MARMARA UNIVERSITY
FACULTY OF ENGINEERING

EFFECTS OF PISTON BOWL GEOMETRIES ON SPRAY AND COMBUSTION IN COMPRESSION IGNITION DIRECT-INJECTION ENGINES

MEHMET EMRE ÇETİN

GRADUATION PROJECT REPORT

Department of Mechanical Engineering

Supervisor
Assoc.Prof. Dr.

Mustafa YILMAZ

ISTANBUL, 2020

MARMARA UNIVERSITY
FACULTY OF ENGINEERING

Effects Of Piston Bowl Geometries On Spray And Combustion In Compression Ignition Direct-Injection Engines

by

Mehmet Emre ÇETİN

July 06, 2020, 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 .... Doç. Dr. Mustafa YILMAZ ....

Project Supervisor, Department of Mechanical Engineering

Accepted By ....

Head of the Department of Mechanical Engineering

ACKNOWLEDGEMENT

First of all, I would like to thank my supervisor Assoc. Prof. Dr. Mustafa YILMAZ and Dr. Ramazan ŞENER, for the valuable guidance and advice on preparing this thesis and giving me moral and material support.

January, 2020   Mehmet Emre ÇETİN

CONTENTS

ABSTRACT

Effects of Piston Bowl Geometries on Spray and Combustion in Compression Ignition Direct – Injection Engines by means of CFD

In this study, investigations were performed on a compression ignition direct-injection engine. Initially, 3 different bowl geometries (HCC,SCC and TCC) were designed in Solidworks 2015.

A computational fluid dynamics based numerical simulation was performed by Converge Engine Simulation v2.4. The engine that we have performed our investigations has a physically real Hemispherical Combustion Chamber. Simulation results for HCC were compared with the experimental data to verify the analysis. Comparison showed good agreements with the experimental results under cylinder pressure for HCC. After the verification, simulations were performed for other two geometries under the same conditions, results were obtained.

In the last section, emissions (NOx, Soot and CO), cylinder pressure and temperature values, P-V diagrams, pressure and temperature contours of these bowl geometries' simulations were compared for different crank angles.

Ensight 2019 R1 (ANSYS) was used to create animations and velocity, pressure, temperature and spray contours.

SYMBOLS

V_c **:** Clearance Volume

V_d **:** Displacement Volume

ABBREVIATIONS

HCC : Hemispherical Combustion Chamber

TCC : Toroidal Combustion Chamber

SCC : Shallow Depth Combustion Chamber

aTDC : After Top-Dead Center

CA : Crank Angle

CR : Compression Ratio

LIST OF FIGURES

LIST OF TABLES

INTRODUCTION

1.1 General Information

After periods of research and literature review, me and my supervisor Assoc. Prof. Dr. Mustafa YILMAZ have decided to study on the Effects of Piston Bowl Geometries on Spray and Combustion Efficiency in Compression Inginition Direct-Injection Engines.

In the end of my literature researches, I found [7] a publication that has Compression Inginition Direct-Injection Engine with its experimental Pressure – Crank Angle data. This gave me the opportunity to verify my simulations and analyses.

This is a CFD study and it represents the comparison of three different bowl geometries and their effects on emissions, spray and combustion.

1.2 Literature Review

Helgi Fridriksson, Bengt Sund´en, Shahrokh Hajireza, Martin Tun´er et al. [1] investigated a diesel engine's response to different heat fluxes with usual diesel combustion mode and partially premixed combustion (PPC) mode. They have used AVL FIRE CFD code and modeled the closed volume cycle, verified their simulation by comparing the results of simulation with experimental data. Lastly, they have compared emission levels (NOx and soot) of these two modes. They have observed increased efficiency in both cases but only PCC mode managed it without any increase in NOx emissions.

C.S. Sharma, T.N.C. Anand and R.V. Ravikrishna et al. [2] have taken in-cylinder process and combustion in a naturally aspirated direct injection diesel engine into consideration. For simulation of suction stroke they have used AVL FIRE code and for closed valve part they have used KIVA-3V code. They have generated an algorithm to create a CFD solution from AVL FIRE to KIVA-3V to provide initial conditions and used validated models which are Characteristic-time model and Shell hydrocarbon auto-ignition model. Used code was integrated and optimized with respect to the experimentally observed pressure data. Finally, they have used the same tools to find multiple injection strategies for the engine used.

Eivaz Akbarian, Bahman Najafi, Mohsen Jafari, Sina Faizollahzadeh Ardabili, Shahaboddin Shamshirband and Kwok-wing Chaue et al. [3] has done their simulation with KIVA3V for a dual-fueled engine and verified their simulations with experimental data. Firstly, they have investigated emissions and performance values of a diesel engine and then to dual-fuel engine. They have done some experimental tests under different fuel loads and pilot to gaseous fuel (PGF)

ratios. They have observed a better NOx and particle materials ratios in dual mode compared to diesel engine. There was a slight decrease in CO_2. There was also reversed effects in these values with respect to the change of load.

Henrik W. R. Dembinski et al. [4] examined the relation between changes in in-cylinder flow and emissions by combustion image velocimetry (CIV) and compared the results to Reynolds-averaged Navier-Stokes (RANS) simulations.

M. Zafer GUL, M. YILMAZ, M. YILDIRIM, S. NAS et al [5] wanted to make suggestions to improve in-cylinder design of a nine-liter diesel engine. So, they analyzed flow field and combustion and made cold flow simulations of the specified engine. To create a surface mesh, they have used a solid CAD model. After that they have created a volume mesh. Commercial software STAR-CD was used for piston and valve motions. Fuel injection was simulated in 3D. Turbulence, air motion with sprayed fuel was in the scope of this study. They aimed low emissions that is caused by high turbulence level.

K. Abay, U. Colak, L. Yüksek et al. [6] has examined flow and combustion by means of CFD. Pressure and temperature values that are crank angle dependent were used as boundary conditions for 3D CFD simulations. Sector mesh which will be taken a look at was used for the closed-valves period of the cycle. For 3D CFD simulations and grid generation, they have used Forte Reaction Design and Chemkin. Simulation results which are pressure and heat release were validated with experimental data. They have lastly investigated NOx and Soot.

Prahhakara Rao Ganji, Rudra Nath Singh, V R K Raju and S Srinivasa Rao et al. [7] have dealth with design of bowl geometries and combustion efficiency in direct-injection compression ignition engines by means of CFD as I did in this study. They firstly, designed 3 different bowl geometries(Hemispherical Combustion Chamber, Shallow Depth Combustion Chamber and Toroidal Combustion Chamber) and then examined their combustion modelling in Converge Engine Simulation and used SAGE combustion model. Their comparison includes emissions (NOx, Soot, HC and CO), pressure. After the comparison of these species, they got into detail of their Toroidal Combustion Chamber design. They made the same comparisons for the same bowl type but with different depth of bowls. After all, they also compared temperature, ISFC, equivalence ratio and swirl ratio. Finally, they found which bowl design was suitable for the emission regulations. This publication was one of my main references in this study.

S Khan et al. [8] have investigated the effects of both bowl geometry and spray pattern on combustion, mixing and emissions in a single cylinder direct injection diesel engine by means of CFD and also they have verified their results with their experimental values. They have used AVL FIRE code as 3D computational fluid dynamics software. They have used ECFM 3Z combustion model and WAWE breakup fuel spray model. Bowl geometries are named as hemispherical

combustion chamber (HCC), toroidal combustion chamber (TCC) and toroidal re-entrant combustion chamber (TRCC). In all of the study, compression ratios were kept constant but there were four different spray angles. Results showed that TRCC has higher cylinder pressure and heat release rate compared to other geometries because re-entrant bowl geometry was like a barrier to decrease the mass transfer from bowl to squish part. It means it has a stronger squish, turbulence and more homogenous fuel air mixture which make the combustion better.
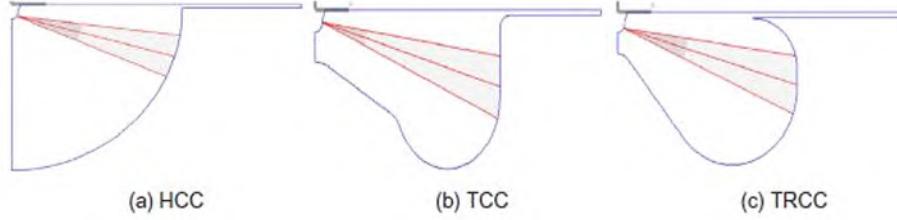
Figure 1: HCC, TCC and TRCC bowl geometries

K.L. Tay et al. [9] studied on different piston bowl geometries and ramp injection rate in a kerosene-diesel fueled direct injection compression ignition engines. Bowl geometries are omega (OCC), shallow-depth (SCC) and shallow-depth re-entrant (SRCC). Turbulent kinetic energy was highest in SRCC and this means they had 2 peak pressure points. In SCC they had less turbulence compared to SRCC. When they had used kerosene instead of diesel, they observed higher heat release in premixed combustion. If the viscosity of the fuel is lower, it gives lower CO and more NOx. These changes also affected power.
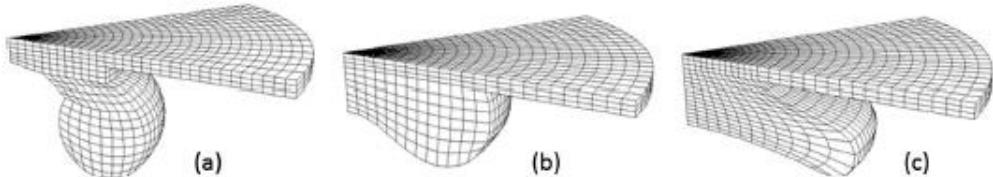
Figure 2 : OCC, SCC and SRCC bowl geometries

De Risi, D. F. Manieri and D. Laforgia et al. [10] have investigated bowl geometry and engine speed effects on temperature and emissions. They have used KIVA-3V as their CFD code. They also verified their simulations with experiments. Five different combustion chamber geometries were used. Initially they had Mexican-hat piston bowl, for the other four geometry, they have edited the initial one. For NOx, they have selected Zeldovich model and they have used Hiroyashu's model for soot.

In this paper I have also learned the difference between the modified RNG $\kappa-\varepsilon$ turbulence model and the normal one. In the modified one, there is an additional term in the dissipation equation.

Engine performance had almost 15% difference at 1690 RPM and there was almost no difference at 3500 RPM. And also, at lower speed, soot production is more dependent on bowl geometry.

A.-H. Kakaee et al. [11] investigated bowl geometry, piston bowl depth and engine speed effects on performance and emissions on a natural gas / diesel RCCI (reactivity controlled compression ignition) engine.I added this publication because it deals with piston bowl geometries and also they have used Converge CFD tool as I did in this study. Compression ratio was constant under every case. Used bowl geometries are cylindrical, bathtub and stock. Results showed that at higher speeds there are much more effects of bowl geometries compared to lower speeds. Piston bowl depth affected CO emissions and bathtub geometry gives the best emission values besides performance.
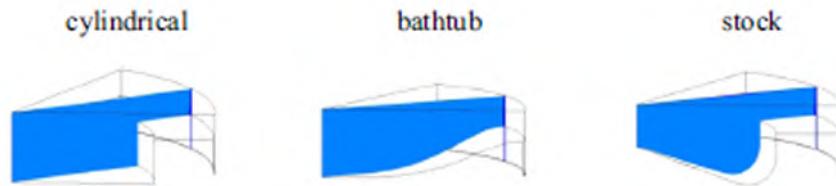
Figure 3 : Cylindrical, bathtub and stock bowl geometries

There are many more publications that I have read but I wanted to add the publications above intentionally. At first, while studying on literature, I focused on CFD studies on diesel engines. After I get used to the concept of diesel engines and CFD logic, I focused on piston bowl geometries and their effects on combustion, spray, emissions in detail.

In the next page, we are going to see our engine specifications.

1.3 Engine Specifications

Bore (mm)	87.5
Stroke (mm)	110
Connecting rod length (mm)	243
Maximum injection pressure (bar)	280
Speed (R.P.M)	1500
Rated power (kW)	3.5
Injected mass of fuel (kg/cycle)	2.57778e−05
Compression ratio	17.5
Start of injection/ (aTDC)	−23°
Injection duration (CAD)	22°
Load %	100
Inlet valve timings	IVO − 184.5°aTDC IVC − 144.5°aTDC
Exhaust valve timings	EVO 144.5°aTDC EVC 184.5°aTDC
Piston temperature (K)	523
Cylinder temperature (K)	443
Head temperature (K)	523
Fuel injection system	Direct injection by multi-hole nozzle
No of nozzle holes	3
Nozzle hole diameter	0.255 mm
Injection pressure (Max)	280 bar

Figure 4 : Engine Specifications

DESIGN PROCESS
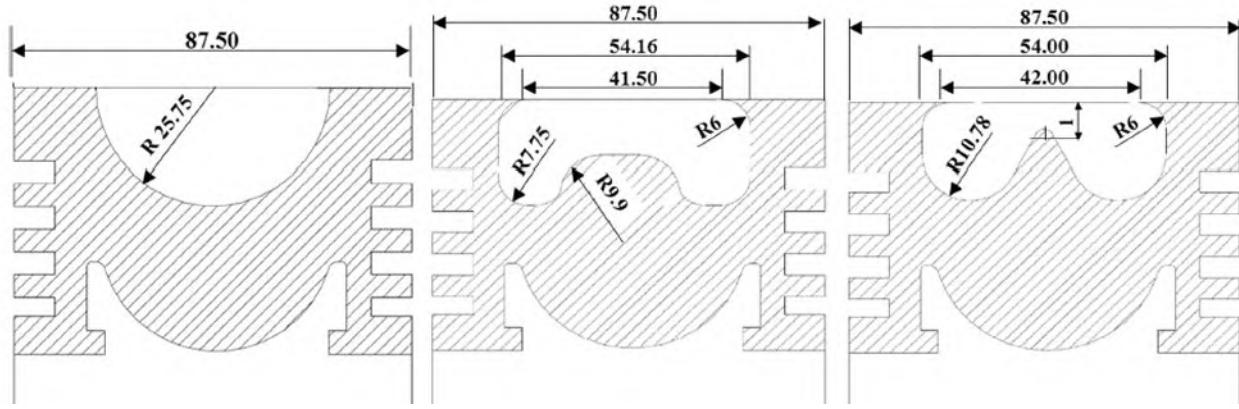
Figure 5 : Technical drawing of bowl geometries

As we can see above, we have 3 bowl geometries and their technical drawings; HCC, SCC and TCC, respectively. I am going to give details about my drawings in Solidworks.

The engine that I found has a physically real Hemispherical Combustion Chamber and it's experimental data. Firstly, I analyzed HCC and compared it with the experimental data to verify my analysis. After the verification, I analyzed other geometries, obtained the results.

Before drawing these geometries I also had to calculate their volumes to get a compression ratio that is equal to 17.5 . Because it was indicated that we have our compression ratio as 17.5 in the engine specifications.

Also I had some problems related to my drawings, required volumes of the geometries etc. and every time we had a solution for these problems. In the next parts, I will comment on them one by one.

2.1 Drawing and preparing HCC geometry for Analysis
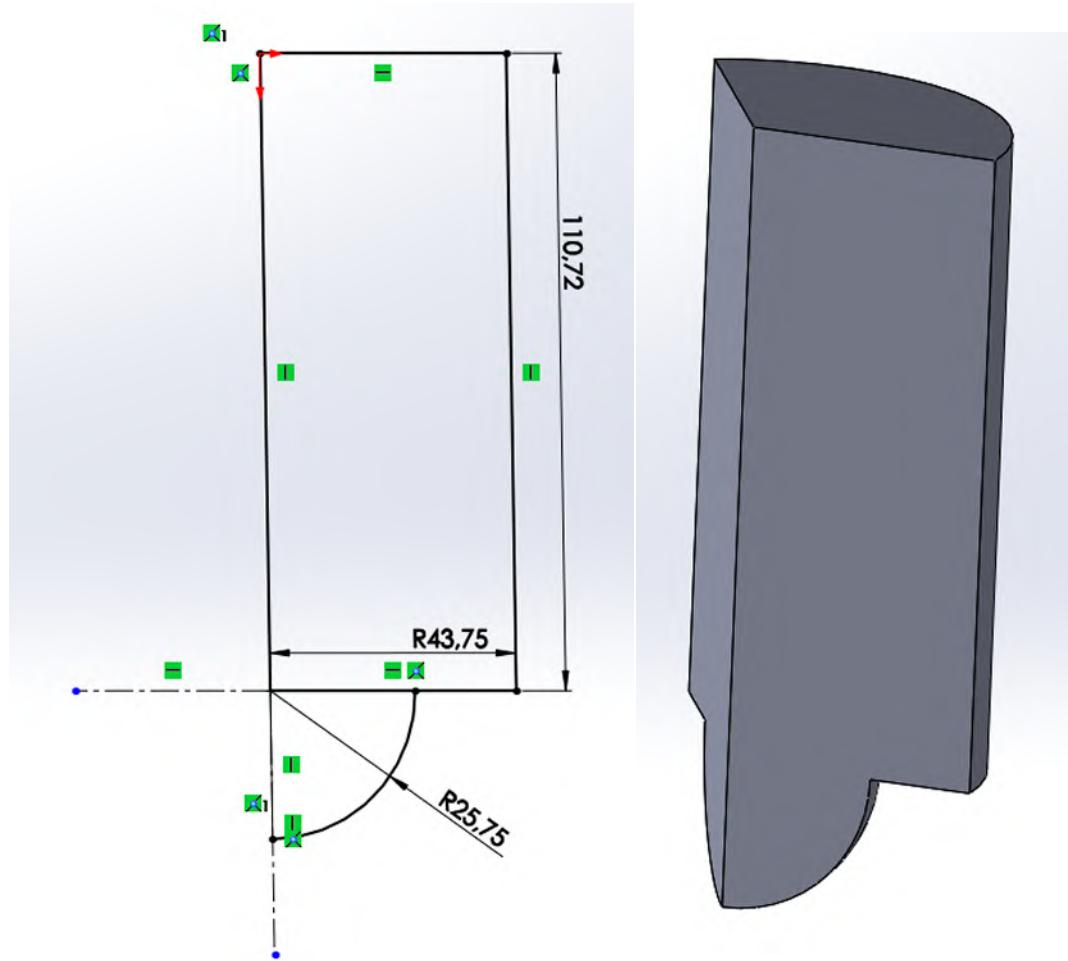
Figure 6 : HCC geometry

As indicated in engine specifications, we have

- Bore   = 110 mm
- Stroke = 87.5 mm
- Radius of hemispherical bowl = 25.75 mm

As you can see above, the height of the cylinder is 110.72 mm which means that **0.72 mm part of the cylinder belongs to clearance volume (Vc).** This means when the piston is at TDC, there is still a gap between the cylinder head and piston and that is equal to 0.72 mm.

After I drawed my geometry, I measured the clearance volume and displacement volumes from Solidworks' measure tool and also verified these values with hand calculations as

Vc = 40088 mm^3

Vd = 661453 mm^3

And we know compression ratio is equal to $\dfrac{Vc+Vd}{Vc}$. So,

C.R = $\dfrac{40088+661453}{40088}$ = 17.50

After the compression ratio is verified, I had to open my drawing in ANSYS Space Claim to convert it to an .STL file from .SLDPART . Because, if you open your .SLDPART file directly in Converge Engine Simulation, you may get an error during the anaylsis. That is why it is necessary to do as mentioned.

## 2.2 Drawing and preparing SCC geometry for Analysis
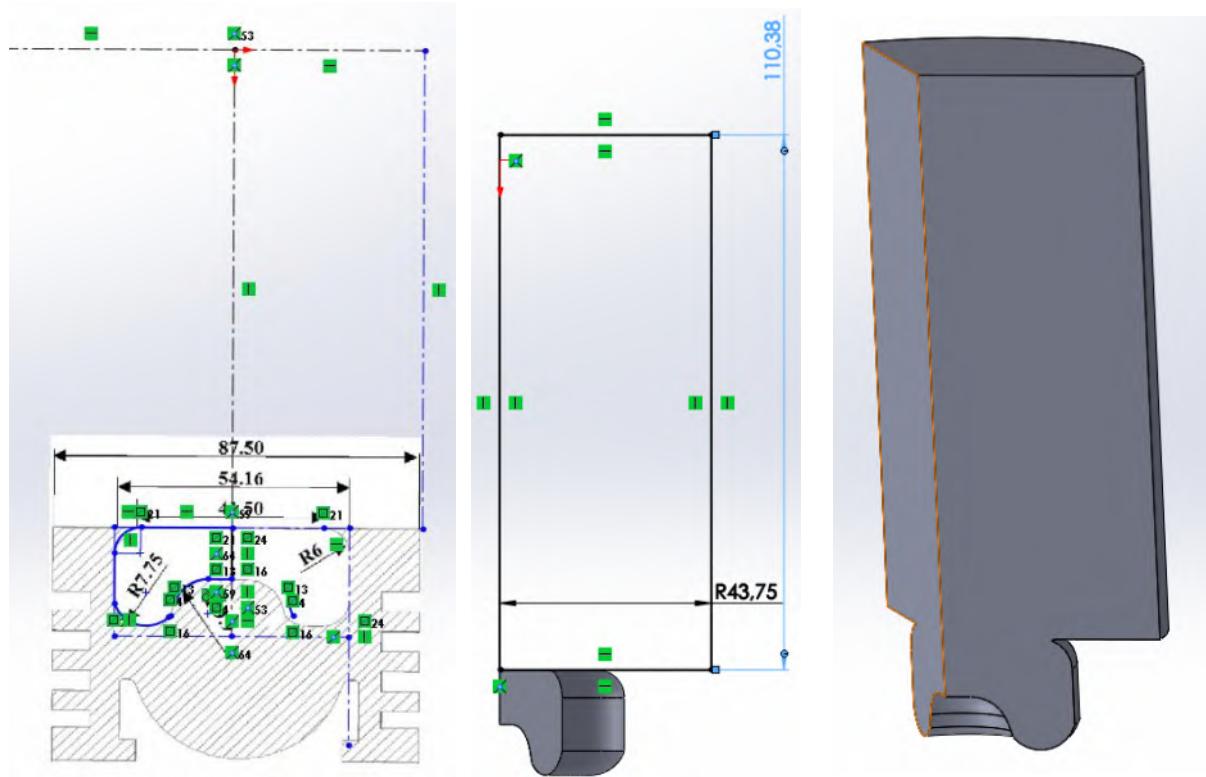
Figure 7 : SCC geometry

While drawing, to get an accurate result, I placed the technical drawing of my geometry behind top plane and checked if my drawing is accurate.

We already calculated the necessary values for Vc and Vd. So, I just needed to check if the drawing give the same values. There was a little difference between the correct value of clearance volume and my drawing's clearance volume. Therefore, I used scale command in Solidworks for bowl geometry. The true **scale factor for this geometry was 0.956**.

As you can see above, the height of the cylinder is 110.38 mm which means that **0.38 mm part of the cylinder belongs to clearance volume (Vc).** This means when the piston is at TDC, there is still a gap between the cylinder head and piston and that is equal to 0.38 mm.

After I drawed my geometry, I measured the clearance volume and displacement volume from Solidworks' measure tool and also verified these values with hand calculations as

Vc = 40090 mm^3

Vd = 661453 mm^3

And we know compression ratio is equal to $\dfrac{Vc+Vd}{Vc}$. So,

$$C.R = \dfrac{40090+661453}{40088} = 17.49$$

which is close enough to get accurate analysis results.

After the compression ratio is verified, I again had to open my drawing in ANSYS Space Claim to convert it to an .STL file from .SLDPART not to get an error during the analysis.

2.2.1   Problems encountered with SCC

Figure below is a sectional view of SCC geometry from Converge Engine Simulation. We can see the x, y and z axes. Z axis looks upwards and x axis lies in the midplane of the sector mesh of our piston.

Not to have any problem to start anaylsis, the directions of the axes should be the same as indicated and also origin must be on top, adjacent to the upper part of the piston and also should be at the center. If not, we can not start our analysis because Converge will give errors related to negative volume and etc.

At first, when I used scale command in Solidworks, the origin relocated itself. And when I upload my geometry into Converge, it gave a negative volume error. I made some research and found how to relocate the origin. I thought I solved the problem but there was still an error. So, I discussed the problem with Dr. Ramazan ŞENER and we decided to attain a new coordinate system to our drawing. Finally, second method was successful and I continued my analysis without any problem.
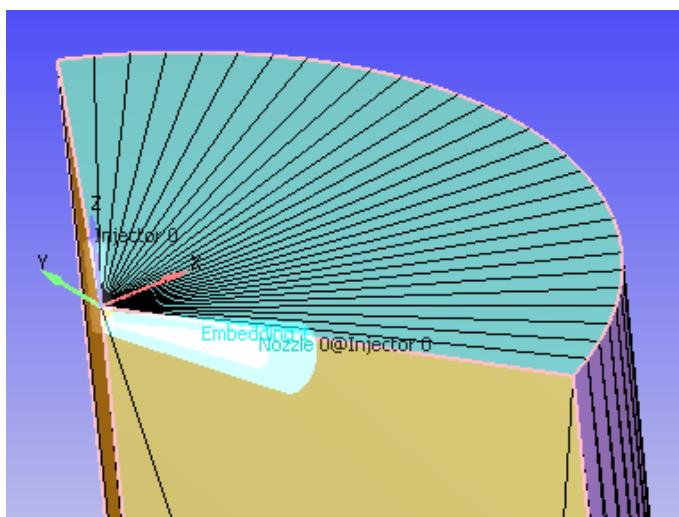
Figure 8 : Coordinates

## 2.3 Drawing and preparing TCC geometry for Analysis
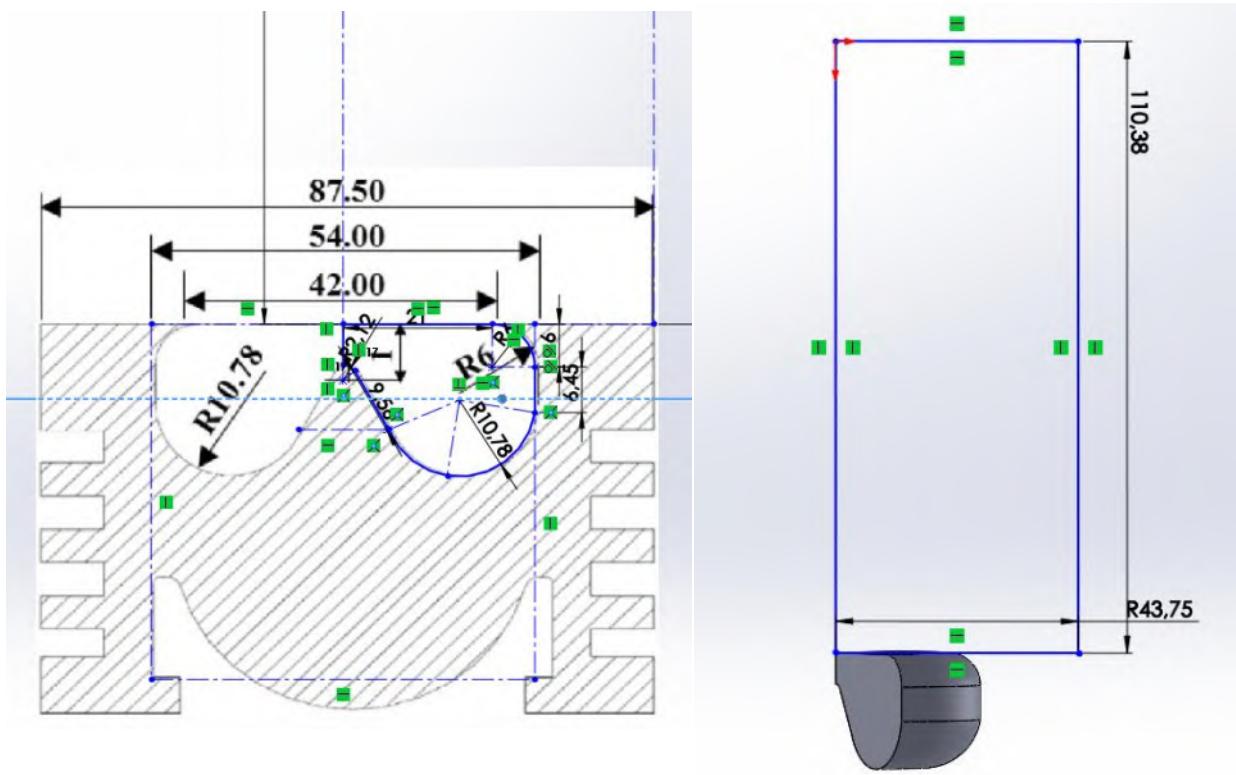
Figure 9 : TCC geometry

While drawing, to get an accurate result, again I placed the technical drawing of my TCC geometry behind top plane and checked if my drawing is accurate.

We already calculated the necessary values for Vc and Vd. So, I just needed to check if the drawing give the same values. There was a little difference between the correct value of clearance volume and my drawing's clearance volume. Therefore, I used scale command in Solidworks for bowl geometry. The true **scale factor for this geometry was 0.9655**.

As you can see above, the height of the cylinder is 110.38 mm which means that **0.38 mm part of the cylinder belongs to clearance volume (Vc).** This means when the piston is at TDC, there is still a gap between the cylinder head and piston and that is equal to 0.38 mm.

After I drawed my geometry, I measured the clearance volume and displacement volume from Solidworks' measure tool and also verified these values with hand calculations as

Vc = 40092 mm^3

Vd = 661453 mm^3

And we know compression ratio is equal to $\frac{Vc+Vd}{Vc}$. So,

$C.R = \frac{40092+661453}{40092} = 17.499$

which is close enough to get accurate analysis results.

After the compression ratio is verified, I again had to open my drawing in ANSYS Space Claim to convert it to an .STL file from .SLDPART not to get an error during the analysis.
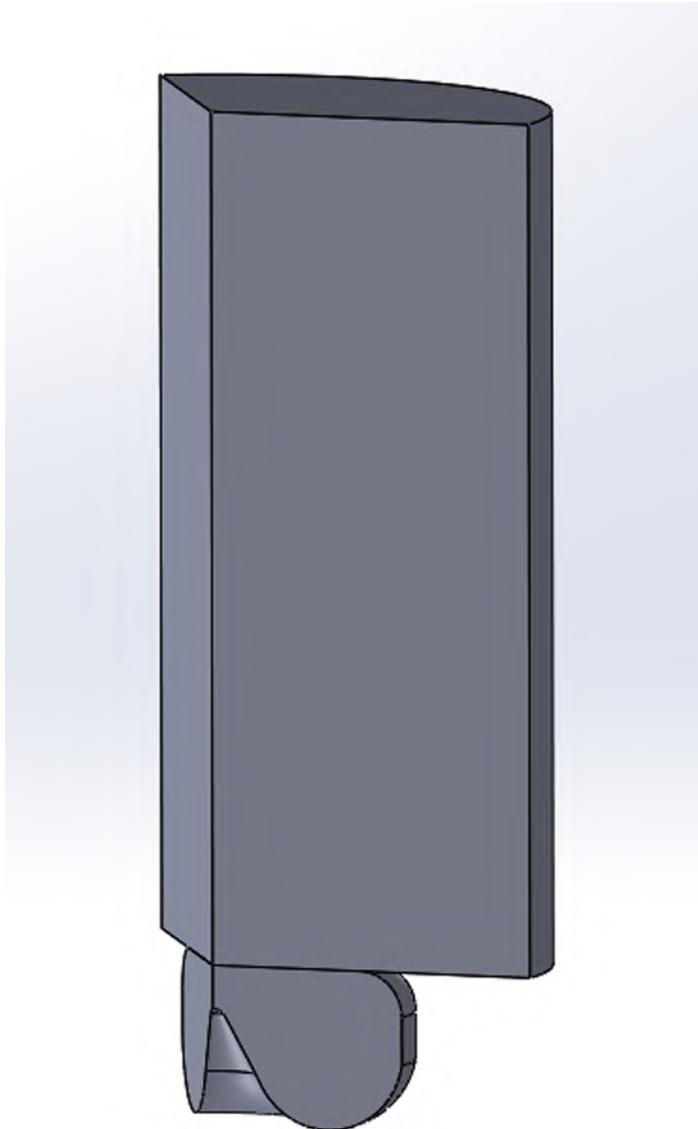
Figure 10 : TCC geometry 2

2.3.1 Problems encountered with TCC

I already explained the situation about x,y and z axis and also origin's location.Figure on your right is a sectional view of HCC geometry from Converge Engine Simulation.

At first, when I used scale command in Solidworks, the origion relocated itself again as it did in SCC. There was a gap between cylinder volume and bowl as shown below.
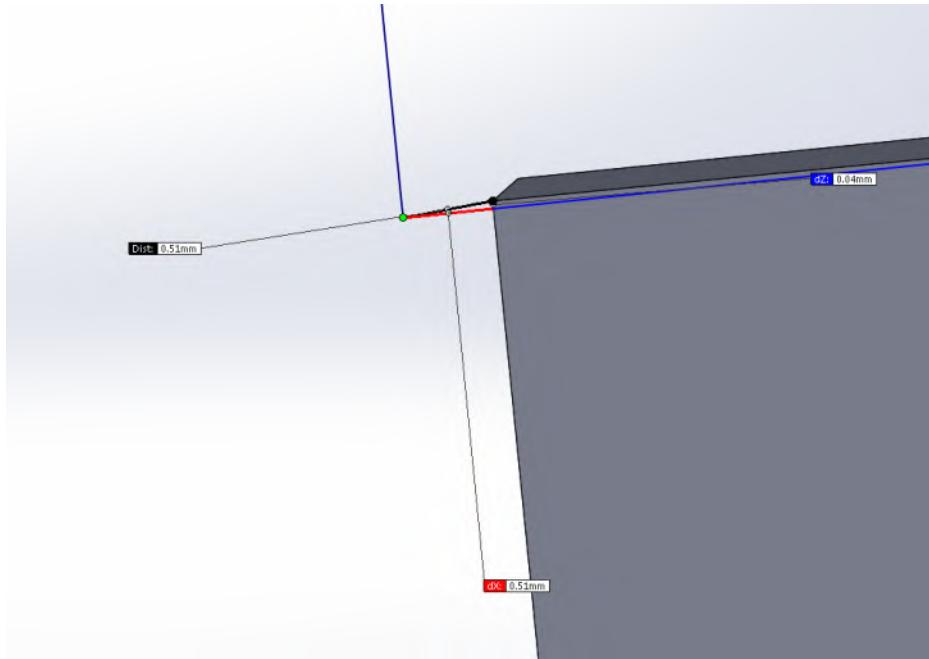
Figure 11a : Problem faced with TCC geometry

To solve this problem, I relocated the origin and bowl geometry. There was no numerical gap between the upper and lower part of the total geometry. However, as you can see below.
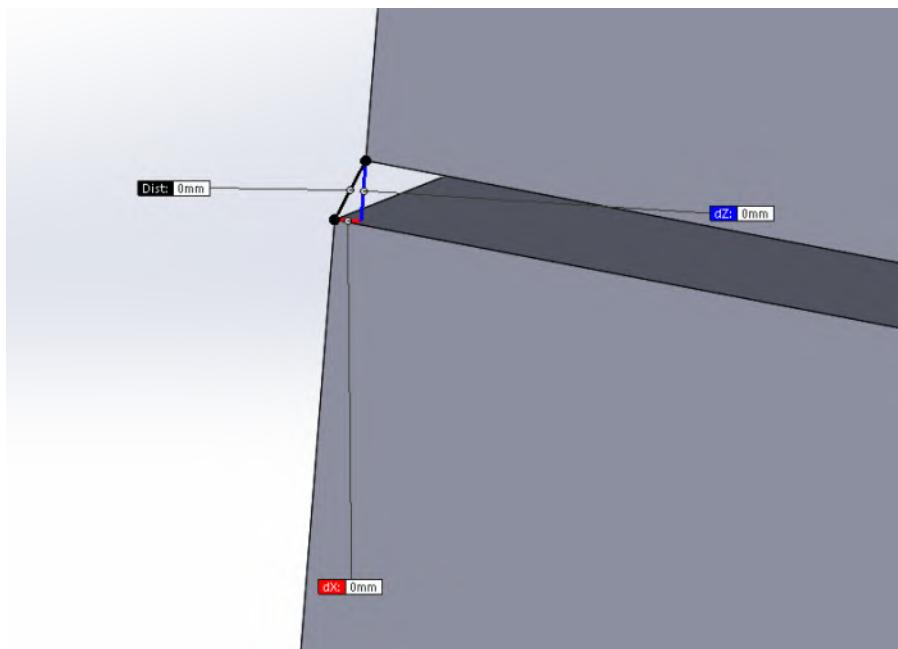
Figure 11b : Problem faced with TCC geometry

There was still a visible gap eventhough it is too small. And with this, it was unavailable to start my analysis.

So, again I discussed the problem with Dr. Ramazan ŞENER and we decided to attain a new coordinate system to our drawing again. Finally, second method was successful and I continued my analysis without any problem.

Now we are ready to get into the details of analysis section.

ANALYSIS

In this section, we are going to have a detailed review of this engine's analysis. Before beginning the analysis, I would like to give details about Converge Engine Simulation and its differences from other CFD softwares.

3.1 Specifications of Converge and Model Explanation

3.1.1 Adaptive Mesh Refinement

Adaptive mesh refinement (AMR) is a method of adapting the accuracy of a solution within certain sensitive or turbulent regions of simulation, dynamically and during the time the solution is being calculated. When solutions are calculated numerically, they are often limited to pre-determined quantified grids as in the Cartesian plane which constitute the computational grid, or 'mesh'. Many problems in numerical analysis, however, do not require a uniform precision in the numerical grids used for graph plotting or computational simulation, and would be better suited if specific areas of graphs which needed precision could be refined in quantification only in the regions requiring the added precision. Adaptive mesh refinement provides such a dynamic programming environment for adapting the precision of the numerical computation based on the requirements of a computation problem in specific areas of multi-dimensional graphs which need precision while leaving the other regions of the multi-dimensional graphs at lower levels of precision and resolution. Converge provides us this opportunity.

3.1.2 ECFM – 3Z Combustion Model

The **ECFM**-**3Z** model is a general purpose combustion model capable of simulating the complex mechanisms of turbulent mixing, flame propagation, diffusion combustion and pollutant emission that characterize modern internal combustion engines. In my study, after some comparisons, we have selected ECFM-3Z model as our combustion model.

3.1.2 Sector Mesh

Sector mesh is a method that provides less cells number with reliable results. Sector mesh means we have a portion of a whole thing and our portion is the same as the others and it allows our solver to make calculations only for this portion and assume the same results for the whole thing. In my cases, we have a piston and if you look at it from top view, you will see that it is symmetric. And we have divided our piston into 3 equal portions (120 degree) because we have 3 nozzles.

## 3.2 Preparing Geometries for CFD

We first import our .STL geometry files into Converge. In all three cases, we are going to use the same setup to compare different cases accurately. The only difference between them will be their geometries.

In the geometry section, firstly we select find option to introduce our geometry to software and it creates boundary fences. Secondly, we should select selection criteria as boundary fence and click anywhere inside the boundary of one section(for example piston) to assign flags as piston, front face, back face, cylinder wall and cylinder head as you can understand from the second and third figure. This helps program get to know the surfaces and what they are. After every surface is defined and checking for errors, our geometry will ready for case setup. We have checked for each of our cases and there were no errors given by Converge.
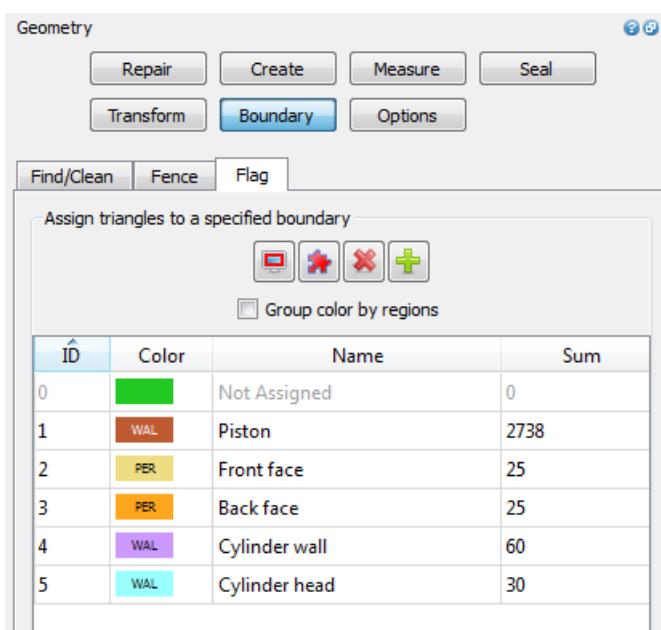
Figure 12a : Flagging
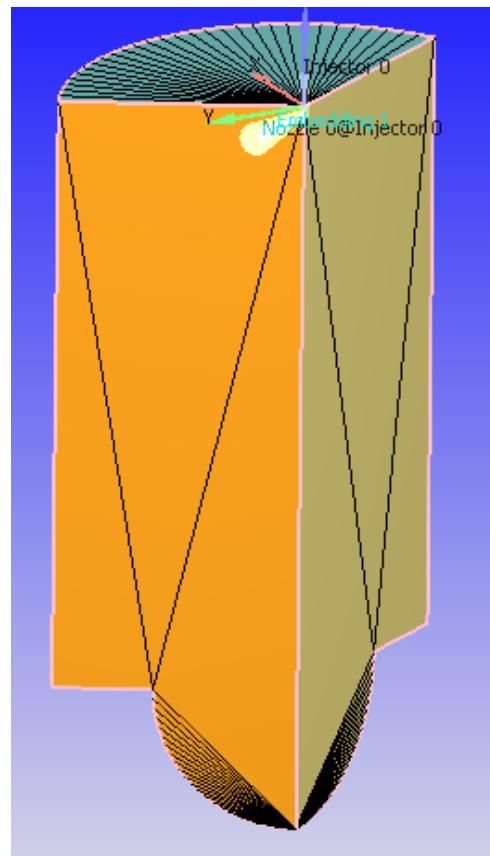
Figure 12b : Flagging

3.3 Setting up Cases
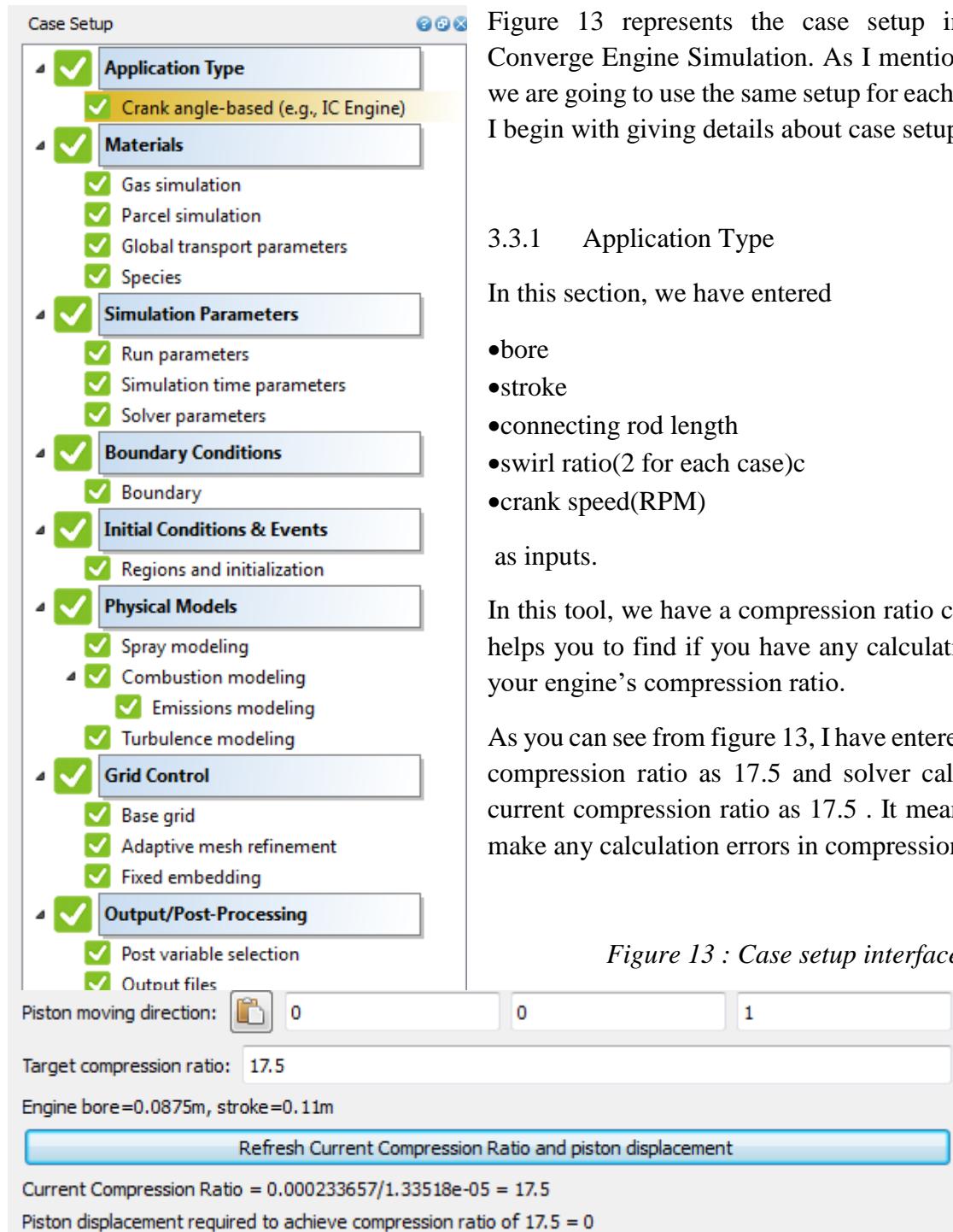
Figure 13 represents the case setup interface of Converge Engine Simulation. As I mentioned before, we are going to use the same setup for each case. Here, I begin with giving details about case setup.

3.3.1 Application Type

In this section, we have entered

- bore
- stroke
- connecting rod length
- swirl ratio(2 for each case)c
- crank speed(RPM)

 as inputs.

In this tool, we have a compression ratio calculator. It helps you to find if you have any calculation error of your engine's compression ratio.

As you can see from figure 13, I have entered my target compression ratio as 17.5 and solver calculated my current compression ratio as 17.5 . It means I did not make any calculation errors in compression ratio.

Figure 13 : Case setup interface

3.3.2   Materials

Equation of state :	Redlich - Kwong
Critical Temperature :	133 K
Critical Pressure :	3.77 MPA
Turbulent prandtl number :	0.9
Turbulent Schmidt number :	0.78

Table 1 : Materials

3.3.3   Simulation Parameters

Start Time :	-100 (aTDC)
End Time :	140 (aTDC)
Time-step Selection :	Variable time-step algorithm
Initial Time-step :	1e-06
Minimum Time-step :	1e-08
Maximum Time-step :	1e-04

Table 2 : Simulation parameters

As you can see above, we do not have a full cycle for our analysis. Our solver starts solving at -100 degree aTDC and stops at aTDC.

We also used variable time step algorithm. We start with a medium time step but if there is something really important happens solver gets into detail and starts to solve for smaller dt. Also, in the relatively unimportant conditions, it starts to solve for bigger dt. There are some reasons related to this and listed below:

- Spray and combustion are crucial for us and we want to get into detail at spray and combustion eventough the solver starts to spend more time gets slower. We have to do this because we have to get accurate, precise and most importantly reliable results. So, simulation for spray and combustion must be as perfect as possible
- Solving time is also one of the considerations. In industrial life, time is really important when it is not necessary, we do not want to spend it.

These are why we are applying variable time-step algorithm. Lastly, we used SOR solver type for momentum.

3.3.4   Boundary Conditions

| Piston Temperature	523 K
Cylinder Wall Temperature	443 K
Cylinder Head Temperature	523 K

Table 3 : Boundary Conditions

These are entered as inputs as indicated in the engine specifications.

3.3.5   Initial Conditions

Temperature	330 K
Pressure	120 kPa

Table 4 : Initial Conditions

As I indicated before, we start our anaylsis at -100 CA aTDC and finish at 140 CA aTDC. So, we had to find specific initial condition values for -100 CA aTDC but there was no specific temperature and pressure values for each crank angle but we have obtained some values from temperature and pressure plots (shown below). We have tested all of the initial conditions and the values above are reliable ones.
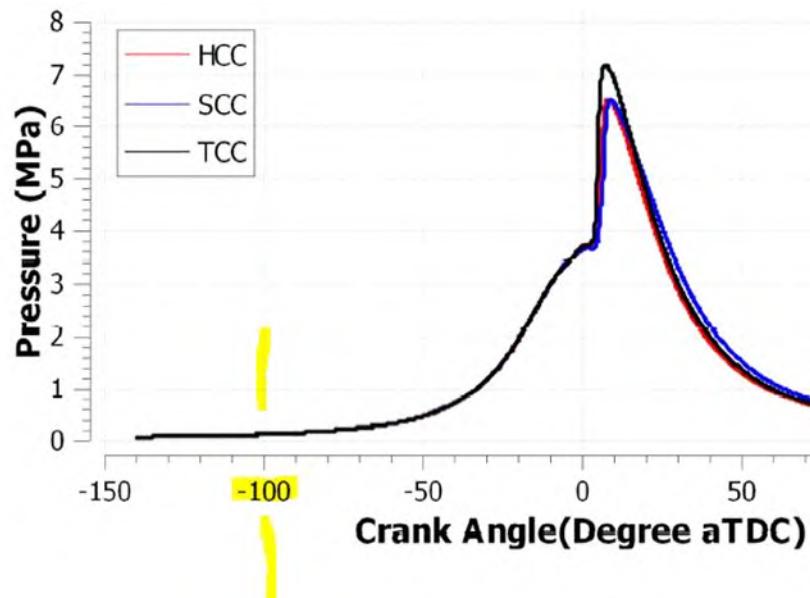
*Figure 14a : Initial conditions pressure data*
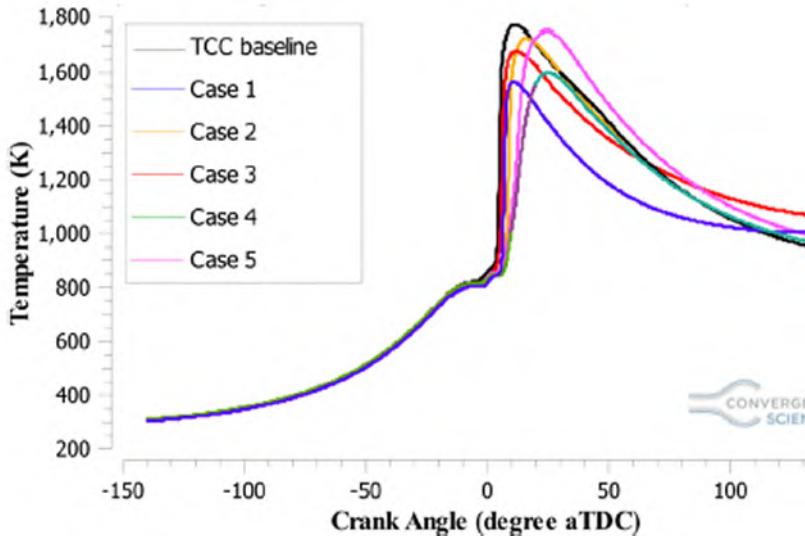
Figure 14b : Initial conditions temperature data

3.3.6   Sub-models of the Simulation

Evaporation Model	Frossling Model
Turbulence Model	RNG k-ε
Combustion Model	ECFM - 3Z
Collision Model	NTC
Spray-wall Interaction Model	Rebound / Slide
Spray Breakup Model	KH + RT
NOx Model	Extended Zeldovich
Soot Model	Hiroyasu Soot Model

Table 5 : Sub-models of the simulation

Here, After I have defined models, I am going to give details about combustion model timing/activation and injectors.

3.3.6.1 Combustion model timing

As given in engine specifications, spray starts at -23 CA aTDC and continues 22 CA which means it ends at -1 CA aTDC.

So, till that point we do not need to let the combustion model work. At -23 CA aTDC with the start of spray, our combustion model has to start working. We can not take our chance here, that is why we let our combustion model start 1 CA before the spray starts which means it will start working at -24 CA aTDC and it will continue till the end of our simulation.

Start Time for combustion model	-24 CA aTDC
End Time for combustion model	140 CA aTDC

Table 6 : Combustion model timing

3.3.6.2 Injectors

After some discussions with my supervisors, we decided to locate our nozzle 2mm away from cylinder head. As indicated before, we have 3 nozzles but also we use sector mesh (120 degree which means 1/3 of the piston) so we will locate only 1 nozzle for our sector mesh.

We also have the same condition for total injected mass. It is indicated as 2.57788e-05 kg/cycle and we use sector mesh and have 1 nozzle instead of 3. So this will also affect our injected mass of fuel. We will also take it as 1/3 of our total injected mass of fuel.

Start of Injection	-23 CA aTDC
Injection Duration	22 CA
Total Injected Mass of Fuel	2.57788e-05 kg/cycle
Injected Mass of Fuel as input	8.5926e-06 kg/cycle

Table 7 : Injectors

3.3.7 Grid Control

	Base Grid Size
dx	0.004 m
dy	0.004 m
dz	0.004 m

Table 8 : Grid control

As we indicated before, we are applying Adaptive Mesh Refinement. So, we will have a base grid and have some embedding levels and sub-grid criterions for temperature and velocity which means our mesh will be changing related to the changes in temperature and velocity.

3.3.8 Embedding Level

	Temperature	Velocity
Max Embedding Level	2	2
Sub-grid Criterion	5 K	2 m/s

Table 9: Embedding Level

Embedding level and sub-grid criterion for temperature means our mess will be finer as 2^2 =4 if there is a temperature increase of 5K anywhere in the cylinder.

Also for velocity, our mesh will be 2^2 = 4 times finer if there is a velocity increase of 2 m/s anywhere in the cylinder.

To get a better understanding of embedding level, I want to give an example. If our embedding level was 4, our change in mesh would be 2^4 = 16 times finer. We discussed this topic with my supervisors and decided that 2 is enough as embedding level for both temperature and velocity.

3.3.9   Post-Processing Selections

	Post-variable Selection
Cells	Pressure, Temperature, Velocity, Volume, Internal Energy, Density
Parcels	Radius, Temperature, Velocity
Time interval for 3D	5.0 seconds

Table 10: Post-processing selections

VERIFICATION, RESULTS and CONCLUSION

4.1 Verification of Simulation with Experimental Data (HCC pressure data comparison)
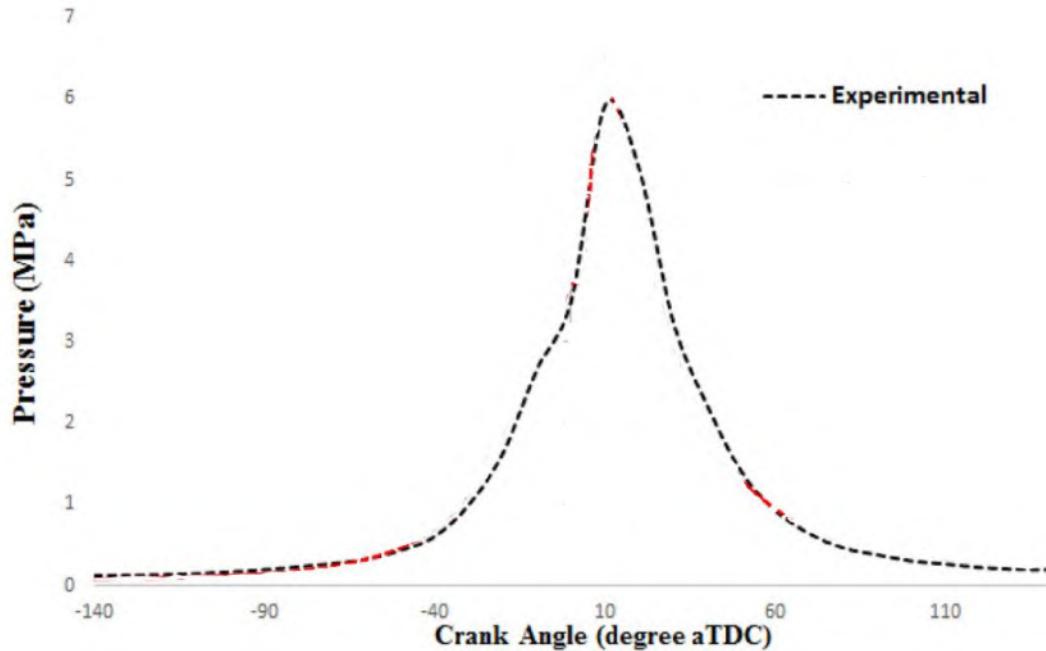

*Figure 15a : Experimental Pressure vs. CA data*
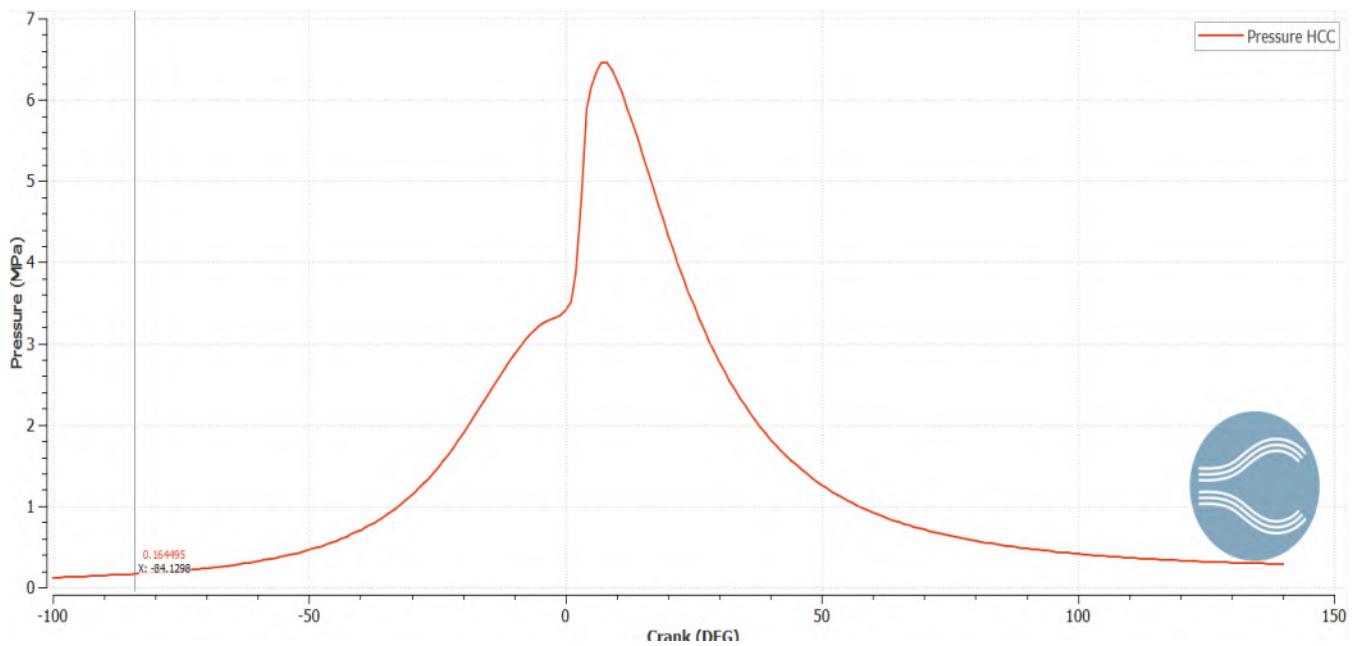
Figure 15b : Simulation data

We do not have experimental pressure values for each crank angle as an excel file. So in this section, I could not add the experimental data with every data point to my plot for one-to-one comparison but still I have both of the plots and we can understand my simulation holds.

We also discussed this situation with my supervisors, sent an e-mail to the owner's of experimental data but they did not respond. So, we have only 1 way to compare.

Do not let the area under the curve surprise you because the graphs are not at the same size and also do not start and end at the same crank angles.

There is a 0.5 MPa difference between the peak points of Pressure and there is almost no difference at 0 CA. It is observable that most of the trend holds for my simulation. Ignition delay is visible in my simulation near 0 CA.

After I obtained the results, I discussed them with my supervisor and we decided that this simulation is reliable and he said that my simulation is better than the one in the conference paper that i took as reference. We are still looking for a better way to compare. If we can find, we will add it to this study as soon as possible.

4.2 Results from Analyses

4.2.1   Pressure Comparison of Three Geometries

From now on, we are going to see

HCC	Yellow
SCC	Red
TCC	Blue

Table 11 : Geometry Colours
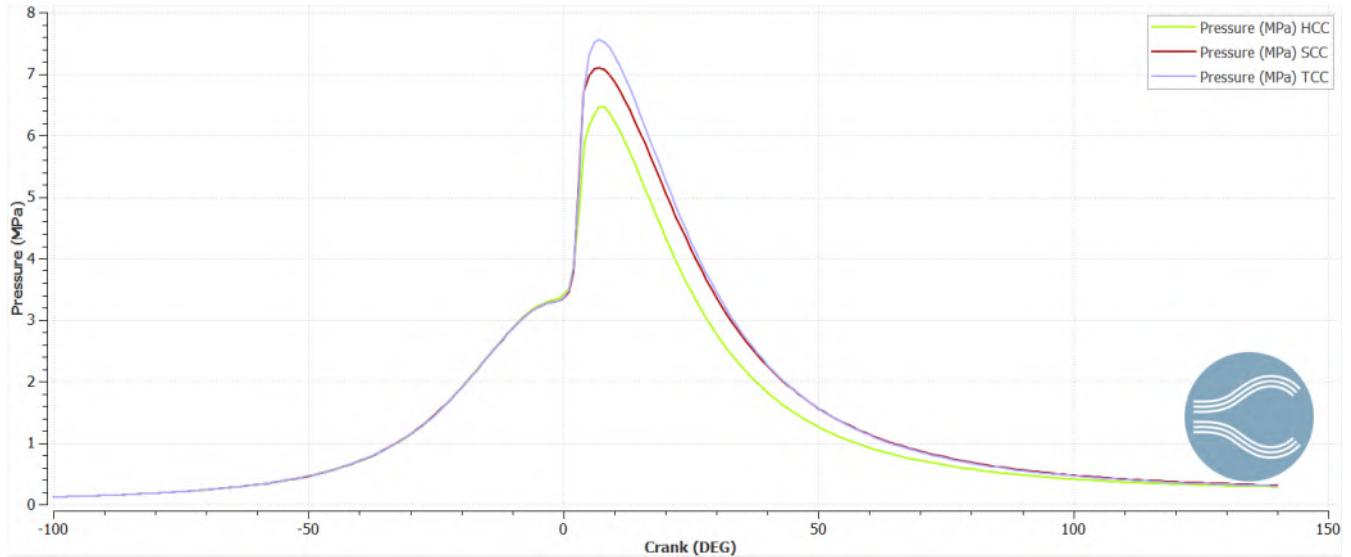
Figure 16 : P vs CA

As you can see above, we have higher pressure values for SCC and TCC compared to HCC. There are many reasons like turbulent kinetic energy which is related to combustion efficiency, geometry curvature and complexity etc.

After giving all of the plots, I am going to comment on them in detail.

4.2.2 Temperature vs CA Diagram of Three Geometries
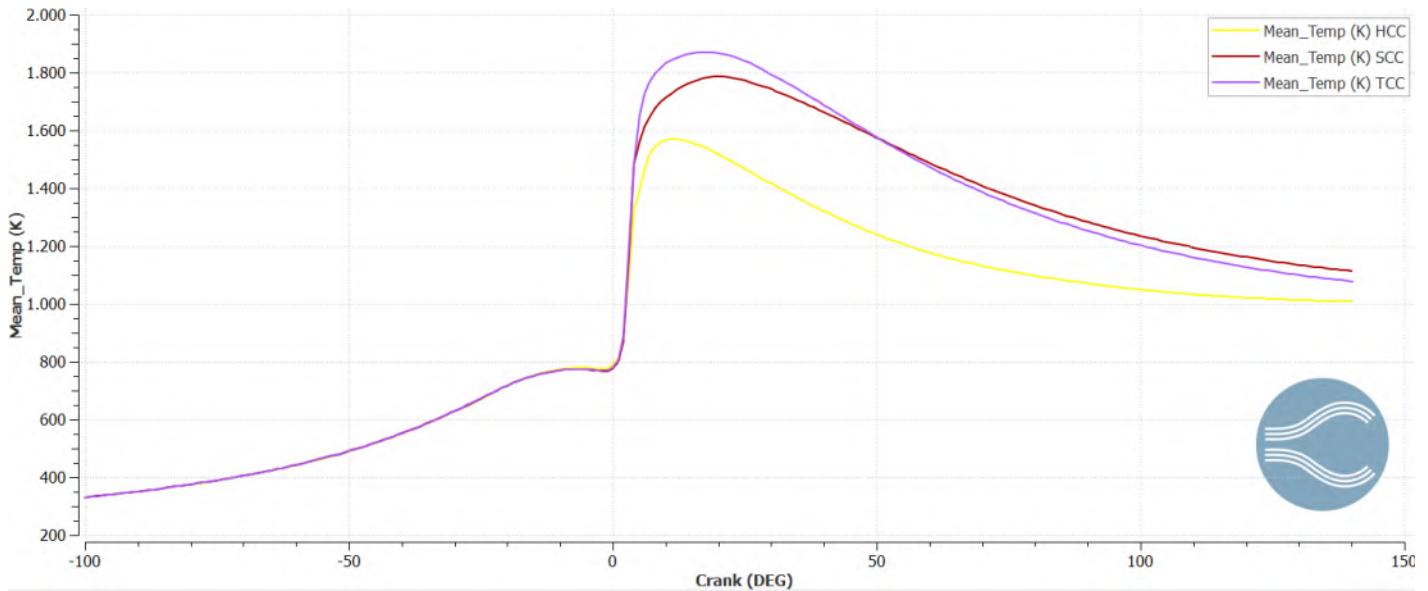
Figure 17 : Temperature vs CA

### 4.2.3 NOx vs CA Diagram of Three Geometries
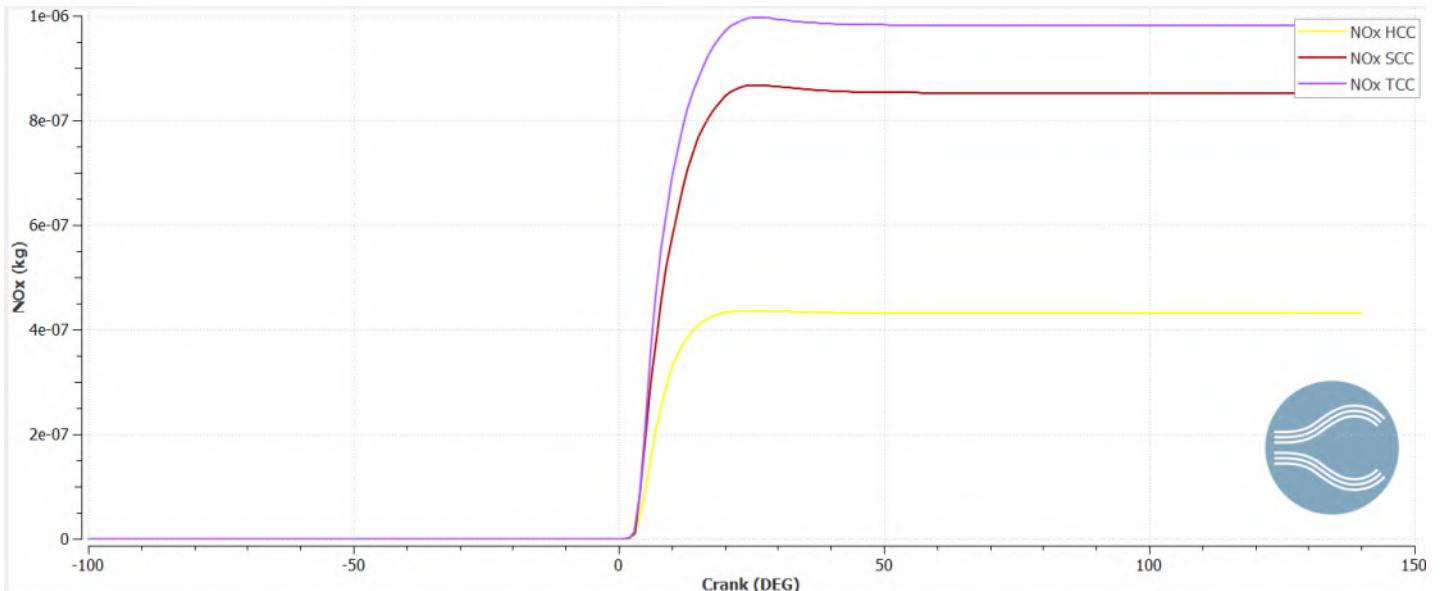
Figure 18 : NOx vs CA

4.2.4   Soot vs CA Diagram of Three Geometries
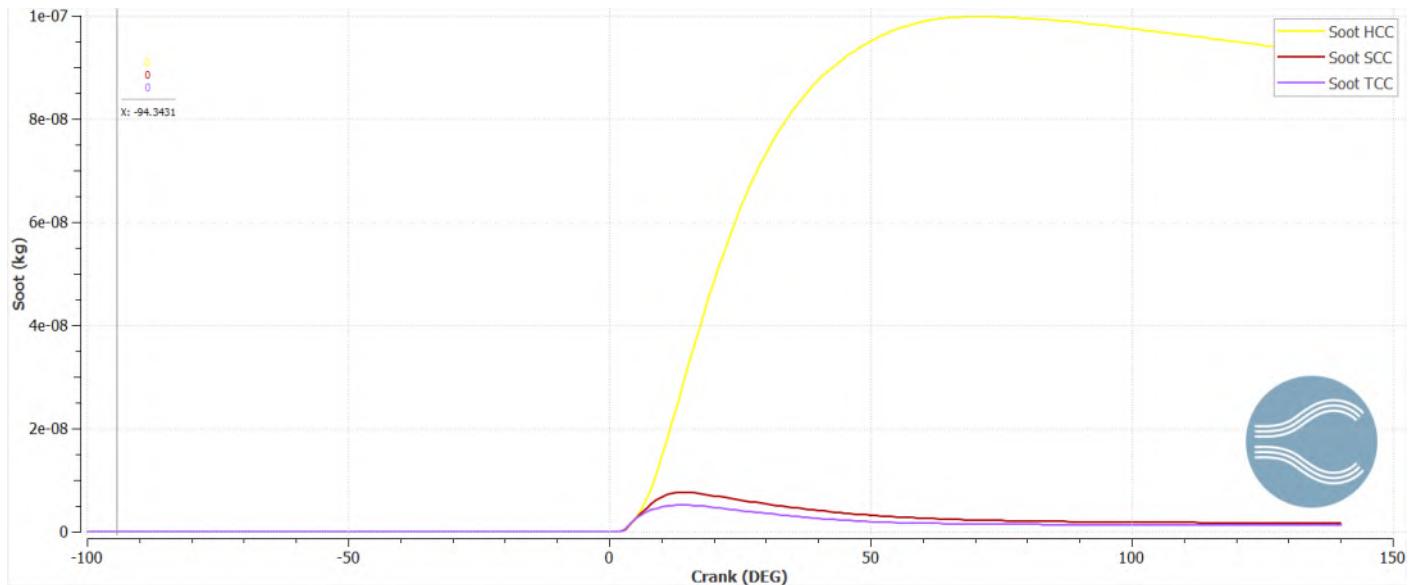
Figure 19 : Soot vs CA

### 4.2.5   CO vs CA Diagram of Three Geometries
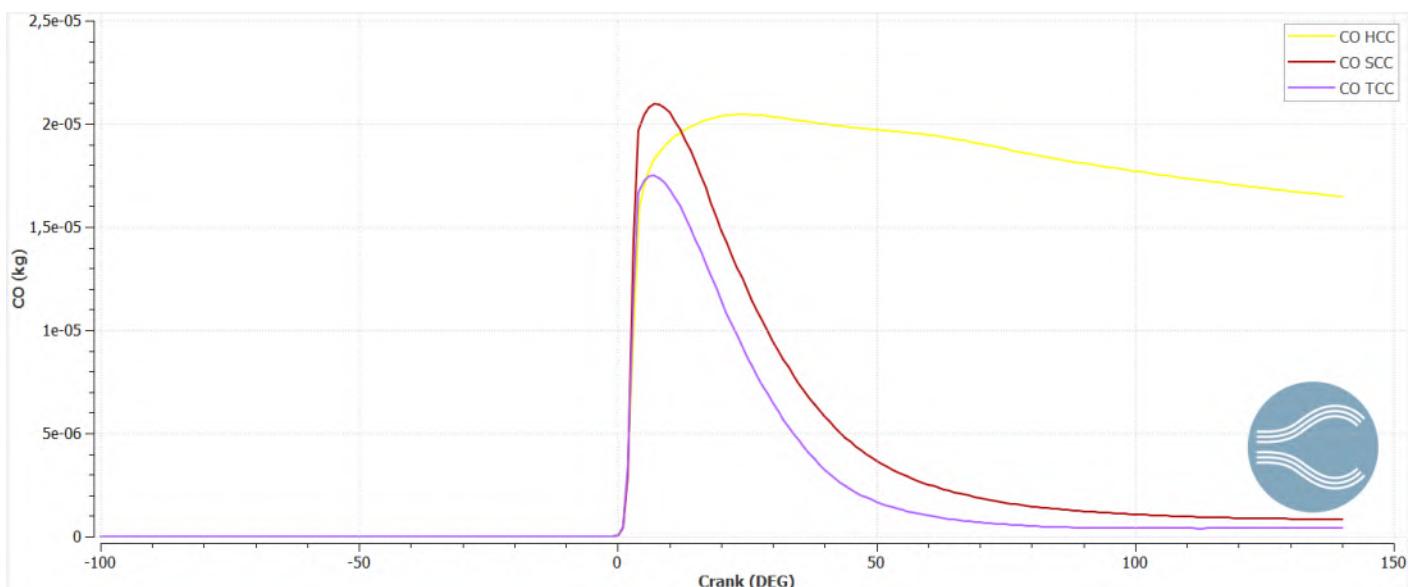
*Figure 20 : CO vs CA*

4.2.6   P - V Diagram of Three Geometries
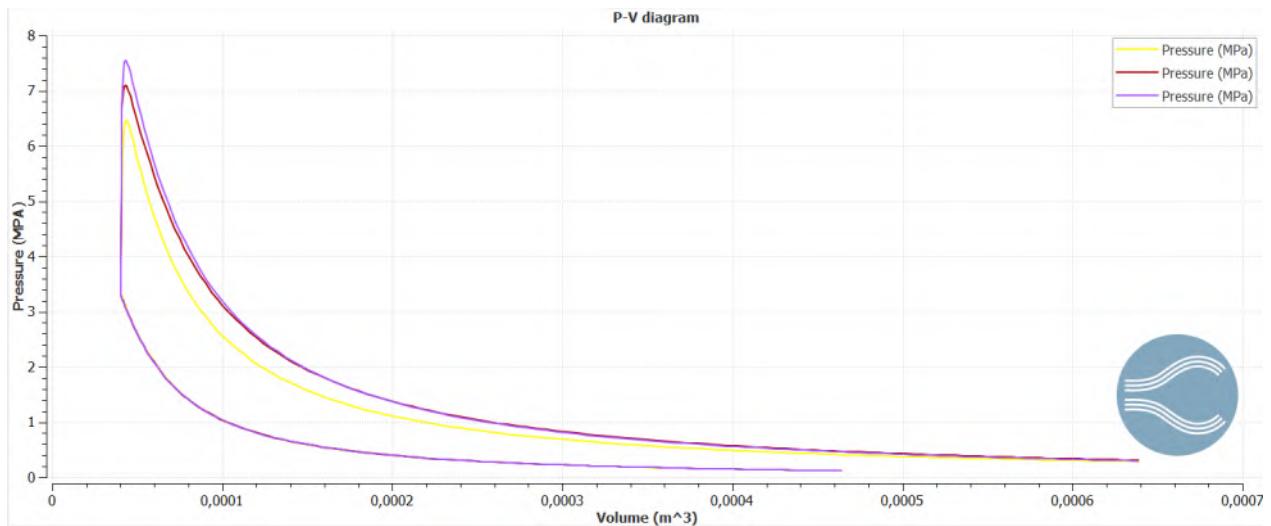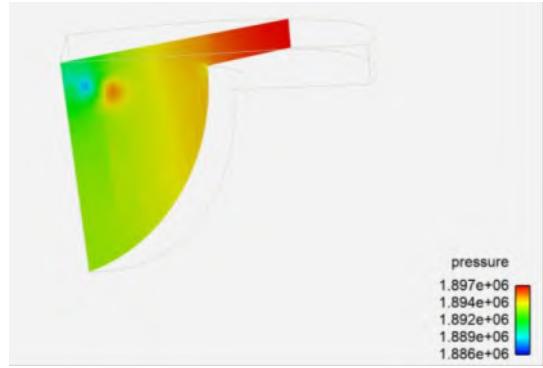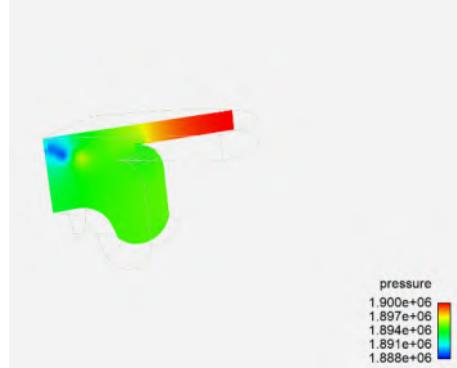
Figure 21 : P – V diagram

4.2.7 Pressure Contours for -20, -10, 0, 5 and 10 CA aTDC for 3 Geometries

HCC	SCC	TCC
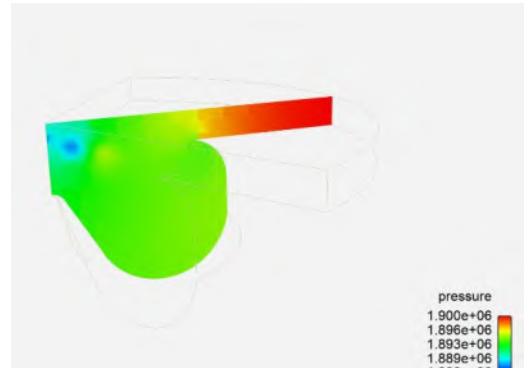
Figure 22a : Pressure contours for -20 CA

From left to right HCC, SCC, TCC pressure contours for **-20 CA**. The order will always be the same and to save place, I will not write their names again. It is also visible from geometries.

Figure 22b: Pressure contours for -10 CA
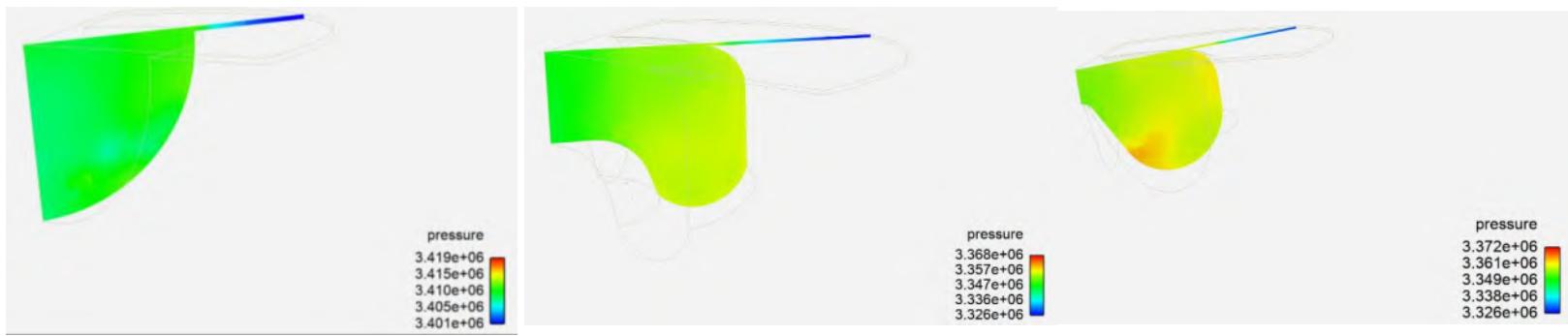
*Figure 22c : Pressure contours for 0 CA*
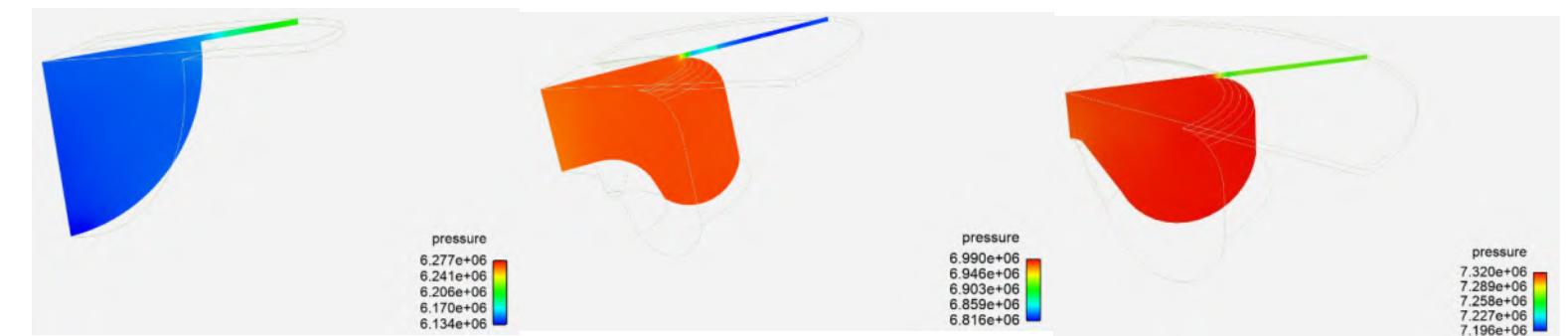
Figure 22d : Pressure contours for 5 CA
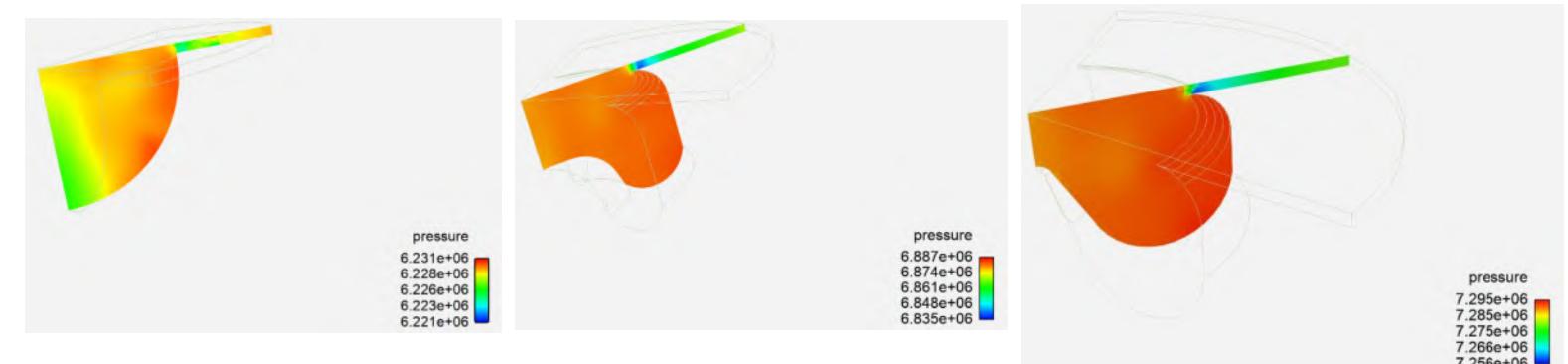
Figure 22e : Pressure contours for 10 CA

4.2.8 Temperature Contours for -20, -10, 0, 5 and 10 CA aTDC for 3 Geometries

HCC   SCC   TTC


*Figure 23a : Temperature contours for -20 CA*
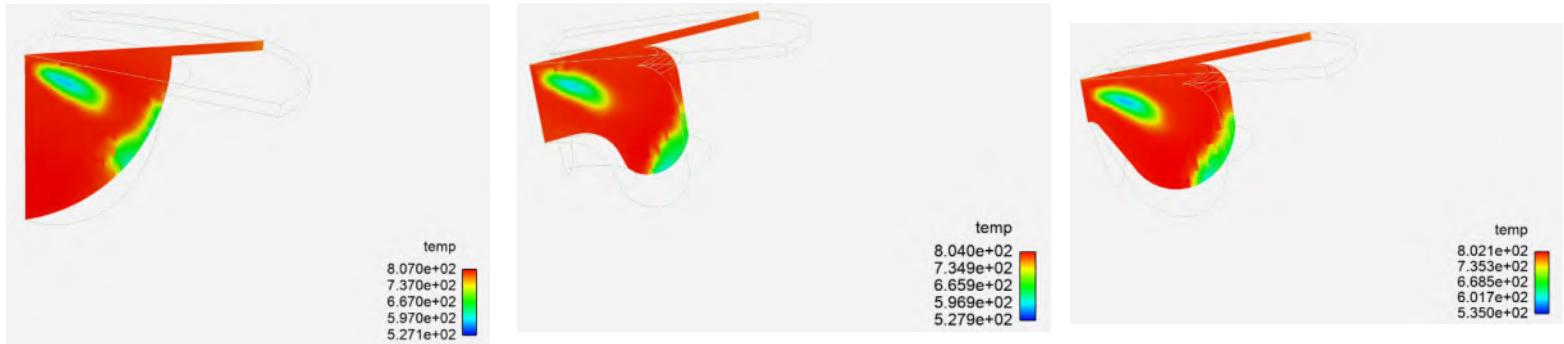
Figure 23b : Temperature contours for -10 CA
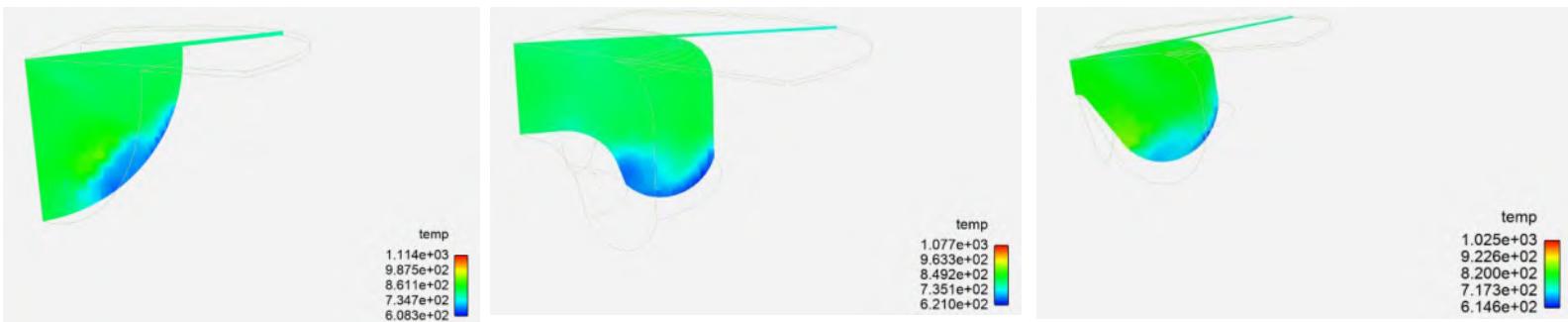
Figure 23c : Temperature contours for 0 CA
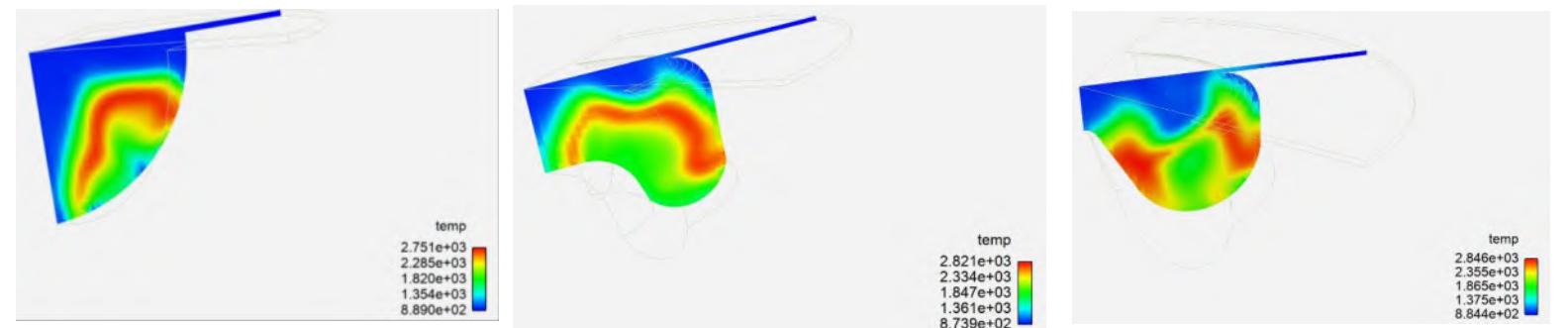

Figure 23d : Temperature contours for 5 CA
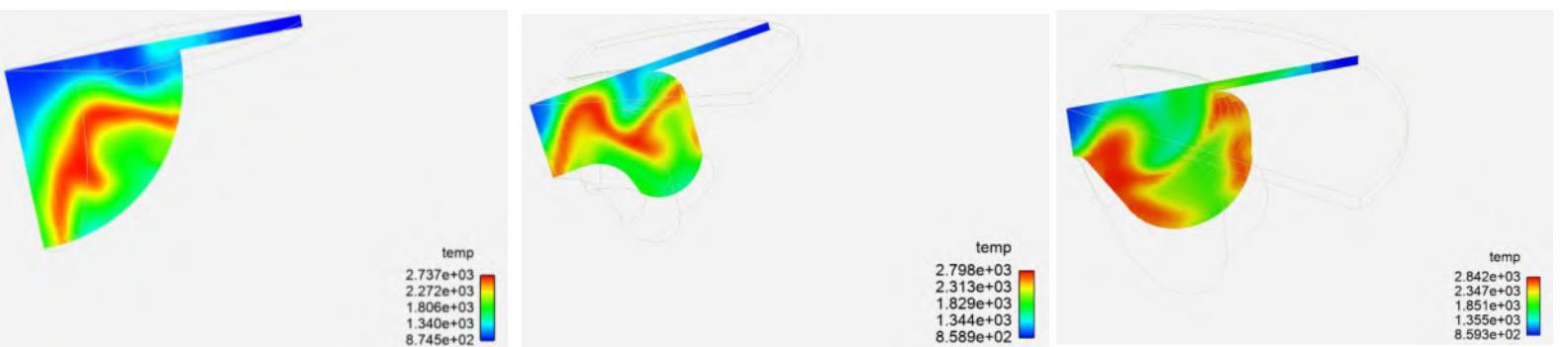
Figure 23e : Temperature contours for 10 CA

4.3 Discussion & Conclusion

As we can see from section 4.2.1 we have higher pressure values for SCC and TCC compared to HCC. The peak pressure of TCC bowl geometry is higher than others. It is due to better combustion in TCC bowl and better mixture formation of fuel and air. Better mixture formation is caused by more curvy geometry.

In section 4.2.2, because of high in-cylinder pressure, it has caused an increase in temperature inside the cylinder. Peak temperature is the highest in TCC compared to the others.

NOx variation is also higher in TCC bowl geometry compared to HCC and SCC. Because high cylinder pressure caused an increase in cylinder temperature and we know that higher peak temperature values causes higher NOx variations. That is why we have the highest NOx in TCC and the less in HCC.

It is critical to know that NOx and Soot have a reversed relation which means when NOx increases Soot decreases and also when Soot increases NOx decreases. This can also be another verification of our simulation. Because the values and the plots are reliable. While TCC has the highest NOx, we see that TCC has the lowest Soot. And also, while having HCC having the lowest NOx value, we see that HCC has the highest Soot.

We can also relate these two with the combustion efficiency. There is less soot in TCC compared to HCC so we can say our combustion is more efficient in TCC than HCC.

If we take a look at CO plot, for TCC and SCC we can see that, at 5 CA aTDC, CO has its peak value for both of this two geometries but, after 5 CA aTDC, we can also easily see that CO is almost going to vanish in TCC and SCC. However we can not say the same thing for HCC.

There is also a relation between Soot and CO. It is critical to know that Soot and CO have similar trends. As we can easily see from the graphs, HCC has the highest Soot and CO value at the same time. This could be used as a generalization.

I also want to comment on the igniton delay in CI combustion and also my geometries. If you take a look at my P vs CA graph you can easily see the ignition delay between -5 and 0 CA. As we know from Internal Combustion Engines 1 Course, The ignition delay in a CI engine is defined as the time interval between the start of injection and the start of combustion. In this delay period, at first, the injected fuel is in liquid form. It is a must that our fuel hast to vaporize for the start of ignition. That is the reason that we have a delay here.

During this delay time our fuel gets heat from it's environment to vaporize. This is also visible in our Temperature vs Crank Angle plot. As you can easily see there is a little decrease in temperature near 0 CA. This is caused by fuel that has to vaporize.

Let us check our P vs V diagram. It looks like an SI engine, is not it? There is a reason related to this. As we all know, our simulation works from -100 CA aTDC to 140 CA aTDC which means we do not have a full cycle like 360 deg. or 720 deg. So we can say that this similarity is acceptable due to this reason. I also discussed this with my supervisor and we decided as above.

There is another thing that we should also take a look at. At first I wanted to calculate indicated power from IMEP but the results taken from Converge are not reliable (**for only this part**) because this is also related to our not having a full cycle simulation. Eventough, I made some hand calculations, I could not get an accurate result compared to the power value that is given in engine specifications as 3.5 kW which is the engines break power. If I had a chance to get closer to this value, for example indicated power as 3.8 kW, then I would say there is also mechanical efficiency that differs between 0.8 to 0.95 and this value would be acceptable.

For pressure and temperature contours, if you take a detailed look you will see that till TDC, there are small differences for pressure and temperature values in the contours of HCC, SCC and TCC. But, at 5 and 10 CA there is a visible difference between the geometries.

This verifies our P vs CA, T vs CA plots. We also see the same situation in them. Till TDC there are not big differences, but after the instant that combustion started, the differences get visible more and more.

In this study, I did not give any detail about hydrocarbon (HC) because the values were too small and can not be considered as a reference. And we also know that, in diesel combustion we do not have too much HC.

For the cell numbers, I already gave details about mesh and it's not under the control of the user. We just define temperature and velocity's sub-grid criterions and embedding levels. So I do not have a specific plot for this simulations' cell numbers.

Last but not least, I also have different animations that I have taken from ANSYS Ensight for each of our bowl geometries. I will also share them with you to take a look at.

Mehmet Emre ÇETİN

Yours sincerely,

REFERENCES

[1] Helgi Fridriksson, Bengt Sund´en, Shahrokh Hajireza, Martin Tun´er '*CFD Investigation of Heat Transfer in a Diesel Engine with Diesel and PPC Combustion*' JSAE 20119177 SAE 2011-01-1838

[2] C.S. Sharma, T.N.C. Anand and R.V. Ravikrishna '*A methodology for analysis of diesel engine in-cylinder flow and combustion*' Progress in Computational Fluid Dynamics 10(3):157–167, 2010

[3] Eivaz Akbarian, Bahman Najafi, Mohsen Jafari, Sina Faizollahzadeh Ardabili, Shahaboddin Shamshirband and Kwok-wing Chaue '*Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled diesel engine*' ENGINEERING APPLICATIONS OF COMPUTATIONAL FLUID MECHANICS 2018, VOL. 12, NO. 1, 517–534

[4] Henrik W. R. Dembinski '*In-cylinder Flow Characterisation of Heavy Duty Diesel Engines Using Combustion Image Velocimetry*' TRITA – MMK 2013:17

[5] M. Zafer GUL, M. YILMAZ, M. YILDIRIM, S. NAS '*IN-CYLINDER COLD FLOW MODELING WITH SPRAY INTERACTION IN A HEAVY DUTY DI-CI ENGINE*' Conference Paper · November 2008

[6] K. Abay, U. Colak, L. Yüksek '*COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF FLOW AND COMBUSTION OF A DIESEL ENGINE*' Journal of Thermal Engineering, Vol. 4, No. 2, Special Issue 7, pp. 1878-1895, February, 2018

[7] PRABHAKARA RAO GANJI, RUDRA NATH SINGH, V R K RAJU and S SRINIVASA RAO '*Design of piston bowl geometry for better combustion in direct-injection compression ignition engine*' https://doi.org/10.1007/s12046-018-0907

[8] Shahanwaz Khan, Rajsekhar Panua, Probir Kumar Bose '*Combined effects of piston bowl geometry and spray pattern on mixing, combustion and emissions of a diesel engine: A numerical approach*' Fuel 225 (2018) 203–217

[9] Kun Lin Tay, Wenming Yang, Feiyang Zhao, Wenbin Yu, Balaji Mohan '*Numerical investigation on the combined effects of varying piston bowl geometries and ramp injection rate shapes on the combustion characteristics of a kerosene-diesel fueled direct injection compression ignition engine*' Energy Conversion and Management 136 (2017) 1–10

[10] A. De Risi, D. F. Manieri and D. Laforgia '*A Theoretical Investigation on the Effects of Combustion Chamber Geometry and Engine Speed on Soot and NOx Emissions*'

[11] Amir-Hasan Kakaee, Ali Nasiri-Toosi, Babak Partovi, Amin Paykani '*Effects of piston bowl geometry on combustion and emissions characteristics of a natural gas/diesel RCCI engine*' Applied Thermal Engineering 102 (2016) 1462–1472

[12] Federico Perini, Kan Zha, Stephen Busch, Eric Kurtz, Richard C Peterson, Alok Warey and Rolf D Reitz '*Piston geometry effects in a light-duty, swirl-supported diesel engine: Flow structure characterization*' International J of Engine Research 2018, Vol. 19(10) 1079–1098

[13] M. Zafer GUL, Mustafa YILMAZ, Hasan KOTEN '*EXAMINATION OF CONE ANGLE AND SPRAY PROFILE ON EMISSIONS OF A HEAVY DUTY CI ENGINE AND GEOMETRICAL IMPROVEMENTS*' Proceedings of ICFD 10: Tenth International Congress of Fluid Dynamics December 16-19, 2010

[14] Paul C Miles and Öivind Andersson '*A review of design considerations for light-duty diesel combustion systems*' 13 June 2015

[15] Yong Lu and Daniel B. Olsen 'OPTIMIZATION METHOD AND SIMULATION STUDY OF A DIESEL ENGINE USING FULL VARIABLE VALVE MOTIONS' Journal of Engineering for Gas Turbines and Power. Received November 13, 2016

[16] Zhao, L., Torelli, R., Zhu, X., Scarcelli, R. '*An Experimental and Numerical Study of Diesel Spray Impingement on a Flat Plate*' SAE Int. J. Fuels Lubr. 10(2):2017, doi:10.4271/2017-01 0854.

[17] S. Mauro, R. Şener, M. Z. Gül, R. Lanzafame, M. Messina and S. Brusca '*Internal combustion engine heat release calculation using single-zone and CFD 3D numerical models*' International Journal of Energy and Environmental Engineering February,12-2018

[18] N. Ramesh, J.M. Mallikarjuna, '*Low Temperature Combustion Strategy in an Off-Highway Diesel Engine – Experimental and CFD study*' Applied Thermal Engineering (2017)

[19] Daniyal Khan and M. Zafer Gül '*Zero-dimensional modelling of a four-cylinder turbocharged diesel engine with variable compression ratio and its effects on emissions*' SN Applied Sciences (2019) 1:1162