

**Influence of Cyclone Diameter on Pressure Drop by Computational Fluid
Dynamics**

By

Kantapong Chumchuen



**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

อิทธิพลของเส้นผ่านศูนย์กลางของไซโคลนต่อความดันลดโดยพลศาสตร์ของไหลเชิง

คำนวณ



ปริญญาานิพนธ์นี้เป็นส่วนหนึ่งของการศึกษาตามหลักสูตร

วิศวกรรมศาสตรบัณฑิต สาขาวิชาวิศวกรรมปิโตรเคมี

ภาควิชาวิศวกรรมเคมี คณะวิศวกรรมศาสตร์

สถาบันเทคโนโลยีพระจอมเกล้าเจ้าคุณทหารลาดกระบัง

ปีการศึกษา 2563

Title Influence of cyclone diameter on pressure drop by computational fluid dynamic

By Kantapong Chumchuen

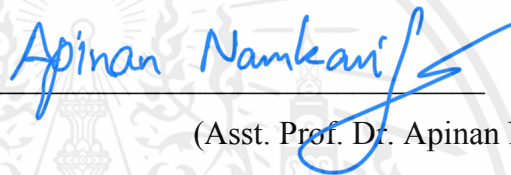
Field of Study Petrochemical Engineering

Advisor Asst. Prof. Dr. Apinan Namkanisorn

Co-advisor Asst. Prof. Dr. Santi Wattananusorn

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



Apinan Namkanisorn

Chairman
(Asst. Prof. Dr. Apinan Namkanisorn)



W. Santi

Committee
(Asst. Prof. Dr. Santi Wattananusorn)



Yaneeporn Patcharavorachot

Committee
(Assoc. Prof. Dr. Yaneeporn Patcharavorachot)

Title Influence of cyclone diameter on pressure drop
by computational fluid dynamic

By Kantapong Chumchuen

Advisor Asst. Prof. Dr. Apinan Namkanisorn,

Co-advisor Asst. Prof. Dr. Santi Wattananusorn

Field of Study Petrochemical Engineering

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

Abstract

This project uses computational fluid dynamic (CFD) to simulate fluid flow inside a cyclone in order to study the influence of cyclone diameter on pressure drop. Reynold stress model (RSM) is used in the simulation numerical calculation to investigate turbulent flow. Grid independent study is carried out and its result is compared with reference research. The grid independent produces fine mesh, medium mesh and coarse mesh. At different heights, the input inlet velocity is 16, 20, 22, 32 m/s, and the diameter is 0.205 m and 0.290 m. For computational fluid dynamics simulations inside the cyclone, the suitable mesh is used. The influence of cyclone diameter on pressure drop is also compared. The fine mesh is a suitable mesh size for simulating the influence of diameter on pressure drop at an inlet velocity of 16 m/s. The variation in diameters has a pronounced influence on pressure drop.

| | |
|----------------------|--|
| เรื่อง | อิทธิพลของเส้นผ่านศูนย์กลางของไซโคลนต่อความดันลดโดยพลศาสตร์ของไหลเชิงคำนวณ |
| โดย | กันตพงศ์ ชุ่มชื่น |
| อาจารย์ที่ปรึกษา | ผศ.ดร.อภิรักษ์ นัมคณิศรณ |
| อาจารย์ที่ปรึกษาร่วม | ผศ.ดร.สันติ วัฒนานุสรณ์ |
| สาขาวิชา | วิศวกรรมวิชาปิโตรเคมี |
| สังกัด | ภาควิชาวิศวกรรมเคมี คณะวิศวกรรมศาสตร์ สถาบันเทคโนโลยีพระจอมเกล้าเจ้าคุณทหารลาดกระบัง |

บทคัดย่อ

งานวิจัยนี้จำลองการไหลของของไหลภายในโดยใช้การจำลองพลศาสตร์ของไหลเชิงคำนวณ (Computational fluid dynamic, CFD) เพื่อตรวจสอบอิทธิพลของเส้นผ่านศูนย์กลางของไซโคลนต่อความดันลด ใช้แบบจำลอง Reynold stress model (RSM) เพื่อตรวจสอบการไหลแบบปั่นป่วน เปรียบเทียบกรกับการวิจัยอ้างอิงเพื่อสร้าง กริดละเอียด กลาง หยาบ ที่ความสูงต่างกัน มีความเร็วขาเข้าคือ 16, 20, 22, 32 เมตร/วินาที และเส้นผ่านศูนย์กลางคือ 0.205 เมตร และ 0.290 เมตร สำหรับการจำลองของไหลเชิงคำนวณภายในไซโคลนจะใช้กริดที่เหมาะสม นอกจากนี้เปรียบเทียบอิทธิพลของเส้นผ่านศูนย์กลางของไซโคลนที่มีต่อความดันลด กริดละเอียดนั้นเหมาะสมสำหรับการจำลองอิทธิพลของเส้นผ่านศูนย์กลางต่อความดันลดที่ความเร็วขาเข้าคือ 16 เมตร/วินาที ในการเปรียบเทียบนี้ พบว่าความแปรผันของเส้นผ่านศูนย์กลางนั้นมีผลต่อความดันลดอย่างชัดเจน

Acknowledgements

Asst. Prof. Dr. Apinan Namkanisorn, thesis advisor that provides advice, guideline, and knowledge in other areas until this research was successful.

Asst. Prof. Dr. Santi Wattananusorn thesis advisor that introduces the knowledge of cyclones guide various approaches and find research for seniors for further study until this research was successful.

Dr. Eakarach Bumrunghthaichaichan who teaches the use of fluid dynamics techniques and other fields of knowledge and is a person who has proposed different ways for me to complete this research.

I would like to thanks to Department of Chemical Engineering, King Mongkut's Institute of Technology Ladkrabang, technician scientist, and technicians of the laboratory for giving me some advice and assistance for resources in this thesis.

Finally, I most gratefully acknowledge my parents, my family, and my Department of Petrochemical Engineering friends for suggestions, advice, moral support, and all their help throughout the period of this thesis.

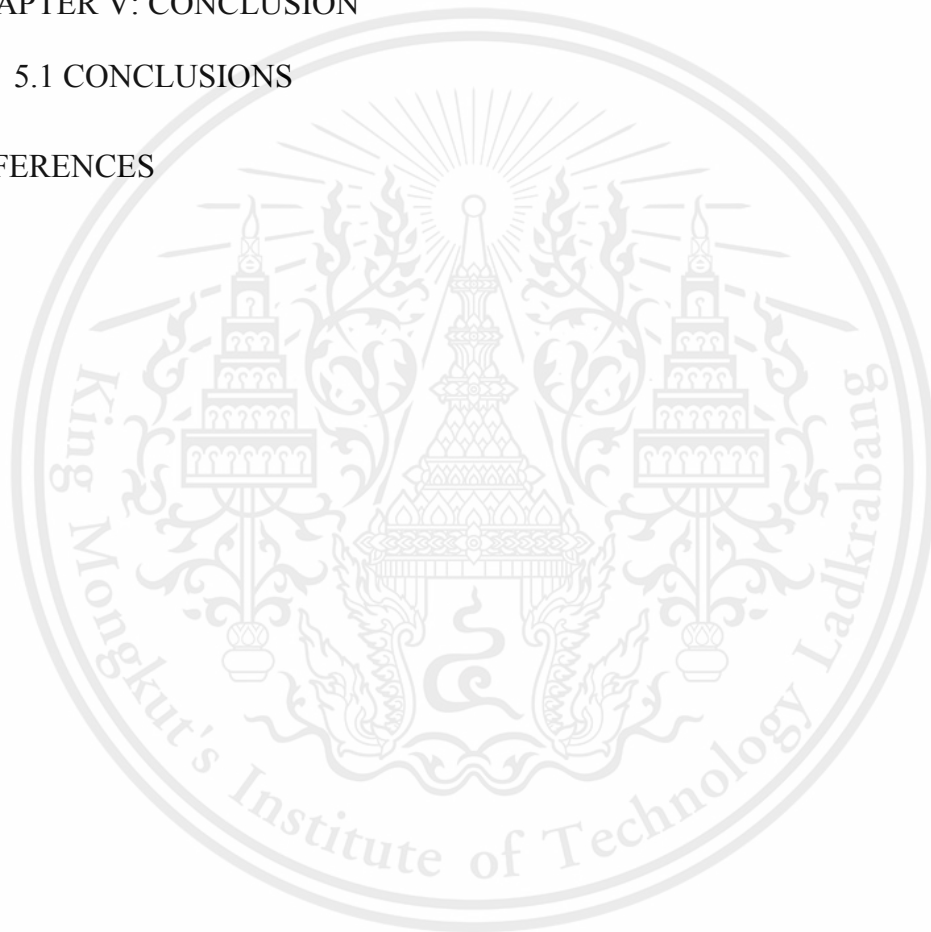
Kantapong Chumchuen

Table of Contents

| | Page |
|---|-------------|
| Abstract | IV |
| Acknowledgements | VI |
| NOMENCLATURE | 10 |
| CHAPTER I: INTRODUCTION | 1 |
| 1.1 Background | 1 |
| 1.2 Objectives | 1 |
| 1.3 Scopes of Work | 1 |
| 1.4 Expected Outputs | 2 |
| CHAPTER II: THEORY | 3 |
| 2.1 Cyclone | 3 |
| 2.1.1 Mechanical collectors | 3 |
| 2.1.2 Collection mechanisms | 3 |
| 2.1.3 Principle | 3 |
| 2.2 Computational Fluid Dynamics (CFD) | 4 |
| 2.2.1 Pre-Processor | 5 |
| 2.2.2 Solver | 5 |
| 2.2.3 Post-Processor | 5 |
| 2.3 Navier-Stokes Equation | 6 |
| 2.3.1 Continuity equation | 6 |
| 2.3.2 Momentum equation | 6 |
| 2.3.3 Energy equation | 7 |
| 2.4 Reynold-Averaged Navier Stokes Equation | 8 |
| 2.5 Reynolds Stress Model (RSM) | 9 |
| 2.6 Literature researcher | 12 |
| CHAPTER III: RESEARCH METHODOLOGY | 13 |

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

| | |
|---|----|
| 3.1 Grid generation and Domain decomposition | 13 |
| 3.2 ANSYS Fluent | 16 |
| CHAPTER IV: RESULTS AND DISCUSSION | 18 |
| 4.1 Grid independent | 18 |
| 4.2 Comparison result with reference research | 22 |
| 4.3 Calculation of diameter impact on pressure drop | 25 |
| CHAPTER V: CONCLUSION | 27 |
| 5.1 CONCLUSIONS | 27 |
| REFERENCES | 28 |



Lis of Figure

| | Page |
|---|-------------|
| Figure 2.1: Components of cyclone | 4 |
| Figure 3.1: Operation flowchart | 13 |
| Figure 3.2: Cyclone structure | 14 |
| Figure 3.3: 3D cyclone model, domain decomposition and grid generation in program GAMBIT | 15 |
| Figure 3.4: Mesh size A, B, and C are 447556, 243780, and 143592 respectively (Fine mesh, medium mesh, coarse mesh) | 17 |
| Figure 4.1: Axial velocity of 3 mesh using (fine, medium, and coarse mesh) size at different Z/D (0.226, 0.436, 0.506, and 0.616) | 19 |
| Figure 4.2: Tangential velocity of 3 mesh using (fine, medium, and coarse mesh) size at different Z/D (0.226, 0.436, 0.506, and 0.616) | 21 |
| Figure 4.3: Axial velocity of fine mesh and experiment data at different Z/D (0.226, 0.436, 0.506, and 0.616) | 23 |
| Figure 4.4: Tangential velocity of fine mesh and experiment data at different Z/D (0.226, 0.436, 0.506, and 0.616) | 25 |
| Figure 4.5: Contour of Axial velocity and Tangential velocity of fine mesh | 25 |

NOMENCLATURE

English alphabet symbols

| | |
|-----------------------------|---|
| u, v, w | Velocity in the x, y, z directions respectively |
| t | Time |
| V | Fluid velocity |
| g | Gravity force |
| Re | Reynold number |
| $\bar{u}, \bar{v}, \bar{w}$ | Average velocity in the x, y, z direction, respectively |
| u', v', w' | Swing velocity from the average in the x, y, z directions, respectively |
| \bar{P} | Average pressure |
| R_{ij} | Reynold Stress |
| P_{ij} | Stress Production |
| D_{ij} | Diffusion |
| C_{μ} | Convection Term |
| k | Turbulent kinetic energy |
| T_i | Turbulent intensity |
| F_D | Drag force per particle unit mass |
| d_p | Particles diameter |
| C_D | Drag coefficient |
| u | Gas velocity |
| u_p | Particles velocity |
| Re_r | Absolute Reynold number |
| F_x | Increasing force per unit mass of particles |

Greek alphabet symbol

| | |
|--------------------|---|
| ρ | Fluid density |
| τ | Tensor of stress |
| μ | Viscosity |
| ε_{ij} | Dissipation |
| π_{ij} | Pressure strain |
| Ω_{ij} | Production by system Rotation |
| δ_{ij} | Kronecker delta |
| σ_{ij} | Turbulent Prandtl number for kinetic energy |
| ε | Rate of kinetic energy of the disorder |
| Ω_k | Vector showing the direction of rotation |
| ρ_p | Particle density |
| Ω | Interference factor |
| θ | Angle of flat plate |

CHAPTER I

INTRODUCTION

1.1 Background

In a present, there are many industries cause of pollution such as small dust from operation process. Small dust affects to health of the operator and community surrounding area by respired. Therefore, many industries use high efficiency small dust control or protection equipment such as cyclone, wet scrubber, fabric filter, and electrostatic precipitator. The most used cyclone because low cost and decrease maintenance problem.

Mechanical of cyclone separator are removing dust or particles from air or gas. Centrifugal force caused by vortex inside the cyclone that swing to the wall of cyclone. The particle will fall to the bottom with gravity force. For studying to increase the efficiency of cyclone is not predict the actual simulation form improvement.[1]

At the present, the Computational Fluid Dynamic (CFD) is applied to help easily and accurately study cyclone efficiency. This technique is modeling on three-dimensional using the Reynold Stress model (RSM), Turbulence, and Large Eddy simulation (LES) and Direct numerical simulation (DNS).[2]

1.2 Objectives

- 1.2.1. To use GAMBIT program to create the model and simulate the flow in cyclone using program ANSYS Fluent 14.5
- 1.2.2. To study the influence of cyclone diameter on pressure drop by computational fluid dynamics (CFD)

1.3 Scopes of Work

- 1.3.1 Simulation of cyclone without dustbin.
- 1.3.2 Grid generation and domain decomposition created by the program GAMBIT.
- 1.3.2 Calculation of fluid flow in cyclone by the numerical method using ANSYS Fluent 14.5 program.

1.4 Expected Outputs

- 1.4.1. Learn and understand how to create Grids using program GAMBIT.
- 1.4.2. Learn and understand the calculation of fluid flow in cyclone by the Numerical method using ANSYS Fluent 14.5 program.
- 1.4.3. Gain knowledge on the influence of cyclone diameter on pressure drop.



CHAPTER II

THEORY

2.1 Cyclone

2.1.1 Mechanical collectors

Mechanical collector separate dust or particles from air or gas using gravity or inertia force or both. Example of mechanical separators include gravity setting chambers and cyclone collectors.

Cyclone is an equipment for separating particle from gas using centrifugal force which is caused by vortex.[3]

2.1.2 Collection mechanisms

- Centrifugal force which is caused by vortex. Particle is casted into wall of the cyclone.
- Gravity force, the particle adheres to the wall of the cyclone which is separated from air and dropdown with gravity force.[1]

2.1.3 Principle

The cyclone is consisting of cylinder at the tip cone as shown in figure 2.1. Air flow into cyclone creating spiral vortex (Main vortex or outer vortex) which occurring the centrifugal force to swing particle to the wall cyclone. The main vortex is moves downward to the bottom of cyclone and create the inner vortex closer to center of cyclone. The inner vortex is spiral upward to the top (It is called inner vortex or core vortex). This is double vortex that occur in same direction. For particles are dropdown to outlet of cyclone.[3],[4]

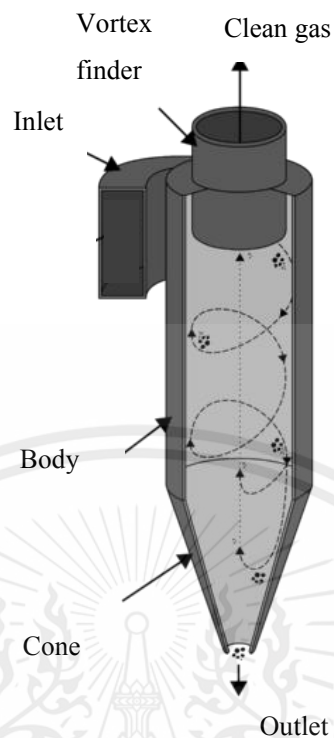


Figure 2.1: Components of cyclone[3]

2.2 Computational Fluid Dynamics (CFD)

CFD uses numerical and methods to solve mathematical exact numbers. The problems related to fluid heat transfer or phenomena related to chemical reactions based on mass, momentum, and energy was solved by CFD. This technique can be calculated the position output such as pressure and temperature. Techniques CFD is widely to apply in all industries.

In general, analysis program has high cost but not higher than experiment and has require high quality. Therefore, this technique CFD has the advantage as follow:

- Able to study system which cannot be proved.
- Able to decrease time and cost of new designs.
- Able to study system risk or exceed normally limit which system is operating before unexpected danger occurs.
- The results that require to use for work are not limited to details. This makes it possible to design the equipment to operate in the optimal conditions, both in terms of efficiency and economics.[5],[6]

The principle of the CFD can be divided into 3 parts.

2.2.1 Pre-Processor

Pre-Processor is input the data of flow problem to calculation program in this part includes:

- Define the shape of the scope that interest and determines the scope of the problem requires to be calculated.
- Divided scope of the problem into more small parts and not collapse.
- Choose the equation or model of physical or chemical for calculated the phenomena that interested.
- Define the properties of fluid.
- Define the boundary conditions that suitable for problems.

The number of cells controlled the accuracy of the answers from the fluid dynamics calculation method. In general, a greater number of cells the answers will be more accurate. The accurate of the answers, cost, and time for calculated depend on the resolution of cells. Computer performance is limited by cell division for calculations.

2.2.2 Solver

Solver is divided into 4 types of numerical calculation: Finite Element Method, Finite Difference Method, Finite Volume Method, and Spectral method. In general, the steps of calculation are as follows:

- Estimate an unknown flow variable with simple function.
- Convert Partial differential equations to algebraic equations using numerical method.
- Solve the algebraic equations with repetitive actions.

2.2.3 Post-Processor

Post-Processor is the results part. This program can show the results from calculations in other forms such as:

1. Shape of boundary of problems and grid.
2. Vector
3. Contour and Lines

4. 2D and 3D textures.
5. Particle Tracking
6. Alignment, viewing (moving, rotating, adjusting, etc.).

2.3 Navier-Stokes Equation

Navier-Stokes Equation is used for describing the movement of fluid that created by applied the Newton's second law of motion on fluid and combined with assumption of the stress on fluid. The assumption of the stress on fluid is the sum of viscosity of the spreading and pressure. The Navier-Stokes Equation is a differential equation that does not specify the relation between certain variables, but specific as the rate of change. The Navier-Stokes Equation cannot specify the positions, but this equation can specify the velocity. The answer of Navier-Stokes Equation is called that velocity field or flow field. That explain to velocity of fluid at the position and time. In general, Navier-Stokes Equation consist of three equations: Continuity equation, Momentum equation, and Energy equation, there is shown as follows:[6],[7]

2.3.1 Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \quad 2-1$$

2.3.2 Momentum equation

X component:

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \quad 2-2$$

Y component:

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \quad 2-3$$

Z component:

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + S_{Mz} \quad 2-4$$

2.3.3 Energy equation

$$\begin{aligned} \rho \frac{DE}{DT} = & -\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} + \frac{\partial}{\partial x} \left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(k \frac{\partial T}{\partial z} \right) \\ & + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} \\ & + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} + S_E \end{aligned} \quad 2-5$$

Where

| | | | |
|--------|-------------------|-----------|--|
| V | velocity of fluid | g | gravity force |
| ρ | density of fluid | u, v, w | velocity in direction x, y, z respectively |
| p | pressure | τ | normal stresses on planes |
| T | temperature | k | turbulent kinetic energy |
| E | total energy | | |

2.4 Reynold-Averaged Navier Stokes Equation

Characteristics of flow is calculated by Direct Numerical Simulation, (DNS). It requires large amounts of resources for calculation such as very small grids are required for calculations. The size of grid is exceedingly very small to measure characteristics of flow. Navier-Stokes Equation is applied by increasing the average velocity of current flow and increased the oscillation of velocity that explain characteristics of flow at any time. The velocity of oscillation is specified into other models which decrease the resources for calculation but the more error such as LES or RANS.

The problems of Direct Numerical Simulation, (DNS), the researcher choose calculation only Large Eddy Simulation (LES). Whatever this model required the high-performance computer and more time to calculated. The problems of Large Eddy Simulation (LES) developed as Reynolds averaged Navier-Stokes equations (RANS). This model is difficult to calculation of oscillation of velocity in terms Reynolds stress.[2]

Reynolds averaged Navier-Stokes equations (RANS) is the model which shown characteristics of flow at average time. This model is used for oscillation of velocity. The equation can estimate quality for solve the Navier-Stokes equations like Newtonian fluid as follows:

$$\rho \frac{\partial \overline{u'_j u'_i}}{\partial x_j} = \overline{\rho f_i} + \frac{\partial}{\partial x_j} \left[-\overline{p} \delta_{ij} + \mu \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) - \overline{\rho u'_i u'_j} \right] \quad 2-6$$

Where

| | | | |
|------------------|--------------------------------------|---------------|-----------------|
| ρ | density of the fluid | δ_{ij} | Kronecker delta |
| \overline{u} | average velocity | u' | swing velocity |
| $\overline{f_i}$ | force from the average object weight | μ | fluid viscosity |

$\rho \frac{\partial \overline{u'_j u'_i}}{\partial x_j}$ shows the change of average momentum value of component of fluid

causing instability of average flow and conveying a current average flow. This change is replaced by the mean of the body force show as follow:

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

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

- Value of isotropic stress caused by average pressure field.
- Value of viscous stresses
- Value of apparent stress, $(-\rho\overline{u_j u_i})$ caused by the field of oscillation of velocity.

Three values are called that Reynolds stress by terms of Nonlinear Reynolds stress required the other models. It is similar to RANS which branching into other turbulent models such as:

1. Zero equation model Mixing length
2. One equation model Spalart-Almaras
3. Two equation model k- ε style (Standard, RNG, Realizable), k- ω and ASM
4. Seven equation model Reynolds stress

2.5 Reynolds Stress Model (RSM)

Reynold Stress Model (RSM) describes the characteristics of turbulent flow that is similar to Navier-Stokes equations (RANS). The equation is adding for calculate characteristics caused by Reynolds Stress value. This equation is using for simulation the other factor caused by high-level stress of flow and the factors due to Reynolds Stress of characteristic of flow not equal in each direction. The different calculation of Reynolds Stress value at each position, $(-\rho\overline{u_j u_i})$ or write in Kinematic Reynolds Stress

$$\text{as } R_{ij} = \frac{-\tau_{ij}}{\rho} = \overline{u'_i u'_j} . [2]$$

where τ is the Reynolds Stress Tensor value and the equation of motion as follows;

$$\frac{\partial}{\partial t}(\rho\overline{u'_i u'_j}) + \frac{\partial}{\partial t}(\rho u_k \overline{u'_i u'_j}) = -\frac{\partial}{\partial x_k} \left[\rho\overline{u'_i u'_j u'_k} + p(\delta_{kj} u'_i + \delta_{ik} u'_j) \right]$$

$$\begin{aligned}
& -\rho \left(\overline{u'_i u'_j} \frac{\partial u_j}{\partial x_k} + \overline{u'_j u'_k} \frac{\partial u_i}{\partial x_k} \right) + p \left(\frac{\partial u'_i}{\partial x'_i} + \frac{\partial u'_j}{\partial x'_j} \right) - 2\mu \overline{\frac{\partial u'_i}{\partial x_k} \cdot \frac{\partial u'_j}{\partial x_k}} \\
& - 2p\omega_k \left(\overline{u'_j u'_m} \varepsilon_{ikm} + \overline{u'_i u'_m} \varepsilon_{jkm} \right) + \frac{\partial}{\partial x_k} \left[\mu \frac{\partial}{\partial x_k} \left(\overline{u'_i u'_j} \right) \right] \\
& - \rho\beta \left(g_i \overline{u'_j \theta} + g_j \overline{u'_i \theta} \right) + S_{user}
\end{aligned} \tag{2-7}$$

where

| | | | |
|----------|-------------------------------|---------------------|------------------------|
| ρ | Fluid density | u' | Fluctuating Velocity |
| u | Velocity | μ | Viscosity |
| p | Pressure of fluid | ε_{jkm} | Permutation Symbol |
| ω | Ratio by mass | δ_{ij} | Kronecker delta |
| β | Thermal expansion coefficient | g | Force from object mass |

Or can be written in the form

$$\frac{\partial}{\partial t} \left(\rho \overline{u'_i u'_j} \right) + C_{ij} = D_{T,ij} + P_{ij} + \Pi_{ij} - \varepsilon_{ij} + \Omega_{ij} + D_{L,ij} + G_{ij} + S_{user} \tag{2-8}$$

Where

| | | | |
|--------------------|-------------------------------|------------|--------------------------|
| ε_{ij} | Dissipation | S_{user} | User-Defined Source Term |
| C_{ij} | Convection Term | $D_{T,ij}$ | Turbulent Diffusion |
| $D_{L,ij}$ | Molecular Diffusion | P_{ij} | Stress Production |
| G_{ij} | Buoyancy Production | Π_{ij} | Pressure Strain |
| Ω_{ij} | Production by System Rotation | | |

Where value of $D_{L,ij}$, G_{ij} and S_{user} are added into the equation for completely calculation of fluid flow patters. The RSM equation is written as follows:

$$\frac{\partial}{\partial t}(\overline{\rho u'_i u'_j}) + C_{ij} = P_{ij} + D_{T,ij} + \Pi_{ij} - \varepsilon_{ij} + \Omega_{ij} \quad 2-9$$

Describing the changes in field of flow follow Reynolds Stress value that specify in equation of motions in six equations.

Table 2.1: Show the equations is used above as models

| Term | Equation | Equation for calculation | The original of equations |
|--------------------|---|--|--|
| $D_{L,ij}$ | $-\frac{\partial}{\partial x} \left[\overline{\rho u'_i u'_j u'_k} + p(\delta_{kj} u'_i + \delta_{ik} u'_j) \right]$ | $-\frac{\partial}{\partial x_m} \left(\frac{\nu_t}{\sigma_k} \cdot \frac{\partial \overline{u'_i u'_j}}{\partial x_m} \right)$ | The assumption of the diffusion gradient. |
| Π_{ij} | $-p \left(\frac{\partial u'_i}{\partial x'_j} + \frac{\partial u'_j}{\partial x'_i} \right)$ | $-C_1 \frac{\varepsilon}{k} \left(\overline{u'_i u'_j} - \frac{2}{3} k \delta_{ij} \right) - C_2 \left(P_{ij} - \frac{2}{3} P \delta_{ij} \right)$ | Usually uses the Launder model |
| ε_{ij} | $2\mu \frac{\partial u'_i}{\partial x_k} \cdot \frac{\partial u'_j}{\partial x_k}$ | $\frac{2}{3} \varepsilon \delta_{ij}$ | Calculate form standard ε equation |

By

ν_t Turbulent kinematic viscosity

P Pressure equal $0.5 P_{ij}$

C_1 and C_2 is 1.8 and $0.6 P$, respectively. (In case of Launder model)

For this model, term of Pressure Strain Π_{ij} is term that adjusts the pressure oscillation. Due to interaction between eddy current that occurs together and interaction between eddy current and field of fluid flow at different average velocity. The result occurs by adjusting Normal Reynolds Stress value to increase oscillation in isotropic. And reduce the Shear Reynolds Stress at close to the wall that occur the Anisotropic oscillation of pressure in perpendicular to the wall. So that is not required assumption

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

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

of the isotropic oscillation of pressure that occurs. This model is accurate and close to reality.

Although RSM is little complicate, it is the easiest model for group simulation of turbulent phenomena, especially for high stress and vortex flow such as flow in cyclone. There is ability to predict the characteristics of flow.

Reynolds Stress Model (RSM) has advantages and disadvantages as follow:

Advantages

1. Potential of this model is covering the flow pattern.
2. The calculation required only condition or the original variable value.
3. Cover the turbulent that occurs from oscillation isotropic and anisotropic of pressure. Not require the assumption of isotropic.
4. High accuracy in calculation quality of flow including Reynolds Stress value, simple flow, and complicate flow.

Disadvantages

1. Required high end computer.
2. This model is not verified and experimental compared by k-epsilon
3. The result form simulation of characteristics flow is similar to the k-epsilon model such as unconfined jet.

The source of computers is required high performance for calculation. Currently many programs are developed to predict the flow.[8]

2.6 Literature researcher

SLACK, M. D.[2] reviewed the application of Computational Fluid Dynamics (CFD) for 3D cyclone model and uninstalled the dustbin. That use the three-dimensional the meshes using the Reynolds Stress Turbulence model and Large Eddy Simulation. The result of this reviews shows the compared with Laser Doppler Anemometry measurement at axial belong the length of Stairmand high efficiency cyclone.

CHAPTER III RESEARCH METHODOLOGY

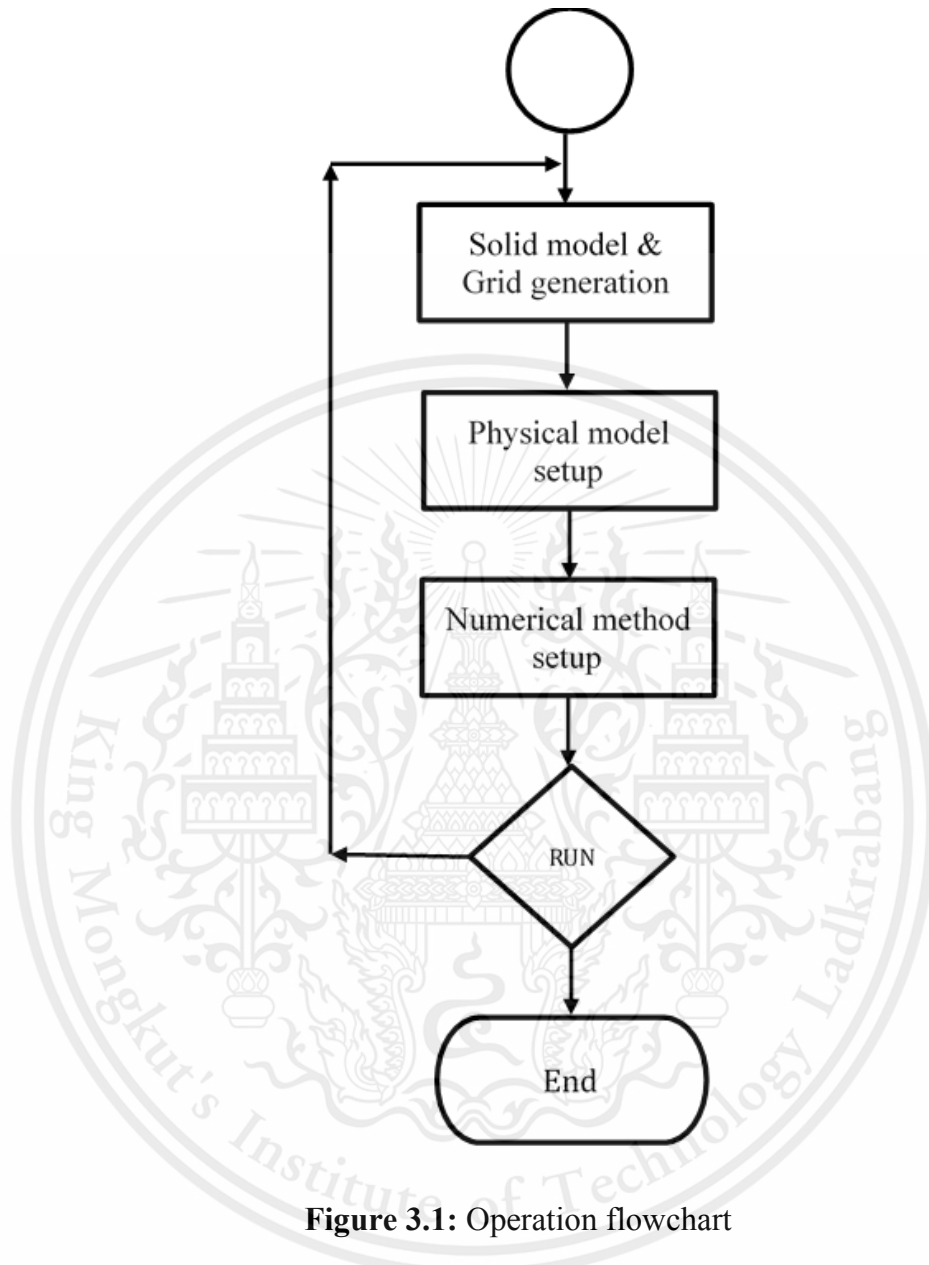


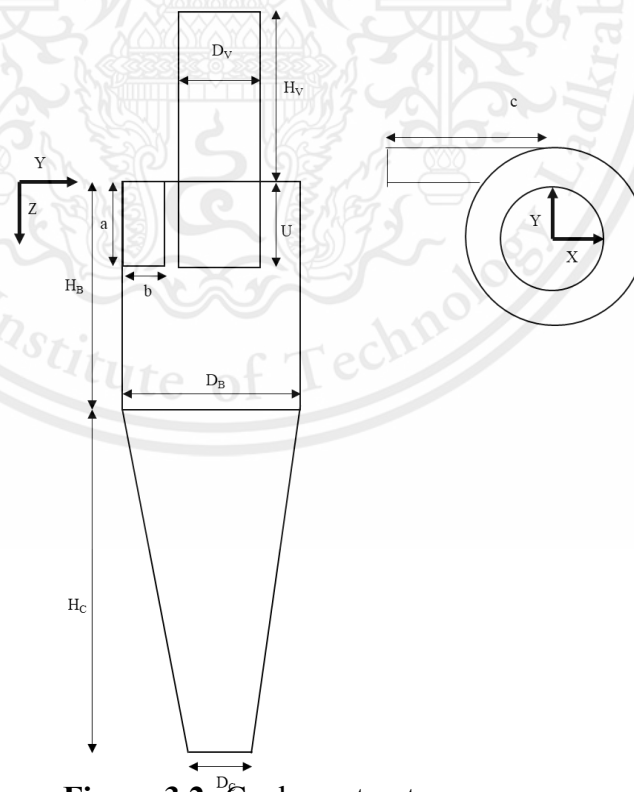
Figure 3.1: Operation flowchart

3.1 Grid generation and Domain decomposition

Solid model and grid generation of cyclone use the program GAMBIT to analysis the result. First, created the 3D cyclone model using the program GAMBIT reference scale from the Table 3.1 and Figure 3.2.[2] that are reference researcher. The 3D cyclone model and the domain decomposition are created. Next, creating optimum mesh or grid is place in the domain decomposition as shown in Figure 3.3

Table 3.1: The cyclone model is sized by the reference research.

| Type | Dimension | Length (mm) | Dimension/D |
|----------------------|-----------------------------------|-------------|-------------|
| Vortex finder | Inner vortex finder height, U | 102.50 | 0.5 |
| | Vortex finder diameter, D_v | 102.50 | 0.5 |
| | Outer vortex finder height, H_v | 307.50 | 1.5 |
| Barrel | Barrel diameter, D_B | 205.00 | 1 |
| | Barrel height, H_B | 307.50 | 1.5 |
| Cone | Cone tip diameter, D_c | 73.80 | 0.37 |
| | Cone height, H_c | 512.50 | 2.5 |
| Inlet | Inlet height, a | 102.50 | 0.5 |
| | Inlet width, b | 41.00 | 0.2 |
| | Inlet length, c | 174.25 | 0.85 |

**Figure 3.2:** Cyclone structure

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

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

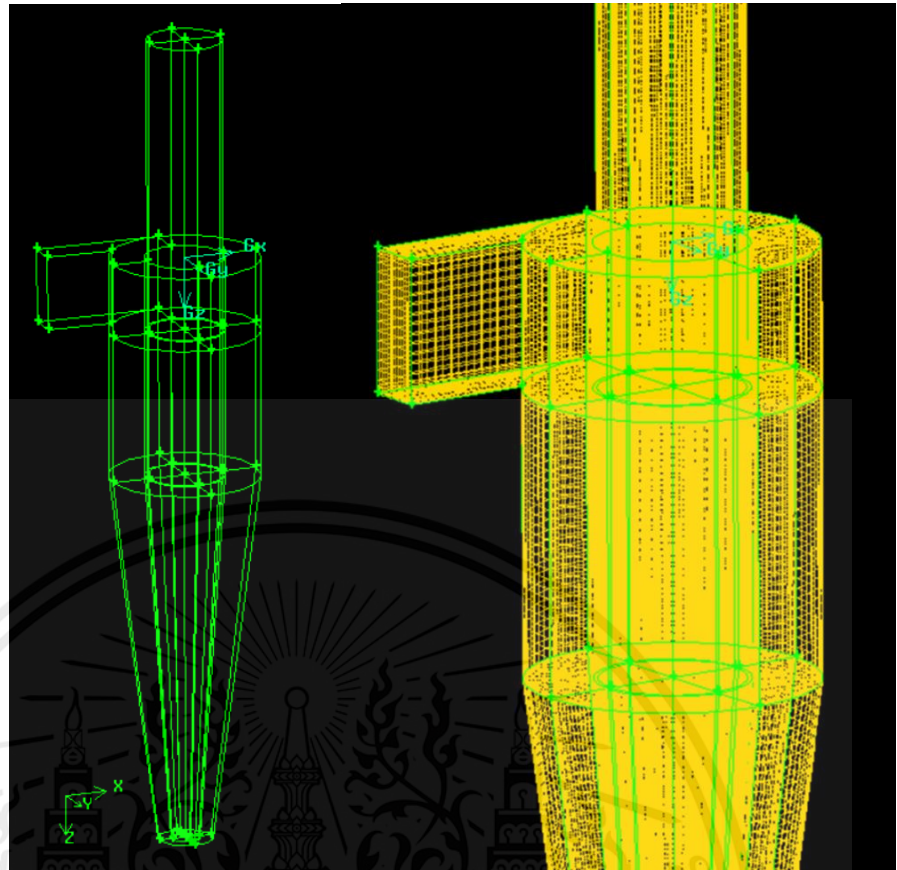


Figure 3.3: 3D cyclone model, domain decomposition and grid generation in program
GAMBIT

3.2 ANSYS Fluent

Use the data from table 3.2, table 3.3, and table 3.4 to set the program ANSYS Fluent.

Table 3.2: Gas properties enter to the cyclone.

| Gas properties | | |
|-------------------------|-----------------------|-------------------|
| Gas density (ρ) | 1.2250 | kg/m ³ |
| Gas viscosity (μ) | 1.79×10^{-5} | kg/(ms) |

Table 3.3: Inlet conditions

| Inlet conditions | | |
|------------------------------|--------|----------------|
| Inlet height (a) | 0.1025 | m |
| Inlet width (b) | 0.0410 | m |
| Inlet area (A) | 0.0042 | m ² |
| Inlet perimeter (P) | 0.2870 | m |
| Hydraulic diameter (D_h) | 0.0586 | m |
| Inlet velocity (U_{in}) | 20 | m/s |

Table 3.4: Boundary conditions is used for calculating the cyclone model from research reference.

| Boundary conditions | | |
|--|------------|--------------------------------|
| Turbulence intensity (l) | 0.0390 | - |
| Turbulence kinetic energy (k) | 0.9128 | m ² /s ² |
| Turbulence dissipation rate (ε) | 49.9267 | m ² /s ³ |
| Reynold number based on hydraulic diameter (Re_{Dh}) | 80194.4800 | - |
| Normal Reynold stresses ($2k/3$) | 0.6085 | m ² /s ² |
| C_μ | 0.0900 | - |

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

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

From calculation of ANSYS Fluent have result as Axial velocity, Tangential velocity, and Pressure drop that trend to be close to the experiment. The grid independent process uses for generating new mesh to 3 mesh size as 447556(fine), 243780(medium), and 143592(coarse) at Z/D position as 0.226, 0.436, 0.506, and 0.616.

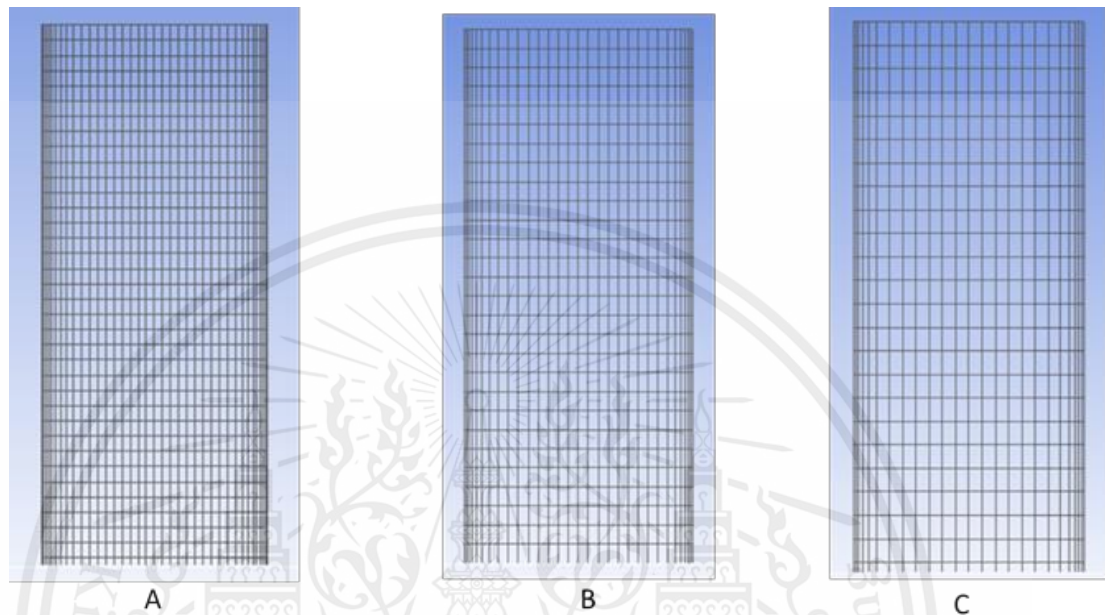


Figure 3.4: Mesh size A, B, and C are 447556, 243780, and 143592 respectively (Fine mesh, medium mesh, coarse mesh)

Compare the axial velocity, tangential velocity, and pressure drop trends to 3 mesh and choose the suitable mesh size for experiment investigation.

The next step is to compare pressure drop in cyclone with a diameter of 0.205 m and input inlet velocity of 16, 20, 22, and 32 m/s using the suitable mesh. This cyclone is based on a Hoekstra model with a diameter of 0.290 m, an inlet velocity of 16 m/s, a mass flow rate, and Reynold stress. Moreover, changing the diameter of 0.205 m to investigate the diameter has and influence on pressure drop.

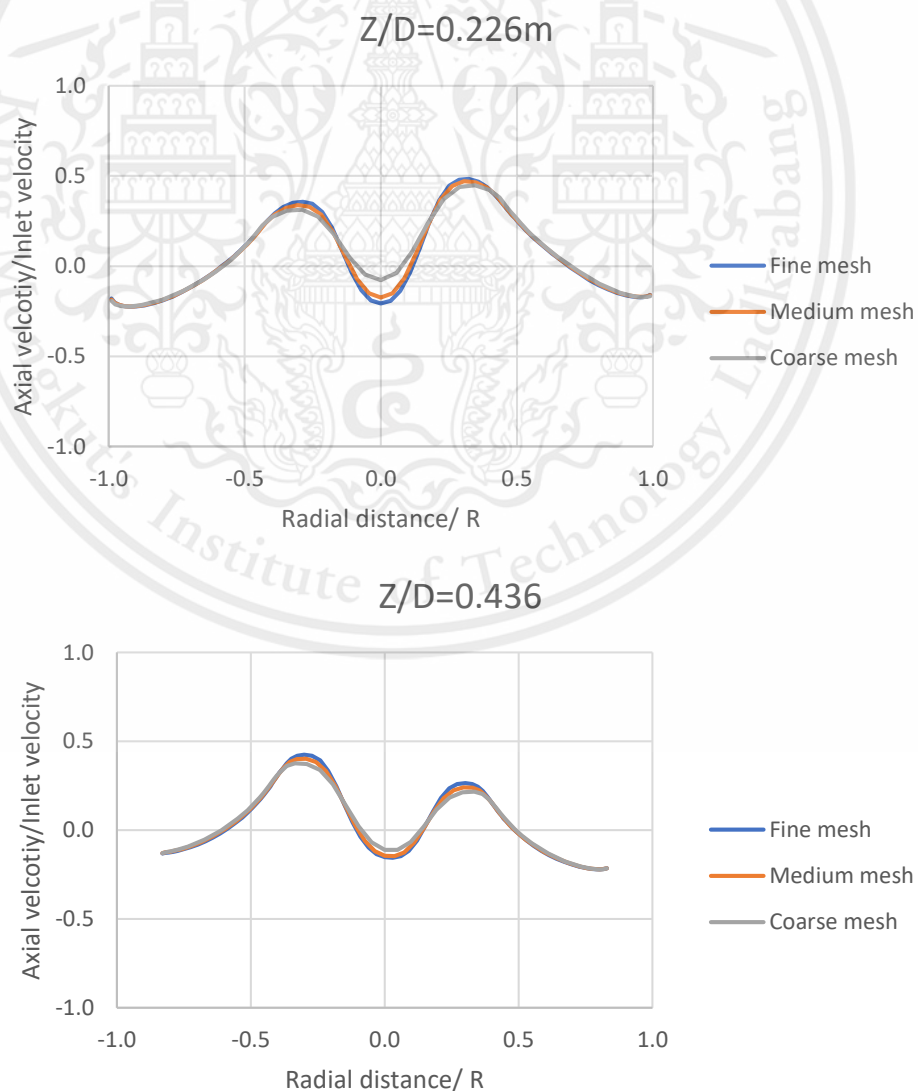
CHAPTER IV

RESULTS AND DISCUSSION

Calculation and analysis result has different structure of generated mesh impact to efficiency of cyclone by using ANSYS Fluent 14.5 for investigating suitable grid generation for computational fluid dynamics simulation in cyclone by Reynold stress model (RSM).

4.1 Grid independent

Comparison grid independent for calculating the prediction pressure drop in cyclone and generate a different mesh with fine mesh, medium mesh, and coarse mesh are 447556, 243780, and 143592 respectively, then plot axial velocity, tangential velocity and pressure drop at $Z/D = 0.226$, 0.436 , 0.506 , and 0.616 show in Figure 4.1 and 4.2.



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

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

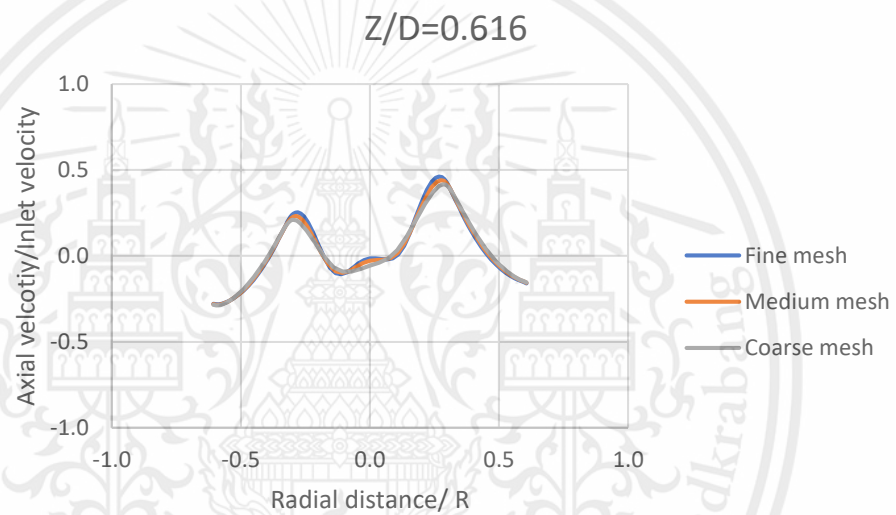
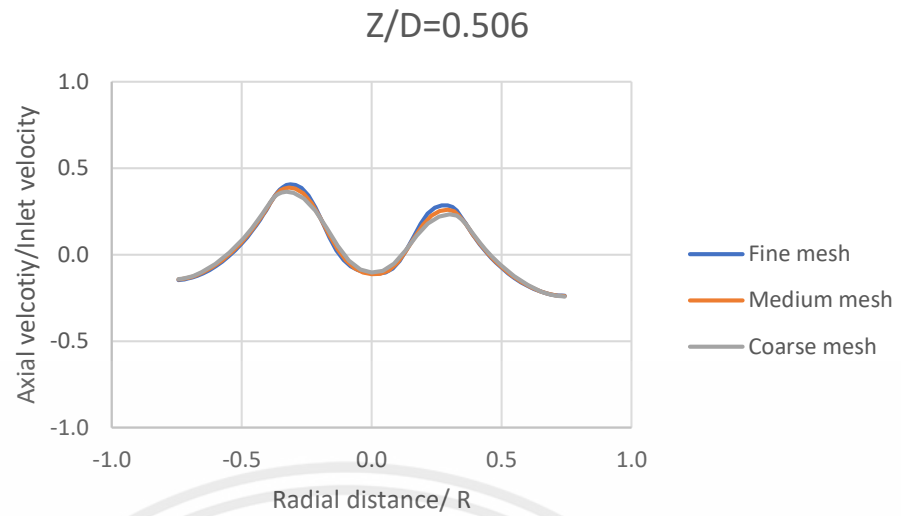
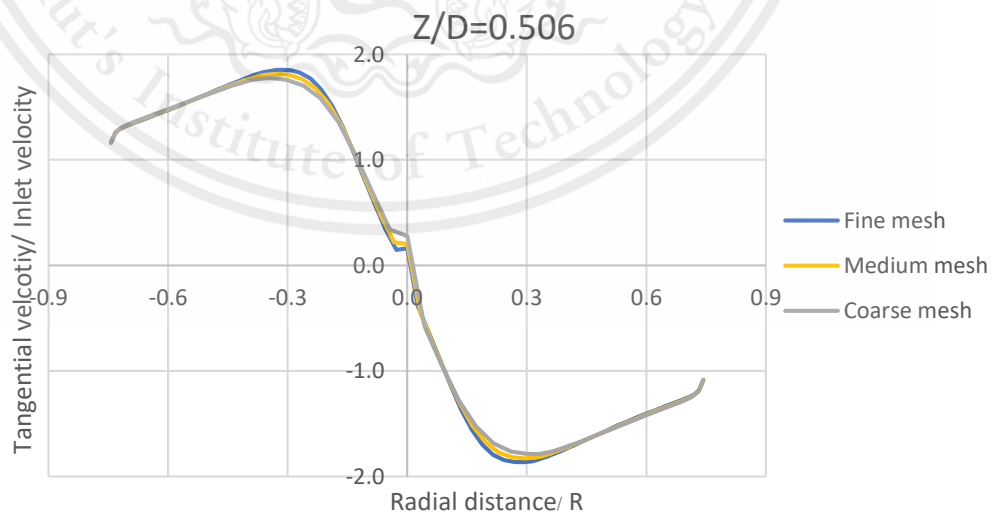
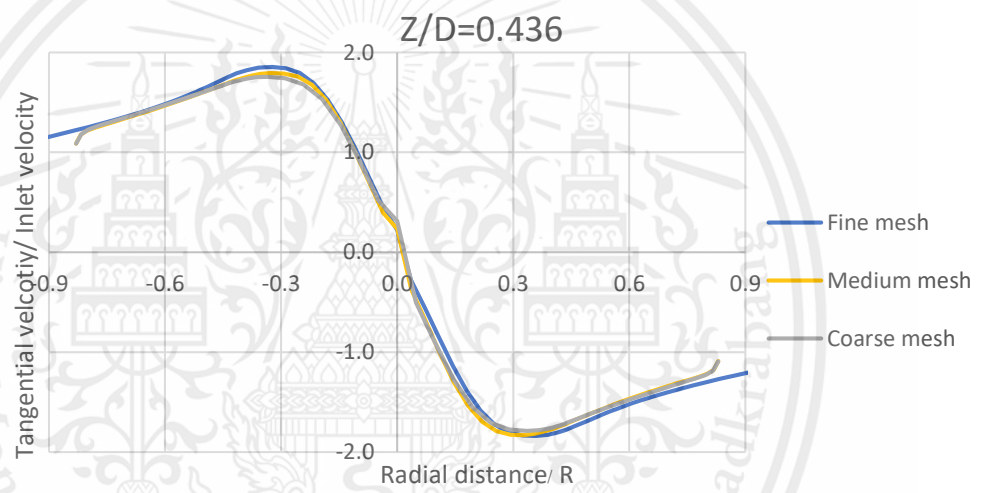
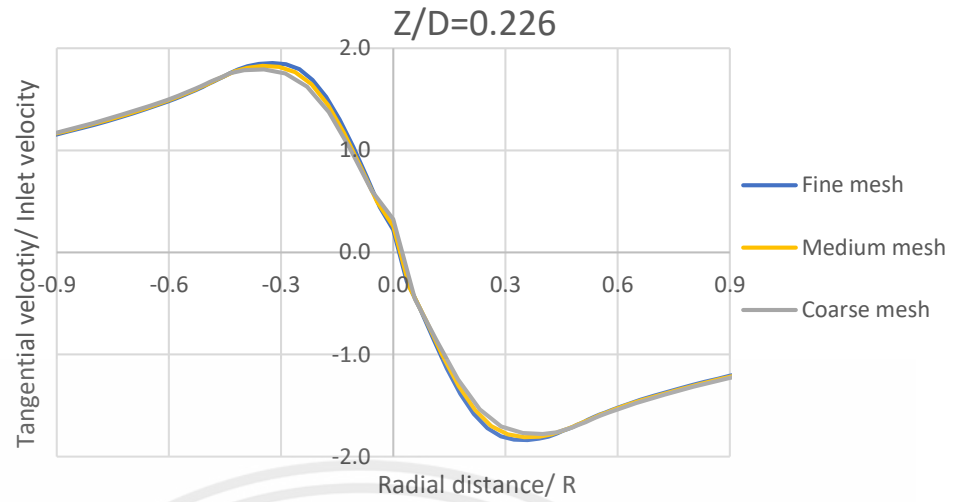


Figure 4.1: Axial velocity of 3 mesh using (fine, medium, and coarse mesh) size at different Z/D (0.226, 0.436, 0.506, and 0.616)



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

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

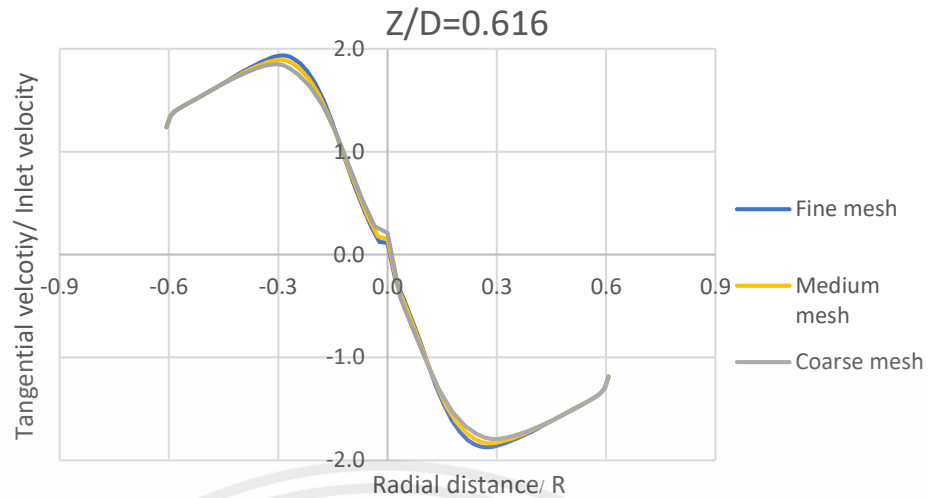


Figure 4.2: Tangential velocity of 3 mesh using (fine, medium, and coarse mesh) size at different Z/D (0.226, 0.436, 0.506, and 0.616)

Table 4.1: Pressure drop of different grid dependent

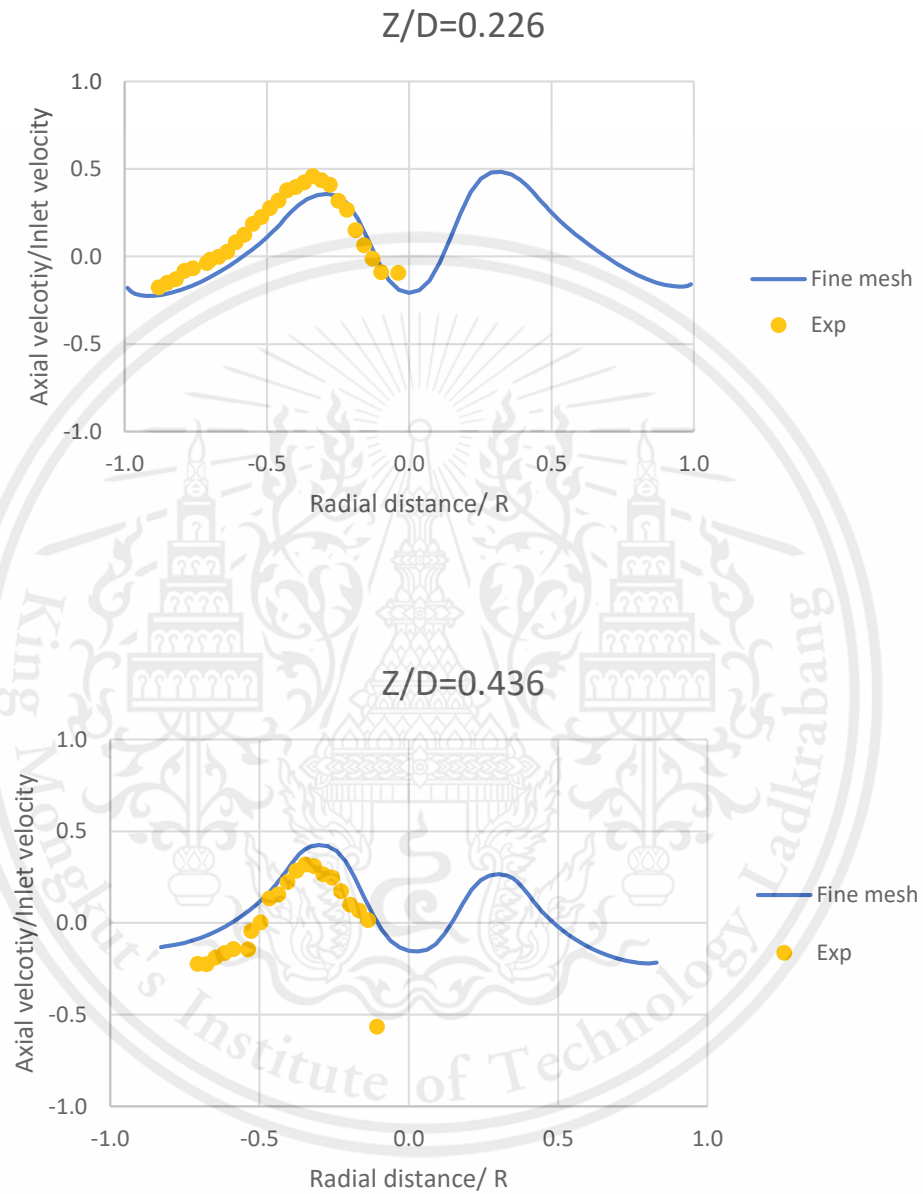
| Velocity = 20 m/s | Diameter = 0.205 m | | |
|--------------------|--------------------|----------|----------|
| | Fine | Medium | Coarse |
| Pressure drop (Pa) | 1084.301 | 1096.268 | 1109.250 |

A comparison of axial velocity of three mesh sizes is shown in Figure 4.1. The fine mesh trend is similar to the medium mesh. At Z/D of 0.226 m the coarse mesh is not similar to fine mesh, and medium mesh. therefore, the coarse mesh is excluded from consideration. As for the tangential velocity graph from Figure 4.2, the result of 3 mesh sizes has similar value but the coarse mesh is lower than the fine mesh and medium mesh. This mesh is excluded for consideration. The fine mesh is the suitable mesh for protecting the divergence from the referent research and using less time to perform simulation.

Finally, Table 4.1 the pressure drop at inlet velocity of 20 m/s is compare with grid independent as fine mesh, medium mesh, and coarse mesh. Which the result show that grid independent has no more impact on pressure drop, so grid independent is not consideration.

4.2 Comparison result with reference research

From comparison result the validate mesh size of axial velocity and tangential velocity with reference research, as show in Figure 4.3 and 4.4.



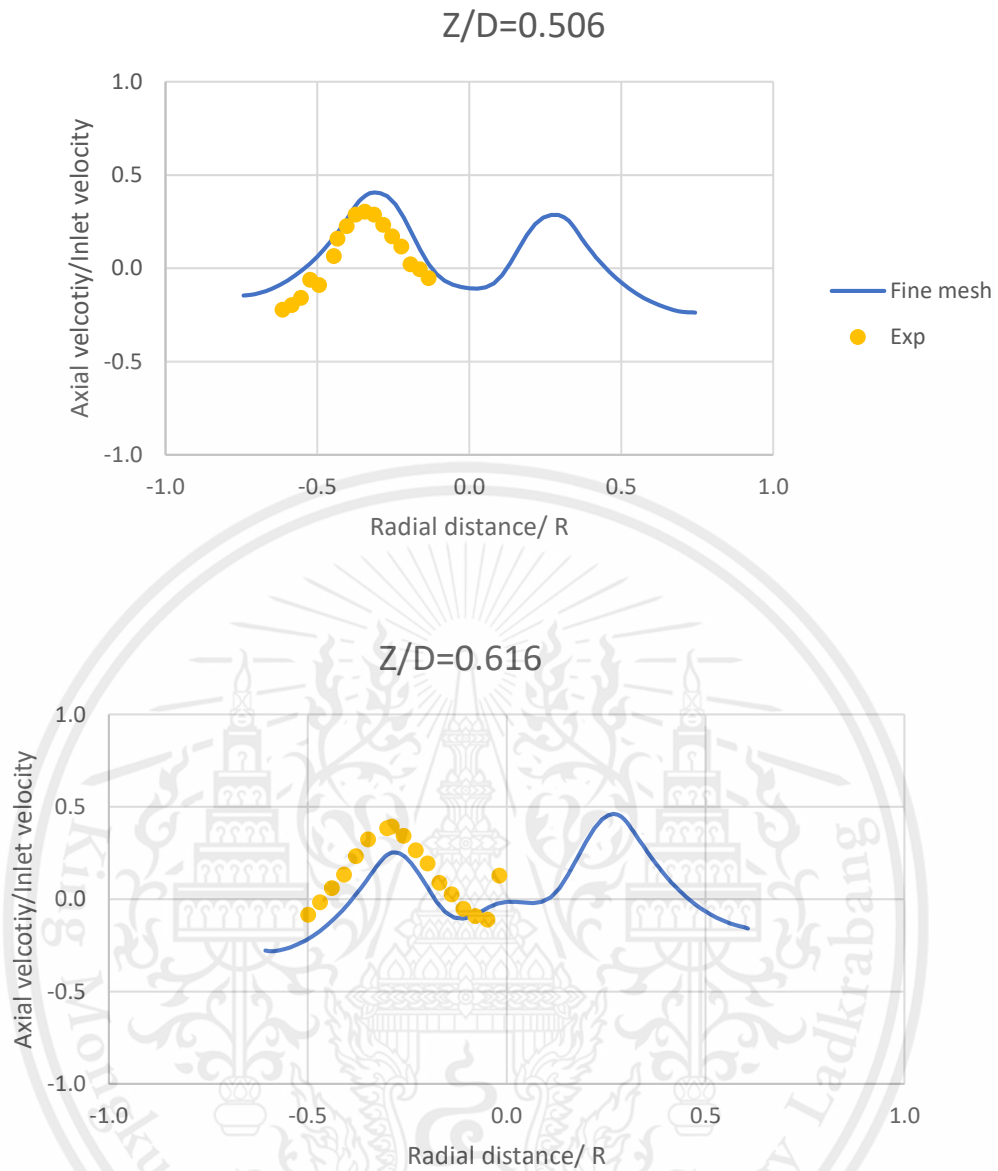
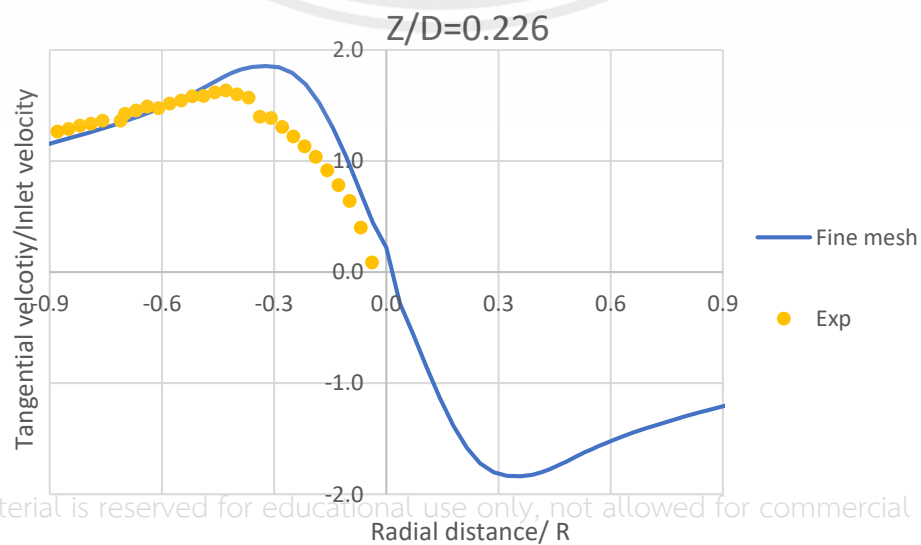
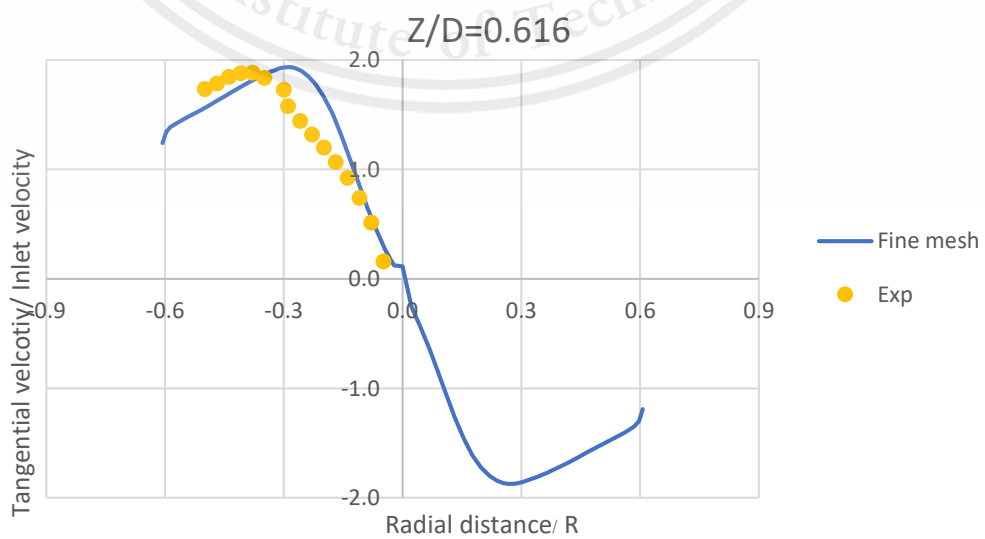
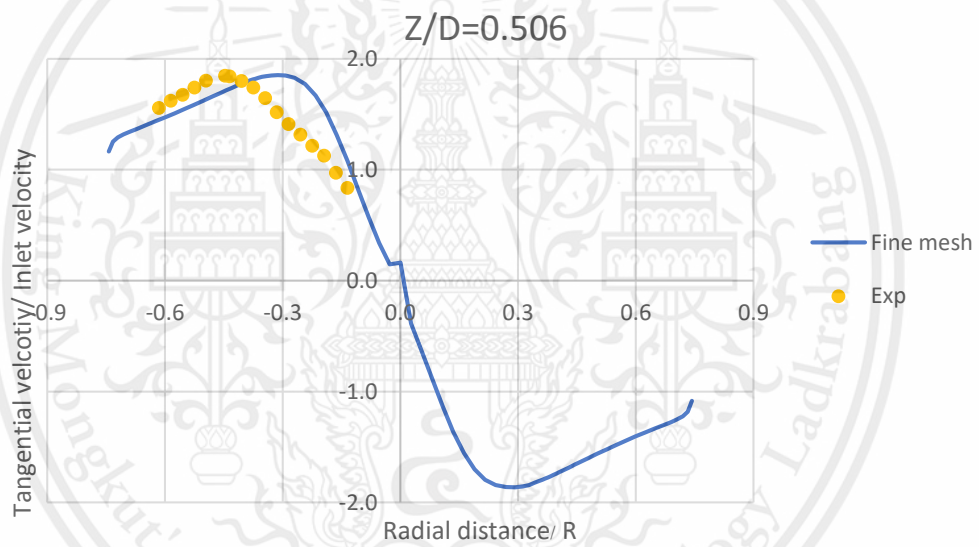
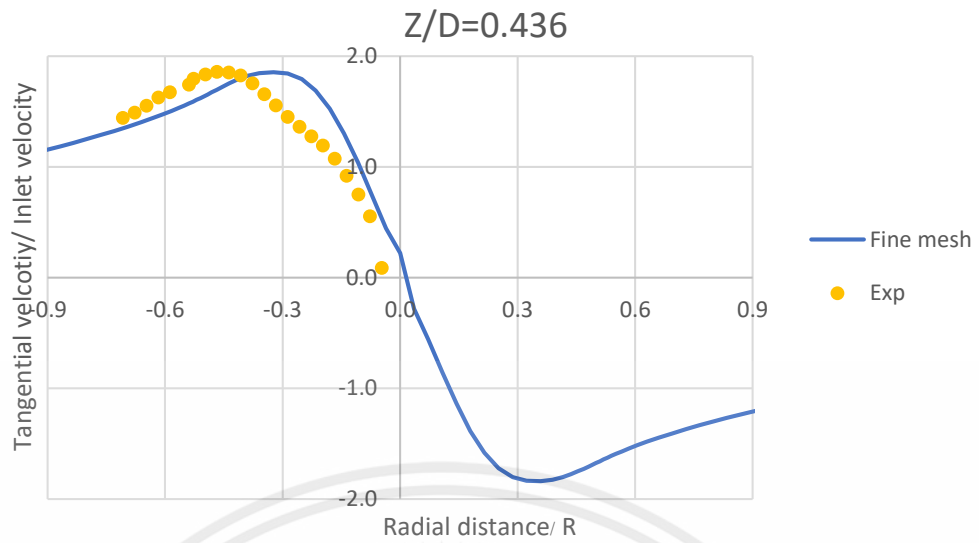


Figure 4.3: Axial velocity of fine mesh and experiment data at different Z/D (0.226, 0.436, 0.506, and 0.616)



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

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



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

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

Figure 4.4: Tangential velocity of fine mesh and experiment data at different Z/D (0.226, 0.436, 0.506, and 0.616)

According to the calculated results of fine mesh with experiment research. Comparison the fine mesh is suitable for use in computational fluid dynamics, to compare the pressure drop with inlet velocity, and to explore the diameter effect on the pressure drop, and the contour is shown in Figure 4.5.

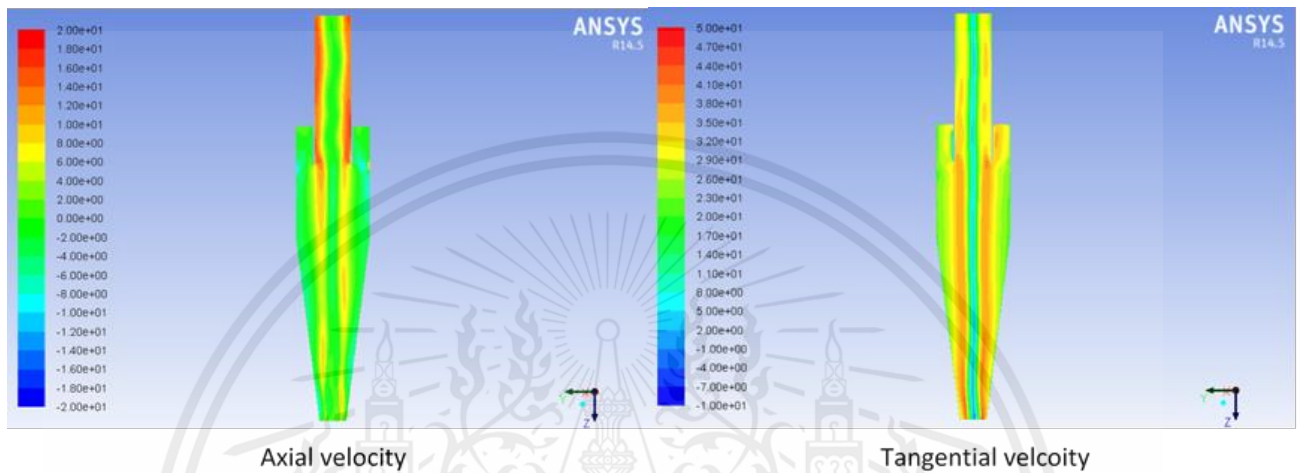


Figure 4.5: Contour of Axial velocity and Tangential velocity of fine mesh

4.3 Calculation of diameter impact on pressure drop

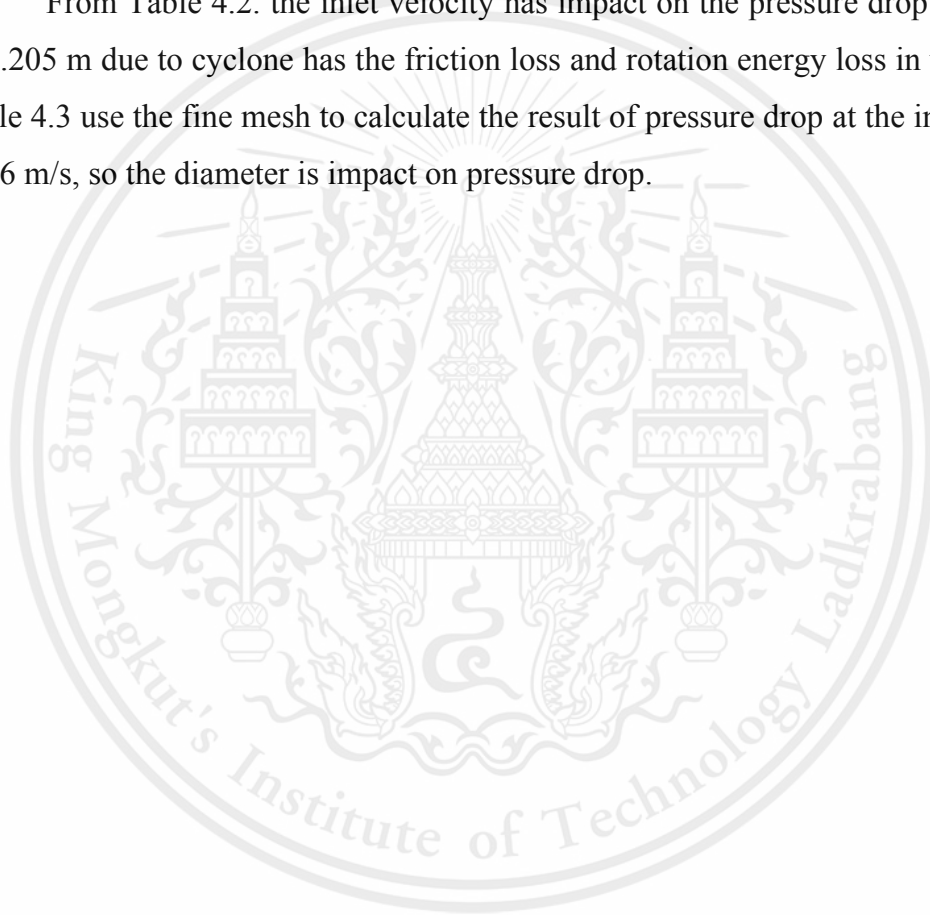
Table 4.2: Pressure drop of different inlet velocity

| | Diameter = 0.205 m | | | |
|----------------------|--------------------|----------|----------|----------|
| Inlet velocity (m/s) | 16 | 20 | 22 | 32 |
| Pressure drop (Pa) | 673.381 | 1084.301 | 1390.696 | 2857.178 |

Table 4.3: Pressure drop of different diameter

| Inlet velocity (16 m/s) | Reference | Fine mesh |
|-------------------------|-----------|-----------|
| Diameter (m) | 0.290 | 0.205 |
| Pressure drop (Pa) | 696.488 | 673.381 |

From Table 4.2. the inlet velocity has impact on the pressure drop at diameter of 0.205 m due to cyclone has the friction loss and rotation energy loss in wall. As for Table 4.3 use the fine mesh to calculate the result of pressure drop at the inlet velocity as 16 m/s, so the diameter is impact on pressure drop.



CHAPTER V

CONCLUSION

5.1 CONCLUSIONS

This project uses computational fluid dynamic (CFD) to simulate fluid flow inside a cyclone in order to study the influence of cyclone diameter on pressure drop. The operation involves using GAMBIT and ANSYS Fluent 14.5 to create the simulation model. Reynold stress model (RSM) is used in the simulation numerical calculation to investigate turbulent flow.

Grid independence is carried out to determine the suitable grid and mesh size. Fine mesh with size of 447556 is selected and used for the later part of this simulation. The resulting values of axial velocity, tangential velocity, and pressure drop are compared with values obtained from references.

The axial and tangential velocities of each point at different heights $Z/D = 0.226$, 0.436 , 0.506 , and 0.616 are close to the reference experiment. Although these results imply the right approach, the pressure drop from computation grid independence is not taken into consideration because fine mesh, medium mesh, and coarse mesh have no effect on pressure drop.

Finally, the fine mesh is used with a 16 m/s input velocity and diameters of 0.290 m and 0.205 m. Variation in diameter value has a pronounced impact on pressure drop.

5.2 SUGGESTION

The resolution and quality of the grid affect the accuracy. So that of the calculation should be increased resolution by add cell numbers and add more quality for accurate and uncomplicated calculation results.

REFERENCES

- [1] L. WANG, "THEORETICAL STUDY OF CYCLONE DESIGN," no. May, pp. 1–137, 2004, [Online]. Available: <http://hdl.handle.net/1969.1/2192>.
- [2] M. D. SLACK, R. O. PRASAD, A. BAKKER, and F. BOYSAN, "Advances in cyclone modelling using unstructured grids," *Chem. Eng. Res. Des.*, vol. 78, no. 8, pp. 1098–1104, 2000, doi: 10.1205/026387600528373.
- [3] "CYCCLONE SEPARATOER FOR ENERGY EDUCATION." https://energyeducation.ca/encyclopedia/Cyclone_separator (accessed Nov. 19, 2020).
- [4] "CLCLONE SEPARATOR WORKING PRINCIPLE (DUST SEPARATOR)." <https://savree.com/en/encyclopedia/cyclone-separator-working-principle-dust-separator> (accessed Nov. 21, 2020).
- [5] T. R. Taha, *AN INTRODUCTION TO PARALLEL COMPUTATIONAL FLUID DYNAMICS*, vol. 6, no. 4. 2005.
- [6] S. K. Shukla, P. Shukla, and P. Ghosh, "EVALUATION OF NUMERICAL SCHEMES USING DIFFERENT SIMULATION METHODS FOR THE CONTINUOUS PHASE MODELING OF CYCLONE SEPARATORS," *Adv. Powder Technol.*, vol. 22, no. 2, pp. 209–219, 2011, doi: 10.1016/j.appt.2010.11.009.
- [7] C. J. Geankoplis, *Transport Processes and Separation Process Principles*, 4th ed. Prentice Hall Professional Technical Reference, 2003.
- [8] N. Jaiklom, D. Onprom, and T. Seethao, "Nunnumerical investigation of performance characteristics of a cyclone prolonged with a diverfent dipleg," 2011.