2nd International Congress on Marine Energy CEMIE-Océano

CEMIE-Océano Asociación Civil

CEMIE-Océano

NUMERICAL SIMULATION OF THERMOFLUIDS OF A CENTRIFUGAL PUMP FOR A 1 KWE OTEC PROTOTYPE IN THE MEXICAN CARIBBEAN SEA.

Oscar Sierra Llanes [1], Cristian A. Carrillo Martínez [2] and Víctor M. Romero Medina [3].

[1] Universidad del Caribe, 140300108@ucaribe.edu.mx
[2] Universidad del Caribe, México, 140300250@ucaribe.edu.mx
[3]Departamento de Ciencias Básicas e Ingeniería, Universidad del Caribe, vromero@ucaribe.edu.mx

Abstract

Currently, the CASES of the Universidad del Caribe is collaborating with the interdisciplinary group CEMIE-Océano developing the prototype of a 1 kWe OTEC plant for the Mexican Caribbean Sea. As part of the research for the development of the prototype, CASES has a research line focused on the numerical simulation of each of its four main components: pump, evaporator, turbine-generator and condenser; Therefore, the main objective of this work is to carry out the numerical simulation of the Lederle Hermetic CAM 1 commercial axial flow centrifugal pump, which drives the working fluid R152a throughout the thermodynamic cycle. The simulation process consists of 5 fundamental stages, to carry it out, it begins with the definition of the geometry of the pump. Through the use of a CAD modeling tool such as SolidWorks, the 3D models of the different components that make it up are developed along with their corresponding assembly. Next, we proceed to the discretization of the volume of fluid that consists of the generation of a mesh of finite elements in the space that the fluid occupies inside the pump. Subsequently, the selection of the numerical models to be solved is made, the properties of the materials that make up the pump, the working fluid R152a and the boundary conditions that are necessary for the simulation. Finally, it ends with the analysis of the results obtained. Through the validation of the numerical simulation with the operating conditions of the commercial pump, it will be possible to use the simulation programs for the development of new projects.

Introduction

The Ocean Thermal Energy Conversion, OTEC for its acronym in English (Ocean Thermal Energy Conversion), essentially works by harnessing the solar energy absorbed by the ocean to be converted into electrical energy. All OTEC plants use the thermal gradient that converts heat into work through thermal energy between a source that is water located on the sea surface at a temperature between 27 and 30 °C and a sink that is seawater. at a depth of 700 m at a temperature of 5 °C. () The basic principle of operation is the Rankine thermodynamic cycle, specifically the closed cycle (CC) and it is made up of 4 main components: 1) R-152a pump, 2) Evaporator, 3) Turbine-generator and 4) Condenser.

The general objective of this project is to validate the numerical simulation of the Lederle Hermetic CAM 1 axial flow centrifugal pump by comparing the results of the simulation with the operating parameters of the real pump in order to use the device simulations of future projects with a high degree of reliability. confidence. The specific objectives are the following: 1) Draw the 3D geometry of the axial flow centrifugal pump using the SolidWorks 2017 computer-aided drawing program, 2) Discretize the volume of fluid for the simulation using Ansys Mesh, 3) Establish the numerical models, the boundary conditions and the properties of the participating materials with the Ansys Fluent program, 4) Carry out the simulation, 5) Compare the results obtained from the simulation with the operating parameters of the commercial pump. (ANSYS, 2017)

Figure 1 shows an aerial view of the three-dimensional design of the prototype of the OTEC-CC-MX-1kWe plant that is being developed by the Academic Body of Energy Systems and Sustainability, CASES by its acronym in Spanish of the Universidad del Caribe.

Figure 1. Three-dimensional representation of the prototype of the OTEC plant. (CASES, 2021)

Methodology

The methodology used in the development of this project consists of 3 fundamental stages adjusted to meet the 5 specific objectives mentioned above. It is based on the solution methodology in Ansys Fluent (Ansys, 2017), as well as on the pump modeling and simulation work carried out by García, A. (2020) and Garrido, S. (2017).

Results

Once the simulation with the laminar flow model was finished, it was decided to carry out a new test using a k-omega turbulence model, respecting the main operating conditions (gauge pressure of 412 kPa and rotational speed of 3440 rpm). in order to check if using a turbulent flow, it is possible to approach the mass flow obtained in the laboratory test.

Based on the results of the simulation (see table 1), an increase in mass flow of 0.3327 kg/s was obtained, which represents a decrease in the margin of error of 76.07% compared to that obtained in the laminar flow model test.

Table 8. Comparison of the simulation results with the k-omega turbulence model with the laboratory test. Source: self made

Parameters	Simulation results	Laboratory test results	Absolut mistake	Relative error	Percent relative error/margin of error (%)
Mass flow (kg/s)	0.4044	0.4373	0.0329	0.0753	7.53
Volumetric Flow (m^3/s)	0.0004317	0.0004668	0.0000351	0.0752	7.52

Conclusion

Based on the results obtained according to the methodology used, a total of 22 simulations were carried out that have been necessary to carry out the different analyses. It is observed that the k-omega turbulence model, considered as a suitable model for simulations of rotating systems such as the pump, was very close to the operating parameters evaluated in the laboratory, thus confirming that this type of flow corresponds to that of the prototype pump. Due to the low margins of error obtained between the results of the simulations and the empirical measurements, it is concluded that numerical simulation programs such as Ansys Fluent are a tool that allows us to carry out research and development of engineering devices with a high degree of precision and reliability.

References

ANSYS (2017). ANSYS TurboSystem User's Guide. ANSYS Inc.

CASES (2021). Metodología de simulación de los componentes principales del prototipo de planta OTEC mediante el uso del programa ANSYS-Fluent

García, A. (2020). Diseño and simulación de una bomba centrífuga mediante su prototipo virtual. Universidad de Cantabria.

Garrido, S. (2017). Diseño and estudio de una bomba hidráulica centrífuga mediante la generación de su prototipo virtual. Universidad Politécnica de Valencia