“Mircea cel Batran” Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by “Mircea cel Batran” Naval Academy Press, Constanta, Romania // The journal is indexed in:
PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref / Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

COMBUSTION STAGE NUMERICAL ANALYSIS OF A MARINE ENGINE

Dorel Dumitru VELCEA[1]
[1] PhD Student, Military Technical Academy, Bucharest

Abstract: *The primary goal of engine design is to maximize each efficiency factor, in order to extract the most power from the least amount of fuel. In terms of fluid dynamics, the volumetric and combustion efficiency are dependent on the fluid dynamics in the engine manifolds and cylinders. Cold flow analysis involves modeling the airflow in the transient engine cycle without reactions. The goal is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process. The changing characteristics of the air flow jet that tumbles into the cylinder with swirl via intake valves and the exhaust jet through the exhaust valves as they open and close can be determined, along with the turbulence production from swirl and tumble due to compression and squish. 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 were rather scarce in terms of real processes taking place in the combustion stage, but at the end we managed to have a full picture of the main parameters evolution during the combustion phase inside this existing marine engine.*

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

Introduction

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

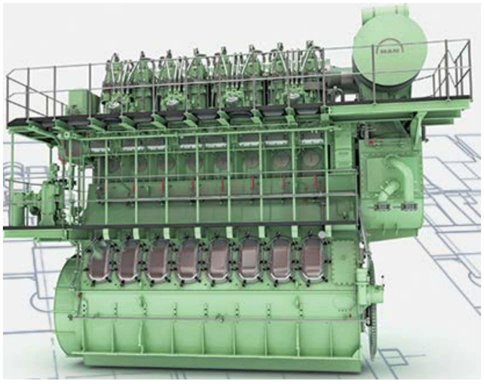


Fig.1 MAN B&W 6S60MC-C engine

The software to be used is ANSYS 15.

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

CFD simulation of combustion in an In-Cylinder engine [8] can be done to achieve variety of goals. One of the goals is to extract thermal information on the solid surfaces to perform engine cooling analysis and do thermal stress calculations. The other goal is to optimize the operating parameters of the engine to minimize formation of pollutants like soot, NOx and HC. Over years, analysts have devised best-practices in order to perform combustion calculations using CFD. Rather than using the entire in-cylinder model with ports, valves and other details, the CFD model for combustion analysis generally is restricted to a closed domain, without valves and ports, since the most important phase of cycle is from Intake-valve-closed (IVC) to exhaust-valve-open (EVO) position. Most of the times, analysts consider only a sector model of the combustion chamber for CFD analysis if the geometric model is symmetric, injector axis is aligned to the cylinder axis and the swirling flow can be assumed to be purely about the cylinder axis.

Considering a sector models leads to significant run time reduction, however in order to ensure that the air fuel ratio does not change because of the approximation of sector model, analysts sometimes have to increase the squish height (in order to account for the increase in volume due to valve recess and other features which are compromised in the sector model). Increase in squish height leads to change in compression ratio, change in the relative position of spray in the bowl region and can therefore adversely affect the combustion behavior.

DOI: 10.21279/1454-864X-16-I1-053

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania // The journal is indexed in: PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref / Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

The more seamlessly the different CFD simulations are connected the faster is the turnaround time. Seamless exchange of information from one simulation to other also leads to less loss of data and therefore more confidence on the predicted results. Tools like ANSYS Workbench allow seamless exchange of information from a CFD model to a structural model. It also facilitates performing parametric simulations where the parameters can be geometric or can be loads and boundary conditions.

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

The goal of this paper is to simulate the aerodynamics and thermodynamics of the combustion stage 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):

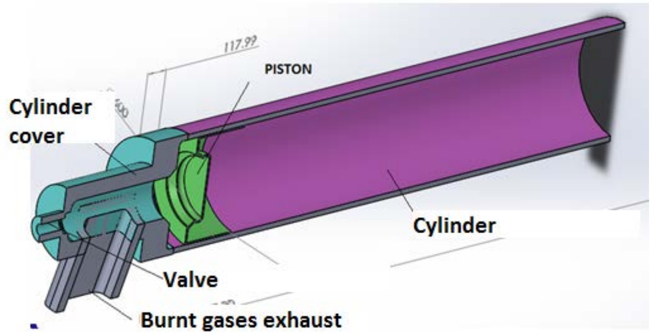


Fig.2-In cylinder and ports CAD in SolidWorks 2015

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:

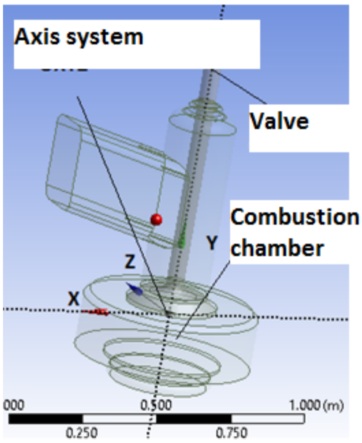


Fig.3 The ports and the combustion chamber

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:

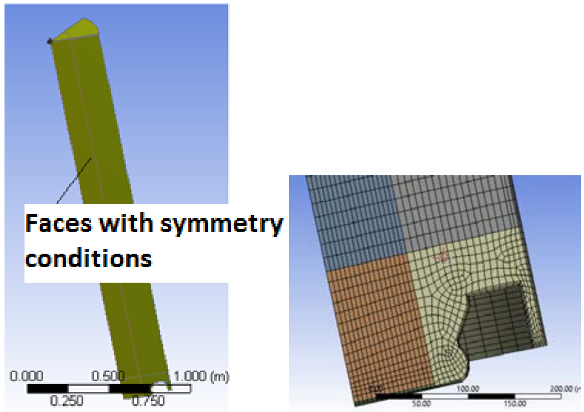


Fig.4 ICE modeled sector of the combustion chamber

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	3,21e5

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

Table 1 Model input data

The fuel injection process is very important in terms of combustion. The model of injections comprise four injectors positioned half radius inside the combustion chamber, each injector having four nozzles as seen in the figure below:

DOI: 10.21279/1454-864X-16-I1-053

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania // The journal is indexed in:
PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref / Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

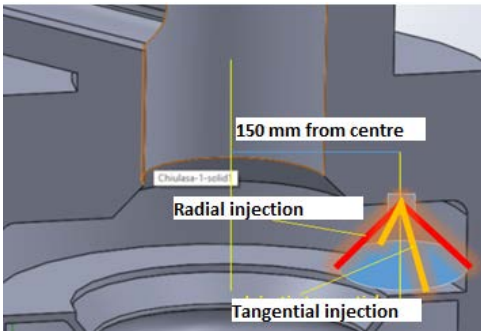


Fig.5 Injector nozzles position

The main parameters of the injection process are as follows:

- Fuel-Diesel
- Injection CA start angle-350
- Injection CA stop angle-370
- Fuel temperature-366.7 deg K
- Angle of the nozzle in report to the cylinder axis-70 deg
- Position of the injector axis in report to the cylinder axis-150 mm
- Fuel velocity at the exit from nozzle-500 m/sec
- The fuel cone angle at the exit from nozzle-9 deg
- The fuel cone radius at the exit from nozzle-0,127 mm
- Fuel mass flux-1 kg/sec
- Jet breaking model- KHRT standard
- The drag law of the fuel jet-Dynamic drag
- Tracking stochastic of fuel droplets-Discrete Random Walk Model and Random Eddy Lifetime

CFD Simulation Results

The combustion is starting at 360 deg. CA and ends at 530 deg CA when the scavenging process starts.
The results analysis will cover the following crank angles (CA):

- CA = 375 (deg) in which the injection process ended since 5 deg CA
- CA = 420 (deg) in which the injection process ended since 50 deg CA
- CA = 450 (deg) in which the injection process ended since 80 deg CA
- CA = 480 (deg)-end of combustion

Results for CA = 375 (deg)

- **Pressure fields**

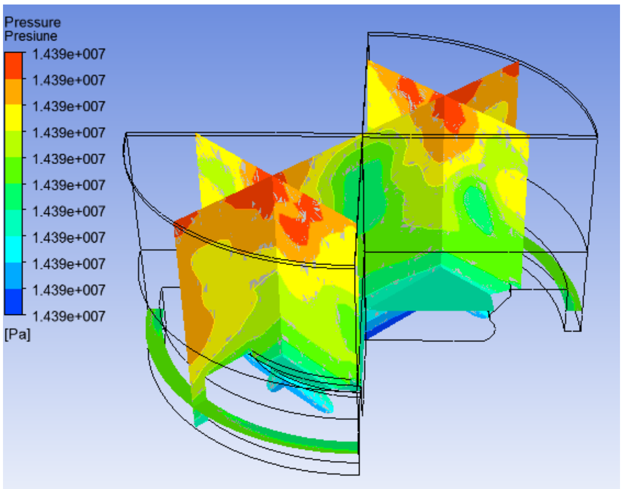


Fig.6 Pressure fields for CA=375 deg

At this stage the pressure inside is getting to a maximum of 1,439e7 Pa. This is due to the fact that the fuel injection stopped and even though the combustion of the remaining still is in place, the combustion energy is gradually transformed into work by displacing the piston downwards. The pressure field distribution is almost even.

- **Velocity fields**

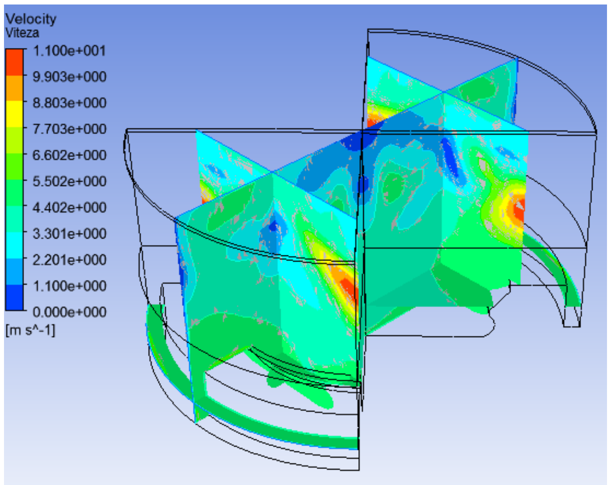


Fig.7-Velocity fields for CA=375deg

The burnt gas mixture velocity field distribution shows an increased value toward the center of the combustion chamber up to 11 m/sec, since in this region the burning of the fuel continues.

DOI: 10.21279/1454-864X-16-I1-053

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania // The journal is indexed in:
PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref /
Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

- **Temperature fields**

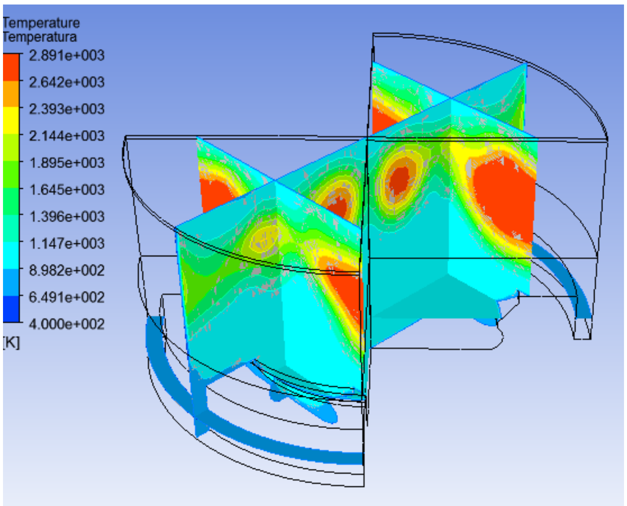


Fig.8-Temperature fields for CA=375 deg

As mentioned even though the injection process stopped, the burning of the remaining fuel is still in place. The flames front is moving towards the center and the walls of the combustion chamber from the injectors zones where the peak temperature is reaching 2891 deg K.

- **Turbulence kinetic energy fields**

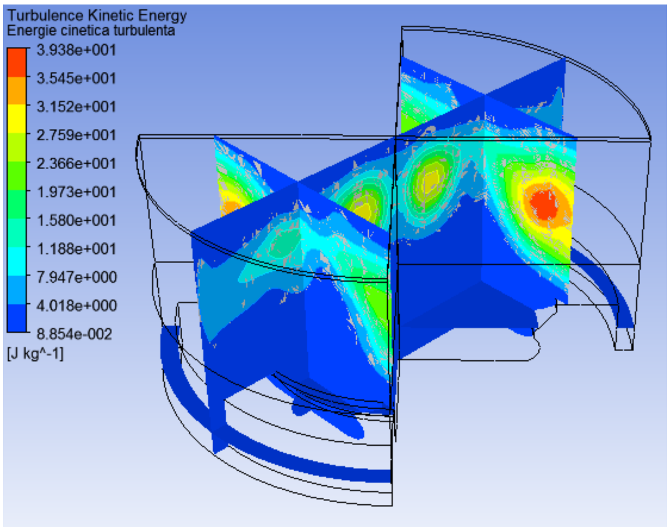


Fig.9-Turbulence kinetic energy fields for CA=375 deg

In report to the previous reading the turbulence kinetic energy stays almost constant to a peak value of 39.38 J/kg in the same central zone of the combustion chamber where the biggest velocities where recorded.

Results for CA = 420 (deg)

- **Pressure fields**

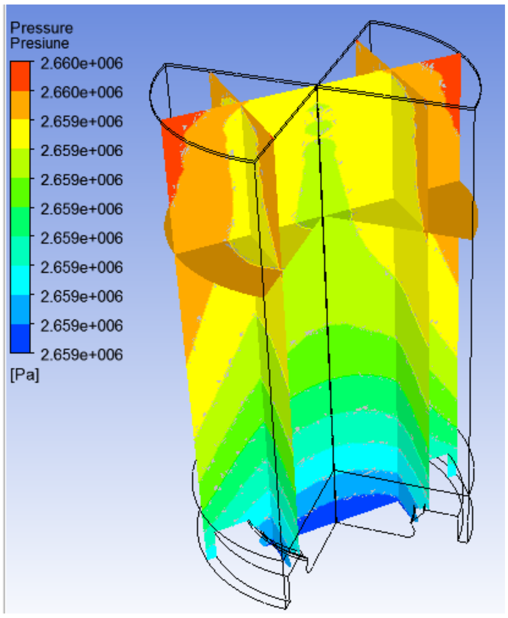


Fig.10 Pressure fields for CA=420 deg

The piston continues to travel downwards so that the pressure keeps decreasing to a value of 2.66e6Pa. The burnt gases energy is so translated into useful work transmitted to the ship propeller. The pressure field distribution is almost even due to the slow velocity of the engine (100 RPM) which allows to these fields to distribute evenly inside the chamber.

- **Velocity fields**

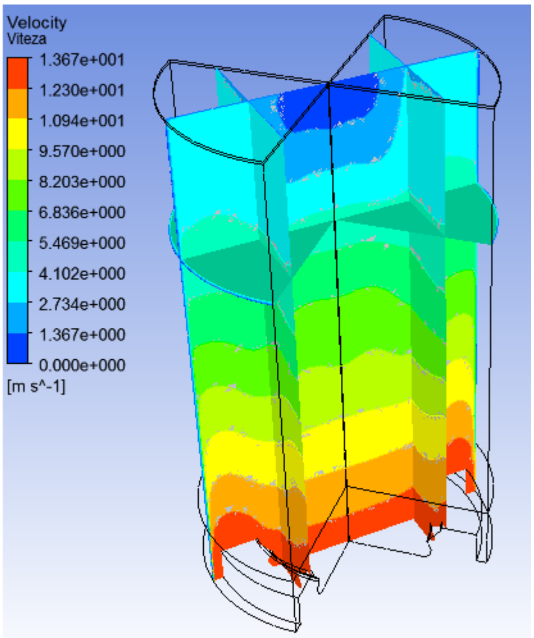


Fig.11-Velocity fields for CA=420 deg

DOI: 10.21279/1454-864X-16-I1-053

The velocity fields in turn have a dramatic change since now the increased velocity fields are moving from the central upper part of the cylinder to the lower central zone immediately above the piston head. The peak value there is 13.67 m/sec.

- **Temperature fields**

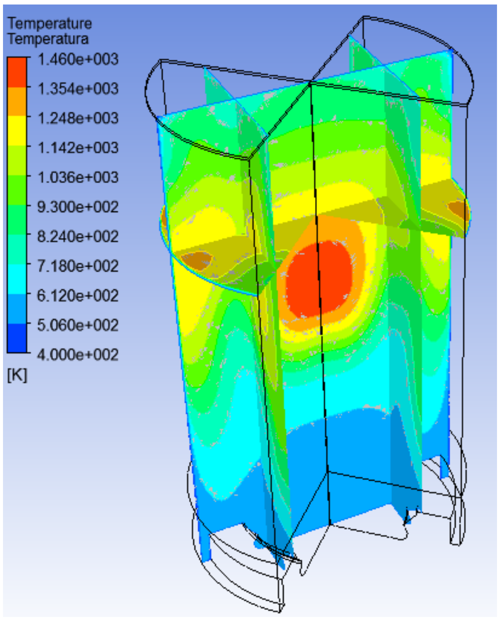


Fig.12-Temperature fields for CA=420deg

Since by piston motion downwards the combustion chamber is elongating, the combustion process still in place has to accommodate to an larger volume, so that the peak value of the temperature decrease to 1460 deg K and the shape of the temperature field is elongated.

- **Turbulence kinetic energy fields**

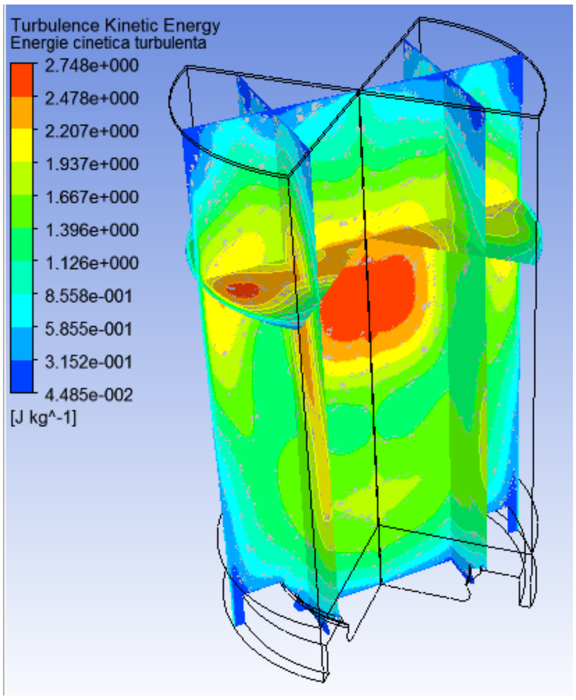


Fig.13-Turbulence kinetic energy fields for CA=420deg

This new configuration of the combustion process and chamber will lead to an increased turbulence inside, the maximum values being 2.74 J/kg in the very center of the fluid domain.

Results for CA = 450 (deg)

- **Pressure fields**

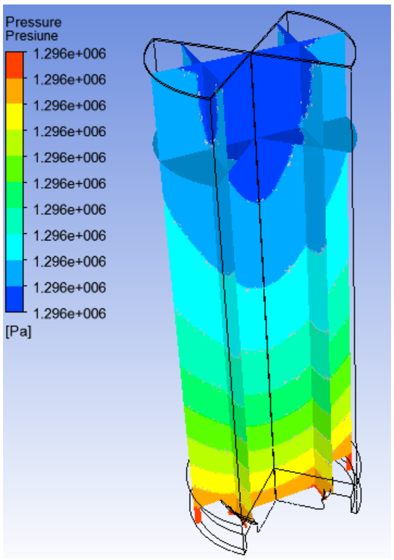


Fig.14 Pressure fields for CA=450 deg

DOI: 10.21279/1454-864X-16-I1-053

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania // The journal is indexed in: PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref / Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

The pressure continues to drop to a maximum of 1.29e6 Pa with the same even distribution, with a very slight difference right near the piston head.

- **Velocity fields**

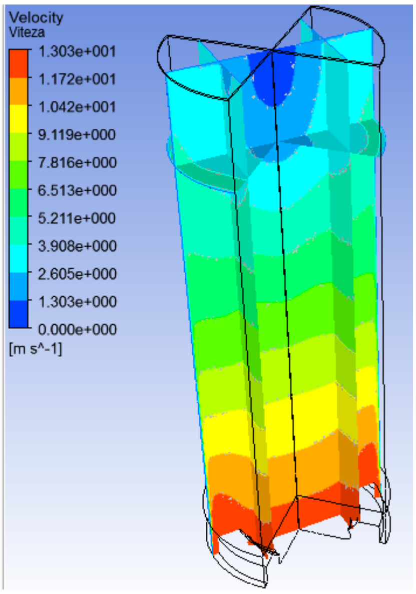


Fig.15-Velocity fields for CA=450 deg

The distribution of the velocities fields are pretty much the same as the previous reading, the peak value being 13 m/sec.

- **Temperature fields**

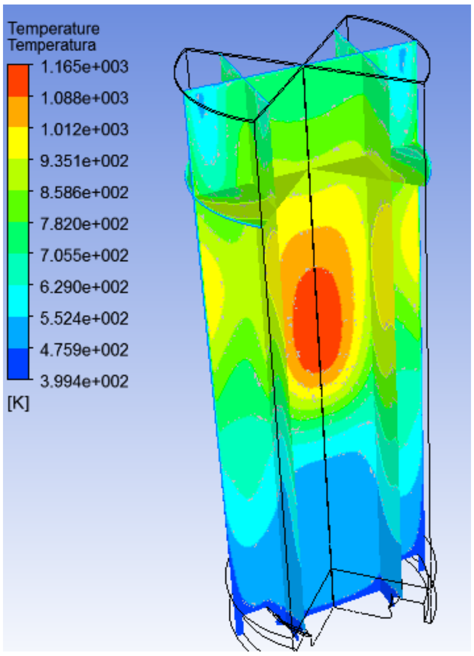


Fig.16-Temperature fields for CA=450 deg

The temperature fields continues to elongate and cool, the peak value being calculated in the same upper-central zone of the chamber as seen above. The maximum value now is 1165 deg K.

- **Turbulence kinetic energy fields**

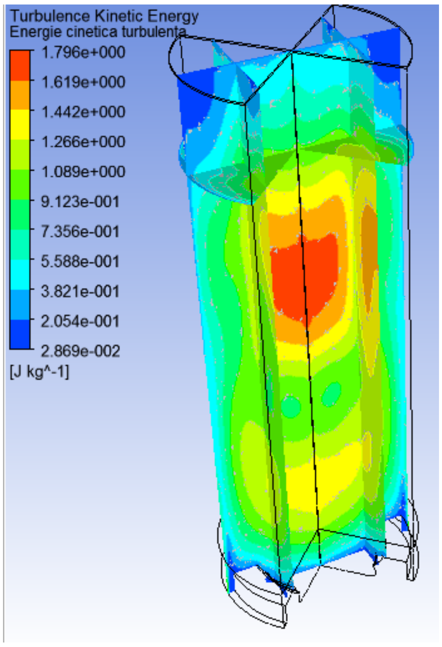
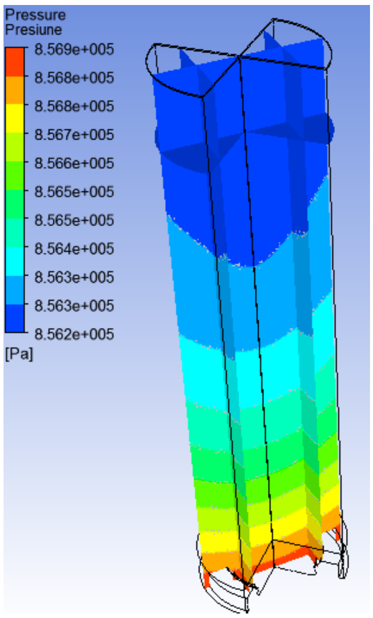


Fig.17-Turbulence kinetic energy fields for CA=450 deg

The turbulence fields now are confined into the central space of the chamber, and is decreasing to a maximum of 1.79 J/kg.

Results for CA = 480 (deg)-Combustion stage ending

- **Pressure fields**



DOI: 10.21279/1454-864X-16-I1-053

"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XIX – 2016 – Issue 1
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania // The journal is indexed in:
PROQUEST / DOAJ / DRJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Crossref /
Academic Keys / ROAD Open Access / OAJI / Academic Resources / Scientific Indexing Services / SCIPIO

Fig.18 Pressure fields for CA=480 deg

Immediately after this point the exhaust valve placed on the top of the cylinder will open and the scavenging process will follow. The pressure inside the chamber reaches a maximum of 8.56e5 Pa.

- **Velocity fields**

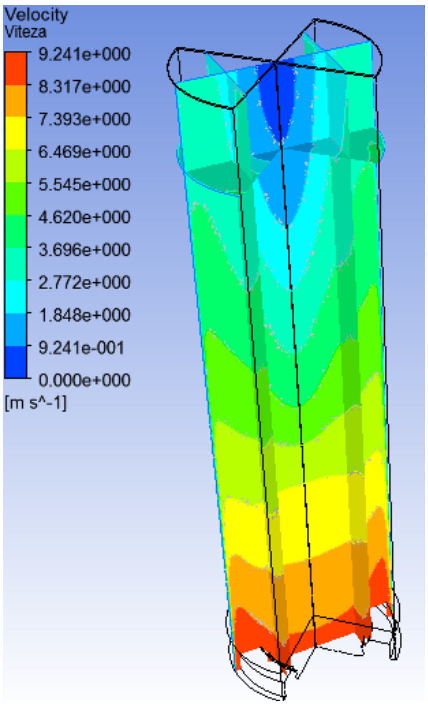


Fig.19-Velocity fields for CA=480 deg

The top velocity values of 9.24 m/sec will be recorded inside the top-lower half of the chamber, near the walls and in the center as seen in the figure above.

- **Temperature fields**

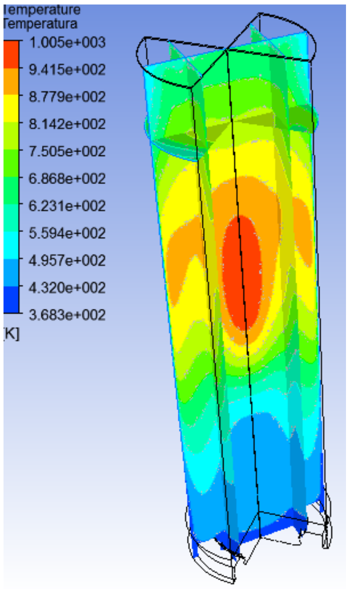


Fig.20-Temperature fields for CA=480 deg

The burnt gases temperature have their peak value of 1005 deg K at the top central zone of the chamber. The calculated **Average temperature** is 693 deg K. This average value will be used for validating the model.

- **Turbulence kinetic energy fields**

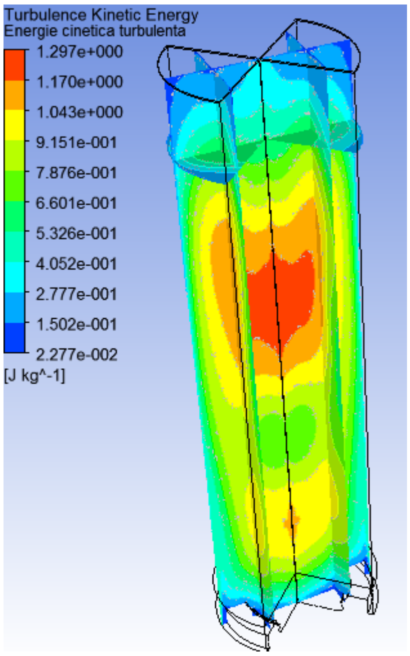


Fig.21-Turbulence kinetic energy fields for CA=480 deg

The turbulence is fading away, the turbulent zone is narrowing and the peak value is 1.297 J/kg.

Model Validation

The model validation are taking into account the results gathered from the sea trials of the oil carrier SCORPIO with 105.000 TDW in March

DOI: 10.21279/1454-864X-16-I1-053

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 combustion [Pa]	856800	891000	-3.838
Temperature end combustion [grdK]	693,400	688.15	+0.763

Table 2- Model validation

As seen above the pressure field model at the end of combustion is quite accurate, the error being only -3.8%.
The final combustion temperature of the burnt gases therefore seem to be accurate and credible.

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 were rather scarce in terms of real processes taking place in the combustion stage, but at the end we managed to have a full picture of the main parameters evolution during the combustion phase 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, Minrou Wada and Buhei Kobayashi. Develope-ment of a low emission two-stroke cycle engine. SAE paper no. 980761, 1998.
[7] ANSYS 15.0 Help Library
[8] Sourabh Shrivastava, Padmesh Mandloi, Ajey Walavalkar-Modeling IC Engine Thermal Management using ANSYS CFD-ANSYS Inc.

DOI: 10.21279/1454-864X-16-I1-053