DESIGN, OPTIMIZATION AND VERIFICATION OF THE STRUCTURAL INTEGRITY OF THE PISTON OF A SYSTEM THAT HAS THE FUNCTIONAL ROLE OF A HEAT PUMP

ANCUȚA Andrei-Alexandru

Facultatea: Inginerie Industrială şi Robotică, Specializarea: Siguranța și Integritatea Structurilor, Anul de studii: Master II, e-mail: andrei.ancuta2210@upb.ro

Conducător ştiinţific: Prof. dr. ing. **Gheorghe-Gabriel JIGA**

SUMMARY: This research article represents a study that aims to identify an optimal topology for a heat pump device, considering factors as lifetime, cost and thermodynamics reliability. The device in question is constructed similarly to hydraulic cylinders. Within this paper the overall design of the system, a working cycle and the initial stage of the designing, parametric optimization and validation of the first device's piston integrity are presented. The first parametric optimization is realized using Ansys direct optimization module, The second one is represented by manuals, iteratives improvements of the model based on post-processing the results obtained from the direct optimization module. The objective function of the optimizations is reducing of mass while the restrictions consisting in a minimum lifetime of 36000 cycles and a safety coefficient of stability equal or higher than four.

KEY WORDS: Ansys, parametric optimization, F.E.A., heat pump

1. Introduction

This article focuses on an early phase of optimizing the piston of a heat pump device, presenting also the general design of the device and its working cycle. The finality of this particular research is to identify the optimal piston topology regarding its mass. In order to achieve this objective, a parametric optimization and multiple manuals, iteratives improvements of the model were performed. The parametric optimization was made in Ansys [1], has an objective function of reducing mass, and two restrictions: a minimum lifetime of 36000 cycles and a safety factor of stability equal to or greater than four [2].

2. The general design of the device

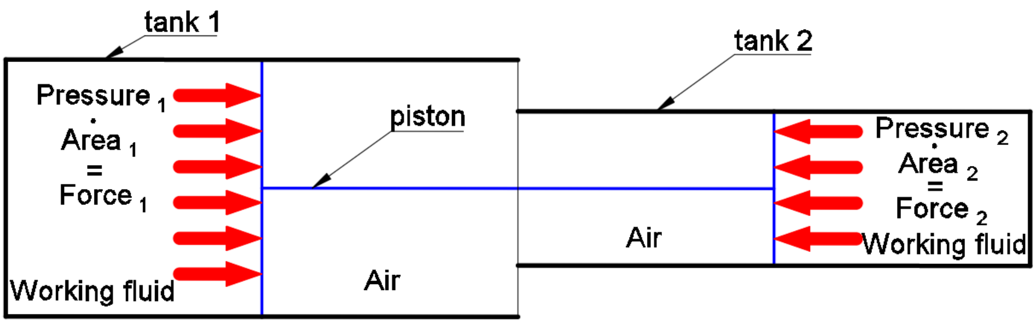

Fig. 1. Sketch of the general design of the device.

In Fig. 1. are illustrated the primary components of the device, which include two tanks of different diameters, each with a sliding lid. The two lids are rigidized one to another by a rod, forming what it will be further called as the piston of the device. The working fluid is constantly in a double state of aggregation, being on his vapor-liquid curve. For the moment, the device is designed, and the piston is optimized to have carbon dioxide as working fluid. However, once the optimal device topology is identified, the research will continue by identifying a working fluid with superior thermodynamic properties.

3. The working cycle

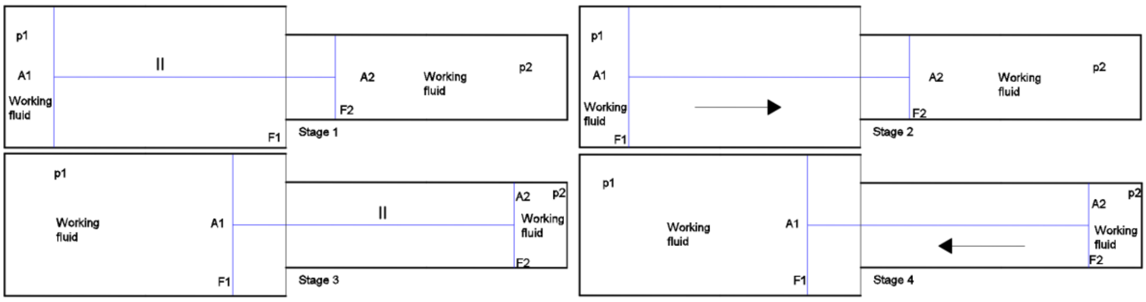

Fig. 2. Collage of four figures depicts the principal phases of the working cycle.

In Fig. 2. are illustrated the different stages of the working cycle. Each tank and lid assembly are characterized by a pressure and area, therefore a force. Initially, the piston is help in place by an external force, as the tanks are at equal pressure determined by the ambiental temperature. When the piston is released, a dynamic imbalance occurs due to the different lid areas, resulting in the movement of piston in the direction of compressing the working fluid from the smaller tank. The compression generates variation of enthalpy on the vapor-liquid curve of the carbon dioxide, leading to heat release. After a period of time the difference between the forces tends to 0 so the piston become balanced from a dynamic point of view. This represents the third stage. In the fourth stage, an external force returns the piston to its initial position, preparing it for the next cycle.

4. The device topologies that will be designed, optimized and verified to find the best option

In this series of research, to identify a suitable topology for the functional role of such a system and the loads to which is subjected, there will be designed, optimized and verified a variety of topologies.

A topology of system includes predominantly the design and the structural verification of the tank and the piston (including its lids).

Topology classification criteria:

1) By the type of the main degree of freedom:
    a. System with the main degree of freedom of the translation type;
    b. System with the main degree of freedom of the rotation type.
2) According to the perimeter shape of the piston ends (criterion valid only for systems with the main degree of freedom of translation type)
    a. System with circular perimeter of the piston ends;
    b. System with stepped shape of the piston ends.
3) By the number of stiffeners placed between the piston ends (criterion valid only for systems with the main degree of freedom of translation type)
    a. System with 16 stiffeners placed between the piston ends;
    b. System with 12 stiffeners placed between the piston ends;
    c. System with eight stiffeners placed between the piston ends;
    d. System with four stiffeners placed between the piston ends;
    e. System with one stiffener placed between the piston ends (the stiffener is present on all the circumference of the end of the piston).

4) By the presence of a stiffener plate positioned on the middle of the piston (criterion valid only for systems with the main degree of freedom of translation type)
    a. System with a stiffener plate positioned on the middle of the piston;
    b. System without a stiffener plate positioned on the middle of the piston.

In the followings, not all combinations that can result from the upper criterions will be studied. The study will start from one of the possible topologies, after this, using the initial configuration will be compared all the possibilities present at the fourth criterion. With the optimal version, will be analyzed in the same manner all possibilities from the third criterion and the procedure will repeat until it will be identified an overall optimal topology.

5. First topology – general elements

Based on the mentioned criteria, the first topology of the system is characterized as having a primary degree of freedom of the translation type, with circular perimetral ends of the piston, 16 stiffeners between the piston lids and no central stiffener plate on the rod.

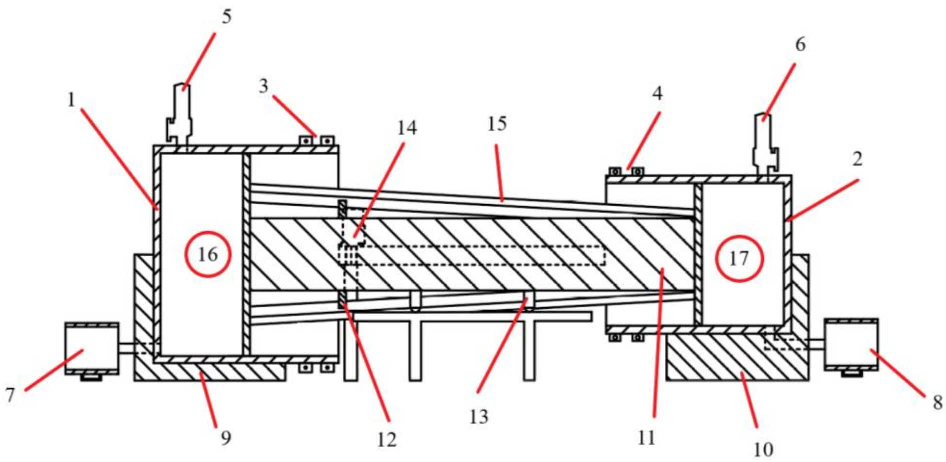

Fig. 3. Schematic drawing of the heat pump with the primary components identified.

Legend: 1, 2 – principal tanks, 3, 4 – stiffening collars of the main tanks, 5, 6 – safety valves, 7, 8 – auxiliary tanks featured with pumps and sense vales, 9, 10 – grounding system, 11 – piston, 12 – stopping element, 13 – the system of ensuring the orientation of the device, represented by guideways (roller supports), 14 – gear electromotor with rack and pinion, 15 – stiffeners of the ends of the piston, 16, 17 – working fluid (represented by carbon dioxide)

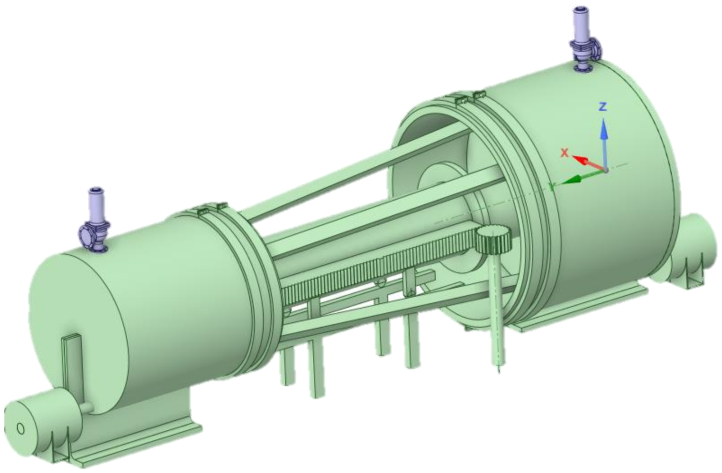

Fig. 4. The 3D representation of the first topology.

6. Parametric optimization and structural verification of the piston of the first topology

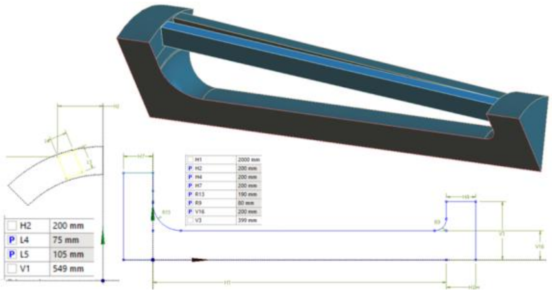

Fig. 5. Collage illustrating the geometry that was utilized to generate the finite element model.

The geometry used for to generate the finite element model was created in Design Modeler instead of Space Claim (as initially tried). Design Modeler offers advantages over Space Claim, such as clarity and ease of setting input parameters. Additionally, mathematical relations can be configured between input parameters to gain more control over the geometry. All the elements indicated with a blue “P” represents input or output parameters. For the input parameters that define the geometry manufacturable values were set before running the optimization analysis. The finite model used represents only one eight of the whole piston.

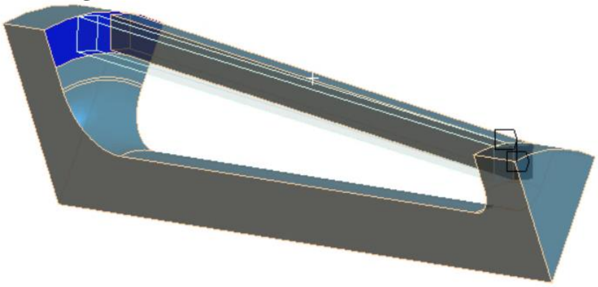

Fig. 6. One of the two contacts of “bonded” type, defined between the stiffeners and the lids of the piston.

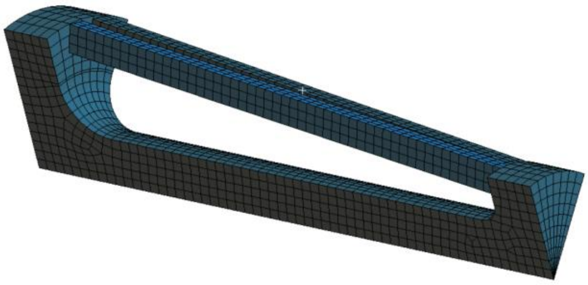

Fig. 7. The mesh of the model (controlled in local and global sizing, mesh method and element order).

The global element size is also configured as an input parameter, the reason will be covered later in this article.

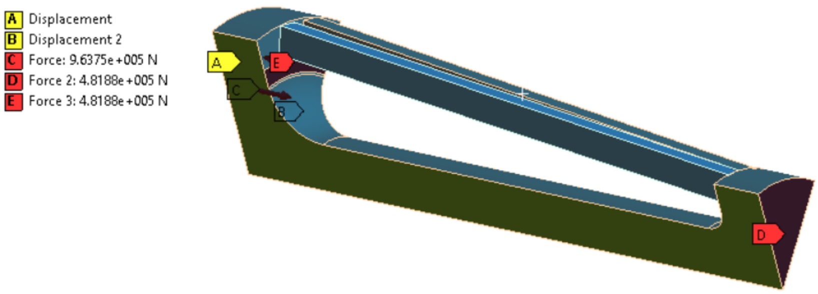

Fig. 8. The boundary conditions (including the loads acting on the structure).

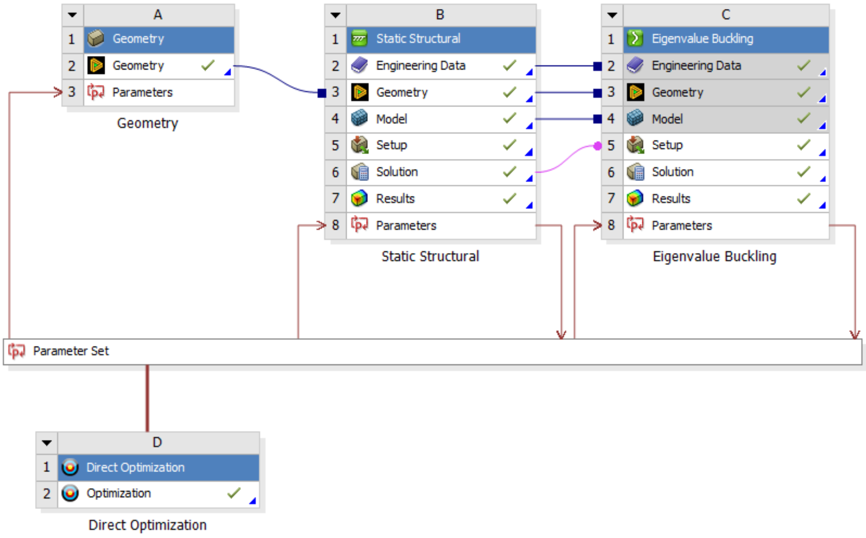

Fig. 9. The optimization project includes a static module with fatigue calculus and a stability calculus module based on eigenvalues and eigenvectors.

The objective function of the optimization was mass reduction, and it has two restrictions: a minimum lifetime of 36000 cycles and a safety factor of stability equal of higher than four [2].

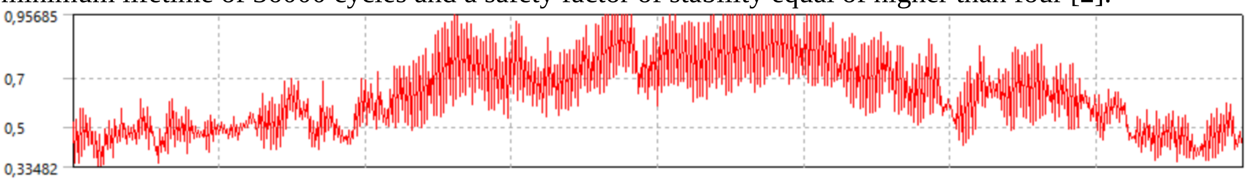

Fig. 10. The diagram illustrating the fatigue cycle.

The daily temperature fluctuations over the course of a year results in pressure changes on the piston lids. To simulate this, was implemented a fatigue block consisting of 365 daily cycles, which is shown in Fig. 10. This fatigue block depicts the fluctuation of the pressure exerted on the piston lids as a consequence of the daily temperature variation experienced throughout each day of a year. The daily temperatures were obtained by the National Meteorological Administration of Romania, using the Meteorological station located in Targul Jiu, and published in the article [3].

To obtain this fatigue block it was created a C++ program that was designed to intersect the daily minimum and maximum temperature with the pressure-temperature expression of the vapor-liquid curve for the working fluid [4] resulting corresponding pressure values for each recorded ambiental temperature.

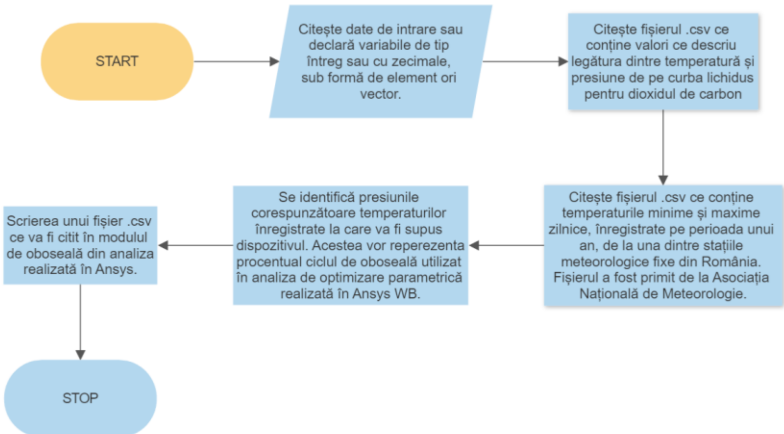

Fig. 11. The pseudocode of the program that generate the pressure fluctuation of a year.

It is indeed true the fact that the approximative working cycle is currently unknown in this stage of the research. However, this particular aspect is not considered extremely relevant in the current phase. Only a rough approximation is needed with the scope of identify an optimal topology, which will be further analyzed subjected to a more realistic work cycle.

In the study of the lifetime, the Soderberg mean stress correction theory was used. This theory, from the ones implemented in Ansys, is the most conservative and returns a zero lifetime for any zone that experiences stress above the yield stress.

The optimization method utilized in the study was the M.O.G.A. (Multi-Objective Genetic Algorithm), which successfully converged after computing 370 design points. The optimization process involved eight input parameters, resulting in a vast number of possible permutations (approximative 0.1 billion). However, this number was significantly reduced by defining the first two inequations depicted in Fig. 12. These constraints ensure that all the design point will have a geometry that can be generated. The final two inequations, which are also visible in Fig. 12, represents the control mechanism of the global sizing of the meshing elements. By controlling the global mesh element size, it had been achieved that all the design points exhibit similar accuracy in terms of stress and load multiplication coefficient, and all this advantages in the smallest computing time.

P20 <= (200[mm]-P22)+80[mm]	P20	<=	(200[mm]-P22)+80[mm]
P21 <= (200[mm]-P22)+190[mm]	P21	<=	(200[mm]-P22)+190[mm]
1	(P20*P20*(1-3.14/4)*2*3.14*(P20+P22)/2.37+P21*P21*(1-3.14/4)*2*3.14*(P21+P22)/2.37+564.2[mm]*564.2[mm]*3.14*P19+399[mm]*399[mm]*3.14*P17+2000[mm]*3.14*P22*P22)/(8*9000/4)	>=	P33*P33*P33
2	(P20*P20*(1-3.14/4)*2*3.14*(P20+P22)/2.37+P21*P21*(1-3.14/4)*2*3.14*(P21+P22)/2.37+564.2[mm]*564.2[mm]*3.14*P19+399[mm]*399[mm]*3.14*P17+2000[mm]*3.14*P22*P22)/(8*15000/4)	<=	P33*P33*P33

Fig. 12. The constraining inequations used in the optimization.

7. Manual optimization of the piston of the first topology

The results of the parametric optimization realized in Ansys optimization module were post-processed and was concluded that the model can be further enhanced. All the results can be analyzed in chapter 8. Results. Because of the fact that the model could be further enhanced the following solution and procedure in two steps was implemented. The first step had consisted in obtaining a dimensionless factor, defined for each input parameter, that signifies both the sensitivity of the of the structure's mass and its reliability. The equation that defines this parameter is Eq. (1), where *m* represents the mass and *c* the safety factor for two design points 1 and 2. These two design points differ in only one input parameter.

$$S = \frac{\sum_{i=1}^{n}\left(\frac{m_{i1}-m_{i2}}{c_{i1}-c_{i2}}\right)}{n} \tag{1}$$

This step was realized using a C++ code, which has the pseudocode presented in Fig. 13, that imported all the 370 design points (as input and output parameters) calculated in the Ansys optimization module. After this, the program identifies all sets of two design points that differs in one input parameter, and calculates the dimensionless factor presented in Eq. (1) for each input parameter. This factor is a measure of how much a parameter influences the structure both in mass and stability safety coefficient. Using the resulted sensitivities, it was clear which parameter should be decrease in order to obtain a more reliable model. After each modification, the analysis was run and the sensitivity for the changed parameter was recalculated. Then, the parameter with the greatest sensitivity was reidentified and procedure was reapplied. In only nine of these iterations, the Ansys optimized model was decreased in mass with an additional 30%. Additionally, it is no longer necessary to have predefined values for the parameters, so the last obtained model was again manually optimized.

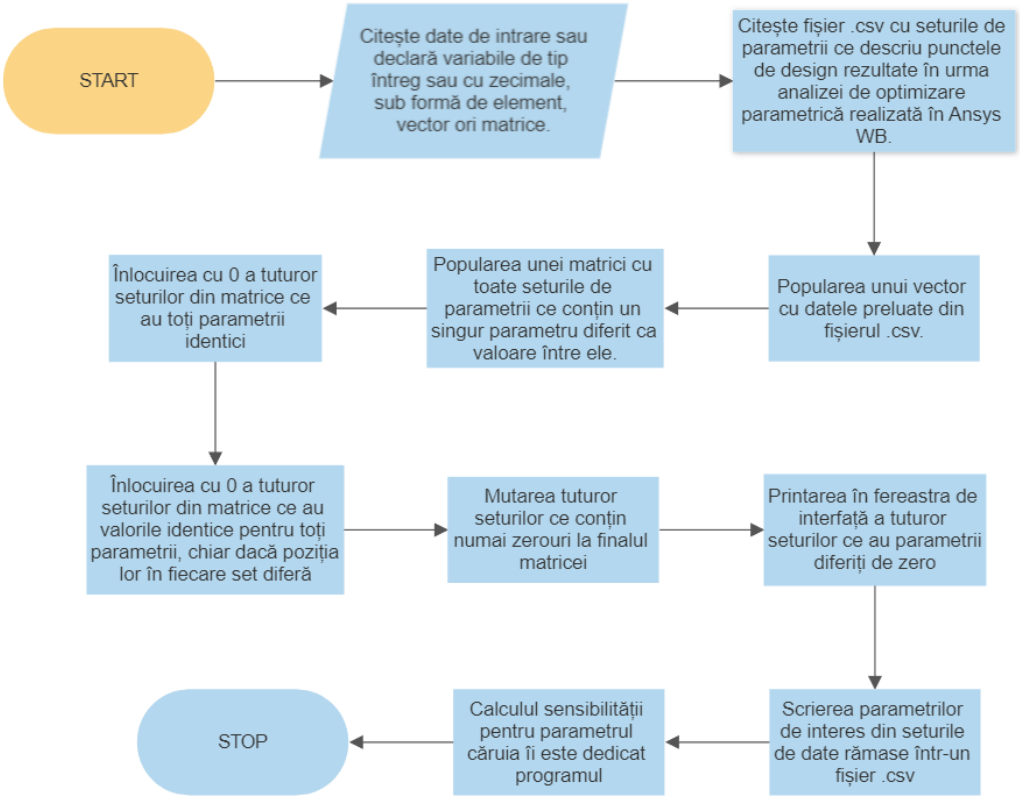

Fig. 13. The pseudocode of the program that calculates the sensitivity factor.

8. Results

The results of the optimizations are presented tabular, following this paragraph, being observed an overall mass reduction of 81.5%, percentage not necessarily relevant since I intentionally chosen a robust initial model, resulting section of the stiffeners of the piston are similar to the ones obtained analytically.

Table 1. Concatenated Results

Parameters	Initial model	Ansys optimized model	Manually obtained model (9 steps)	Manually obtained model (no predefined values)
Tank two - diameter lid thickness (H4)	200 mm	80 mm	40 mm	38 mm
Tank one - diameter lid thickness (H7)	200 mm	80 mm	40 mm	37 mm
Tank one – stiffening radius (R13)	190 mm	190 mm	140 mm	125 mm
Tank two – stiffening radius (R9)	160 mm	160 mm	140 mm	140 mm
Core radius (V16)	200 mm	80 mm	60 mm	59 mm
Stiffeners of the lids section width (L4)	75 mm	45 mm	45 mm	45 mm
Stiffeners of the lids section length (L5)	105 mm	55 mm	55 mm	48 mm
Minimum lifetime	10^6 cycles	10^6 cycles	38061 cycles	36378 cycles
Equivalent von Mises stress	24,73 MPa	98,54 MPa	202 MPa	202 MPa
Multiplying loads coefficient	57,25	5,84	4.67	4.0216
One eight of the resulting structure mass	6585.2 kg	2026.25 kg	1357 kg	1218.8 kg

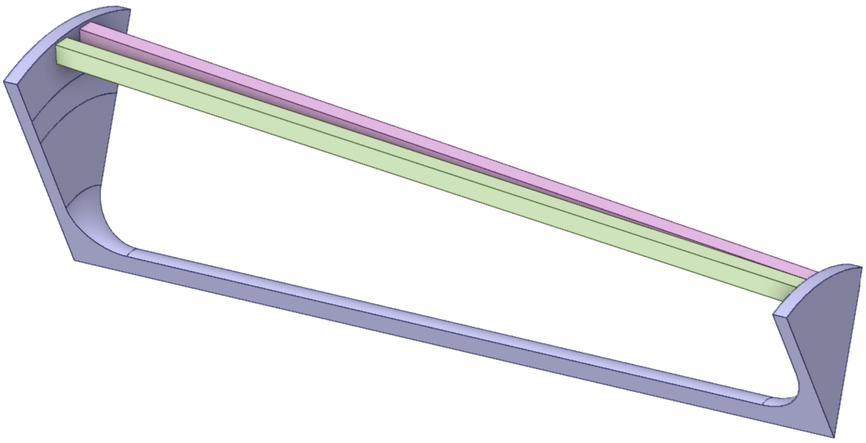

Fig. 14. The geometry of the resulted optimized model.

9. Conclusions

In this initial paper of a series, the author introduced the overall design of a device specifically developed as a heat pump, intended to utilize ambient temperature and energy for its functional purpose. The paper also presented the operational principles and the functioning of this device through the described work cycle. Additionally, an examination of various topologies was conducted to enable a future comparison and identification of the most optimal among them.

The design and initial phase of verification and optimization for the structural piston of the first technology were also introduced. The optimization process contains two optimizations, one realized in Ansys, and another one realized in a manual manner. The objective of the optimization analysis done in Ansys was to minimize the mass and had to two constraints: a minimum lifetime of 36,600 cycles and a stability safety factor of four or higher [2]. The achieved mass reduction between the initial and final models amounted to 81.5%, although the significance of this percentage is currently not substantial.

On short term, the next step is creating a Python program, using the PyMAPDL library, which will be used as an optimization module for this device. In this optimization module, with the data obtained after an initial set of design points which will be controlled by the user of the program (this represents a consistent advantage over using Ansys optimization module for example) an neural network will make prognostics about other design points (not calculated yet), for each prognostic the model will be run and compared the results with the values predicted by the artificial intelligence. This procedure will be repeated until the results and the predicted values will converge, and from that point on will be chosen the optimal model based on the predicted values that can be made on millions of design point in a very short time.

On long term the author will apply the same procedure on different tanks and, after this step, on other topologies with the scope of identification of the optimal version for these devices.

10. Bibliography

[1] * * * - ANSYS Workbench User's Guide, Release 2021 R2, ANSYS, Inc., 2022.

[2] Bambach, M.R., Strengthening of thin-walled (hollow) steel sections using fibre-reinforced polymer (FRP) composites, in: *Rehabilitation of Metallic Civil Infrastructure Using Fiber Reinforced Polymer (FRP) Composites*, Editor: Vistasp M. Karbhari, Woodhead Publishing, 2014, pp. 140-168.

[3] Temperature data for Targu Jiu weather station, National Meteorological Administration of Romania,https://www.meteoromania.ro/catalog/?tip=1&cod_geo=502317&cod_clasa=CLIMATOLOGICA&pas=4, Accessed May 14 ,2022

[4] Thermodynamic propreties of the carbon dioxide. https://encyclopedia.airliquide.com/carbon-dioxide#properties, Accessed May 14 ,2022.