شبكة المعلومات الجامعية
التوثيق الإلكتروني والميكروفيلم

بسم الله الرحمن الرحيم

Ain Shams University Information Network
جامعة عين شمس
شبكة المعلومات الجامعية
@ ASUNET

HANAA ALY

شبكة المعلومات الجامعية
التوثيق الإلكتروني والميكروفيلم

شبكة المعلومات الجامعية

التوثيق الالكتروني والميكروفيلم

HANAA ALY

شبكة المعلومات الجامعية
التوثيق الإلكتروني والميكروفيلم

جامعة عين شمس

التوثيق الإلكتروني والميكروفيلم

قسم

نقسم بالله العظيم أن المادة التي تم توثيقها وتسجيلها
علي هذه الأقراص المدمجة قد أعدت دون أية تغيرات

يجب أن

تحفظ هذه الأقراص المدمجة بعيدا عن الغبار

HANAA ALY

AIN SHAMS UNIVERSITY

FACULTY OF ENGINEERING

MECHANICAL POWER DEPARTMENT

CFD Modeling of Wall Steam Condensation

A thesis submitted in partial fulfillment of the requirement of the degree of M.Sc. in Mechanical Engineering

By

Mohamed Ismail Mohamed Ismail Mousa

Bachelor of Science in Automotive Engineering

Faculty of Engineering, Ain Shams University, 2016

Supervised by

Prof. Dr. Nabil Abdel Aziz

Dr. Aya Khairy Diab

Cairo. 2021

AIN SHAMS UNIVERSITY

FACULTY OF ENGINEERING

MECHANICAL POWER DEPARTMENT

CFD Modeling of Wall Steam Condensation

A thesis submitted in partial fulfillment of the requirement of the degree of M.Sc. in Mechanical Engineering

By

Mohamed Ismail Mohamed Ismail Mousa

Bachelor of Science in Automotive Engineering

Faculty of Engineering, Ain Shams University, 2016

Supervised by

Prof. Dr. Nabil Abdel Aziz

Dr. Aya Diab

AIN SHAMS UNIVERSITY

FACULTY OF ENGINEERING

MECHANICAL POWER DEPARTMENT

CFD Modeling of Wall Steam Condensation

By

Mohamed Ismail Mohamed Ismail Mousa
Bachelor of Science in Automotive Engineering
Faculty of Engineering, Ain Shams University, 2016

Examiners' Committee

Name and Affiliation	*Signature*
Dr. Ali Mahmoud Ahmed Abd El Nabi Former Vice-Chairman of Nuclear Power Plants Authority, EGYPT	....
Prof. Dr. Mahmoud Abdel Rasheed Nossier Mechanical Power Department. Faculty of Engineering, Ain Shams University	....
Prof. Dr. Nabil Abdel Aziz Mahmoud Mechanical Power Department. Faculty of Engineering, Ain Shams University (Supervisor)	....

Date: …... /…... / 2021

STATEMENT

This thesis is submitted in partial fulfillment of the requirements for the Master of Science degree in Thermal Facilities and Combustion, Mechanical Power Engineering Department, Faculty of Engineering, Ain Shams University.

The author carried out the work included in this thesis, and no part of it has been submitted for a degree or a qualification at any other scientific entity.

Mohamed Ismail Mohamed Ismail Mousa

Signature

……………….

Date / / 2021

RESEARCHER DATA

Name	Mohamed Ismail Mohamed Ismail Mousa
Date of Birth	02 July 1993
Place of Birth	Saudi Arabia
Last academic degree	Bachelor of Science
Field of specialization	Mechanical Power Engineering
University issued the degree	Ain Shams University
Date of issued degree	2016
Current job	CFD Engineer

ABSTRACT

In this thesis, a wall condensation model for steam in the presence of non-condensable gas is successfully integrated into ANSYS FLUENT commercial computational fluid dynamics (CFD) software. The condensation process occurring in a single-phase multi-component flow is modeled by adding sink terms to consider the impact of vapor removed from the computational domain on the mass, momentum, energy conservation, as well as turbulence transport. These sink terms are programmed using C language and included within ANSYS FLUENT via its User-Defined Functions (UDFs) capability.

The condensation process is assumed to be driven by the vapor concentration gradient within the mixture, neglecting the thermal resistance of the liquid film. Given the importance of the conditions at the wall, the process is modeled as a conjugate heat transfer problem.

The condensation model is thoroughly validated using experimental data from three different facilities to prove the model's capability of predicting the condensation rate and associated heat flux. The first two occur under steady-state conditions in a simple one-dimensional geometry, while the last one focuses on transient three-dimensional flow conditions that may occur in the containment of a nuclear power plant in the rare event of a postulated accident.

The first validation test uses data from the CONAN test facility. Five cases with different inlet conditions have been simulated, and the model predictions of condensation rate as well as the wall heat flux are found to be in good agreement with the available experimental data. Moreover, a comparison with a previous numerical study indicates noticeable improvements in the current condensation model.

The second validation is performed using data from the Kuhn test facility, in which the condensation occurs in a vertical tube. The experiments provide temperature measurements along the test sections, runoff test 2.1-8R-Air, whose results were found to be in reasonable agreement with the model predictions.

The model is applied to a large-scale test facility, PANDA, to reflect the model capabilities under the complex flow conditions inherent in a three-dimensional flow domain. Contrary to the previous cases, the PANDA experiments are concerned with the condensation in low Reynolds number flow under transient conditions. The numerical results are discussed and compared with the available test measurements and also found to be in reasonable agreement.

Keywords: ANSYS FLUENT, condensation, CFD, Mass diffusion, User Defined Functions.