Fluid structure interaction analysis of a naval economizer

C Faitar, N Buzbuchi, L C Stan*
Maritime University of Constanța- România, Faculty of Naval Electromechanics, Department Electromechanics, Blvd. Mîroslav, No 104, 900663, Constanța, România
liviustan14@yahoo.com

Abstract. By definition the economizers are those devices able to exchange the heat between two fluids and thus increasing the enthalpy of the useful fluid. In the naval domain the economizers are used in order to save energy by means of scavenging the heat of the exhaust gases coming from the main engine of the ship. The purpose of this paper is to analyze by using the advanced features of ANSYS and with finite elements the structural behavior of a naval economizer low pressure tube using the Fluid Structure Interaction technology. This paper is a numerical investigation of the complex processes of feed water vaporisation inside the naval economizer tubes. The biphasic flow was modeled by using the top of the art new technology ANSYS RPI developed for vaporisation processes inside the heat exchanger tubes and the results were successfully validated against the recorded data. This model furthermore was used as a numerical platform for the calculations of Fluid Structure Interaction with the conclusion that major stress concentrators are acting on the tube bends requiring tube replacement after breaking.

1. Introduction

By definition the economizers are those devices able to exchange the heat between two fluids and thus increasing the enthalpy of the useful fluid. In the naval domain the economizers are used in order to save energy by means of scavenging the heat of the exhaust gases coming from the main engine of the ship.

They are placed inside the ship chimney with indirect contact between the exhaust gases and the fluid. Usually they are placed inside the chimney having indirect contact between the useful fluid and the exhaust gases recovering thus the residual heat of the combustion by-products.

The relatively high temperature compared to the cooling water temperature is the driving force for recovering the energy contained inside the exhaust gases.

The economizers have some distinct treats:

- The heat exchange is done only by convection, due to the moderate exhaust gas temperature, but not less than 200°-250°C;
- at the economizer outlet the gas temperature must exceed the agent temperature by 30° - 40°C, in order not to increase the heat exchange surface without purpose;
- the outlet temperature of the economizer is advisable not be less than 160 °C to 170 °C in order to avoid acid dew temperature, mostly when the engine is running on heavy fuel oil with a certain high percentage of sulphur and thus to avoid corrosion the surfaces of the channels through which the gases circulate;
- they are not supposed not to endanger the normal operation of the engine; the resistance the maximum calibre gas dynamics must be less than 250 mm water column on the two-stroke engine and 400 mm water column on the four-stroke engine. The heat flow thus recovered by economizer
can be used for production of saturated water vapor for fuel heating hard and ship needs and the production of superheated water vapor for the purpose of supplying a turbo-generator for power generation.

On board any ship and in order to increase the efficiency of internal combustion engine the economizers are used, which, from a constructive point of view, can have all the elements of an aqua-tubular heat, which is no longer necessary for the outbreak combustion of fuel. Onboard ships the economizer is placed inside an arrangement as seen in Fig. 1.

The purpose of this paperwork is to analyse by using the advanced features of ANSYS and with finite elements the structural behaviour of a naval economizer low pressure tube using the Fluid Stricture Interaction technology.

![Fig.1 Economizer arrangement onboard ships](image)

2. MATERIALS AND METHODS

2.1. Economizer description

This study via CFD (Computer Fluid Dynamics) study is approaching real economizer placed on a tanker of 305,000 MT deadweight with the following parameters:

| EXH. GAS ECONOMISER | DUAL STEM PRESSURE / 1 SET |
|---------------------|-----------------------------|
| TYPE/ NO. OF SET    | 5909/9800/3682 mm            |
| SIZE                |                             |
| HEATING SURFACE     | 4345.0 m2                   |
| EVAPORATION MAX/NOR | 18210/13540 kg/h            |
| PRESSURE MAX/NOR    | 2.16/0.69 MPa               |
| TEMPERATURE MAX/NOR | 282/263                     |
| WEIGHT kt           | 92                          |

Table 1 Economizer parameters

Inside the economizer there are 18 rows of tubes with the external diameter of 38.1 mm and 3.5 mm wall thickness, made of steel, with water inside being washed by a stream of burnt gases coming from the main engine of the ship. It has three stages: Low- and High-pressure stages and one stage of the Superheater.

The working parameters for any and each stage are shown in the Table below:
| Parameter                       | MU | Low pressure stage | High pressure stage | Superheater |
|---------------------------------|----|--------------------|---------------------|-------------|
| Working pressure                | MPa| 0.29               | 0.74                | 0.69        |
| Produced Steam temperature      | °C | Saturated          | Saturated           | 245         |
| Steam flow                      | Kg/h| 2420               | 5710                | 5410        |
| Burnt gases flow                | Kg/h| 179800             |                     |             |
| Burnt gases temp                | °C | 263                |                     |             |
| Inlet water temp                | °C | 130                |                     |             |
| Heating surface                 | m² | 1612               | 2579                | 154         |

Table 2 Working parameters

The economizer has the dimensions: 9.8 m tall and 5.9 m wide being a massive piece of equipment which is not easy to simulate with finite volume due to its excessive big dimensions. Thus the CFD simulation strategy is to simplify the model by selecting one row of pressure tubes comprising one single row of tubes for every stage as seen in Figure 2 by slicing in vertical direction the economizer and placing symmetry conditions on the new formed surfaces inside the CFD model. The width of the slice should be 1/18 the width of the economizer corresponding to each such a vertical row of stages.

The inlet water stream inside the tubes is in even current for the Low and High stages meaning that if the burnt gases are circulating from down to up direction and the water inside the tubes is circulating from down to up direction. The Superheater is in counter-current the water is circulating from up to down direction.
2.2 The CAD Model

The CAD model was generated inside ANSYS DesignModeler module, the burnt gas fluid domain surrounding the water pressure tubes with the three stages of Low and High pressure and the Superheater defined with the dimensions exactly corresponding to the real case. As seen in Figure 3 the Low Pressure (LP) and High Pressure (HP) stages have 2 tubes with a crisscross direction while the Superheater (Super) has just one tube.

2.3 The Fluid Stricture Interaction Model (FSI)

For the thermo-hydrodynamic calculation the mesh is comprising 6 fluid domains (2 for LP stage, 2 for HP stage, 1 for Super stage and 1 for the Gas region) made out of 1,033,394 finite volumes and 354,486 nodes as seen in the Figure 4.

The boundary conditions are given in the Table 3 and Figure 5 and they are:

- In the burnt gas fluid domain the inlet and the outlet of the gas were imposed as InletGas and OutletGas plus a supplementary symmetry condition imposed on the sliced new surfaces.
- In the Low-pressure fluid domain for each tube (LP1 and LP2) the inlet and the outlet of the feed water were imposed as InletLP1, OutletLP1 and InletLP2, OutletLP2 with values from Table 3. The fluid is flowing in the same direction with the gas.
- In the Superheater fluid domain, the inlet and the outlet of the feed water were imposed as InletSuper, OutletSuper with values from Table 3. The fluid is flowing in the opposed direction with the gas.

| Fluid domain | Flow (on each tube) [m$^3$/s] | Feed water speed [m/s] |
|--------------|-------------------------------|------------------------|
| Gas          | 0.05 x 10$^5$                 | 55                     |
| LP1          | 0.77 x 10$^5$                 | 0.28                   |
| LP2          | 0.77 x 10$^5$                 | 0.28                   |
| HP1          | 2 x 10$^7$                    | 0.73                   |
| HP2          | 2 x 10$^7$                    | 0.73                   |

Fig.3 The CAD model of the economizer “slice”
The model used for tube vaporisation simulation is the newly released ANSYS RPI model. After calculating the pressure fields inside the tubes one Low Pressure is taken with the purpose of calculating the stresses and displacements of such a tub subjected to the calculated internal pressure. The tube is given in the Figure 5 below:
For the tube we have 72105 tri-dimensional solid finite elements (most of them Hex20) with 505790 nodes.

The pressures acting on the tube walls will be imported from the CFX module and are given in the figure 6.

![Imported pressures acting on the tube walls imported from the CFX module](image1)

Fig. 6 Imported pressures acting on the tube walls imported from the CFX module

In order to complete the mechanical model some other boundary mechanical conditions are imposed upon the mechanical model of the tube and are comprising Fixed supports and Frictionless supports as in the Figure 7.

![Mechanical boundary conditions imposed upon the tube walls](image2)

Fig. 7 Mechanical boundary conditions imposed upon the tube walls

![Fluid Pressure variation on the tube](image3)

Fig. 8 Fluid Pressure variation on the tube

The tube material is the structural steel with yield strength of 280 MPa, ultimate strength 480 MPa, Poisson ratio 0.3 and Young modulus 2.1 e11 Pa.
3. RESULTS AND DISCUSSION

As previously stated, the CFX module of ANSYS for the considered LP tube the following pressures were calculated on the tube walls (Figure 8):

The pressure is within the range of 286480 Pa and 292719 Pa. This pressure field was imported inside the mechanical model as shown in Figure 6.

- **The total deformation**

  Under the inside pressure and the boundary conditions influence the calculated deformation is shown in the Figure 9.

  ![Total deformation of the tube](image)

  The middle zone of the tube will suffer a total deformation with the magnitude of 184 mm and the first conclusion is that this tube arrangement is in need of some central supports usually supplied by the neighbouring tubes.

- **The equivalent elastic strain**

  The equivalent elastic strain fields are pinpointing the regions where the biggest stresses are expected as in Figure 10:

  The biggest value of the elastic strain is calculated as seen with the red label “Max” on the figure on the bent of the tube where big stress concentrators are acting. The value of 0.006 is bigger than the yielding value of 0.002 therefore in this region the material structural yielding will happen.

- **The equivalent von Mises stresses**

  In the above stated tube bent the yielding will occur so that the plastic behaviour of the material will re-distribute the stress inside the neighbouring material decreasing the local high value of 1286 MPa as seen in the Figure 10.

  The maximum stresses are calculated inside the tube bends where the yielding process occurs redistributing thus the stress inside the material. For better results an elastic-plastic approach should be considered.
In order to have a whole image of the tube structural integrity a fatigue calculation was further performed. The Mean Stress Correction Theory involved in this analysis is the classical Soderberg with 10% variation of the load as in the Figure 11:

With this hypothesis the maximum number of cycles for the tube representing its calculated life is 9.7e5 in the same location where the stress concentrator is located (Figure 12).
4. CONCLUSIONS

This paper is a numerical investigation of the complex processes of feed water vaporisation inside the naval economizer tubes. The biphasic flow was modeled by using the top of the art new technology ANSYS RPI developed for vaporisation processes inside the heat exchanger tubes and the results were successfully validated against the recorded data.

This model furthermore was used as a numerical platform for the calculations of Fluid Structure Interaction with the conclusion that major stress concentrators are acting on the tube bends requiring tube replacement after breaking.

References

[1] Faităr, C., Novac, I. A new approach on the upgrade of energetic system based on green energy. A complex comparative analysis of the EEDI and EEOI , ModTech International Conference Modern Technologies in Industrial Engineering 15-18 June, 2016, Iasi, Romania

[2] Faităr C., Novac, I. Basic aspects and contributions to the optimization of energy systems exploitation of a super tanker ship, ModTech International Conference Modern Technologies in Industrial Engineering 14-17 June 2017, Sibiu, Romania

[3] Baldi, F., Jhonson, H., Gabrielli, C., Anderson, K. Energy and Exergy Analysis of Ship Energy Systems – The Case study of a Chemical Tanker, Department of Shipping and Marine Technology, Chalmers University of Technology Gothenburg, Sweden, Received 25 September 2014, Accepted 17 April 2015; p.81-85

[4] Buzbuchi, N., Dragalina, Al., Manea, L., Moroianu, C., Dinescu, C. Motoare navale. Procese și caracteristici, Editura Didactică și Pedagogică, București, 1997

[5] Faităr Cătălin, Iordan Novac - Comparative analysis of energy efficiency indicators for a crude oil super-tanker ship, Constanta Maritime University Annals, Year XVII vol 24, Constanta 2016

[6] Faităr Cătălin, Iordan Novac- Basic aspects and contributions to the optimization of energy systems exploitation of a super tanker ship, ModTech International Conference Modern Technologies in Industrial Engineering 14-17 June 2017, Sibiu, Romania

[7] ANSYS 16 documentation

The safety factor for the tube is 0.996 therefore in that tube bent zone the tube might break requiring the replacement often enough (Figure 13). The most susceptible breaking points are inside the tube bends where the minimum safety factors are recorded.

Fig.13 Tube safety factor