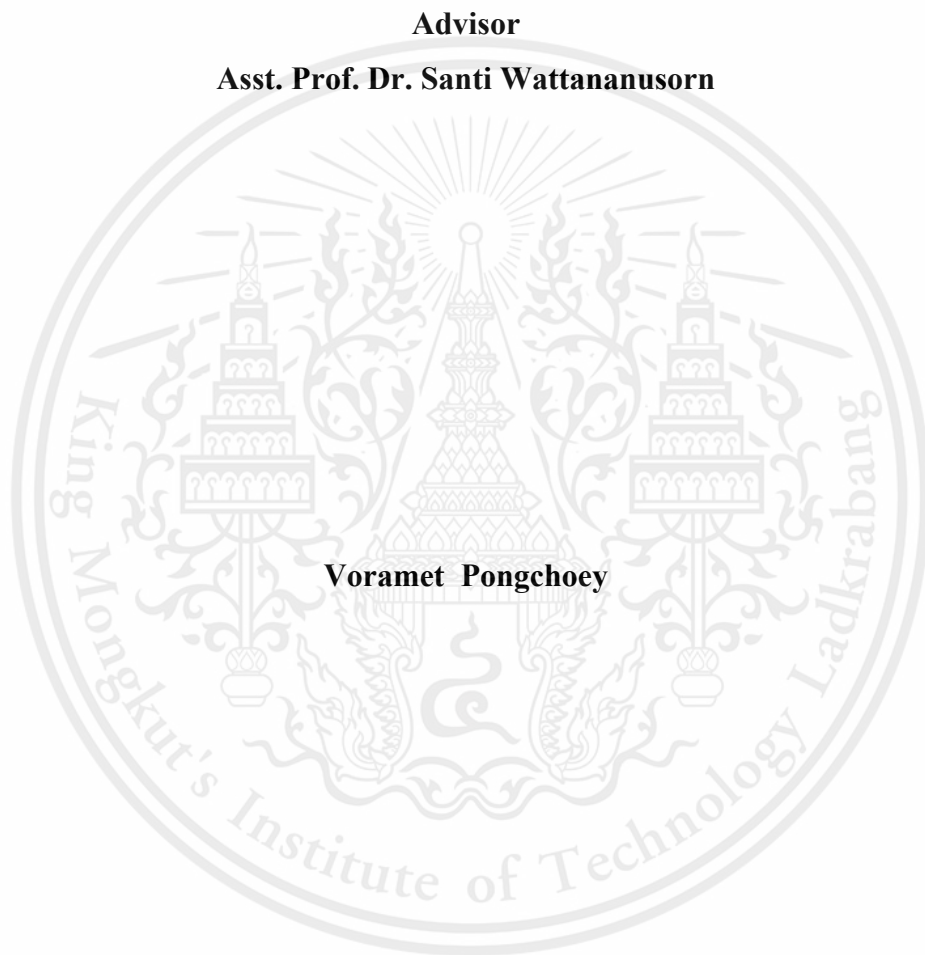


Computational Fluid Dynamics simulation of water steam ejector

Advisor

Asst. Prof. Dr. Santi Wattananusorn



Voramet Pongchoey

**A Report Submitted in Partial Fulfillment of the Requirements
for the Degree of Bachelor of Engineering (Petrochemical Engineering)
Department of Chemical Engineering, School of Engineering,
King Mongkut's Institute of Technology Ladkrabang**

Academic Year 2020

This material is reserved for educational use only, not allowed for commercial use.

Forbidden to modify the content, and cite the document when use

การจำลองพลศาสตร์ของไหลเชิงคำนวณของอีเจคเตอร์ไอน้ำ

อาจารย์ที่ปรึกษา

ผศ.ดร.สันติ วัฒนานุสรณ์



วรมะธ พงษ์เฉย

ปริญญานิพนธ์นี้เป็นส่วนหนึ่งของการศึกษาตามหลักสูตร

วิศวกรรมศาสตร์บัณฑิต สาขาวิชาวิศวกรรมปิโตรเคมี

ภาควิชาวิศวกรรมเคมี คณะวิศวกรรมศาสตร์

สถาบันเทคโนโลยีพระจอมเกล้าเจ้าคุณทหารลาดกระบัง

ปีการศึกษา 2563

This material is reserved for educational use only, not allowed for commercial use.

Forbidden to modify the content, and cite the document when use

Title	Computational Fluid Dynamics simulation of water steam ejector
By	Voramet Pongchoey
Field of Study	Petrochemical Engineering
Advisor	Asst. Prof. Dr. Santi Wattanusorn

Accepted by the School of Engineering, King Mongkut's Institute of Technology Ladkrabang in Partial Fulfillment of the Requirements for the Degree of Bachelor of Engineering (Petrochemical Engineering).

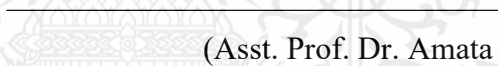
Thesis Committee



Chairman

(Asst. Prof. Dr. Santi Wattanusorn)

Committee



(Asst. Prof. Dr. Amata Anantpinijwatna)

Committee



(Asst. Prof. Dr. Apinan Namkanisorn)

Title Computational Fluid Dynamics simulation of water steam ejector

By Voramet Pongchoey

Advisor Asst. Prof. Dr. Santi Wattananusorn

Field of Study Petrochemical Engineering

Affiliation Department of Chemical Engineering, School of Engineering, King Mongkut's Institute of Technology Ladkrabang

Abstract

Computational fluid dynamic or CFD is system analysis by simulating via computer. This technique saves budget from experimentation and shows many things that cannot measure in the experiment. This research studies the simulation of ejector flow that create model by using Gambit and calculate by using ANSYS Fluent. Solid model is created by using paper of Ruangtrakoon et al. for reference. The conditions of the calculation are saturation temperature of evaporator is 7.5 °C, temperature of boiler is 130 °C and Nozzle size is D1.7M4.

From calculation results between 2D model and 3D model, Entrainment ratio (R_m) and critical pressure of 2D model is 0.521 and 3250 Pa respectively. Entrainment ratio (R_m) and critical pressure of 3D model is 0.471 and 3500 Pa respectively. Entrainment ratio (R_m) and critical pressure of reference paper is 0.422 and 3500 Pa respectively. The simulated results revealed that 3D model is more suitable for calculation than 2D model.

Keywords: CFD, Ejector

เรื่อง	การจำลองพลศาสตร์ของไหลเชิงคำนวณของอีเจคเตอร์ไอ้่น้ำ
โดย	วรมเชษ พงษ์เฉย
อาจารย์ที่ปรึกษา	ผศ.ดร.สันติ วัฒนานุสรณ์
สาขาวิชา	วิศวกรรมวิชาปิโตรเคมี
สังกัด	ภาควิชาวิศวกรรมเคมี คณะวิศวกรรมศาสตร์ สถาบันเทคโนโลยีพระ จอมเกล้าเจ้าคุณทหารลาดกระบัง

บทคัดย่อ

พลศาสตร์การไหลเชิงคำนวณ หรือ CFD คือ การวิเคราะห์แบบจำลองผ่านคอมพิวเตอร์. เทคนิคนี้ช่วยให้ประหยัดงบประมาณในการทำการทดลอง และ ยังทำให้มองเห็นในสิ่งที่การทดลองไม่สามารถตรวจวัดได้. การวิจัยนี้ศึกษาเกี่ยวกับการจำลองการไหลของของไหลภายใน อีเจคเตอร์ ซึ่งสร้างแบบจำลองโดยใช้โปรแกรม Gambit และ จำลองการไหลโดยใช้โปรแกรม ANSYS Fluent. แบบจำลองที่ใช้ในงานวิจัยนี้อ้างอิงมาจากเอกสารการวิจัยของ ดร.ณัฐวุฒิ เรื่องตระกูล และคณะ. เงื่อนไขของการวิจัยนี้ คือ อุณหภูมิเครื่องกำเนิดไอ 130 องศาเซลเซียส, อุณหภูมิเครื่องระเหย 7.5 องศาเซลเซียส และ หัวฉีดหลักขนาด D1.7M4.

จากผลการคำนวณระหว่างแบบจำลอง 2 มิติ และ แบบจำลอง 3 มิติ ค่า Entrainment ratio (R_m) และ ความดันวิกฤตของ แบบจำลอง 2 มิติ คือ 0.521 และ 3250 ตามลำดับ ส่วนของแบบจำลอง 3 มิติ คือ 0.471 และ 3500 Pa พอนำมาเทียบกับงานวิจัยที่ยกมาอ้างอิงที่มี ค่า Entrainment ratio (R_m) และ ความดันวิกฤต คือ 0.422 และ 3500 Pa. จากการศึกษาสรุปได้ว่าการใช้แบบจำลอง 3 มิติ ในการคำนวณนั้นมีความเหมาะสมมากกว่าแบบจำลอง 2 มิติ

คำสำคัญ: CFD, Ejector

Acknowledgements

I would like to express the deepest appreciation to

Dr. Eakarach Bumrunghthaichaichan who taught me about how to use Gambit and ANSYS Fluent for supporting me until this research finished.

Asst. Prof. Dr. Santi Wattananusorn who is my thesis advisor for helping me to modify this research from committee commend.

Chayanut Wangthanasarn, Thanadul Tanjaipetch, Thanawin Amradis for suggestion about ejector and setting in ANSYS Fluent.

My family who raised me for helping me with everything.

All of my friends who study in faculty of petrochemical engineering for helping me to do this research and presentation.

Voramet Pongchoey

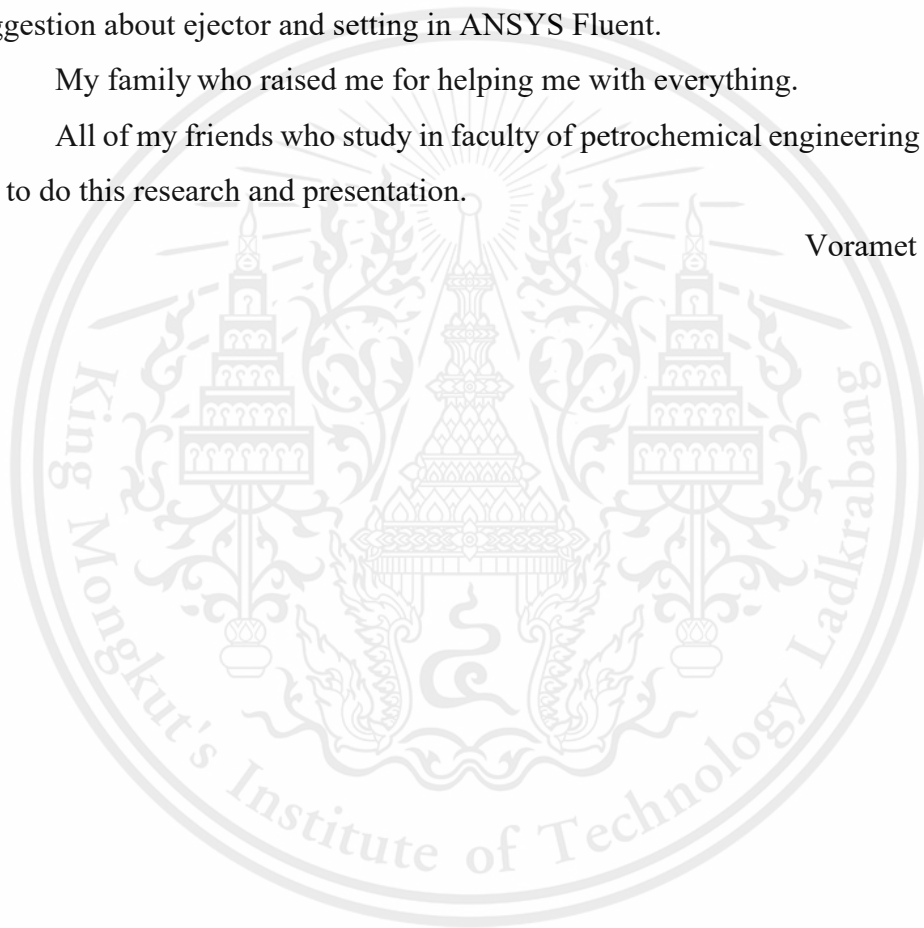


Table of Contents

	Page
Abstract.....	II
Abstract in Thai Language.....	III
Acknowledgements.....	IV
NOMENCLATURE.....	IX
CHAPTER I INTRODUCTION.....	1
1.1 Background.....	1
1.2 Objectives.....	1
1.3 Scopes of Work.....	1
1.4 Expected Outputs.....	1
CHAPTER II LITERATURE REVIEW.....	2
2.1 Computational fluid dynamic: CFD.....	2
2.1.1 Pre-computation process (Pre-Processor).....	2
2.1.2 Calculation process (Solver).....	2
2.1.3 Post-processing process (Post-Processor).....	3
2.2 Control equation.....	4
2.2.1 Mass conservation equations.....	4
2.2.2 Momentum equations.....	4
2.2.3 Navier-Stokes equations.....	5
2.3 Ejector.....	7
2.3.2 Flare Gas Recovery using Ejectors.....	8
2.3.3 Ejector refrigerator.....	9
2.4 Related research.....	11
CHAPTER III RESEARCH METHODOLOGY.....	12
3.1 Overall process.....	12

3.2 2D Model creation and Domain decomposition.....	13
3.3 3D Model creation and Domain decomposition.....	15
3.4 Setting in ANSYS Fluent.....	15
3.4.1 Model for calculation.....	15
3.4.2 Properties of material.....	15
3.4.3 Boundary Conditions Setting.....	16
CHAPTER IV RESULTS AND DISCUSSION.....	17
4.1 Example of divergence result in ANSYS Fluent.....	17
4.2 Compare the calculation results from ANSYS Fluent.....	18
4.2.1 Comparison of Entrainment Ratio and Critical pressure.....	18
4.2.2 Comparison of axial velocity.....	18
4.2.3 Comparison of radial velocity.....	19
4.2.4 Comparison of Temperature.....	20
4.2.5 Comparison of Mach Number.....	20
CHAPTER V CONCLUSION.....	22
5.1 Conclusion.....	22
5.2 Suggestions.....	22
REFERENCES.....	23

List of Figure

	Page
Figure 2.1: Principle of computational fluid dynamic (CFD).....	3
Figure 2.2: Ejector structure.....	8
Figure 2.3: Flare gas recovery ejector solution.....	9
Figure 2.4: Ejector refrigeration system.....	10
Figure 2.5: Experimental ejector refrigerator.....	11
Figure 3.1: Overall process for this project.....	12
Figure 3.2: Configuration of ejector from reference paper.....	13
Figure 3.3: Solid model of ejector.....	14
Figure 3.4: Decomposed model.....	14
Figure 3.5: Mesh generation model.....	14
Figure 3.6: 3D model of ejector.....	15
Figure 4.1: Divergence result.....	17
Figure 4.2: Axial velocity of 2D model and 3D model.....	19
Figure 4.3: Radial velocity of 2D model and 3D model.....	19
Figure 4.4: Temperature of 2D model and 3D model.....	20
Figure 4.5: Mach Number of 2D model and 3D model.....	21

List of Tables

	Page
Table 3-1: Properties of water-vapor.....	15
Table 3-2: Boundary Conditions.....	16
Table 4-1: Comparison of Rm and Critical pressure.....	18

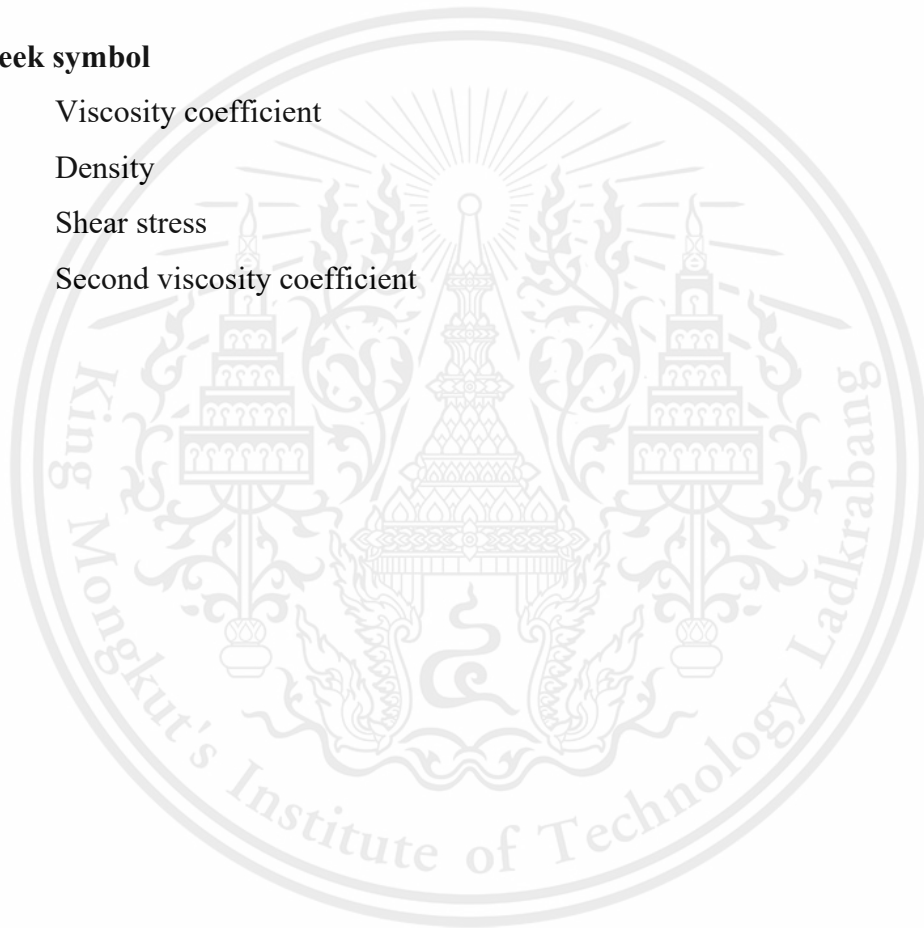


NOMENCLATURE

P	Pressure
S_M	External forces
t	Time
U	Velocity vector
C_p	Specific heat
R_m	Entrainment Ratio

Greek symbol

μ	Viscosity coefficient
ρ	Density
τ	Shear stress
λ	Second viscosity coefficient



CHAPTER I

INTRODUCTION

1.1 Background

Nowadays, energy is a limited supply so it leading to various way to save energy. Choosing an equipment that uses lower power is one option. Ejector simple pieces of equipment could use for less energy requiring systems and more efficient. Moreover, ejector could be used in many applications such as flare gas recovery system, refrigeration system, boosting production, etc. To study the design and implementation of ejector, this project would like to create a flow model in the ejector.

Paper of Natthawut Ruangtrakoon and team studies about experimental of steam ejector refrigeration system. This project will remodel of ejector calculation by using Computational Fluid Dynamics (CFD). This technique is used for saving the budget for experimenting and designing of the ejector before ordering or using. [1]

Models used for simulation in CFD could be both 2D and 3D model, but in this case, inlet positions are not the same. So, this project would like to compare results from both of models.

1.2 Objectives

1. To find suitable model for ejector in ejector refrigeration system.
2. To compare how 2D vs 3D model simulations affect the flow field.

1.3 Scopes of Work

1. Model and grid are created by using Gambit.
2. Flow simulation is calculated by using ANSYS FLUENT 14.5.

1.4 Expected Outputs

1. To obtain suitable model to design ejector.
2. To understand the difference between 2D model and 3D model.

CHAPTER II

LITERATURE REVIEW

2.1 Computational fluid dynamic: CFD

Computational fluid dynamic or CFD is system analysis involving fluid flow, heat transfer and related phenomena such as chemical reaction by simulating via computer. This technique is very useful and widely applied in industry.

Principle of computational fluid dynamic is divided into three parts: Pre-computation process (Pre-Processor), Calculation process (Solver) and Post-processing process (Post-Processor) shows the details of procedures as follows[2]

2.1.1 Pre-computation process (Pre-Processor)

At this part, it is a part before the flow problem is calculated. This is a procedure for entering data that is required for calculations to the program such as computing domain creation, creating mesh, choosing an equation or model, determination of flow properties, determination of boundary conditions, etc.[2]

2.1.2 Calculation process (Solver)

There are 4 methods of numerical computation techniques: Finite Difference method, Finite Element method, Spectral method, and Finite Volume method. The basic form of problem solving has procedures as follows

1. Estimate unknown variables with simple function.
2. Convert the partial differential equation to an algebraic equation by numerical methods (substituting values of variable in the control equations and then formulating them).
3. Solving algebraic equations.

In this project use Finite Volume method. The calculation of this method has procedures as follows

1. Divide the computational domain into small volumes.
2. Integrate control equation of all volume flows within scope of problem.
3. Convert the partial differential equation to an algebraic equation by substituting the different terms that are integrated with approximation value.
4. Find the solution of the algebraic equation by repetition method.[2]

2.1.3 Post-processing process (Post-Processor)

Post-Processor is process that shows the results obtained from the calculation. These results could be displayed in different formats such as shape of domain, vector, grid, graph, contour, 2D surface, 3D surface, etc.[2]

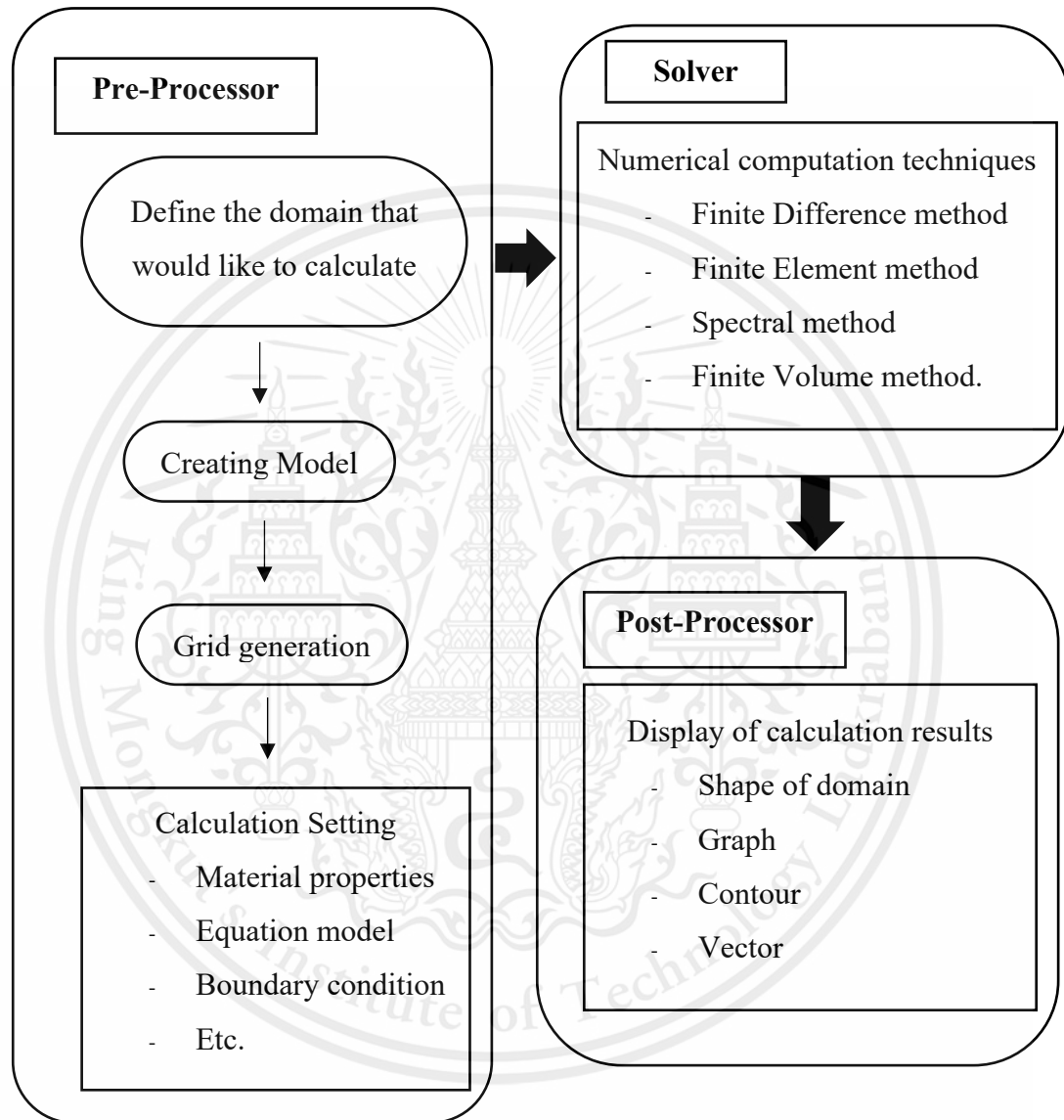


Figure 2.1: Principle of computational fluid dynamic (CFD)

2.2 Control equation

Fluid flow phenomena is based on 3 conservation laws: law of conservation of mass, Newton's second law of motion and law of conservation of energy. All 3 laws can be explained as control equations.[2]

2.2.1 Mass conservation equations

The mass balance of fluid is rate of mass increasing in fluid = rate of net mass flow entering the fluid.

So, the equation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (2.1)$$

Or

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (2.2)$$

Where

ρ is density
 t is time
 U is velocity vector

Both equation 2.1 and 2.2 are called Continuity equation

In case of incompressible fluid that has constant density, equation (2.1) is simplified to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (2.3)$$

Equation (2.3) is called Continuity equation for incompressible fluid [2]

2.2.2 Momentum equations

From Newton's second law of motion, “force is equal to the rate of momentum change”, the momentum equations are

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial(\tau_{xx})}{\partial x} + \frac{\partial(\tau_{yx})}{\partial y} + \frac{\partial(\tau_{zx})}{\partial z} + S_{Mx} \quad (2.4)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial(\tau_{xy})}{\partial x} + \frac{\partial(\tau_{yy})}{\partial y} + \frac{\partial(\tau_{zy})}{\partial z} + S_{My} \quad (2.5)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial(\tau_{xz})}{\partial x} + \frac{\partial(\tau_{yz})}{\partial y} + \frac{\partial(\tau_{zz})}{\partial z} + S_{Mz} \quad (2.6)$$

Where

- P is pressure
 τ is shear stress
 S_M is external forces that affect volume of fluid such as gravity force, magnetic force, etc.

Substitute equation (2.4), (2.5) and (2.6) with equation (2.2) then,

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u U) = -\frac{\partial p}{\partial x} + \frac{\partial(\tau_{xx})}{\partial x} + \frac{\partial(\tau_{yx})}{\partial y} + \frac{\partial(\tau_{zx})}{\partial z} + S_{Mx} \quad (2.7)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v U) = -\frac{\partial p}{\partial y} + \frac{\partial(\tau_{xy})}{\partial x} + \frac{\partial(\tau_{yy})}{\partial y} + \frac{\partial(\tau_{zy})}{\partial z} + S_{My} \quad (2.8)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w U) = -\frac{\partial p}{\partial z} + \frac{\partial(\tau_{xz})}{\partial x} + \frac{\partial(\tau_{yz})}{\partial y} + \frac{\partial(\tau_{zz})}{\partial z} + S_{Mz} \quad (2.9)$$

These equations are called Momentum equations in conservation form. [2]

2.2.3 Navier-Stokes equations

“The shear stress in fluid is proportional to rate of stress change” is said by Isaac Newton. This fluid is called Newtonian fluids. In 1845, George Gabriel Stokes found

$$\tau_{xx} = \lambda(\nabla \cdot U) + 2\mu \frac{\partial u}{\partial x} \quad (2.10)$$

$$\tau_{yy} = \lambda(\nabla \cdot U) + 2\mu \frac{\partial v}{\partial y} \quad (2.11)$$

$$\tau_{zz} = \lambda(\nabla \cdot U) + 2\mu \frac{\partial w}{\partial z} \quad (2.12)$$

$$\tau_{xy} = \tau_{yx} = \mu \left(\frac{\partial v}{\partial w} + \frac{\partial u}{\partial y} \right) \quad (2.13)$$

$$\tau_{xz} = \tau_{zx} = \mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \quad (2.14)$$

$$\tau_{yz} = \tau_{zy} = \mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \quad (2.15)$$

Where

μ is viscosity coefficient

λ is second viscosity coefficient

Assumption of George Gabriel Stokes is

$$\lambda = -\frac{2}{3} \mu \quad (2.16)$$

Although this assumption has not been proven, this equation has been used until now.

Substitute equation (2.10), (2.11), (2.12), ... and (2.15) with Momentum equations in conservation form then

$$\begin{aligned} \frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u U) = & -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[\lambda (\nabla \cdot U) + 2\mu \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[\mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right] \\ & + \frac{\partial}{\partial z} \left[\mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \right] + S_{Mx} \end{aligned} \quad (2.17)$$

$$\begin{aligned} \frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v U) = & -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[\mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left[\lambda (\nabla \cdot U) + 2\mu \frac{\partial v}{\partial y} \right] \\ & + \frac{\partial}{\partial z} \left[\mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \right] + S_{My} \end{aligned} \quad (2.18)$$

$$\begin{aligned} \frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w U) = & -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left[\mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \right] + \frac{\partial}{\partial y} \left[\mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \right] \\ & + \frac{\partial}{\partial z} \left[\lambda (\nabla \cdot U) + 2\mu \frac{\partial w}{\partial z} \right] + S_{Mz} \end{aligned} \quad (2.19)$$

These equations are called Navier-Stokes equations in conservation form.

Substitute shear stress from viscosity and rearrange equations form then

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u U) = -\frac{\partial p}{\partial x} + \nabla \cdot (\mu \nabla \cdot u) + S_{Mx} \quad (2.20)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v U) = -\frac{\partial p}{\partial y} + \nabla \cdot (\mu \nabla \cdot v) + S_{My} \quad (2.21)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w U) = -\frac{\partial p}{\partial z} + \nabla \cdot (\mu \nabla \cdot w) + S_{Mz} \quad (2.22)$$

In general, Navier-Stokes equation consists of Continuity equation and 3 momentum equations.[2]

2.3 Ejector

Ejector is an equipment that has been used in industry process for a long time. This equipment was first invented and developed in 1901 by Sir Charles Parsons for removing air from a steam engine's condenser.[3]

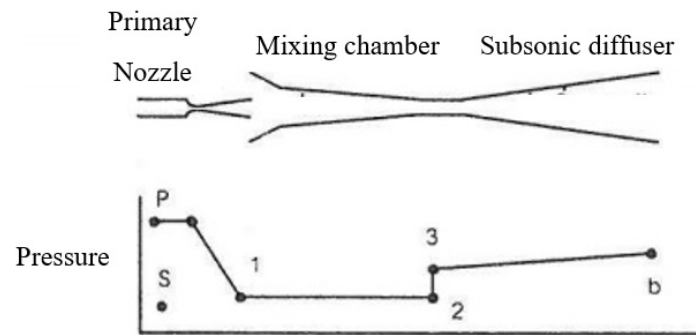


Figure 2.2: Ejector structure [3]

Figure 2.2 shows structure and main components of the ejector. Primary fluid(P) is compressed through primary nozzle to increase supersonic speed so, Low pressure is created at outlet of nozzle (1). This reduced pressure brings secondary fluid (S) into the mixing chamber. The assumption is that, primary fluid and secondary fluid are well mixed (2) and have supersonic velocity. When the mixed fluid arrives at throat (3) where the pressure is high, shock wave would occur then, velocity of fluid decrease to be lower than supersonic and pressure increase. After that, pressure would be increased when velocity is decreased by subsonic diffuser (b).[3]

Ejector could be used in many applications of industry process such as

2.3.2 Flare Gas Recovery using Ejectors

High pressure stream is used to drive ejector's operation. Existing gas, waste gas or liquid stream could use to operate ejector. For high-pressure gas processes, the ejector is one of the first devices to be used. Example of flare gas recovery using ejector is shown in Figure 2.3. [4]

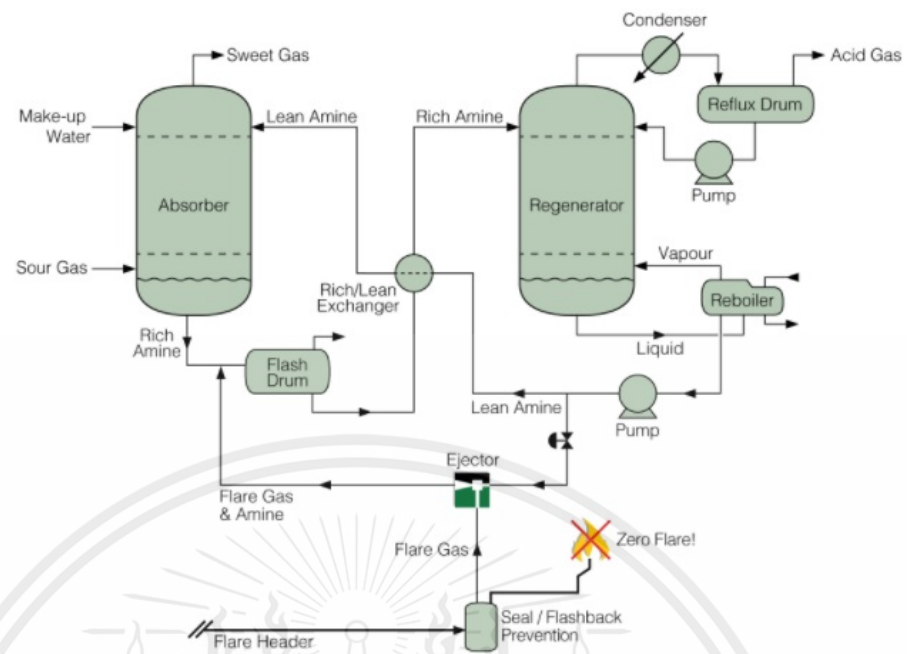


Figure 2.3: Flare gas recovery ejector solution [4]

In this example, amine that has high pressure from pump is used as the primary fluid for ejector, and flare gas from seal drum is used as the secondary fluid. After amine and flare gas pass ejector, they are mixed and sent to absorber flash drum. Then, amine return to regeneration unit and compressed flare gas could be directed to gas processing.[4]

2.3.3 Ejector refrigerator

In 1910, Maurice Leblanc brought ejector to refrigeration system by using water as working fluid. This refrigeration system became very popular in the 30s for large air condition system. Later, ejector refrigeration system decreased in popularity and replaced by vapor compression refrigeration systems. [3]

Example of ejector refrigeration system is shown in Figure 2.4.

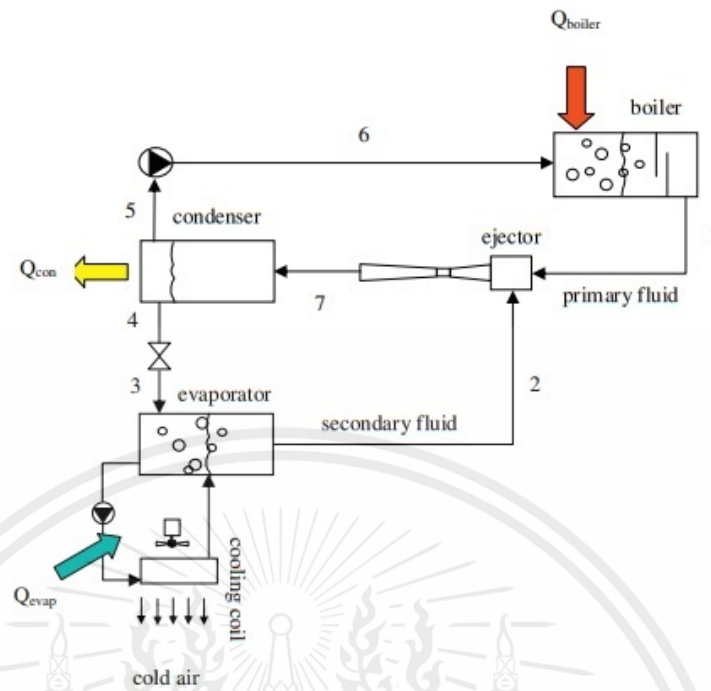


Figure 2.4: Ejector refrigeration system [1]

High pressure and high temperature gas that come from boiler is used as primary fluid. Then, primary fluid pass through primary nozzle in ejector. So, low pressure region at ejector that connect to evaporator. Low pressure causes secondary fluid to vaporize and heat that is used to generate the vapor is coolness that is obtained from system. This secondary fluid is brought into the ejector and is mixed with primary fluid at mixing chamber then mixed fluid is brought. Primary fluid that is separated from condenser is brought to boiler for complete cycle.[3]

2.4 Related research

Paper of Ruangtrakoon et al. (2011) explain about ejector refrigerator. Experimental results show geometries of the primary nozzle have strong effects to the ejector performance.[1]

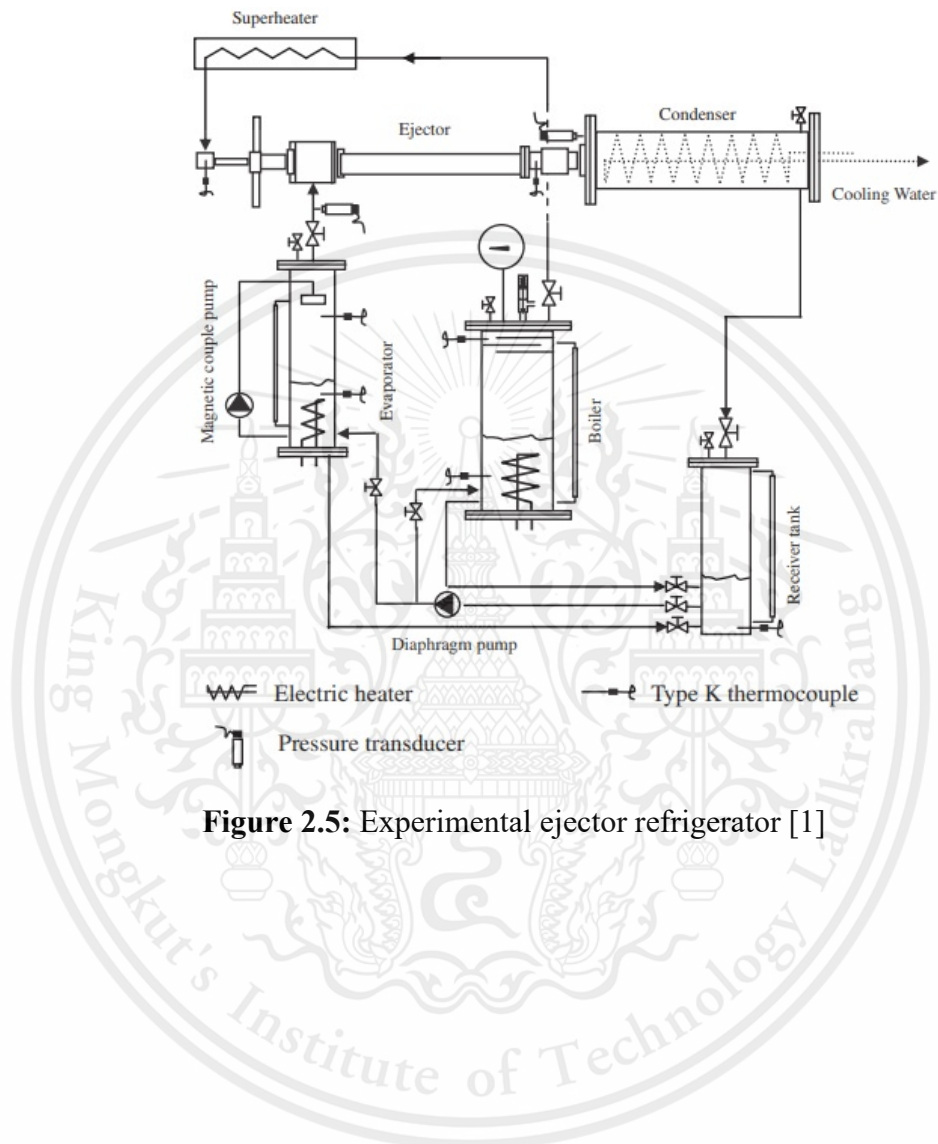


Figure 2.5: Experimental ejector refrigerator [1]

CHAPTER III
RESEARCH METHODOLOGY

3.1 Overall process

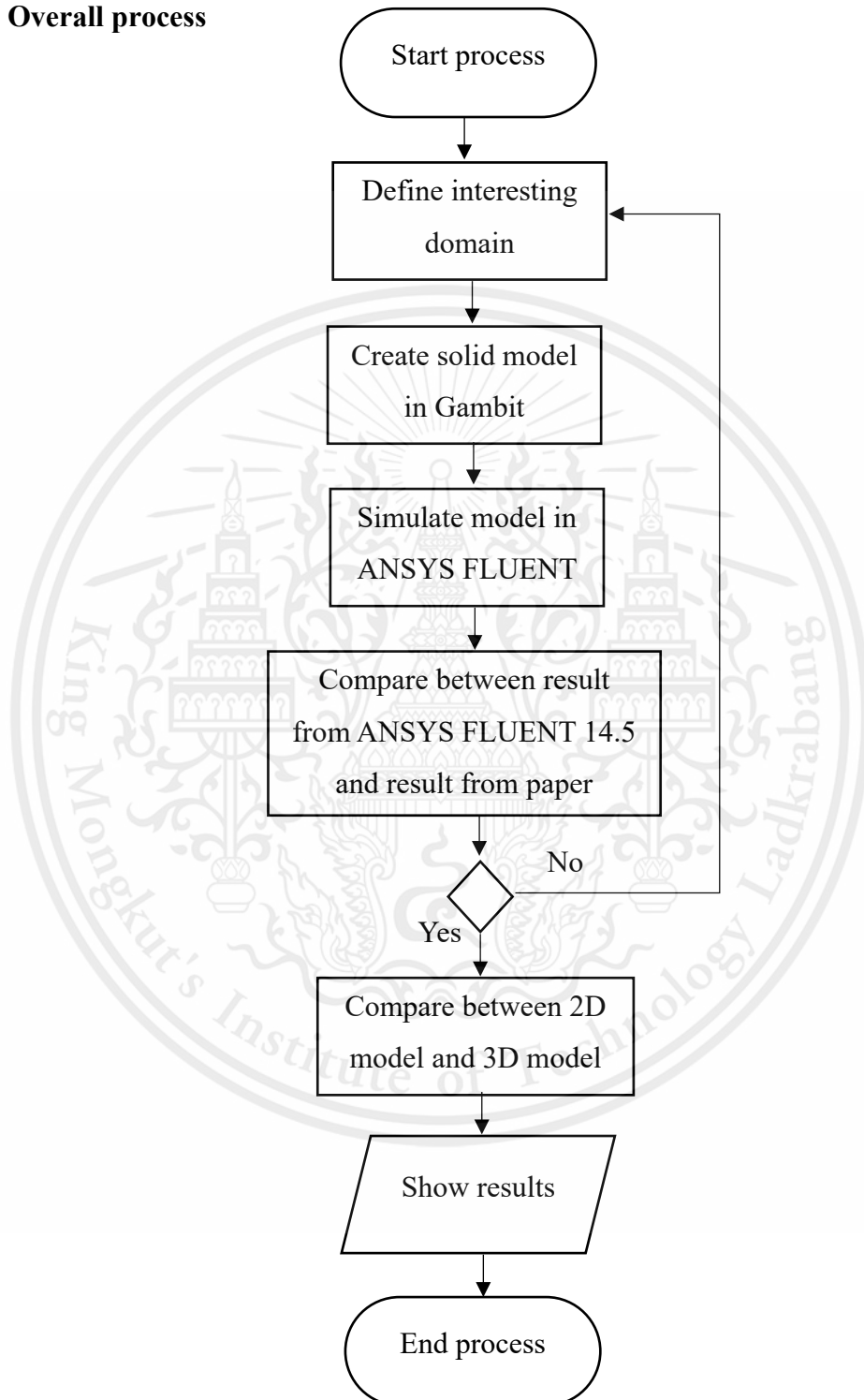


Figure 3.1: Overall process for this project

3.2 2D Model creation and Domain decomposition

Solid model of ejector from referent paper as show in Figure 3.2 is created by using Gambit. Model from Gambit is shown in Figure 3.3. Before this model is brought to ANSYS FLUENT 14.5, model is needed to create mesh or grid. For easier to create square mesh that suitable for calculation in ANSYS FLUENT 14.5, domain is needed to decompose. Model that domain is decomposed is shown in Figure 3.4. After that, mesh is created in decomposed model is shown in Figure 3.5.

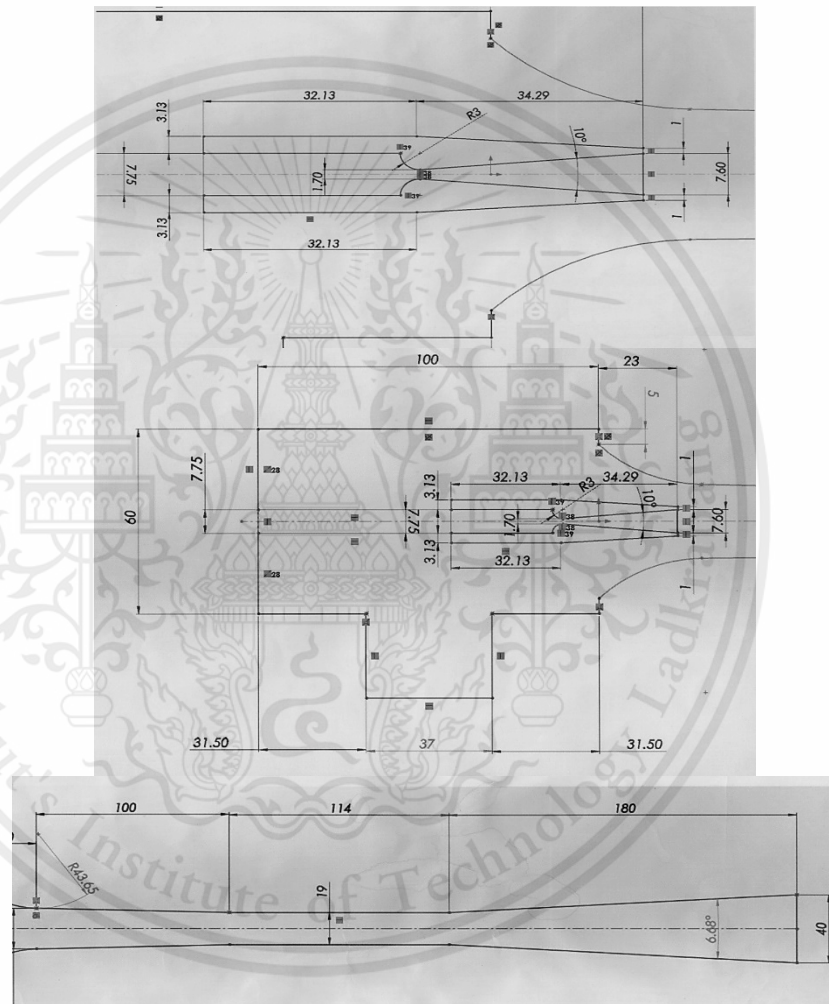


Figure 3.2: Configuration of ejector from reference paper



Figure 3.3: Solid model of ejector

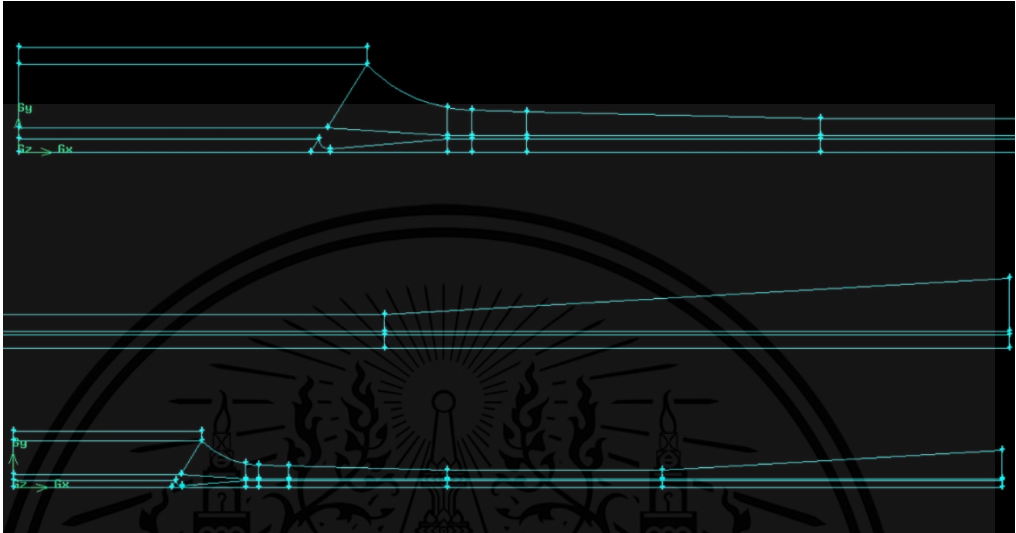


Figure 3.4: Decomposed model

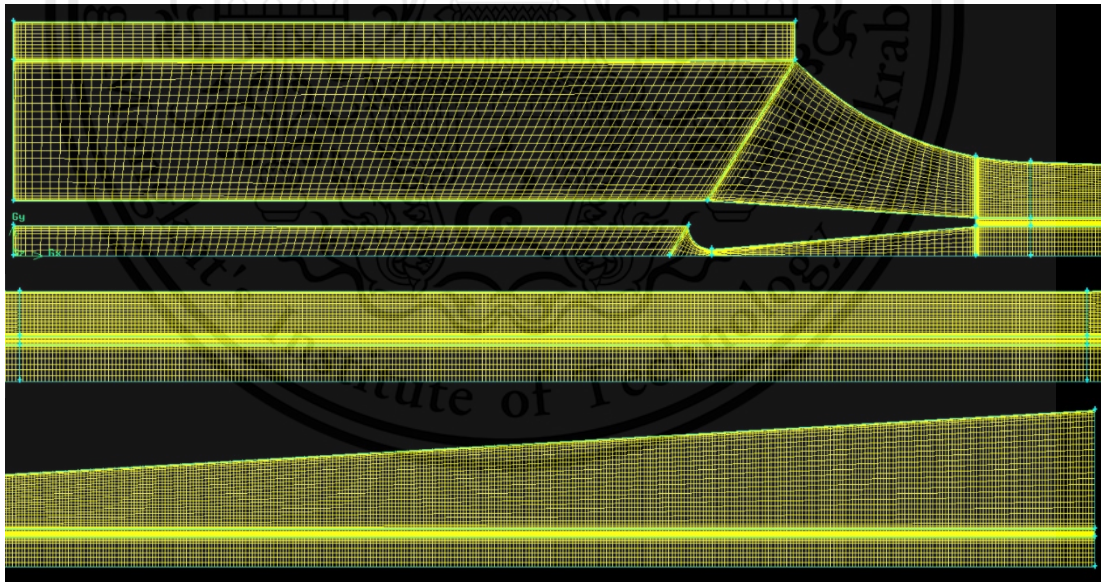


Figure 3.5: Mesh generation model

This material is reserved for educational use only, not allowed for commercial use.

Forbidden to modify the content, and cite the document when use

3.3 3D Model creation and Domain decomposition

In the process of creating a 3D model, it is more difficult than creating a 2D model because it has to create a full model from referent paper and change the input of the secondary inlet. Model from Gambit is shown in Figure 3.6.

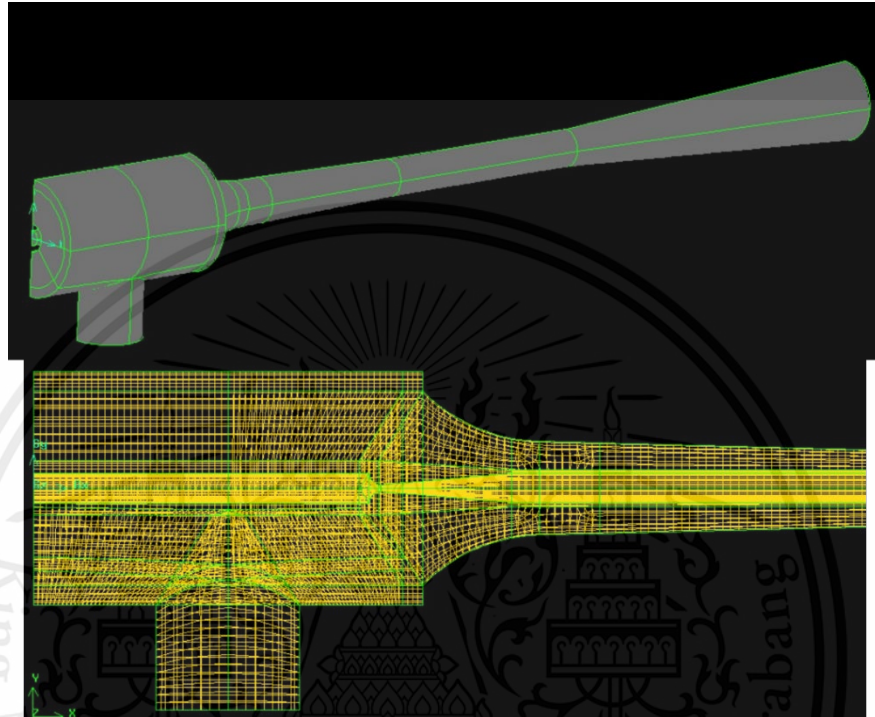


Figure 3.6: 3D model of ejector

3.4 Setting in ANSYS Fluent

3.4.1 Model for calculation

SST k-omega Model is equation used in ANSYS Fluent.[5]

3.4.2 Properties of material

Water-vapor properties (Unit)		
Density	Ideal-gas	kg/m ³
C _p (Specific heat)	2077.4	j/kg·K
Thermal Conductivity	polynomial	W/m·K
Viscosity	polynomial	kg/m·s
Molecular Weight	18.01534	kg/kgmol

Table 3-1: Properties of water-vapor

This material is reserved for educational use only, not allowed for commercial use.

Forbidden to modify the content, and cite the document when use

Density of water-vapor is set to ideal-gas for calculation in ANSYS FLUENT 14.5 that assume water-vapor in ejector is ideal-gas.

Thermal conductivity and viscosity are set to polynomial due to properties of water-vapor needed to calculate by using polynomial function.

3.4.3 Boundary Conditions Setting

Condition	Primary Inlet	Secondary Inlet	Outlet
Gauge Total Pressure (pascal)	270,467.3	1,050.962	3500
Supersonic Gauge Pressure (pascal)	270,259.6	1,037.003	-
Temperature (K)	403.206	280.5248	300
Direction Specification Method	Normal to Boundary	Normal to Boundary	-

Table 3-2: Boundary Conditions

CHAPTER IV

RESULTS AND DISCUSSION

This section discusses the results of computation using ANSYS Fluent of 3D models and their comparison with the results of 2D models taken from the thesis of Wangthanasarn et al.[6] and the laboratory results from the paper of Ruangtrakoon et al.[1]

4.1 Example of divergence result in ANSYS FLUENT 14.5

If result from calculation is divergence, setting will be wrong. So, setting must be modified for new calculation. Example of divergence result is shown in Figure 4.1.

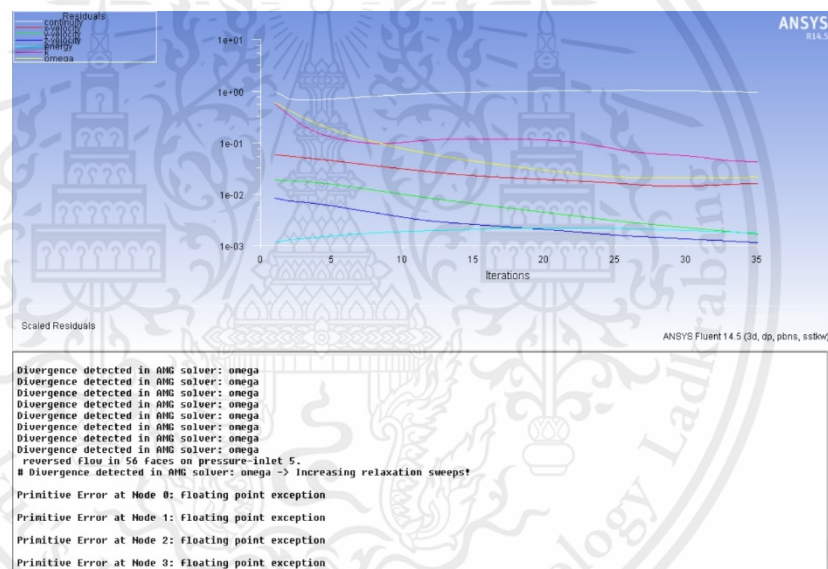


Figure 4.1: Divergence result

4.2 Compare the calculation results from ANSYS FLUENT 14.5

4.2.1 Comparison of Entrainment Ratio and Critical pressure

The conditions of result are following as

- Saturation temperature of evaporator is 7.5 °C.
- Temperature of boiler is 130 °C.
- Nozzle size is D1.7M4.

	3D Model	2D Model	Ref. Paper
Entrainment Ratio (R_m)	0.471	0.521	0.422
%Error	11.61	23.46	-
Critical Pressure	3500	3250	3500

Table 4-1: Comparison of R_m and Critical pressure

4.2.2 Comparison of axial velocity

Axial velocity of both of 2D and 3D models have same trend when fluid pass nozzle. Their axial velocity are increase after pass exit way of nozzle. But their trend are different at the place near outlet. I think it happen because problem in model. Mesh that I made for simulation is not fine enough and not symmetrical. Figure that compares between axial velocity of 2D model and 3D model is shown in Figure 4-1.

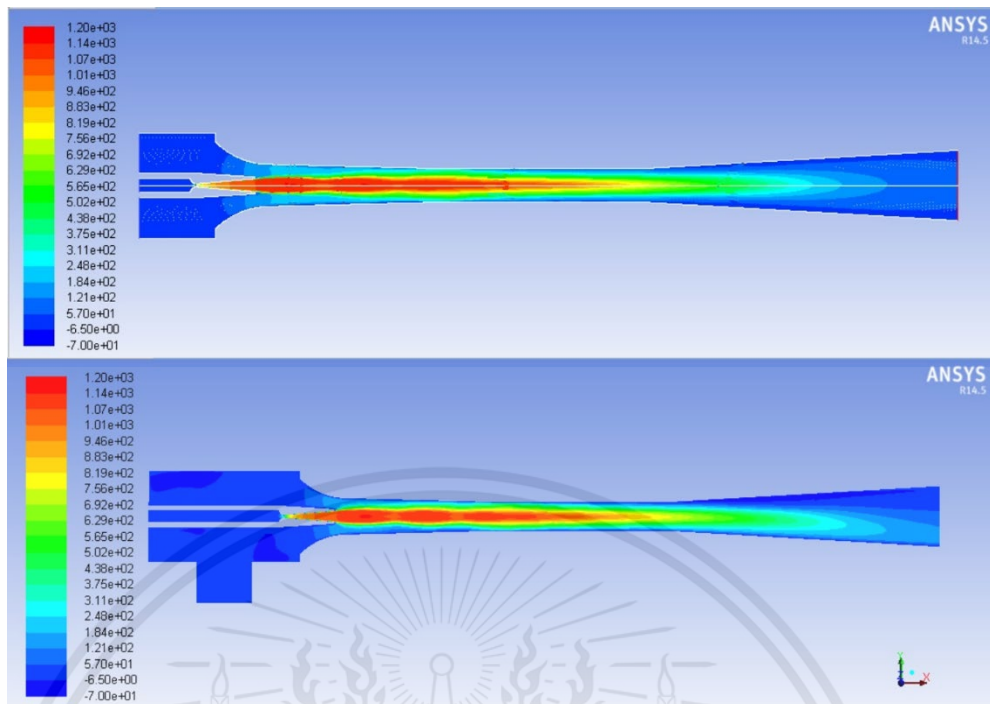


Figure 4.2: Axial velocity of 2D model (top) and 3D model (bottom)

4.2.3 Comparison of radial velocity

The change in radial velocity of both models is very similar. There is a difference in the secondary inlet range because the secondary fluid was pulled by change of velocity from exit way of primary nozzle.

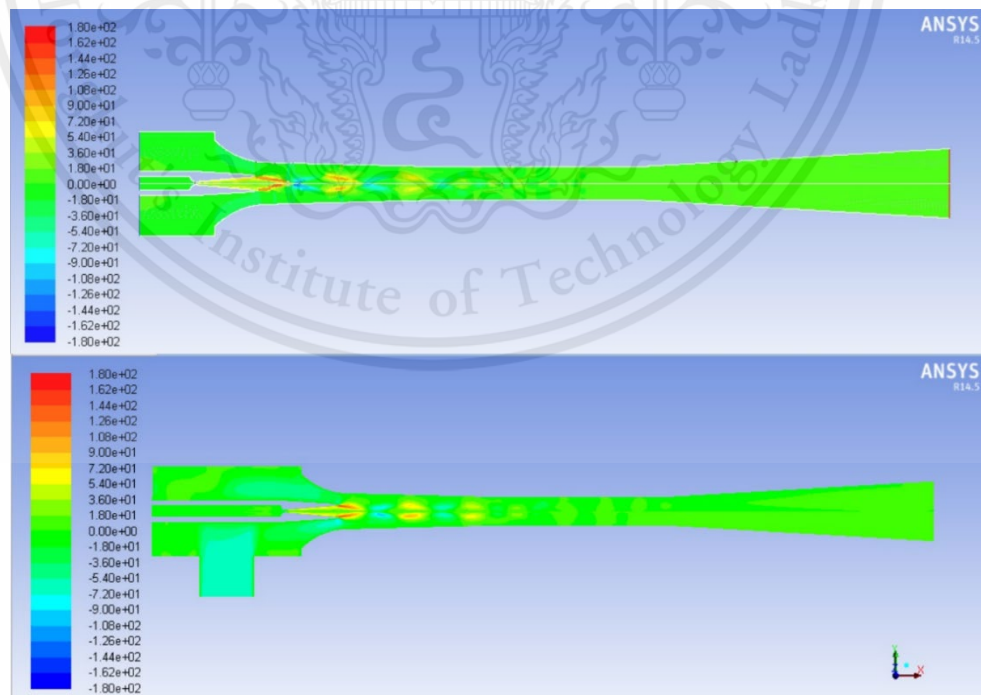


Figure 4.3: Radial velocity of 2D model (top) and 3D model (bottom)

4.2.4 Comparison of Temperature

The temperature of the fluid entering through the nozzle decreases when passes through the mixing chamber because of the decreased pressure. After the primary fluid is combined with the secondary fluid at the mixing chamber, the temperature changes. The temperature increases when the fluid flows to the diffuser because of the increased pressure. The contour shows the temperature change of both models.

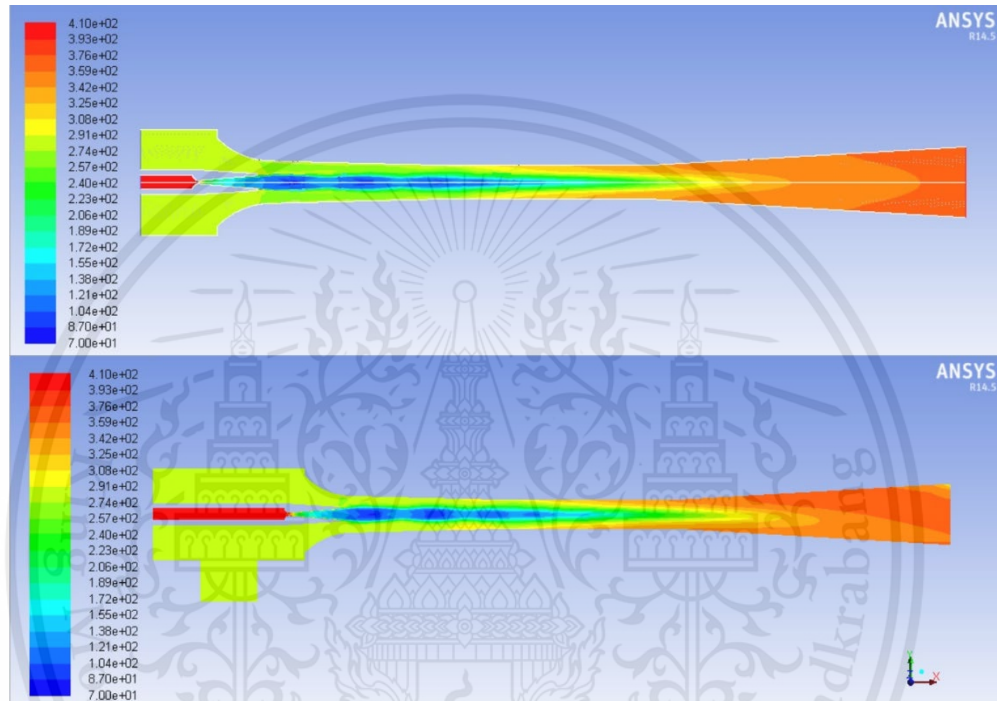


Figure 4.4: Temperature of 2D model (top) and 3D model (bottom)

4.2.5 Comparison of Mach Number

The tendency of Mach number is close to axial velocity due to their correlation. Figure that compares between axial velocity of 2D model and 3D model is shown in Figure 4.5.

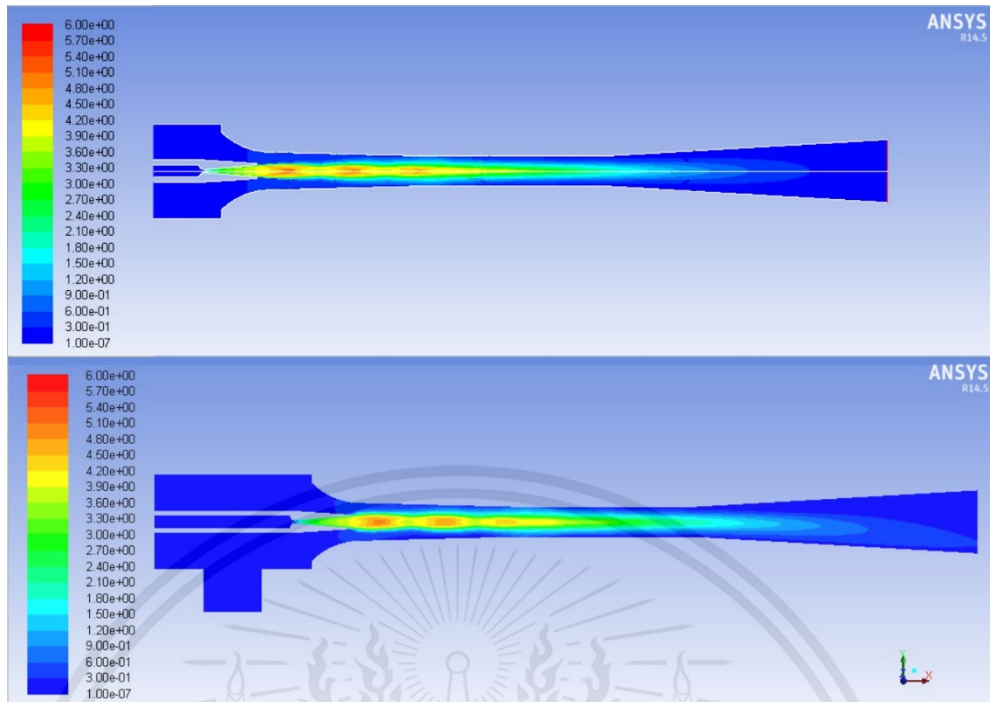


Figure 4.5: Mach Number of 2D model (top) and 3D model (bottom)

CHAPTER V

CONCLUSION

5.1 Conclusion

This study investigates fluid flow field of ejector. Solid model of ejector is created by using Gambit. Fluid flow simulation is calculated by using ANSYS Fluent. Solid model of ejector has reference model from paper of Natthawut Ruangtrakoon and team. Objectives of this study are finding suitable model for ejector in ejector refrigeration system and comparing how 2D vs 3D model simulations affect the flow field.

From results in Chapter 4, percentage error of Entrainment Ratio (R_m) of 3D model (11.61%) is less than 2D model (23.46%). Critical Pressure of 3D model (3500 Pa) is equal to critical pressure from paper but 2D model (3250 Pa) is lower. That shows calculations from 3D model are more accurate than 2D model. The result becomes like this because position of secondary inlet in the model 2D is different from 3D model. In addition to the displayed values, the condition in the ejector of the 2D model and the 3D model are different. But the trend of condition is similar and follow the theory of the ejector.

So, suitable model for simulating ejector flow is 3D model. Both models have different position of secondary inlets results in different conditions in the ejector. The difference is noticeable at the secondary inlet and diffuser range.

5.2 Suggestions

1. This study simulates model in 1 condition. If a better result is desired, it should be calculated in other conditions as well.
2. Models of equations given in fluent should be studied. Changing the equation model might make more accurate results.

REFERENCES

- [1] N. Ruangtrakoon, S. Aphornratana, and T. Sriveerakul, “Experimental studies of a steam jet refrigeration cycle: Effect of the primary nozzle geometries to system performance,” *Exp. Therm. Fluid Sci.*, vol. 35, no. 4, pp. 676–683, 2011, doi: 10.1016/j.expthermflusci.2011.01.001.
- [2] N. Jaiklom, D. Onprom, and T. Seethao, “Numerical investigation of performance characteristics of a cyclone prolonged with a divergent dipleg,” 2011.
- [3] “ผลการศึกษาเชิงปฏิบัติของระบบทำความเย็นแบบอีเจกเตอร์.”
<https://ph02.tci-thaijo.org/index.php/eit-researchjournal/article/view/174236/124794> (accessed Dec. 12, 2020).
- [4] Transvac, “Flare Gas Recovery with Ejectors .”
<https://www.transvac.co.uk/flare-gas-recovery/> (accessed Dec. 12, 2020).
- [5] “ANSYS FLUENT 12.0 User’s Guide.”
https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/main_pre.htm
 (accessed Dec. 20, 2020).
- [6] T. Wangthanasarn, C. Tanjaipetch, T. Amradis, “The Numerical Studies on The Performances of The Constant Pressure Mixing Ejector and Constant Rate of Momentum Change Ejectors Used in Steam Ejector Refrigeration System,” 2020.