Journal of POLISH CIMAC

Faculty of Ocean Engineering & Ship Technology
GDAŃSK UNIVERSITY OF TECHNOLOGY

3D MESH MODEL FOR RANS NUMERICAL RESEARCH ON MARINE 4-STROKE ENGINE

Jerzy Kowalski

Gdynia Maritime University
Department of Engineering Sciences
Morska Street 81-87, 81-225 Gdynia, Poland
tel.: +48 58 6901434, fax: +48 58 6901399
e-mail: jerzy95@am.gdynia.pl

Piotr Jaworski

Warsaw University of Technology, Institute of Heat Engineering
ul. Nowowiejska 21/25, 00-655 Warsaw, Poland
tel.: +48 22 234 52 41
e-mail: piotr.jaworski@itc.pw.edu.pl

Abstract

The article consist the 3d mesh analysis prepared for simulation of the processes in combustion chamber of marine compression ignition engine. The three moving meshes models where prepared: A – mesh for engine cycle work simulation; B – mesh of combustion chamber volume for work stroke simulations, no valves included; C- mesh of combustion chamber including mountings screw whole for work stroke simulations, no valves including. Prepared mesh where used for numerical simulations of injection and combustion processes in engine combustion chamber. Type C model, even if the total number of cells is lower in comparison to B model, result in calculation time increase. B and C models are solution for fast and robust validation of injection and auto ignition model parameters. Type A model is only one suitable for full cycle simulation. Only with accurate initial and boundary conditions the qualitative results of the injection, mixing and combustion process can be obtain on mesh type B and C.

Key works: *CFD, RANS, marine engine, moving mesh, 3d mesh*

1. Introduction

Nowadays and future regulations regarding environment protection and combustion emission products limitations are source of a need to design engine constructions with higher efficiency. Solution to decrease the research and development costs is to use computer fluid dynamics (CFD) technics, which is an effective tool for analyse and verification of the fluid flow and combustion processes in piston engine. According to Drake et al. [1], dynamic development of CFD methods in modelling of the piston engine processes started with the development of port fuel injection piston engines. The processes of the energy conversion to mechanical work from combustion in piston engine are complicated to describe numerically. In simulation there is a need for injection, mixing, ignition and combustion models verification. Numerical representation of the fluid flow, pressure temperature field, heat exchange in flow and on boundary wall need to be described also precisely. Assuming the number of thermodynamic processes which need to be defined in CFD and continues hardware development, simulation methods and models are also continuously developed. The newest injection and combustion models for compression ignition diesel driven engine are fully validated [2]. According to Collin et al. [3] today every new construction of the compression ignition engines is optimised by use of CFD methods.

Proper numerical analyse of engine processes needs preparation of the accurate representation of engine geometry, by mean of the finite volume elements (mesh) model. Also piston and valve movement need to be included. Mesh should be prepared together with the knowledge about simulated process. Still the final resulting mesh is an optimal solution for robust calculation in time. In presented paper the methodology of mesh preparation will be shown for marine compression ignition engine together with comparison of different solution for simulation purpose.

2. Methods of mesh preparation

Geometrical data about piston chamber and valves dimension together with data about piston and valves movement were prepared. The chosen compression ignition engine is from Maritime University laboratory, Sulzer A125/30, construction typical for marine purpose. Basic geometrical engine parameters are presented in Tab. 1.

Tab.1. AL25/30 Engine parameters

Parameters	Value	Unit
Max. electric power	250	kW
Rotational speed	750	rpm
Cylinder number	3	–
Cylinder bore	250	mm
Stroke	300	mm
Compression ratio	12,7	–
Injector nozzle		
Holes number	9	–
Holes diameter	0.325	mm
Holes position diameter	7	mm
Holes position angle	150	deg.
Spray cone angle	6	deg.
Opening pressure	25	MPa

Three moving mesh models where proposed for the purpose of numerical simulations:

A – mesh for full engine cycle simulation presented on Fig. 1,

B – assuming axis symmetry of combustion chamber, moving mesh representing combustion chamber volume for one injector hole was prepared Fig. 2a,

C – assuming axis symmetry of combustion chamber, moving mesh representing combustion chamber with injector and mounting screw elements volume for one injector hole was prepared Fig. 2b.

Mesh A, presented on Fig. 1 where prepared for whole engine cycle, 720°. It was done with use of AVL Fire software, “Fame Engine Plus” (FEM+) module. Geometrical models were drawn for every engine stroke on fluid side. As the results four geometrical fluid models, representing exhaust, scavenging, intake, compression and combustion stroke were imported in to the Fire workflow manager. The reason for different models preparation is the result of assumption that after the valves are closed the simulation of the fluid flow in intake or exhaust channels can be omitted. It also decreases significantly calculation time.

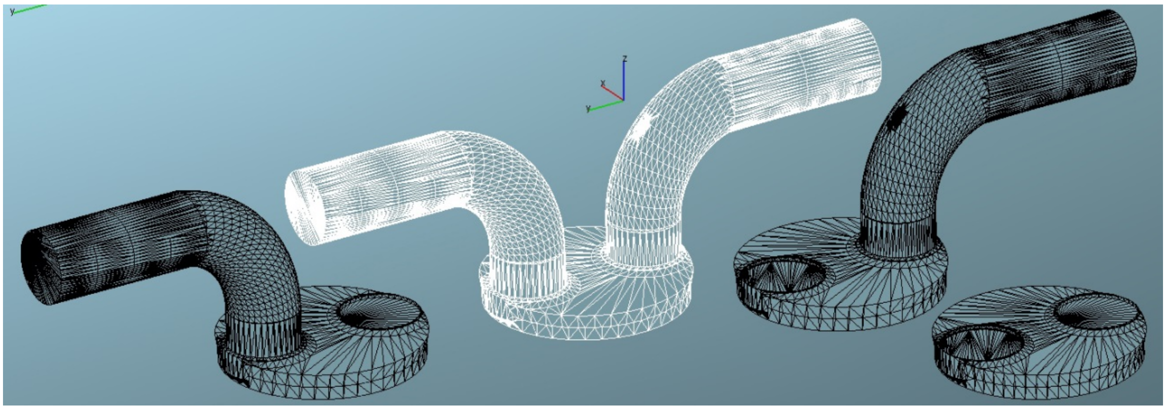

Fig.1. 3D geometrical models for whole engine cycle calculation.

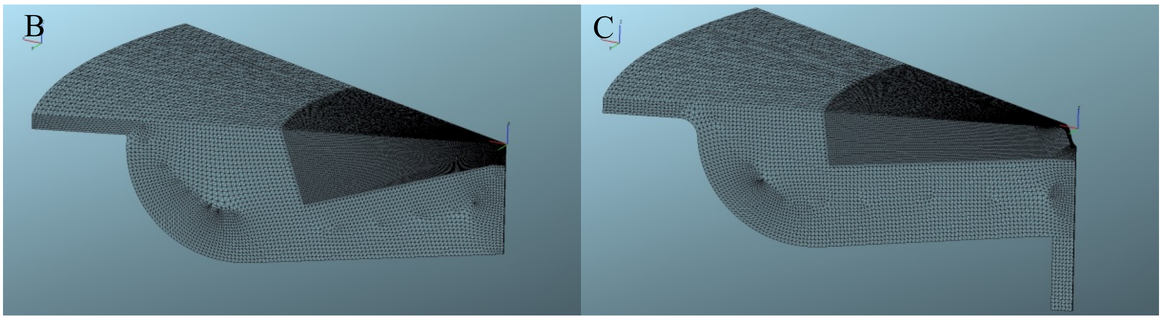

Fig.2. 3D geometrical models of combustion chamber, type B and C.

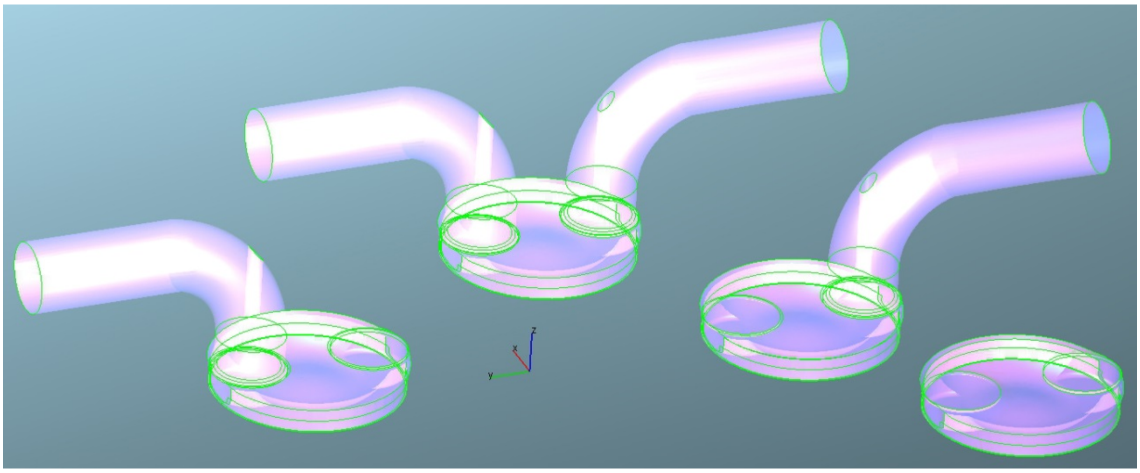

Fig.3. Edges representation for type A geometrical model

On Fig. 3 the edges (green) representation for type A geometrical model is shown. Mesh calculated in FEM+ module is based on geometrical and edge input data. Edges are elements for proper definitions of surface cross section. Piston movement where conducted with use of crankshaft geometrical data by methodology presented by Heywood [5]. Valves movement where prepared by construction analysis of laboratory engine. For proper movement representation also selections on movement elements are defined. On Fig. 4, the piston and valves selection for non-moving, buffer and moving elements is presented. Proper choice of movement definitions is key element for robust and fine mesh generation.

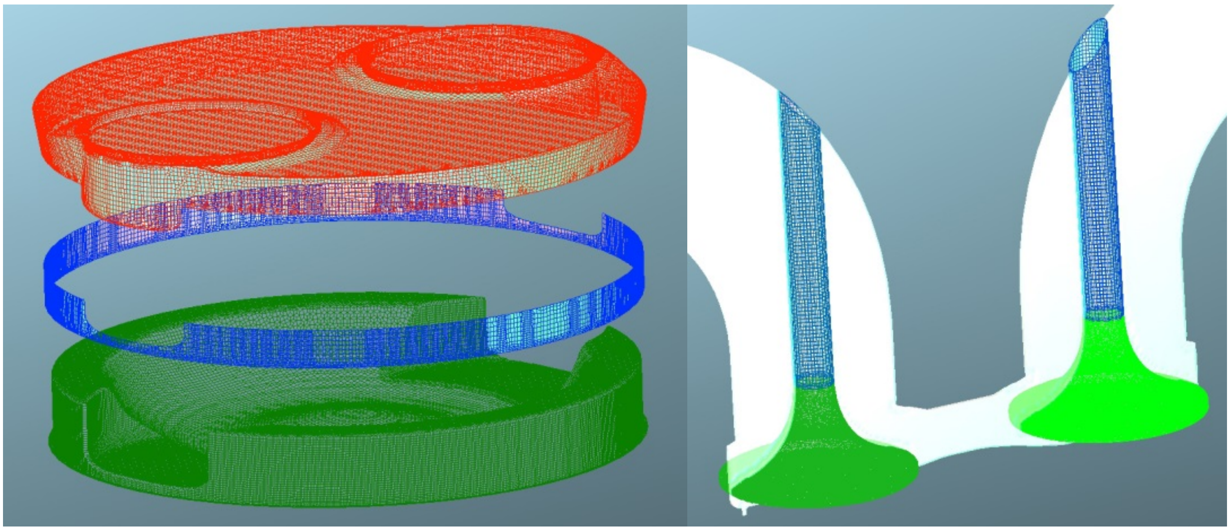

Fig.4. Selection for mesh movement description on type A geometrical model

Calculated mesh model consist of 500 000 cells at work stroke and 1 500 000 cells at scavenging stroke. Base cells size is 2 [mm] minimum and 8 [mm] maximum. Cells size should be defined by geometrical and simulated process complicity.

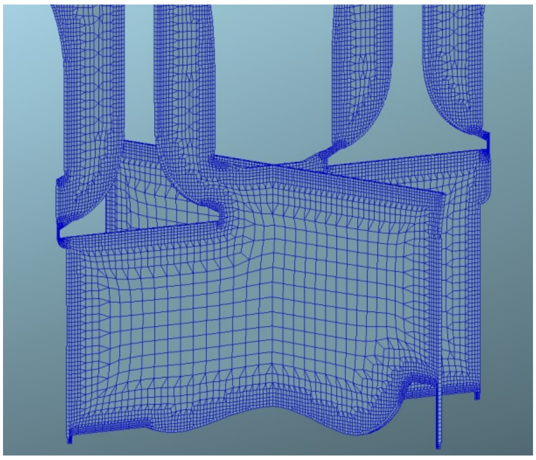

Fig.5. Mesh cross-section for type A model

Flow velocity, mixture preparation and combustion processes scale related to combustion chamber complicity are the key parameters to decrease the cells characteristic size. Simultaneously, smaller cells lead to increase of the total number of cells and calculation time. To fulfil those statements the maximum cell size is limited to 2 [mm] for injection and combustion period and 0.125 [mm] for valves ports during the opening and closing crank angle degree. On Fig. 5 it is shown the mesh cross section view for scavenging stroke. Piston and liner includes also cut-off for valve opening. On Fig. 6 it is shown that the distance between open valve and liner cut-off surface is around 1 [mm] and distance between open valve and piston cut-off surface and top dead centre position is 2 [mm]. Mesh resolution for such area is also refined to avoid convergence problem because of the velocity increase.

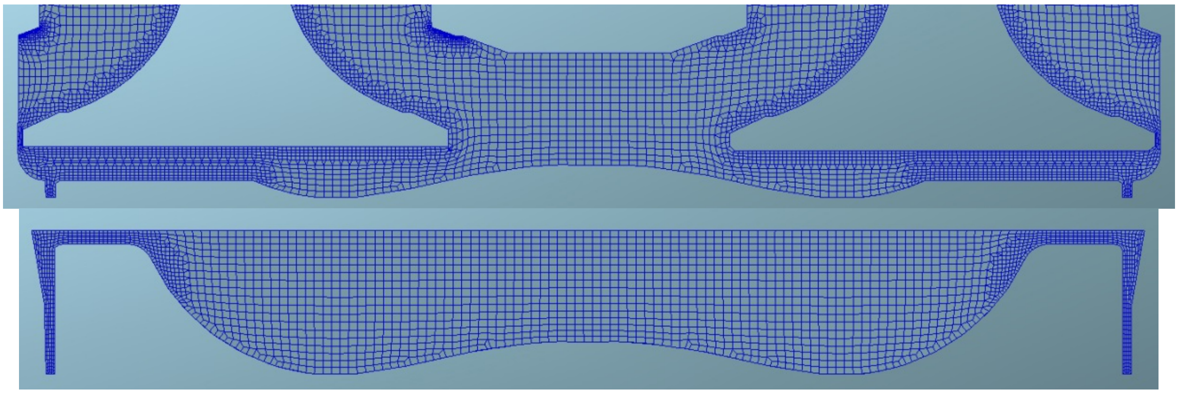

Fig.6. Mesh type A cross section for TDC position at scavenging and work stroke

Valves and piston surface engine elements which distance between the liner surface are from 1 to 2 [mm] and the surface move is parallel to each other can cause cells skewness. To avoid such result, which can cause negative cells preparation, not only cell size need to be refined (0.25 [mm]), but also mesh movement resolution need to be decrease to 0.5°.

Models type B and C were prepared with use of ESE Diesel tool form AVL Fire software. Models are based on assumption that geometrical and fluid flow characteristic is axis symmetric. Also it is possible to divide the combustion chamber volume according to angle symmetry of the injector nozzle holes. It results with model covering 40° of the combustion volume chamber for one nozzle injector hole. Methodology to prepare such mesh is simpler and it is based on the surface regular mesh rotate by 40° and 25 divisions. Models type B and C were divided in to two areas, injection area with 0.5 [mm] size and the rest volume with 1 [mm] size. For those models the valve movement is not described, also it is assumed that piston/liner cut-off and clearance between piston and liner is neglected.

For B type model the injector nozzle and mounting screw whole is also neglected.

3. Results

Calculations with all considered grids require the input of initial and boundary conditions data as well as the fuel injection parameters. Mentioned data were collected during laboratory tests, which the course and the results are presented in [5]. In addition, it is necessary to implement models of fuel injection, fuel spray brake-up, evaporation and combustion. The analysis of these problems is presented in the work [6]. It should be noted that the spatial geometry of the B and C meshes does not allow to model the full cycle of the engine operation. Omission of the valve geometry does not allow the modeling of the cylinder scavenging.

The computation time depends largely on the computing power of the computer, the number and complexity of solved equations, consisting of the combustion process and the size and complexity of the spatial grids. It is important problem because the calculation of one the engine cycle of 720° crankshaft angle (CA) with the single processor computer (4-core processor with the 3.4 GHz clock frequency, 16Gb of RAM memory and 64-bit operating system) take about 250 hours.

The Tab.2 presents a summary of the built grids parameters and the results of calculation time for the same input data. The calculations were carried out for the compression and combustion processes in the engine cylinder, starting from 140° CA before top dead center of the piston (TDC) to 130° CA after TDC. The presented calculations were performed on the same computer set and using the same equations and input data. Calculation time was measured using the internal software module of the Fire AVL package.

Tab.2. Calculation time

Grid	A	B	C
Number of cells [thousands]	500 – 1500	317 – 396	257 – 360
Accumulated sum of iterations	25110	46458	47604
Mean CPU time per iteration [s]	11,59	10,67	16,05
Combustion	3,12	1,29	3,54
Emission	0,17	0,09	0,23
Energy	0,33	0,17	0,47
Momentum	0,49	0,24	0,64
Pressure	3,34	1,01	2,09
Spray	0,14	5,78	3,63
Thermo-Chemistry	2,83	1,52	4,14
Turbulence	0,69	0,40	0,97

According to the presented results, average time for one calculation iteration is the longest for the C grid and the shortest for the B grid. It means that calculation time is largely dependent on the shape and structure of the individual cells of the mesh. It should be noted that the overall computation time is also influenced by the number of calculated iterations. According to the presented results, the best convergence of numerical calculations is obtained for grid A.

Tab.3. The iteration number for characteristic steps of calculations

Grid		A	B	C
Start of calculations – first 1°CA	Steps	100	10	10
	iterations/step	19,9	74,3	75,2
Compression – next 119°CA	Steps	151	119	119
	iterations/step	73,6	94,5	50,8
Start of fuel injection – next 28°CA	Steps	240	240	240
	iterations/step	22,5	98,0	94,9
Combustion – next 106,5°CA	Steps	213	221	221
	iterations/step	16,3	64,5	66,0
Exhaust valve opening – next 20,5°CA	Steps	62	41	41
	iterations/step	50,7	55,9	84,1

As mentioned earlier, presented the average computation time is also apparent from the amount of implemented equations in the model. The average computation time for each model, which make up the model of the combustion process in the engine, for a single iteration is shown in the Tab.2. The presented results show that considered grids are optimal solutions for the selected computational models. Grid A is the best solution for the calculation of the fuel injection and grid B for the calculation of emissions, energy and turbulence phenomena. Increasing the accuracy of mapping the shape of the cylinder about the shape of the fuel injector in a C grid resulted in a significant increase in computation time for all considered model's equations of the combustion process. An additional effect was an increase in the number of iterations.

The size of the calculation steps for the characteristic engine operation phases expressed in the angle of CA are presented in the Tab.3. The average number of iterations which are needed to obtain a result of the assumed accuracy, per one step calculation, is also presented.

According to the presented results, increase of the calculation step leads to an increase in the number of iterations. It must be remembered that the use of the excessive computational step size, decreases the convergence of the calculation, leading to an increase in the number of iterations.

This conduct may result in a significant increase in computation time and thus counterproductive. An example of this is the average number of iterations for the opening of the engine exhaust valve. Not implemented valves geometry in the B and C grids, allowed reducing the amount of calculation steps. Despite this, there has been an increase in the average number of iterations for one step. As a result, the computation time has not reduced significantly.

It should be remembered that the analysis of the optimal choice of the grid should also be conducted based on the validation of computational results obtained with the measured data. Calculations with B and C grids cannot be prepared for subsequent cycles of the combustion process. The advantage of these grids is the ability to quickly tune the boundary conditions and the parameters of the models of the combustion process. We can conclude that the B and C moving grid, based on the axial symmetry assumption are not consistent with the real model, but they can give a quick solution, but only in the case of a regular structure. Grids of this type may be a prelude to the calculation of the A grid.

It should also be noted that the advantage of the A grid is fully possible to estimate the velocity field, pressure and temperature at the start of the compression stroke with consideration of the fluid flow in a perpendicular direction to the cylinder axis.

4. Conclusions

The article consist the 3d mesh analysis prepared for simulation of the processes in combustion chamber of marine compression ignition engine. The three moving meshes models where prepared: A – mesh for engine cycle work simulation; B – mesh of combustion chamber volume for work stroke simulations, no valves included; C- mesh of combustion chamber including mountings screw whole for work stroke simulations, no valves including. Prepared mesh where used for numerical simulations of injection and combustion processes in engine combustion chamber. Type C model, even if the total number of cells is lower in comparison to B model, result in calculation time increase. B and C models are solution for fast and robust validation of injection and auto ignition model parameters. Type A model is only one suitable for full cycle simulation. Only with accurate initial and boundary conditions the qualitative results of the injection, mixing and combustion process can be obtain on mesh type B and C.

Acknowledgments

The project was supported by the National Science Centre in Poland, granted on the basis of decision No. DEC-2011/01/D/ST8/07142

The project was supported by AVL Company according to University Partnership Program and license of AVL Fire software

References

[1] Drake M.C., Haworth D.C., *Advanced gasoline engine development using optical diagnostics and numerical modeling*, Proceedings of the Combustion Institute, Vol. 31, pp. 99-124, 2007.

[2] Priesching P., Ramusch G., Ruetz J., Tatschl R., *3D-CFD Modeling of Conventional and Alternative Diesel Combustion and Pollutant Formation – A Validation Study*, SAE Technical Paper 2007-01-1907, 2007.

[3] Colin O., Benkeida A., *The 3-Zones Extended Coherent Flame Model (ECFM3Z) for Computing Premixed/Diffusion Combustion*, Oil & Gas Science and Technology, Vol. 59, No.6, pp.593-609, 2004.

[4] Devesa A., Moreau J., Helie J., Faivre V., Poinsot T., *Initial conditions for Large Eddy Simulations of piston engine flows*, Computers & Fluids, Vol. 36 pp. 701-713, 2007.

[5] Heywood J. B., *Internal Combustion Engine Fundamentals*, McGraw-Hill 1988.
[6] Kowalski J., *Laboratory study on influence of the exhaust duct throttling on exhaust gas composition in marine four-stroke diesel engine*, Journal of Polish CIMAC Vol. 7, No 1. pp. 109-115, Gdańsk 2012.
[7] Kowalski J., *Assessment of the 3-D combustion model in the marine 4-stroke engine*, Journal of Polish CIMAC Vol. 9, 2014. (in press)