CHAPTER 1

INTRODUCTION

1.1 Project Background

Computational Fluid Dynamics (CFD) is a systematic application of computing systems and computational solution techniques to mathematical models formulated to describe and simulate fluid dynamic phenomena. CFD also is an engineering method for simulating the behavior of systems, processes and equipment involving flow of gases and liquids, heat and mass transfer, chemical reactions and related physical phenomena [1]. In the early days, fluid dynamics analysis was conducted manually and it is limited to simple geometry with tedious calculation. As the computer application start to grow, people started to use computer as the tool to carry out the calculation of the CFD analysis. The process is getting faster and more accurate as computer with higher clock speed being used to compute and solve CFD problems. By using the numerical approach the fluids flow can be analyzed without conducting the actual experiment, hence it can reduce the time and cost for experimental based research. CFD also permits one to incorporate performance enhancement into a design without the need for extensive testing in physical models.

The separator has a wide range of applications in the oil and gas industry. In gas industry production facilities objective are to separate the production fluids into its own component such as natural gas, water, impurities and gas condensate. Gravity-based vessels (two- or three-phase, horizontal or vertical) perform most of the separation duty above the water. Demands of deep water production require subsea water dehydrating facilities in order to separate liquid (mainly water) and gas mixture before it is transmitted from subsea through pipeline to the processing facilities [2]. Mixture of fluid and gas has a big impact to the pipeline and the equipment, as the mixture might cause liquid hammering in piping, pipeline corrosion, and damage the equipment especially the rotating equipment such as pump and motor.

The supersonic swirling separator has been introduced to treat natural gas for dehydration and solid particle removal. It is a compact tubular separator device with no moving parts, enabling high reliability and availability. It is smaller, lighter, cheaper and with fewer emissions than conventional dehydration plant [3-5]. It is suited for platforms, due to its lightweight and the viability of unmanned operation. Significant potential has been identified for future application of this technology on various other gas processing separation applications including deep LPG extraction, bulk removal of CO_2 and H_2S, and subsea gas processing [6].

‘Swirler’ is a new type of compact wet gas separator which is developed by PETRONAS Gas Separation Research Centre. Swirler which utilizes supersonic separation method is designed to remove free water at subsea gas transmission pipeline. Swirler consists of static mechanical parts and purely operated by the well pressure. Swirling flow is generated during fluid entrance by tangential inlet before it flows across the supersonic nozzle. At the supersonic nozzle the natural gas will be throttle to supersonic velocity where the pressure and temperature drop will cause the water vapor to condense. Centrifugal force from the swirl flow will pushed the water condensate and impurities outwards to the wall where water collector is placed. Formation of supersonic flow at the nozzle part is crucial in order to provide pressure and temperature drop which allows water vapor to condense. Centrifugal force generated by the swirl is important to further separate the liquid and solid from the natural gas. Combination of these effects of swirling and throttling formed a great separation capability of compact gas separator.

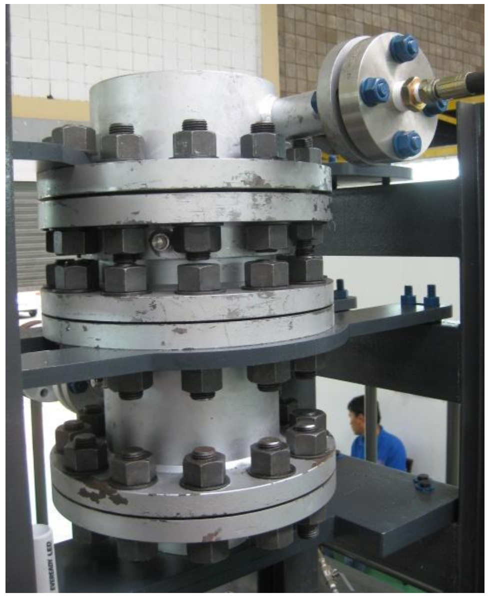


Figure 1.1: The Swirler prototype

Currently, Swirler is in the experimental stage where the development is for platform application. In order to study the flow behavior inside this separator, a series of CFD modeling and simulation need to be done with reference to the prototype. The formation of swirl inside the separator is crucial as it is the one that initiates the centrifugal force which drive the separation process. The flow across the supersonic nozzle is the most critical as it determine the dehydration performance. This separator is design to handle supersonic flow regime, hence, the flow inside the separator need to be simulate in order to determine the type of the flow across the supersonic nozzle whether it is subsonic or supersonic. The results of the CFD analysis later to be used to optimize the separator design.

1.2 Problem statement

Swirler is a new type of supersonic separator with tangential inlet flow. Data regarding the flow behavior across the separator is insufficient in order to improve the current design. Dehydration performance of the separator mainly determined by the velocity of fluid flows across the nozzle which is expected to be a supersonic flow. Series of CFD analysis was conducted in order to predict the fluid flow behavior especially the fluid velocity across the nozzle. Data from the CFD analysis later were used for current design improvement.

1.3 Objectives

1. To model and simulate the supersonic wet gas separator for CFD analysis.
2. To predict the velocity of the fluid flow across the separator nozzle for the flow regime evaluation.
3. To propose suitable recommendation for future design improvement.

1.4 Scope of study

CFD modeling and simulation was conducted to predict and analyzed fluid behavior across a supersonic separator. Although the analyses are only interested at the nozzle section, however the whole geometry of the prototype was modeled to allow the effects of the upstream part of the nozzle to be taken into account during the analysis. The whole geometry simulation provides better realization of the flow parameters. The propose recommendation only involve minor parts modification.

CHAPTER 2

LITERATURE REVIEW

2.1 CFD software

Various CFD program has different features for CFD analysis depends on the problem to be solved. The FLUENT solver is based on the centre node Finite Volume Method, (FVM) discretization technique and offers both segregated and coupled solution methods. Three Euler-Euler multiphase models are available; the Eulerian model, the mixture model and the VOF model. In addition, one particle tracking model is available [7].

In the case of supersonic separator CFD, Chuang Wen [4], had used FLUENT to carry out the simulation. FLUENT is the CFD solver of choice for complex flows ranging from incompressible (low subsonic) to mildly compressible (transonic) and to highly compressible (supersonic and hypersonic) flows [4]. The flows, in a supersonic separator, are very complicated, including swirling flow, supersonic velocity and shockwaves. Ferhat [8] performed the numerical study of supersonic separator using CFX instead of FLUENT. Kefalas [6] reported that the geometry of the separator was built with GAMBIT and imported to FLUENT for post processing.

2.2 Turbulence model

A turbulence model is a computational procedure to close the system of the mean flow equation, so that variety of flow problems can be calculated. The turbulence model applies certain assumption on the flow; hence the solution could be simplified. Table 2.1 shows the literature summary of turbulence model which has been used to solve supersonic flow.

Table 2.1: Turbulence model literature summary

Ref No	Author	Year	Assessment	Method	Results
7	Chuang Wen et al	2012	Analyze the gas dynamics parameters of natural gas flows in the supersonic separators	Standard K-ε flow model. Inlet pressures on natural gas characteristics were computational simulated with ideal gas and Redlich-Kwong equation of states	Ideal gas and Peng-Robinson models predict the same gas parameters in the subsonic zone, while the ideal gas result diverges from the Redlich-Kwong case in supersonic zones.
8	Ferhat et al	1997	CFD analysis on Gas Liquid Cylindrical Cyclone, GLCC different configuration	Euler – Euler model for two phase turbulence flow	CFD predicted velocity agree with experimental data.
9	Chuang Wen et al	2011	The effects of swirls on natural gas flow in Supersonic Separators	Natural gas flow were computational simulated with the Reynolds Stress model (RSM)	Numerical result predicts the same fluid behavior with the experimental results.

6	P.I Kefalas & D.P Margaris	2008	CFD analysis of the liquid-gas flow, in various inlet flow patterns	RNG K-ε turbulence model to simulate multiphase flow	CFD results predict correctly that the separator efficiency will increase with the flow rate as well as the pressure drop
10	A.B Majid et al	2010	Predict the behavior of high pressure natural gas flowing through supersonic nozzles	Standard K-ε turbulence model	CFD results agree with experiment result. Pressure variation well predicted.
11	M. Haghighi et al	2013	Predict the flow behavior of wet gaseous mixture in supersonic separator	Navier – Stokes equations combined with the realizable K-ε turbulence model for conservation of momentum	Compared with experimental data, CFD analysis predicts the pressure profile well.
12	Chuang Wen et al	2010	The effects of the shock waves on the natural gas flow fields were analyzed in the supersonic separator	Flow investigated using RNG K-ε turbulence model	CFD analysis successfully predicts fluids behavior shock wave location.

2.3 Governing equation

2.3.1 Conservation equation

From the physical viewpoint, the equations describing fluid flows, heat, and mass transfer are simply versions of the conservation laws of classical physics namely:

- Conservation of chemical species (law of conservation of mass)
- Conservation of momentum (Newton's second law of motion)
- Conservation of energy (first law of thermodynamics)

This conservation laws leads to governing equation describing fluid flows. The mass equation, momentum equation, and energy equation to be calculated are described as equations 1~3.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j + p\delta_{ij} - \tau_{ji}) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j}(\rho u_j E + u_j p + q_j - u_i \tau_{ij}) = 0 \tag{3}$$

Where ρ, u, p are the gas density, velocity, and pressure, respectively. τ_{ij} is the viscous stress; δ_{ij} is the Kronecker delta; E is the total energy; q_j is the heat flux; t is the time [2].

2.3.2 Equation of state

An equation of state is developed in order to calculate the physical property of fluids flows. In the flow of compressible fluid the equation of states provides the linkage between energy equation, conservation of mass equation, and momentum equation. The ideal gas law and Redlich-Kwong real gas equation of state model can be employed to predict gas dynamics parameters. The classical ideal gas law may be written:

$$pV = nRT \tag{4}$$

Where *p, V, T* are the gas absolute pressure, volume, absolute temperature, respectively while *n* is the number of moles of a substance. *R* is gas constant [4]. The Redlich–Kwong equation of state is an equation that is derived from the van der Waals equation [13]. It is generally more accurate than the van der Waals equation and the ideal gas equation. Redlich - Kwong equation of state can be described as:

$$p = \frac{RT}{V_m - b} - \frac{a}{\sqrt{T}V_m(V_m + b)} \tag{5}$$

The constants *a* and *b* are different depending on which gas is being analyzed, which can be calculated from the critical point data of the gas:

$$a = \frac{0.4275R^2T_c^{2.5}}{p_c} \tag{6}$$

$$b = \frac{0.08664RT_c}{p_c} \tag{7}$$

T_c and p_c are the temperature and pressure at the critical point, respectively.

2.4 Computational grid

Computational grid is required in order to discretize the geometry into finite volume cells where the flow variables are solved at the nodes of the grid. Having a well-constructed grid is necessary and significant step toward an accurate, efficient, and robust numerical solution. The quality of the mesh plays a significant role in the accuracy and stability of the numerical calculation. According to Chuang Wen [4], the structured mesh was applied to the supersonic separator. The supersonic separator is meshed by the coarse, moderate and fine grids, respectively.

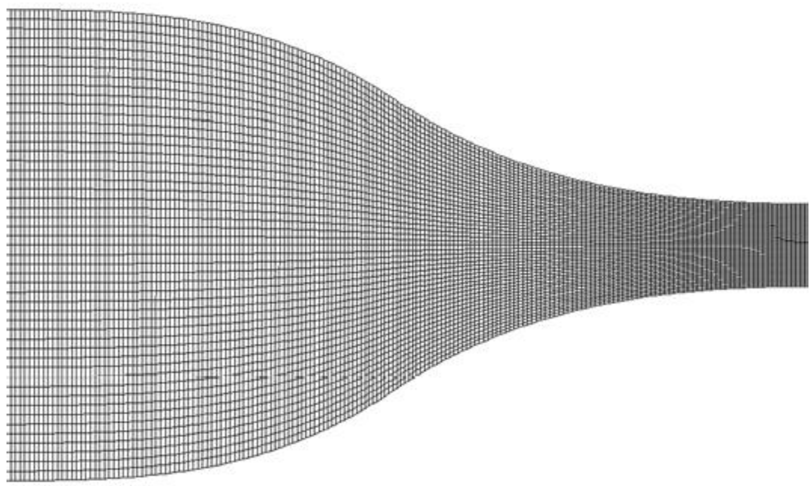


Figure 2.1: Moderate mesh at the nozzle section [4]

The polyhedral meshes allow the flexibility of an unstructured mesh to be applied to a complex geometry. Chuang Wen [12], reported that the swirling vanes and the cyclone separation section was complicated. Therefore, the supersonic separator is meshed by the hexahedral and tetrahedral grids, and divided by the coarse, moderate and fine grids, respectively.

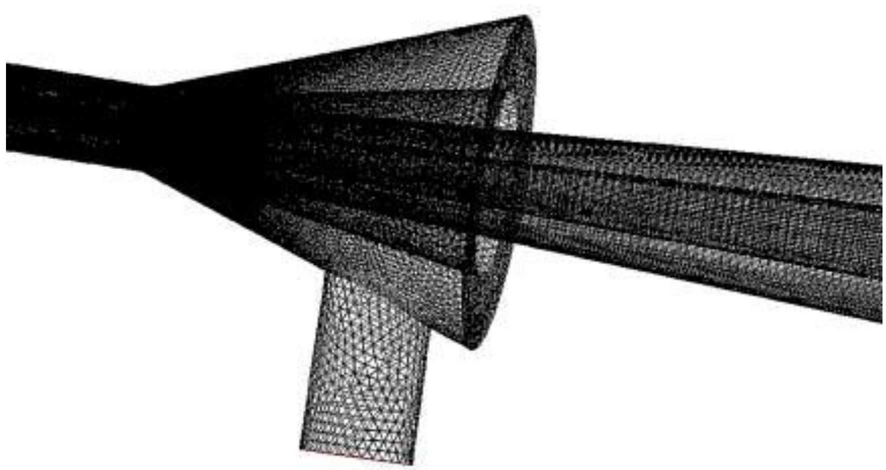


Figure 2.2: Fine grid on the supersonic section [12]

In the case of simple geometry which is mainly consist of primitive geometry; structured mesh is selected based on Kefalas [6] work on compact phase separator. The meshes are mainly consisting of hexahedral cells. Excess time and effort were needed compared to the unstructured mesh approach but the results are considered as more accurate and reliable [10]. The geometry was split in domains. Most of them were similar in geometry.

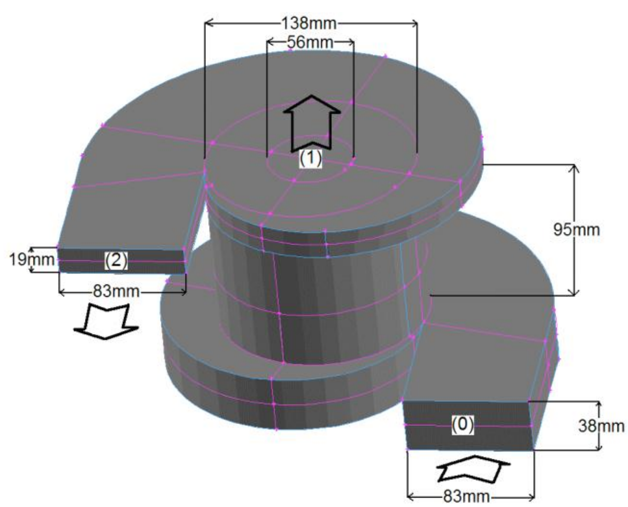


Figure 2.3: Kefalas compact phase separator [10]

Accuracy increases with larger grids. However, grid independency studies have shown that larger grids do not necessarily influence the accuracy and the iteration steps for convergence of the solution in the case of the supersonic nozzle flow; however, large number of grid will require more computational time [14].

2.5 Boundary and Initial Condition

CFD problems are defined in terms of initial and boundary conditions where some of the values are supplied by the user. Although there are few choices of turbulent model such as RSM, standard K-ε, and RNG K-ε, previous work by Chuang Wen [4],[6],[12] shows that the same boundary condition are specified throughout different flow model.

According to the flow characteristics of the supersonic compressible swirling flow, boundary conditions are imposed as follows: pressure boundary conditions for inlet and outlet of the supersonic separator, respectively. No-slip and adiabatic boundary conditions are specified for the walls [4],[6],[12].

CHAPTER 3

METHODOLOGY

This project is based on numerical analysis where computational approach is carried out instead of actual experiment on the prototype. For this case, standard modeling and simulation procedure were adapted as the methodology of this project. The CFD method was applied as a tool to predict the flow behavior across the supersonic separator.

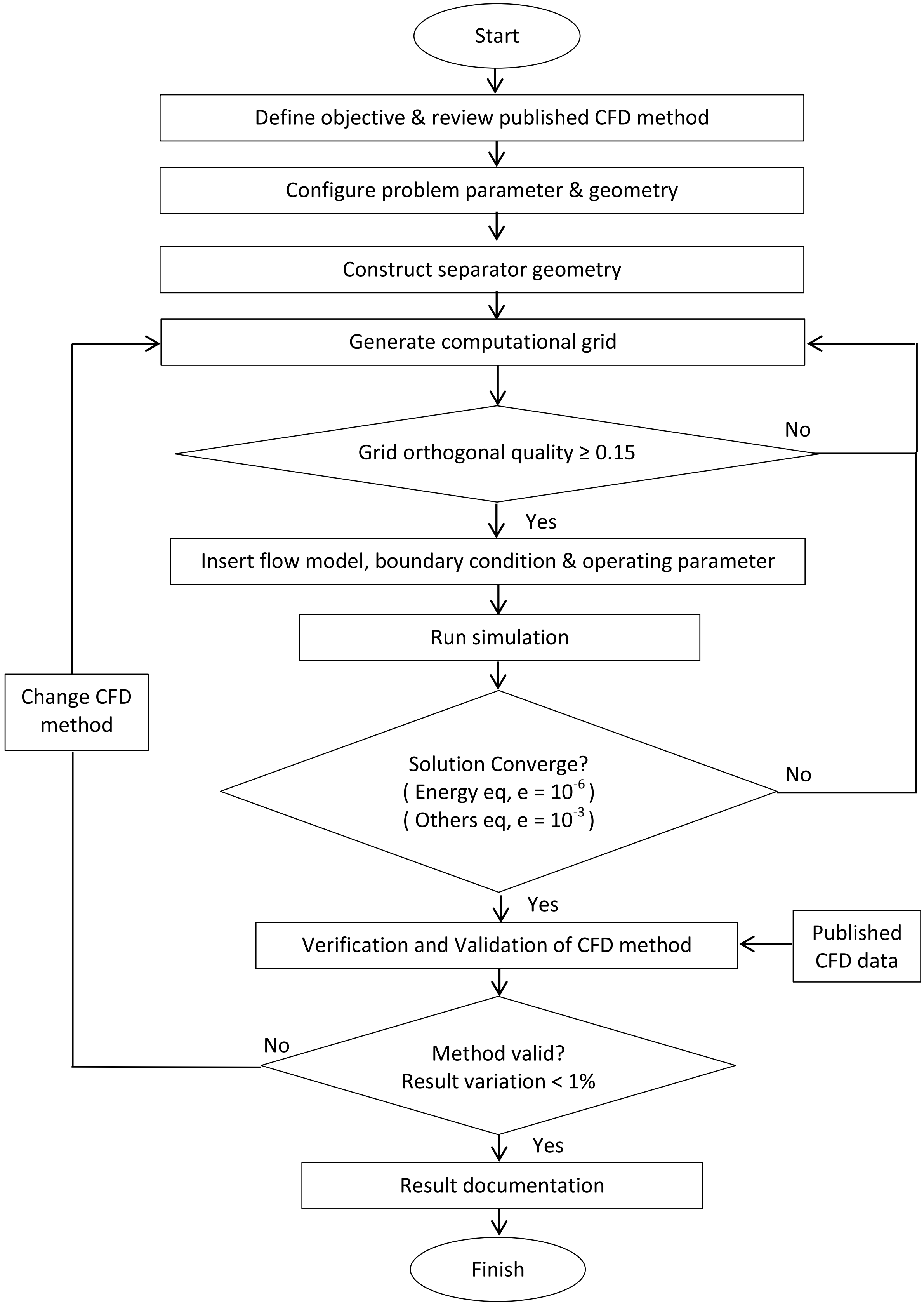
3.1 Software Selection

ANSYS FLUENT commercial CFD software is selected as the numerical solver of this simulation. FLUENT provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems [15]. In addition, FLUENT is utilized in this project because it is the only CFD software currently available at the simulation lab.

3.2 General procedure

Before the simulation process started, several information regarding the wet gas separator need to be determined. Among the information are; prototype parameters and geometry, type of process fluid, operating condition, types of flow pattern and governing equation to be used to solve the numerical problems. Later, the model is constructed in simulation program, or imported from existing CAD files. The type of flow model and the approaches must be selected depends on the problem to be solved.

3.3 Project Flow Chart



3.4 Project Schedule

Table 3.1: Project schedule

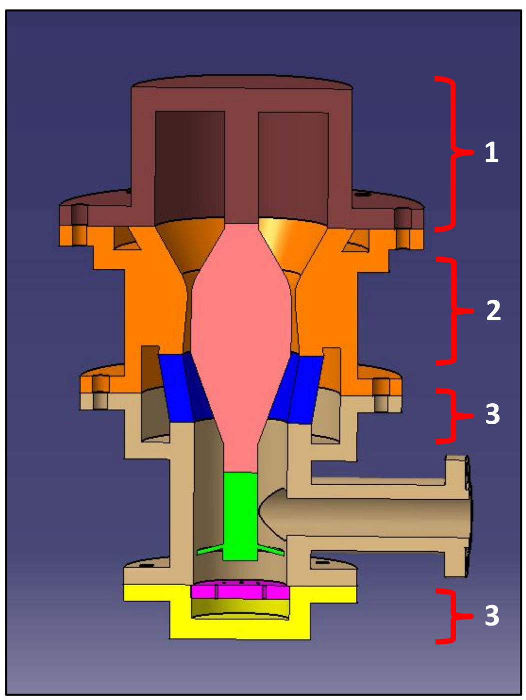
No	Activities	No. of weeks																											
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28
1	Define objective & review published CFD method,																												
2	Configure separator geometry and operating parameter																												
3	Geometry modelling & computational grid generation																												
4	Flow model selection & configure governing equation																												
5	Define initial & boundary condition, insert operating parameter																												
6	Start simulation																												
7	Result verification & validation																												
8	Result documentation & final report presentation																												

Project milestone

3.5 Geometry Modeling

During the modeling stage, the geometry model was simplified from the actual geometry of the separator. In this case, the CFD analysis only interested at the flow path, therefore, the wall thickness and external geometry were neglected. Hence, only the critical profile of the separator was modeled instead of modeling the whole geometry of the separator.

The fluid path started at the inlet, then to the cyclone where swirling flow is generated. After that, the fluid will be accelerated through the convergence divergence nozzle where at the divergence section the fluid is expected to be accelerated to supersonic velocity. Then, the fluid will exit at the outlet water collector geometry does not taken into account because only velocity of the natural gas was interested to be determine. Figure 3.1 shows the internal profile of the Swirler which is obtained by sectioning the Swirler CAD model.



1. Cyclone generator
2. Convergence divergence nozzle
3. Water collector

Figure 3.1: Internal profile of the Swirler

The main body of the separator was modeled by revolving the profile of the horizontal cross section on the X-axis. The profiles shown in Figure 3.2 only consider the flow path and the profile are closed to allow it to be revolved. Note that, the profile of the inlet and outlet was not included, because these profiles will be constructed after the main body was build.

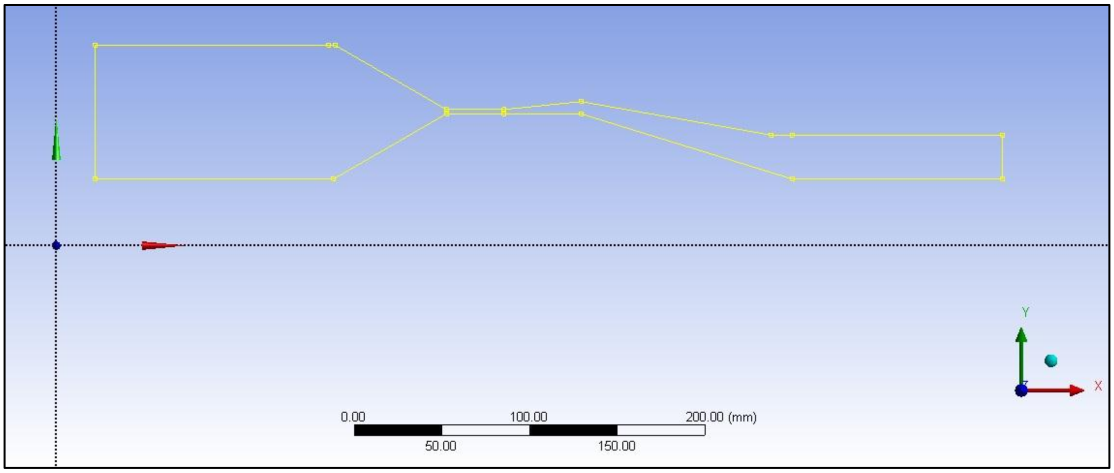


Figure 3.2: Horizontal cross section profile of separator

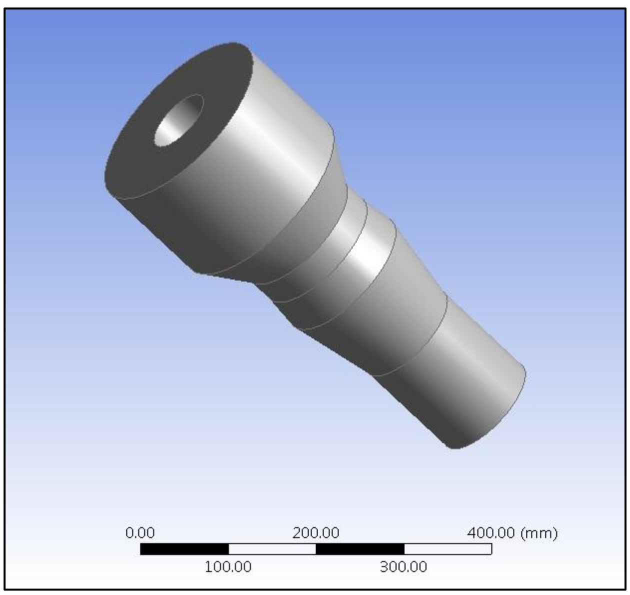


Figure 3.3: Main body of separator

After the main body was constructed, the inlet and outlet geometry were then constructed from the main body. Figure 3.4 shows the complete constructed model. Note that, the inlet and outlet was constructed as a solid instead of hollow pipe geometry because they were defined as inlet and outlet boundary condition; which then recognize by FLUENT as a hollow geometry.

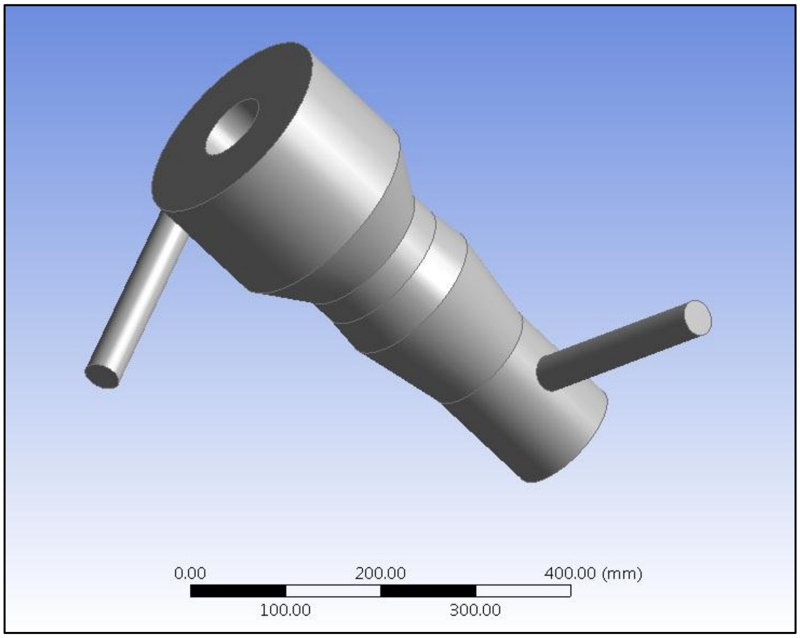


Figure 3.4: Constructed geometry

3.6 Geometry Discretization

The model is mesh in order to discretize the geometry into finite volume element. The quality of the mesh plays a significant role in the accuracy and stability of the numerical calculation. The geometry was meshed using proximity and curvature based meshed with program controlled. Proximity and curvature based mesh allows flexibility of unstructured mesh to be applied to the geometry.

The quality of the meshed is measured from the orthogonal quality and skewness of the cells. Generally it is advised by FLUENT developer to keep minimum orthogonal quality greater than 0.1 and maximum skewness less than 0.95 [13]. Figure 3.5 and Figure 3.6 show the orthogonal quality and skewness metrics spectrum respectively.

Inacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Figure 3.5: Orthogonal quality metrics spectrum [13]

Excellent	Very good	Good	Acceptable	Bad	Inacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Figure 3.6: Skewness metrics spectrum [13]

From the metrics spectrum in Figure 3.5 and Figure 3.6, it can be summarized that the acceptable orthogonal quality should be greater than 0.14, and the acceptable skewness should be less than 0.95.

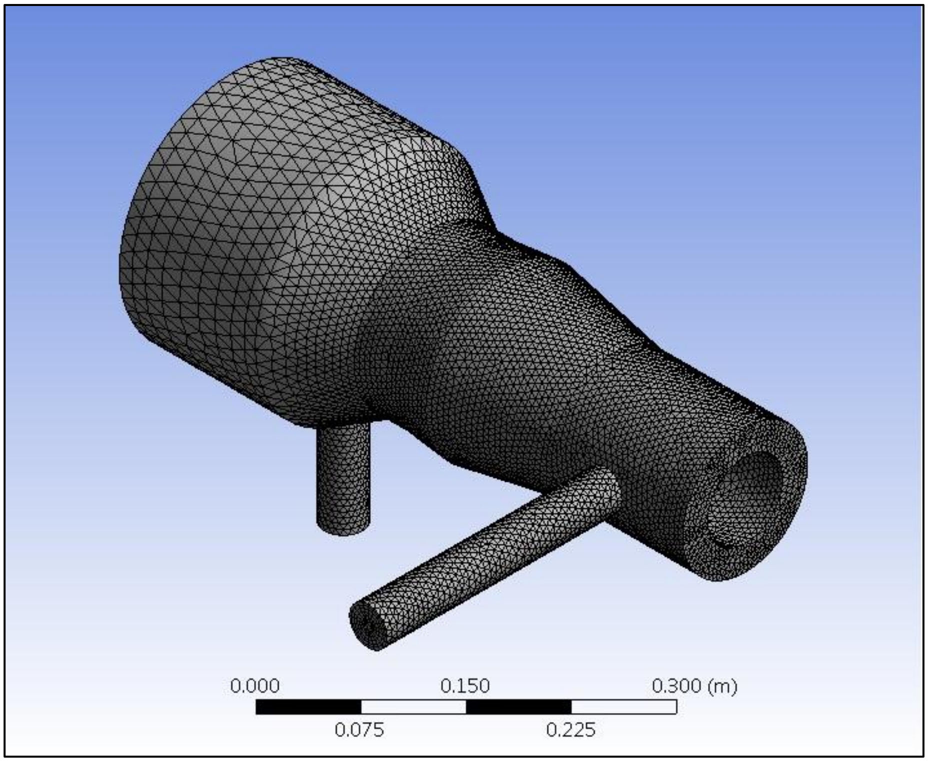


Figure 3.7: Meshed model

Structured mesh was favorable because the grid can be varies in term of size, type and location on the geometry; however, this method are time consuming and required high experience skills. The structured mesh construction was not feasible with this project timeline due to short period of study. To overcome this problem, the unstructured mesh method was applied to the geometry. Unstructured mesh allow mesh flexibility and suitable for complex geometry [9]. The cells were generated finer at the nozzle part because the nozzle geometry and the gap between the walls are small. Finer mesh cells also were required at the nozzle section because the flow is critical at this part. The statistics show that 132982 elements were generated with orthogonal quality of 0.87 which is very good quality and the geometry are well captured by the mesh. The skewness of the mesh also is very low with an average value of 0.25. Inflation layers were created from wall internal surface in order to capture fluid properties at near – wall location.

3.7 Equation of state

An equation of state was applied in order to relate the governing equations so that property of the fluid in supersonic flow can be predicted. The Redlich-Kwong real gas model was used to analyze the gas dynamics parameters of natural gas flows in the supersonic separators. It is generally more accurate than the van der Waals equation and the ideal gas equation [13].

3.8 Turbulence Model

Based on the literature review of the turbulence model, it shows that the k-ε turbulence model is the most selected model for supersonic problem. Therefore, the k-ε turbulence model was selected because it has performed particularly well in confined flows where the Reynolds shear stresses are most important [16]. This model also is the simplest turbulence model for which only initial or boundary conditions need to be supplied [16]. It performance is excellent in industrial application and it is the most validated turbulence model [16].

3.9 Solution Algorithm and Convergence Criteria

In the numerical calculation, the finite volume method and the second order upwind scheme were used. The wall function was introduced to model the flow near the wall, while the Semi-Implicit Method for Pressure Linked Equations algorithm, SIMPLE was applied to couple the velocity field and pressure. SIMPLE is an approximation of the velocity field which obtained by solving the momentum equation. The pressure gradient term is calculated using the pressure distribution from the previous iteration or from initial guess [16]. SIMPLE algorithm is the most popular algorithm for pressure and velocity calculations with finite volume method [16].The convergence criterion were set at 10^{-6} for the energy equation and 10^{-3} for all other equations while the number of iteration was set at 3000.

3.10 Boundary Condition

According to the flow characteristics of the supersonic compressible swirling flow [17], boundary conditions are imposed as follows:

1. Pressure boundary conditions for inlet and outlet of the supersonic separator respectively.

2. No-slip and adiabatic boundary conditions were specified for the walls.

Inlet stagnation pressure and temperature were set at 4 Mpa and 300 K respectively. The dynamics pressure was assumed to be zero, thus, the static pressure will equal to the total pressure. The hydraulic diameter was set at 0.038 mm both at inlet and outlet boundary condition. The process fluid is specified as methane gas because methane is the main component of natural gas.

3.11 Simulation Parameterization

Simulation parameter was set up in order to analyze different model parameters and identify how certain parameters affect the fluid behavior. In this simulation the model was run multiple times with pressure parameters being varied. Normal operating pressure for high pressure gas separator recorded range between 4 – 8 Mpa with 30 percent allowable pressure drop [18]. Inlet and outlet pressure parameter were set up in order to vary the pressure drop ranging from 10 – 30 percent.

Table 3.2: Simulation input parameter

$P_{stagnation}$ (Mpa)	P_{outlet} (Mpa)	Pressure drop (%)
4.0	3.60	10
5.0	4.25	15
6.0	4.80	20
7.0	5.25	25
8.0	5.60	30

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Validation of Computational Model

In this section numerical model technique which is used in this project are validated with others published numerical solutions. The selected numerical technique is applied to validated problem and the results is compared in such a way that the physics of the problem behave accordingly with the published data. In this case, numerical data from Arina's [18] work are used to validate the propose technique. The geometry used in Arina's investigation is shown in Figure 4.1.

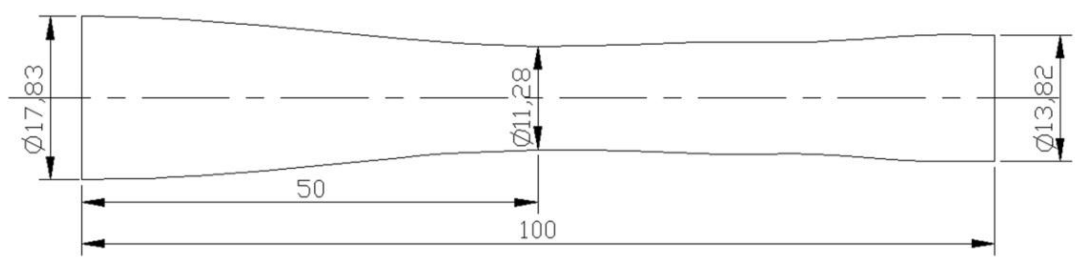


Figure 4.1: Arina's nozzle geometry [18]

The working fluid was air. The inlet total temperature and pressure were 288 K and 1bar, respectively. The exit gage pressure is assigned at 83049 Pa. The Redlich-Kwong equation of state is employed to predict the dynamics parameters of gas fluids in this study. The same condition and setup proposed previously were used in this problem. The result was compared with fluid dynamics characteristics of Arina's work. Numerical result shows that same fluid behavior was predicted accordingly as recorded by Arina's method. This validation support that the method used was applicable for supersonic nozzle problem.

4.2 Results and Discussion

Supersonic separator dynamic parameter was successfully simulated with standard *k*- ε model combined with Redlich-Kwong equation of state. Based on the parametric study it shows that maximum achievable velocity across the nozzle is 0.42 Mach which is at 8Mpa inlet pressure. The highest velocity was achieved at 8 Mpa inlet pressure with 30 percent pressure drop. Figure 4.2 shows the Mach number plot across the separator cross section. Due to the unstructured mesh generated, the nodes of the grid where the calculation were conducted were scattered. Hence, the Mach number plot across the cross section also scattered.

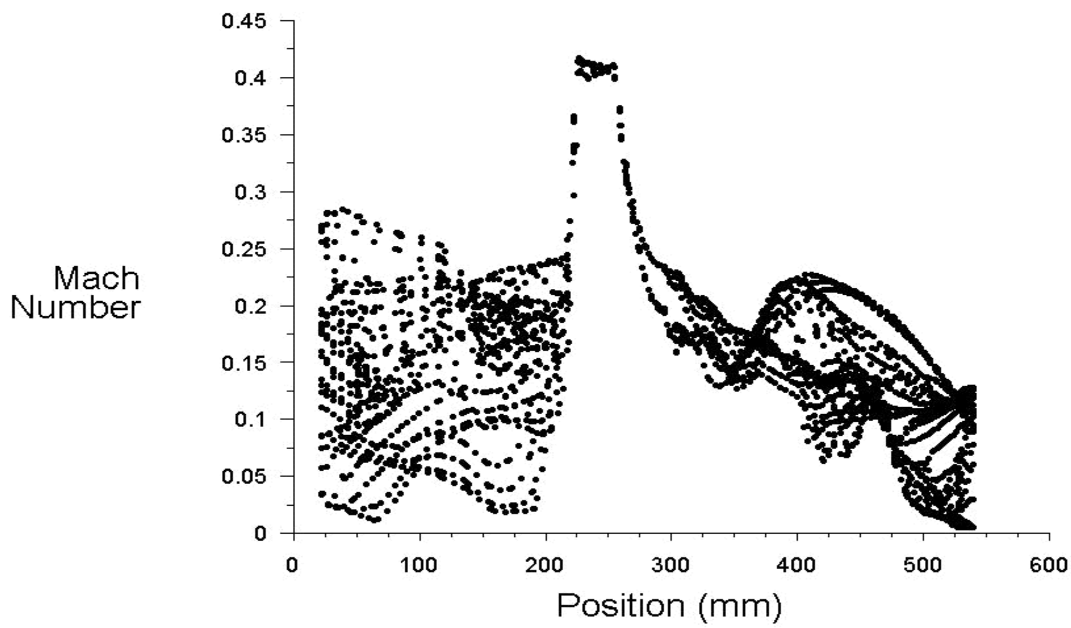


Figure 4.2: Mach number plot across separator cross section

From Figure 4.2 it shows that the highest velocity was achieved at 230 mm along X – axis which is the location of the smallest nozzle area which is the throat. From the Mach number plot, it shows that at the cyclone part the fluid velocity decreases due to expansion over a large volume. As the fluid approaching nozzle entrance it start to accelerate due to throttling effect of the nozzle. At the nozzle section the velocity increase rapidly from 0.24 Mach to 0.42 Mach which is located at the nozzle throat. Figure 4.3 shows the separator profile with respect to X-Y coordinate.

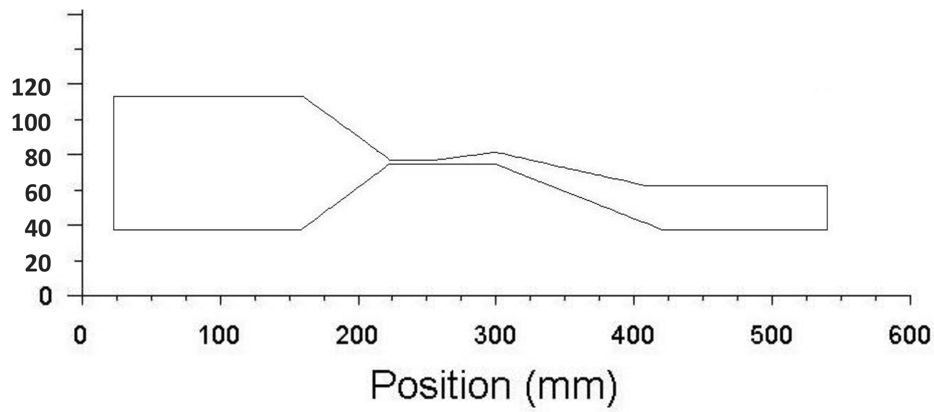


Figure 4.3: Separator profile coordinate

Figure 4.3 shows the location of the nozzle with respect to its Mach number plot. It shows that the nozzle throat is located at 230 mm from separator top wall.

Table 4.1: Parametric study result

Stagnation pressure (kpa)	Mach Number
40000	0.27
50000	0.32
60000	0.35
70000	0.38
80000	0.42

Variation of inlet pressure from 4 to 8 Mpa result in range of fluid velocity across the separator nozzle from 0.27 to 0.42 Mach. Although the velocity can be increased by increasing the inlet pressure; however, the inlet pressure greater than 8 Mpa are not acceptable as the normal platform operating pressure is maximum at 8 Mpa.

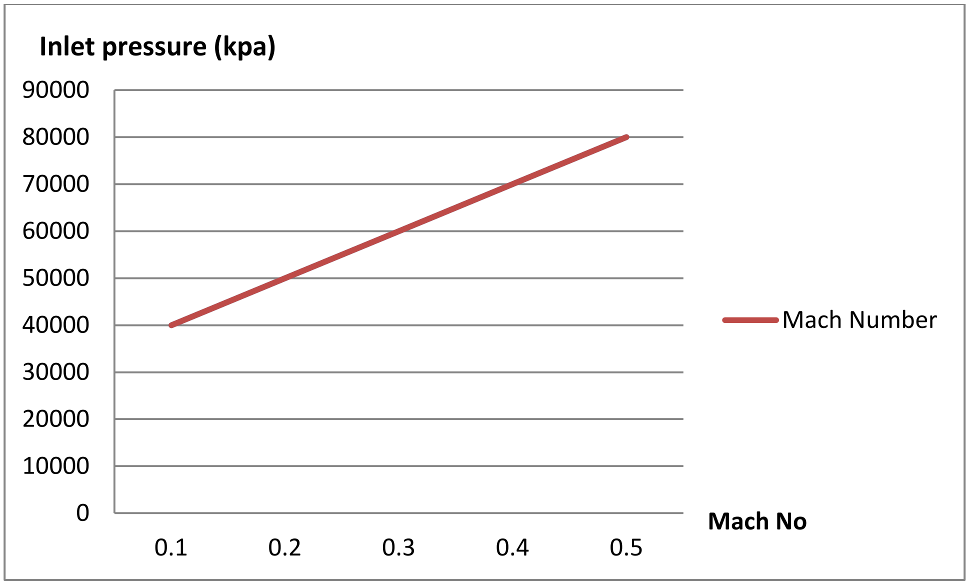


Figure 4.4: Inlet pressure against Mach number

Figure 4.3 shows the relationship between the inlet pressure and velocity across the nozzle. Velocity of fluid flow is directly proportional with the inlet pressure, the higher pressure inlet the higher inlet mass flow rate.

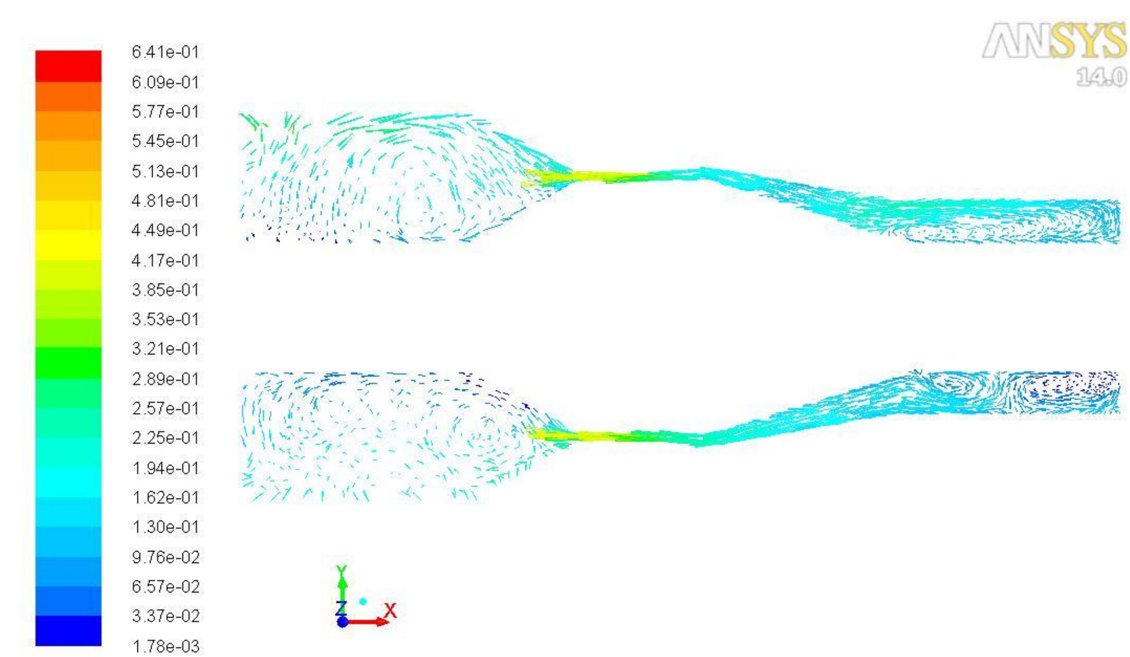


Figure 4.5: Velocity vector across separator cross section

Velocity vector plot in Figure 4.4 shows that the highest velocity of the fluid flow achieved at the nozzle geometry which is 4.2 Mach. As the fluid approaching the nozzle entrance its velocity increases rapidly and as it reach the nozzle the velocity increases further. However, due to subsonic velocity at the convergence section the divergence section acts as a diffuser which slows down the fluid velocity.

4.3 Grid Independence Test

Number of grid generated affect the simulation result in such a way that the finer mesh will capture the geometry more accurately; hence, the solution also could be said more accurate provided that the numerical method applied was appropriate. Grid independence test was carried out in order to determine the best result by refining the mesh. The solution is said grid independence when the increase in number of grid no longer affects the solution.

Table 4.3: Grid independence test results

Test no.	No. of element	Increment (%)	Mach No.	Result variation (%)
1	132982	0	0.420	0
2	233388	75	0.417	0.7
3	358240	169	0.418	0.5

Based on the test, further increment of number of grid does not affect the result significantly. Based on Table 4.3, increasing number of grid by 169 percent vary the result by only 0.5 percent. Hence, the solution can be said grid independence.

4.4 Design Recommendation

Based on the results obtained from the simulation, minor modification can be done in order to achieve supersonic flow across the nozzle. To increase the flow velocity modification can be done at the nozzle section and the inlet.

4.4.1 Nozzle

To obtain sonic velocity at the throat, the ratio of the throat diameter to inlet diameter is crucial, it is necessary to maintain the inlet diameter, D_1 larger than $\sqrt{5}$ of the throat diameter [12].

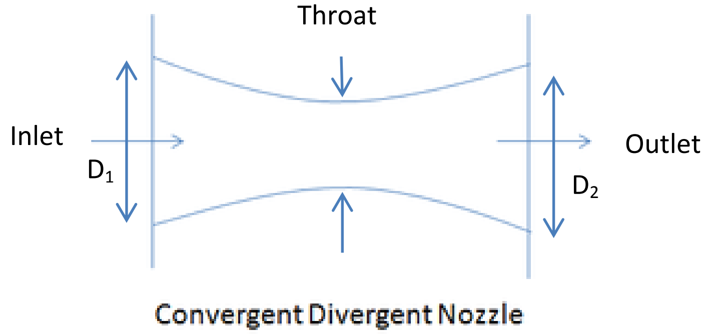


Figure 4.5: Convergence divergence nozzle [12]

The angle of the convergence section should be revised in order to improve the flow at throat entrance. According to Dieter K.H et al [19] the nozzle angle can be varied from 20 – 45 degree. This angle can be increase up to 45 degree in order to find the optimum angle for the convergent nozzle. To obtain high performance and short length nozzle, a contoured nozzle design can be adapted. Bell shape nozzle allows fast expansion of gas at divergent section [19].

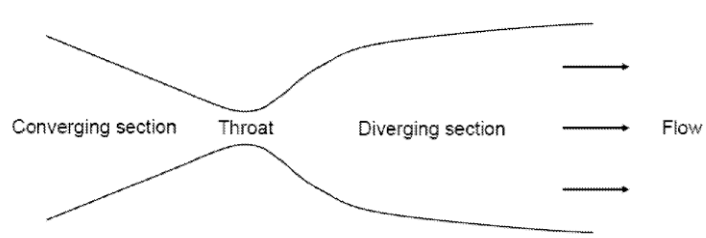


Figure 4.6: Example of bell shape nozzle [19]

5.2 Inlet diameter

By maintaining other parts dimension, the velocity across the nozzle can be increase by increasing the mass flow rate at nozzle inlet. In order to increase the velocity, the total mass flow rate flowing inside the system must be increase by increasing the inlet area [20]. According to conservation of mass equation, mass flow rate, area, and velocity can be relate by equation 8~9. ρ, A, V, and $\dot{m}$ are fluid density, duct area, velocity and mass flow rate respectively.

$$\dot{m} = \rho A V \tag{8}$$

$$V = \frac{\dot{m}}{\rho A} \tag{9}$$

From the above equation it shows that, by increasing the inlet mass flow rate the inlet velocity also increasing proportionally. By increasing the inlet mass flow rate only minor changes need to be done to the current design which is by increasing the inlet diameter.

Although these proposed modifications are not tested on the simulation; however, they can be considered in the future for design improvement.

CHAPTER 5

CONCLUSION

Supersonic separator with tangential inlet was successfully modeled with k- ε turbulence model combined with Redlich-Kwong equation of states. Natural gas dynamics parameter was simulated and the results are properly recorded graphically. The computational method developed predicts the flow behavior sufficiently according with the published validated method. Although, the experimental data is not available, however the numerical results are proved to be accurate as shown by the grid independence test result.

Based on the simulation parametric study, it shows that the maximum achievable velocity across the separator nozzle is 0.42 Mach which is in subsonic flow regime. The conducted parametric study provides range of operating pressure of the separator; it allows the model to be simulated at it maximum applicable pressure. Hence, the result can be claimed as the highest achievable velocity at the given range of operating parameter. The numerical results proved that the Swirler is not able to achieve a supersonic flow even though the maximum operating pressure is applied.

In order to optimize the current design, few minor modifications can be considered to achieve supersonic velocity at the nozzle section. Redesign the nozzle section according to the supersonic nozzle design criteria possibly could increase the throttling effect of the nozzle. Other than that, by increasing inlet diameter, the inlet mass flow rate also increases. Hence, the velocity at nozzle entrance can be increased proportionally. These modifications can be done without altering the whole parts of the separator.

Throughout the project, few challenges have been faced before the simulation was successful. Although the simplified geometry has eased the modeling works, however the computational grid generation of the geometry had caused tedious work in order to produce a good quality meshes. The meshes are critical at the nozzle where the gap is small and the grid skewness happened to be high at the tangential inlet geometry. Insufficient computational effort was surface during the grid independence test where the number of grid was increased by 169 percent. To overcome this problem, the calculation was run with computer with higher clock speed with multiple CPU units. As the result, the calculation was successful and the computational time was reduced as the parallel processing function was used.

The obtained CFD result has provide sufficient data which can be used to determine the separator functionality. Although the experimental data are not available to be compared, however the data obtained from the CFD analysis can be used as a guideline for future development of the Swirler. The CFD method developed in this project later can be used to carry out various analyses regarding this problem.

REFERENCES

[1] Ansys Inc. (2010). ANSYS CFD. Canonsburg, United States.

[2] Rawlins, C. (2003, May). The Case for Compact Separation. Technology Today Series.

[3] F. Okimoto, J.M. Brouwer, Supersonic gas conditioning, World Oil 223 (2002) 89–91.

[4] 4. H. Liu, Z. Liu, Y. Feng, K. Gu, T. Yan, Characteristic of a supersonic swirling dehydration system of natural gas, Chin. J. Chem. Eng. 13 (2005) 9–12.

[5] W. Jiang, Z. Liu, H. Liu, J. Zhang, X. Zhang, Y. Feng, Two dimensional simulation

[6] Kefalas, P. (2008). Numerical Analysis of Fluid Flow in a Compact Phase Separator. The Open Mechanical Engineering Journal, 21-23.

[7] Chuang Wen Xuewen Cao Yan Yang Wenlong Li. (2012). Supersonic Separators for Natural Gas Processing: Real Gas Effects. Twenty-second (2012) International Offshore and Polar Engineering Conference.

[8] M.E. Ferhat et al. (1996). CFD simulation of two phase and three phase gas liquid cylindrical cyclone separator. SPE annual technical conference & exibition.

[9] Chuang Wen et al. (2011). Swirling flow of natural gas in supersonic separators. Chemical Engineering and Processing:, 644-649.

[10] A.B Majid et al. (2010). Applications of CFD in Natural Gas. Canada: InTech.

[11] M.Haghighi et al. (2013). Natural gas dehydration using supersonic separator. Offshore Technology Conference. Texas, USA: Offshore Technology Conference.

[12] Wen, C., et al. (2010). Swirling flow of natural gas in supersonic separators.

[13] Redlich, O.K. (1949). On the thermodynamics of solution. An equation of state. Fugacities of gaseous solution, 233-234.

[14] E. Jassim, M.A. Abdi, Y. Muzychka. (2008). Computational fluid dynamics study for flow of natural gas through high pressure supersonic nozzle. Technol, 1773-1785.

[15] Ansys Inc. (2011, November). ANSYS FLUENT Theory Guide. Southpointe, Canonsburg, United States of America.

[16] 16.H.K Versteeg, W.Malalasekera. (1995). An introduction to Computational Fluid Dynamics. UnitedStates: Longman Scientific & Technical.

[17] Zikanov, O. (2010). Essential Computational Fluid Dynamics. New Jersey: John Wiley.

[18] Lyons, W. (2010). Working Guide to Petroleum and Natural Gas Production Engineering. Oxford, UK: Elsevier Inc.

[19] Arina, R. (2004). Numerical simulation of near-critical fluids. Applied Numerical Mathematics 51, 409–426.

[20] Dieter K.H et al. (1992). Modern Engineering for Design of Liquid-propellant Rocket Engines. Washinton DC: American Institutes of Aeronautics.

[21] B.R. Munson et al. (2010). Fundamental of fluid mechanic. River street: John Wiley & Son.