Modeling of Flow through Cooling Plant with Sea Water

Prof. PhD.Eng. **Mariana PANAITESCU**[1], Prof. PhD.Eng. **Fanel-Viorel PANAITESCU**[2],
Master Degree Student **Gabriel TUDOSE**[3]

[1] Constanta Maritime University, marianapan@yahoo.com

[2] Constanta Maritime University, viopanaitescu@yahoo.ro

[3] Constanta Maritime University, tudosegabriel74@gmail.com

Abstract: *This paper aims at modeling of fluid flow through the installation of sea-water cooling system of an engine. The overall goal is to optimize the flow by modifying driving pipes and decreasing hydraulic losses. It has created a three-dimensional model to reproduce as accurately as the real facility using ANSYS FLUENT 13.0. There were calculated the variation of pressure and flow inside the plant. Flow model chosen is k-ε model. The results are presented graphically. In order to optimize the flow were analyzed the hydraulic losses both analytical and simulation in ANSYS FLUENT 13.0.The differences obtained ranged between 9...15%. For a result as close as possible to reality, the losses of pressure and flow rate variation are measured on a pipe segment as bounded. In practice, this study will help to correct the operation of the plant and its aggregates.*

Keywords: *Modeling, flow, cooling plant, sea, water, loss, pressure, simulation.*

1. Introduction

Cooling plant with sea water, contains a whole series of elements, by their construction, the material used and the types of pumps, leading to reliable operation of all equipments in the machine compartment, interdependent towards some.
Cooling facility is provided so many filtering systems back-up: main pumps, an emergency pump, coolers, valves "one-way", valves type "Butterfly", pressure and temperature transmitters (Fig. 1)[1, 2].

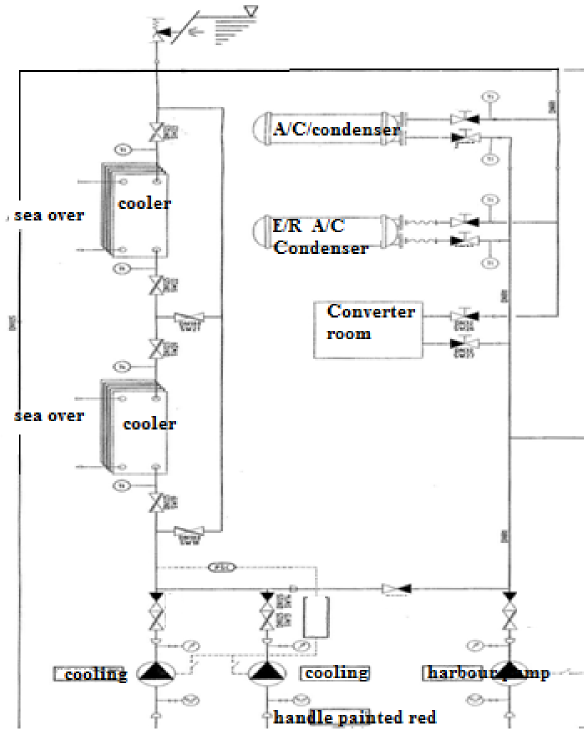

Fig. 1. Cooling plant scheme [2]

Pumps used are: water pump (Q = 125 m^3/h), fire and bilge pump, bilge pump and ancillary installations (Q = 5 m^3/h), foam pump (Q = 210 m^3/h), the ejector pump (Q = 36 m^3/h) and general service pump (Q = 60 m^3/h).

2. Method and research

For the modeling of fluid flow in the cooling plant ANSYS FLUENT 13.0 has been used [3,4].

2.1 Choice of model and mesh scope study

Build a three-dimensional model to reproduce as accurately as possible the real facility. The principle of construction itself was to merge some geometric shapes (square, sphere, cylinder, truncated cone) (Fig. 2) the initial diameter of the pipes is 100 mm, it is likely to drop to 60 mm by means of reductions [5,6]. After we have chosen the desired geometric form, we can define the whole beach of dimensions.

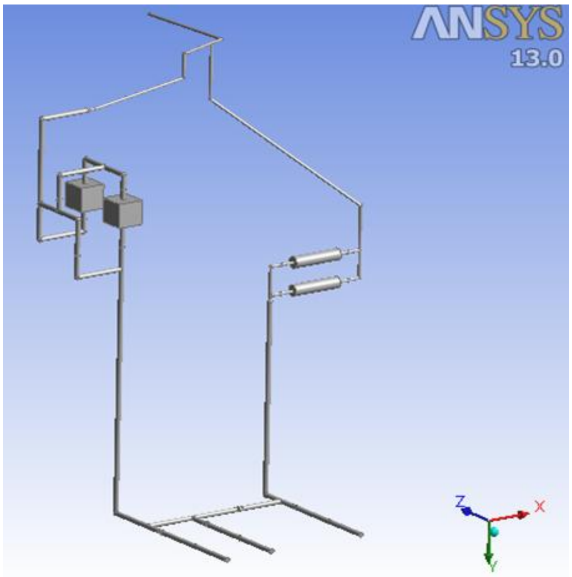

Fig. 2. Sea water cooling plant scheme in 3D [4, 5]

2.2 Mesh fields of study

There are meshing areas of plant study: pipes, valves, cooling (Fig. 3, Fig. 4).

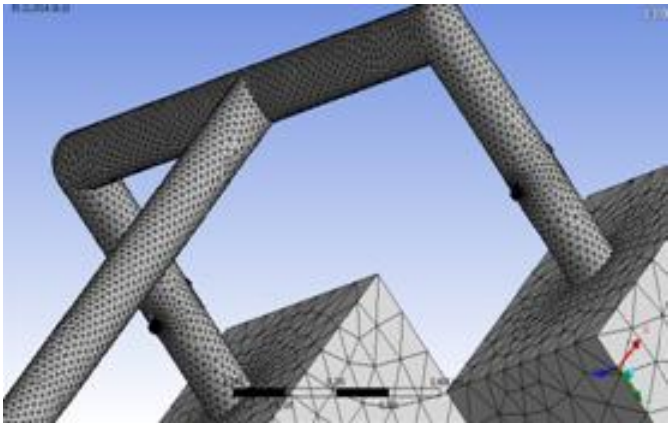

Fig. 3. Mesh of drive pipes

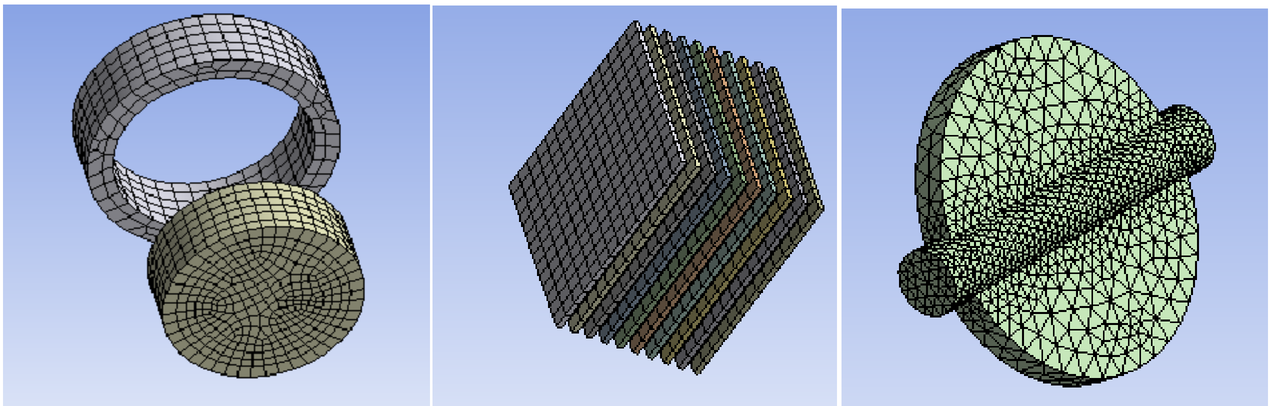

Fig. 4. Mesh of one-way valve, cooler and "butterfly" valve

2.3 Setting water flow parameters

Select the following variables: the working fluid (seawater), density (1015 kg/m^3), the acceleration due to gravity (9.81 m^2 /s), kinematics viscosity constant (0.001003 kg/m.s), mass flow (32.24 kg/s, respectively 22.5 kg/s, absolute speed, flow model k- ε,, transitional fluid, fluid discharge in atmosphere.

3. Results and interpretations

Further analyze the graphical results for chosen facility. In the following figures (Fig. 5, Fig. 6, and Fig. 7) can be seen falling pressure and speed changes that take place as the fluid flows through the pipes. Converges solutions aimed at 537 iterations.

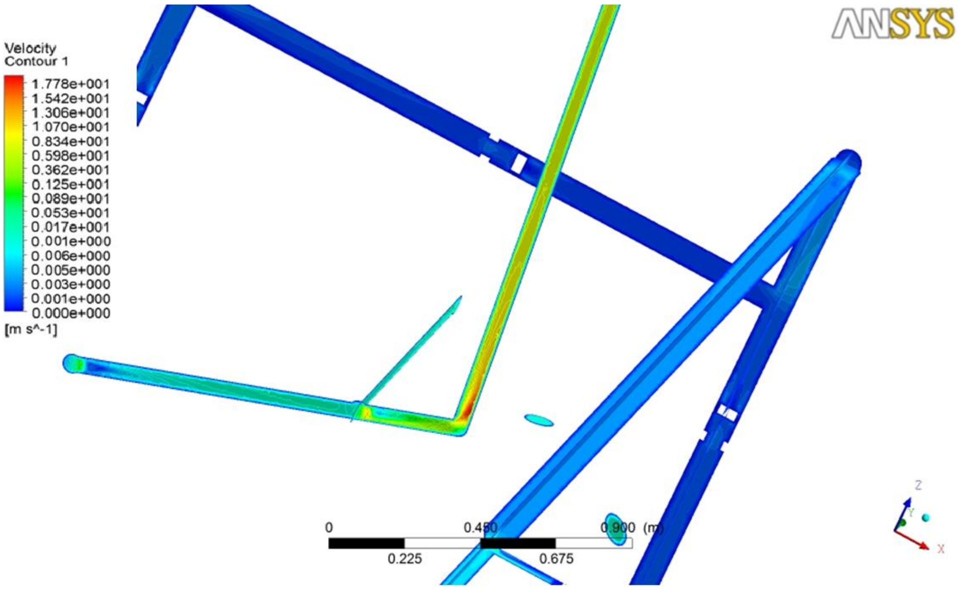

Fig. 5. Maximal speed through the pipes of plant

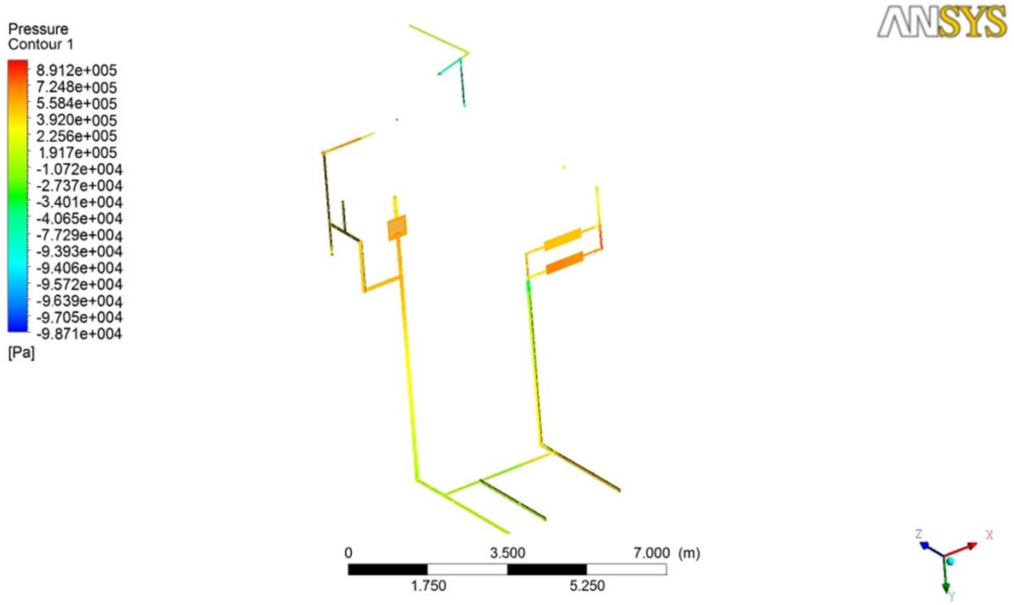

Fig. 6. Falling pressure through the pipes of plant

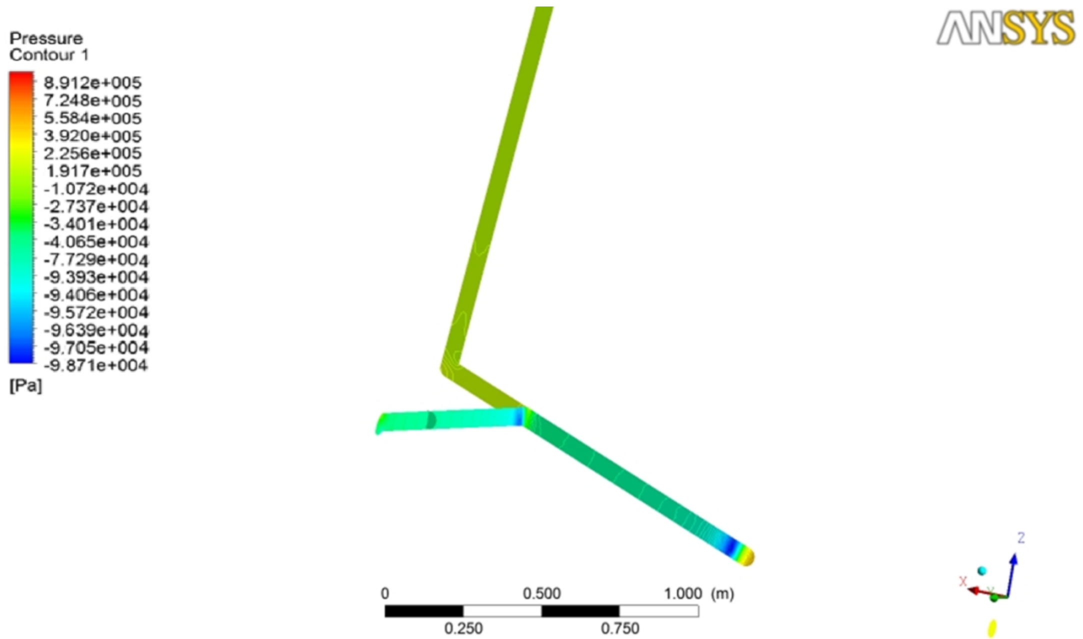

Fig. 7. The partial driving pipes with maximal falling pressure

In order to optimize the flow were analyzed the plant’s losses, both analytical and hydraulic simulation in ANSYS FLUENT 13.0 (Table 1).
There have been compared hydraulic losses on several sections of pipes (1, 2, 3), on elbows and valves. After this, the total losses have been calculated and compared.

Table 1: Comparative study for hydraulic losses

Hydraulic loss	Calculated Value [Pa]	ANSYS FLUENT Value [Pa]
Loss section 1	$0.44*10^5$	$0.65*10^5$
Loss section 2	$0.89*10^5$	$0.93*10^5$
Loss section 3	$0.14*10^4$	$0.12*10^4$
Total linear losses	$1.34*10^5$	$1.59*10^5$
Elbow loss on 90^0	$1.66*10^5$	$1.54*10^5$
Loss on "butterfly" valves	$1.83*10^5$	$1.62*10^5$
Loss on "one-way" valves	$0.70*10^5$	$0.93*10^5$
Total local losses	$8.02*10^5$	$8.91*10^5$

The values obtained using ANSYS FLUENT 13.0 are not identical, but are plausible. For precision of study, pressure losses or cha nges in flow rate shall be measured on a limited sections of pipes.

4. Conclusions

For analysis of graphics were inserted various sections of the plane built, observing thus eddy currents. ANSYS FLUENT values were taken with the option "variable sampling" in the points of interest.
The main purpose of this work was to carry out the construction of a segment of the plant as close to the real.
It has also pursued the fluid flow around fittings, elbows's T, taking into account linear and local losses of pressure.
The margin of error was received between 8-15%, about what it means that the estimates were done correctly.

Acknowledgments

Acknowledgments of the crew and management team from LS Jamie Oil Tanker.

References

[1] A. Costică, Machines and plants for naval propulsion" ("Mașini și instalații navale de propulsie"), Technical Publishing House, Bucharest, 1991;
[2] http://www.marinetraffic.com/pl/ais/details/ships/236507000;
[3] http://148.204.81.206/Ansys/150/ANSYS%20Fluent%20Tutorial%20Guide.pdf;
[4] http://www.ansys.com/Products/Fluids/ANSYS-Fluent;
[5] D. Dinu, "Fluids Mechanics for seafarers" ("Mecanica Fluidelor pentru navigatori"), NAUTICA Publishing House, Constanta, 2010;
[6] M. Panaitescu, F.V. Panaitescu, „Fluids Mechanics and Hydraulic machines for Environmental Engineering" („Mecanica Fluidelor si Masin Hidraulice pentru ingineria mediului"), volume 1, 2, NAUTICA Publishing House, Constanta, 2016.