Armour College of Engineering
ILLINOIS INSTITUTE OF TECHNOLOGY

RESEARCH PROJECT

DESIGN AND ANALYSIS OF NEW FUEL INJECTION SYSTEM IN THE INTAKE MANIFOLD

JON DOMINGUEZ

Supervised by Dra. Carrie Hall

Chicago, Illinois

08/18/2018

ACKNOWLEDGMENT

I would like to sincerely thank Illinois Institute of Technology for the opportunity to complete my master's degree and provide me with new tools and engineering skills which undoubtedly will be helpful for not only my future career but also for my personal development.

I would like to extend my thanks to Dra. Carrie Hall, my supervisor and mentor, who give me the opportunity to work with her, who is a recognized professional and an incredible professor. I truly appreciate his time, concern, confidence and advice. This project would not be able without his support and encouragement.

This project success also required the guidance and assistance from my partners of Advanced Control Engine Laboratory of Illinois Institute of Technology. I am grateful since their willingness to contribute towards an overall goal have been essential to achieve successful results. I am grateful to Jorge Pulpeiro, Vipul Vagga and Alvaro Blazquez for them help and support along the term of the project. I would like to express especial thanks to Vicente Chapa, who has been my fellow in the last part of the project.

Last but not least, I wish to thank my family for their unconditional support and belief shown in me and my work, without they I would not be able to achieve anything. Also, I would like to truly thanks Maialen Maestre, my girlfriend who has always accompanied me through this way and is an essential support in my life.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ABSTRACT

The aim of this project is design and analysis of the new fuel injection system in the intake manifold of the 2.0 Liter TDI Common Rail BIN5 ULEV Engine. This research is part of a long-term research study carried out by Dra. Carrie Hall and the Advanced Engine Control Laboratory research team of Illinois Institute of Technology, in which ‘’Nonlinear Model-based Control Strategies for Advanced Fuel-Flexible Multi Cylinder Engines’’ is studied.

In this paper the research work is summarized and explained. The first part of the paper includes the theoretical framework in order to understand the current situation of the elements and resources used in this study. This section contains a description of the engine used to carry out the long-term research, the air manifold of the engine explained in more detail, the chosen fuel injectors and the basics to understand the fluid simulation software used in the research.

Then the procedure followed for analyzing and designing the new system is detailly explained. In this practical approach, first, the fluid analysis using a CFD simulation software for the actual manifold is carried out and explained in the report. Then, the new design with fuel injector is analyzed. Finally, the conclusions of the research and the guidelines for the future work are set up.

CHAPTER

1. INTRODUCTION

This report contains and summarizes the research done for the design and analysis of new fuel injection system in the intake manifold of the IIT's Advanced Engine Control Laboratory test engine. The 2.0 Liter TDI Common Rail BIN5 ULEV Engine is the actual engine used by Dra. Carrie Hall's research team.

The purpose of this paper is to set up the bases of the preliminary design and analysis of the new fuel ports in the intake manifold and help future work that will be carried out by Engine Laboratory team. In particular, this report explains the procedure of computational fluid analysis of the actual manifold and the design for a new fuel system.

This research work is part of the research project on the ''Nonlinear Model-based Control Strategies for Advanced Fuel-Flexible Multi Cylinder Engines'' [1]. The long-term objective is to reduce the use of fossil fuel in transportation sector and to lower the pollutant emissions levels. In the pursuit of this goal, the research team investigates the dynamics of advanced combustion model in order to increase the efficiency of dual-fuel diesel engines and creates model-based control techniques that could be applied into this nonlinear and complex system.

Advanced combustion strategies can ensure a higher engine efficiency and lower exhaust emissions, comparing with conventional CI[1] combustion. Among these advanced solutions, Reactivity Controlled Compression Ignition (RCCI) achieves these objectives by using two fuels with different reactivity to increase combustion delays periods and promote premixing. RCCI uses port-injected gasoline fuel as low reactivity fuel and directed injected high reactivity diesel fuel.

The dynamics of this advanced combustion mode are not completely understood, in particular in multi-cylinder engines with alternative fuels. The main challenge in this research is analyzing the nonlinear dynamics, modeling the representative parameters and controlling these complex systems.

[1] CI: compression ignition engines.

2. SCOPE

The scope of the research is the design, analysis and implementation of the low reactivity fuel injection system into the manifold of the actual diesel engine. In order to evaluate the efficiency of the actual and future system, a CFD[1] analysis using ANSYS® Fluent[2] of the manifold model is carried out.

The figure 1 shows the stages of the current work in this area:

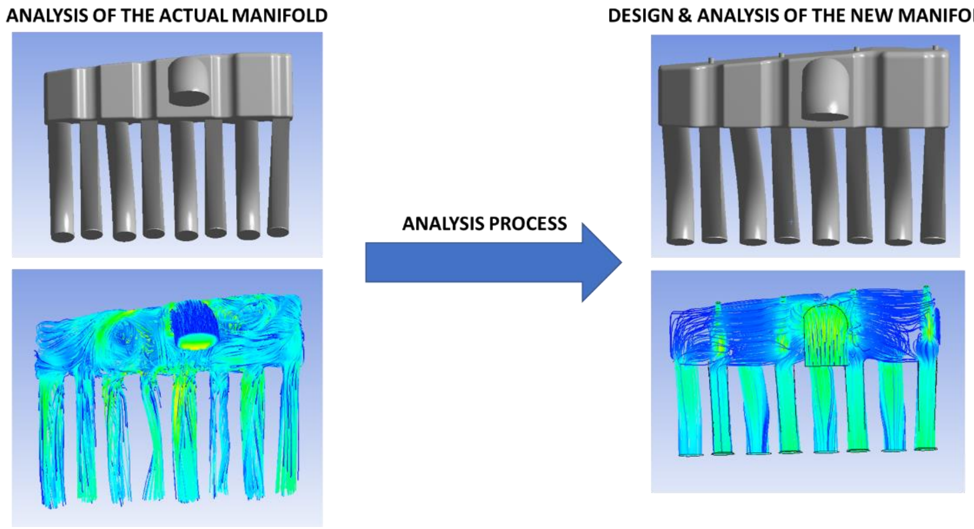


Figure 1. Research current state.

In this paper, the work done up to the current state of the research is explained in detail. As the bases are set up, the future work and research in this area are also explained. As the long-term goal of the advanced combustion research project, the final conclusions of the analysis of the new port system would be able to ensure the following statements:

- Extract and model the key parameters of the air and fuel intake new system.
- Incorporate in MoTec[3] the tracking of these parameters.
- Incorporate the new manifold in the overall performance of the dual-fuel advanced combustion diesel engine.

[1] Computational Fluid Dynamics: branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involves fluid flows.

[2] ANSYS® Fluent: advanced CFD simulation software. The software used in the analysis of the manifold.

[3] MoTec: engine management and data acquisition system. The control software installed and used in the Advanced Engine Control Laboratory.

3. BENEFITS OF THE PROJECT

3.1. Technical and Economic Benefits

The research project benefits are potentially increase the efficiency of fuel-flexible diesel engine by up to 20% and reduce considerably the harmful pollutant emissions of the engine. Once when the complex control for the advanced combustion system is understood, the RCCI strategy would be fully develop and implement in actual automobile models, with the principal aim of reducing the massive fossil fuel use in the vehicles along the world.

RCCI operates in conditions where optimal premixed combustion eliminates the fuel rich area where PM[1] is formed. Thus, local temperatures decrease, and NO_x[2] emission reduce.

A huge world of applications not only for automobile engines, but also for industrial heavy diesel engines will be displayed with the optimal development of the RCCI combustion strategy along with the use of alternative fuels.

In particular, the design of the new low reactivity fuel injection in the air intake manifold not only implies the design and fluid analysis of the system, but also implies the implementation of the new approach in the overall engine performance. Therefore, the incorporation of the optimal system would ensure the appropriate conditions for the premixed air and fuel combustion, which is one of the key factors of the RCCI performance.

The use of alternative fuels, particularly in the new manifold is one of the more challenging parts of the research, but once it is achieved, would enhance the use of new ways of fueling the engines. Thus, the use of fossil in automobiles would be drastically reduced, which is one of the main goals of the long-term research project.

---

[1] PM: particulate matter or particulate pollution, is the term for a mixture of solid particles and liquid droplets found in the air. Once of the emissions formed in the combustion process of the engine.

[2] NO_x: nitrogen oxides gases produced from the reaction among nitrogen and oxygen during combustion of fuels, at high temperatures, such as occur in CI engines. Contribute to the formation of smog and acid rain are the main emission problem for actual vehicles.

4. ANALYSIS OF ALTERNATIVES

In this section, different alternatives to carry out the analysis and design of the new fuel injection system in the intake manifold are introduced. Analysis of alternatives is one of the essential stages in the design process, because it helps to decide what would be the most appropriate path to follow in order to solve the problem.

There will be stated and briefly explained the different alternatives that would be carried out:

1. **CFD analysis using another software:** there could be found several computation fluid dynamics software, which could be academic or commercial. Actually, Vicente Chapa, a partner of the Engine Lab, has analyzed this case using Autodesk® CFD software, for which a student version is available. The problem is solved, and the results shown very similar behavior of the fluids inside the manifold, which indicated that the solution looks correct. The difference between ANSYS® Fluent and Autodesk® CFD is the accuracy of the results. Fluent provides more precise and accurate results and Autodesk® has the advantage of computational 'lightweight'. For instance, problem processing in Fluent takes around 1 hour and solutions in Autodesk around 15 minutes.

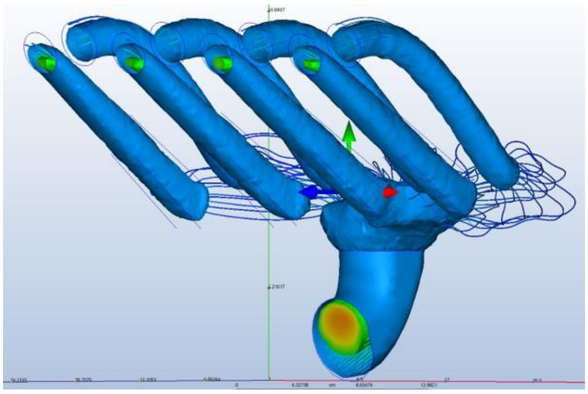


Figure 2. Autodesk® CFD Results.

2. **Experimental analysis:** another alternative is to analyze experimentally the behavior of the actual manifold. This implies the tracking of different parameter during the performance of the engine. This tracking is possible because MoTec, the control software, can gather the data form the engine performance. Besides, the signals could be measure using traditional methods, which could be using an oscilloscope. This alternative is undoubtedly very accurate in the analysis of the actual design of the manifold, but it is not very appropriate for the design of the new manifold, because the new system need to be set up without the security of a proper performance with other engine parts.

Figure 3. MoTec user interface example.

More than explained approaches could be carried out. Finally, all the alternatives being considered, the analysis was decided to be carried out using ANSYS® Fluent. The reasons of the decision are stated below:

- **Software availability:** a complete version of the software was available, without any node or iteration restrictions.
- **ANSYS® Fluent reputation:** this software is used by several engineering firms, and it has been improved during the years and now it is a very powerful CFD software.
- **ANSYS® Fluent learning material:** there is a huge amount of information on the Internet with useful tips about the software. The preparation to carry out a successful analysis is done using video tutorials and more material online.

5. THEORETICAL FRAMEWORK

This section is essential to understand the context where the main analysis would be carried out. The idea is to show and explain the systems and elements used in the research, which are the theorical support followed to successfully complete the design and analysis of the new manifold.

5.1. RCCI advanced combustion strategy

Reactivity Controlled Compression Ignition (RCCI) uses two fuels with different reactivity to increase combustion delays periods and promote premixing. RCCI uses port-injected gasoline fuel as low reactivity fuel and directed injected high reactivity diesel fuel. This advanced combustion model ensures a higher engine efficiency and lower exhaust emissions.

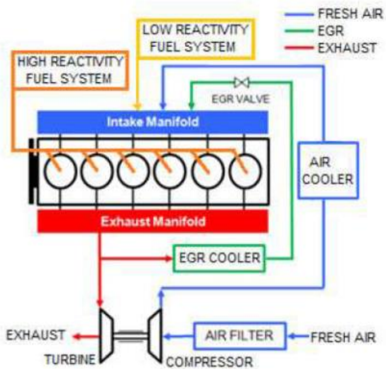


Figure 4. RCCI Engine System

The implementation of a control model for this complex system is what the long-term research project is pursuing. The Advanced Engine Control Laboratory is working to achieve this overall objective. The detailed explanation for this combustion model is not in the scope of this research, but a basic knowledge of its performance is needed.

5.2. Advanced Engine Control Laboratory

The research team supervised by Dra. Carrie Hall is formed by undergraduate, graduate and PhD students from Armour College of Engineering of Illinois Institute of Technology. All of them contribute with their work to advance in the knowledge of the new combustion strategy and perform their work in the Advanced Engine Control Laboratory.

This laboratory consists in two rooms where the resources for the research are available. The first room is where the test engine is located, and the second room contains the computer equipment to control the engine performance. The software used to control the performance of the engine is MoTec.

Figure 5. Advanced Engine Control Laboratory.

5.3. 2.0 Liter TDI Common Rail BIN5 ULEV Engine

The 2.0 Liter TDI Common Rail BIN5 ULEV Diesel Engine [2] is tested in the Engine Laboratory. This engine was part of 2013 Volkswagen Jetta TDI. This engine has been mounted, prepared and tested by research people over the time when long-term study has been carried out.

The technical characteristics of this engine

- Common rail injection system with piezo fuel injectors.
- Diesel particulate filter with upstream oxidation catalyst.
- Intake manifold with flap valve control
- Electric exhaust gas return valve.
- Adjustable exhaust gas turbocharger with displacement feedback.
- Low and high-pressure Exhaust Gas Recirculation (EGR) system.

Technical data is summarized in the table 1:

Design	4-Cylinder In-Line Engine
Displacement	1968 cm^3
Bore	81 mm
Stroke	95.5 mm
Valves per Cylinder	4
Compression Ratio	16.5:1
Maximum Output	140 HP (103 kW) at 4000 rpm
Maximum Torque	320 Nm at 1750 rpm up to 2500 rpm
Engine Management	Bosch EDC 17
Fuel	ULSD/ASTM D975-06b 2_D_S<15
Exhaust Gas Treatment	High and Low Pressure EGR, Oxidation Catalytic Converter, Diesel Particulate Filter, NOx Storgafe Catalytic Converter

Table 1. Engine Technical Data.

Pursuing the objective of completely understand the RCCI combustion strategy, the engine has been modified to adapt the components and to track and control the engine behavior and characteristics. The engine is shown in the next images.

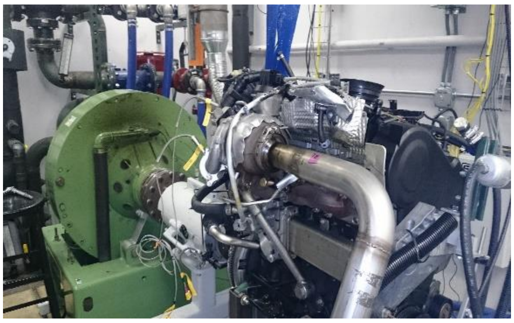


Figure 6. Test Engine.

5.4. Intake Manifold

The main function of the intake manifold of a diesel engine is to supply air to each intake port in the cylinder head. In order to ensure the correct distribution of the air a partial vacuum exist in the manifold. The intake manifold of the engine is manufactured of aluminum.

The intake system used by 2.0 Liter TDI engine is shown in the figure 7:

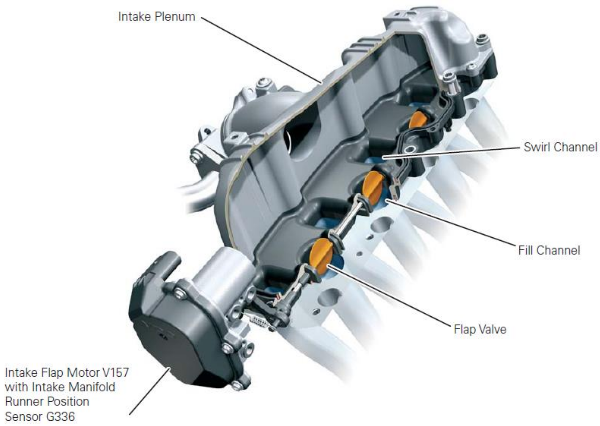


Figure 7. Intake system of 2.0 Liter TDI Engine.

This figure illustrates the variable flap system of the intake manifold. The Intake Flap Motor V157 controls the positioning of the flaps and adjusts the intake air based on the engine speed and load.

At low speed the flap valves are closed, thus the formation of the mixture is enhanced. The flap valves are adjusted depending on the engine speed and load. When almost 3000 rpm is reached, the flap valves are completely open. The system ensures an optimal performance for the intake system.

Figure 8 shows the location of the intake manifold in the lab engine, and the actual manifold used for modeling the geometry for the analysis:

Figure 8. Actual intake manifold and intake manifold model.

5.5. Exhaust gas turbocharger

Another important component which directly influences the intake manifold pressure is the turbocharger. The turbocharger increases the efficiency and power output by recovering the remain energy of exhaust gases and feeding it back into the intake system.

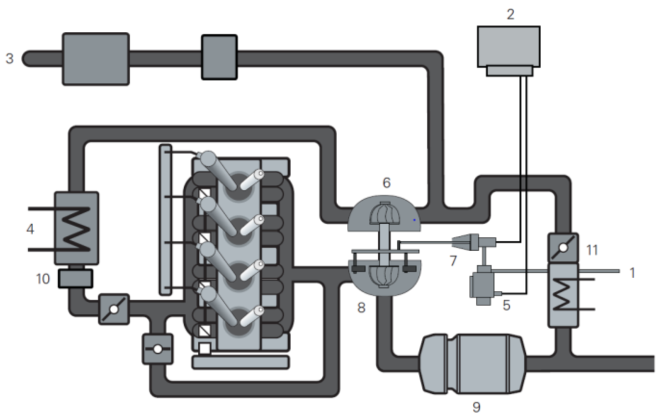


Figure 9. Sketch of turbocharger system of the TDI engine.

The engine boost pressure control is the system responsible for the management of the volume of air that is compressed by the turbocharger. The increase of air mass ensures all fuel is burned before the exhaust stroke. Thus, temperature and pressure increase, and this results in a higher efficiency for the engine.

The pressure increase caused by the turbocharger performance, as well as the one caused by the Exhaust Gas Recirculation (EGR) system, is an important feature to consider in the analysis of the intake manifold, thus this varies the boundary conditions of the analysis.

5.6. Fuel injectors

For the new fuel injection system implementation in the intake manifold, 4 piezoelectric injectors will be set up. The piezoelectric injectors inject highly pressurized fuel, when voltage is applied across piezo crystal. This material generates an electrical charge once mechanical stress is applied.

The injector model chosen for the study is ID1700X from Bosch Motorsports and Injector Dynamics. The technical features of the injector are summarized in table 2:

Nominal Flow Rate	1725 cc/min at 3 bar (43.5 psi)
Maximum Differential Fuel Pressure	7 bar (101.5 psi)
Fuel Compatibility	Methanol/Ethanol/All Hydrocarbon Fuels
Prize	$250

Table 2. Fuel injector technical data.

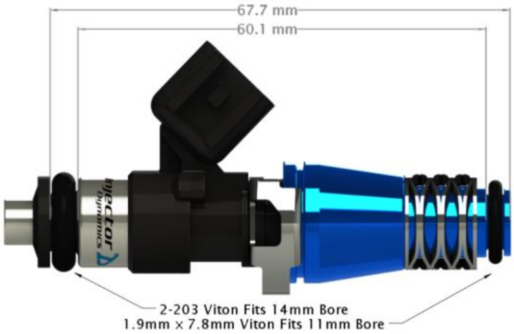


Figure 10. Fuel injector Dimensions.

5.7. ANSYS® FLUENT

ANSYS® Fluent 18.2 is the computational fluid dynamics simulation software used in the fluid analysis of this research. Among different options to carry out the analysis, the huge computational capability, result accuracy and industrial application of the software are the features that decided the use of ANSYS® Fluent in the study.

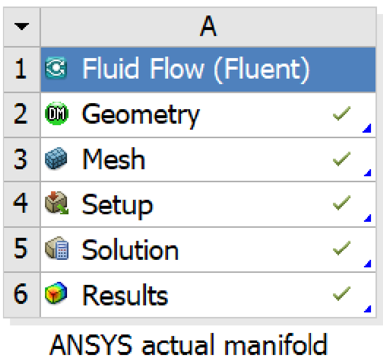


Figure 11. ANSYS® Fluent Project Schematic Window.

As it is illustrated in figure 11, the software requires the completion of different modules before processing the physics problem. First, common modules used in different physics problems such as Geometry and Mesh need to be completed. Then, Setup holds the Fluent module for processing part of the problem. Finally, Solution and Results allow the postprocessing of the analysis. The setup of Fluent parameters will be explained in detail in the analysis section of the report.

6. METHODOLOGY

In this section the design and analysis process are explained in detail. The fluid analysis of the intake manifold is carried out in two stages. In the first stage the actual manifold is analyzed, in which air the only fluid. The second stage incorporates the fuel injector inlets; thus, air and gasoline would be the flux analyzed. The results of each analysis are shown and discusses along with the analysis.

6.1. First Stage: Fluid analysis of Intake Manifold

The analysis was carried out to simulate and understand the fluid process in the intake manifold. The actual design for the manifold is evaluated. In the following section this process and the achieved results would be shown, and the order of the different phases of the process are set up in the same order as ANSYS® modules have been completed.

6.1.1. Geometry

The actual design of the intake manifold is imported into the Design Modeler of ANSYS®. This design was done by Riddish Parekh, a research student who worked in the Engine Laboratory. All the shapes and details of the manifold are modeled in the step file and exported them into the software CAD modeler.

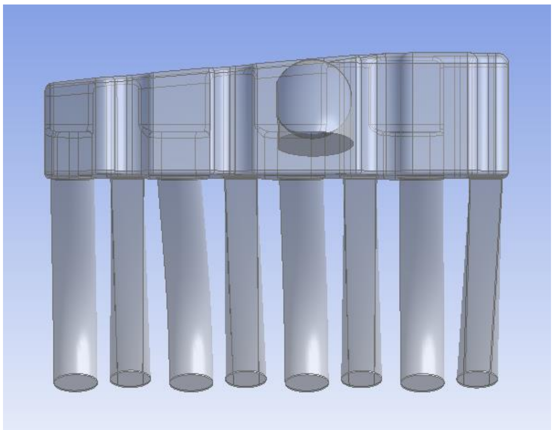


Figure 12. Geometry of the manifold.

The geometry imported into Design Modeler only represents the cavity of the manifold, that is the geometry is designed without thickness. The reason is because the analysis becomes simpler in Fluent, and the importance of the walls is negligible.

6.1.2. Mesh Model

The next step of the process is meshing the geometry. The mesh is an important step of the analysis because the success of the fluid analysis starts when the mesh is successfully exported to Fluent. Because of this, the mesh quality has to meet some requirements, so mesh elements have to be small enough to enable the analysis and achieve accurate results.

The size function used in Meshing module is curvature. The curvature on faces and edges is examined and the mesh is computed by setting up the element size on these curvatures and in any case exceeding the maximum size or the curvature normal angle. In this analysis, these parameters are automatically calculated by the software.

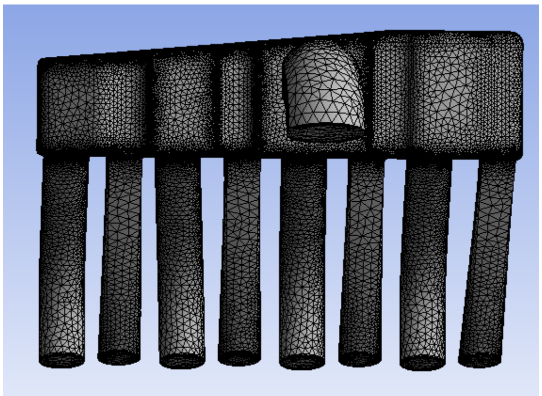


Figure 13. Mesh of the manifold.

6.1.3. Fluent

The module where solution is processed. The Fluent module is launched as a Double Precision Parallel solver, in order to achieve accurate results of the fluid analysis. Table 3 illustrates the solution set up:

Solver type	Pressure-based
Solver time	Steady
Energy equation	Off
Gravity	Off
Flux	Laminar
Method	Simple

Table 3. Solution Set Up.

In the analysis air is the unique fluid and the physical properties are shown below:

Density	1.225 kg/m^3
Viscosity	1.79E-05 kg/m·s

Table 4. Air properties.

Once the solution parameters and the fluid properties are established, the boundary conditions for this flow need to be set up.

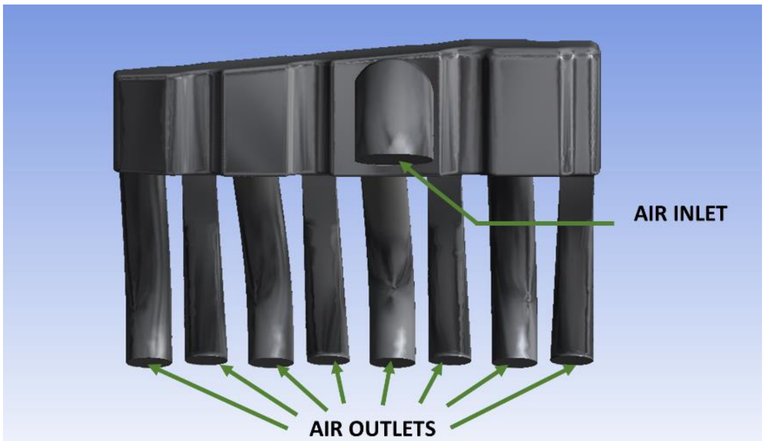


Figure 14. Inlets and outlets of the manifold

As it shown in the figure above, the manifold is composed by one air inlet and eight outlets. The air outlets represent the runners of the intake manifold, which extend from the plenum of the intake manifold to the cylinders head intake ports. There are two runners for each cylinder, one is the swirl channel and the other the fill channel of the manifold, as it is explained in the theoretical framework's intake manifold section.

In order to represent different operating points of the engine, the pressure in the air inlet is changed, and 4 different analysis are carried out. With this different analysis the influence of the turbocharger and the exhaust gases recirculation systems is evaluated. These 4 different pressures are represented in Table 5:

Condition	Gauge Pressure	Absolute pressure
Atmospheric pressure	0 MPa	101325 MPa
1.5 Atmospheric Pressure	50662.5 MPa	151988 MPa
2 Atmospheric Pressure	101325 MPa	202650 MPa
3 Atmospheric Pressure	202650 MPa	303975 MPa

Table 5. Different pressure conditions.

The difference between Gauge pressure and Absolute pressure is the first one uses the reference of atmospheric pressure, and the second one takes the perfect vacuum as its reference.

The operating conditions in the plenum, the manifold cavity, is set up as atmospheric pressure. Then, the air inlet boundary conditions are set up as pressure inlet, and the pressure value is modified depending the case that is been analyzed. The outlets are established as pressure outlet, and the pressure is indicated as atmospheric pressure in all of the, despite the fact that the pressure in the outlets would be an output of the analysis, but a first approximation is needing to initialize the iteration process.

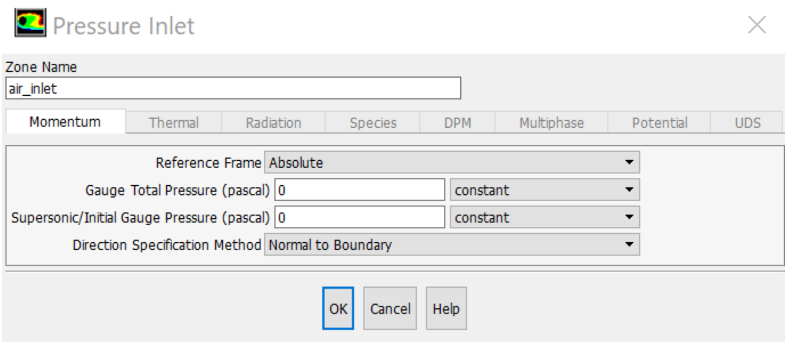


Figure 15. Air inlet as pressure inlet.

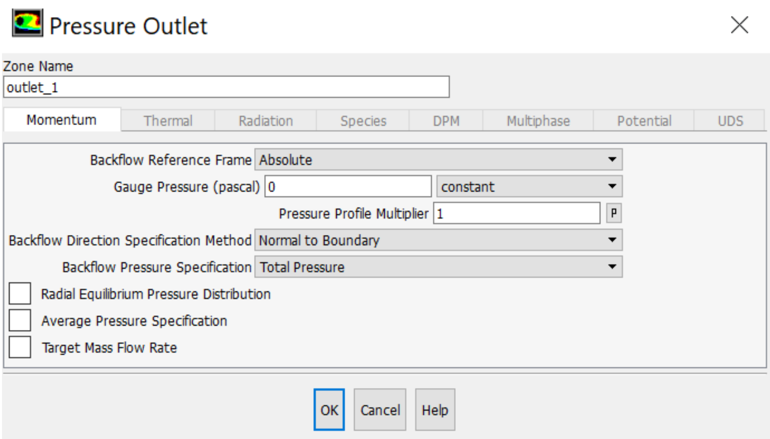


Figure 16. Outlets as pressure outlets.

When the boundary conditions are set up, the problem can be initialized, and the processing part of the problem is set up in 100 iterations. The computational time to run the analysis is almost 10 minutes for each case.

6.1.4. Results

The postprocessing of the fluid analysis is done using CFD-Post module of ANSYS®. Among several possibilities that could be shown, since the postprocessing tool of the software provides the users with multiple options, in this research the following results for each case will be shown, which are:

- Velocity streamlines.
- Pressure values of the manifold plenum, inlets and outlets.

Finally, the results would be commented and a conclusion for this analysis is presented in the report.

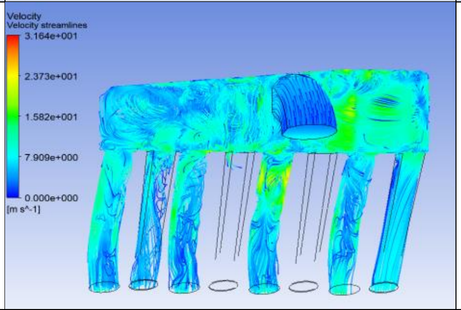
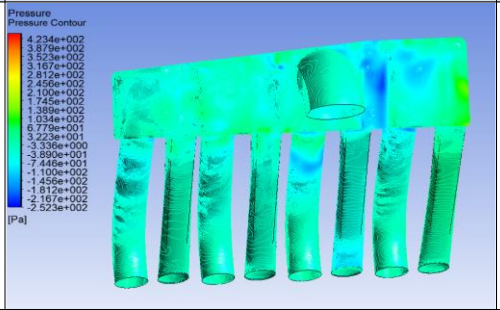
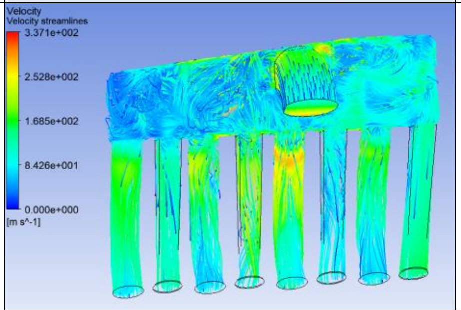
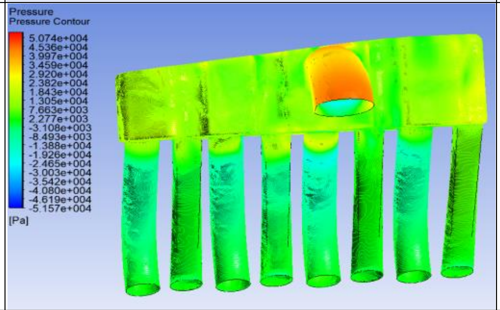
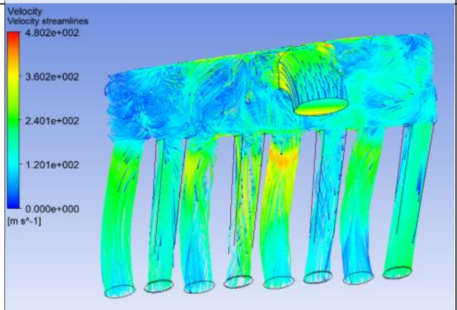
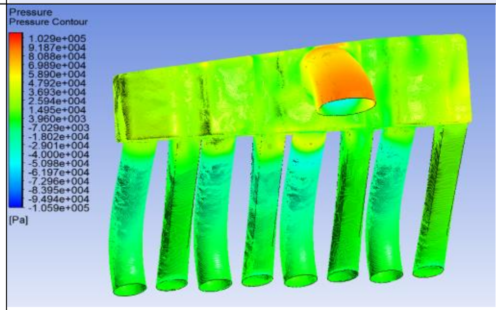
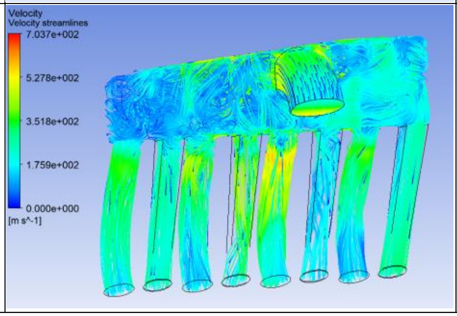
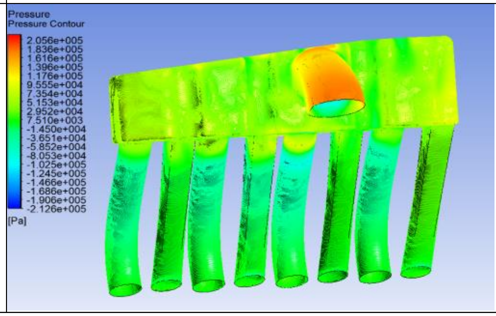
Case	Velocity Streamlines	Vmax	Pressure	Comments
Atmospheric Pressure		31 m/s		It can clearly observed that two central of the cental outlets do not have any streamline, so air is not going by these runners
1.5 x Atmospheric Pressure		330 m/s		The previous problems is fixed here, note that velocities and pressures are much more higher in this case
2 x Atmospheric Pressure		460 m/s		Similarly to the previous case, all runners contain fluid flux, and here the velocities and pressure are also higher
3 x Atmospheric Pressure		680 m/s		In this last case all the runners are fulfilled, and the increase in pressure and velocity is significant.

Table 6. Results Stage 1.

6.1.5. Stage 1 Conclusions

Once the results are evaluated, some conclusions for the first stage should be discussed. In this moment, the analytical and critical though of the engineer plays a fundamental role, because the results from a Finite Element Analysis need interpretation and evaluation.

- **Velocity streamlines:** as it is clearly observed in the results table in the previous page, the higher the air inlet gauge pressure is, the better distribution happen for the velocity streamlines. In the first case, there are two central runners without any air flux. When the pressure is increased in the inlet, air flows along all the runners. Comparing with the engine performance, the higher engine speed, the higher intake pressure in the air inlet of the manifold, and the more air flux flows along the runners to the cylinders.
- **Maximum velocity value:** focusing on the values taken from the postprocessing module, the values are extremely high. In the three last cases, the values are higher than the sound's velocity, which is around 340 m/s. Thus, this values are illogical, however, the interpretation could be done as taking the velocity proportional to the pressure, that is the higher intake pressure, the higher the velocity of the flux inside the plenum, which is logical as part as Bernoulli's equation says.
- **Wall pressure:** all the cases deliver the same results, indicating that the maximum pressure of the plenum is achieved just in the air inlet pipe and in the back part of the plenum, just in front of the air exit from the inlet.

The main purpose for this analysis was analyzing the flux inside an actual intake manifold, and it has been achieved. The actual conditions inside the manifold depend on several parameters, and it is very difficult to define in Fluent. This stage set the bases for the important analysis of the research, which it is explained in the next section.

6.2. Second Stage: Fluid analysis of the New Intake Manifold

Once the actual intake manifold is analyzed, the new design of the manifold has to be done. In this section, the new design is shown, and then the fluid analysis is explained in detail. The fluid analysis is carried out in different ways, in order to understand the flux behavior inside the plenum and the runners. These different case studies will be explained in the Fluent module section of this stage. The process will be tracked as in the precious stage, following the order of the ANSYS® modules.

6.2.1. Geometry

The new design of the manifold is the first step. 4 gasoline injectors need to be placed in the intake manifold. The locations of the injectors are decided along with Vicente Chapa, a research partner in the Engine Laboratory. After discussing the appropriate location of these fuel ports, the final place was set up in the horizontal upper plane of the manifold, just behind the runners. The best way to understand the new design is focusing on the figure 17:

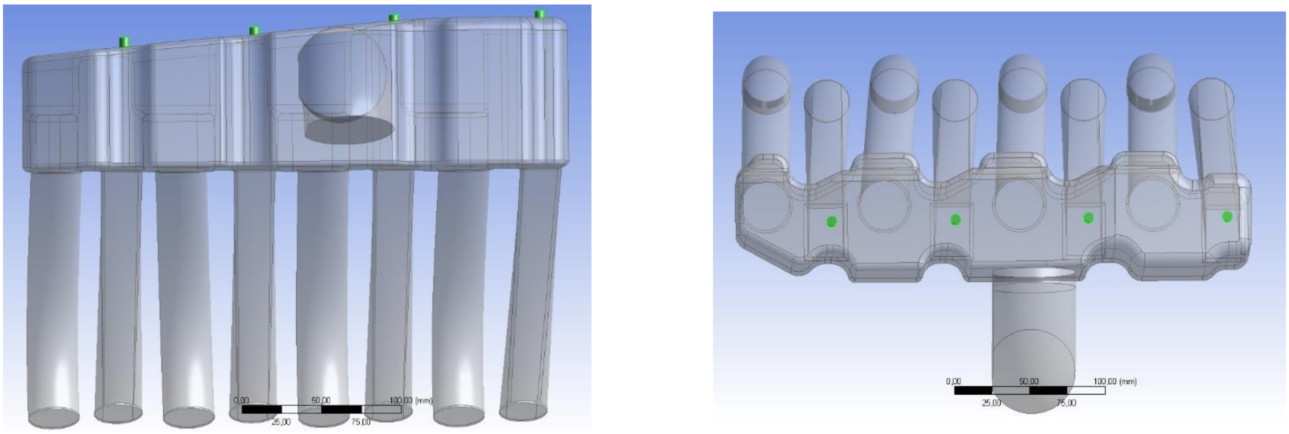


Figure 17. Geometry of the new manifold.

The fuel ports are represented in green in the image above. The geometry of the injector is modeled only with a cylindrical port, which simplifies a lot the fuel flux across the injectors. The fuel ports are centered in the upper plane and are separated at equal distance between two consecutive ones.

As in the previous case, the geometry imported into Design Modeler only represents the cavity of the manifold, since the analysis becomes simpler, and the importance of the walls is negligible.

6.2.2. Mesh Model

This step of the process is set up automatically by Meshing module of ANSYS® and the properties of the mesh are explained in the Module 2 section of the first stage. The figure 18 shows the final shape of the mesh for the new design:

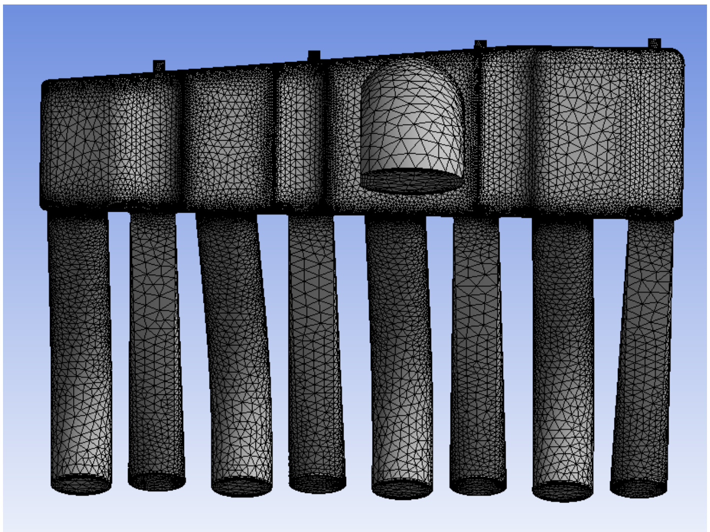


Figure 18. Mesh of the new manifold.

6.2.3. Fluent & Results

Unlike the first stage of the methodology, in the following section there will be explained the fluent module and results together. That is because, different approaches are evaluated, in order to completely understand the characteristics of the flux in the manifold, and in order to easily associate each analysis and its corresponding results.

Step 1: Unique fluid analysis

When more than one fluid or fluid phases are introduced in Fluent, the resolution of the problem is much more complex. For this reason, the first step is carrying out an analysis using only one fluid in the manifold. Thus, in this step only air or gasoline, which will be in vapor and liquid phase, flows along the intake manifold.

The gasoline used in the analysis is a common fuel employed in automobiles. The gasoline is n-octane, C_8H_{18}, analyzed in liquid and vapor phases. The physical properties for each fluid are shown in the table below:

Fluid	**Air**	**C_8H_{18} vapor**	**C_8H_{18} liquid**
Density	1.225 kg/m^3	4.84 kg/m^3	720 kg/m^3
Viscosity	1.79E-05 kg/m·s	6.75E-06 kg/m·s	5.40E-04 kg/m·s

Table 7. Properties of the fluids.

Three analysis are carried out, each of then uses one of this fluid in the analysis:

1. **Air:** similar to stage one, in this case there are 5 air inlets in the manifold. The actual one and the fuel ports as air inlets.
2. **C_8H_{18} vapor:** now the air inlet is set up as a wall, and the fuel is injected in vapor phase from the fuel ports.
3. **C_8H_{18} liquid:** here again the air inlet is a wall, and the liquid fuel is injected pressurized from the fuel ports.

In the table below the solution set up is illustrated. As it is shown, the solver conditions are similar in all the case studies, except the flux condition. In the air analysis a laminar flux is set up, but when gasoline is analyzed, convergence problems appeared trying to solve a laminar flux, so a turbulent analysis was carried out.

Case	**Air**	**C_8H_{18} vapor**	**C_8H_{18} liquid**
Solver type	Pressure-based	Pressure-based	Pressure-based
Solver time	Transient	Transient	Transient
Energy equation	Off	Off	Off
Gravity	Off	Off	Off
Flux	Laminar	Turbulent	Turbulent
Method	Simple	Simple	Simple

Table 8. Solution set up.

The next step in the process is set up the boundary conditions:

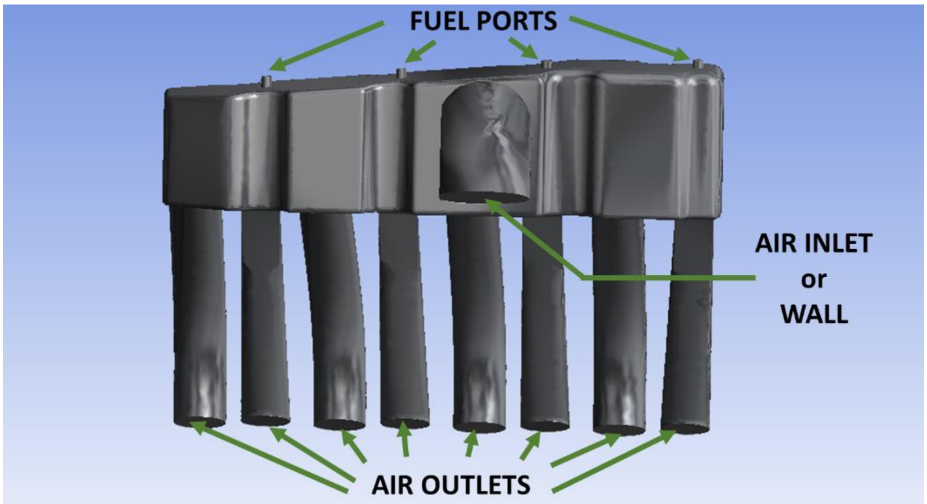


Figure 19. Inlets and outlets of the manifold.

As it is illustrated in the image above, an air inlet, 4 fuel ports and eight outlets compose the manifold boundaries. As it is explained in the previous stage, there are two runners per cylinder, which are the swirl channel and the fill channel of the manifold. The differences among the three cases is in the air inlet, in the first analysis the inlet is set up as a pressure inlet, and the other two analysis the air inlet is set up as wall, because only the fuel ports are open in this simulations.

The operating conditions in the plenum, the manifold cavity, is set up as atmospheric pressure. The following table summarized the boundary conditions for each case, and then each case conditions will be explained in order to clarify the decisions made.

Case	**Air**	**C_8H_{18} vapor**	**C_8H_{18} liquid**
Air inlet	Pressure Inlet	Wall	Wall
Air inlet Gauge pressure	101325 Pa	None	None
Fuel ports	Pressure Inlet	Pressure Inlet	Pressure Inlet
Fuel ports Gauge Pressure	101325 Pa	398675 Pa	598675 Pa
Outlets	Pressure Outlet	Pressure Outlet	Pressure Outlet

Table 9. Boundary conditions.

1. **Air:** air inlet and fuel ports are set up with a gauge pressure of 101325 Pa, which means that the air and fuel is injected at the double of atmospheric pressure.
2. **C_8H_{18} vapor:** the vapor fuel is injected with 5 bar of total pressure, in order to represent a pressurized fuel injected by this injectors.
3. **C_8H_{18} liquid:** the liquid fuel is injected pressurized at 7 bar of total pressure, which is an actual high value taken from the characteristics of the injectors.

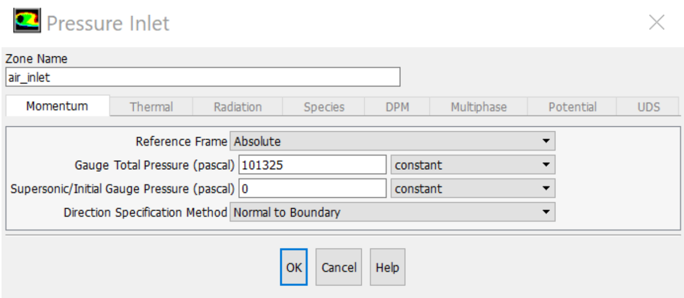


Figure 20. Air inlet as pressure inlet.

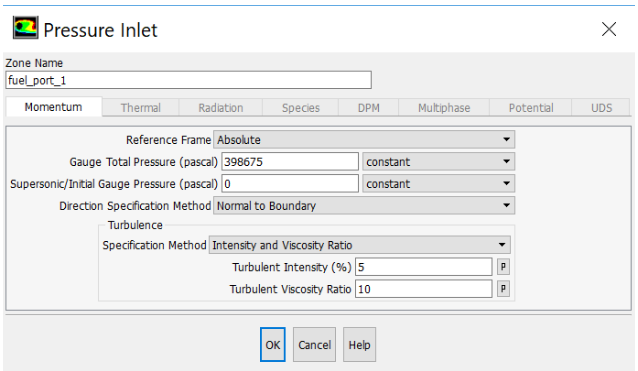


Figure 21. Fuel ports as pressure inlets.

When the boundary conditions are set up, the problem can be initialized. Once the problem is initialized the processing part starts. At this point, some parameters have to be explained.

- **Time step:** in unsteady flow calculations, time step is the incremental change in time for which the governing equations are being solved. It is a key parameter to achieve the convergence in the solution.
- **Time step size:** the time for each time step.
- **Flow time:** once the number of time steps multiply by the time step size, gives the time in which the flow is simulated. For example, with a time step size of 0.01 seconds and 100 time steps, the software would simulate a flow of 1 second.
- **Maximum iterations per time step:** fluent is trying to achieve stability in the solution. If the boundary conditions are set up correctly, the convergence could be achieved with a small number of iterations in one time step, but many times, the software needs more iterations to achieve convergent solution.

In all the cases, there was decided to simulate 0.1 seconds, because is enough to evaluate the flux within the plenum and the runners. Thus, 10 time steps were done with a size of 0.01 seconds. The maximum iterations per time steps were 50.

At this point the problem is resolved and the postprocess of the problem has to be done. In the next page, the results for each case is shown. This figures illustrate the velocity streamlines for each flux.

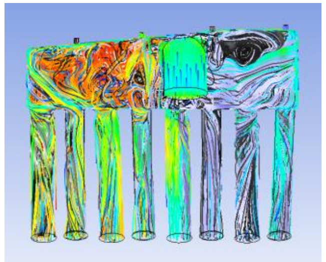
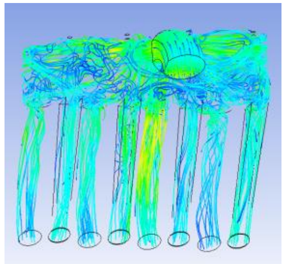
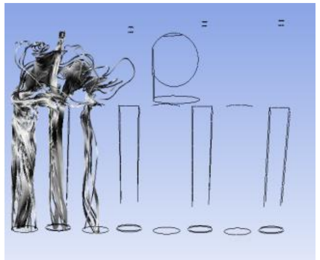
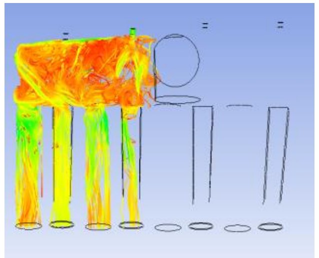
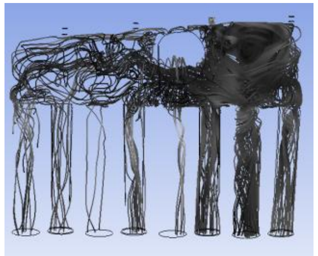
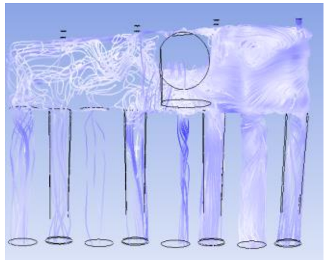
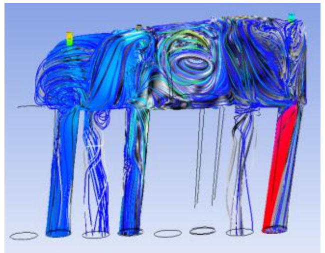
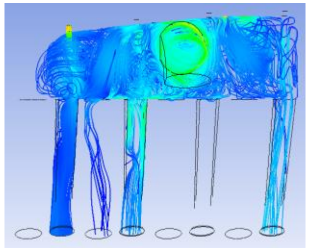
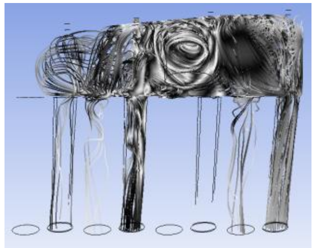
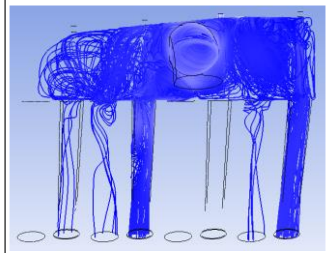
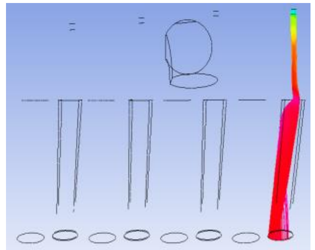
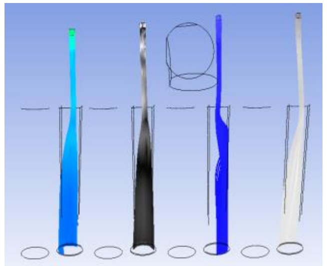
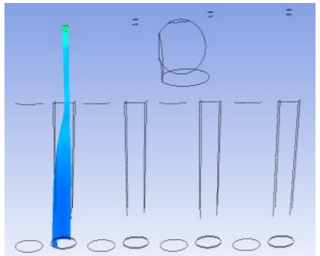
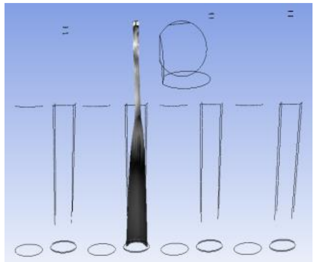
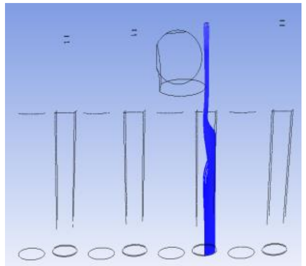
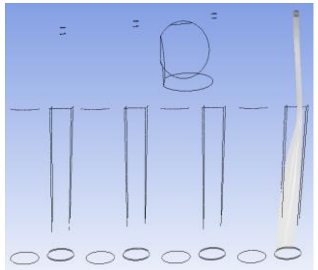
Case	All flows	Air inlet	Fuel port 1	Fuel port 2	Fuel port 3	Fuel port 4
Air						
C_8H_{18} vapor						
C_8H_{18} liquid						

Table 10. Results Stage 1 Step 1.

Stage 2 Step 1 Conclusions

The variety in the results need an interpretation, and in this chapter the results will be discusses. As each case is totally different from the others, an individual analysis is done. Finally, an overall conclusion will be stated for this case study.

- **Air:** apparently the case with better results. In this case, the unique fluid is air, which makes the problem easier than the other ones. The air flow from the air inlet cause interaction among air particles inside the plenum. All the flux interacts among each other, and all the runners are filled, which represent an optimal performance of the manifold. Besides, it can be observed that the fuel port 4 fills all the runners. On the contrary, the first fuel injector only fills the most closer outlets. That is because of the design of the manifold, which its upper plane is not horizontal, it is inclined. Thus, the last two fuel ports achieve a better filling than the first two ones.
- **C_8H_{18} vapor:** results suggest a difficult interpretation. The air inlet is set up as wall, and the only flows come from the injectors. Three runners are empty, without gasoline flowing within then. Besides, each injection fills the closest runner, and the runner in the right. There is not almost any interaction between the injections flows. Finally, injection 4 only fills the runners in the right. The logical interpretation is as any air inlet provides the necessary air flow to cause interaction among fuel and air particles, the fuel behavior is not the proper one to perform an optimal filling of the cylinders.
- **C_8H_{18} liquid:** here the results clarify something logical. The density of the liquid fuel is too high to perform any movement within the plenum. Thus, the fuel directly falls over each runner above the injection. The preliminary aim was simulating the behavior of a pressurized fuel, but in this final case this is not achieved.

To sum up, air injection is need in the manifold. If there is an air flow coming from the intake inlet, interaction among particles would be generated inside the manifold, and an optimal filling of the manifold would be achieved.

As it is explained, the design of the manifold causes a different distribution of the fuel injection due to the fact that the upper plane of the plenum is inclined, and the injection occurs in different heights. This can be a design factor in future work, because an equal filling of the runner is needed in order to balance the fuel and air mixture flow distributed to each cylinder.

Step 2: Volume of Fluids

Finally, the second step of the stage 2 is the more complex analysis carried out with ANSYS® Fluent. The introduction of two different fluids, in two different phases increase the complicity of the problem, due to the computational limitations and a precisely set up of boundaries.

The way of simulating the flux of two different fluids in Fluent is using the VOF, (Volume of Fluids) multiphase model. The VOF model allow the simulation of two or more immiscible fluids. The software solves a single set of momentum equations, and then tracks the volume fraction of simulated fluids' flows along the domain. Undoubtedly, it is a powerful tool to model fluids flows, but the model has some limitations, and the achieving a solution is a complex process., even more in a 3D design.

For this analysis, the problem is solved using three different fluid or phases. Air, water and n-octane in vapor phase are taken from the Fluent database, and they have the following properties:

Fluid	**Air**	**H_2O liquid**	**C_8H_{18} vapor**
Density	1.225 kg/m^3	998.2 kg/m^3	4.84 kg/m^3
Viscosity	1.79E-05 kg/m·s	0.001003 kg/m·s	6.75E-06 kg/m·s

Table 11. Properties of the fluids.

Due to the computational limitations, two analysis are carried out. In both, it is tried to represent an actual performance of the manifold, thus practical boundary conditions are set up:

1. **Air-C_8H_{18} vapor:** the most representative and complex analysis. Two different gas fluid are analyzed, water and gasoline in vapor phase. As it is explained, injectors inject pressurized liquid gasoline. After trying to simulate this situation and getting errors in all the attempts, the solution taken was injecting pressurized vapor gasoline through the injectors. Here again, air enters from the air inlet and the gasoline is injected through the fuel ports.
2. **Air-H_2O liquid:** two phases of air and liquid water are simulated here. The water represents the behavior of liquid fuel flowing through the plenum and runners. In this case, air comes from the air inlet, and water is injected through the fuel ports.

Then, the solution is set up. This part is become very important, because the VOF model posses some limitations, such as only accepts steady-state fluids in certain conditions, laminar calculations are also limited to particular problems or the set up of correct boundary conditions to ensure the convergence. Table 12 shows the solver conditions:

Case	Air-C_8H_{18} vapor	Air-H_20 liquid
Solver type	Pressure-based	Pressure-based
Solver time	Transient	Transient
Energy equation	On	Off
Gravity	Off	Off
Flux	Turbulent	Laminar
Method	PISO	SIMPLE

Table 12. Solution set up.

The first case represents an actual situation within the plenum, thus turbulent transient flow is represented, and also energy equation is applied here. The second case is the simplest one, where laminar flow is analyzed. The solution process takes huge time to be processed, for that reason, the problem should be simplified in order to reduce the computational time.

In order to get a stable solution, several attempts have been done. In the vast majority, convergence errors or computational limitations came across. It has been applied different changes in the boundary conditions and solver parameters, to achieve a successful results.

Boundary conditions are shown in the figure below:

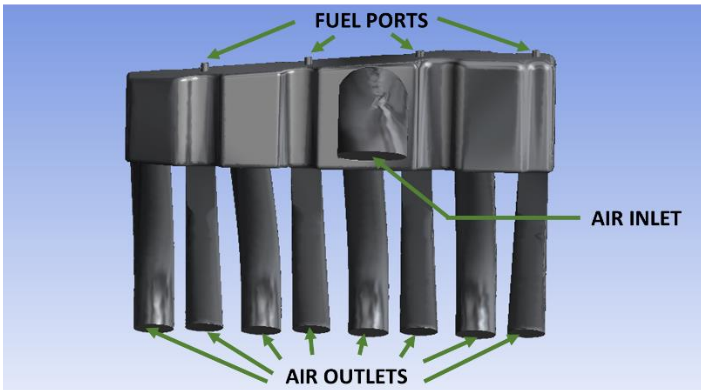


Figure 22. Inlets and outlets of the new manifold.

As it is illustrated in the image above, an air inlet, 4 fuel ports and eight outlets compose the manifold boundaries. As it is explained in the previous stage, there are two runners per cylinder, which are the swirl channel and the fill channel of the manifold.

The table below contains the boundary conditions for each case, which will be explained after that in detail to clarify the decisions taken:

Case	Air-$H_2$0 liquid	Air-C_8H_{18} vapor
Interior Pressure	101325 Pa	101325 Pa
Air inlet	Velocity-inlet	Pressure-inlet
Air inlet Gauge pressure	2 m/s	101325 Pa
Fuel ports	Velocity-inlet	Pressure-inlet
Fuel ports Gauge Pressure	1 m/s	398675
Outlets	Pressure Outlet	Pressure Outlet
Outlets Gauge Pressure	0 Pa	-101325 Pa
Plenum absolute pressure	101325 Pa	0 Pa

Table 13. Boundary conditions.

1. **Air-C_8H_{18} vapor:** a trial to represent not only the most realistic operating conditions of the manifold, but also the most complex analysis for the software. Here, the plenum and outlets have been set up with vacuum pressure, in order to create depression inside the manifold and induce the flows through the plenum and the runners.

   The inlets have been set up as pressure inlets. Air is injected with 101325 Pa higher than atmospheric pressure, and fuel is injected with 5 bar of total pressure, which means 398675 Pa of Gauge pressure.

2. **Air-H_2O liquid:** it was tried to represent actual conditions, which means setting up the inlets as pressure-inlet, since the fluids are injected due to the pressure applied. However, convergence errors such as floating-point exception or large Courant Number appeared, terms that will be explained later.

   In order to fix these problems, velocity-inlets have been set up. The velocity as it can be observed is very low, since the convergence problems still came across when a higher velocity magnitude was tried to establish. Manifold plenum and outlets are set up with atmospheric pressure.

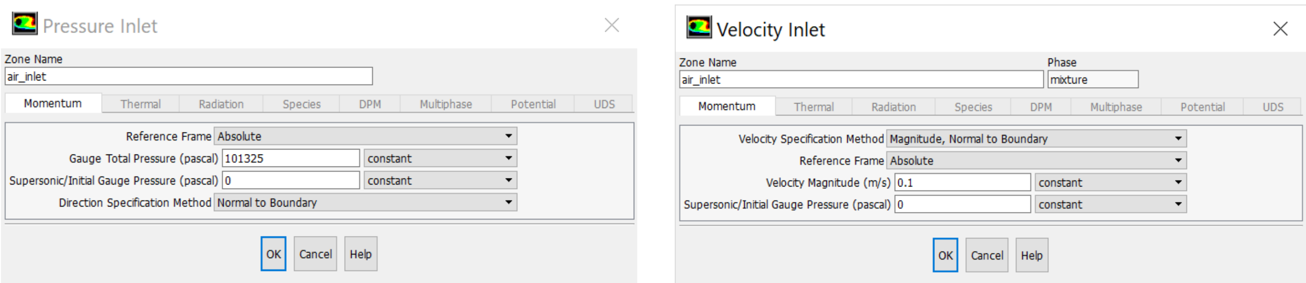


Figure 23. Air inlet as velocity inlet and pressure inlet.

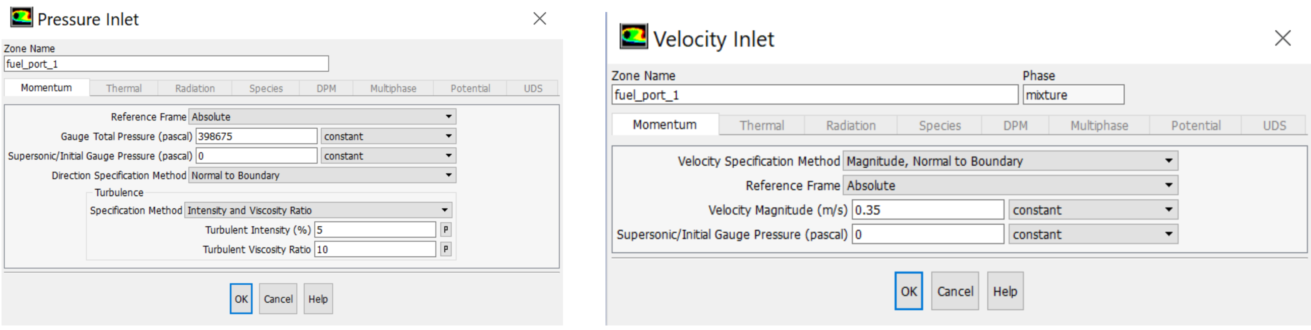


Figure 24. Fuel ports as velocity-inlets and pressure-inlets.

Along with the convergence parameter explained in the previous step of the stage 2, here other terms should be discussed in order to understand the solution stability:

- **Courant Number:** is a dimensionless number that compares the time step in the VOF calculation to the characteristic time of transit of a fluid element across a control volume. The default value is 2, and it could be reduced if more accurate results are needed. In some calculation, Courant Number starts increasing as time steps are completed, which means that the velocity field is increasing, so the problem could end up in an unstable solution. This number depends on the mesh element size, velocity magnitudes and the time step size.

  This number influence the solution convergence, and it is an indicator of a correct set up of the boundary conditions of the problem. The attempts focused on maintaining this number below 5, because of computational limitations.

- **Floating-point exception:** this convergence error message appears when the iterations became unstable. The solution seems to solve correctly, but suddenly the iteration residuals start to increase uncontrollably. Figure 25 shows the error details. Floating-point set up a boundary in processing capability, and it has been a setback to overcome during this second step.

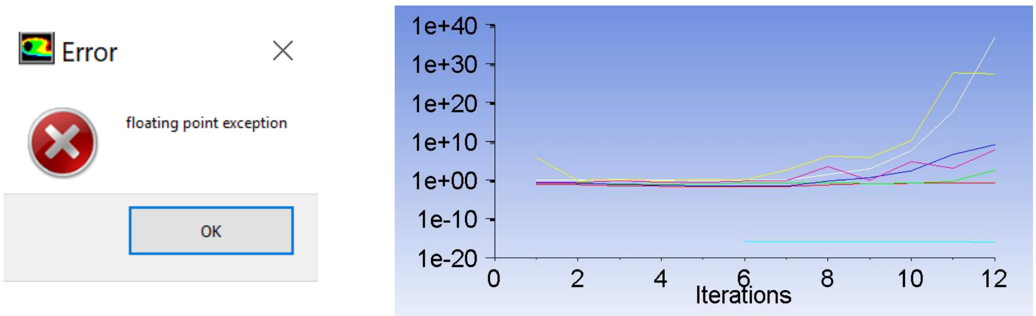


Figure 25. Floating-point exception and residuals.

In the first case, air and gasoline vapor, the flow simulation is 0.001 milliseconds, since only one time step of 1x10-7 second was simulated. This is because the convergence is very limited here, and if the number of iterations per time steps or the time step increase the solution is not stable. There is something to fix in future simulations.

In the second case, air and liquid water, the flow simulation time is 1 second, which is established since the time step size is 0.001 seconds and there are simulated 300 time steps in which 20 iterations are done in each one. Considering all these parameters, the solution time is around 12 hours, which is a huge simulation time.

At this point, the problem is resolved and the postprocess of the problem has to be done. The results for both cases are shown separately.

Air & C_8H_{18} vapor Results

Unfortunately, this case is not as well solved as the other cases. The reason are the convergence and computational limitations. In the set-up process of the boundaries and other conditions, a lot of errors appeared in the solving process, problems such as floating-point exception, computation limitation that enables the resolution of the case, and more.

In this section the results for this case are show. Since the flow has only been simulated 1x10-7 seconds, the results do not show the actual behavior of the fluid iteration.

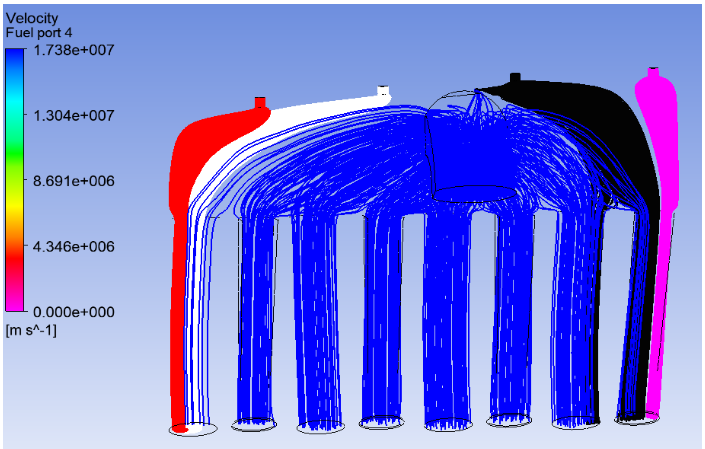


Figure 26. Results velocity streamlines.

As it can be observed in the figure, the velocity is zero in all the streamlines drawn. The reason is the lack of flow simulation time, as it has been explained before.

Air & H_2O liquid Results

The results of this case study are very conclusive. It can be observed from them a clear behavior of the flows inside the manifold. The flow is simulated 0.3 seconds and it allows a complete visualization of new manifold design performance.

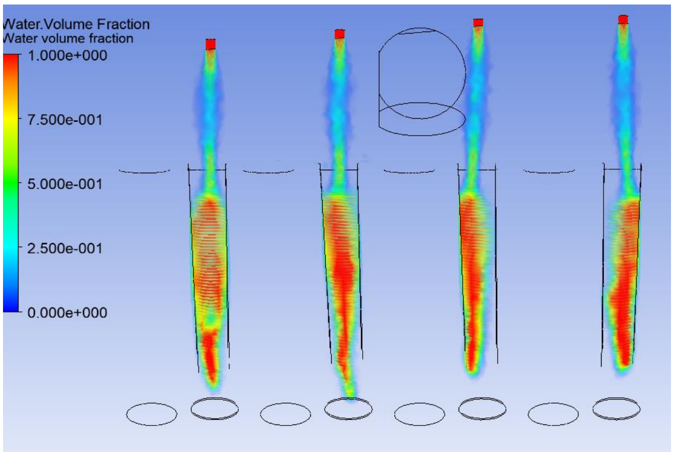


Figure 27. Water volume fraction.

Figure 27 shows a volume rendering for the water volume fraction. Here it can be clearly appreciated the behavior of the water particles inside the plenum. The liquid particles are too heavy to interact with the air particles, so they go straight to their respective runners. The interaction between the two fluids is negligible, which is not an appropriate result for the analysis.

In the result table below, there are illustrated the velocity streamlines starting from the different inlets.

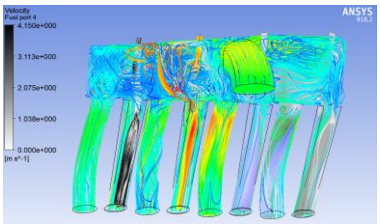
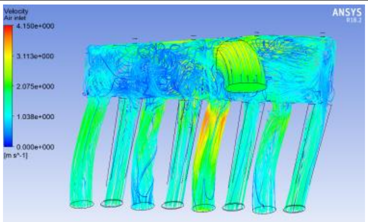
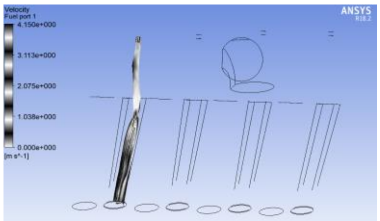
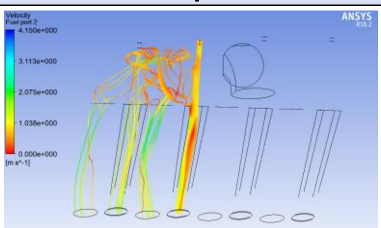
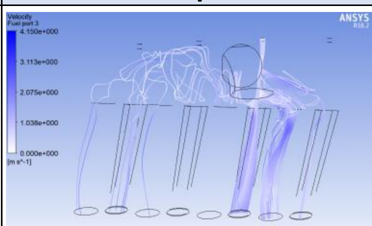
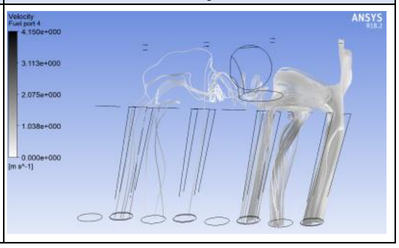
All flows	Air inlet	Water port 1
		
Water port 2	**Water port 3**	**Water port 4**
		

Table 14. Velocity streamlines.

Focusing on these results, the air streamline is distributed among all the runners, which is an important result because the mixture air-gasoline would have to distribute properly among all the cylinder to ensure a successful combustion inside them. The distribution of the water among the fuel ports is clear. Almost all the water injected from them exit from their respective runners. In the second, third and fourth water ports, a small quantity of this water is distributed among other runners. This only occurs in these ports because there are located higher than the first port.

Stage 2 Step 2 Conclusions

Analyzing separately both cases of this step, some conclusions are drawn. Finally, an overall conclusion will be stated to clarify this case.

- **Air-C_8H_{18} vapor:** clearly this case has to be checked and redefined. The flow simulation time is not enough to observe the behavior of the fluids in the manifold. As the actual operating conditions have been represented in Fluent, the computation limitations and convergence play an important role in the analysis. When these error are fixed, the analysis will lead to successful results for the research.
- **Air-H_2O liquid:** once a huge analysis is run, which spend more than 12 hours processing, results shows a clear behavior of the fluids inside the manifold. It is illustrated in the results that air and water do not interact between them. This could be because of the boundary conditions. The inlets are set up as velocity inlets, which is not an actual characteristic of the manifold. This analysis was done in order to show the behavior of a liquid inside the manifold, and it can clearly conclude that the liquid need to be pressurized in order to ensure a proper interaction among air and liquid particles.

To summarize, it is difficult to represent an actual situation inside the manifold. However, when this is achieved, the results will shed light and the research would a complete success.

7. CONCLUSIONS

The particular results and conclusion for each analyzed cases have been explained during the methodology section of the report. In this section, overall conclusions for the research are discussed.

First, a totally successful fluid analyzed of the actual manifold has been carried out. These results show the behavior and shape of the air flow along the manifold, which have set up the bases for the next more complex analysis. The second stage of the analysis has been undoubtedly more complicated. Several analysis were carried out, where a lot of them were unsuccessful, and finally two cases are shown in the report, which set up the bases for the future work.

Once all the cases are analyzed, there is a clear conclusion which may solve the flow problems in the manifold:

- **The upper inclined plane of the manifold need to be flat:** as it is observed in more than one case, the injectors position in the upper plane is not at the same level. Because of that, the fuel injected in each injection has different behavior. The right fuel ports achieved a better filling, because there are located higher than the other ones. One of the solution that might be taken is design a horizontal upper plane in the manifold, which would distribute equally the fuel flows within the cavity of the manifold.

Apart from the empirical results and conclusions, there are more skills developed, among them the most important ones are summarized here:

- **Working in a research laboratory environment:** the research group work together to achieve a common objective. All the members are involved not only in their particular duties, but also in the research success. Weekly the group meet and discuss about the work done during the week, and the next duties are established.
- **Actual engine and control equipment:** although the research is done by computer software, the actual engine, control equipment and laboratory staff are used during the project. Apart from the current research, there are more duties that needed to be completed during the semester. These duties are common examples of actual work in any engineering company, which is really enriching for the engineer formation.
- **Dra. Carrie Hall help and supervise:** the research is supervised by an expert in this field of the engineering. The help and support during the semester has been essential to complete the research. Supervised project is another work environment characteristic which will be applied in the future career of the engineer.

8. FUTURE WORK

In this last section, it will be presented the future work that could complete this research project. The design process is a continuous iterative process, where new improvements can be applied in order to achieve optimal results. There are summarized the most important next step that could be done to achieve this optimal results:

- **Achieve a successful analysis using liquid gasoline:** as it is explained in the methodology, the computational and convergence limited the solution for the air and liquid gasoline approach. The next step is trying to fix these problem and achieve more revealing results for this case.
- **New design analysis:** a new design with the upper plane horizontal is the next step in the design process. A complete fluid analysis has to be carried out, in order to determine if the new design is viable.
- **Manufacturing the new design:** once a revealing complete analysis is achieved, the next step will be manufacturing this new design and incorporate the injectors. There are many ways to do this, thus an alternative analysis has to be done to find the most cost-effective and useful solution.
- **Fluid reactivity:** in the research the fuel reactivity has not been considered. In future analysis, since the fuel could react within some pressure and temperature conditions, critical operating conditions might be analyzed to ensure a correct behavior of the mixture during engine demanding performances.
- **Fuel injection timing:** another important variable to introduce in the analysis is the fuel injection timing. The fuel injector has a particular shape of the injection timing, and there would be very helpful introducing this timing shapes in the fluid analysis.
- **Incorporate the new injection system to the engine:** a long-term objective is installing the new injection system in the test engine. For that, several modifications and new fuel system has to be placed in the engine. It is an ambitious long-term project.
- **Implement the changes into MoTec and track the results:** once the new injection is installed in the engine, the next step would be tracking and gathering the variables involves in the process.

REFERENCES

[1] Hall, C. “*Modeling and Control of Fuel Distribution in a Dual-fuel Internal Combustion Engine”*

[2] Volkswagen of America Inc. (2008). “*2.0 Liter TDI Common Rail BIN5 ULEV Engine*”