Escola de Primavera de Transição e Turbulência
Blumenau 2022

13th Spring School on Transition and Turbulence
September 19th-23rd, 2022, Blumenau, SC, Brazil

EPTT-2022-0022
NUMERICAL ANALYSIS OF THE TURBULENT FLOW OF THE COOLING JACKET APPLIED IN A STAINLESS STEEL THERMOSYPHON

Victor Vaurek Dimbarre, victordimbarre@alunos.utfpr.edu.br[1]
Felipe Mercês Biglia, felipebiglia@gmail.com[2]
Luiz Eduardo Melo Lima, lelima@utfpr.edu.br[1]
Paulo Henrique Dias dos Santos, psantos@utfpr.edu.br[2]
Thiago Antonini Alves, antonini@utfpr.edu.br[1]

[1] Federal University of Technology - Paraná (UTFPR), Academic Department of Mechanics, Street Doutor Washington Subtil Chueire, 330, Jardim Carvalho, CEP 84.017-220, Ponta Grossa, PR, Brazil
[2] Federal University of Technology - Paraná (UTFPR), Academic Department of Mechanics, Street Deputado Heitor Alencar Furtado, 5000, Ecoville, CEP 81.280-340, Curitiba, PR, Brazil

Abstract. *Thermosyphons are passive heat transfer devices that operate using latent heat exchange and are divided into three sections: evaporator, adiabatic and condenser. The evaporator section has the characteristic of absorbing heat from an external source, the adiabatic section has the characteristic of not performing thermal exchanges with the environment and, finally, the condenser section aims to transfer the absorbed heat to the cooling flow. In this context, the objective of this work is to perform a numerical analysis of the fluid dynamic behavior of the internal flow of a cooling shirt of the condenser section of a stainless steel thermosyphon that uses water as a working fluid. The analysis was performed using the Ansys Fluent® 2021 R2 application and the turbulence was modeled using a Reynolds Stress Transport Model. This turbulence model complies with the required purposes, and can determine characteristic scales of turbulence, being suitable for the application in question that requires the knowledge of the average quantities. From the simulations it was possible to estimate the velocity profiles and current lines in the different regions of the condenser, thus showing which are the regions of greatest turbulence in the internal flow to the cooling jacket.*

Keywords: *Computational Fluid dynamics, RANS, Turbulence, Thermosyphon.*

1. INTRODUCTION

With the increasing global energy demand, the development of increasingly efficient devices and the use of devices that can recover part of this energy that would be wasted to the environment can reach 60% of all energy in the primary state (Cullen and Allwood, 2010), in this context, the use of thermosyphon in various applications stands out, for example, in the conversion of solar energy in thermal energy for heating water in homes, as well as the use of these devices in chimneys for energy recovery in industrial environments.

Thermosyphons are defined as passive gravity-assisted heat transfer devices that have high thermal conductivity when compared to other massive devices, such as solid bars. They are produced in clean (metallic or not) pipes, evacuated and filled with a certain working fluid with a determined pressure and have their operation based on the latent heat exchange of a working fluid operating in a biphasic cycle. It can be divided into three sections, as shown in Figure 1 (Machado, 2022), where it is possible to visualize its operating principle, at the bottom is located the evaporator, where there is heat absorption by the working fluid, causing the formation of vapor bubbles in the liquid pool that, by difference in density, are directed to the top of the thermosyphon, thus forming a flow of steam. At the top is located the condenser, part of which is responsible for dissipating the heat from the working fluid to the cooling fluid, causing the work fluid to condense and return to the evaporator, passing through the adiabatic section, with the aid of gravity forming a film of liquid in the walls, and the adiabatic section has the characteristic of not performing thermal exchanges between the work fluid and the external environment (Mantelli, 1994; Reay et al., 2014; Zohuri, 2016).

The concept of convection is defined for the description of the energy transfer enters a moving fluid and a surface, knowing that diffusion contributes to the transfer, is predominant the transfer caused by the general movement of both mass and particles. When it comes to problems related to convection, it is necessary to evaluate the regime to which it is found, and may be laminar or turbulent, and the movement of the fluid along a current line in its velocity components is evaluated in the different directions (Bergman et al., 2019).

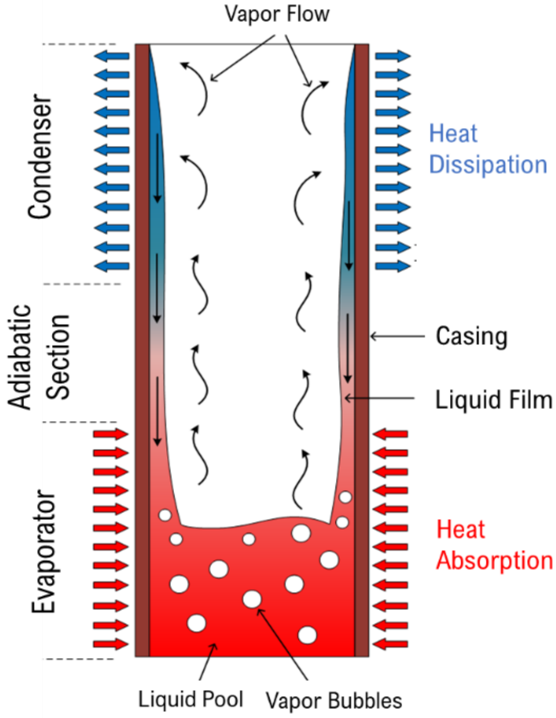

Figure 1. Thermosyphon's parts and work principles.

In laminar regime the movement of the fluid is organized and it can be identified current lines along the movement of particles, since when the flow is identified as turbulent the movement of particles is irregular and is characterized by fluctuation of velocity and can be amplified or reduced depending on the direction of the flow, and this highlights the transfer of momentum, energy and mass, thus causing increased friction on the surface and also increasing convective transfer rates (Çengel and Ghajar, 2020).

Reynolds' number is defined by Fox et al. (2020), as the ratio between the inertial and viscous forces in a flow of a fluid and is a dimensional number that was popularized by the Irish scientist Osborne Reynolds. This dimensional number shows the effect of viscosity on the control of the speed that a fluid flows. When the flow is laminar, that is, the Reynolds number is below 2000 for a circular tube, it can be stated that there is a linear viscosity profile, so that the fluid has parallel current lines, that is, they do not intersect and move without any interruption between them. When a flow is characterized as turbulent there is a mixture between the current lines and this flow occurs when the Reynolds number is above 4000, being favorable to the flows where you want to mix fluids and thermal exchanges between surfaces and fluids.

In this context, the present paper consists of a numerical study of the turbulent flow of the cooling jacket applied in a 304 stainless steel thermosyphon, performed using the ANSYS FLUENT® 2021 R2 software.

2. COMPUTATIONAL MODEL AND SOLUTION

The turbulence modeling was made through the Navier-Stokes equations with Reynolds means, where the flow quantities are decomposed into mean values and fluctuations by temporal mean processes, being resolved through the Reynolds Stress model, via linear pressure stress, which in the commercial software Ansys Fluent® is composed of 7 equations.

The representative equations, shown in Equation 1 and 2, represent mass and momentum transports, respectively.

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 , \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial}{\partial x_i}\left(\overline{u_i u_j}\right) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right], \tag{2}$$

where u is the speed in given component, ρ *is* the density of the fluid in the boundary condition, p is the pressure and ν the kinematic viscosity, for the determination of the type of flow was calculated Reynolds, the Eq. 3, in which is related to the speed of the fluid (u)*,* the hydraulic diameter (D_h)*,* being finite in Eq. 4, with the kinematic viscosity of the fluid.

$$\mathrm{Re} = \frac{u\ D_h}{\nu} \tag{3}$$

$$D_h = \frac{4\ \left(\pi/4\right)\ \left(D^2 - d^2\right)}{\pi\ \left(D + d\right)} \tag{4}$$

According to the literature, the values for internal flows in circular and annular ducts may vary, but according to Fox et al. (2020), the reference values are: Laminar flow Reynolds values are less than 2000, in the range between 2000 and 4000 the flow is in a transitional range and when the flow has Reynolds greater than 4000 is defined as turbulent.

The creation of the 3D model was based on a cooling jacket of an experimental thermosyphon already built according to the Figure 2, which is related the main measures necessary to perform the analysis, such as the internal diameter that is 19.05mm and the external diameter of 44.96mm with a total length of 400mm, with inlet and output sections of the same dimensions, which are 3.8mm in diameter.

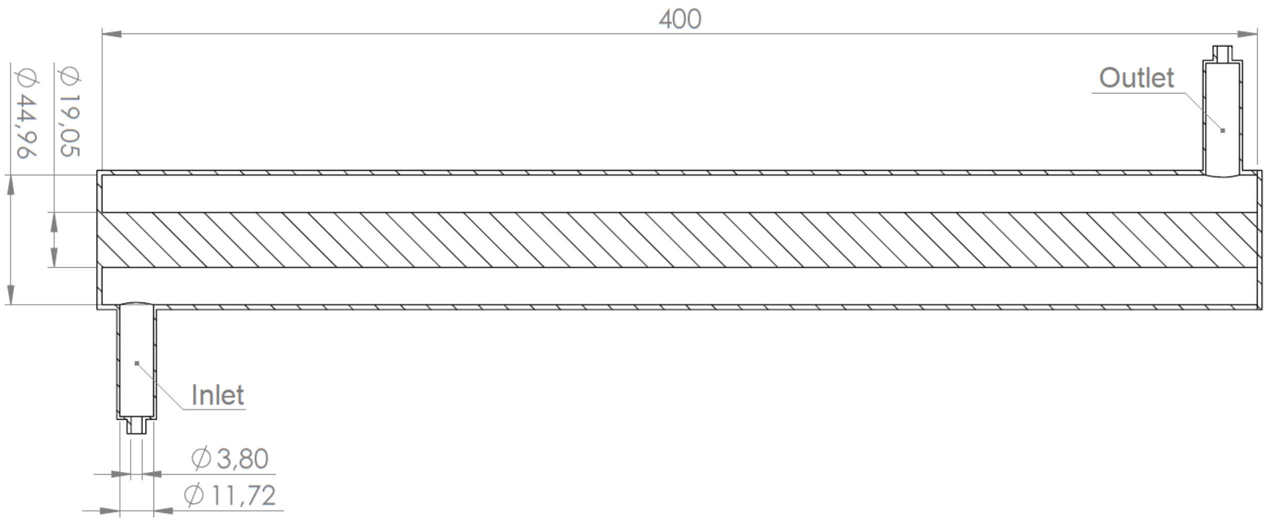

Figure 2. Thermosyphon's project.

A study of meshes was carried out to evaluate the variation of 0.001m in the size of the elements by seeing the impact on the total number of elements and nodes of the model. It was possible to notice that the proximity between the elements of Mesh#1 and Mesh#2 when compared to Mesh#3, which has a considerable reduction, already characteristic of average quality element of all meshes are close.

Table 1. Meshing refinement analysis.

Mesh	Element Size [m]	Elements	Nodes	Average Element Quality
#1	$9*10^{-4}$	1.883.050	2.719.118	0.83905
#2	$1*10^{-3}$	1.471.117	2.130.610	0.83911
#3	$2*10^{-3}$	278.944	417.035	0.84196

Given this information, Mesh#2 was selected to perform the simulations of the disposal of refrigerant fluid in the shirt of a thermosyphon and such mesh generated is represented in Fig. 3.

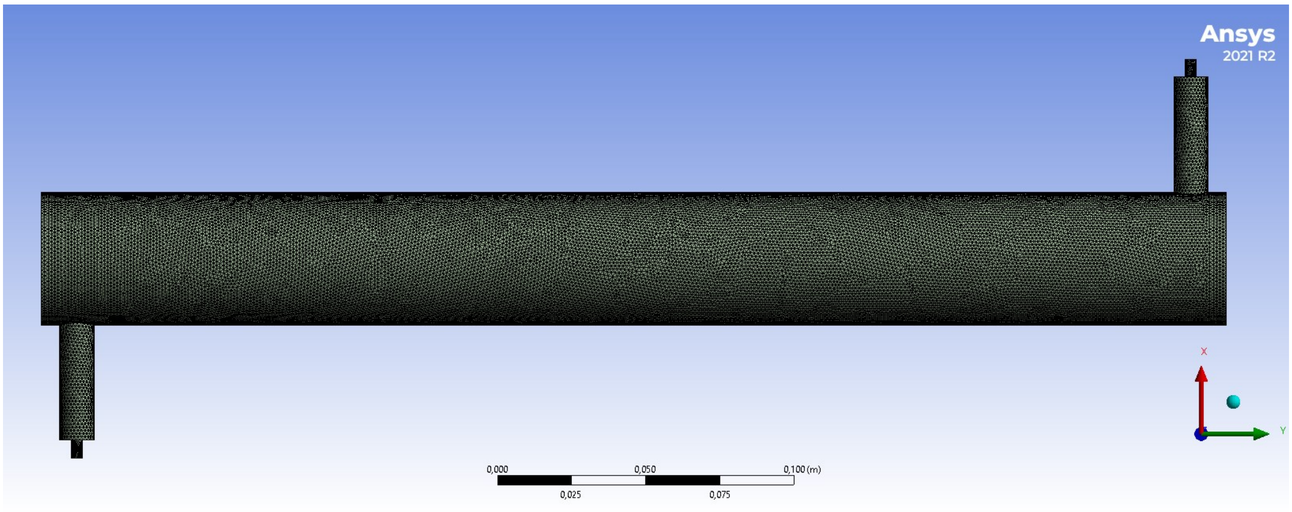

Figure 3. Mesh#2.

As conditions of contour of the problem, conditions of flow inlet velocity were used, being 1.3 m/s. As a speed method the coupling pressure used was coupled, with spatial gradient discretization at Least Square Cell Based, second-order pressure, second-order upwind momentum, turbulent kinetic energy, turbulent dissipation rate and Reynolds Stresses to first-order upwind, which are considered in pseudo-transient regime. The number of time-steps and the number of iterations used was 10^{-4} and 10000, respectively.

The computer used for the simulations has an Intel(R) Core TM i7-8700 CPU @ 3.20GHz and 16.0 GB of RAM.

3. RESULTS AND DISCUSSION

The beginning of the evaluation was by the calculation of Reynolds in 5 sections, which are demonstrated in Fig. 4, and their values are expressed in Table 2. And as expected the flow is defined as turbulent and as the hydraulic diameter is increased the number of Reynolds increases according to Eq. (3).

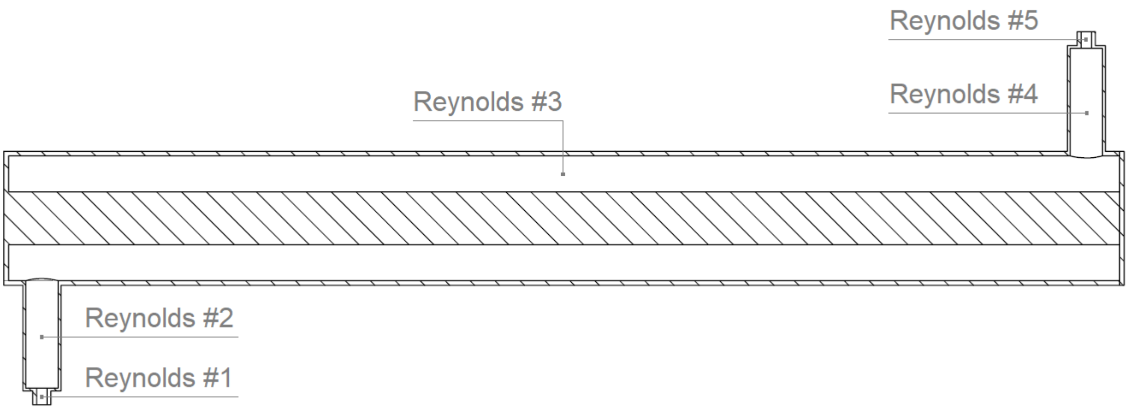

Figure 4. Reynolds rating points.

Table 2. Reynolds Rating Points Values.

Rating points	Reynolds value
#1	5,700.00
#2	15,001.60
#3	4,145.60
#4	4,336.40
#5	5,586.00

Figure 5 illustrate the velocity distribution in the jacket, it is observed that the change in flow direction is generated as a consequence of convective acceleration that occurs due to abrupt geometry changes (White, 2016).

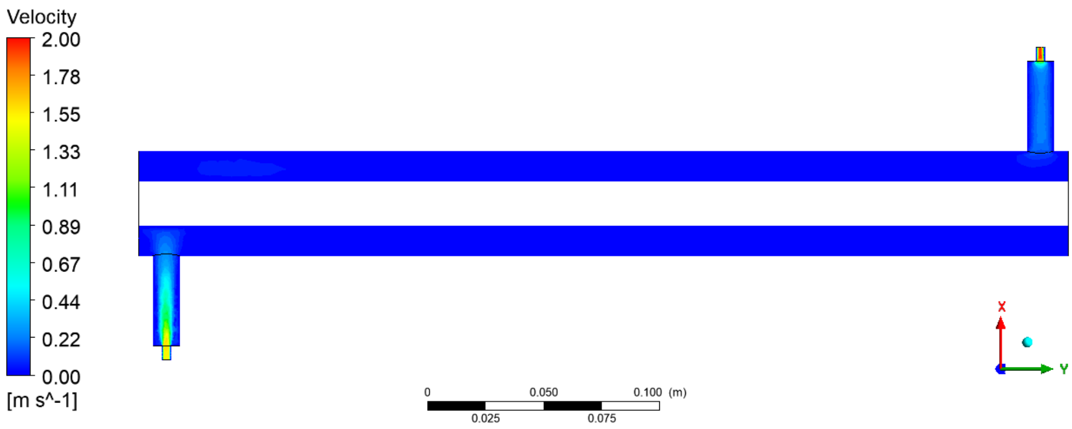

Figure 5. Velocity distribution in the jacket.

The recirculation zone is accelerated by a velocity leveling due to wall friction, generating a region of recirculating flow (separated region) (Fox et al., 2020).

Figure 6 shows the velocity gradient present at the entrance of the thermosyphon cooling jacket. It can be observed that as there is a change in the section area there is a change in the flow rate. Another important information provided is the recirculation areas of the cooling fluid present at the entrance (White and Majdalani, 2021).

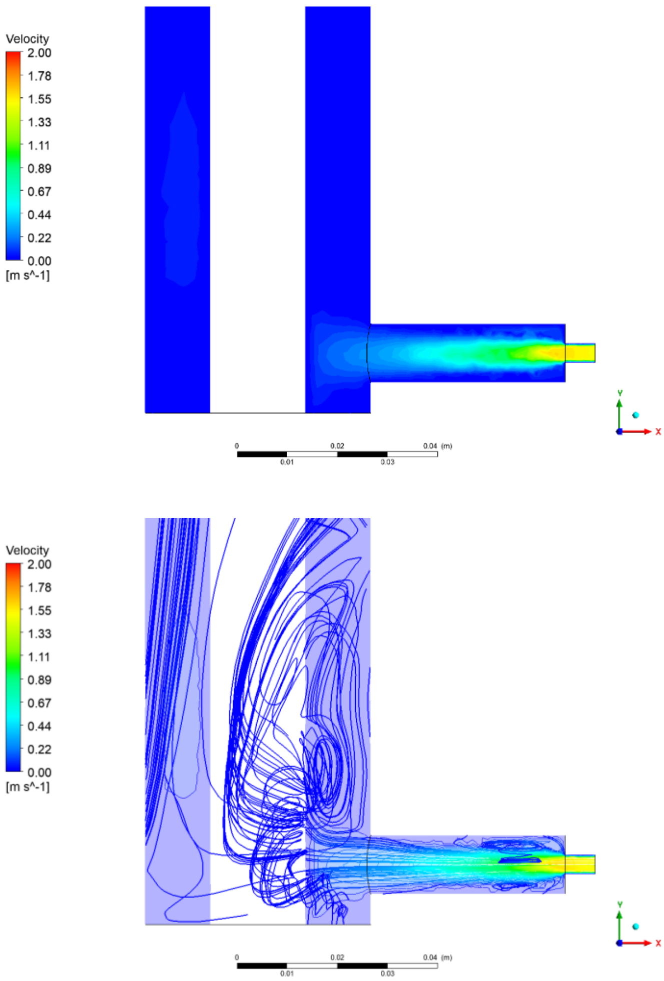

Figure 6. Inlet velocity and recirculation point at inlet.

Figure 7 shows the flow behavior at the top and exit section of the cooling jacket of the thermosyphon. It is noted that the flow is developed and in the upper right corner of the image there are some recirculation points, which is characteristic in turbulent flows. Finally, in the region where the throttling is located there is an increase in the speed of the coolant outlet. Consequently, since the center of curvature of the streamline is in the vicinity of the opening, a pressure variation occurs (Potter, 2014).

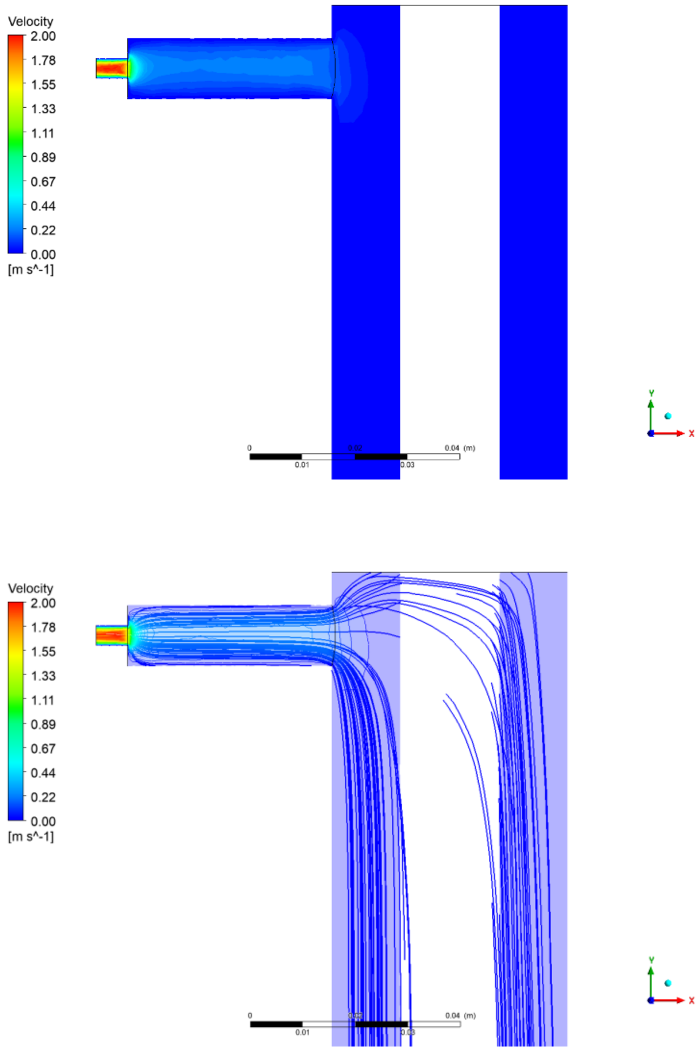

Figure 7. Outlet velocity recirculation point at outlet.

4. CONCLUSION

This work presented the analysis of a dynamic fluid flow in a cooling jacket of a condenser that uses water from the cooling fluid and that has an inlet flow of 1.3m/s in the lower region of the device.

In agreement with the literature, it was possible to visualize that the flow is considered turbulent and as there is an increase in the flow section there is an increase in Reynolds.

It is also possible to visualize that there are regions of recirculation at the entrance and at the top of the flow due to the turbulence present, being possible to visualize from the crossing of the current lines. With this it is possible to affirm when considering thermal contour conditions there will be a greater thermal exchange due to the characteristic of the flow corroborating with the literature.

5. ACKNOWLEDGEMENTS

Thanks are given to CAPES, the Pro-Rectory of Research and Graduate Studies of UTFPR, the Research and Graduate Board, the Graduate Program in Mechanical Engineering and the Academic Department of Mechanics of UTFPR/*Campus* Ponta Grossa.

6. REFERENCES

Bergman, T. et al. *Heat transfer analysis in a Maxwell fluid over an oscillating vertical plate using fractional Caputo-Fabrizio derivatives*. EUROPEAN PHYSICAL JOURNAL PLUS, v. 134, n. 3, 2019.

Çengel, Y. A., Ghajar, A. J. *Heat and Mass Transfer: Fundamentals & Applications*. 6 ed., McGraw-Hill Education, Penn Plaza, New York, NY, 2020.

Cullen, J. M., Allwood, J.M. *Theoretical efficiency limits for energy conversion devices*. Energy, pp. 2059-2069, 2010.

Fox, R. W., McDonald, A. T., Mitchell, J. W. *Fox and McDonald's introduction to fluid mechanics*. John Wiley & Sons, 2020.

Machado, P. L. O. *Desenvolvimento de um Sistema Híbrido Fotovoltaico/Térmico para Geração de Energia Elétrica e Aquecimento de Água utilizando Termossifões*. Dissertação de Mestrado, Universidade Tecnológica Federal do Paraná, Ponta Grossa, 2022.

Mantelli, M. B. H. *Thermosyphons and Heat Pipes: Theory and Applications*. Springer Nature. Application. New York, USA: John Wiley & Sons, 1994.

Potter, C. M., Wiggert, D. C., Ramadan, B. H. *Fluid Mechanics*. 4ed., Cengage Learning, Oklahoma, MI, 2012.

Reay, D. A., Kew, P. A., McGlen, R. J. *Heat pipes: theory, design and applications*. Waltham, MA, USA: Elsevier, 2014.

White, F. M. *Fluid Mechan*ics. 8 ed., McGraw-Hill Education, Penn Plaza, New York, NY, 2016.

White, F. M., Majdalani, J. *Viscous Fluid Flow*. 4 ed., McGraw-Hill LLC, Avenue of the Americas, New York, NY, 2021.

Zohuri, B. *Heat Pipe Design and Technology: Modern Applications for Practical Thermal Management*. 2ed., Springer, 2016.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.