"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 2
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

COMPRESSION STAGE NUMERICAL ANALYSIS OF A MARINE ENGINE

Dorel Dumitru VELCEA[1]

[1] Ph.D. Student, Military Technical Academy, Bucharest

Abstract: *The departing information were rather scarce in terms of real processes taking place in the compression stage, but at the end we managed to have a full picture of the main parameters evolution during the compression inside this existing marine engine.*
The target of this paper was to show how, by using the reverse engineering techniques, one may replicate and simulate the functioning conditions and parameters of an existing marine engine.
The air flow jet that enters the cylinder swirling via the intake valves has changing characteristics, same can be said about the exhaust jet through the exhaust valves as they open and close, and they can be ascertained, along with the turbulence creation from swirl and tumble due to compression and squish.
The mail goal of an engine's design is to maximize the efficiency factors, to develop the most power from the least amount of fuel. Regarding fluid dynamics, the combustion and volumetric efficiency depend on the fluid dynamics in the engine manifolds and cylinders. Analysis in cold flow involves modeling the airflow in the transient engine cycle without reactions. The endgame is to catch the process of mixture formation by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process.

Keywords: *Compression Simulation; Marine Engines; Finite Volume Analysis*

Introduction

Internal combustion engines have been very successful as power producers in the marine application due to their simplicity of construction, high efficiency and long track of proven technology.
Designing an engine is very challenging. Many parameters that will contribute to the performance of the engine. However, performance through power and torque, is not the only target. When an engine is conceived for commercial purposes, the number of challenges rises. Consumers and legislation will put demands on noise, fuel consumption, levels of emissions etcetera. There are useful tools that help design an engine which fulfills these requirements. Usually an engine is developed using a combination of various practical experiments, simulations with the help of powerful computers and of course previous experiences. These tools are very useful because they predict the engine characteristics but also help to evaluate its design. The focus will now be put on computer aided simulations and flow simulations in particular.
Calculations, which models the flow of fluids in an engine, can be used to predict and model a number of properties. A tool that is commonly used the 1-dimensional flow calculation software, [2] [3]. This program can predict many properties such as: engine power, torque and levels of emissions. When there is a need of a more detailed description of a flow, a 3-dimensional flow calculation method is preferred, [6]. The 3-dimensional flow simulation can give information about weak spots in the engine design regarding flow characteristics. The 3-dimensional flow calculations will from now on be referred to as computational fluid dynamics, CFD. The CFD is used to calculate properties like temperature, pressure, velocities which are used to characterize a flow.
The combustion process can also be modeled by CFD. The information which could be acquired from these features could be very helpful when designing or performing research on two-stroke engines.
A two-stroke engine contains three moving parts, the piston (1) which is connected to the crankshaft (8) by the connection rod (3). The crankshaft (8) is located in the crankcase (9). The crankcase (9) volume and the cylinder (2) volume are separated by a dashed line in Figure 1.1. The cylinder walls are also known as the liner. In figure 1.1 we can also see scavenging port (4), the scavenging channel (5), the scavenging inlet (11), the exhaust port (6), the exhaust channel (7) and the intake channel (10).

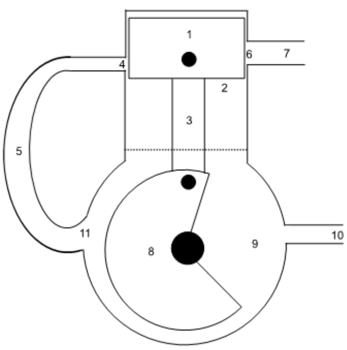

Fig.1 Schematic of a two-stroke engine

DOI: 10.21279/1454-864X-16-I2-048

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 2
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

The name of the two stoke engine derives from the fact that the engine completes a cycle in two strokes which means one revolution of the crankshaft. These two strokes are known as the compression and expansion strokes. As the crankshaft turns, the piston will move in the cylinder. The piston has two directions of motion. Downwards, the first direction, is the case when the piston moves towards the crankcase. This part is defined as the expansion because the cylinder volume above the piston is expanding. Upwards, the second direction, is when the piston is moving away from the crankcase. This part is defined as the compression because the cylinder volume above the piston is decreasing. The uppermost position of the piston is defined as the top dead center (defined from now on as TDC). This is indicated in Figure 2 as TDC with a dashed line. The piston is at TDC in Figure 2. In Figure 2 is also indicated the lowermost position of the piston. It is called the bottom dead center and is shown with a dashed line and the acronym BDC in figure 2. This figure shows the engine when the piston is at BDC. The crankshaft has turned half a revolution compared to the crankshaft position. To indicate the crankshaft position the crank angle degrees, CAD, are introduced. In this work the starting point, 0 CAD, is set to TDC. This reveals that that BDC occurs at 180 CAD. The CAD is indicated as θ [deg].

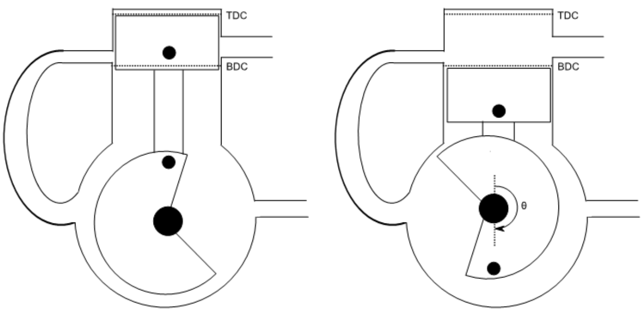

Fig.2 A two-stroke engine with the piston at TDC and BDC

In this paper we'll investigate the compression stage taking place between BDC and TDC for an engine type MAN B&W 6S60MC-C as seen in the figure below:

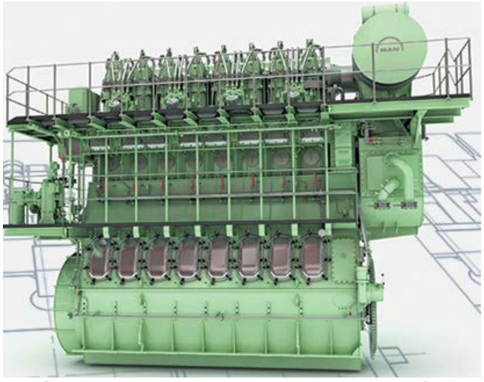

Fig.3 MAN B&W 6S60MC-C engine

The software to be used is ANSYS 15.
The primary goal of engine design is to maximize each efficiency factor, to extract the most power from the least amount of fuel. Regarding fluid dynamics, the volumetric and combustion efficiency are dependent on the fluid dynamics in the engine manifolds and cylinders. Cold flow analysis represents modeling of the airflow in the transient engine cycle without reactions. The final desire is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process. We can determine the changing characteristics of the air flow jet that tumbles into the cylinder with swirl via intake valves and through the exhaust valves, the exhaust jet, as they open and close along with the turbulence production from swirl and tumble due to compression and squish.

CAD and Finite Volume Analysis (FVA) Model of the IC

The goal of this paper is to simulate the aerodynamics of the beginning and end compression stages for MAN B&W 6S60MC-C engine. Therefore, the CFD will have as fluid domain the "negative-half" of the In-Cylinder (the combustion chamber along with its ports) as given below (CAD):

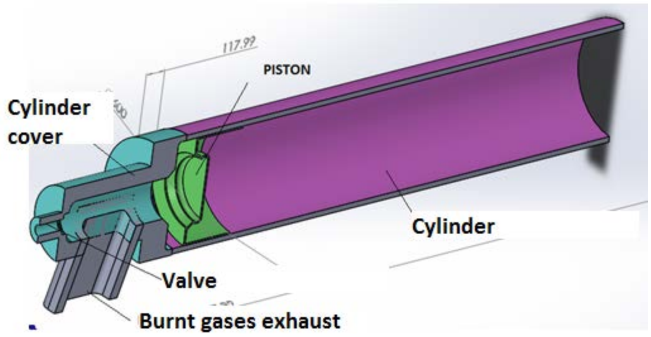

Fig.4-In cylinder and ports CAD in SolidWorks 2015

DOI: 10.21279/1454-864X-16-I2-048

Once the cylinder CAD is generated in SolidWorks 2015, the CAD model is imported in ANSYS Modeler module, and the "negative" of the geometry is obtained as shown below:

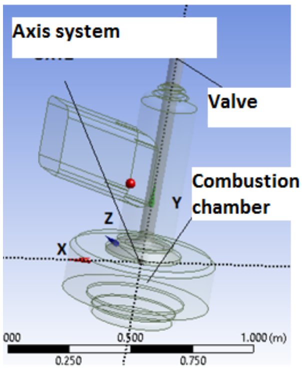

Fig.5 The ports and the combustion chamber

Furthermore, using the ICE module of ANSYS software a 90^0 sector of the combustion chamber is modeled as given below and is populated with finite volume elements:

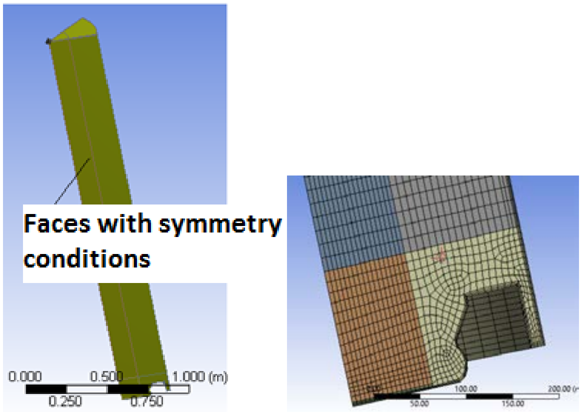

Fig.6 ICE modeled sector of the combustion chamber

The resulting model is imported in Fluent 15 module of ANSYS for further processing. The input data are as given below:

Connecting rod length [mm]	Crank length [mm]	Engine [RPM]	Exhaust valve opening angle [CAD]	Closing angle for the scavenging ports [CAD]
3000	1200	100	480	210
Temperature of the combustion chamber [^{0} K]	O_2 mass fraction in the air	CO_2 mass fraction in the air	H_2O mass fraction in the air	Pressure at the compression starting [Pa]
400	0,232	0,00046	5e-7	2.3e5

Fuel
Diesel
Temperature at the beginning of the compression [^{0}K]
302
Cylinder diameter [mm]
600

Table 1 Model input data

CFD Simulation Results

The compression is starting at 190 deg. CA.
The results analysis will cover the following crank angles (CA):

- CA = 240 (deg) in which the compression started and already covered 30 deg.
- CA = 285 (deg) in which the compression started and already covered 75 deg.
- CA = 330 (deg) in which the compression started and already covered 120 deg
- CA = 350 (deg) in which the compression started and already covered 170 deg. This is the end of the compression stage.

Results for CA = 240 (deg)

- **Pressure fields**

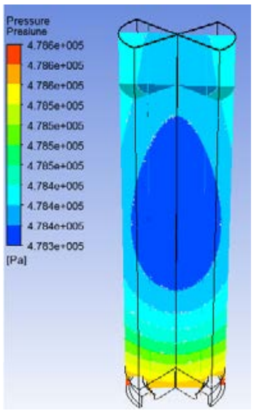

Fig.7 Pressure fields for CA=240 deg

By evolving from 210 deg when the compression started to 240 deg-CA, the piston begins to compress the air inside the combustion chamber reaching a top pressure of 4,78e5 Pa with an even distribution of pressure fields.

DOI: 10.21279/1454-864X-16-I2-048

- **Velocity fields**

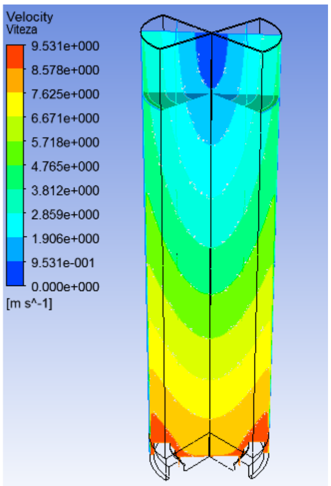

Fig.8-Velocity fields for CA=240 deg

Being in motion upwards to TDC, the piston moves the air right above inducing an increased velocity field up to 9.5 m/sec whereas the air in the upper part of the combustion chamber is relatively still and stagnant.

- **Temperature fields**

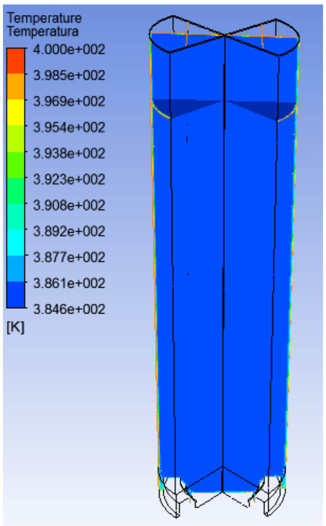

Fig.9-Temperature fields for CA=240 deg

The air temperature inside the combustion chamber rises from the initial 302^0 K to 384.9^0 K, with an even distribution across the combustion chamber.

- **Turbulence kinetic energy fields**

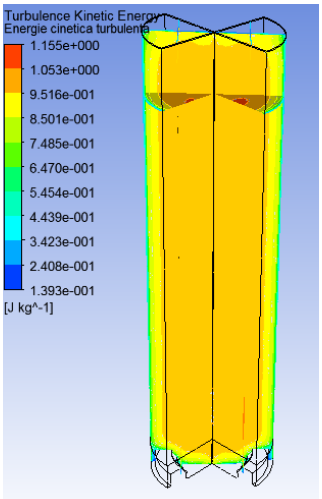

Fig.10-Turbulence kinetic energy fields for CA=240deg

The air turbulence will determine decisively the potential of the combustion once the fuel starts to be injected inside in the injection phase. The peak value for this CA angle is 1,155 J/kg and the distribution is depicted in the figure above.

Results for CA = 285 (deg)

- **Pressure fields**

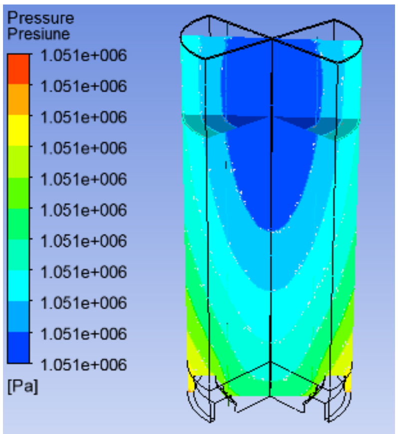

Fig.11 Pressure fields for CA=285 deg

The piston continues to rise toward the TDC so that the air is compressed more so that the peak value for the pressure inside is 1.05e6 Pa with an almost even distribution.

DOI: 10.21279/1454-864X-16-I2-048

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 2
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

- **Velocity fields**

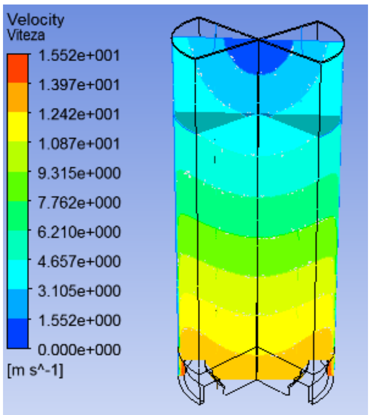

Fig.12-Velocity fields for CA=285 deg

The air above the piston head is accelerated up to 15.52 m/sec whereas at the top of the combustion chamber the central zone is stagnant.

- **Temperature fields**

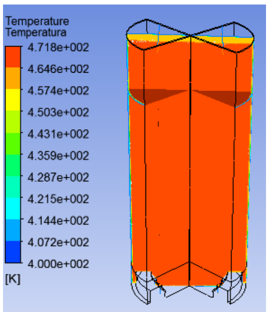

Fig.13-Temperature fields for CA=285 deg

The air temperature inside the combustion chamber to 471,8^{0} K, with an almost even distribution across the combustion chamber.

- **Turbulence kinetic energy fields**

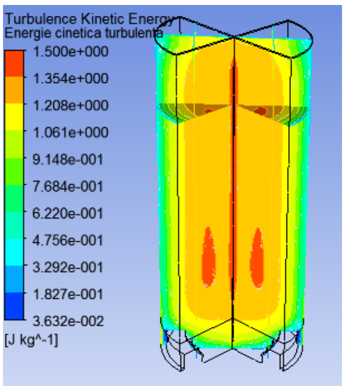

Fig.14-Turbulence kinetic energy fields for CA=285 deg

The air turbulence is rising to 1,5 J/kg within the central zone not far from the piston head.

Results for CA = 330 (deg)

- **Pressure fields**

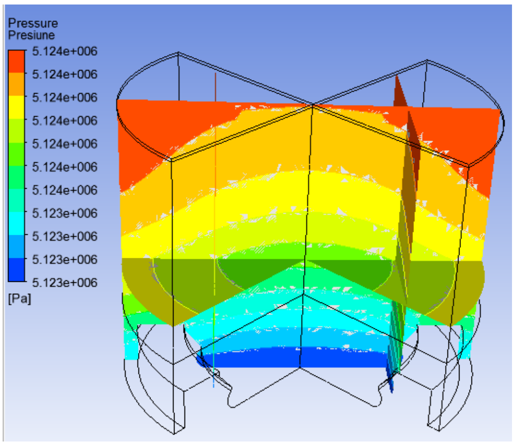

Fig.15 Pressure fields for CA=330 deg

The piston continues to rise toward the TDC so that the air is compressed more so that the peak value for the pressure inside is 5.12e6 Pa with an almost even distribution.

- **Velocity fields**

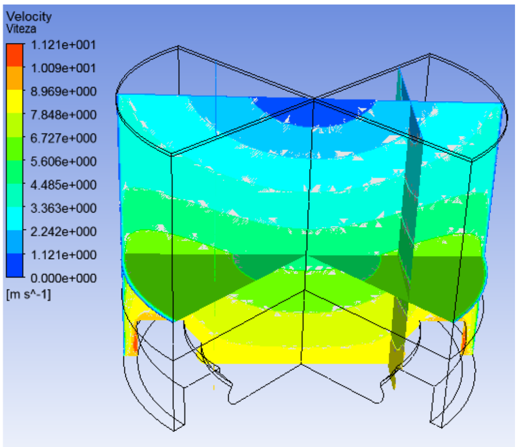

Fig.16-Velocity fields for CA=330 deg

The air above the piston head is accelerated up to 11.2 m/sec m/sec whereas at the top of the combustion chamber the central zone is stagnant.

DOI: 10.21279/1454-864X-16-I2-048

- **Temperature fields**

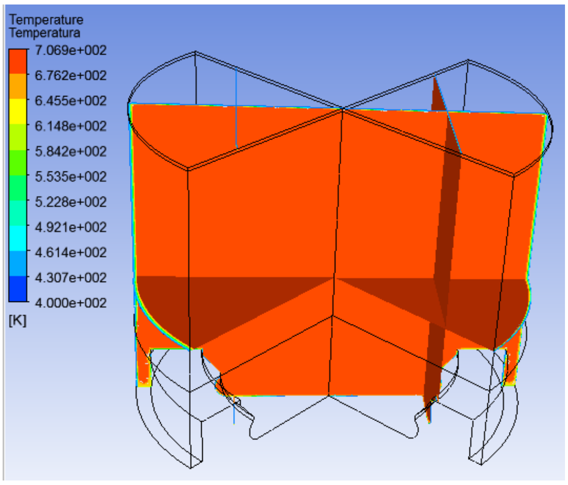

Fig.17-Temperature fields for CA=330 deg

The air temperature inside the combustion chamber to 706.9^{0} K, with an almost even distribution across the combustion chamber.

- **Turbulence kinetic energy fields**

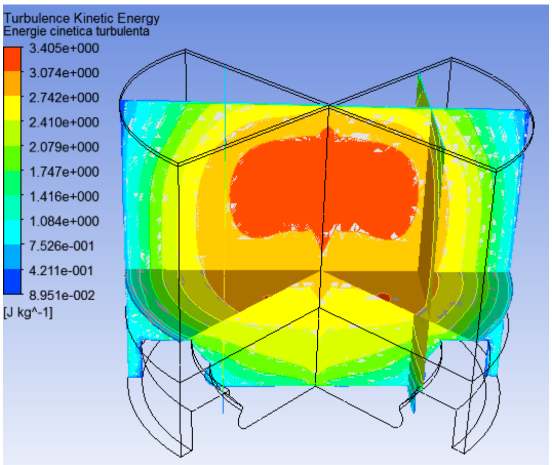

Fig.18-Turbulence kinetic energy fields for CA=330 deg

The air turbulence is rising to 3.305 J/kg within the central zone not far from the piston head. By now this high turbulence region is fully individualized in this region where the injectors are to be placed.

Results for CA = 350 (deg)-Compression stage ending

- **Pressure fields**

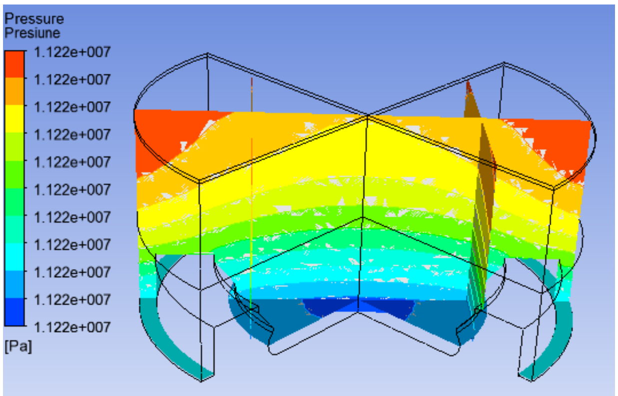

Fig.19 Pressure fields for CA=350 deg

The piston is in the TDC so that the air is compressed to its peak value for of 1,122e7 Pa with an almost even distribution.

- **Velocity fields**

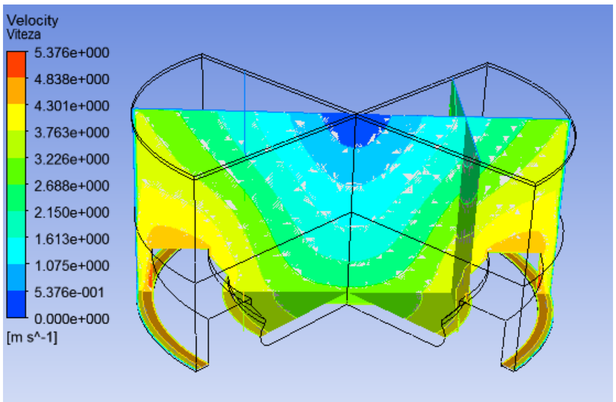

Fig.20-Velocity fields for CA=350deg

The air tends to decrease its velocity once TDC is reached with the maximum values of 5.37 m/sec distributed on the outskirts of the combustion chamber as seen above.

DOI: 10.21279/1454-864X-16-I2-048

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 2
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

- **Temperature fields**

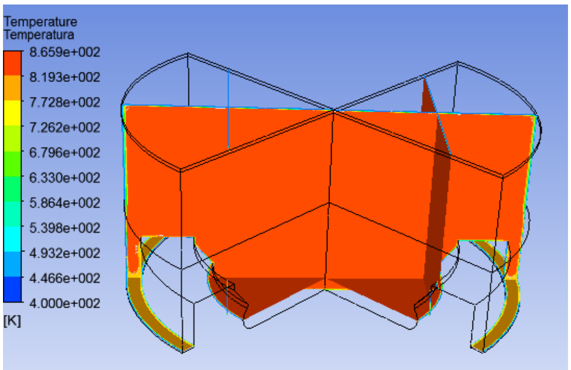

Fig.21-Temperature fields for CA=350 deg

The air temperature inside the combustion chamber to 865.9^0 K with an almost even distribution across the combustion chamber.

- **Turbulence kinetic energy fields**

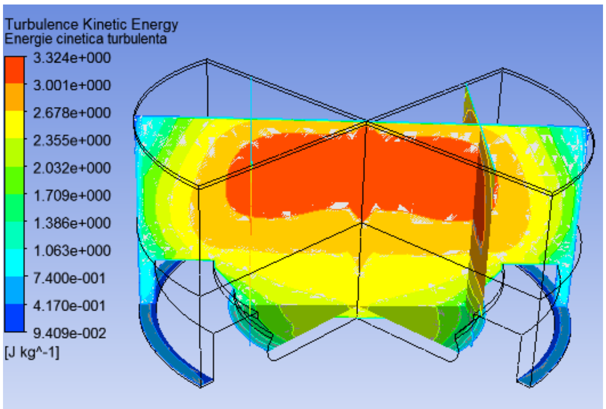

Fig.22-Turbulence kinetic energy fields for CA=350 deg

The air turbulence is rising to 3,32 J/kg within the central zone not far from the piston head. By now this high turbulence region is fully individualized in this region where the injectors are to be placed.

Model Validation

The model validation is taking into account the results gathered from the sea trials of the oil carrier SCORPIO with 105.000 TDW in March 2009. The engine of that ship is MAN B&W 6S60MC-C type.

In a synthetic way, the results of the sea trial and the model are shown below:

	Simulation results	Experimental results	Error ANSYS-Experiment %
Pressure end compression [Pa]	**11220000**	11660000.000	**+0.538**
Temperature end compression [grdK]	**865**	**901,100**	**-3.996**

Table 2- Model validation

As seen above the pressure field model at the end of compression is quite accurate, the error being only 0.53%. The temperature calculated is within the engineering precision with a value of only -4 %. In this way the model is deemed to be accurate within the engineering precision.

CONCLUSIONS

The target of this paper was to show how, by using the reverse engineering techniques, one may replicate and simulate the functioning conditions and parameters of an existing marine engine. The departing information was rather scarce regarding real processes taking place in the compression stage, but at the end, we managed to have a full picture of the main parameters evolution during the compression inside this existing marine engine.

BIBLIOGRAPHY

[1] John B. Heywood. Internal Combustion Engine Fundamentals. McGraw-Hill, Inc, 1988.
[2] R.U.K. Gustafsson M. Bergman and B.I.R Jonsson. Scavenging system layout of a 25 cc two-stroke engine intended for stratified scavenging. SAE paper no. 2002-32-1840, 2002.
[3] R.U.K. Gustafsson M. Bergman and B.I.R Jonsson. Emission and performance evaluation of a 25 cc stratified scavenging two-stroke engine. SAE paper no. 2003-32-0047,
2003.
[4] Open CFD Limited. Open FOAM The Open Source CFD Toolbox Programmer's Guide Version 1.5, 2008.
[5] Open CFD Limited. OpenFOAM The Open Source CFD Toolbox User Guide Version 1.5, 2008.
[6] Masanori Noguchi Toshiharu Sawada, Minoru Wada and Buhei Kobayashi. Development of a low emission two-stroke cycle engine. SAE paper no. 980761, 1998.
[7] ANSYS 15.0 Help Library

DOI: 10.21279/1454-864X-16-I2-048