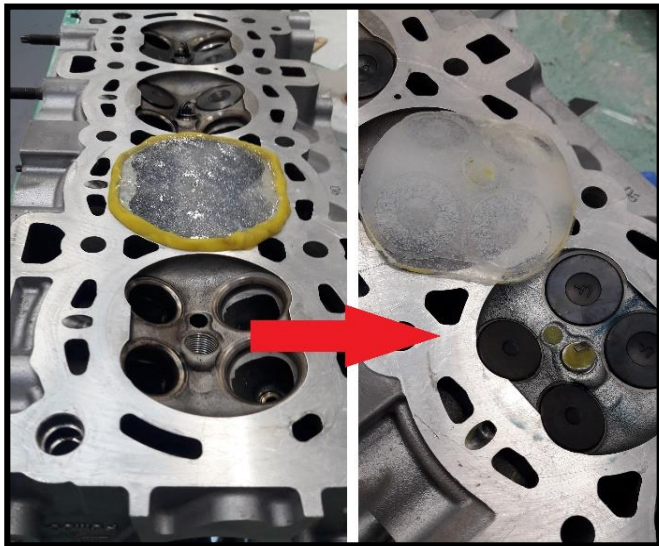


Figure 6 CC molding process

Second stage was developing a geometrical model in CAD using Solidworks with the aid of Geomagic-for-Solidworks Ad-In (integrated software), this phase is referred to as *CAE port design*. The acquired CAD model (figure 8) was then evaluated against the actual scanning data using GOM-Inspect software, by aligning

both of the scan data and designed geometries, the software highlights the deviation points and calculate the accuracy of the Intake port CAD model (Appendix J). A re-design process will be carried to alter on the models for better accuracy, before the re-evaluation process. This procedure cycle will be repeated till the accuracy is high enough for further work.

Final stage of the project (*CFD analyses*) will be the development of computational mesh utilising ANSYS software to simulate and analyse the intake flow (tumble ratio, velocity magnitude...etc.). The obtained data will be Investigated and used for the development of the new port geometry. Post assessing the flow performance of the base-port, an optimised port will be developed using Solidworks to modify on the base-port CAD model. The optimised port will run into a CFD ANSYS analyses, and the results of both base-port and optimised port will be compared and evaluated.



Figure 5 NextEngine 3D scanner HD (cc mold scan)

This methodology of using reverse engineering (RE) techniques to digitalise real life data for further CFD/CAE investigation and development, has been widely recognised as an important technique in the product design cycle. RE allows to create surface models by three-dimensional scanning techniques, which will make it possible to redesign and manufacture variety of parts and tools in a short development period. Thus, the application of reverse engineering significantly decrease the

manufacturing cost and gain speed for product realization system (Kumar et al. 2013).

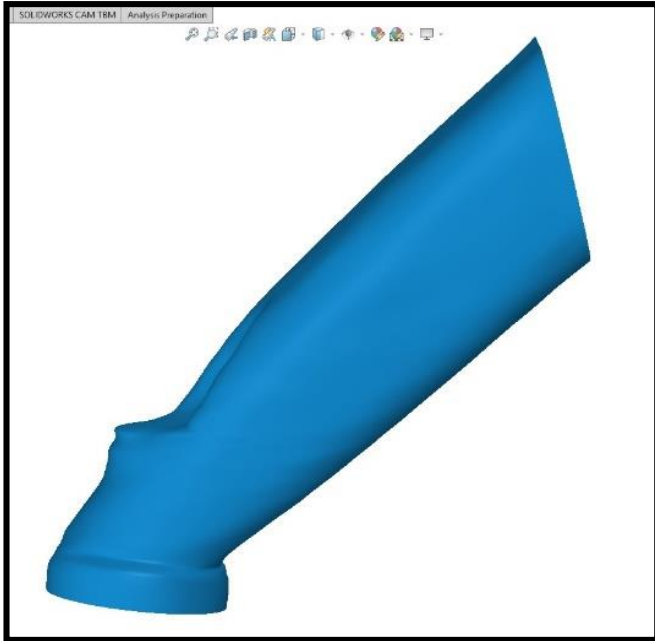


Figure 8 Intake port CAD model (Solidworks)

Bialy et al. 2014, followed this methodology in the simulation work of intake-port of a diesel engine with compressed natural gas direct injection. The computational model was developed using three stages: a. three-dimensional scanning of the real object using ZScanner 700; b. development of geometrical model in CAD using CATIA V5; c. the development of computational mesh in the AVL_Fire software. Similar methodology was adopted by Masi, Artico, and Gobbato. 2015, in the study of measurement of the intake and in-cylinder flow field to investigate the reliability of CFD steady-state simulations for actual engines. In this research however, the computation have been performed by using release 9 of CD-Adapco Star CCM+ finite volume code. In 2017, De Lima et al, completed a project on combustion analysis in IC engine adapted with pre-chamber, utilising three dimensional model obtained from the same reverse engineering method discussed in the previous studies. The numerical simulations were performed via the commercial software Converge-CFD.

4. Results:

The results of this study will point the key parameters of intake-port geometry optimisation for tumble enhancement. The predicted results of this study so far are between 5 to 10% increase in term of tumble ratio, while keeping the same percentage of flow co-efficient to prevent performance losses. Also, the presented paper will show the benefits of reverse engineering, CAE, and computational fluid dynamic tools on cylinder-head intake port development specifically, and IC engine combustion development generally.

Previous similar studies has shown positive results in term of CAE/CFD intake port development. Qi et al, 2010. Stated that the results of the completed study shows that the magnitude of tumble ratio in the modified intake port has twice the ratio of the original port.

A positive effect on efficiency was noticed and tumble influenced multiple combustion parameters, in a similar study performed on a direct injection turbocharged downsized gasoline engine. Different level of tumble also aid on reducing combustion retard due to reduced knock sensitivity, and increased EGR-tolerance for medium loads (Berntsson et al. 2011).

An intake port geometry was developed in a recent study by CFD analysis to increase the tumble ratio on PFI engine, the result revealed a great achievement of 45% increase in tumble performance (Singh et al. 2019).

5. Conclusion:

Tumble flow of an intake port is an important parameter if a further development can be achieved, as this will highly contribute to the overall engine efficiency and reduce emission gases.

Reverse engineering, CAE, and CFD tools, are being widely used to develop on the intake tumble flow, multiple projects has positively affected engine performance by developing on a tumble-orientated intake port geometry.

This Research critically investigate some of the previous papers on this subject, and will continue looking at more recent papers regarding tumble flow development and CAE methods.

References

- Adomeit, A., Hopp, M., Schmidt, A., Lang, O. and Schmitt, P. (2005) 'CAE-Based port development and flow design for SI engines', *SAE International Journal*.
DOI: 10.4271/2005-01-0243
- Adomeit, A., Jakob, M., Pischinger, S., Brunn, A. and Ewald, J. (2011) 'Effect of intake port design on the flow field stability of a gasoline DI engine', *SAE International Journal*.
DOI: 10.4271/2011-01-1284
- Bandel, W., Fraidl, G., Kapus, P., Sikinger, H. and Cowland, C. (2006) 'The Turbocharged GDI Engine: Boosted Synergies for high fuel economy plus ultra-low emission', *SAE International Journal*.
DOI: 10.4271/2006-01-1266
- Berntsson, A., Josefsson, G., Ekdahl, R., Ogink, R. and Grandin, B. (2011) 'The effect of tumble flow on efficiency for a direct injection turbocharged downsized gasoline engine', *SAE International Journal of Engines*, 4(2), pp. 2298-2311.
DOI: 10.4271/2011-24-0054
- Bialy, M., Wendeker, M., Magryta, P. and Czyz, Z. (2014) 'CFD model of the mixture formation process of the CNG direct injection engine', *SAE International Journal*.
DOI: 10.4271/2014-01-2575
- De Lima, B., Thiriet, A., Teixeira, A. and Valle, R. (2017) 'Three-dimensional model obtained from reverse engineering for analysis of combustion in internal combustion engine adapted with pre-chamber', *SAE International Journal*.
DOI: 10.4271/2017-36-0346
- Kirwan, J., Shost, M., Roth, G. and Zizelman, J. (2010) '3-Cylinder turbocharge gasoline direct injection value solution for low CO₂ and NO_x emissions', *SAE International Journal*.
DOI: 10.4271/2010-01-0590
- Kumar, A., Jain, P. and Pathak, P. (2013) 'Reverse Engineering in Product Manufacturing: An Overview', *DAAAM International Scientific Book 2013*, pp. 665-678.
DOI: 10.2507/daaam.scibook.2013.39
- Kuwahara, K., Watanabe, T., Tanada, H., Ueda, K. and Ando, H. (2000) 'Intake-port design for Mitsubishi GDI engine to realize distinctive in-cylinder flow and high charge coefficient', *SAE International Journal*.
DOI: 10.4271/2000-01-2801
- Masi, M., Artico, L. and Gobbato, P. (2015) 'Measurements of the intake and in-cylinder flow field to investigate the reliability of CFD steady-state simulations for actual engine', *SAE International Journal*.
DOI: 10.4271/2015-24-2404
- Qi, Y., Liu, H., Midkiff, K. and Puzinauskas, P. (2010) 'A feasible CFD methodology for gasoline intake flow optimization in a HEV application – Part 2: Prediction and optimization', *SAE International Journal*.
DOI: 10.4271/2010-01-2238
- Sadakane, S., Sugiyama, M., Kishi, H., Abe, S., Harada, J. and Sonoda, Y. (2005)

'Development of a new V-6 high performance stoichiometric gasoline direct injection engine', *SAE International Journal*.

DOI: 10.4271/2005-01-1152

Schogl, O., Edtmayer, H., Schmidt, S., Leiber, S. and Schabetsberger, T. (2011) 'Design of a tumble-orientated intake port layout for a gasoline combustion process used in power sport application', *SAE International Journal*.

DOI: 10.4271/2011-32-0589

Selvaraj, B., Sridhara, S., Indraprakash, G., Senthilkumar, A. and Pangaonkar, A. (2011) 'Effects of intake port geometry on the performance of an SI engine', *SAE International Journal*.

DOI: 10.4271/2011-32-0506

Singh, A., Poonia, S., Jalan, A., Singh, J. and Kumar, N. (2019) 'Intake and Exhaust Ports Design for Tumble and Mass Flow Rate Improvements in Gasoline Engine', *SAE International Journal*.

DOI: 10.4271/2019-01-0763

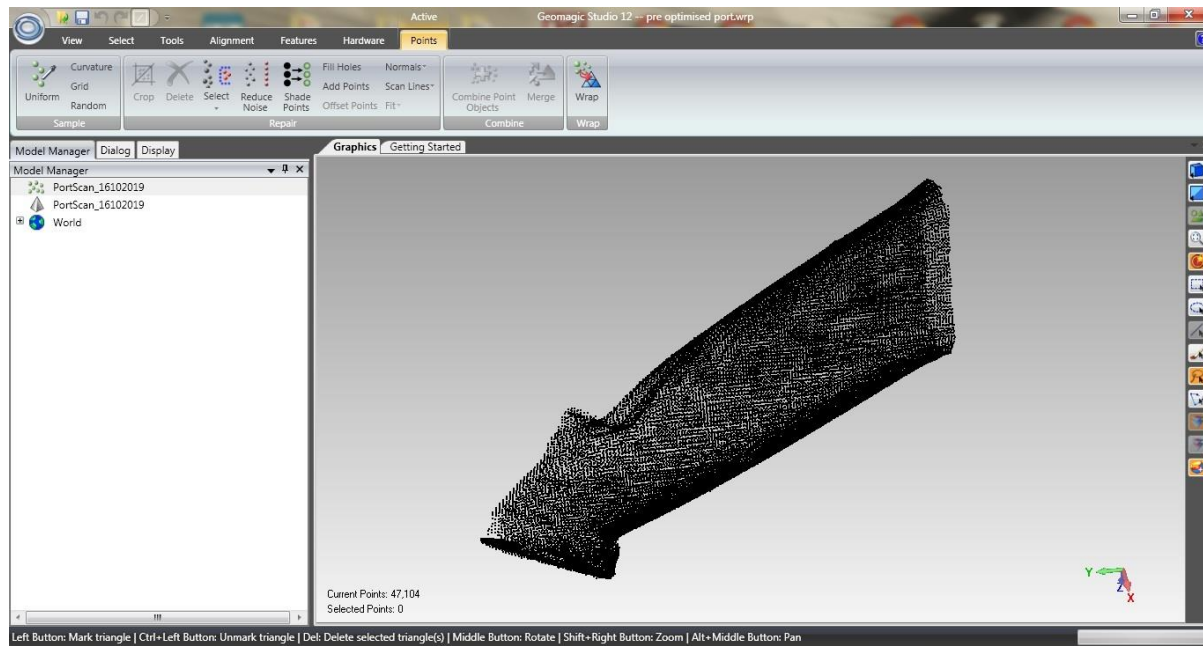
Wirth, M., Mayerhofer, U., Piock, W. and Fraidl, G. (2019) 'Turbocharging the DI Gasoline Engine', *SAE International Journal*.

DOI: 10.4271/2000-01-0251

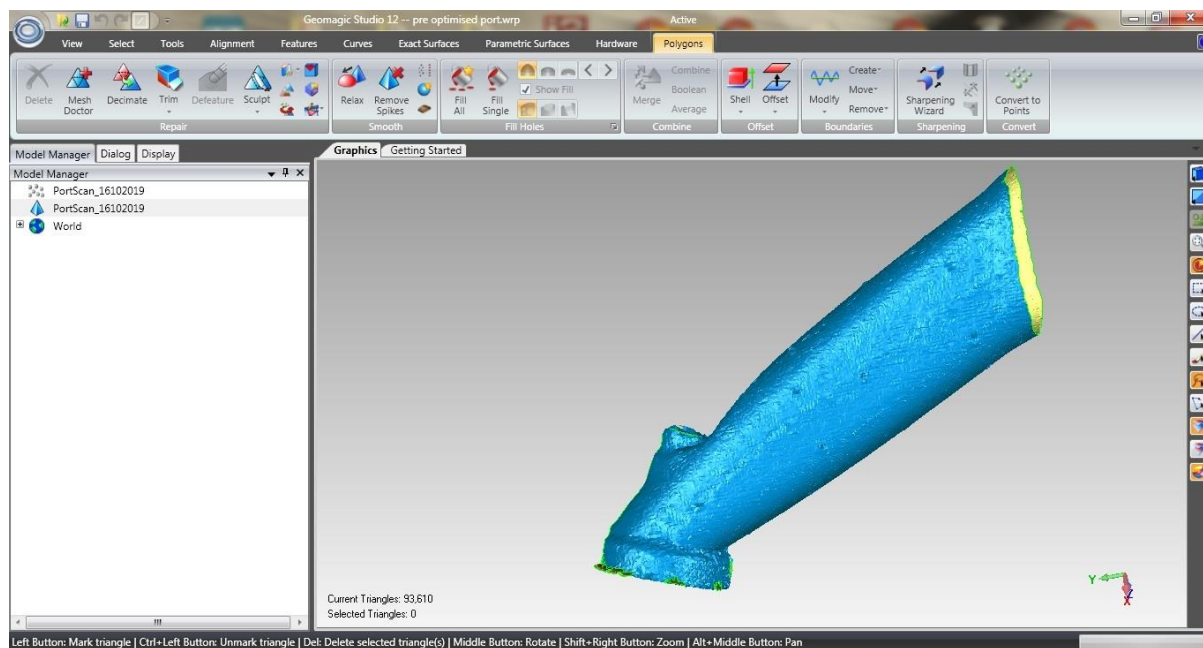
Zhao, F., Lai, M. and Harrington, D. (1999) 'Automotive spark-ignition direct-injection gasoline engines', *Progress in energy and combustion science*, 25(5), pp. 437-562.

Appendices

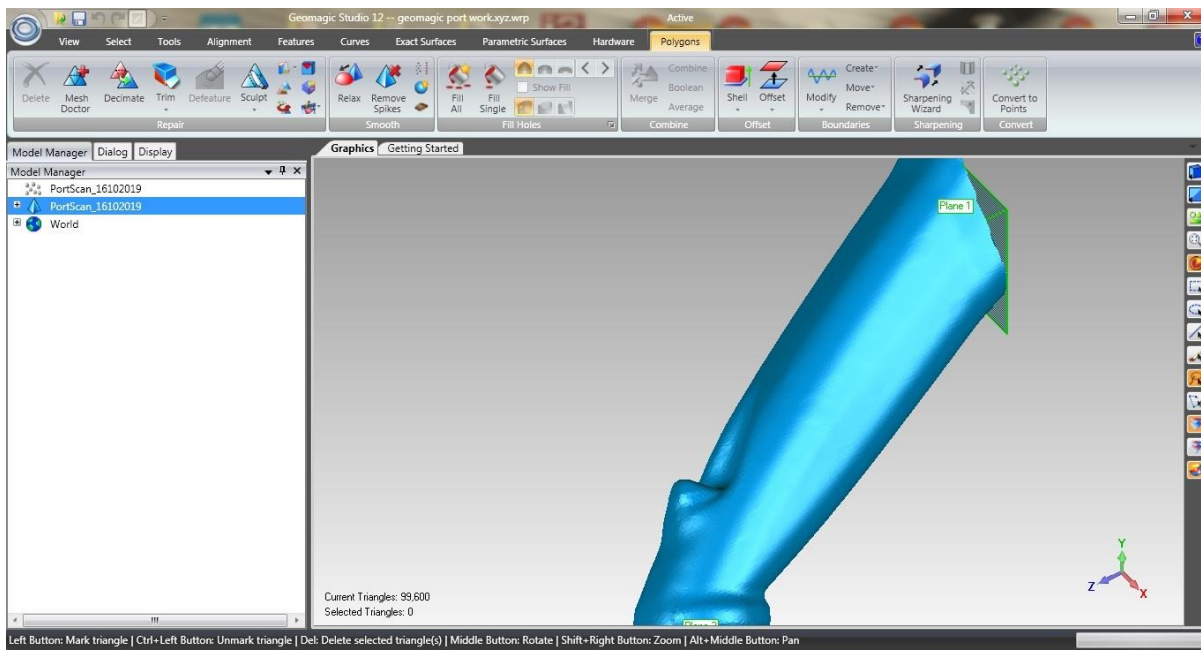
Appendix A:



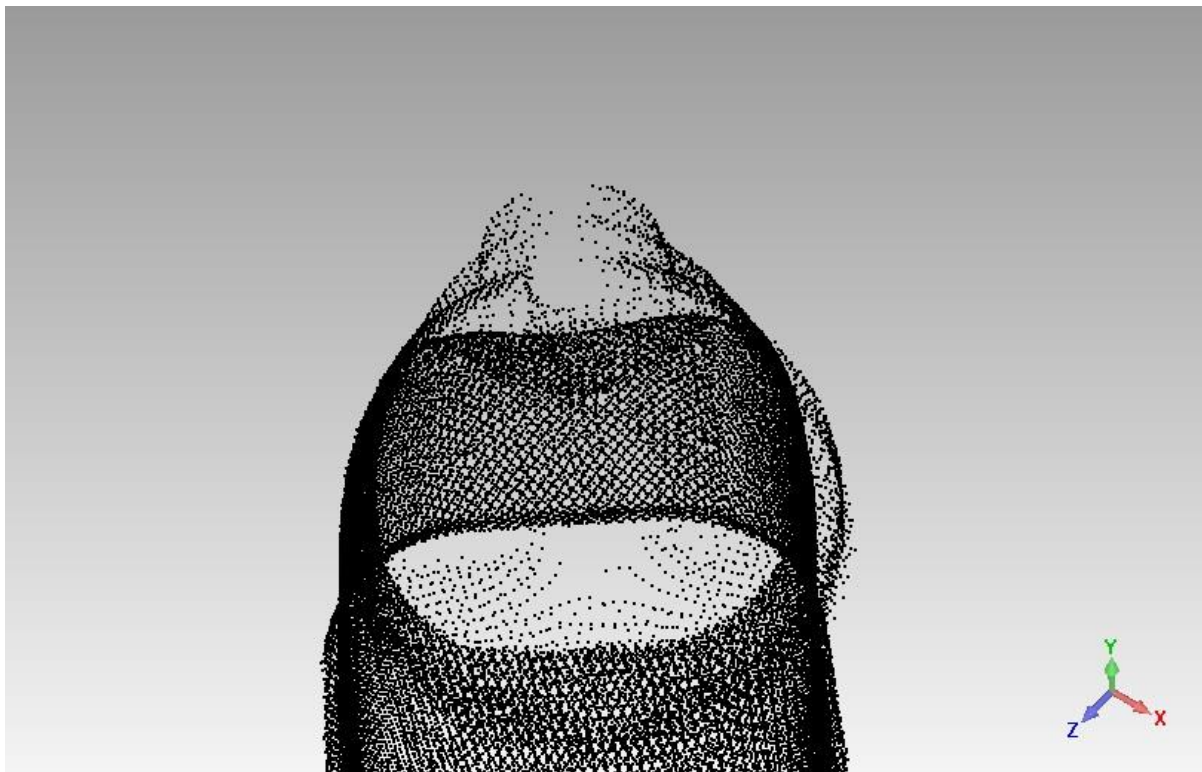
Appendix B:



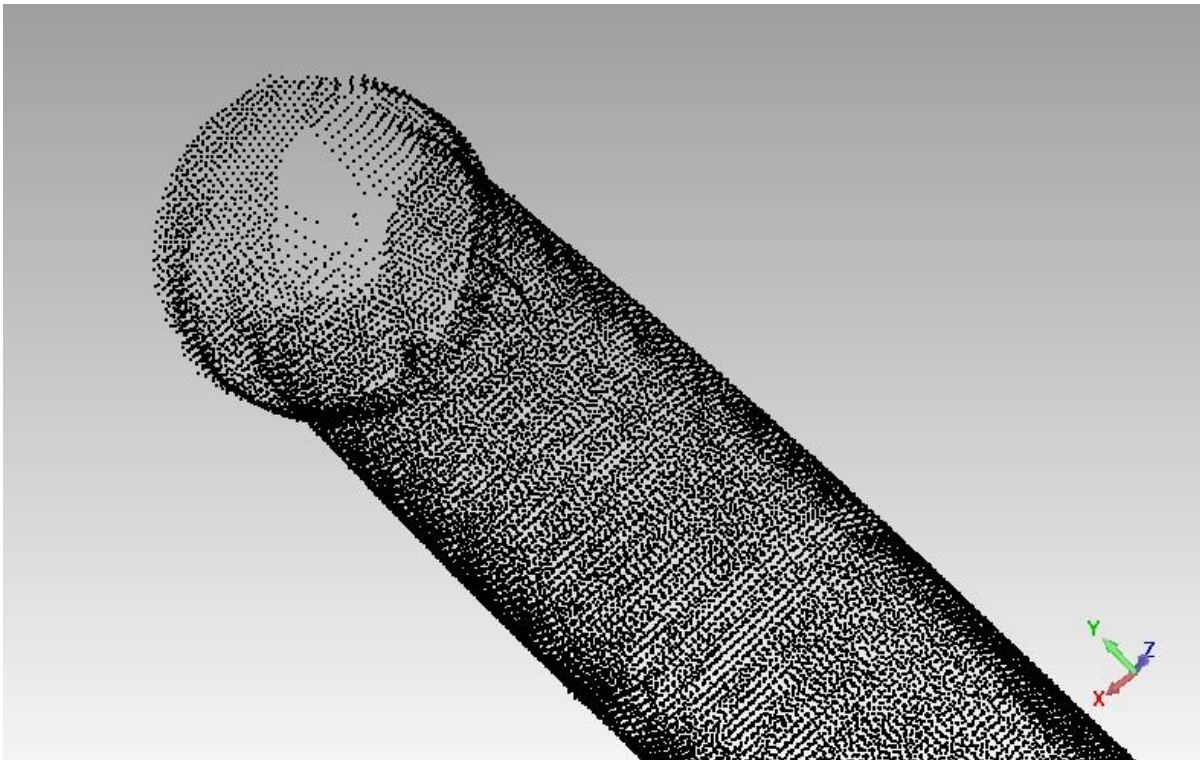
Appendix C:



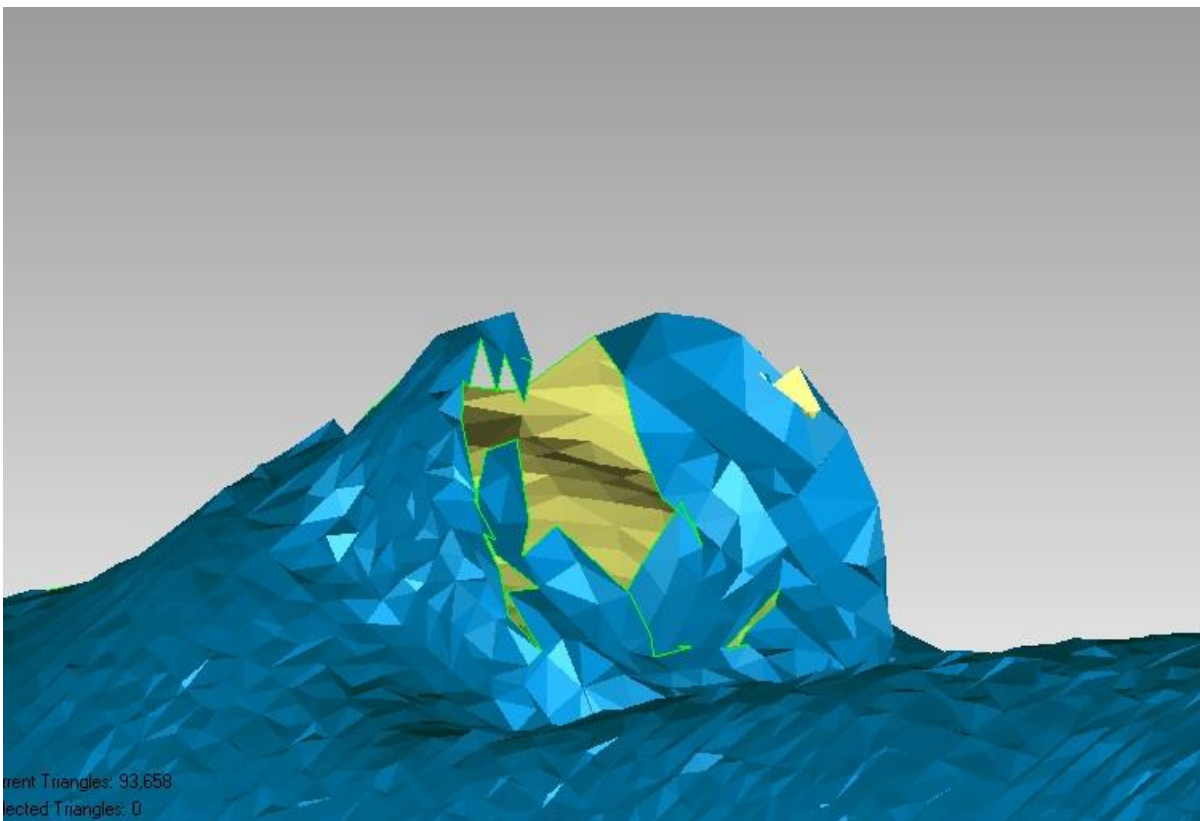
Appendix D:



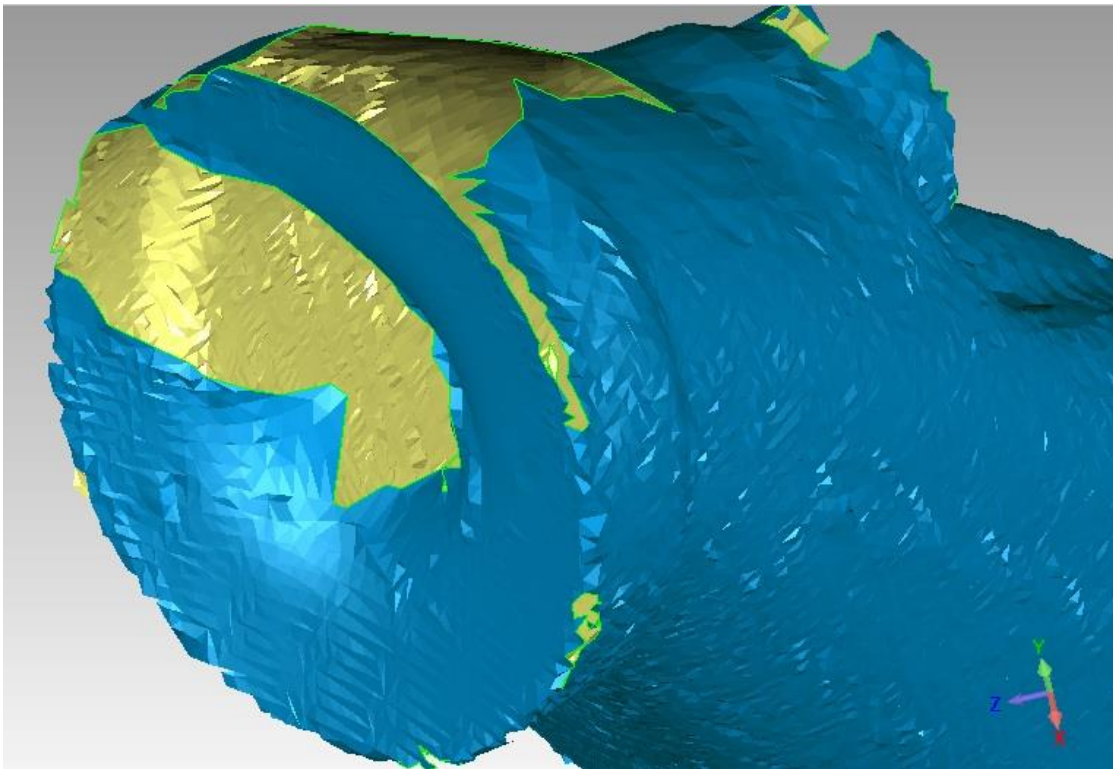
Appendix E:



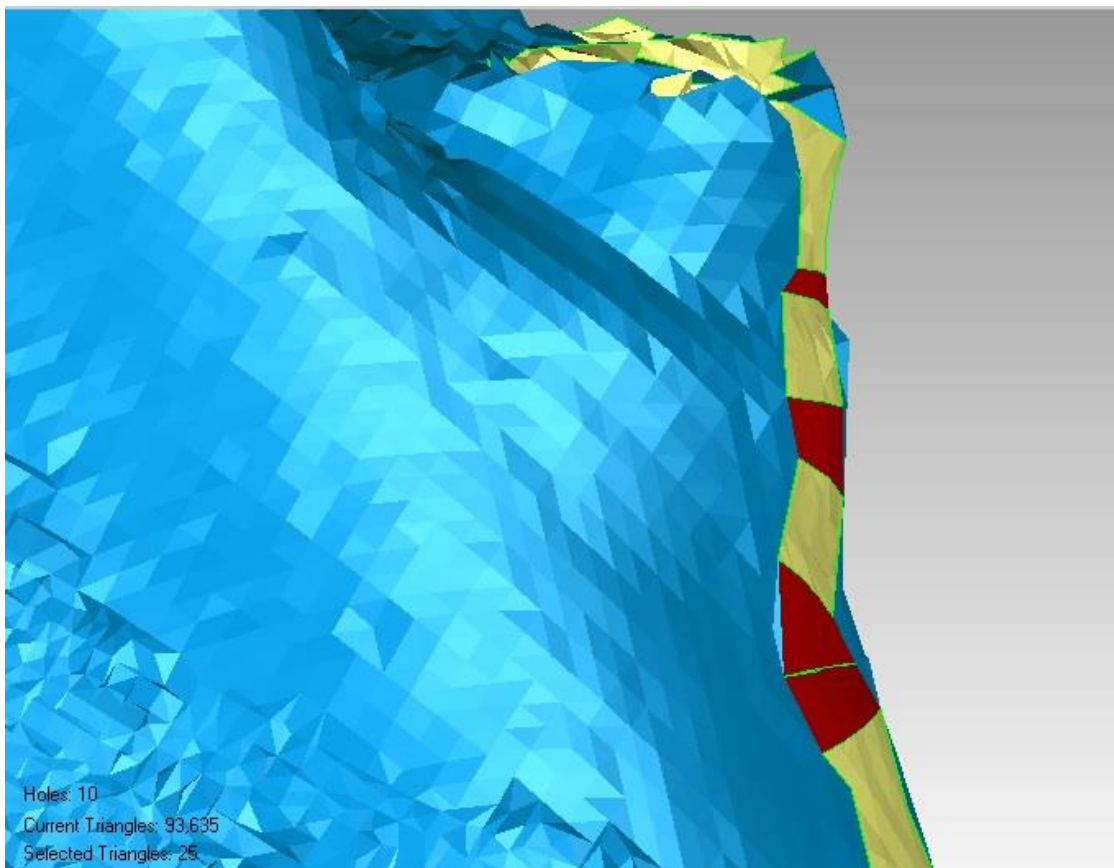
Appendix F:



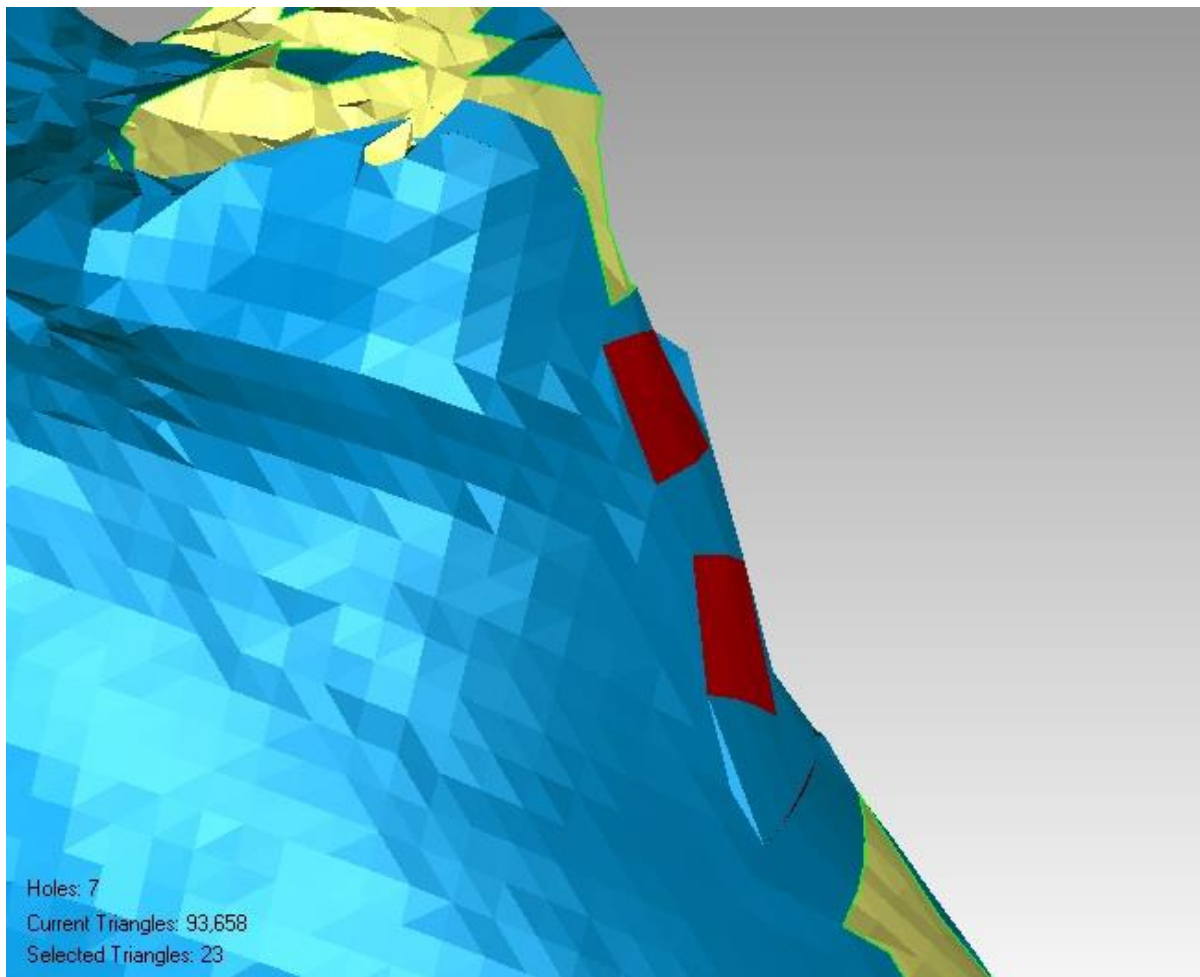
Appendix G:



Appendix H:



Appendix I:



Appendix J:

