SIMULATION OF ENERGY PROMOTERS IN THE INTERMEDIATE PASSAGE OF GAS TURBINE

by

TAN FOK HON (9761)

Dissertation Report submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

MAY 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Simulation of Energy Promoters in the Intermediate Passage of Gas Turbine

by

Tan Fok Hon

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

(Dr. Hussain H Al-Kayiem)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

May 2011

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

TAN FOK HON

ABSTRACT

The main objective of this project is to perform simulations on the effect of energy promoters in an aggressive intermediate gas turbine duct (diffuser). Many approaches and ideas have been used to improve diffuser's size, but flow separation will occur and significantly reduces the diffuser's performance. Therefore energy promoter is introduced to reduce the boundary layer. This can be done by increasing the fluid's momentum and redirect them to oppose the boundary layer, thus reducing its size and adverse effect onto the diffuser. A typical S-shaped diffuser is designed and simulated to become the benchmark for this simulation testing. Then, the diffuser is shortened to create an aggressive diffuser design. Then the energy promoter is introduced, and simulated with various configurations to obtain the best height, and position. The objective is to obtain the best exit static pressure, which ultimately affect the diffuser's efficiency. The result shows the energy promoters works as intended, but still far from reaching the benchmark efficiency of a normal/ideal diffuser. Further testing will be required in three dimensional as well as experimental to realize this technology into the real world.

ACKNOWLEDGEMENT

Firstly, the author would like to convey my highest gratitude to my supervisor, Dr. Hussain H Al-Kayiem, for always assisting and advise me throughout this project. Next, the author would like to express humble gratitude to the author's family members, who had been supporting me, my FYP examiners, Ir. Mohd Shiraz Aris and Dr. Khairul Habib, who will be evaluating my works during this Puasa month. All of you had been gracious to allocate time for me.

As an undergraduate, it is important for the author to have a positive mind set during this period to avoid bad habits which will jeopardize the work pace of this project. Throughout the two semesters, the author had gained an immeasurable value of experience as well as knowledge in various aspects, especially in engineering project management. This exposure is very useful to the author's future career, and the author hope to make good use of the knowledge gained.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF APPENDIX

ABBREVIATION

The following abbreviations are used throughout this document:

AR	Aout/Ain exit to entry area ratio of duct (dimensionless)
AIDA	Aggressive Intermediate Duct Aerodynamics (EU project)
CFD	Computational Fluid Dynamics
HP	High Pressure
LP	Low Pressure
TTTF	Transonic Test Turbine Facility (Graz University of Technology)

CHAPTER 1

INTRODUCTION

1. BACKGROUND STUDY

In modern commercial gas turbine engines have to inherit traits of small specific fuel consumption as well as low life cycle cost to allow for the best economical solution. Furthermore, with increased global awareness on the importance of clean environment, gas turbine engines have strict design standards, which directly refer to reduced CO2 emissions by less fuel burning and generating less noise pollution during operation.

Gas turbine engine design has always become competitive for a few well known manufacturers over the world, coming out with the best gas turbine engine design, with low fuel consumption, meets the environmental code, while keeping the gas turbine engine light in weight, and compact in size. This delivers great advantage in aero engines, which lowers the overall weight and structure integrity of the aircraft, as well as keeping the space efficient for related industry.

Gas turbine engines are widely used in oil and gas industry, to generate electricity power as well as producing gas lift. Gas lift refers to harvesting natural gas from the sea. The bulky size of gas turbine–compressor/generator set has always become a great factor and limit in designing oil and gas offshore platform. It also limits the flexibility of improving the existing oil and gas platform by exchanging different model of the gas turbine according to needs.

Figure 1.1: Two Shaft Gas Turbine Engine

The best design idea to favour the function range of gas turbine is to reduce the size of gas turbine. Figure 1.1 shows a typical design two shaft gas turbine engine. A diffuser is located in between the LP turbine and HP turbine, namely intermediate turbine diffuser with the function of pressure recovery. This diffuser's size increases with higher power output, due to higher combustor exit flow. Numerous researches is done to study the behaviour and different configuration of this diffuser as well as improving them.

1.1 Problem Statement

Throughout the year, many engineers from all over the world is researching and finding ways to further improve the turbine, in terms of its efficiency, size, portability, all in favour of many industrial advantages. For this project, the author will focus on improving the gas turbine by reducing its size at the exhaust compartment.

Reducing the size of the gas turbine can have many advantages, this include increasing the stability of the shaft (shorter shaft has better stability/balance), and reducing its total weight. Other than that, this can contribute to easier position allocation for gas turbine at offshore platform structures, as the space available is very limited.

In a gas turbine, a diffuser is used at the exhaust compartment in between two power turbine blades to increase the flow's static pressure. However, by shortening this diffuser, while retaining its inlet and outlet size (the diffuser's cone angle will increase), flow separation will occur that will significantly reduce the turbine's performance.

1.2 Objectives

The objective that the author would like to achieve in this project is to investigate the energy promoters in a diffuser by CFD simulation. This includes:

a) Create a 2D intermediate diffuser.
b) Simulate the model with assist of energy promoters.
c) Simulate the model with various configurations of energy promoters.

1.3 Scope of Work

The scope of study will cover searching and studying relevant journals, design a 2D S Shaped Diffuser in GAMBIT, simulation of a normal angled diffuser, a shorten version of the diffuser, with and without the energy promoters. Lastly, the author will simulate the diffuser by varying the position, height and shape of the energy promoters.

CHAPTER 2
LITERATURE REVIEW

2.1 Flow Separation

David Balmer [1] explains how flow separation can occur through the Navier Stokes Equation. In summary of his study, he states that whenever a severe adverse pressure gradient exist in a flow, the velocity profile of the flow will become increasingly distorted, and thus creating the effect known as **separation**. Other than that, he also discussed that the separation accounts for the majority of the drag on a bluff body due to inertial effects. Lastly, he suggested that the effect of separation can be prevented by accelerating the boundary layer, which gives a high kinetic energy to fluid in the boundary layer to overcome the adverse pressure gradient.

In Peter Bradshaw and Rabi Mehta's Wind Tunnel Design [2], he used an aggresive diffuser as part of his design. He mentioned one of the problems that he faced while using this type of diffuser is flow separation due to the cone angle of the diffuser is higher than 5 degree, and he managed to solve it by introducing screens made up by woven wire gauze in the diffuser. This screen can improve the uniformity of the flow's velocity profile, thus eliminate the separation.

N.F. Zulkefli and K.A. Ahmad [3] had done a numerical simulation of the effect of streamwise vortices on turbulent flow structure. Their objective in his simulation is to obtain the optimum parameter of sub-boundary layer vortex generator. According to their journal, they had used Commercial Code Fluent 6.3^{TM} to simulate their model. In their journal, they stated two different type of flow control device, which is passive and active control device. They had also elaborated on the importance of flow control devices, as separation contributes to great energy losses.

Olaf Sieker and Joerg R. Seume [4] discussed that the power and efficiency of turbines strongly depend on the performance of the exhaust turbine diffuser in one of their journal. They did an experiment analysis to relate the influence of rotating wakes on separation in turbine exhaust diffusers.

Keiko Fukudome, Masashi Watanabe, Akiyoshi Iida and Akisatu Mizuno [5] did an analysis on separation control of high angle of attack air foil for vertical axis wind turbines. Some of the methods they used in their analysis is by using turbulence promoters, oil film and numerical simulation. They had concluded that the present of turbulence promoter is useful to modify the aerodynamic performance of the vertical axis wind turbine.

Lord et al. [6] investigated on active or passive flow controls to design more aggressive transition duct geometries with larger radial offsets. The first type can either be energization of boundary layer by injecting high energy fluid or removal of low energy fluid from critical wall region.

Two patents held by General Electric, namely Graziosi and Kirtley [7] as well as Widenhoefer et al. [8] introduces inter turbine diffusers with different type of method in eliminating flow separation. In both inventions, secondary air is injected to energize the boundary layer in order to prevent it from separation. It was mentioned that the air will be taken from the compressor section of the gas turbine, due to suitable static pressure ratio between suction port and the injection slot.

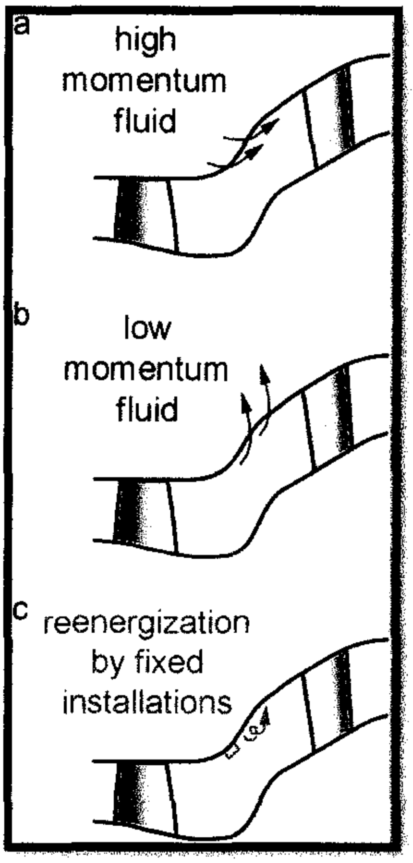


Figure 2.1: Flow control mechanism active (a and b) and passive (c) measures for re-energization of boundary layer.

2.2 Intermediate Passageway and Energy Promoters Geometry

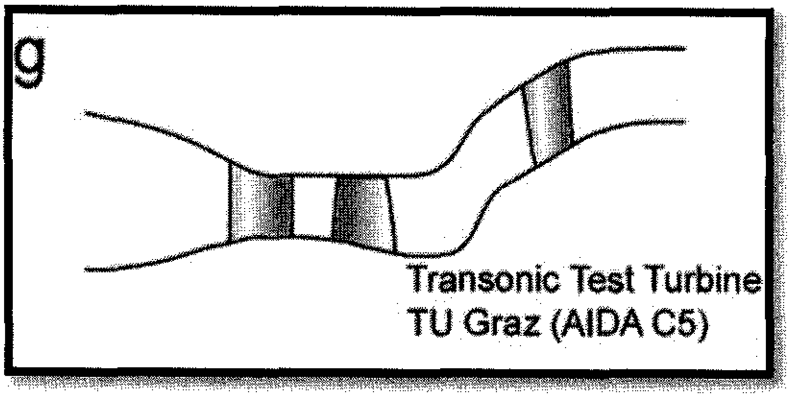


Figure 2.2: Test setups for intermediate turbine diffusers at different test turbine rigs.

Emil Gottlich explains that passive flow control is less complex than active flow control, because there is no need for handling additional fluid streams at unsteady flow rates. Merely the installation of fixed components at the right position would be very beneficial to re-energize boundary layer. He also installed a work package, namely EU project AIDA [9] to evaluate the application of passive flow control devices in both compressor and turbine transition ducts. Low vortex generators have been designated for one of their super aggressive intermediate turbine diffuser setup, AIDA C5 (see Fig. 2.2) and TTTF at Graz University of Technology. These ducts shows fully separated flow on casing wall and therefore suitable for the study of passive flow control devices in order to show improvements after installation.

Lin [10] performed a thorough review on low-profile vortex generator and their ability to prevent flow separation. The working principle explained is to transport high momentum fluid from the core flow into the boundary layer by means of stream wise vortices. One of the recommended vortex generator geometry is the vane type (see Fig. 2.3). Vortex is formed behind these small vanes as the flow has to pass the tip region from the pressure to the suction side. With right adjustment and parameters to the vortex generators, counter rotating stream wise vortices can be generated. He also mentioned on the additional drag generated by this vortex

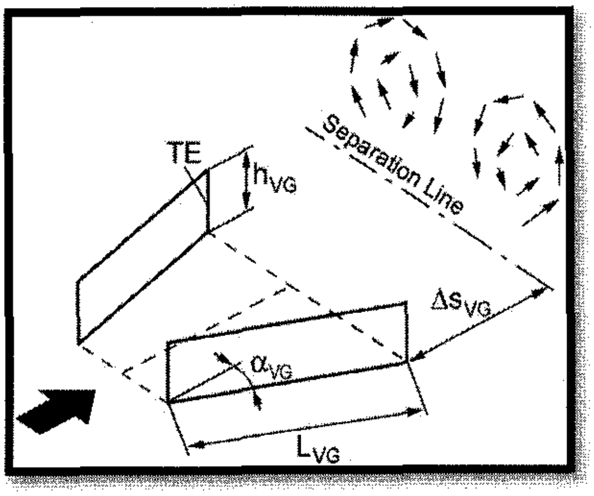


Figure 2.3: Vane-type vortex generator

generator can result in total pressure loss. For that reason, he suggested for the extension of the low profile vortex generator into the flow is only 10% to 50% of the boundary layer thickness to minimize their losses, while retaining its ability to provide high momentum fluid over the boundary layer.

A vortex generator model of [11] has been adopted for the investigation of various configurations within a design of experiments for a flow controlled intermediate turbine diffuser by Wallin and Eriksson [12]. Some of the factors in designing this experiment include but not limited to their position, height, length and angle of attack, but it was not within the scope of the work to present the complete design process for vortex generator controlled intermediate turbine diffusers. A very aggressive and separating duct design with an area ratio AR of 1.62 and L/h_{in} of 2.56 was chosen based on the Sovran and Klomp diagram (see Fig 2.4).

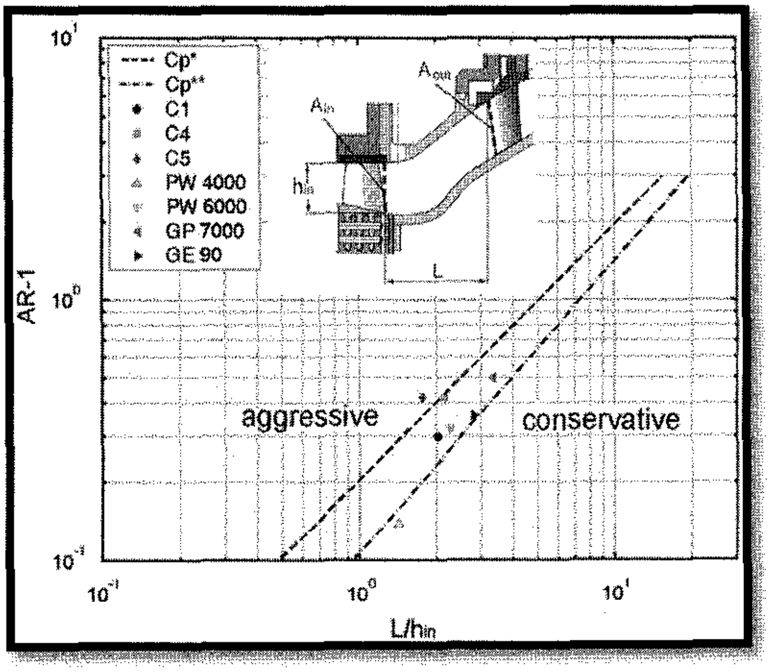


Figure 2.4: Sovran and Klomp Performance Chart

Four parameters with influence on the vortex generator performance were allowed to vary within the bounds:

1) A non-dimensional location of the trailing edges relative to the baseline separation line ($\frac{\Delta S_{VG}}{h_{VG}} = 0 - 11$).

2) The minimum height corresponds with the boundary layer thickness. Height ($h_{VG} = 1.3 - 2.9mm$).

3) Non dimensional length ($\frac{L_{VG}}{h_{VG}} = 2.5 - 5.5 * h_{VG}$).

4) Angle of attack (10 – 26 degree).

The procedure resulted in the optimum design parameter settings of $h_{VG} = 1.9mm, \frac{\Delta S_{VG}}{h_{VG}} = 0, \frac{L_{VG}}{h_{VG}} = 4$, and angle of attack of 5 degree. In comparison with former experiments, where optimal angle of attack is around 25 degree, here the optimum design was found to be only 5 degree.

Santer et al. [13] investigated on the performance and application of a low profile vortex generator installation within a super aggressive intermediate turbine diffuser (very high diffusion rate) within the AIDA project. According to Wallin and Eriksson [12], the angle of attack and the position of the energy promoters has the highest influence on its efficiency. In their tests, they used low profile vanes similar to those defined in the work of Canepa et al. [14], angle attack of 25 degree. CFD simulations where conducted to find the optimal position and height of the vortex generator. The optimum parameters found is a height of 0.7 mm, and length of 5 mm. They tried to mimic the result experimentally, a simple method for the installation of the vortex generator on the existing duct geometry was needed due to cost and time simple. They manufactured by stamping their shape and glued them on the surface.

CHAPTER 3
METHODOLOGY

3. ANALYSIS METHOD

3.1 Numerical Analysis

CFD simulations will be carried out using both GAMBIT version 2.2.30 and FLUENT version 6.3.26 software. Simulations will be done to prove the existence of flow separation as the diffuser's cone angle increase, and reduce the separation by introducing energy promoters.

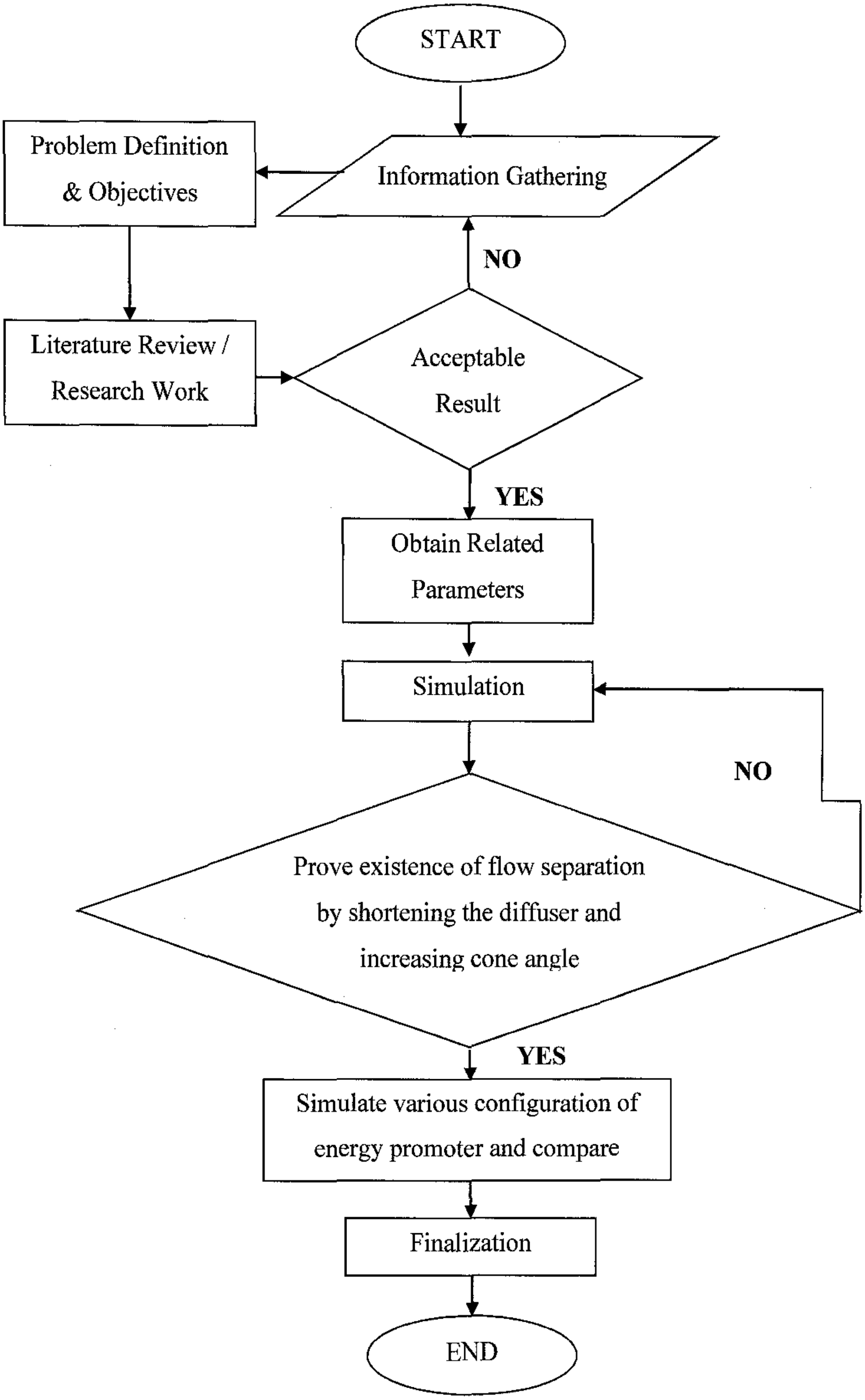
3.1.1 GAMBIT 2.2.30 (Modelling Software)

- A pre-processor for engineering analysis
- Easy to use interface with advanced geometry and meshing tools in a powerful, flexible, tightly integrated with any major CAD / CAE system.

3.1.2 FLUENT 6.3.26 (Simulating Software)

- Broad physical modelling capabilities needed to model flow, turbulence, and heat transfer.
- Advanced solver technology provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability.

3.2 Execution Flow Chart



3.3 Gantt Chart FYP I

No.	Detail / Week	1	2	3	4	5	6		7	8	9	10	11	12	13	14
1.	Selection of Project Topic	▨	▨					Mid Semester Break								
2.	Preliminary Research Work		▨	▨	▨											
3.	Submission of Preliminary Report				√											
4.	Identify and obtain related parameters					▨	▨									
5.	Submission of Progress Report									√						
6.	Seminar									√						
7.	Create a diffuser model in CFD					▨	▨		▨	▨	▨	▨	▨	▨	▨	▨
8.	Submission of Interim Report Final Draft															√
9.	Oral Presentation															√

Gantt Chart FYP II

No.	Detail / Week	1	2	3	4	5	6		7	8	9	10	11	12	13	14
1.	Simulate the effect of energy promoter	▨	▨	▨	▨	▨	▨	Mid Semester Break								
2.	Submission of Progress Report 1				√											
3.	Simulate various type of energy promoter				▨	▨	▨		▨	▨	▨	▨				
4.	Submission of Progress Report 2									√						
5.	Seminar									√						
6.	Poster Exhibition and EDX													√		
7.	Submission of Dissertation Final Draft														√	
8.	Oral Presentation								During Study Week							
9.	Submission of Dissertation (hard bound)								7 Days after Oral Presentation							

CHAPTER4

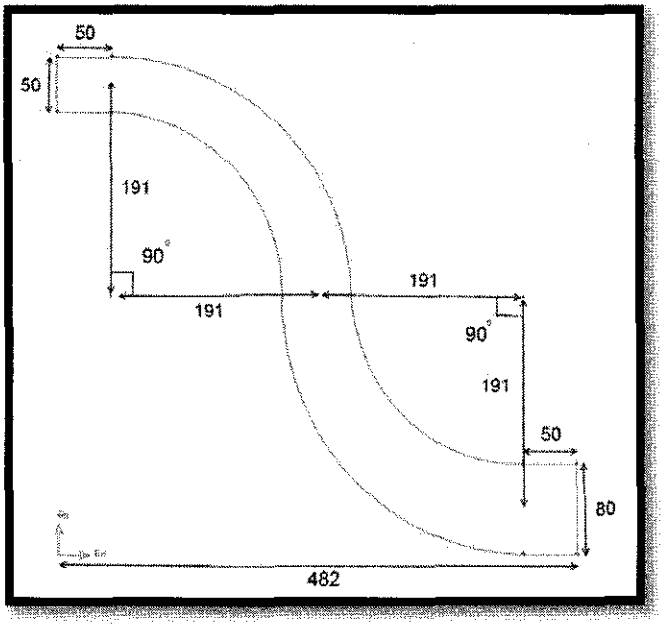
RESULT & DISCUSSION

4.1 RESULT

4.1.1 Geometric Details

Following is the geometric details of the chosen models for this project. Since the complexity of S-shape diffuser design dimension is not revealed in all previous literatures, a typical S-shape diffuser dimension is used instead in the following project, and the length is shortened to mimic the design of an aggressive S-shape diffuser.

4.1.1.1 Normal Diffuser

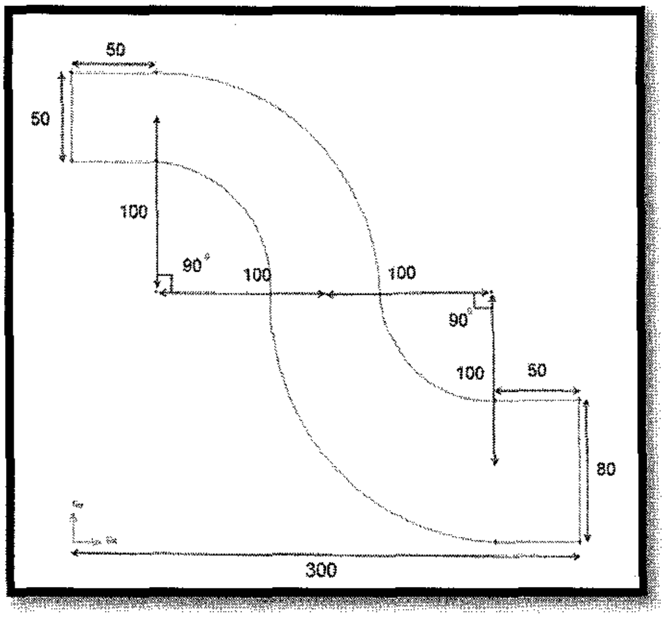


Radius of curvature = 191 mm

Centreline length = 600 mm

Figure 4.1: Normal Diffuser Design Dimension

4.1.1.2 Aggressive Diffuser



Radius of curvature = 100 mm

Centreline length = 314 mm

Figure 4.2: Aggressive Diffuser Design Dimension

4.1.2 Boundary Conditions

4.1.2.1 Inlet Boundary Conditions

i)	Type of boundary	Velocity Inlet
ii)	Inlet speed	200 m/s
iii)	Turbulence intensity	10%

4.1.2.2 Outlet Boundary Conditions

i)	Type of boundary	Pressure Outlet
ii)	Pressure specified	0 Pa Gauge Scale

4.1.2.3 Wall Boundary Conditions

i)	Type of boundary	Rough; 0.01 roughness height
ii)	Shear condition	No-slip

4.1.2.4 Working Fluid Conditions

i) Working fluid Flue Gas

Temperature	Density	Specific heat	Dynamic Viscosity	Kinematic Viscosity
C	kg/m3	kJ/kgK	Pas e-6	m2/s e-6
0	1.295	1.042	15.8	12.2
100	0.95	1.068	20.4	21.54
200	0.748	1.097	24.5	32.8
300	0.617	1.122	28.2	45.81
400	0.525	1.151	31.7	60.38
500	0.457	1.185	34.8	76.3
600	0.405	1.214	37.9	93.61
700	0.363	1.239	40.7	112.1
800	0.33	1.264	43.4	131.8
900	0.301	1.29	45.9	152.5
1000	0.275	1.306	48.4	174.3
1100	0.257	1.323	50.7	197.1
1200	0.24	1.34	53	221

Table 4.1: Flue Gas Properties [15]

4.1.2.5 Constants

i)	C1-Epsilon, $C_{1\varepsilon}$	1.44
ii)	C2-Epsilon, $C_{2\varepsilon}$	1.92
iii)	CMU, C_{μ}	0.09
iv)	TKE Prandtl Number, σ_k	1.0
v)	TDR Prandtl Number, σ_ε	1.33

4.1.3 Simulation Result

4.1.3.1 Velocity Vectors

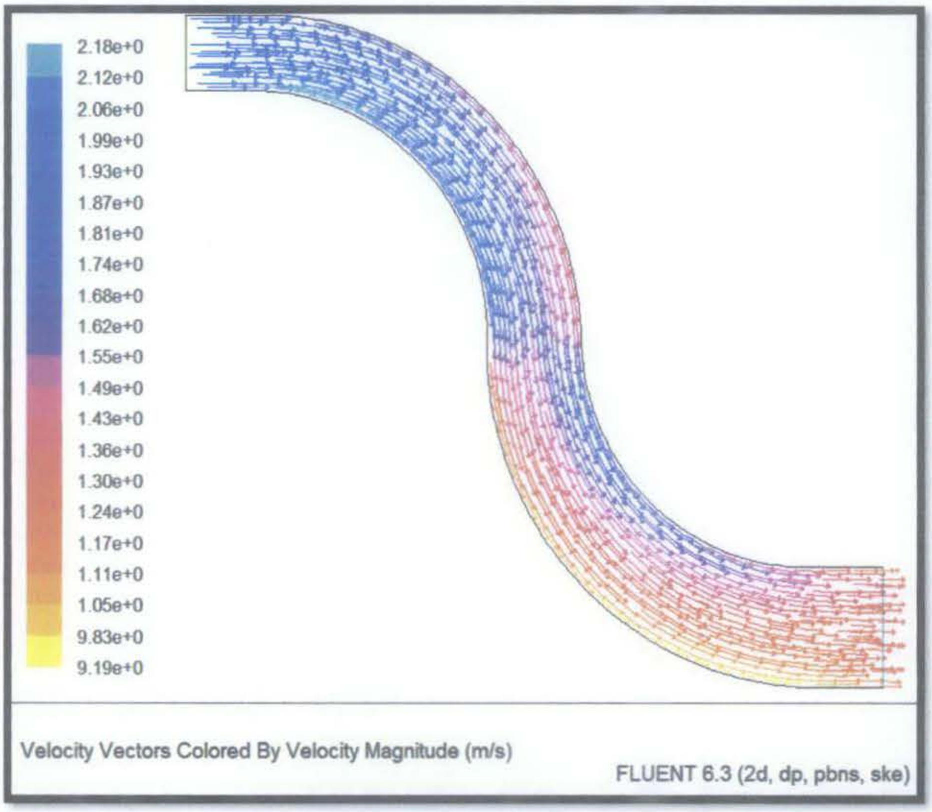


Figure 4.3: Normal Diffuser Velocity Vector

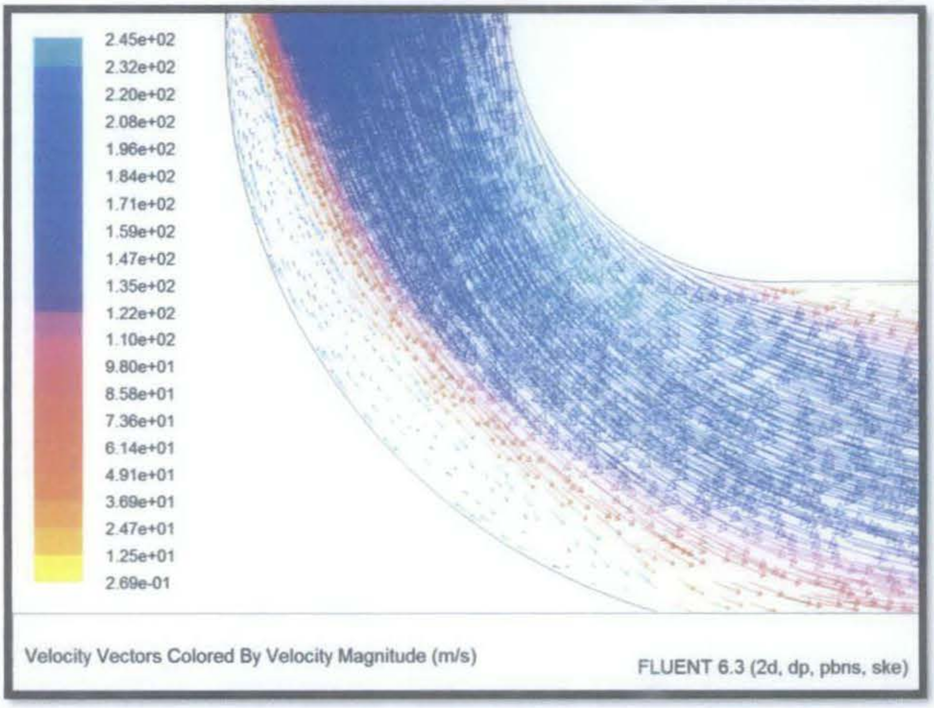


Figure 4.4: Aggressive Diffuser Velocity Vector

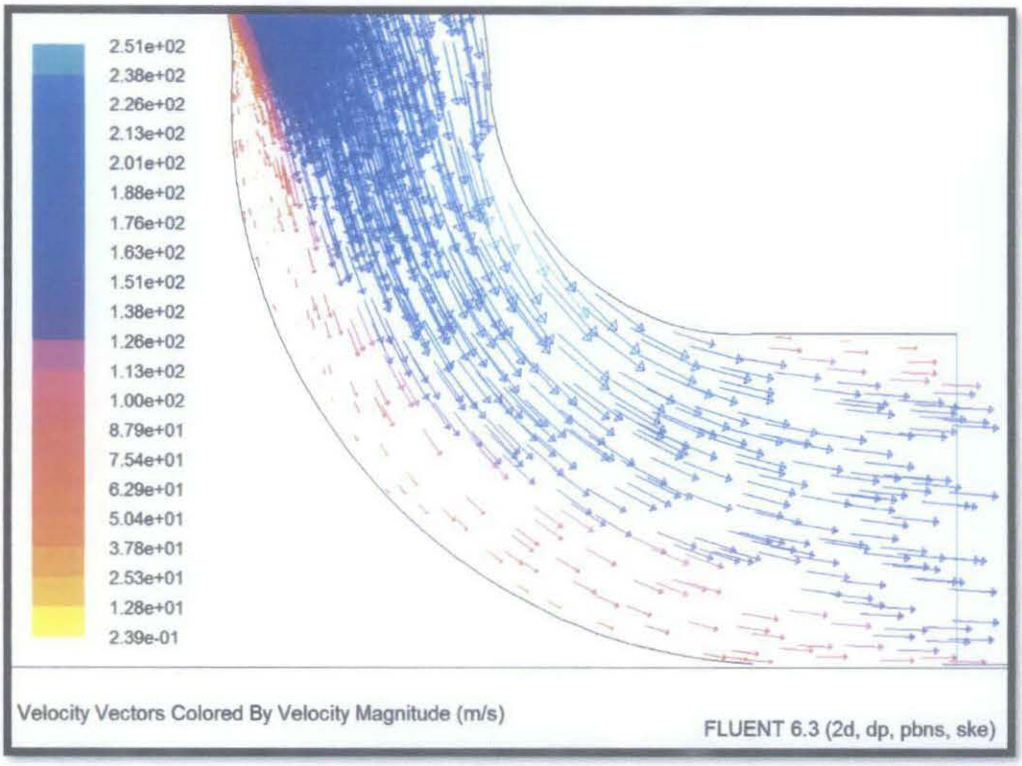


Figure 4.5: Aggressive Diffuser with Energy Promoter Velocity Vector

4.1.3.2 Total Pressure Contour

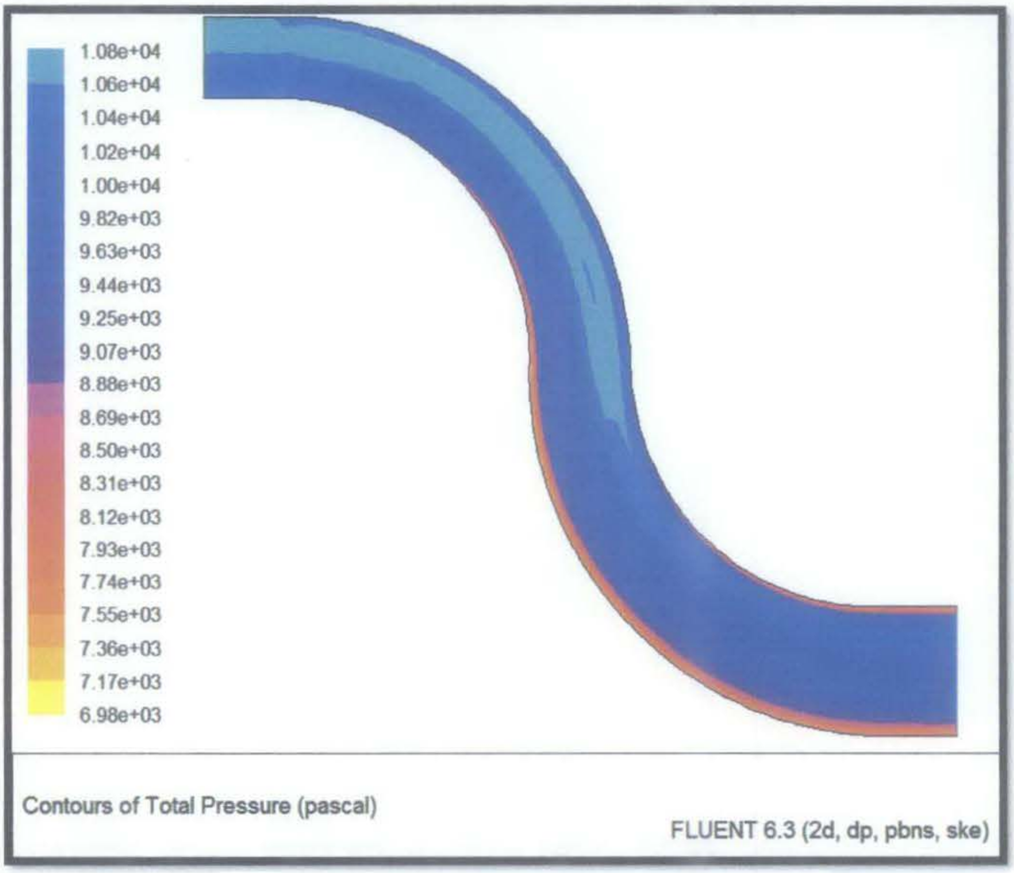


Figure 4.6: Normal Diffuser Total Pressure Contour

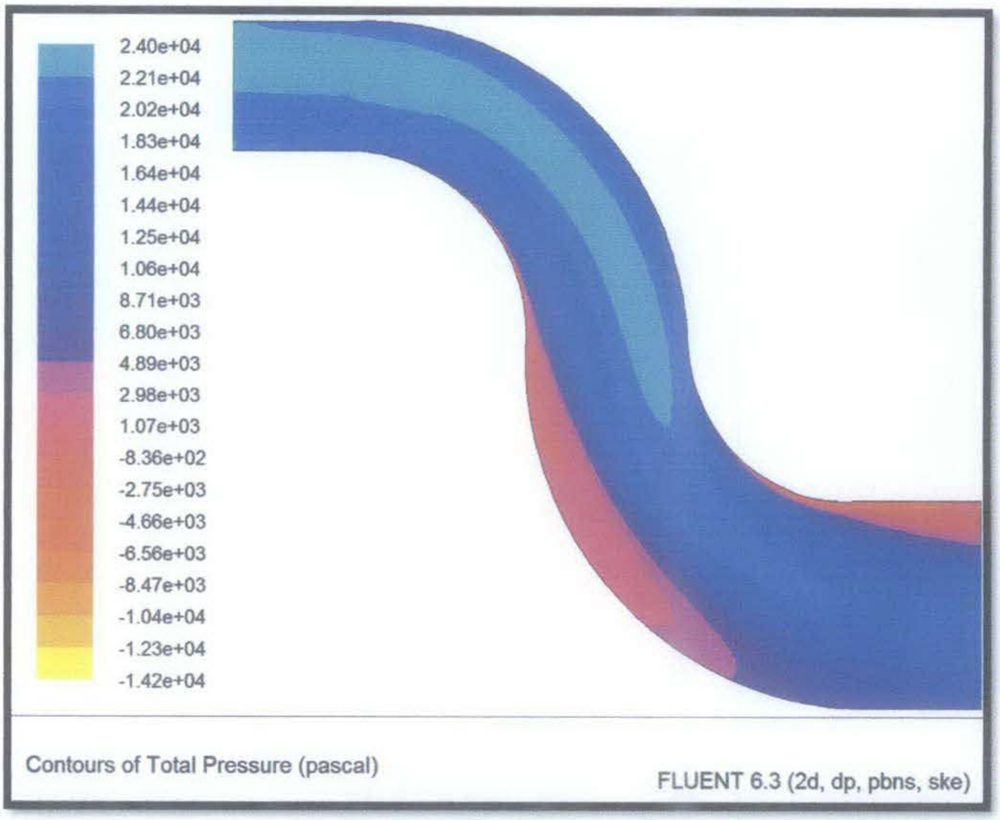


Figure 4.7: Aggressive Diffuser Total Pressure Contour

4.1.3.3 Static Pressure Contour

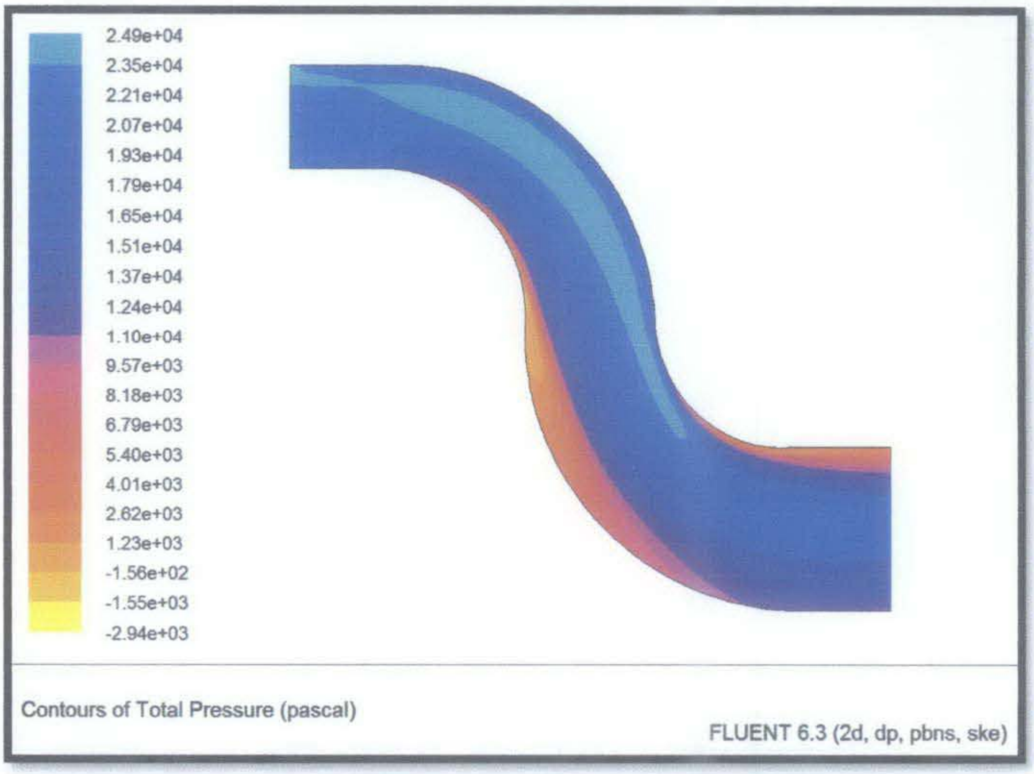


Figure 4.8: Aggressive Diffuser with Energy Promoter Total Pressure Contour

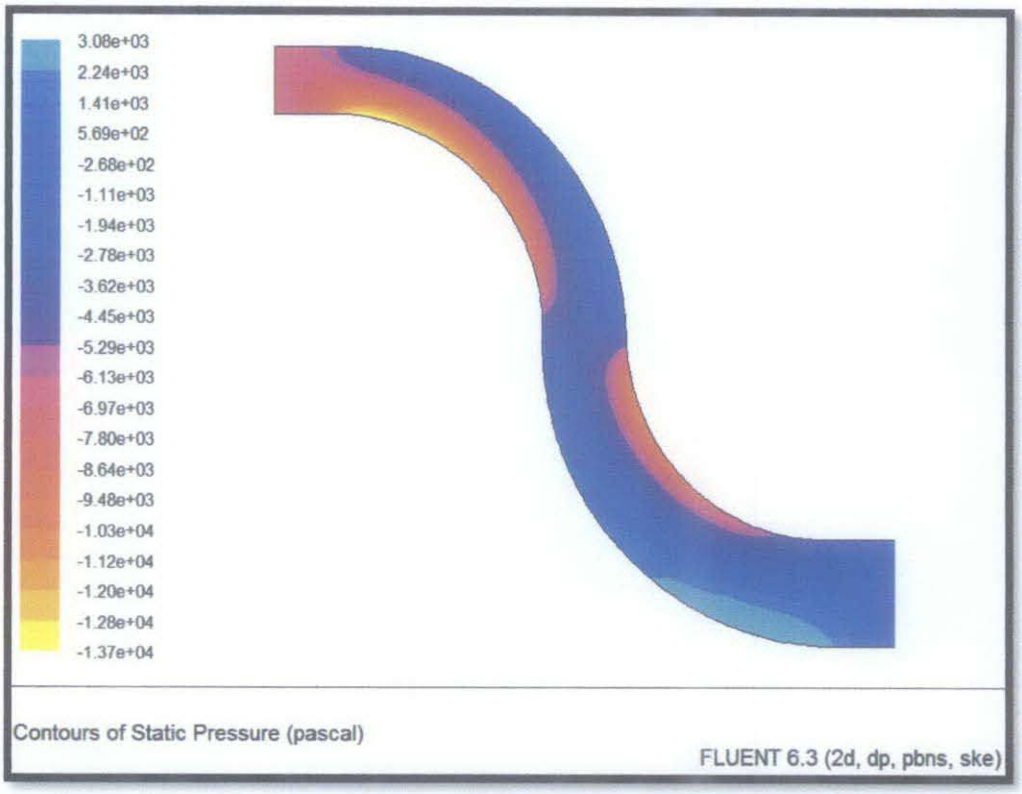


Figure 4.9: Normal Diffuser Static Pressure Contour

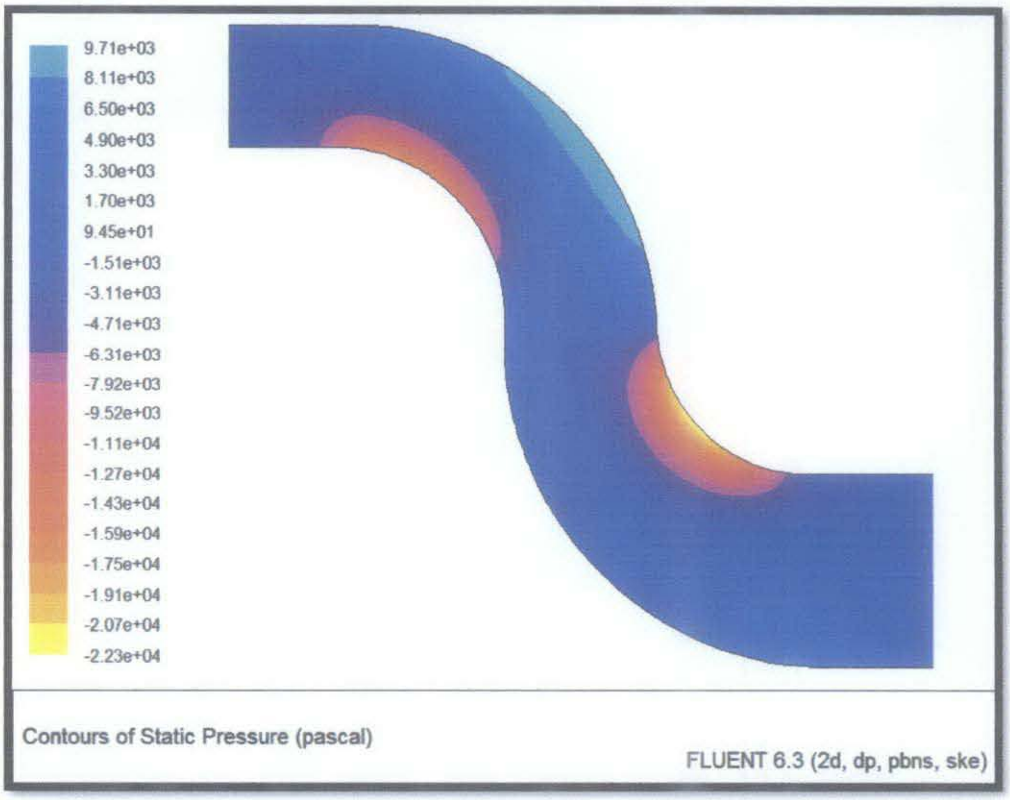


Figure 4.10: Aggressive Diffuser Static Pressure Contour

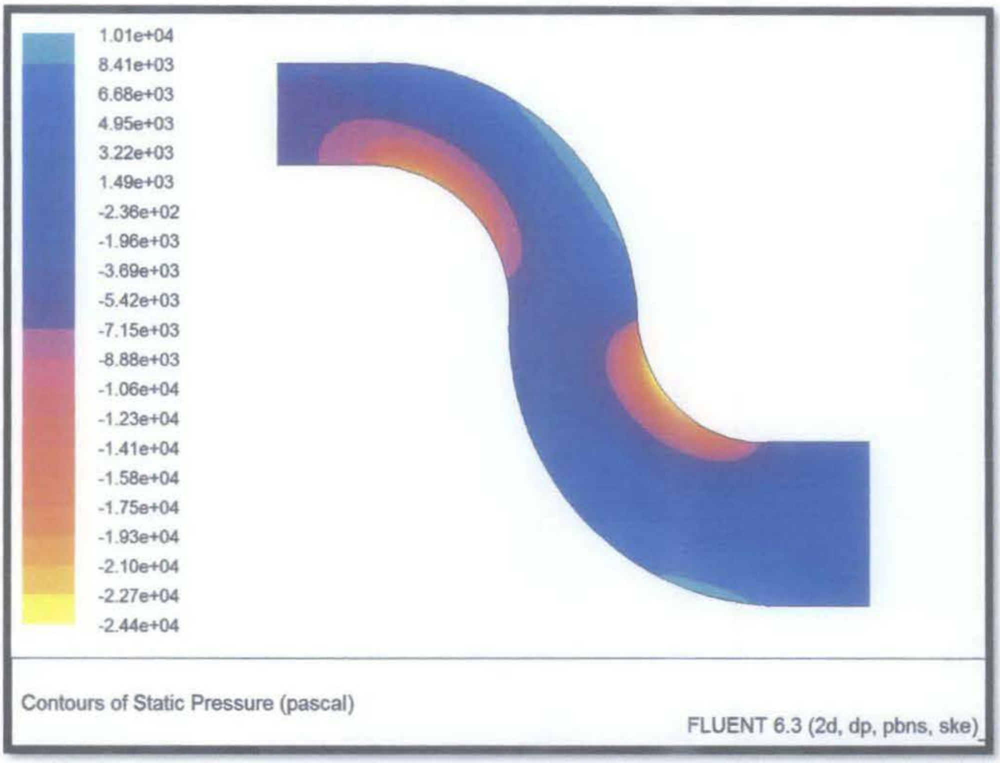


Figure 4.11: Aggressive Diffuser with Energy Promoter Static Pressure Contour

4.2 DISCUSSION

As per the CFD simulation conducted, the following variables were used to investigate the best position and height of the energy promoters. The objective is to obtain the best static pressure difference.

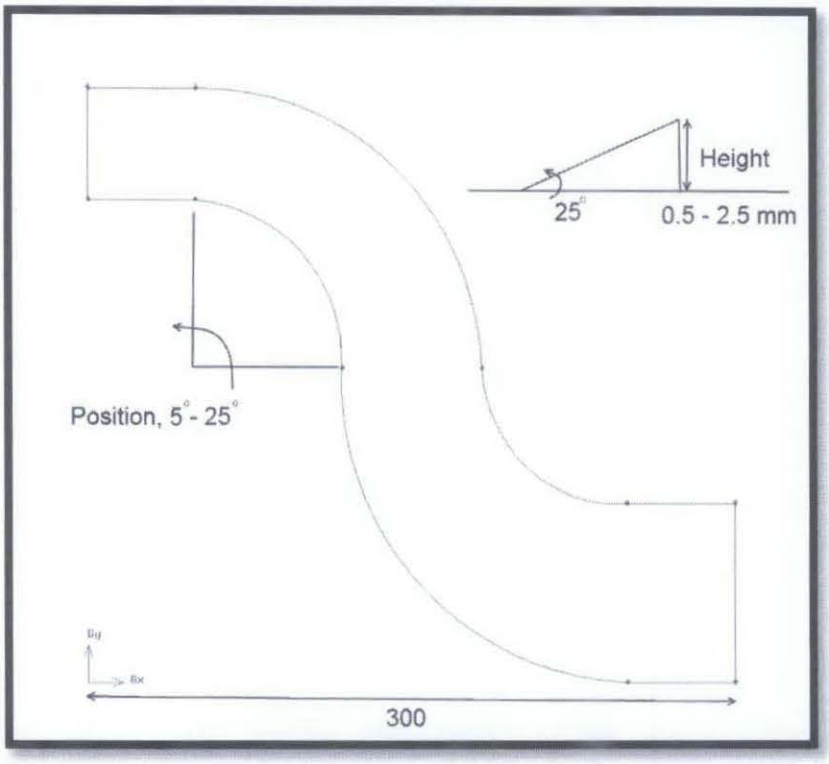


Figure 4.12: Energy Promoter's Variables

Figure 4.3, 4.4 and 4.5 shows the velocity vectors of three different types of diffuser, with figure 4.3 as benchmark. At the second bend of the aggressive diffuser (Fig 4.4) shows that the separated region with reversed flow creates the boundary layer which lowers the efficiency of the diffuser in gaining static pressure. After adding in the energy promoter in the diffuser at the optimized position and height, the separated region is lowered, creating less boundary layer area, and increases the diffuser's efficiency.

Figure 4.6, 4.7 and 4.8 shows the total pressure of the three different diffusers. Low total pressure region shows where the separated region is. An improvement is seen over the diffuser with energy promoter (Fig 4.8) with comparison with the aggressive diffuser (Fig 4.7).

The static pressure recovery contour is also shown respectively at Figure 4.9, 4.10 and 4.11. The result is plotted through XY plot in Fluent software, and the following results were obtained:

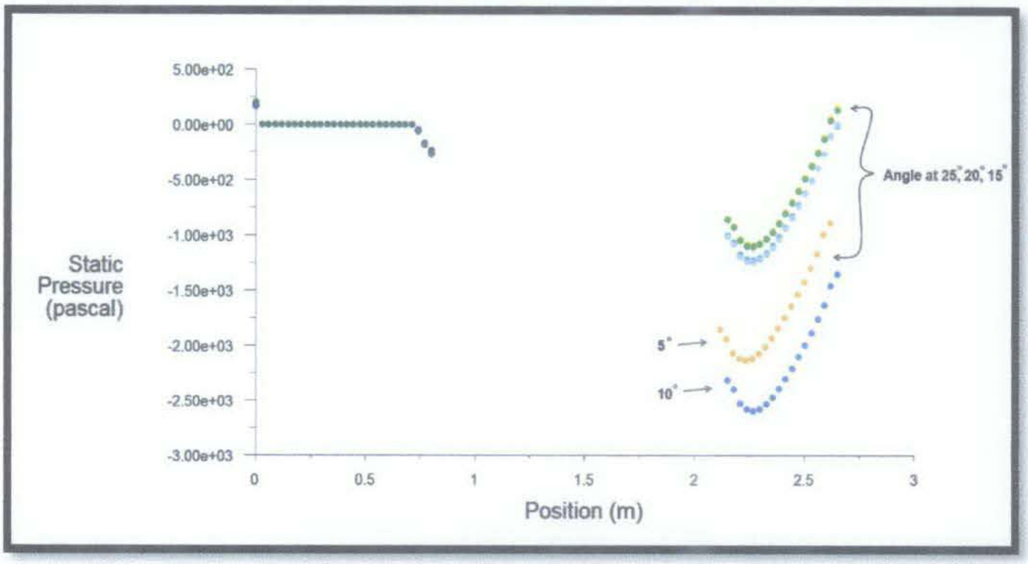


Figure 4.13: Static Pressure at Inlet and Outlet with Varying Energy Promoter's Position

The exit static pressure is at lowest value at position 10 degree at the first bend. At angle 25, 20 and 15, the exit static pressure shows slight changes, due to its misplaced position. Instead of increase the momentum of the fluid to attack the boundary layer, some values reached higher than the inlet static pressure, which shows that the diffuser acted as a nozzle, ineffective in recovering pressure but accelerating, creating a worse boundary layer. At angle of 5 degree, with comparison to 10 degree, the static pressure increased, showing that the effective position of the energy promoter is at 10 degree.

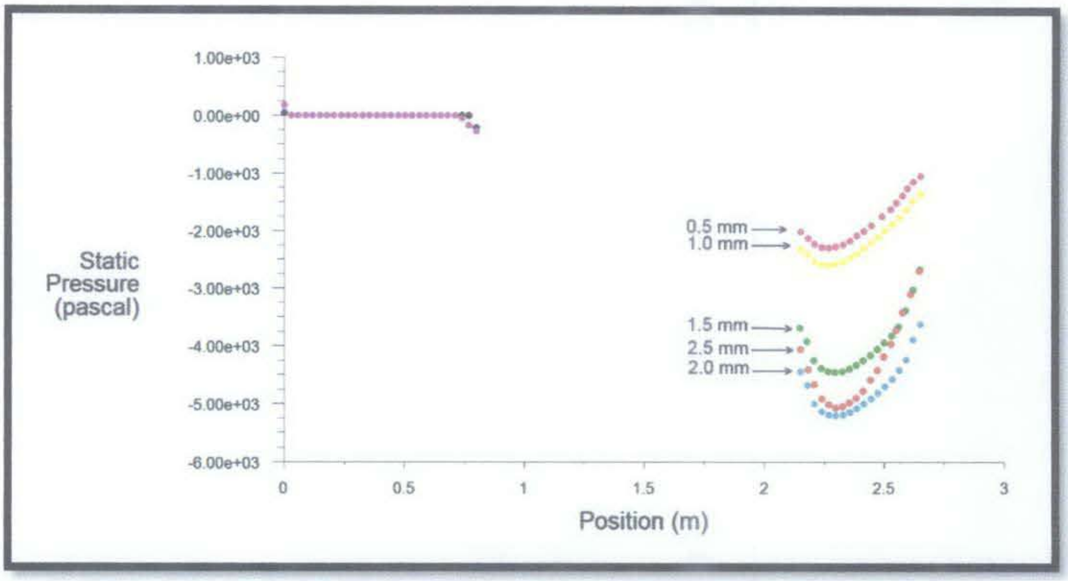


Figure 4.14: Static Pressure at Inlet and Outlet with Varying Energy Promoter's Height

By varying the height of the energy promoter at 10 degree, we can obtain the best height targeting at lowest static pressure. With the aid of figure 4.14, we can see that energy promoter at height of 2.0 mm delivers the biggest static pressure difference, which refers to highest static pressure recovery. Low height of the energy promoters shows that the vortex generated is not sufficient to deliver high momentum fluid towards the boundary layer, eliminating it. As the height increases to 2.5 mm, the exit static pressure shows a more adverse distribution, and lower static pressure recovery. This indicated that the vortex generated by the energy promoter extends the separated region towards the diffuser's exit, resulting in uneven static pressure distribution.

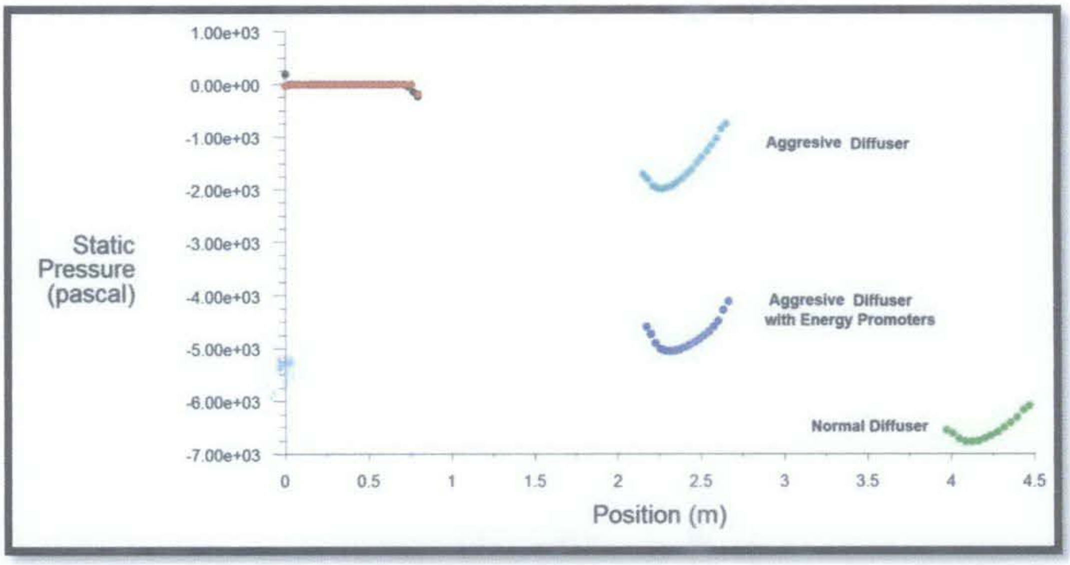


Figure 4.15: Static Pressure at Inlet and Outlet with Diffusers with and without Energy Promoter

Figure 4.15 shows a static pressure difference comparison between a normal diffuser, aggressive diffuser, as well as an aggressive diffuser with energy promoter. In contrast of the aggressive diffuser, with and without energy promoters, it is obvious that the energy promoters helped in delivering high momentum fluid towards the boundary layer, creating a higher diffusion capability, despite its inability to perform the exact same result of a normal diffuser.

Some of the reason contributes to this pressure recovery losses may due to the over-aggressive diffuser design (very short centre-line length) (see Fig 4.1 and 4.2) that was used in this project, which creates a large separated region that is not fully eliminate-able. Other than that, due to insufficient resources on current existing aggressive diffuser design geometry, a typical S-shaped diffuser was chosen instead, and perhaps this diffuser design is out of the effective range of energy promoter's works, which leads to vast difference in exit static pressure between the normal diffuser and the aggressive diffuser with energy promoter.

Table 4.2 shows the inlet and exit velocity of the fluid, static pressure recovered, pressure rise coefficient, ideal pressure rise coefficient and diffuser's efficiency (referred to as the diffuser effectiveness by Sovran and Klomp (1967).

Configuration	C1 m/s	C2 m/s	Pressure Recovered Pascal	Cp	Cpi	nD
Normal Diffuser	200	124	6500	0.43	0.61	71.23
Aggressive Diffuser	200	159	1500	0.10	0.61	16.44
Aggressive Diffuser with Energy Promoter	200	138	3500	0.23	0.61	38.35

Table 4.2: Diffuser's Efficiency Comparison

These parameters were obtained through these formulas:

Pressure rise coefficient:

$$Cp = (p_2 - p_1)/q_1,$$

Where $q_1 = \frac{1}{2}\rho c_1^2$, Since T = 200 Celsius, $\rho = 0.748\ kg/m^3$

Diffuser's efficiency:

$$\eta_D = (\frac{C_p}{C_{pi}}),$$

Where the ideal pressure rise coefficient:

$$C_{pi} = 1 - \left[\frac{c_1}{c_2}\right]^2 = 1 - \left[\frac{1}{A_R^2}\right]$$

From the result obtained, we can see an increment of 21.91% in diffuser's efficiency with addition of energy promoter into the aggressive diffuser design. However, the resulted efficiency is still 32.88 % lower than the benchmark normal diffuser's efficiency.

Grid Independency Check:

Serial no:	No of volume cells	Static Pressure (Pascal)	Difference %
1	20510	5700	0.00
2	27250	6200	8.77
3	30294	6450	4.03
4	35016	6500	0.78

Table 4.3: Grid Independency Check

CHAPTER 5

CONCLUSION

The purpose of this final year project is to study on the behaviour of Energy Promoter on a short/aggressive diffuser with flow separation. The discussion begins with a review on past literatures such as the fundamentals of the energy promoter's theory and ideas, in terms of how they are utilized in order to eliminate flow separations.

The next topic would be sourcing upon the dimensions and geometry of diffusers that can be improve by the utilization of the energy promoter. Through literature reviews, the exact dimensions of complex S-shaped diffusers were not successfully obtained, therefore, a typical S-shaped diffuser is used, and the dimension is shortened to mimic an aggressive diffuser design. These topics will contribute to the first step of this project, in obtaining data to design and begin the simulation testing.

By taking up a typical S-shape diffuser design, the diffuser is simulated, shortened and simulated again to create flow separation. Then the diffuser is simulated with various configurations of energy promoters to obtain the result of its effect. Looking at the result, we can conclude that Energy Promoter is able to create a higher momentum fluid directed at the boundary layer in order to reduce its adverse effect against the diffuser's performance. Since the experiment is performed in two dimensional, a similar simulation should be conducted in 3D CFD to further investigate its effect and relate them to this 2D simulation.

It can be concluded that, in this final year project, studies and simulation done shows that energy promoters can be utilized to accelerate the fluid's momentum before the boundary layer separation, to re-energize the boundary layer, and reduce its adverse effect on the diffuser's performance. The result obtained is not 100 % recovered performance, with unknown relation of diffuser's design in terms of shape and diffusion rate's relation towards the effectiveness of different configuration and shape of energy promoter, which require further studies and analysis.

However, on the experimental side of the testing of intermediate turbine diffusers downstream of HP turbine stages, it is necessary to capture all effects occurring in a real aero engine. The test data is merely used to validate new designs and calibrate numerical tools for the engine design process. The further step is to test diffuser setups together with up and downstream component like in the work of Antonov et al. [16] (unsteady simulations of transient flows).

REFERENCE

[1] David Balmer: <http://www.see.ed.ac.uk/~johnc/teaching/fluidmechanics4/2003-04/fluids14/separation.html>.

[2] Peter Bradshaw and Rabi Mehta's: <http://www-htgl.stanford.edu/bradshaw/tunnel/wadiffuser.html>.

[3] N.F.Zulkefli and K.A. Ahmad: University Science Malaysia, *Numerical Simulation of The Effect Of Streamwise Vortices On Turbulent Flow Structure*. Universiti Sains Malaysia.

[4] Olaf Sieker and Joerg R. Seume: *Influence of Rotating Wakes on Separation in Turbine Exhaust Diffusers.* Leibniz University Hannover.

[5] Keiko Fukudome, Masashi Watanabe, Akiyoshi Iida and Akisatu Mizuno: *Separation Control of High Airfoil for Vertical Axis Wind Turbines.* Kogakuin University, Japan.

[6] Lord WK, MacMartin DG, Tillman G. *Flow control opportunities in gas turbine engines*. AIAA paper AIAA-2000-2234, 2000.

[7] Graziosi P, Kirtley KR. *High-area-ratio inter-tuurbine duct with inlet blowing.* United States patent 7137245 B2, 2006.
<http://www.freepatentsonline.com/7137245.html>.

[8] Widenhoefer JF. Graziosi P, Kirtley KR. *Multi-slot inter-turbine duct assembly for use in a turbine engine*. United States Patent 7549282 B2, 2009. <http://www.freepatentsonline.com/7549282.html>.

[9] EU Transport Research: *Aggressive Intermediate Duct Aerodynamics for Competitive and Environmentally Friendly Jet Engines*. <http://ec.europa.eu/research/transport/projects/article_3683_en.html>.

[10] Lin JC. *Review of research on low-profile vortex generators to control boundary-layer separation.* Prog. Aerosp Sci. 2002;38:389-420.

[11] Wallin F, Erriksson L-E. *A tuning-free body-force vortex generator model.* AIAA paper AIAA-2006-0873, 2006.

[12] Wallin F, Eriksson L-E. *Design of an aggressive flow controlled turbine duct,* ASME paper GT2008-51202, 2008.

[13] Santer C, Gottlich E, Marn A, Hubinka J, Paradiso B. *The application of low-profile vortex generators in an intermediate turbine diffuser.* ASME paper GT2010-22892, 2010.

[14] Canepa E, Lengani D, Satta F, Spano E, Ubaldi M, Zunino P. *Boundary layer separation on a flat plate with adverse pressure grandients using vortex generators.* ASME paper GT-2006-90809, 2006.

[15] *Table of physical properties of Flue Gases* <http://www.pipeflowcalculations.com/tables/flue-gas.php>.

[16] Antonov SS, Sadovnichii VN, Chernyakov VA. *The operating characteristics of an intermediate diffuser between turbine stages under transient conditions.* Therm Eng 1996;43:471-5.

APPENDIXES A

Meshings

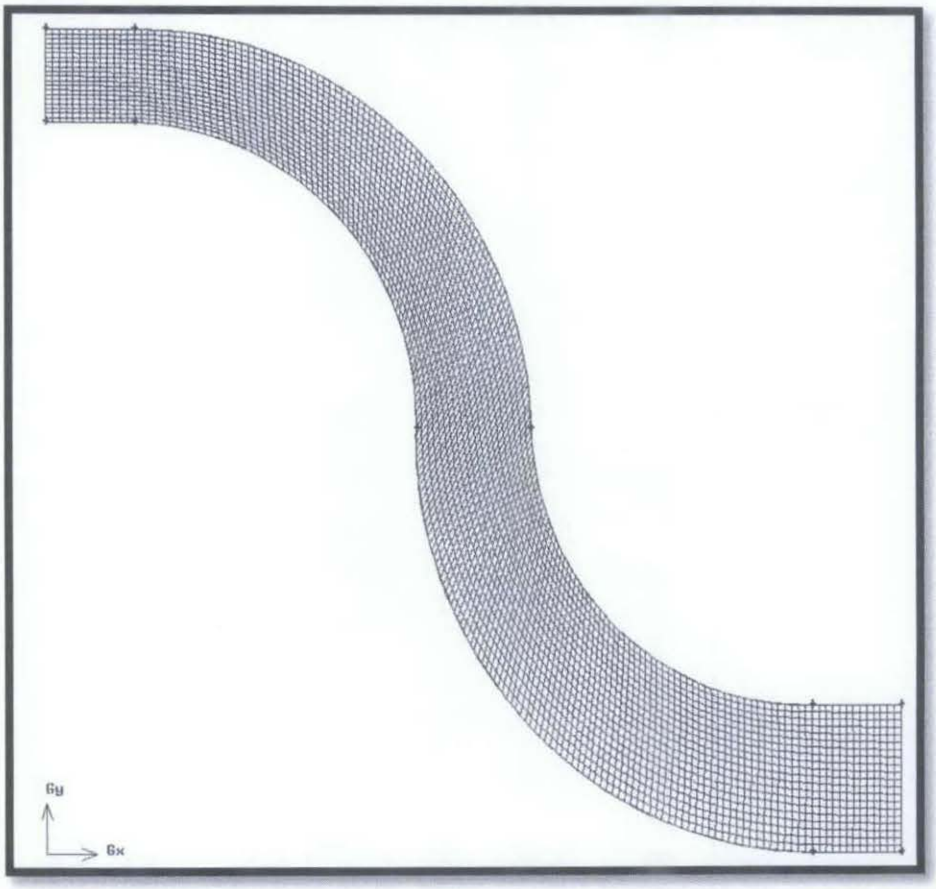


Figure A-1: Normal Diffuser's Meshing

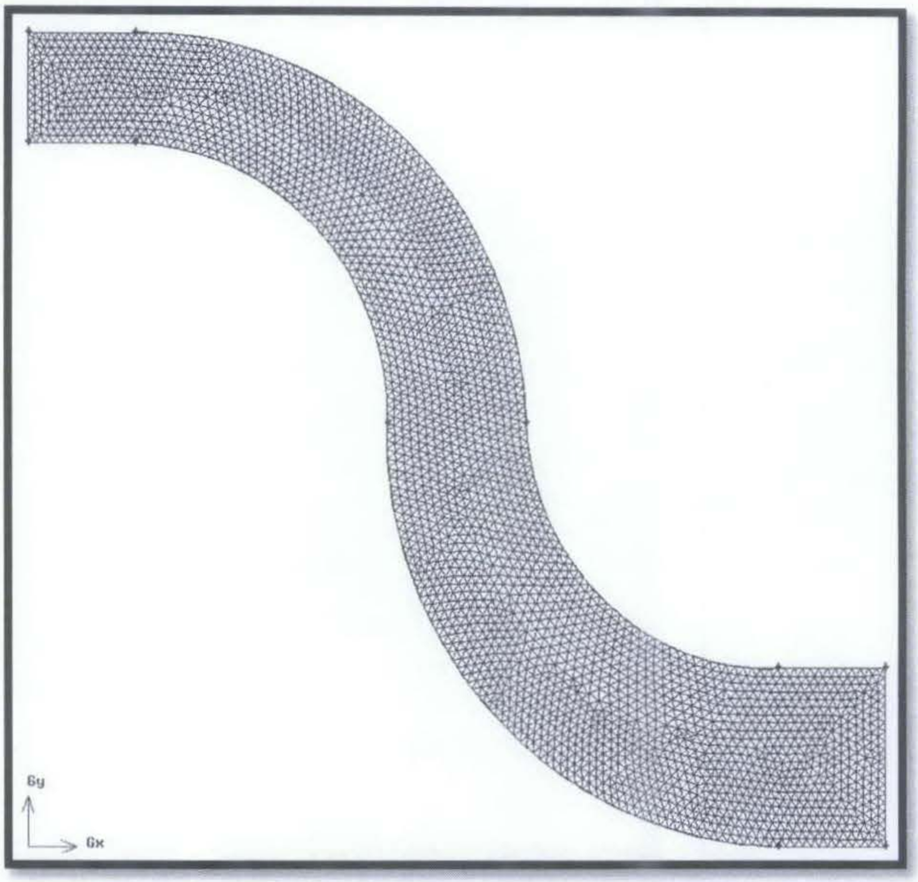


Figure A-2: Aggressive Diffuser's Meshing

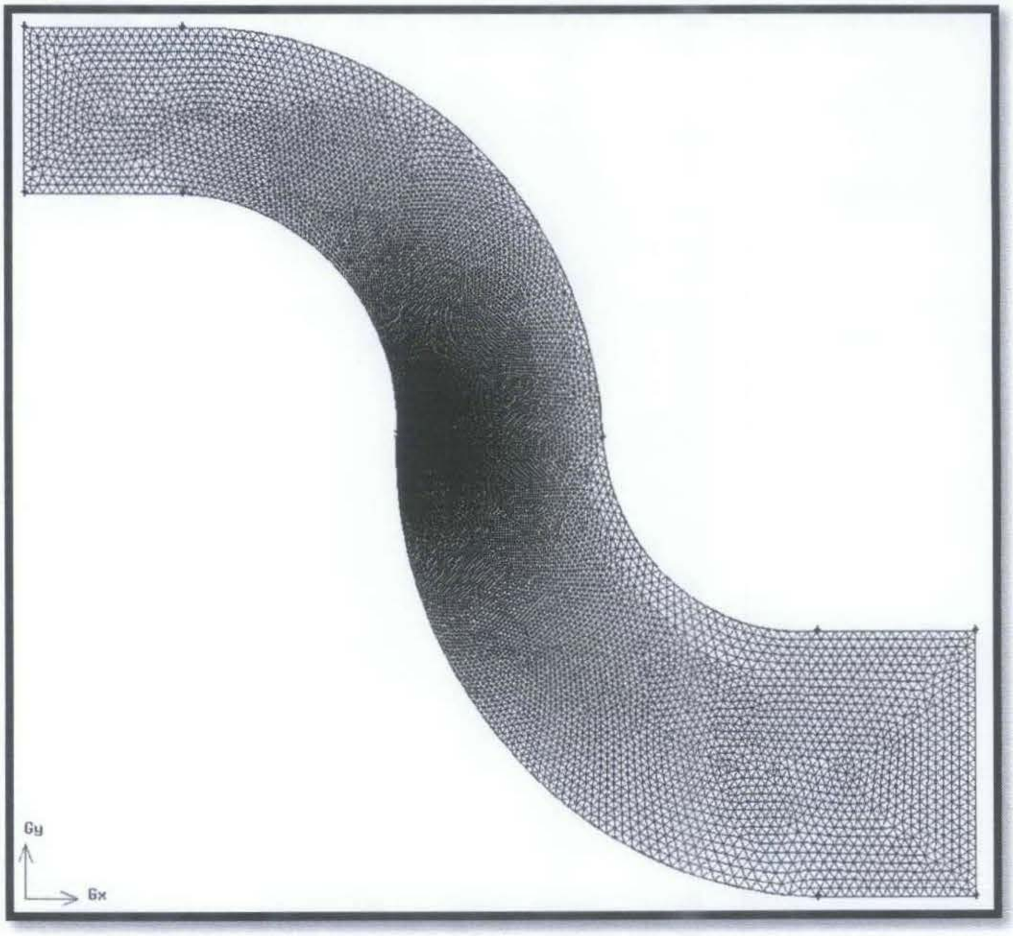


Figure A-3: Aggressive Diffuser with Energy Promoter's Meshing

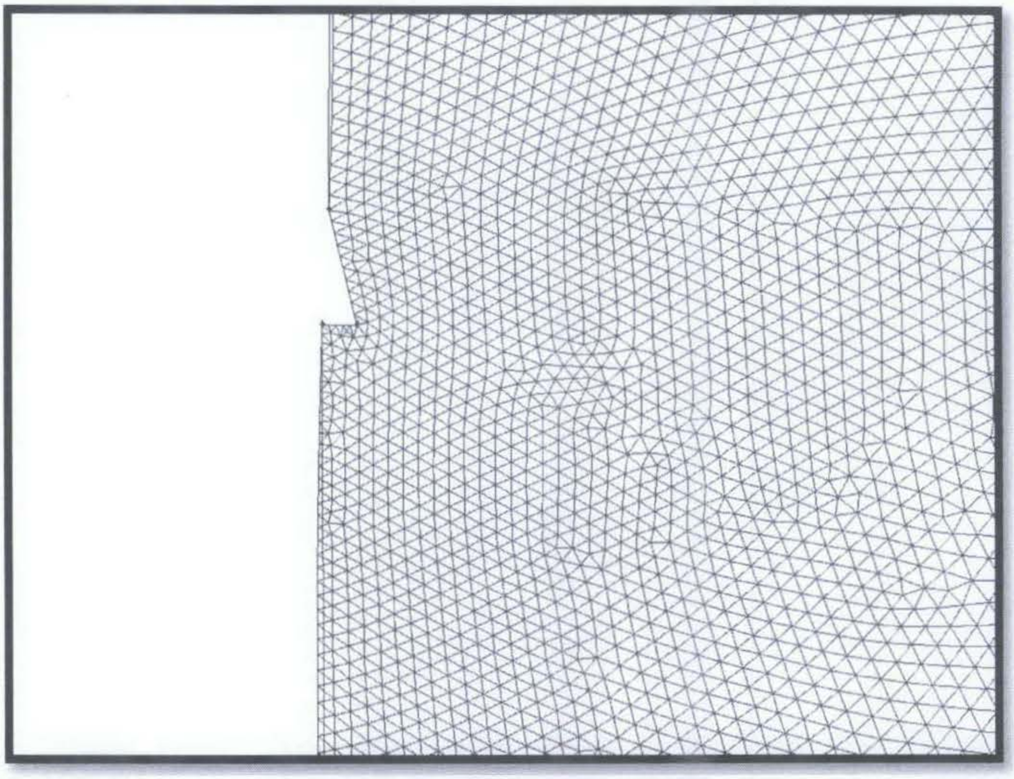


Figure A-4: Energy Promoter's Meshing

APPENDIXES B

Results

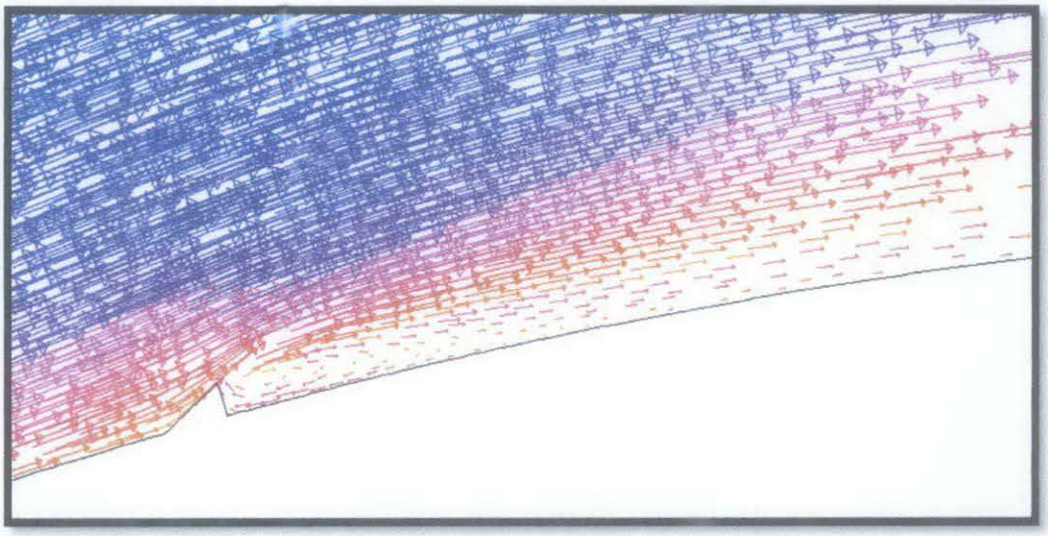


Figure B-1: Velocity Vector at Energy Promoter

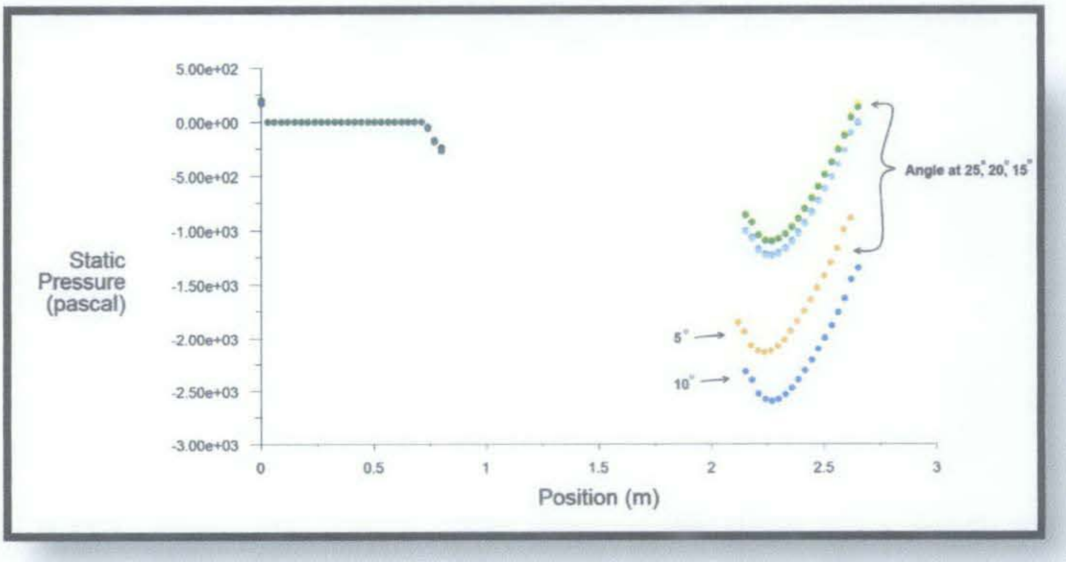


Figure B-2: Inlet and Exit Static Pressure with Various Energy Promoter's Position

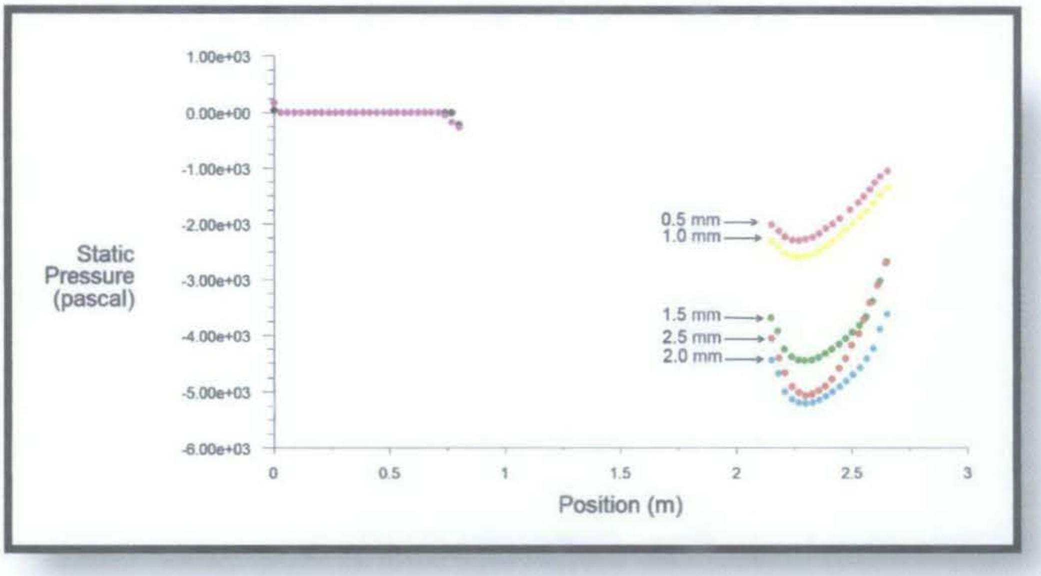


Figure B-3: Inlet and Exit Static Pressure with Various Energy Promoter's Height