MARMARA UNIVERSITY

FACULTY OF ENGINEERING

PERFORMANCE AND IMPROVEMENT ANALYSIS OF SINGLE-CYLINDER DIESEL ENGINE WITH 1D AND 3D SIMULATION SOFTWARE

MEHMET GÖKMEN, YUNUS EMRE DEMİR, GÖKTUĞ BİNGÖL

GRADUATION PROJECT REPORT

Department of Mechanical Engineering

Supervisor

Assoc. Prof. Mustafa YILMAZ

ISTANBUL, 2019

MARMARA UNIVERSITY

FACULTY OF ENGINEERING

Performance and improvement analysis of single-cylinder diesel engine with 1d and 3d simulation software

by

Mehmet GÖKMEN, Yunus Emre DEMİR, Göktuğ BİNGÖL

Jun 24, 2019, Istanbul

SUBMITTED TO THE DEPARTMENT OF MECHANICAL ENGINEERING IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE

OF

BACHELOR OF SCIENCE

AT

MARMARA UNIVERSITY

Signature of Author(s) ....

Department of Mechanical Engineering

Certified By ....

Project Supervisor, Department of Mechanical Engineering

Accepted By..

Head of the Department of Mechanical Engineering

ACKNOWLEDGEMENT

This study was supported by Marmara University, Faculty of Engineering. We would like to thank our supervisor Assoc. Prof. Mustafa YILMAZ for advising us and Resarch Assistant Ramazan Şener for his technical support on this project.

June, 2019

Mehmet GÖKMEN,

Yunus Emre DEMİR,

Göktuğ BİNGÖL

Table of Contents

ABSTRACT

In this study, combustion, cold flow and port flow analyses are performed for the single cylinder 4 stroke diesel engine that has 510 cm3 displacement volume, 85 mm bore, 90 mm stroke, 32 Nm maximum torque, using one dimensional (GT-POWER) and three dimensional (ANSYS FORTE) simulation softwares. After that, it is aimed to increase the maximum torque by making changes in valve lift profile and injection timing. The results of changes in valve lift profile and injection timing is compared with the results of the original motor. Maximum torque has been increased as a result of changes done in optimization.

SYMBOLS

Symbol	Unit	Description
Rc	-	Compression ratio
Vcy	m3	Cylinder volume
ρg	kg/m3	Density of gas
ρl	kg/m3	Density of liquid fuel
γ	-	Specific heat ratio
P	bar-Pa	Pressure
ηth	-	Thermal efficiency
B	m	Bore
S	m	Stroke
L	m	Connecting Rod Length
CO_2	-	Carbondixoide
H_2O	-	Water
NO_x	-	Nitrogen Oxide
Rpm	rev/min	Revolution per Minute
T	N.m	Torque
W	N.m/s	Power

ABBREVIATIONS

ICE: Internal Combustion Engine

CFD: Computational Fluid Dynamics

CI: Compression Ignition

SI : Spark Ignition

1D: One Dimensional

2D: Two Dimensional

SFC: Specific Fuel Consumption

VVT: Variable Valve Timing

VVL: Variable Valve Lift

BDC: Bottom Dead Center

TDC: Top Dead Center

NVO: Negative valve overlapping

OEM: Original Equipment Manufacturer

GT: Gamma Technologies

EGR: exhaust gas recirculation

LES: Lotus Engine Simulation

BSFC: Brake Specific Fuel Consumption

BMEP: Brake Mean Effective Pressure

CA: Crank Angle

IVO: Intake Valve Open

IVC: Intake Valve Closed

EVO: Exhaust Valve Open

EVC: Exhaust Valve Closed

LIST OF FIGURES

LIST OF TABLES

1- INTRODUCTION

Nowadays, it is seen that diesel engines are used in many fields that require power. With the increase in diesel fuel prices in our country, increasing the amount of power obtained per fuel has gained more importance. Many methods can be used to increase the maximum torque, such as piston and cylinder internal geometry changes. In this study, it is aimed to increase the maximum torque by making more minimal changes. Therefore, the present study intends to provide a 0D–3D numerical procedure for the analysis of the heat release in ICEs which will have to be further refined and validated on other types of engines with different operating conditions. Both models must be implemented, refined, calibrated and validated specifically on the engine characteristics. In order to achieve this goal, the fluid dynamic models, also known as computational fluid dynamics (CFD) models are designed in terms of both time and cost. Computer solution of a problem gives detailed and complete information. [6]. It can be used as a velocity. In this way, it helps us to question whether the changes we make are logical. CFD models are, inherently unsteady tridimensional models and are based on the conservation of mass, chemical species, momentum, and energy at any location within the engine cylinder domain. Thus, the CFD models solve the Navier–Stokes equations, and the general transport equations for each physical quantity.

2- DIESEL ENGINES

Diesel engine, any internal-combustion engine in which air is compressed to a sufficiently high temperature to ignite diesel fuel injected into the cylinder, where combustion and expansion actuate a piston. It converts the chemical energy stored in the fuel into mechanical energy, which can be used to power freight trucks, large tractors, locomotives, and marine vessels. A limited number of automobiles also are diesel-powered, as are some electric-power generator sets.

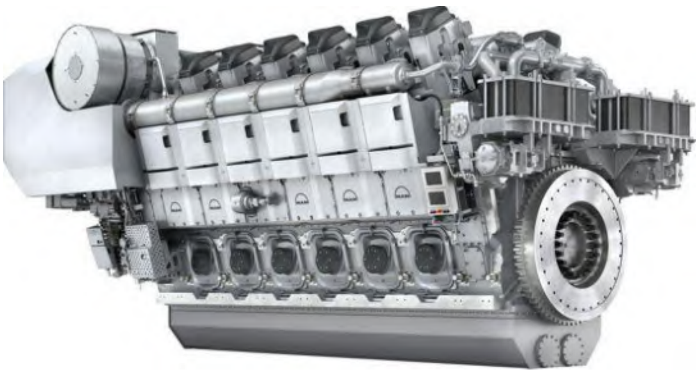


Figure 1 *MAN 48/60CR Four Stroke Diesel Engine (marine.man-es.com)*

2.1- Diesel Combustion

The diesel engine is an intermittent-combustion piston-cylinder device. It operates on either a two-stroke or four-stroke cycle; however, unlike the spark-ignition gasoline engine, the diesel engine induces only air into the combustion chamber on its intake stroke. Diesel engines are typically constructed with compression ratios in the range 14:1 to 22:1. Both two-stroke and four-stroke engine designs can be found among engines with bores (cylinder diameters) less than 600 mm (24 inches). Engines with bores of greater than 600 mm are almost exclusively two-stroke cycle systems.

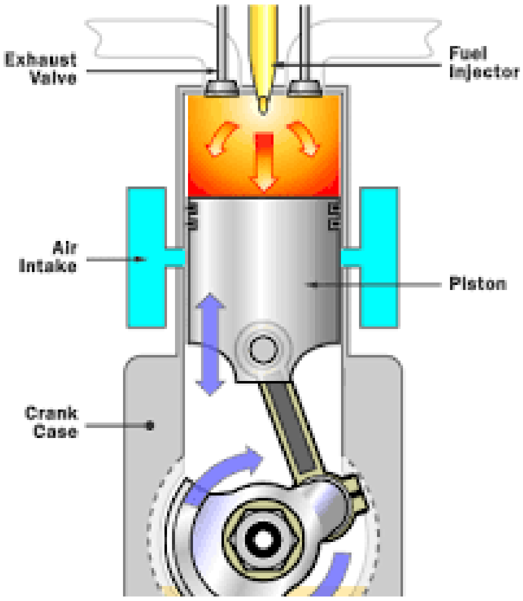


Figure 2 *(Diesel Combustion (auto.howstuffworks.com)*

2.2- Engine Cycle

A **four-stroke engine** (also known as **four-cycle**) is an internal combustion engine in which the piston completes four separate strokes which constitute a single thermodynamic cycle. A stroke refers to the full travel of the piston along the cylinder, in either direction. The four separate strokes are termed:

- Intake: this stroke of the piston begins at top dead center. The piston descends from the top of the cylinder to the bottom of the cylinder, increasing the volume of the cylinder. A mixture of fuel and air is forced by atmospheric (or greater by some form of air pump) pressure into the cylinder through the intake port.
- Compression: with both intake and exhaust valves closed, the piston returns to the top of the cylinder compressing the air or fuel-air mixture into the cylinder head.
- Power: this is the start of the second revolution of the cycle. While the piston is close to Top Dead Centre, the compressed air–fuel mixture in a gasoline engine is ignited, by a spark plug in gasoline engines, or which ignites due to the heat generated by compression in a diesel engine. The resulting pressure from the combustion of the compressed fuel-air mixture forces the piston back down toward bottom dead centre.
- Exhaust: during the *exhaust* stroke, the piston once again returns to top dead centre while the exhaust valve is open. This action expels the spent fuel-air mixture through the exhaust valve(s).

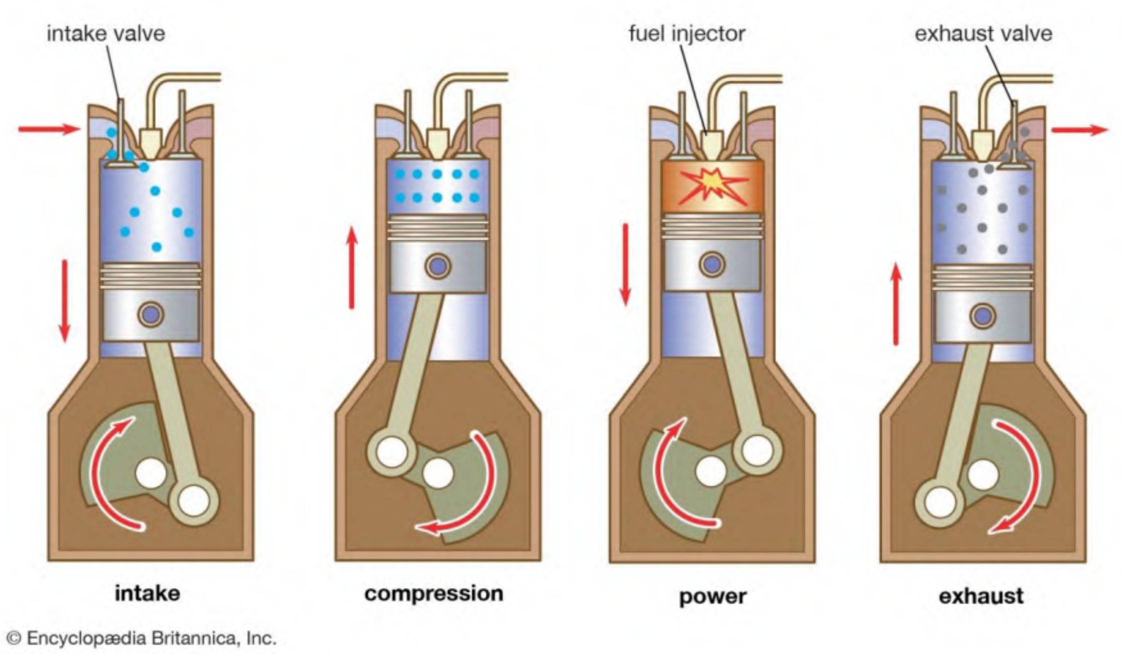


Figure 3 *Four Stroke Engine Cycle (http://mechstuff.com)*

3- Factors Affecting the Engine Performance

The heat is exchanged in both directions between the gases and engine cylinder walls and the other parts of the engine coming in contact with the gases. During combustion, expansion, exhaust and the later part of the compression, heat transfer takes place from the gases to the walls and from the wall to the cooling water or ambient air. During suction and the earlier part of the compression, heat transfer takes place from the walls to the gases. The heat lost to the walls during latter part of compression is almost equal to<the heat received by the gases from the walls during early part of compression. The amount of heat lost during exhaust stroke is unavoidable and unavailable. The heat lost during combustion and expansion lowers the thermal efficiency of the engine. The factors that affect the heat losses to the walls are as follows:

(i) Duration of combustion of the charge. This increases the heat loss.
(ii) Temperature of combustion. This is turn depends upon the fuel, compression ratio and the load on the engine. The temperature increases with load and compression ratio. This increases the thermal loss.
(iii) Speed of the engine. The increase of the engine speed decreases the duration of combustion hence decreases the heat loss.
(iv) Shape of the combustion space. The increase in ratio of combustion chamber surface to volume decreases the heat loss. However, turbulence and flame propagation also effect the heat transfer to combustion chamber wall.
(v) Size of the cylinder. The effect of cylinder size is rather complicated. An increase in the cylinder size decreases the ratio of surface to volume but increases the frame travel. This increases the combustion duration and hence engine speed is decreased.
(vi) Ignition timing in S.I. engines and fuel injection timing in C.I. engines. Proper ignition and injection timings give rise in quicker combustion with less after burning and hence less heat loss. The heat flow from the walls to fresh charge during suction stroke increases the temperature of the charge and hence decreases the quantity of charge. This decreases the power that the engine can develop.

3.1- Shape of the Combustion Space (Shape of Cylinder Head)

In order to get proper air fuel mixing, a systematic air movement also called swirl is essential, which produce higher relative velocity between fuel droplets and air. The spray cone of the injected liquid fuel gets disturbed because of air movement and turbulence inside the chamber.

3.1.1 Squish : It is an effect in IC engine which creates sudden and abrupt turbulence of fuel - air mixture as the piston approaches Top Dead Centre (TDC). Squish is a radial flow.

- During this phenomenon the piston of the engine is closest to the cylinder head.
- As the gases are suddenly squished in the combustion chamber, there is a thorough mixing of the air and fuel due to the sudden turbulence produced.
- Squish effect is generally observed in engines with overhead valves and overhead camshafts.

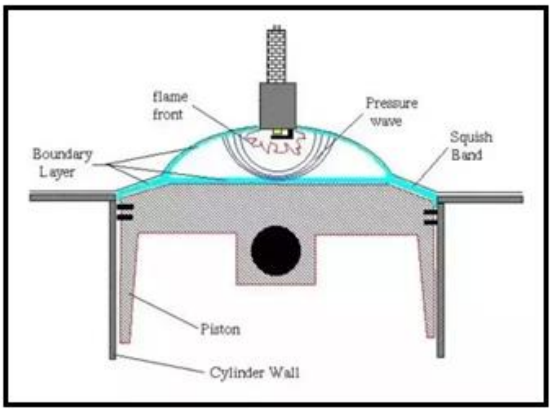


Figure 4 *IC Engine Squish Representation*

Generally, the IC engines are designed in such a way that they can obtain a maximum advantage of Squish effect as it promotes the effective mixing of air and fuel in combustion chamber. The effect particularly squeezes the mixture in that minimum area between cylinder head and piston head and promotes better and more complete combustion.

3.1.2 Tumble:

Tumble is the secondary flow generated by the squish when the piston squeezes the air - fuel mixture in the combustion chamber when it reaches the Top Dead Centre (TDC). Tumble is the circumferential flow mainly occurring near the cavity of the piston.

Introduction of the tumble in the combustion chamber is an effective way of enhancing the intensity of turbulence prior to the ignition before the end of the compression stroke. This helps in stabilizing the combustion, extending dilution limit and thereby increasing the burn rates.

Rotating flow can considerably reduce the burning period and hence increases the thermal efficiency of the engine.

Rotating flow are generally of two types *swirl* and *tumble*. The optimum rotating flow field is the combination of these two flows.

This rotational flow can substantially increase the flame propagation, reduce unexpected cyclic vibrations and expand lean limit.

Excessive rotational flows sometimes may have unexpected effects on induction system flow resistance and may hamper the heat transfer and thermal efficiency.

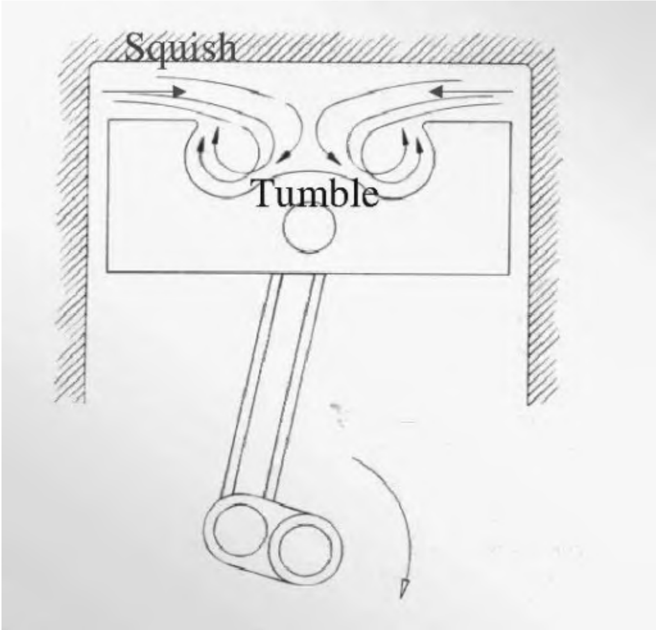


Figure 5 *Squish and Tumble Representation*

3.1.3 Swirl:

Swirl coefficient and swirl ratio Swirl coefficient is defined as the ratio of circumferential air speed in the cylinder to the axial speed of the air flow in the cylinder. The global swirl ratio, RS, is an integrated value and an estimate of the angular momentum of the total flow field during the entire intake process.

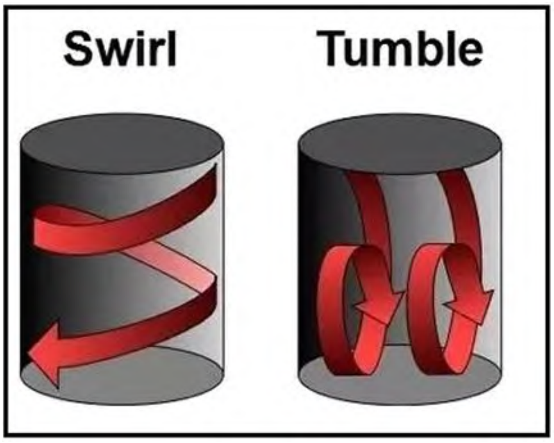


Figure 6 *Swirl and Tumble Motion in the Combustion Chamber*

During induction stroke air flows around the inlet valve and gets into the cylinder through the gap between the valve and the valve seat. Air expands into the cylinder and impinges directly on the cylinder wall creating a large angular momentum. As the air flows only through the inlet valve, the momentum of the air just below the exhaust valve is small. The large angular momentum on one side of the cylinder moves through the other side of the cylinder as the flow develops downstream and it becomes uniform there after having center of the swirl near the center of the cylinder.

3.2- INJECTION STRATEGY IN DIESEL ENGINES

A diesel fuel injection system differs from a gas engine in that the diesel fuel gets injected directly into the cylinder. In a gas engine, a port injection or a carburetor is used to inject fuel prior to the intake stroke (outside the cylinder) where the fuel mixes with air before it enters the cylinder.

The injector on a diesel engine is its most complex part. The pressure and heat within a cylinder are quite intense and the injector has to be able to handle those conditions while still distributing the fuel in a fine mist. Equally important, the mist needs to be distributed evenly throughout the cylinder. Some diesel engines use induction valves, pre-combustion chambers or another type of device to swirl air through the combustion chamber. This process helps distribute the fuel mist evenly and keeps the ignition and combustion process smooth.

Figure 7 Direct Injection (www.carid.com)

Some diesel engines help the combustion process by using a glow plug, or some other device that helps keep the air temperature in the cylinder at a sufficiently high temperature. If the air temperature is too low, the fuel won't ignite. When the engine isn't running, it obviously gets colder; as a result, a glow plug (which looks like heated coils, like on a toaster) is needed to heat the air when the car is started. By keeping the air temperature high, the engine has a better chance of starting in cold weather.

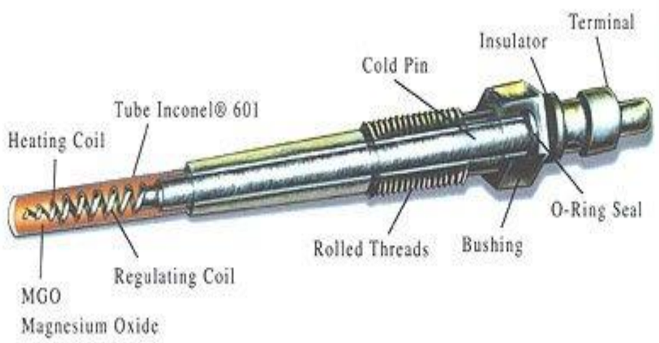


Figure 8 Glow Plug (thoughtco.com)

3.3- Valve Timing and Valve Lift

The valves opening in the cylinder determine the amount of gas that may enter (intake valve) or exit (exhaust valve) to or from the combustion chamber. It also determines the flow rate, and hence the pressure and temperature while combustion occurs. The effective compression ratio can be affected by the valves timing. In old engines, the duration of valves opening is fixed due to geometries of mechanical parts, that is, camshaft and valve lobe. Current variable valve timing systems (VVT) are limited to designed valve profile due to the dimension of lobe. There are two main methodologies in cam VVT, cam phasing which uses gear to shift the valve opening angle, and cam changing which shifts to different valve dimension when needed. The valve profile curve leads to a one-off open and close action. Electronic camless valve is under research for more valve timing flexibility for future use.

3.3.1- Reaction on Compression Ratio Using Different Valve Timing

The intake valve opening timing affects the effective compression ratio. The intake valve opening is designed to have the highest compression ratio in specification. With earlier or later intake valve closing, compression ratio is smaller than the specification. The amount of air or air/fuel mixture that enters into cylinder depends on the lift of the valve where maximal lift is reduced when the opening duration is reduced, and hence maximum volume is reduced. The compression ratio is an important parameter of engine torque.

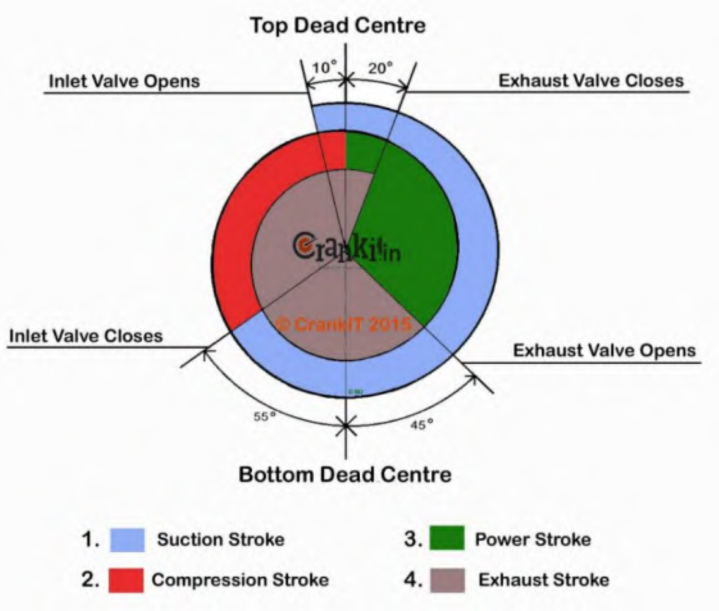


Figure 9 Valve Timing (berdengkape.com)

3.3.2- Valve Overlapping

Valve overlapping happens when both intake and exhaust valve open for the same period , Typically, the period before the piston reaches the TDC in exhaust stroke and after the piston moves beyond the TDC in intake stroke. The advantage for valve overlapping is continuous variable valve timing (VVT) which provides adaptive adjustment to improve engine torque delivery across different revolution range . When the engine speed is high, more air is needed for higher energy combustion; therefore, duration of intake valve opening needs to be longer. A portion of exhaust gas will suck back to the combustion chamber while the exhaust valve is opening, and hence increase the mixture pressure and temperature. At present, several vehicle manufacturers offer mechanical VVT, by means of cam phasing VVT. In this approach, the gear attached to the valve moves the position to change the two mechanical lobes and hence switch to different profiles. These profiles correspond to the amount of valve lifting.

Negative valve overlapping (NVO) consists of closing both valves for a period of time. Exhaust valve closes before piston reaches the TDC, and intake valve opens after piston moves beyond the TDC. The concept is similar to the role of the exhaust valve in exhaust gas recirculation systems. When the intake valve opens, the hot residual gas heats up the fresh mixture, and the high temperature mixture is enhanced for compression ignition. The results, presented in , show that the injection timing is an important parameter. The new fuel injection strategy by injecting a portion of fuel during the negative valve closing interval and inject the rest in the intake stroke was also examined in.

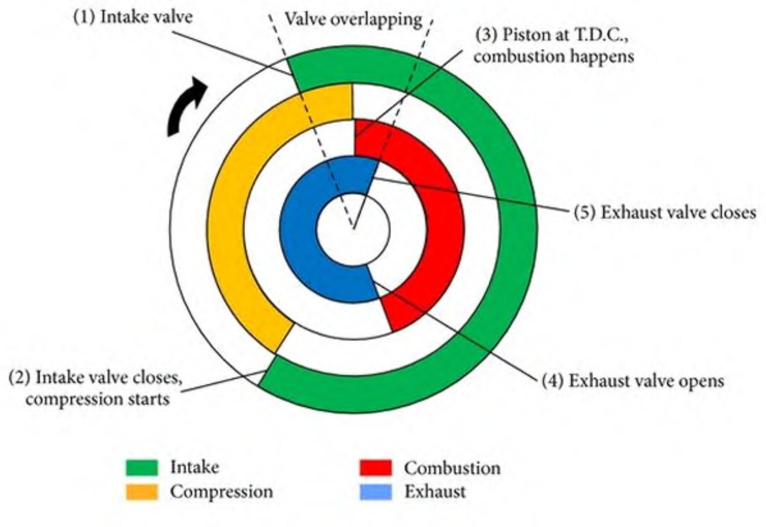


Figure 10 *Valve Overlapping Comparison of Engine Simulation Software for Development of Control System (2013)*

3.4- Compression Ratio

The compression ratio typically ranges from 8 to 12 for spark-ignition ICEs, and it is in the range of 12–24 for compression-ignition ICEs. When compression ratio is high, the fuel and air particles are closely packed together and obtain a higher explosive velocity during combustion, that is, better performance and better fuel consumption. However, it is limited by two considerations: material strength due to maximum stress at certain pressure and engine knock due to the maximum temperature during process exceeds autoignition temperature.

3.5- Fuel Properties

Historically, fuel properties have been continuously changing for various reasons, including crude oil prices, crude oil quality, refinery technologies, relative demand for diesel and gasoline fuel, and changing engine technologies. In the recent years environmental considerations and emission legislation have been increasingly more important in the formulation and properties of fuels. The interaction mechanisms between fuel quality, engine technologies, and emissions need to be understood to find the most effective approach towards low-emission diesel engines.

3.5.1- Comparison Between Biodiesel and Diesel fuel Cylinder Pressure

As we can see from the researchs made about this comparison, also one of them [cfd simulation biodiesel] make a statement in cylinder pressure with 3 biodiesel and traditional diesel fuel. The differences in cylinder pressure between diesel and biodiesel blends can be seen from the below figure.

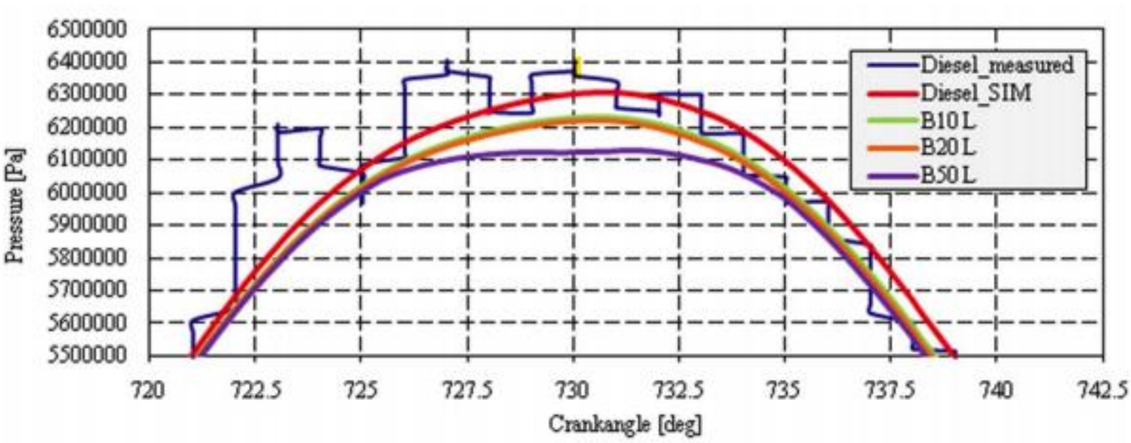


Figure 10. Cylinder pressure for diesel, B10 B20 and B50

Figure 11 *CFD SIMULATION OF A SINGLE CYLINDER RESEARCH ENGINE (pg.7)*

The pressure inside the combustion chamber using pure diesel is 63.29 bars, for B10 it is 62.18 bars, for B20 it is 62.32 bars and for B50 it is 61.26 bars; so the pressure decreases to 97% using B50 in comparison to pure diesel.

3.5.2 Temperature in the combusiton chamber

The temperature inside the combustion chamber during the diesel and biodiesel burn is presented in figure below.

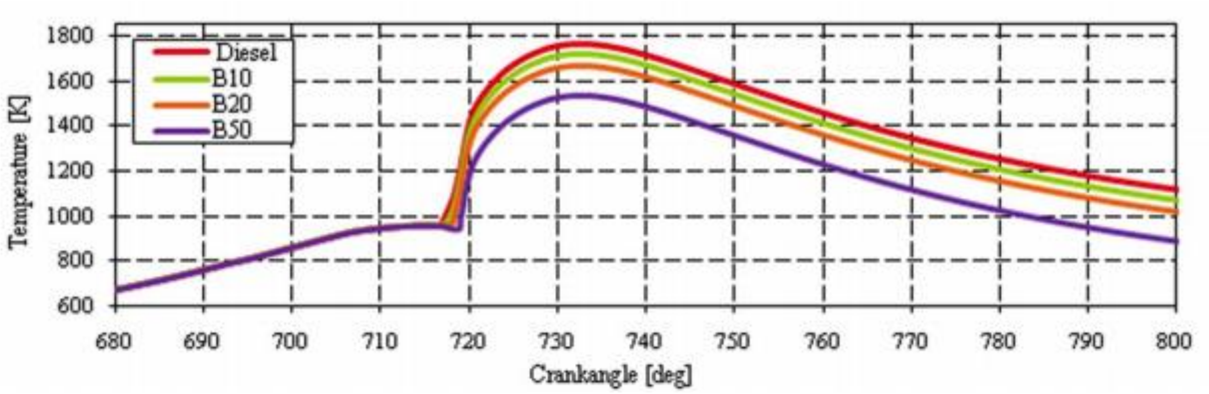


Figure 12 *CFD SIMULATION OF A SINGLE CYLINDER RESEARCH ENGINE(pg. 8)*

The temperature inside the combustion chamber reaches from 1761 K (when using diesel) to 1534K (when using B50), so the temperature decreases to 87% when using B50, to 94% when using B20 and to 97% when using B10 in comparison to diesel.

4- ANTOR 3LD 510

Antor 3LD 510 is a single cylinder, four-stroke, direct injection, diesel engine**. 3LD 510** is used for many applications, especially in the field of agriculture. Its main applications are: farm machinery**, tractors**, motor mowers, **rotary hoes**, lawn mowers, **cement mixer**, dumper, minidumper, mini excavator, motor compressor, vibrating plate, roller, asphalt cutter, generator, refrigerating groups, motor welder, motor sweepers, aerial platform.

***Table 1** The Technical Properties of Antor 3LD510*

Main Specs	
Number of cylinders	1
Bore and stroke (mm)	85 x 90
Displacement (cm3)	510
Aspiration	Naturally aspirated
Cycle	4 stroke
Combustion system	Direct Injection
Rotation	Counter-clockwise (view from main PTO side)
Cooling system	Air
Fuel tank capacity (l)	5.3
Oil sump capacity (l)	1.75
Lenght (mm)	466
Width (mm)	422
Height (mm)	568
Dry weight (kg)	60
Max. power kW (hp) @rpm	7.3 (10.0) @3000
Maximum Torque (Nm@rpm)	33.5@1800
Emission Compliance	No Regulations

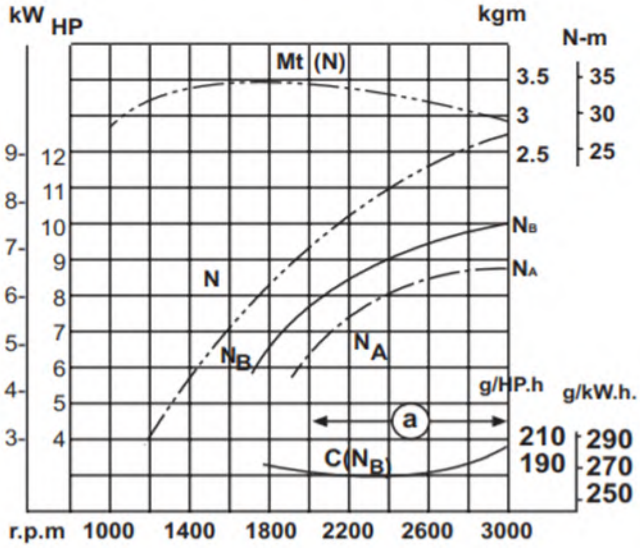


Figure 13 *Performance curve Engine Lombardini 3LD 510 Diesel (3LD 510 Catalogue)*

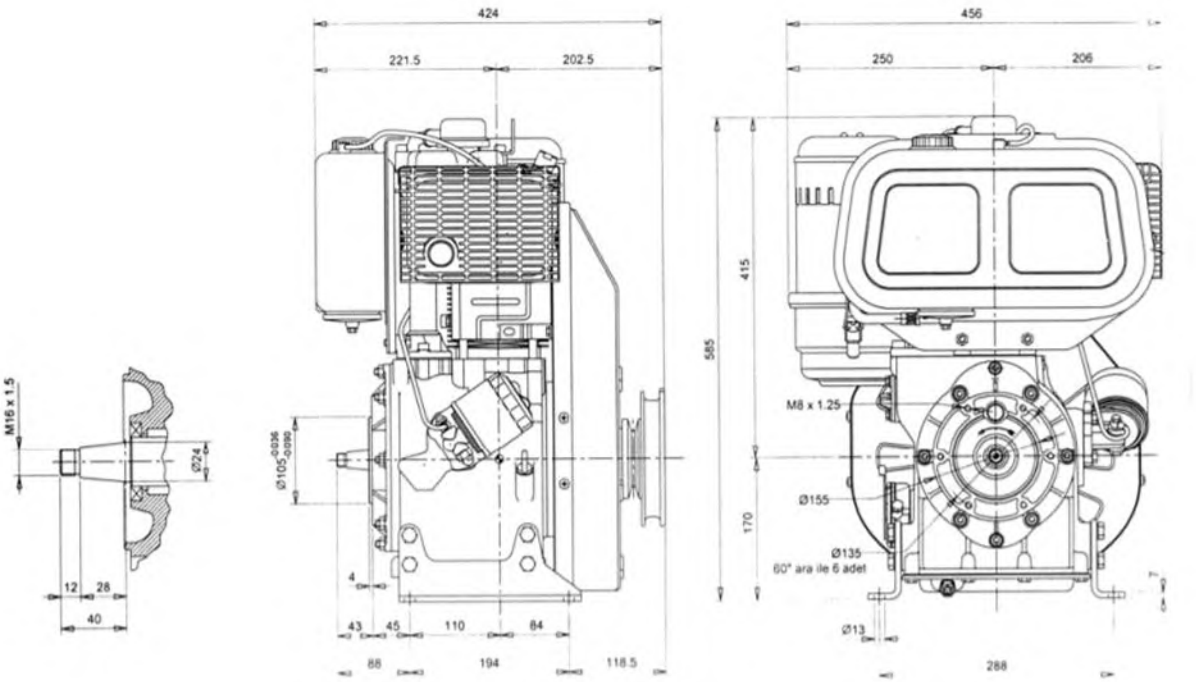


Figure 14 *Technical Drawing of 3LD 510*
(3LD 510 Catalogue)

5- SOFTWARES

5.1 Software Selection

We initially chose to use the gt power program for 1d analysis. The reason for starting with 1d dimensional solution is to pass the cold start which is going to last for a lot of time in 3D dimensional analysis and to find the initial values for the heated engine conditions. The reason we choose Gt power is that it is the most advanced program for user intervention between 1d programs. In addition to this, lotus engine simulation data results will be used to compare with each other. Also the use of LES is easy to learn and use.

Ansys forte has been selected for 3D analysis. Of course, the biggest reason is that the university has a ANSYS license. Secondly, it is the basis of the analysis that has the technologies that will provide a lot of convenience in the meshing. Of course, tons of articles on the internet and video explanations helped us in our selection.

5.2- ANSYS FORTE

ANSYS Forte incorporates proven ANSYS Chemkin-Pro solver technology for modeling and simulating gas phase and surface chemistry. While legacy engine-combustion CFD simulations utilize chemistry solvers that are too slow to handle the chemistry details required for accurate predictions of ignition and emissions, Forte uses multicomponent fuel models combined with comprehensive spray dynamics, without sacrificing simulation time-to-solution.

Key benefits of the FORTE :

• Automatic mesh generation that eliminates weeks of effort typically spent on manual mesh preparation .

• True multicomponent fuel-vaporization modelsthat enable a self-consistent representation of the physical spray and the kinetics for accurateprediction of fuel effects.

• Advanced spray models that dramatically reduce grid and time-step dependency when compared to existing approaches.

• The ability to track soot particle formation, growth, agglomeration and oxidation without a computetime penalty to predict particle size and number.

5.3- GT POWER

GT POWER is the industry standard engine p4.3erformance simulation**,** used by all major engine manufacturers and vehicle OEM**s**. GT-POWER is used to predict engine performance quantities such as power, torque, airflow, volumetric efficiency, fuel consumption, turbocharger performance and matching, and pumping losses, to name just a few. Beyond basic performance predictions, GT-POWER includes physical models for extending the predictions to include cylinder and tailpipe-out emissions, intake and exhaust system acoustic characteristics (level and quality), in-cylinder and pipe/manifold structure temperature, measured cylinder pressure analysis, and control system modeling.

Key benefits of the GT POWER :

- Highly accurate, fully predictive, multi-pulse diesel combustion model
- Tumble sensitive, turbulent SI combustion model
- Complete chemical kinetics library
- Vehicle model for integrated engine/vehicle simulations
- Directly coupled with Simulink, Star CD, Converge, Fluent and other codes
- Flexibility to study any valving concept, infinitely variable VVT and VVL, as well as cylinder deactivation concepts

6 – GT-POWER SIMULATION

6.1 – Defining Objects

6.1.1 Inlet Environment

The first stage to create a single-cylinder engine model is to define the boundary conditions of inlet. To do this, a template ' EndEnvironment ' will be used in this model. We named the ' env-inlet ' item and entered the following values in the attributes.

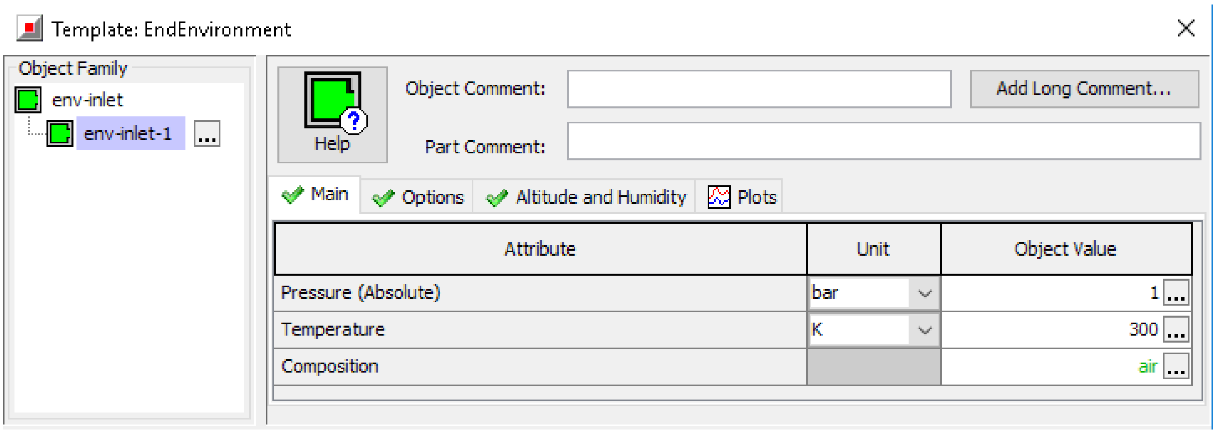


Figure 15. End Environment Template for Inlet

We selected composition for inlet as air under 'Fluid Mixture' in 'Value Selector'.

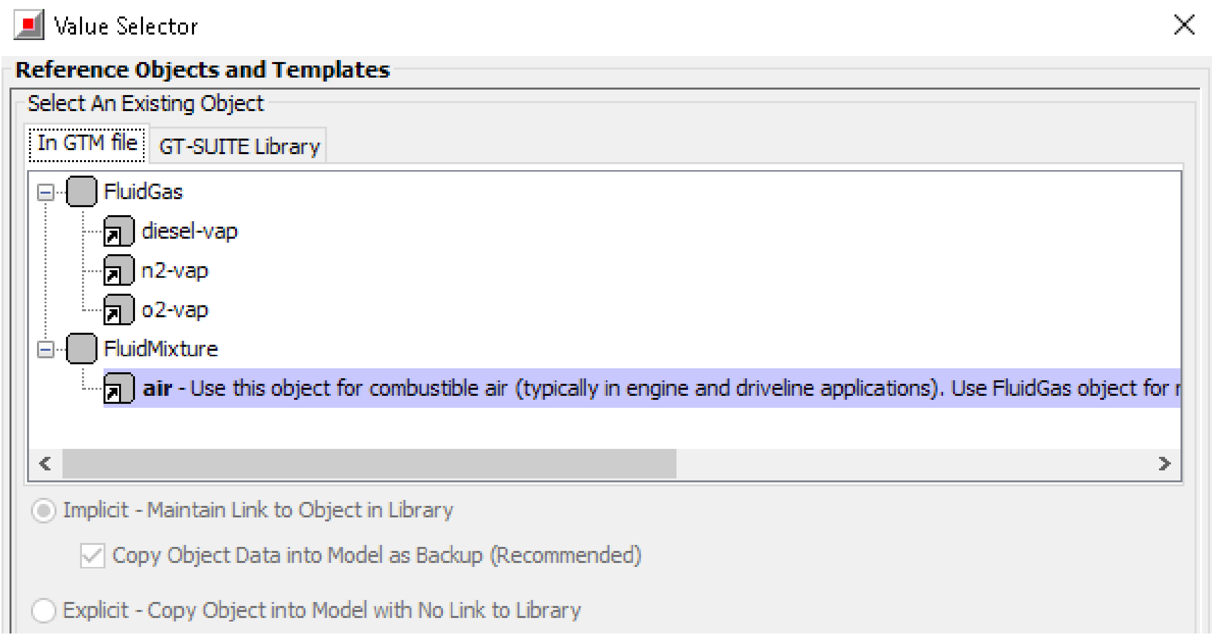


Figure 16. Composition Selection for Inlet EndEnvironment

6.1.2 Intake Runner

The next step is to build an intake runner to connect the ' EndEnvironment ' to the intake port. To generate a pipe object, we selected the ' PipeRound ' template within the project library. We named the ' intrunner ' item and filled in the attributes in the Main and Thermal folder characteristics with the following values.

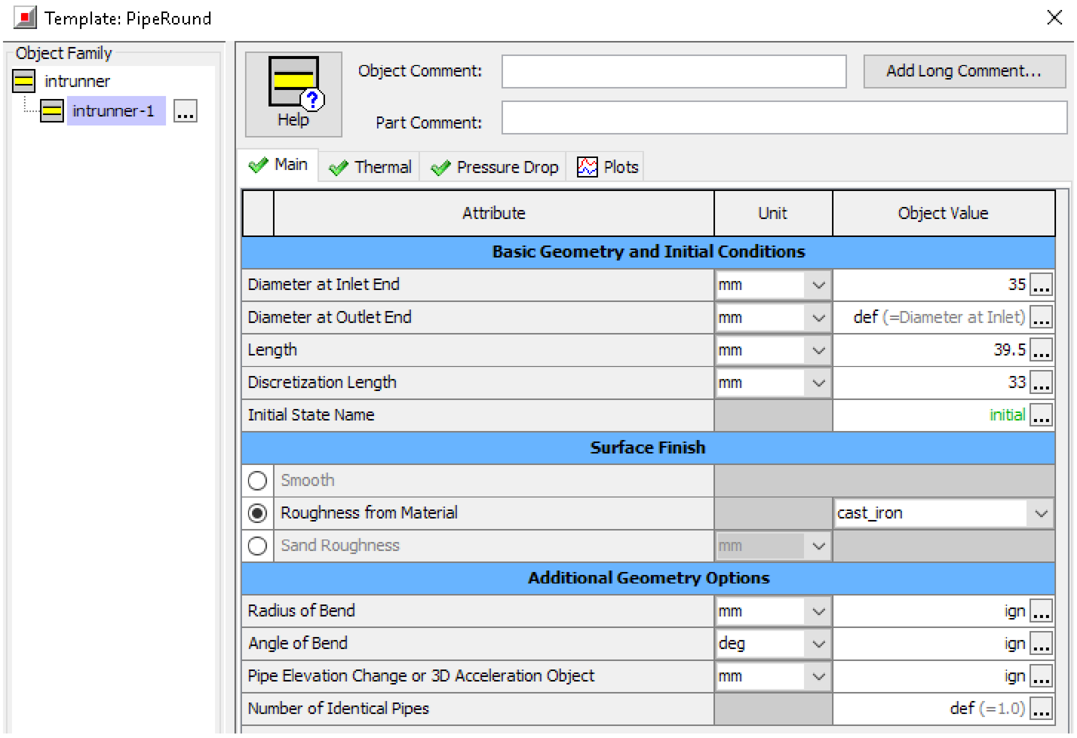


Figure 17. Intake Runner Template

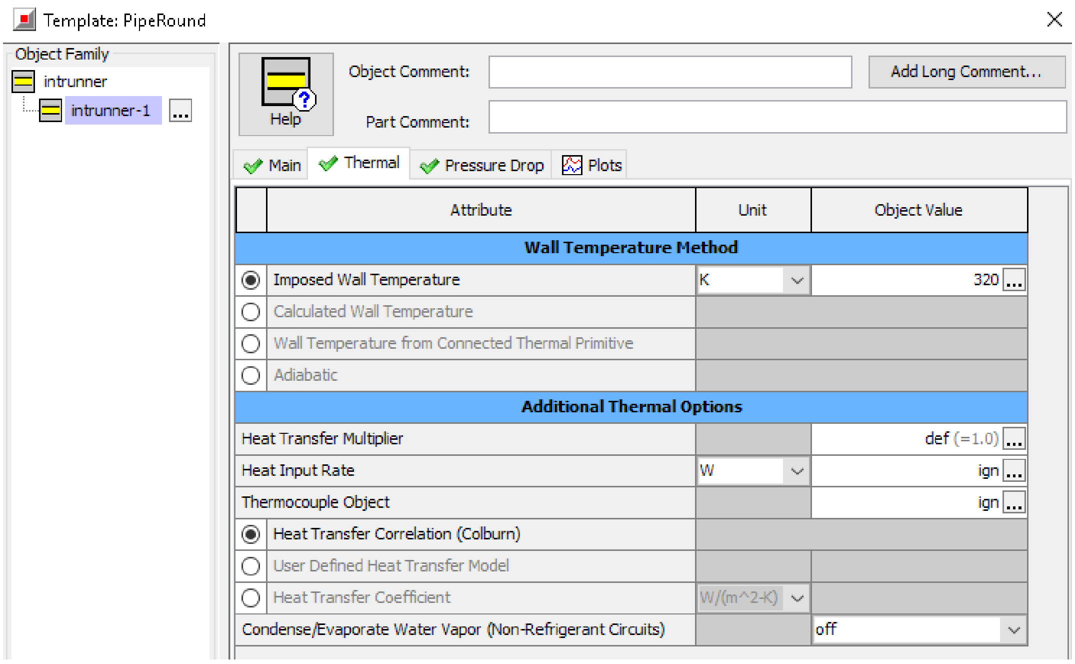


Figure 18. Thermal Conditions for Intake Runner

6.1.3 Intake Port

The next step is to create an object that identifies the intake port of the engine. We selected Pipe Round object in the Project Library and selected Copy and Edit Object. We changed the object name such as "intport-1 and filled in the values as in the following figures.

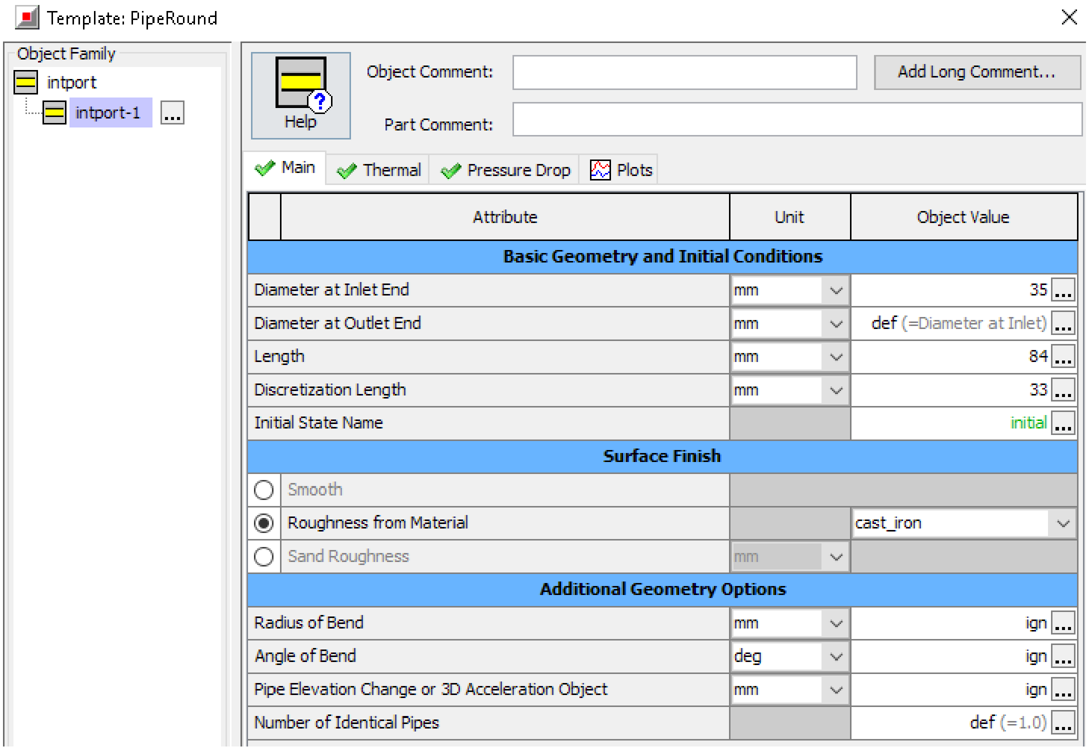


***Figure 19**. Intake Port Template*

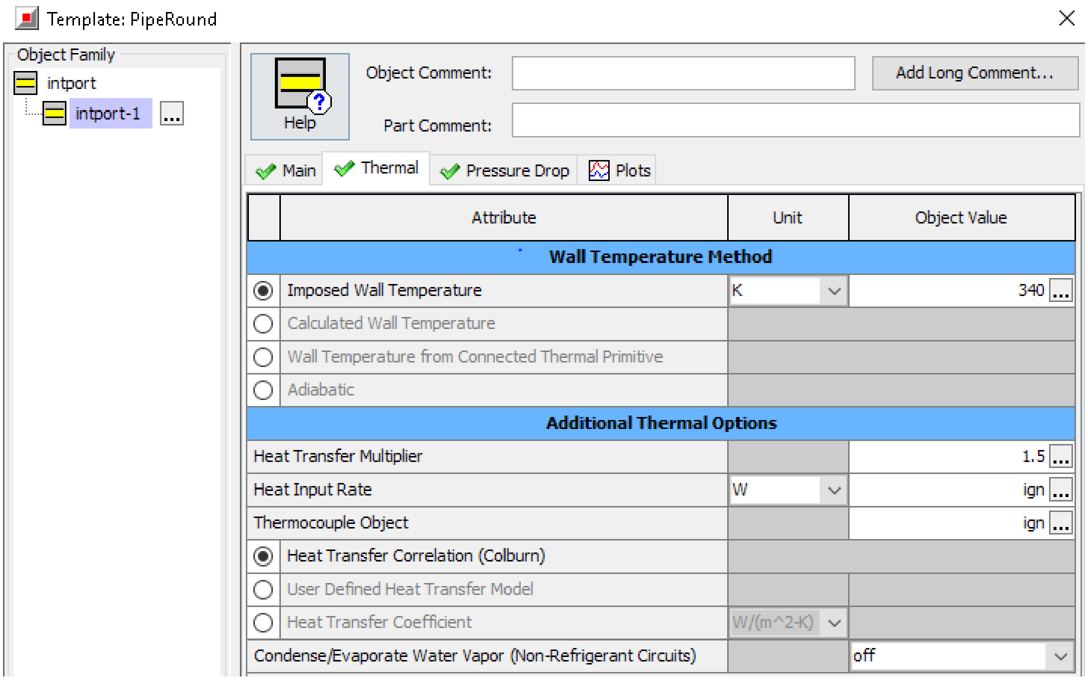


***Figure 20**. Thermal Conditions for Intake Port*

6.1.4 Intake Valve

We used the ValveCamConn object to create the intake valve. The ValveCamConn object contains information such as cam timing angle, lift array data, and forward and reverse flow coefficient data.

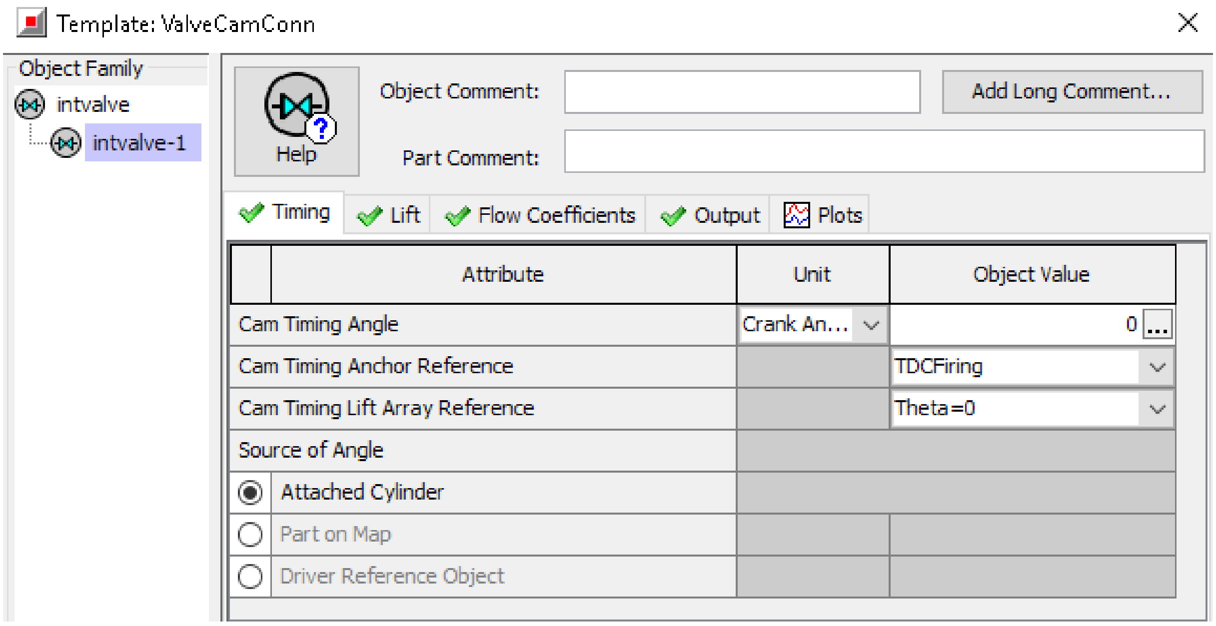


Figure 21. ValveCamConn Template

We entered lift data of intake valve of the engine as shown in the Figure 20.

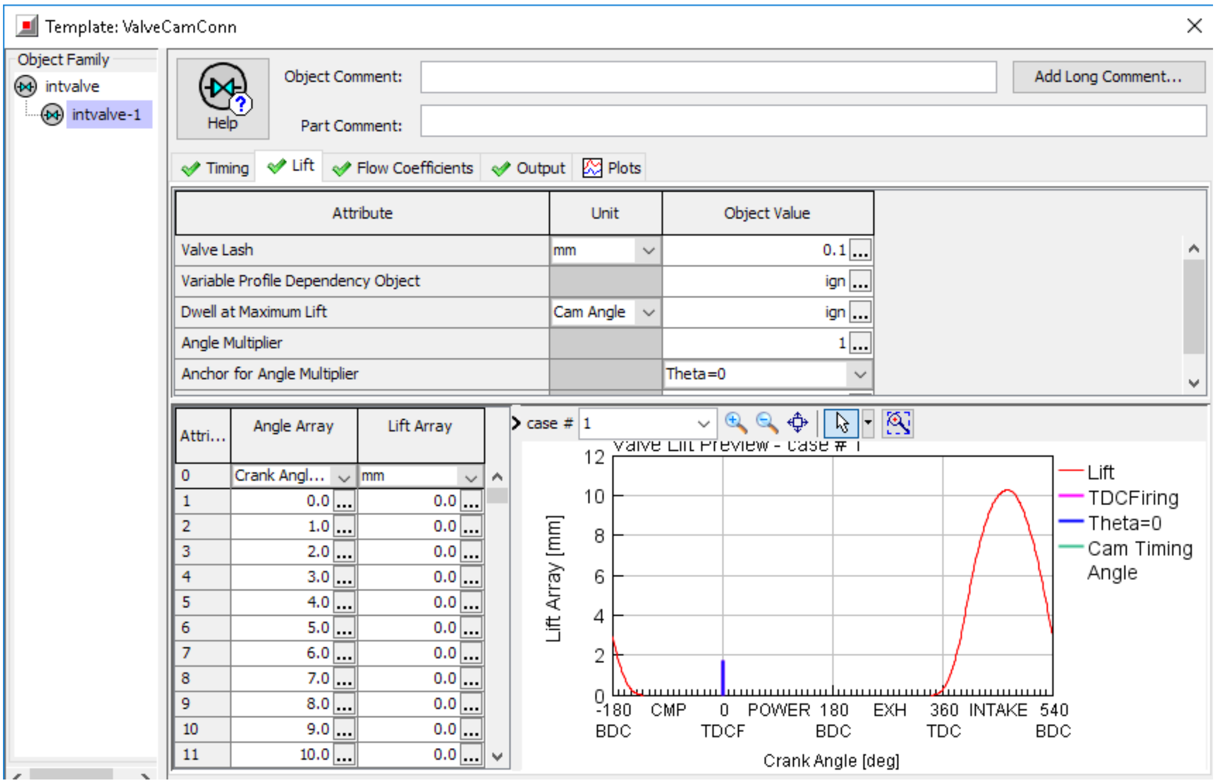


Figure 22. Intake Valve Lift Profile

We used GT Discharge Coefficient Generator to obtain intake flow coefficients. Required engine parameters were entered as shown in the Figure 21.

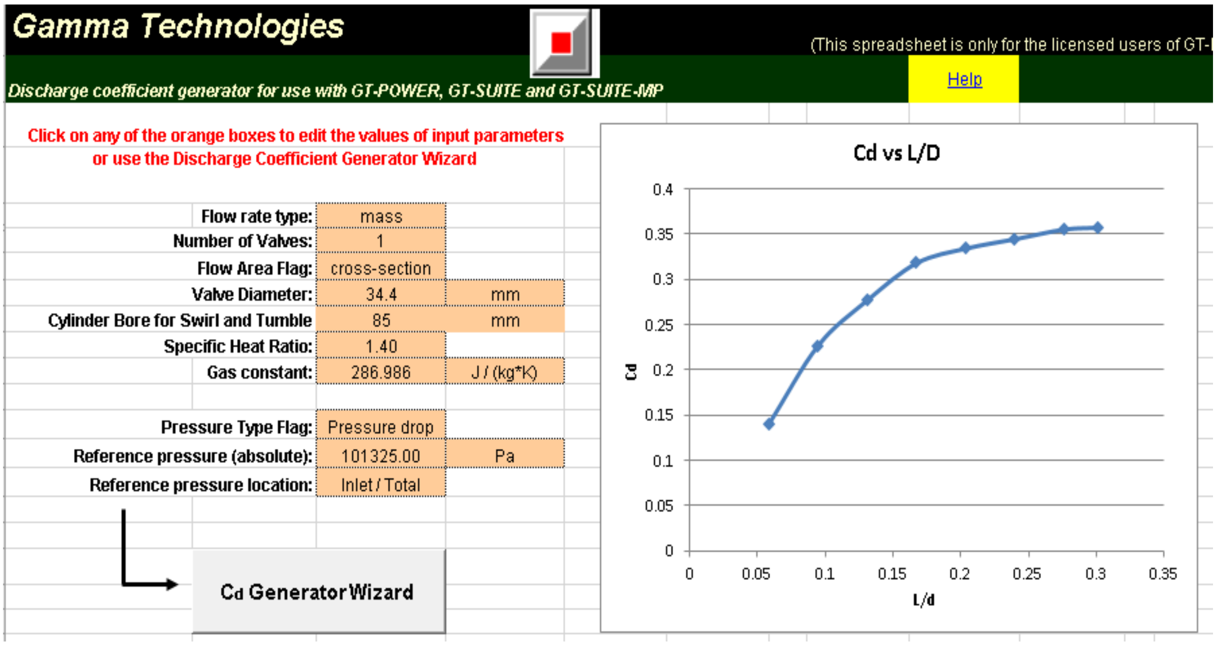


***Figure 23**.GT Discharge Coefficient Generator Interface*

In Input Section, pressure drops, flow rates and upstream stagnation temperatures entered for eight different intake valve lifts. Flow rate values were obtained from ANSYS Fluent Port Flow Simulation. This simulation will be mentioned section …

Input Section					
Lift	Upstream Stagnation Temperature	Pressure Drop (See cells C14, C15 & C16)	Flow Rate (See cell C5)	Swirl Torque (Optional)	Tumble Torque (Optional)
mm	K	Pa	kg / s	Nm	Nm
2	300	5000	0.01373		
3.25	300	5000	0.0223		
4.5	300	5000	0.0272		
5.75	300	5000	0.0312		
7	300	5000	0.0328		
8.25	300	5000	0.0338		
9.5	300	5000	0.0349		
10.345	300	5000	0.0350		

***Figure 24** Input Section of GT Discharge Coefficient Generator*

Intake valve discharge coefficients for 8 different lifts were obtained as shown in the Figure 23.

Output Section					
Reference Flow Area	Density at Throat	Isentropic Velocity	Effective Area	L/D	Discharge Coefficient
mm^2	kg/m^3	m/s	mm^2		
929.407986	1.135105887	93.0113532	130.046359	0.0581395	0.139923867
929.407986	1.135105887	93.0113532	210.839909	0.0944767	0.22685399
929.407986	1.135105887	93.0113532	257.440644	0.130814	0.276994224
929.407986	1.135105887	93.0113532	295.611571	0.1671512	0.318064376
929.407986	1.135105887	93.0113532	310.576848	0.2034884	0.334166321
929.407986	1.135105887	93.0113532	320.237976	0.2398256	0.344561248
929.407986	1.135105887	93.0113532	330.183254	0.2761628	0.355261908
929.407986	1.135105887	93.0113532	331.793442	0.3007267	0.356994396

Figure 25. Intake Valve Discharge Coefficients

After this process, obtained discharge flow coefficients for eight different valve lifts were entered in ValveCamConn / Flow Coefficient template.

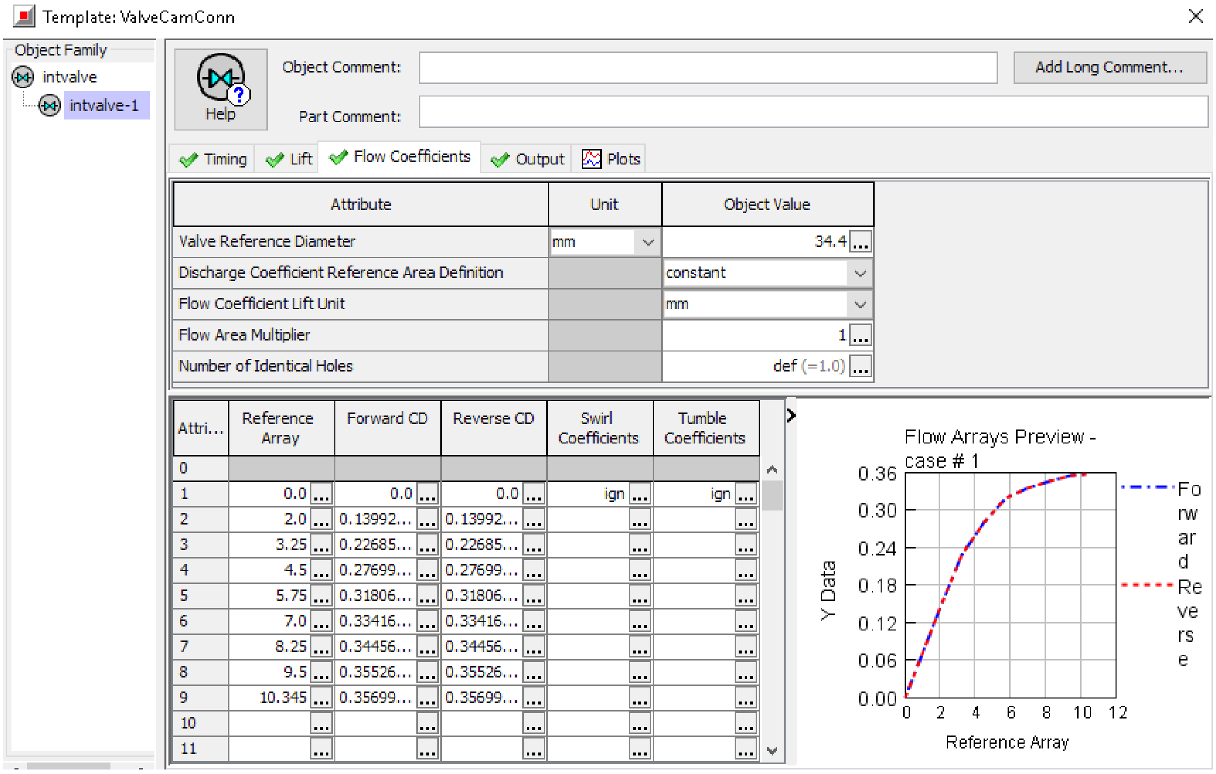


Figure 26. Intake Valve Discharge Coefficients for 8 Different Valve Lifts

6.1.5 Cylinder

We used the 'EngCylinder' template located in the project library to create cylinder. The EngCylinder contains sub-models for the cylinder such as geometry, wall temperature, heat transfer, in-cylinder flow, and combustion.

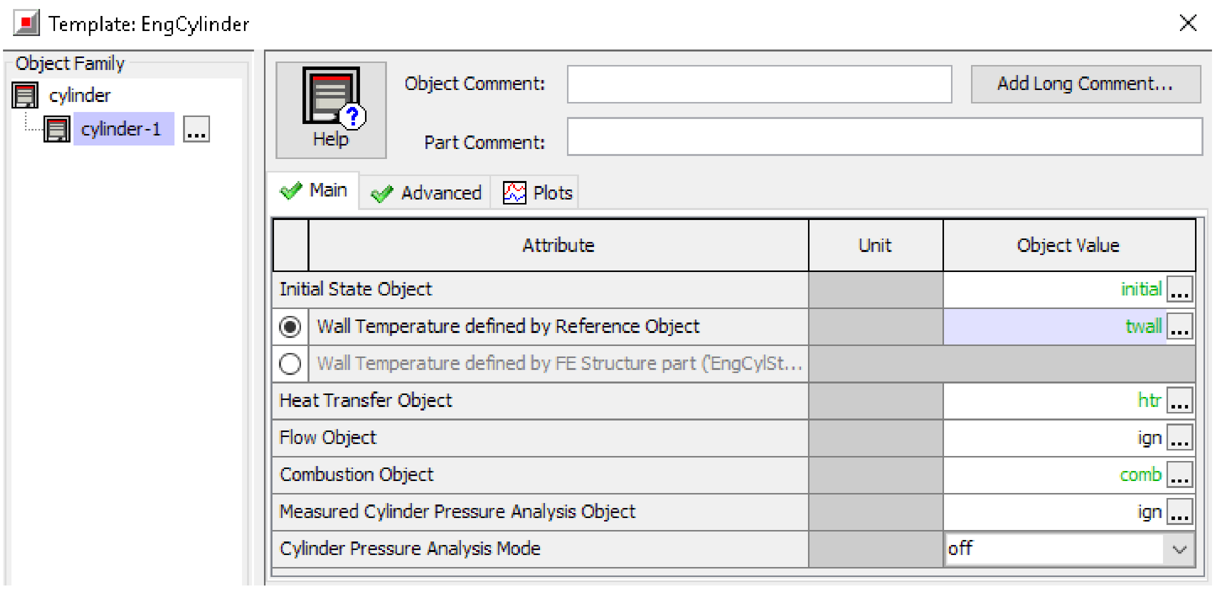


Figure 27. Engcylinder Template

For the next attribute, 'Wall Temperature defined by Reference Object', we used Value Selector and selected 'EngCylTWall' under 'In Template Library'. This object is a simple cylinder wall temperature object used to define three constant imposed wall temperatures, one for the head, one for the piston, and one for the cylinder/liner.

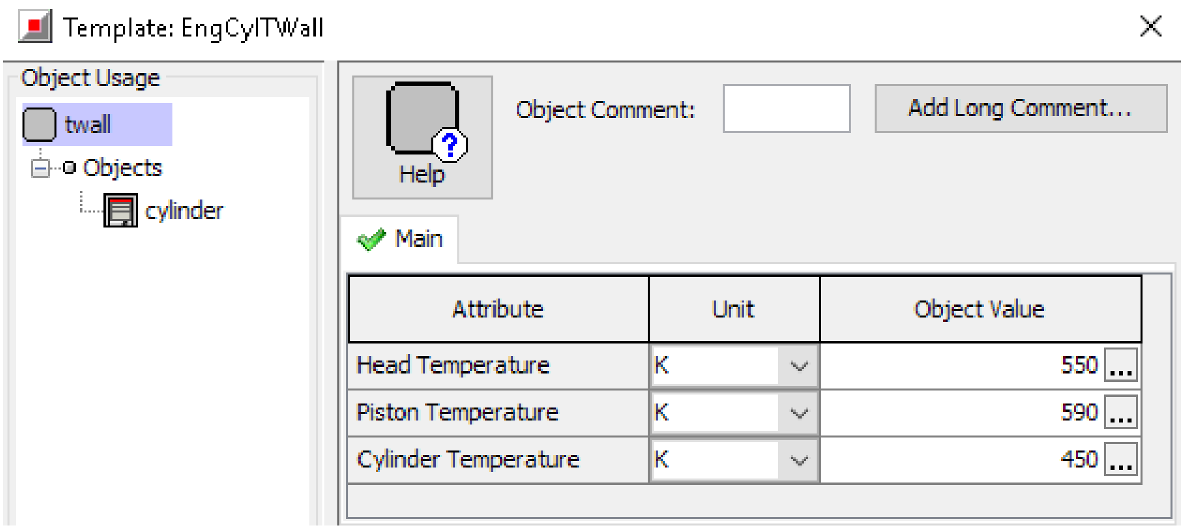
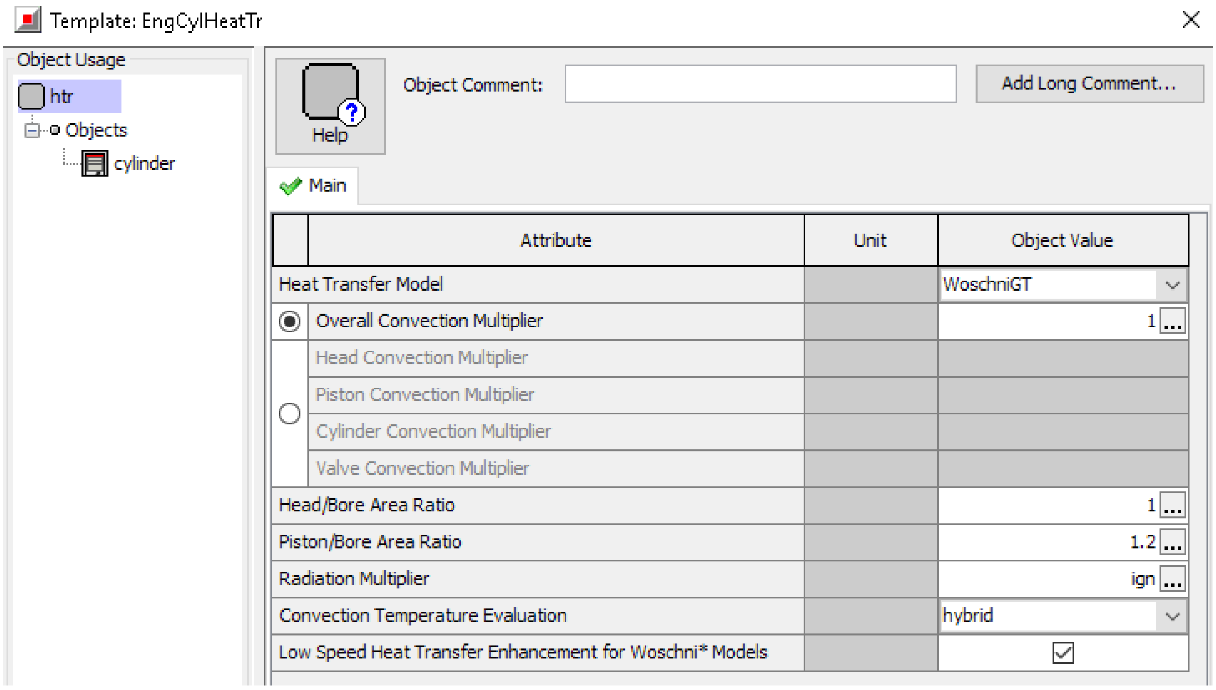


Figure 28. EngCylTwall Template

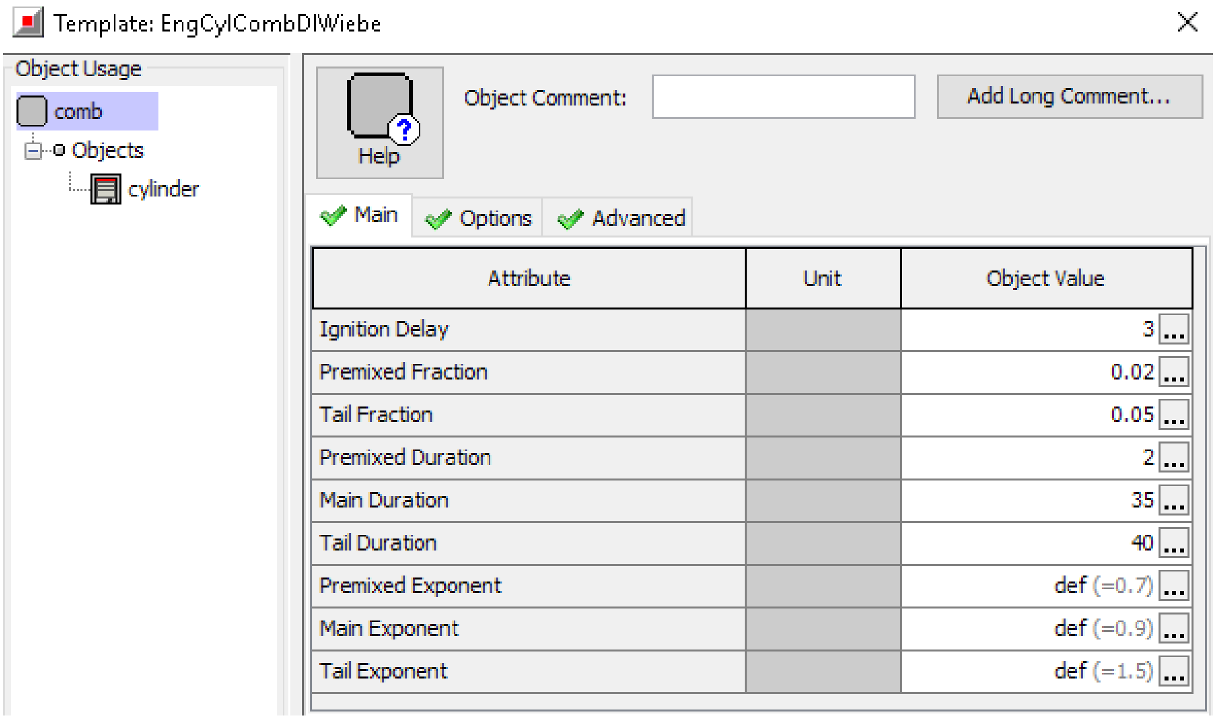
We selected heat transfer model as WoschiGT using EngCylHeatTr template under ‘In Template Library’.



Attribute	Unit	Object Value
Heat Transfer Model		WoschniGT
Overall Convection Multiplier		1
Head Convection Multiplier		
Piston Convection Multiplier		
Cylinder Convection Multiplier		
Valve Convection Multiplier		
Head/Bore Area Ratio		1
Piston/Bore Area Ratio		1.2
Radiation Multiplier		ign
Convection Temperature Evaluation		hybrid
Low Speed Heat Transfer Enhancement for Woschni* Models		☑

Figure 29. *EngCylHeatTr Template*

The combustion burn rate will be modeled with a Wiebe curve. EngCylCombDIWiebe template was selected under ‘In Template Library’ because our engine is a diesel engine.



Attribute	Unit	Object Value
Ignition Delay		3
Premixed Fraction		0.02
Tail Fraction		0.05
Premixed Duration		2
Main Duration		35
Tail Duration		40
Premixed Exponent		def (=0.7)
Main Exponent		def (=0.9)
Tail Exponent		def (=1.5)

Figure 30. *EngCylCombDIWiebe Template*

6.1.6 Fuel Injector

We selected di-inject object under InjDieselSimpleConn to set injection properties of the engine. Injected mass, fluid object, injected fluid temperature, injection timing and injection duration values were entered as shown in the Figure 29.

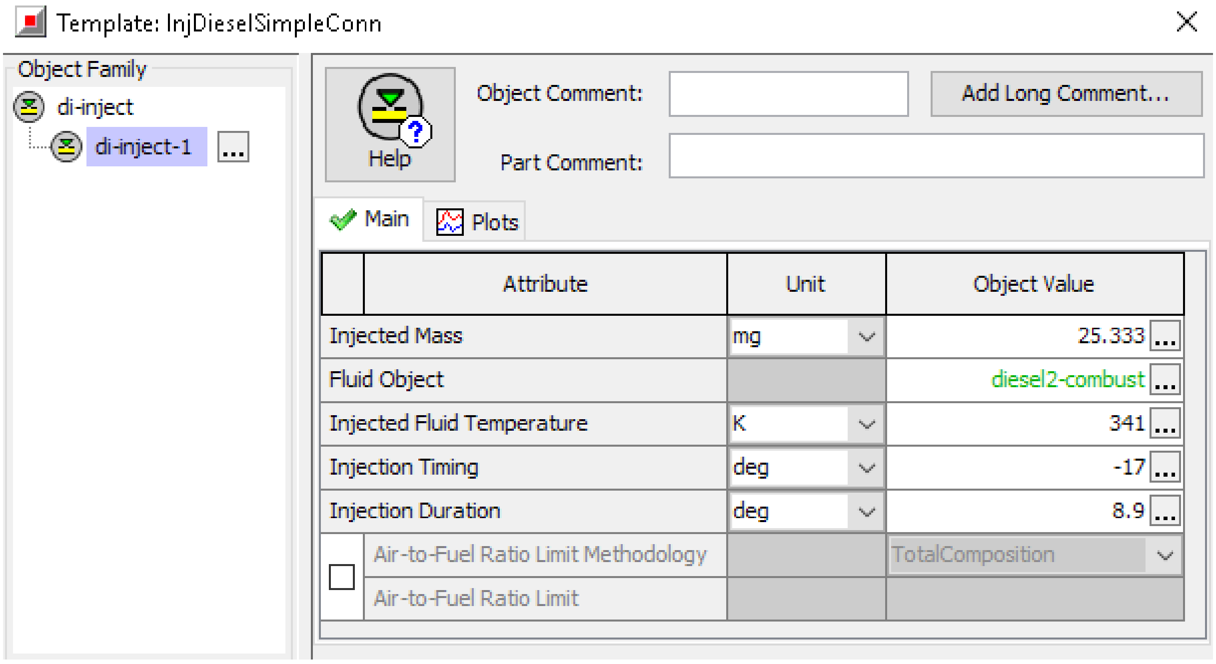


Figure 31. InjDieselSimpleConn Template

6.1.7 Engine Crank Train

EngineCrankTrain template was used to create engine crank train. This template contains Main, Cylinder Geometry, Firing Order, RLT Norms, Inertia and Bearing Loads section. For the Main section, entered values are shown in the Figure 30.

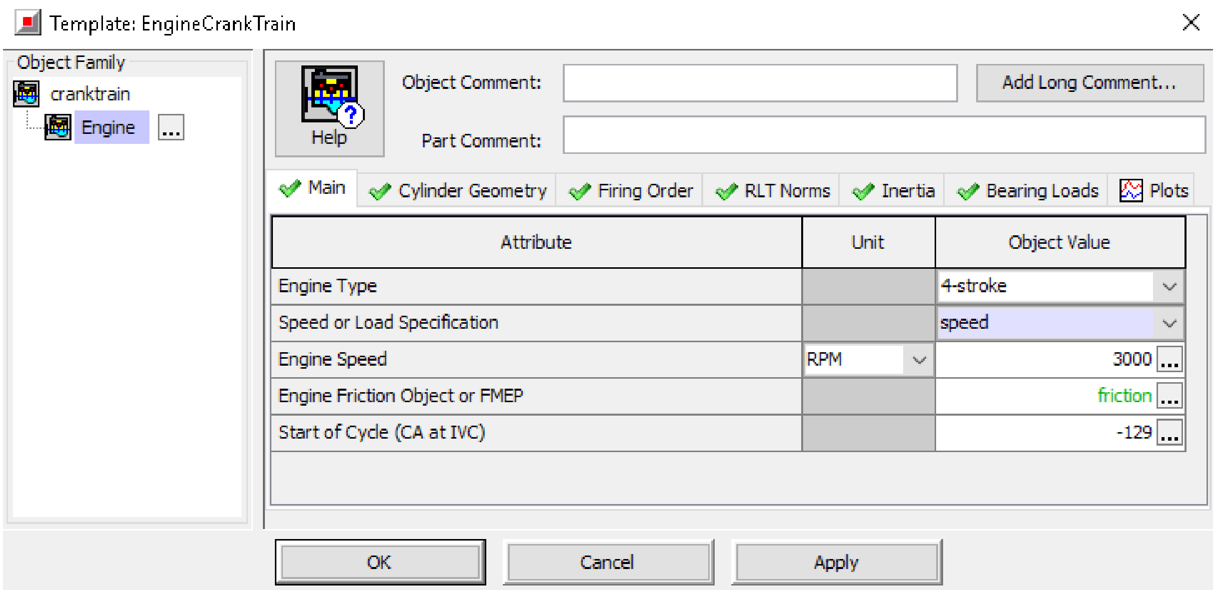


Figure 32. EngineCrankTrain Template

Properties of the cylinder geometry are shown in the Figure 31.

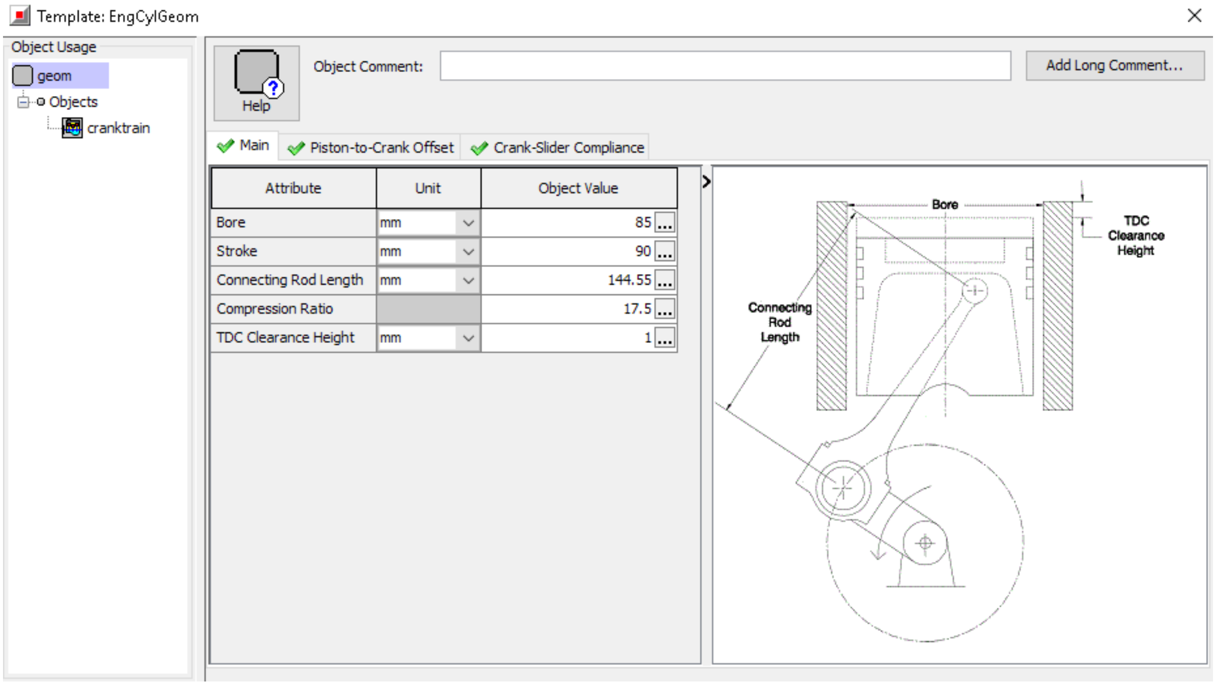


Figure 33. Cylinder Geometry Properties

Since we used a single cylinder engine, we did not make any changes to the combustion order.

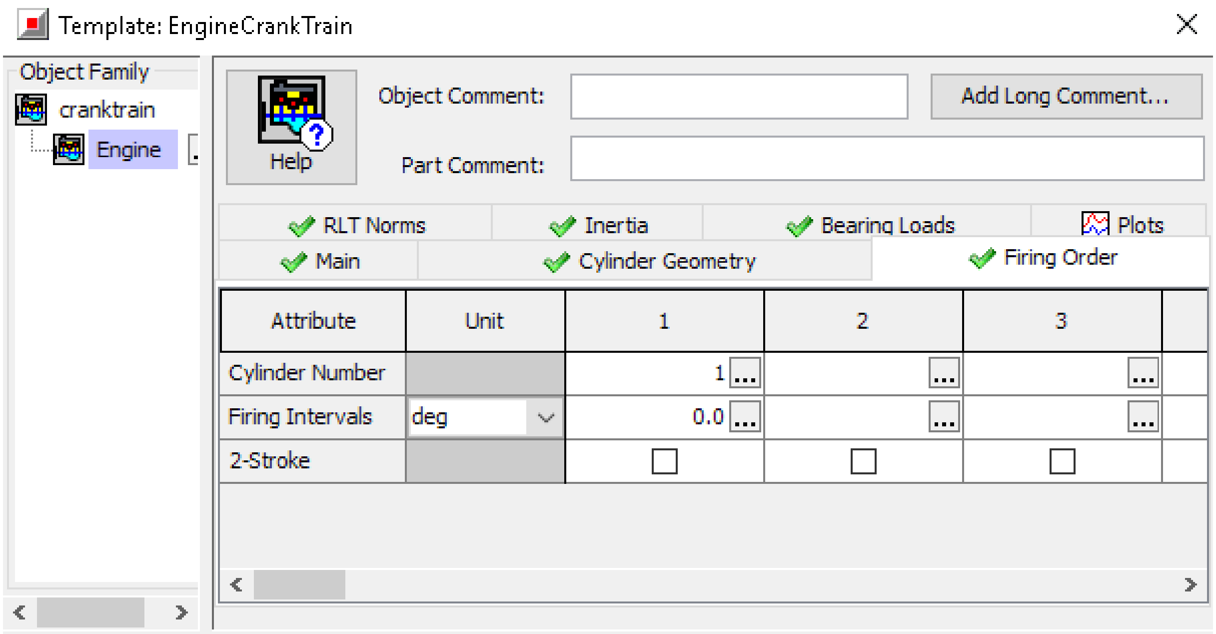


Figure 34. Firing Order Properties

6.1.8 - Exhaust Valve

We used the ValveCamConn object to create the exhaust valve. The ValveCamConn object contains information such as cam timing angle, lift array data, and forward and reverse flow coefficient data.

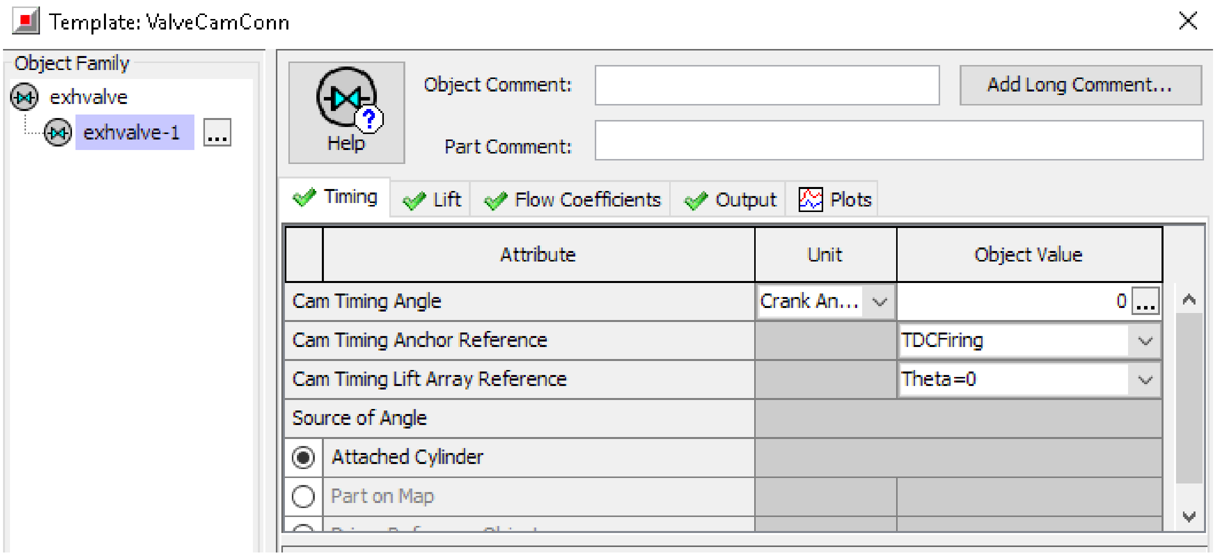


Figure 35. ValveCamConn Template for Exhaust Valve

We entered lift data of exhaust valve of the engine as shown in the Figure 34.

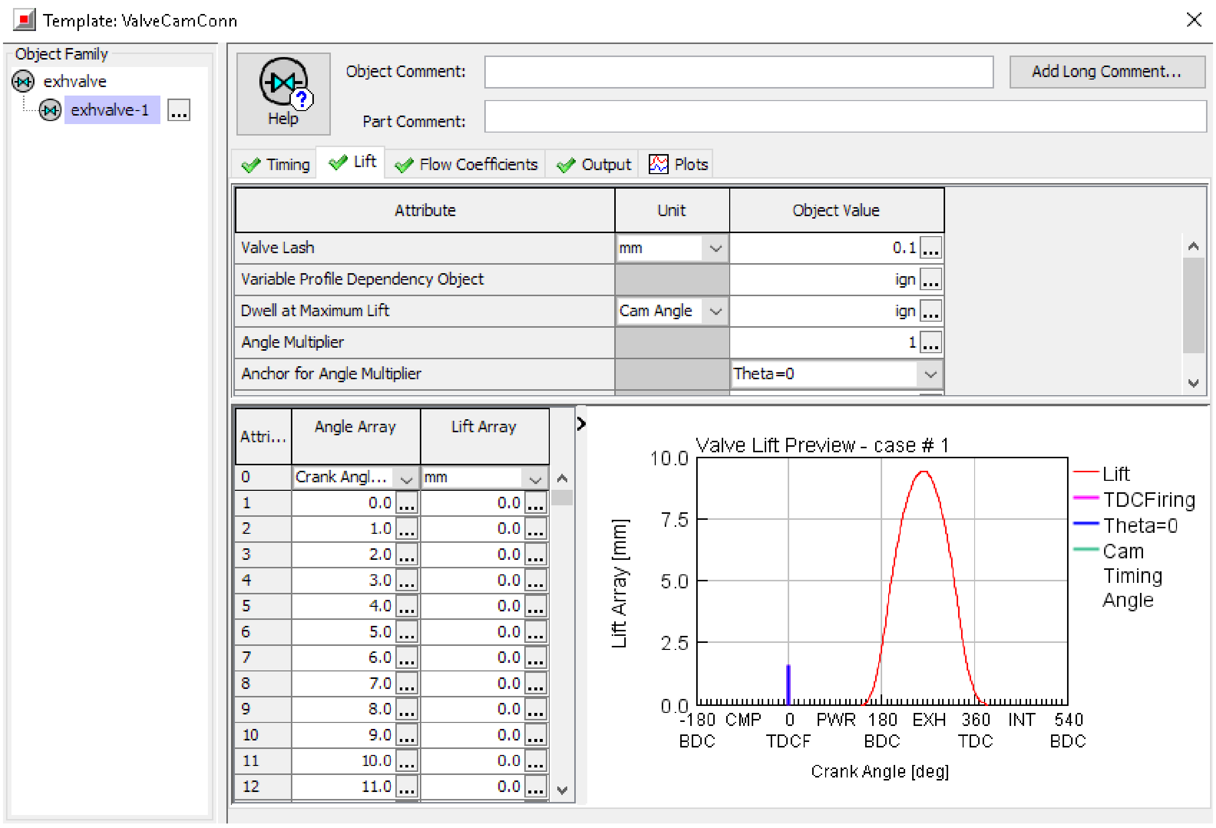
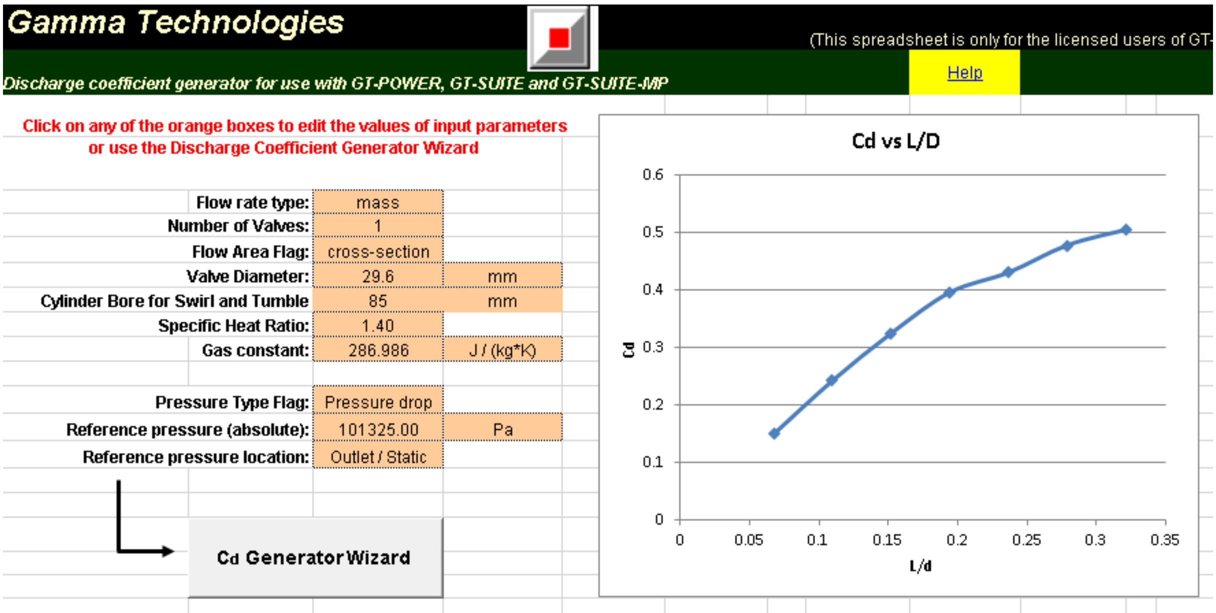


Figure 36. Exhaust Valve Lift Profile

We used GT Discharge Coefficient Generator to obtain exhaust flow coefficients. Required engine parameters were entered as shown in the Figure 35.



.GT Discharge Coefficient Generator Interface for Exhaust Valve

In Input Section, pressure drops, flow rates and upstream stagnation temperatures entered for seven different exhaust valve lifts. Flow rate values were obtained from ANSYS Fluent Port Flow Simulation. This simulation will be mentioned section …

Input Section					
Lift	Upstream Stagnation Temperature	Pressure Drop (See cells C14, C15 & C16)	Flow Rate (See cell C5)	Swirl Torque (Optional)	Tumble Torque (Optional)
mm	K	Pa	kg / s	Nm	Nm
2	300	5000	0.011245128		
3.25	300	5000	0.0181		
4.5	300	5000	0.0243		
5.75	300	5000	0.0295		
7	300	5000	0.0321		
8.25	300	5000	0.0356		
9.51	300	5000	0.0377		

***Figure 37** Input Section of GT Discharge Coefficient Generator For Exhaust Valve*

Intake valve discharge coefficients for 7 different lifts were obtained as shown in the Figure 37.

			Output Section		
Reference Flow Area	Density at Throat	Isentropic Velocity	Effective Area	L/D	Discharge Coefficient
mm^2	kg/m^3	m/s	mm^2		
688.133874	1.19319507	90.7589578	103.839706	0.0675676	0.150900443
688.133874	1.19319507	90.7589578	167.550094	0.1097973	0.243484735
688.133874	1.19319507	90.7589578	223.989892	0.152027	0.325503365
688.133874	1.19319507	90.7589578	272.800065	0.1942568	0.396434582
688.133874	1.19319507	90.7589578	296.550719	0.2364865	0.430949167
688.133874	1.19319507	90.7589578	329.059879	0.2787162	0.478191659
688.133874	1.19319507	90.7589578	347.726121	0.3212838	0.505317547

Figure 38. Exhaust Valve Discharge Coefficients

After this process, obtained discharge flow coefficients for seven different valve lifts were entered in ValveCamConn / Flow Coefficient template.

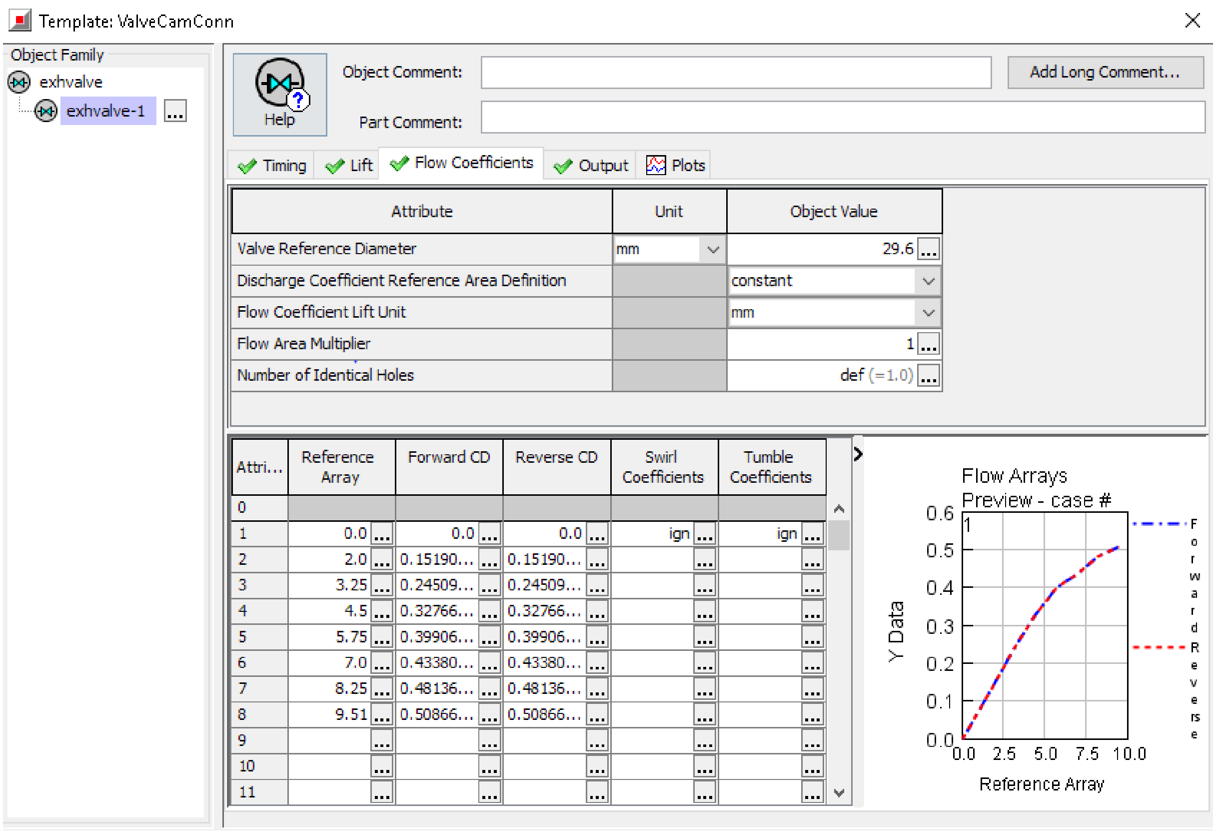


Figure 39. Exhaust Valve Discharge Coefficients for 7 Different Valve Lifts

6.1.9 Exhaust Port

The next step is to create an object that identifies the exhaust port of the engine. We selected Pipe Round object in the Project Library and selected Copy and Edit Object. We changed the object name such as “exhport-1 and filled in the values as in the following figures.

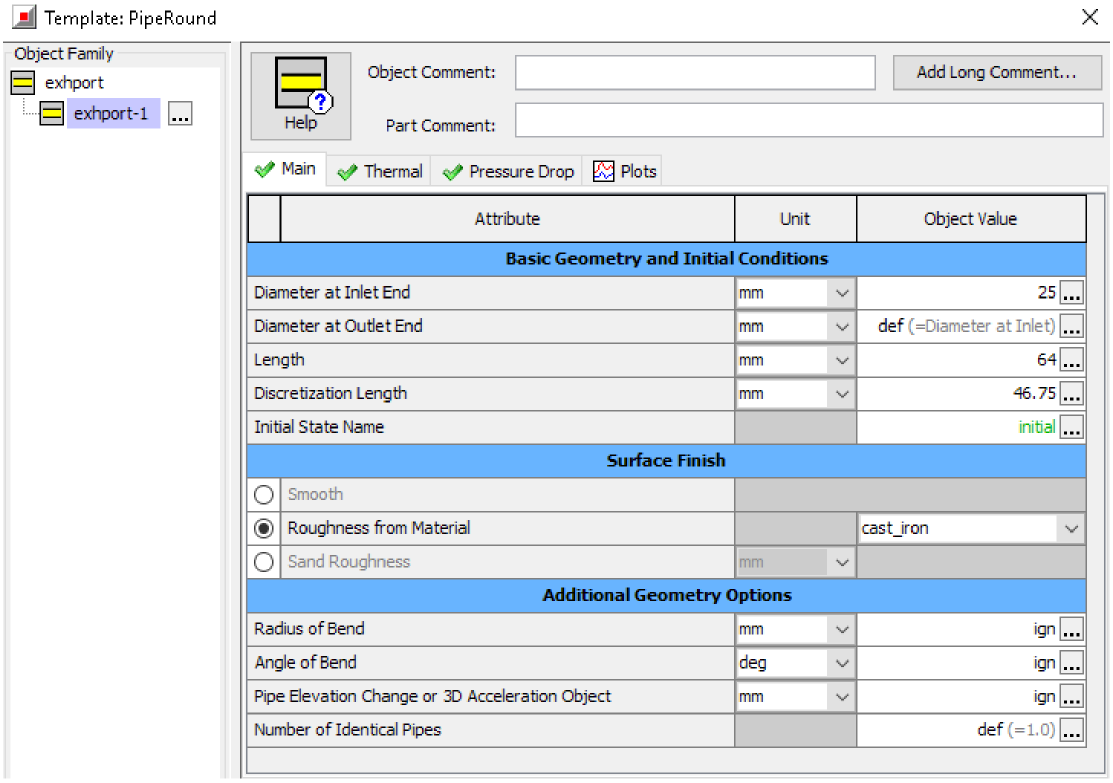


Figure 40. Exhaust Port Template

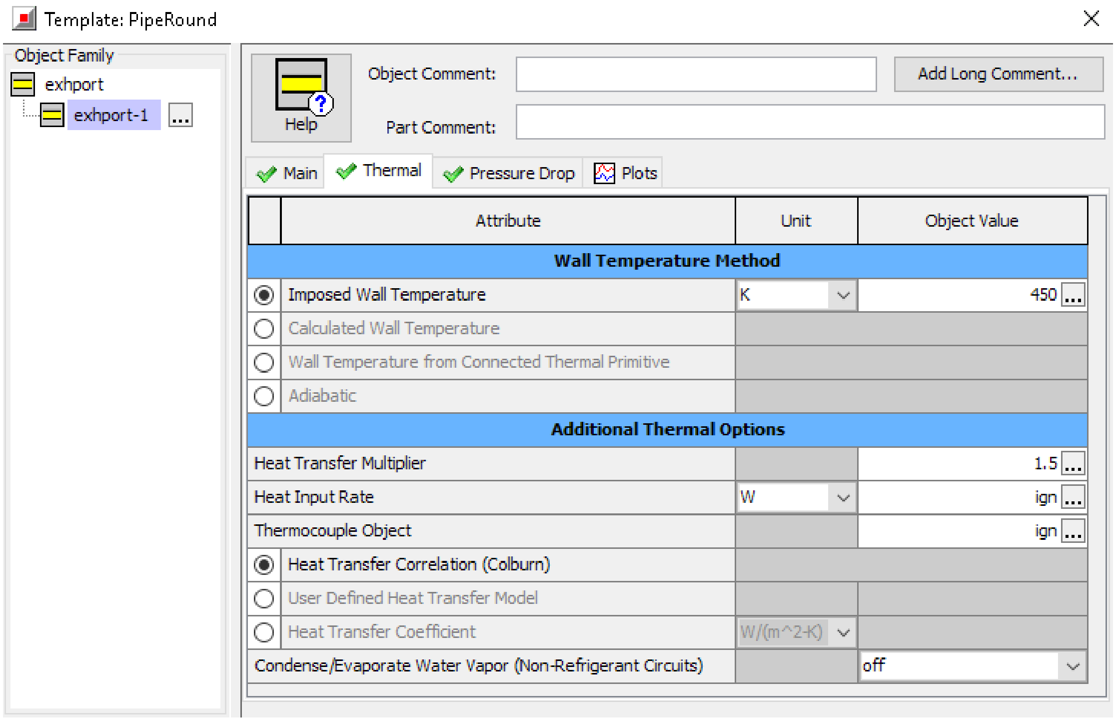


Figure 41. Thermal Conditions for Exhaust Port

6.1.10 Exhaust Runner

The next step is to build an exhaust runner to connect the ' EndEnvironment ' to the exhaust port. To generate a pipe object, we selected the ' PipeRound ' template within the project library. We named the ' exhrunner-1 ' item and filled in the attributes in the Main and Thermal folder characteristics with the following values.

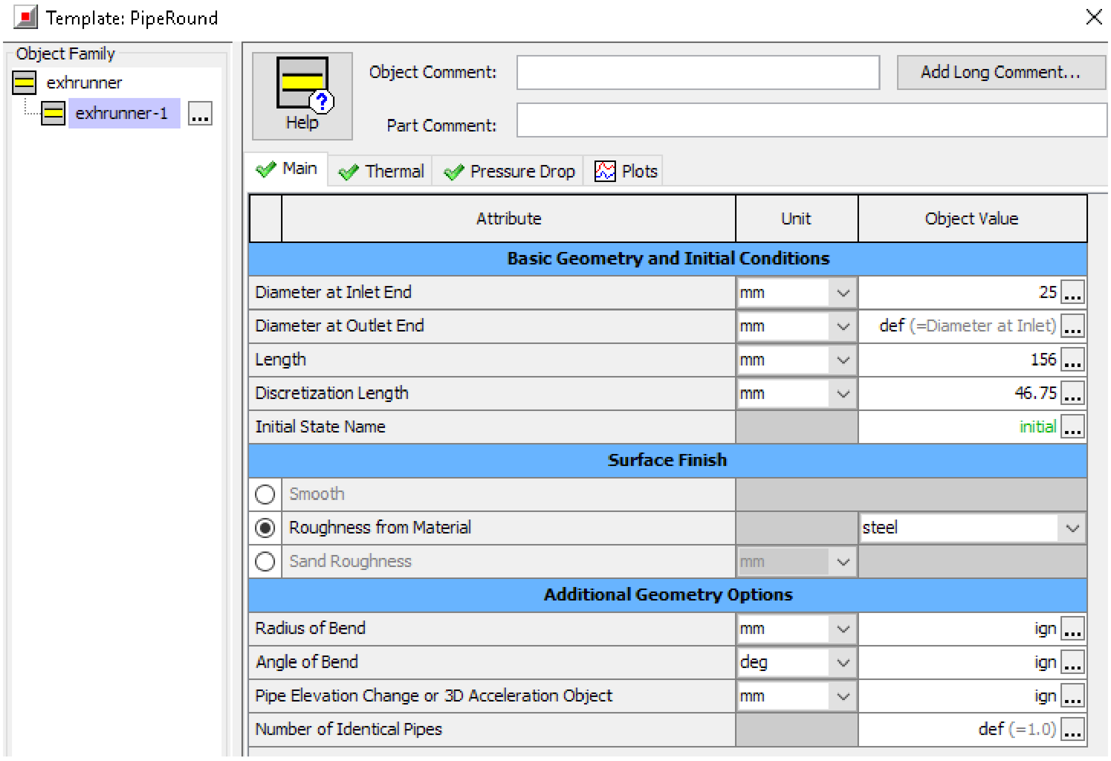


Figure 42. Exhaust Runner Template

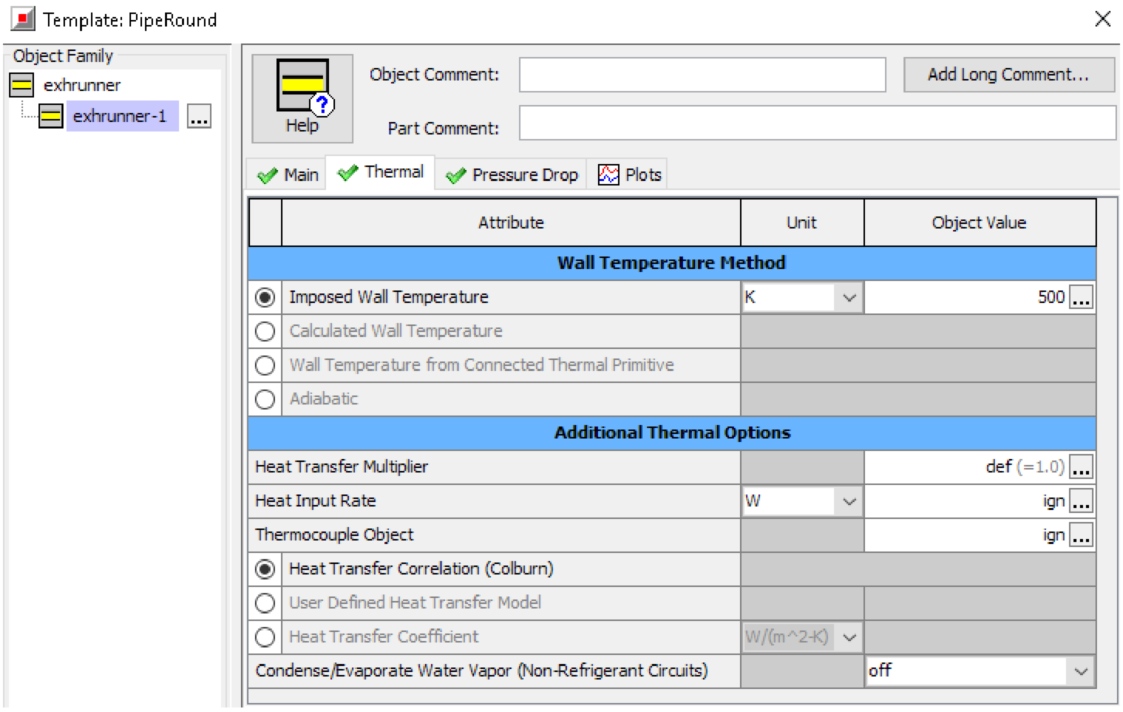


Figure 43. Thermal Conditions for Exhaust Runner

6.1.11 Outlet Environment

The last stage to create a single-cylinder engine model is to define the boundary conditions of outlet. To do this, a template ' EndEnvironment ' will be used in this model. We named the ' env-outlet-1 ' item and entered the following values in the attributes.

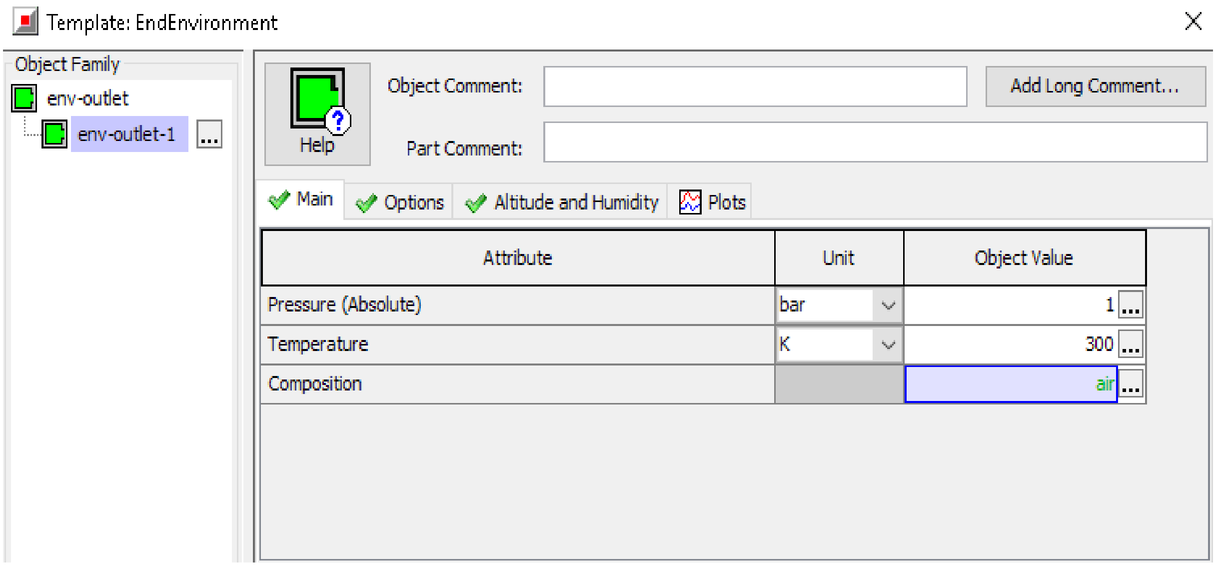


Figure 44. End Environment Template for Outlet

We selected composition for outlet as air under 'Fluid Mixture' in 'Value Selector'.

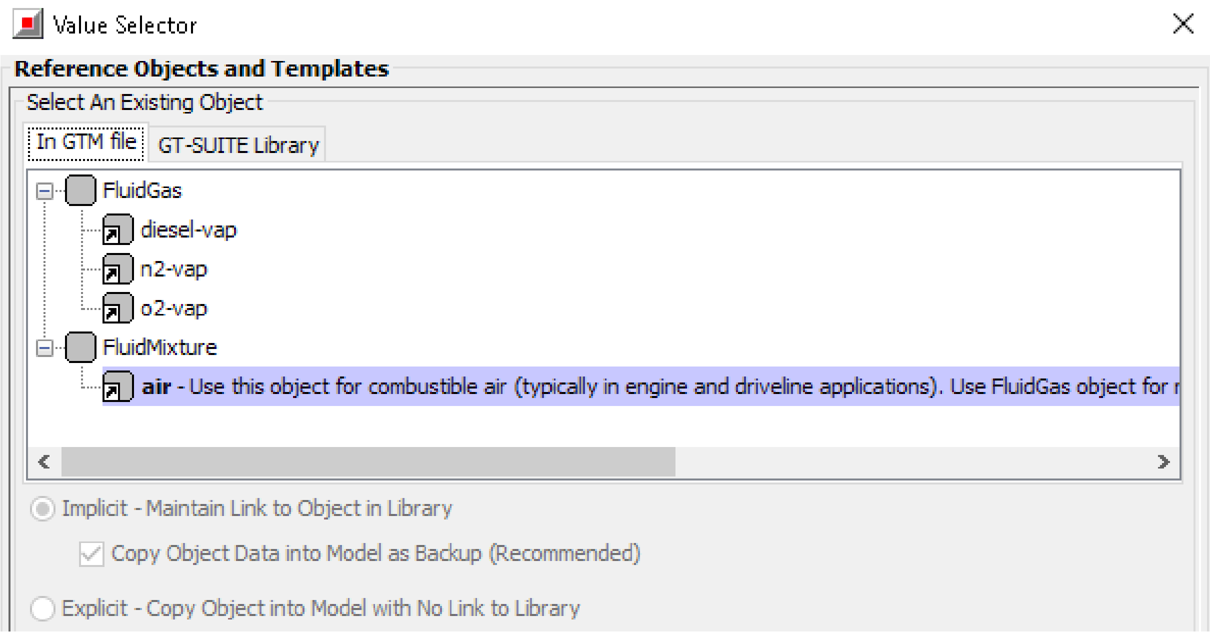


Figure 45. Composition Selection for Outlet EndEnvironment

6.2 GT-Power Engine System Combustion Setup

Antor 3LD510 Engine final setup are shown in the Figure 45 and Figure 46.

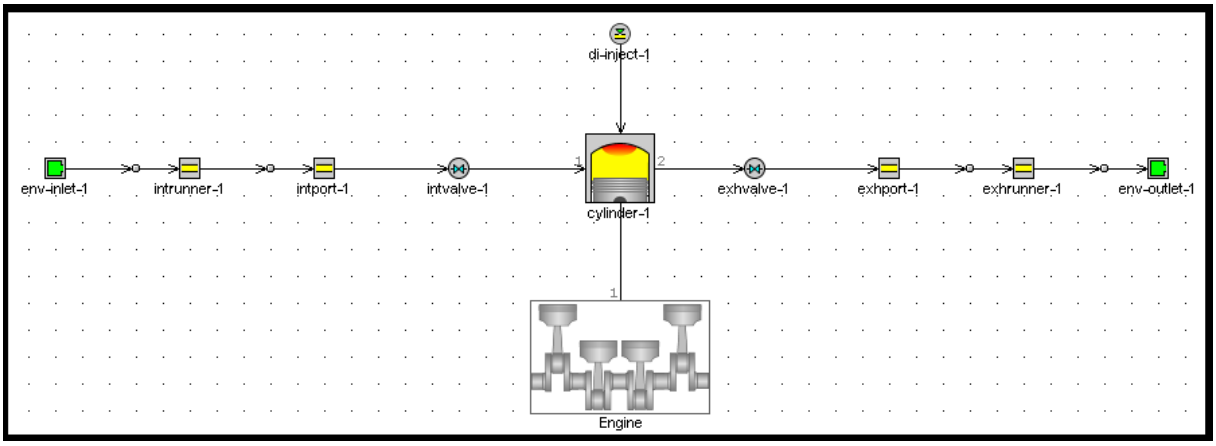


Figure 46. *Antor 3LD510 Final Setup*

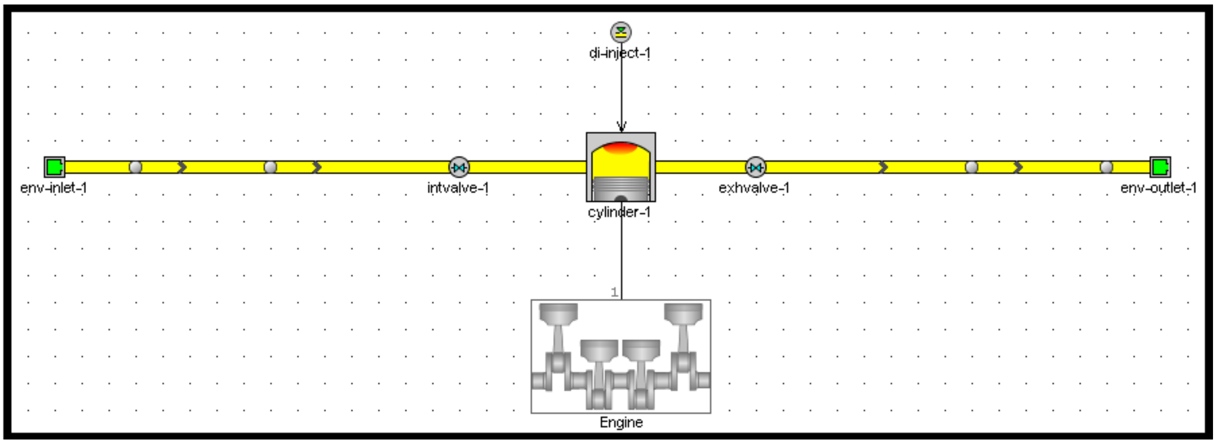


Figure 47. *Antor 3LD510 Final Setup (Pipe View)*

The system consists of the following:

1. One Environment Inlet
2. One Environment Exit
3. One Inlet Runner
4. One Exhaust Runner
5. One Inlet Port
6. One Exhaust Port
7. One Inlet Valve
8. One Exhaust Valve
9. One Cylinder
10. One Direct Injection Injector
11. One Crank-Shaft Engine System

6.2.1 Case Setup

The next step is to go to 'Home' tab (from ribbon toolbar) and select Case Setup. For our engine only one case will be run at constant engine speed of 3000 RPM. We ensured that the check box next to Case 1 is checked ON, indicating that Case 1 of the model will run when the simulation is run.

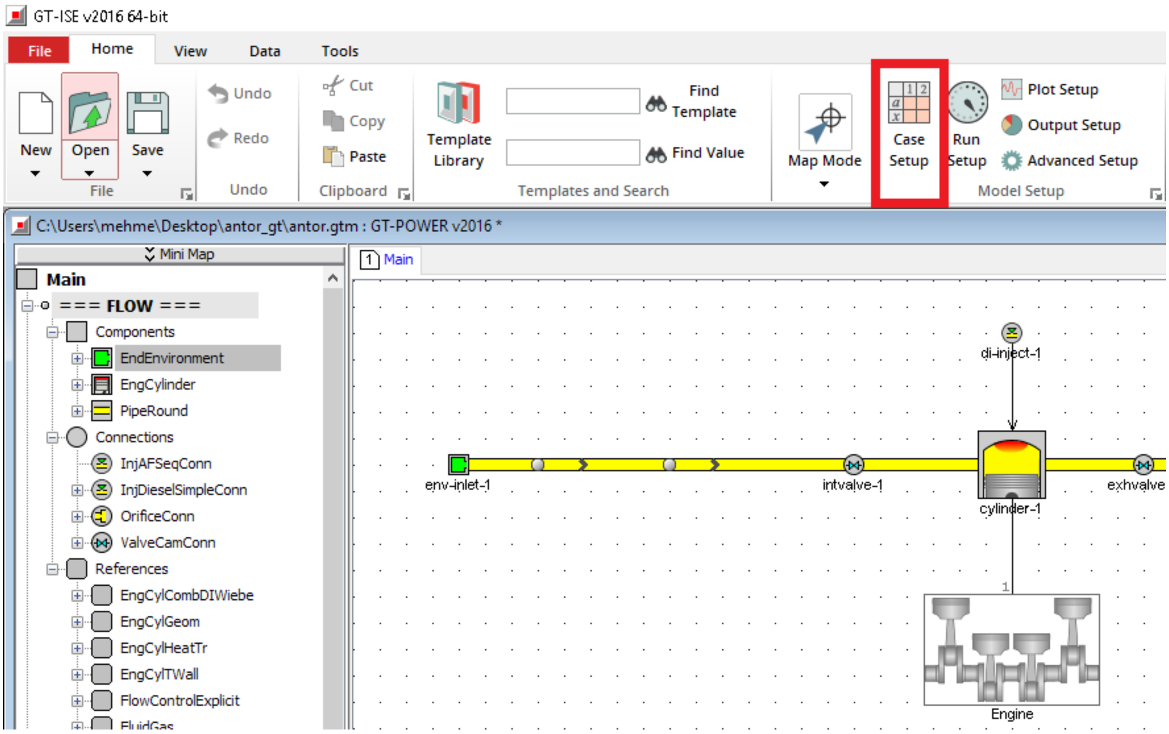


Figure 48. Case Setup Selection

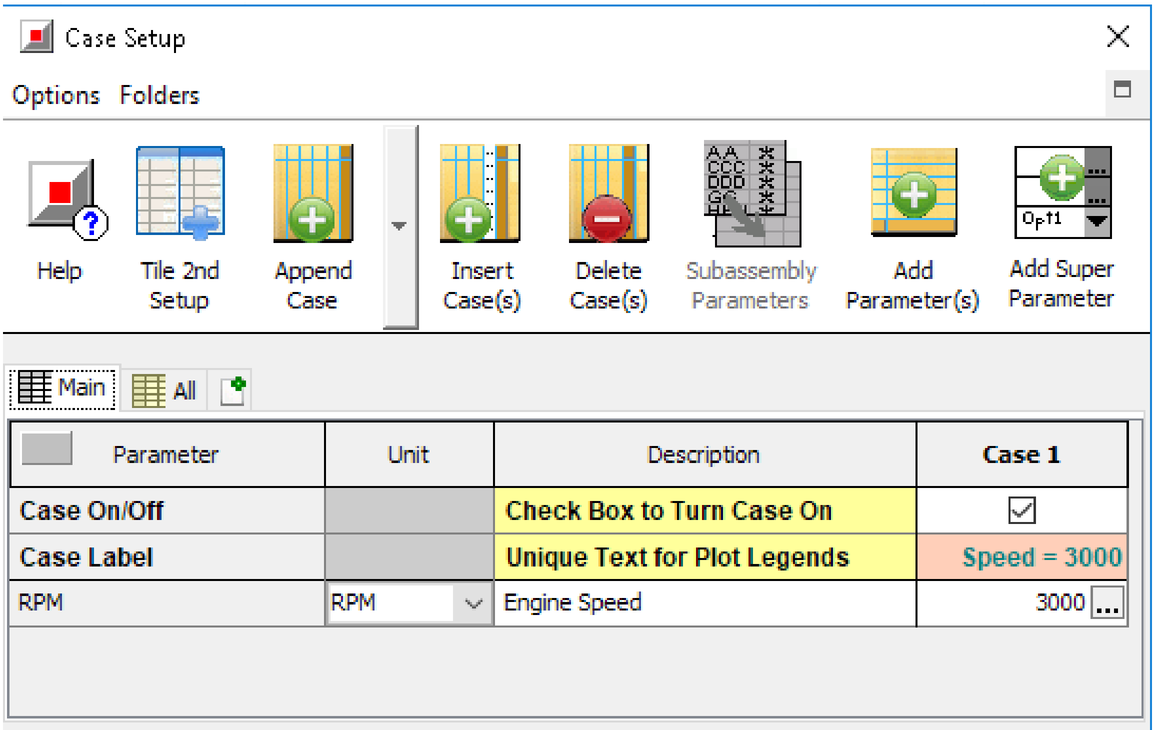


Figure 49. Case Setup Parameters

6.2.2 Run Simulation

We clicked on the Run Simulation button on the GTISE Toolbar to run it.

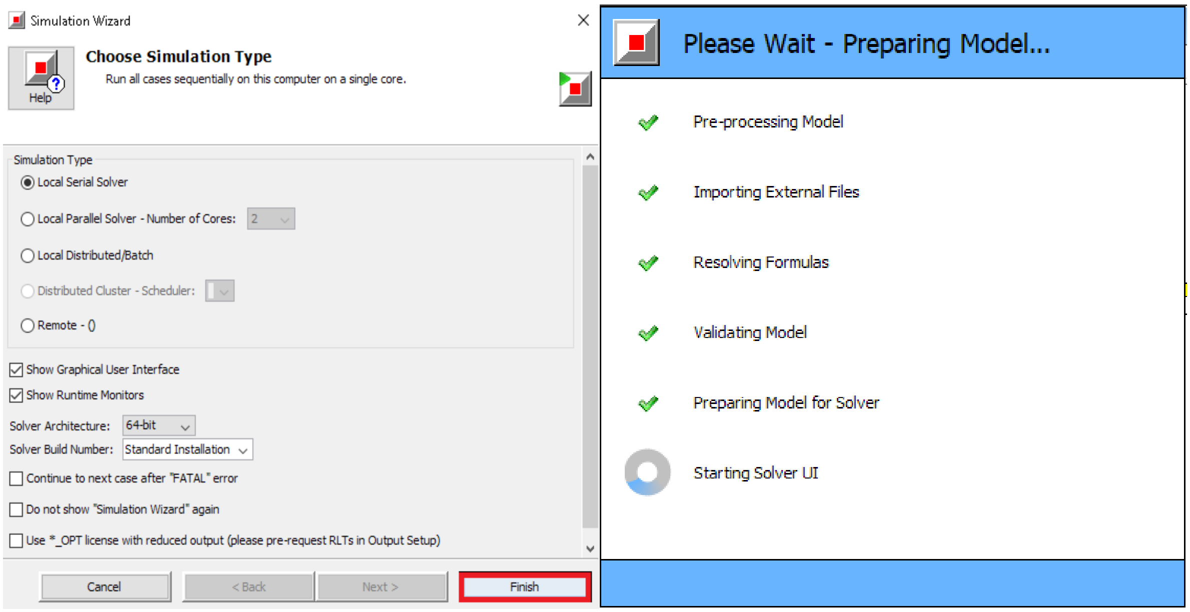


Figure 50. Simulation Wizard and Information Display

Calculation status are shown in the Figure 50.

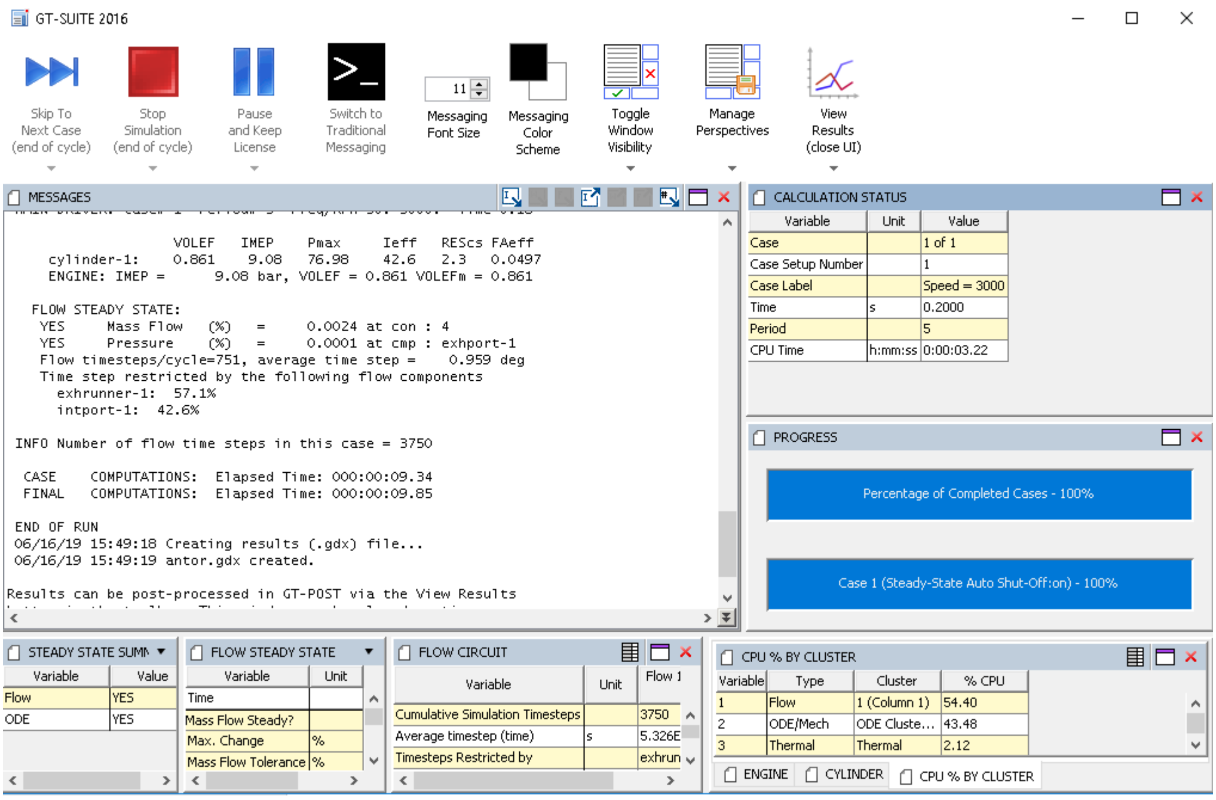


Figure 51. Calculation Process

6.3 GT-Power Engine System Combustion Optimization Setup

6.3.1 Case Setup

We opened the 'ValveCamConn' connection part named "intvalve-1" and replaced the value for the Cam Timing Angle with the parameter [intvalve] and selected OK to complete the change to the object.

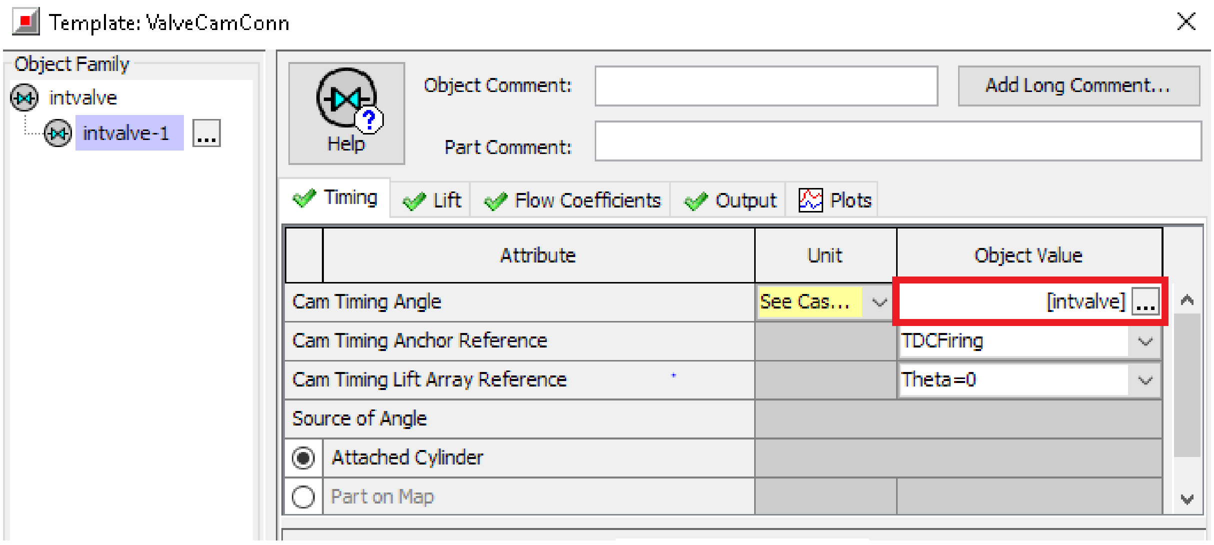


Figure 52. Parameterizing the Intake Cam Timing Angle

We opened the 'ValveCamConn' connection part named "exhvalve-1" and replaced the value for the Cam Timing Angle with the parameter [exhvalve] and selected OK to complete the change to the object.

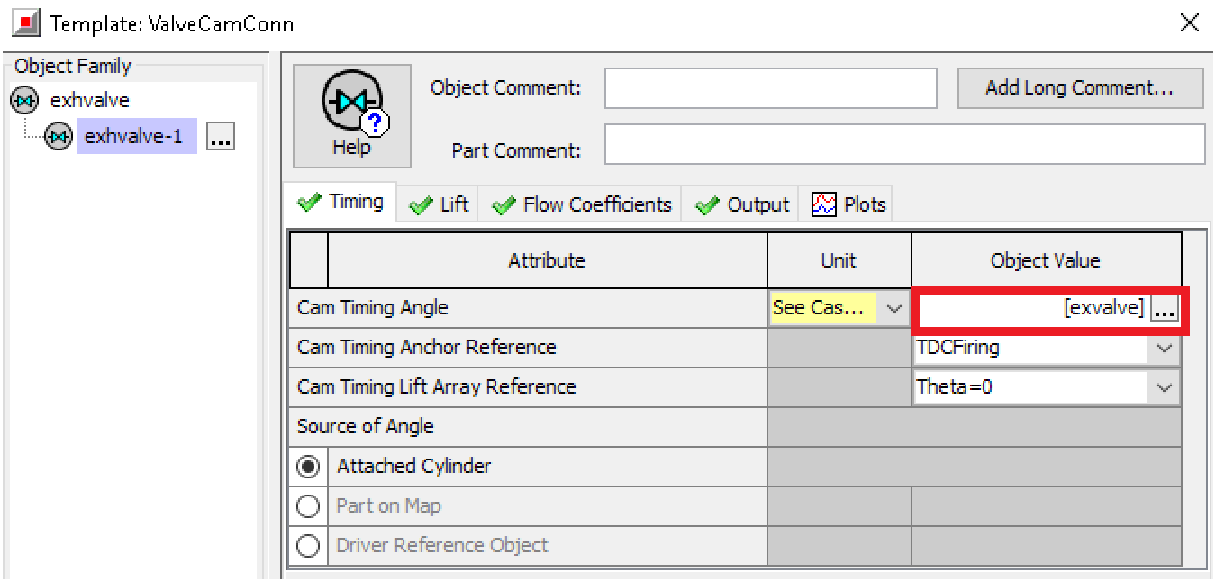


Figure 53. Parameterizing the Exhaust Cam Timing Angle

We opened the 'InjDieselSimpleConn' connection part named "di-inject-1" and replaced the value for the Injection Timing with the parameter [injectiontiming] and replaced the value for the Injection Duration with the parameter [injectionduration].

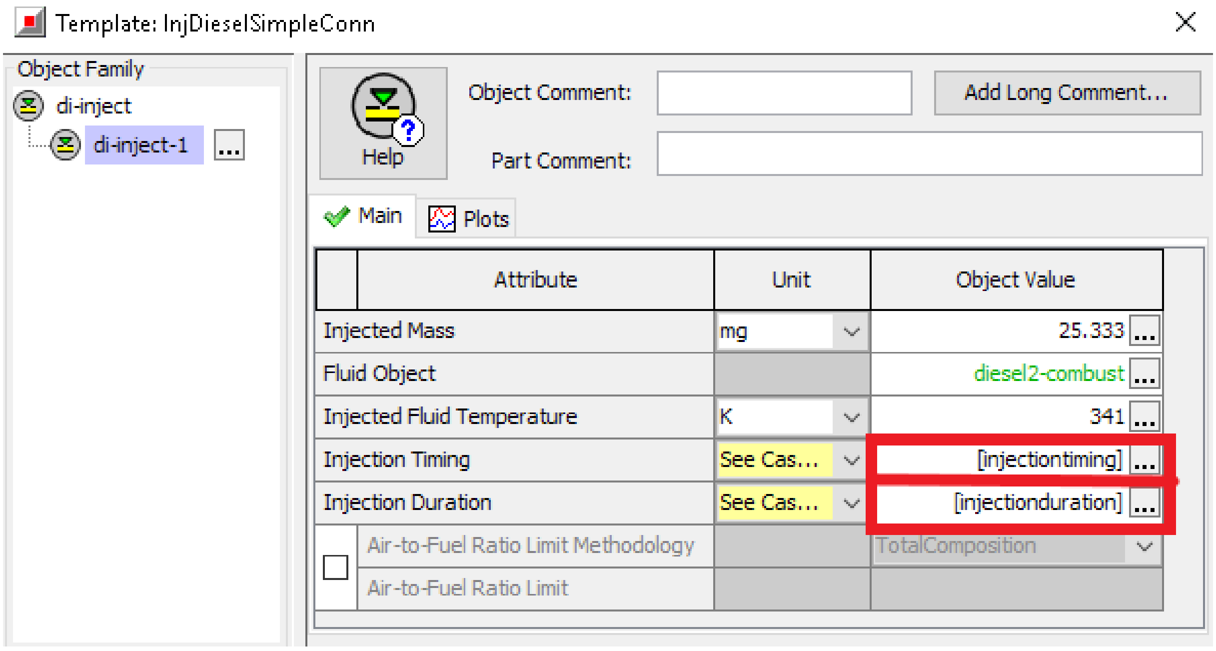


Figure 54.Parameterizing Injection Timing and Injection Duration

We selected Optimization -> Direct Optimizer, enable the Optimizer by clicking on the check-box and enter the values in the following two figures in the Optimization and Opt-Arrays folders, respectively.

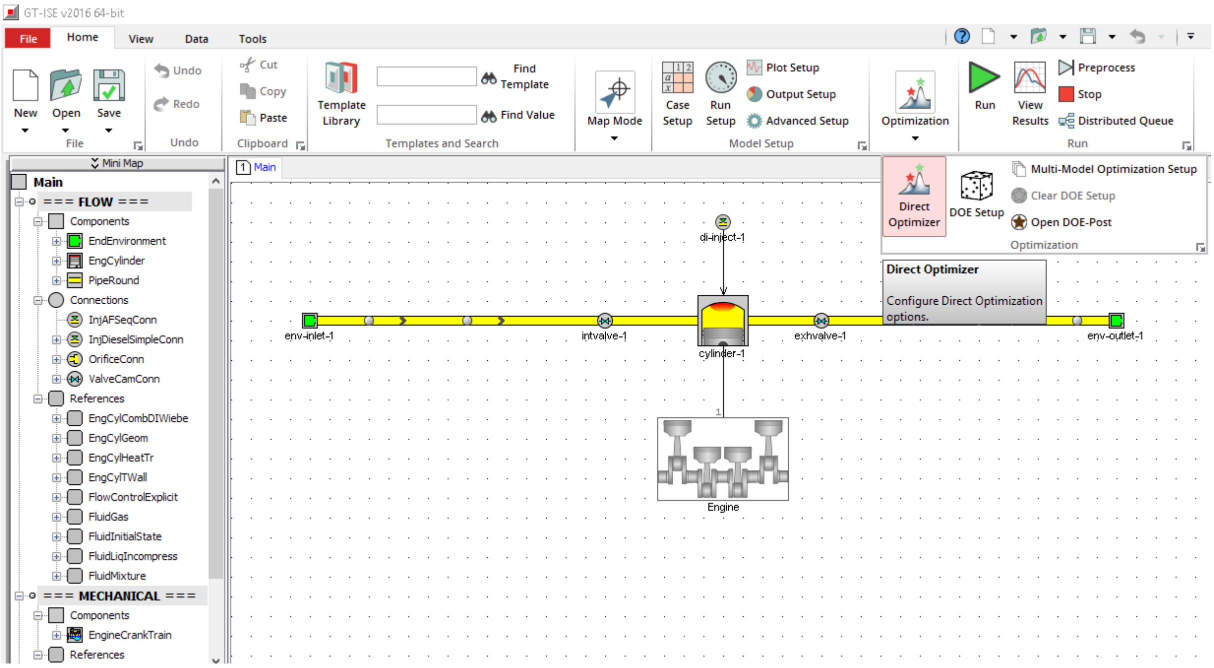


Figure 55. Starting GT- Direct Optimizer

For the optimization setup, we selected optimization objective as ‘Maximize’, Search Method Algorithm as ‘Brent-method’ and dependent variable as ‘btq:Engine’. Aim of this optimization is maximize the maximum brake torque.

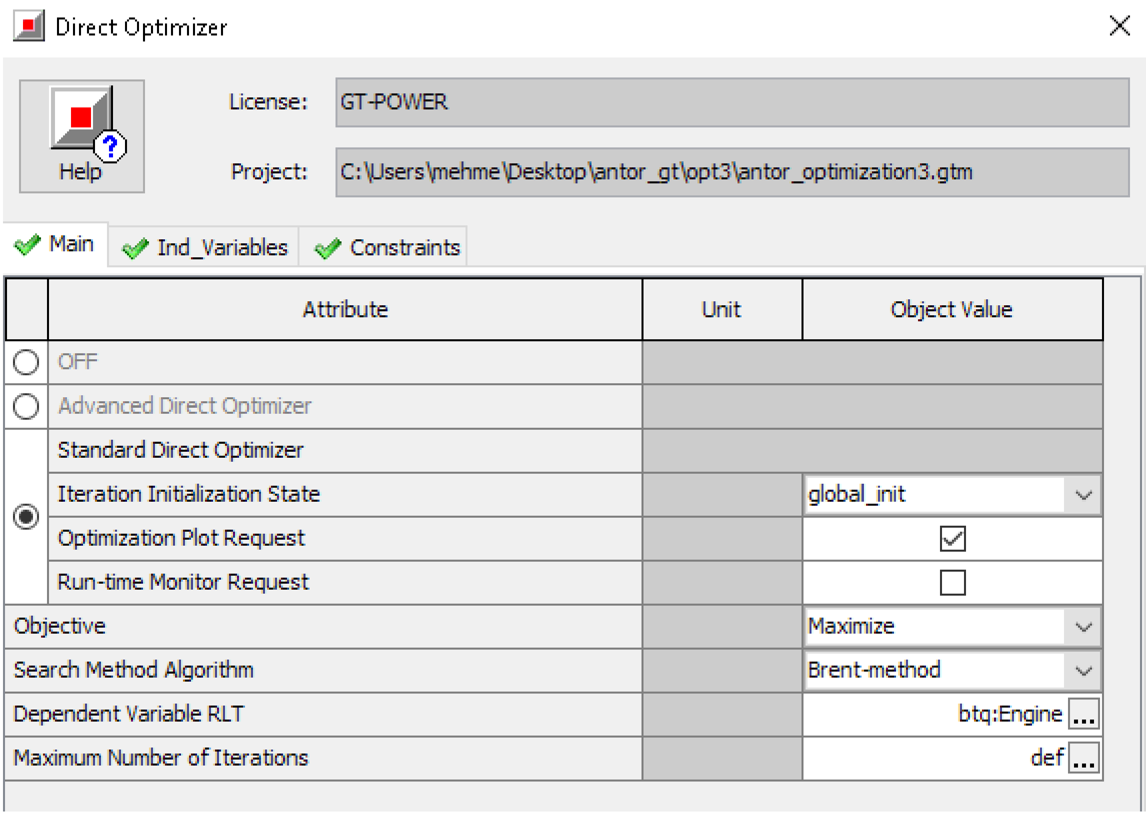


Figure 56. Direct Optimizer Menu

After that, we selected indirect variables that are intvalve, exvalve, injectiontiming and injection duration. Parameter ranges and resolutions are also shown in Figure 56.

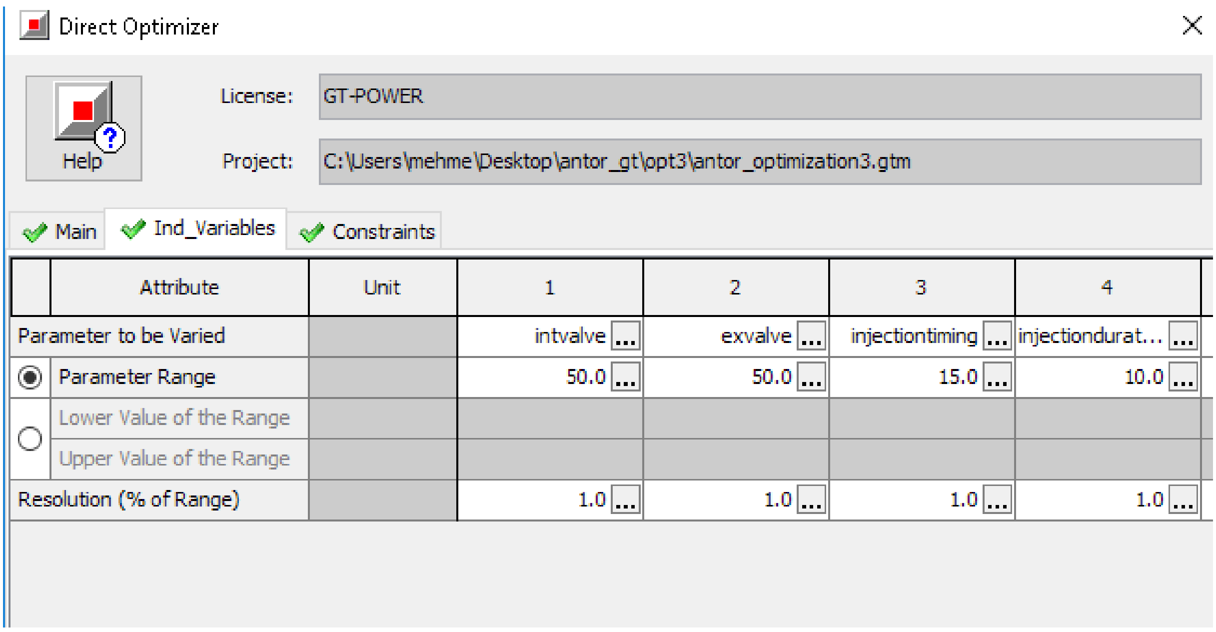


Figure 57. Indirect Variables

We entered initial values of the parameters for case setup.

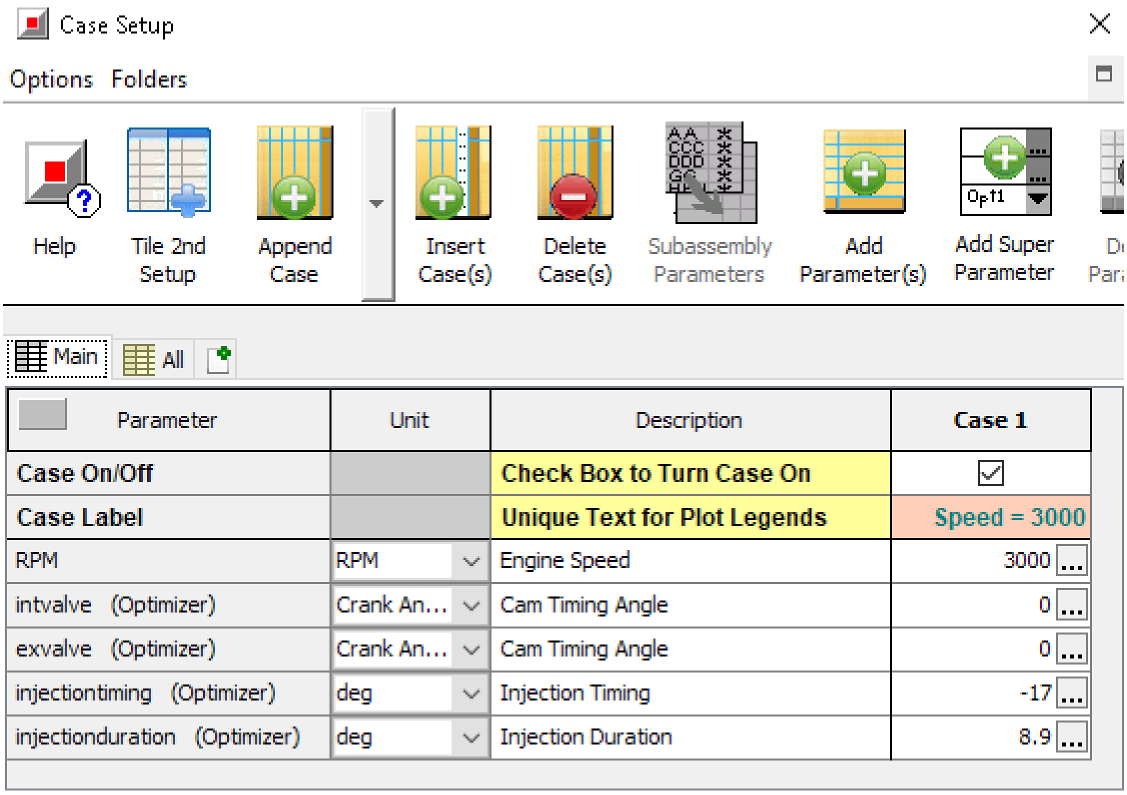


Figure 58*. Optimization Case Setup*

6.3.2 Run Simulation

We clicked on the Run Simulation button on the GTISE Toolbar to run it.

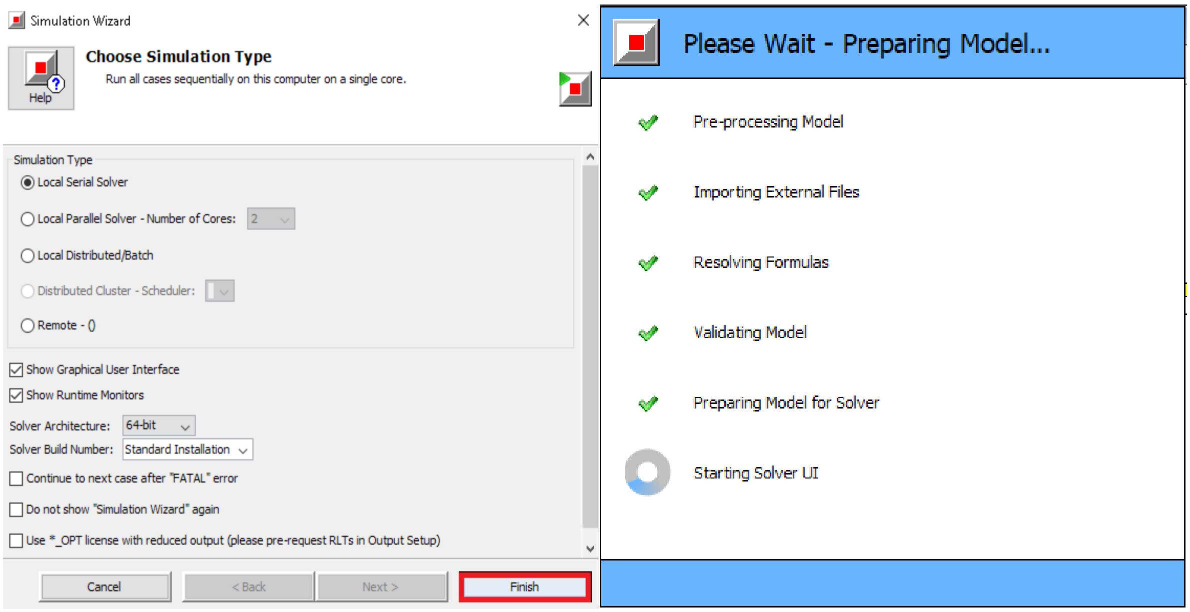


Figure 59*. Simulation Wizard and Information Display*

Calculation status are shown in the Figure 59.

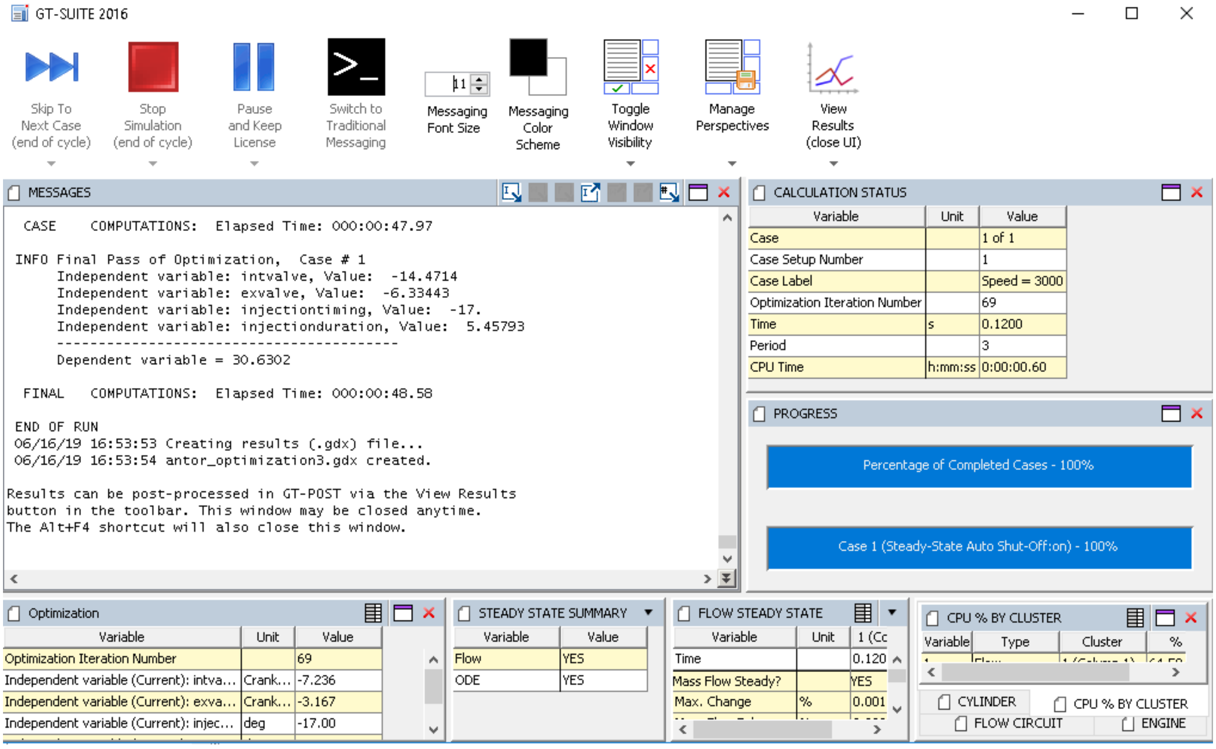


Figure 60. Calculation of Optimization Process

7 - ANSYS WORKBENCH INTAKE PORT FLOW SIMULATION

For the intake port flow simulation, we used IC Engine Fluent module of the Workbench.

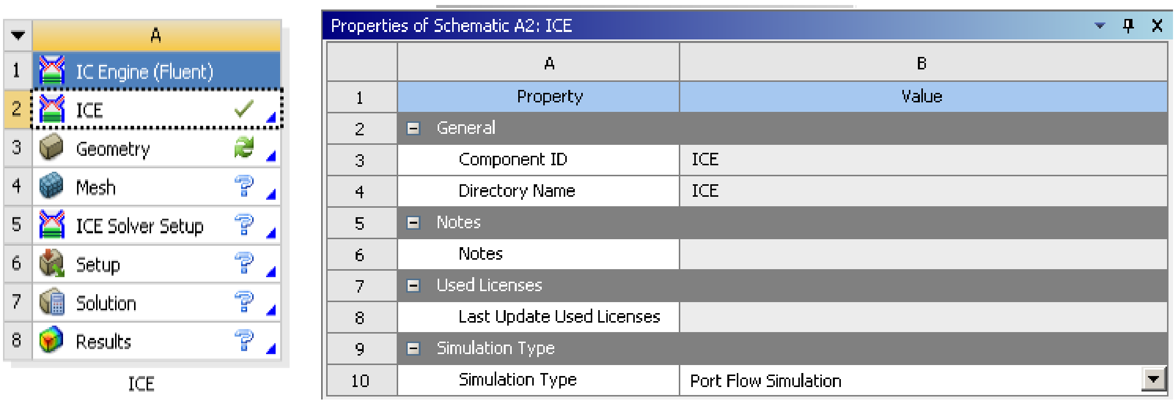


Figure 61. IC Engine Fluent Module in Workbench

7.1 Preparing the Geometry

We set the unit as mm, depending upon the geometry units, in ICE-DesignModeler. We loaded the geometry file and clicked ‘Generate’ to complete the import.

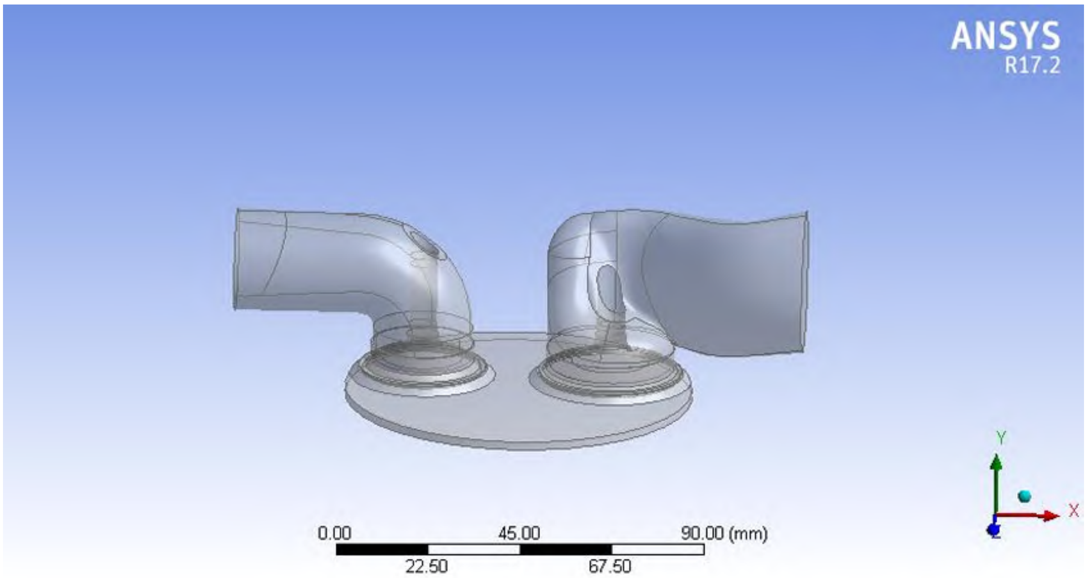


Figure 62. Imported Geometry File

7.1.2 Intake and Exhaust Valve Creation

Intake and exhaust valves were created with Pre Manager.

Details View

Details of ICPreManager1	
Name	ICPreManager1
Pre Manager Operation	Create Valve
Valve Hole Edges	2 Edges
Valve Seats	2 Faces
Valve Stem Extrusion Lenght	0.012486 mm

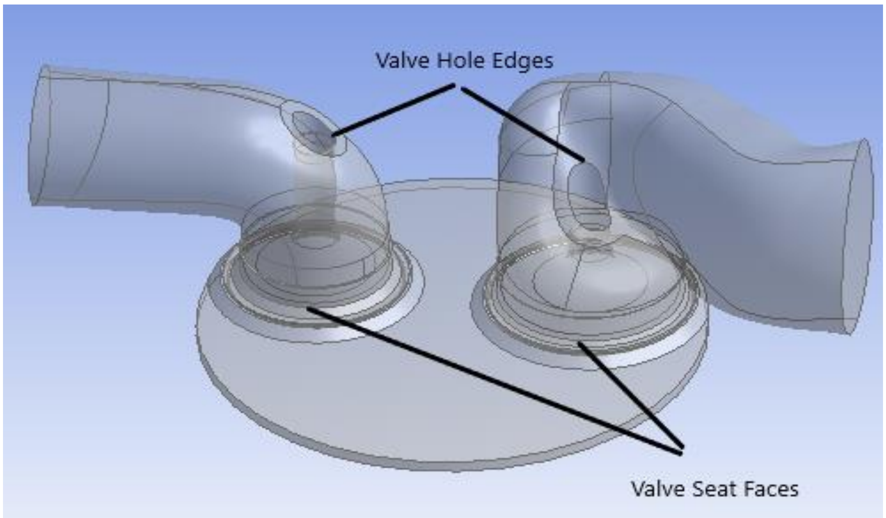


Figure 63. Valve Hole Edges and Valve Seat Selections

Created intake and exhaust valves are shown in the Figure 51.

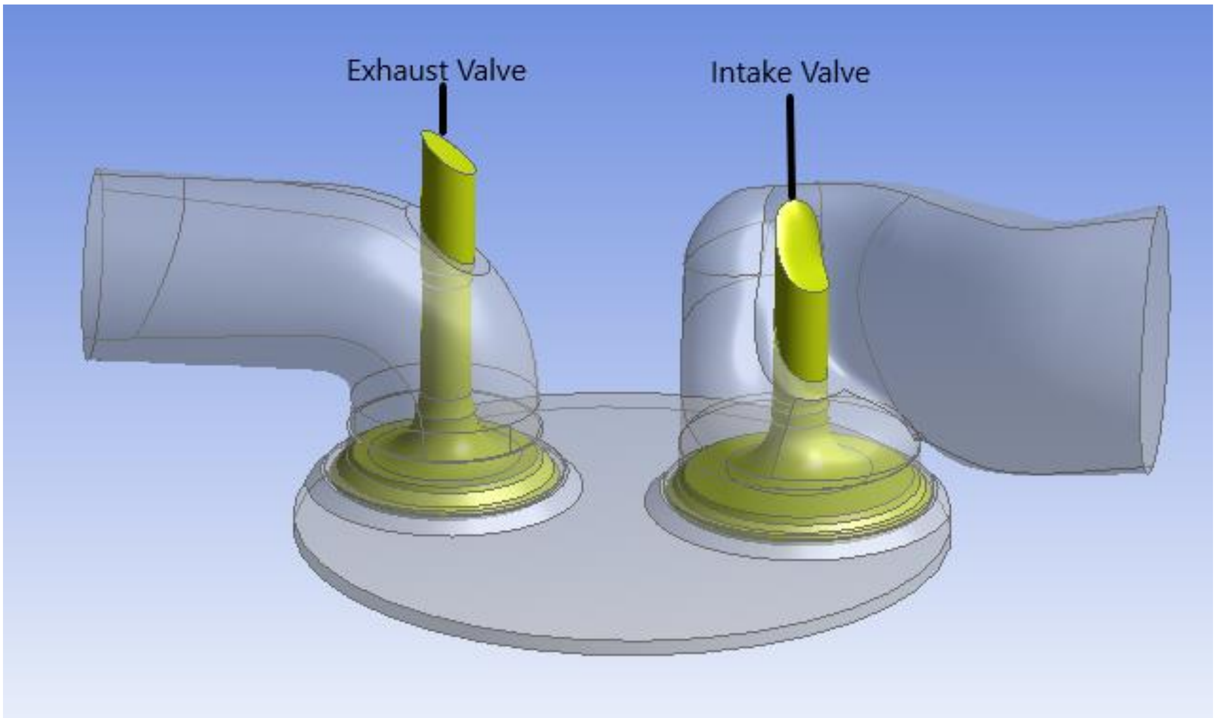


Figure 64. Intake Valve and Exhaust Valve

7.1.3 Input Manager

Input Manager dialog box takes inputs of the engine to set it up for decomposition.

Details View	
Details of InputManager1	
Slice	InputManager1
Cylinder Liner Faces	1 Face
Symmetry Face Option	No
Post Planes Dist. From Ref.	15.0; 30.0 (mm)
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
D FD1, Valve Lift	2 mm
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
FD2, Valve Lift	0 mm (Deactivate Port)
IC Inlet Plenum 1 (RMB)	
Inlet/(Plenum Inlet) Faces	1 Face
Plenum Type	Hemisphere
Inlet Extension Length	39.8 mm
Plenum Size	140 mm
Plenum Blend Rad	25 mm
IC Outlet Plenum (RMB)	
Outlet Plenum Option	Yes
Cylinder Extension Length	90 mm
Plenum Type	Cylinder
Plenum Size	120 mm

Figure 65. Details of Input Manager

Cylinder liner face selected. Two post planes were created. These planes are required for creating swirl monitors in Fluent. Intake valve and exhaust valve were selected. Intake valve lift value entered as 2 mm. Exhaust valve lift value entered as 0 mm because we performed intake port flow simulation. For the inlet plenum, inlet face, plenum type (Hemisphere), inlet extension length (39.8 mm), plenum size (140mm) and plenum blend radius (25 mm) were selected. Cylinder extension length entered as 90 mm because engine has 90 mm stroke. Plenum type (cylinder) and plenum size (120 mm) were entered.

Selected regions in Input Manager are shown in the Figure 53.

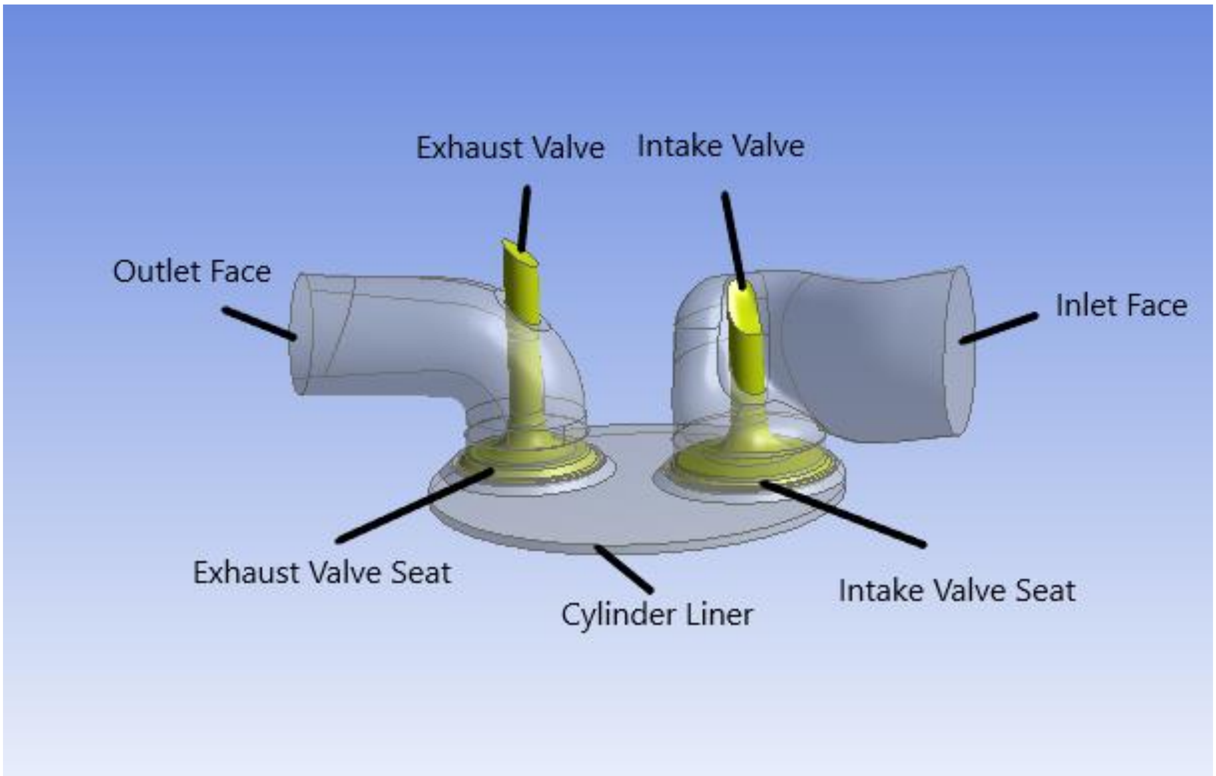


Figure 66. *Input Manager Selected Regions*

7.1.4 Decomposition

We performed decomposition (located in the IC Engine toolbar). After decomposition, the engine is divided into one part, Port and several bodies.

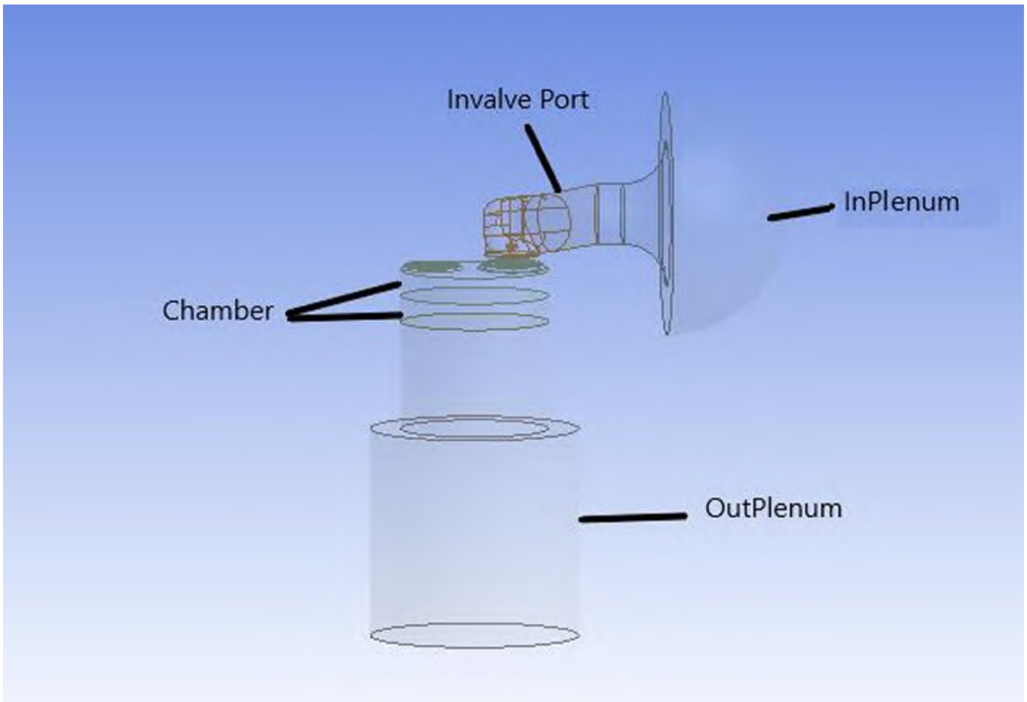


Figure 67. *Several Bodies after Decomposition*

Outplenum: The plenum created at the outlet.

Inplenum: The plenum created at the inlet.

Chamber: The chamber is divided into parts depending upon the number of postprocessing planes.

Invalve-Port: Depending upon the number of inlet valves in the geometry, Invalve-Port is created.

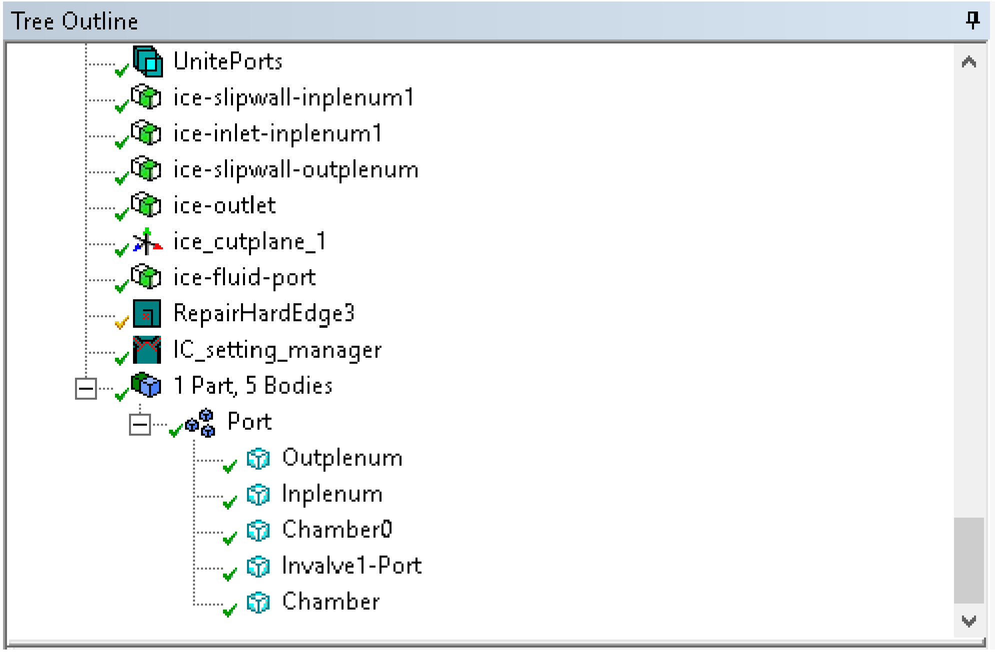
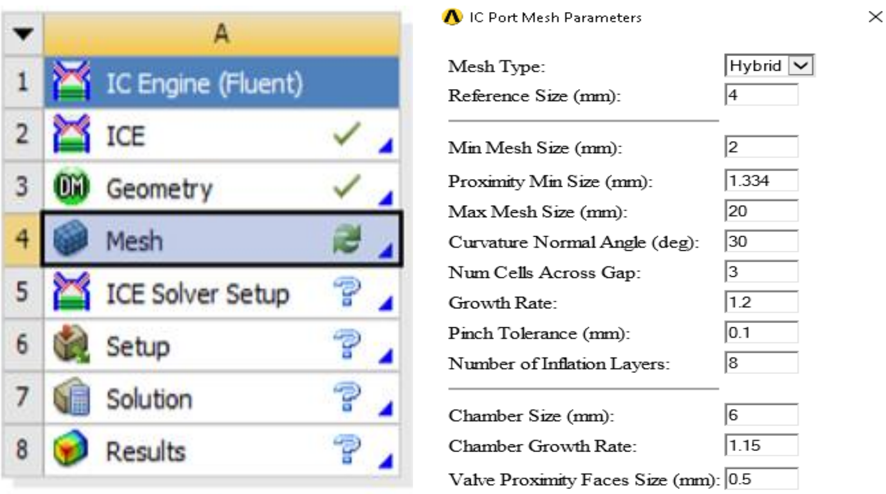


Figure 68. Tree Outline of the Decomposed Geometry

7.2 Meshing Procedure with ANSYS Meshing Application for Port Flow Simulation

We use the Mesh cell in the IC Engine analysis system to open the ANSYS Meshing application.



We started IC Setup Mesh (located in the IC Engine toolbar).This opens the IC Mesh Parameters dialog box. We selected mesh type as Hybrid and reference size as 4mm. We used default settings for other parameters.

Figure 69. Mesh Properties for Intake Port Flow Simulation

After selecting mesh parameters, we generated mesh with ‘Generate’ button.

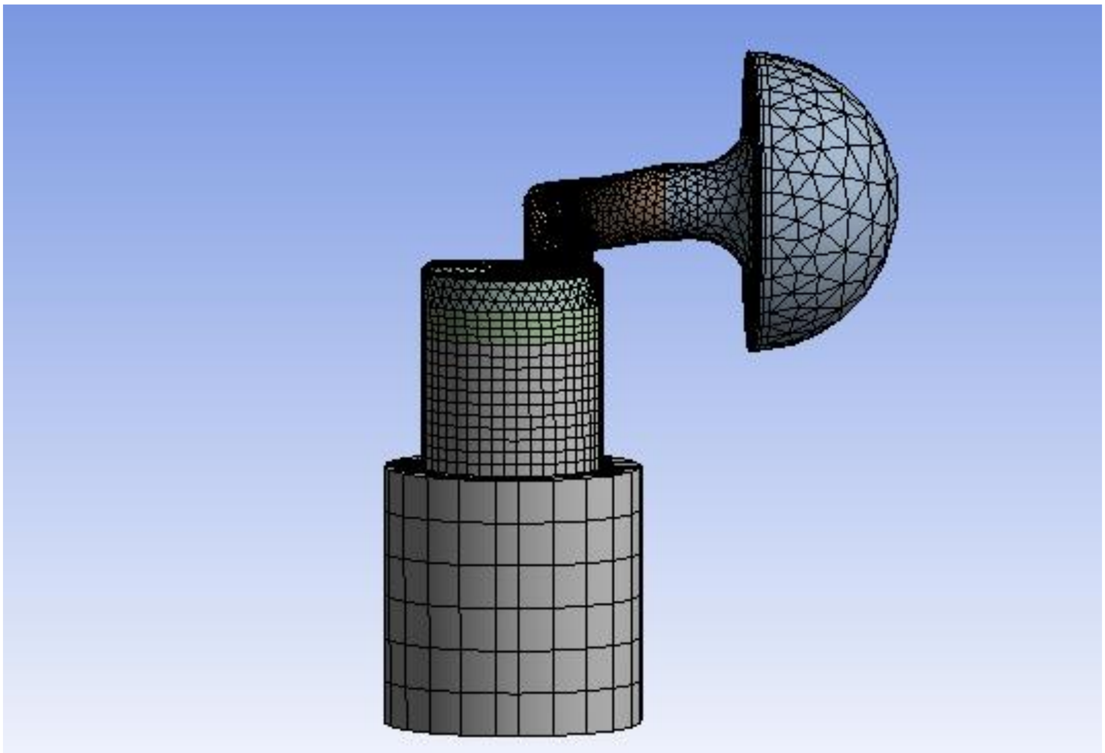


Figure 70. Meshing for Intake Port Flow Simulation

7.3 Setting Up the Analysis in IC Engine (Fluent)

In the Basic Settings tab we have the following settings:

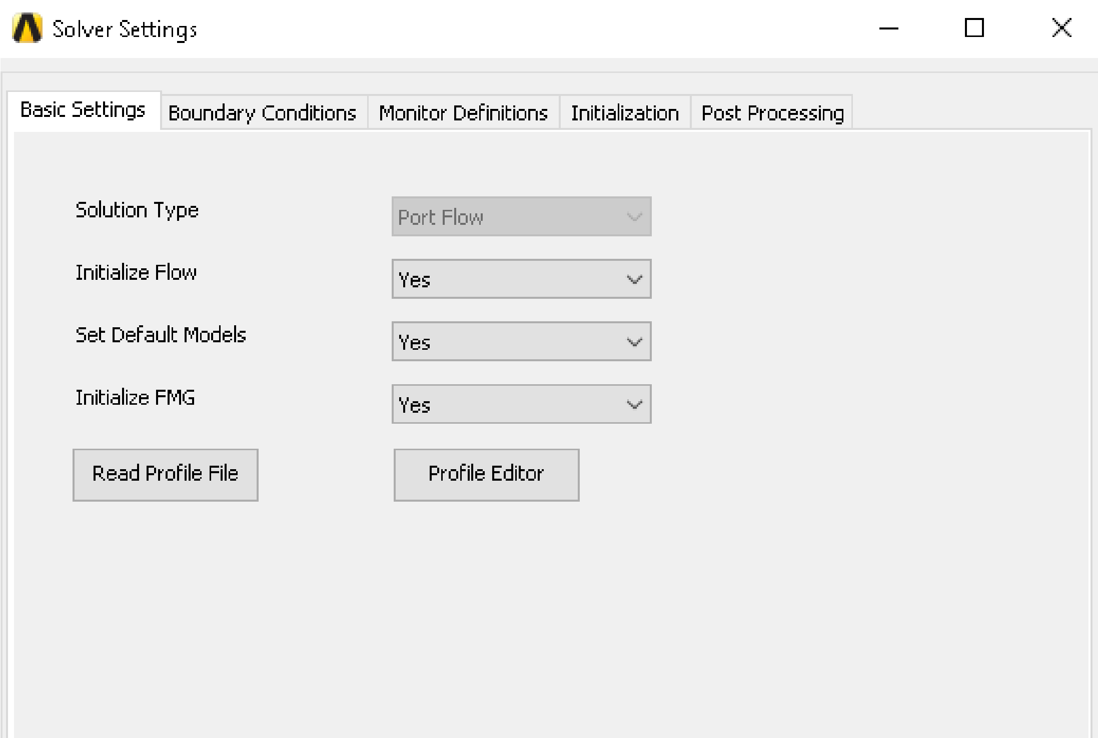


Figure 71. Basic Settings for Intake Port Flow

In the Boundary Conditions tab, it can be seen boundary conditions set for four zones.

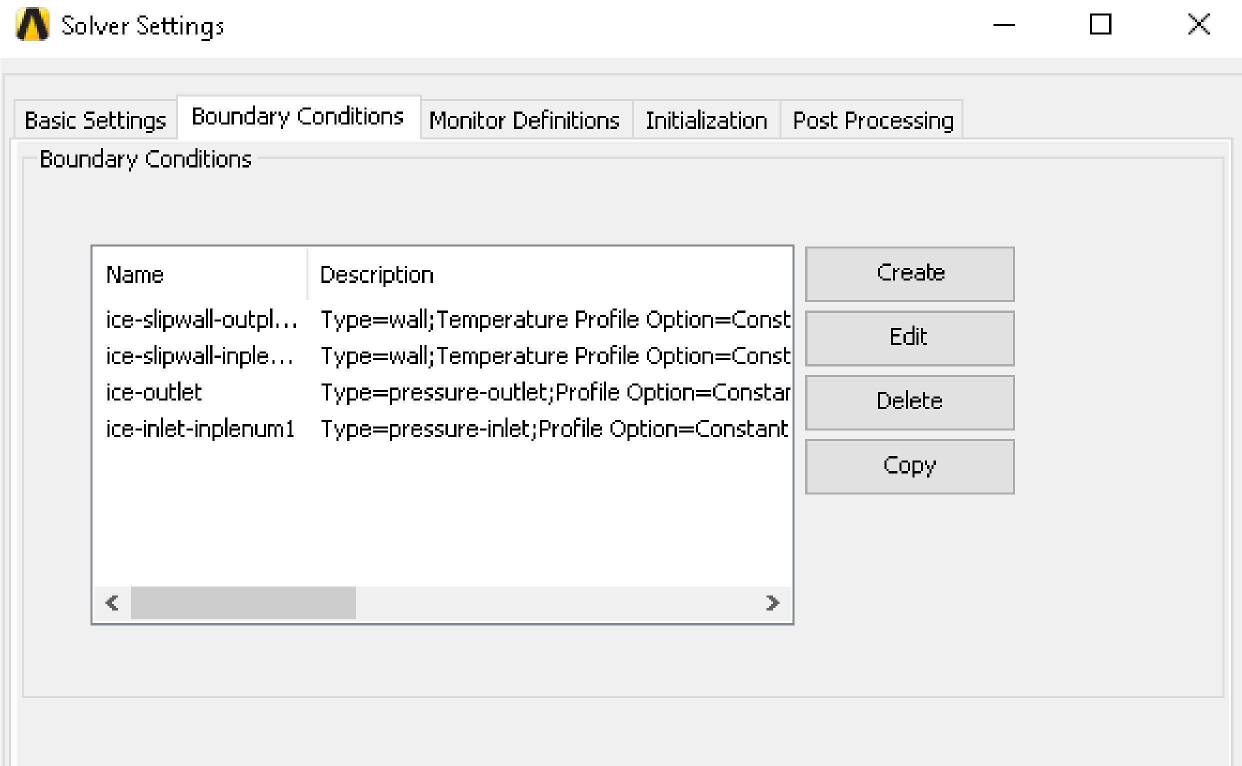


Figure 72. Boundary Conditions Tab for Intake Port Flow

The wall is set as slipwall and the Temperature is set to 300.

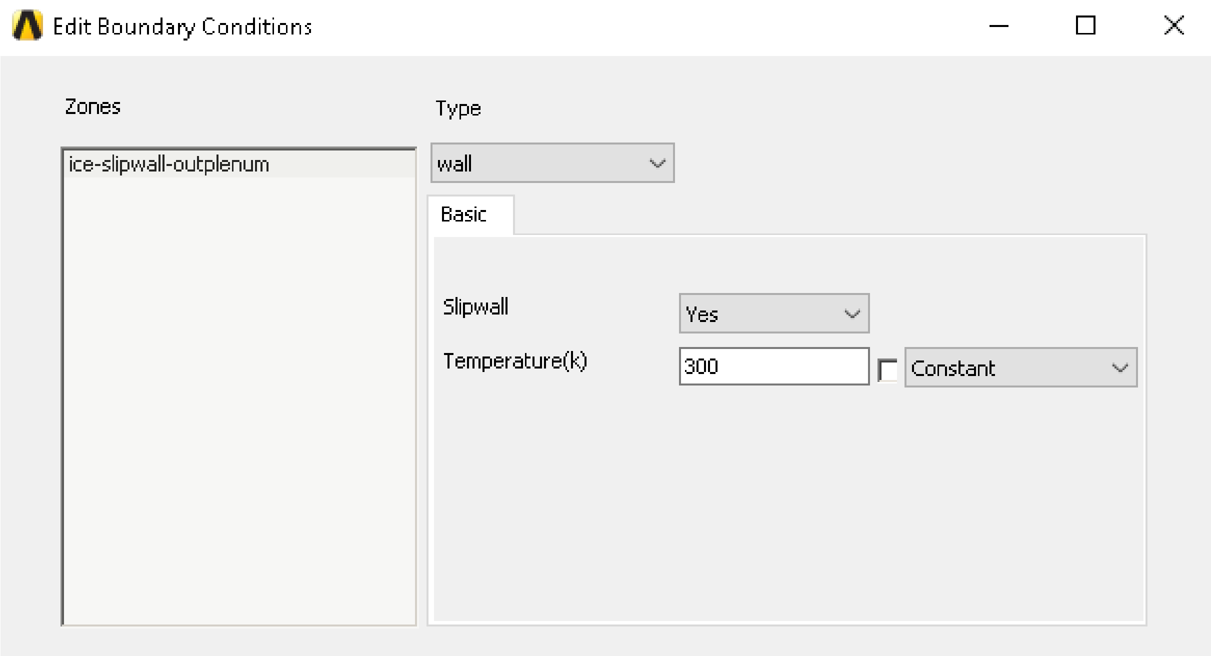


Figure 73. Boundary Conditions for slipwall-outplenum

The wall is set as slipwall and the Temperature is set to 300.

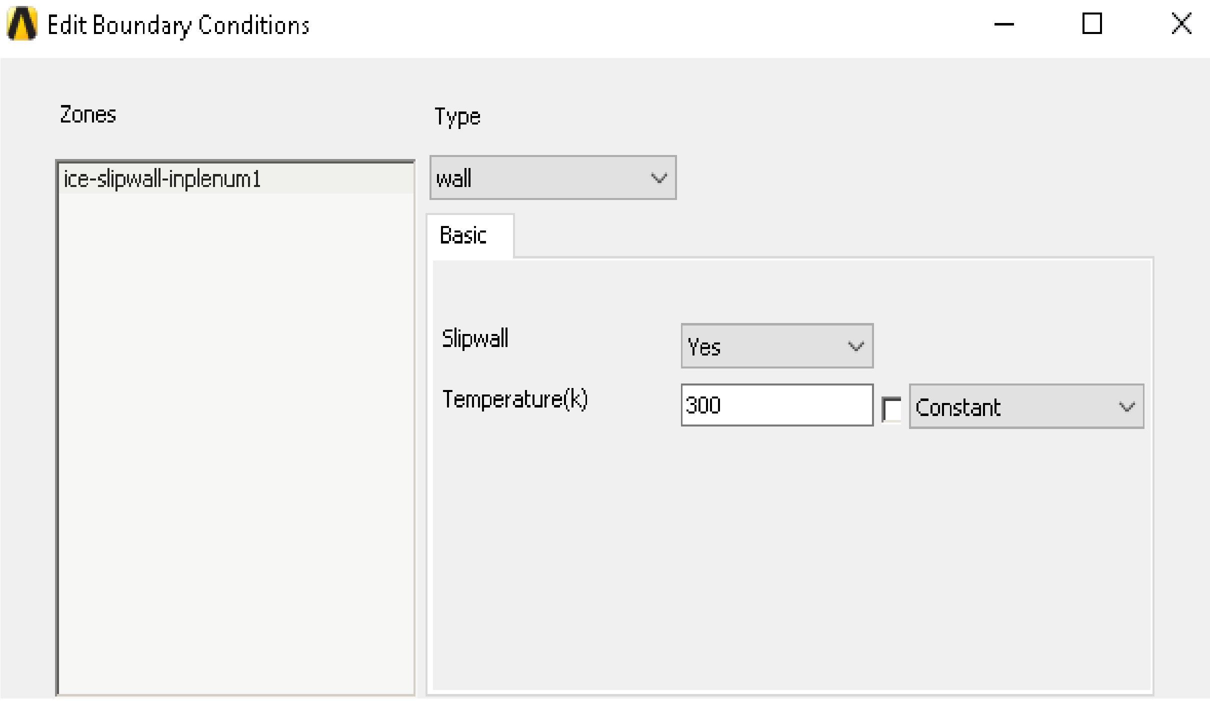


Figure 74. Boundary Conditions for slipwall-inplenum

Pressure-inlet type is selected, gage pressure is set to 0 pascal and temperature is set to 300 K for inlet-inplenum.

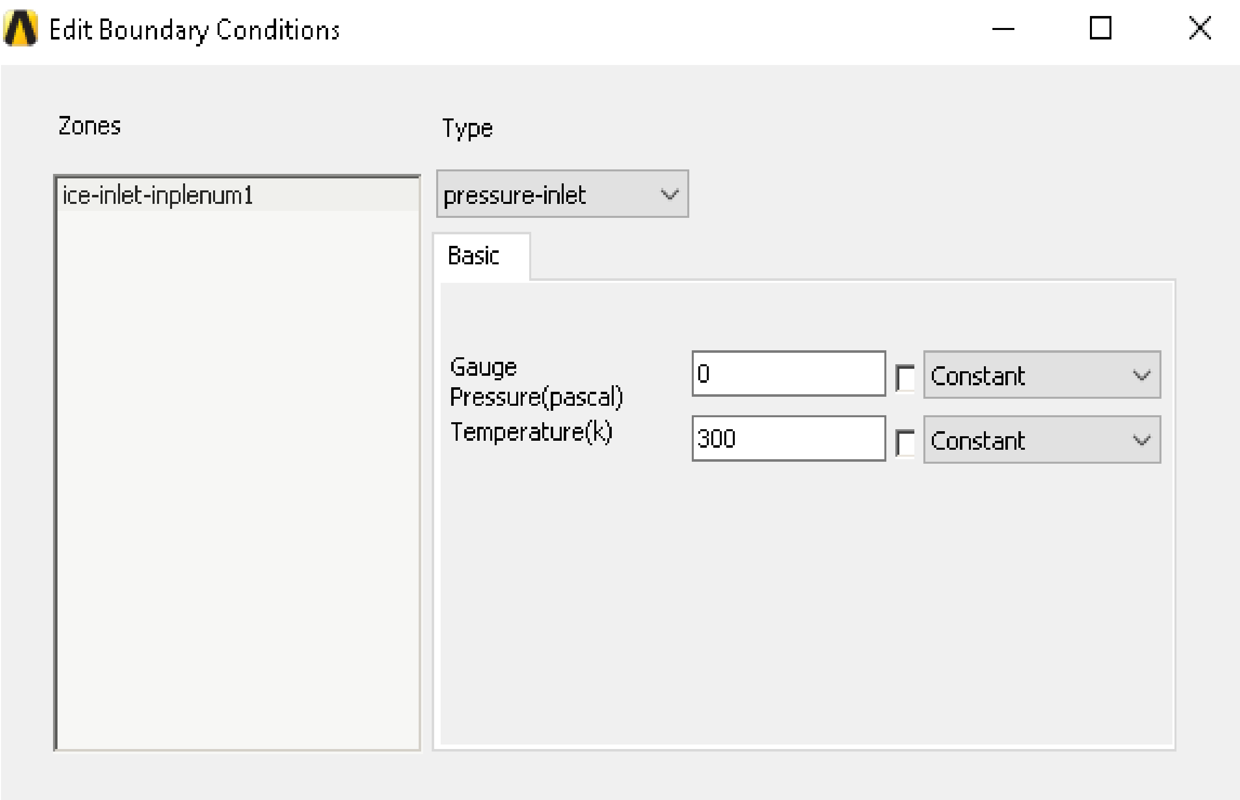


Figure 75.Boundary Conditions for inlet-inplenum

For the outlet zone, pressure-outlet type is selected, gauge pressure is set to -5000 pascal and temperature is set to300 K.

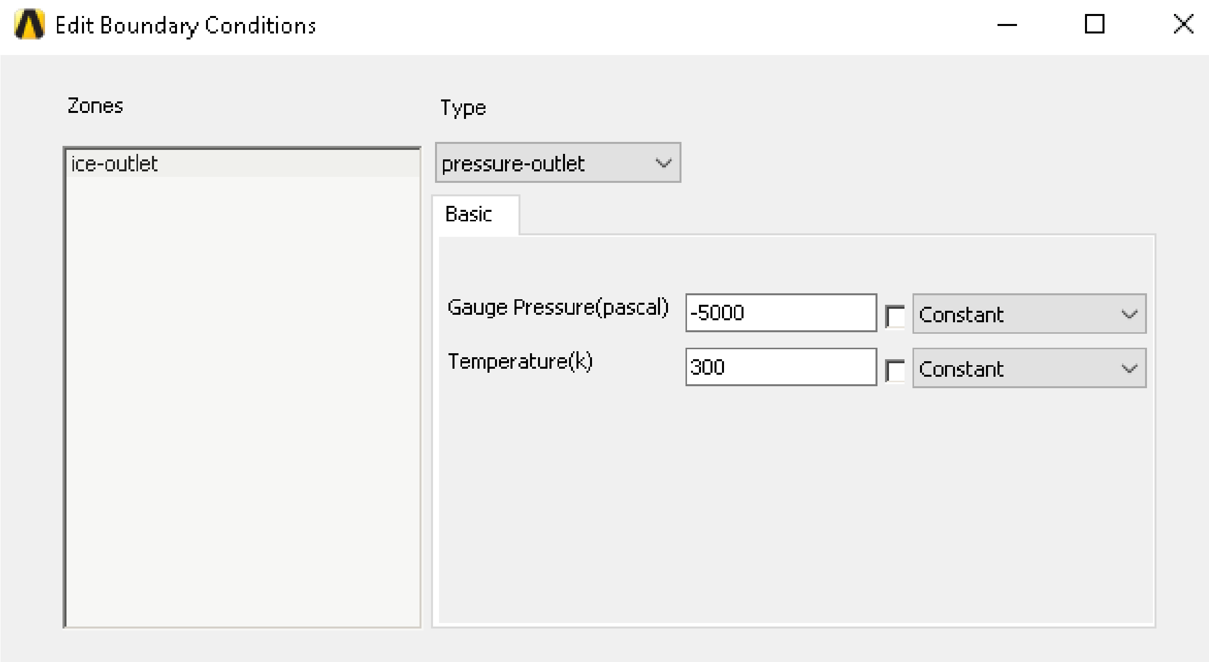


Figure 76. Boundary Conditions for outlet

In the Monitor Definitions tab, it can be seen monitors set for three zones. Mass flow rate will be obtained ice-inlet-inplenum and ice-outlet zones, type is selected as surface and report type is selected mass flow rate.

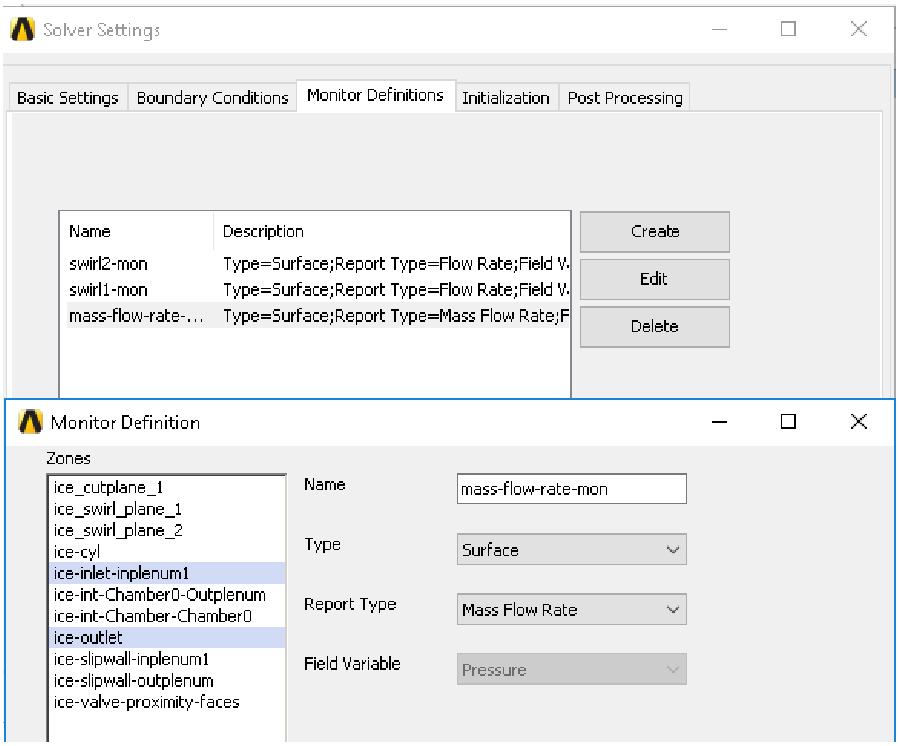


Figure 77. Monitor Definitions for Intake Port Flow

Initial values are selected as shown in Figure 65.

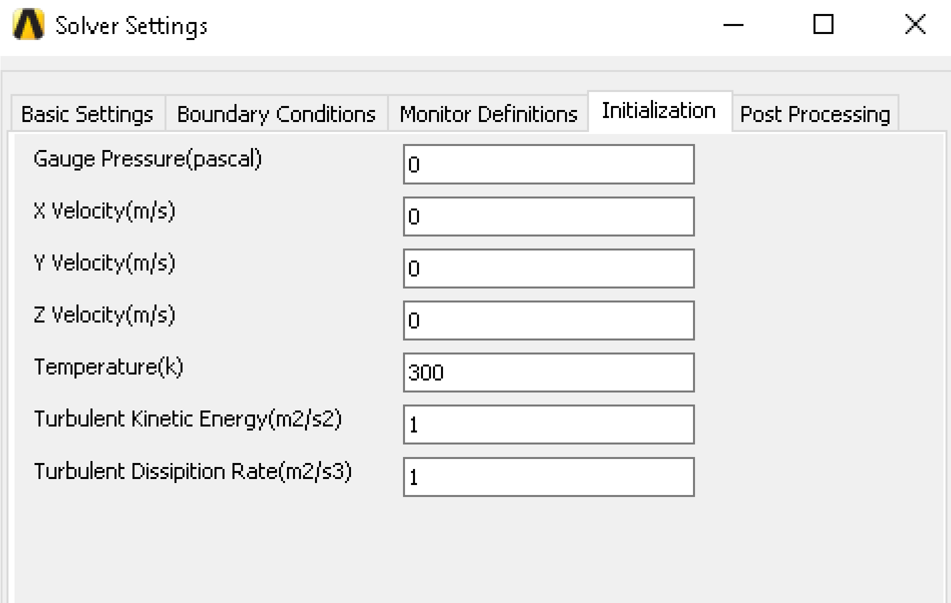


Figure 78*. Initial Values of Simulation*

7.3.1 Parameter Set

For parameter set of the intake port flow, 8 different valve lifts are considered from 2 mm valve lift to maximum valve lift . Mass flow rate values will be obtained at the end of the simulation.

Table 2*. Table of Design Points*

Table of Design Points						
	A	B	C	D	E	F
1	Name	Update Order	P1 - InValveLift	P2 - MassFlowRate	Retain	Retained Data
2	Units		mm	kg s^-1		
3	DP 0 (Current)	1	2		☑	⚡
4	DP 1	2	3.25		☑	⚡
5	DP 2	3	4.5		☑	⚡
6	DP 3	4	5.75		☑	⚡
7	DP 4	5	7		☑	⚡
8	DP 5	6	8.25		☑	⚡
9	DP 6	7	9.5		☑	⚡
10	DP 7	8	10.345		☑	⚡
*					☐	

7.4 Solver Default Settings

We clicked OK and ANSYS Fluent reads the mesh file and sets up the IC Engine case. It will:

- Set up the required models.
- Set up the default boundary conditions and material.
- Set up the default monitors.

Figure 79. Fluent Luncher

8 - ANSYS WORKBENCH EXHAUST PORT FLOW SIMULATION

For the exhaust port flow simulation, we used IC Engine Fluent module of the Workbench.

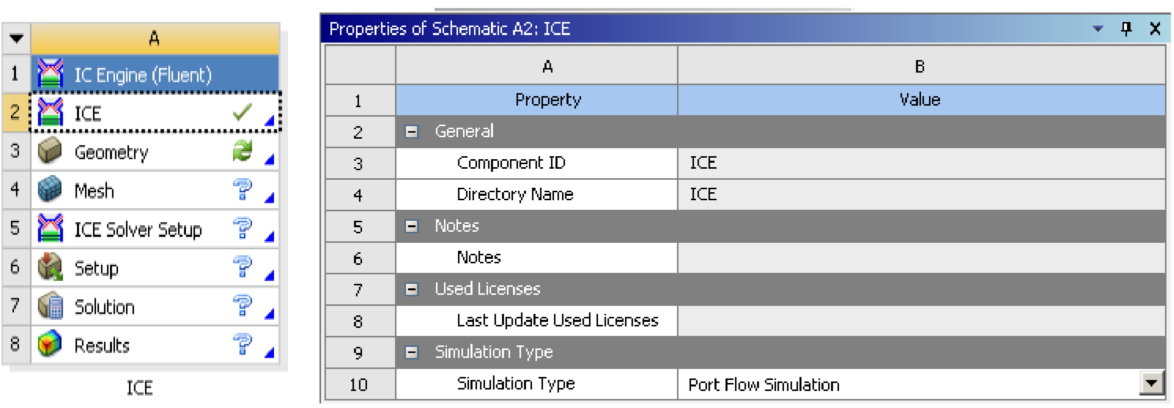


Figure 80. IC Engine Fluent Module in Workbench

8.1 Preparing the Geometry

We set the unit as mm, depending upon the geometry units, in ICE-DesignModeler. We loaded the geometry file and clicked ‘Generate’ to complete the import.

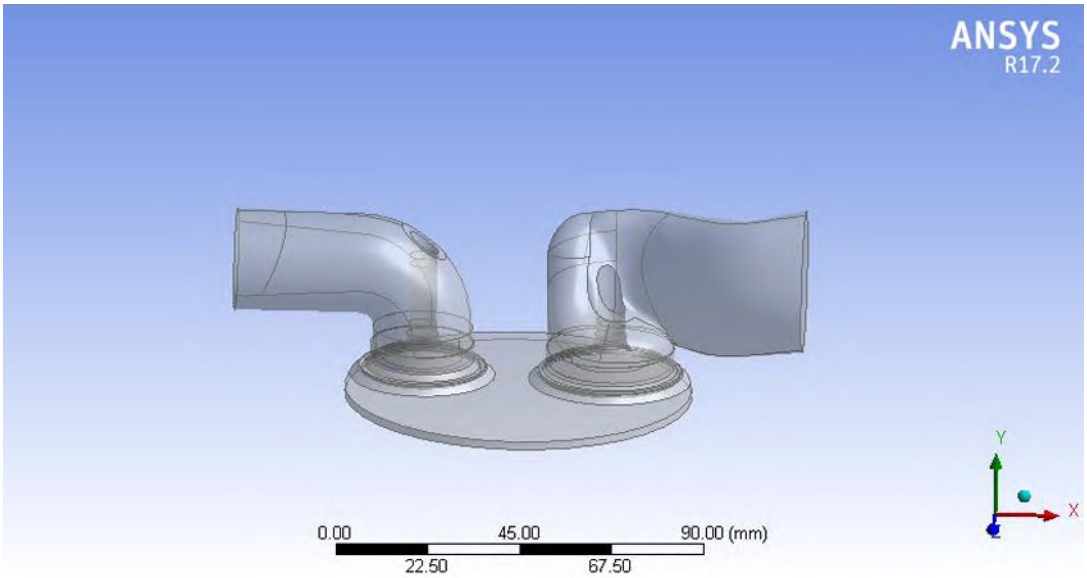


Figure 81. *Imported Geometry File*

8.1.2 Intake and Exhaust Valve Creation

Intake and exhaust valves were created with Pre Manager.

Details View

Details of ICPreManager1	
Name	ICPreManager1
Pre Manager Operation	Create Valve
Valve Hole Edges	2 Edges
Valve Seats	2 Faces
Valve Stem Extrusion Lenght	0.012486 mm

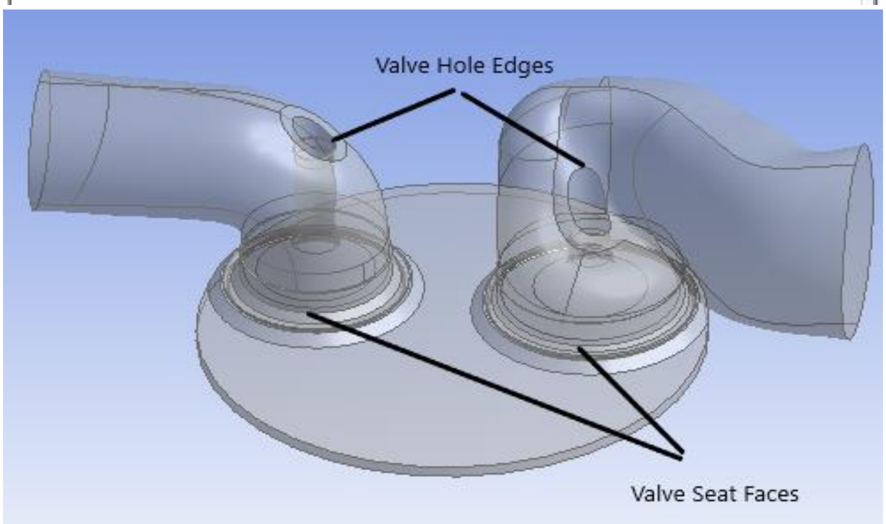


Figure 82. *Valve Hole Edges and Valve Seat Selections*

Created intake and exhaust valves are shown in the Figure 70.

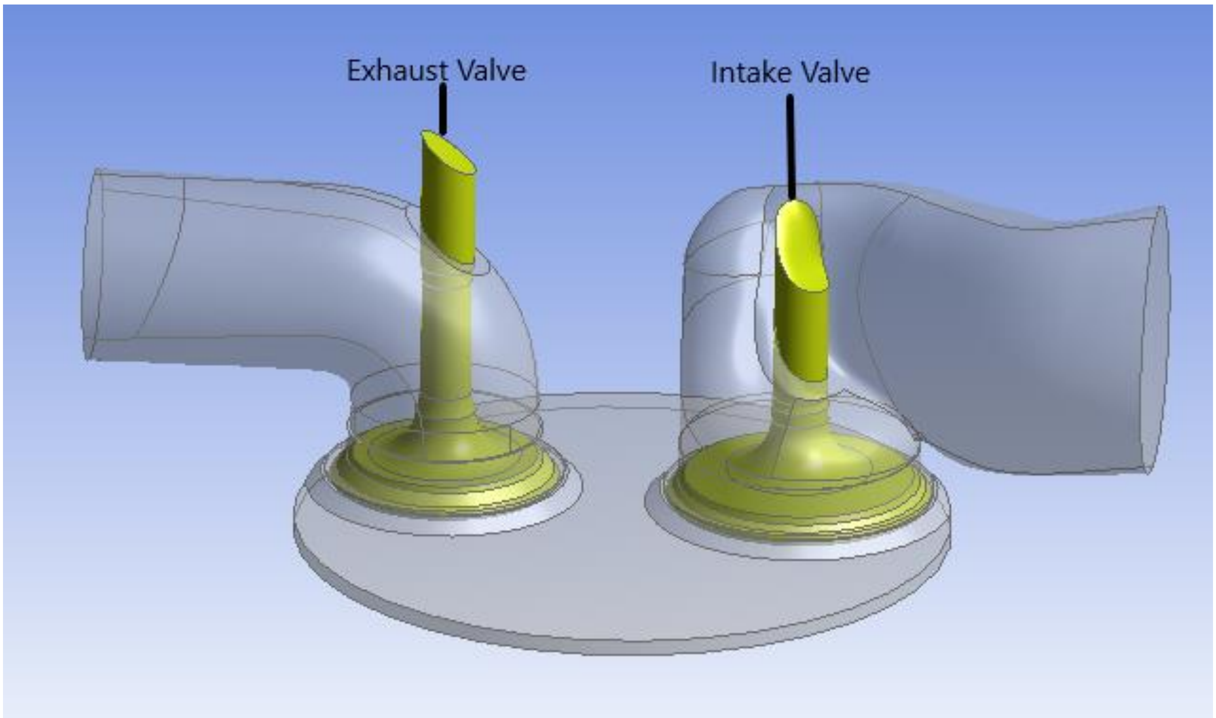


Figure 83. *Intake Valve and Exhaust Valve*

8.1.3 Input Manager

Input Manager dialog box takes inputs of the engine to set it up for decomposition.

Details View	
Details of InputManager1	
Slice	InputManager1
Cylinder Liner Faces	1 Face
Symmetry Face Option	No
Post Planes Dist. From Ref.	None
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
FD1, Valve Lift	0 mm (Deactivate Port)
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
D FD2, Valve Lift	2 mm
IC Inlet Plenum 1 (RMB)	
Inlet/(Plenum Inlet) Faces	1 Face
Plenum Type	Hemisphere
Inlet Extension Length	156 mm
Plenum Size	140 mm
Plenum Blend Rad	25 mm
IC Outlet Plenum (RMB)	
Outlet Plenum Option	Yes
Cylinder Extension Length	90 mm
Plenum Type	Cylinder
Plenum Size	120 mm

Figure 84. Details of Input Manager

Cylinder liner face selected. There is no post planes that are created. These planes are required for creating swirl monitors in Fluent. Intake valve and exhaust valve were selected. Exhaust valve lift value entered as 2 mm. Intake valve lift value entered as 0 mm because we performed exhaust port flow simulation. For the inlet plenum, inlet face, plenum type (Hemisphere), inlet extension length (156 mm), plenum size (140mm) and plenum blend radius (25 mm) were selected. Cylinder extension length entered as 90 mm because engine has 90 mm stroke. Plenum type (cylinder) and plenum size (120 mm) were entered.

Selected regions in Input Manager are shown in the Figure 72.

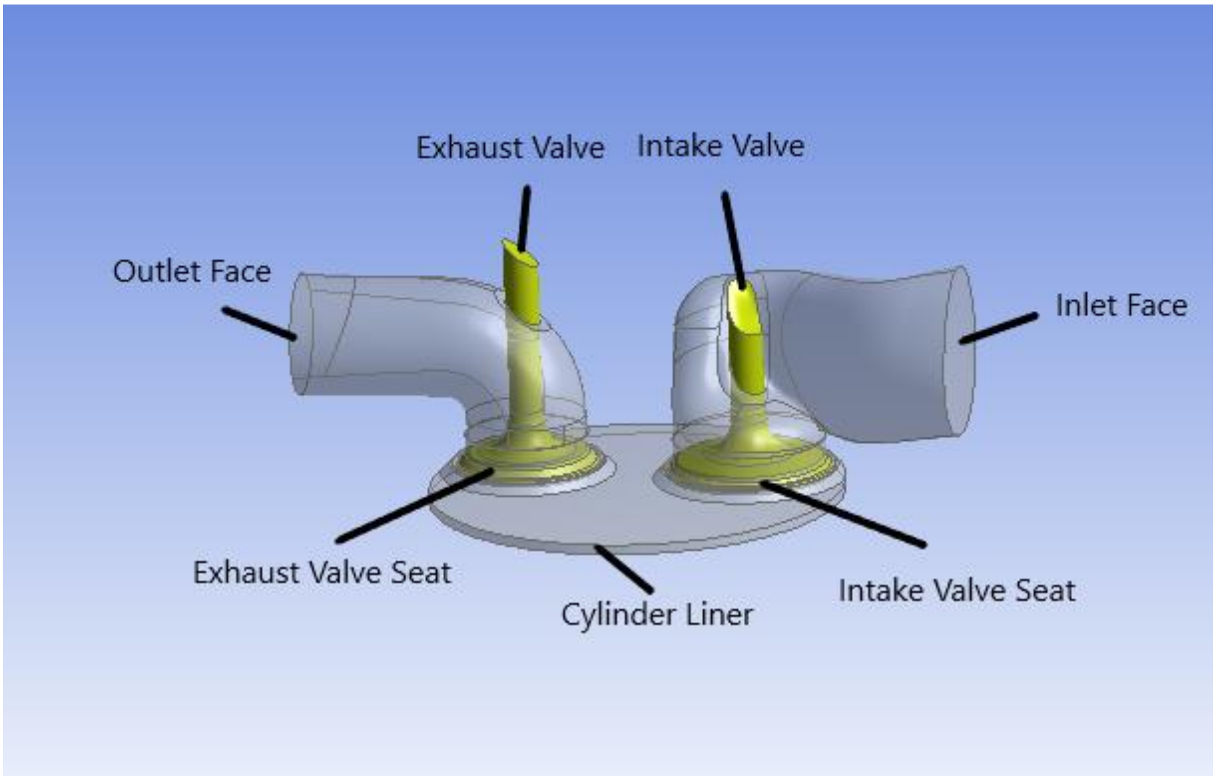


Figure 85. Input Manager Selected Regions

8.1.4 Decomposition

We performed decomposition (located in the IC Engine toolbar). After decomposition, the engine is divided into one part, Port and several bodies.

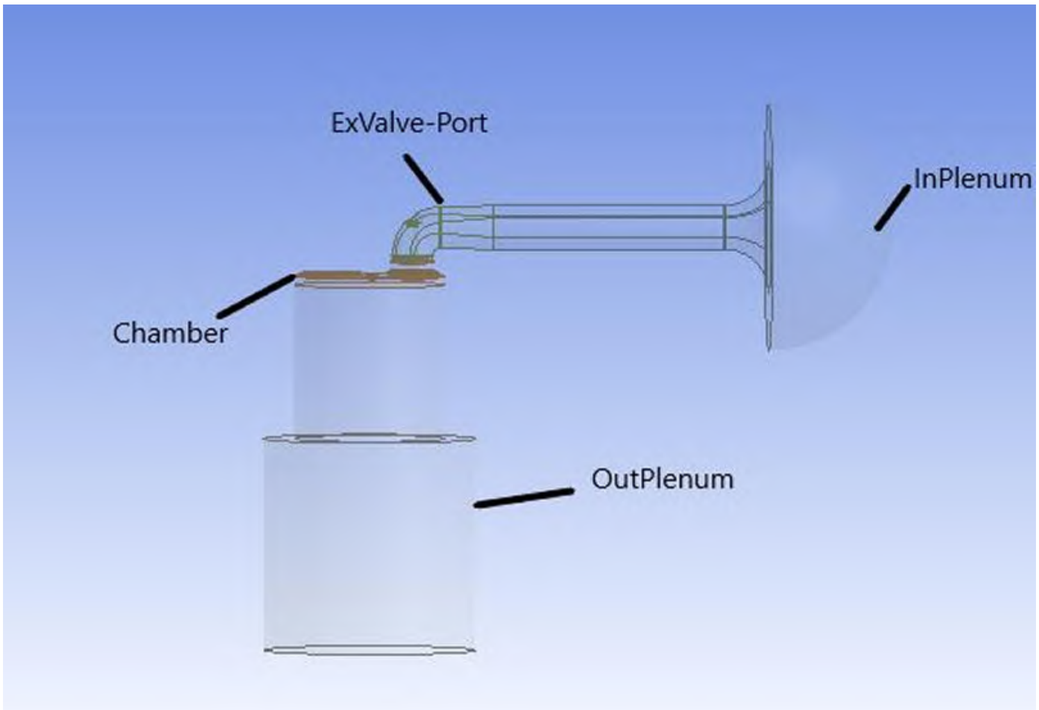


Figure 86. Several Bodies after Decomposition

Outplenum: The plenum created at the outlet.

Inplenum: The plenum created at the inlet.

Chamber: The chamber is divided into parts depending upon the number of postprocessing planes.

Exvalve-Port: Depending upon the number of exhaust valves in the geometry, Exvalve-Port is created.

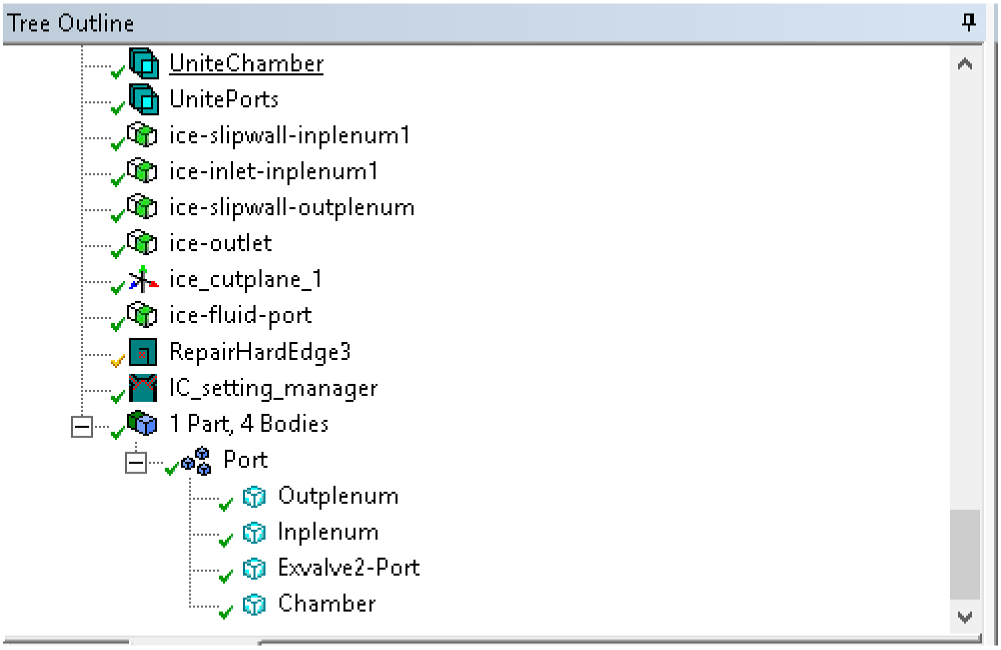
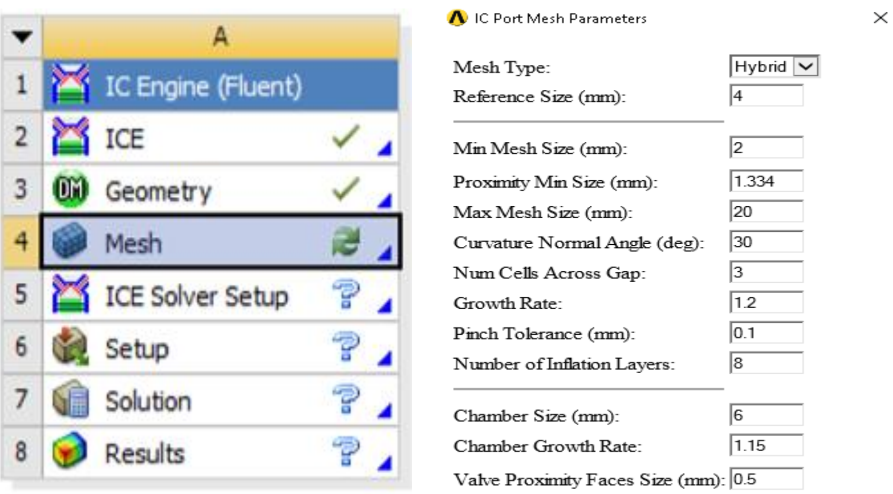


Figure 87. Tree Outline of the Decomposed Geometry

8.2 Meshing Procedure with ANSYS Meshing Application for Port Flow Simulation

We used the Mesh cell in the IC Engine analysis system to open the ANSYS Meshing application.



We started IC Setup Mesh (located in the IC Engine toolbar).This opens the IC Mesh Parameters dialog box. We selected mesh type as Hybrid and reference size as 4mm. We used default settings for other parameters.

Figure 88. Mesh Properties for Exhaust Port Flow Simulation

After selecting mesh parameters, we generated mesh with 'Generate' button.

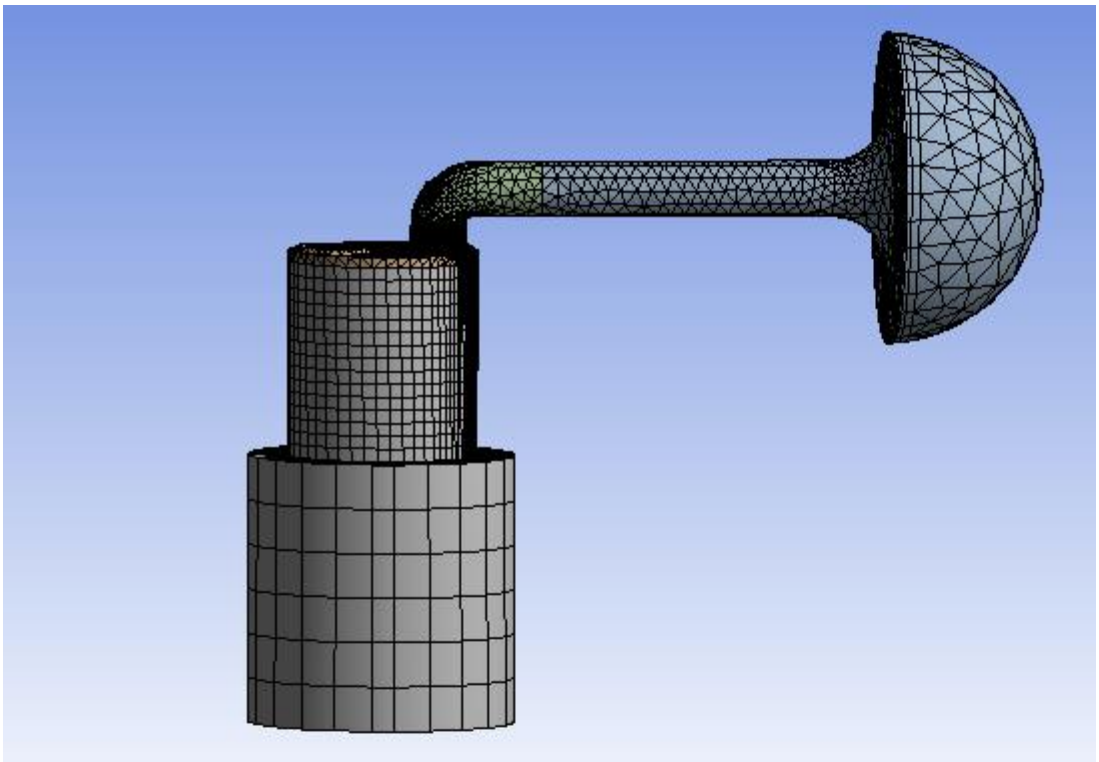


Figure 89. Meshing for Exhaust Port Flow Simulation

8.3 Setting Up the Analysis in IC Engine (Fluent)

In the Basic Settings tab we have the following settings:

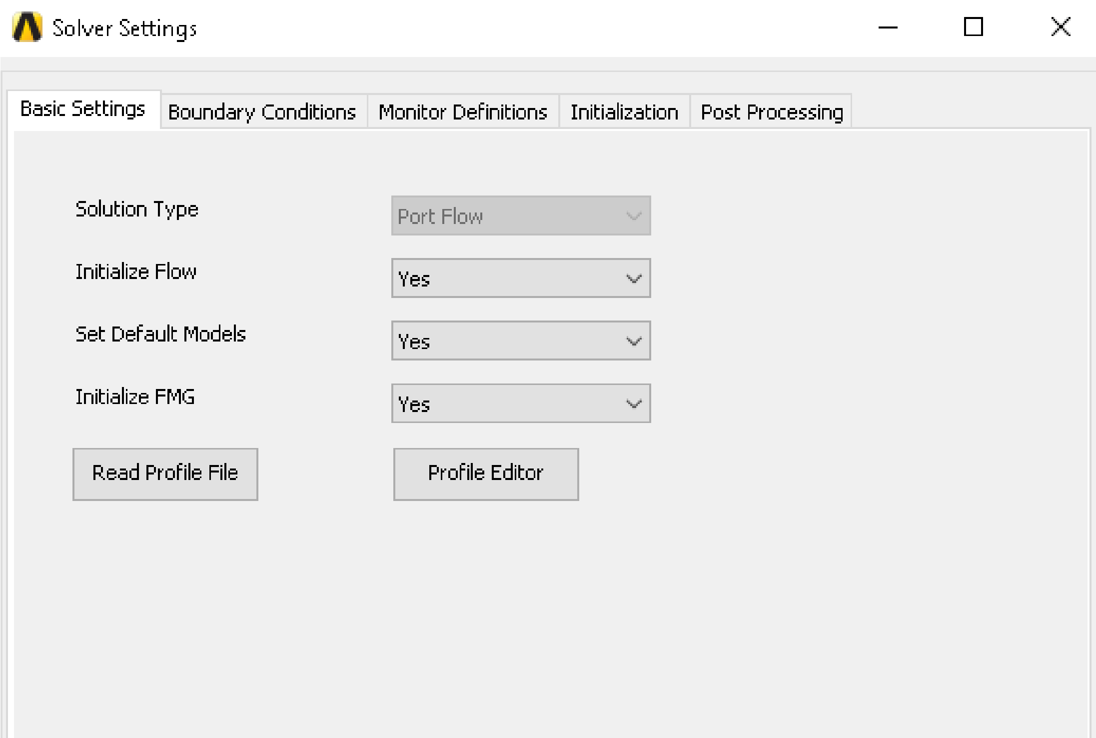


Figure 90. Basic Settings for Intake Port Flow

In the Boundary Conditions tab, it can be seen boundary conditions set for four zones.

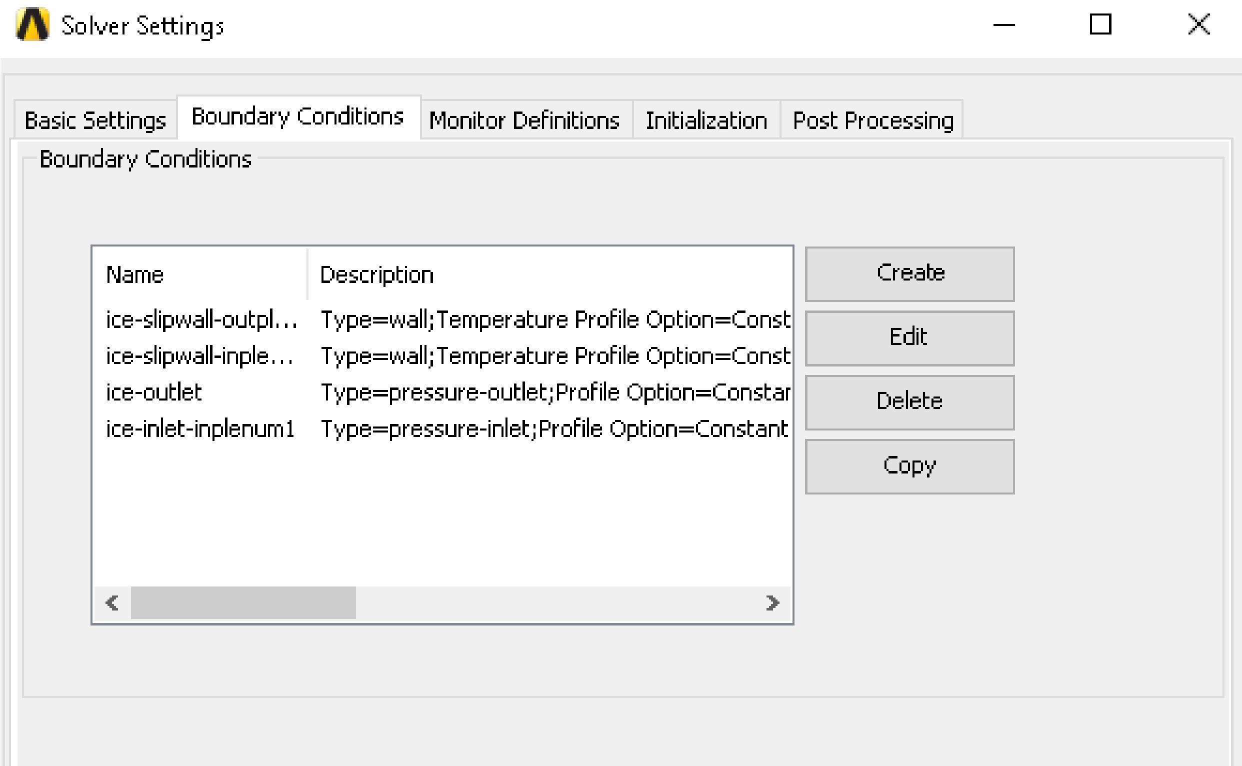


Figure 91. *Boundary Conditions Tab for Intake Port Flow*

The wall is set as slipwall and the Temperature is set to 300.

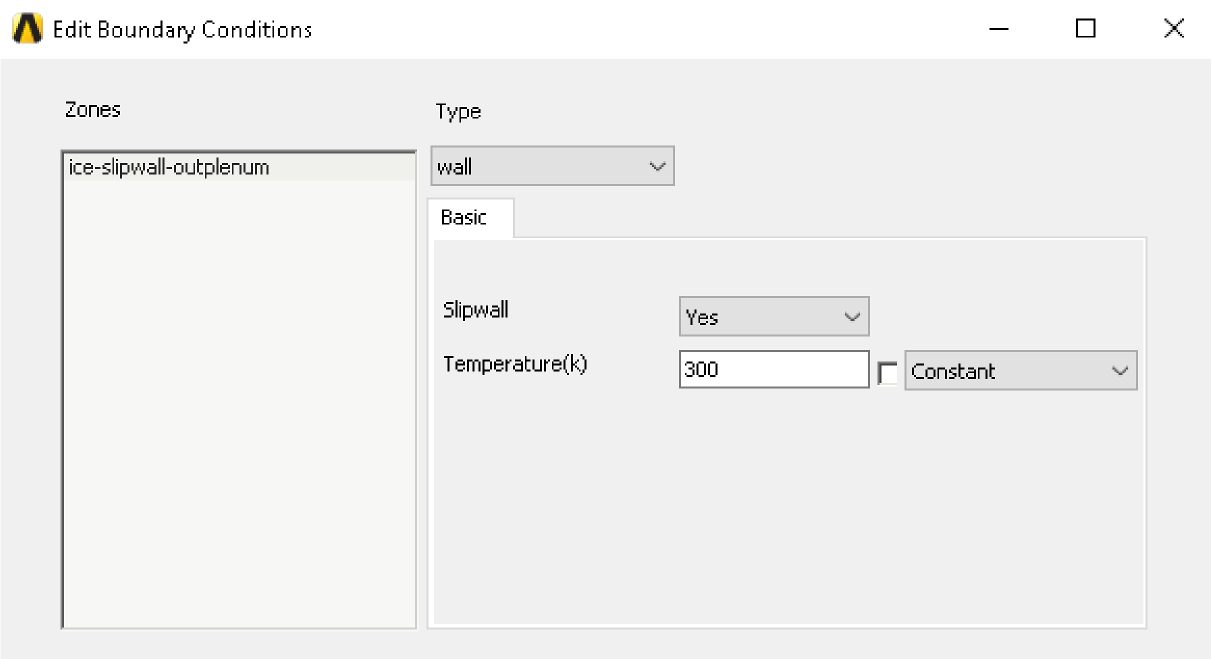


Figure 92. *Boundary Conditions for slipwall-outplenum*

The wall is set as slipwall and the Temperature is set to 300.

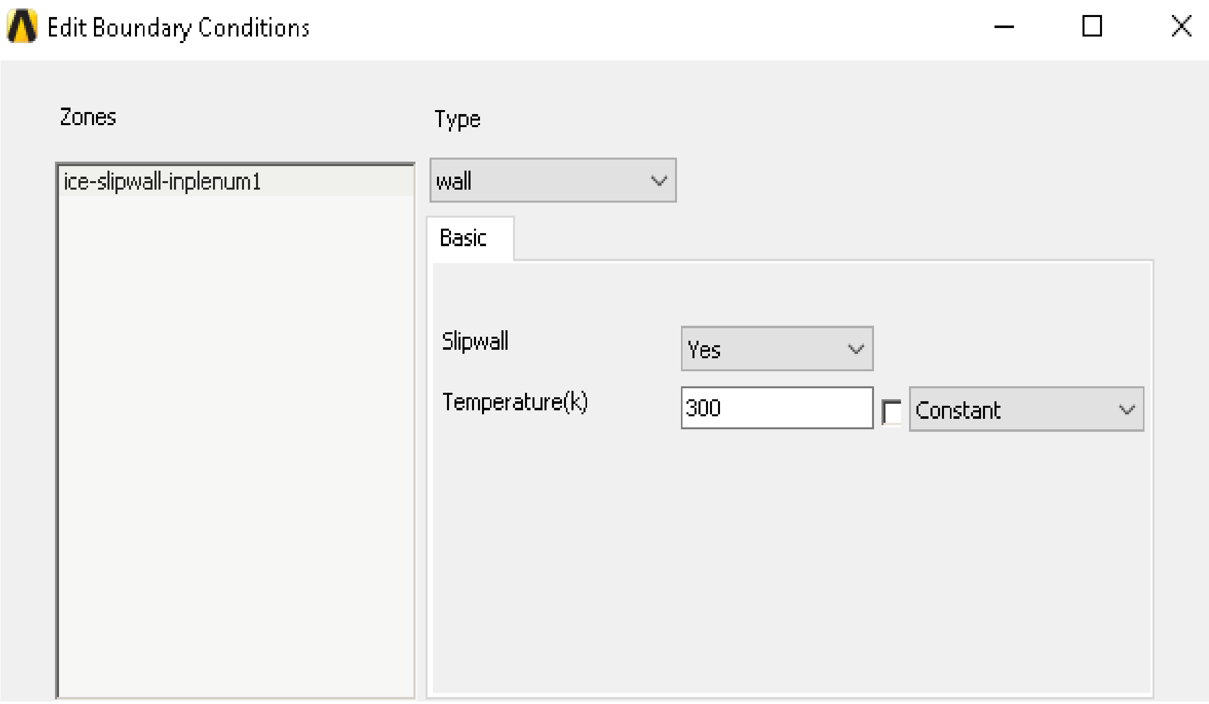


Figure 93. Boundary Conditions for slipwall-inplenum

Pressure-outlet type is selected, gage pressure is set to 0 pascal and temperature is set to 300 K for inlet-inplenum.

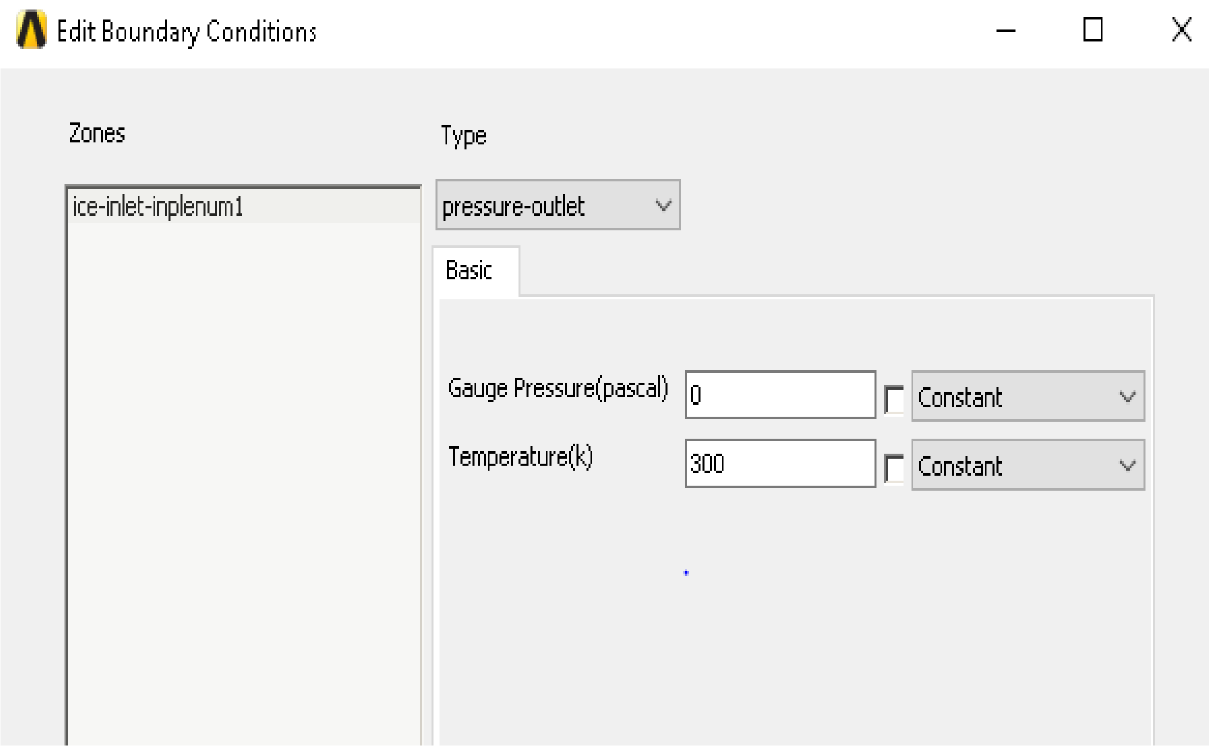


Figure 94.Boundary Conditions for inlet-inplenum

For the outlet zone, pressure-inlet type is selected, gauge pressure is set to 5000 pascal and temperature is set to300 K.

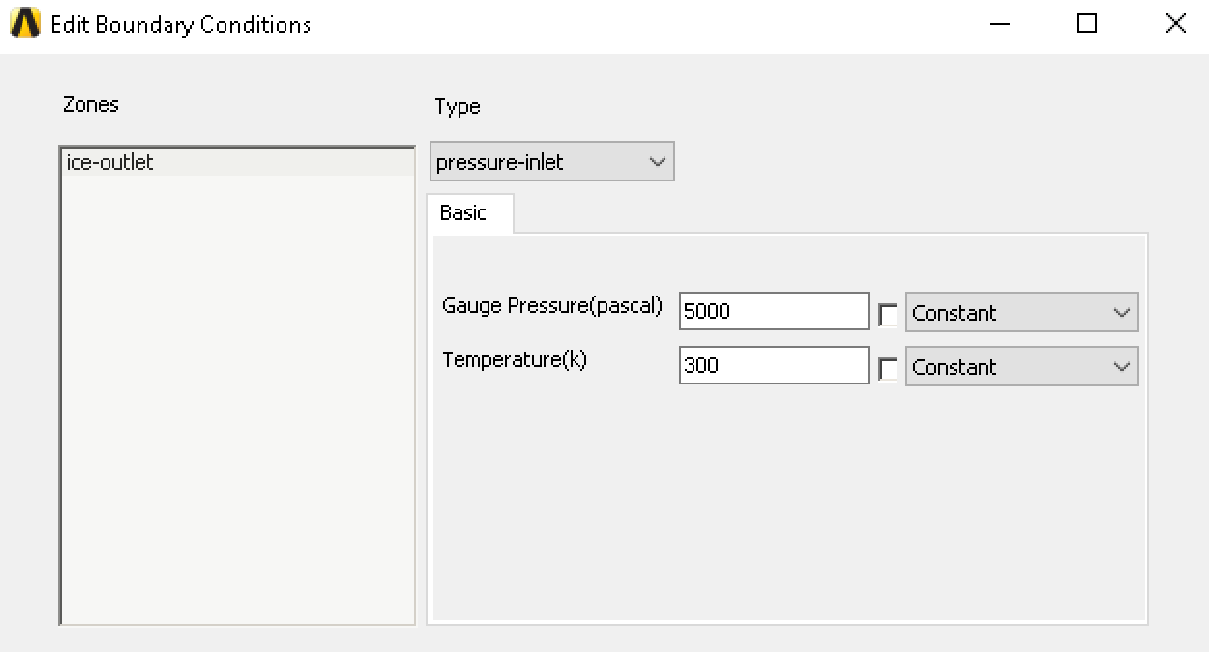


Figure 95. Boundary Conditions for outlet

In the Monitor Definitions tab, it can be seen monitors set for three zones. Mass flow rate will be obtained ice-inlet-inplenum and ice-outlet zones, type is selected as surface and report type is selected mass flow rate.

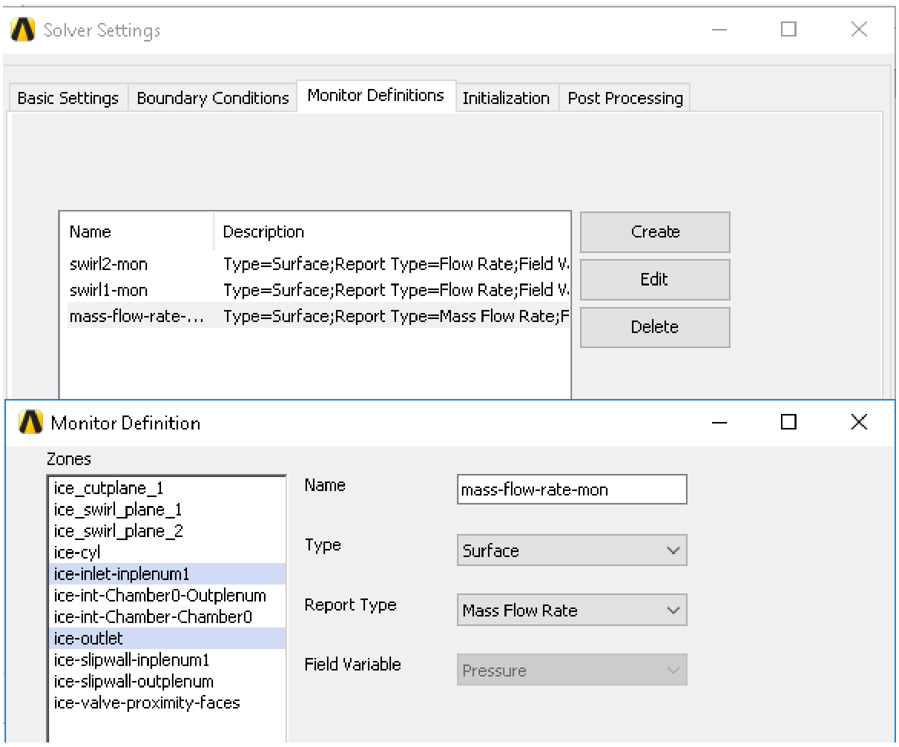


Figure 96. Monitor Definitions for Exhaust Port Flow

Initial values are selected as shown in Figure 65.

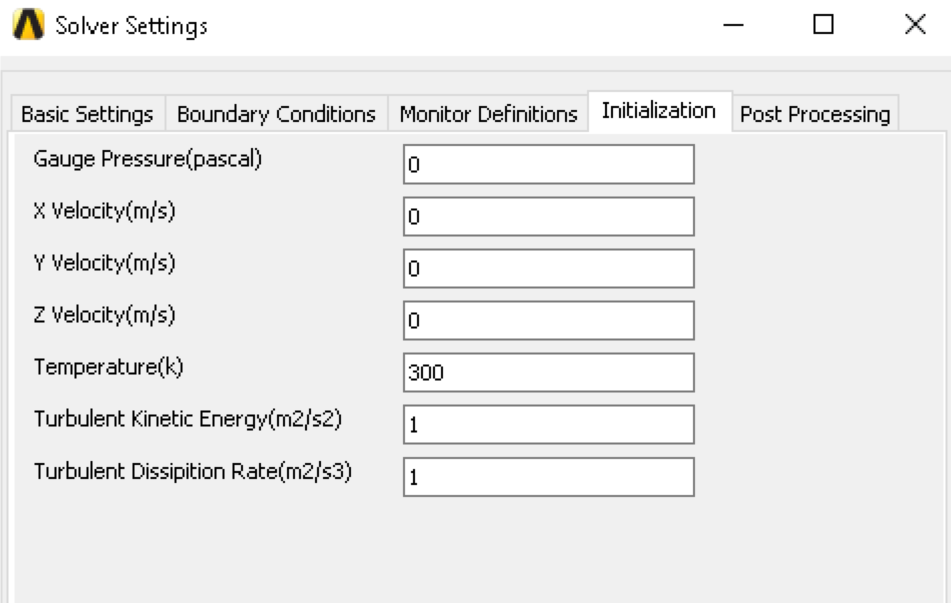


***Figure 97**. Initial Values of Simulation*

8.3.1 Parameter Set

For parameter set of the exhaust port flow, 7 different valve lifts are considered from 2 mm valve lift to maximum valve lift . Mass flow rate values will be obtained at the end of the simulation.

***Table 2**. Table of Design Points*

Table of Design Points

	A	B	C	D	E	F
1	Name	Update Order	P2 - ExValveLift	P1 - MassFlowRate	Retain	Retained Data
2	Units		mm	kg s^-1		
3	DP 0 (Current)	1	2		☑	
4	DP 1	2	3.25		☑	
5	DP 2	3	4.5		☑	
6	DP 3	4	5.75		☑	
7	DP 4	5	7		☑	
8	DP 5	6	8.25		☑	
9	DP 6	7	9.51		☑	
*					☐	

8.4 Solver Default Settings

We clicked OK and ANSYS Fluent reads the mesh file and sets up the IC Engine case. It will:

• Set up the required models.

• Set up the default boundary conditions and material.

• Set up the default monitors.

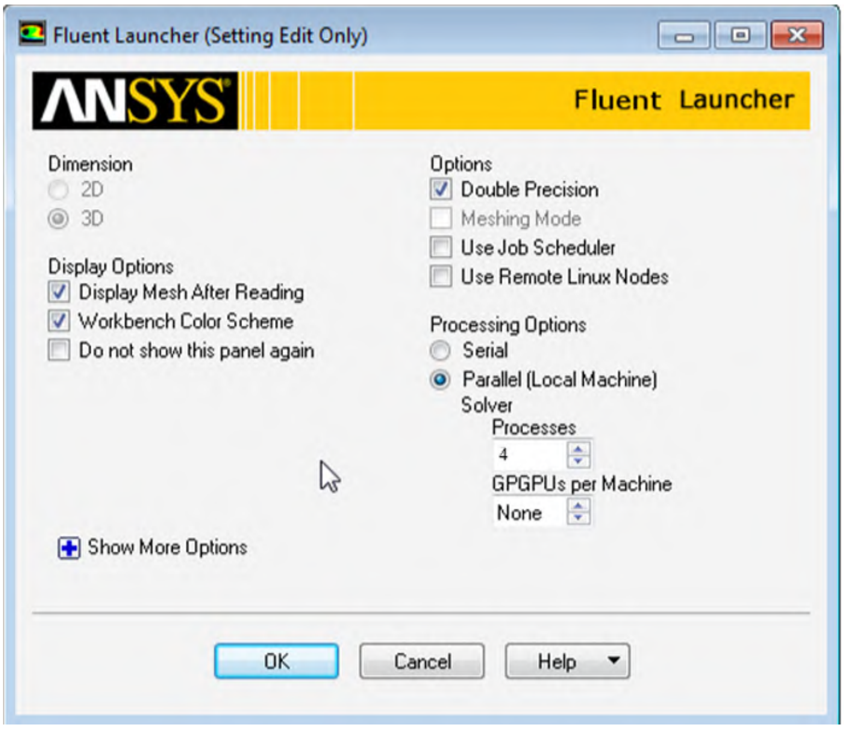


Figure 98. Fluent Luncher

9 - ANSYS WORKBENCH COLD FLOW SIMULATION

.For the coldflow simulation, we used IC Engine Florte module of the Workbench. Simulation settings are as shown in Figure 87.

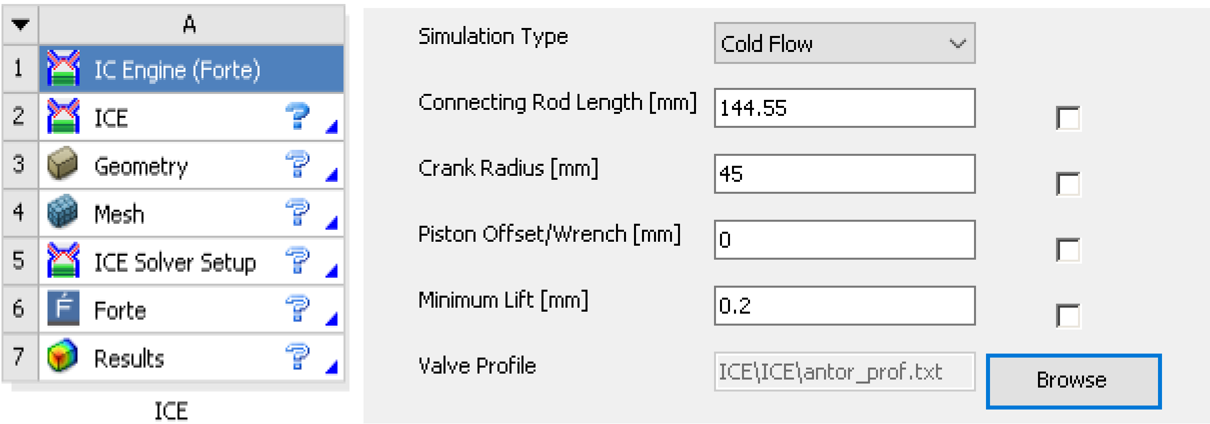


Figure 99. IC Engine Forte Module in Workbench with Simulation Settings

9.1 Preparing the Geometry

We set the unit as mm, depending upon the geometry units, in ICE-DesignModeler. We loaded the geometry file and clicked 'Generate' to complete the import.

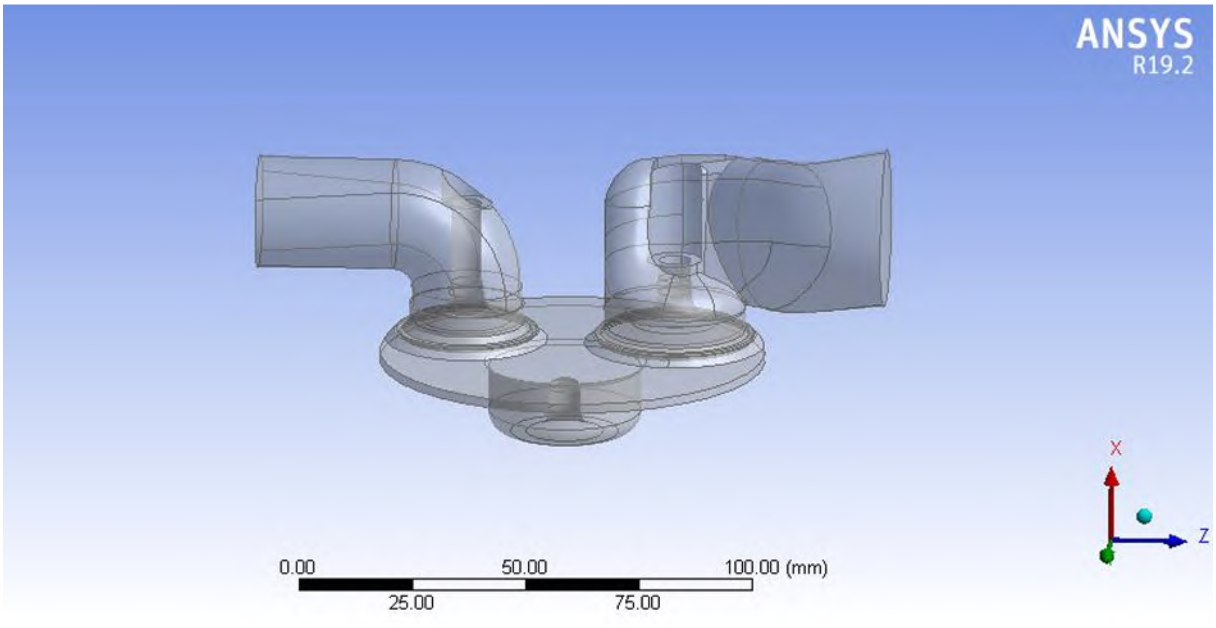


Figure 100. *Imported Geometry File*

9.1.2 Intake and Exhaust Valve Creation

Intake and exhaust valves were created with Pre Manager.

Details View

Details of ICPreManager1	
Name	ICPreManager1
Pre Manager Operation	Create Valve
Valve Hole Edges	2 Edges
Valve Seats	2 Faces
Valve Stem Extrusion Lenght	0.012486 mm

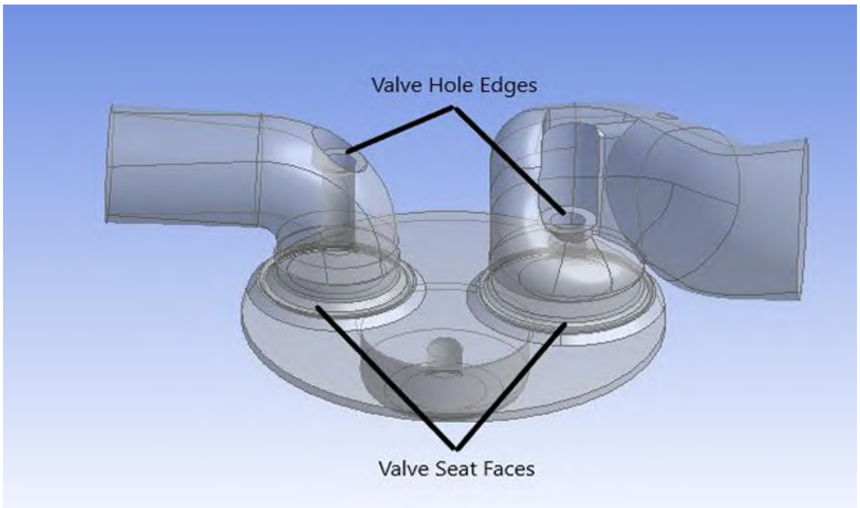


Figure 101. *Valve Hole Edges and Valve Seat Selections*

Created intake and exhaust valves are shown in the Figure 70.

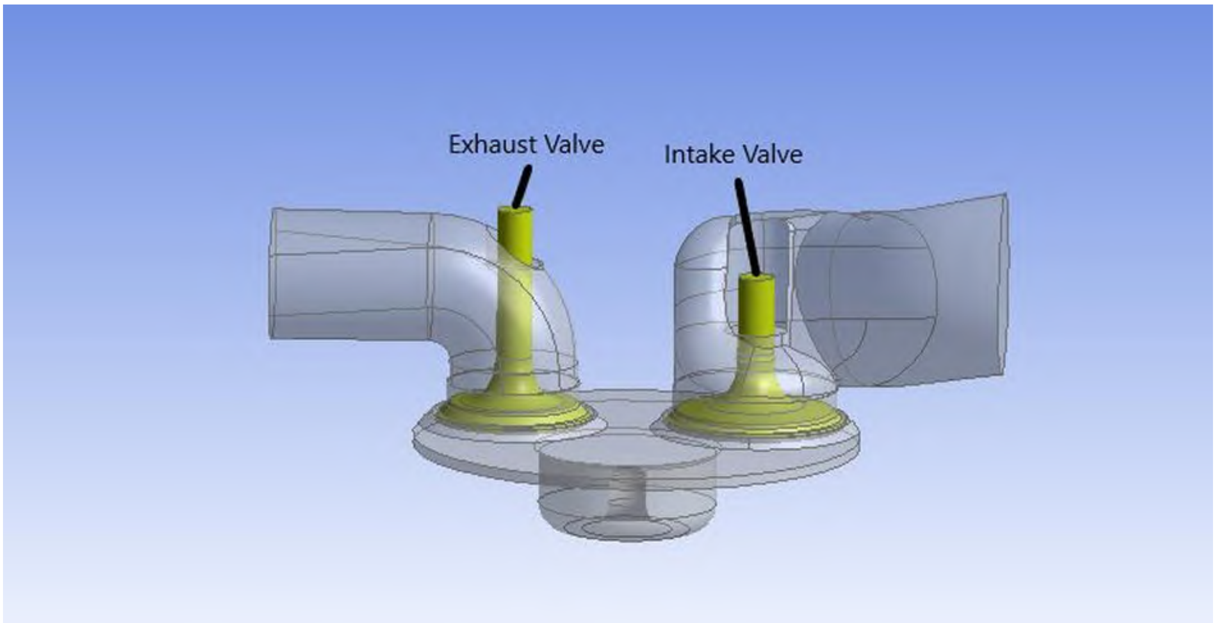


Figure 102. Intake Valve and Exhaust Valve

9.1.3 Input Manager

Input Manager dialog box takes inputs of the engine to set it up for decomposition.

Details View

Details of InputManager1	
Slice	InputManager1
Inlet Faces	1 Face
Outlet Faces	1 Face
Cylinder Liner Faces	1 Face
Symmetry Face Option	No
Validate Compression Ratio	Yes
Compression Ratio	17.5
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	exvalve1
IC Animation Inputs (RMB)	
Start Crank Angle	-360 °
End Crank Angle	720 °
Intervals	1 °
Spray Cones Option	No
IC Advanced Options (RMB)	
Piston Profile Option	No

Figure 103. Input Manager for Cold Flow Simulation

Inlet, outlet and cylinder liner faces are selected. There is no post planes that are created. Compression ratio is entered as 17.5. Intake valve and exhaust valve are selected. Engine valve profiles are entered. Simulation start crank angle is set -360 CA and end crank angle is set 720 CA with 1 CA intervals. Simulation starts with -360 CA because the intake valve open at this CA.

Selected regions in Input Manager are shown in the Figure 72.

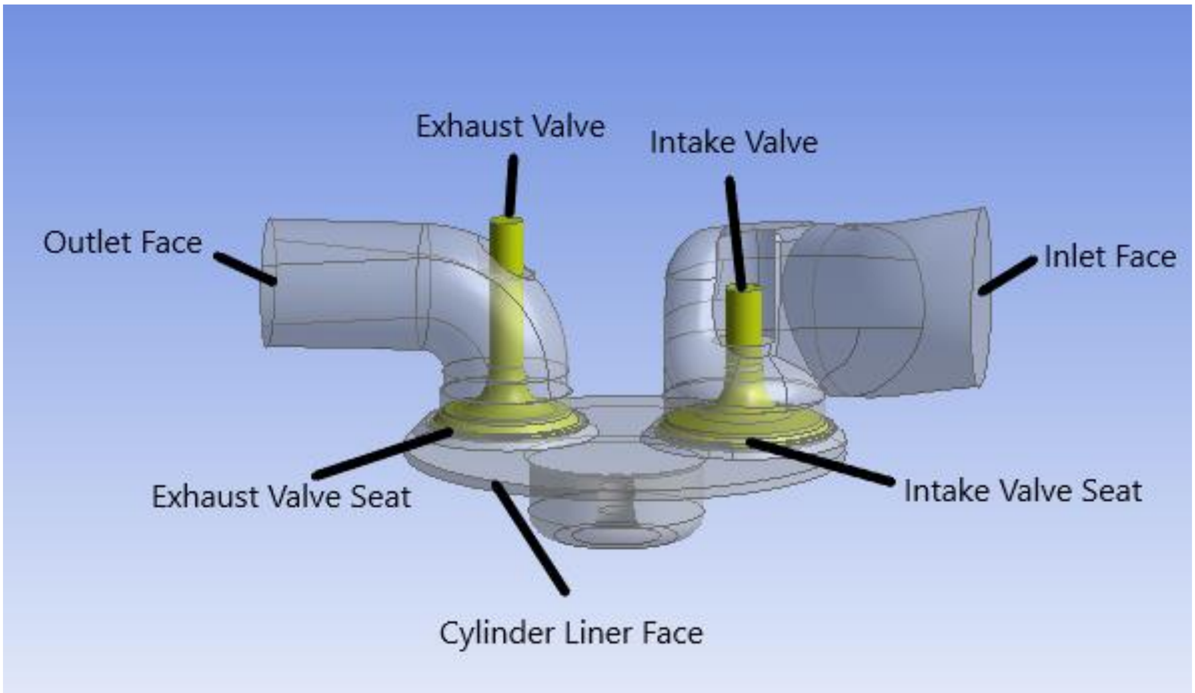


Figure 104. *Input Manager Selected Regions*

9.1.4 Decomposition

We performed decomposition (located in the IC Engine toolbar). After decomposition, we have one port body that is shown in Figure 95.

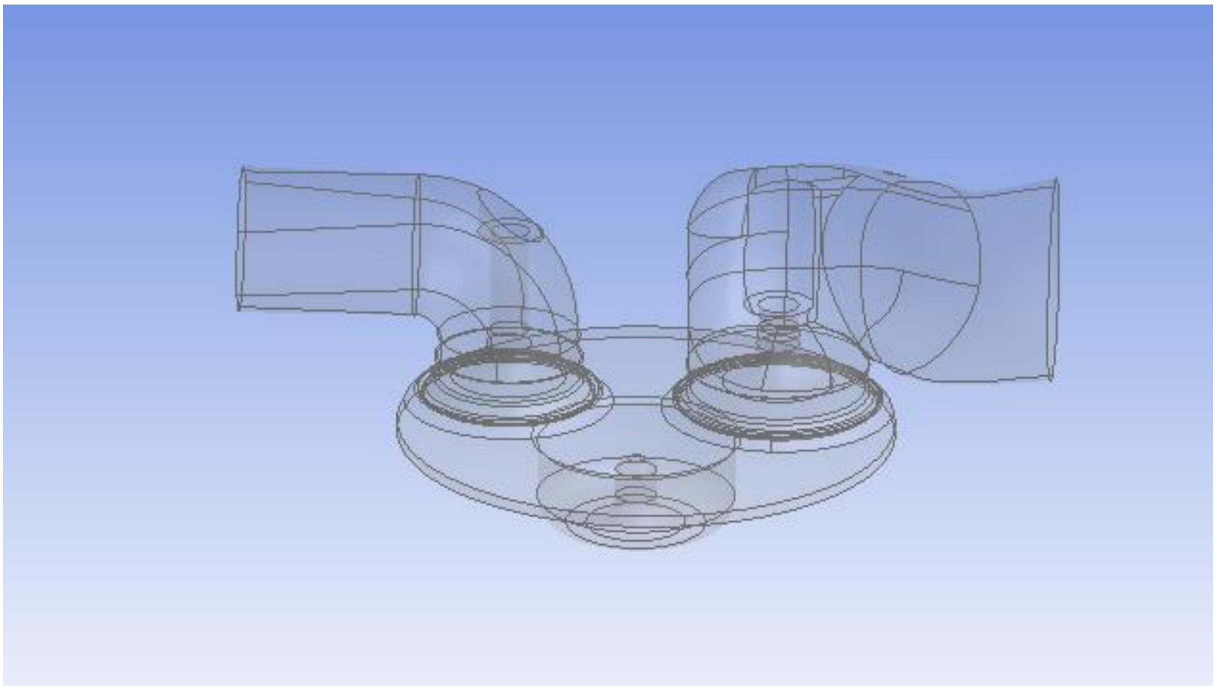
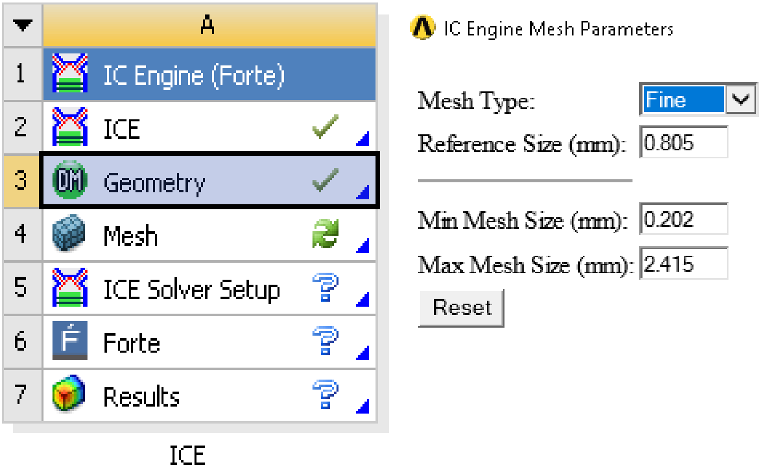


Figure 105. *After the Decomposition*

9.2 Meshing Procedure with ANSYS Meshing Application for Cold Flow Simulation

We used the Mesh cell in the IC Engine analysis system to open the ANSYS Meshing application.



We started IC Setup Mesh (located in the IC Engine toolbar).This opens the IC Mesh Parameters dialog box. We selected reference size as 0.805 mm. We used default settings for other parameters.

Figure 106*. Mesh Properties for Exhaust Port Flow Simulation*

After selecting mesh parameters, we generated mesh with ‘Generate’ button.

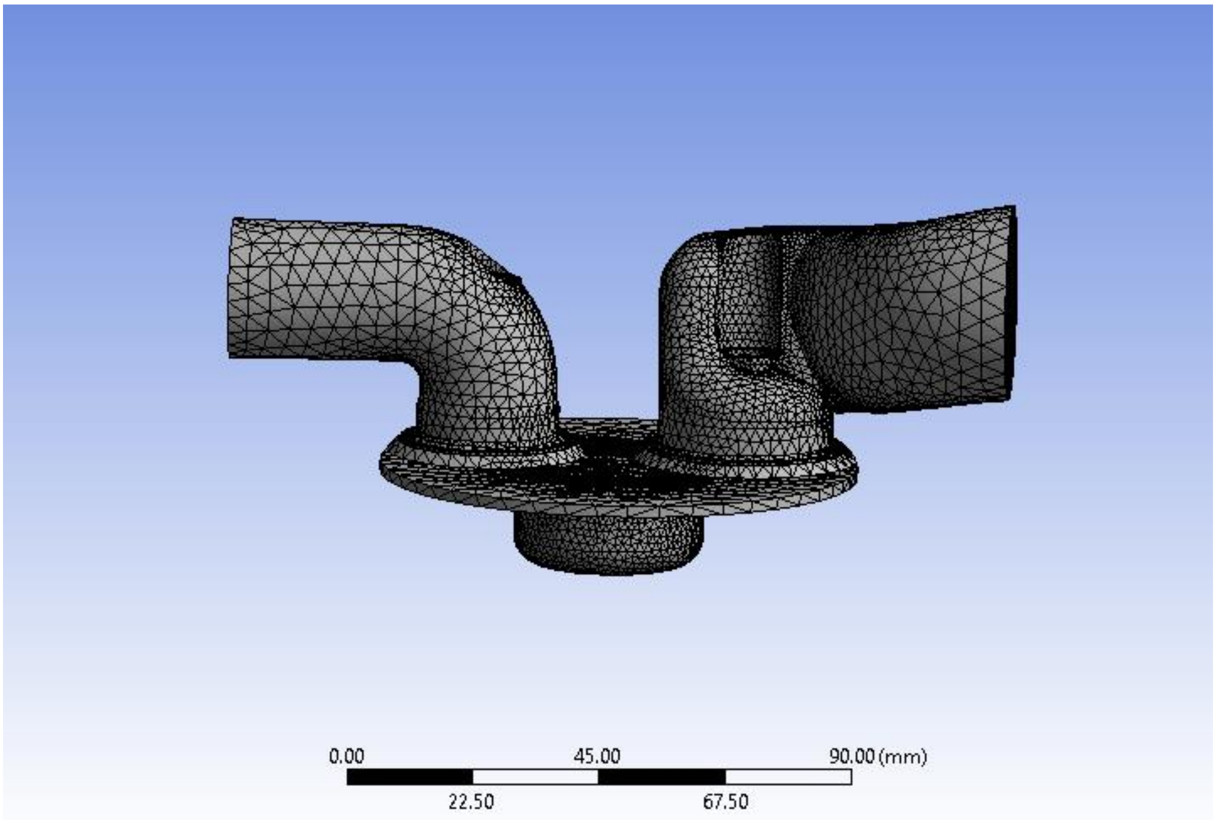


Figure 107. *Meshing for Cold Flow Simulation*

9.3 Setting Up the Analysis in IC Engine (Forte)

Global mesh size, start crank angle and end crank angle values are shown in Figure 98.

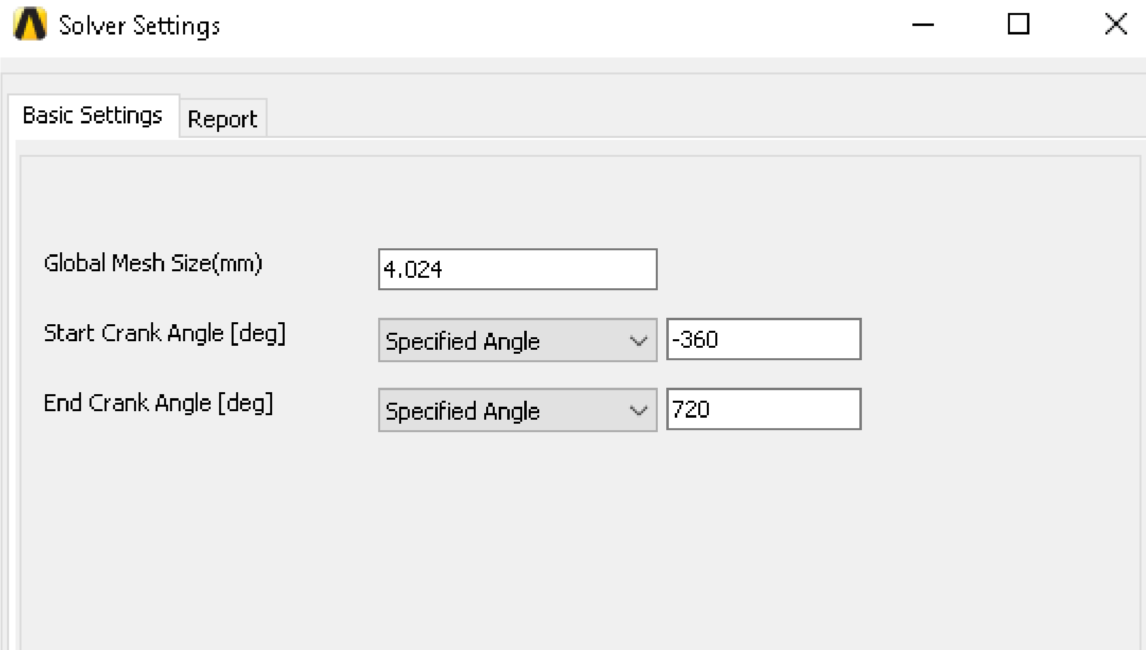


Figure 108*. Solver Settings for Cold Flow Simulation*

9.4 Boundary Conditions for Cold Flow (Forte)

All boundaries are generated automatically with decomposition that is mentioned section 8.1.4. For the cold flow simulation, piston, liner and head temperatures are set 320 K. Other boundaries have 300 K wall temperature. All boundaries have 1 bar pressure. Air is used for composition.

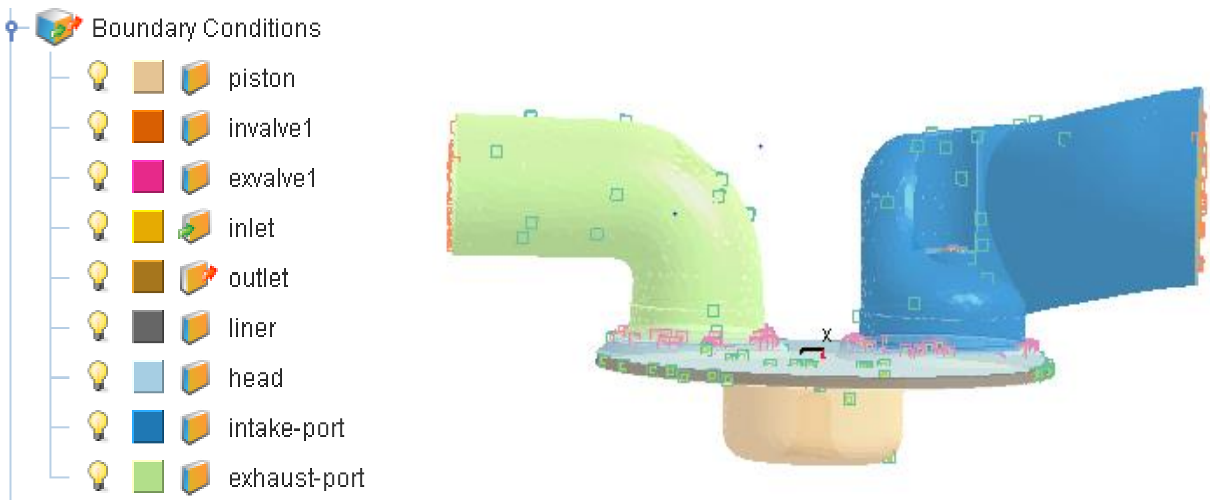


Figure 109*. Boundary Conditions for Cold Flow Simulation*

9.5 Final Setup and Run for Cold Flow (Forte)

Simulation controls is final step before running process. Engine speed is set 3000 RPM and initial CA and final CA values are shown in Figure 100.

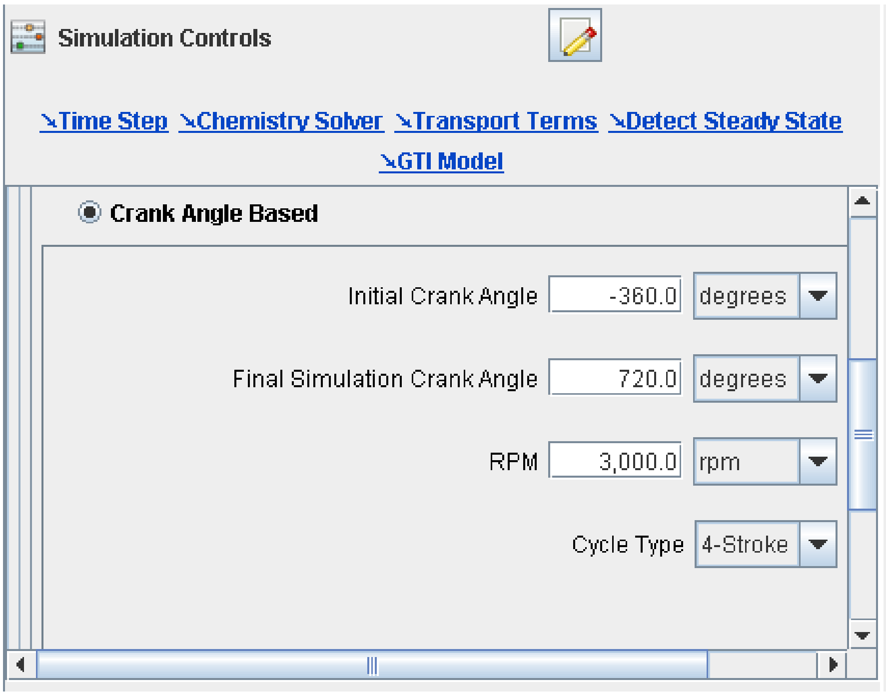


Figure 110. Simulation Controls for Cold Flow Simulation

Final settings for running process are shown in Figure 101.

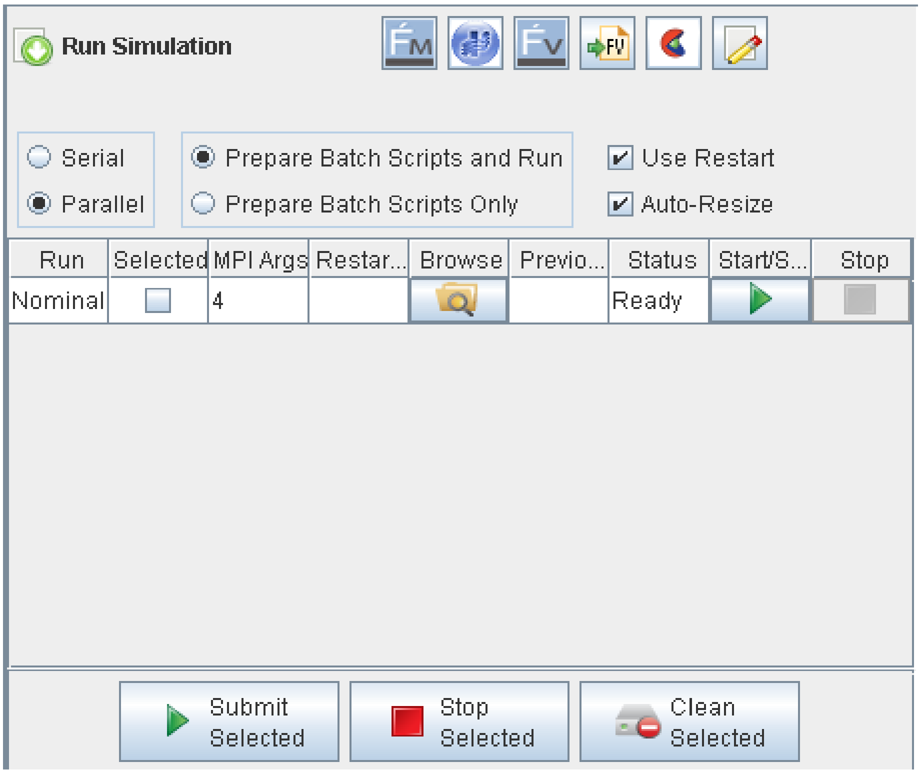


Figure 111. Run Simulation Settings for Cold Flow

10 - ANSYS WORKBENCH COMBUSTION SIMULATION

For the combustion simulation, we used similar forte case with cold flow simulation. Boundary conditions are changed only and added injection parameters.

10.1 Boundary Conditions for Combustion (Forte)

Boundaries are shown in the Figure 102.

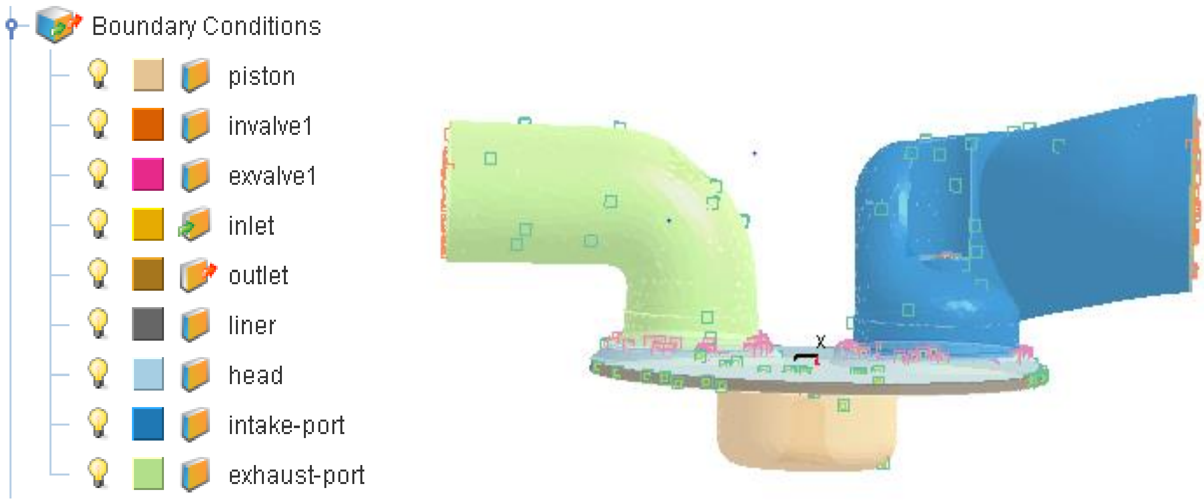


Figure 112. Boundary Conditions for Combustion Solution

Boundary temperatures are set as shown in the Table 2.

Table 3. Boundary Temperatures

Boundary	Temperature (K)
piston	590
invalve1	400
exvalve1	500
inlet	300
outlet	300
liner	450
head	550
intake-port	340
exhaust-port	600

10.2 Injection Timing

Injection timing parameters are shown in the Figure

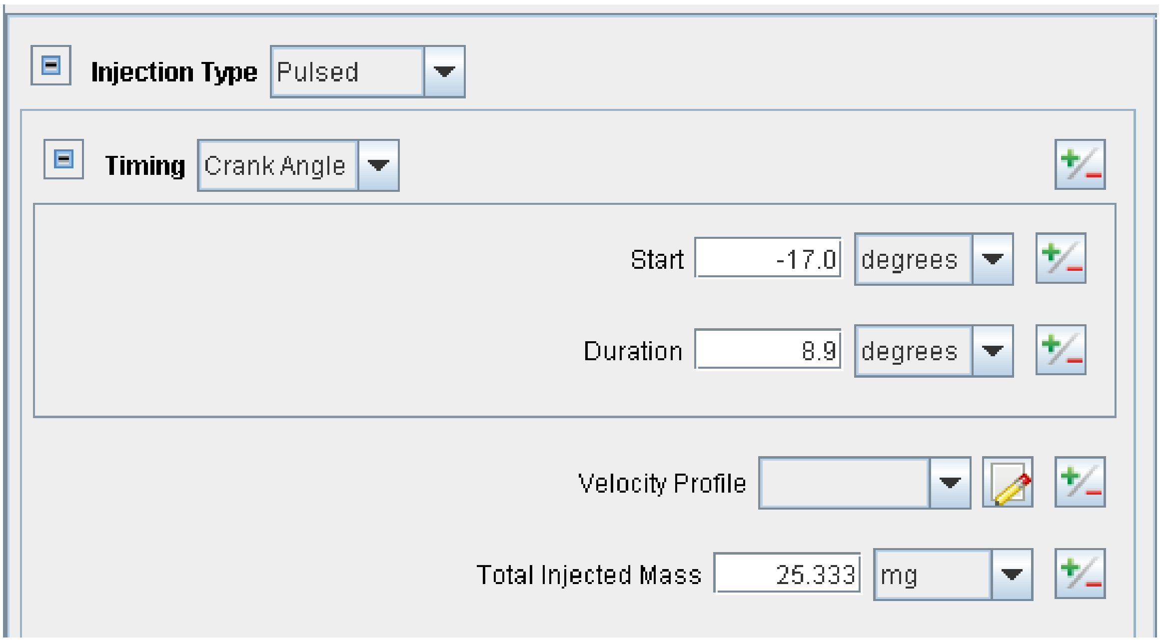


Figure 113. Injection Properties

11- Results

11.1- GT-Power Combustion

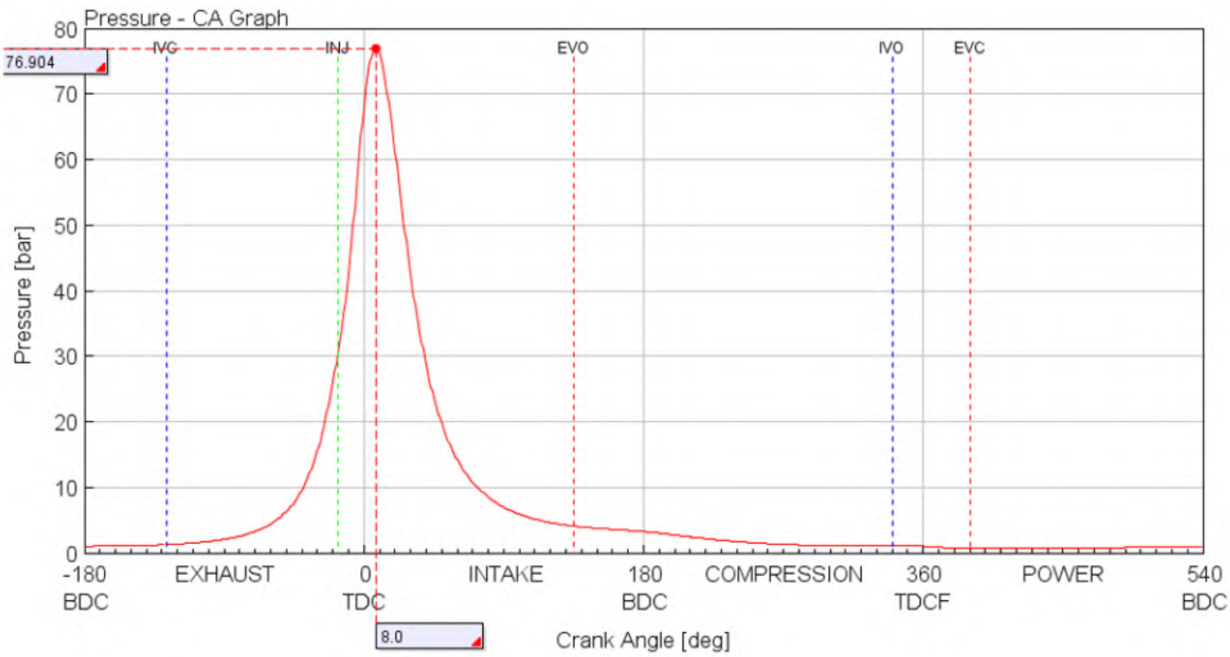


Figure 114 PRESSURE VS CRANK ANGLE

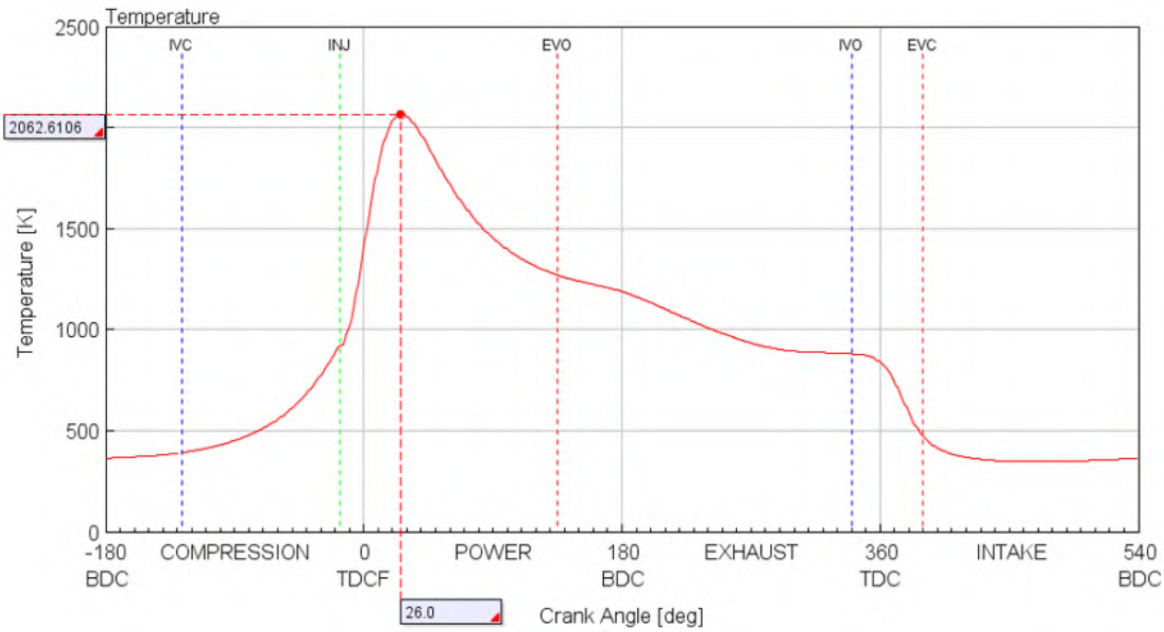


***Figure 115** TEMPERATURE VS CRANK ANGLE*

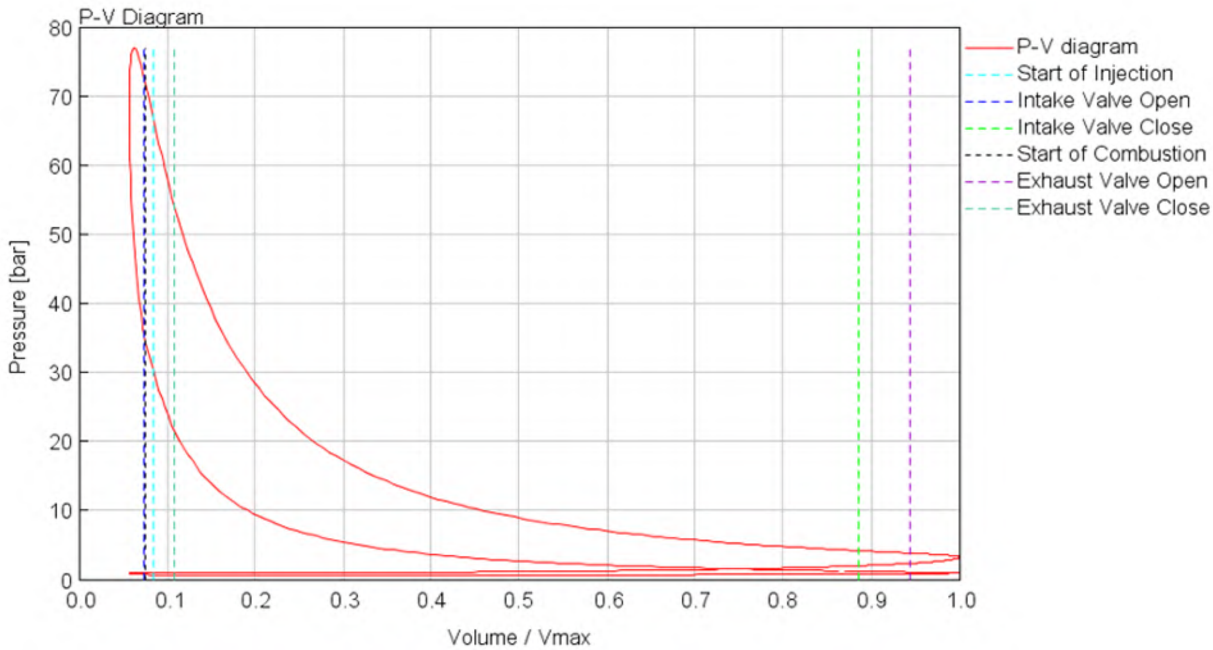


***Figure 116** PRESSURE VS VOLUME*

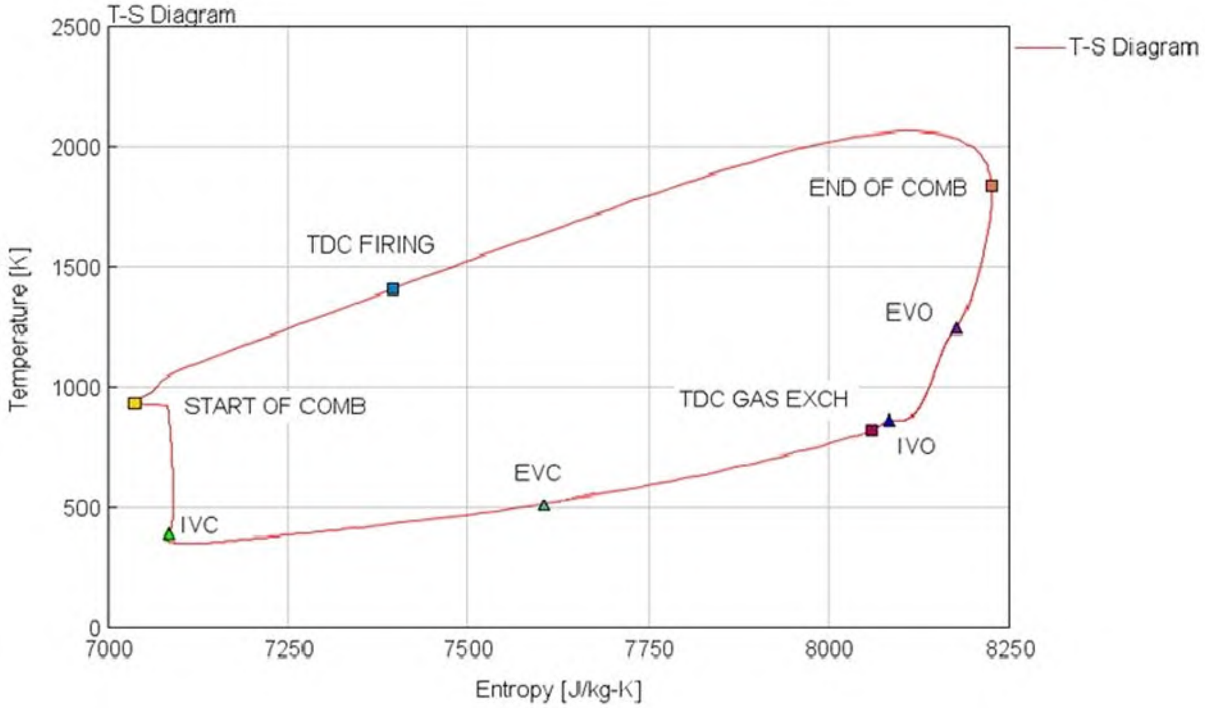


Figure 117 *TEMPERATURE VS ENTROPY*

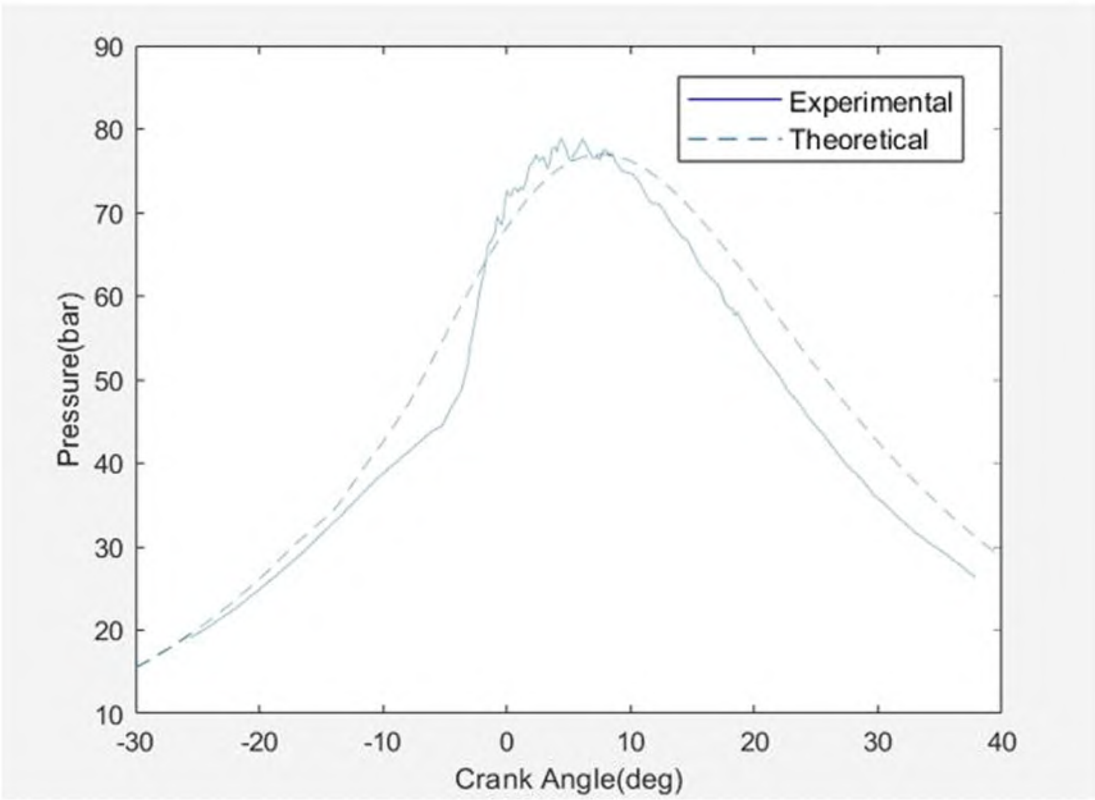


Figure 118 *Comparison Between Experimental and Simulated Pressure Values*

11.2- GT-POWER OPTIMIZATION

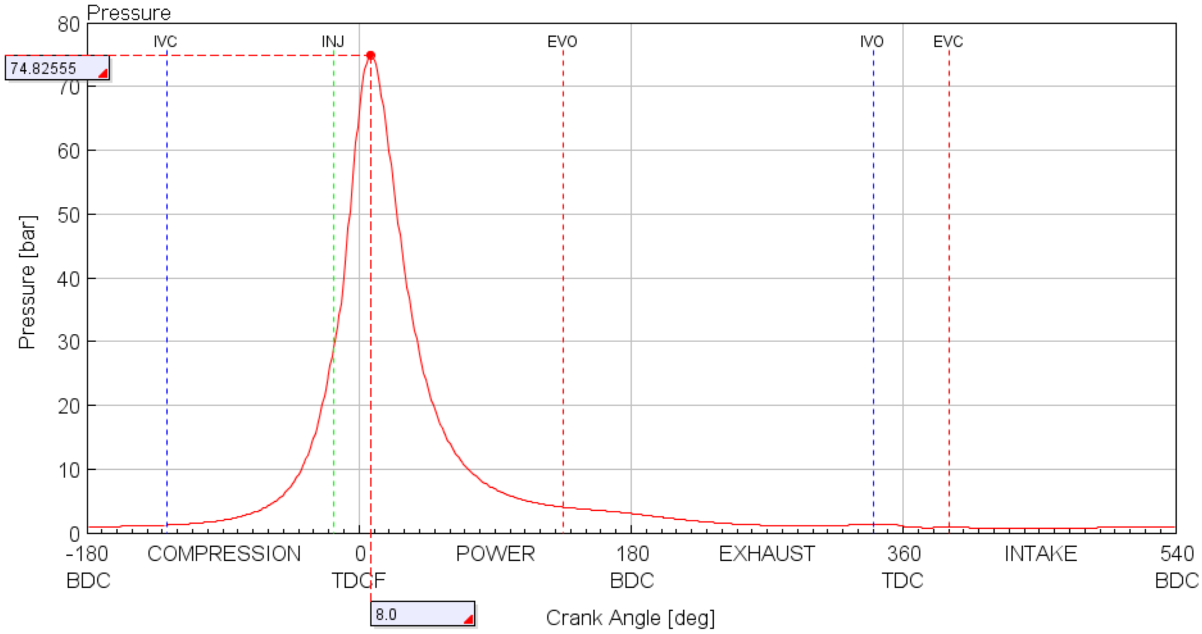


Figure 119 PRESSURE VS CRANK ANGLE

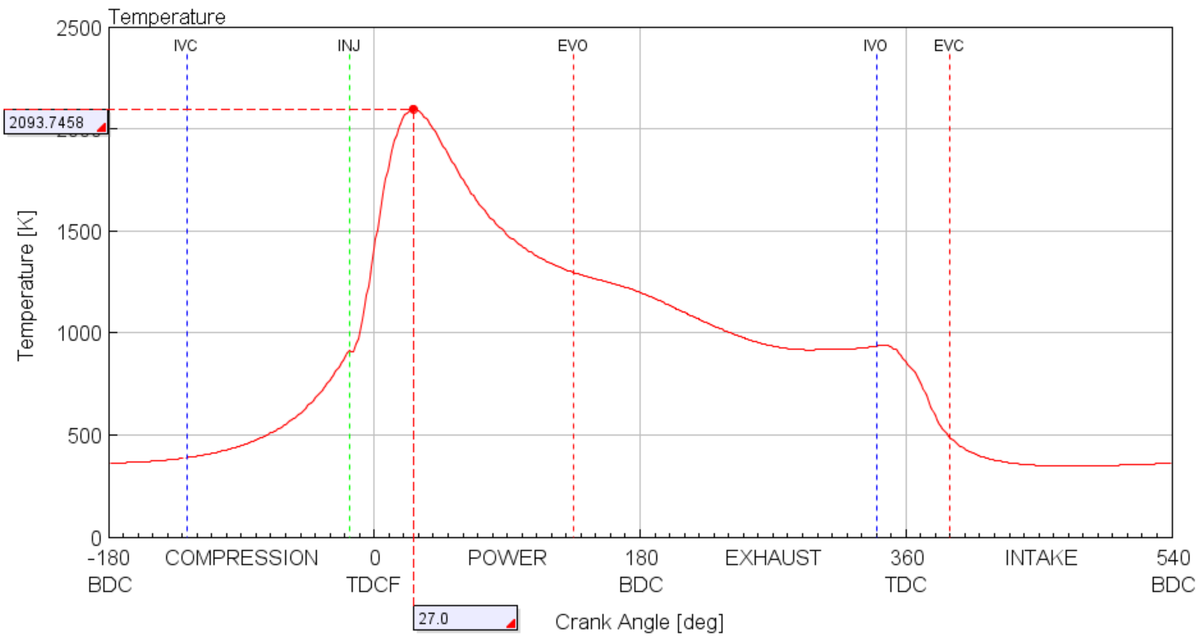


Figure 120 TEMPERATURE VS CRANK ANGLE

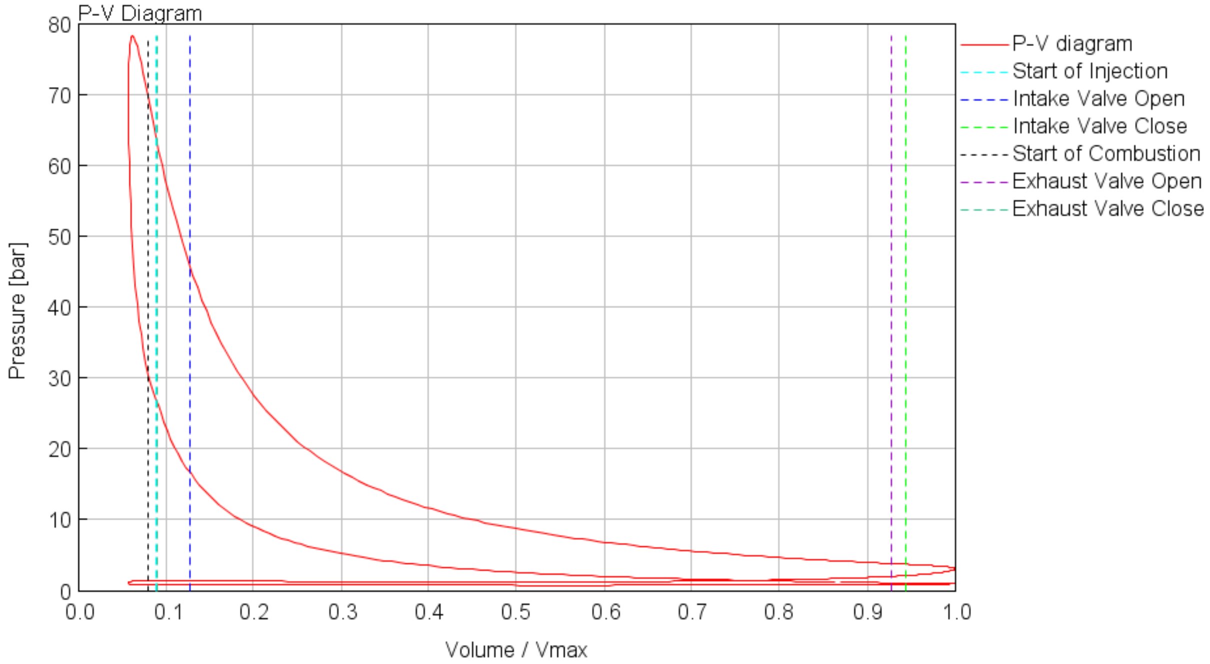


Figure 121 *PRESSURE VS VOLUME*

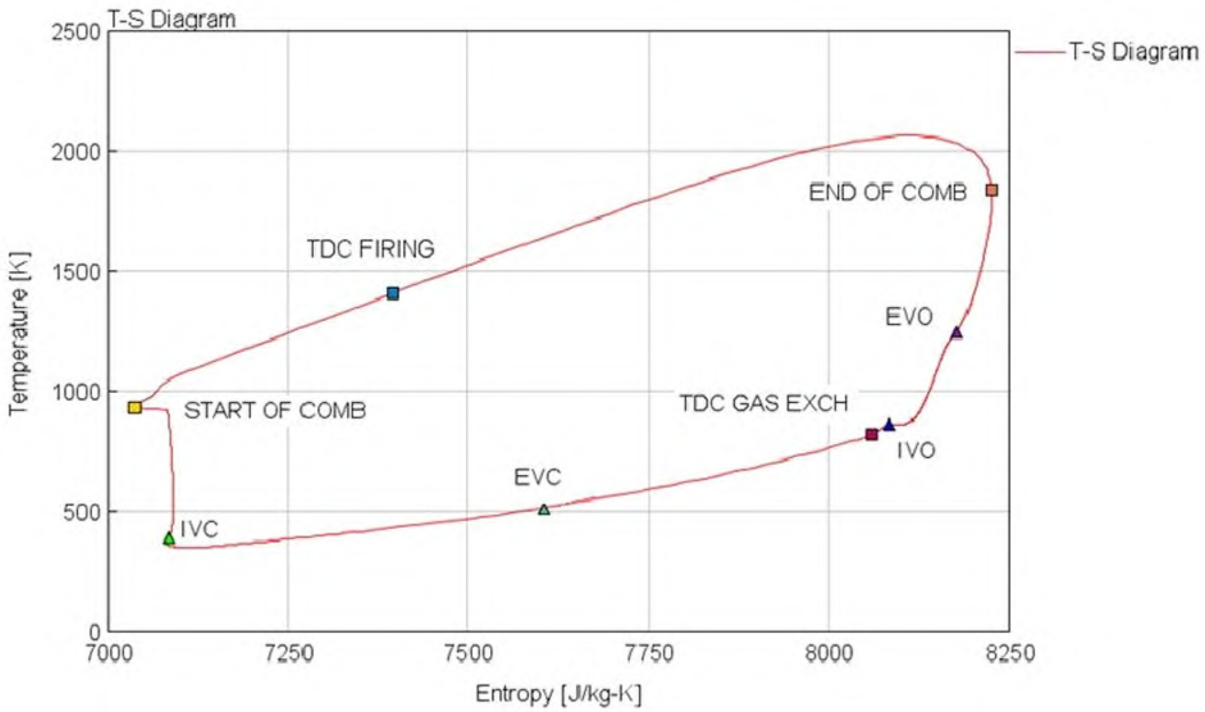


Figure 122 *TEMPERATURE VS ENTROPY*

Table 4 STANDARD VS OPTIMIZED VALUES OF PARAMATERS

Optimized Parameter	Standard Value	Optimized Value
IVO (CA)	340	325.529
EVO (CA)	135	128.666
Injection Timing (CA)	-17	-17
Injection Duration (CA)	8.9	5.457
Brake Torque (N.m)	30.322	30.63

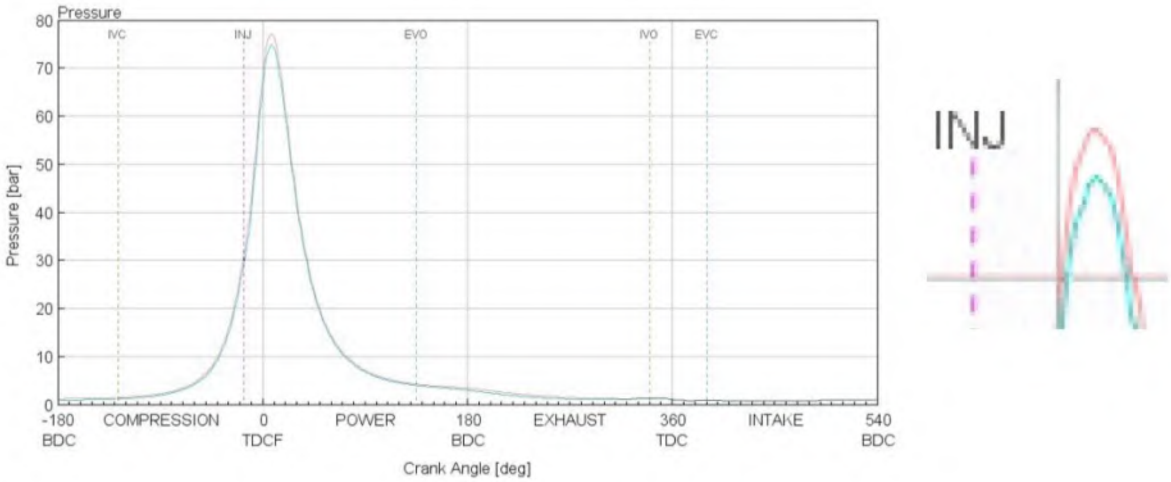


Figure 123 PRESSURE VS CRANK ANGLE COMPARISON BETWEEN OPTIMIZED AND STANDARD COMBUSTION ANALYSIS

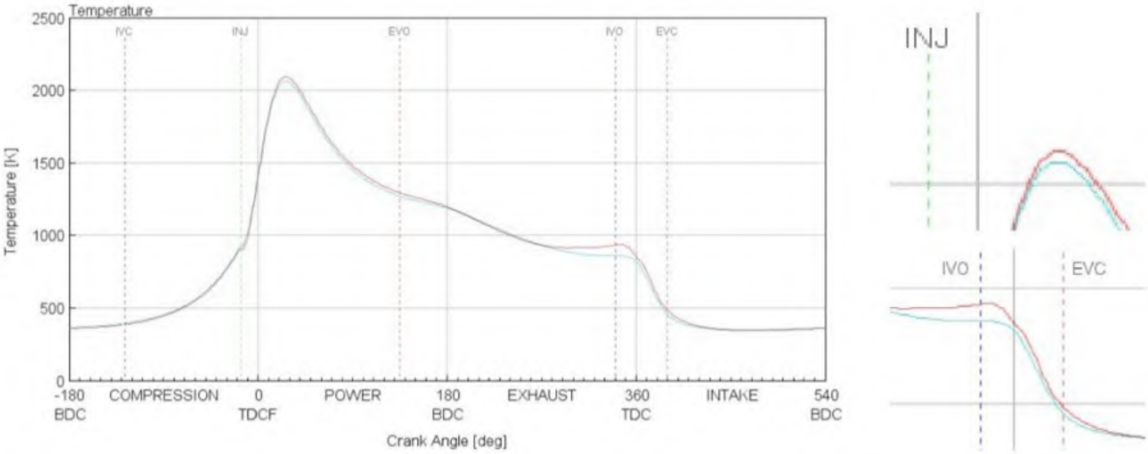


Figure 124 TEMPERATURE VS CRANK ANGLE COMPARISON BETWEEN OPTIMIZED AND STANDARD COMBUSTION ANALYSIS

11.3- 3D ANALYSIS - INTAKE PORT FLOW

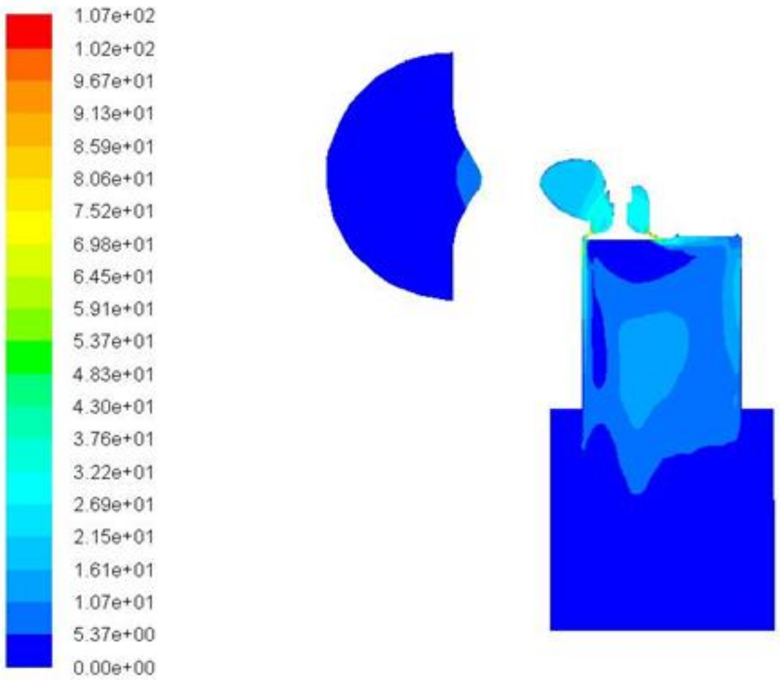


Figure 125 *Contour map of the velocity magnitude of the vertical cut-planes for 2 mm intake valve lift*

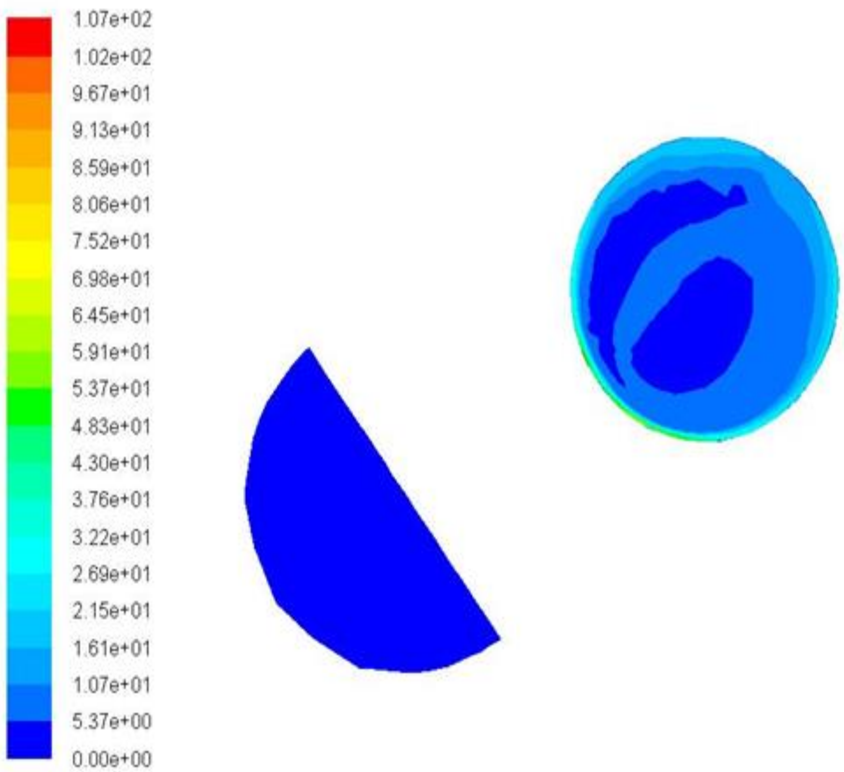


Figure 126 *Contour map of the velocity magnitude of the horizontal cut-planes for 2 mm intake valve lift*

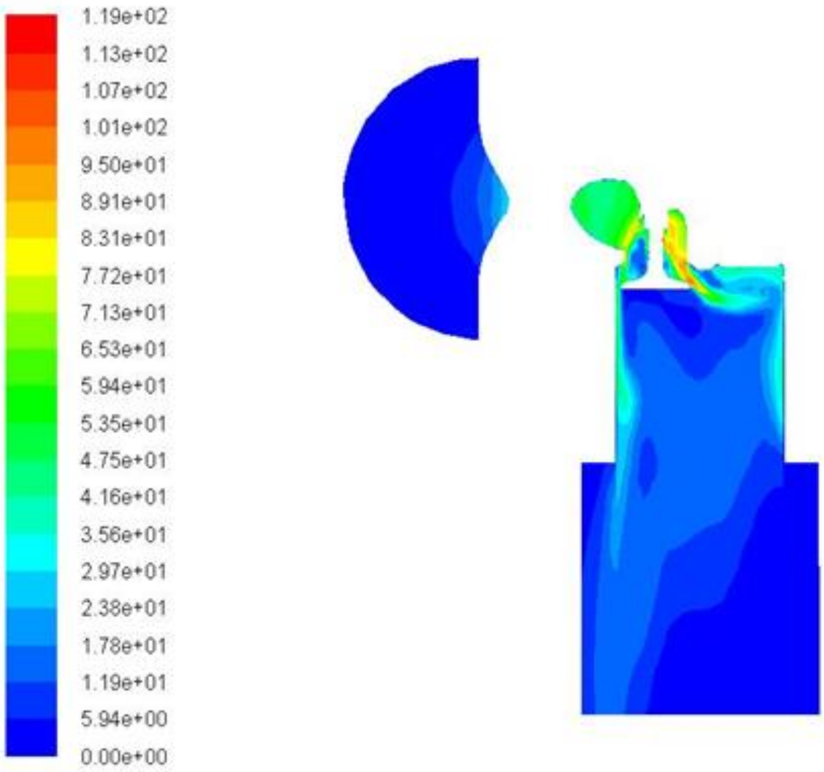


Figure 127 *Contour map of the velocity magnitude of the vertical cut-planes for 10.345 mm intake valve lift*

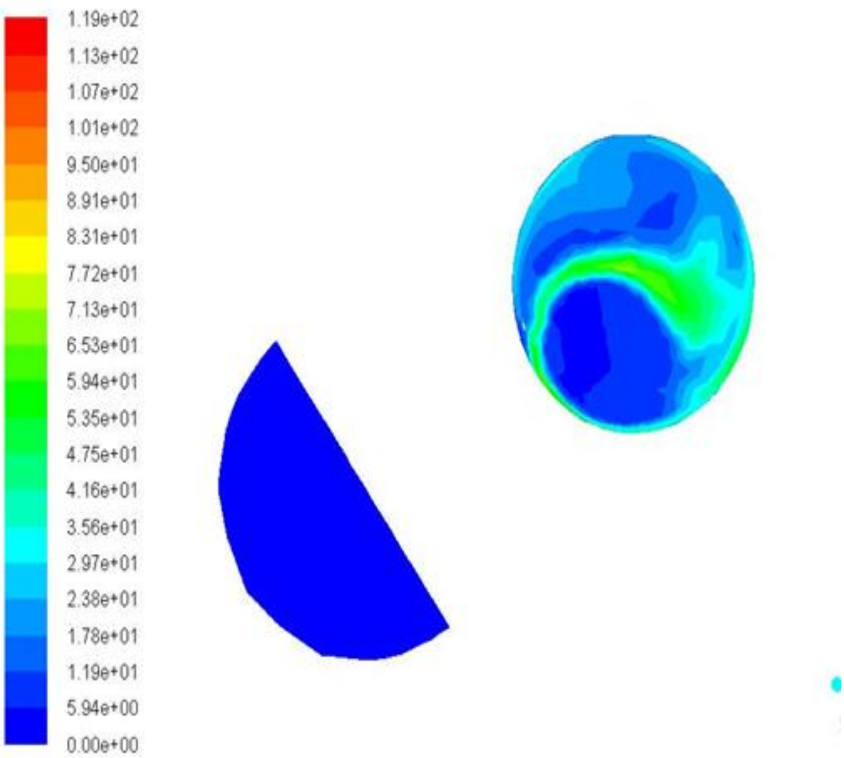


Figure 128 *Contour map of the velocity magnitude of the horizontal cut-planes for 10.345 mm intake valve lift*

11.4- 3D ANALYSIS - EXHAUST PORT FLOW

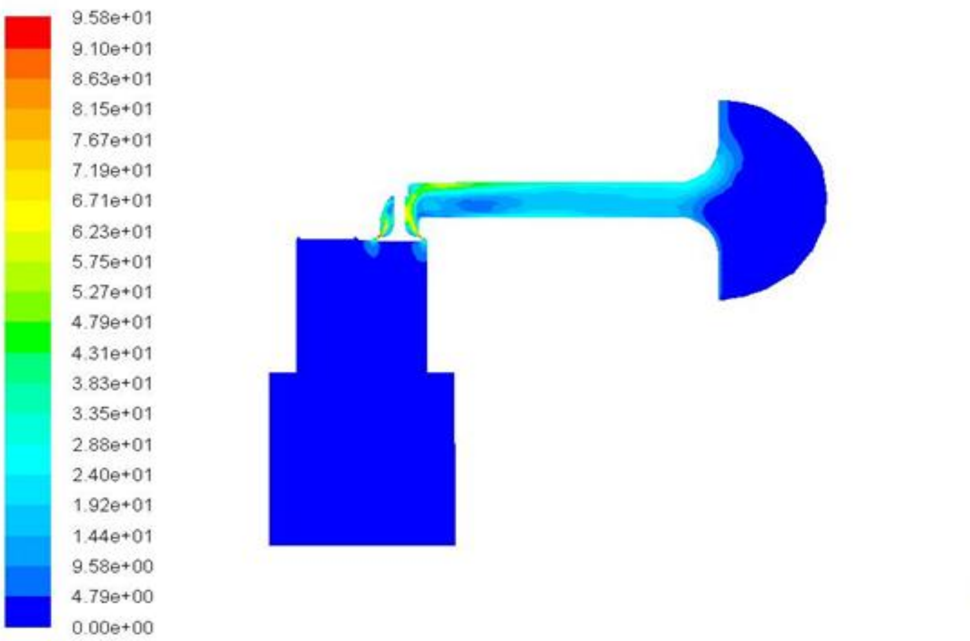


Figure 129 *Contour map of the velocity magnitude of the vertical cut-planes for 2 mm exhaust valve lift*

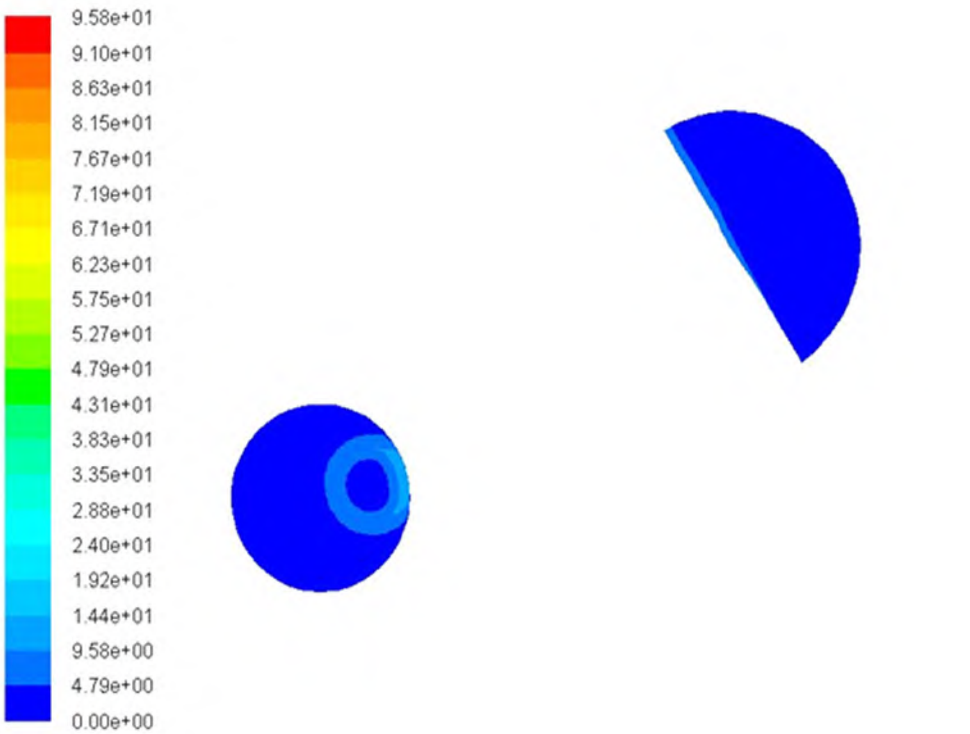


Figure 130 *Contour map of the velocity magnitude of the horizontal cut-planes for 2 mm exhaust valve lift*

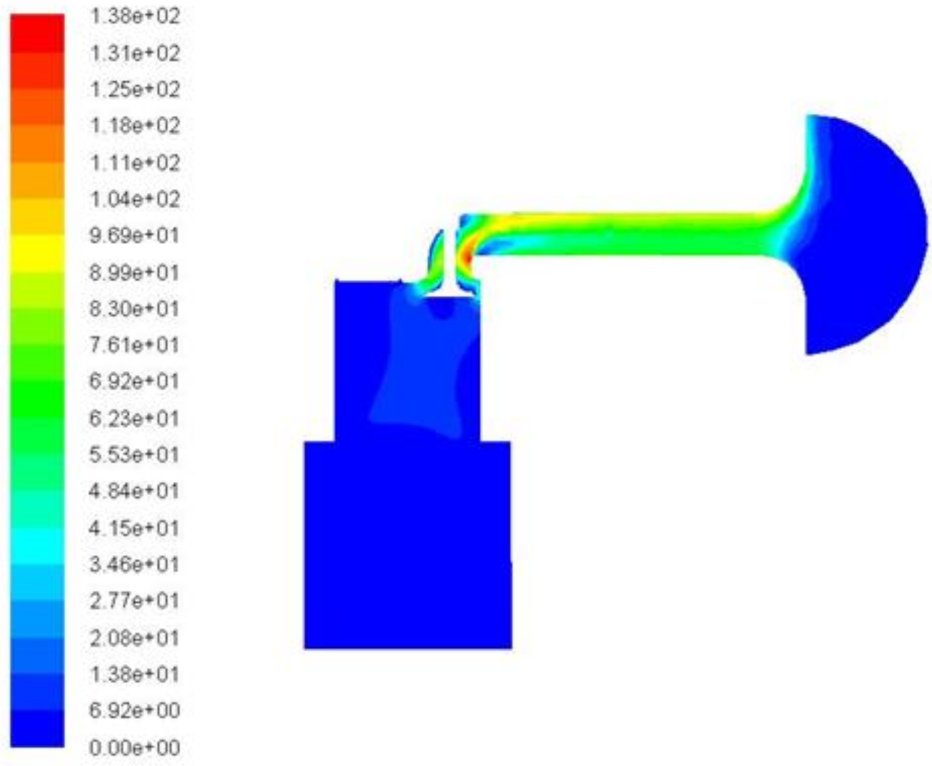


Figure 131 *Contour map of the velocity magnitude of the vertical cut-planes for 9.51 mm exhaust valve lift*

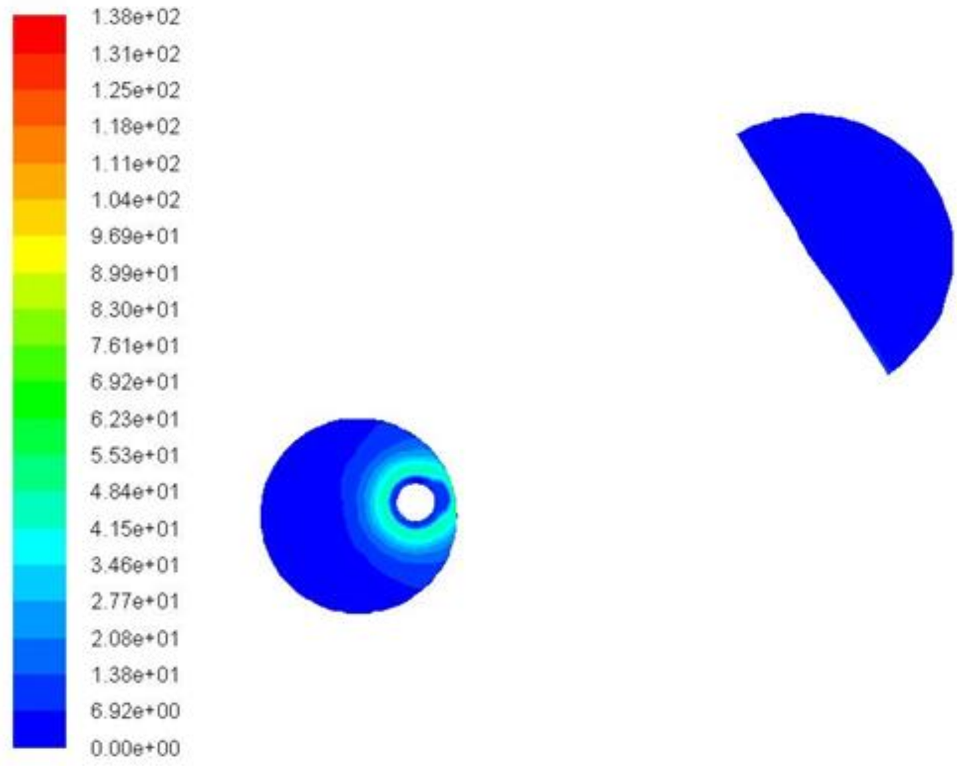


Figure 132 *Contour map of the velocity magnitude of the horizontal cut-planes for 9.51 mm exhaust valve lift*

11.5- 3D ANALYSIS - COLD FLOW RESULTS

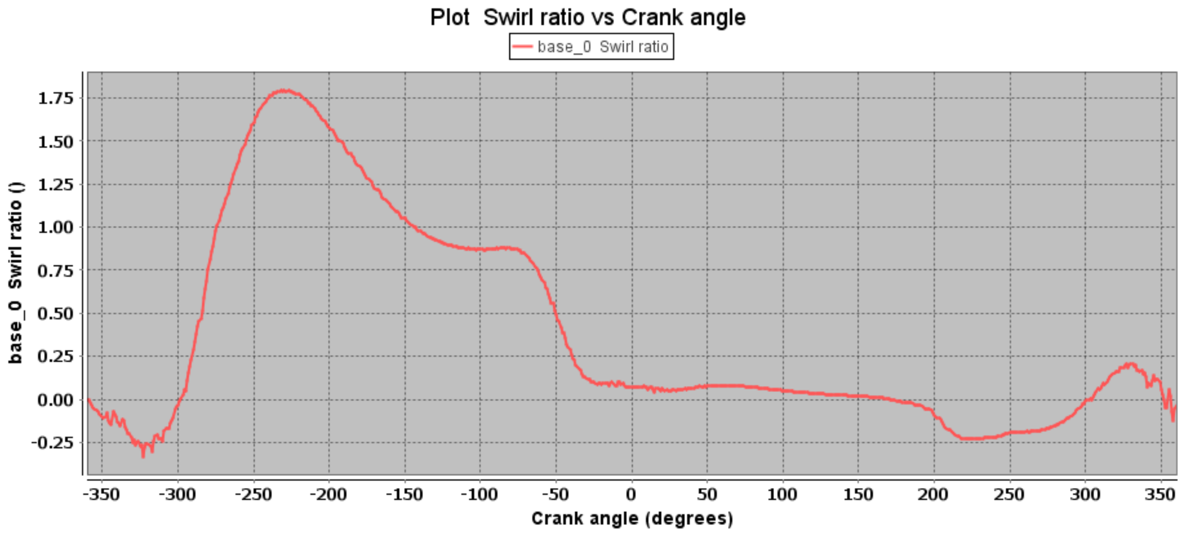


Figure 133 *SWIRL RATIO vs CRANK ANGLE*

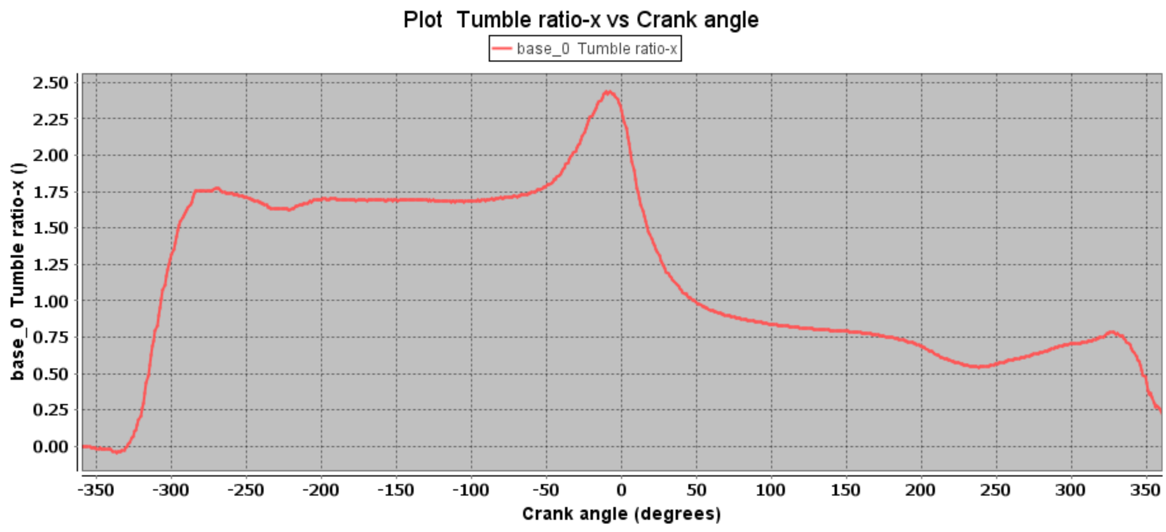


Figure 134 *TUMBLE RATIO-X vs CRANK ANGLE*

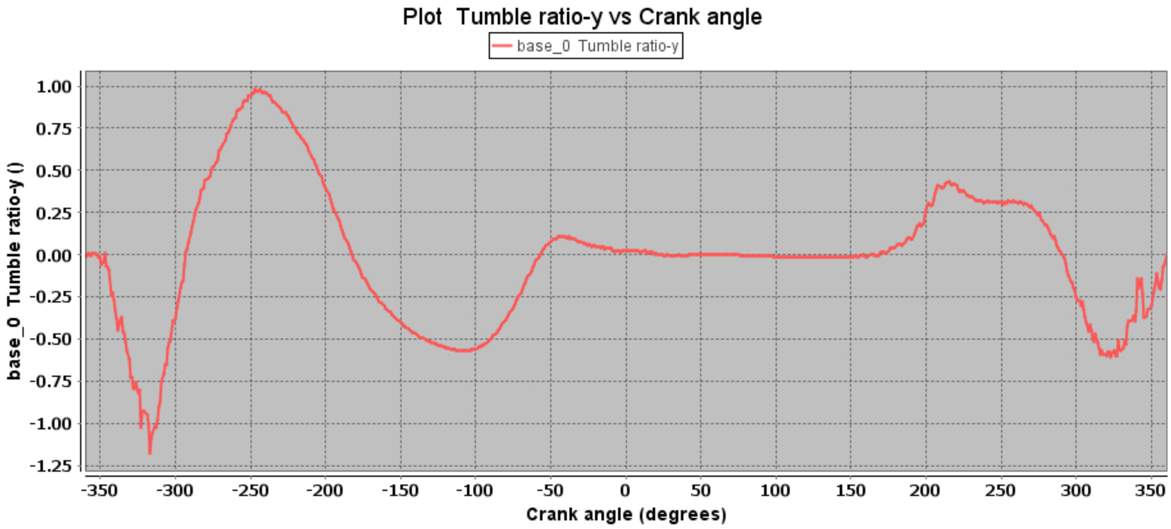


Figure 135 TUMBLE RATIO-Y vs CRANK ANGLE

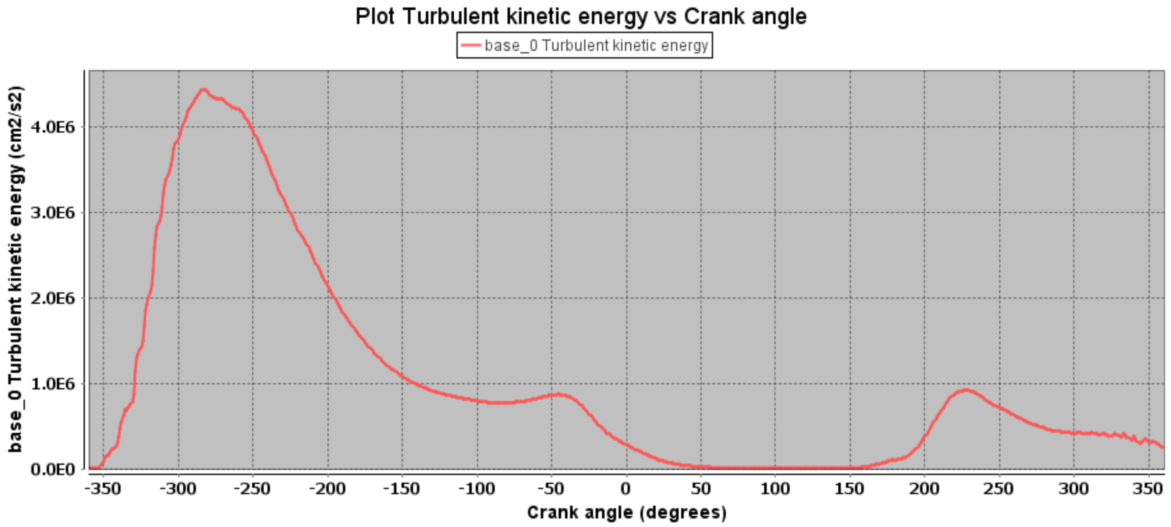


Figure 136 TURBULENT KINETIC ENERGY vs CRANK ANGLE

11.6- 3D ANALYSIS – COMBUSTION RESULTS

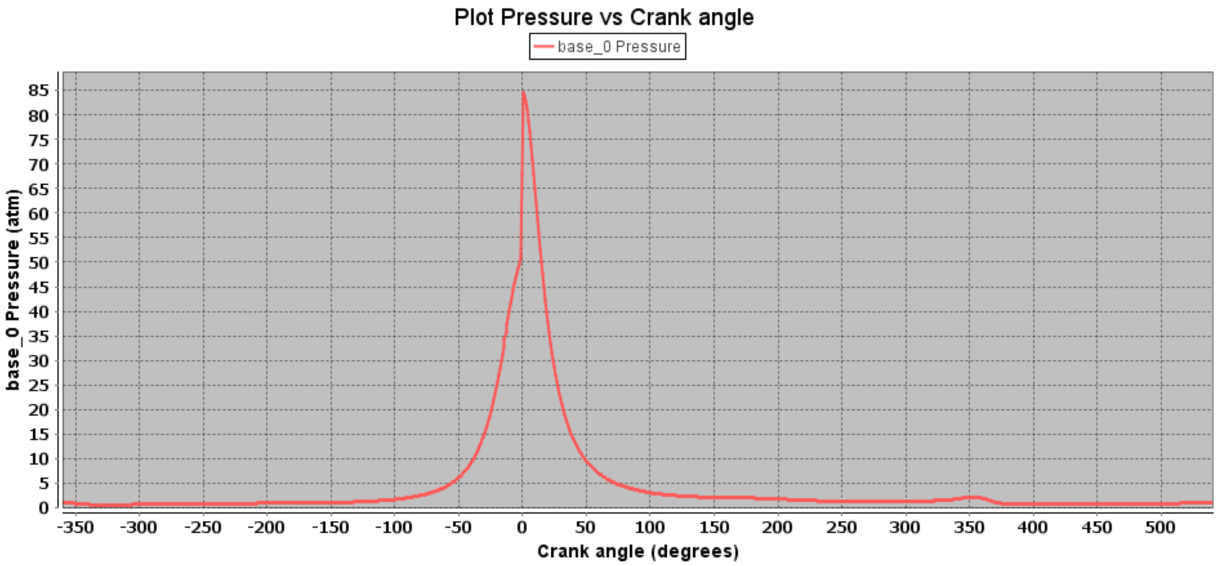


Figure 137 PRESSURE vs CRANK ANGLE

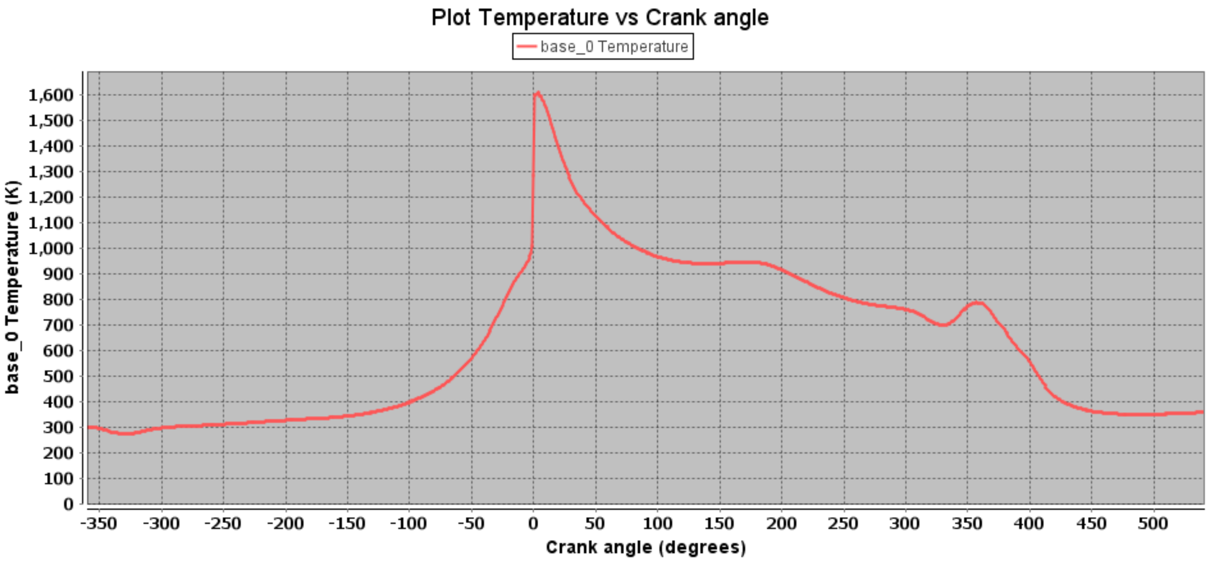


Figure 138 TEMPERATURE vs CRANK ANGLE

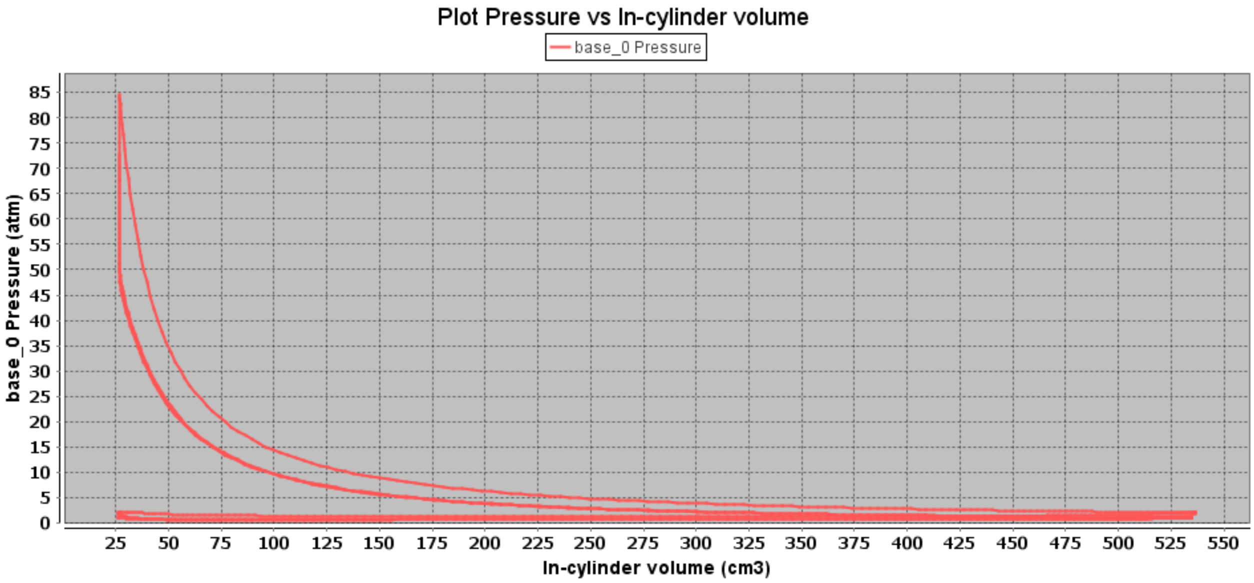


Figure 139 *PRESSURE vs VOLUME*

12- Conclusion

In this study, one-dimensional combustion and optimization of a single-cylinder engine and three-dimensional cold flow and port flow analyzes were performed. For combustion analysis, pressure crank angle and temperature crank angle plots, pressure-volume and temperature-entropy diagrams were obtained. In port flow analysis, mass flow rates for different valve lifts, velocity contour maps of the intake and exhaust ports were obtained. For cold flow analysis, swirl ratio values were obtained. As a result of optimization, the aim of increasing the maximum torque has been successfully carried out. The experimental results are consistent with the results obtained from the simulation.

13- References

13.1- Articles

- Bakar, R. and Ismail, A. (2018). Investigation of Diesel Engine Performance Based on Simulation. *Automotive Focus Group, Faculty of Mechanical Engineering,*.

- Pearson, R.J. & Bassett, Mike & P. Fleming, N & Rodemann, T. (2018). Lotus Engineering Software - An Approach to Model-Based Design.

- Chan, K., Ordys, A., Volkov, K. and Duran, O. (2013). Comparison of Engine Simulation Software for Development of Control System. *Modelling and Simulation in Engineering*, 2013, pp.1-21.

- Meyer, J. (2007). Engine Modeling of an Internal Combustion Engine With Twin Independent Cam Phasing.

- Sohail, Umer. (2016). Simulation of Four Stroke Internal Combustion Engine. International Journal of Scientific and Engineering Research. 7. 1212-1219.

- Moldovanu, D. and Burnete, N. (2013). Computational fluid dynamics simulation of a single cylinder research engine working with biodiesel. *Thermal Science*, 17(1), pp.195-203.

- Saddique, Abdul & Vijaya, K & Reddy, Kumar. (2015). EXPERIMENTAL VALIDATION AND COMBUSTION CHAMBER GEOMETRY OPTIMIZATION OF DIESEL ENGINE BY USING DIESEL-RK.

- Moldovanu, D. and Burnete, N. (2013). Computational fluid dynamics simulation of a single cylinder research engine working with biodiesel. *Thermal Science*, 17(1), pp.195-203.

- Plengsa-ard, C. and Kaewbumrung, M. (2018). CFD modelling wall heat transfer inside a combustion chamber using ANSYS forte. *IOP Conference Series: Materials Science and Engineering*, 297, p.012036.

13.2- Theses

- Johansson, E. and Wagnborg, S. (2014) *Analysis of Engine Cold Start Simulation in GT-Power*. Göteborg : Chalmers University of Technology (Diploma work - Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden, no: 2014:13).

- Venkateshmohan, V. och Kumar, M. (2015) *Predictive Diesel Combustion Using DI-Pulse in GT-Power*. Göteborg : Chalmers University of Technology (Diploma work - Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden, nr: 2015:88).

13.3- Websites

- http://www.eurekamagazine.co.uk/design-engineering-news/converge-cfd-software-enables-in-cylinder-simulations-to-run-faster-than-ever/147111

- https://convergecfd.com/benefits/third-party-integration

- http://hpcadvisorycouncil.com/pdf/CD_adapco_applications.pdf

- https://carbiketech.com/valve-timing/

- https://www.hindawi.com/journals/mse/2013/401643/

13.4- Tutorials

- Forte Tutorials, Release 19.0, January 2018, Canonsburg, PA 15317, https://www.ansys.com/

- GETTING STARTED USING LOTUS ENGINE SIMULATION, VERSION 5.05, 2001, Lotus Cars Ltd.

- GT-SUITE Engine Performance Tutorials, VERSION 2016, Gamma Technologies, www.gtisoft.com

- GT-SUITE GT-POST Tutorials, VERSION 2016, Gamma Technologies, www.gtisoft.com